® 2024 ANSYS, Inc. or its affiliated companies
Unauthorized use, distribution, or duplication is prohibited.

Ansys Fluent User's Guide

Ansys

ANSYS, Inc.
Southpointe
2600 Ansys Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
http://www.ansys.com
(T) 724-746-3304
(F) 724-514-9494

Release 2024 R2
July 2024

ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001: 2015 companies.

Copyright and Trademark Information

© 2024 ANSYS, Inc. Unauthorized use, distribution or duplication is prohibited.

ANSYS, Ansys Workbench, AUTODYN, CFX, FLUENT and any and all ANSYS, Inc. brand, product, service and feature names, logos and slogans are registered trademarks or trademarks of ANSYS, Inc. or its subsidiaries located in the United States or other countries. ICEM CFD is a trademark used by ANSYS, Inc. under license. CFX is a trademark of Sony Corporation in Japan. All other brand, product, service and feature names or trademarks are the property of their respective owners. FLEXlm and FLEXnet are trademarks of Flexera Software LLC.

Disclaimer Notice

THIS ANSYS SOFTWARE PRODUCT AND PROGRAM DOCUMENTATION INCLUDE TRADE SECRETS AND ARE CONFIDENTIAL AND PROPRIETARY PRODUCTS OF ANSYS, INC., ITS SUBSIDIARIES, OR LICENSORS. The software products and documentation are furnished by ANSYS, Inc., its subsidiaries, or affiliates under a software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, compliance with exporting laws, warranties, disclaimers, limitations of liability, and remedies, and other provisions. The software products and documentation may be used, disclosed, transferred, or copied only in accordance with the terms and conditions of that software license agreement.

ANSYS, Inc. and ANSYS Europe, Ltd. are UL registered ISO 9001: 2015 companies.

U.S. Government Rights

For U.S. Government users, except as specifically granted by the ANSYS, Inc. software license agreement, the use, duplication, or disclosure by the United States Government is subject to restrictions stated in the ANSYS, Inc. software license agreement and FAR 12.212 (for non-DOD licenses).

Third-Party Software

See the legal information in the product help files for the complete Legal Notice for ANSYS proprietary software and third-party software. If you are unable to access the Legal Notice, contact ANSYS, Inc.

Published in the U.S.A.

Table of Contents

I. Getting Started 139
1. Introduction to Ansys Fluent 141
1.1. The Ansys Product Improvement Program 143
1.2. Program Capabilities 147
1.3. Known Limitations in Ansys Fluent 2024 R2 150
1.4. Supported Third-Party Software 169
2. Basic Steps for CFD Analysis using Ansys Fluent 171
2.1. Steps in Solving Your CFD Problem 171
2.2. Planning Your CFD Analysis 171
3. Guide to a Successful Simulation Using Ansys Fluent 177
4. Starting and Executing Ansys Fluent 179
4.1. Starting Ansys Fluent 179
4.1.1. Selecting the Licensing Level 179
4.1.2. Starting Ansys Fluent Using Fluent Launcher 180
4.1.2.1. Setting General Options in Fluent Launcher 184
4.1.2.2. Single-Precision and Double-Precision Solvers 185
4.1.2.3. Setting Parallel Options in Fluent Launcher 186
4.1.2.4. Setting Remote Options in Fluent Launcher 187
4.1.2.5. Setting Scheduler Options in Fluent Launcher 188
4.1.2.6. Setting Environment Options in Fluent Launcher 188
4.1.3. Starting Ansys Fluent on a Windows System 189
4.1.4. Starting Ansys Fluent on a Linux System 191
4.1.5. Command Line Startup Options 191
4.1.5.1. ACT Option 195
4.1.5.2. Application Option 195
4.1.5.3. Application Script Option 195
4.1.5.4. Graphics and Files Options 196
4.1.5.5. Meshing Mode Option 197
4.1.5.6. Performance Options 197
4.1.5.7. Parallel Options 198
4.1.5.8. Postprocessing Option 199
4.1.5.9. Remote Visualization Options 200
4.1.5.10. Scheduler Options 200
4.1.5.11. Text Command Option 201
4.1.5.12. Version, Release Options, and Environment Variables 202
4.1.5.13. System Coupling Options 202
4.1.5.14. Other Startup Options 203
4.1.6. Aborting During Startup 203
4.2. Running Ansys Fluent in Batch Mode 203
4.2.1. Background Execution on Linux Systems 204
4.2.2. Background Execution on Windows Systems 205
4.2.3. Batch Execution Options 206
4.3. Switching Between Meshing and Solution Modes 207
4.4. Checkpointing an Ansys Fluent Simulation 208
4.5. Cleaning Up Processes From an Ansys Fluent Simulation 209
4.6. Exiting Ansys Fluent 210
Glossary of Terms 211
II. Meshing Mode 215
1. Introduction to Meshing Mode in Fluent 217

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.1. Meshing Approach 217
1.2. Meshing Mode Capabilities 218
1.3. Starting Fluent in Meshing Mode 218
1.3.1. Starting the Dual Process Build 219
1.3.2. Dynamically Spawning Processes Between Fluent Meshing and Fluent Solution Modes 220
1.4. Graphical User Interface 221
1.4.1. User Interface Components 222
1.4.1.1. The Ribbon 223
1.4.1.2. The Workflow Tab 229
1.4.1.3. The Outline View Tab 230
1.4.1.4. The Graphics Window 238
1.4.1.5. Quick Search 238
1.4.1.6. The Console 239
1.4.1.7. The Toolbars 239
1.4.1.7.1. Pointer Tools 240
1.4.1.7.2. View Tools 241
1.4.1.7.3. Graphics Effects Tools 242
1.4.1.7.4. Mesh Display Tools 243
1.4.1.7.5. Visibility Tools 243
1.4.1.7.6. Copy Tools 244
1.4.1.7.7. Object Selection/Display Tools 244
1.4.1.7.8. Filter Toolbar 245
1.4.1.7.9. CAD Tools 245
1.4.1.7.10. Tools 246
1.4.1.7.11. Context Toolbar 246
1.4.1.8. ACT Start Page 246
1.4.2. Customizing the User Interface 246
1.4.3. Setting User Preferences/Options 247
1.4.4. Using the Help System 249
1.4.4.1. Help for Text Interface Commands 249
1.4.4.2. Obtaining a Listing of Other License Users 249
1.5. Text User Interface 250
1.6. Reading and Writing Files 250
1.6.1. Shortcuts for Reading and Writing Files 250
1.6.1.1. Binary Files 251
1.6.1.2. Reading and Writing Compressed Files 251
1.6.1.2.1. Reading Compressed Files 251
1.6.1.2.2. Writing Compressed Files 251
1.6.1.3. Tilde Expansion (LINUX Systems Only) 252
1.6.1.4. Disabling the Overwrite Confirmation Prompt 252
1.6.2. Mesh Files 252
1.6.2.1. Reading Mesh Files 253
1.6.2.1.1. Reading Multiple Mesh Files 254
1.6.2.1.2. Reading 2D Mesh Files in the 3D Version of Fluent 254
1.6.2.2. Reading Boundary Mesh Files 255
1.6.2.3. Reading Faceted Geometry Files from Ansys Workbench in Fluent 255
1.6.2.4. Appending Mesh Files 255
1.6.2.5. Writing Mesh Files 256
1.6.2.6. Writing Boundary Mesh Files 257
1.6.3. Case Files 257
1.6.3.1. Reading Case Files 257

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.6.3.1.1. Reading Files Using the Legacy Format 258
1.6.3.2. Writing Case Files 258
1.6.3.2.1. Writing Files Using the Legacy Format 259
1.6.4. Reading and Writing Size-Field Files 260
1.6.5. Reading Scheme Source Files 260
1.6.6. Creating and Reading Journal Files 260
1.6.7. Creating Transcript Files 262
1.6.8. Reading and Writing Domain Files 263
1.6.9. Importing Files 263
1.6.9.1. Importing CAD Files 264
1.6.10. Saving Picture Files 271
1.6.10.1. Using the Save Picture Dialog Box 272
2. Getting Started with the Fluent Guided Workflows 277
2.1. Prerequisites for the Fluent Guided Workflows 278
2.2. Limitations of the Fluent Guided Workflows 279
2.3. Customizing Workflows 280
2.3.1. Customizing Your Journal Files Using Python 281
2.4. Understanding Task States 284
2.5. Operating on Tasks 285
2.6. Grouping Tasks 285
2.7. Editing Tasks 286
2.8. Monitoring Task Updates 287
2.9. Accessing Advanced Options 287
2.10. Filtering Lists and Using Wildcards 287
2.11. Saving and Loading Workflows 288
2.12. Setting Preferences for Workflows 289
2.13. Getting Help for Workflow Tasks 292
3. Using the Watertight Geometry Meshing Guided Workflows 293
3.1. Watertight Geometry Workflow Limitations 294
3.2. Importing Geometries 298
3.3. Importing Body of Influence Geometries 301
3.4. Adding Local Sizing 302
3.5. Generating the Surface Mesh 307
3.6. Setting Up Periodic Boundaries 315
3.7. Describing the Geometry 318
3.7.1. Best Practices for Using Non-conformal Interfaces 320
3.8. Applying Share Topology 321
3.8.1. Troubleshooting Gap Marking 324
3.9. Enclosing Fluid Regions 326
3.10. Creating Regions 330
3.11. Updating Regions 331
3.12. Adding Thin Volume Meshing Controls 332
3.12.1. Thin Volume Meshing of Parallel Zones 336
3.12.2. Thin Volume Meshing of Stacked Plates 338
3.13. Adding Boundary Layers 340
3.14. Adding Multizone Controls 344
3.14.1. Understanding Multizone Meshing 348
3.14.2. Strategies for Using Multizone Meshing 349
3.14.3. Working With Boundary Layers in Multizone Meshing 350
3.15. Generating the Multizone Mesh 350
3.16. Generating the Volume Mesh 351

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.17. Updating Boundaries 357
3.18. Improving the Surface Mesh 359
3.19. Adding Boundary Types 360
3.20. Improving the Volume Mesh 361
3.21. Transforming the Volume Mesh 362
3.22. Extruding the Volume Mesh 364
3.23. Adding Linear Mesh Patterns 368
3.23.1. Creating Custom Patterns Using Scripts 371
3.23.1.1. Examples of Creating Custom Patterns Using Scripts 374
3.23.1.1.1. Pattern Example Using Explicit Name Rule 375
3.23.1.1.2. Pattern Example of Using Both Explicit and Regular Name Rule 376
3.24. Managing Zones 379
3.25. Modifying Mesh Refinement 382
3.26. Creating Local Refinement Regions 383
3.27. Running Custom Journal Commands 388
4. Using the Fault-tolerant Meshing Workflow 389
4.1. Fault-tolerant Meshing Workflow Limitations 390
4.2. Importing CAD Geometries and Managing CAD Parts 392
4.2.1. Appending CAD Files 396
4.2.2. Working with the CAD Model Tree 397
4.2.3. Working with the Meshing Model Tree 401
4.2.4. Setting Properties for Meshing Model Objects 404
4.2.4.1. Creating Automatic Meshing Objects on a Per Part Basis 404
4.2.4.2. Creating Customized Meshing Objects 407
4.2.5. Performing Operations on Meshing Model Objects 410
4.2.5.1. Performing Transformation Operations on Meshing Model Objects 411
4.2.5.1.1. Applying a Translation Transformation 412
4.2.5.1.2. Applying a Rotation Transformation 413
4.2.5.1.3. Applying a Rotation About an Axis Transformation 414
4.2.5.1.4. Applying a Scaling Transformation 415
4.2.5.1.5. Applying a Mirror Transformation 416
4.2.5.2. Performing Refaceting Operations on Meshing Model Objects 417
4.2.5.3. Performing Object Setting Operations on Meshing Model Objects 419
4.2.5.4. Listing Meshing Operations 422
4.2.6. Faceting Considerations 422
4.2.7. Setting Display Options for CAD Model and Meshing Objects 423
4.2.8. Using Hot Key Shortcuts in the Model Trees and the Graphics Window 424
4.3. Describing the Geometry and the Flow 424
4.4. Enclosing Fluid Regions 426
4.5. Creating External Flow Boundaries 429
4.6. Creating Local Refinement Regions 431
4.7. Identifying Construction Surfaces 436
4.8. Extracting Edge Features 439
4.9. Adding Thickness to Your Geometry 443
4.10. Creating Porous Regions 446
4.11. Identifying Regions 450
4.12. Defining Leakage Thresholds 452
4.13. Updating Your Region Settings 454
4.14. Choosing Mesh Control Options 456
4.14.1. Working With Wrap and Target Sizes 458
4.15. Adding Local Size Controls 459

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.16. Creating Contact Patches 461
4.17. Creating Gap Covers 463
4.18. Generating the Surface Mesh 466
4.19. Updating Boundaries 470
4.20. Describing Overset Features 471
4.20.1. Creating Collar Meshes 472
4.20.2. Creating Component Meshes 474
4.21. Adding Boundary Layers 476
4.22. Identifying Deviated Faces 480
4.23. Generating the Volume Mesh 481
4.24. Creating Overset Mesh Interfaces 485
4.25. Identifying Orphans 485
4.26. Transforming the Volume Mesh 487
4.27. Extruding the Volume Mesh 488
4.28. Managing Zones 492
4.29. Separating Contacts 494
4.30. Choosing Part Replacement Options 494
4.30.1. Appending Replacement Parts 495
4.30.2. Applying Part Replacement Settings 495
4.30.3. Adding Local Sizing for Replacement Parts 497
4.30.4. Adding Boundary Layers for Replacement Parts 497
4.30.5. Updating the Volume Mesh 497
5. Using the 2D Meshing Guided Workflow 499
5.1. 2D Meshing Workflow 499
5.1.1. Loading the CAD Geometry 499
5.1.2. Updating Regions 501
5.1.3. Updating Boundaries 502
5.1.4. Defining Global Sizing 503
5.1.5. Adding Local Sizing 504
5.1.6. Adding 2D Boundary Layers 506
5.1.7. Generating the Surface Mesh 508
5.1.8. Exporting the Fluent 2D Mesh 509
6. Improving and Examining the Mesh and its Quality 511
6.1. Improving the Mesh 511
6.1.1. Smoothing Nodes 512
6.1.1.1. Laplace Smoothing 512
6.1.1.2. Variational Smoothing of Tetrahedral Meshes 512
6.1.1.3. Skewness-Based Smoothing of Tetrahedral Meshes 512
6.1.2. Swapping 513
6.1.3. Improving the Mesh 514
6.1.4. Removing Slivers from a Tetrahedral Mesh 514
6.1.4.1. Automatic Sliver Removal 515
6.1.4.2. Removing Slivers Manually 515
6.1.5. Modifying Cells 516
6.1.5.1. Using the Modify Cells Dialog Box 516
6.1.6. Moving Nodes 519
6.1.6.1. Automatic Correction 519
6.1.6.2. Semi-Automatic Correction 520
6.1.6.3. Repairing Negative Volume Cells 520
6.1.7. Cavity Remeshing 521
6.1.7.1. Tetrahedral Cavity Remeshing 521

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.1.7.2. Hexcore Cavity Remeshing 523
6.1.8. Manipulating Cell Zones 526
6.1.8.1. Active Zones and Cell Types 526
6.1.8.2. Copying and Moving Cell Zones 527
6.1.9. Separating Cell Zones 528
6.1.10. Manipulating Cell Zone Conditions 528
6.1.11. Using Domains to Group and Mesh Boundary Faces 528
6.1.11.1. Using Domains 529
6.1.11.2. Defining Domains 529
6.1.12. Checking the Mesh 530
6.1.13. Selectively Checking the Volume Mesh 531
6.1.14. Checking the Mesh Quality 534
6.1.15. Clearing the Mesh 534
6.2. Examining the Mesh 535
6.2.1. Displaying the Mesh 535
6.2.1.1. Generating the Mesh Display using Onscreen Tools 535
6.2.1.2. Generating the Mesh Display Using the Display Grid Dialog Box 537
6.2.1.2.1. Mesh Display Attributes 538
6.2.2. Controlling Display Options 541
6.2.3. Modifying and Saving the View 543
6.2.3.1. Mirroring a Non-symmetric Domain 543
6.2.3.2. Controlling Perspective and Camera Parameters 544
6.2.4. Composing a Scene 545
6.2.4.1. Changing the Display Properties 545
6.2.4.2. Transforming Geometric Entities in a Scene 545
6.2.4.3. Adding a Bounding Frame 546
6.2.4.4. Using the Scene Description Dialog Box 546
6.2.5. Controlling the Mouse Buttons 548
6.2.6. Controlling the Mouse Probe Function 551
6.2.7. Annotating the Display 553
6.2.8. Setting Default Controls 553
6.3. Determining Mesh Statistics and Quality 553
6.3.1. Performing Meshing Diagnostics 554
6.3.1.1. Performing a Mesh Check and Diagnostics 555
6.3.1.2. Diagnosing the Mesh Quality 556
6.3.1.3. Using the Diagnostics Tools Dialog Box 557
6.3.1.4. Setting the Scope of Your Diagnostics 561
6.3.2. Determining Mesh Statistics 563
6.3.3. Determining Mesh Quality 563
6.3.3.1. Determining Surface Mesh Quality 563
6.3.3.2. Determining Volume Mesh Quality 564
6.3.3.3. Determining Boundary Cell Quality 565
6.3.3.4. Quality Measure 566
6.3.4. Reporting Mesh Information 575
7. Advanced Meshing Topics 577
7.1. CAD Assemblies 577
7.1.1. CAD Assemblies Tree 578
7.1.1.1. FMDB File 579
7.1.1.2. CAD Entity Path 579
7.1.1.3. CAD Assemblies Tree Options 579
7.1.2. Visualizing CAD Entities 580

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.1.3. Updating CAD Entities 581
7.1.4. Manipulating CAD Entities 581
7.1.4.1. Creating and Modifying Geometry/Mesh Objects 582
7.1.4.2. Managing Labels 582
7.1.4.3. Setting CAD Entity States 583
7.1.4.4. Modifying CAD Entities 583
7.1.5. CAD Association 584
7.2. Size Functions and Scoped Sizing 584
7.2.1. Types of Size Functions or Scoped Sizing Controls 585
7.2.1.1. Curvature 586
7.2.1.2. Proximity 587
7.2.1.3. Meshed 590
7.2.1.4. Hard 590
7.2.1.5. Soft 591
7.2.1.6. Body of Influence 592
7.2.2. Defining Size Functions 593
7.2.2.1. Creating Default Size Functions 594
7.2.3. Defining Scoped Sizing Controls 594
7.2.3.1. Size Control Files 595
7.2.4. Computing the Size Field 595
7.2.4.1. Size Field Files 595
7.2.4.2. Using Size Field Filters 596
7.2.4.3. Visualizing Sizes 597
7.2.5. Using the Size Field 598
7.3. Objects and Material Points 600
7.3.1. Objects 600
7.3.1.1. Object Attributes 601
7.3.1.1.1. Creating Objects 603
7.3.1.2. Object Entities 604
7.3.1.2.1. Using Face Zone Labels 605
7.3.1.3. Managing Objects 606
7.3.1.3.1. Using hotkeys and onscreen tools 606
7.3.1.3.1.1. Creating Objects for CAD Entities 606
7.3.1.3.1.2. Creating Objects for Unreferenced Zones 607
7.3.1.3.1.3. Creating Multiple Objects 607
7.3.1.3.1.4. Easy Object Creation and Modification 608
7.3.1.3.1.5. Changing Object Properties 608
7.3.1.3.1.6. Automatic Alignment of Objects 608
7.3.1.3.1.7. Remeshing Geometry Objects 609
7.3.1.3.1.8. Creating Edge Zones 609
7.3.1.3.2. Using the Manage Objects Dialog Box 609
7.3.1.3.2.1. Defining Objects 610
7.3.1.3.2.2. Object Manipulation Operations 611
7.3.1.3.2.3. Object Transformation Operations 612
7.3.2. Material Points 613
7.3.2.1. Creating Material Points 615
7.4. Object-Based Surface Meshing 616
7.4.1. Surface Mesh Processes 617
7.4.2. Preparing the Geometry 619
7.4.2.1. Using a Bounding Box 619
7.4.2.2. Closing Annular Gaps in the Geometry 619

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.2.3. Patching Tools 620
7.4.2.3.1. Using the Patch Options Dialog Box 620
7.4.2.3.2. Using the Loop Selection Tool 624
7.4.2.4. Using User-Defined Groups 625
7.4.3. Diagnostic Tools 625
7.4.3.1. Geometry Issues 626
7.4.3.2. Face Connectivity Issues 626
7.4.3.3. Quality Checking 628
7.4.3.4. Summary 629
7.4.4. Connecting Objects 629
7.4.4.1. Using the **Join/Intersect** Dialog Box 632
7.4.4.2. Using the **Join** Dialog Box 633
7.4.4.3. Using the **Intersect** Dialog Box 634
7.4.5. Advanced Options 634
7.4.5.1. Object Management 634
7.4.5.2. Removing Gaps Between Mesh Objects 635
7.4.5.3. Removing Thickness in Mesh Objects 637
7.4.5.4. Sewing Objects 639
7.4.5.4.1. Resolving Thin Regions 641
7.4.5.4.2. Processing Slits 641
7.4.5.4.3. Removing Voids 641
7.5. Object-Based Volume Meshing 642
7.5.1. Volume Mesh Process 642
7.5.2. Volumetric Region Management 643
7.5.2.1. Computing and Verifying Regions 644
7.5.2.2. Volumetric Region Operations 646
7.5.3. Generating the Volume Mesh 647
7.5.3.1. Meshing All Regions Collectively Using Auto Mesh 648
7.5.3.2. Meshing Regions Selectively Using Auto Fill Volume 650
7.5.4. Cell Zone Options 651
7.6. Manipulating the Boundary Mesh 652
7.6.1. Manipulating Boundary Nodes 653
7.6.1.1. Free and Isolated Nodes 653
7.6.2. Intersecting Boundary Zones 654
7.6.2.1. Intersecting Zones 654
7.6.2.2. Joining Zones 655
7.6.2.3. Stitching Zones 657
7.6.2.4. Using the Intersect Boundary Zones Dialog Box 659
7.6.2.5. Using Shortcut Keys/Icons 660
7.6.3. Modifying the Boundary Mesh 660
7.6.3.1. Using the Modify Boundary Dialog Box 660
7.6.3.2. Operations Performed: Modify Boundary Dialog Box 661
7.6.3.3. Locally Remeshing a Boundary Zone or Faces 667
7.6.3.4. Moving Nodes 668
7.6.4. Improving Boundary Surfaces 668
7.6.4.1. Improving the Boundary Surface Quality 669
7.6.4.2. Smoothing the Boundary Surface 669
7.6.4.3. Swapping Face Edges 669
7.6.5. Refining the Boundary Mesh 669
7.6.5.1. Procedure for Refining Boundary Zones 670
7.6.6. Creating and Modifying Features 672

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.6.6.1. Creating Edge Zones 672
7.6.6.2. Modifying Edge Zones 675
7.6.6.3. Using the Feature Modify Dialog Box 676
7.6.7. Remeshing Boundary Zones 678
7.6.7.1. Creating Edge Zones 679
7.6.7.2. Modifying Edge Zones 679
7.6.7.3. Remeshing Boundary Face Zones 680
7.6.7.4. Using the Surface Retriangulation Dialog Box 680
7.6.8. Faceted Stitching of Boundary Zones 682
7.6.9. Triangulating Boundary Zones 683
7.6.10. Separating Boundary Zones 683
7.6.10.1. Separating Face Zones using Hotkeys 683
7.6.10.2. Using the Separate Face Zones dialog box 684
7.6.11. Projecting Boundary Zones 687
7.6.12. Creating Groups 687
7.6.13. Manipulating Boundary Zones 688
7.6.14. Manipulating Boundary Conditions 689
7.6.15. Creating Surfaces 690
7.6.15.1. Creating a Bounding Box 690
7.6.15.1.1. Using the Bounding Box Dialog Box 690
7.6.15.1.2. Using the Construct Geometry Tool 691
7.6.15.2. Creating a Planar Surface Mesh 692
7.6.15.2.1. Using the Plane Surface Dialog Box 693
7.6.15.3. Creating a Cylinder/Frustum 693
7.6.15.3.1. Using the Cylinder Dialog Box 696
7.6.15.3.2. Using the Construct Geometry Tool 697
7.6.15.4. Creating a Swept Surface 698
7.6.15.4.1. Using the Swept Surface Dialog Box 698
7.6.15.5. Creating a Revolved Surface 699
7.6.15.5.1. Using the Revolved Surface Dialog Box 699
7.6.15.6. Creating Periodic Boundaries 700
7.6.16. Removing Gaps Between Boundary Zones 703
7.6.17. Using the Loop Selection Tool 703
7.7. Wrapping Objects 704
7.7.1. The Wrapping Process 705
7.7.1.1. Extract Edge Zones 707
7.7.1.2. Create Intersection Loops 709
7.7.1.2.1. Individually 709
7.7.1.2.2. Collectively 709
7.7.1.3. Setting Geometry Recovery Options 710
7.7.1.4. Fixing Holes in Objects 711
7.7.1.5. Shrink Wrapping the Objects 715
7.7.1.6. Improving the Mesh Objects 718
7.7.1.7. Object Wrapping Options 719
7.7.1.7.1. Resolving Thin Regions During Object Wrapping 719
7.7.1.7.2. Detecting Holes in the Object 720
7.7.1.7.3. Improving Feature Capture For Mesh Objects 720
7.8. Creating a Mesh 720
7.8.1. Choosing the Meshing Strategy 721
7.8.1.1. Boundary Mesh Containing Only Triangular Faces 722
7.8.1.2. Mixed Boundary Mesh 723

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.8.1.3. Hexcore Mesh 724
7.8.1.4. CutCell Mesh 725
7.8.1.5. Rapid Octree Mesh 726
7.8.1.6. Additional Meshing Tasks 727
7.8.1.7. Inserting Isolated Nodes into a Tet Mesh 728
7.8.2. Using the Auto Mesh Dialog Box 731
7.8.3. Generating a Thin Volume Mesh 734
7.8.4. Generating Pyramids 735
7.8.4.1. Creating Pyramids 735
7.8.4.2. Zones Created During Pyramid Generation 737
7.8.4.3. Pyramid Meshing Problems 737
7.8.5. Creating a Non-Conformal Interface 739
7.8.5.1. Separating the Non-Conformal Interface Between Cell Zones 739
7.8.6. Creating a Heat Exchanger Zone 740
7.8.7. Parallel Meshing 741
7.8.7.1. Auto Partitioning 741
7.8.7.1.1. Availability of Graphical User Interface Options After Parallel Meshing 742
7.8.7.1.2. Availability of Text Interface Options After Parallel Meshing 744
7.8.7.2. Controlling the Threads 749
7.9. Generating Prisms 750
7.9.1. The Prism Generation Process 751
7.9.1.1. Zones Created During Prism Generation 752
7.9.2. Procedure for Creating Zone-based Prisms 753
7.9.3. Prism Meshing Options for Zone-Specific Prisms 756
7.9.3.1. Growth Options for Zone-Specific Prisms 757
7.9.3.1.1. Growing Prisms Simultaneously from Multiple Zones 757
7.9.3.1.2. Growing Prisms on a Two-Sided Wall 760
7.9.3.1.3. Ignoring Invalid Normals 760
7.9.3.1.4. Detecting Proximity and Collision 761
7.9.3.1.5. Splitting Prism Layers 764
7.9.3.1.6. Preserving Orthogonality 764
7.9.3.2. Offset Distances 765
7.9.3.3. Direction Vectors 768
7.9.3.4. Using Adjacent Zones as the Sides of Prisms 770
7.9.3.5. Post Prism Mesh Quality Improvement 773
7.9.3.5.1. Improving the Prism Cell Quality 773
7.9.3.5.2. Removing Poor Quality Cells 774
7.9.3.5.3. Improving Warp 775
7.9.4. Prism Meshing Options for Scoped Prisms 775
7.9.5. Prism Meshing Problems 778
7.10. Generating Tetrahedral Meshes 780
7.10.1. Automatically Creating a Tetrahedral Mesh 781
7.10.1.1. Automatic Meshing Procedure for Tetrahedral Meshes 781
7.10.1.2. Using the Auto Mesh Tool 783
7.10.1.3. Automatic Meshing of Multiple Cell Zones 783
7.10.1.4. Automatic Meshing for Hybrid Meshes 784
7.10.1.5. Further Mesh Improvements 785
7.10.2. Manually Creating a Tetrahedral Mesh 785
7.10.2.1. Manual Meshing Procedure for Tetrahedral Meshes 785
7.10.3. Initializing the Tetrahedral Mesh 788
7.10.3.1. Initializing Using the Tet Dialog Box 789

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.10.4. Refining the Tetrahedral Mesh 790
7.10.4.1. Using Local Refinement Regions 791
7.10.4.2. Refinement Using the Tet Dialog Box 792
7.10.5. Common Tetrahedral Meshing Problems 793
7.11. Generating the Hexcore Mesh 795
7.11.1. Hexcore Meshing Procedure 795
7.11.2. Using the Hexcore Dialog Box 797
7.11.3. Controlling Hexcore Parameters 798
7.11.3.1. Maximum or Minimum Cell Length 798
7.11.3.2. Buffer Layers 799
7.11.3.3. Peel Layers 799
7.11.3.4. Defining Hexcore Extents 800
7.11.3.4.1. Hexcore to Selected Boundaries 801
7.11.3.5. Local Refinement Regions 803
7.12. Generating Polyhedral Meshes 804
7.12.1. Meshing Process for Polyhedral Meshes 804
7.12.2. Steps for Creating the Polyhedral Mesh 805
7.12.2.1. Further Mesh Improvements 808
7.12.2.2. Transferring the Poly Mesh to Solution Mode 808
7.13. Generating Poly-Hexcore Meshes 808
7.13.1. Steps for Creating the Poly-Hexcore Mesh 808
7.14. Generating the CutCell Mesh 810
7.14.1. The CutCell Meshing Process 811
7.14.2. Using the CutCell Dialog Box 815
7.14.2.1. Handling Zero-Thickness Walls 817
7.14.2.2. Handling Overlapping Surfaces 818
7.14.2.3. Resolving Thin Regions 819
7.14.3. Improving the CutCell Mesh 820
7.14.4. Post CutCell Mesh Generation Cleanup 821
7.14.5. Generating Prisms for the CutCell Mesh 821
7.14.6. The Cut-Tet Workflow 826
7.15. Generating Rapid Octree Meshes 828
7.15.1. Using the Rapid Octree Mesher 829
7.15.1.1. Geometry 830
7.15.1.1.1. Specifying the Input Object 830
7.15.1.1.2. Specifying the Volume 830
7.15.1.1.3. Defining the Bounding Box 833
7.15.1.1.4. Reporting the Base Length 835
7.15.1.2. Boundary Treatment 835
7.15.1.2.1. Boundary Mesh Optimization 837
7.15.1.2.2. Improve Geometry Resolution 838
7.15.1.3. Mesh Parameters 840
7.15.1.3.1. Refinement Regions 842
7.15.1.3.2. Custom Boundary Sizes 844
7.15.1.3.2.1. Creating Size Functions 845
7.15.1.3.2.2. Draw, Change, and Delete Functions 845
7.15.1.3.3. Boundary Layer Mesh 845
7.15.2. Limitations of the Rapid Octree Mesher 850
A. Importing Boundary and Volume Meshes 853
A.1. GAMBIT Meshes 853
A.2. TetraMesher Volume Mesh 853

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A.3. Meshes from Third-Party CAD Packages 853
A.3.1. I-deas Universal Files 855
A.3.1.1. Recognized I-deas Datasets 855
A.3.1.2. Grouping Elements to Create Zones for a Surface Mesh 855
A.3.1.3. Grouping Nodes to Create Zones for a Volume Mesh 856
A.3.1.4. Periodic Boundaries 856
A.3.1.5. Deleting Duplicate Nodes 856
A.3.2. PATRAN Neutral Files 856
A.3.2.1. Recognized PATRAN Datasets 856
A.3.2.2. Grouping Elements to Create Zones 857
A.3.2.3. Periodic Boundaries 857
A.3.3. ANSYS Files 857
A.3.3.1. Recognized Datasets 857
A.3.3.2. Periodic Boundaries 858
A.3.4. ARIES Files 858
A.3.5. NASTRAN Files 858
A.3.5.1. Recognized NASTRAN Bulk Data Entries 858
A.3.5.2. Periodic Boundaries 859
A.3.5.3. Deleting Duplicate Nodes 859
B. Mesh File Format 861
B.1. Guidelines 861
B.2. Formatting Conventions in Binary Files and Formatted Files 861
B.3. Grid Sections 862
B.3.1. Comment 862
B.3.2. Header 863
B.3.3. Dimensions 863
B.3.4. Nodes 863
B.3.5. Periodic Shadow Faces 864
B.3.6. Cells 865
B.3.7. Faces 866
B.3.8. Edges 868
B.3.9. Face Tree 869
B.3.10. Cell Tree 870
B.3.11. Interface Face Parents 870
B.4. Non-Grid Sections 871
B.4.1. Zone 871
B.5. Example Files 873
C. Shortcut Keys 877
C.1. Shortcut Key Actions 877
C.1.1. Entity Information 888
Bibliography 891
III. Solution Mode 893
Using This Manual ..... dcccxcvii
1. Typographical Conventions ..... dcccxcvii
2. Mathematical Conventions ..... dcccxcix
1. Graphical User Interface (GUI) 901
1.1. GUI Components 901
1.1.1. The Ribbon 902
1.1.2. The Outline View 903
1.1.2.1. Drag and Drop, Copy and Paste 905
1.1.3. Graphics Windows 909

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.1.4. Quick Search 913
1.1.5. Toolbars 913
1.1.5.1. The Standard Toolbar 913
1.1.5.2. The Graphics Toolbars 914
1.1.5.2.1. Mesh Display 914
1.1.5.2.2. Pointer Tools 915
1.1.5.2.3. View Tools 917
1.1.5.2.4. Visibility Tools 918
1.1.5.2.5. Copy Tools 918
1.1.5.2.6. Object Selection/Display Tools 919
1.1.5.2.7. Graphics Effects Tools 919
1.1.5.2.8. Additional Display Options 920
1.1.6. Task Pages 920
1.1.7. The Console 921
1.1.8. Dialog Boxes 923
1.1.8.1. Input Controls 925
1.1.8.1.1. Tabs 925
1.1.8.1.2. Buttons 925
1.1.8.1.3. Check Boxes 925
1.1.8.1.4. Radio Buttons 925
1.1.8.1.5. Text Entry Boxes 926
1.1.8.1.6. Integer Number Entry Boxes 926
1.1.8.1.7. Real Number Entry Boxes 926
1.1.8.1.8. Filter Text Entry Boxes 926
1.1.8.1.9. Single-Selection Lists 927
1.1.8.1.10. Multiple-Selection Lists 927
1.1.8.1.11. Drop-Down Lists 928
1.1.8.1.12. Scales 929
1.1.8.2. Types of Dialog Boxes 929
1.1.8.2.1. Information Dialog Boxes 929
1.1.8.2.2. Warning Dialog Boxes 930
1.1.8.2.3. Error Dialog Boxes 930
1.1.8.2.4. The Working Dialog Box 930
1.1.8.2.5. Question Dialog Box 931
1.1.8.2.6. The Select File Dialog Box 931
1.1.9. Quick Property Editor for Boundaries 934
1.2. Customizing the Graphical User Interface 935
1.3. Setting User Preferences/Options 937
1.4. Fluent Graphical User Interface Other Languages 938
1.5. Having the Session Close After Sitting Idle 941
1.5.1. Timeout Using the Set Idle Timeout Dialog Box 941
1.5.2. Timeout Using FLUENT_MAX_IDLE_TIMEOUT 942
1.5.3. Idle Timeout Limitations 942
1.6. Using the Help System 942
1.6.1. Learning and Support 943
1.6.2. Task Page and Dialog Box Help 943
1.6.3. Obtaining License Use Information 943
1.6.4. Version and Release Information 943
2. Text User Interface (TUI) 945
3. Reading and Writing Files 947
3.1. Shortcuts for Reading and Writing Files 948

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.1.1. Default File Suffixes 948
3.1.2. Binary Files 949
3.1.3. Detecting File Format 950
3.1.4. Recent File List 950
3.1.5. Reading and Writing Compressed Files 950
3.1.5.1. Reading Compressed Files 950
3.1.5.2. Writing Compressed Files 951
3.1.6. Tilde Expansion (Linux Systems Only) 952
3.1.7. Automatic Numbering of Files 952
3.1.8. Disabling the Overwrite Confirmation Prompt 953
3.2. Reading Mesh Files 953
3.3. Reading and Writing Case and Data Files 954
3.3.1. Reading and Writing Case Files 956
3.3.2. Reading and Writing Data Files 957
3.3.3. Reading and Writing Case and Data Files Together 957
3.3.4. Reading and Writing Files in the Legacy Format 958
3.3.5. Automatic Saving of Case and Data Files 958
3.4. Reading Settings Only for Mesh or Case Files with Large Cell Counts 962
3.5. Reading Fluent/UNS and RAMPANT Case and Data Files 963
3.6. Reading and Writing Profile Files 964
3.6.1. Reading Profile Files 964
3.6.2. Writing Profile Files 964
3.6.2.1. Writing Circumferential-Averaged Profiles 967
3.7. Reading Files in Tabular Format 969
3.7.1. Matrix Table Files 970
3.7.2. Real Gas Property (RGP) Table Files 970
3.8. Reading and Writing Boundary Conditions 971
3.9. Writing a Boundary Mesh 972
3.10. Reading Scheme Source Files 972
3.11. Creating and Reading Journal Files 972
3.11.1. Procedure 974
3.11.2. Multiple Journal Files 975
3.12. Creating Transcript Files 976
3.13. Importing Files 976
3.13.1. ABAQUS Files 979
3.13.2. CFX Files 979
3.13.3. Meshes and Data in CGNS Format 980
3.13.4. EnSight Files 981
3.13.5. GAMBIT and GeoMesh Mesh Files 981
3.13.6. HYPERMESH ASCII Files 982
3.13.7. NASTRAN Files 982
3.13.8. PLOT3D Files 982
3.13.9. Tecplot Files 982
3.13.10. Partition Files 983
3.13.11. CHEMKIN Mechanism 983
3.14. Exporting Solution Data 983
3.14.1. Exporting Limitations 984
3.15. Exporting Solution Data after a Calculation 985
3.15.1. ABAQUS Files 986
3.15.2. Mechanical APDL Input Files 987
3.15.3. ASCII Files 988

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.15.4. CDAT for CFD-Post and EnSight 988
3.15.5. CGNS Files 989
3.15.6. Common Fluids Format - Post Files 991
3.15.7. EnSight Case Gold Files 992
3.15.8. EnSight DVS 995
3.15.9. FAST Files 996
3.15.10. FAST Solution Files 996
3.15.11. FieldView Unstructured Files 996
3.15.12. NASTRAN Files 998
3.15.13. TAITherm Files 998
3.15.14. Tecplot Files 999
3.16. Exporting Steady-State Particle History Data 999
3.17. Exporting Data During a Transient Calculation 1001
3.17.1. Creating Automatic Export Definitions for Solution Data 1003
3.17.2. Creating Automatic Export Definitions for Transient Particle History Data 1004
3.18. Exporting to Ansys CFD-Post 1007
3.19. Parallel Exporting to Ansys EnSight 1008
3.20. Managing Solution Files 1009
3.21. Mesh-to-Mesh Solution Interpolation 1010
3.21.1. Performing Mesh-to-Mesh Solution Interpolation 1010
3.21.2. Format of the Interpolation File 1012
3.22. Mapping Data for Fluid-Structure Interaction (FSI) Applications 1013
3.22.1. FEA File Formats 1014
3.22.2. Using the FSI Mapping Dialog Boxes 1014
3.23. Saving Picture Files 1019
3.23.1. Using the Save Picture Dialog Box 1020
3.23.1.1. Choosing the Picture File Format 1021
3.23.1.2. Specifying the Color Mode 1024
3.23.1.3. Choosing the File Type 1024
3.23.1.4. Defining the Resolution 1024
3.23.1.5. Picture Options 1025
3.23.2. Picture Options for PostScript Files 1025
3.23.2.1. Window Dumps (Linux Systems Only) 1025
3.23.2.2. Previewing the Picture Image 1026
3.24. Setting Data File Quantities 1026
3.25. The .fluent File 1028
3.26. Coupled Simulations in Ansys Fluent with Functional Mock-up Unit (FMU) Files 1028
4. Unit Systems 1033
4.1. Restrictions on Units 1033
4.2. Units in Mesh Files 1034
4.3. Built-In Unit Systems in Ansys Fluent 1034
4.4. Customizing Units 1035
4.4.1. Listing Current Units 1035
4.4.2. Changing the Units for a Quantity 1035
4.4.3. Defining a New Unit 1035
4.4.3.1. Determining the Conversion Factor 1036
5. Fluent Expressions Language 1037
5.1. Introduction to Expressions 1037
5.1.1. Expression Syntax 1037
5.1.1.1. Expression Data Types 1037
5.1.1.2. Expression Values 1038

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.1.1.3. Expression Operations and Functions 1038
5.1.2. Units Validation 1044
5.2. Expression Sources 1045
5.2.1. Field Variables 1045
5.2.2. Solution Variables 1045
5.2.3. Scientific Constants 1046
5.2.4. Aliases 1047
5.2.5. Profiles 1048
5.3. Creating and Using Expressions 1049
5.3.1. Directly Applied Expressions 1050
5.3.1.1. Expressions for Cell Zones and Boundary Conditions 1050
5.3.1.2. Expressions for Material Properties 1053
5.3.1.3. Expressions in the Console 1054
5.3.2. Named Expressions 1054
5.3.3. Context Specification 1060
5.3.4. Plotting Expressions 1060
5.3.5. Postprocessing Expressions 1062
5.3.6. Expression Manager 1063
5.4. Expression Examples 1064
5.4.1. Parabolic Inflow Profile 1064
5.4.2. Time-Varied Parabolic Inflow 1068
5.4.3. Controlled Outlet Temperature 1071
5.4.4. Computing Forces with Parameterized Angle of Attack 1075
5.5. Appendix: Supported Field Variables 1078
6. Reading and Manipulating Meshes 1129
6.1. Mesh Topologies 1129
6.1.1. Examples of Acceptable Mesh Topologies 1131
6.1.2. Face-Node Connectivity in Ansys Fluent 1136
6.1.2.1. Face-Node Connectivity for Triangular Cells 1137
6.1.2.2. Face-Node Connectivity for Quadrilateral Cells 1138
6.1.2.3. Face-Node Connectivity for Tetrahedral Cells 1139
6.1.2.4. Face-Node Connectivity for Wedge Cells 1140
6.1.2.5. Face-Node Connectivity for Pyramidal Cells 1141
6.1.2.6. Face-Node Connectivity for Hex Cells 1142
6.1.2.7. Face-Node Connectivity for Polyhedral Cells 1143
6.1.3. Choosing the Appropriate Mesh Type 1143
6.1.3.1. Setup Time 1143
6.1.3.2. Computational Expense 1144
6.1.3.3. Numerical Diffusion 1145
6.2. Mesh Requirements and Considerations 1145
6.2.1. Geometry/Mesh Requirements 1145
6.2.2. Mesh Quality 1146
6.2.2.1. Mesh Element Distribution 1149
6.2.2.2. Cell Quality 1149
6.2.2.3. Smoothness 1150
6.2.2.4. Flow-Field Dependency 1150
6.3. Mesh Sources 1150
6.3.1. Ansys Meshing Mesh Files 1151
6.3.2. Fluent Meshing Mode Mesh Files 1151
6.3.3. Fluent Meshing Mesh Files 1151
6.3.4. GAMBIT Mesh Files 1151

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.5. GeoMesh Mesh Files 1152
6.3.6. NASTRAN Files 1152
6.3.6.1. Recognized NASTRAN Bulk Data Entries 1152
6.3.6.2. Deleting Duplicate Nodes 1153
6.3.7. CFX Files 1153
6.3.8. Using the fe2ram Filter to Convert Files 1154
6.3.9. Removing Hanging Nodes / Edges 1155
6.3.9.1. Limitations 1155
6.3.10. Fluent/UNS and RAMPANT Case Files 1155
6.3.11. Ansys FIDAP Neutral Files 1156
6.3.12. Reading Multiple Mesh/Case/Data Files 1156
6.3.12.1. Reading Multiple Mesh Files via the Solution Mode of Fluent 1157
6.3.12.2. Reading Multiple Mesh Files via the Meshing Mode of Fluent 1157
6.3.12.3. Reading Multiple Mesh Files via tmerge 1158
6.3.13. Reading Surface Mesh Files 1159
6.4. Reference Frames 1160
6.4.1. Creating and Using Reference Frames 1161
6.5. Curvilinear Coordinate Systems 1164
6.5.1. Defining a Curvilinear Coordinate System 1164
6.5.2. Curvilinear Coordinate System Example 1167
6.5.3. Limitations of Curvilinear Coordinate Systems 1168
6.5.4. Limitations with Jump Boundary 1168
6.6. Non-Conformal Meshes 1170
6.6.1. Non-Conformal Mesh Calculations 1170
6.6.1.1. The Periodic Boundary Condition Option 1172
6.6.1.2. The Periodic Repeats Option 1174
6.6.1.3. The Coupled Wall Option 1176
6.6.1.4. Matching Option 1177
6.6.1.5. The Mapped Option 1178
6.6.1.6. The Static Option 1180
6.6.1.7. Interface Zones Automatic Naming Conventions 1180
6.6.1.7.1. Default (No Options Enabled) 1181
6.6.1.7.2. Periodic Boundary Condition 1181
6.6.1.7.3. Periodic Repeats 1181
6.6.1.7.4. Coupled Wall 1181
6.6.1.7.5. Matching 1182
6.6.1.7.6. Mapped 1182
6.6.1.7.7. Static 1182
6.6.2. Non-Conformal Interface Algorithm 1183
6.6.3. Requirements and Limitations of Non-Conformal Meshes 1183
6.6.4. Using a Non-Conformal Mesh in Ansys Fluent 1185
6.6.4.1. Manually Creating Many-to-Many Mesh Interfaces 1201
6.6.4.2. Manually Creating Many Non-Overlapping Mesh Interfaces 1204
6.6.4.3. Transferring Motion Across a Mesh Interface 1205
6.7. Overset Meshes 1206
6.7.1. Introduction 1207
6.7.2. Overset Topologies 1208
6.7.3. Overset Domain Connectivity 1212
6.7.3.1. Hole Cutting 1212
6.7.3.1.1. Hole Cutting Control 1213
6.7.3.2. Overlap Minimization 1214

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.7.3.3. Donor Search 1216
6.7.4. Diagnosing Overset Interface Issues 1216
6.7.4.1. Flood Filling Fails During Hole Cutting 1217
6.7.4.1.1. Incorrect Seed Cells 1217
6.7.4.1.2. Leakage Between Overlapping Boundaries 1217
6.7.4.2. Donor Search Fails Due to Orphan Cells 1218
6.7.5. Overset Mesh Adaption 1219
6.7.5.1. Marking for Orphan Adaption 1219
6.7.5.2. Marking for Size Adaption 1219
6.7.5.3. Marking for Gap Adaption 1219
6.7.5.4. Using Manual Overset Adaption 1220
6.7.5.5. Using Automatic Overset Adaption 1220
6.7.5.6. Overset Adaption Controls 1221
6.7.6. Overset Meshing Best Practices 1222
6.7.7. Overset Meshing Limitations and Compatibilities 1224
6.7.7.1. Limitations 1224
6.7.7.2. Compatibilities 1225
6.7.8. Setting up an Overset Interface 1227
6.7.9. Postprocessing Overset Meshes 1229
6.7.9.1. Overset Mesh Display 1229
6.7.9.2. Overset Field Functions 1230
6.7.9.3. Overset Cell Marks 1233
6.7.9.4. Overset Interface Listing 1233
6.7.9.5. Overset Postprocessing Limitations 1233
6.7.10. Writing and Reading Overset Files 1234
6.8. Controlling Flow in Narrow Gaps for Valves and Pumps 1234
6.8.1. The Gap Model Approach 1234
6.8.2. Limitations of the Gap Model 1235
6.8.3. Recommendations for the Setup of a Simulation with Gaps 1236
6.8.4. Using the Gap Model 1236
6.9. Checking the Mesh 1245
6.9.1. Mesh Check Report 1246
6.9.2. Repairing Meshes 1247
6.10. Reporting Mesh Statistics 1250
6.10.1. Mesh Size 1250
6.10.2. Memory Usage 1251
6.10.2.1. Linux Systems 1252
6.10.2.2. Windows Systems 1252
6.10.3. Mesh Zone Information 1252
6.10.4. Partition Statistics 1253
6.11. Converting the Mesh to a Polyhedral Mesh 1253
6.11.1. Converting the Domain to a Polyhedra 1253
6.11.1.1. Limitations 1257
6.11.2. Converting Skewed Cells to Polyhedra 1258
6.11.2.1. Limitations 1258
6.11.3. Converting Cells with Hanging Nodes / Edges to Polyhedra 1259
6.11.3.1. Limitations 1259
6.12. Modifying the Mesh 1260
6.12.1. Merging Zones 1261
6.12.1.1. When to Merge Zones 1261
6.12.1.2. Using the Merge Zones Dialog Box 1262

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.12.2. Separating Zones 1262
6.12.2.1. Separating Face Zones 1263
6.12.2.1.1. Methods for Separating Face Zones 1263
6.12.2.1.2. Inputs for Separating Face Zones 1263
6.12.2.2. Separating Cell Zones 1265
6.12.2.2.1. Methods for Separating Cell Zones 1265
6.12.2.2.2. Inputs for Separating Cell Zones 1265
6.12.3. Fusing Face Zones 1267
6.12.3.1. Inputs for Fusing Face Zones 1268
6.12.3.1.1. Fusing Zones on Branch Cuts 1269
6.12.4. Creating Periodic Zones and Interfaces 1269
6.12.5. Decoupling Periodic Zones 1272
6.12.6. Slitting Face Zones 1273
6.12.6.1. Inputs for Slitting Face Zones 1274
6.12.7. Orienting Face Zones 1274
6.12.8. Extruding Face Zones 1274
6.12.8.1. Specifying Extrusion by Displacement Distances 1275
6.12.8.2. Specifying Extrusion by Parametric Coordinates 1275
6.12.9. Replacing, Deleting, Deactivating, and Activating Zones 1275
6.12.9.1. Replacing Zones 1275
6.12.9.2. Deleting Zones 1277
6.12.9.3. Deactivating Zones 1277
6.12.9.4. Activating Zones 1278
6.12.10. Copying Cell Zones 1279
6.12.11. Replacing the Mesh 1279
6.12.11.1. Inputs for Replacing the Mesh 1280
6.12.11.2. Limitations 1281
6.12.12. Managing Adjacent Zones 1281
6.12.12.1. Renaming Zones Using the Adjacency Dialog Box 1283
6.12.13. Reordering the Domain 1285
6.12.14. Scaling the Mesh 1285
6.12.14.1. Scaling the Entire Mesh 1286
6.12.14.1.1. Changing the Unit of Length 1287
6.12.14.1.2. Unscaling the Mesh 1287
6.12.14.1.3. Changing the Physical Size of the Mesh 1288
6.12.14.2. Scaling Individual Cell Zones 1288
6.12.15. Translating the Mesh 1288
6.12.15.1. Translating the Entire Mesh 1288
6.12.15.2. Translating Individual Cell Zones 1289
6.12.16. Rotating the Mesh 1289
6.12.16.1. Rotating the Entire Mesh 1290
6.12.16.2. Rotating Individual Cell Zones 1290
6.12.17. Improving the Mesh by Smoothing and Swapping 1291
6.12.17.1. Smoothing 1291
6.12.17.1.1. Quality-Based Smoothing 1292
6.12.17.1.2. Laplacian Smoothing 1293
6.12.17.1.3. Skewness-Based Smoothing 1295
6.12.17.2. Face Swapping 1296
6.12.17.2.1. Triangular Meshes 1296
6.12.17.2.2. Tetrahedral Meshes 1297
6.12.17.3. Combining Skewness-Based Smoothing and Face Swapping 1297

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.12.18. Boundary Layer Redistribution 1297
6.12.19. Deleting Cells 1298
7. Cell Zone and Boundary Conditions 1299
7.1. Overview 1299
7.1.1. Available Cell Zone and Boundary Types 1300
7.1.2. The Cell Zone and Boundary Conditions Task Pages 1300
7.1.3. Changing Cell and Boundary Zone Types 1301
7.1.4. Setting Cell Zone and Boundary Conditions 1303
7.1.5. Copying Cell Zone and Boundary Conditions 1305
7.1.6. Exporting Boundary Zones in CSV Format 1306
7.1.7. Changing Cell or Boundary Zone Names 1308
7.1.8. Defining Non-Uniform Cell Zone and Boundary Conditions 1308
7.1.9. Defining and Viewing Parameters 1309
7.1.9.1. Creating a New Parameter 1311
7.1.9.2. Working With Advanced Parameter Options 1313
7.1.9.2.1. Defining Scheme Procedures With Input Parameters 1313
7.1.9.2.2. Defining UDFs With Input Parameters 1315
7.1.9.2.3. Using the Text User Interface to Define UDFs and Scheme Procedures With Input Parameters 1316
7.1.10. Selecting Cell or Boundary Zones in the Graphics Display 1317
7.1.11. Operating and Periodic Conditions 1319
7.1.12. Saving and Reusing Cell Zone and Boundary Conditions 1320
7.2. Cell Zone Conditions 1320
7.2.1. Fluid Conditions 1320
7.2.1.1. Inputs for Fluid Zones 1321
7.2.1.1.1. Defining the Fluid Material 1322
7.2.1.1.2. Defining Sources 1322
7.2.1.1.3. Defining Fixed Values 1322
7.2.1.1.4. Specifying a Laminar Zone 1322
7.2.1.1.5. Specifying a Reaction Mechanism 1323
7.2.1.1.6. Specifying the Rotation Axis 1323
7.2.1.1.7. Defining Zone Motion 1323
7.2.1.1.8. Defining Radiation Parameters 1326
7.2.2. Solid Conditions 1326
7.2.2.1. Inputs for Solid Zones 1327
7.2.2.1.1. Defining the Solid Material 1327
7.2.2.1.2. Defining a Heat Source 1327
7.2.2.1.3. Defining a Fixed Temperature 1327
7.2.2.1.4. Specifying the Rotation Axis for Boundary Zones 1328
7.2.2.1.5. Defining Zone Motion 1328
7.2.2.1.6. Defining Radiation Parameters 1333
7.2.3. Porous Media Conditions 1333
7.2.3.1. Limitations and Assumptions of the Porous Media Model 1334
7.2.3.2. Momentum Equations for Porous Media 1335
7.2.3.2.1. Darcy's Law in Porous Media 1336
7.2.3.2.2. Inertial Losses in Porous Media 1336
7.2.3.3. Relative Viscosity in Porous Media 1337
7.2.3.4. Treatment of the Energy Equation in Porous Media 1337
7.2.3.4.1. Equilibrium Thermal Model Equations 1337
7.2.3.4.2. Non-Equilibrium Thermal Model Equations 1338
7.2.3.5. Treatment of Turbulence in Porous Media 1339

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.2.3.6. Effect of Porosity on Transient Scalar Equations 1339
7.2.3.7. Modeling Porous Media Based on Physical Velocity 1340
7.2.3.7.1. Single Phase Porous Media 1340
7.2.3.7.2. Multiphase Porous Media 1341
7.2.3.7.2.1. The Continuity Equation 1342
7.2.3.7.2.2. The Momentum Equation 1342
7.2.3.7.2.3. The Energy Equation 1342
7.2.3.8. User Inputs for Porous Media 1343
7.2.3.8.1. Defining the Porous Zone 1344
7.2.3.8.2. Defining the Porous Velocity Formulation 1345
7.2.3.8.3. Defining the Fluid Passing Through the Porous Medium 1345
7.2.3.8.4. Enabling Reactions in a Porous Zone 1345
7.2.3.8.5. Including the Relative Velocity Resistance Formulation 1345
7.2.3.8.6. Defining the Viscous and Inertial Resistance Coefficients 1346
7.2.3.8.7. Deriving Porous Media Inputs Based on Superficial Velocity, Using a Known Pressure Loss 1350
7.2.3.8.8. Using the Ergun Equation to Derive Porous Media Inputs for a Packed Bed 1351
7.2.3.8.9. Using an Empirical Equation to Derive Porous Media Inputs for Turbulent Flow Through a Perforated Plate 1351
7.2.3.8.10. Using Tabulated Data to Derive Porous Media Inputs for Laminar Flow Through a Fibrous Mat 1352
7.2.3.8.11. Deriving the Porous Coefficients Based on Experimental Pressure and Velocity Data 1353
7.2.3.8.12. Using the Power-Law Model 1354
7.2.3.8.13. Defining Porosity 1354
7.2.3.8.14. Specifying the Heat Transfer Settings 1354
7.2.3.8.14.1. Equilibrium Thermal Model 1355
7.2.3.8.14.2. Non-Equilibrium Thermal Model 1355
7.2.3.8.15. Specifying the Relative Viscosity 1358
7.2.3.8.16. Specifying the Relative Permeability 1358
7.2.3.8.17. Specifying the Capillary Pressure 1363
7.2.3.8.17.1. Brooks-Corey Model 1364
7.2.3.8.17.2. Van-Genuchten Model 1364
7.2.3.8.17.3. Leverett J-Function 1364
7.2.3.8.17.4. Skjaeveland Model 1365
7.2.3.8.17.5. Capillary Pressure Data in a Tabular Format 1366
7.2.3.8.17.6. Capillary Pressure Usage 1367
7.2.3.8.17.7. Modeling Capillary Pressure as Diffusion 1369
7.2.3.8.18. Defining Sources 1370
7.2.3.8.19. Defining Fixed Values 1370
7.2.3.8.20. Suppressing the Turbulent Viscosity in the Porous Region 1370
7.2.3.8.21. Specifying the Rotation Axis and Defining Zone Motion 1370
7.2.3.9. Solution Strategies for Porous Media 1371
7.2.3.10. Postprocessing for Porous Media 1371
7.2.4. 3D Fan Zones 1372
7.2.4.1. Momentum Equations for 3D Fan Zones 1373
7.2.4.2. User Inputs for 3D Fan Zones 1374
7.2.4.2.1. Defining the Geometry of a 3D Fan Zone 1375
7.2.4.2.2. Defining the Properties of a 3D Fan Zone 1376
7.2.4.3. 3D Fan Zone Limitations 1377
7.2.5. Fixing the Values of Variables 1377

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.2.5.1. Overview of Fixing the Value of a Variable 1378
7.2.5.1.1. Variables That Can Be Fixed 1379
7.2.5.2. Procedure for Fixing Values of Variables in a Zone 1380
7.2.5.2.1. Fixing Velocity Components 1380
7.2.5.2.2. Fixing Temperature and Enthalpy 1381
7.2.5.2.3. Fixing Species Mass Fractions 1381
7.2.5.2.4. Fixing Turbulence Quantities 1381
7.2.5.2.5. Fixing User-Defined Scalars 1382
7.2.6. Locking the Temperature for Solid and Shell Zones 1382
7.2.7. Defining Mass, Momentum, Energy, and Other Sources 1382
7.2.7.1. Sign Conventions and Units 1383
7.2.7.2. Procedure for Defining Sources 1383
7.2.7.2.1. Mass Sources 1384
7.2.7.2.2. Momentum Sources 1385
7.2.7.2.3. Energy Sources 1385
7.2.7.2.4. Turbulence Sources 1385
7.2.7.2.4.1. Turbulence Sources for the k- ε Model 1385
7.2.7.2.4.2. Turbulence Sources for the Spalart-Allmaras Model 1385
7.2.7.2.4.3. Turbulence Sources for the k- ω Model 1386
7.2.7.2.4.4. Turbulence Sources for the Reynolds Stress Model 1386
7.2.7.2.5. Mean Mixture Fraction and Variance Sources 1386
7.2.7.2.6. P-1 Radiation Sources 1386
7.2.7.2.7. Progress Variable Sources 1387
7.2.7.2.8. NO, HCN, and NH3 Sources for the NOx Model 1387
7.2.7.2.9. Discrete Bin Fraction Sources for the Population Balance Model 1387
7.2.7.2.10. User-Defined Scalar (UDS) Sources 1387
7.3. Operating Conditions 1387
7.3.1. Buoyancy-Driven Flows and Natural Convection 1387
7.3.1.1. Modeling Natural Convection in a Closed Domain 1387
7.3.1.2. The Boussinesq Model 1388
7.3.1.3. Limitations of the Boussinesq Model 1388
7.3.1.4. Steps in Solving Buoyancy-Driven Flow Problems 1388
7.3.1.5. If you are using the incompressible ideal gas law, check the Operating Density 1391
7.3.1.5.1. Setting the Operating Density for a Single Phase Flow 1391
7.3.1.5.2. Setting the Operating Density for a Multiphase Flow 1391
7.3.1.6. Solution Strategies for Buoyancy-Driven Flows 1391
7.3.1.6.1. Guidelines for Solving High-Rayleigh-Number Flows 1392
7.4. Boundary Conditions 1392
7.4.1. Flow Inlet and Exit Boundary Conditions 1394
7.4.2. Using Flow Boundary Conditions 1394
7.4.2.1. Determining Turbulence Parameters 1395
7.4.2.1.1. Specification of Turbulence Quantities Using Profiles 1395
7.4.2.1.2. Uniform Specification of Turbulence Quantities 1396
7.4.2.1.3. Turbulence Intensity 1396
7.4.2.1.4. Turbulence Length Scale and Hydraulic Diameter 1397
7.4.2.1.5. Turbulent Viscosity Ratio 1397
7.4.2.1.6. Relationships for Deriving Turbulence Quantities 1398
7.4.2.1.7. Estimating Modified Turbulent Viscosity from Turbulence Intensity and Length Scale 1398
7.4.2.1.8. Estimating Turbulent Kinetic Energy from Turbulence Intensity 1398
7.4.2.1.9. Estimating Turbulent Dissipation Rate from a Length Scale 1399

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.2.1.10. Estimating Turbulent Dissipation Rate from Turbulent Viscosity Ratio 1399
7.4.2.1.11. Estimating Turbulent Dissipation Rate for Decaying Turbulence 1399
7.4.2.1.12. Estimating Specific Dissipation Rate from a Length Scale 1400
7.4.2.1.13. Estimating Specific Dissipation Rate from Turbulent Viscosity Ratio 1400
7.4.2.1.14. Estimating Reynolds Stress Components from Turbulent Kinetic Energy 1400
7.4.2.1.15. Specifying Inlet Turbulence for Scale Resolving Simulations 1401
7.4.3. Pressure Inlet Boundary Conditions 1401
7.4.3.1. Inputs at Pressure Inlet Boundaries 1402
7.4.3.1.1. Summary 1402
7.4.3.1.1.1. Pressure Inputs and Hydrostatic Head 1403
7.4.3.1.1.2. Defining Total Pressure and Temperature 1404
7.4.3.1.1.3. Defining the Flow Direction 1405
7.4.3.1.1.4. Defining Static Pressure 1408
7.4.3.1.1.5. Prevent Reverse Flow 1408
7.4.3.1.1.6. Defining Turbulence Parameters 1409
7.4.3.1.1.7. Defining Radiation Parameters 1409
7.4.3.1.1.8. Defining Species Mass or Mole Fractions 1409
7.4.3.1.1.9. Defining Non-Premixed Combustion Parameters 1409
7.4.3.1.1.10. Defining Premixed Combustion Boundary Conditions 1409
7.4.3.1.1.11. Defining Discrete Phase Boundary Conditions 1409
7.4.3.1.1.12. Defining Multiphase Boundary Conditions 1409
7.4.3.1.1.13. Defining Open Channel Boundary Conditions 1409
7.4.3.2. Default Settings at Pressure Inlet Boundaries 1409
7.4.3.3. Calculation Procedure at Pressure Inlet Boundaries 1410
7.4.3.3.1. Incompressible Flow Calculations at Pressure Inlet Boundaries 1410
7.4.3.3.2. Compressible Flow Calculations at Pressure Inlet Boundaries 1410
7.4.4. Velocity Inlet Boundary Conditions 1411
7.4.4.1. Inputs at Velocity Inlet Boundaries 1411
7.4.4.1.1. Summary 1411
7.4.4.1.2. Defining the Velocity 1413
7.4.4.1.3. Setting the Velocity Magnitude and Direction 1413
7.4.4.1.4. Setting the Velocity Magnitude Normal to the Boundary 1413
7.4.4.1.5. Setting the Velocity Components 1414
7.4.4.1.6. Setting the Angular Velocity 1414
7.4.4.1.7. Defining Static Pressure 1414
7.4.4.1.8. Defining the Temperature 1415
7.4.4.1.9. Defining Outflow Gauge Pressure 1415
7.4.4.1.10. Defining Turbulence Parameters 1415
7.4.4.1.11. Defining Radiation Parameters 1415
7.4.4.1.12. Defining Species Mass or Mole Fractions 1415
7.4.4.1.13. Defining Non-Premixed Combustion Parameters 1415
7.4.4.1.14. Defining Premixed Combustion Boundary Conditions 1415
7.4.4.1.15. Defining Discrete Phase Boundary Conditions 1415
7.4.4.1.16. Defining Multiphase Boundary Conditions 1416
7.4.4.2. Default Settings at Velocity Inlet Boundaries 1416
7.4.4.3. Calculation Procedure at Velocity Inlet Boundaries 1416
7.4.4.3.1. Treatment of Velocity Inlet Conditions at Flow Inlets 1416
7.4.4.3.2. Treatment of Velocity Inlet Conditions at Flow Exits 1417
7.4.4.3.3. Density Calculation 1417
7.4.5. Mass-Flow Inlet Boundary Conditions 1417
7.4.5.1. Limitations and Special Considerations 1418

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.5.2. Inputs at Mass-Flow Inlet Boundaries 1418
7.4.5.2.1. Summary 1418
7.4.5.2.2. Selecting the Reference Frame 1419
7.4.5.2.3. Defining the Mass Flow Rate or Mass Flux 1419
7.4.5.2.4. More About Mass Flux and Average Mass Flux 1420
7.4.5.2.5. Defining the Total Temperature 1421
7.4.5.2.6. Defining Static Pressure 1421
7.4.5.2.7. Defining the Flow Direction 1422
7.4.5.2.8. Defining Turbulence Parameters 1423
7.4.5.2.9. Defining Radiation Parameters 1423
7.4.5.2.10. Defining Species Mass or Mole Fractions 1423
7.4.5.2.11. Defining Non-Premixed Combustion Parameters 1423
7.4.5.2.12. Defining Premixed Combustion Boundary Conditions 1423
7.4.5.2.13. Defining Discrete Phase Boundary Conditions 1423
7.4.5.2.14. Defining Open Channel Boundary Conditions 1423
7.4.5.3. Default Settings at Mass-Flow Inlet Boundaries 1424
7.4.5.4. Calculation Procedure at Mass-Flow Inlet Boundaries 1424
7.4.5.4.1. Flow Calculations at Mass Flow Boundaries for Ideal Gases 1424
7.4.5.4.2. Flow Calculations at Mass Flow Boundaries for Incompressible Flows 1425
7.4.5.4.3. Flux Calculations at Mass Flow Boundaries 1425
7.4.6. Mass-Flow Outlet Boundary Conditions 1425
7.4.6.1. Limitations 1425
7.4.6.2. Inputs at Mass-Flow Outlet Boundaries 1426
7.4.6.2.1. Summary 1426
7.4.6.2.2. Selecting the Reference Frame 1426
7.4.6.2.3. Defining the Mass Flow Rate or Mass Flux 1426
7.4.6.2.4. Defining Radiation Parameters 1428
7.4.6.2.5. Defining Discrete Phase Boundary Conditions 1428
7.4.6.3. Calculation Procedure at Mass-Flow Outlet Boundaries 1428
7.4.6.3.1. Exit Corrected Mass Flow Rate 1428
7.4.7. Inlet Vent Boundary Conditions 1429
7.4.7.1. Inputs at Inlet Vent Boundaries 1429
7.4.7.1.1. Specifying the Loss Coefficient 1431
7.4.8. Intake Fan Boundary Conditions 1432
7.4.8.1. Inputs at Intake Fan Boundaries 1432
7.4.8.1.1. Specifying the Pressure Jump 1434
7.4.9. Pressure Outlet Boundary Conditions 1434
7.4.9.1. Inputs at Pressure Outlet Boundaries 1434
7.4.9.1.1. Summary 1434
7.4.9.1.2. Defining Static Pressure 1436
7.4.9.1.3. Defining Backflow Conditions 1437
7.4.9.1.3.1. Prevent Reverse Flow 1439
7.4.9.1.4. Defining Radiation Parameters 1439
7.4.9.1.5. Defining Discrete Phase Boundary Conditions 1439
7.4.9.1.6. Defining Open Channel Boundary Conditions 1439
7.4.9.2. Default Settings at Pressure Outlet Boundaries 1439
7.4.9.3. Calculation Procedure at Pressure Outlet Boundaries 1439
7.4.9.3.1. Average Pressure Specification 1440
7.4.9.3.1.1. Strong Averaging 1440
7.4.9.3.1.2. Weak Averaging 1441
7.4.9.4. Other Optional Inputs at Pressure Outlet Boundaries 1441

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.9.4.1. Non-Reflecting Boundary Conditions Option 1441
7.4.9.4.2. Target Mass Flow Rate Option 1442
7.4.10. Pressure Far-Field Boundary Conditions 1444
7.4.10.1. Limitations 1444
7.4.10.2. Inputs at Pressure Far-Field Boundaries 1444
7.4.10.2.1. Summary 1444
7.4.10.2.2. Defining Static Pressure, Mach Number, and Static Temperature 1445
7.4.10.2.3. Defining the Flow Direction 1446
7.4.10.2.4. Defining Turbulence Parameters 1446
7.4.10.2.5. Defining Radiation Parameters 1446
7.4.10.2.6. Defining Species Transport Parameters 1446
7.4.10.3. Defining Discrete Phase Boundary Conditions 1446
7.4.10.4. Default Settings at Pressure Far-Field Boundaries 1446
7.4.10.5. Calculation Procedure at Pressure Far-Field Boundaries 1447
7.4.10.6. Flux-based Pressure Far-Field 1447
7.4.10.7. Tangency Correction 1448
7.4.11. Outflow Boundary Conditions 1448
7.4.11.1. Ansys Fluent's Treatment at Outflow Boundaries 1448
7.4.11.2. Using Outflow Boundaries 1449
7.4.11.3. Mass Flow Split Boundary Conditions 1450
7.4.11.4. Other Inputs at Outflow Boundaries 1451
7.4.11.4.1. Radiation Inputs at Outflow Boundaries 1451
7.4.11.4.2. Defining Discrete Phase Boundary Conditions 1451
7.4.12. Outlet Vent Boundary Conditions 1451
7.4.12.1. Inputs at Outlet Vent Boundaries 1451
7.4.12.1.1. Specifying the Loss Coefficient 1453
7.4.13. Exhaust Fan Boundary Conditions 1454
7.4.13.1. Inputs at Exhaust Fan Boundaries 1454
7.4.13.1.1. Specifying the Pressure Jump 1455
7.4.14. Degassing Boundary Conditions 1456
7.4.14.1. Limitations 1457
7.4.14.2. Inputs at Degassing Boundaries 1457
7.4.15. Wall Boundary Conditions 1457
7.4.15.1. Inputs at Wall Boundaries 1458
7.4.15.1.1. Summary 1458
7.4.15.2. Stationary Wall 1458
7.4.15.3. Wall Roughness Effects in Turbulent Wall-Bounded Flows 1458
7.4.15.3.1. Setting the Roughness Parameters 1459
7.4.15.3.2. Additional Roughness Models for Icing Simulations 1460
7.4.15.3.2.1. Specified Roughness 1461
7.4.15.3.2.2. NASA Correlation 1461
7.4.15.3.2.3. Shin-et-al 1461
7.4.15.3.2.4. ICE3D Roughness File 1461
7.4.15.4. Wall Motion 1462
7.4.15.4.1. Velocity Conditions for Moving Walls 1462
7.4.15.4.2. Shear Conditions at Walls 1464
7.4.15.4.3. No-Slip Walls 1464
7.4.15.4.4. Specified Shear 1465
7.4.15.4.5. Specularity Coefficient 1465
7.4.15.4.6. Marangoni Stress 1466
7.4.15.4.7. Partial Slip for Rarefied Gases 1467

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.15.5. Thermal Boundary Conditions at Walls 1467
7.4.15.5.1. Heat Flux Boundary Conditions 1469
7.4.15.5.2. Temperature Boundary Conditions 1469
7.4.15.5.3. Convective Heat Transfer Boundary Conditions 1469
7.4.15.5.4. External Radiation Boundary Conditions 1470
7.4.15.5.5. Combined Convection and External Radiation Boundary Conditions 1470
7.4.15.5.6. Augmented Heat Transfer 1470
7.4.15.5.7. Thin-Wall Thermal Resistance Parameters 1470
7.4.15.5.8. Thermal Conditions for Two-Sided Walls 1472
7.4.15.5.8.1. Orthogonality-Based Secondary Gradient Limiting at Coupled Two-Sided Walls 1473
7.4.15.5.9. Boundary Advection for Solid Motion 1474
7.4.15.5.10. Shell Conduction 1475
7.4.15.5.11. Heat Transfer Boundary Conditions Through System Coupling 1475
7.4.15.5.12. Heat Transfer Boundary Conditions Across a Mapped Interface 1476
7.4.15.5.13. Temperature Jump for Rarefied Gases 1477
7.4.15.6. Species Boundary Conditions for Walls 1477
7.4.15.6.1. Partial Catalytic Boundary Conditions for Walls 1479
7.4.15.7. Radiation Boundary Conditions for Walls 1479
7.4.15.8. Discrete Phase Model (DPM) Boundary Conditions for Walls 1479
7.4.15.8.1. Wall Adhesion Contact Angle for VOF Model 1479
7.4.15.9. User-Defined Scalar (UDS) Boundary Conditions for Walls 1479
7.4.15.10. Wall Film Conditions for Walls 1480
7.4.15.11. Structural Model Conditions for Walls 1480
7.4.15.12. Default Settings at Wall Boundaries 1480
7.4.15.13. Shear-Stress Calculation Procedure at Wall Boundaries 1480
7.4.15.13.1. Shear-Stress Calculation in Laminar Flow 1480
7.4.15.13.2. Shear-Stress Calculation in Turbulent Flows 1480
7.4.15.14. Heat Transfer Calculations at Wall Boundaries 1481
7.4.15.14.1. Temperature Boundary Conditions 1481
7.4.15.14.2. Heat Flux Boundary Conditions 1481
7.4.15.14.3. Convective Heat Transfer Boundary Conditions 1482
7.4.15.14.4. External Radiation Boundary Conditions 1482
7.4.15.14.5. Combined External Convection and Radiation Boundary Conditions 1482
7.4.15.14.6. Calculation of the Fluid-Side Heat Transfer Coefficient 1483
7.4.16. Perforated Wall Boundary Conditions 1483
7.4.16.1. Overview and Limitations 1483
7.4.16.2. Modeling Concept 1483
7.4.16.3. Setting Perforated Walls 1487
7.4.16.4. Procedure for Manual Setup of Perforated Walls 1489
7.4.16.5. Perforated Wall File Format 1495
7.4.16.6. Postprocessing for Perforated Walls 1498
7.4.17. Symmetry Boundary Conditions 1499
7.4.17.1. Examples of Symmetry Boundaries 1499
7.4.17.2. Calculation Procedure at Symmetry Boundaries 1501
7.4.18. Periodic Boundary Conditions 1501
7.4.18.1. Examples of Periodic Boundaries 1501
7.4.18.2. Inputs for Periodic Boundaries 1502
7.4.18.3. Default Settings at Periodic Boundaries 1503
7.4.18.4. Calculation Procedure at Periodic Boundaries 1503
7.4.19. Axis Boundary Conditions 1504

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.19.1. Calculation Procedure at Axis Boundaries 1504
7.4.20. Fan Boundary Conditions 1504
7.4.20.1. Limitations of Fan Boundary Conditions 1504
7.4.20.2. Fan Equations 1505
7.4.20.2.1. Modeling the Pressure Rise Across the Fan 1505
7.4.20.2.2. Modeling the Fan Swirl Velocity 1505
7.4.20.3. User Inputs for Fans 1506
7.4.20.3.1. Identifying the Fan Zone 1506
7.4.20.3.2. Defining the Pressure Jump 1507
7.4.20.3.2.1. Polynomial, Piecewise-Linear, or Piecewise-Polynomial Function 1507
7.4.20.3.2.2. Constant Value 1508
7.4.20.3.2.3. User-Defined Function or Profile 1508
7.4.20.3.2.4. Example: Determining the Pressure Jump Function 1509
7.4.20.3.3. Defining Discrete Phase Boundary Conditions for the Fan 1510
7.4.20.3.4. Defining the Fan Swirl Velocity 1510
7.4.20.3.4.1. Polynomial Function 1510
7.4.20.3.4.2. Constant Value 1510
7.4.20.3.4.3. User-Defined Function or Profile 1511
7.4.20.4. Postprocessing for Fans 1511
7.4.20.4.1. Reporting the Pressure Rise Through the Fan 1511
7.4.20.4.2. Graphical Plots 1511
7.4.21. Radiator Boundary Conditions 1511
7.4.21.1. Radiator Equations 1512
7.4.21.1.1. Modeling the Pressure Loss Through a Radiator 1512
7.4.21.1.2. Modeling the Heat Transfer Through a Radiator 1512
7.4.21.1.2.1. Calculating the Heat Transfer Coefficient 1513
7.4.21.2. User Inputs for Radiators 1513
7.4.21.2.1. Identifying the Radiator Zone 1514
7.4.21.2.2. Defining the Pressure Loss Coefficient Function 1514
7.4.21.2.2.1. Polynomial, Piecewise-Linear, or Piecewise-Polynomial Function 1515
7.4.21.2.2.2. Constant Value 1515
7.4.21.2.2.3. Example: Calculating the Loss Coefficient 1515
7.4.21.2.3. Defining the Heat Flux Parameters 1517
7.4.21.2.3.1. Polynomial, Piecewise-Linear, or Piecewise-Polynomial Function 1517
7.4.21.2.3.2. Constant Value 1517
7.4.21.2.3.3. Example: Determining the Heat Transfer Coefficient Function 1517
7.4.21.2.4. Defining Discrete Phase Boundary Conditions for the Radiator 1518
7.4.21.3. Postprocessing for Radiators 1518
7.4.21.3.1. Reporting the Radiator Pressure Drop 1518
7.4.21.3.2. Reporting Heat Transfer in the Radiator 1518
7.4.21.3.3. Graphical Plots 1518
7.4.22. Porous Jump Boundary Conditions 1519
7.4.22.1. User Inputs for the Porous Jump Model 1519
7.4.22.1.1. Identifying the Porous Jump Zone 1520
7.4.22.1.2. Defining Discrete Phase Boundary Conditions for the Porous Jump 1521
7.4.22.2. Postprocessing for the Porous Jump 1521
7.5. Editing Multiple Boundary Conditions at Once 1521
7.6. Transient Cell Zone and Boundary Conditions 1523
7.6.1. Standard Transient Profiles 1524
7.6.2. Tabular Transient Profiles 1525
7.6.3. Profiles for Moving and Deforming Meshes 1526

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.7. Boundary Acoustic Wave Models 1527
7.7.1. Turbo-Specific Non-Reflecting Boundary Conditions 1527
7.7.1.1. Overview 1528
7.7.1.2. Limitations 1528
7.7.1.3. Theory 1531
7.7.1.3.1. Equations in Characteristic Variable Form 1531
7.7.1.3.2. Inlet Boundary 1533
7.7.1.3.3. Outlet Boundary 1535
7.7.1.3.4. Updated Flow Variables 1536
7.7.1.4. Using Turbo-Specific Non-Reflecting Boundary Conditions 1536
7.7.1.4.1. Using the NRBCs with the Mixing-Plane Model 1538
7.7.1.4.2. Using the NRBCs in Parallel Ansys Fluent 1538
7.7.2. General Non-Reflecting Boundary Conditions 1538
7.7.2.1. Overview 1538
7.7.2.2. Restrictions and Limitations 1538
7.7.2.3. Theory 1539
7.7.2.4. Using the General Non-Reflecting Boundary Condition 1543
7.7.3. Impedance Boundary Conditions 1545
7.7.3.1. Restrictions and Limitations 1545
7.7.3.2. Theory 1545
7.7.3.3. Using the Impedance Boundary Condition 1546
7.7.3.4. Calculating Impedance Parameters 1548
7.7.4. Transparent Flow Forcing Boundary Conditions 1554
7.7.4.1. Restrictions and Limitations 1554
7.7.4.2. Theory 1554
7.7.4.3. Using the Transparent Flow Forcing Boundary Condition 1554
7.8. User-Defined Fan Model 1556
7.8.1. Steps for Using the User-Defined Fan Model 1557
7.8.2. Example of a User-Defined Fan 1557
7.8.2.1. Setting the User-Defined Fan Parameters 1558
7.8.2.2. Sample User-Defined Fan Program 1559
7.8.2.3. Initializing the Flow Field and Profile Files 1561
7.8.2.4. Selecting the Profiles 1561
7.8.2.5. Performing the Calculation 1562
7.8.2.6. Results 1563
7.9. Profiles 1564
7.9.1. Profile Specification Types 1564
7.9.2. Profile File Formats 1565
7.9.2.1. Standard Profiles 1566
7.9.2.1.1. Example 1567
7.9.2.2. CSV Profiles 1567
7.9.3. Using Profiles 1570
7.9.3.1. Checking and Deleting Profiles 1571
7.9.3.2. Viewing Profile Data 1573
7.9.3.3. Example 1576
7.9.3.4. Reorienting Profiles 1576
7.9.3.4.1. Steps for Changing the Profile Orientation 1576
7.9.3.4.2. Profile Orienting Example 1579
7.9.3.5. Replicating Profiles 1581
7.9.3.5.1. Steps for Replicating a Profile 1581
7.9.3.5.2. Complex Mode Shape Profile Replication 1584

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.9.3.5.2.1. Steps for Replicating a Complex Mode Shape Profile 1585
7.10. Coupling Boundary Conditions with GT-POWER 1587
7.10.1. Requirements and Restrictions 1587
7.10.2. User Inputs 1588
7.10.3. Torque-Speed Coupling with GT-POWER 1591
7.11. Coupling Boundary Conditions with WAVE 1592
7.11.1. Requirements and Restrictions 1592
7.11.2. User Inputs 1593
8. Physical Properties 1597
8.1. Defining Materials 1597
8.1.1. Physical Properties for Solid Materials 1598
8.1.2. Material Types and Databases 1598
8.1.3. Using the **Create/Edit Materials** Dialog Box 1599
8.1.3.1. Modifying Properties of an Existing Material 1601
8.1.3.2. Renaming an Existing Material 1602
8.1.3.3. Copying Materials from the Ansys Fluent Database 1602
8.1.3.4. Copying Materials from the Ansys GRANTA MDS Database 1604
8.1.3.5. Creating a New Material 1606
8.1.3.6. Saving Materials and Properties 1606
8.1.3.7. Deleting a Material 1607
8.1.3.8. Changing the Order of the Materials List 1607
8.1.4. Using a User-Defined Materials Database 1607
8.1.4.1. Opening a User-Defined Database 1608
8.1.4.2. Viewing Materials in a User-Defined Database 1609
8.1.4.3. Copying Materials from a User-Defined Database 1609
8.1.4.4. Copying Materials from the Case to a User-Defined Database 1610
8.1.4.5. Modifying Properties of an Existing Material 1611
8.1.4.6. Creating a New Materials Database and Materials 1612
8.1.4.7. Deleting Materials from a Database 1615
8.2. Defining Properties Using Temperature-Dependent Functions 1616
8.2.1. Inputs for Polynomial Functions 1617
8.2.2. Inputs for Piecewise-Linear Functions 1618
8.2.3. Inputs for Piecewise-Polynomial Functions 1620
8.2.4. Inputs for NASA-9-Piecewise-Polynomial Functions 1622
8.2.5. Checking and Modifying Existing Profiles 1624
8.3. Density 1624
8.3.1. Defining Density for Various Flow Regimes 1625
8.3.1.1. Mixing Density Relationships in Multiple-Zone Models 1625
8.3.2. Input of Constant Density 1626
8.3.3. Inputs for the Boussinesq Approximation 1626
8.3.4. Compressible Liquid Density Method 1626
8.3.4.1. Compressible Liquid Inputs 1627
8.3.4.2. Compressible Liquid Density Method Availability 1630
8.3.5. Density as a Profile Function of Temperature 1630
8.3.6. Incompressible Ideal Gas Law 1630
8.3.6.1. Density Inputs for the Incompressible Ideal Gas Law 1631
8.3.7. Ideal Gas Law for Compressible Flows 1631
8.3.7.1. Density Inputs for the Ideal Gas Law for Compressible Flows 1632
8.3.8. Composition-Dependent Density for Multicomponent Mixtures 1632
8.4. Viscosity 1634
8.4.1. Input of Constant Viscosity 1635

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.4.2. Viscosity as a Function of Temperature 1635
8.4.2.1. Sutherland Viscosity Law 1636
8.4.2.1.1. Inputs for Sutherland's Law 1636
8.4.2.2. Power-Law Viscosity Law 1637
8.4.2.2.1. Inputs for the Power Law 1637
8.4.3. Defining the Viscosity Using Kinetic Theory 1638
8.4.4. Defining Viscosity Using Gupta Curve Fits 1638
8.4.5. Composition-Dependent Viscosity for Multicomponent Mixtures 1638
8.4.6. Viscosity for Non-Newtonian Fluids 1639
8.4.6.1. Temperature Dependent Viscosity 1640
8.4.6.2. Power Law for Non-Newtonian Viscosity 1641
8.4.6.2.1. Inputs for the Non-Newtonian Power Law 1641
8.4.6.3. The Carreau Model for Pseudo-Plastics 1641
8.4.6.3.1. Inputs for the Carreau Model 1642
8.4.6.4. Cross Model 1642
8.4.6.4.1. Inputs for the Cross Model 1643
8.4.6.5. Herschel-Bulkley Model for Bingham Plastics 1643
8.4.6.5.1. Inputs for the Herschel-Bulkley Model 1644
8.5. Thermal Conductivity 1644
8.5.1. Constant Thermal Conductivity 1645
8.5.2. Thermal Conductivity as a Function of Temperature 1646
8.5.3. Thermal Conductivity Using Kinetic Theory 1646
8.5.4. Defining Thermal Conductivity Using Gupta Curve Fits 1646
8.5.5. Composition-Dependent Thermal Conductivity for Multicomponent Mixtures 1646
8.5.6. Anisotropic Thermal Conductivity for Solids 1648
8.5.6.1. Anisotropic Thermal Conductivity 1648
8.5.6.2. Biaxial Thermal Conductivity 1650
8.5.6.3. Orthotropic Thermal Conductivity 1650
8.5.6.4. Cylindrical Orthotropic Thermal Conductivity 1652
8.5.6.5. Principal Axes and Principal Values 1654
8.5.6.6. User-Defined Anisotropic Thermal Conductivity 1656
8.6. User-Defined Scalar (UDS) Diffusivity 1656
8.6.1. Isotropic Diffusion 1656
8.6.2. Anisotropic Diffusion 1658
8.6.2.1. Anisotropic Diffusivity 1658
8.6.2.2. Orthotropic Diffusivity 1659
8.6.2.3. Cylindrical Orthotropic Diffusivity 1661
8.6.3. User-Defined Anisotropic Diffusivity 1662
8.7. Specific Heat Capacity 1663
8.7.1. Input of Constant Specific Heat Capacity 1663
8.7.2. Specific Heat Capacity as a Function of Temperature 1663
8.7.3. Defining Specific Heat Capacity Using Kinetic Theory 1664
8.7.4. Specific Heat Capacity as a Function of Composition 1664
8.8. Radiation Properties 1665
8.8.1. Absorption Coefficient 1665
8.8.1.1. Inputs for a Constant Absorption Coefficient 1666
8.8.1.2. Inputs for a Composition-Dependent Absorption Coefficient 1666
8.8.1.2.1. Path Length Inputs 1666
8.8.1.2.1.1. Inputs for a Non-Gray Radiation Absorption Coefficient 1666
8.8.1.2.1.2. Effect of Particles and Soot on the Absorption Coefficient 1667
8.8.2. Scattering Coefficient 1667

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.8.2.1. Inputs for a Constant Scattering Coefficient 1667
8.8.2.2. Inputs for the Scattering Phase Function 1667
8.8.2.2.1. Isotropic Phase Function 1667
8.8.2.2.2. Linear-Anisotropic Phase Function 1667
8.8.2.2.3. Delta-Eddington Phase Function 1668
8.8.2.2.4. User-Defined Phase Function 1668
8.8.3. Refractive Index 1668
8.8.4. Reporting the Radiation Properties 1668
8.9. Mass Diffusion Coefficients 1668
8.9.1. Mass Diffusion Coefficient Inputs 1669
8.9.1.1. Constant Dilute Approximation Inputs 1671
8.9.1.2. Dilute Approximation Inputs 1671
8.9.1.3. Multicomponent Method Inputs 1672
8.9.2. Mass Diffusion Coefficient Inputs for Turbulent Flow 1673
8.9.3. Anisotropic Species Diffusion 1673
8.9.4. Thermal Diffusion Coefficient Inputs 1674
8.10. Standard State Enthalpies 1675
8.11. Standard State Entropies 1676
8.12. Unburnt Thermal Diffusivity 1676
8.13. Kinetic Theory Parameters 1676
8.13.1. Inputs for Kinetic Theory 1677
8.14. Operating Pressure 1677
8.14.1. The Significance of Operating Pressure 1678
8.14.2. Operating Pressure, Gauge Pressure, and Absolute Pressure 1678
8.14.3. Setting the Operating Pressure 1678
8.15. Using a Reference Pressure to Adjust the Gauge Pressure Field 1679
8.16. Real Gas Models 1681
8.16.1. Introduction 1681
8.16.2. Choosing a Real Gas Model 1683
8.16.3. Cubic Equation of State Models 1684
8.16.3.1. Overview and Limitations 1684
8.16.3.2. Equation of State 1686
8.16.3.3. Enthalpy, Entropy, and Specific Heat Calculations 1687
8.16.3.4. Critical Constants for Pure Components 1689
8.16.3.5. Calculations for Mixtures 1690
8.16.3.5.1. Using the Cubic Equation of State Real Gas Models 1692
8.16.3.5.2. Solution Strategies and Considerations for Cubic Equations of State Real Gas Models 1696
8.16.3.5.3. Using the Cubic Equation of State Models with the Lagrangian Dispersed Phase Models 1698
8.16.3.5.4. Postprocessing the Cubic Equations of State Real Gas Model 1700
8.16.4. The NIST Real Gas Models 1701
8.16.4.1. Limitations of the NIST Real Gas Models 1701
8.16.4.2. The REFPROP v10.0 Database 1702
8.16.4.3. Using the NIST Real Gas Models 1706
8.16.4.3.1. Creating NIST Look-up Tables 1710
8.16.4.3.1.1. Capabilities and Limitations with NIST Look-up Tables 1712
8.16.4.4. Legacy TUI for the NIST Real Gas Models 1713
8.16.4.4.1. Limitations of the Legacy TUI NIST Real Gas Models 1713
8.16.4.4.2. Activating the NIST Real Gas Model 1714
8.16.4.4.3. Creating Full NIST Look-up Tables 1715

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.16.4.4.4. Creating Binary Mixture Saturation Tables for Binary Mixtures 1718
8.16.4.5. Solution Strategies and Considerations for NIST Real Gas Model Simulation 1720
8.16.4.5.1. Writing Your Case File 1721
8.16.4.5.2. Postprocessing 1721
8.16.5. The User-Defined Real Gas Model 1721
8.16.5.1. Limitations of the User-Defined Real Gas Model 1721
8.16.5.2. Writing the UDRGM C Function Library 1724
8.16.5.3. Compiling Your UDRGM C Functions and Building a Shared Library File 1728
8.16.5.3.1. Compiling the UDRGM Using the Graphical Interface 1728
8.16.5.3.2. Compiling the UDRGM Using the Text Interface 1729
8.16.5.3.3. Loading the UDRGM Shared Library File 1730
8.16.5.4. UDRGM Example: Ideal Gas Equation of State 1730
8.16.5.4.1. Ideal Gas UDRGM Code Listing 1731
8.16.5.5. Additional UDRGM Examples 1734
8.16.6. Using Real Gas Property (RGP) Table Files 1734
8.16.6.1. Overview 1734
8.16.6.2. Defining Material Properties Using RGP Tables 1734
8.16.6.3. Defining Saturation Properties via RGP Tables 1735
9. Modeling Basic Fluid Flow 1737
9.1. User-Defined Scalar (UDS) Transport Equations 1737
9.1.1. Introduction 1737
9.1.2. UDS Theory 1737
9.1.2.1. Single Phase Flow 1738
9.1.2.2. Multiphase Flow 1738
9.1.3. Setting Up UDS Equations in Ansys Fluent 1739
9.1.3.1. Single Phase Flow 1740
9.1.3.2. Multiphase Flow 1745
9.2. Periodic Flows 1747
9.2.1. Overview and Limitations 1747
9.2.1.1. Overview 1747
9.2.1.2. Limitations for Modeling Streamwise-Periodic Flow 1748
9.2.2. User Inputs for the Pressure-Based Solver 1749
9.2.2.1. Setting Parameters for the Calculation of β 1751
9.2.3. User Inputs for the Density-Based Solvers 1751
9.2.4. Monitoring the Value of the Pressure Gradient 1752
9.2.5. Postprocessing for Streamwise-Periodic Flows 1752
9.3. Swirling and Rotating Flows 1753
9.3.1. Overview of Swirling and Rotating Flows 1754
9.3.1.1. Axisymmetric Flows with Swirl or Rotation 1754
9.3.1.1.1. Momentum Conservation Equation for Swirl Velocity 1754
9.3.1.2. Three-Dimensional Swirling Flows 1754
9.3.1.3. Flows Requiring a Moving Reference Frame 1755
9.3.2. Turbulence Modeling in Swirling Flows 1755
9.3.3. Mesh Setup for Swirling and Rotating Flows 1756
9.3.3.1. Coordinate System Restrictions 1756
9.3.3.2. Mesh Sensitivity in Swirling and Rotating Flows 1756
9.3.4. Modeling Axisymmetric Flows with Swirl or Rotation 1756
9.3.4.1. Problem Setup for Axisymmetric Swirling Flows 1757
9.3.4.2. Solution Strategies for Axisymmetric Swirling Flows 1757
9.3.4.2.1. Step-By-Step Solution Procedures for Axisymmetric Swirling Flows 1758

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.3.4.2.2. Improving Solution Stability by Gradually Increasing the Rotational or Swirl Speed 1759
9.3.4.2.2.1. Postprocessing for Axisymmetric Swirling Flows 1759
9.4. Compressible Flows 1759
9.4.1. When to Use the Compressible Flow Model 1760
9.4.2. Physics of Compressible Flows 1761
9.4.2.1. Basic Equations for Compressible Flows 1761
9.4.2.2. The Compressible Form of the Gas Law 1761
9.4.3. Modeling Inputs for Compressible Flows 1762
9.4.3.1. Boundary Conditions for Compressible Flows 1763
9.4.4. Floating Operating Pressure 1763
9.4.4.1. Limitations 1764
9.4.4.2. Theory 1764
9.4.4.3. Enabling Floating Operating Pressure 1764
9.4.4.4. Setting the Initial Value for the Floating Operating Pressure 1764
9.4.4.5. Storage and Reporting of the Floating Operating Pressure 1765
9.4.4.6. Monitoring Absolute Pressure 1765
9.4.5. Solution Strategies for Compressible Flows 1765
9.4.6. Reporting of Results for Compressible Flows 1766
9.5. Inviscid Flows 1766
9.5.1. Setting Up an Inviscid Flow Model 1766
9.5.2. Solution Strategies for Inviscid Flows 1767
9.5.3. Postprocessing for Inviscid Flows 1767
10. Modeling Flows with Moving Reference Frames 1769
10.1. Introduction 1769
10.2. Flow in Single Moving Reference Frames (SRF) 1771
10.2.1. Mesh Setup for a Single Moving Reference Frame 1772
10.2.2. Setting Up a Single Moving Reference Frame Problem 1772
10.2.2.1. Choosing the Relative or Absolute Velocity Formulation 1775
10.2.2.1.1. Example 1775
10.2.3. Solution Strategies for a Single Moving Reference Frame 1776
10.2.3.1. Gradual Increase of the Rotational Speed to Improve Solution Stability 1777
10.2.4. Postprocessing for a Single Moving Reference Frame 1777
10.3. Flow in Multiple Moving Reference Frames 1779
10.3.1. The Multiple Reference Frame Model 1779
10.3.1.1. Overview 1779
10.3.1.2. Limitations 1780
10.3.2. Mesh Setup for a Multiple Moving Reference Frame 1781
10.3.3. Setting Up a Multiple Moving Reference Frame Problem 1781
10.3.3.1. Setting Up Multiple Reference Frames 1782
10.3.4. Solution Strategies for MRF and Problems 1786
10.3.5. Postprocessing for MRF Problems 1786
11. Managing Motion Definitions 1787
12. Managing Auxiliary Geometry Definitions 1791
13. Modeling Flows Using Sliding and Dynamic Meshes 1797
13.1. Introduction 1797
13.2. Sliding Mesh Examples 1798
13.3. The Sliding Mesh Technique 1800
13.4. Sliding Mesh Interface Shapes 1801
13.5. Using Sliding Meshes 1804
13.5.1. Requirements, Constraints, and Considerations 1804

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.5.2. Setting Up the Sliding Mesh Problem 1805
13.5.3. Solution Strategies for Sliding Meshes 1808
13.5.3.1. Saving Case and Data Files 1809
13.5.3.2. Time-Periodic Solutions 1809
13.5.4. Postprocessing for Sliding Meshes 1810
13.6. Using Dynamic Meshes 1812
13.6.1. Setting Dynamic Mesh Modeling Parameters 1814
13.6.2. Dynamic Mesh Update Methods 1816
13.6.2.1. Smoothing Methods 1816
13.6.2.1.1. Diffusion-Based Smoothing 1818
13.6.2.1.1.1. Diffusivity Based on Boundary Distance 1822
13.6.2.1.1.2. Diffusivity Based on Cell Volume 1823
13.6.2.1.1.3. Applicability of the Diffusion-Based Smoothing Method 1824
13.6.2.1.2. Spring-Based Smoothing 1824
13.6.2.1.2.1. Applicability of the Spring-Based Smoothing Method 1828
13.6.2.1.3. Linearly Elastic Solid Based Smoothing Method 1828
13.6.2.1.3.1. Applicability of the Linearly Elastic Solid Based Smoothing Method 1830
13.6.2.1.4. Radial Basis Function Smoothing 1830
13.6.2.1.4.1. Local Smoothing with the Radial Basis Function Smoothing Method .. 1832
13.6.2.1.5. Smoothing from a Reference Position 1832
13.6.2.1.6. Laplacian Smoothing Method 1833
13.6.2.1.7. Boundary Layer Smoothing Method 1834
13.6.2.2. Dynamic Layering 1837
13.6.2.2.1. Applicability of the Dynamic Layering Method 1840
13.6.2.3. Remeshing 1841
13.6.2.3.1. Methods-Based Remeshing 1844
13.6.2.3.1.1. Local Remeshing Method 1844
13.6.2.3.1.1.1. Local Cell Remeshing Method 1845
13.6.2.3.1.1.2. Local Face Remeshing Method 1845
13.6.2.3.1.1.3. Local Remeshing Based on Sizing Function 1846
13.6.2.3.1.2. Cell Zone Remeshing Method 1852
13.6.2.3.1.2.1. Limitations of the Cell Zone Remeshing Method 1852
13.6.2.3.1.3. Face Region Remeshing Method 1852
13.6.2.3.1.3.1. Face Region Remeshing with Wedge Cells in Prism Layers 1854
13.6.2.3.1.3.2. Applicability of the Face Region Remeshing Method 1856
13.6.2.3.1.4. 2.5D Surface Remeshing Method 1857
13.6.2.3.1.4.1. Applicability of the 2.5D Surface Remeshing Method 1858
13.6.2.3.1.4.2. Using the 2.5D Model 1859
13.6.2.3.2. Unified Remeshing 1861
13.6.2.3.2.1. Sizing Controls 1862
13.6.2.3.2.2. Prism Controls 1864
13.6.2.4. Volume Mesh Update Procedure 1866
13.6.2.5. Transient Considerations for Remeshing and Layering 1867
13.6.3. Feature Detection 1868
13.6.3.1. Applicability of Feature Detection 1868
13.6.4. In-Cylinder Settings 1868
13.6.4.1. Using the In-Cylinder Option 1874
13.6.4.1.1. Overview 1874
13.6.4.1.2. Defining the Mesh Topology 1875
13.6.4.1.3. Defining Motion/Geometry Attributes of Mesh Zones 1877
13.6.4.1.4. Defining Valve Opening and Closure 1883

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.6.5. Six DOF Solver Settings 1883
13.6.5.1. Setting Rigid Body Motion Attributes for the Six DOF Solver 1885
13.6.6. Implicit Update Settings 1888
13.6.7. Contact Detection Settings 1890
13.6.7.1. Flow Control Using Contact Zones 1892
13.6.7.2. Flow Control Using Contact Marks 1892
13.6.7.2.1. Selecting Parameters for Flow Control 1893
13.6.7.2.2. Modifying and Displaying Contact Cell Marks 1894
13.6.8. Defining Dynamic Mesh Events 1894
13.6.8.1. Procedure for Defining Events 1895
13.6.8.2. Defining Events for In-Cylinder Applications 1897
13.6.8.2.1. Events 1898
13.6.8.2.2. Changing the Zone Type 1898
13.6.8.2.3. Copying Zone Boundary Conditions 1898
13.6.8.2.4. Activating a Cell Zone 1898
13.6.8.2.5. Deactivating a Cell Zone 1898
13.6.8.2.6. Creating a Sliding Interface 1898
13.6.8.2.7. Deleting a Sliding Interface 1900
13.6.8.2.8. Changing the Motion Attribute of a Dynamic Zone 1900
13.6.8.2.9. Changing the Time Step Size 1900
13.6.8.2.10. Changing the Under-Relaxation Factor 1900
13.6.8.2.11. Inserting a Boundary Zone Layer 1900
13.6.8.2.12. Removing a Boundary Zone Layer 1901
13.6.8.2.13. Inserting an Interior Zone Layer 1901
13.6.8.2.14. Removing an Interior Zone Layer 1902
13.6.8.2.15. Inserting a Cell Layer 1903
13.6.8.2.16. Removing a Cell Layer 1903
13.6.8.2.17. Executing a Command 1903
13.6.8.2.18. Replacing the Mesh 1903
13.6.8.2.19. Resetting Inert EGR 1903
13.6.8.2.20. Diesel Unsteady Flamelet Reset 1903
13.6.8.3. Exporting and Importing Events 1904
13.6.9. Specifying the Motion of Dynamic Zones 1904
13.6.9.1. General Procedure 1904
13.6.9.1.1. Creating a Dynamic Zone 1904
13.6.9.1.2. Modifying a Dynamic Zone 1904
13.6.9.1.3. Checking the Center of Gravity 1904
13.6.9.1.4. Deleting a Dynamic Zone 1904
13.6.9.2. Stationary Zones 1905
13.6.9.3. Rigid Body Motion 1906
13.6.9.4. Deforming Motion 1910
13.6.9.5. User-Defined Motion 1913
13.6.9.5.1. Specifying Boundary Layer Deformation Smoothing 1915
13.6.9.6. System Coupling Motion 1916
13.6.9.7. Intrinsic FSI Motion 1917
13.6.9.8. Solution Stabilization for Dynamic Mesh Boundary Zones 1918
13.6.9.9. Solid-Body Kinematics 1919
13.6.10. Previewing the Dynamic Mesh 1922
13.6.10.1. Previewing Zone Motion 1922
13.6.10.2. Previewing Mesh Motion 1923
13.6.11. Steady-State Dynamic Mesh Applications 1924

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.6.11.1. An Example of Steady-State Dynamic Mesh Usage 1926
14. Modeling Turbomachinery Flows 1929
14.1. Using the Turbomachinery Guided Workflow 1929
14.1.1. Describing the Components of the Turbo Machine 1931
14.1.2. Defining Your Blade Row Scope 1933
14.1.3. Importing Your Mesh 1934
14.1.4. Associating Your Mesh 1938
14.1.5. Mapping Your Regions 1940
14.1.6. Creating the CFD Model 1943
14.1.7. Defining the Turbomachinery Physics 1945
14.1.8. Defining the Turbomachinery Regions and Zones 1946
14.1.9. Defining the Turbomachinery Topology 1949
14.1.10. Defining Turbomachinery Surfaces 1950
14.1.11. Creating Turbomachinery Report Definitions and Monitors 1951
14.1.12. Text Command List for the Turbo Workflow 1951
14.1.13. Editing Tasks in the Turbo Workflow 1951
14.1.14. Saving and Loading Turbo Workflows 1952
14.1.15. Applying Preferences to the Turbo Workflow 1952
14.2. Frozen Gust / Inlet Disturbance Flow Modeling 1955
14.3. Blade Row Interaction Modeling 1956
14.3.1. Pitch-Change Models 1957
14.3.1.1. Pitch-Scale interface 1958
14.3.1.2. No Pitch-Scale interface 1958
14.3.1.3. Mixing-Plane interface 1959
14.3.2. Modeling an Ensemble of Blades Per Row using GTI 1961
14.3.3. Creating and Editing General Turbo Interfaces 1962
14.3.4. Legacy Mixing Plane Model 1966
14.3.4.1. Limitations 1966
14.3.4.2. Setting Up the Legacy Mixing Plane Model 1967
14.3.4.2.1. Modeling Options 1970
14.3.4.2.1.1. Fixing the Pressure Level for an Incompressible Flow 1970
14.3.4.2.1.2. Conserving Swirl Across the Mixing Plane 1971
14.3.4.2.1.3. Conserving Total Enthalpy Across the Mixing Plane 1971
14.3.4.3. Solution Strategies for Mixing Plane Problems 1972
14.4. Aerodynamic Damping (Blade Flutter Analysis) 1972
14.4.1. Common Settings for a Blade Flutter Case 1972
14.4.1.1. Reading the Mode Shapes 1973
14.4.1.2. Configuring Run Calculation Settings 1976
14.4.1.3. Using Dynamic Mesh Zones in a Blade Flutter Simulation 1977
14.4.1.3.1. Turning on Dynamic Mesh 1978
14.4.1.3.2. Defining the Periodic Displacement of the Blades 1978
14.4.1.3.3. Creating and Applying Dynamic Mesh Zones 1986
14.4.2. Traveling Wave Method (TWM) 1990
14.4.2.1. Setup Specific to the TWM Method 1992
14.4.2.2. Visualizing and Exporting Blade Flutter Harmonics with TWM 1994
14.4.2.3. TWM Method Post-processing 1996
14.4.3. Influence Coefficient Method (ICM) 1997
14.4.3.1. Setup Specific to the ICM Method 1998
14.4.3.1.1. Limitations of the ICM Method 1999
14.4.3.2. ICM Method Post-processing 1999
14.4.4. Common Postprocessing for a Blade Flutter Case 2003

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.5. Phase-lag Method 2003
14.5.1. Phase-lag Theory 2004
14.5.2. Phase-lag Capabilities and Limitations 2005
14.5.3. Using the Phase-lag Method 2005
14.5.3.1. Pre-requisites and Recommendations 2006
14.5.3.2. Creating the Phase-lag Interface 2007
14.5.3.3. Phase-lag Spectral Description 2010
14.5.3.3.1. Inlet/Outlet Disturbance Specific Phase-lag Method Workflow 2010
14.5.3.3.2. Blade Flutter (TWM) Specific Phase-lag Workflow 2011
14.5.3.3.3. Additional Commands for Phase-lag Spectral Descriptions 2011
14.5.3.4. Initializing and Running Phase-lag Simulations 2011
14.5.3.5. 21.2.3.6. Fourier Coefficient Postprocessing for Inlet/Outlet Disturbance and Blade Flutter Use-Cases 2012
14.6. Non-equilibrium Wet Steam Model for Steam Turbines 2012
14.7. Blade Film Cooling for Gas Turbines 2013
14.7.1. Specifying the Virtual Hole Geometry 2014
14.7.2. Specifying the Boundary Interface 2021
14.7.2.1. Hole Locations for Rotationally Periodic Interface Pairs 2022
14.7.3. Boundary Interface Definitions 2023
14.7.4. Specifying Overlap Boundary Conditions 2024
14.7.5. Limitations for Boundary Interfaces and their Geometries 2025
14.8. Turbomachinery Description 2026
14.9. Turbomachinery-Specific Numerics 2027
14.10. Turbomachinery Postprocessing 2029
14.10.1. Defining the Turbomachinery Topology 2029
14.10.1.1. Boundary Types 2032
14.10.1.2. Turbomachinery-Specific Variables 2033
14.10.2. Contours and Vectors Visualization for Turbomachinery 2034
14.10.2.1. Creating Turbo Surfaces 2034
14.10.2.2. Periodic Instancing 2035
14.10.3. General Fourier Coefficient Postprocessing for Turbomachinery Cases 2035
14.10.3.1. Fourier Coefficient Postprocessing Theory 2035
14.10.3.2. Fourier Coefficient Postprocessing Pre-requisites 2036
14.10.3.3. Fourier Coefficient Postprocessing Use-Cases 2036
14.10.3.4. Creating Graphics Spectral Content 2037
14.10.3.5. Extra Settings 2038
14.10.3.6. Graphical Postprocessing of Fourier Coefficients 2039
14.10.4. Circumferential-Averaged Profile Extraction 2040
14.10.5. Turbo Post 2040
14.10.5.1. Generating Reports of Turbomachinery Data 2041
14.10.5.1.1. Computing Turbomachinery Quantities 2042
14.10.5.1.1.1. Mass Flow 2042
14.10.5.1.1.2. Swirl Number 2042
14.10.5.1.1.3. Average Total Pressure 2042
14.10.5.1.1.4. Average Total Temperature 2043
14.10.5.1.1.5. Average Flow Angles 2043
14.10.5.1.1.6. Passage Loss Coefficient 2044
14.10.5.1.1.7. Axial Force 2045
14.10.5.1.1.8. Torque 2045
14.10.5.1.1.9. Efficiencies for Pumps and Compressors 2045
14.10.5.1.1.9.1. Incompressible Flows 2046

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.10.5.1.1.9.2. Compressible Flows 2046
14.10.5.1.1.10. Efficiencies for Turbines 2047
14.10.5.1.1.10.1. Incompressible Flows 2048
14.10.5.1.1.10.2. Compressible Flows 2048
14.10.5.2. Displaying Turbomachinery Averaged Contours 2049
14.10.5.2.1. Steps for Generating Turbomachinery Averaged Contour Plots 2049
14.10.5.2.2. Contour Plot Options 2051
14.10.5.3. Displaying Turbomachinery 2D Contours 2051
14.10.5.3.1. Steps for Generating Turbo 2D Contour Plots 2051
14.10.5.3.2. Contour Plot Options 2053
14.10.5.4. Generating Averaged XY Plots of Turbomachinery Solution Data 2053
14.10.5.4.1. Steps for Generating Turbo Averaged XY Plots 2053
14.10.5.5. Globally Setting the Turbomachinery Topology 2055
14.10.6. Calculating Turbomachine Performance 2055
14.10.6.1. Turbo Performance 2055
14.10.6.2. Calculating Efficiency using Named Expressions 2057
14.10.6.2.1. Limitations 2058
14.10.6.3. Calculating Efficiency using Turbo Report 2058
14.10.6.4. Calculating Total Pressure and Total Temperature Ratios using Named Expressions 2059
14.10.6.5. Calculating Mass Flow Rates using Named Expressions 2059
15. Modeling Turbulence 2061
15.1. Introduction 2061
15.2. Choosing a Turbulence Model 2064
15.2.1. Reynolds Averaged Navier-Stokes (RANS) Turbulence Models 2064
15.2.1.1. Spalart-Allmaras One-Equation Model 2065
15.2.1.2. k-ϵ Models 2065
15.2.1.3. k-ω Models 2065
15.2.1.4. Generalized k-ω (GEKO) Model 2066
15.2.1.5. Reynold Stress Models 2069
15.2.1.6. Laminar-Turbulent Transition Models 2069
15.2.1.7. Curvature Correction for the Spalart-Allmaras and Two-Equation Models 2071
15.2.1.8. Corner Flow Correction 2071
15.2.1.9. Production Limiters for Two-Equation Models 2071
15.2.1.10. Model Enhancements 2071
15.2.1.11. Wall Treatment for RANS Models 2072
15.2.1.12. Grid Resolution for RANS Models 2072
15.2.2. Scale-Resolving Simulation (SRS) Models 2073
15.2.2.1. Large Eddy Simulation (LES) 2073
15.2.2.2. Hybrid RANS-LES Models 2074
15.2.2.2.1. Scale-Adaptive Simulation (SAS) 2074
15.2.2.2.2. Detached Eddy Simulation (DES) 2075
15.2.2.2.3. Shielded Detached Eddy Simulation (SDES) and Stress-Blended Eddy Simulation (SBES) 2076
15.2.2.3. Zonal Modeling and Embedded LES (ELES) 2076
15.2.3. Grid Resolution SRS Models 2076
15.2.3.1. Wall Boundary Layers 2077
15.2.3.2. Free Shear Flows 2077
15.2.4. Numerics Settings for SRS Models 2077
15.2.4.1. Time Discretization 2078
15.2.4.2. Spatial Discretization 2078

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15.2.4.3. Iterative Scheme 2079
15.2.4.3.1. Convergence Control 2079
15.2.5. Model Hierarchy 2080
15.3. Steps in Using a Turbulence Model 2081
15.4. Setting Up the Spalart-Allmaras Model 2084
15.5. Setting Up the k-ε Model 2085
15.5.1. Setting Up the Standard or Realizable k-ε Model 2085
15.5.2. Setting Up the RNG k-ε Model 2087
15.6. Setting Up the k-ω Model 2089
15.6.1. Setting Up the Standard k-ω Model 2089
15.6.2. Setting up the Generalized k-ω (GEKO) Model 2090
15.6.3. Setting Up the Baseline (BSL) k-ω Model 2093
15.6.4. Setting Up the Shear-Stress Transport k-ω Model 2095
15.6.5. Setting up the WJ-BSL-EARSM Model 2097
15.7. Setting Up the Transition k-kl-ω Model 2100
15.8. Setting Up the Transition SST Model 2101
15.9. Setting Up the Algebraic or Intermittency Transition Model 2103
15.10. Setting Up the Reynolds Stress Model 2105
15.11. Setting Up Scale-Adaptive Simulation (SAS) Modeling 2109
15.12. Setting Up the Detached Eddy Simulation Model 2112
15.12.1. Setting Up DES with the Spalart-Allmaras Model 2112
15.12.2. Setting Up DES with the Realizable k-ε Model 2114
15.12.3. Setting Up DES with the SST k-ω Model 2115
15.12.4. Setting Up DES with the BSL k-ω Model 2118
15.12.5. Setting Up DES with the Transition SST Model 2120
15.13. Setting Up the Large Eddy Simulation Model 2122
15.14. Model Constants 2124
15.15. Setting Up the Embedded Large Eddy Simulation (ELES) Model 2124
15.16. Setup Options for All Turbulence Modeling 2128
15.16.1. Including the Viscous Heating Effects 2129
15.16.2. Including Buoyancy Effects on Turbulence 2129
15.16.3. Including the Curvature Correction for the Spalart-Allmaras and Two-Equation Turbulence Models 2131
15.16.4. Including Corner Flow Correction 2132
15.16.5. Including the Compressibility Effects Option 2134
15.16.6. Including Production Limiters for Two-Equation Models 2134
15.16.7. Vorticity- and Strain/Vorticity-Based Production 2135
15.16.8. Delayed Detached Eddy Simulation (DDES) 2135
15.16.9. Differential Viscosity Modification 2135
15.16.10. Swirl Modification 2136
15.16.11. Low-Re Corrections 2136
15.16.12. Shear Flow Corrections 2136
15.16.13. Turbulence Damping 2136
15.16.14. Including Pressure Gradient Effects 2137
15.16.15. Including Thermal Effects 2137
15.16.16. Including the Wall Reflection Term 2137
15.16.17. Solving the k Equation to Obtain Wall Boundary Conditions 2137
15.16.18. Quadratic Pressure-Strain Model 2138
15.16.19. Stress-Omega and Stress-BSL Models 2138
15.16.20. Subgrid-Scale Model 2138
15.16.21. Customizing the Turbulent Viscosity 2139

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15.16.22. Customizing the Turbulent Prandtl and Schmidt Numbers 2139
15.16.23. Modeling Turbulence with Non-Newtonian Fluids 2139
15.16.24. Including Scale-Adaptive Simulation with ω-Based URANS Models 2140
15.16.25. Including Detached Eddy Simulation with the Transition SST Model 2140
15.16.26. Including the SDES or SBES Model with RANS Models 2140
15.16.27. Shielding Functions for the BSL / SST / Transition SST Detached Eddy Simulation Model 2143
15.17. Defining Turbulence Boundary Conditions 2143
15.17.1. Wall Roughness Effects 2143
15.17.2. The Spalart-Allmaras Model 2144
15.17.3. k-ε Models and k-ω Models 2144
15.17.4. Reynolds Stress Model 2144
15.17.5. Scale Resolving Simulations 2146
15.18. Providing an Initial Guess for k and ε (or k and ω) 2147
15.19. Solution Strategies for Turbulent Flow Simulations 2147
15.19.1. Mesh Generation 2148
15.19.2. Accuracy 2148
15.19.3. Convergence 2148
15.19.4. RSM-Specific Solution Strategies 2149
15.19.4.1. Under-Relaxation of the Reynolds Stresses 2149
15.19.4.2. Disabling Calculation Updates of the Reynolds Stresses 2150
15.19.4.3. Residual Reporting for the RSM 2150
15.19.5. LES-Specific Solution Strategies 2150
15.19.5.1. Temporal Discretization 2151
15.19.5.2. Spatial Discretization 2151
15.20. Postprocessing for Turbulent Flows 2151
15.20.1. Custom Field Functions for Turbulence 2160
15.20.2. Postprocessing Turbulent Flow Statistics 2160
15.20.3. Troubleshooting 2162
16. Modeling Thermal Energy 2163
16.1. Introduction 2163
16.2. Modeling Conductive and Convective Heat Transfer 2163
16.2.1. Solving Heat Transfer Problems 2164
16.2.1.1. Limiting the Predicted Temperature Range 2166
16.2.1.2. Modeling Heat Transfer in Two Separated Fluid Regions 2167
16.2.2. Solution Strategies for Heat Transfer Modeling 2167
16.2.2.1. Under-Relaxation of the Energy Equation 2167
16.2.2.2. Under-Relaxation of Temperature When the Enthalpy Equation is Solved 2168
16.2.2.3. Disabling the Species Diffusion Term 2168
16.2.2.4. Step-by-Step Solutions 2168
16.2.2.4.1. Decoupled Flow and Heat Transfer Calculations 2168
16.2.2.4.2. Coupled Flow and Heat Transfer Calculations 2169
16.2.2.5. Conjugate Heat Transfer 2169
16.2.2.5.1. Specifying the Solid Time Step Size 2169
16.2.2.5.1.1. Automatic Time Step Size Calculation 2172
16.2.2.5.2. Loosely Coupled Conjugate Heat Transfer 2172
16.2.2.5.3. Time Averaged Explicit Thermal Coupling 2177
16.2.2.5.4. Settings for Anisotropic Solid Zones 2179
16.2.3. Postprocessing Heat Transfer Quantities 2180
16.2.3.1. Available Variables for Postprocessing 2180
16.2.3.2. Definition of Enthalpy and Energy in Reports and Displays 2180

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16.2.3.3. Reporting Heat Transfer Through Boundaries 2180
16.2.3.4. Reporting Heat Transfer Through a Surface 2181
16.2.3.5. Reporting Averaged Heat Transfer Coefficients 2181
16.2.3.6. Exporting Heat Flux Data 2181
16.2.4. Natural Convection 2182
16.2.5. Shell Conduction 2182
16.2.5.1. Introduction 2182
16.2.5.2. Physical Treatment 2183
16.2.5.3. Limitations of Shell Conduction Walls 2184
16.2.5.4. Managing Conduction Walls 2184
16.2.5.5. Initializing Shells 2187
16.2.5.6. Locking the Temperature for Shells 2187
16.2.5.7. Postprocessing Shells 2187
16.2.6. Anisotropic Thermal Conductivity with Curvilinear Coordinate System (CCS) 2190
16.2.6.1. Workflow for Anisotropic Thermal Conductivity with CCS 2190
16.3. Modeling Radiation 2192
16.3.1. Using the Radiation Models 2193
16.3.2. Setting Up the P-1 Model with Non-Gray Radiation 2194
16.3.3. Setting Up the DTRM 2195
16.3.3.1. Defining the Rays 2196
16.3.3.2. Controlling the Clusters 2197
16.3.3.3. Controlling the Rays 2197
16.3.3.4. Writing and Reading the DTRM Ray File 2198
16.3.3.5. Displaying the Clusters 2198
16.3.4. Setting Up the S2S Model 2199
16.3.4.1. View Factors and Clustering Settings 2200
16.3.4.1.1. Forming Surface Clusters 2201
16.3.4.1.1.1. Setting the Split Angle for Clusters 2204
16.3.4.1.2. Setting Up the View Factor Calculation 2204
16.3.4.1.2.1. Selecting the Basis for Computing View Factors 2204
16.3.4.1.2.2. Selecting the Method for Computing View Factors 2205
16.3.4.1.2.3. Accounting for Blocking Surfaces 2206
16.3.4.1.2.4. Specifying Boundary Zone Participation 2206
16.3.4.2. Computing View Factors 2209
16.3.4.3. Reading View Factors into Ansys Fluent 2211
16.3.5. Setting Up the DO Model 2211
16.3.5.1. Angular Discretization 2212
16.3.5.2. Defining Non-Gray Radiation for the DO Model 2212
16.3.5.3. Enabling DO/Energy Coupling 2214
16.3.6. Setting Up the MC Model 2214
16.3.7. Defining Material Properties for Radiation 2216
16.3.7.1. Absorption Coefficient for a Non-Gray Model 2217
16.3.7.2. Refractive Index for a Non-Gray Model 2217
16.3.8. Defining Boundary Conditions for Radiation 2217
16.3.8.1. Inlet and Outlet Boundary Conditions 2218
16.3.8.1.1. Emissivity 2218
16.3.8.1.2. Black Body Temperature 2218
16.3.8.1.3. Inlet and Outlet Boundary Conditions for the DO and MC Models 2219
16.3.8.2. Wall Boundary Conditions for the DTRM, P-1, S2S, and Rosseland Models 2220
16.3.8.2.1. Boundary Conditions for the S2S Model 2220
16.3.8.3. Wall Boundary Conditions for the DO Model 2220

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16.3.8.3.1. Opaque Wall for the DO Model 2220
16.3.8.3.2. Semi-Transparent Walls for the DO Model 2222
16.3.8.4. Wall Boundary Conditions for the MC Model 2226
16.3.8.4.1. Opaque Walls for the MC Model 2226
16.3.8.4.2. Semi-Transparent Walls for the MC Model 2233
16.3.8.5. Solid Cell Zones Conditions for the DO or MC Models 2235
16.3.8.6. Thermal Boundary Conditions 2236
16.3.9. Solution Strategies for Radiation Modeling 2236
16.3.9.1. P-1 Model Solution Parameters 2236
16.3.9.2. DTRM Solution Parameters 2237
16.3.9.3. S2S Solution Parameters 2238
16.3.9.4. DO Solution Parameters 2239
16.3.9.5. MC Solution Parameters 2239
16.3.9.6. Iteration Parameters For Radiation Solve Frequency 2240
16.3.9.7. Running the Calculation 2241
16.3.9.7.1. Residual Reporting for the P-1 Model 2242
16.3.9.7.2. Residual Reporting for the DO Model 2242
16.3.9.7.3. Residual Reporting for the DTRM 2242
16.3.9.7.4. Residual Reporting for the S2S Model 2242
16.3.9.7.5. Disabling the Update of the Radiation Fluxes 2243
16.3.10. Postprocessing Radiation Quantities 2243
16.3.10.1. Available Variables for Postprocessing 2243
16.3.10.2. Reporting Radiative Heat Transfer Through Boundaries 2244
16.3.10.3. Overall Heat Balances When Using the DTRM 2245
16.3.10.4. Displaying Rays and Clusters for the DTRM 2245
16.3.10.4.1. Displaying Clusters 2245
16.3.10.4.2. Displaying Rays 2246
16.3.10.4.3. Including the Mesh in the Display 2246
16.3.10.5. Reporting Radiation in the S2S Model 2246
16.3.11. Solar Load Model 2247
16.3.11.1. Introduction 2248
16.3.11.2. Solar Ray Tracing 2248
16.3.11.2.1. Shading Algorithm 2249
16.3.11.2.2. Glazing Materials 2250
16.3.11.2.3. Inputs 2250
16.3.11.3. Solar Irradiation 2252
16.3.11.4. Solar Calculator 2252
16.3.11.4.1. Inputs/Outputs 2253
16.3.11.4.2. Theory 2253
16.3.11.4.3. Computation of Load Distribution 2254
16.3.11.5. Using the Solar Load Model 2255
16.3.11.5.1. User-Defined Functions (UDFs) for Solar Load 2256
16.3.11.5.2. Setting Up the Solar Load Model 2256
16.3.11.5.3. Setting Boundary Conditions for Solar Loading 2261
16.3.11.5.4. Solar Ray Tracing 2261
16.3.11.5.5. Solar Irradiation 2266
16.3.11.5.6. Text Interface-Only Commands 2268
16.3.11.5.6.1. Automatically Saving Solar Ray Tracing Data 2268
16.3.11.5.6.2. Automatically Reading Solar Data 2268
16.3.11.5.6.3. Aligning the Camera Direction With the Position of the Sun 2269
16.3.11.5.6.4. Specifying the Scattering Fraction 2269

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16.3.11.5.6.5. Applying the Solar Load on Adjacent Fluid Cells 2269
16.3.11.5.6.6. Specifying Quad Tree Refinement Factor 2270
16.3.11.5.6.7. Specifying Ground Reflectivity 2270
16.3.11.5.6.8. Reverting to Single Band Implementation of DO Model 2270
16.3.11.5.6.9. Additional Text Interface Commands 2270
16.3.11.6. Postprocessing Solar Load Quantities 2271
16.3.11.6.1. Solar Load Animation at Different Sun Positions 2272
16.3.11.6.2. Reporting and Displaying Solar Load Quantities 2273
16.4. Modeling Periodic Heat Transfer 2274
16.4.1. Overview and Limitations 2274
16.4.1.1. Overview 2274
16.4.1.2. Constraints for Periodic Heat Transfer Predictions 2274
16.4.2. Theory 2275
16.4.2.1. Definition of the Periodic Temperature for Constant- Temperature Wall Conditions 2275
16.4.2.2. Definition of the Periodic Temperature Change σ for Specified Heat Flux Conditions 2276
16.4.3. Using Periodic Heat Transfer 2276
16.4.4. Solution Strategies for Periodic Heat Transfer 2278
16.4.5. Monitoring Convergence 2278
16.4.6. Postprocessing for Periodic Heat Transfer 2279
16.5. Modeling Heat Exchangers 2279
16.5.1. Choosing a Heat Exchanger Model 2280
16.5.2. The Dual Cell Model 2282
16.5.2.1. Restrictions 2282
16.5.2.2. Using the Dual Cell Heat Exchanger Model 2283
16.5.3. The Macro Heat Exchanger Models 2292
16.5.3.1. Restrictions 2293
16.5.3.2. Using the Ungrouped Macro Heat Exchanger Model 2294
16.5.3.2.1. Selecting the Zone for the Heat Exchanger 2299
16.5.3.2.2. Specifying Heat Exchanger Performance Data 2299
16.5.3.2.3. Specifying the Auxiliary Fluid Inlet and Pass-to-Pass Directions 2300
16.5.3.2.4. Defining the Macros 2300
16.5.3.2.4.1. Viewing the Macros 2301
16.5.3.2.5. Specifying the Auxiliary Fluid Properties and Conditions 2302
16.5.3.2.6. Setting the Pressure-Drop Parameters and Effectiveness 2303
16.5.3.2.6.1. Using the Default Core Porosity Model 2303
16.5.3.2.6.2. Defining a New Core Porosity Model 2304
16.5.3.2.6.3. Reading Heat Exchanger Parameters from an External File 2305
16.5.3.2.6.4. Viewing the Parameters for an Existing Core Model 2305
16.5.3.3. Using the Grouped Macro Heat Exchanger Model 2305
16.5.3.3.1. Selecting the Fluid Zones for the Heat Exchanger Group 2311
16.5.3.3.2. Selecting the Upstream Heat Exchanger Group 2311
16.5.3.3.3. Specifying the Auxiliary Fluid Inlet and Pass-to-Pass Directions 2312
16.5.3.3.4. Specifying the Auxiliary Fluid Properties 2312
16.5.3.3.5. Specifying Supplementary Auxiliary Fluid Streams 2312
16.5.3.3.6. Initializing the Auxiliary Fluid Temperature 2312
16.5.4. Postprocessing for the Heat Exchanger Model 2313
16.5.4.1. Heat Exchanger Reporting 2313
16.5.4.1.1. Computed Heat Rejection 2313
16.5.4.1.2. **Inlet/Outlet Temperature** 2314

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16.5.4.1.3. **Mass Flow Rate** 2316
16.5.4.1.4. **Specific Heat** 2317
16.5.4.2. Total Heat Rejection Rate 2317
16.5.5. Useful Reporting TUI Commands 2318
16.6. Thermal Analysis of Printed Circuit Boards 2319
16.6.1. PCB Model Workflow with ECAD File 2320
16.6.1.1. Using the PCB Model with ECAD File 2320
16.6.2. PCB Model Workflow with Board Configuration File 2326
16.6.2.1. Using the PCB Model with Board Configuration File 2326
16.6.3. Postprocessing for the PCB Model 2330
16.6.4. Limitations for the PCB Model 2330
17. Modeling Hypersonic Flow ..... ?
17.1. Introduction to Hypersonic Flows ..... ?
17.2. High Speed Numerics ..... ?
17.3. Modeling Non-Equilibrium Gas Dissociation Using Finite Rate Chemistry ..... ?
17.4. Modeling Transport Properties Using Gupta Curve Fits ..... ?
17.4.1. **Using Gupta Curve Fits** ..... ?
17.5. Modeling Hypersonic Flows Using the Two-Temperature Model ..... ?
17.5.1. Using the Two-Temperature Model ..... ?
17.6. Partial Slip for Rarefied Gases ..... ?
17.7. Temperature Jump for Rarefied Gases ..... ?
17.8. Partial Catalytic Boundary Condition for Walls ..... ?
17.9. The Ablation Condition at Wall Boundaries ..... ?
17.10. Best Practices ..... ?
17.10.1. Setting Up Gas Properties ..... ?
17.10.2. Solver Settings ..... ?
17.10.3. Solution Initialization ..... ?
17.10.4. Solution Monitoring and Postprocessing ..... ?
18. Fluent's Virtual Blade Model 2331
18.1. Introduction 2331
18.2. The Virtual Blade Model (VBM) 2332
18.2.1. VBM Mode 2334
18.2.2. Rotor Disks 2334
18.2.3. Blade Geometry 2336
18.2.4. Blade Pitch 2336
18.2.5. Blade Flapping 2337
18.2.6. Rotor Trimming 2338
18.2.7. Tip Losses 2340
18.3. Meshing Guidelines and Creating the VBM Disk 2341
18.3.1. Embedded Disk 2342
18.3.2. Floating Disk 2342
18.3.3. General Considerations 2343
18.4. Airfoil File Format 2344
18.5. Enabling the VBM 2345
18.6. VBM Configuration 2346
18.7. VBM Field Variables 2350
18.8. VBM Monitoring and Report Definition 2351
18.9. References 2352
19. Modelling with Finite-Rate Chemistry 2355
19.1. Modeling Species Transport and Finite-Rate Chemistry 2355
19.1.1. Volumetric Reactions 2356

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

19.1.1.1. Overview of User Inputs for Modeling Species Transport and Reactions 2356
19.1.1.1.1. Mixture Materials 2357
19.1.1.2. Enabling Species Transport and Reactions and Choosing the Mixture Material 2359
19.1.1.3. Importing a Volumetric Kinetic Mechanism in CHEMKIN Format 2368
19.1.1.3.1. Using Ansys Encrypted Mechanisms 2369
19.1.1.3.2. Procedure for Importing Volumetric CHEMKIN Mechanisms 2369
19.1.1.3.3. CHEMKIN Mechanisms Included with Ansys Fluent 2373
19.1.1.4. Defining Properties for the Mixture and Its Constituent Species 2374
19.1.1.4.1. Defining the Species in the Mixture 2375
19.1.1.4.1.1. Overview of the Species Dialog Box 2376
19.1.1.4.1.2. Adding Species to the Mixture 2377
19.1.1.4.1.3. Removing Species from the Mixture 2378
19.1.1.4.1.4. Assigning the Last Species 2378
19.1.1.4.1.5. The Naming and Ordering of Species 2378
19.1.1.4.2. Defining Reactions 2379
19.1.1.4.2.1. Inputs for Reaction Definition 2379
19.1.1.4.2.2. Defining Species and Reactions for Fuel Mixtures 2387
19.1.1.4.3. Defining Zone-Based Reaction Mechanisms 2387
19.1.1.4.3.1. Inputs for Reaction Mechanism Definition 2387
19.1.1.4.4. Defining Physical Properties for the Mixture 2390
19.1.1.4.5. Defining Physical Properties for the Species in the Mixture 2390
19.1.1.5. Setting up Coal Simulations with the Coal Calculator Dialog Box 2391
19.1.1.6. Defining Cell Zone and Boundary Conditions for Species 2394
19.1.1.6.1. Diffusion at Inlets with the Pressure-Based Solver 2395
19.1.1.7. Defining Other Sources of Chemical Species 2395
19.1.1.8. Solution Procedures for Chemical Mixing and Finite-Rate Chemistry 2395
19.1.1.8.1. Stability and Convergence in Reacting Flows 2395
19.1.1.8.2. Two-Step Solution Procedure (Steady-state Only) 2396
19.1.1.8.3. Density Under-Relaxation 2396
19.1.1.8.4. Ignition in Steady-State Combustion Simulations 2397
19.1.1.8.5. Solution of Stiff Chemistry Systems 2397
19.1.1.8.6. Eddy-Dissipation Concept Model Solution Procedure 2398
19.1.1.9. Postprocessing for Species Calculations 2398
19.1.1.9.1. Averaged Species Concentrations 2399
19.1.2. Wall Surface Reactions and Chemical Vapor Deposition 2400
19.1.2.1. Overview of Surface Species and Wall Surface Reactions 2400
19.1.2.2. Importing a Surface Kinetic Mechanism in CHEMKIN Format 2400
19.1.2.2.1. Compatibility and Limitations for Gas Phase Reactions 2403
19.1.2.2.2. Compatibility and Limitations for Surface Reactions 2403
19.1.2.3. Manual Inputs for Wall Surface Reactions 2403
19.1.2.4. Including Mass Transfer To Surfaces in the Continuity Equation 2405
19.1.2.5. Wall Surface Mass Transfer Effects in the Energy Equation 2405
19.1.2.6. Modeling the Heat Release Due to Wall Surface Reactions 2406
19.1.2.7. Solution Procedures for Wall Surface Reactions 2406
19.1.2.8. Postprocessing for Surface Reactions 2406
19.1.3. Particle Reactions 2406
19.1.3.1. Combusting Particle Surface Reactions 2406
19.1.3.1.1. Limitations 2407
19.1.3.1.2. User Inputs for Particle Surface Reactions 2407
19.1.3.1.3. Modeling Gaseous Solid Catalyzed Reactions 2408

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

19.1.3.1.4. Using the Multiple Surface Reactions Model for Discrete-Phase Particle Combustion 2408
19.1.3.2. Multicomponent Particles with Chemical Reactions 2409
19.1.4. Electrochemical Reactions 2414
19.1.4.1. Overview and Limitation of Electrochemical Reactions 2414
19.1.4.2. User Inputs for Electrochemical Reactions 2414
19.1.4.3. Electrochemical Reaction Effects in the Energy Equation 2422
19.1.4.4. Electrochemical Reaction Effects in the Species Transport Equation 2422
19.1.4.5. Including Mass Transfer in Continuity 2422
19.1.4.6. Solution Procedures for Electrochemical Reactions 2422
19.1.4.7. Modeling Corrosion with the Water Corrosion Pre Tool 2423
19.1.4.7.1. Background 2423
19.1.4.7.2. Procedure for Setting Corrosion Simulations using the Water Corrosion Pre Tool 2424
19.1.5. Species Transport Without Reactions 2427
19.1.6. Reacting Channel Model 2429
19.1.6.1. Overview and Limitations of the Reacting Channel Model 2429
19.1.6.2. Enabling the Reacting Channel Model 2430
19.1.6.3. Boundary Conditions for Channel Walls 2435
19.1.6.4. Postprocessing for Reacting Channel Model Calculations 2435
19.1.7. Reactor Network Model 2437
19.1.7.1. Overview and Limitations of the Reactor Network Model 2438
19.1.7.2. Solving Reactor Networks 2438
19.1.7.3. Postprocessing Reactor Network Calculations 2442
19.2. Modeling a Composition PDF Transport Problem 2442
19.2.1. Limitation 2442
19.2.2. Steps for Using the Composition PDF Transport Model 2442
19.2.3. Enabling the Lagrangian Composition PDF Transport Model 2444
19.2.4. Enabling the Eulerian Composition PDF Transport Model 2447
19.2.4.1. Defining Species Boundary Conditions 2449
19.2.4.1.1. Equilibrating Inlet Streams 2450
19.2.5. Initializing the Solution 2450
19.2.6. Monitoring the Solution 2451
19.2.6.1. Running Unsteady Composition PDF Transport Simulations 2453
19.2.6.2. Running Compressible Lagrangian PDF Transport Simulations 2453
19.2.6.3. Running Lagrangian PDF Transport Simulations with Conjugate Heat Transfer 2453
19.2.7. Postprocessing for Lagrangian PDF Transport Calculations 2453
19.2.7.1. Reporting Options 2453
19.2.7.2. Particle Tracking Options 2454
19.2.8. Postprocessing for Eulerian PDF Transport Calculations 2455
19.2.8.1. Reporting Options 2455
19.3. Using Chemistry Acceleration 2456
19.3.1. Using ISAT 2457
19.3.1.1. ISAT Parameters 2458
19.3.1.2. Monitoring ISAT 2458
19.3.1.3. Using ISAT Efficiently 2459
19.3.1.4. Reading and Writing ISAT Tables 2460
19.3.2. Using Dynamic Mechanism Reduction 2461
19.3.2.1. Mechanism Reduction Parameters 2462
19.3.2.2. Monitoring and Postprocessing Dynamic Mechanism Reduction 2464
19.3.2.3. Using Dynamic Mechanism Reduction Effectively 2465

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

19.3.3. Using Chemistry Agglomeration 2465
19.3.4. Dimension Reduction 2466
19.3.5. Using Dynamic Cell Clustering 2467
19.3.6. Using Dynamic Adaptive Chemistry with Ansys Fluent CHEMKIN-CFD Solver 2468
20. Modelling of Turbulent Combustion With Reduced Order 2471
20.1. Modeling Non-Premixed Combustion 2471
20.1.1. Steps in Using the Non-Premixed Model 2471
20.1.1.1. Preliminaries 2472
20.1.1.2. Defining the Problem Type 2472
20.1.1.3. Overview of the Problem Setup Procedure 2472
20.1.2. Setting Up the Equilibrium Chemistry Model 2476
20.1.2.1. Choosing Adiabatic or Non-Adiabatic Options 2477
20.1.2.2. Specifying the Operating Pressure for the System 2478
20.1.2.3. Enabling a Secondary Inlet Stream 2478
20.1.2.4. Choosing to Define the Fuel Stream(s) Empirically 2479
20.1.2.5. Enabling the Rich Flammability Limit (RFL) Option 2480
20.1.3. Setting Up the Steady and Unsteady Diffusion Flamelet Models 2481
20.1.3.1. Choosing Adiabatic or Non-Adiabatic Options 2482
20.1.3.2. Specifying the Operating Pressure for the System 2482
20.1.3.3. Specifying a Chemical Mechanism File for Flamelet Generation 2482
20.1.3.4. Importing a Flamelet 2482
20.1.3.5. Using the Unsteady Diffusion Flamelet Model 2482
20.1.3.6. Using the Diesel Unsteady Laminar Flamelet Model 2484
20.1.3.6.1. Recommended Settings for Internal Combustion Engines 2486
20.1.3.7. Resetting Diesel Unsteady Flamelets 2487
20.1.4. Defining the Stream Compositions 2487
20.1.4.1. Setting Boundary Stream Species 2490
20.1.4.1.1. Including Condensed Species 2490
20.1.4.2. Modifying the Database 2490
20.1.4.3. Composition Inputs for Empirically-Defined Fuel Streams 2490
20.1.4.4. Modeling Liquid Fuel Combustion Using the Non-Premixed Model 2491
20.1.4.5. Modeling Coal Combustion Using the Non-Premixed Model 2491
20.1.4.5.1. Defining the Coal Composition: Single-Mixture-Fraction Models 2492
20.1.4.5.2. Defining the Coal Composition: Two-Mixture-Fraction Models 2493
20.1.4.5.3. Additional Coal Modeling Inputs in Ansys Fluent 2495
20.1.4.5.4. Postprocessing Non-Premixed Models of Coal Combustion 2496
20.1.4.5.5. The Coal Calculator 2496
20.1.5. Setting Up Control Parameters 2498
20.1.5.1. Forcing the Exclusion and Inclusion of Equilibrium Species 2499
20.1.5.2. Defining the Flamelet Controls 2500
20.1.5.3. Zeroing Species in the Initial Unsteady Flamelet 2501
20.1.6. Calculating the Flamelets 2501
20.1.6.1. Steady Diffusion Flamelet 2502
20.1.6.2. Unsteady Diffusion Flamelet 2504
20.1.6.3. Saving the Flamelet Data 2506
20.1.6.4. Postprocessing the Flamelet Data 2506
20.1.7. Calculating the Look-Up Tables 2509
20.1.7.1. Full Tabulation of the Two-Mixture-Fraction Model 2514
20.1.7.2. Stability Issues in Calculating Chemical Equilibrium Look-Up Tables 2514
20.1.7.3. Saving the Look-Up Tables 2514
20.1.7.4. Postprocessing the Look-Up Table Data 2515

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

20.1.8. Standard Files for Diffusion Flamelet Modeling 2519
20.1.8.1. Sample Standard Diffusion Flamelet File 2520
20.1.8.2. Missing Species 2521
20.1.9. Setting Up the Inert Model 2521
20.1.9.1. Setting Boundary Conditions for Inert Transport 2523
20.1.9.2. Initializing the Inert Stream 2524
20.1.9.2.1. Inert Fraction 2524
20.1.9.2.2. Inert Composition 2524
20.1.9.3. Resetting Inert EGR 2525
20.1.10. Defining Non-Premixed Boundary Conditions 2526
20.1.10.1. Input of Mixture Fraction Boundary Conditions 2526
20.1.10.2. Diffusion at Inlets 2527
20.1.10.3. Input of Thermal Boundary Conditions and Fuel Inlet Velocities 2527
20.1.11. Defining Non-Premixed Physical Properties 2527
20.1.12. Solution Strategies for Non-Premixed Modeling 2528
20.1.12.1. Single-Mixture-Fraction Approach 2528
20.1.12.2. Two-Mixture-Fraction Approach 2529
20.1.12.3. Starting a Non-Premixed Calculation From a Previous Case File 2529
20.1.12.3.1. Retrieving the PDF File During Case File Reads 2530
20.1.12.4. Solving the Flow Problem 2530
20.1.12.4.1. Under-Relaxation Factors for PDF Equations 2531
20.1.12.4.2. Density Under-Relaxation 2531
20.1.12.4.3. Tuning the PDF Parameters for Two-Mixture-Fraction Calculations 2531
20.1.13. Enabling Robust Numerics for Combustion with a PDF Table 2532
20.1.14. Postprocessing the Non-Premixed Model Results 2533
20.1.14.1. Postprocessing for Inert Calculations 2535
20.2. Modeling Premixed Combustion 2536
20.2.1. Limitations of the Premixed Combustion Model 2536
20.2.2. Using the Premixed Combustion Model 2536
20.2.2.1. Enabling the Premixed Combustion Model 2537
20.2.2.2. Choosing an Adiabatic or Non-Adiabatic Model 2538
20.2.3. Setting Up the C-Equation and G-Equation Models 2538
20.2.3.1. Modifying the Constants for the Zimont Flame Speed Model 2539
20.2.3.2. Modifying the Constants for the Peters Flame Speed Model 2540
20.2.3.3. Additional Options for the G-Equation Model 2540
20.2.3.4. Defining Physical Properties for the Unburnt Mixture 2540
20.2.3.5. Setting Boundary Conditions for the Progress Variable 2541
20.2.3.6. Initializing the Progress Variable 2541
20.2.4. Postprocessing for Premixed Combustion Calculations 2541
20.2.4.1. Computing Species Concentrations 2542
20.3. Modeling Partially Premixed Combustion 2543
20.3.1. Limitations 2543
20.3.2. Using the Partially Premixed Combustion Model 2543
20.3.2.1. Setup and Solution Procedure 2543
20.3.2.2. Importing a Flamelet 2546
20.3.2.3. Flamelet Generated Manifold 2546
20.3.2.3.1. Premixed Flamelet Generated Manifolds 2547
20.3.2.3.1.1. Editing the Flamelet Grid Distribution 2549
20.3.2.3.2. Diffusion Flamelet Generated Manifolds 2551
20.3.2.3.3. Using the Heat Loss Modeling Capability for Nonadiabatic FGM 2552
20.3.2.4. Calculating the Look-Up Tables 2554

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

20.3.2.4.1. Postprocessing the Look-Up Tables with Flamelet Generated Manifolds 2558
20.3.2.5. Standard Files for Flamelet Generated Manifold Modeling 2561
20.3.2.5.1. Sample Standard FGM File 2561
20.3.2.6. Setting Premix Flame Propagation Parameters 2562
20.3.2.7. Modifying the Unburnt Mixture Property Polynomials 2564
20.3.2.8. Modeling Strained Laminar Flame Speed 2567
20.3.2.9. Modeling In Cylinder Combustion 2569
20.3.2.10. Postprocessing for FGM Scalar Transport Calculations 2569
21. Modeling Engine Ignition 2571
21.1. Using the Spark Model 2571
21.2. Using the Autoignition Models 2574
21.3. Using the Crevice Model 2577
21.3.1. Setup Procedure 2577
21.3.2. Crevice Model Solution Details 2580
21.3.3. Postprocessing for the Crevice Model 2580
21.3.3.1. Using the Crevice Output File 2581
22. Modeling Pollutant Formation 2585
22.1. NOx Formation 2585
22.1.1. Using the NOx Model 2585
22.1.1.1. Decoupled Analysis: Overview 2585
22.1.1.2. Enabling the NOx Models 2586
22.1.1.3. Defining the Fuel Streams 2588
22.1.1.4. Specifying a User-Defined Function for the NOx Rate 2590
22.1.1.5. Setting Thermal NOx Parameters 2591
22.1.1.6. Setting Prompt NOx Parameters 2591
22.1.1.7. Setting Fuel NOx Parameters 2592
22.1.1.7.1. Setting Gaseous and Liquid Fuel NOx Parameters 2592
22.1.1.7.2. Setting Solid (Coal) Fuel NOx Parameters 2593
22.1.1.8. Setting N2O Pathway Parameters 2594
22.1.1.9. Setting Parameters for NOx Reburn 2595
22.1.1.10. Setting SNCR Parameters 2596
22.1.1.11. Setting Turbulence Parameters 2598
22.1.1.12. Defining Boundary Conditions for the NOx Model 2600
22.1.2. Solution Strategies 2600
22.1.3. Postprocessing 2601
22.2. Soot Formation 2602
22.2.1. Using the Soot Models 2603
22.2.1.1. Setting Up the One-Step Model 2604
22.2.1.2. Setting Up the Two-Step Model 2606
22.2.1.3. Setting Up the Moss-Brookes Model and the Hall Extension 2608
22.2.1.3.1. Specifying a User-Defined Function for the Soot Oxidation Rate 2612
22.2.1.3.2. Specifying a User-Defined Function for the Soot Precursor Concentration 2613
22.2.1.3.3. Species Definition for the Moss-Brookes Model with a User-Defined Precursor Correlation 2613
22.2.1.4. Setting Up the Method of Moments Soot Model 2616
22.2.1.5. Defining Boundary Conditions for the Soot Model 2621
22.2.1.6. Reporting Soot Quantities 2621
22.3. Using the Decoupled Detailed Chemistry Model 2622
23. Predicting Aerodynamically Generated Noise 2625
23.1. Overview 2625
23.1.1. Direct Method 2625

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

23.1.2. Integral Method by Ffowcs Williams and Hawkings 2626
23.1.3. Method Based on Wave Equation 2627
23.1.4. Broadband Noise Source Models 2627
23.2. Using the Ffowcs Williams and Hawkings Acoustics Model 2628
23.2.1. Enabling the FW-H Acoustics Model 2629
23.2.1.1. Setting Model Constants 2630
23.2.1.2. Computing Sound "on the Fly" 2631
23.2.1.3. Writing Source Data Files 2632
23.2.1.3.1. Exporting Source Data Without Enabling the FW-H Model: Using the Ansys Fluent ASD Format 2633
23.2.1.3.2. Exporting Source Data Without Enabling the FW-H Model: Using the CGNS Format 2633
23.2.2. Specifying Source Surfaces 2635
23.2.2.1. Saving Source Data 2637
23.2.3. Specifying Acoustic Receivers 2638
23.2.4. Specifying the Time Step Size 2639
23.2.5. Postprocessing the FW-H Acoustics Model Data 2641
23.2.5.1. Writing Acoustic Signals 2641
23.2.5.2. Reading Unsteady Acoustic Source Data 2641
23.2.5.2.1. Pruning the Signal Data Automatically 2643
23.2.5.3. Reporting the Static Pressure Time Derivative 2644
23.2.5.4. Using the FFT Capabilities for Sound Pressure Signals 2644
23.2.6. FFT of Acoustic Sources: Band Analysis and Export of Surface Pressure Spectra 2644
23.2.6.1. Using the FFT of Acoustic Sources 2645
23.3. Using the Acoustics Wave Equation Model 2653
23.3.1. Specifying Source Mask and Sponge Regions 2654
23.3.2. Solution Controls for the Acoustics Wave Equation 2657
23.3.3. Solution Initialization 2659
23.3.4. Postprocessing 2660
23.3.5. Using the Kirchhoff Integral Model 2661
23.4. Using the Broadband Noise Source Models 2664
23.4.1. Enabling the Broadband Noise Source Models 2665
23.4.1.1. Setting Model Constants 2665
23.4.2. Postprocessing the Broadband Noise Source Model Data 2666
23.5. Sponge Layers 2667
24. Modeling Discrete Phase 2671
24.1. Introduction 2671
24.1.1. Concepts 2672
24.1.1.1. Uncoupled vs. Coupled DPM 2672
24.1.1.2. Steady vs. Unsteady Tracking 2673
24.1.1.3. Parcels 2673
24.1.2. Limitations 2674
24.1.2.1. Limitation on the Particle Volume Fraction 2674
24.1.2.2. Limitation on the Particle Knudsen Number 2675
24.1.2.3. Limitation on Modeling Continuous Suspensions of Particles 2675
24.1.2.4. Limitations on Modeling Particle Rotation 2675
24.1.2.5. Limitations on Using the Discrete Phase Model with Other Ansys Fluent Models 2675
24.1.2.6. Limitations on Using the Hybrid Parallel Method 2676
24.2. Steps for Using the Discrete Phase Models 2677
24.2.1. Options for Interaction with the Continuous Phase 2678
24.2.2. Steady/Transient Treatment of Particles 2679

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.2.3. Tracking Settings for the Discrete Phase Model 2683
24.2.4. Drag Laws 2695
24.2.5. Physical Models for the Discrete Phase Model 2695
24.2.5.1. Including Radiation Heat Transfer Effects on the Particles 2697
24.2.5.2. Including Thermophoretic Force Effects on the Particles 2697
24.2.5.3. Including Saffman Lift Force Effects on the Particles 2697
24.2.5.4. Including the Virtual Mass Force and Pressure Gradient Effects on Particles 2697
24.2.5.5. Monitoring Erosion/Accretion of Particles at Walls 2697
24.2.5.6. Pressure Options for Vaporization Models 2698
24.2.5.7. Considering Pressure Dependence in Boiling 2698
24.2.5.8. Including the Effect of Droplet Temperature on Latent Heat 2700
24.2.5.9. Including the Effect of Particles on Turbulent Quantities 2700
24.2.5.10. Including Collision and Droplet Coalescence 2700
24.2.5.11. Including the DEM Collision Model 2701
24.2.5.12. Including Droplet Breakup 2701
24.2.5.13. Modeling Collision Using the DEM Model 2701
24.2.5.13.1. Limitations 2707
24.2.5.13.2. Numeric Recommendations 2707
24.2.5.14. Including the Volume Displacement of Particles 2707
24.2.6. User-Defined Functions 2708
24.2.7. Numerics of the Discrete Phase Model 2710
24.2.7.1. Numerics for Tracking of the Particles 2712
24.2.7.2. Including Coupled Heat-Mass Solution Effects on the Particles 2713
24.2.7.3. Tracking in a Reference Frame 2714
24.2.7.4. Node Based Averaging of Particle Data 2714
24.2.7.5. Linearized Source Terms 2715
24.2.7.6. Using the Dynamic Interaction Range 2716
24.2.7.7. Staggering of Particles in Space and Time 2716
24.2.7.8. Packing Limit 2718
24.2.7.9. Under-Relaxing Lagrangian Wall Film Height 2718
24.3. Setting Initial Conditions for the Discrete Phase 2718
24.3.1. Injection Types 2720
24.3.2. Particle Types 2722
24.3.3. Point Properties for Single Injections 2724
24.3.4. Point Properties for Group Injections 2725
24.3.5. Point Properties for Cone Injections 2726
24.3.6. Point Properties for Surface Injections 2733
24.3.6.1. Using the Rosin-Rammler Diameter Distribution Method for Surface Injections 2733
24.3.6.2. Injecting Wall Film Particles 2734
24.3.7. Point Properties for Volume Injections 2735
24.3.7.1. Using the Rosin-Rammler Diameter Distribution Method for Volume Injections 2735
24.3.8. Point Properties for Plain-Orifice Atomizer Injections 2736
24.3.9. Point Properties for Pressure-Swirl Atomizer Injections 2737
24.3.10. Point Properties for Air-Blast/Air-Assist Atomizer Injections 2738
24.3.11. Point Properties for Flat-Fan Atomizer Injections 2739
24.3.12. Point Properties for Effervescent Atomizer Injections 2741
24.3.13. Point Properties for File Injections 2742
24.3.13.1. Steady File Format 2742
24.3.13.2. Unsteady File Format 2743
24.3.13.3. User Input for File Injections 2744
24.3.14. Point Properties for Condensate Injections 2745

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.3.15. Using the Rosin-Rammler Diameter Distribution Method 2745
24.3.15.1. The Stochastic Rosin-Rammler Diameter Distribution Method 2748
24.3.16. Using the Tabulated (Discrete) Diameter Distribution 2748
24.3.17. Creating and Modifying Injections 2753
24.3.17.1. Creating Injections 2755
24.3.17.2. Modifying Injections 2755
24.3.17.3. Copying Injections 2755
24.3.17.4. Deleting Injections 2755
24.3.17.5. Listing Injection Initial Conditions 2755
24.3.17.6. Listing Injection Properties 2756
24.3.17.7. Reading and Writing Injections 2756
24.3.18. Defining Injection Properties 2756
24.3.19. Specifying Injection-Specific Physical Models 2764
24.3.19.1. Drag Laws 2765
24.3.19.2. Heat Transfer Coefficient 2766
24.3.19.3. Particle Rotation 2767
24.3.19.4. Rough Wall Model 2768
24.3.19.5. Brownian Motion Effects 2768
24.3.19.6. Breakup 2769
24.3.20. Specifying Turbulent Dispersion of Particles 2772
24.3.20.1. Stochastic Tracking 2772
24.3.21. Custom Particle Laws 2774
24.3.22. Defining Properties Common to More than One Injection 2775
24.3.22.1. Modifying Properties 2776
24.3.22.2. Modifying Properties Common to a Subset of Selected Injections 2778
24.3.23. Point Properties for Transient Injections 2778
24.4. Setting Boundary Conditions for the Discrete Phase 2779
24.4.1. Discrete Phase Boundary Condition Types 2780
24.4.1.1. The **reflect** Boundary Condition 2782
24.4.1.2. The **trap** Boundary Condition 2782
24.4.1.3. The **escape** Boundary Condition 2783
24.4.1.4. The **wall-jet** Boundary Condition 2783
24.4.1.5. The **wall-film** Boundary Condition 2783
24.4.1.6. The **interior** Boundary Condition 2783
24.4.1.7. The **reinject** Boundary Condition 2784
24.4.1.8. The **user-defined** Boundary Condition 2785
24.4.2. Default Discrete Phase Boundary Conditions 2785
24.4.3. Coefficients of Restitution 2785
24.4.4. Friction Coefficient 2786
24.4.5. Particle-Wall Impingement Heat Transfer 2786
24.4.6. Setting Particle Erosion and Accretion Parameters 2787
24.5. Particle Erosion Coupled with Dynamic Meshes 2793
24.5.1. Preliminaries 2793
24.5.2. Limitations 2794
24.5.3. Procedure for the Erosion Coupled with Dynamic Mesh Setup and Solution 2794
24.5.4. Postprocessing for Erosion Dynamic Mesh Calculations 2800
24.6. Modeling Lagrangian Wall Films 2800
24.6.1. Limitations on Using the Lagrangian Wall Film Model 2800
24.6.2. Setting the Lagrangian Wall Film Model 2802
24.6.2.1. Setting the Film-to-Wall Heat Transfer Model 2805
24.6.2.2. Removing the Wall Temperature Limiter for Lagrangian Wall-Film Walls 2806

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.6.3. Film Condensation Model 2806
24.6.4. Gas-Side Boundary Layer Model 2809
24.6.5. Patching the Wall Film 2809
24.7. Setting Material Properties for the Discrete Phase 2811
24.7.1. Summary of Property Inputs 2811
24.7.2. Setting Discrete-Phase Physical Properties 2814
24.7.2.1. The Concept of Discrete-Phase Materials 2814
24.7.2.1.1. Defining Additional Discrete-Phase Materials 2816
24.7.2.2. Description of the Properties 2817
24.8. Solution Strategies for the Discrete Phase 2827
24.8.1. Performing Trajectory Calculations 2827
24.8.1.1. Uncoupled Calculations 2828
24.8.1.2. Coupled Calculations 2828
24.8.1.2.1. Procedures for a Coupled Two-Phase Flow 2829
24.8.1.2.2. Stochastic Tracking in Coupled Calculations 2830
24.8.1.2.3. Under-Relaxation of the Interphase Exchange Terms 2830
24.8.2. Resetting the Interphase Exchange Terms 2832
24.8.3. Random Trajectories 2832
24.9. Postprocessing for the Discrete Phase 2833
24.9.1. Displaying of Trajectories 2834
24.9.1.1. Options for Particle Trajectory Plots 2837
24.9.1.2. Controlling the Particle Tracking Style 2838
24.9.1.3. Controlling the Vector Style of Particle Tracks 2841
24.9.1.4. Importing Particle Data 2844
24.9.1.5. Particle Filtering 2845
24.9.1.6. Graphical Display for Axisymmetric Geometries 2846
24.9.2. Particle Tracking Statistics 2846
24.9.3. Summary Reports 2846
24.9.3.1. Trajectory Fates 2848
24.9.3.2. Elapsed Time 2849
24.9.3.3. Mass Transfer Summary 2850
24.9.3.4. Energy Transfer Summary 2850
24.9.3.5. Heat Rate and Energy Reporting 2850
24.9.3.5.1. Change of Heat and Change of Energy Reporting 2852
24.9.3.6. Combusting Particles 2853
24.9.3.7. Combusting Particles with the Multiple Surface Reaction Model 2853
24.9.3.8. Multicomponent Particles 2854
24.9.3.9. Reinjected Particles 2854
24.9.3.10. Evaporated Mass 2855
24.9.4. Step-by-Step Reporting of Trajectories 2856
24.9.5. Reporting of Current Positions for Unsteady Tracking 2858
24.9.6. Reporting of Interphase Exchange Terms (Discrete Phase Sources) 2860
24.9.7. Reporting of Particle Variables 2861
24.9.8. Reporting of Discrete Phase Variables 2869
24.9.9. Reporting of Unsteady DPM Statistics 2871
24.9.10. Sampling of Trajectories 2873
24.9.11. Histogram Reporting of Samples 2875
24.9.11.1. Analysis, Investigation, and Reporting of Samples 2876
24.9.11.2. Data Reduction of Samples 2879
24.9.12. Contour Plots of DPM Particle Sampling Results on a Planar Surface 2880
24.9.13. Summary Reporting of Current Particles 2883

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.9.14. Postprocessing of Erosion/Accretion Rates 2884
24.9.15. Assessing the Risk for Solids Deposit Formation During Selective Catalytic Reduction Process 2885
24.10. Parallel Processing for the Discrete Phase Model 2888
25. Modeling Macroscopic Particles 2893
25.1. Overview and Limitations 2893
25.2. Loading the MPM add-on Module 2894
25.3. Setting up Macroscopic Particle Model Simulations 2895
25.4. Modeling Macroscopic Particles 2895
25.4.1. Specifying Particle Tracking Parameters 2897
25.4.2. Specifying the Drag Law 2898
25.4.3. Defining Parameters for Particle-Particle and Particle-Wall Collisions 2900
25.4.4. Specifying Deposition Parameters 2901
25.4.5. Specifying Injection Parameters 2903
25.4.5.1. Defining MPM Injection Properties 2904
25.4.5.2. Inputs for `point` Injections 2906
25.4.5.3. Inputs for `plane` Injections 2907
25.4.5.4. Inputs for `packing` Injections 2909
25.4.5.5. Inputs for `from-file` Injections 2910
25.4.6. Defining Field Forces 2911
25.4.7. Initializing the MPM 2911
26. Modeling Multiphase Flows 2915
26.1. Introduction 2915
26.2. Steps for Using a Multiphase Model 2915
26.2.1. Enabling the Multiphase Model 2918
26.2.1.1. Inputs for the VOF Model 2919
26.2.1.2. Inputs for the Mixture Multiphase Model 2920
26.2.1.3. Inputs for the Eulerian Multiphase Model 2921
26.2.2. Choosing Volume Fraction Formulation 2921
26.2.2.1. Interface Modeling Type 2922
26.2.2.2. Spatial Discretization Schemes for Volume Fraction 2924
26.2.2.3. Volume Fraction Limits 2926
26.2.2.4. Expert Options 2926
26.2.3. Solving a Homogeneous Multiphase Flow 2929
26.2.4. Modeling Buoyancy-Driven Multiphase Flow 2930
26.2.4.1. Setting the Operating Density for a Buoyancy-Driven Multiphase Flow 2930
26.2.4.2. The Boussinesq Approximation in a Multiphase Flow 2931
26.2.5. Modeling Compressible Flows 2931
26.2.6. Defining the Phases 2932
26.2.7. Including Body Forces 2933
26.2.8. Modeling Multiphase Species Transport 2934
26.2.9. Specifying Heterogeneous Reactions 2937
26.2.10. Including Mass Transfer Effects 2940
26.2.10.1. Alternative Modeling of Energy Sources 2943
26.2.10.2. Mass Transfer Mechanisms 2945
26.2.10.2.1. Constant-Rate Option 2945
26.2.10.2.2. User-Defined Option 2945
26.2.10.2.3. Population-Balance Mechanism 2946
26.2.10.2.4. Cavitation Mechanism 2946
26.2.10.2.5. Evaporation-Condensation Mechanism 2950
26.2.10.2.6. Species-Mass-Transfer Mechanism 2953

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.2.10.2.7. Boiling Mechanism 2957
26.2.11. Defining Multiphase Cell Zone and Boundary Conditions 2957
26.2.11.1. Steps for Setting Boundary Conditions 2957
26.2.11.2. Steps for Setting Cell Zone Conditions 2963
26.2.11.3. Boundary and Cell Zone Conditions for the Mixture and the Individual Phases ... 2965
26.2.11.3.1. VOF Model 2965
26.2.11.3.2. Mixture Model 2967
26.2.11.3.3. Eulerian Model 2968
26.2.11.4. Steps for Copying Cell Zone and Boundary Conditions 2974
26.2.12. Setting Initial Conditions 2974
26.2.12.1. Setting Initial Volume Fractions 2974
26.2.12.1.1. Options for Patching Volume Fraction 2975
26.2.12.2. Setting the Initial Turbulence Field 2977
26.3. Setting Up the VOF Model 2977
26.3.1. Solving Steady-State VOF Problems 2978
26.3.2. Guidelines for Using the Multiphase Pseudo Time Method 2978
26.3.3. Including Coupled Level Set with the VOF Model 2978
26.3.4. Mesh Adaption with the VOF Model 2979
26.3.5. Modeling Open Channel Flows 2979
26.3.5.1. Defining Inlet Groups 2981
26.3.5.2. Defining Outlet Groups 2981
26.3.5.3. Setting the Inlet Group 2981
26.3.5.4. Setting the Outlet Group 2982
26.3.5.5. Determining the Free Surface Level 2982
26.3.5.6. Determining the Bottom Level 2983
26.3.5.7. Specifying the Total Height 2984
26.3.5.8. Determining the Velocity Magnitude 2984
26.3.5.9. Determining the Secondary Phase for the Inlet 2984
26.3.5.10. Determining the Secondary Phase for the Outlet 2985
26.3.5.11. Choosing the Pressure Specification Method 2986
26.3.5.12. Choosing the Density Interpolation Method 2986
26.3.5.13. Open Channel Flow Compatibility with Velocity Inlet 2987
26.3.5.13.1. Velocity Inlet, Open Channel Flow, Steady-State 2987
26.3.5.13.2. Velocity Inlet, Open Channel Flow, Transient 2987
26.3.5.14. Limitations 2988
26.3.5.15. Recommendations for Setting Up an Open Channel Flow Problem 2988
26.3.6. Modeling Open Channel Wave Boundary Conditions 2990
26.3.6.1. Summary Report and Regime Check 2998
26.3.6.2. Transient Profile Support for Wave Inputs 3000
26.3.6.3. Alternative Stokes Wave Theory Variant 3000
26.3.7. Recommendations for Open Channel Initialization 3001
26.3.7.1. Reporting Parameters for Open Channel Wave BC Option 3004
26.3.8. Numerical Beach Treatment for Open Channels 3005
26.3.8.1. Solution Strategies 3008
26.3.9. Defining the Phases for the VOF Model 3009
26.3.9.1. Defining the Primary Phase 3009
26.3.9.2. Defining a Secondary Phase 3010
26.3.10. Defining Phase Interaction Terms 3011
26.3.10.1. Including Surface Tension and Adhesion Effects 3011
26.3.10.2. Discretizing Using the Phase Localized Compressive Scheme 3015
26.3.11. Setting Time-Dependent Parameters for the Explicit Volume Fraction Formulation 3017

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.3.12. Modeling Solidification/Melting 3019
26.3.13. Using the VOF-to-DPM Model Transition for Dispersion of Liquid in Gas 3020
26.3.13.1. Limitations on Using the VOF-to-DPM Model Transition 3021
26.3.13.2. Setting up the VOF-to-DPM Model Transition 3021
26.3.13.3. Best Practice Guidelines for Considering Diffuse Lumps 3028
26.3.13.4. Postprocessing for VOF-to-DPM Model Transition Calculations 3029
26.3.14. Using the DPM-to-VOF Model Transition 3029
26.3.14.1. Setting up the DPM-to-VOF Model Transition 3030
26.3.14.2. Limitations 3037
26.4. Setting Up the Mixture Model 3038
26.4.1. Defining the Phases for the Mixture Model 3038
26.4.1.1. Defining the Primary Phase 3038
26.4.1.2. Defining a Non-Granular Secondary Phase 3038
26.4.1.3. Defining a Granular Secondary Phase 3039
26.4.1.4. Defining the Interfacial Area Concentration via the Transport Equation 3042
26.4.1.5. Defining the Algebraic Interfacial Area Concentration 3045
26.4.1.6. Defining Drag Between Phases 3046
26.4.1.7. Defining the Slip Velocity 3047
26.4.1.8. Including Surface Tension and Wall Adhesion Effects 3048
26.4.2. Including Mixture Drift Force 3048
26.4.3. Using the Flow Regime Modeling 3048
26.4.3.1. Limitations for the Flow Regime Modeling 3048
26.4.3.2. Setting the Flow Regime Modeling 3048
26.4.3.3. Solution Strategies 3055
26.4.3.4. Postprocessing for Flow Regime Modeling Simulations 3056
26.4.4. Using the Singhal et al. Expert Cavitation Model 3056
26.4.5. Including Semi-Mechanistic Boiling 3057
26.4.5.1. Overview and Limitations for the Semi-Mechanistic Boiling Model 3057
26.4.5.2. Using the Semi-Mechanistic Boiling Model 3057
26.4.5.3. Cell Zone Specific Boiling 3061
26.4.5.4. Expert Options for the Semi-Mechanistic Boiling Model 3061
26.4.5.5. Solution Strategies for the Semi-Mechanistic Boiling Model 3064
26.5. Setting Up the Eulerian Model 3065
26.5.1. Additional Guidelines for Eulerian Multiphase Simulations 3065
26.5.2. Defining the Phases for the Eulerian Model 3066
26.5.2.1. Defining the Primary Phase 3066
26.5.2.2. Defining a Non-Granular Secondary Phase 3066
26.5.2.3. Defining a Granular Secondary Phase 3067
26.5.2.4. Defining the Interfacial Area Concentration 3072
26.5.2.5. Defining the Interaction Between Phases 3074
26.5.2.5.1. Specifying the Drag Function 3075
26.5.2.5.1.1. Drag Modification 3079
26.5.2.5.2. Specifying the Restitution Coefficients (Granular Flow Only) 3079
26.5.2.5.3. Including the Lift Force 3080
26.5.2.5.4. Including the Lift Correlation 3081
26.5.2.5.5. Including the Wall Lubrication Force 3081
26.5.2.5.6. Including the Turbulent Dispersion Force 3085
26.5.2.5.7. Including Surface Tension and Wall Adhesion Effects 3088
26.5.2.5.8. Including the Virtual Mass Force 3088
26.5.3. Modeling Turbulence 3089
26.5.3.1. Including Turbulence Interaction Source Terms 3091

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.5.3.2. Customizing the k-ε Multiphase Turbulent Viscosity 3093
26.5.4. Including Heat Transfer Effects 3093
26.5.5. Using an Algebraic Interfacial Area Model 3096
26.5.6. Using the Algebraic Interfacial Area Density (AIAD) Model 3097
26.5.6.1. Limitations 3097
26.5.6.2. Procedure for Setting the AIAD Model 3097
26.5.6.3. Solution Strategies 3104
26.5.7. Using the Generalized Two Phase Flow (GENTOP) Model 3104
26.5.7.1. Limitations 3104
26.5.7.2. Steps for Using the GENTOP Model 3105
26.5.7.3. Solution Strategies 3107
26.5.8. Including the Dense Discrete Phase Model 3107
26.5.8.1. Defining a Granular Discrete Phase 3111
26.5.9. Including the Boiling Model 3113
26.5.9.1. Limitations of the Boiling Model 3113
26.5.9.2. Procedure for Setting the Boiling Model 3113
26.5.9.3. Postprocessing for the Boiling Model 3127
26.5.10. Setting Up Polydisperse Boiling 3128
26.5.11. Including the Multi-Fluid VOF Model 3129
26.6. Population Balance Model 3131
26.6.1. Population Balance Model Setup 3131
26.6.1.1. Enabling the Population Balance Model 3131
26.6.1.1.1. Generated DQMOM Values 3145
26.6.1.2. Defining Population Balance Boundary Conditions 3149
26.6.1.2.1. Initializing Bin Fractions With a Log-Normal Distribution 3150
26.6.1.3. Defining Population Balance Cell Zones Conditions 3151
26.6.1.4. Specifying Population Balance Solution Controls 3151
26.6.1.5. Coupling With Fluid Dynamics 3152
26.6.1.6. Specifying Interphase Mass Transfer Due to Nucleation and Growth 3153
26.6.1.7. Size Calculator 3157
26.6.2. Solution Strategies 3159
26.6.3. Postprocessing for the Population Balance Model 3160
26.6.3.1. Population Balance Solution Variables 3160
26.6.3.2. Reporting Derived Population Balance Variables 3160
26.6.3.2.1. Computing Moments 3161
26.6.3.2.2. Displaying a Number Density Function 3161
26.6.4. UDFs for Population Balance Modeling 3163
26.6.4.1. Population Balance Variables 3163
26.6.4.2. Population Balance DEFINE Macros 3164
26.6.4.2.1. DEFINE_PB_BREAK_UP_RATE_FREQ 3164
26.6.4.2.1.1. Usage 3164
26.6.4.2.1.2. Example 3165
26.6.4.2.2. DEFINE_PB_BREAK_UP_RATE_PDF 3165
26.6.4.2.2.1. Usage 3165
26.6.4.2.2.2. Example 3166
26.6.4.2.3. DEFINE_PB_COALESCENCE_RATE 3166
26.6.4.2.3.1. Usage 3166
26.6.4.2.3.2. Example 3167
26.6.4.2.4. DEFINE_PB_NUCLEATION_RATE 3167
26.6.4.2.4.1. Usage 3167
26.6.4.2.4.2. Example 3168

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.6.4.2.5. DEFINE_PB_GROWTH_RATE 3169
26.6.4.2.5.1. Usage 3169
26.6.4.2.5.2. Example 3169
26.6.4.3. Hooking a Population Balance UDF to Ansys Fluent 3170
26.6.5. DEFINE_HET_RXN_RATE Macro 3170
26.6.5.1. Description 3171
26.6.5.2. Usage 3171
26.6.5.3. Example 3172
26.6.5.4. Hooking a Heterogeneous Reaction Rate UDF to Ansys Fluent 3172
26.7. Setting Up the Wet Steam Model 3173
26.7.1. Using User-Defined Thermodynamic Wet Steam Properties 3175
26.7.2. Writing the User-Defined Wet Steam Property Functions (UDWSPF) 3175
26.7.3. Compiling Your UDWSPF and Building a Shared Library File 3178
26.7.4. Loading the UDWSPF Shared Library File 3179
26.7.5. UDWSPF Example 3179
26.8. Solution Strategies for Multiphase Modeling 3184
26.8.1. General Solution Strategies 3184
26.8.1.1. Coupled Solution for Eulerian Multiphase Flows 3184
26.8.1.2. Coupled Solution for VOF and Mixture Multiphase Flows 3186
26.8.1.3. Selecting the Pressure-Velocity Coupling Method 3187
26.8.1.3.1. Limitations and Recommendations of the Coupled with Volume Fraction Options for the VOF and Mixture Models 3189
26.8.1.3.2. Solving N-Phase Volume Fraction Equations 3190
26.8.1.4. Controlling the Volume Fraction Coupled Solution 3190
26.8.1.5. Default and Stability Controls 3194
26.8.1.5.1. Default Controls 3194
26.8.1.5.2. VOF Solution Stability Controls 3195
26.8.1.5.3. Text User Interface for VOF Stability Controls 3197
26.8.1.6. Heat Transfer and Radiative Flux Distribution for Non-Eulerian Multiphase Models 3201
26.8.1.7. Steady-State Solution Strategies 3201
26.8.2. Model-Specific Solution Strategies 3201
26.8.2.1. VOF Model 3202
26.8.2.1.1. Setting the Reference Pressure Location 3202
26.8.2.1.2. Pressure Interpolation Scheme 3202
26.8.2.1.3. Discretization Scheme Selection 3203
26.8.2.1.4. High-Order Rhie-Chow Face Flux Interpolation 3203
26.8.2.1.5. Treatment of Unsteady Terms in Rhie-Chow Face Flux Interpolation 3204
26.8.2.1.6. Pressure-Velocity Coupling and Under-Relaxation for the Time-dependent Formulations 3204
26.8.2.1.7. Under-Relaxation for the Steady-State Formulation 3205
26.8.2.2. Mixture Model 3205
26.8.2.2.1. Setting the Under-Relaxation Factor for the Slip Velocity 3205
26.8.2.2.2. Calculating an Initial Solution 3205
26.8.2.3. Eulerian Model 3205
26.8.2.3.1. Calculating an Initial Solution 3205
26.8.2.3.2. Temporarily Ignoring Lift and Virtual Mass Forces 3206
26.8.2.3.3. Using W-Cycle Multigrid 3206
26.8.2.3.4. Including the Anisotropic Drag Law 3206
26.8.2.3.5. Using Reference Density 3207
26.8.2.3.6. Controlling NITA Solution Options via the Text Interface 3207

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.8.2.4. Wet Steam Model 3208
26.8.2.4.1. Boundary Conditions, Initialization, and Patching 3208
26.8.2.4.2. Solution Limits for the Wet Steam Model 3208
26.8.2.4.3. Solution Strategies for the Wet Steam Model 3209
26.9. Multiphase Case Check 3210
26.10. Postprocessing for Multiphase Modeling 3211
26.10.1. Model-Specific Variables 3211
26.10.1.1. VOF Model 3211
26.10.1.2. Mixture Model 3212
26.10.1.3. Eulerian Model 3212
26.10.1.4. Multiphase Species Transport 3213
26.10.1.5. Wet Steam Model 3214
26.10.1.6. Dense Discrete Phase Model 3215
26.10.2. Displaying Velocity Vectors 3216
26.10.3. Reporting Fluxes 3216
26.10.4. Reporting Forces on Walls 3217
26.10.5. Reporting Flow Rates 3217
27. Modeling Solidification and Melting 3219
27.1. Setup Procedure 3219
27.2. Procedures for Modeling Continuous Casting 3222
27.3. Modeling Thermal and Solutal Buoyancy 3223
27.4. Solution Procedure 3224
27.5. Postprocessing 3225
28. Modeling Fluid-Structure Interaction (FSI) Within Fluent 3227
28.1. Overview and Limitations 3227
28.2. Setting Up an Intrinsic Fluid-Structure Interaction (FSI) Simulation 3229
28.2.1. Using Intrinsic FSI With Non-Conformal Interfaces 3236
29. Modeling Eulerian Wall Films 3239
29.1. Limitations 3239
29.2. Overview of Using the Eulerian Wall Film Model 3240
29.3. Setting Eulerian Wall Film Model Options 3241
29.4. Setting Eulerian Wall Film Solution Controls 3248
29.5. Setting Eulerian Wall Film Boundary, Initial, and Source Term Conditions 3252
29.5.1. Specifying the Boundary Type 3253
29.5.2. Setting the Source Terms 3254
29.5.3. Setting the Phase Change 3255
29.5.4. Setting the Surface Contact 3257
29.5.5. Setting the DPM interaction 3258
29.5.6. Setting the VOF interaction 3259
29.6. Coupling of Eulerian Wall Film with the VOF Multiphase Model 3261
29.7. Postprocessing the Eulerian Wall Film 3261
30. Modeling Electric Potential Field and Electrochemistry Models 3265
30.1. Simulating the Electric Potential Field 3265
30.1.1. Limitation of the Electric Potential Model 3265
30.1.2. Setting Up the Electric Potential Model 3266
30.2. Simulating the Lithium-ion Battery 3267
30.2.1. Limitations of the Detailed Lithium-ion Battery Model 3268
30.2.2. Setting Up the Lithium-ion Battery Model 3268
30.3. Setting the Electrolysis and H2 Pump Model 3275
30.3.1. Geometry Definition for the Electrolysis and H2 Pump Model 3275
30.3.2. Workflow for Using the Electrolysis and H2 Pump Model 3275

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

30.3.3. Setting up the Electrolysis and H2 Pump Model 3276
30.3.3.1. Specifying Model Options (Model Tab) 3278
30.3.3.2. Specifying Model Parameters (Parameters Tab) 3281
30.3.3.3. Specifying Anode Properties (Anode Tab) 3284
30.3.3.3.1. Specifying Current Collector Properties for the Anode 3285
30.3.3.3.2. Specifying Flow Channel Properties for the Anode 3286
30.3.3.3.3. Specifying Porous Layer Properties for the Anode 3287
30.3.3.3.4. Specifying Catalyst Layer Properties for the Anode 3289
30.3.3.4. Specifying Electrolyte/Membrane Properties (**Electrolyte** Tab) 3290
30.3.3.4.1. Resolved MEA Layer Method 3290
30.3.3.4.2. Unresolved MEA Layer Method 3292
30.3.3.5. Specifying Cathode Properties (Cathode Tab) 3294
30.3.3.6. Setting the External Electrical Tabs (**Electrical Tabs** Tab) 3294
30.3.3.7. (Customization Tab) 3295
30.3.3.8. Setting Advanced Options (Advanced Tab) 3295
30.3.4. Solution Strategies for the Electrolysis and H2 Pump Model 3296
30.4. Postprocessing Electric Potential Field and Li-ion Battery Quantities 3297
31. Modeling Batteries 3299
31.1. Introduction 3299
31.1.1. Overview 3299
31.1.2. General Procedure 3300
31.2. Using the MSMD-Based Battery Models 3300
31.2.1. Limitations 3301
31.2.2. Geometry Definition 3301
31.2.3. Setting up the Battery Model 3301
31.2.3.1. Specifying Battery Model Options 3303
31.2.3.1.1. Setting the Life Model 3308
31.2.3.1.2. Specifying a Profile or Data Table 3311
31.2.3.1.3. Using Tables in the Battery Model 3315
31.2.3.2. Specifying Conductive Zones 3317
31.2.3.3. Specifying Electric Contacts 3318
31.2.3.4. Specifying Battery Model Parameters 3321
31.2.3.4.1. Inputs for the CHT Coupling Method 3322
31.2.3.4.2. Inputs for the FMU-CHT Coupling Method 3323
31.2.3.4.3. Inputs for the NTGK/DCIR Model 3324
31.2.3.4.4. Inputs for the Equivalent Circuit Model 3334
31.2.3.4.4.1. The HPPC Library 3340
31.2.3.4.5. Inputs for the Newman's P2D Model 3342
31.2.3.4.6. Input for the User-Defined E-Model 3349
31.2.3.5. Hooking User-Defined Functions 3350
31.2.3.6. Specifying Advanced Options 3351
31.2.3.6.1. Running the Standalone Echem Model 3352
31.2.3.6.2. Simulating the Aging Process of a Battery (Standalone Mode) 3356
31.2.3.6.3. Using the Battery Pack Builder Tool 3361
31.2.3.6.4. Using the Battery ROM Tool Kit 3365
31.2.3.6.4.1. Create ROM Training Data (ROM Data Creator Tab) 3368
31.2.3.6.4.2. Computing Coefficient Matrices of the State Space Equations (**LTI-ROM Generation** Tab) 3372
31.2.3.6.4.3. Postprocessing of the SVD-ROM Data (SVD-ROM Post-Processing Tab) 3373
31.2.3.6.5. Defining Orthotropic Thermal Conductivity 3374

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.2.3.6.6. Using the Empirical-Based Battery Swelling Model 3376
31.2.3.6.7. Battery Venting Model 3377
31.2.3.6.8. Including the Entropic Heat Effects 3380
31.2.3.6.9. Using the Thermal Abuse Model 3382
31.2.3.6.9.1. One-Equation Kinetics Model 3383
31.2.3.6.9.2. Four-Equation Kinetics Model 3385
31.2.3.6.9.3. Standalone Thermal Abuse Model 3386
31.2.3.6.9.4. Running the Thermal Abuse Model Only 3388
31.2.3.7. Specifying External and Internal Short-Circuit Resistances 3388
31.2.3.8. Setting a Battery Swelling Case 3389
31.2.4. Using Parameter Estimation Tools 3390
31.2.4.1. Using Parameter Estimation Tools in the GUI 3390
31.2.4.2. Using Parameter Estimation Tools in the TUI 3390
31.2.4.2.1. Using the Parameter Estimation Tool for the NTGK Model in the TUI 3391
31.2.4.2.2. Using the Parameter Estimation Tool for the ECM Model in the TUI 3392
31.2.4.2.3. Using the Parameter Estimation Tool for the Thermal Abuse Model in the TUI 3394
31.2.5. Initializing the Battery Model 3395
31.2.6. Modifying Material Properties 3395
31.2.7. Solution Controls for the MSMD Battery Model 3396
31.2.8. Predefined Report Definitions for the Battery Model 3396
31.2.9. Postprocessing the MSMD Battery Model 3396
32. Modeling Fuel Cells 3401
32.1. Using the PEMFC Model 3401
32.1.1. Overview and Limitations 3402
32.1.2. Geometry Definition for the PEMFC Model 3402
32.1.3. Installing the PEMFC Model 3402
32.1.4. Loading the PEMFC Module 3403
32.1.5. Workflow for Using the PEMFC Module 3403
32.1.6. Setting Up the PEMFC Module 3404
32.1.6.1. Specifying Model Options (**Model** Tab) 3406
32.1.6.2. Specifying Model Parameters (**Parameters** Tab) 3411
32.1.6.3. Specifying Anode Properties (**Anode** Tab) 3415
32.1.6.3.1. Specifying Current Collector Properties for the Anode 3416
32.1.6.3.2. Specifying Flow Channel Properties for the Anode 3417
32.1.6.3.3. Specifying Porous Electrode Properties for the Anode 3418
32.1.6.3.4. Specifying Catalyst Layer Properties for the Anode 3420
32.1.6.3.5. Specifying Micro Porous Layer (Optional) Properties for the Anode 3423
32.1.6.3.6. Specifying Cell Zone Conditions for the Anode 3424
32.1.6.4. Specifying Electrolyte/Membrane Properties (**Electrolyte** Tab) 3424
32.1.6.4.1. Specifying Cell Zone Conditions for the Membrane 3426
32.1.6.5. Specifying Cathode Properties (**Cathode** Tab) 3426
32.1.6.5.1. Specifying Current Collector Properties for the Cathode 3426
32.1.6.5.2. Specifying Flow Channel Properties for the Cathode 3426
32.1.6.5.3. Specifying Porous Electrode Properties for the Cathode 3426
32.1.6.5.4. Specifying Catalyst Layer Properties for the Cathode 3427
32.1.6.5.5. Specifying Micro Porous Layer (Optional) Properties for the Cathode 3429
32.1.6.5.6. Specifying Cell Zone Conditions for the Cathode 3429
32.1.6.6. Setting the External Electrical Tabs (**Electrical Tabs** Tab) 3430
32.1.6.7. Setting Advanced Properties (**Advanced** Tab) 3430
32.1.6.7.1. Setting Contact Resistivities for the PEMFC Model 3431

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.1.6.7.2. Setting Coolant Channel Properties for the PEMFC Model (Optional) 3432
32.1.6.7.3. Managing Stacks for the PEMFC Model 3433
32.1.6.8. Customizing the PEM Fuel Cell Module 3434
32.1.6.9. Reporting on the Solution (**Reports** Tab) 3436
32.1.6.10. User-Defined Functions Hooked to the PEMFC Module 3438
32.1.7. PEMFC Model Boundary Conditions 3440
32.1.8. Solution Guidelines for the PEMFC Model 3441
32.1.9. Postprocessing the PEMFC Model 3441
32.1.10. User-Accessible Functions 3443
32.2. Using the Fuel Cell and Electrolysis Model 3447
32.2.1. Overview and Limitations 3448
32.2.2. Geometry Definition for the Fuel Cell and Electrolysis Model 3448
32.2.3. Installing the Fuel Cell and Electrolysis Model 3448
32.2.4. Loading the Fuel Cell and Electrolysis Module 3449
32.2.5. Workflow for Using the Fuel Cell and Electrolysis Module 3449
32.2.6. Setting Up the Fuel Cell and Electrolysis Module 3450
32.2.6.1. Specifying Model Options (**Model** Tab) 3452
32.2.6.2. Specifying Model Parameters (**Parameters** Tab) 3454
32.2.6.3. Specifying Anode Properties (**Anode** Tab) 3456
32.2.6.3.1. Specifying Current Collector Properties for the Anode 3457
32.2.6.3.2. Specifying Flow Channel Properties for the Anode 3458
32.2.6.3.3. Specifying Porous Electrode Properties for the Anode 3459
32.2.6.3.4. Specifying Catalyst Layer Properties for the Anode 3460
32.2.6.3.5. Specifying Cell Zone Conditions for the Anode 3461
32.2.6.4. Specifying Electrolyte/Membrane Properties (**Electrolyte** Tab) 3461
32.2.6.4.1. Specifying Cell Zone Conditions for the Membrane 3462
32.2.6.5. Specifying Cathode Properties (**Cathode** Tab) 3462
32.2.6.5.1. Specifying Current Collector Properties for the Cathode 3463
32.2.6.5.2. Specifying Flow Channel Properties for the Cathode 3464
32.2.6.5.3. Specifying Porous Electrode Properties for the Cathode 3465
32.2.6.5.4. Specifying Catalyst Layer Properties for the Cathode 3466
32.2.6.5.5. Specifying Cell Zone Conditions for the Cathode 3467
32.2.6.6. Setting Advanced Properties (**Advanced** Tab) 3467
32.2.6.6.1. Setting Contact Resistivities for the Fuel Cell and Electrolysis Model 3467
32.2.6.6.2. Setting Coolant Channel Properties for the Fuel Cell and Electrolysis Model 3468
32.2.6.6.3. Managing Stacks for the Fuel Cell and Electrolysis Model 3469
32.2.6.7. Customizing the Fuel Cell and Electrolysis Module 3471
32.2.6.8. Reporting on the Solution (**Reports** Tab) 3471
32.2.7. Modeling Current Collectors 3472
32.2.8. Fuel Cell and Electrolysis Model Boundary Conditions 3474
32.2.9. Solution Guidelines for the Fuel Cell and Electrolysis Model 3475
32.2.10. Postprocessing the Fuel Cell and Electrolysis Model 3475
32.2.11. User-Accessible Functions 3477
32.3. Using the Solid Oxide Fuel Cell With Unresolved Electrolyte Model 3480
32.3.1. Limitation on Modeling Solid Oxide Fuel Cells 3480
32.3.2. Installing the Solid Oxide Fuel Cell With Unresolved Electrolyte Model 3480
32.3.3. Loading the Solid Oxide Fuel Cell With Unresolved Electrolyte Module 3480
32.3.4. Solid Oxide Fuel Cell With Unresolved Electrolyte Module Set Up Procedure 3482
32.3.5. Setting the SOFC Model 3485
32.3.5.1. Setting the Parameters for the SOFC With Unresolved Electrolyte Model 3486

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.3.5.2. Setting Up the Electrochemistry Parameters 3488
32.3.5.3. Setting Up the Electrode-Electrolyte Interfaces 3490
32.3.5.4. Setting Up the Electric Field Model Parameters 3492
32.3.5.5. Customizing the SOFC Module 3494
32.3.6. User-Accessible Functions for the Solid Oxide Fuel Cell With Unresolved Electrolyte Model 3494
33. Modeling Magnetohydrodynamics 3497
33.1. Introduction 3497
33.2. Implementation 3497
33.2.1. Solving Magnetic Induction and Electric Potential Equations 3498
33.2.2. Calculation of MHD Variables 3498
33.2.3. MHD Interaction with Fluid Flows 3499
33.2.4. MHD Interaction with Discrete Phase Model 3499
33.2.5. General User-Defined Functions 3499
33.3. Using the Ansys Fluent MHD Module 3499
33.3.1. MHD Module Installation 3499
33.3.2. Loading the MHD Module 3500
33.3.3. MHD Model Setup 3501
33.3.3.1. Enabling the MHD Model 3502
33.3.3.2. Selecting an MHD Method 3503
33.3.3.3. Applying an External Magnetic Field 3504
33.3.3.4. Setting Up Boundary Conditions 3507
33.3.3.5. Solution Controls 3511
33.3.4. MHD Solution and Postprocessing 3512
33.3.4.1. MHD Model Initialization 3513
33.3.4.2. Iteration 3513
33.3.4.3. Postprocessing 3513
33.3.5. Limitations 3514
33.4. Guidelines For Using the Ansys Fluent MHD Model 3515
33.4.1. Installing the MHD Module 3515
33.4.2. An Overview of Using the MHD Module 3515
33.5. Definitions of the Magnetic Field 3518
33.6. External Magnetic Field Data Format 3518
34. Modeling Continuous Fibers 3521
34.1. Installing the Continuous Fiber Module 3522
34.2. Loading the Continuous Fiber Module 3522
34.3. Getting Started With the Continuous Fiber Module 3524
34.3.1. User-Defined Memory and the Adjust Function Setup 3524
34.3.2. Source Term UDF Setup 3524
34.4. Fiber Models and Options 3525
34.4.1. Choosing a Fiber Model 3526
34.4.2. Including Interaction With Surrounding Flow 3527
34.4.3. Including Lateral Drag on Surrounding Flow 3527
34.4.4. Including Fiber Radiation Interaction 3527
34.4.5. Viscous Heating of Fibers 3527
34.4.6. Drag, Heat and Mass Transfer Correlations 3527
34.5. Fiber Material Properties 3528
34.5.1. The Concept of Fiber Materials 3528
34.5.2. Description of Fiber Properties 3528
34.6. Defining Fibers 3531
34.6.1. Overview 3531

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

34.6.2. Fiber Injection Types 3532
34.6.3. Working with Fiber Injections 3533
34.6.3.1. Creating Fiber Injections 3533
34.6.3.2. Modifying Fiber Injections 3533
34.6.3.3. Copying Fiber Injections 3533
34.6.3.4. Deleting Fiber Injections 3534
34.6.3.5. Initializing Fiber Injections 3534
34.6.3.6. Computing Fiber Injections 3534
34.6.3.7. Print Fiber Injections 3534
34.6.3.8. Read Data of Fiber Injections 3535
34.6.3.9. Write Data of Fiber Injections 3535
34.6.3.10. Write Binary Data of Fiber Injections 3535
34.6.3.11. List Fiber Injections 3535
34.6.4. Defining Fiber Injection Properties 3536
34.6.5. Point Properties Specific to Single Fiber Injections 3540
34.6.6. Point Properties Specific to Line Fiber Injections 3540
34.6.7. Point Properties Specific to Matrix Fiber Injections 3540
34.6.8. Define Fiber Grids 3541
34.6.8.1. Equidistant Fiber Grids 3541
34.6.8.2. One-Sided Fiber Grids 3542
34.6.8.3. Two-Sided Fiber Grids 3542
34.6.8.4. Three-Sided Fiber Grids 3542
34.7. User-Defined Functions (UDFs) for the Continuous Fiber Model 3543
34.7.1. UDF Setup 3544
34.7.1.1. Linux Systems 3544
34.7.1.2. Windows Systems 3544
34.7.2. Customizing the fiber_fluent_interface.c File for Your Fiber Model Application 3544
34.7.2.1. Example: Heat Transfer Coefficient UDF 3545
34.7.2.2. Example: Fiber Specific Heat Capacity UDF 3546
34.7.3. Compile Fiber Model UDFs 3547
34.7.3.1. Linux Systems 3547
34.7.3.2. NT/Windows Systems 3548
34.7.4. Hook UDFs to the Continuous Fiber Model 3549
34.8. Fiber Model Solution Controls 3550
34.9. Postprocessing for the Continuous Fibers 3552
34.9.1. Display of Fiber Locations and Grid Points 3553
34.9.2. Exchange Terms of Fibers 3555
34.9.3. Analyzing Fiber Variables 3556
34.9.3.1. XY Plots 3556
34.9.3.2. Fiber Display 3557
34.9.4. Running the Fiber Module in Parallel 3558
35. Creating Reduced Order Models (ROMs) 3561
35.1. Defining a ROM 3561
35.2. Reduced Order Model (ROM) Evaluation in Fluent 3563
35.3. Exporting Reduced Order Model (ROM) Results from Fluent 3568
35.4. ROM Limitations 3569
36. Using the Solver 3571
36.1. Overview of Using the Solver 3571
36.1.1. Choosing the Solver 3573
36.2. Choosing the Spatial Discretization Scheme 3575
36.2.1. First-Order Accuracy vs. Second-Order Accuracy 3576

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.2.1.1. First- to Higher-Order Blending 3576
36.2.2. Other Discretization Schemes 3577
36.2.3. Choosing the Pressure Interpolation Scheme 3577
36.2.4. Choosing the Density Interpolation Scheme 3578
36.2.5. High Order Term Relaxation (HOTR) 3578
36.2.5.1. Limitations 3582
36.2.6. User Inputs 3582
36.3. Pressure-Based Solver Settings 3584
36.3.1. Choosing the Pressure-Velocity Coupling Method 3585
36.3.1.1. SIMPLE vs. SIMPLEC 3586
36.3.1.2. PISO 3586
36.3.1.3. Fractional Step Method 3587
36.3.1.4. Coupled 3587
36.3.1.5. User Inputs 3587
36.3.2. Mass Flux Types 3588
36.3.3. Setting Under-Relaxation Factors 3589
36.3.3.1. User Inputs 3590
36.3.4. Setting Solution Controls for the Non-Iterative Solver 3592
36.3.4.1. User Inputs 3593
36.3.4.2. Hybrid NITA for the VOF and Mixture Multiphase Models 3595
36.3.4.3. NITA Expert Options 3598
36.3.4.4. Compatibility of the NITA Scheme with Other Ansys Fluent Models 3599
36.3.5. Equation Order 3600
36.3.6. Using the Correction Form of Momentum Discretization 3600
36.4. Density-Based Solver Settings 3601
36.4.1. Changing the Courant Number 3602
36.4.1.1. Courant Numbers for the Density-Based Explicit Formulation 3602
36.4.1.2. Courant Numbers for the Density-Based Implicit Formulation 3602
36.4.1.3. Courant Number Monitor 3603
36.4.1.4. User Inputs 3603
36.4.2. Convective Flux Types 3604
36.4.3. Convergence Acceleration for Stretched Meshes (CASM) 3605
36.4.3.1. Enhanced Convergence Acceleration for Stretched Meshes 3608
36.4.4. Enabling High-Speed Numerics 3608
36.4.5. Preventing Divergence Using Local Under-Relaxation 3608
36.4.6. Specifying the Explicit Relaxation 3609
36.4.7. Turning On FAS Multigrid 3609
36.4.7.1. Setting Coarse Grid Levels 3609
36.4.7.2. Using Residual Smoothing to Increase the Courant Number 3610
36.5. Setting Algebraic Multigrid Parameters 3610
36.5.1. Specifying the Multigrid Cycle Type 3613
36.5.2. Setting the Termination and Residual Reduction Parameters 3613
36.5.3. Setting the Stabilization Method 3613
36.5.4. Additional Algebraic Multigrid Parameters 3614
36.5.4.1. Fixed Cycle Parameters 3615
36.5.4.2. Coarsening Parameters 3615
36.5.4.3. Smoother Types 3616
36.5.4.4. Flexible Cycle Parameters 3617
36.5.4.5. Setting the Verbosity 3617
36.5.4.6. Returning to the Default Multigrid Parameters 3618
36.5.5. Setting FAS Multigrid Parameters 3618

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.5.5.1. Combating Convergence Trouble 3618
36.5.5.2. "Industrial-Strength" FAS Multigrid 3618
36.6. Setting Solution Limits 3621
36.6.1. Limiting the Values of Solution Variables 3623
36.6.2. Adjusting the Positivity Rate Limit 3623
36.6.3. Resetting Solution Limits 3624
36.7. Setting Multi-Stage Time-Stepping Parameters 3624
36.7.1. Changing the Multi-Stage Scheme 3624
36.7.1.1. Changing the Coefficients and Number of Stages 3625
36.7.1.2. Controlling Updates to Dissipation and Viscous Stresses 3625
36.7.1.3. Resetting the Multi-Stage Parameters 3626
36.8. Selecting Gradient Limiters 3626
36.9. Initializing the Solution 3627
36.9.1. Initializing the Entire Flow Field Using Standard Initialization 3628
36.9.1.1. Saving and Resetting Initial Values 3630
36.9.2. Patching Values in Selected Cells 3630
36.9.2.1. Using Registers 3632
36.9.2.2. Using Field Functions 3632
36.9.2.3. Using Patching Later in the Solution Process 3632
36.10. Full Multigrid (FMG) Initialization 3633
36.10.1. Steps in Using FMG Initialization 3633
36.10.2. Convergence Strategies for FMG Initialization 3634
36.10.3. Additional FMG Initialization Options with the Density-Based Solver 3634
36.11. Hybrid Initialization 3635
36.11.1. Steps in Using Hybrid Initialization 3636
36.11.2. Solution Strategies for Hybrid Initialization 3638
36.12. Performing Steady-State Calculations 3639
36.12.1. Updating UDF Profiles and Named Expressions 3640
36.12.2. Resetting Data 3640
36.12.3. Data Sampling for Steady Statistics 3641
36.13. Performing Time-Dependent Calculations 3643
36.13.1. Inputs for Time-Dependent Problems 3645
36.13.1.1. Additional Inputs 3660
36.13.2. CFL-Based Time Stepping 3660
36.13.2.1. The CFL-Based Time Stepping Algorithm 3660
36.13.2.2. Specifying Parameters for CFL-Based Time Stepping 3661
36.13.3. Error-Based Time Stepping 3663
36.13.3.1. The Error-Based Time Stepping Algorithm 3664
36.13.3.2. Specifying Parameters for Error-Based Time Stepping 3664
36.13.4. Multiphase-Specific Time Stepping 3665
36.13.4.1. The Multiphase-Specific Time Stepping Algorithm 3665
36.13.4.2. Specifying Parameters for Multiphase-Specific Time Stepping 3668
36.13.5. Postprocessing for Time-Dependent Problems 3669
36.13.6. Runtime Discrete Fourier Transformation 3671
36.13.6.1. Runtime DFT Formulation 3672
36.13.6.2. Input Parameters for the Runtime DFT 3673
36.13.6.3. DFT Field Variables for Postprocessing 3673
36.14. Performing Calculations with a Pseudo Time Method 3673
36.14.1. Local Time Step Method Setting 3675
36.14.2. Global Time Step Method Settings 3676
36.14.2.1. Pseudo Time Explicit Relaxation Factors 3677

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.14.2.2. Advanced Solution Controls for the Pseudo Time Method 3679
36.14.2.3. Pseudo Time Settings for the Calculation 3680
36.15. Monitoring Solution Convergence 3684
36.15.1. Monitoring Residuals 3684
36.15.1.1. Definition of Residuals for the Pressure-Based Solver 3684
36.15.1.2. Definition of Residuals for the Density-Based Solver 3686
36.15.1.3. Overview of Using the Residual Monitors Dialog Box 3687
36.15.1.4. Printing and Plotting Residuals 3689
36.15.1.5. Storing Residual History Points 3690
36.15.1.6. Controlling Normalization and Scaling 3690
36.15.1.7. Choosing a Convergence Criterion 3691
36.15.1.8. Modifying Convergence Criteria 3693
36.15.1.9. Disabling Monitoring 3694
36.15.1.10. Plot Parameters 3694
36.15.1.11. Postprocessing Residual Values 3694
36.15.2. Monitoring Statistics 3695
36.15.3. Monitoring Solution Quantities 3696
36.16. Convergence Conditions 3696
36.16.1. Setting Up the Convergence Conditions Dialog Box 3698
36.17. Executing Commands During the Calculation 3700
36.17.1. Defining Macros 3703
36.17.2. Saving Files During the Calculation 3705
36.18. Automatic Initialization of the Solution and Case Modification 3705
36.19. Animating the Solution 3714
36.19.1. Creating an Animation Definition 3714
36.19.1.1. Guidelines for Creating an Animation Definition 3717
36.19.2. Playing an Animation Sequence 3717
36.19.2.1. Modifying the View 3719
36.19.2.2. Modifying the Playback Speed 3719
36.19.2.3. Playing Back an Excerpt 3719
36.19.2.4. "Fast-Forwarding" the Animation 3719
36.19.2.5. Continuous Animation 3720
36.19.2.6. Stopping the Animation 3720
36.19.2.7. Advancing the Animation Frame by Frame 3720
36.19.2.8. Deleting an Animation Sequence 3720
36.19.3. Saving an Animation Sequence 3720
36.19.3.1. Solution Animation File 3721
36.19.3.2. Picture File 3721
36.19.3.3. Video File 3722
36.19.4. Reading an Animation Sequence 3725
36.20. Checking Your Case Setup 3725
36.20.1. Automatic Implementation 3726
36.20.2. Manual Implementation 3726
36.20.2.1. Checking the Mesh 3727
36.20.2.2. Checking Model Selections 3729
36.20.2.3. Checking Boundary and Cell Zone Conditions 3731
36.20.2.4. Checking Material Properties 3734
36.20.2.5. Checking the Solver Settings 3735
36.21. Convergence and Stability 3738
36.21.1. Judging Convergence 3738
36.21.2. Step-by-Step Solution Processes 3739

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.21.2.1. Selecting a Subset of the Solution Equations 3740
36.21.2.2. Turning Reactions On and Off 3740
36.21.3. Modifying Algebraic Multigrid Parameters 3741
36.21.4. Modifying the Multi-Stage Parameters 3741
36.21.5. Robustness with Meshes of Poor Quality 3741
36.21.6. Warped-Face Gradient Correction 3745
36.21.7. Numerical Noise Filter for the Energy Equation 3746
36.22. Solution Steering 3746
36.22.1. Overview of Solution Steering 3747
36.22.2. Solution Steering Strategy 3747
36.22.2.1. Initialization 3747
36.22.3. Using Solution Steering 3748
37. Using the Fluent Native GPU Solver 3755
37.1. Introduction to the Fluent GPU Solver 3755
37.2. Supported GPUs and Drivers 3755
37.3. Basic Steps for CFD Analysis Using the Fluent GPU Solver 3756
37.4. Starting the Fluent GPU Solver 3756
37.4.1. Starting the Fluent GPU Solver Using the Fluent Launcher 3757
37.4.2. Starting the Fluent GPU Solver from the Command Line 3758
37.5. Exiting the Fluent GPU Solver 3759
37.6. Graphical User Interface (GUI) 3759
37.7. Using CPU Processes for Setup and Postprocessing 3759
37.8. Reading Fluent Case Files Into the Fluent GPU Solver 3760
37.9. Features Supported by the Fluent GPU Solver 3760
37.9.1. Models 3761
37.9.2. Material Properties 3762
37.9.3. Solver Settings 3764
37.9.4. Parametric Studies 3764
37.9.5. Cell Zone and Boundary Conditions 3764
37.9.6. Mesh Interfaces 3765
37.9.7. Solution Monitors and Report Definitions 3766
37.10. Fluent GPU Solver Limitations 3767
37.11. Transitioning from a Steady-State Solution to a Transient Calculation 3769
37.12. Troubleshooting Cases 3769
37.13. Resolving GPU Solver Performance Issues 3770
37.14. GPU Memory Usage 3771
38. Adapting the Mesh 3773
38.1. Using Adaption 3773
38.1.1. Adaption Example 3774
38.1.2. Adaption Guidelines 3775
38.2. Refining and Coarsening 3777
38.2.1. Predefined Criteria for Adaption 3784
38.2.1.1. Boundary Layer Adaption Based on Cell Distance 3785
38.2.1.2. Aerodynamics Adaption 3787
38.2.1.3. Combustion Adaption 3791
38.2.1.4. VOF Adaption 3795
38.2.1.5. Overset Adaption 3799
38.3. Adaption Examples 3801
38.3.1. Boundary Cell Register 3801
38.3.2. Region Cell Register 3803
38.3.3. Field Variable Cell Registers (Gradients, Scaling, and So On) 3806

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

38.3.4. Expression Adaption Refinement 3808
38.4. Geometry-Based Adaption with the Hanging Node Method 3810
38.4.1. Performing Geometry-Based Adaption with the Hanging Node Method 3811
39. Creating Surfaces and Cell Registers for Displaying and Reporting Data 3813
39.1. Using Surfaces 3814
39.1.1. Zone Surfaces 3815
39.1.2. Partition Surfaces 3817
39.1.3. Imprint Surfaces 3819
39.1.4. Point Surfaces 3821
39.1.4.1. Using the Point Tool 3823
39.1.5. Structural Point Surfaces 3825
39.1.6. Line and Rake Surfaces 3826
39.1.6.1. Using the Line Tool 3828
39.1.6.1.1. Initializing the Line Tool 3828
39.1.6.1.2. Translating the Line Tool 3829
39.1.6.1.3. Rotating the Line Tool 3829
39.1.6.1.4. Resizing the Line Tool 3830
39.1.6.1.5. Resetting the Line Tool 3830
39.1.7. Plane Surfaces 3830
39.1.7.1. Using the Plane Tool 3833
39.1.8. Quadric Surfaces 3836
39.1.9. Iso-surfaces 3838
39.1.10. Clipping Surfaces 3840
39.1.11. Transforming Surfaces 3843
39.1.12. Expression Volumes 3845
39.1.13. Grouping, Prioritizing, Editing, Renaming, and Deleting Surfaces 3847
39.1.13.1. Grouping Surfaces 3849
39.1.13.2. Editing and Renaming Surfaces 3849
39.1.13.3. Setting Surface Rendering Priority 3850
39.1.13.4. Deleting Surfaces 3850
39.1.13.5. Surface Statistics 3850
39.2. Using Cell Registers 3851
39.2.1. Region 3852
39.2.1.1. Defining a Region 3852
39.2.1.2. Setting Up a Region Cell Register 3852
39.2.2. Boundary 3854
39.2.3. Variable Limiter 3855
39.2.4. Field Variable 3855
39.2.4.1. Approaches For Deriving Field Values 3855
39.2.4.2. Setting Up a Field Variable Cell Register 3857
39.2.5. Residuals 3860
39.2.6. Volume 3861
39.2.6.1. Volume Cell Register Approach 3861
39.2.6.2. Setting Up a Volume Cell Register 3861
39.2.7. Yplus/Ystar 3862
39.2.7.1. Yplus/Ystar Approach 3862
39.2.7.2. Setting up a Yplus/Ystar Cell Register 3863
39.2.8. Manage Cell Registers 3864
39.2.9. Cell Register Operations 3865
39.2.10. Copying and Renaming Cell Registers 3867
40. Displaying Graphics 3869

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

40.1. Basic Graphics Generation 3869
40.1.1. Graphics Performance 3870
40.1.2. Displaying the Mesh 3872
40.1.2.1. Generating Mesh or Outline Plots 3875
40.1.2.2. Modifying the Mesh Colors 3877
40.1.2.3. Realistic Rendering of Materials 3882
40.1.2.3.1. List of Default Realistic Materials 3887
40.1.2.3.2. Creating Custom Realistic Materials 3889
40.1.2.3.3. Limitations in Realistic Material Rendering 3890
40.1.2.4. Controlling Mesh Transparency 3891
40.1.2.5. Mesh and Outline Display Options 3893
40.1.2.5.1. Adding Features to an Outline Display 3893
40.1.2.5.2. Drawing Partition Boundaries 3894
40.1.2.5.3. Shrinking Faces and Cells in the Display 3894
40.1.2.6. Creating and Using Mesh Plot Definitions 3896
40.1.3. Displaying Contours and Profiles 3896
40.1.3.1. Quickly Coloring Surfaces by Field Variable Value 3897
40.1.3.2. Generating Contour and Profile Plots 3898
40.1.3.3. Contour and Profile Plot Options 3901
40.1.3.3.1. Drawing Filled Contours or Profiles 3901
40.1.3.3.2. Specifying the Range of Magnitudes Displayed 3902
40.1.3.3.3. Including the Mesh in the Contour Plot 3904
40.1.3.3.4. Choosing Node or Cell Values and Node or Boundary Values 3905
40.1.3.3.5. Storing Contour Plot Settings 3910
40.1.3.4. Creating and Using Contour Plot Definitions 3910
40.1.4. Displaying Vectors 3912
40.1.4.1. Generating Vector Plots 3912
40.1.4.2. Displaying Relative Velocity Vectors 3914
40.1.4.3. Vector Plot Options 3914
40.1.4.3.1. Scaling the Vectors 3915
40.1.4.3.2. Skipping Vectors 3915
40.1.4.3.3. Drawing Vectors in the Plane of the Surface 3916
40.1.4.3.4. Displaying Fixed-Length Vectors 3916
40.1.4.3.5. Displaying Vector Components 3916
40.1.4.3.6. Specifying the Range of Magnitudes Displayed 3917
40.1.4.3.7. Changing the Scalar Field Used for Coloring the Vectors 3917
40.1.4.3.8. Controlling 3D Vector Tessellation for Performance and Appearance 3917
40.1.4.3.9. Displaying Vectors Using a Single Color 3918
40.1.4.3.10. Including the Mesh in the Vector Plot 3919
40.1.4.3.11. Changing the Arrow Characteristics 3919
40.1.4.4. Creating and Managing Custom Vectors 3919
40.1.4.4.1. Creating Custom Vectors 3919
40.1.4.4.2. Manipulating, Saving, and Loading Custom Vectors 3920
40.1.4.5. Creating and Using Vector Plot Definitions 3922
40.1.5. Displaying Pathlines 3922
40.1.5.1. Steps for Generating Pathlines 3923
40.1.5.2. Options for Pathline Plots 3924
40.1.5.2.1. Including the Mesh in the Pathline Display 3924
40.1.5.2.2. Controlling the Pathline Style 3925
40.1.5.2.3. Controlling Pathline Colors 3926
40.1.5.2.4. "Thinning" Pathlines 3926

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

40.1.5.2.5. Coarsening Pathlines 3927
40.1.5.2.6. Reversing the Pathlines 3927
40.1.5.2.7. Plotting Oil-Flow Pathlines 3927
40.1.5.2.8. Controlling the Pulse Mode 3927
40.1.5.2.9. Controlling the Accuracy 3928
40.1.5.2.10. Plotting Relative Pathlines 3928
40.1.5.2.11. Generating an XY Plot Along Pathline Trajectories 3928
40.1.5.2.12. Saving Pathline Data 3928
40.1.5.2.12.1. Standard Type 3929
40.1.5.2.12.2. Geometry Type 3930
40.1.5.2.12.3. EnSight Type 3930
40.1.5.2.13. Choosing Node or Cell Values 3932
40.1.5.3. Creating and Using Pathline Definitions 3932
40.1.6. Displaying Line Integral Convolutions (LICs) 3932
40.1.6.1. Generating an LIC Plot 3933
40.1.7. Displaying a Scene 3934
40.1.7.1. Generating a Scene 3935
40.1.8. Displaying Results on a Sweep Surface 3937
40.1.8.1. Steps for Generating a Plot Using a Sweep Surface 3937
40.1.8.2. Animating a Sweep Surface Display 3939
40.1.9. Hiding the Graphics Window Display 3939
40.2. Customizing the Graphics Display 3940
40.2.1. Embedded Graphics Window Dashboards 3941
40.2.1.1. Automatically Embedding Windows in the Solution View 3942
40.2.1.2. Embedding Windows with the Dashboard Definition Dialog Box 3944
40.2.1.3. Manually Embedding Windows 3947
40.2.1.4. Animating Embedded Window Dashboards 3950
40.2.1.5. Limitations for Embedded Windows 3953
40.2.2. Advanced Graphics Overlays 3954
40.2.3. Managing Multiple Graphics Windows 3955
40.2.3.1. Setting the Active Window 3957
40.2.3.2. Synchronizing Window Views 3957
40.2.4. Showing Boundary Markers 3957
40.2.5. Changing the Legend Display 3959
40.2.5.1. Controlling the Titles, Axes, Ruler, Logo, and Colormap 3959
40.2.5.2. Editing the Legend 3959
40.2.5.3. Adding a Title to the Caption 3960
40.2.5.4. Enabling/Disabling the Axes 3960
40.2.5.5. Enabling/Disabling the Ruler 3960
40.2.5.6. Modifying and Displaying/Hiding the Logo 3960
40.2.5.7. Colormap Alignment 3960
40.2.6. Adding Text to the Graphics Window 3961
40.2.6.1. Adding Text Using the Annotate Dialog Box 3961
40.2.6.2. Editing Existing Annotation Text 3963
40.2.6.3. Clearing Annotation Text 3963
40.2.7. Changing the Colormap and Range 3963
40.2.7.1. Colormap Nomenclature 3966
40.2.7.2. Predefined Colormaps 3967
40.2.7.3. Selecting a Colormap 3980
40.2.7.3.1. Specifying the Colormap Size and Scale 3980
40.2.7.3.2. Changing the Number Format 3980

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

40.2.7.4. Displaying Colormap Labels and Titles 3981
40.2.7.5. Creating a Customized Colormap 3982
40.2.7.6. Colormap References 3984
40.2.8. Adding Lights 3984
40.2.8.1. Controlling Lighting Effects with the Display Options Dialog Box 3985
40.2.8.2. Controlling Lighting Effects with the **Lights** Dialog Box 3985
40.2.8.3. Defining Light Sources 3985
40.2.8.3.1. Removing a Light 3987
40.2.8.3.2. Resetting the Light Definitions 3987
40.2.9. Modifying the Rendering Options 3987
40.2.9.1. Graphics Device Information 3989
40.3. Enhanced Graphics Visual Effects 3989
40.3.1. Optimizing Graphical Priorities 3990
40.3.2. Graphics Effects Options 3990
40.4. Realistic Rendering Using Raytracing 3995
40.4.1. Animating with Realistic Raytracer Images 4000
40.4.1.1. Realistic Raytracer Images Captured at Runtime 4000
40.4.1.2. Raytracer Image Conversion Using the Playback Dialog Box 4002
40.4.2. Saving Pictures with Realistic Raytracer Images 4006
40.4.3. Listing of Realistic Environments and Backplates 4010
40.4.4. Limitations with Realistic Raytracing 4020
40.5. Controlling the Mouse Button Functions 4020
40.6. Viewing the Application Window 4023
40.7. Measuring Distance and Angle 4024
40.8. Controlling the Display State and Modifying the View 4025
40.8.1. Specifying a Display State 4026
40.8.2. Selecting a View 4027
40.8.3. Manipulating the Display 4029
40.8.3.1. Scaling and Centering 4030
40.8.3.2. Rotating the Display 4030
40.8.3.2.1. Spinning the Display with the Mouse 4031
40.8.3.3. Translating the Display 4031
40.8.3.4. Zooming the Display 4032
40.8.4. Controlling Perspective and Camera Parameters 4033
40.8.4.1. Perspective and Orthographic Views 4033
40.8.4.2. Modifying Camera Parameters 4034
40.8.5. Saving and Restoring Views 4034
40.8.5.1. Restoring the Default View 4035
40.8.5.2. Returning to Previous Views 4035
40.8.5.3. Saving Views 4035
40.8.5.4. Reading View Files 4036
40.8.5.5. Deleting Views 4036
40.8.6. Mirroring and Periodic Repeats 4036
40.8.6.1. Periodic Repeats for Graphics 4039
40.8.6.2. Mirroring for Graphics 4041
40.9. Advanced Scene Composition 4042
40.9.1. Selecting the Object(s) to be Manipulated 4043
40.9.2. Changing an Object's Display Properties 4043
40.9.2.1. Controlling Visibility 4044
40.9.2.2. Controlling Object Color and Transparency 4045
40.9.3. Transforming Geometric Objects in a Scene 4046

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

40.9.3.1. Translating Objects 4047
40.9.3.2. Rotating Objects 4047
40.9.3.3. Scaling Objects 4047
40.9.3.4. Displaying the Meridional View 4048
40.9.4. Modifying Iso-Values 4048
40.9.4.1. Steps for Modifying Iso-Values 4048
40.9.5. Modifying Pathline Attributes 4048
40.9.6. Deleting an Object from the Scene 4049
40.9.7. Adding a Bounding Frame 4049
40.10. Animating Graphics 4051
40.10.1. Creating an Animation 4053
40.10.1.1. Deleting Key Frames 4053
40.10.2. Playing an Animation 4054
40.10.2.1. Playing Back an Excerpt 4054
40.10.2.2. "Fast-Forwarding" the Animation 4054
40.10.2.3. Continuous Animation 4054
40.10.2.4. Stopping the Animation 4055
40.10.2.5. Advancing the Animation Frame by Frame 4055
40.10.3. Saving an Animation 4055
40.10.3.1. Animation File 4055
40.10.3.2. Picture File 4055
40.10.3.3. MPEG File 4056
40.10.4. Reading an Animation File 4056
40.10.5. Notes on Animation 4056
40.11. Histogram and XY Plots 4057
40.11.1. Plot Types 4057
40.11.1.1. XY Plots 4057
40.11.1.2. Histograms 4058
40.11.1.3. Enhanced Interactive Plots 4058
40.11.2. XY Plots of Solution Data 4060
40.11.2.1. Steps for Generating Solution XY Plots 4060
40.11.2.2. Options for Solution XY Plots 4063
40.11.2.2.1. Including External Data in the Solution XY Plot 4063
40.11.2.2.2. Choosing Node or Cell Values 4064
40.11.2.2.3. Saving the Plot Data to a File 4064
40.11.3. Creating an XY Plot From Multiple Data Sources (Including Files) 4064
40.11.3.1. Steps for Generating XY Plots of Data from Multiple Sources 4065
40.11.4. XY Plots of Profiles 4066
40.11.4.1. Steps for Generating Plots of Profile Data 4066
40.11.4.2. Steps for Generating Plots of Interpolated Profile Data 4067
40.11.5. XY Plots of Circumferential Averages 4068
40.11.5.1. Steps for Generating an XY Plot of Circumferential Averages 4068
40.11.5.2. Customizing the Appearance of the Plot 4071
40.11.6. XY Plot File Format 4071
40.11.7. Residual Plots 4072
40.11.8. Histograms 4072
40.11.8.1. Steps for Generating Histogram Plots 4072
40.11.8.2. Options for Histogram Plots 4074
40.11.8.2.1. Specifying the Range of Values Plotted 4074
40.11.8.2.2. Adding Histograms to Your Simulation Reports 4074
40.11.9. Modifying Axis Attributes 4074

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

40.11.9.1. Using the Axes Dialog Box 4075
40.11.9.1.1. Changing the Axis Title and Font Size 4075
40.11.9.1.2. Changing the Format of the Data Labels 4075
40.11.9.1.3. Choosing Logarithmic or Decimal Scaling 4075
40.11.9.1.4. Resetting the Range of the Axis 4076
40.11.9.1.5. Controlling the Major and Minor Rules 4076
40.11.10. Modifying Curve Attributes 4076
40.11.10.1. Using the Curves Dialog Box 4077
40.11.10.1.1. Changing the Line Style 4077
40.11.10.1.2. Changing the Marker Style 4077
40.11.10.1.3. Previewing the Curve Style 4078
40.12. Fast Fourier Transform (FFT) Postprocessing 4078
40.12.1. Limitations of the FFT Algorithm 4078
40.12.2. Windowing 4079
40.12.3. Fast Fourier Transform (FFT) 4080
40.12.4. Using the FFT Utility 4081
40.12.4.1. Loading Data for Spectral Analysis 4082
40.12.4.2. Customizing the Input and Defining the Spectrum Smoothing 4083
40.12.4.2.1. Customizing the Input Signal Data Set 4083
40.12.4.2.2. Spectrum Smoothing Through Signal Segmentation 4084
40.12.4.2.3. Viewing Data Statistics 4084
40.12.4.2.4. Customizing Titles and Labels 4084
40.12.4.2.5. Applying the Changes in the Input Signal Data 4085
40.12.4.3. Customizing the Output 4085
40.12.4.3.1. Specifying a Function for the Y Axis 4085
40.12.4.3.2. Specifying a Function for the X Axis 4087
40.12.4.3.3. Specifying Output Options 4089
40.12.4.3.4. Specifying a Windowing Technique 4089
40.12.4.3.5. Specifying Labels and Titles 4089
40.12.4.4. Performing an Ansys Sound Analysis 4089
40.13. Cumulative Force, Moment, and Coefficients Plots 4094
40.13.1. Steps for Generating Cumulative Plots 4094
41. Reporting Alphanumeric Data 4097
41.1. Reporting Conventions 4097
41.2. Monitoring and Reporting Solution Data 4098
41.2.1. Creating Report Definitions 4098
41.2.1.1. Surface Report Definitions 4103
41.2.1.2. Volume Report Definitions 4104
41.2.1.3. Force and Moment Report Definitions 4106
41.2.1.4. Flux Report Definition 4111
41.2.1.5. Mesh Report Definitions 4114
41.2.1.6. Aerodamping (Travelling Wave Method) Report Definition 4117
41.2.1.7. DPM Report Definition 4118
41.2.1.8. User Defined Report Definition 4120
41.2.1.8.1. User Defined Report Definition Function 4120
41.2.1.8.2. User Defined Report Definition Function Hooking 4121
41.2.1.9. Expression Report Definition 4121
41.2.2. Report Files and Report Plots 4123
41.2.2.1. Creating Report Files 4123
41.2.2.2. Creating Report Plots 4126
41.2.2.3. Moving Average Monitors 4129

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

41.2.2.4. Clearing File and Plot Histories 4132
41.3. Creating Output Parameters 4133
41.4. Fluxes Through Boundaries 4135
41.4.1. Generating a Flux Report 4136
41.4.2. Flux Reporting for Reacting Flows 4139
41.4.3. Flux Reporting with Particles 4140
41.4.4. Flux Reporting with Multiphase 4142
41.4.5. Flux Reporting with the DDPM 4142
41.4.6. Flux Reporting with the Potential Solver 4144
41.4.7. Flux Reporting with Other Volumetric Sources 4145
41.5. Forces on Boundaries 4145
41.5.1. Generating a Force, Moment, or Center of Pressure Report 4145
41.5.1.1. Example 4148
41.6. Projected Surface Area Calculations 4149
41.7. Surface Integration 4150
41.7.1. Generating a Surface Integral Report 4151
41.8. Volume Integration 4153
41.8.1. Generating a Volume Integral Report 4153
41.9. Efficiency Calculations 4154
41.9.1. Device Efficiency 4154
41.9.2. Calculating Efficiency using Named Expressions 4156
41.9.3. Limitations 4157
41.10. Histogram Reports 4158
41.11. Discrete Phase 4158
41.12. S2S Information 4158
41.13. Reference Values 4158
41.13.1. Setting Reference Values 4159
41.13.2. Setting the Reference Zone 4161
41.14. Summary Reports of Case Settings 4161
41.14.1. Modified Settings Summary 4161
41.14.2. Generating a Summary Report 4162
41.15. System Resource Usage 4163
41.15.1. Processor Information 4163
41.15.2. Memory Information 4164
41.15.3. Process and Model Timers 4165
42. Field Function Definitions 4167
42.1. Node, Cell, and Facet Values 4167
42.1.1. Cell Values 4167
42.1.2. Node Values 4168
42.1.2.1. Vertex Values for Points That Are Not Mesh Nodes 4168
42.1.3. Facet Values 4168
42.1.3.1. Facet Values on Zone Surfaces 4169
42.1.3.2. Facet Values on Postprocessing Surfaces 4169
42.2. Velocity Reporting Options 4169
42.3. Field Variables Listed by Category 4171
42.4. Alphabetical Listing of Field Variables and Their Definitions 4205
42.5. Custom Field Functions 4285
42.5.1. Creating a Custom Field Function 4286
42.5.1.1. Using the Calculator Buttons 4288
42.5.1.2. Using the Field Functions List 4288
42.5.2. Manipulating, Saving, and Loading Custom Field Functions 4289

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

42.5.3. Sample Custom Field Functions 4290
43. Parallel Processing 4293
43.1. Introduction to Parallel Processing 4293
43.1.1. Recommended Usage of Parallel Ansys Fluent 4295
43.2. Starting Parallel Ansys Fluent Using Fluent Launcher 4296
43.2.1. Setting Parallel Scheduler Options in Fluent Launcher 4298
43.2.2. Setting Additional Options When Running on Remote Linux Machines 4301
43.2.2.1. Setting Job Scheduler Options When Running on Remote Linux Machines 4303
43.3. Starting Parallel Ansys Fluent on a Windows System 4306
43.3.1. Starting Parallel Ansys Fluent on a Windows System Using Command Line Options 4306
43.3.1.1. Starting Parallel Ansys Fluent with the Microsoft Job Scheduler 4308
43.4. Starting Parallel Ansys Fluent on a Linux System 4311
43.4.1. Starting Parallel Ansys Fluent on a Linux System Using Command Line Options 4311
43.4.2. Setting Up Your Secure Shell Clients 4314
43.4.2.1. Configuring the ssh Client 4314
43.5. Mesh Partitioning and Load Balancing 4315
43.5.1. Overview of Mesh Partitioning 4315
43.5.2. Partitioning the Mesh Automatically 4316
43.5.2.1. Preferences for Advanced Auto-Partitioning Methods 4318
43.5.2.2. Reporting During Auto Partitioning 4318
43.5.3. Partitioning the Mesh Manually and Balancing the Load 4319
43.5.3.1. Guidelines for Partitioning the Mesh 4319
43.5.4. Using the Partitioning and Load Balancing Dialog Box 4319
43.5.4.1. Partitioning 4319
43.5.4.1.1. Example of Setting Selected Cell Registers to Specified Partition IDs 4325
43.5.4.1.2. Partitioning Within Zones or Registers 4328
43.5.4.1.3. Reporting During Partitioning 4328
43.5.4.1.4. Resetting the Partition Parameters 4329
43.5.4.2. Load Balancing 4329
43.5.5. Mesh Partitioning Methods 4331
43.5.5.1. Partition Methods 4332
43.5.5.2. Optimizations 4336
43.5.5.3. Pretesting 4337
43.5.5.4. Using the Partition Filter 4338
43.5.6. Checking the Partitions 4338
43.5.6.1. Interpreting Partition Statistics 4339
43.5.6.2. Examining Partitions Graphically 4342
43.5.7. Load Distribution 4342
43.5.8. Troubleshooting 4342
43.6. Using General Purpose Graphics Processing Units (GPGPUs) With the Algebraic Multigrid (AMG) Solver 4343
43.6.1. Requirements 4343
43.6.2. Limitations 4343
43.6.3. Using and Managing GPGPUs 4344
43.7. Controlling the Threads 4345
43.8. Checking Network Connectivity 4345
43.9. Checking and Improving Parallel Performance 4346
43.9.1. Parallel Check 4347
43.9.2. Checking Parallel Performance 4347
43.9.2.1. Checking Latency and Bandwidth 4349
43.9.3. Optimizing the Parallel Solver 4351

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

43.9.3.1. Increasing the Report Interval 4351
43.9.3.2. Accelerating View Factor Calculations Using General Purpose Graphics Processing Units (GPGPUs) 4351
43.9.3.3. Accelerating Discrete Ordinates (DO) Radiation Calculations 4352
43.9.3.3.1. Accelerated DO Model Limitations 4353
43.9.4. Clearing the Linux File Cache Buffers 4353
44. Running Ansys Fluent on Arm Compute Nodes 4355
44.1. Setting up Fluent for Arm 4356
44.2. Message Passing Interface (MPI) Supported with Fluent for Arm 4356
44.3. Starting Fluent for Arm 4356
44.4. Running Fluent for Arm with a Job Scheduler 4357
44.5. Fluent for Arm Known Limitations 4357
45. Using Simulation Reports 4359
45.1. Overview of Simulation Reports 4359
45.1.1. Limitations for Simulations Reports 4362
45.2. Preparing Simulation Reports 4362
45.2.1. Setting General Report Properties 4364
45.2.2. Organizing Your Simulation Report 4364
45.3. Generating Simulation Reports 4366
45.4. Viewing Simulation Reports 4367
45.4.1. Viewing System Information 4367
45.4.2. Viewing Geometry and Mesh Information 4368
45.4.3. Viewing Simulation Setup Information 4369
45.4.4. Viewing Run Information 4373
45.4.5. Viewing Solution Status 4374
45.4.6. Viewing Named Expression Information 4375
45.4.7. Viewing Report Definition Information 4375
45.4.8. Viewing Plot Information 4375
45.4.9. Viewing Contours, Vectors, Pathlines, LICs, XY Plots, Scenes, Histograms, and Animations 4378
45.5. Saving Simulation Reports 4383
45.6. Additional Simulation Report Options 4383
45.7. Customizing Simulation Reports 4384
45.7.1. Customizing Your Report Image Settings 4384
45.7.2. Changing the Layout of Your Results 4384
45.7.3. Customizing Your Report Settings Using Templates 4385
45.7.4. Adding Additional Graphics to Your Report 4386
45.7.5. Hiding and Showing Report Sections 4386
45.8. Generating Reports Using the Text User Interface (TUI) 4386
46. Performing Parametric Studies 4387
46.1. Prerequisites 4388
46.2. Limitations 4389
46.3. Getting Started With Your Parametric Study 4389
46.4. Using the Parametric Ribbon 4390
46.5. Using the Outline View for Parametric Studies 4391
46.6. Working With the Design Point Table 4393
46.6.1. Customizing the Design Point Table 4394
46.6.2. Updating Design Points 4395
46.6.2.1. Updating Design Points in the Same Session Versus a New Session 4396
46.6.2.1.1. Running Locally Concurrent Design Point Updates with GPUs 4400
46.6.2.2. Updating the Current Design Point 4400

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

46.6.2.3. Updating All Design Points 4401
46.6.3. Adding Design Points 4401
46.6.3.1. Manually Adding Design Points 4401
46.6.3.2. Automatically Adding Design Points 4402
46.6.3.2.1. Creating Designs of Experiments for optiSLang 4402
46.6.3.2.2. Creating and Optimizing Designs of Experiments for optiSLang 4404
46.6.3.2.2.1. Configuring optiSLang AMOP Settings 4407
46.6.3.2.2.2. Configuring optiSLang One-Click Settings 4409
46.6.4. Operating on Design Points 4409
46.6.5. Importing and Exporting Design Point Tables 4410
46.6.6. Saving Journals When Updating Design Points 4411
46.6.7. Accounting for Mesh Morphing During Parametric Updates 4411
46.6.8. Parametric Design Point Process Details and Case Change Considerations 4412
46.6.8.1. Initializing Design Point Data 4413
46.6.8.2. Reusing Your Case File 4414
46.6.8.3. Managing Case File Changes 4416
46.6.8.4. Managing Case Changes During Design Point Updates 4416
46.7. Monitoring and Viewing Design Point Update Status 4417
46.7.1. Exporting Parametric Designs to optiSLang 4420
46.7.2. Comparing Parametric Plots 4421
46.7.3. Customizing Graphical Plots 4423
46.8. Creating Simulation Reports for Design Points and Parametric Studies 4424
46.8.1. Creating Design Point Reports for Your Simulation 4424
46.8.1.1. Accessing Design Point Report Settings 4424
46.8.1.2. Overview of Design Point Report Settings 4427
46.8.1.3. Generating, Viewing, and Saving Your Design Point Reports 4428
46.8.2. Creating Parametric Reports for Your Simulation 4428
46.8.2.1. Accessing Parametric Report Settings 4429
46.8.2.2. Overview of Parametric Report Settings 4430
46.8.2.3. Comparing Parametric Results 4433
46.8.2.4. Generating, Viewing, and Saving Your Parametric Reports 4436
46.9. Setting Preferences for Parametric Studies 4436
46.10. Viewing the Current Case Parameters 4438
46.11. Managing Files for Your Parametric Studies 4438
46.11.1. Organizing Parametric Studies Using Projects 4438
46.11.1.1. Reading and Writing Project Files 4438
46.11.1.2. Using the Parametric Project View 4439
46.11.2. Managing Design Point Files 4442
46.11.3. Using Lightweight Parametric Projects 4442
47. Remotely Accessing Your Simulations Using Ansys Fluent's Web Interface 4445
47.1. Overview 4445
47.1.1. Requirements 4447
47.1.2. Limitations 4447
47.2. Getting Started With Ansys Fluent's Web Interface 4447
47.2.1. Specifying Web Server Details 4449
47.2.2. Connecting to Your Simulation Web Sessions 4449
47.3. Using Ansys Fluent's Web Interface 4450
47.3.1. Overview of the Graphical Interface 4450
47.3.1.1. The Main Menu 4453
47.3.1.2. The Outline View 4454
47.3.1.3. The Graphics Window 4456

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

47.3.1.4. The View Arc 4458
47.3.1.5. The Status Bar 4459
47.3.1.6. The Stage Navigator 4460
47.3.1.7. The Console Panel 4460
47.3.1.7.1. Using the Console to Interact with Your Session Using Python 4462
47.3.1.8. The Results Arc 4463
47.3.1.9. The Plots View 4465
47.3.1.10. Property Panels 4466
47.3.1.11. Working with Units 4467
47.3.1.12. Working with Expressions 4467
47.3.1.13. Editing Multiple Objects at Once 4468
47.3.2. Interacting with Your Simulation Setup 4469
47.3.2.1. Accessing General Settings 4470
47.3.2.2. Accessing Model Settings 4470
47.3.2.3. Accessing Material Settings 4470
47.3.2.4. Accessing Cell Zone Settings 4470
47.3.2.5. Accessing Boundary Condition Settings 4471
47.3.2.6. Accessing Reference Values Settings 4471
47.3.2.7. Accessing Reference Frame Settings 4471
47.3.2.8. Accessing Named Expression Settings 4472
47.3.3. Interacting with Your Solution Settings 4472
47.3.3.1. Setting Solution Methods 4472
47.3.3.2. Setting Solution Controls 4473
47.3.3.3. Setting Report Definitions 4473
47.3.3.4. Setting Solution Monitors 4473
47.3.3.5. Initializing the Solution 4474
47.3.3.6. Running the Calculations 4474
47.3.3.7. Interrupting and Pausing Calculations 4474
47.3.4. Interacting with the Results of Your Simulation 4475
47.3.4.1. Working With Surfaces and Your Simulation Results 4475
47.3.4.1.1. Creating and Displaying Point Surfaces 4476
47.3.4.1.2. Creating and Displaying Line Surfaces 4476
47.3.4.1.3. Creating and Displaying Rake Surfaces 4476
47.3.4.1.4. Creating and Displaying Plane Surfaces 4476
47.3.4.1.5. Creating and Displaying Iso Surfaces 4477
47.3.4.1.6. Creating and Displaying Iso-Clip Surfaces 4477
47.3.4.1.7. Creating and Displaying Transform Surfaces 4477
47.3.4.1.8. Creating and Displaying Plane Slice Surfaces 4478
47.3.4.1.9. Creating and Displaying Sphere Slice Surfaces 4478
47.3.4.1.10. Creating and Displaying Quadric Surfaces 4478
47.3.4.2. Working With Graphics Objects and Your Simulation Results 4479
47.3.4.2.1. Creating and Displaying Mesh Objects 4479
47.3.4.2.2. Creating and Displaying Contours Objects 4480
47.3.4.2.3. Creating and Displaying Vector Objects 4480
47.3.4.2.4. Creating and Displaying Pathline Objects 4480
47.3.4.2.5. Creating and Displaying Mirror Plane Objects 4480
47.3.4.2.5.1. Creating New Mirror Planes 4481
47.3.4.2.5.2. Applying Your Mirror Planes 4482
47.3.4.3. Working With Plots and Your Simulation Results 4482
47.3.4.3.1. Creating and Displaying XY Plots 4483
47.3.4.3.2. Generating Histogram Data 4483

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

47.3.4.3.3. Generating Cumulative Data 4483
47.3.4.3.4. Viewing Residual Plots 4484
47.3.4.3.5. Viewing Report Definition Plots 4484
47.3.4.3.6. Displaying Multiple Plots 4485
47.3.4.3.7. Analyzing Data in Plots 4485
47.3.4.4. Creating and Displaying Scenes and Your Simulation Results 4486
47.3.4.5. Creating and Displaying Reports for Your Simulation Results 4486
47.3.5. Working With Parameters 4488
47.4. Setting Up Your Environment (System Administrators) 4488
47.4.1. Supporting HTTPS 4490
48. Design Analysis and Optimization 4491
48.1. The Adjoint Solver 4491
48.1.1. General Observables 4493
48.1.2. General Operations 4497
48.1.3. Discrete Versus Continuous Adjoint Solver 4499
48.1.4. Discrete Adjoint Solver Overview 4500
48.1.5. Adjoint Solver Stabilization 4503
48.1.6. Solution-Based Adaption 4504
48.1.7. Using The Data To Improve A Design 4505
48.1.7.1. Smoothing and Mesh Morphing 4506
48.1.7.1.1. Polynomials-Based Approach 4507
48.1.7.1.2. Direct Interpolation Method 4508
48.1.7.1.3. Radial Basis Function 4508
48.2. Using the Adjoint Solver 4509
48.2.1. Model Considerations for Using Adjoint Solver 4510
48.2.1.1. Basic Assumptions and Consistency Checks 4510
48.2.1.2. User-Defined Sources 4513
48.2.2. Defining Observables 4515
48.2.2.1. Creating New Observables 4516
48.2.2.2. Editing Observable Definitions 4517
48.2.2.3. Defining Observables Using Expressions 4520
48.2.2.4. Selecting an Observable for Sensitivity Calculation 4524
48.2.2.5. Defining Observables for Turbulence Model Optimization 4525
48.2.3. Solving the Adjoint 4526
48.2.3.1. Using the Adjoint Solution Methods Dialog Box 4526
48.2.3.2. Using the Adjoint Solution Controls Dialog Box 4530
48.2.3.2.1. Stabilization Strategies, Schemes, and Settings 4533
48.2.3.2.1.1. Dissipation Scheme 4534
48.2.3.2.1.2. Residual Minimization Scheme 4535
48.2.3.3. Working with Adjoint Residual Monitors 4537
48.2.3.4. Printing and Postprocessing the Adjoint Equation Residuals 4538
48.2.3.5. Running the Adjoint Calculation 4539
48.2.3.5.1. Automatic Saving of Case and Data Files During an Adjoint Calculation 4540
48.2.4. Postprocessing of Adjoint Solutions 4541
48.2.4.1. Using the Adjoint Postprocess Options Dialog Box 4542
48.2.4.2. Field Data 4543
48.2.4.3. Scalar Data 4552
48.2.5. Modifying the Geometry Using the Design Tool 4553
48.2.5.1. Defining the Region for the Design Change 4556
48.2.5.2. Defining Region Conditions 4558
48.2.5.3. Defining Observable Objectives 4559

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

48.2.5.4. Defining Conditions for the Deformation 4560
48.2.5.5. Design Tool Numerics 4570
48.2.5.6. Shape Modification 4575
48.2.6. Using the Gradient-Based Optimizer 4583
48.2.6.1. Using the Turbulence Model Design Tool 4599
48.2.6.1.1. Turbulence Model Optimization Strategies 4601
48.2.6.1.1.1. Online Mode Optimization 4602
48.2.6.1.1.2. Offline Mode Optimization 4602
48.2.6.1.2. Defining the Design Region 4603
48.2.6.1.3. Defining Optimizer Design Variable Settings 4604
48.2.6.1.3.1. Defining Design Variables for Optimization 4605
48.2.6.1.3.2. Defining the Model for Design Variables 4606
48.2.6.1.3.2.1. Defining Neural Network Model Settings 4607
48.2.6.1.3.2.2. Defining Offline Training Settings 4611
48.2.6.1.3.3. Verifying the Trained Turbulence Model 4613
48.2.6.1.3.4. Managing Trained Turbulence Models 4613
48.3. Geometry Parameterization and Exploration 4614
48.4. The Mesh Morpher/Optimizer 4616
48.4.1. Limitations 4617
48.4.2. The Optimization Process 4617
48.4.3. Optimizers 4618
48.4.3.1. The Compass Optimizer 4618
48.4.3.2. The NEWUOA Optimizer 4618
48.4.3.3. The Simplex Optimizer 4619
48.4.3.4. The Torczon Optimizer 4619
48.4.3.5. The Powell Optimizer 4619
48.4.3.6. The Rosenbrock Optimizer 4620
48.5. Using the Mesh Morpher/Optimizer 4620
49. Performing System Coupling Simulations Using Fluent 4647
49.1. Supported Capabilities and Limitations 4647
49.2. Performing System Coupling in the GUI or CLI 4650
49.2.1. Generating a System Coupling Setup File 4651
49.2.2. Using Custom Input Files in System Coupling's User Interfaces 4652
49.2.2.1. Setting the Fluent Input Option 4652
49.2.2.2. Generating a Fluent Journal Script 4653
49.3. Performing System Coupling in Ansys Workbench 4653
49.4. Variables Available for System Coupling 4654
49.4.1. Force transferred to System Coupling from a Wall Boundary 4655
49.4.2. Force transferred to System Coupling from a Porous Jump Boundary 4656
49.4.3. Displacement transferred from System Coupling 4657
49.4.4. Displacement transferred from System Coupling to a Sliding Mesh Zone 4658
49.4.5. Absolute Pressure Example 4658
49.5. System Coupling Related Settings in Fluent 4658
49.6. FSI Setup Recommendations for Fluent-Mechanical Couplings 4660
49.6.1. Using Contact Detection for Fluent-Mechanical FSI Problems 4661
49.6.2. Recommendations for Dynamic Mesh Settings for Fluent-Mechanical FSI 4663
49.6.3. Pathologies & Candidate Resolutions for Fluent-Mechanical FSI 4665
49.6.3.1. Mesh Folds within the First Coupling Steps 4666
49.6.3.2. Deformed Prism Layers 4666
49.6.3.2.1. Using Boundary Layer Smoothing and Region Face Remeshing 4667
49.6.3.2.2. Overset Meshes 4671

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

49.6.3.3. Interior Elements have High Skewness or Are Too Large/small 4673
49.6.3.4. Divergence if Flow Block-Off is Established at the Beginning of a Run 4675
49.7. How Fluent's Execution is Affected by System Couplings 4676
49.8. Restarting Fluent Analyses as Part of System Couplings 4676
49.8.1. Generating Fluent Restart Files 4676
49.8.2. Specify a Restart Point in Fluent 4676
49.8.3. Making Setup Changes Before Restarting 4677
49.8.4. Recovering the Fluent Restart Point after a Workbench Crash 4677
49.9. System Coupling case with Fluent using Patched Data 4678
49.10. Running Fluent as a Participant from System Coupling's GUI or CLI 4679
49.11. Troubleshooting Two-Way Coupled Analysis Problems 4680
49.12. Product Licensing Considerations when using System Coupling 4680
50. Customizing Fluent 4681
51. Task Page Reference Guide 4683
51.1. Meshing Task Page 4683
51.2. Setup Task Page 4683
51.3. General Task Page 4684
51.3.1. Scale Mesh Dialog Box 4686
51.3.2. Mesh Display Dialog Box 4688
51.3.3. Set Units Dialog Box 4693
51.3.4. Define Unit Dialog Box 4694
51.3.5. Mesh Colors Dialog Box 4695
51.4. Models Task Page 4696
51.4.1. Multiphase Model Dialog Box 4700
51.4.2. Energy Dialog Box 4714
51.4.3. Viscous Model Dialog Box 4715
51.4.4. Radiation Model Dialog Box 4736
51.4.5. View Factors and Clustering Dialog Box 4741
51.4.6. Participating Boundary Zones Dialog Box 4745
51.4.7. Solar Calculator Dialog Box 4747
51.4.8. Heat Exchanger Model Dialog Box 4748
51.4.9. Dual Cell Heat Exchanger Dialog Box 4749
51.4.10. Set Dual Cell Heat Exchanger Dialog Box 4750
51.4.11. Heat Transfer Data Table Dialog Box 4753
51.4.12. NTU Table Dialog Box 4755
51.4.13. Copy From Dialog Box 4756
51.4.14. Ungrouped Macro Heat Exchanger Dialog Box 4756
51.4.15. Velocity Effectiveness Curve Dialog Box 4761
51.4.16. Core Porosity Model Dialog Box 4761
51.4.17. Macro Heat Exchanger Group Dialog Box 4763
51.4.18. Species Model Dialog Box 4768
51.4.19. Coal Calculator Dialog Box 4791
51.4.20. Integration Parameters Dialog Box 4795
51.4.21. Flamelet 3D Surfaces Dialog Box 4797
51.4.22. Flamelet 2D Curves Dialog Box 4800
51.4.23. Unsteady Flamelet Parameters Dialog Box 4801
51.4.24. Flamelet Fluid Zones Dialog Box 4802
51.4.25. Select Transported Scalars Dialog Box 4803
51.4.26. Distribution of Points Dialog Box 4803
51.4.27. PDF Table Dialog Box 4805
51.4.28. Spark Ignition Dialog Box 4809

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

51.4.29. Set Spark Ignition Dialog Box 4810
51.4.30. Autoignition Model Dialog Box 4811
51.4.31. Inert Dialog Box 4814
51.4.32. NOx Model Dialog Box 4816
51.4.33. Soot Model Dialog Box 4825
51.4.34. Sticking Coefficients Dialog Box 4835
51.4.35. Mechanism Dialog Box 4835
51.4.36. Reactor Network Dialog Box 4837
51.4.37. Decoupled Detailed Chemistry Dialog Box 4840
51.4.38. Reacting Channel Model Dialog Box 4841
51.4.39. Reacting Channel 2D Curves Dialog Box 4844
51.4.40. Discrete Phase Model Dialog Box 4845
51.4.41. DEM Collisions Dialog Box 4855
51.4.42. Create Collision Partner Dialog Box 4856
51.4.43. Copy Collision Partner Dialog Box 4856
51.4.44. Rename Collision Partner Dialog Box 4857
51.4.45. DEM Collision Settings Dialog Box 4857
51.4.46. Solidification and Melting Dialog Box 4858
51.4.47. Acoustics Model Dialog Box 4860
51.4.48. Acoustic Sources Dialog Box 4864
51.4.49. Acoustic Receivers Dialog Box 4866
51.4.50. Basic Shapes Dialog Box 4867
51.4.51. Integration Surface Dialog Box 4868
51.4.52. Interior Cell Zone Selection Dialog Box 4869
51.4.53. Structural Model Dialog Box 4869
51.4.54. Eulerian Wall Film Dialog Box 4870
51.4.55. Battery Model Dialog Box 4878
51.4.56. Standalone Echem Model Dialog Box 4880
51.4.57. Potential/Electrochemistry Dialog Box 4881
51.5. Materials Task Page 4883
51.5.1. Create/Edit Materials Dialog Box 4885
51.5.2. Fluent Database Materials Dialog Box 4897
51.5.3. GRANTA MDS Materials Dialog Box 4899
51.5.4. Open Database Dialog Box 4901
51.5.5. User-Defined Database Materials Dialog Box 4901
51.5.6. Copy Case Material Dialog Box 4903
51.5.7. Material Properties Dialog Box 4904
51.5.8. Edit Property Methods Dialog Box 4905
51.5.9. New Material Name Dialog Box 4906
51.5.10. Polynomial Profile Dialog Box 4906
51.5.11. Piecewise-Linear Profile Dialog Box 4907
51.5.12. Piecewise-Polynomial Profile Dialog Box 4909
51.5.13. NASA-9-Coefficient Piecewise-Polynomial Profile Dialog Box 4911
51.5.14. Model Options Dialog Box 4913
51.5.15. Compressible Liquid Dialog Box 4914
51.5.16. User-Defined Functions Dialog Box 4915
51.5.17. Sutherland Law Dialog Box 4916
51.5.18. Power Law Dialog Box 4917
51.5.19. Non-Newtonian Power Law Dialog Box 4919
51.5.20. Carreau Model Dialog Box 4920
51.5.21. Cross Model Dialog Box 4921

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

51.5.22. Herschel-Bulkley Dialog Box 4922
51.5.23. Biaxial Conductivity Dialog Box 4924
51.5.24. Cylindrical Orthotropic Conductivity Dialog Box 4924
51.5.25. Orthotropic Conductivity Dialog Box 4926
51.5.26. Anisotropic Conduction - Principal Components Dialog Box 4927
51.5.27. Anisotropic Conductivity Dialog Box 4928
51.5.28. Species Dialog Box 4929
51.5.29. Reactions Dialog Box 4930
51.5.30. Backward Reaction Parameters Dialog Box 4936
51.5.31. Third-Body Efficiency Dialog Box 4936
51.5.32. Pressure-Dependent Reaction Dialog Box 4937
51.5.33. Coverage-Dependent Reaction Dialog Box 4940
51.5.34. Reference Molar Concentrations Dialog Box 4941
51.5.35. Reaction Mechanisms Dialog Box 4941
51.5.36. Site Parameters Dialog Box 4943
51.5.37. Mass Diffusion Coefficients Dialog Box 4944
51.5.38. Thermal Diffusion Coefficients Dialog Box 4945
51.5.39. UDS Diffusion Coefficients Dialog Box 4946
51.5.40. WSGGM User Specified Dialog Box 4947
51.5.41. Gray-Band Absorption Coefficient Dialog Box 4948
51.5.42. Delta-Eddington Scattering Function Dialog Box 4949
51.5.43. Gray-Band Refractive Index Dialog Box 4949
51.5.44. Single Rate Model Dialog Box 4950
51.5.45. Secondary Rate Model Dialog Box 4951
51.5.46. Two Competing Rates Model Dialog Box 4952
51.5.47. CPD Model Dialog Box 4954
51.5.48. Kinetics/Diffusion-Limited Combustion Model Dialog Box 4955
51.5.49. Intrinsic Combustion Model Dialog Box 4955
51.5.50. Multiple Surface Reactions Dialog Box 4957
51.5.51. Edit Material Dialog Box 4957
51.6. Cell Zone Conditions Task Page 4958
51.6.1. Fluid Dialog Box 4961
51.6.2. Solid Dialog Box 4973
51.6.3. Copy Conditions Dialog Box 4977
51.6.4. Operating Conditions Dialog Box 4978
51.6.5. Select Input Parameter Dialog Box 4980
51.6.6. Profiles Dialog Box 4981
51.6.7. Replicate Profile Dialog Box 4984
51.6.8. Orient Profile Dialog Box 4986
51.6.9. Write Profile Dialog Box 4988
51.6.10. Select Profiles for Writing 4990
51.7. Boundary Conditions Task Page 4991
51.7.1. Axis Dialog Box 4994
51.7.2. Degassing Dialog Box 4995
51.7.3. Exhaust Fan Dialog Box 4995
51.7.4. Fan Dialog Box 5002
51.7.5. Inlet Vent Dialog Box 5005
51.7.6. Intake Fan Dialog Box 5013
51.7.7. Interface Dialog Box 5019
51.7.8. Interior Dialog Box 5020
51.7.9. Mass-Flow Inlet Dialog Box 5020

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

51.7.10. Mass-Flow Outlet Dialog Box 5028
51.7.11. Outflow Dialog Box 5033
51.7.12. Outlet Vent Dialog Box 5035
51.7.13. Overset Dialog Box 5041
51.7.14. Periodic Dialog Box 5041
51.7.15. Porous Jump Dialog Box 5042
51.7.16. Pressure Far-Field Dialog Box 5045
51.7.17. Pressure Inlet Dialog Box 5051
51.7.18. Pressure Outlet Dialog Box 5060
51.7.19. Radiator Dialog Box 5069
51.7.20. RANS/LES Interface Dialog Box 5070
51.7.21. Symmetry Dialog Box 5071
51.7.22. Velocity Inlet Dialog Box 5072
51.7.23. Wall Dialog Box 5085
51.7.24. Periodic Conditions Dialog Box 5104
51.7.25. **Perforated Walls** Dialog Box 5105
51.8. Overset Interfaces Task Page 5108
51.8.1. Create/Edit Overset Interfaces Dialog Box 5110
51.9. Dynamic Mesh Task Page 5111
51.9.1. Mesh Method Settings Dialog Box 5114
51.9.2. Mesh Smoothing Parameters Dialog Box 5118
51.9.3. Advanced Remeshing Settings Dialog Box 5120
51.9.4. Mesh Scale Info Dialog Box 5121
51.9.5. Options Dialog Box 5121
51.9.6. In-Cylinder Output Controls Dialog Box 5125
51.9.7. Six DOF Properties Dialog Box 5127
51.9.8. Periodic Displacement Group Dialog Box 5130
51.9.9. Periodic Displacement Properties Dialog Box 5132
51.9.10. Flow Control Settings Dialog Box 5134
51.9.11. Dynamic Mesh Events Dialog Box 5135
51.9.12. Define Event Dialog Box 5137
51.9.13. Events Preview Dialog Box 5139
51.9.14. Dynamic Mesh Zones Dialog Box 5140
51.9.15. Orientation Calculator Dialog Box 5149
51.9.16. Zone Scale Info Dialog Box 5150
51.9.17. Zone Motion Dialog Box 5150
51.9.18. Mesh Motion Dialog Box 5151
51.9.19. Autosave Case During Mesh Motion Preview Dialog Box 5153
51.10. Reference Values Task Page 5154
51.11. Solution Task Page 5156
51.12. Solution Methods Task Page 5157
51.12.1. Relaxation Options Dialog Box 5162
51.13. Solution Controls Task Page 5163
51.13.1. Equations Dialog Box 5166
51.13.2. Solution Limits Dialog Box 5166
51.13.3. Advanced Solution Controls Dialog Box 5168
51.14. Solution Initialization Task Page 5179
51.14.1. Acoustics Initialization Dialog Box 5182
51.14.2. Patch Dialog Box 5183
51.14.3. Full-Multigrid (FMG) Initialization Dialog Box 5185
51.14.4. Hybrid Initialization Dialog Box 5186

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

51.15. Calculation Activities Task Page 5188
51.15.1. Autosave Dialog Box 5190
51.15.2. Data File Quantities Dialog Box 5192
51.15.3. Automatic Export Dialog Box 5193
51.15.4. Automatic Particle History Data Export Dialog Box 5198
51.15.5. Execute Commands Manager Dialog Box 5200
51.15.6. Execute Commands Dialog Box 5201
51.15.7. Define Macro Dialog Box 5203
51.15.8. Case Modification Manager Dialog Box 5204
51.15.9. Auto Initialization Method Dialog Box 5205
51.15.10. Modify Case Dialog Box 5206
51.16. Run Calculation Task Page 5207
51.16.1. Case Check Dialog Box 5219
51.16.2. Adaptive Time Stepping Dialog Box 5220
51.16.3. Simulation Status Dialog Box 5221
51.16.4. Solution Steering Dialog Box 5222
51.16.5. Acoustic Sources FFT Dialog Box 5224
51.16.6. Acoustic Signals Dialog Box 5230
51.16.7. Sampling Options Dialog Box 5232
51.16.8. Zone-Specific Sampling Options Dialog Box 5233
51.17. Results Task Page 5235
51.18. Graphics and Animations Task Page 5235
51.18.1. Profile Options Dialog Box 5238
51.18.2. Vector Options Dialog Box 5239
51.18.3. Custom Vectors Dialog Box 5240
51.18.4. Vector Definitions Dialog Box 5241
51.18.5. Path Style Attributes Dialog Box 5243
51.18.6. Ribbon Attributes Dialog Box 5243
51.18.7. Particle Filter Attributes Dialog Box 5244
51.18.8. Reporting Variables Dialog Box 5245
51.18.9. Track Style Attributes Dialog Box 5246
51.18.10. Particle Sphere Style Attributes Dialog Box 5247
51.18.11. Particle Vector Style Attributes Dialog Box 5248
51.18.12. Sweep Surface Dialog Box 5250
51.18.13. Create Surface Dialog Box 5251
51.18.14. Animate Dialog Box 5251
51.18.15. Save Picture Dialog Box 5254
51.18.16. Playback Dialog Box 5257
51.18.17. Video Options Dialog Box 5260
51.18.18. Advanced Video Quality Options Dialog Box 5261
51.18.19. Display Options Dialog Box 5262
51.18.20. Scene Description Dialog Box 5265
51.18.21. Display Properties Dialog Box 5267
51.18.22. Transformations Dialog Box 5269
51.18.23. Iso-Value Dialog Box 5270
51.18.24. Pathline Attributes Dialog Box 5271
51.18.25. Bounding Frame Dialog Box 5271
51.18.26. Views Dialog Box 5272
51.18.27. Write Views Dialog Box 5274
51.18.28. Mirror Planes Dialog Box 5275
51.18.29. Periodic Instancing Dialog Box 5276

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

51.18.30. Camera Parameters Dialog Box 5277
51.18.31. Lights Dialog Box 5279
51.18.32. Colormap Dialog Box 5280
51.18.33. Colormap Editor Dialog Box 5284
51.18.34. Annotate Dialog Box 5286
51.19. Plots Task Page 5289
51.19.1. Solution XY Plot Dialog Box 5290
51.19.2. Histogram Dialog Box 5295
51.19.3. Plot Data Sources Dialog Box 5297
51.19.4. Plot Profile Data Dialog Box 5299
51.19.5. Plot Interpolated Data Dialog Box 5300
51.19.6. Fourier Transform Dialog Box 5302
51.19.7. Ansys Sound Analysis Dialog Box 5304
51.19.8. Cumulative Plot Dialog Box 5307
51.19.9. Plot/Modify Input Signal Dialog Box 5309
51.19.10. Axes Dialog Box 5313
51.19.11. Curves Dialog Box 5316
51.20. Reports Task Page 5317
51.20.1. Flux Reports Dialog Box 5320
51.20.2. Force Reports Dialog Box 5322
51.20.3. Projected Surface Areas Dialog Box 5324
51.20.4. Surface Integrals Dialog Box 5325
51.20.5. Volume Integrals Dialog Box 5329
51.20.6. Sample Trajectories Dialog Box 5331
51.20.7. Trajectory Sample Histograms Dialog Box 5333
51.20.8. Particle Summary Dialog Box 5336
51.20.9. Heat Exchanger Report Dialog Box 5337
51.20.10. Parameters Dialog Box 5338
51.20.11. Use Input Parameter in Scheme Procedure Dialog Box 5341
51.20.12. Use Input Parameter for UDF Dialog Box 5342
51.20.13. Rename Dialog Box 5343
51.20.14. Parameter Expression Dialog Box 5343
51.20.15. Save Output Parameter Dialog Box 5345
51.21. Parameters and Customization Task Page 5346
52. Ribbon Reference Guide 5347
52.1. File Ribbon Tab 5347
52.1.1. File/Read/Mesh... 5349
52.1.1.1. Read Mesh Options Dialog Box 5349
52.1.2. File/Read/Case... 5349
52.1.3. File/Read/Data... 5350
52.1.4. File/Read/Case & Data... 5350
52.1.5. File/Read/Case Settings Only... 5351
52.1.6. File/Read/Case/Mesh Info... 5351
52.1.7. File/Read/PDF... 5352
52.1.8. File/Read/ISAT Table... 5352
52.1.9. File/Read/DTRM Rays... 5352
52.1.10. File/Read/View Factors... 5353
52.1.11. File/Read/Profile... 5353
52.1.12. File/Read/Scheme... 5353
52.1.13. File/Read/Journal... 5353
52.1.14. File/Write/Case... 5353

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

52.1.15. File/Write/Data... 5354
52.1.16. File/Write/Case & Data... 5354
52.1.17. File/Write/PDF... 5354
52.1.18. File/Write/ISAT Table... 5354
52.1.19. File/Write/Flamelet... 5354
52.1.20. File/Write/Profile... 5354
52.1.21. File/Write/Autosave... 5355
52.1.22. File/Write/Boundary Mesh... 5355
52.1.23. File/Write/Start Journal... 5355
52.1.24. File/Write/Stop Journal 5355
52.1.25. File/Write/Start Transcript... 5355
52.1.26. File/Write/Stop Transcript 5355
52.1.27. File/Import/ABAQUS/Input File... 5355
52.1.28. File/Import/ABAQUS/Filbin File... 5355
52.1.29. File/Import/ABAQUS/ODB File... 5356
52.1.30. File/Import/CFX/Definition File... 5356
52.1.31. File/Import/CFX/Result File... 5356
52.1.32. File/Import/CGNS/Mesh... 5356
52.1.33. File/Import/CGNS/Data... 5356
52.1.34. File/Import/CGNS/Mesh & Data... 5356
52.1.35. File/Import/EnSight... 5356
52.1.36. File/Import/FIDAP... 5356
52.1.37. File/Import/GAMBIT... 5356
52.1.38. File/Import/HYPERMESH ASCII... 5357
52.1.39. File/Import/LSTC/Input File... 5357
52.1.40. File/Import/LSTC/State File... 5357
52.1.41. File/Import/Marc POST... 5357
52.1.42. File/Import/NASTRAN/Bulkdata File... 5357
52.1.43. File/Import/NASTRAN/Op2 File... 5357
52.1.44. File/Import/PLOT3D/Grid File... 5357
52.1.45. File/Import/PLOT3D/Result File... 5357
52.1.46. File/Import/Tecplot... 5357
52.1.47. File/Import/Partition/Metis... 5358
52.1.48. File/Import/Partition/Metis Zone... 5358
52.1.49. File/Import/CHEMKIN Mechanism... 5358
52.1.49.1. Import CHEMKIN Format Mechanism Dialog Box 5358
52.1.50. File/Import/FMU... 5360
52.1.51. File/Export/Solution Data... 5360
52.1.51.1. Export Dialog Box 5361
52.1.52. File/Export/Particle History Data... 5366
52.1.52.1. Export Particle History Data Dialog Box 5366
52.1.53. File/Export/During Calculation/Solution Data... 5368
52.1.54. File/Export/During Calculation/Particle History Data... 5368
52.1.55. File/Export to CFD-Post... 5368
52.1.55.1. Export to CFD-Post Dialog Box 5368
52.1.56. File/Table File Manager... 5369
52.1.57. File/Solution Files... 5369
52.1.57.1. Solution Files Dialog Box 5369
52.1.58. File/Interpolate... 5370
52.1.58.1. Interpolate Data Dialog Box 5371
52.1.59. File/FSI Mapping/Volume... 5372

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

52.1.59.1. Volume FSI Mapping Dialog Box 5372
52.1.60. File/FSI Mapping/Surface... 5375
52.1.60.1. Surface FSI Mapping Dialog Box 5375
52.1.61. File/Save Picture... 5379
52.1.62. File/Data File Quantities... 5379
52.1.63. File/Batch Options... 5379
52.1.63.1. Batch Options Dialog Box 5379
52.1.64. File/Idle Timeout... 5380
52.1.64.1. Set Idle Timeout Dialog Box 5380
52.1.65. File/Exit 5381
52.2. Dialog Boxes Available from the Ribbon 5381
52.2.1. 1D Simulation Library Dialog Box 5385
52.2.2. Activate Cell Zones Dialog Box 5386
52.2.3. Adaption Criteria Settings Dialog Box 5386
52.2.4. Adjacency Dialog Box 5390
52.2.5. Advanced Options Dialog Box 5393
52.2.6. Aerodamping (Influence Coefficient Method) Report Definition Dialog Box 5394
52.2.7. Aerodamping (Travelling Wave Method) Report Definition Dialog Box 5395
52.2.8. Animation Definition Dialog Box 5398
52.2.9. Application About to Exit Dialog Box 5401
52.2.10. Auto Partition Mesh Dialog Box 5402
52.2.11. Automatic Mesh Adaption Dialog Box 5403
52.2.12. Cell Count Report Definition Dialog Box 5404
52.2.13. Cell Register Display Options Dialog Box 5407
52.2.14. Compiled UDFs Dialog Box 5408
52.2.15. Conduction Layers Dialog Box 5409
52.2.16. Conduction Manager Dialog Box 5411
52.2.17. Contours Dialog Box 5412
52.2.18. Convergence Conditions Dialog Box 5416
52.2.19. Create Mesh Interfaces 5418
52.2.20. Create/Edit Mesh Interfaces Dialog Box 5419
52.2.21. Create/Edit Turbo Interfaces Dialog Box 5423
52.2.22. Curvilinear Coordinate System Dialog Box 5425
52.2.23. Custom Field Function Calculator Dialog Box 5428
52.2.24. Custom Laws Dialog Box 5429
52.2.25. Dashboard Definition Dialog Box 5430
52.2.26. Deactivate Cell Zones Dialog Box 5432
52.2.27. Define Control Points Dialog Box 5432
52.2.28. Delete Cell Zones Dialog Box 5434
52.2.29. Display Options - Adaption Dialog Box 5435
52.2.30. Display States Dialog Box 5437
52.2.31. DPM Report Definition Dialog Box 5441
52.2.32. DPM Source Report Definition Dialog Box 5444
52.2.33. Drag Report Definition Dialog Box 5446
52.2.34. DTRM Graphics Dialog Box 5448
52.2.35. DTRM Rays Dialog Box 5450
52.2.36. Edit Gap Region Dialog Box 5451
52.2.37. Edit Mesh Interfaces Dialog Box 5454
52.2.38. Edit Report File Dialog Box 5456
52.2.39. Edit Report Plot Dialog Box 5458
52.2.40. Execute on Demand Dialog Box 5460

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

52.2.41. Expression Dialog Box 5461
52.2.42. Expression Editor Dialog Box 5465
52.2.43. Expression Manager Dialog Box 5468
52.2.44. Expression Report Definition Dialog Box 5469
52.2.45. Expression Volume Dialog Box 5472
52.2.46. Face Count Report Definition Dialog Box 5473
52.2.47. Field Function Definitions Dialog Box 5476
52.2.48. Flux Report Definition Dialog Box 5477
52.2.49. Force Report Definition Dialog Box 5481
52.2.50. Fuse Face Zones Dialog Box 5483
52.2.51. Gap Model Dialog Box 5484
52.2.52. General Adaption Controls Dialog Box 5486
52.2.53. Generalized/Modal Force Report Definition Dialog Box 5488
52.2.54. Geometry Based Adaption Controls Dialog Box 5490
52.2.55. Geometry Based Adaption Dialog Box 5491
52.2.56. Import Particle Data Dialog Box 5492
52.2.57. Imprint Surface Dialog Box 5494
52.2.58. Improve Mesh Dialog Box 5495
52.2.59. Injections Dialog Box 5496
52.2.60. Input Summary Dialog Box 5497
52.2.61. Interface Creation Options Dialog Box 5498
52.2.62. Interpreted UDFs Dialog Box 5499
52.2.63. Iso-Clip Dialog Box 5500
52.2.64. Iso-Surface Dialog Box 5501
52.2.65. Lift Report Definition Dialog Box 5503
52.2.66. Line Integral Convolutions Dialog Box 5506
52.2.67. Line/Rake Surface Dialog Box 5514
52.2.68. Manual Mesh Adaption Dialog Box 5515
52.2.69. Manage Adaption Criteria Dialog Box 5516
52.2.70. Manage Geometry-Based Adaption Dialog Box 5518
52.2.71. Manage Sponge Layers Dialog Box 5519
52.2.72. Mapped Interface Options Dialog Box 5520
52.2.73. Material Editor Dialog Box 5521
52.2.74. Measure Dialog Box 5523
52.2.75. Merge Zones Dialog Box 5525
52.2.76. Mesh Interfaces Dialog Box 5525
52.2.77. Mesh Morpher/Optimizer Dialog Box 5528
52.2.78. Mixing Planes Dialog Box 5538
52.2.79. Moment Report Definition Dialog Box 5541
52.2.80. Motion Settings Dialog Box 5543
52.2.81. Multi Edit Dialog Box 5549
52.2.82. New Report File Dialog Box 5550
52.2.83. New Report Plot Dialog Box 5552
52.2.84. Objective Function Definition Dialog Box 5553
52.2.85. Optimization History Monitor Dialog Box 5556
52.2.86. Parallel Connectivity Dialog Box 5557
52.2.87. Parameter Bounds Dialog Box 5557
52.2.88. Particle Tracks Dialog Box 5558
52.2.89. Partition Surface Dialog Box 5565
52.2.90. Partitioning and Load Balancing Dialog Box 5567
52.2.91. Pathlines Dialog Box 5572

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

52.2.92. PCB Model Dialog Box 5579
52.2.93. Plane Surface Dialog Box 5580
52.2.94. Point Surface Dialog Box 5582
52.2.95. Quadric Surface Dialog Box 5584
52.2.96. Raytracing Display Dialog Box 5586
52.2.97. Reduced Order Model Dialog Box 5590
52.2.98. Reference Frame Dialog Box 5592
52.2.99. Replace Cell Zone Dialog Box 5593
52.2.100. Report Definitions Dialog Box 5594
52.2.101. Report File Definitions Dialog Box 5596
52.2.102. Report Plot Definitions Dialog Box 5597
52.2.103. Residual Monitors Dialog Box 5599
52.2.104. Rotate Mesh Dialog Box 5602
52.2.105. S2S Information Dialog Box 5603
52.2.106. Select UDM Zones Dialog Box 5604
52.2.107. Select Window Dialog Box 5605
52.2.108. Separate Cell Zones Dialog Box 5605
52.2.109. Separate Face Zones Dialog Box 5606
52.2.110. Set Injection Properties Dialog Box 5608
52.2.111. Set Multiple Injection Properties Dialog Box 5621
52.2.112. Sponge Layer Dialog Box 5622
52.2.113. Structural Point Surface Dialog Box 5623
52.2.114. Surface Meshes Dialog Box 5624
52.2.115. Surface Rendering Properties Dialog Box 5625
52.2.116. Surface Report Definition Dialog Box 5626
52.2.117. Surfaces Dialog Box 5629
52.2.118. Thread Control Dialog Box 5631
52.2.119. Time Averaged Explicit Thermal Coupling Dialog Box 5632
52.2.120. Transform Surface Dialog Box 5633
52.2.121. Translate Mesh Dialog Box 5634
52.2.122. Turbo 2D Contours Dialog Box 5635
52.2.123. Turbo Averaged Contours Dialog Box 5637
52.2.124. Turbo Averaged XY Plot Dialog Box 5639
52.2.125. Turbo Options Dialog Box 5641
52.2.126. Turbo Report Dialog Box 5641
52.2.127. Turbo Topology Dialog Box 5644
52.2.128. UDF Library Manager Dialog Box 5646
52.2.129. User-Defined Fan Model Dialog Box 5647
52.2.130. User-Defined Function Hooks Dialog Box 5648
52.2.131. User-Defined Memory Dialog Box 5652
52.2.132. User Defined Report Definition Dialog Box 5652
52.2.133. User-Defined Scalars Dialog Box 5654
52.2.134. Vectors Dialog Box 5655
52.2.135. Volume Report Definition Dialog Box 5659
52.2.136. Warning Dialog Box 5662
52.2.137. Zone Surface Dialog Box 5663
52.2.138. Zone Type Color and Material Assignment Dialog Box 5664
A. Ansys Fluent Model Compatibility 5667
B. Ansys Fluent File Formats 5671
B.1. CFF File Format 5671
B.1.1. CFF Case File Layout 5671

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

B.1.2. CFF Solution Data File Layout 5673
B.1.3. Variable Sized Data on Collected Element / Node Sets 5674
B.2. Legacy Case and Data File Formats 5675
B.2.1. Guidelines 5675
B.2.2. Formatting Conventions in Binary and Formatted Files 5676
B.2.3. Grid Sections 5676
B.2.3.1. Comment 5676
B.2.3.2. Header 5677
B.2.3.3. Dimensions 5677
B.2.3.4. Nodes 5677
B.2.3.5. Periodic Shadow Faces 5678
B.2.3.6. Cells 5679
B.2.3.7. Faces 5681
B.2.3.8. Face Tree 5683
B.2.3.9. Cell Tree 5683
B.2.3.10. Interface Face Parents 5684
B.2.3.11. Example Files 5684
B.2.3.11.1. Example 1 5684
B.2.3.11.2. Example 2 5685
B.2.3.11.3. Example 3 5686
B.2.4. Other (Non-Grid) Legacy Case Sections 5688
B.2.4.1. Zone 5688
B.2.4.2. Partitions 5690
B.2.5. Data Sections 5691
B.2.5.1. Grid Size 5691
B.2.5.2. Data Field 5691
B.2.5.3. Residuals 5692
B.3. Mesh Morpher/Optimizer File Formats 5693
B.4. Conduction Settings File Format 5694
B.5. 3D Fan Curve File Format 5694
C. Controlling CHEMKIN-CFD Solver Parameters Using Text Commands 5697
C.1. Advanced Parameters Used in the Steady-State Solution Algorithm 5701
C.2. Setting Up Monitor Cells for the Ansys CHEMKIN-CFD Chemistry Solver 5702
C.3. Diagnostic Files and Error Messages 5702
C.4. Error Messages Printed in the Ansys Fluent Graphical User Interface 5704
C.5. Diagnostic Messages in the `KINetics-log.txt` File 5706
D. Nomenclature 5709
Bibliography 5713

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

List of Figures

1.1. Ansys Fluent Architecture 142
4.1. The General Options Tab of Fluent Launcher 184
4.2. The Parallel Settings Tab of Fluent Launcher 186
4.3. The Remote Tab of Fluent Launcher 187
4.4. The Scheduler Tab of Fluent Launcher (Windows Version) 188
4.5. The Environment Tab of Fluent Launcher 189
4.6. The Batch Options Dialog Box 206
8. Cell Types 212
1.1. The User Interface Components 222
1.2. Clipping Plane and Clipping Plane Tool 225
1.3. The Watertight Geometry Workflow 230
1.4. The Outline View Tree 230
1.5. Filtering Field 231
1.6. Model Level Menu 231
1.7. CAD Assemblies Tree 232
1.8. CAD Assemblies Menu 232
1.9. CAD Component/Body Level Menu 233
1.10. CAD Label Level Menu 233
1.11. Global Object Level Menu 234
1.12. Individual Object Level Menu 235
1.13. Face Zone Labels Level Menu 236
1.14. Individual Label Menu 237
1.15. Unreferenced Zones Menu 237
1.16. The Graphics Effects Tools 242
1.17. The Mesh Display Tools 243
1.18. The Copy Tools 244
1.19. The Object Selection/Display Tools 244
1.20. Preferences Dialog Box 248
1.21. Splitting the Face of a Coiled Geometry 271
1.22. Imported Coiled Geometry 271
2.1. The Dockable Workflow Editor 286
2.2. Setting the Default Workflow Editor View 286
2.3. Messages and Progress Bar During Meshing Tasks 287
3.1. Example of a Self-Intersection: Double Faces Appear When Share Topology is Not Enabled 310
3.2. Example of a Self-Intersection: Local Mesh Size is Significantly Larger Than the Pipe Thickness 311
3.3. Showing Marked Gaps 324
3.4. Applying Share Topology to Marked Gaps 325
3.5. Incomplete Gap Marking (Maximum Gap Distance = 0.12) 325
3.6. Complete Gap Marking (Maximum Gap Distance = 0.15) 325
3.7. Excessive Gap Marking Around a Washer (Maximum Gap Distance = 1.2) 326
3.8. Proper Gap Marking Around a Washer (Maximum Gap Distance = 0.8) 326
3.9. Example of a Single Surface Cap with Multiple Faces 327
3.10. Example of a Single Surface Cap with Multiple Faces 328
3.11. Example of an Annular Cap Type 328
3.12. Example of a Problematic Tilted Annular Opening 328
3.13. Example of a Self-Intersection: Additional Cap Intersects With Other Surfaces 329
3.14. The Add Thin Volume Meshing Controls Task Page 333
3.15. Comparison of Standard and Aggressive Starstepping 334
3.16. Target and Source Zones for Electromagnetic Air-Gap Model 335

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.17. Stack of Thin and Thick Plates 336
3.18. Thin Plates 337
3.19. Thin Volume Meshing Controls for Parallel Zones 338
3.20. Key Source Faces 339
3.21. Stacked Plates with Co-planar 340
3.22. Examples of Fill Types 346
3.23. Sample Geometries 349
3.24. Example of a Fluid and a Solid Volume Mesh 356
3.25. Example of a Copying and Rotating a Volume Mesh (Periodic) 362
3.26. Example of Copying and Translating a Volume Mesh (Periodic) 362
3.27. Example of a Geometry With Part Selected for Adding a Mesh Pattern 368
3.28. Example of a Geometry With a Preview of the Mesh Pattern 369
3.29. Example of a Geometry With a Mesh Pattern 370
3.30. An Example of a Custom Singular Pattern 374
3.31. An Example of a Custom Dual Pattern 375
3.32. An Example of a Refinement Region Around a Car 384
3.33. An Example of Multiple Refinement Regions Around a Car 387
3.34. An Example of Multiple Refinement Regions Around a Vehicle 387
4.1. Example of Using a Component Name Prefix 394
4.2. A CAD File Loaded into the CAD Model Tree 397
4.3. Example of Using a Component Name Prefix 398
4.4. Selected Portions of a CAD File Loaded into the Meshing Model Tree 401
4.5. CAD Properties of a Selected Per-Part Meshing Model Object 405
4.6. Properties of a Selected Custom Meshing Model Object 408
4.7. Transformation Example: Truck Bed Translation Along the Global X Axis for 5 Meters 413
4.8. Transformation Example: Truck Bed Rotation of -45 Degrees About Local X Axis 414
4.9. Transformation Example: Truck Bed Rotation of 45 Degrees With a Global Pivot in X About the Z Axis 415
4.10. Transformation Example: Truck Bed Scaled by Half for Global X, Y, and Z 416
4.11. Transformation Example: Truck Bed Mirrored About the Global XY Plane 417
4.12. Faceting Example 422
4.13. Example of a Single Surface Cap with Multiple Faces 427
4.14. Example of a Single Surface Cap with Multiple Faces 427
4.15. Example of an Annular Cap Type 428
4.16. Example of a Problematic Tilted Annular Opening 428
4.17. Example of a Self-Intersection: Additional Cap Intersects With Other Surfaces 429
4.18. An Example of an External Flow Boundary 430
4.19. An External Flow Bounding Box 431
4.20. An External Flow Boundary Around a Car 431
4.21. An Example of a Refinement Region Around a Car 432
4.22. An Example of Multiple Refinement Regions Around a Car 435
4.23. An Example of Multiple Refinement Regions Around a Vehicle 435
4.24. An Example of Addressing Sharp Angles - CAD Geometry 442
4.25. An Example of Addressing Sharp Angles - Final Mesh 442
4.26. Sharp Angles With and Without the Zones Separated By Face 443
4.27. An Example of a Porous Region 446
4.28. An Example of a Porous Region: Fins and Tubes in a Heat Exchanger 446
4.29. The Points of a Porous Region 448
4.30. The Buffer Size for a Porous Region 449
4.31. Identifying a Fluid Region in the Wake Behind a Car 450
4.32. Identifying Potential Leakages Within a Car 453
4.33. Collar and Component Meshes for a Propeller and Hub Geometry 472

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.34. A Collar Mesh for a Propeller and Hub Geometry 473
4.35. A Component Mesh for a Propeller and Hub Geometry 474
4.36. Example of Surface Mesh Quality Contours 481
4.37. Example of Orphan Cells in an Offset Mesh 486
6.1. 2–3 and 3–2 Swap Configurations 513
6.2. 4–4 Swap Configuration 514
6.3. Sliver Formation 514
6.4. Movement of Boundary Nodes 520
6.5. Cavity Around a Mirror Remeshed With Tetrahedra 523
6.6. Cavity Around a Mirror Remeshed With Hexcore Mesh 526
6.7. Copying and Translating a Cell Zone 527
6.8. The Selective Mesh Check Dialog 532
6.9. Mesh Display (A) With Shrink Factor = 0 (B) With Shrink Factor = 0.01 539
6.10. Camera Definition 544
6.11. Graphics Display with Bounding Frame 546
6.12. The Navigation Branch of Preferences 549
6.13. The Mesh Diagnostic Tools in the Ribbon 555
6.14. The Mesh Check Diagnostic Tools in the Ribbon 555
6.15. The Mesh Quality Diagnostic Tools in the Ribbon 556
6.16. The Diagnostics Tools Dialog Box 558
6.17. Example of Marked Mesh Cells (Multi Face) 560
6.18. The Diagnostics Scope Dialog Box 562
6.19. Ideal and Skewed Triangles and Quadrilaterals 567
6.20. Vectors Used to Compute Ortho Skew/Inverse Orthogonal Quality for a Cell 572
6.21. Vectors Used to Compute Ortho Skew Quality for a Face 573
6.22. Calculating the Fluent Aspect Ratio for a Unit Cube 574
7.1. Use of Curvature Sizing 586
7.2. Use of Proximity Sizing 587
7.3. Use of the Face Boundary Option for Face Proximity 588
7.4. Use of the Ignore Orientation Option for Face Proximity 589
7.5. Use of Meshed Sizing 590
7.6. Use of Soft Sizing 592
7.7. Use of Body of Influence Sizing 593
7.8. Contours of Size 598
7.9. Display of Mesh Size Based on Size Field 598
7.10. Mesh With Different Cell Zone Types 602
7.11. Use of the Object Priority for Overlapping Objects 603
7.12. Creating Objects—Example 603
7.13. Objects Defined Using the Subtract Method 604
7.14. Using Material Points—Example 613
7.15. Example—CutCell Mesh, Only Objects Defined 614
7.16. Example—Fluid Surface Extracted From Geometry Objects and Material Point 615
7.17. Closing a Radial Gap 620
7.18. Creating a Surface Using an Edge 622
7.19. Creating a Surface Using Nodes 623
7.20. Overlapping Surfaces 630
7.21. Connected Surfaces After Join 630
7.22. Intersecting Surfaces 631
7.23. Connected Surfaces After Intersect 631
7.24. Orientation of Normals in Gap 636
7.25. Removing Gaps Between Objects—Face-Face Option 637

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.26. Removing Gaps Between Objects—Face-Edge Option 637
7.27. Gap and Thickness Configurations 638
7.28. Removing Thickness in Objects 639
7.29. Mesh Objects to be Connected 640
7.30. Mesh Object Created by Sewing 640
7.31. Free Nodes 653
7.32. Example of a Thin Wall 654
7.33. Intersection of Boundary Zones 655
7.34. Intersection (A) Without and (B) With the Refine Option 655
7.35. Partially Overlapping Faces 656
7.36. Joining of Overlapping Faces 656
7.37. Remeshing of Joined Faces 657
7.38. Nearest Point Projection for Stitching 658
7.39. Surfaces Before Stitch 658
7.40. Surfaces After Stitch 659
7.41. Refining a Triangular Boundary Face 670
7.42. Boundary Mesh (A) Before and (B) After Refining Based on Proximity 671
7.43. Surface Mesh - Feature Angle = 60 673
7.44. Edge Zone for Face Zone Approach and Fixed Angle = 65 674
7.45. Edge Zones for Face Zone Approach and Fixed Angle = 55 (or Adaptive Angle) 674
7.46. Edge Zone for Face Seed Approach and Fixed Angle = 65 675
7.47. Edge Zones for Face Seed Approach and Fixed Angle = 55 (or Adaptive Angle) 675
7.48. Mesh (A) Before and (B) After Using the Faceted Stitch Option 682
7.49. Triangulating a Boundary Zone 683
7.50. Face Separation Based on Region 685
7.51. Face Separation Based on Cell Neighbor 685
7.52. Planar Points Method 692
7.53. Cylinder Defined by 3 Arc Nodes, Radial Gap, and Axial Delta 694
7.54. Cylinder Defined by 3 Arc Nodes and a Height Node 695
7.55. Cylinder Defined by Axial Points and Radii 696
7.56. Loop Selection Toolbar 703
7.57. Schematic Representation of Wrapping Process 706
7.58. Individual Object Loop 709
7.59. Collective Object Loops 710
7.60. Overlaid Geometry Clipped with the Pan Plane 713
7.61. Leak Detection Using the Pan Regions Dialog Box 713
7.62. Wrapping Individual Objects 716
7.63. Multiple Solids 716
7.64. Single Solid Surface 717
7.65. Extracting the Flow Volume 717
7.66. Possible Mesh Cell Shapes 721
7.67. Mesh with Prisms in a Boundary Layer Region 722
7.68. Surface Mesh Containing Only Tetrahedra 723
7.69. Surface Mesh 724
7.70. Hexcore Mesh 725
7.71. CutCell Mesh 726
7.72. Rapid Octree Mesh with Projection Boundary Treatment Method 726
7.73. Extending an Existing Tetrahedral Mesh Using Prisms 727
7.74. Example of a Non-Conformal Interface 728
7.75. Mesh Generated Using Isolated Nodes to Concentrate Cells 730
7.76. Mesh Generated Without Using Isolated Nodes 730

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.77. Pyramid Cell—Transition from a Hexahedron to a Tetrahedron 735
7.78. Pyramid Cells Intersecting Each Other and Boundary 738
7.79. Fixed Intersecting Pyramid Cells Using Triangular Faces 739
7.80. Creating the Heat Exchanger Mesh 741
7.81. The Thread Control Dialog Box 749
7.82. Prism Shapes 750
7.83. Layer Heights Computed Using the Four Growth Methods 754
7.84. Different Growth Parameters on Adjacent Zones 758
7.85. Different Growth Parameters on Nonadjacent Zones—Using the Auto Mesh Option 759
7.86. Prism Growth on a Dangling Wall 760
7.87. Ignoring Invalid Normals 761
7.88. Collision of Prism Layers 762
7.89. Prism Layers Shrunk to Avoid Collision 762
7.90. Ignoring Areas of Proximity 763
7.91. Uniform Offset Distance Method 765
7.92. Minimum-Height Offset Distance Method 766
7.93. Last Ratio Method 767
7.94. Effect of Offset Smoothing 768
7.95. Uniform Direction Vector for a Straight-Sided Prism Region 768
7.96. Normal Direction Vectors for a Curved Prism Region 769
7.97. Normal Direction Vectors Before Smoothing 769
7.98. Normal Direction Vectors After Smoothing 770
7.99. Effect of Adjacent Zone Angle 771
7.100. Symmetry Zone and Car Wall Before Prism Generation 772
7.101. Symmetry Zone and Car Wall After Prism Generation Without Retriangulation 772
7.102. Symmetry Zone and Car Wall After Prism Generation and Retriangulation 773
7.103. Use of Multiple Scoped Prism Controls 777
7.104. Stair Stepped Prism Layers in Sharp Corner 778
7.105. Local Refinement Region for the Tetrahedral Mesh 792
7.106. Hexcore Mesh Using (A) Buffer Layers = 1 (B) Buffer Layers = 2 799
7.107. Hexcore Mesh Using (A) Peel Layers = 0 (B) Peel Layers = 2 800
7.108. Hexcore to the Far-Field Boundary 801
7.109. Hexcore to Boundaries 803
7.110. Local Refinement Region for the Hexcore Mesh 803
7.111. Schematic Representation of the Cartesian Grid Refinement Using Size Functions 812
7.112. Mesh After Refinement 812
7.113. Mesh After Projection 813
7.114. Cells Separated After Decomposition 814
7.115. CutCell Mesh After Boundary Recovery 815
7.116. Mesh Generated for Geometry Having Zero-Thickness Baffles 818
7.117. Recovering Overlapping Surfaces 819
7.118. Resolving Thin Regions 820
7.119. Rezoning Multiply Connected Faces 821
7.120. Generating Prisms for the CutCell Mesh 824
7.121. Prism Growth Limitations—Volumes Sharing an Edge 824
7.122. Prism Growth Limitations—Volumes Sharing an Edge 825
7.123. Prism Growth Limitations—Volumes Sharing the Prism Base 825
7.124. The Rapid Octree Dialog Box 829
7.125. The Geometry Group Box 830
7.126. Selecting Active Material Points 832
7.127. Bounding Box Dialog Box 834

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.128. Bounding Box Surrounding Geometry 835
7.129. The Boundary Treatment Group Box 836
7.130. Mesh Generated by Cartesian Snapping 836
7.131. Mesh Generated by Boundary Projection 837
7.132. The Boundary Treatment Group Box 838
7.133. The Boundary Treatment Group Box 838
7.134. Improving the Geometry Resolution 839
7.135. The Mesh Parameters Group Box 840
7.136. Surface Mesh with Two Levels of Feature Angle Refinement and 10° Threshold 841
7.137. Mesh Coarsening Controls 842
7.138. The Rapid Octree Refinement Regions Dialog Box 843
7.139. The Surface Sizing Dialog Box 844
7.140. Multiple Prism Layers on Exterior Boundary Zones 846
7.141. The Boundary Layer Mesh Definition Dialog Box 847
7.142. Defeaturing Considerations 850
1. Quadrilateral Mesh 873
2. Quadrilateral Mesh with Periodic Boundaries 874
3. Quadrilateral Mesh with Hanging Nodes 875
1.1. The GUI Components 902
1.2. The Fluent Ribbon 903
1.3. The Fluent Outline View 904
1.4. Hover-Over Highlight 909
1.5. Graphics Window Context Menu: Single-Selection 910
1.6. Graphics Window Context Menu: Multiple-Selection 910
1.7. Displaying Two Graphics Windows 912
1.8. The View Ribbon Tab 913
1.9. The Standard Toolbar 914
1.10. The Graphics Toolbars 914
1.11. Mesh Display 914
1.12. The Pointer Tools 915
1.13. The View Tools 917
1.14. The Visibility Tools 918
1.15. The Copy Tools 918
1.16. The Object Selection/Display Tools 919
1.17. The Graphics Effects Tools 919
1.18. Additional Display Options 920
1.19. Messages Dialog Box with Tabs for Console, Warnings, and Errors 922
1.20. Location of Messages Dialog Box Access with Console in Message Window 922
1.21. The Select File Dialog Box for Windows 931
1.22. The Select File Dialog Box for Linux Platforms 932
1.23. Another Version of the Select File Dialog Box for Linux Platforms 933
1.24. Quick Editor for a Velocity Inlet 934
1.25. Fluent in Dark Theme 935
1.26. Component Dock 936
1.27. Preferences Dialog Box 937
1.28. Fluent GUI in Japanese 939
1.29. Fluent GUI in Korean 939
1.30. Fluent GUI in Chinese 940
1.31. Set Idle Timeout Dialog Box 941
1.32. Application About to Exit Dialog Box 942
1.33. Learning and Support Ribbon Tab 943

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.1. The Select File Dialog Box 949
3.2. The Autosave Dialog Box 959
3.3. The Select File Dialog Box for Reading Case Settings Only 963
3.4. The Write Profile Dialog Box 965
3.5. The Select Profiles for Writing Dialog Box 966
3.6. The Write Profile Dialog Box 968
3.7. The Table File Manager Dialog Box 970
3.8. Multiple Selection of Journal Files 975
3.9. The Import Menu 978
3.10. The Export Dialog Box 986
3.11. The Export Particle History Data Dialog Box 1000
3.12. The Calculation Activities Task Page 1002
3.13. The Automatic Export Dialog Box 1003
3.14. The Automatic Particle History Data Export Dialog Box 1005
3.15. The Export to CFD-Post Dialog Box 1007
3.16. The Solution Files Dialog Box 1009
3.17. The Interpolate Data Dialog Box 1011
3.18. The Volume FSI Mapping Dialog Box for Cell Zone Data 1015
3.19. The Surface FSI Mapping Dialog Box for Face Zone Data 1016
3.20. The Save Picture Dialog Box 1020
3.21. The Data File Quantities Dialog Box 1027
3.22. **Import FMU File** Dialog Box 1030
3.23. **Import FMU File** Dialog Box 1031
4.1. The Set Units Dialog Box 1034
4.2. The Define Unit Dialog Box 1036
5.1. Example Profile Expression 1048
5.2. The Expression Editor Dialog Box 1052
5.3. Example Expression for Water Density 1054
5.4. The Expression Dialog Box 1055
5.5. Velocity Magnitude Defined as "expr1", An Expression Input Parameter 1058
5.6. Plotting an Expression 1061
5.7. Expressions... Postprocessing Field 1062
5.8. Expression Manager Dialog Box 1064
5.9. Contours of Velocity - Parabolic Inflow 1065
5.10. Parabolic Inflow Velocity Over Time 1069
5.11. Pipe Geometry Colored by ID (Heated Wall is Green) 1072
5.12. Contours of Temperature (outlet is closest) 1072
5.13. Plots of Inlet Temperature, Average Outlet Temperature, and Maximum Outlet Temperature 1073
6.1. Cell Types 1130
6.2. Structured Quadrilateral Mesh for an Airfoil 1131
6.3. Unstructured Quadrilateral Mesh 1132
6.4. Multiblock Structured Quadrilateral Mesh 1132
6.5. O-Type Structured Quadrilateral Mesh 1132
6.6. Parachute Modeled With Zero-Thickness Wall 1133
6.7. C-Type Structured Quadrilateral Mesh 1133
6.8. 3D Multiblock Structured Mesh 1134
6.9. Unstructured Triangular Mesh for an Airfoil 1134
6.10. Unstructured Tetrahedral Mesh 1135
6.11. Hybrid Triangular/Quadrilateral Mesh with Hanging Nodes 1135
6.12. Non-Conformal Hybrid Mesh for a Rotor-Stator Geometry 1136
6.13. Polyhedral Mesh 1136

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.14. Face and Node Numbering for Triangular Cells 1137
6.15. Face and Node Numbering for Quadrilateral Cells 1138
6.16. Face and Node Numbering for Tetrahedral Cells 1139
6.17. Face and Node Numbering for Wedge Cells 1140
6.18. Face and Node Numbering for Pyramidal Cells 1141
6.19. Face and Node Numbering for Hex Cells 1142
6.20. An Example of a Polyhedral Cell 1143
6.21. Setup of Axisymmetric Geometries with the x Axis as the Centerline 1146
6.22. The Vectors Used to Compute Orthogonality 1147
6.23. Calculating the Aspect Ratio for a Unit Cube 1148
6.24. The Surface Meshes Dialog Box 1160
6.25. The Reference Frame Dialog Box 1161
6.26. The Motion Tab of the Reference Frame Dialog Box 1162
6.27. Point Surface Creation on Local Reference Frame 1163
6.28. Profile Definition on Local Reference Frame 1164
6.29. The Curvilinear Coordinate System Dialog Box 1165
6.30. Completely Overlapping Mesh Interface Intersection 1170
6.31. Partially Overlapping Mesh Interface Intersection 1171
6.32. Two-Dimensional Non-Conformal Mesh Interface 1171
6.33. Non-Conformal Periodic Boundary Condition (Translational) 1173
6.34. Non-Conformal Periodic Boundary Condition (Rotational) 1174
6.35. Translational Non-Conformal Interface with the Periodic Repeats Option 1175
6.36. Rotational Non-Conformal Interface with the Periodic Repeats Option 1176
6.37. Non-Conformal Coupled Wall Interfaces 1177
6.38. Matching Non-Conformal Wall Interfaces 1178
6.39. Non-Conformal Mapped Interface with a Gap and Penetration 1179
6.40. A Circular Non-Conformal Interface 1184
6.41. Pairing Boundary Zones Using the Right-Click Menu 1187
6.42. The Mesh Interfaces Dialog Box for the One-to-One Method 1189
6.43. The Mesh Interfaces Dialog Box for the Many-to-Many Method 1190
6.44. The Interface Creation Options Dialog Box 1191
6.45. Managing One-to-One Interfaces from the Outline View Tree 1192
6.46. Managing One-to-One Interfaces from the Mesh Interfaces Dialog Box 1193
6.47. The Edit Mesh Interface Dialog Box 1193
6.48. Creating a Coupled Wall at a One-to-One Mesh Interface 1194
6.49. The Edit Mesh Interfaces Dialog Box 1195
6.50. Contours of Interface Overlap Fraction 1198
6.51. Displaying the Intersected Zone 1199
6.52. The Create/Edit Mesh Interfaces Dialog Box 1202
6.53. The Create Mesh Interfaces Dialog Box 1204
6.54. Transferring Displacements 1205
6.55. Projecting Nodes 1206
6.56. Overset Component and Background Mesh 1207
6.57. Solve Cells After Initialization 1208
6.58. Valid Overset Meshes with Components in Close Proximity 1209
6.59. Second Component Modifying Existing Body 1210
6.60. Existing Body Modification After Initialization 1210
6.61. Multiple Components Bridged by Collars Meshes 1211
6.62. Multiple Components with Collar Meshes Initialized 1211
6.63. Adding Fluid to a Region Using Cut Control 1212
6.64. Cut Controlled Region After Initialization 1212

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.65. Overset Component and Background Meshes Before Hole Cutting 1213
6.66. Overset Component and Background Meshes After Hole Cutting 1213
6.67. Overset Component and Background Meshes After Overlap Minimization 1214
6.68. Overset Mesh Before Hole Cutting 1215
6.69. Overset Mesh After Minimization Based on Boundary Distance 1215
6.70. Valid Overlap 1216
6.71. Invalid Overlap Creating Orphans 1218
6.72. Overset Gap Adaption 1220
6.73. Prismatic Adaption Examples 1222
6.74. Create/Edit Overset Interfaces Dialog Box 1228
6.75. Contours of Overset Cell Type: Background Mesh 1232
6.76. Contours of Overset Cell Type: Component Mesh 1232
6.77. The Gap Model Dialog Box 1237
6.78. The Edit Gap Region Dialog Box 1239
6.79. Controlling Gap Shape by Excluding Zones 1240
6.80. The Advanced Options Dialog Box 1241
6.81. The Mesh Display Dialog Box for a Gap Model Simulation 1243
6.82. Contours and Mesh with a Flow-Blocking Gap 1244
6.83. The Solution Methods Task Page 1248
6.84. Connection of Edge Centroids with Face Centroids 1254
6.85. A Polyhedral Cell 1255
6.86. A Converted Polyhedral Cell with Preserved Hexahedral Cell Shape 1255
6.87. Treatment of Wedge Boundary Layers 1256
6.88. The Original Tetrahedral Mesh 1256
6.89. The Converted Polyhedral Mesh 1257
6.90. The Merge Zones Dialog Box 1261
6.91. The Separate Face Zones Dialog Box 1264
6.92. Cell Zone Separation Based on Region 1265
6.93. The Separate Cell Zones Dialog Box 1266
6.94. The Fuse Face Zones Dialog Box 1267
6.95. The Create Periodic Dialog Box 1271
6.96. The Replace Cell Zone Dialog Box 1276
6.97. The Delete Cell Zones Dialog Box 1277
6.98. The Deactivate Cell Zones Dialog Box 1278
6.99. The Activate Cell Zones Dialog Box 1279
6.100. The Select File Dialog Box 1280
6.101. The Adjacency Dialog Box 1282
6.102. The Scale Mesh Dialog Box 1287
6.103. The Translate Mesh Dialog Box 1289
6.104. The Rotate Mesh Dialog Box 1290
6.105. The Improve Mesh Dialog Box 1292
6.106. Result of Smoothing Operator on Node Position 1293
6.107. Initial Mesh Before Smoothing Operation 1294
6.108. Mesh Smoothing Causing Mesh-Line Crossing 1294
6.109. Examples of Cell Configurations in the Circle Test 1296
6.110. Swapped Faces to Satisfy the Delaunay Circle Test 1296
6.111. 3D Face Swapping 1297
7.1. The Boundary Conditions Task Page 1301
7.2. The Copy Conditions Dialog Box 1305
7.3. Export Mesh Zones Dialog Box 1307
7.4. The Parameters Dialog Box 1309

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.5. The New Input Parameter... Selection 1311
7.6. The Parameter Expression Dialog Box 1312
7.7. Use Input Parameter in Scheme Procedure Dialog Box 1314
7.8. Use Input Parameter for UDF Dialog Box 1315
7.9. Selecting Multiple Boundaries for Display in the Graphics Window 1318
7.10. Example Operations for Multiple Selected Surfaces in the Graphics Window 1319
7.11. The Fluid Dialog Box 1321
7.12. Rotation Specified in the Absolute Reference Frame 1324
7.13. Rotation Specified Relative to a Moving Zone 1325
7.14. The Solid Dialog Box 1327
7.15. Single Rotating Solid Zone 1330
7.16. Rotating solid zone separated from another fluid or solid zone separated by a surface of revolution. 1330
7.17. Multiple rotating solid zones having the same material and motion specifications, separated by mesh interfaces or coupled walls. 1331
7.18. Two solids in contact where one is stationary and the other is moving with translational motion. The translational motion of the moving solid should be described in the Solid Motion tab. 1331
7.19. Two solids in contact where one is stationary and the other is rotating. The rotational motion of the moving solid should be described in the Solid Motion tab. Both solids may also have rotation about the y-axis described in the Frame Motion tab. 1332
7.20. Rotating solid with boundaries which are not tangential to the motion. 1332
7.21. Two solids in contact with some squish. At the contact, the rotational motion has some normal component, so the solver will not achieve global energy conservation. However, the temperature field might still be acceptable for engineering purposes. 1333
7.22. The Fluid Dialog Box for a Porous Zone 1344
7.23. Cone Half Angle 1348
7.24. The Heat Transfer Settings Group Box of the Fluid Dialog Box 1356
7.25. The Fluid Dialog Box: Relative Permeability 1360
7.26. The Table Input Dialog Box for Relative Permeability 1362
7.27. Skjaeveland Correlation Behavior 1366
7.28. The Table Input Dialog Box for Capillary Pressure 1368
7.29. The Fluid Dialog Box for a 3D Fan Zone 1375
7.30. The Inflection Point Ratio of a Pitched Blade Turbine 1376
7.31. Fixing Values for the Flow in a Stirred Tank 1379
7.32. Defining a Source for a Tiny Inlet 1383
7.33. The Operating Conditions Dialog Box 1389
7.34. Specifying Inlet Turbulence for Scale Resolving Simulation 1401
7.35. The Pressure Inlet Dialog Box 1403
7.36. Cylindrical Velocity Components in 3D, 2D, and Axisymmetric Domains 1407
7.37. The Velocity Inlet Dialog Box 1412
7.38. The Mass-Flow Inlet Dialog Box 1419
7.39. The Mass-Flow Outlet Dialog Box 1426
7.40. The Inlet Vent Dialog Box 1431
7.41. The Intake Fan Dialog Box 1433
7.42. The Pressure Outlet Dialog Box 1436
7.43. Pressures at the Face of a Pressure Outlet Boundary 1441
7.44. The Pressure Outlet Dialog Box with the Target Mass Flow Rate Option Enabled 1443
7.45. The Pressure Far-Field Dialog Box 1445
7.46. Choice of the Outflow Boundary Condition Location 1449
7.47. The Outflow Dialog Box 1450
7.48. The Outlet Vent Dialog Box 1453

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.49. The Exhaust Fan Dialog Box 1456
7.50. The Wall Dialog Box for **High Roughness (Icing)** Models 1460
7.51. The Wall Dialog Box for a Moving Wall 1462
7.52. The Wall Dialog Box for Specified Shear 1465
7.53. The Wall Dialog Box for the Specularity Coefficient 1466
7.54. The Wall Dialog Box for Marangoni Stress 1467
7.55. The Wall Dialog Box (Thermal Tab) 1469
7.56. A Thin Wall 1471
7.57. Uncoupled Thin Walls 1473
7.58. 2D Interface with Penetration and Gaps 1477
7.59. The Wall Dialog Box for Species Boundary Condition Input 1478
7.60. Modeling Perforated Walls 1484
7.61. Uniform and Discrete Approaches 1484
7.62. The Perforated Walls Dialog Box 1488
7.63. The Injection Holes Dialog Box 1490
7.64. The Static Setup Dialog Box (Uniform Injection) 1492
7.65. The Dynamic Setup Dialog Box (Uniform Injection) 1493
7.66. Injection Surface and Tangential Angles 1494
7.67. Use of Symmetry to Model One Quarter of a 3D Duct 1499
7.68. Use of Symmetry to Model One Quarter of a Circular Cross-Section 1500
7.69. Inappropriate Use of Symmetry 1500
7.70. Use of Periodic Boundaries to Define Swirling Flow in a Cylindrical Vessel 1501
7.71. Example of Translational Periodicity - Physical Domain 1502
7.72. Example of Translational Periodicity - Modeled Domain 1502
7.73. The Periodic Dialog Box 1503
7.74. Use of an Axis Boundary as the Centerline in an Axisymmetric Geometry 1504
7.75. The Fan Dialog Box 1506
7.76. Polynomial Profile Dialog Box for Pressure Jump Definition 1507
7.77. A Fan Located In a 2D Duct 1509
7.78. The Radiator Dialog Box 1514
7.79. Polynomial Profile Dialog Box for Loss Coefficient Definition 1515
7.80. A Simple Duct with a Radiator 1516
7.81. The Porous Jump Dialog Box 1520
7.82. The Multi Edit Wall Settings Dialog Box 1522
7.83. Mesh and Prescribed Boundary Conditions in a 3D Axial Flow Problem 1529
7.84. Mesh and Prescribed Boundary Conditions in a 3D Radial Flow Problem 1530
7.85. Mesh and Prescribed Boundary Conditions in a 2D Case 1531
7.86. Prescribed Inlet Angles 1534
7.87. The Local Orthogonal Coordinate System onto which Euler Equations are Recasted for the General NRBC Method 1540
7.88. Waves Leaving and Entering a Boundary Face on Inflow and Outflow Boundaries. The Wave Amplitudes are Shown with the Associated Eigenvalues for a Subsonic Flow Condition 1541
7.89. The Pressure Outlet Dialog Box With the Non-Reflecting Boundary Enabled 1543
7.90. The Impedance Data Fitting Dialog Box 1549
7.91. The Inlet, Fan, and Pressure Outlet Zones for a Circular Fan Operating in a Cylindrical Domain 1558
7.92. The User-Defined Fan Model Dialog Box 1559
7.93. The Fan Dialog Box 1562
7.94. Transverse Velocities at the Site of the Fan 1563
7.95. Static Pressure Jump Across the Fan 1564
7.96. The Profiles Dialog Box 1572
7.97. Preview of Profile as a Surface (Gray) with Mesh Overlayed (Pastel Green) 1573

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.98. Preview of Profile as a Point Cloud (Black) with Mesh Overlayed (Pastel Green) 1574
7.99. Preview of Profile Field as a Point Cloud 1575
7.100. Example of Using Profiles as Boundary Conditions 1576
7.101. The Orient Profile Dialog Box 1577
7.102. Scalar Profile at the Outlet 1579
7.103. Problem Specification 1581
7.104. The Replicate Profile Dialog Box 1582
7.105. The Profiles Dialog Box with Complex Mode Shape Profile 1585
7.106. The Replicate Profile Dialog Box with Complex Pair 1586
7.107. The 1D Simulation Library Dialog Box 1589
7.108. Using GT-POWER Data for Boundary Conditions 1590
7.109. Cell Zone Conditions for Torque-Speed Coupling with GT-POWER 1591
7.110. The 1D Simulation Library Dialog Box with WAVE Selected 1593
7.111. Using WAVE Data for Boundary Conditions 1594
8.1. The Materials Task Page 1600
8.2. The Materials Branch in the **Outline View** 1601
8.3. Fluent Database Materials Dialog Box 1603
8.4. GRANTA MDS Materials Dialog Box 1605
8.5. Open Database Dialog Box 1608
8.6. User-Defined Database Materials Dialog Box 1609
8.7. New Material Name Dialog Box 1610
8.8. Copy Case Material Dialog Box 1611
8.9. User-Defined Database Materials Dialog Box: Blank 1613
8.10. Material Properties Dialog Box: Blank 1614
8.11. Edit Property Methods Dialog Box 1615
8.12. The Polynomial Profile Dialog Box 1617
8.13. The Piecewise-Linear Profile Dialog Box 1619
8.14. Piecewise-Linear Definition of Viscosity as a Function of Temperature 1620
8.15. The Piecewise-Polynomial Profile Dialog Box 1621
8.16. The NASA-9-Coefficient Piecewise-Polynomial Profile Dialog Box 1623
8.17. Compressible Liquid Materials Setting 1628
8.18. Compressible Liquid Density Settings Panel 1629
8.19. Variation of Viscosity with Shear Rate According to the Carreau Model 1642
8.20. The Carreau Model Dialog Box 1642
8.21. Variation of Shear Stress with Shear Rate According to the Herschel-Bulkley Model 1644
8.22. The Create/Edit Materials Dialog Box 1645
8.23. The Anisotropic Conductivity Dialog Box 1649
8.24. The Biaxial Conductivity Dialog Box 1650
8.25. The Orthotropic Conductivity Dialog Box 1651
8.26. The Cylindrical Orthotropic Conductivity Dialog Box 1653
8.27. Unaligned Principal Axes 1654
8.28. The Anisotropic Conductivity - Principal Components Dialog Box 1655
8.29. The UDS Diffusion Coefficients Dialog Box 1657
8.30. The Anisotropic UDS Diffusivity Dialog Box 1659
8.31. The Orthotropic UDS Diffusivity Dialog Box 1660
8.32. The Cylindrical Orthotropic UDS Diffusivity Dialog Box 1661
8.33. The UDS Diffusion Coefficients Dialog Box 1662
8.34. The Mass Diffusion Coefficients Dialog Box for Dilute Approximation 1671
8.35. The Mass Diffusion Coefficients Dialog Box for the Multicomponent Method 1672
8.36. Anisotropic Species Diffusion Matrix 1674
8.37. The Thermal Diffusion Coefficients Dialog Box 1675

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.38. Connected and Disconnected Cell Zones 1680
8.39. Typical PT Diagram of a Pure Material 1682
8.40. Typical PV Diagram of a Pure Material 1683
8.41. The Cubic Equation of State Model for a Real-Gas Fluid 1693
8.42. The Cubic Equation of State Model for a Real-Gas Mixture 1694
8.43. The Operating Conditions for a Real Gas State 1696
8.44. The PV Diagram for the Cubic Equation of State Real Gas Model 1697
8.45. The Create/Edit Materials Dialog Box with NIST Model Enabled 1707
8.46. The NIST Fluid Data Dialog Box 1708
8.47. The Create/Edit Materials Dialog Box with NIST Mixture Model Enabled 1709
8.48. Operating Conditions dialog Box with Liquid State Enabled 1710
8.49. The NIST Fluid Data Dialog Box With Lookup Table Enabled 1711
8.50. The NIST Fluid Data Dialog Box For Fixed Composition 1712
8.51. The Table Settings Dialog Box 1735
8.52. The RGP Table Saturation Data Dialog Box 1736
9.1. The User-Defined Scalars Dialog Box 1741
9.2. The Fluid Dialog Box with Inputs for Source Terms for a User-Defined Scalar 1742
9.3. The User Scalar Sources Dialog Box 1743
9.4. The Materials Dialog Box with Input for Diffusivity for UDS Equations 1744
9.5. The User-Defined Scalars Dialog Box for a Multiphase Flow 1746
9.6. Example of Periodic Flow in a 2D Heat Exchanger Geometry 1748
9.7. The Periodic Conditions Dialog Box 1750
9.8. The Periodic Dialog Box 1752
9.9. Periodic Pressure Field Predicted for Flow in a 2D Heat Exchanger Geometry 1753
9.10. Rotating Flow in a Cavity 1756
9.11. Swirling Flow in a Gas Burner 1757
9.12. Flow in a Converging-Diverging Nozzle 1760
10.1. Single Component (Blower Wheel Blade Passage) 1770
10.2. Multiple Component (Blower Wheel and Casing) 1771
10.3. Single Blade Model with Rotationally Periodic Boundaries 1772
10.4. The Fluid Dialog Box Displaying Frame Motion Inputs 1774
10.5. Geometry with the Rotating Impeller 1775
10.6. Absolute Velocity Vectors 1778
10.7. Relative Velocity Vectors 1778
10.8. The Solution Initialization Task Page for Moving Reference Frames 1785
11.1. The Manage Motion Definitions Dialog Box 1787
11.2. The Motion Definition Dialog Box 1788
12.1. The Manage Auxiliary Geometry Definitions Dialog Box 1791
12.2. The Auxiliary Geometry Definition Dialog Box 1792
12.3. The Compute from a Face Zone Dialog Box 1793
12.4. The Surfaces Meshes Dialog Box 1794
12.5. The Display Options - Auxiliary Geometry Definitions Dialog Box 1795
13.1. Two Passing Trains in a Tunnel 1799
13.2. Rotor-Stator Interaction (Stationary Guide Vanes with Rotating Blades) 1799
13.3. Blower 1800
13.4. Initial Position of the Meshes 1801
13.5. Rotor Mesh Slides with Respect to the Stator 1801
13.6. 2D Linear Mesh Interface 1802
13.7. 2D Circular-Arc Mesh Interface 1802
13.8. 3D Conical Mesh Interface 1803
13.9. 3D Planar-Sector Mesh Interface 1803

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.10. The Mesh Interfaces Dialog Box 1807
13.11. Lift Coefficient Plot for a Time-Periodic Solution 1810
13.12. Contours of Static Pressure for the Rotor-Stator Example 1811
13.13. The Dynamic Mesh Task Page 1815
13.14. The Smoothing Tab of the Mesh Method Settings Dialog Box 1817
13.15. The Mesh Smoothing Parameters Dialog Box 1817
13.16. The Initial Mesh 1820
13.17. Valid Mesh After 45 Degree Rotation Using Diffusion-Based Smoothing 1820
13.18. Degenerated Mesh After 40 Degree Rotation Using Spring-Based Smoothing 1821
13.19. Effect of Diffusion Parameter of 0 on Interior Node Motion 1822
13.20. Effect of Diffusion Parameter of 1 on Interior Node Motion 1823
13.21. Spring-Based Smoothing on Interior Nodes: Start 1825
13.22. Spring-Based Smoothing on Interior Nodes: End 1826
13.23. Interior Nodes Extend Beyond Boundary (Spring Constant Factor = 1) 1827
13.24. Interior Nodes Remain Within Boundary (Spring Constant Factor = 0) 1827
13.25. An Example of Where Local Smoothing is Applied 1832
13.26. The Undeformed Mesh 1834
13.27. The Deformed Mesh 1835
13.28. Zooming into the Undeformed Compliant Strip 1836
13.29. Zooming into the Deformed Compliant Strip with Boundary Layer Smoothing Applied 1837
13.30. Dynamic Layering 1838
13.31. Results of Splitting Layer with the Height-Based Option 1838
13.32. Results of Splitting Layer with the Ratio-Based Option 1839
13.33. The Layering Tab in the Mesh Method Settings Dialog Box 1840
13.34. Use of Sliding Interfaces to Transition Between Adjacent Cell Zones and the Dynamic Layering Cell Zone 1841
13.35. The Remeshing Tab in the Mesh Method Settings Dialog Box for Methods-Based Remeshing 1843
13.36. The Remeshing Tab in the Mesh Method Settings Dialog Box for Unified Remeshing 1844
13.37. Mesh at the End of a Dynamic Mesh Simulation Without Sizing Functions 1847
13.38. Mesh at the End of a Dynamic Mesh Simulation With Sizing Functions 1847
13.39. Sizing Function Determination at Background Mesh Vertex I 1848
13.40. Interpolating the Value of the Sizing Function 1849
13.41. Determining the Normalized Distance 1850
13.42. Expanding Cylinder Before Region Face Remeshing 1853
13.43. Expanding Cylinder After Region Face Remeshing 1854
13.44. Volume Decomposition for Prism Layers 1855
13.45. Volume Decomposition for the Base of the Prism Layers 1856
13.46. Close-Up of 2.5D Extruded Flow Meter Pump Geometry Before Remeshing and Laplacian Smoothing 1857
13.47. Close-Up of 2.5D Extruded Flow Meter Pump Geometry After Remeshing and Laplacian Smoothing . 1858
13.48. The Remeshing Tab for the 2.5D Model 1859
13.49. 2.5D Extruded Gear Pump Geometry 1860
13.50. The Advanced Remeshing Settings Dialog Box 1861
13.51. Cross Section of a 3D Corner 1868
13.52. The In-Cylinder Tab of the Options Dialog Box 1869
13.53. Determining the Sign of the Piston Pin Offset 1870
13.54. The In-Cylinder Output Controls Dialog Box 1871
13.55. Sample Output File Showing Various Quantities 1874
13.56. A 2D In-Cylinder Geometry 1874
13.57. Mesh Topology Showing the Various Mesh Regions 1875
13.58. Mesh Associated With the Chosen Topology 1876

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.59. The Use of Sliding Interfaces to Connect the Exhaust Valve Layering Zone to the Remeshing Zone ... 1877
13.60. Mesh Sequence 1 ... 1878
13.61. Mesh Sequence 2 ... 1878
13.62. Mesh Sequence 3 ... 1879
13.63. Mesh Sequence 4 ... 1879
13.64. Mesh Sequence 5 ... 1880
13.65. Mesh Sequence 6 ... 1880
13.66. Piston Position (m) as a Function of Crank Angle (deg) ... 1881
13.67. Intake and Exhaust Valve Lift (m) as a Function of Crank Angle (deg) ... 1882
13.68. Definition of Valve Zone Attributes (Intake Valve) ... 1883
13.69. The Six DOF Tab of the Options Dialog Box ... 1884
13.70. The Six DOF Properties Dialog Box ... 1886
13.71. A Check Valve with One DOF Translation ... 1887
13.72. The Implicit Update Tab of the Options Dialog Box ... 1889
13.73. The Contact Detection Tab of the Options Dialog Box ... 1891
13.74. The Flow Control Settings Dialog Box with Contact Zones ... 1892
13.75. The Flow Control Settings Dialog Box with Contact Marks ... 1893
13.76. The Dynamic Mesh Events Dialog Box ... 1895
13.77. The Define Event Dialog Box ... 1896
13.78. The Events Preview Dialog Box for In-Cylinder Flows ... 1897
13.79. The Define Event Dialog Box for the Creating Sliding Interface Option ... 1899
13.80. Boundary Zone Before Insertion ... 1901
13.81. Boundary Zone After Insertion ... 1901
13.82. Interior Zone Before Insertion ... 1902
13.83. Interior Zone After Insertion ... 1902
13.84. The Dynamic Mesh Zones Dialog Box for a Stationary Zone ... 1905
13.85. The Dynamic Mesh Zones Dialog Box for a Rigid Body Motion ... 1906
13.86. Orientation Calculator Dialog Box ... 1908
13.87. The Dynamic Mesh Zones Dialog Box for a Rigid Body Motion Using the Six DOF Solver ... 1909
13.88. The Dynamic Mesh Zones Dialog Box for a Deforming Motion with Cell Zone Options ... 1911
13.89. The Dynamic Mesh Zones Dialog Box for an Intrinsic FSI Zone ... 1917
13.90. Solid Body Rotation Coordinates ... 1921
13.91. The Zone Motion Dialog Box ... 1922
13.92. The Mesh Motion Dialog Box ... 1923
13.93. The Mesh Motion Dialog Box for Steady-State Dynamic Meshes ... 1926
13.94. Initial Object Position ... 1927
13.95. The Mesh Motion Dialog Box After 40 Updates ... 1928
13.96. Final Object Position After 40 Executions ... 1928
14.1. Examples of Schematics for Various Multi-Row Component Types ... 1933
14.2. Setting the Default Workflow Editor View ... 1954
14.3. Frozen Gust variations ... 1955
14.4. Ensemble Blade Rows with a GTI Interface ... 1962
14.5. Create/Edit Turbo Interfaces Dialog Box ... 1963
14.6. Create/Edit Turbo Interfaces Dialog Box with Adjacent Zone Filtering ... 1963
14.7. Create/Edit Turbo Interfaces Dialog Box for a Specified Interface ... 1964
14.8. The Mixing Planes Dialog Box ... 1969
14.9. Passage Numbering from Profile Expansion ... 1975
14.10. Periodic Displacement Example for Real Only Mode Shape ... 1985
14.11. Periodic Displacement Example for a Complex Mode Shape ... 1986
14.12. Aerodamping (Travelling Wave Method) Report Definition Dialog Box ... 1996
14.13. Inlet Disturbance Configuration ... 2004

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.14. Converting Periodic Boundary to Phase Lag Periodic Interface 2007
14.15. Creating Phase Lag Periodic Interface from Boundaries 2008
14.16. Create Periodic Dialog Box for a Phase-lag Interface 2009
14.17. The Multiphase Model Dialog Box with the Wet Steam Model Selected (Pressure-Based) 2013
14.18. The Multiphase Model Dialog Box with the Wet Steam Model Selected (Density-Based) 2013
14.19. Schematic Diagrams of a Cooled Turbine Blade 2014
14.20. Hole Shape Dependency from Hole Direction 2018
14.21. Creation of Overlap and Non-overlap Surfaces 2022
14.22. Holes Outside the Parent Cell Zone Rotated to Overlap with Mesh 2023
14.23. The Turbomachinery-Specific Numerics Option in the Solution Methods Task Page 2028
14.24. The Turbo Topology Dialog Box 2030
14.25. Turbomachinery Boundary Types 2033
14.26. Iso-Surface Dialog Box for Turbo Surfaces 2034
14.27. Options for Spectral Content in the Turbomachinery Ribbon 2037
14.28. Fourier Coefficients Available for Graphical Postprocessing 2040
14.29. The Turbo Report Dialog Box 2041
14.30. Pump or Compressor 2045
14.31. Turbine 2047
14.32. The Turbo Averaged Contours Dialog Box 2050
14.33. Turbo Averaged Filled Contours of Static Pressure 2051
14.34. The Turbo 2D Contours Dialog Box 2052
14.35. The Turbo Averaged XY Plot Dialog Box 2054
14.36. The Turbo Options Dialog Box 2055
14.37. The Turbo Performance Dialog Box 2056
14.38. Expression Dialog Box with Efficiency Calculation 2057
14.39. Turbo Report Dialog Box 2058
15.1. Velocity Profiles for Axi-symmetric Diffuser Flow (Case CS0 – Driver). Impact of Variation of C_{SEP} 2067
15.2. Impact of Changes in C_{MIX} on Free Mixing Layer. Left: Velocity Profiles, Right: Turbulence Kinetic Energy Profiles 2067
15.3. Impact of Changes in C_{NW} on Backward Facing Jet with Heat Transfer. Left: Wall Shear Stress Coefficient, C_f, Right: Wall Heat Transfer Coefficient, S_t 2068
15.4. Impact of Changes in C_{JET} on Free Jet Flows. Left: Plane Jet, Right: Round Jet 2068
15.5. Illustration of SST-URANS vs. SST-SAS Models 2074
15.6. The Viscous Model Dialog Box 2082
15.7. The Viscous Model Dialog Box Displaying the Spalart-Allmaras Production 2085
15.8. The Viscous Model Dialog Box Displaying the Standard k-ε Model 2086
15.9. The Viscous Model Dialog Box Displaying the RNG k-ε Model 2088
15.10. The Viscous Model Dialog Box Displaying the Standard k-ω Model 2090
15.11. The Viscous Model Dialog Box with GEKO Options for the Full Model 2093
15.12. The Viscous Model Dialog Box Displaying the BSL k-ω Model 2095
15.13. The Viscous Model Dialog Box Displaying the SST k-ω Model 2097
15.14. The Viscous Model Dialog Box With WJ-BSL-EARSM Enabled 2099
15.15. The Viscous Model Dialog Box for the Transition k-kl-ω Model 2100
15.16. The Viscous Model Dialog Box for the Transition SST Model 2102
15.17. Transition Option enabled in Combination with the SST k-ω Model 2104
15.18. The Viscous Model Dialog Box Displaying the Reynolds Stress Model Options 2106
15.19. The Viscous Model Dialog Box Displaying the Stress-Omega Model Options 2108
15.20. Scale-Adaptive Simulation (SAS) in Combination with the SST Turbulence Model 2110
15.21. Scale-Adaptive Simulation (SAS) in Combination with the Transition SST Model 2111
15.22. The Viscous Model Dialog Box Displaying Options for DES with the Spalart-Allmaras Model 2113

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15.23. The Viscous Model Dialog Box Displaying Options for DES with the Realizable k-ε Model 2115
15.24. The Viscous Model Dialog Box Displaying Options for DES with the SST k-ω Model 2117
15.25. The Viscous Model Dialog Box Displaying Options for DES with the BSL k-ω Model 2119
15.26. The Viscous Model Dialog Box Displaying Options for DES with the Transition SST Model 2121
15.27. The Viscous Model Dialog Box Displaying the Large Eddy Simulation Model Options 2123
15.28. Specifying an ELES Zone in the Fluid Dialog Box 2126
15.29. Specifying the RANS/LES Interface 2127
15.30. The RANS/LES Interface Dialog Box 2128
15.31. SST Model with the **Buoyancy Effects: Only Turbulence Production** Option Enabled 2130
15.32. The Viscous Model Dialog Box with Corner Flow Correction option enabled 2133
15.33. The Viscous Model Dialog Box with the SBES Options for ω-based RANS models 2142
15.34. Specifying Inlet Boundary Conditions for the Reynolds Stresses 2146
15.35. The Sampling Options Dialog Box 2161
15.36. The Zone-Specific Sampling Options Dialog Box 2162
16.1. Enabling the Energy Equation 2164
16.2. The Conduction Manager Dialog Box 2166
16.3. Typical Counterflow Heat Exchanger Involving Heat Transfer Between Two Separated Fluid Streams .. 2167
16.4. The Run Calculation Task Page Showing Solid Time Stepping 2171
16.5. The Loosely Coupled Conjugate Heat Transfer Option for a Transient Flow 2174
16.6. The Loosely Coupled Conjugate Heat Transfer Option for a Steady Flow 2175
16.7. The Time Averaged Explicit Thermal Coupling Dialog Box 2178
16.8. A Boundary Wall with Shell Conduction 2183
16.9. A Two-Sided Wall with Shell Conduction 2183
16.10. The Conduction Manager Dialog Box 2185
16.11. Conduction Layers Dialog Box 2186
16.12. Shell Surface Names for a Boundary Wall 2188
16.13. Shell Surface Names for a Two-Sided Wall 2189
16.14. Create/Edit Materials Dialog Box with Orthotropic Conductivity 2191
16.15. Solid Cell Zone Dialog Box with Defined CCS 2192
16.16. The Radiation Model Dialog Box (DO Model) 2193
16.17. The Radiation Model Dialog Box (Non-Gray P-1 Model) 2195
16.18. The DTRM Rays Dialog Box 2196
16.19. The Radiation Model Dialog Box (S2S Model) 2199
16.20. The View Factors and Clustering Dialog Box 2201
16.21. The Wall Dialog Box 2203
16.22. The Participating Boundary Zones Dialog Box 2207
16.23. The Thread Control Dialog Box 2211
16.24. The Radiation Model Dialog Box (Non-Gray DO Model) 2213
16.25. The Radiation Model Dialog Box with DO/Energy Coupling Enabled 2214
16.26. The Radiation Model Dialog Box (MC) 2216
16.27. Treatment of Radiation at Transparent Boundary 2219
16.28. The Wall Dialog Box Showing Radiation Conditions for an Opaque Wall 2221
16.29. The Wall Dialog Box Showing External Emissivity and External Radiation Temperature Thermal Conditions 2222
16.30. The Wall Dialog Box for a Semi-Transparent Wall Boundary 2223
16.31. Refraction of Irradiation Entering Computational Domain 2224
16.32. The Wall Dialog Box for an Interior Semi-Transparent Wall 2226
16.33. The Wall Dialog Box for an Opaque Wall with MC Model (Gray) 2227
16.34. The Wall Dialog Box for an Opaque Wall with MC Model (Boundary Source) 2228
16.35. The Wall Dialog Box for an Opaque Wall with MC Model (Polar Distribution with Expression) 2229
16.36. Polar Coordinate Plot of Radiation Intensity for Polar Distribution Function 2230

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16.37. The Wall Dialog Box for an Opaque Wall with MC Model (Polar Distribution with Table) 2231
16.38. The Wall Dialog Box for an Opaque Wall with MC Model (Non-gray) 2232
16.39. The Wall Dialog Box Showing External Emissivity and External Radiation Temperature Thermal Conditions 2233
16.40. The Wall Dialog Box for a Semi-transparent Wall with MC Model 2234
16.41. The Solid Dialog Box 2235
16.42. The Radiation Model Dialog Box (DTRM) 2238
16.43. Iteration Parameters for Steady Simulations 2240
16.44. Iteration Parameters for Transient Simulations 2241
16.45. The DTRM Graphics Dialog Box 2245
16.46. Ray Display 2246
16.47. The S2S Information Dialog Box 2247
16.48. The Radiation Model Dialog Box 2256
16.49. The Radiation Model Dialog Box (With Solar Load Model Solar Ray Tracing Option) 2257
16.50. The Radiation Model Dialog Box (with Solar Load Model Solar Irradiation Option) 2258
16.51. The Solar Calculator Dialog Box 2260
16.52. The Velocity Inlet Dialog Box 2262
16.53. The Wall Dialog Box 2263
16.54. The Wall Dialog Box 2264
16.55. The Porous Jump Dialog Box 2265
16.56. The Wall Dialog Box Radiation tab with Solar Irradiation 2267
16.57. The Contours Dialog Box 2272
16.58. The Execute Commands Dialog Box 2273
16.59. Temperature Field in a 2D Heat Exchanger Geometry With Fixed Temperature Boundary Conditions 2279
16.60. An Example of a Four-Pass Heat Exchanger 2280
16.61. Heat Exchanger Modeling Options 2282
16.62. The Heat Exchanger Model Dialog Box 2283
16.63. The Dual Cell Heat Exchanger Dialog Box 2284
16.64. The Set Dual Cell Heat Exchanger Dialog Box 2285
16.65. The Heat Rejection Tab 2286
16.66. The Performance Data Tab 2287
16.67. The Heat Transfer Data Table Dialog Box 2288
16.68. The Frontal Area Tab 2289
16.69. The Coupling Tab 2290
16.70. An Example of a Four-Pass Heat Exchanger 2290
16.71. The Heat Exchanger Model Dialog Box 2294
16.72. The Ungrouped Macro Heat Exchanger Dialog Box Displaying the Model Data Tab 2295
16.73. The Heat Transfer Data Table Dialog Box for the NTU Model 2296
16.74. The Ungrouped Macro Heat Exchanger Dialog Box Displaying the Geometry Tab 2297
16.75. The Ungrouped Macro Heat Exchanger Dialog Box Displaying the Auxiliary Fluid Tab 2298
16.76. 1x4x3 Macros 2301
16.77. Mesh Display With Macros 2302
16.78. The Core Porosity Model Dialog Box 2304
16.79. The Macro Heat Exchanger Group Dialog Box 2306
16.80. The Heat Transfer Data Table Dialog Box for the NTU Model 2307
16.81. The Macro Heat Exchanger Group Dialog Box - Geometry Tab 2308
16.82. The Macro Heat Exchanger Group Dialog Box - Auxiliary Fluid Tab 2309
16.83. The Macro Heat Exchanger Group Dialog Box - Supplementary Auxiliary Fluid Stream Tab 2310
16.84. The Heat Exchanger Report Dialog Box for Reporting Computed Heat Rejection 2314
16.85. The Heat Exchanger Report Dialog Box for Reporting the Inlet Temperature 2315

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16.86. The Heat Exchanger Report Dialog Box for Reporting Mass Flow Rate 2316
16.87. The Heat Exchanger Report Dialog Box for Reporting Specific Heat 2317
16.88. The Volume Report Definition Dialog Box 2318
16.89. The PCB Model Dialog Box 2319
16.90. The PCB Model Dialog Box for ECAD File 2321
16.91. Material Properties for PCB Model Material 2324
16.92. Orthotropic Conductivity for PCB Model Material 2325
16.93. The PCB Model Dialog Box for Board Configuration File 2327
16.94. Material Properties for PCB Model Material 2328
16.95. Orthotropic Conductivity for PCB Model Material 2329
17.1. **Create/Edit Materials** Dialog Box with Gupta Curve Fit Properties ..... ?
17.2. Gupta Curve Fit Conductivity Dialog Box ..... ?
17.3. The Energy Dialog Box ..... ?
17.4. The Create/Edit Materials Dialog Box ..... ?
17.5. The Blottner Curve Fit Dialog Box ..... ?
17.6. The Multiple Vibrational Modes Dialog Box ..... ?
17.7. Species Model Dialog Box with Settings for Two-Temperature Model ..... ?
17.8. Reactions Dialog Box with Settings for Two-Temperature Model ..... ?
17.9. Create/Edit Materials Dialog Box for Mixture ..... ?
17.10. The Collision Cross Section Dialog Box for air-5species-park93 Mixture ..... ?
17.11. The Fluid Dialog Box ..... ?
17.12. Pressure Far-Field Dialog Box ..... ?
17.13. Pressure Far-Field Dialog Box with Non-Equilibrium Boundary Option ..... ?
17.14. Two-sided Wall Dialog Box with Coupled Option ..... ?
17.15. The Solution Methods Task Page ..... ?
17.16. The Solution Controls Task Page ..... ?
17.17. The Equations Dialog Box ..... ?
17.18. The Residual Monitors Dialog Box ..... ?
17.19. The Wall Dialog Box for Partial Slip Shear Condition ..... ?
17.20. The Wall Dialog Box for Temperature Jump Thermal Condition ..... ?
17.21. Illustration of Partially Catalytic Wall ..... ?
17.22. The Species Model Dialog Box with Air Mixture ..... ?
17.23. The Wall Dialog Box with Partially Catalytic Wall ..... ?
17.24. The Wall Dialog Box with Partially Catalytic Wall for Thermal Protection System ..... ?
17.25. The Perforated Walls Dialog Box ..... ?
18.1. Rotor Disks Schematic 2335
18.2. 3D Blade Geometry Represented by a Stack of 2D Airfoils 2336
18.3. Effective Pitch Angle as a Function of r/R and ψ 2337
18.4. Blade Flapping – Baseline β_0 (Top), Longitudinal and Lateral Components β_{1c}, β_{1s} (Bottom) 2338
18.5. Schematic of a Cut Through a Structured VBM Grid. The Green Cells Are Assigned to the VBM 2343
18.6. VBM Graphical User Interface Configuration Panels 2346
19.1. The Species Model Dialog Box 2360
19.2. The Species Model Dialog Box Displaying the Thickened Flame Model 2366
19.3. The Select Boundary Species Dialog Box 2367
19.4. The Select Residual Monitored Species 2368
19.5. The Import CHEMKIN Format Mechanism Dialog Box for Volumetric Kinetics 2370
19.6. The Material Dialog Box When Importing CHEMKIN Transport Properties 2373
19.7. The Create/Edit Materials Dialog Box (Showing a Mixture Material) 2375
19.8. The Species Dialog Box 2376
19.9. The Reactions Dialog Box 2380
19.10. The Third-Body Efficiency Dialog Box 2383

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

19.11. The Pressure-Dependent Reaction Dialog Box 2384
19.12. The Coverage Dependent Reaction Dialog Box 2385
19.13. Backward Reaction Parameters Dialog Box 2386
19.14. The Reaction Mechanisms Dialog Box 2388
19.15. The Site Parameters Dialog Box 2389
19.16. The Coal Calculator Dialog Box 2392
19.17. The Import CHEMKIN Format Mechanism Dialog Box for Surface Kinetics 2401
19.18. The Species Dialog Box for the Mixture Material 2411
19.19. The Species Dialog Box for the Particle-Mixture Material with Reacting Components 2411
19.20. The Reaction Dialog Box of a Mixture Material with Particle Reactions Enabled 2412
19.21. The Species Model Dialog Box with Electrochemical Reactions Enabled 2415
19.22. The Reactions Dialog Box 2417
19.23. The Reaction Mechanisms Dialog Box 2419
19.24. Wall Potential Boundary Condition 2420
19.25. The Water Chemistry and Corrosion Mechanism Dialog Box 2425
19.26. Optimal Surface Mesh on the Reacting Channel Wall 2430
19.27. The Reacting Channel Model Dialog Box 2431
19.28. The Reacting Channel Model Dialog Box (Group Inlet Conditions Tab) 2433
19.29. The Wall Boundary Condition Dialog Box for the Reacting Channel Model 2435
19.30. Reacting Channel 2D Curves Dialog Box (Plot) 2436
19.31. Reacting Channel 2D Curves Dialog Box (Report) 2437
19.32. Reactor Network Dialog Box (Steady-State Flow) 2439
19.33. Reactor Network Dialog Box - Expert Options 2440
19.34. The Species Model Dialog Box for Lagrangian Composition PDF Transport 2445
19.35. The Integration Parameters Dialog Box 2446
19.36. The Species Model Dialog Box for Eulerian Composition PDF Transport 2448
19.37. The Velocity Inlet Dialog Box for Eulerian Composition PDF Transport 2450
19.38. The Solution Initialization Task Page for Eulerian Composition PDF Transport 2451
19.39. The Run Calculation Task Page for Composition PDF Transport 2452
19.40. The Particle Tracks Dialog Box for Tracking PDF Particles 2455
19.41. The Integration Parameters Dialog Box 2457
19.42. The **Select DAC Target Species** Dialog Box 2469
20.1. Defining Equilibrium Chemistry 2473
20.2. Defining Steady Diffusion Flamelet Chemistry 2473
20.3. Defining Chemical Boundary Species 2474
20.4. Calculating Steady Diffusion Flamelets 2475
20.5. Calculating the Chemistry Look-Up Table 2475
20.6. The Species Model Dialog Box (Chemistry Tab) 2477
20.7. The Chemistry Tab for the Unsteady Diffusion Flamelet Model 2481
20.8. The Enabled Diesel Unsteady Flamelet Model 2484
20.9. The Unsteady Flamelet Parameters Dialog Box 2485
20.10. The Flamelet Fluid Zones Dialog Box 2486
20.11. The Species Model Dialog Box (Boundary Tab) 2488
20.12. The Coal Calculator Dialog Box 2497
20.13. The Species Model Dialog Box (Control Tab) 2499
20.14. The Species Model Dialog Box (Control Tab) for the Steady Diffusion Flamelet Model 2500
20.15. Method to Zero Out the Slow Chemistry Species 2501
20.16. The Species Model Dialog Box (Flamelet Tab) 2502
20.17. The Flamelet Tab for the Unsteady Diffusion Flamelet Model 2505
20.18. The Flamelet 2D curves Dialog Box 2506
20.19. The Flamelet 3D Surfaces Dialog Box 2507

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

20.20. Example 2D Plot of Flamelet Data 2508
20.21. Example 3D Plot of Flamelet Data 2509
20.22. The Species Model Dialog Box (Table) Tab Displaying Automated Grid Refinement 2510
20.23. The Species Model Dialog Box (Table) Tab Excluding Automated Grid Refinement 2511
20.24. The PDF Table Dialog Box (Non-Adiabatic Case With Flamelets) 2515
20.25. Mean Species Mole Fraction Derived From an Equilibrium Chemistry Calculation 2518
20.26. Mean Temperature Derived From an Equilibrium Chemistry Calculation 2518
20.27. 3D Plot of Look-Up Table for Temperature Generated for a Simple Hydrocarbon System 2519
20.28. The Inert Model Dialog Box 2522
20.29. The Inert Model Dialog Box 2523
20.30. The Velocity Inlet Dialog Box Showing Mixture Fraction Boundary Conditions 2526
20.31. The Species Model Dialog Box for a Two-Mixture-Fraction Calculation 2532
20.32. Predicted Contours of Mixture Fraction in a Methane Diffusion Flame 2534
20.33. Predicted Contours of CO2 Mass Fraction Using the Non-Premixed Combustion Model 2535
20.34. The Species Model Dialog Box for Premixed Combustion 2538
20.35. The Species Model Dialog Box for the G-Equation Model 2539
20.36. The Progress Variable Definition Dialog Box 2547
20.37. Premixed Flamelet Generated Manifolds (Flamelet Tab) 2548
20.38. The Distribution of Points Dialog Box 2550
20.39. Diffusion Flamelet Generated Manifolds (Flamelet Tab) 2551
20.40. Non-adiabatic Premixed Flamelet Generated Manifolds (**Flamelet** Tab) 2553
20.41. The Species Model Dialog Box: Table Tab with no Automated Grid Refinement 2554
20.42. The Species Model Dialog Box: Table Tab Displaying Automated Grid Refinement 2556
20.43. The Select Transported Scalars Dialog Box 2557
20.44. The PDF Table Dialog Box (Adiabatic Case With FGM) 2559
20.45. The PDF Table Dialog Box (Non-Adiabatic Case With FGM) 2560
20.46. The Species Model Dialog Box(Premix Tab) 2563
20.47. The Species Model Dialog Box (Properties Tab) 2565
20.48. The Quadratic of Mixture Fraction Dialog Box 2565
20.49. The Piecewise Linear Dialog Box 2566
20.50. The Species Model Dialog Box: Strained Laminar Flame Speed 2567
20.51. The Flamelet 2D Curves Dialog Box 2568
21.1. The Spark Ignition Dialog Box 2572
21.2. The Set Spark Ignition Dialog Box 2573
21.3. The Ignition Delay Model in the Autoignition Model Dialog Box 2575
21.4. The Knock Model in the Autoignition Model Dialog Box 2575
21.5. The Ignition Delay Model for the Partially Premixed Combustion Model 2576
21.6. The Knock Model with the Partially Premixed Combustion Model Enabled 2577
21.7. Experimental Engine Mesh 2578
21.8. Cylinder Mass vs. Crank Angle 2580
21.9. Cylinder Pressure vs. Crank Angle 2581
21.10. Crevice Pressures 2583
22.1. The NOx Model Dialog Box 2587
22.2. The NOx Model Dialog Box Displaying the Fuel Streams 2590
22.3. The NOx Dialog Box Displaying the Reburn Reduction Method 2596
22.4. The NOx Dialog Box Displaying the SNCR Reduction Method 2596
22.5. The NOx Model Dialog Box and the Turbulence Interaction Mode Tab 2598
22.6. The Soot Model Dialog Box for the One-Step Model 2604
22.7. The Soot Model Dialog Box for the Two-Step Model 2606
22.8. The Soot Model Dialog Box for the Moss-Brookes Model 2609
22.9. The Soot Model Dialog Box for the Moss-Brookes Model with a User-Defined Precursor Correlation ... 2614

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

22.10. The Piecewise-Polynomial Profile Dialog Box 2615
22.11. The Soot Model Dialog Box for the Method of Moments Model 2617
22.12. Sticking Coefficients for Soot Precursors 2618
22.13. Settings for the Nucleation Mechanism 2619
22.14. The Decoupled Detailed Chemistry Dialog Box 2623
23.1. The Acoustics Model Dialog Box 2630
23.2. The Acoustics Model Dialog Box for a 3D Steady-State Case with a Single Moving Reference Frame 2632
23.3. The Acoustics Model Dialog Box for Exporting in CGNS Format 2634
23.4. The Acoustics Model Dialog Box 2635
23.5. The Interior Cell Zone Selection Dialog Box 2636
23.6. An Interior Source Surface 2637
23.7. The Acoustic Receivers Dialog Box 2638
23.8. The Run Calculation Task Page 2640
23.9. The Acoustic Signals Dialog Box 2642
23.10. The Read ASD Files Tab of the Acoustic Source FFT Dialog Box 2646
23.11. The Compute FFT Fields Tab of the Acoustic Source FFT Dialog Box 2647
23.12. The FFT Surface Variables Tab of the Acoustic Source FFT Dialog Box for the Octave Bands 2648
23.13. Bar Chart of Surface Pressure Level for Octave Bands 2650
23.14. The FFT Surface Variables Tab of the Acoustic Source FFT Dialog Box for a Set of Individual Modes ... 2650
23.15. The Write CGNS Files Tab of the Acoustic Source FFT Dialog Box 2652
23.16. The Acoustics Model Dialog Box 2654
23.17. The Basic Shapes Dialog Box 2656
23.18. The Acoustics Wave Equation Solver Controls Task Page 2658
23.19. The Acoustics Initialization Dialog Box 2659
23.20. Acoustics Model Dialog Box with Kirchhoff Integral Options 2662
23.21. Integration Surface Dialog Box 2663
23.22. The Acoustics Model Dialog Box for Broadband Noise 2665
23.23. The Manage Sponge Layers Dialog Box 2667
23.24. The Sponge Layer Dialog Box 2668
24.1. The Discrete Phase Model Dialog Box - Tracking Tab 2684
24.2. A Subtet Formed From Decomposing a Hexagonal Cell 2687
24.3. Degenerate Subtet in a Polyhedral Mesh 2688
24.4. Degenerate Subtet in a Hex Mesh 2689
24.5. Lagrangian Wall Film Tracking 2690
24.6. The Discrete Phase Model Dialog Box - Physical Models Tab 2696
24.7. Discrete Phase Model Dialog Box with DEM Collision Model 2702
24.8. Wall Boundary Condition for the DEM Model 2704
24.9. Collision Dialog Box 2705
24.10. DEM Collision Settings Dialog Box 2705
24.11. The Discrete Phase Model Dialog Box - UDF Tab 2709
24.12. The Discrete Phase Model Dialog Box - Numerics Tab 2711
24.13. Particle Injection Defining a Single Particle Stream 2722
24.14. Particle Injection Defining an Initial Spatial Distribution of the Particle Streams 2722
24.15. Particle Injection Defining an Initial Spray Distribution of the Particle Velocity 2722
24.16. Cone Injector Geometry 2727
24.17. Hollow Cone With Specified Cone Angle 2728
24.18. Hollow Cone With Specified Cone Angle And Dispersion Angle 2729
24.19. Point Injection 2731
24.20. Solid Injection 2731
24.21. Ring Injection 2732
24.22. Hollow Injection 2732

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.23. Flat Fan Viewed from Above and from the Side 2741
24.24. Example of Cumulative Size Distribution of Particles 2746
24.25. Rosin-Rammler Curve Fit for the Example Particle Size Data 2747
24.26. Discrete Number/Mass Fraction Distribution 2749
24.27. Discrete Cumulative Number/Mass Fraction Distribution 2749
24.28. Association of Particle Size Distribution Data Dialog Box 2750
24.29. Tabulated Size Distribution Histogram of Diameter Over Number Fraction 2752
24.30. The Injections Branch in the **Outline View** 2754
24.31. The Injections Dialog Box 2754
24.32. The Set Injection Properties Dialog Box 2757
24.33. Setting Surface Injection Properties 2758
24.34. Setting Surface Injection Properties 2764
24.35. Mean Trajectory in a Turbulent Flow 2773
24.36. Stochastic Trajectories in a Turbulent Flow 2774
24.37. The Custom Laws Dialog Box 2775
24.38. The Set Multiple Injection Properties Dialog Box 2776
24.39. Discrete Phase Boundary Conditions in the Wall Dialog Box 2781
24.40. "Trap" Boundary Condition for the Discrete Phase 2783
24.41. "Escape" Boundary Condition for the Discrete Phase 2783
24.42. The Wall Dialog Box: the Particle-Wall Heat Exchange Option 2787
24.43. The **Generic Erosion Model Parameters** Dialog Box 2788
24.44. The **Finnie Model Parameters** Dialog Box 2789
24.45. The **McLaury Model Parameters** Dialog Box 2789
24.46. The **Oka Model Parameters** Dialog Box 2790
24.47. The **DNV Model Parameters** Dialog Box 2791
24.48. The **Shear Stress Model Parameters** Dialog Box 2792
24.49. The **Erosion Dynamic Mesh Coupling Setup** Dialog Box 2794
24.50. The **Run Erosion-Dynamic Mesh Simulation** Dialog Box 2797
24.51. The **Graphics Objects** Dialog Box 2799
24.52. Valid Configuration for the Lagrangian Wall film Model with Overset Mesh 2801
24.53. Invalid Configuration for the Lagrangian Wall film Model with Overset Mesh 2802
24.54. The Set Injection Properties Dialog Box: Condensate Injection 2808
24.55. The Components Tab 2816
24.56. Uncoupled Discrete Phase Calculations 2828
24.57. Coupled Discrete Phase Calculations 2829
24.58. Effect of Number of Source Term Updates on Source Term Applied to Flow Equations 2831
24.59. The Particle Tracks Dialog Box 2835
24.60. The Track Style Attributes Dialog Box 2839
24.61. The Particle Sphere Style Attributes Dialog Box 2840
24.62. Particles with the Vector Style 2841
24.63. Particles with the Centered Vector Style 2842
24.64. Particles with the Centered Cylinder Style 2843
24.65. The Particle Vector Style Attributes Dialog Box 2844
24.66. The Import Particle Data Dialog Box 2844
24.67. The Particle Filter Attributes Dialog Box 2845
24.68. The Reporting Variables Dialog Box 2857
24.69. The Sample Trajectories Dialog Box 2874
24.70. The Trajectory Sample Histograms Dialog Box 2876
24.71. The Trajectory Sample Histograms Dialog Box: Correlation 2878
24.72. The Trajectory Sample Histograms Dialog Box: Data File Reduction 2879
24.73. The Particle Summary Dialog Box 2884

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.74. The Shared Memory Option with Workpile Algorithm Enabled 2890
25.1. Macroscopic Particle Model Dialog Box (Particle Tracking Tab) 2896
25.2. Macroscopic Particle Model Dialog Box (Drag Tab) 2898
25.3. Macroscopic Particle Model Dialog Box (Collision Tab) 2900
25.4. Macroscopic Particle Model Dialog Box (Deposition Tab) 2902
25.5. Macroscopic Particle Model Dialog Box (Injection Tab) 2903
25.6. Macroscopic Particle Model Dialog Box (Attraction Forces Tab) 2911
25.7. Macroscopic Particle Model Dialog Box (Initialize MPM Tab) 2912
26.1. Multiphase Model Dialog Box for the VOF Model 2919
26.2. Multiphase Model Dialog Box for the Mixture Model 2920
26.3. Multiphase Model Dialog Box for the Eulerian Model 2921
26.4. Expert Options Dialog Box 2927
26.5. Numerical Flotsams in the Volume Fraction Field 2928
26.6. The Volume Fraction Field for the Cell Based Flotsam Detection 2928
26.7. The Volume Fraction Field for the Node Based Flotsam Detection 2929
26.8. The Volume Fraction Field for the Node-Averaged Filtering 2929
26.9. The Operating Conditions Dialog Box for Multiphase Flows 2930
26.10. The **Multiphase Model** dialog box - **Phases** tab 2933
26.11. The Species Model Dialog Box with a Multiphase Model Enabled 2935
26.12. The Phase Properties Dialog Box 2936
26.13. The Reactions Tab for Heterogeneous Reactions 2938
26.14. The Mass Tab for Mass Transfer 2941
26.15. The Cavitation Model Dialog Box 2946
26.16. Table Input for Vaporization Pressure 2947
26.17. The Evaporation-Condensation Model Dialog Box (Eulerian Multiphase Model) 2951
26.18. The Species Mass Transfer Model Dialog Box 2954
26.19. The Pressure Inlet Dialog Box for a Mixture 2958
26.20. The Wall Dialog Box for a Mixture in a Multiphase Calculation with Wall Adhesion 2959
26.21. Measuring the Contact Angle 2960
26.22. The Porous Jump Dialog Box Displaying Jump Adhesion 2961
26.23. The Wall Dialog Box for a Phase 2962
26.24. The Pressure Outlet Dialog Box for a Phase 2963
26.25. The Cell Zone Conditions Task Page 2964
26.26. Mass-Flow Inlet Boundary Condition Dialog Box 2970
26.27. Determining the Free Surface Level and the Bottom Level 2984
26.28. Pressure Inlet for Open Channel Flow 2985
26.29. Density Interpolation Method for Open Channel Flow 2987
26.30. The Velocity Inlet for Open Channel Wave BC 2991
26.31. Segregated Velocity Inputs for Open Channel Wave BC 2992
26.32. The Velocity Inlet for Open Channel Wave BC (Explicit Formulation) 2994
26.33. The Solution Initialization Task Page 3002
26.34. The Fluid Dialog Box to Enable Numerical Beach 3005
26.35. Numerical Beach Sketch 3008
26.36. Defining the Primary Phase in the Phases Tab 3010
26.37. Defining the Secondary Phase in the Phases Tab 3011
26.38. The Multiphase Model Dialog Box (Forces Tab) 3012
26.39. The Multiphase Model Dialog Box for the VOF Model (Discretization Tab) 3016
26.40. The VOF-to-DPM Transition Parameters Dialog Box 3024
26.41. The DPM-to-VOF Transition Parameters Dialog Box 3032
26.42. DPM Particles at the Gas-Liquid Interface 3035
26.43. Surface Mesh Before the Model Transition is Triggered 3035

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.44. Gas-Liquid Interface after the Model Transition is Triggered 3036
26.45. Surface Mesh After the Mesh Adaption and Phase Transition 3037
26.46. Defining the Secondary Phase for the Mixture Model 3039
26.47. Defining a Granular Phase in the Mixture Model 3040
26.48. The Multiphase Model Dialog Box Displaying the Interfacial Area Concentration Settings 3043
26.49. The Multiphase Model Dialog Box for the Mixture Model (Forces Tab) 3047
26.50. The Multiphase Model Dialog Box - Hybrid Phase Morphology 3050
26.51. The Hybrid Drag Model Parameters Dialog Box 3052
26.52. The Hybrid Area Density Parameters Dialog Box 3054
26.53. The Evaporation-Condensation Model Dialog Box 3058
26.54. Boiling Model Expert Options 3062
26.55. Transition Function vs. Volume Fraction of Liquid 3064
26.56. The Multiphase Model Dialog Box for a Non-Granular Phase (**Phases** Tab) 3067
26.57. The Multiphase Model Dialog Box for a Granular Phase (**Phases** Tab) 3068
26.58. **Syamlal Obrien Model** Dialog Box 3077
26.59. Antal et al. Model Dialog Box 3082
26.60. Tomiyama Model Dialog Box 3082
26.61. Frank Model Dialog Box 3083
26.62. Hosokawa Model Dialog Box 3084
26.63. Lopez de Bertodano Model Dialog Box 3085
26.64. Simonin Model Dialog Box 3086
26.65. Burns et al. Model Dialog Box 3086
26.66. Diffusion—in—vof Model Dialog Box 3087
26.67. The Viscous Model Dialog Box for an Eulerian Multiphase Calculation 3090
26.68. Troshko-Hassan Model Dialog Box 3091
26.69. Sato Model Dialog Box 3092
26.70. Simonin-et-al Model Dialog Box 3092
26.71. The Multiphase Model Dialog Box for Heat Transfer 3093
26.72. The Two Resistance Model Dialog Box 3095
26.73. The Time Scale Model Dialog Box 3095
26.74. The Multiphase Model Dialog Box for Interfacial Area 3096
26.75. Defining a Secondary Phase with the AIAD Continuous Phase Treatment 3099
26.76. Example of Defining a Secondary Entrained Phase for Droplets 3100
26.77. The **AIAD Model Parameters** Dialog Box 3102
26.78. The **AIAD Entrainment Parameters** Dialog Box 3103
26.79. The Dense Discrete Phase Model - Models Tab 3108
26.80. The Multiphase Model Dialog Box for DDPM 3109
26.81. The Set Injection Properties Dialog Box 3110
26.82. The Multiphase Model Dialog Box for a Granular Phase 3112
26.83. The Boiling Model 3114
26.84. The Boiling Options Dialog Box 3115
26.85. The Multiphase Model Dialog Box for the Boiling Model (Phases Tab) 3117
26.86. Defining Forces in the Forces Tab 3118
26.87. The Boiling Model Dialog Box 3120
26.88. The Kocamustafaogullari-Ishii Model Parameter Dialog Box 3121
26.89. The Lavieville Model Parameters Dialog Box 3122
26.90. The Tentner Model Parameters Dialog Box 3123
26.91. The **Multiphase Model** Dialog Box - **Population Balance Model** Tab 3133
26.92. The **Liao Aggregation Model Parameters** Dialog Box 3137
26.93. The Surface Tension for Population Balance Dialog Box 3138
26.94. The Hamaker Constant for Population Balance Dialog Box 3138

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.95. The **Prince and Blanch Model Parameters** Dialog Box 3139
26.96. **Liao Breakage Model Parameters** Dialog Box 3140
26.97. The Surface Tension and Weber Number Dialog Box 3141
26.98. The Ghadiri Breakage Constant for Population Balance Dialog Box 3141
26.99. The Shape Factor for Parabolic PDF Dialog Box 3142
26.100. The Generalized pdf for multiple breakage Dialog Box 3143
26.101. The Population Balance Model Tab for the DQMOM Model 3145
26.102. DQMOM Values Produced From a PDF File 3146
26.103. Specifying Inlet Boundary Conditions for the Population Balance Model 3149
26.104. The Equations Dialog Box 3152
26.105. Setting the Secondary Phase for Hydrodynamic Coupling 3153
26.106. The Phase Interaction Tab for Non-reacting Species 3155
26.107. The Reactions Tab for a Heterogeneous Reaction 3157
26.108. The Size Calculator Dialog Box 3158
26.109. The Population Balance Moments Dialog Box 3161
26.110. The Number Density Function Dialog Box 3162
26.111. The Multiphase Model Dialog Box with the Wet Steam Model Selected (Pressure-Based) 3173
26.112. The Multiphase Model Dialog Box with the Wet Steam Model Selected (Density-Based) 3173
26.113. The Solution Methods Task Page Displaying The Pressure-Velocity Coupling Options 3189
26.114. The Solution Controls Task Page Displaying the Coupled Volume Fraction Method for the VOF and Mixture Models 3192
26.115. The Solution Controls Task Page Displaying the Coupled Volume Fraction Method for the Eulerian Multiphase Model 3194
26.116. The Stabilization Methods Dialog Box 3196
26.117. The Velocity Limiting Treatment Dialog Box 3197
27.1. The Solidification and Melting Dialog Box 3219
27.2. The Create/Edit Materials Dialog Box for Melting and Solidification 3221
27.3. The Solidification and Melting Dialog Box 3224
27.4. Liquid Fraction Contours for Continuous Crystal Growth 3226
28.1. The Structural Model Dialog Box 3230
28.2. The Structure Tab of the Wall Dialog Box 3231
28.3. The Wall Dialog Box for a Two-Sided Wall 3232
29.1. The Eulerian Wall Film Dialog Box - Model Options and Setup tab (with DPM Interaction) 3242
29.2. The Eulerian Wall Film Dialog Box - Model Options and Setup tab (with Phase Interaction) 3247
29.3. Eulerian Wall Film Solution Controls (Steady Flow) 3249
29.4. Eulerian Wall Film Solution Controls (Unsteady Flow) 3250
29.5. Wall Dialog Box - Boundary Type Tab 3253
29.6. Wall Dialog Box - Sources Terms Tab 3255
29.7. Wall Dialog Box - Phase Change Tab 3256
29.8. Wall Dialog Box - Surface Contact Tab 3257
29.9. Wall Dialog Box - DPM Interaction Tab 3258
29.10. Wall Dialog Box - VOF Interaction Tab 3260
30.1. The **Potential/Electrochemistry** Dialog Box - Lithium-ion Battery Model 3269
30.2. The **Potential/Electrochemistry** Dialog Box - Echem Rate Tab 3270
30.3. The **Potential/Electrochemistry** Dialog Box - Material Properties Tab 3271
30.4. The **Potential/Electrochemistry** Dialog Box - Report Tab 3274
30.5. The Potential/Electrochemistry Dialog Box (Model Tab) 3278
30.6. The Potential/Electrochemistry Model Dialog Box (Parameters Tab) 3282
30.7. The Anode Tab of the Potential/Electrochemistry Dialog Box for the Current Collector 3285
30.8. The Anode Tab of the Potential/Electrochemistry Dialog Box for the Flow Channel 3286
30.9. The Anode Tab of the Potential/Electrochemistry Dialog Box for the Porous Layer 3287

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

30.10. The Anode Tab of the Potential/Electrochemistry Dialog Box with the Catalyst Layer Selected - Resolved Approach 3289
30.11. The Electrolyte Tab of the Potential/Electrochemistry Dialog Box - Resolved MEA Layer 3291
30.12. The Species Permeation Dialog Box 3292
30.13. The Electrolyte Tab of the Potential/Electrochemistry Dialog Box - Unresolved MEA Layer 3293
30.14. The Electrical Tabs Tab of the Potential/Electrochemistry Dialog Box 3295
30.15. The Advanced Tabs Tab of the Potential/Electrochemistry Dialog Box 3296
31.1. The Battery Model Option in the **Outline View** 3302
31.2. The Battery Model Dialog Box (Model Options Tab) 3304
31.3. The Calendar Life Parameters Dialog Box 3309
31.4. The Battery Aging Table Dialog Box - Cycle Life 3310
31.5. The Fast Charging Data Table Dialog Box 3314
31.6. The Battery Model Dialog Box (Conductive Zones Tab) 3318
31.7. The Battery Model Dialog Box (Electric Contacts Tab) 3319
31.8. Model Parameters Tab—CHT Coupling Method 3322
31.9. Model Parameters Tab—FMU-CHT Coupling Method 3323
31.10. Model Parameters Tab—NTGK/DCIR Model 3324
31.11. NTGK U-parameter Data Table Dialog Box 3326
31.12. The Import Raw Data Dialog Box 3327
31.13. The Internal Resistance Data Table Dialog Box 3328
31.14. The Limiting Current Data Table Dialog Box 3330
31.15. The Parameter Estimation Dialog Box for the NTGK Model 3331
31.16. Model Parameters Tab—Equivalent Circuit Model 3334
31.17. The Parameter Estimation Dialog Box for the ECM 3336
31.18. The HPPC Data Library Dialog Box 3338
31.19. Overview of the Folder Structure in the HPPC Library 3341
31.20. Model Parameters Tab—Newman's P2D Model 3342
31.21. The Echem Material Database Dialog Box 3346
31.22. Electrode Layer Orientation 3348
31.23. Model Parameters Tab—User-Defined E-Model 3349
31.24. The Battery Model Dialog Box (UDF Tab) 3350
31.25. The Battery Model Dialog Box (Advanced Options Tab) 3351
31.26. The Standalone Echem Model Dialog Box (Battery Life Model) 3357
31.27. Battery Pack with a Cold Plate 3362
31.28. The Pack Builder Dialog Box 3362
31.29. The Battery ROM Tool Kit Dialog Box 3367
31.30. The Select Volumes Dialog Box 3368
31.31. The Value Distribution Dialog Box 3369
31.32. The Battery ROM Tool Kit Dialog Box - LTI-ROM Generation Tab 3372
31.33. The Orthotropic Thermal Conductivity Dialog Box 3375
31.34. The Swelling Model Parameters Dialog Box 3376
31.35. The Venting Model Setup Dialog Box 3378
31.36. The Venting Gas Composition Dialog Box 3379
31.37. The dUdT Data Table Dialog Box 3381
31.38. The Battery Model Dialog Box - One-Equation Kinetics Model 3383
31.39. The Abuse Rate Data Table Dialog Box 3384
31.40. The Material Database for Abuse Kinetics Dialog Box 3386
31.41. The Standalone Thermal Abuse Model Dialog Box 3387
32.1. The PEMFC Option in the **Outline View** 3405
32.2. The Model Options in the PEM Fuel Cell Model Dialog Box 3407
32.3. The Parameters Tab of the PEM Fuel Cell Model Dialog Box 3412

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.4. The Anode Tab of the PEM Fuel Cell Model Dialog Box with Current Collector Selected 3416
32.5. The Anode Tab of the PEM Fuel Cell Model Dialog Box with Flow Channel Selected 3417
32.6. The Anode Tab of the PEM Fuel Cell Model Dialog Box with Porous Electrode Selected 3418
32.7. The Anode Tab of the PEM Fuel Cell Model Dialog Box with TPB Layer (Catalyst) Selected 3420
32.8. The Anode Tab of the PEM Fuel Cell Model Dialog Box with Micro Porous Layer Selected 3423
32.9. The Electrolyte Tab of the PEM Fuel Cell Model Dialog Box 3425
32.10. The Cathode Tab of the PEM Fuel Cell Model Dialog Box with TPB Layer (Catalyst) Selected 3427
32.11. The Electrical Tabs Tab of the PEM Fuel Cell Model Dialog Box 3430
32.12. The Advanced Tab of the PEM Fuel Cell Model Dialog Box for Contact Resistivities 3431
32.13. The Advanced Tab of the PEM Fuel Cell Model Dialog Box for the Coolant Channel 3432
32.14. The Advanced Tab of the PEM Fuel Cell Model Dialog Box for Stack Management 3433
32.15. The Customized UDFs Tab in the PEM Fuel Cell Dialog Box 3435
32.16. The Compiled Customized Addon Module Dialog Box 3435
32.17. The Reports Tab of the PEM Fuel Cell Model Dialog Box 3436
32.18. The Fuel Cell and Electrolysis Option in the Tree 3451
32.19. The Model Options in the Fuel Cell and Electrolysis Models Dialog Box—PEMFC Enabled 3452
32.20. The Parameters Tab of the Fuel Cell and Electrolysis Models Dialog Box 3455
32.21. The Anode Tab of the Fuel Cell and Electrolysis Models Dialog Box With Current Collector Selected .. 3457
32.22. The Anode Tab of the Fuel Cell and Electrolysis Models Dialog Box With Flow Channel Selected 3458
32.23. The Anode Tab of the Fuel Cell and Electrolysis Models Dialog Box With Porous Electrode Selected .. 3459
32.24. The Anode Tab of the Fuel Cell and Electrolysis Models Dialog Box With TPB Layer (Catalyst) Selected 3460
32.25. The Electrolyte Tab of the Fuel Cell and Electrolysis Models Dialog Box 3461
32.26. The Cathode Tab of the Fuel Cell and Electrolysis Models Dialog Box With Current Collector Selected 3463
32.27. The Cathode Tab of the Fuel Cell and Electrolysis Models Dialog Box With Flow Channel Selected 3464
32.28. The Cathode Tab of the Fuel Cell and Electrolysis Models Dialog Box With Porous Electrode Selected 3465
32.29. The Cathode Tab of the Fuel Cell and Electrolysis Models Dialog Box With TPB Layer (Catalyst) Selected 3466
32.30. The Advanced Tab of the Fuel Cell and Electrolysis Models Dialog Box for Contact Resistivities 3467
32.31. The Advanced Tab of the Fuel Cell and Electrolysis Models Dialog Box for the Coolant Channel 3468
32.32. The Advanced Tab of the Fuel Cell and Electrolysis Models Dialog Box for Stack Management 3469
32.33. The Customized UDFs Tab in the Fuel Cell and Electrolysis Models Dialog Box 3471
32.34. The Reports Tab of the Fuel Cell and Electrolysis Models Dialog Box 3472
32.35. The Electric Conductivity Field in the Create/Edit Materials Dialog Box 3473
32.36. Opening the SOFC Model Dialog Box in the **Outline View** 3482
32.37. The Model Parameters Tab in the SOFC Model Dialog Box 3486
32.38. The Electrochemistry Tab in the SOFC Model Dialog Box 3489
32.39. The Electrolyte and Porous Zones Tab in the SOFC Model Dialog Box 3491
32.40. The Electric Field Tab in the SOFC Model Dialog Box 3493
32.41. The Customized UDFs Tab in the SOFC Model Dialog Box 3494
33.1. Enabling the MHD Model Dialog Box 3502
33.2. The MHD Model Dialog Box 3503
33.3. The MHD Model Dialog Box for Patching an External Magnetic Field 3504
33.4. The MHD Model Dialog Box for Specifying a Moving Field 3505
33.5. The MHD Model Dialog Box for Importing an External Magnetic Field 3506
33.6. Apply External B0 Field Dialog Box 3507
33.7. Cell Boundary Condition Setup 3508
33.8. Editing Material Properties within Boundary Condition Setup 3509
33.9. Wall Boundary Condition Setup 3510

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

33.10. Conducting Wall Boundary Conditions in Electrical Potential Method 3511
33.11. Solution Control Tab in the MHD Model Dialog Box 3512
34.1. The Fiber Model Dialog Box 3526
34.2. The Fiber Injections Dialog Box 3533
34.3. The Set Fiber Injection Properties Dialog Box 3537
34.4. The Set Fiber Injection Properties Dialog Box With Take-Up Point Properties 3539
34.5. Line Injections 3540
34.6. Matrix Injections 3541
34.7. Equidistant Fiber Grid 3541
34.8. One-Sided Fiber Grid 3542
34.9. Two-Sided Fiber Grid 3542
34.10. Three-Sided Fiber Grid 3543
34.11. Defining a Three-Sided Fiber Grid Using the Set Fiber Injection Properties Dialog Box 3543
34.12. The Fiber Model Dialog Box 3550
34.13. Fiber Solution Controls Dialog Box 3551
34.14. Displaying Fiber Locations Using the Contours Dialog Box 3553
34.15. The Fiber Mesh Display Dialog Box 3554
34.16. The Fiber Style Attributes Dialog Box 3555
34.17. The Fiber Display Dialog Box 3558
35.1. The Reduced Order Model Dialog Box 3563
35.2. The Write Profile Dialog Box 3569
36.1. The General Task Page 3574
36.2. The Solution Methods Task Page for the HOTR Option 3580
36.3. The Relaxation Options Dialog Box with the Pressure-Based Solver 3581
36.4. The Solution Methods Task Page for a Pressure-Based Segregated Algorithm 3583
36.5. The Solution Controls Task Page for the Pressure-Based Solver 3591
36.6. The Advanced Solution Controls Dialog Box for the Pressure-Based Segregated Non-Iterative Solver . 3594
36.7. The NITA Options Dialog Box 3596
36.8. Creating CFL Report Plots and Files 3603
36.9. The Solution Controls Task Page for the Density-Based Explicit Formulation 3604
36.10. The Solution Methods Task Page for the Density-Based Implicit Formulation 3606
36.11. The Multigrid Tab 3612
36.12. The Advanced Solution Controls Dialog Box 3620
36.13. The Solution Limits Dialog Box 3622
36.14. The Multi-Stage Tab 3625
36.15. The Solution Initialization Task Page 3629
36.16. The Patch Dialog Box 3631
36.17. The Solution Initialization Task Page for Hybrid Initialization 3636
36.18. The Hybrid Initialization Dialog Box 3637
36.19. The Run Calculation Task Page 3639
36.20. Time-Dependent Calculation of Vortex Shedding (t=36.6 sec) 3644
36.21. Time-Dependent Calculation of Vortex Shedding (t=41.6 sec) 3644
36.22. The General Task Page for a Transient Calculation 3645
36.23. The Solution Methods Task Page for a Transient Calculation 3646
36.24. The Run Calculation Task Page for Implicit Transient Calculations 3650
36.25. The Zone-Specific Sampling Options Dialog Box for Standard Statistics 3656
36.26. The Sampling Options Dialog Box 3658
36.27. Lift Coefficient Plot for a Time-Periodic Solution 3670
36.28. Zone-Specific Sampling Options Dialog Box for Runtime DFT - Frequency Band 3671
36.29. Zone-Specific Sampling Options Dialog Box for Runtime DFT - Single Tone 3672
36.30. The Solution Methods Task Page 3674

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.31. The Solution Controls Task Page for the Local Time Step Method 3676
36.32. The Solution Controls Task Page for the Global Time Step Method 3678
36.33. The Advanced Solution Controls Dialog Box for the Pseudo Time Method 3680
36.34. The Run Calculation Task Page for the User-Specified Pseudo Time Step Method 3681
36.35. The Run Calculation Task Page for the Automatic Pseudo Time Step Method 3683
36.36. The Residual Monitors Dialog Box 3688
36.37. The Residual Monitors Dialog Box with Advanced Options Shown 3689
36.38. The Residual Monitors Dialog Box Displaying Relative or Absolute Convergence 3692
36.39. Report File for 'flow-time', 'delta-time', and 'iters-per-timestep' 3696
36.40. Fluctuating Simulation Example 3699
36.41. The Execute Commands Dialog Box 3701
36.42. Hovering Over Your Commands in the Outline View 3702
36.43. The Execute Commands Manager Dialog Box 3703
36.44. The Define Macro Dialog Box 3704
36.45. The Automatically Initialize and Modify Case Option 3706
36.46. The Case Modification Manager Dialog Box With Pending Modifications 3707
36.47. The Auto Initialization Method Dialog Box 3708
36.48. The Modify Case Dialog Box for Pre-Initialization 3709
36.49. The Modify Case Dialog Box for Original Settings 3709
36.50. The Modify Case Dialog Box 3710
36.51. The Modify Case Dialog Box (Transient Cases) 3710
36.52. The Case Modification Manager Dialog Box With Pending Modifications 3711
36.53. The Run Calculation Task Page 3713
36.54. The Animation Definition Dialog Box 3715
36.55. The Playback Dialog Box 3718
36.56. The Video Options Dialog Box 3723
36.57. The Advanced Video Quality Options Dialog Box 3724
36.58. The Case Check Dialog Box 3725
36.59. The Information Dialog Box 3726
36.60. The Mesh Tab in the Case Check Dialog Box 3727
36.61. The Models Tab in the Case Check Dialog Box 3729
36.62. The Boundaries and Cell Zones Tab in the Case Check Dialog Box 3731
36.63. The Materials Tab in the Case Check Dialog Box 3734
36.64. The Solver Tab in the Case Check Dialog Box 3735
36.65. The Run Calculation Task Page with Solution Steering Enabled 3749
36.66. The Solution Steering Dialog Box 3751
36.67. The FMG Settings Tab in the Solution Steering Dialog Box 3753
38.1. Turbine Cascade Mesh Before Adaption 3774
38.2. Turbine Cascade Mesh after Adaption 3775
38.3. The Manual Mesh Adaption Dialog Box 3777
38.4. The Automatic Mesh Adaption Dialog Box 3778
38.5. The General Adaption Controls Dialog Box 3780
38.6. Additional Refinement Layers: 1, 2, 3 3781
38.7. The Manage Geometry-Based Adaption Dialog Box 3782
38.8. The Manage Adaption Criteria Dialog Box 3784
38.9. The Predefined Criteria Drop-Down List 3786
38.10. The Adaption Criteria Settings Dialog Box for the Cell Distance Criterion 3786
38.11. The Boundary Register Dialog Box for the Refinement Criterion 3787
38.12. The Predefined Criteria Drop-Down List 3790
38.13. The Field Variable Register Dialog Box for a Refinement Criterion 3791
38.14. The Predefined Criteria Drop-Down List 3792

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

38.15. The Adaption Criteria Settings Dialog Box for the Flame Indicator Criterion 3793
38.16. The Expression Dialog Box for the Refinement Criterion 3794
38.17. The Field Variable Register Dialog Box for the Refinement Expression 3795
38.18. The Predefined Criteria Drop-Down List 3796
38.19. The Adaption Criteria Settings Dialog Box for the VOF-to-DPM [Advanced] Criterion 3796
38.20. The Expression Dialog Box for the Refinement Criterion 3798
38.21. The Field Variable Register Dialog Box for the Refinement Criterion 3799
38.22. The Predefined Criteria Drop-Down List 3800
38.23. The Adaption Criteria Settings Dialog Box for the Overset Criterion 3800
38.24. Marking Boundary Cells 3802
38.25. Mesh Before Adaption 3802
38.26. Mesh after Boundary Adaption 3803
38.27. Wing Mesh Before Adaption 3804
38.28. Marking Cells Based on Region 3805
38.29. Wing Mesh After Region Cell Register-Based Adaption 3806
38.30. Field Variable Refinement Register 3807
38.31. Field Variable Coarsening Register 3807
38.32. Controls for Refining and Coarsening the Mesh 3808
38.33. Simple Expression Refinement Setting 3808
38.34. Cells Marked for Simple Expression Refinement 3809
38.35. Advanced Expression Refinement Setting 3809
38.36. Cells Marked for Advanced Expression Refinement 3810
38.37. The Geometry Based Adaption Dialog Box 3811
38.38. The Geometry Based Adaption Controls Dialog Box 3812
39.1. The Zone Surface Dialog Box 3816
39.2. Contours of Cell Partitions on Partition Surface Overlaid on Mesh 3817
39.3. The Partition Surface Dialog Box 3818
39.4. The Imprint Surface Dialog Box 3819
39.5. Imprinted Surface (pink) Superimposed Over Imported Surface (white) 3821
39.6. The Point Surface Dialog Box 3822
39.7. The Point Tool 3823
39.8. Using the Point Tool with Front Faces Transparent 3824
39.9. The Structural Point Surface Dialog Box 3825
39.10. The Line/Rake Surface Dialog Box 3827
39.11. The Line Tool 3829
39.12. The Plane Surface Dialog Box 3831
39.13. The Plane Tool for the YZ Plane Method 3834
39.14. The Plane Tool for the Point and Normal Method 3835
39.15. The Plane Tool for the Three Points Method 3835
39.16. The Plane Tool for Creating Multiple Planes 3836
39.17. The Quadric Surface Dialog Box 3837
39.18. The Iso-Surface Dialog Box 3839
39.19. External Wall Surface Isoclipped to Values of x Coordinate 3841
39.20. The Iso-Clip Dialog Box 3842
39.21. The Transform Surface Dialog Box 3844
39.22. The Expression Volume Dialog Box 3846
39.23. Example Temperature Contour on Volume Surface 3847
39.24. The Surfaces Dialog Box 3848
39.25. Example of Z-fighting 3850
39.26. Region Register Dialog Box 3852
39.27. Boundary Register Dialog Box 3854

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

39.28. Limit Register Dialog Box 3855
39.29. Field Variable Register Dialog Box 3857
39.30. Residual Register Dialog Box 3860
39.31. Volume Change—Ratio of the Volumes of the Cells 3861
39.32. Volume Register Dialog Box 3861
39.33. Airfoil Wall Cells Marked by Y+ Values 3863
39.34. Yplus/Ystar Register Dialog Box 3864
39.35. Manage Cell Registers Dialog Box 3865
39.36. Report Register Dialog Box 3866
39.37. Manage Register Operations Dialog Box 3867
40.1. Performance Preferences 3871
40.2. Outline Display 3872
40.3. Mesh Edge Display 3873
40.4. Mesh Face (Filled Mesh) Display 3874
40.5. Node Display 3875
40.6. The Mesh Display Dialog Box 3876
40.7. The Mesh Colors Dialog Box (Classic Color Scheme) 3878
40.8. The Mesh Colors Dialog Box (Pastel Color Scheme) 3878
40.9. The Zone Type Color and Material Assignment Dialog Box (Classic Color Scheme) 3879
40.10. Zone Type Color and Material Assignment Dialog Box (Pastel Color Scheme) 3879
40.11. Surface Rendering Properties Dialog Box (Classic Color Scheme) 3880
40.12. Mesh Display with Pastel Color Scheme 3881
40.13. Jeep with Multiple Materials Assigned 3882
40.14. Surface Rendering Properties Dialog Box 3884
40.15. Zone Type Color and Material Assignment Dialog Box 3885
40.16. Material Editor Dialog Box 3889
40.17. Transparent Mesh Display 3891
40.18. Global Mesh Display Dialog Box 3891
40.19. Mesh Display with Transparency: 2 Layers 3892
40.20. Mesh Display with Transparency: 3 Layers 3892
40.21. Standard Outline of Complex Duct 3893
40.22. Feature Outline of Complex Duct 3894
40.23. Mesh Display with Shrink Factor = 0 3895
40.24. Mesh Display with Shrink Factor = 0.01 3895
40.25. Contours of Static Pressure 3896
40.26. Profile Plot of y Velocity 3897
40.27. Coloring Surfaces by Static Pressure 3898
40.28. The Contours Dialog Box 3899
40.29. The Profile Options Dialog Box 3900
40.30. Filled Contours of Static Pressure 3902
40.31. Filled Contours with Clip to Range On 3903
40.32. Filled Contours with Clip to Range Off 3904
40.33. Contours of Velocity Magnitude on a Wall + Outlet: Node Values + Boundary Values Enabled 3906
40.34. Contours of Velocity Magnitude: Node Values Disabled + Boundary Values Enabled 3907
40.35. Contours of Velocity Magnitude: Node Values Enabled + Boundary Values Disabled 3908
40.36. Contours of Velocity Magnitude: Node Values Disabled + Boundary Values Disabled 3909
40.37. Velocity Vector Plot 3912
40.38. The Vectors Dialog Box 3913
40.39. The Vector Options Dialog Box 3915
40.40. Velocity Vectors Generated Using the In Plane Option 3916
40.41. 3D Arrow Vectors with Tessellation of 24 3918

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

40.42. 3D Arrow Vectors with Tessellation of 6 3918
40.43. The Custom Vectors Dialog Box 3920
40.44. The Vector Definitions Dialog Box 3921
40.45. Pathline Plot 3922
40.46. The Pathlines Dialog Box 3923
40.47. LICs Plot Colored by Velocity 3933
40.48. The Line Integral Convolutions Dialog Box 3934
40.49. Scene Consisting of Contours and Pathlines 3935
40.50. The Scene Dialog Box 3936
40.51. The Sweep Surface Dialog Box 3938
40.52. The Create Surface Dialog Box 3939
40.53. Example of Embedded Windows 3942
40.54. Dashboard Definition 3945
40.55. Dashboard with "none" for Base Window 3946
40.56. Embedding a Window - Outline View Context Menu 3948
40.57. Embedding a Window - Tab Context Menu 3949
40.58. Selecting a Dashboard to Animate 3950
40.59. Exploded Scene Display of Temperature and Velocity 3955
40.60. Outline View 'Display in' Example 3956
40.61. The Select Window Dialog Box 3956
40.62. Boundary Markers on an Inlet and Outlet 3958
40.63. Graphics Window with Text Annotation 3961
40.64. The Annotate Dialog Box 3962
40.65. Temporarily Hiding the Colormap 3964
40.66. The Colormap Dialog Box 3965
40.67. The Colormap Quick-Edit Panel 3966
40.68. The Colormap with All Titles and Skipped Labels 3981
40.69. The Colormap Editor Dialog Box 3982
40.70. Double-Click to Add Color Stops 3983
40.71. The Lights Dialog Box 3986
40.72. Model with Reflections Enabled 3991
40.73. Model with Static Shadows Enabled 3992
40.74. Model with Dynamic Shadows Enabled 3993
40.75. Model with Ground Plane Grid Displayed 3994
40.76. Model with All Graphics Effects Enabled 3995
40.77. Example of Realistic Rendering on a Highway Midday 3996
40.78. Raytracer Scene Display with Backplate 4008
40.79. Raytracer Scene Display with Embedded Plots 4009
40.80. Move Out Embedded Window 4009
40.81. Embed in Raytracing Window 4010
40.82. The Navigation Branch of Preferences 4021
40.83. Example Distance and Angle Measurement 4024
40.84. The Measure Dialog Box 4025
40.85. The Display States Dialog Box 4027
40.86. Using the Triad to Change the Orientation of the Object 4028
40.87. The Views Dialog Box 4029
40.88. The Camera Parameters Dialog Box 4030
40.89. Zooming In (Magnifying the Display) 4032
40.90. Zooming Out (Shrinking the Display) 4033
40.91. Camera Definition 4034
40.92. The Write Views Dialog Box 4036

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

40.93. Mirroring Across a Symmetry Boundary 4037
40.94. The Periodic Instancing Dialog Box 4037
40.95. The Views Dialog Box 4038
40.96. Before Applying Periodicity 4039
40.97. After Applying Periodicity 4039
40.98. The Periodic Instancing Dialog Box 4040
40.99. The Mirror Planes Dialog Box 4041
40.100. The Scene Description Dialog Box 4042
40.101. The Display Properties Dialog Box 4044
40.102. Velocity Vectors Translated Outside the Domain for Better Viewing 4046
40.103. The Transformations Dialog Box 4047
40.104. The Iso-Value Dialog Box 4048
40.105. The Pathline Attributes Dialog Box 4049
40.106. Graphics Display with Bounding Frame 4050
40.107. The Bounding Frame Dialog Box 4050
40.108. The Animate Dialog Box 4052
40.109. Sample XY Plot 4058
40.110. Sample Histogram 4058
40.111. Enabling Enhanced Plots 4059
40.112. The Solution XY Plot Dialog Box 4061
40.113. Geometry Used for XY Plot 4063
40.114. Data Plotted at Outlet Using a Plot Direction of (1,0,0) 4063
40.115. Data Plotted at Outlet Using a Plot Direction of (0,1,0) 4063
40.116. The Plot Data Sources Dialog Box 4065
40.117. The Plot Profile Data Dialog Box 4067
40.118. The Plot Interpolated Data Dialog Box 4068
40.119. Iso-Clips Created For Circumferential Averaging 4070
40.120. XY Plot of Circumferential Averages 4070
40.121. The Histogram Dialog Box 4073
40.122. The Axes Dialog Box 4074
40.123. The Curves Dialog Box 4076
40.124. The Fourier Transform Dialog Box 4082
40.125. The Plot/Modify Input Signal Dialog Box 4083
40.126. A-, B-, and C-Weighting Functions 4087
40.127. The Ansys Sound Analysis Dialog Box 4090
40.128. The Cumulative Plot Dialog Box 4094
41.1. Report Definitions Dialog Box 4102
41.2. Surface Report Definition Dialog Box 4104
41.3. Volume Report Definition Dialog Box 4105
41.4. Force Report Definition Dialog Box 4107
41.5. Drag Report Definition Dialog Box 4108
41.6. Lift Report Definition Dialog Box 4109
41.7. Moment Report Definition Dialog Box 4110
41.8. Flux Report Definition Dialog Box 4112
41.9. DPM Source Report Definition Dialog Box 4113
41.10. Cell Count Report Definition Dialog Box 4115
41.11. Face Count Report Definition Dialog Box 4116
41.12. Aerodamping (Travelling Wave Method) Report Definition Dialog Box 4117
41.13. DPM Report Definition Dialog Box 4119
41.14. User Defined Report Definition Dialog Box 4121
41.15. Expression Report Definition Dialog Box 4122

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

41.16. New Report File Dialog Box 4124
41.17. Report File Definitions Dialog Box 4125
41.18. Edit Report File Dialog Box 4126
41.19. New Report Plot Dialog Box 4126
41.20. Report Plot Definitions Dialog Box 4128
41.21. Edit Report Plot Dialog Box 4128
41.22. Plot with Instantaneous and Averaged Values 4130
41.23. The Flux Reports Dialog Box 4136
41.24. The Save Output Parameter Dialog Box 4138
41.25. The Flux Reports Dialog Box 4140
41.26. The Flux Reports Dialog Box with DPM 4141
41.27. A Two-Phase Flow with Two Discrete Phase Injections 4143
41.28. The Force Reports Dialog Box 4146
41.29. An Airfoil with its Computed Center of Pressure 4148
41.30. The Force Reports Dialog Box for a Center of Pressure Report 4149
41.31. The Projected Surface Areas Dialog Box 4150
41.32. The Surface Integrals Dialog Box 4151
41.33. The Volume Integrals Dialog Box 4153
41.34. The Device Efficiency Dialog Box 4155
41.35. Expression Dialog Box with Efficiency Calculation 4157
41.36. The Reference Values Task Page 4160
41.37. The Modified Settings Summary Table 4162
41.38. The Input Summary Dialog Box 4163
42.1. Computing Node Values 4168
42.2. Cylindrical Velocity Components in 3D, 2D, and Axisymmetric Domains 4170
42.3. The Custom Field Function Calculator Dialog Box 4287
42.4. The Field Function Definitions Dialog Box 4289
43.1. Ansys Fluent Architecture 4294
43.2. The Parallel Settings Tab of Fluent Launcher 4296
43.3. The Scheduler Tab of Fluent Launcher (Linux Version) 4299
43.4. The Remote Tab of Fluent Launcher 4302
43.5. Partitioning the Mesh 4316
43.6. The Auto Partition Mesh Dialog Box 4317
43.7. The Partitioning and Load Balancing Dialog Box 4320
43.8. The Weighting Tab in the Partitioning and Load Balancing Dialog Box 4323
43.9. The Partitioned Mesh 4326
43.10. The Partitioned ID Set to Zero 4327
43.11. The Partitioned ID Set to 1 4328
43.12. The Dynamic Load Balancing Tab 4330
43.13. Partitions Created with the Cartesian Axes Method 4334
43.14. Partitions Created with the Cartesian Strip or Cartesian X-Coordinate Method 4335
43.15. Partitions Created with the Principal Axes Method 4335
43.16. Partitions Created with the Principal Strip or Principal X-Coordinate Method 4336
43.17. Partitions Created with the Polar Axes or Polar Theta-Coordinate Method 4336
43.18. The Smooth Optimization Scheme 4337
43.19. The Merge Optimization Scheme 4337
43.20. The Thread Control Dialog Box 4345
43.21. The Parallel Connectivity Dialog Box 4346
44.1. The Workflow for Fluent on Arm Compute Nodes 4355
45.1. Accessing the Simulation Report Outline Task Page 4360
45.2. The Simulation Report Outline Task Page 4361

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

45.3. The Report Sections Outline View 4363
45.4. Drag-and-Drop Rearrangement of Nodes 4364
45.5. Drag-and-Drop Onto a Node 4365
45.6. Drag and Drop Indicators 4365
45.7. Positioning Between Two Nodes 4365
45.8. A Generated Report 4366
45.9. An Example of the System Information Report Section 4368
45.10. An Example of the Geometry and Mesh Report Section (Mesh Quality) 4368
45.11. An Example of the Geometry and Mesh Report Section (Orthogonal Quality) 4369
45.12. An Example of the Physics / Models Settings 4370
45.13. An Example of the Physics / Material Properties Settings 4370
45.14. An Example of the Physics / Cell Zone Conditions Section 4371
45.15. An Example of the Physics / Boundary Conditions Section 4372
45.16. An Example of the Physics / Reference Values Section 4373
45.17. An Example of the Solver Settings Section 4373
45.18. An Example of the Run Information Section 4374
45.19. An Example of the Solution Status Section 4374
45.20. An Example of the Named Expressions Section 4375
45.21. An Example of the Report Definitions Section 4375
45.22. An Example of the Plots Section (Residuals) 4376
45.23. Hovering and Showing Data in a Plot 4377
45.24. Hovering and Comparing Data in a Plot 4377
45.25. Example of Contours 4378
45.26. Example of Vectors 4379
45.27. Example of Pathlines 4379
45.28. Example of LICs 4380
45.29. Example of XY Plots 4381
45.30. Example of a Scene 4381
45.31. Example of a Histogram 4382
45.32. Example of an Animation 4383
45.33. Layout Settings for Result Plots 4384
45.34. Example of Contours (Columns) 4385
46.1. Various Views and Components of a Parametric Study 4387
46.2. An Example of the Optimization Results Table (AMOP) 4406
46.3. An Example of the Optimization Results Table (OCO) 4406
46.4. Transcript Tab 4419
46.5. Monitors Tab 4419
46.6. Residuals Tab 4420
46.7. Example of a Comparative Plot: Inlet Temperature vs Design Point 4422
46.8. Parametric Plots Dialog 4423
46.9. An Example of Conditional Formatting for the Design Point Table 4431
46.10. An Example of Using Both Slider Views for Pathline Comparison 4433
46.11. Results Comparison Tools 4434
46.12. Example of Using a Pinned Probe to Compare Pathline Results 4435
46.13. Example of Comparing XY Plots Between Design Points 4436
46.14. Preferences for Parametric Studies 4437
47.1. Ansys Fluent's Web Interface 4445
47.2. The Specify Web Server Details Dialog Box 4448
47.3. The Session Page for Ansys Fluent's Web Interface 4450
47.4. Components of Ansys Fluent's Web-Based User Interface 4451
47.5. An Example of Field-Level User Assistance 4451

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

47.6. Outline View 4455
47.7. A Contour Object Displayed in the Graphics Window 4456
47.8. A Context Menu for a Selected Boundary 4457
47.9. Accessing the Mouse Probe in the Graphics Window 4458
47.10. View Arc 4458
47.11. Displaying Session and Solution Information in the Status Bar 4459
47.12. Displaying Session Information in the Status Bar 4459
47.13. Displaying Solution Information in the Status Bar 4459
47.14. Stage Navigator 4460
47.15. The Console Window 4460
47.16. The Console Icon With Two Alerts 4460
47.17. The Console Window With Alerts 4460
47.18. Searching the Contents of the Console Window 4461
47.19. The Python Tab of the Console Window 4461
47.20. The Output Tab of the Console Window 4461
47.21. The Console Window with Python Command Access 4462
47.22. The Python Tab of the Console Window 4462
47.23. Results Arc 4463
47.24. Initializing or Running a Calculation 4463
47.25. Interrupting or Pausing a Calculation 4464
47.26. Interrupting or Resuming a Calculation 4464
47.27. Plots View 4466
47.28. Changing Units for a Given Property 4467
47.29. Using Expressions for Material Properties 4467
47.30. Using Expressions for Boundary Conditions 4468
47.31. Editing Multiple Boundary Conditions from the Context Menu 4468
47.32. An Example of the Multi-Edit Settings Panel 4468
47.33. An Example of Using the Multi-Edit Settings Panel 4469
47.34. Using The Context Menu to Create Report Definitions 4473
47.35. Using The Context Menu to Create Surfaces 4475
47.36. Pausing Graphics Rendering 4479
47.37. Creating Mirror Planes Using the Results Arc 4481
47.38. Accessing Mirror Planes Using the Results Arc 4481
47.39. Accessing Mirror Planes Using the Outline View 4481
47.40. The New Mirror Plane Dialog Box 4482
47.41. The New Mirror Plane Dialog Box 4482
47.42. Residual Plot 4484
47.43. A Report Definition Plot 4484
47.44. Displaying Multiple Plots (1x3) 4485
47.45. Analyzing Data for a Plot 4485
47.46. Building a Scene 4486
47.47. Reports 4486
47.48. Example of Input and Output Parameters 4488
48.1. Adjoint Observables Dialog Box 4516
48.2. Manage Adjoint Observables Dialog Box 4517
48.3. Adjoint Observables Dialog Box 4524
48.4. Adjoint Solution Methods Dialog Box 4527
48.5. The Stabilized Strategy and Scheme Settings Dialog Box 4533
48.6. The Dissipation Scheme Settings 4535
48.7. The Residual Minimization Scheme Settings 4536
48.8. Adjoint Residual Monitors Dialog Box 4537

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

48.9. Run Adjoint Calculation Dialog Box 4539
48.10. Adjoint Autosave Dialog Box 4541
48.11. Adjoint Postprocess Options Dialog Box 4542
48.12. Adjoint Reporting Dialog Box 4552
48.13. Design Tool Dialog Box 4554
48.14. A Cylindrical Region 4557
48.15. Iso-clip by Region dialog box 4561
48.16. The Design Condition Options Dialog Box 4563
48.17. Specifying a Bounding Plane for Design Changes 4564
48.18. The Bounding Orientation Dialog Box 4565
48.19. Fix Surfaces in the Morphing Region and Away From Applied Moving Conditions Dialog Box 4577
48.20. The Preview Morphing Dialog Box 4580
48.21. The Mesh History Dialog Box 4581
48.22. The Export STL Dialog Box 4583
48.23. Gradient-Based Optimizer Workflow 4584
48.24. Gradient-Based Optimizer Dialog Box 4586
48.25. Adjoint Optimizer Observables Dialog Box 4587
48.26. Adjoint Optimizer Conditions Dialog Box 4588
48.27. Adjoint Optimization History Monitor Dialog Box 4593
48.28. Adjoint Optimizer Autosave Dialog Box 4594
48.29. Optimize Dialog Box 4598
48.30. Gradient-Based Optimizer Dialog Box 4600
48.31. Optimizer Design Region Dialog Box 4604
48.32. Optimizer Design Variable Settings Dialog Box 4604
48.33. Design Variable Options Dialog Box 4605
48.34. Set Design Variable Limits Dialog Box 4606
48.35. Manage Training Data Dialog Box 4612
48.36. Parameterize and Explore Dialog Box 4614
48.37. The Regions Tab of the Mesh Morpher/Optimizer Dialog Box 4622
48.38. The Regions Tab of the Mesh Morpher/Optimizer Dialog Box for an Unstructured Distribution 4624
48.39. Displaying the Control Points for a Regular Distribution 4626
48.40. The Define Control Points Dialog Box 4628
48.41. Displaying the Control Points for an Unstructured Distribution 4630
48.42. The Constraints Tab of the Mesh Morpher/Optimizer Dialog Box 4632
48.43. The Deformation Tab of the Mesh Morpher/Optimizer Dialog Box 4633
48.44. The Parameter Bounds Dialog Box 4634
48.45. The Motion Settings Dialog Box for a Regular Distribution 4635
48.46. The Motion Settings Dialog Box for an Unstructured Distribution 4638
48.47. The Optimizer Tab of the Mesh Morpher/Optimizer Dialog Box 4641
48.48. The Objective Function Definition Dialog Box 4642
48.49. The Optimization History Monitor Dialog Box 4645
49.1. Exporting System Coupling Files from Workbench 4654
49.2. Force transferred to System Coupling when Porous Jump Thickness is Non-Zero 4657
49.3. Undeformed mesh 4664
49.4. Deformed mesh with diffusion smoothing 4664
49.5. Deformed mesh with linearly elastic smoothing and non-FSI boundary set to Unspecified 4665
49.6. Skewed prism cells due to translational motion between two bodies in close proximity 4667
49.7. Example geometry 4668
49.8. Cell Zone containing the inner boundary layer mesh 4669
49.9. Cell Zone containing the outer boundary layer mesh 4669
49.10. Cell Zone containing the interior mesh 4670

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

49.11. Prism layer mesh quality maintained for large deformations 4671
49.12. Overset mesh generated using three separate meshes 4672
49.13. Prism layer mesh quality is maintained 4672
49.14. Original Mesh (left) and deformed mesh (right) with smoothing and remeshing enabled 4674
49.15. Original Mesh (left) and deformed mesh (right) using smoothing and Region Face Remeshing 4675
51.1. The Basic Shapes Dialog Box 4867
51.2. Integration Surface Dialog Box 4868
51.3. Orientation Calculator Dialog Box 5149
51.4. The Acoustics Initialization Dialog Box 5183
52.1. Example with the Image Filter Set to **None** 5509
52.2. Example with the Image Filter Set to **Mild Sharpen** 5510
52.3. Example with the Image Filter Set to **Strong Sharpen** 5511
52.4. Example with the Image Filter Set to **Mild Emboss** 5512
52.5. Example with the Image Filter Set to **Strong Emboss** 5513
1. The CFF Case File Layout 5672
2. The CFF Data File Layout 5673
3. Variable Sized Data Appended to the Case File 5674
4. Variable Sized Data Appended to the Data File 5675
5. Quadrilateral Mesh 5685
6. Quadrilateral Mesh with Periodic Boundaries 5686
7. Quadrilateral Mesh with Hanging Nodes 5687

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

List of Tables

1.1. Fluent Capabilities by Licensing Level 149
1.2. Supported Versions of Third-Party Software 170
4.1. Available Command Line Options for Linux and Windows Platforms 192
6.1. Default Style Attributes 540
6.2. Orthogonal Quality Ranges and Cell Quality 566
6.3. Skewness Ranges and Cell Quality 567
7. Mini Flow Chart Symbol Descriptions ..... dcccxcvii
3.1. CGNS Variables Supported by Ansys Fluent 980
3.2. FEA File Extensions for FSI Mapping 1014
3.3. Units Associated with the Temperature Units Drop-Down List Selections 1018
5.1. Operations and Functions 1038
5.2. Solution Variables 1045
5.3. Scientific Constants 1046
5.4. Variable Aliases 1047
7.1. Zone Types by Category 1300
7.2. Air-side Radiator Data 1516
7.3. Reduced Radiator Data 1516
7.4. **CSV Profile Section Identifiers** 1568
7.5. Profile Types and the Corresponding Required Field Labels 1568
8.1. Recommended Settings for Operating Pressure 1679
8.2. Temperature Limits for Droplet Materials in Ansys Fluent Database prodb.scm 1699
8.3. v10.0 1702
14.1. Pitch Ratio of Ensemble Blade Rows with GTI 1962
16.1. NTU Model Vs. Simple Effectiveness Model 2281
18.1. Format of the Airfoil Data File 2344
18.2. VBM Field Variables 2350
18.3. VBM Global Outputs for Each Rotor 2352
19.1. Modified Specific Heat Capacity (Cp) Polynomial Coefficients 2391
23.1. Source Data Saved in Source Data Files 2632
24.1. Property Inputs for Inert Particles 2811
24.2. Property Inputs for Droplet Particles 2811
24.3. Property Inputs for Combusting Particles (Laws 1–4) 2812
24.4. Property Inputs for Combusting Particles (Law 5) 2813
24.5. Property Inputs for Multicomponent Particles (Law 7) 2814
24.6. Randomization of Particle Trajectories in Steady and Unsteady Cases 2833
24.7. General Particle Variables 2861
24.8. Particle Variables with Heat and Mass Transfer 2864
24.9. Particle Variables with the DEM model 2865
24.10. Particle Variables with Particle Rotation 2867
24.11. Particle Variables with User-Defined Particles 2867
24.12. Particle Variables with Particle Breakup 2867
24.13. Common Mean Diameters and Their Fields of Application 2883
26.1. Spatial Discretization Schemes for the VOF and Eulerian with Multi-Fluid VOF Models 2924
26.2. Spatial Discretization Schemes for the Eulerian Model without Multi-Fluid VOF 2925
26.3. Spatial Discretization Schemes for the Mixture Model 2925
26.4. Phase-Specific and Mixture Conditions for the VOF Model 2966
26.5. Phase-Specific and Mixture Conditions for the Mixture Model 2968
26.6. Phase-Specific and Mixture Conditions for the Eulerian Model (for Laminar Flow) 2971
26.7. Phase-Specific and Mixture Conditions for the Eulerian Model (with the Mixture Turbulence Model) .. 2972

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.8. Phase-Specific and Mixture Conditions for the Eulerian Model (with the Dispersed Turbulence Model) 2972
26.9. Phase-Specific and Mixture Conditions for the Eulerian Model (with the Per-Phase Turbulence Model) 2973
26.10. Open Channel Boundary Parameters for the VOF Model 2980
26.11. Slope Limiter Discretization Scheme 3017
26.12. Parameters for the Coalescence and Breakage Kernels 3044
26.13. Parameters for the Coalescence and Breakage Kernels 3073
26.14. Macros for Population Balance Variables Defined in `sg_pb.h` 3163
32.1. User-Defined Functions for the PEMFC Module 3438
32.2. User-Defined Scalar Allocations 3441
32.3. User-Defined Memory Allocations 3442
32.4. User-Defined Scalar Allocations 3475
32.5. User-Defined Memory Allocations 3475
32.6. User-Defined Memory Allocations 3484
32.7. User-Defined Scalar Allocations 3485
33.1. User-Defined Scalars in MHD Model 3503
33.2. MHD Vectors 3514
34.1. Source Terms and Corresponding UDFs 3525
40.1. Standard Views 4028
40.2. Numbers of Data Points Supported by the Prime-Factor FFT Algorithm 4080
40.3. Octave Band Frequencies and Weightings 4088
42.1. **Expressions** Category 4175
42.2. **Pressure** and **Density** Categories 4175
42.3. **Velocity** Category 4176
42.4. **Temperature**, **Radiation**, **Solidification/Melting**, and **Two-Temperature Model** Categories 4178
42.5. **Turbulence** Category 4180
42.6. **Runtime DFT** Category 4182
42.7. **Species**, **Reactions**, **Pdf**, and **Premixed Combustion** Categories 4182
42.8. **NOx**, **Soot**, and **Steady|Unsteady Statistics** Categories 4185
42.9. **Phases**, **Discrete Phase Model**, **Granular Pressure**, **Granular Temperature**, and **Wall Film** Categories 4187
42.10. **Properties** Category 4191
42.11. **Eulerian Wall Film** Category 4192
42.12. **Sensitivities** Category 4193
42.13. **Wall Fluxes**, **User Defined Scalars**, and **User Defined Memory** Categories 4196
42.14. **Cell Info** and **Mesh** Categories 4197
42.15. **Perforated Walls** Category 4200
42.16. **Mesh** Category (Turbomachinery-Specific Variables) 4200
42.17. **Residuals** Category 4201
42.18. **Derivatives** Category 4201
42.19. **Potential** Category 4202
42.20. **Lithium** Category 4203
42.21. **Acoustics** Category 4203
42.22. **Structure** Category 4205
42.23. **Aeromechanics** Category 4205
43.1. Examples for GPGPUs per Machine 4297
43.2. Supported Interconnects for the Windows Platform 4308
43.3. Available MPIs for Windows Platforms 4308
43.4. Supported MPIs for Windows Architectures (Per Interconnect) 4308
43.5. Supported Interconnects for Linux Platforms (Per Platform) 4313

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

43.6. Available MPIs for Linux Platforms 4313
43.7. Supported MPIs for Linux Architectures (Per Interconnect) 4314
49.1. Using the Fluent Input | Option setting to define custom input files 4652
49.2. Variables On Boundary Wall Regions 4654
49.3. Variables On Cell Zone Regions 4655
49.4. Variables On Porous Jump Boundary 4655
52.1. Default Display State Settings 5440
1. Moving Domain Models vs. Multiphase Models 5667
2. Multiphase Models vs. Turbulence Models 5667
3. Combustion Models vs. Multiphase Models 5667
4. Moving Domain Models vs. Turbulence Models 5668
5. Combustion Models vs. Moving Domain Models 5668
6. Combustion Models vs. Turbulence Models 5668
1. Summary of Basic CHEMKIN-CFD Parameters 5698
2. Summary of Advanced CHEMKIN-CFD Parameters 5699
3. Diagnostic Output Files Created During a CHEMKIN-CFD Run 5702
4. Error Messages that May Be Printed to the Fluent GUI 5704
5. Other Error Messages in `KINetics-log.txt` 5707

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Part I: Getting Started

The section describes getting started with Ansys Fluent.

- Introduction to Ansys Fluent (p. 141), introduction to Ansys Fluent.
- Basic Steps for CFD Analysis using Ansys Fluent (p. 171), basic steps for CFD analysis using Ansys Fluent.
- Guide to a Successful Simulation Using Ansys Fluent (p. 177), guide to a successful simulation using Ansys Fluent.
- Starting and Executing Ansys Fluent (p. 179), provides instructions for starting and executing Ansys Fluent.
- Glossary of Terms (p. 211), this glossary contains a listing of terms commonly used throughout the Ansys Fluent documentation.

Chapter 1: Introduction to Ansys Fluent

Ansys Fluent is a state-of-the-art computer program for modeling fluid flow, heat transfer, and chemical reactions in complex geometries.

Ansys Fluent is written in the C computer language and makes full use of the flexibility and power offered by the language. Consequently, true dynamic memory allocation, efficient data structures, and flexible solver control are all possible. In addition, Ansys Fluent uses a client/server architecture, which enables it to run as separate simultaneous processes on client desktop workstations and powerful compute servers. This architecture allows for efficient execution, interactive control, and complete flexibility between different types of machines or operating systems.

Ansys Fluent provides complete mesh flexibility, including the ability to solve your flow problems using unstructured meshes that can be generated about complex geometries with relative ease. Supported mesh types include 2D triangular/quadrilateral, 3D tetrahedral/hexahedral/pyramid/wedge/polyhedral, and mixed (hybrid) meshes. Ansys Fluent also enables you to refine or coarsen your mesh based on the flow solution.

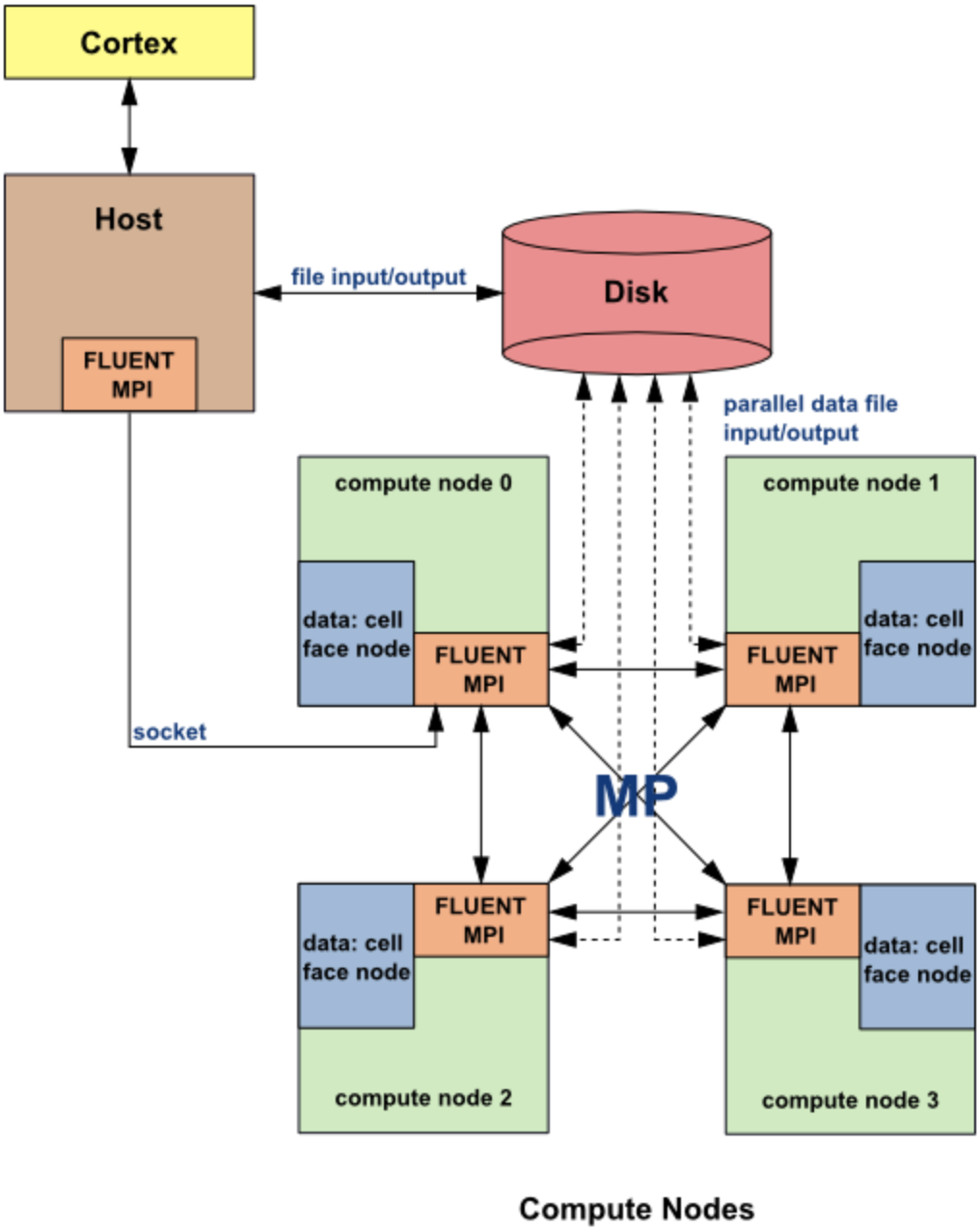
You can read your mesh into Ansys Fluent, or, for 3D geometries, create your mesh using the meshing mode of Fluent (see the Fluent User's Guide (p. 1) for further details). All remaining operations are performed within the solution mode of Fluent, including setting boundary conditions, defining fluid properties, executing the solution, refining the mesh, and postprocessing and viewing the results.

The Ansys Fluent serial solver manages file input and output, data storage, and flow field calculations using a single solver process on a single computer. Ansys Fluent also uses a utility called `cortex` that manages Ansys Fluent's user interface and basic graphical functions. Ansys Fluent's parallel solver enables you to compute a solution using multiple processes that may be executing on the same computer, or on different computers in a network.

The Ansys Fluent solver manages file input and output, data storage, and flow field calculations. Processing involves an interaction between Ansys Fluent, a host process, and one or more compute-node processes. Ansys Fluent interacts with the host process and the compute node(s) using a utility called `cortex`, which manages Ansys Fluent's user interface and basic graphical functions.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.1: Ansys Fluent Architecture



Ansys Fluent's serial solver uses a single compute node, whereas the parallel solver computes a solution using multiple compute nodes that may be executing on the same computer, or on different computers in a network.

For more information about Ansys Fluent's parallel processing capabilities, message passing interfaces (MPI), and so on, refer to Parallel Processing (p. 4293) in the User's Guide (p. 1).

All functions required to compute a solution and display the results are accessible in Ansys Fluent through an interactive interface.

For more information, see the following sections:

1.1. The Ansys Product Improvement Program

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.2. Program Capabilities
1.3. Known Limitations in Ansys Fluent 2024 R2
1.4. Supported Third-Party Software

1.1. The Ansys Product Improvement Program

This product is covered by the Ansys Product Improvement Program, which enables Ansys, Inc., to collect and analyze *anonymous* usage data reported by our software without affecting your work or product performance. Analyzing product usage data helps us to understand customer usage trends and patterns, interests, and quality or performance issues. The data enable us to develop or enhance product features that better address your needs.

How to Participate

The program is voluntary. To participate, select **Yes** when the Product Improvement Program dialog appears. Only then will collection of data for this product begin.

How the Program Works

After you agree to participate, the product collects anonymous usage data during each session. When you end the session, the collected data is sent to a secure server accessible only to authorized Ansys employees. After Ansys receives the data, various statistical measures such as distributions, counts, means, medians, modes, etc., are used to understand and analyze the data.

Data We Collect

The data we collect under the Ansys Product Improvement Program are limited. The types and amounts of collected data vary from product to product. Typically, the data fall into the categories listed here:

Hardware: Information about the hardware on which the product is running, such as the:

- brand and type of CPU
- number of processors available
- amount of memory available
- brand and type of graphics card

System: Configuration information about the system the product is running on, such as the:

- operating system and version
- country code
- time zone
- language used
- values of environment variables used by the product

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Session: Characteristics of the session, such as the:

- interactive or batch setting
- time duration
- total CPU time used
- product license and license settings being used
- product version and build identifiers
- command line options used
- number of processors used
- amount of memory used
- errors and warnings issued

Session Actions: Counts of certain user actions during a session, such as the number of:

- project saves
- restarts
- meshing, solving, postprocessing, etc., actions
- times the Help system is used
- times wizards are used
- toolbar selections

Model: Statistics of the model used in the simulation, such as the:

- number and types of entities used, such as nodes, elements, cells, surfaces, primitives, etc.
- number of material types, loading types, boundary conditions, species, etc.
- number and types of coordinate systems used
- system of units used
- dimensionality (1-D, 2-D, 3-D)

Analysis: Characteristics of the analysis, such as the:

- physics types used
- linear and nonlinear behaviors
- time and frequency domains (static, steady-state, transient, modal, harmonic, etc.)
- analysis options used

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution: Characteristics of the solution performed, including:

- the choice of solvers and solver options
- the solution controls used, such as convergence criteria, precision settings, and tuning options
- solver statistics such as the number of equations, number of load steps, number of design points, etc.

Specialty: Special options or features used, such as:

- user-provided plug-ins and routines
- coupling of analyses with other Ansys products

Data We Do Not Collect

The Product Improvement Program does *not* collect any information that can identify you personally, your company, or your intellectual property. This includes, but is not limited to:

- names, addresses, or usernames
- file names, part names, or other user-supplied labels
- geometry- or design-specific inputs, such as coordinate values or locations, thicknesses, or other dimensional values
- actual values of material properties, loadings, or any other real-valued user-supplied data

In addition to collecting only anonymous data, we make no record of where we collect data from. We therefore cannot associate collected data with any specific customer, company, or location.

Opting Out of the Program

You may *stop* your participation in the program any time you wish. To do so, select **Ansys Product Improvement Program** from the Help menu. A dialog appears and asks if you want to continue participating in the program. Select **No** and then click **OK**. Data will no longer be collected or sent.

The Ansys, Inc., Privacy Policy

All Ansys products are covered by the Ansys, Inc., Privacy Policy.

Frequently Asked Questions

1. *Am I required to participate in this program?*

   No, your participation is voluntary. We encourage you to participate, however, as it helps us create products that will better meet your future needs.

2. *Am I automatically enrolled in this program?*

   No. You are not enrolled unless you explicitly agree to participate.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. *Does participating in this program put my intellectual property at risk of being collected or discovered by Ansys?*

   No. We do not collect any project-specific, company-specific, or model-specific information.

4. *Can I stop participating even after I agree to participate?*

   Yes, you can stop participating at any time. To do so, select **Ansys Product Improvement Program** from the Help menu. A dialog appears and asks if you want to continue participating in the program. Select **No** and then click **OK**. Data will no longer be collected or sent.

5. *Will participation in the program slow the performance of the product?*

   No, the data collection does not affect the product performance in any significant way. The amount of data collected is very small.

6. *How frequently is data collected and sent to Ansys servers?*

   The data is collected during each use session of the product. The collected data is sent to a secure server once per session, when you exit the product.

7. *Is this program available in all Ansys products?*

   Not at this time, although we are adding it to more of our products at each release. The program is available in a product only if this *Ansys Product Improvement Program* description appears in the product documentation, as it does here for this product.

8. *If I enroll in the program for this product, am I automatically enrolled in the program for the other Ansys products I use on the same machine?*

   Yes. Your enrollment choice applies to all Ansys products you use on the same machine. Similarly, if you end your enrollment in the program for one product, you end your enrollment for all Ansys products on that machine.

9. *How is enrollment in the Product Improvement Program determined if I use Ansys products in a cluster?*

   In a cluster configuration, the Product Improvement Program enrollment is determined by the host machine setting.

10. *Can I easily opt out of the Product Improvement Program for all clients in my network installation?*

    Yes. Perform the following steps on the file server:

    a. Navigate to the installation directory: `[Drive:]\v242\commonfiles\globalsettings`

    b. Open the file **ANSYSProductImprovementProgram.txt**.

    c. Change the value from "on" to "off" and save the file.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.2. Program Capabilities

When in meshing mode, Ansys Fluent functions as a robust unstructured-volume-mesh generator (see Meshing Mode Capabilities (p. 218) in the Fluent User's Guide (p. 1) for further details). When in solution mode, Fluent allows you to simulate the following:

- 2D planar, 2D axisymmetric, 2D axisymmetric with swirl (rotationally symmetric), and 3D flows
- Flows on quadrilateral, triangular, hexahedral (brick), tetrahedral, wedge, pyramid, polyhedral, and mixed element meshes
- Steady-state or transient flows
- Incompressible or compressible flows, including all speed regimes (low subsonic, transonic, supersonic, and hypersonic flows)
- Inviscid, laminar, and turbulent flows
- Newtonian or non-Newtonian flows
- Ideal or real gases
- Heat transfer, including forced, natural, and mixed convection, conjugate (solid/fluid) heat transfer, and radiation
- Chemical species mixing and reaction, including homogeneous and heterogeneous combustion models and surface deposition/reaction models
- Free surface and multiphase models for gas-liquid, gas-solid, and liquid-solid flows
- Lagrangian trajectory calculations for dispersed phase (particles/droplets/bubbles), including coupling with continuous phase and spray modeling
- Cavitation model simulations
- Melting/solidification applications using the phase change model
- Porous media with non-isotropic permeability, inertial resistance, solid heat conduction, and porous-face pressure jump conditions
- Lumped parameter models for fans, pumps, radiators, and heat exchangers
- Acoustic models for predicting flow-induced noise
- Inertial (stationary) or non-inertial (rotating or accelerating) reference frames
- Multiple moving frames using multiple reference frame (MRF) and sliding mesh options
- Mixing-plane model simulations of rotor-stator interactions, torque converters, and similar turbomachinery applications with options for mass conservation and swirl conservation
- Dynamic mesh model simulations for domains with moving and deforming meshes
- Volumetric sources of mass, momentum, heat, and chemical species

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Simulations that use a material property database
- Simulations in which the design is revised or optimized, using the adjoint solver or the mesh morpher/optimizer
- Simulations customized by user-defined functions
- Dynamic (two-way) coupling with GT-POWER and WAVE
- Simulations that use the following add-on modules:
  - Battery module
  - Continuous fiber module
  - Macroscopic particle model (MPM) module
  - Fuel cell modules
  - Magnetohydrodynamics (MHD) module
  - Population balance module
- Fluent Icing

Ansys Fluent is available at three different licensing levels, which control the availability of the above-mentioned features and functionalities. These licensing levels are summarized below, see Table 1.1: Fluent Capabilities by Licensing Level (p. 149) for more detail:

- **Enterprise**—full access to all Fluent Meshing and Fluent solver capabilities, as documented in the Fluent User's Guide. Additionally, this licensing level provides access to: Ansys CFX, Ansys Chemkin, Ansys Reaction Workbench, Ansys Energico, Ansys SpaceClaim, Ansys Discovery Modeler, Ansys ICEM CFD, Ansys Meshing, Ansys EnSight Enterprise, Ansys DesignXplorer, Ansys Forte, Ansys Polyflow, Ansys TurboGrid, Ansys FENSAP-ICE, the Model Fuel Library, the Fluent workspaces, and the Fluent Native GPU solver.
- **Premium**—full access to all Fluent Meshing and Fluent solver capabilities, as documented in the Fluent User's Guide. Additionally, this licensing level provides access to: Ansys CFX, Ansys SpaceClaim, Ansys Discovery Modeler, Ansys ICEM CFD, Ansys Meshing, Ansys EnSight Enterprise, Ansys DesignXplorer, Ansys Forte, Ansys TurboGrid, and Ansys Polyflow Pro.
- **Pro**—access to the Water-Tight Meshing workflow in Fluent Meshing and access to a reduced set of Fluent solver capabilities allowing the solution of incompressible and compressible steady-state and transient, single-phase, turbulent, non-reacting flows, and heat transfer. Additionally, this licensing level provides access to: Ansys Meshing and Ansys Polyflow Pro (including Ansys CFD-Post for Polyflow postprocessing)

  Note that only supported options and functionalities are displayed in the interface.

  Important:

  If you require the ability to specify environment variables while running under a **Pro** license, you must define the environment variable on the system (prior to launching Fluent). *This*

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

allows you to, for example, run Fluent with the interface translated to Japanese using the environment variable `lang=ja`.

Note:

The Ansys Student license supports reading cases containing up to 1 million mesh cells.

Table 1.1: Fluent Capabilities by Licensing Level

Capability	Pro	Premium	Enterprise
CAD Import	✓	✓	✓
Steady-state flow and heat transfer (excluding radiation)	✓	✓	✓
Pressure-based Coupled and Segregated solvers	✓*	✓	✓
Basic turbulence models - inviscid, laminar, k-epsilon, k-omega (standard and SST) and Spalart-Allmaras	✓	✓	✓
HPC: 4HPC cores included, additional core accessed via Ansys HPC licensing	✓	✓	✓
Conjugate Heat Transfer (CHT) in solids	✓	✓	✓
Porous media (isotropic, orthotropic, and conical)	✓	✓	✓
Fluid state models: Incompressible Compressible (ideal gas), Bousinesq approximation, non-Newtonian fluids	✓	✓	✓
Rotating reference frames, multiple reference frames (without periodic interfaces)	✓	✓	✓
Inert and Massless particle tracking	✓	✓	✓
2D fan model	✓	✓	✓
Multi-stream mixing (multiple non-reacting species)	✓	✓	✓
Fluent Meshing: Watertight Meshing workflow including Polyhedra, Poly-Hexcore with Mosaic technology, Tetrahedral, and Prism	✓	✓	✓
Fluent Setup and Postprocessing, including Reports	✓	✓	✓
Parameters and expressions	✓	✓	✓
Concurrent Design Point Updates for Parametric Studies		✓	✓
Workbench integration		✓	✓
Fluent batch solving		✓	✓
Fluent Meshing: Parallel meshing and Fault Tolerant meshing		✓	✓
Physics: Transient flows	✓	✓	✓
Physics: Advanced turbulence models		✓	✓
Physics: Density-based solver, adjoint solver, 6-DOF solve for flow-driven solid motion		✓	✓
Physics: Multiphase including free-surface, cavitation, phase change models, thin film model, transition to /from particles, boiling model, surface tension, reactions, granular model, and Dense Particulate Coupling (DDPM)		✓	✓
Physics: Full particle tracking including break-up, coalescence, erosion, evaporation, thin wall films, reactions and macroscopic model		✓	✓

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Capability	Pro	Premium	Enterprise
Physics: Combustion and reacting flows including pollutants/soot modeling, FGM and Finite Rate combustion models, effusion cooling models		✓	✓
Physics: Shell conduction heat transfer	✓	✓	✓
Physics: Thermal radiation models, heat exchanger model		✓	✓
Physics: Real fluid properties (for example, steam, cryogenic fluids)		✓	✓
Physics: Periodic interfaces, turbo interfaces including mixing plane and pitch change		✓	✓
Physics: Turbomachinery blade flutter analysis and blade film cooling model		✓	✓
Physics: Dynamic mesh, overset mesh, and mesh adaption (including Polyhedral Unstructured Mesh Adaption)		✓	✓
Physics: Acoustics		✓	✓
Physics: Electro-chemistry and battery models		✓	✓
Native GPU Solver			✓

(*) With the Pro license, only the Coupled pressure-velocity coupling scheme is supported for steady-state cases, and only the SIMPLE pressure-velocity coupling scheme is supported for transient cases.

Ansys Fluent is ideally suited for incompressible and compressible fluid-flow simulations in complex geometries. Ansys Fluent's parallel solver enables you to compute solutions for cases with very large meshes on multiple processors, either on the same computer or on different computers in a network. Ansys, Inc. also offers other solvers that address different flow regimes and incorporate alternative physical models. Additional CFD programs from Ansys, Inc. include CFX, Ansys Icepak, and Ansys Polyflow.

1.3. Known Limitations in Ansys Fluent 2024 R2

This section lists limitations that are known to exist in Ansys Fluent. Where possible, suggested work-arounds are provided.

- Graphical User Interface

  – For 4K monitors with a resolution higher than 1920x1080, when invoking field-level user assistance ("?") in the Fluent Meshing workflows, the help window cannot be automatically resized, causing the text to be cut off and scroll bars to be added. (1006416)

  – (Linux only) ANSYS Fluent does not support AMD Radeon Pro graphics cards and the Fluent session may close unexpectedly. (348500)

  – Dragging the ANSYS Fluent application between 4K and non-4K displays may result in the application not scaling correctly. As a workaround, restart ANSYS Fluent after relocating it to a new screen to make it appear as expected. (345352)

  – ANSYS Fluent may hang unexpectedly if you have a non-default layout, such as tabbing the console and the graphics window, and you force the MSW graphics driver, either by using `HOOPS_PICTURE=msw/win` or by launching with the `-driver msw` option. (269471)

  – (Windows on 4K monitors) You may need to adjust your display scaling and layout settings to optimize the fit of the Fluent application on your monitor.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - (Linux Only) Automatic scaling of the Ansys Fluent application window and graphics window interactivity are not compatible when viewed on a 4K monitor. If the application is scaled properly, interactions in the graphics window may not behave as expected. Set `QT_AUTO_SCREEN_SCALE_FACTOR` equal to `1` to have the application scale properly. (143229)

- File import/export (for a list of supported files, refer to the table in this section, under **Third-party software**)

  - (Red Hat 8.8 only) Avoid the import/export of CGNS files as CGNS is not supported on Red Hat and it will cause the Fluent session to close unexpectedly. (882695)

  - (1-way Fluent to Mechanical data transfer using the legacy .dat or .dat.gz file formats) For Fluent multiphase cases, pressure values may not appear in Ansys Mechanical. As a workaround, use the following steps:

    1. Right-click the Fluent Solution cell and select Clear Generated Data.

    2. Update the Fluent Solution cell. Ensure you are generating the legacy .dat.gz file format.

    3. Update the Mechanical Setup cell.

  - For some cases with wall/wall-shadow pairs, the variables exported on these surfaces to Common Fluids Format - Post (CFF-Post) may be incorrectly populated with zero-values when, in reality, the variable does not have a value at this location (for example, wall shear stress on a coupled wall shadow). This can lead to differences when comparing postprocessing images and values between Fluent and other postprocessors such as EnSight or CFD-Post. In such comparisons, the postprocessing values seen in Fluent results should be the ones considered as correct. (419385)

  - You cannot read a default CFF case file (.cas.h5) when creating a **bounded-by-surfaces** design condition (as part of the adjoint design tool) or when creating an imprinted surface. **Workaround**: Legacy case files (`.cas`) can be read in these circumstances, so you can save your `.cas.h5` file as a `.cas` file by opening it in a Fluent session and disabling the writing of CFF files (by entering the following text command: `file/cff-files? no`). (199821)

  - (Windows only) You cannot export files larger than 2GB to CGNS format. As a workaround, use a Linux machine to export files larger than 2GB to CGNS format. (259220)

  - Surfaces exported to EnSight format may contain a surface with an ID of zero. This is OK for importing into EnSight, but it will cause an error if you are importing into a non-ANSYS third-party postprocessing software, such as Paraview. As a workaround, create a zone surface for the surface with an ID of zero and export the newly created surface instead of the original. (254966)

  - CGNS files exported from TurboGrid 2019 R3 cannot be read into ANSYS Fluent unless the **Split Blade Faces By Geometry** option is disabled in TurboGrid before you export the CGNS files. You can disable this option in TurboGrid by right-clicking **3D Mesh** in the **Mesh** tab, selecting **Edit in Command Editor** and setting **Split Blade Faces By Geometry** to **false**. (121053)

  - If you change the **File Storage Options** settings in the **Autosave** dialog box, the solution history will be lost.

  - When exporting EnSight Case Gold files for transient simulations, the solver cannot be switched between serial and parallel, and the number of compute nodes cannot be changed for a given

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

parallel run. Otherwise, the exported EnSight Case Gold files for each time step will not be compatible.

– EnSight export with topology changes is not supported.

– To properly view Fieldview Unstructured (`.fvuns`) results from a serial or parallel Ansys Fluent simulation:

→ Mesh files must be exported using the `fieldview-unstruct-grid` text command.

→ Mesh and data files should all be exported from parallel Ansys Fluent sessions with the same number of nodes.

– Tecplot file import does not support the Tecplot360 file format.

– The maximum number of profiles that can be read into a single Fluent session is 50.

– The PARALLEL INDEPENDENT mode for Common Fluids File (CFF) file I/O is known to exhibit slow write performance. On parallel file systems, consider using the PARALLEL COLLECTIVE mode when writing CFF files. On other network file systems, consider using the HOST or NODE0 mode.

– If you are accessing a file using a Universal Naming Convention (UNC) path, you must ensure that you have permission to access to all of the folders in the path or you will not be able to open the file.

– (Windows only) The file filter available in the **Select File** dialog box may not work as expected, requiring you to manually select the desired file(s) without filtering aid. (174291)

– Files written by Ansys products do not support synchronization with Microsoft's OneDrive file hosting service. (187717 / 74067)

• Mesh

– When slitting a coupled wall, Fluent will terminate abnormally if the coupled wall includes a node that is also shared by two or more periodic boundary zone pairs. Note that a slitting operation is performed not only when you use the `mesh/modify-zones/slit-face-zone` text command, but also when you specify a time-averaged explicit thermal coupling (for example, by using the `define/models/cht/explicit-time-averaged-coupling/conformal-coupled-walls` text command). Workaround: decouple all of the periodic pairs that share the node (for example, by using the `define/boundary-conditions/modify-zones/slit-periodic` text command) before you slit the coupled wall. (528636)

– When using the gap model with a flow-blocking gap region or when using contact detection in a dynamic mesh with the contact marks method selected for flow control, you must ensure that none of the cells that are marked as part of the blocked region are adjacent to a periodic boundary zone, as this will result in Fluent reporting an error during initialization or the calculation. (498569)

– If your license preferences are set to **Share a single license between applications when possible** (under **Tools**>**License Preferences**...) and you have a Fluent Meshing session open, then you open the Fluent Solution workspace, Fluent Meshing hangs without a chance for you to save your session. As a workaround, change your license preference to **Use a separate license for each application**, which will prevent you from opening another Fluent session if there is already one open. (147834)

– Hover-over highlighting and Boundary Markers are not available in Meshing Mode. (241447)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - Boundary zone extrusion is not possible from faces that have hanging nodes.

  - For simulations that involve the Fluent, Mechanical, and Meshing applications, meshing problems can arise in instances where there are multiple regions and contacts between them. In Fluent, a zone can only exist in a single contact region. The Mechanical and Meshing applications both use a different approach concerning contact regions when compared to Fluent.

  - Ansys Fluent does not support FSI data mapping of edges and, therefore, it is not supported in 2D.

  - At non-conformal interfaces, the **Matching** option is no longer allowed with the **Mapped** option. When opening a case set up in a previous release with both options enabled, you will be prompted to recreate the interface without the **Matching** option.

  - If your mesh topology has a step-wise prism mesh near the walls, do not use node-based gradients with MUSCL.

- Models

  - Mass-Weighted Average report definitions are not supported for reduced order models because the ROM only has data available on cell face centers and cell centers. Instead, use Area-Weighted Average, Facet Average, Facet Minimum, or Facet Maximum report definitions. (281540)

  - The physical velocity porous formulation may produce non-physical flow fields and poor convergence when porous resistance (Inertial or Viscous) values are less than or comparable to the change in the dynamic pressure across the porous interface (interior face zone separating the porous and non-porous cell zone). Switching the porous interface zone to a porous jump boundary is an effective way to overcome this issue. (61773)

  - Ansys Fluent supports the Chemkin II format for Oppdif flamelet import only.

  - The surface-to-surface (S2S) radiation model does not work with moving/deforming meshes.

  - The DPM work pile algorithm is not compatible with the wall film boundary condition.

  - For transient Lagrangian multiphase analysis with DPM unsteady particle tracking, atomizer or cone injections will release particles from the same position rather than from random positions in one of the following cases:

    → **Particle Time Step Size** (set in the **Discrete Phase Model** dialog box) is smaller than **Time Step Size** (set in the **Run Calculation** task page)

    → Re-randomization of the initial particle positions from iteration to iteration is disabled using the following text command:

    ```
    define/models/dpm/numerics/re-randomize-every-iteration? [yes] no
    ```

    To use random starting points for the particles, enter the following scheme commands in the Ansys Fluent console:

    ```
    (rpsetvar 'dpm/random/seed-timestep-corrected? #t)

    (dpm-parameters-changed)
    ```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that this change will also affect particle trajectories if stochastic tracking, particle breakup, or other mechanisms that involve random numbers are used. (176638)

- The shell conduction model is not applicable on moving walls.
- The heat exchanger model is not compatible with mesh adaption.
- The Fluent/Ansys Reaction Design KINetics coupling is not available on the win64 platform.
- DO-Energy coupling is recommended for large optical thickness cases (> 10) only.
- FMG initialization is not available with the shell conduction model.
- FMG initialization is not compatible with the unsteady solver.
- The MHD module is not compatible with Eulerian multiphase models.
- Bounded 2nd order discretization in time is not compatible with moving and deforming meshes.
- When simulating porous media, the value of the **Porosity** (defined in the **Fluid** dialog box) cannot be `0` or `1` (that is, it must be in between these values) if the non-equilibrium thermal model is enabled.
- When simulating porous media, the non-equilibrium thermal model is not supported with radiation and/or multiphase models.
- For porous media simulations, the relative velocity resistance formulation is not supported with axisymmetric-swirl when there are non-zero swirl resistances.
- The junction of a wall with shell conduction enabled and a non-conformal coupled wall is not supported. Such a junction will not be thermally connected, that is, there will be no heat transfer between the shell and the mesh interface wall.
- After you enable the Eulerian Wall Film model, Fluent will not allow you to save the mesh modifications, such as separating cells, extruding face zones, and changing the cell zones type. If you want to modify the mesh in Fluent, be sure to complete all mesh operations prior to enabling the Eulerian Wall Film model.
- The Transition SST model (also known as the γ-Re_θ model) is not Galilean invariant and should therefore not be applied to surfaces that move relative to the coordinate system for which the velocity field is computed; for such cases, the Intermittency Transition model (also known as the γ model) should be used instead.
- User-defined wall functions are not compatible with the Eulerian multiphase formulation and cannot be used.
- The view factor files generated as part of a surface-to-surface radiation model calculation for version 16.0 or 16.1 of Fluent may not be compatible with newer versions if the **Matching** option was enabled for a mapped interface. For any case file with such a setup, you must recompute the view factors in the newer version to ensure correct results.

- Parametric Studies

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - Since saved journals are designed to be run in a new Fluent session and not in a session where a parametric study is initialized, journals saved as part of a parametric study (using the Save Journal button in the Parametric ribbon) may cause errors. Contact technical support for a workaround. (932695)

- Parallel processing

  - The discrete transfer radiation model (DTRM) is unavailable in the parallel solver.

  - Note that on systems using large pages for memory allocation, the virtual memory usage reported by Fluent may be much higher than actual memory used. In this case resident memory (also reported by Fluent) is a more reliable guide.

  - The Eulerian Wall-film model is not compatible with the DPM Domain option of the hybrid parallel DPM tracking. For such model combination, the **Use DPM Domain** option must be disabled in the **Parallel** tab of the **Discrete Phase Model** dialog box.

  - Starting in version 18.0, Ansys Fluent will require approximately 60 MB more memory per node process compared to version 17.0–17.2.

  - If you have OpenSSH inside C:\windows\system32 and want to run a mixed Windows-Linux simulation, Fluent may not be able to locate and execute the ssh command, and the following warning will be printed:

    ```
    'ssh' is not recognized as an internal or external command, operable
    program or batch file.
    ```

    You must specify the path of the actual ssh to be used: launching from the command line, use the `-rsh` option (for example, `-rsh=c:\cygwin64\bin\ssh.exe`); using Fluent Launcher, select **Other** from the **Remote Spawn Control** list in the **Remote** tab and enter the path in the text box (for example, `c:\cygwin64\bin\ssh.exe`).

    In addition, you should add the location of `ssh` executable to the beginning of the PATH environment variable. (184255/176971)

  - Memory usage may increase compared to previous releases for parallel simulations that use the InfiniBand interconnect. This can be particularly problematic for large cases that are run on a cluster, as you may run out of memory. The memory usage increase is more prominent during intensive input/output (I/O), such as the frequent writing of CFF data files (`.dat.h5`). **Workaround:** use the ethernet interconnect, or if that is not possible, reduce the frequency of I/O operations during the solution and/or stop the calculation when the memory starts to increase and then restart. For details on changing the interconnect see Starting Parallel Ansys Fluent Using Fluent Launcher (p. 4296), Starting Parallel Ansys Fluent on a Windows System Using Command Line Options (p. 4306), and/or Starting Parallel Ansys Fluent on a Linux System Using Command Line Options (p. 4311). (597320 / 610518)

  - Using dynamic process spawning requires that you manually specify the interconnect as `-peth-ernet` for shared memory or distributed memory without InfiniBand, and `-pib.ofed` for distributed memory over InfiniBand.

- Message Passing Interfaces (MPIs)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - For certain Mellanox OFED driver versions or UCX versions, users may need to specify `-pib.ofed` instead of just `-pib` to run Fluent with Infiniband. Most users will not require this workaround.

  - The IBM MPI has been discontinued by the vendor, and is no longer available in the standard installation of Fluent. The Intel MPI is now the default MPI for Windows (with less than 64 logical cores) and Linux, and is known to have the following limitations:

    → The default Intel MPI in Fluent release 2023 R2 fails to install the hydra service when installing Intel MPI from the Ansys Installation Launcher on Windows. Workaround: You must make sure that the appropriate services have been configured to run by the administrator as follows:

    ```
    "c:\Program Files (x86)\Intel\oneAPI\mpi\2021.8.0\bin\hydra_service.exe" -install
    ```

    (812613 / 791313)

    → On Windows or mixed Windows / Linux, it is not possible to dynamically spawn additional processes when switching from meshing mode to solution mode (either by setting the **Meshing Processes** to a lower value than the **Solver Processes** in Fluent Launcher or by using the `/parallel/spawn-solver-process` text command in meshing mode). If you are operating at the **Pro** capability level and you launch with **Solver Processes** > **1**, those cores will also be started in the **Meshing** workspace, but the meshing will occur in serial because parallel meshing is not supported at the **Pro** capability level. (236195 / 257722)

    → By default, the Intel MPI may fail when mixing hardware for compute nodes. **Workaround:** Use the following environment setting:

    ```
    I_MPI_PLATFORM none
    ```

- Platform support and drivers

  - On Windows Server OS, Ansys Fluent supports only MS MPI for parallel runs. Installing the Intel MPI library will result in conflicts.

  - On Windows 7 and later, installing Ansys Fluent on any drive other than `C:` may result in issues arising from spaces in the pathname not getting converted to short file names. This is the result of a change in the default value for `NtfsDisable8dot3NameCreation` starting with Windows 7. If you need to install Ansys Fluent on any drive other than `C:` you must run the following command *prior* to installing Ansys Fluent:

    ```
    fsutil 8dot3name set <driveletter> 0
    ```

    where <*driveletter*> is the target drive letter including the colon (for example, **D:**).

  - The minimum OS requirements for Linux are SLES 11 SP2 or RHEL 6. Note that SLES 12.5 is not supported.

  - The pathname length to the cpropep.so library (including the lib name) is limited to 80 characters. (Linux Opteron cluster using Infiniband interconnect only.)

  - On Linux platforms, including a space character in the current working directory path is not supported.

  - If you are installing Ansys Fluent 2024 R2 on a Windows machine that already has one or more previous versions of Ansys Fluent, then after installing the Intel MPI library from the prerequisites,

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

make sure to delete the environment variable I_MPI_ROOT. Otherwise there will be a conflict while running previous Ansys Fluent versions in parallel mode.

– Remote Solver Facility (RSF) is no longer supported in Ansys Fluent.

– Itanium platform (lnia64) is no longer supported.

– Ansys Fluent uses several TCP/IP ports for communications and error handling. Port conflicts with other programs trying to use the same ports are handled by Ansys Fluent and generate warnings similar to the following:

```
428: mpt_accept: warning: incorrect exercise message "GET /" from 10.1.0.188 on port 56564
```

Long running large sessions are more prone to generating such warnings, but these are generally safe for you to ignore.

– If you lock the computer screen before the Fluent graphics are initialized, the Fluent session will not launch if you are using the OpenGL graphics driver. To avoid this issue with the OpenGL driver, you can use the alternative drivers `x11` or `null` for Linux/unix and `msw` or `null` for Windows. You can specify an alternate graphics driver either by defining it in the `HOOPS_PICTURE` environment variable or using the `-driver` Fluent command line option.

– Fluent may terminate abnormally during launch when running on Community Enterprise OS (CentOS) 7.3 or Red Hat Enterprise Linux (RHEL) 7.3 when DISPLAY is set to a Virtual Network Computing (VNC) session. To attempt to resolve this, verify that you are using a supported graphics card and update the graphics card drivers (directly from the graphics card vendor website). If the issue persists, you can do one of the following: set the DISPLAY to a local machine; set the `LD_PRELOAD` environment variable to `/usr/lib64/libstdc++.so.6`; or use the alternative drivers `x11` or `null` (either by defining it in the `HOOPS_PICTURE` environment variable or using the `-driver` Fluent command line option).

- Remote display

  – Running Ansys Fluent on an accelerated remote display (including but not limited to DCV and EOD) with the OpenGL2 graphics driver may result in Fluent closing unexpectedly. If you experience this issue, change to the OpenGL1 graphics driver (display/set/rendering-options/driver/opengl). (283643)

  – Connecting or disconnecting a VPN network while running a Fluent simulation will result in a failure because of changes to the network interface.

  – If you experience an abnormal termination when running Fluent via a remote display, check you graphics card to ensure that you have a modern professional graphics card that is up-to-date (that is, the latest updated driver for that card, which is available on the company's website). If your system does not meet the graphics card requirements, launch Fluent using a completely software-based driver, such as MSW (Windows) or X11 (Linux).

  – (Exceed onDemand and VNC Viewer) The **Num Lock** number keys and arrows may not function in the graphical user interface or console. As a workaround, you can define the `XKB_DEFAULT_RULES` environment variable set equal to `base`, which resolves the arrow keys' functionality. Number entries must still be completed using the number keys at the top of the standard keyboard. (172528)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - (Linux only) When running Ansys Fluent using the Virtual Network Computing (VNC), Nice DCV, or Exceed on Demand (EoD) applications, you may experience unexpected error messages due to a 3rd party issue. To resolve these errors, install the following packages: (185796)

    → xcb-util-wm-0.4.1-5.el6.x86_64

    → xcb-util-keysyms-0.4.0-1.el6.x86_64

    → xcb-util-0.4.0-2.2.el6.x86_64

    → compat-xcb-util-0.4.0-2.2.el6.x86_64

    → xcb-util-image-0.4.0-3.el6.x86_64

  - (Linux only) On some clusters without accelerated graphics, Fluent may not accept keyboard inputs. If you encounter this behavior, set the `QT_XKB_CONFIG_ROOT` environment variable equal to `/usr/share/X11/xkb`.

- Cell Zones and Boundary Conditions

  - A reference frame is always displayed at its initial state (position and orientation) when displayed from the **Reference Frame** dialog box. While running a transient simulation for multiple time steps, a reference frame is displayed at its current state. After the calculation is completed, if you open the **Reference Frame** dialog box and display it, the reference frame triad will move back to its initial state. (184248)

  - Clicking **OK** within the **Reference Frame** dialog box will reset a reference frame to its initial state. If you do not intend to make changes to a reference frame, leave the dialog box by clicking **Cancel**. (184248)

  - Note that Mesh Motion doesn't work with 6DOF. If you want to have relative motion with respect to a six DOF zone you need to create a Rigid body dynamic zone.

- Reference Frames

  - Fluid zones designated as 3D fan zones cannot have non-conformal interfaces.

- Solver

  - When using the standard CPU-driven solver, you may get erroneous results for cases that have both compressible and incompressible flow zones. Affected field variables include temperature and velocity. Workaround: Use the Fluent native GPU solver instead. (892103)

  - If you have enabled the application of poor mesh numerics using criteria based on the cell gradient quality, the number of cells to which it is applied may vary when performing single-precision calculations with different numbers of processes. Workaround: Use the double-precision solver. (362753)

  - The absolute and relative velocity formulations may yield different results in cases where a strong reversal of flow exists at a pressure outlet boundary.

  - The non-iterative time advancement (NITA) solver is applicable with only a limited set of models. See the Ansys Fluent User's Guide for more details.

  - NITA (using fractional time step method) is not compatible with porous media.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The following models are not available for the density-based solvers:
  - → Volume-of-fluid (VOF) model
  - → Multiphase mixture model
  - → Eulerian multiphase model
  - → Non-premixed combustion model
  - → Premixed combustion model
  - → Partially premixed combustion model
  - → Composition PDF transport model
  - → Soot model
  - → Rosseland radiation model
  - → Melting/solidification model
  - → Enhanced Coherent Flamelet model
  - → Inert model: transport of inert species (EGR in IC engines)
  - → Dense discrete phase model
  - → Shell conduction model
  - → Floating operating pressure
  - → Spark ignition and auto-ignition models
  - → Physical velocity formulation for porous media
  - → Selective multigrid (SAMG)
- The pressure-based coupled solver is not available with the following features:
  - → Fixed velocity
- On some Linux platforms, pressing **Ctrl**+**C** will not interrupt the solution. A suggested workaround is to use the checkpoint mechanism in Fluent to save files and/or exit Fluent. (Checkpointing an Ansys Fluent Simulation in the *Fluent User's Guide* (p. 208))
- In certain cases with tetrahedral or hybrid meshes, the use of the Least-Squares Cell Based gradient method in combination with the cell-to-cell limiter may cause divergence. If this is observed, it is recommended that you either change the gradient method to Green-Gauss Node Based or change the limiter type to the cell-to-face limiter.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– Beginning in version 17.0, the warped-face gradient correction (WFGC) is not supported with shell conduction if the ability to define multi-layer shells has been disabled through the `define/models/shell-conduction/multi-layer-shell` text command.

– For transient and time-dependent cases, the solution advances to the next time step based on various inputs including time step size and convergence criteria, but it is not directly dependent on the flow time interval specified when 'flow-time' is selected for monitoring (report definitions, autosaving, transient export and so on). For these flow time dependent operations, the action will occur when the flow time meets or exceeds the specified flow time interval.

– Note the following issues affecting residuals:

  → Restarting cases that have the **High Order Term Relaxation** option enabled may produce a small residual jump after the restart. (178223)

  → Restarting cases that have turbulent flow with a wall-function-based boundary treatment and a **No Slip** shear condition at the wall may produce a slightly different residual history compared to a continuous run. (165935)

  These issues can occasionally impact the residual history of long transient simulations (such as those that use the LES, SRS, or SBES model) when the solutions for each time step are not deeply converged.

– For case files created in Release 19.2 or earlier and are steady, single phase, and use the pressure-based solver, the **Density** explicit relaxation factor in the **Solution Controls** task page is set to `1` in the following circumstances, even though it should be set to `0.25` (in order to match how a case file created in Release 2019 R1 would behave):

  → if the physics includes reacting flow and/or species transport together with a pseudo time method, and you click the **Default** button in the **Solution Controls** task page

  → if you newly select one or both of the following so that both are used:

    • a reacting flow model and/or the species transport model

    • a pseudo time method

  Work-around: Manually enter `0.25` for **Density** in the **Solution Controls** task page. (182453)

– The following text command works appropriately for setting limits to default values, but fails to take action on solution controls and AMG controls:

```
> solve/set/set-controls-to-default
Set solution controls to default? [no] yes
Set AMG Controls to default? [no] yes
Set limits to default values? [no] yes
```

  Work-around: Click the **Default** button in the **Solution Controls** task page and the **Multigrid** tab of the **Advanced Solution Controls** dialog box, respectively. As noted previously, for older steady-state, single-phase cases that use the pressure-based solver and involve reacting flow and/or species transport with a pseudo time method selected, you must also manually set the **Density** explicit relaxation factor to `0.25` in the **Solution Controls** task page. (183926)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - The use of **Fractional Step** pressure-velocity coupling scheme with dynamic mesh layering is not supported. To run this combination, enter the following Scheme command before running the calculation: `(rpsetvar 'dynamesh/layering/layering-before-move? #t)`. (125886)

- User-Defined

  - For expressions evaluated for zone motion (moving reference frame or moving mesh), "Time" evaluates to the previous time step's flow time, whereas the "Time" argument of a DEFINE_TRANSIENT_PROFILE used for zone motion will have the current time step's flow time. (147162)

  - When you use curly brackets "{}" to specify the name of a defined object, such as a report definition or a cell register, automatic suggestions are disabled and you must provide the display name of that object. (118322)

  - Expressions defined for fields in the **Fixed Values** tab of the **Fluid** and **Solid** cell zone dialog boxes cannot include units in the expression definition. (146295)

  - Any scripts or journals that attempt to add menu items to Fluent pull-down menus (which have been replaced with the Fluent ribbon) will no longer work. You must create separate user-defined menus to house all user-defined menu items. For additional information about user-defined menus, see Adding Menus to the Right of the Ribbon in the *Fluent Customization Manual*.

- User-defined functions (UDFs)

  - Interpreted UDFs cannot be used with an Infiniband interconnect. The compiled UDF approach must be used instead.

  - The **Visual Studio Express 2015 for Windows** installer on Windows 10 installs libraries in non-standard locations. To ensure the use of Microsoft Visual Studio for UDF compiling (rather than the built-in compiler provided with the Fluent installation), you must instead use the **Visual Studio Express 2015 for Desktop** installer, or manually set the library path based on your local installation (for example, LIB="C:\Program Files (x86)\Microsoft Visual Studio 14.0\VC\lib\onecore\amd64";%LIB%).

- Graphics, Reporting, and Postprocessing

  - (Postprocessing application, Linux only) Updates to the location of user-defined surfaces (such as lines) may not show if you only redisplay the scene containing the surface(s). Redisplay the surface(s) after modifying them to ensure they are displayed correctly in all containing graphics objects. (450062)

  - Data files do not store derived quantities, such as pressure gradients, so reading a data file and immediately displaying pressure gradients results will incorrectly show zeros. Run the solver for an iteration and redisplay to see the correct values. (774389)

  - (Parallel only) In some instances, the location selected using the mouse-probe in the graphics window may vary substantially from the location the code selects as the corresponding surface vertex. To make the mouse-probe and corresponding surface vertex match more closely, as they do when running with a single processor, disable the **Remove Partition Lines** option in the **Graphics** branch of **Preferences** (**File** > **Preferences...**). (458754)

  - (Linux only) Postprocessing objects displayed with the Node Values option disabled may not show all of the selected surfaces due to a third-party issue. As a workaround, use the OpenGL graphics driver. (347474)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– Results visualized on bounded plane surfaces or on iso-clips of a plane may incorrectly imply that some hex mesh cells are triangles, when they are not. This is not known to impact results displayed on these surfaces. Planes created with the Bounded option disabled do not experience this issue. (380979)

– If holes or unexpected dark spots appear when postprocessing on plane surfaces (such as displaying a contour plot), you can create an iso-surface to display the desired quantity. *Refer to Iso-surfaces (p. 3838) for additional information on creating iso-surfaces.*

– (Linux only) Ansys Fluent does not support AMD Radeon Pro graphics cards and the Fluent session may close unexpectedly. (348500)

– (Linux only) Contour lines are not displayed when dynamic shadows are enabled. To make the contour lines display, you can disabled dynamic shadows either in the graphics window toolbar or in Preferences. (343770)

– Keyframe **Scene Animations** of pathlines are only available for the non-persistent/"global" version of the **Pathlines** dialog box that does not have a **Name** field. This version of the dialog box is opened as described below: (269934)

  → Right-clicking **Pathlines** in the Outline View tree and selecting **Edit...** (located under the **Results** branch).

  → Clicking **Pathlines** and selecting **Edit...** under **Graphics** in the **Results** ribbon tab.

  Note:

  You must enable **Expose legacy non-persistent graphics** in the **Graphics** branch of **Preferences** (**File** > **Preferences...**) to see the **Edit...** option for pathlines as described above.

– Unsteady statistics mean and rms values for custom field functions defined for dp/dt may show as zero. Contact support for a workaround. (199006)

– (2D Only) Box select is only working for selecting surfaces if you drag from right-to-left. However, you can enable **Edges** or **Faces** in the **Mesh Display** dialog box to restore left-to-right box selection. You can also enable **Show model edges** in the **Appearance** branch of **Preferences** (accessed via **File**>**Preferences...**) to change the default mesh display behavior. (259419)

– When you use curly brackets "{}" to specify the name of a defined object, such as another report definition, automatic suggestions are disabled and you must provide the display name of that object. (118322)

– When exporting a filled contour with contour lines to AVZ format, the contour lines will not be visible in the Ansys Viewer. As a workaround, you can create a scene containing 2 duplicate contour plots—one with filled contours, one with just contour lines (not filled). You can also disable the colormap for one of the plots so the plot only contains a single colormap. (137138)

– Monitors may continue to print/plot values, even if the zones on which they are defined are deactivated.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you are autosaving multiple scenes on a Windows machine, the **Headlight** lighting effect may inconsistently change its state (on/off). This can be avoided by rendering each scene in a separate graphics window.

- Mean and root-mean-squared-error (RMSE) quantities of custom field functions are only available for mixtures. In previous releases it was possible to specify these quantities for phases, which was an incorrect behavior. This behavior is no longer allowed in R16.0 or later releases. If you are running a pre-R16.0 case set to output such quantities in R2024 R2, you may get a segmentation error. To avoid the error, redefine the previously defined monitors reporting mean or RMSE quantities of phases.

- The mouse-annotate feature is no longer available. Annotations can still be created using the Annotate dialog box (see Annotate Dialog Box in the *Fluent User's Guide* (p. 5286) for additional information).

- Beginning in version 15.0, if a flux report for the heat transfer rate is generated on the wall of a moving solid, the reported values will include the convective heat flux due to the motion of the solid. Depending on the mesh and quality of the geometry representation, this may present flux values that are different than the flux specified in the boundary condition definition (for example, a non-zero flux may be reported for an adiabatic wall).

- If you import a case file that was created prior to Release 18.0 and that contains multiple monitors plotting in the same window, you must review the setup to ensure each report plot is assigned to a different window before running the calculation. If the plot windows are not reassigned, then plots assigned to the same window will be lost.

- It is possible to use text commands to create contour, vector, mesh, pathline, particle track, XY plot, and scene graphics objects with spaces in the name (for example, through the `display/objects/create` text command); however, objects with such names cannot be displayed using the `display/objects/display` text command, and attempting to do so will only result in the printing of an error. As a workaround, you can create graphics objects without spaces in the name or use the graphical user interface to display graphics objects with spaces in the name.

- In rare cases, the **Curve Length X Axis Function** for XY plots may not plot correctly, even if the curvilinear surface is piecewise linear and appears to be a single closed curve. A workaround is to use the **Direction Vector X Axis Function**.

- Transient statistics (Mean and RMS) reported for Fluent quantities that are nonlinear functions of the underlying solution variables represent evaluations of those quantities using the Mean or RMS values of the underlying solution variables. For instance, **Mean Velocity Magnitude** is computed as the magnitude of a vector constructed from the mean velocity components, and **Mean Pressure Coefficient** is computed as the pressure coefficient computed using the mean pressure. To construct the true Mean and/or RMS values of such quantities, you can define a custom field function and collect transient statistics of the custom field function. For example, define a custom field function **vmag_cff** = **sqrt (Vx ^ 2 + Vy ^ 2 + Vz ^ 2)**, and report Mean and RMS of **vmag_cff**.

- Scene animations created using **Key Frames** in the **Animate** dialog box are not compatible with graphics displays on isosurfaces (contours, vectors, pathlines, particle tracks). Pathlines are not compatible with scene animations, regardless of the selected surface(s).

- When meshes contain a large number of cells (for example, ~150 million cells or higher), the meshing mode of Fluent may report an incorrect number of skewed cells, based on an incorrect

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

inverse orthoskew (IOS) value. As a workaround, you can use the `(tgapi-util-set-number-of-parallel-compute-threads 1)` command. (168453)

– If you create an XY plot for display on a rake surface and you use the savable XY plot graphics object, accessed by clicking **New...** in the ribbon or outline view, then the results may not be displayed immediately. If they do not appear, click **Curves...** in the **XY Plot** dialog box and select a **Symbol** from the drop-down list in the **Marker Style** group box, click **Apply**, then redisplay the XY plot. (180547)

  If you are viewing cell values (**Node Values** disabled) on an XY plot of a rake surface and the points are being shown as a continuous line, then you can change the **Pattern** to "empty" in the **Curves - Solution XY Plot** dialog box (**Line Style** group box). (180547)

– The transform operation in the **Transform Surface** dialog box is not available for user-created surfaces such as lines, points, iso-surfaces, and so on. To create a transformed line, point, iso-surface, or other user-created surface, you must manually translate the input point(s) and create a new line/point/iso-surface in the respective dialog box (**Line/Rake**, **Point Surface**, **Iso-Surface**, and so on).

– Density contour plots that include a solid region in the display will include the solid zone(s) in any range calculations and will show a density for the solid that does not reflect the actual case setup. Fluid zone densities are still displayed correctly. Selecting **Density...** and **Density All** in the **Contours of** drop-down lists will correctly display density values for solids and fluids. (155346)

– Special characters (/^*%@,<>{}()?&~!=) should not be used in object names: they can affect how an object is rendered in the graphics display, and may make it so the object does not appear at all. (157473)

  Having special characters in object names can also lead to inconsistent report file names, causing validation issues when you attempt to start a calculation, therefore you should avoid using special characters for objects. (531947)

– For annotations, the **foreground** and **background** color options for text are not in sync with those specified in **Preferences**, but are instead controlled using the `display/set/colors/foreground` and `display/set/colors/background` text commands, respectively.

– For any meshing mode or solution mode session that displays graphics in the graphics window (including when running a batch job with the `-gu` command line option) and/or saves picture files, the rendering / saving speed will be significantly slower if you do not follow all of the following best practices:

  → Run Cortex on a suitable machine with an appropriate graphics card and the latest drivers (for details, see the Ansys website). Note that you can assign Cortex to a particular machine using the `-gui_machine=<hostname>` command line option, or by selecting **Specify Machine** from the **Graphics Display Machine** list in the **Scheduler** tab of Fluent Launcher.

  → Ensure that Cortex / the host process is run on a separate machine than that used for compute node 0. For example, do not include the machine assigned using the `-gui_machine` option as the first machine in the hosts file / machine list (specified using the `-cnf=x` command line option).

  → Do not set the graphics driver to `null`, `x11` (for Linux), or `msw` (for Windows).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

→ When saving picture files, enable the **Fast hardcopy** option in the **Preferences** dialog box (under **Graphics**).

- Fluent in Workbench

  – In a Workbench project, if your Fluent case and data files are in the default common fluids format (CFF) and you also export a .cdat file, you may get an error message if you attempt to postprocess the results using CFD-Post. This is because when a .cdat file is exported, a legacy case file (.cas) is also written as the latest case, and so paired with the CFF data file (.dat.h5), and it is not possible to use a combination of legacy and CFF files in CFD-Post. Workaround: Manually write a new .cas.h5 and .dat.h5 file after the .cdat file is exported. (310992)

  – When using Fluent in Workbench, importing Fluent Case and Data into a Fluent system will cause project updates to fail if **Submit** to **Design Point Service** is chosen for **Update Option**. As a workaround, you can right-click the **Solution** cell and choose**Refresh** prior to the project update, or select **RSM** under **Update Option** in the **Project Schematic Properties**. Projects that use the Import Fluent Case functionality only (without importing Data) or that start from Geometry/Mesh without Case file import are unaffected by this limitation. (277667)

  – The **Automatic Skip** option for colormaps is not functioning as intended. As a workaround, you can manually specify the skip value. (273989)

  – Fluent cases with either of:

    → Legacy (pre-18.0) solution monitors that get converted to report definitions.

    → Report file objects in which no filename is specified (from any release).
    may fail if updated in Workbench with the following message: "Update failed for the Solution component in Fluent. Value cannot be null. Parameter name: fileName". To avoid this issue you can first read the case into Fluent and either 1) delete the report file or 2) specify a valid filename for the report file. (146221)

  – Coupling between Fluent and HFSS or Q3D Extractor is not supported.

  – For two-way coupling between Maxwell and Fluent, by default Fluent uses the following zones: when mapping volumetric losses, the same list of zones that you selected for receiving volumetric losses in Fluent's **Maxwell Mapping Volumetric** dialog box are used; and when mapping surface losses, all cell zones are used. To change the zones that are used for feedback mapping, you can use a Scheme command, shown in the following example. This example specifies that only cell zone ID 1 and 2 are used: `(em-set-feedback-map-cell-zone '(1 2))`. Note that you can only specify the IDs of the cell zones as the arguments.

  – In a Fluent analysis system, the **Clear Generated Data** option for the **Solution** cell will not clear the files associated with animations. To have the **Clear Generated Data** option clear the animation files as well, you must define the `FLUENT_WB_REMOVE ALL_GENERATE_FILES` as a system environment variable on your local machine, prior to opening Workbench.

  – Fluent design point studies using the **Remote Solve Manager** may not completely update due to errors. Should this happen, you can rerun the affected design points and they will update as expected (you may have to rerun failed design points multiple times). (350696, 385734, 553033)

- Help

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - (Ubuntu systems only) The following help links in the Fluent Launcher and Fluent Solution workspace are not working and the correct location for these links is provided. As a workaround, you can define the `LD_PRELOAD` environment set equal to `/usr/lib/x86-64-linux-gnu/libstdc++.so.6` prior to starting Fluent: (736116)

    → Fluent Release Notes

    → Fluent Migration Manual

    → Ansys News Center

    → Ansys Blog

    → Fluent Documentation

    → Tutorials

    → Videos

    → Ansys Learning Hub

    → Ansys Training Center

    → Online Technical Resources

- Other

  - Do not use `inf` as the name for any boundaries, cell zones, or objects (report definitions, graphics objects, and so on) due to the conflict of 'inf' representing "infinity" in the Python language.

  - In some cases, you may see a warning when accessing Preferences after loading ACT at startup. As a workaround to open Preferences, close Fluent, disable Load ACT in the Fluent Launcher and start Fluent. Once Fluent is running you can modify Preferences and click OK to save your changes. (462205)

  - Performance on a Windows high performance computing (HPC) cluster may suffer due to poor affinity handling by Microsoft HPC Pack. Workaround: You can try running with the Microsoft HPC affinity management disabled, by selecting No jobs from the Affinity Type list in the Affinity tab of the Job Scheduler Configuration. (395881 / 400841)

  - The following combination is not supported: Open MPI + Infiniband + Mellanox OFED 5.1. Workaround: Either use the Intel MPI or upgrade to Mellanox OFED 5.2. (444933)

  - Images of embedded graphics windows captured using the Save Picture dialog box are saved without borders on the embedded windows, regardless of the setting specified in Preferences. (336851)

  - (Operating under the **Pro** license) While the **user-defined** specification method is available for some fields, such as material properties, you cannot load user-defined functions, as they are not supported at the Pro capability level. (321331)

  - Exports of Common Fluids Format - Post files from Fluent may incorrectly specify the pressure gradient as zero. Contact support for a workaround. (239127, 243318)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If the network connection is lost during a serial or parallel calculation, the Fluent session may terminate abnormally.
- The IRIS Image and HPGL hard copy formats are no longer supported in Ansys Fluent.
- When using Ansys Fluent with the Remote Solve Manager (RSM):

  → Only one copy of a saved project that is in the pending state can reconnect successfully.

  → Maxwell coupling is not supported.

  → UDFs are supported with limitations as detailed in Submitting Fluent Jobs to Remote Solve Manager in Workbench User's Guide.
- The turbo-averaged contour plot in turbomachinery postprocessing may give an unexpected contour region in a selected topology.
- The **Inverse Distance** and **Least Squares** profile interpolation methods are not applicable when a profile is attached to cell zones.
- When opening Ansys Help from Fluent in Linux, you may receive an error message in the Linux console. This can result when another user has created the installation and run Fluent, thus creating a registry file; if you then run this same installation, there will be a permissions conflict. As a workaround, remove the registry file:

  path`/ansys_inc/v242/Tools/mono/Linux64/etc/mono/registry`

  (where *path* is the directory in which you have placed the release directory). Then change the permissions for the Mono platform in order to remove write access from the directory:

  path`/ansys_inc/v242/Tools/mono/Linux64/etc/mono`
- On Windows, mesh reading into a serial Ansys Fluent session may fail if you use more than 20 million cells per core.
- Fluent does not support non-ASCII characters in the names of files, zones, and boundaries.
- When exiting a Fluent session on Linux that was started with the Load ACT option, Fluent may become unresponsive. If this occurs, the Fluent process must be manually terminated. In the console, the error message specifies that a corrupted double-linked list is responsible for the error. Because this unexpected shutdown occurs after the project has been saved, no data is lost.
- When Fluent is run on Linux with the Load ACT option, Fluent may repeatedly issue the following warning during solution iteration:

  ```
  Unexpected error checking licensing server.
  ```

  This warning is harmless and does not impact Fluent or ACT usage.
- When running a stand-alone instance of Fluent in a mixed Windows / Linux configuration or from a remote Windows installation, ACT does not open. To correct the problem, you must set the `AWP_ROOT242` environment variable to point to the Ansys installation directory.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - ACT with Ansys Fluent is disabled on Linux systems starting in Release 19.2. To use ACT with Ansys Fluent, use Release 19.2 or later on Windows or use Release 19.1 on Linux. (177173)

  - If the network connection is lost during a serial or parallel calculation, the Ansys Fluent session may terminate abnormally.

  - Ansys Fluent uses several TCP/IP ports for communications and error handling. Port conflicts with other programs trying to use the same ports are handled by Ansys Fluent and generate warnings similar to the following

    ```
    428: mpt_accept: warning: incorrect exercise message "GET /" from 10.1.0.188 on port 56564
    ```

    Long running large sessions are more prone to generating such warnings, but these are generally save for you to ignore.

- Remote Visualization Client

  - When running ANSYS Fluent utilizing VirtualGL `vglrun` or EOD `ssrun`, Fluent may hang when writing image files. As a workaround, disable **Use hardware acceleration** under **Save Picture Settings** in the **Graphics** branch of **Preferences** (accessed via **File**>**Preferences**...). (252910)

  - For a client session with multiple connected servers, only one server can receive commands through the **Send Command to Server** dialog box—that server is whichever one you send a command to first using this dialog box. As a workaround, you can use the Python console to control the other connected servers. (177632)

- Fluent Meshing

  - When using previous versions of Fluent Meshing (version 2024 R2 or earlier), if an overlap in the display of the workflow tree/outline view and the graphics window occurs, simply remove the `.cxlayout.ini` file located in the home directory.

  - Limitations related to the Fluent guided meshing workflows are documented separately. See Limitations of the Fluent Guided Workflows (p. 279) for details.

  - When creating a rapid octree mesh, while creating a new mesh object is available in the graphical user interface, you are only able to perform the same operation in the text user interface using the following command(s):

    ```
    /parallel/auto-partition? no
    /parallel/agglomerate yes
    /objects/create
    ```

  - When using the Fault-tolerant Meshing workflow, Fluent Meshing may encounter problems when reading mesh files while using large machine configurations, for example, 256 core machines or larger.

  - Performing a mesh check on a polyhexcore mesh may not yield satisfactory results if there are any isolated cells present. Isolated cells can be removed using the **`(%tg-delete-isolated-cells (get-cell-zones-of-filter '*))`**command in the console window.

  - *Before* importing SpaceClaim files (`.scdoc`) into Fluent Meshing, you need to first enable the **Set as the alternative license preference** license preference option in SpaceClaim in order to uphold licensing preferences.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– Using journal (`*.jou`) or workflow (`*.wft`) files from previous releases may slow or even halt the volume mesh generation process. **Workaround:** For journal files, make sure that you add version-specific information to the beginning of your journal files (for example: `/file/set-tui-version "19.5"`). For workflow files, make sure that you have the **Quality Warning Limit** property in the **Create Volume Mesh** task set to a value closer to 0 (such as 0.05) instead of a value near 1. (163320)

– While writing case files in the CFF format (`.cas.h5`) in a mixed Windows/Linux environment, Fluent Meshing will only write files on the Linux nodes. To write such case files properly, you have to assign a mapped driver on the Windows side as the working directory, and then assign the remote working directory to the same path, where the mapped driver path is derived from.

– (Linux only) Geometries imported from Ansys Mechanical, Ansys DesignModeler, and Ansys Meshing depend on the Linux `xmessage` command. The following RedHat package managers should be installed before runtime:

  → For SLES, xmessage-1.0.4-5.58.x86_64 (version may differ)

  → For RHEL, xorg-x11-apps

– Meshes generated from Ansys Meshing that are then imported into Fluent Meshing may not include all interior faces. **Workaround:** In Ansys Meshing, ensure that all interior surfaces are assigned to be internal faces (that is, put into a Named Selection that includes the name "internal") before generating the mesh, such that all internal surfaces will then be accounted for when the mesh is imported into Fluent Meshing.

– After Ansys Fluent 19.2, you can no longer have multiple interior zones defined in a single cell zone. If your case file has such zones, they will not appear as available zones. **Workaround:** Read the mesh file into Ansys Fluent 19.2, and set the type for the interior zones to be 'wall' in order to restore the missing zones. (143347)

  For example, to read files created in an older version of Ansys Fluent Meshing (or even Workbench), perform the following:

```
Meshing>file read-options
Enforce mesh topology [no]
Check read deata [no]
Correct Interiors? [yes] no
```

  After reading the mesh, change the problematic boundary's type to 'internal' and save the mesh file.

1.4. Supported Third-Party Software

- Fluent-Platform LSF integration is not supported on the MS Windows platform.
- Fluent-SGE integration is supported only on Linux platforms.
- Wave and GT-POWER coupling are available only with stand-alone Ansys Fluent and not in the Workbench environment.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Supported versions of third-party software are listed below:

Table 1.2: Supported Versions of Third-Party Software

Third-Party Software	Supported Version
Abaqus	6.14 ODB Library: 6.14.5
Altair HYPERMESH	5.1
CGNS	4.3
EnSight 6 (TUI only)	Ensight 6
EnSight Case Gold	10.1.6
FAST	1.3
Fieldview	16.0
GT-POWER	2022.3
Hierarchical Data Format version 5 (HDF5)	1.12.2
HOOPS	23.00-1
Intel MPI	Linux: 2021.11 Windows: 2021.8
libpng	1.6.18
LSTC-DYNA	970.0
Microsoft MPI	10.0
MPCCI	3.0.5
NASTRAN	Bulk data input file - MSC.NASTRAN 2010 OUTPUT2 data file - NX/NASTRAN 10
NIST	10.0
Open MPI	4.0.5
Sundials	2.5.0
TECPLOT	Tecplot file format, version 11.2
VKI	4.4.6.r1
WAVE	2020.2
zlib	1.2.8

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 2: Basic Steps for CFD Analysis using Ansys Fluent

Before you begin your CFD analysis using Ansys Fluent, careful consideration of the following issues will contribute significantly to the success of your modeling effort. Also, when you are planning a CFD project, be sure to take advantage of the customer support available to all Ansys Fluent users.

For more information, see the following sections:

2.1. Steps in Solving Your CFD Problem
2.2. Planning Your CFD Analysis

2.1. Steps in Solving Your CFD Problem

Once you have determined the important features of the problem you want to solve, follow the basic procedural steps shown below.

1. Define the modeling goals.
2. Create the model geometry and mesh.
3. Set up the solver and physical models.
4. Compute and monitor the solution.
5. Examine and save the results.
6. Consider revisions to the numerical or physical model parameters, if necessary.

Step 2. of the solution process requires a geometry modeler and mesh generator. You can use DesignModeler and Ansys Meshing within Ansys Workbench or you can use a separate CAD system for geometry modeling and mesh generation. When meshing 3D geometries, you can also use the meshing mode of Fluent. Alternatively, you can use supported CAD packages to generate volume meshes for import into Ansys Fluent (see the User's Guide (p. 1)). For more information on creating geometry and generating meshes using each of these programs, refer to their respective manuals.

The details of the remaining steps are covered in the User's Guide (p. 1).

2.2. Planning Your CFD Analysis

For each of the problem-solving steps, there are some questions that you need to consider:

- **Defining the Modeling Goals**
  - What results are you looking for, and how will they be used?

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

→ What are your modeling options?

→ What physical models will need to be included in your analysis?

→ What simplifying assumptions do you have to make?

→ What simplifying assumptions can you make?

→ Do you require a unique modeling capability?

- Could you use user-defined functions (written in C)?

– What degree of accuracy is required?

– How quickly do you need the results?

– How will you isolate a piece of the complete physical system?

– Where will the computational domain begin and end?

→ Do you have boundary condition information at these boundaries?

→ Can the boundary condition types accommodate that information?

→ Can you extend the domain to a point where reasonable data exists?

– Can it be simplified or approximated as a 2D or axisymmetric problem?

- **Creating Your Model Geometry and Mesh**

  Ansys Fluent uses unstructured meshes in order to reduce the amount of time you spend generating meshes, to simplify the geometry modeling and mesh generation process, to enable modeling of more complex geometries than you can handle with conventional, multi-block structured meshes, and to enable you to adapt the mesh to resolve the flow-field features. Ansys Fluent can also use body-fitted, block-structured meshes (for example, those used by Ansys Fluent 4 and many other CFD solvers). Ansys Fluent is capable of handling triangular and quadrilateral elements (or a combination of the two) in 2D, and tetrahedral, hexahedral, pyramid, wedge, and polyhedral elements (or a combination of these) in 3D. This flexibility enables you to pick mesh topologies that are best suited for your particular application, as described in the User's Guide (p. 1).

  For 3D geometries, you can create the mesh using the meshing mode of Fluent; otherwise, you must generate the initial mesh (whatever the element types used) outside of Fluent or use one of the CAD systems for which mesh import filters exist. When in solution mode, Fluent can be used to adapt all types of meshes (except for polyhedral), in order to resolve large gradients in the flow field.

  The following questions should be considered when you are generating a mesh:

  – Can you benefit from other Ansys, Inc. products such as CFX or Ansys Icepak?

  – Can you use a quad/hex mesh or should you use a tri/tet mesh or a hybrid mesh?

  → How complex is the geometry and flow?

  → Will you need a non-conformal interface?

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - What degree of mesh resolution is required in each region of the domain?

    → Is the resolution sufficient for the geometry?

    → Can you predict regions with high gradients?

    → Will you use adaption to add resolution?

  - Do you have sufficient computer memory?

    → How many cells are required?

    → How many models will be used?

- **Setting Up the Solver and Physical Models**

  For a given problem, you will need to:

  - Import and check the mesh.
  - Select the numerical solver (for example, density based, pressure based, unsteady, and so on).
  - Select appropriate physical models.

    → Turbulence, combustion, multiphase, and so on.

  - Define material properties.

    → Fluid

    → Solid

    → Mixture

  - Prescribe operating conditions.
  - Prescribe boundary conditions at all boundary zones.
  - Provide an initial solution.
  - Set up solver controls.
  - Set up convergence monitors.
  - Initialize the flow field.

- **Computing and Monitoring Your Solution**

  - The discretized conservation equations are solved iteratively.

    → A number of iterations are usually required to reach a converged solution.

  - Convergence is reached when:

    → Changes in solution variables from one iteration to the next are negligible.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

    - Residuals provide a mechanism to help monitor this trend.

  → Overall property conservation is achieved.

– The accuracy of a converged solution is dependent upon:

  → Appropriateness and accuracy of physical models.

  → Mesh resolution and independence.

  → Problem setup.

- **Examining and Saving Your Results**

  Examine the results to review the solution and extract useful data.

  – Visualization tools can be used to answer such questions as:

    → What is the overall flow pattern?

    → Is there separation?

    → Where do shocks, shear layers, and so on form?

    → Are key flow features being resolved?

  – Numerical reporting tools can be used to calculate the following quantitative results:

    → Forces and moments

    → Average heat transfer coefficients

    → Surface and volume integrated quantities

    → Flux balances

- **Revising Your Model**

  Once your solution is converged, the following questions should be considered when you are analyzing the solution:

  – Are physical models appropriate?

    → Is flow turbulent?

    → Is flow unsteady?

    → Are there compressibility effects?

    → Are there 3D effects?

  – Are boundary conditions correct?

    → Is the computational domain large enough?

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

→ Are boundary conditions appropriate?

→ Are boundary values reasonable?

– Is the mesh adequate?

→ Can the mesh be adapted to improve results?

→ Does the solution change significantly with adaption, or is the solution mesh independent?

→ Does boundary resolution need to be improved?

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 3: Guide to a Successful Simulation Using Ansys Fluent

The following guidelines can help you make sure your CFD simulation is a success. Before logging a technical support request, make sure you do the following:

1. Examine the quality of the mesh in Fluent.

   There are two basic things that you should do before you start a simulation:

   - Perform a mesh check to avoid problems due to incorrect mesh connectivity, and so on. In particular, you should be sure that the minimum reported cell volume is not negative.
   - Look at maximum cell skewness (for example, using the **Compute** button in the **Contours** dialog box after initializing the model). As a rule of thumb, the skewness should be below 0.98. You can also use the **Report Quality** function to calculate the minimum cell orthogonality. You can find more details about mesh quality considerations in Mesh Quality in the *Fluent User's Guide* (p. 1146).

   If there are mesh problems, you may have to re-mesh the problem.

2. Scale the mesh and check length units.

   In Ansys Fluent, all physical dimensions are initially assumed to be in meters. You should scale the mesh accordingly. Other quantities can also be scaled independently of other units used. Ansys Fluent defaults to SI units.

3. Employ the appropriate physical models.

4. Set the energy under-relaxation factor between 0.95 and 1.

   For problems with conjugate heat transfer, when the conductivity ratio is very high, smaller values of the energy under-relaxation factor practically stall the convergence rate.

5. Use node-based gradients with unstructured tetrahedral meshes.

   The node-based averaging scheme is known to be more accurate than the default cell-based scheme for unstructured meshes, most notably for triangular and tetrahedral meshes.

6. Monitor convergence with residuals history.

   Residual plots can show when the residual values have reached the specified tolerance. After the simulation, note if your residuals have decreased by at least 3 orders of magnitude to at least 10^{-3}. For the pressure-based solver, the scaled energy residual must decrease to 10^{-6}. Also, the scaled species residual may need to decrease to 10^{-5} to achieve species balance.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also monitor lift, drag, or moment forces as well as pertinent variables or functions (for example, surface integrals) at a boundary or any defined surface.

7. Run the CFD simulation using second order discretization for better accuracy rather than a faster solution.

   A converged solution is not necessarily a correct one. You should use the second-order upwind discretization scheme for final results.

8. Monitor values of solution variables to make sure that any changes in the solution variables from one iteration to the next are negligible.

9. Verify that property conservation is satisfied.

   After the simulation, note if overall property conservation has been achieved. In addition to monitoring residual and variable histories, you should also check for overall heat and mass balances. At a minimum, the net imbalance should be less than 1% of the smallest flux through the domain boundary.

10. Check for mesh dependence.

    You should ensure that the solution is mesh-independent and use mesh adaption to modify the mesh or create additional meshes for the mesh-independence study.

11. Check to see that the solution makes sense based on engineering judgment.

    If flow features do not seem reasonable, you should reconsider your physical models and boundary conditions. Reconsider the choice of the boundary locations (or the domain). An inadequate choice of domain (especially the outlet boundary) can significantly impact solution accuracy.

You are encouraged to collaborate with your technical support engineer in order to develop a solution process that ensures good results for your specific application. This type of collaboration is a good investment of time for both yourself and the Ansys Fluent support engineer.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 4: Starting and Executing Ansys Fluent

This chapter provides instructions for starting and executing Ansys Fluent.

4.1. Starting Ansys Fluent

4.2. Running Ansys Fluent in Batch Mode

4.3. Switching Between Meshing and Solution Modes

4.4. Checkpointing an Ansys Fluent Simulation

4.5. Cleaning Up Processes From an Ansys Fluent Simulation

4.6. Exiting Ansys Fluent

4.1. Starting Ansys Fluent

The following sections describe how start Ansys Fluent:

Important:

- Ensure your graphics driver is up-to-date to avoid issues with graphics displays (such as the display coming upside-down).
- If your %HOMEDRIVE% environment variable is set to a network drive and you experience issues such as delays in the Fluent Launcher appearing, add a copy of the preferences file to the %HOMEDRIVE% network location (`%HOMEDRIVE%%HOMEPATH%\.fluent-conf\24.2.0\preferences`). Refer to Setting User Preferences/Options (p. 937) for additional information on the preferences file location.

4.1.1. Selecting the Licensing Level

4.1.2. Starting Ansys Fluent Using Fluent Launcher

4.1.3. Starting Ansys Fluent on a Windows System

4.1.4. Starting Ansys Fluent on a Linux System

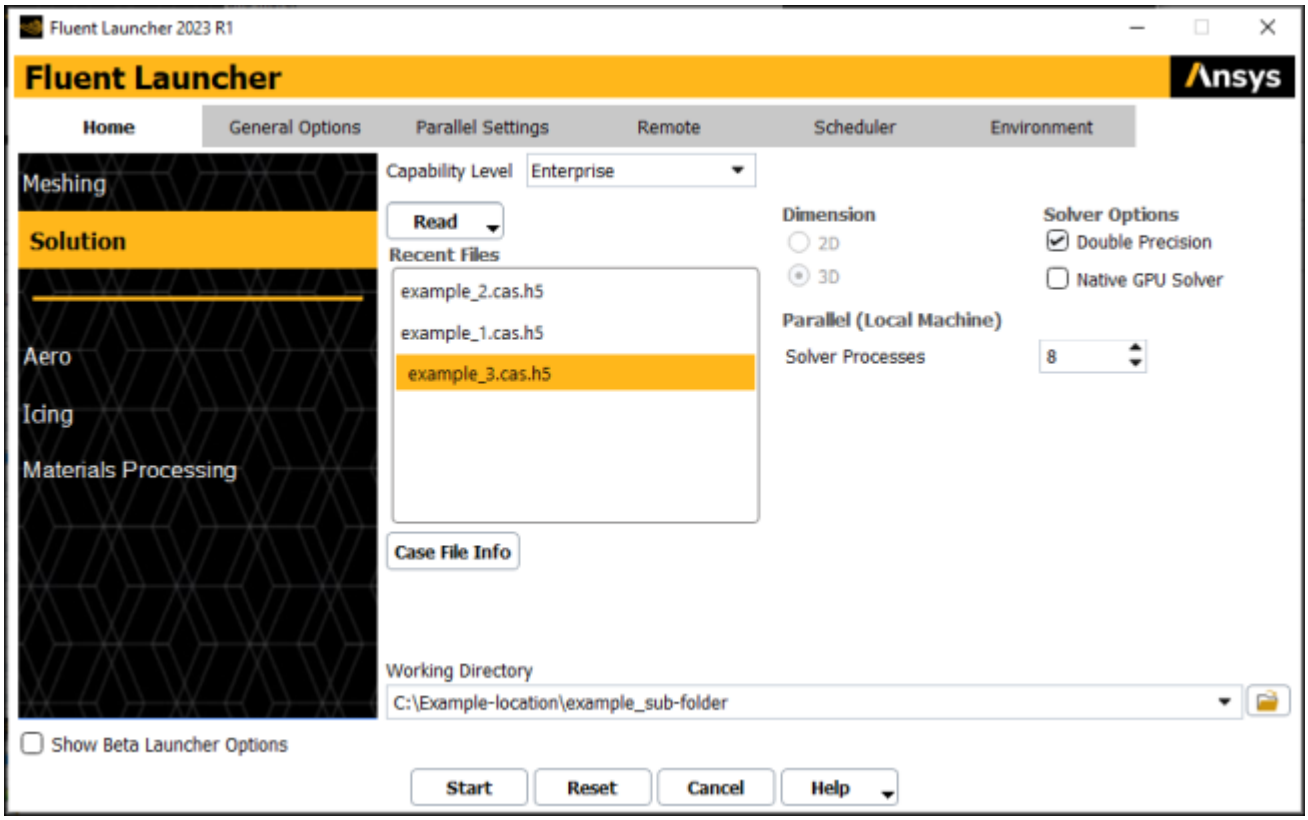
4.1.5. Command Line Startup Options

4.1.6. Aborting During Startup

4.1.1. Selecting the Licensing Level

You specify the licensing level using the **Capability Level** drop-down list in the Fluent Launcher. The feature availability at the different licensing levels is discussed in Program Capabilities (p. 147).

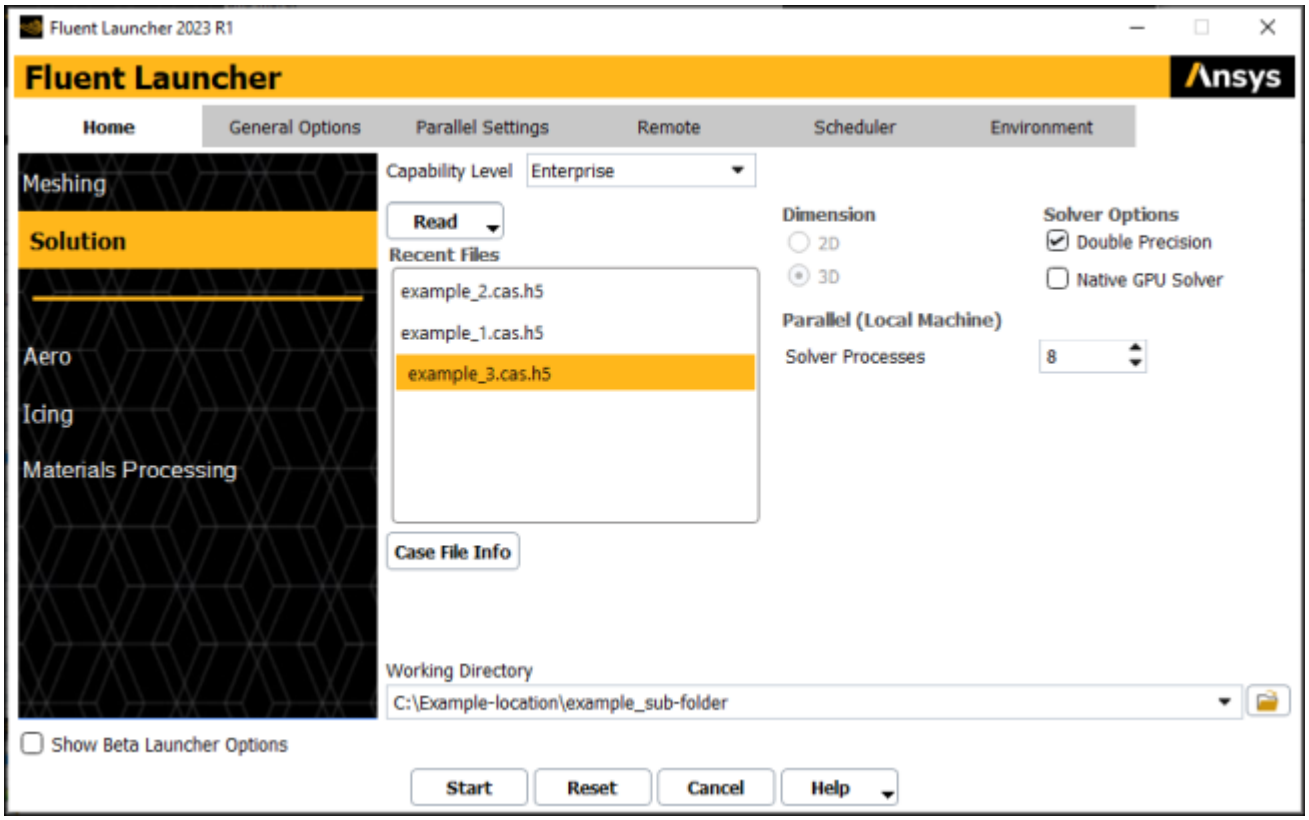
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4.1.2. Starting Ansys Fluent Using Fluent Launcher

You can interactively specify Ansys Fluent dimension, display, processing and other options using the Fluent Launcher.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To start the Fluent Launcher, do one of the following:

- Start Ansys Fluent from the Linux or Windows command line with no arguments.
- Start Ansys Fluent from the Windows Start menu.
- Start Ansys Fluent from the Windows desktop or Quick Launch bar.

Any options set in the Fluent Launcher will be retained for your next session.

Select the workspace you plan to use (the dividing line in the launcher separates the general-purpose Meshing and Solution workspaces from the application-specific option):

- **Meshing**—See Introduction to Meshing Mode in Fluent (p. 217) and subsequent chapters for further details about using the Fluent Meshing workspace.
- **Solution**—See Graphical User Interface (GUI) (p. 901) and subsequent chapters for further details about using the Fluent Solution workspace.
- **Aero**—See Fluent Aero in the *Fluent Workspaces User's Guide* for further details about using the Fluent Aero workspace.
- **Icing**—See Fluent Icing in the *Fluent Workspaces User's Guide* for further details about using the Fluent Icing workspace.
- **Materials Processing**—See Fluent Materials Processing Workspace in the *Fluent Workspaces User's Guide* for further details about using the Fluent Materials Processing workspace.

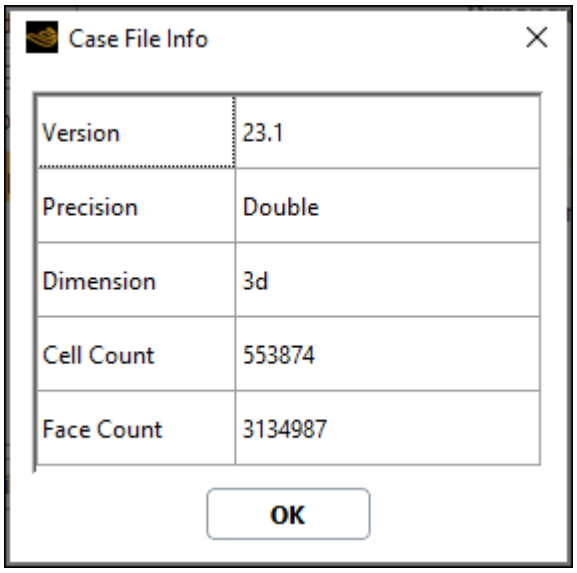
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

*You can enable **Show Beta Launcher Options** to expose additional workspaces and settings that are not yet considered "full" features. These are documented in the Fluent Beta Features Manual.*

Important:

The available workspaces depends on your licensing level. To learn more about your licensing options, refer to the ANSYS Licensing documentation or speak with your sales representative.

Select a case| case and data | mesh | journal file to start with or specify the **Dimension** of the simulation you intend to perform. *Beginning with a journal file allows you to automatically load the case, compile any user-defined functions, iterate until the solution converges, and write results to an output file.* Click the highlighted case | mesh | journal file in the **Recent Files** list to undo the selection of a starting file. Click **Case File Info** to see the following case or mesh file properties:



- Fluent version the file was last saved in
- Precision
- Dimension
- Total number of cells (`.cas.h5` and `.msh.h5` only)
- Total number of faces (`.cas.h5` and `.msh.h5` only)

Note:

Selecting a case file from the **Recent Files** list only loads the case file (after clicking **Start**), even if there is an associated data file in the same directory.

Select your required **Solver Options**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Choose to perform solution calculations in **Double Precision** mode, if desired. (Default is single-precision mode) See Single-Precision and Double-Precision Solvers (p. 185) to help with your decision.

  Note:

  The **Meshing** workspace is always run in **Double Precision**. This option applies for the Solution workspace only.

- Enable **Native GPU** to run the Fluent solver on one or more GPUs. For more information on starting the Fluent GPU solver using the Fluent Launcher see Starting the Fluent GPU Solver Using the Fluent Launcher (p. 3757).

Select your **Parallel (Local Machine)** options.

- Set **Meshing Processes** to **1** to restrict the meshing calculations to a single processor core.
- Set **Solver Processes** to **1** to restrict the solution calculations to a single processor core.
- Set **Meshing Processes** more than 1 to allow multiple simultaneous processes. *Note that increasing the number of meshing processes also increases the solver processes to the same processor count.* See Setting Parallel Options in Fluent Launcher (p. 186) for additional information.
- Set **Solver Processes** more than 1 to allow multiple simultaneous processes. See Setting Parallel Options in Fluent Launcher (p. 186) for additional information.
- If **Native GPU Solver** is enabled:
  - Set **CPU Processes** to more than 1 to allow multiple simultaneous processes.
  - Set the number of GPUs (if your machine has a graphics card with more than one GPU) by clicking **...** next to the name of the card to see the individual available GPUs.
  - **Important:**

    The maximum number of GPUs used will be either the number of selected GPUs or the number of CPU processes, whichever count is smaller.

Specify the path of your current working directory using the **Working Directory** field or click to browse through your directory structure. *Selecting a file from the **Recent Files** list automatically updates the working directory to the location of the selected file, however you can change to another location, if desired.*

Note:

a Uniform Naming Convention (UNC) path cannot be set as a working directory. You need to map a drive to the UNC path (Windows only)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Click the **Help** drop-down to reveal links to: Documentation, News, Tutorials, Online Resources, and Videos.

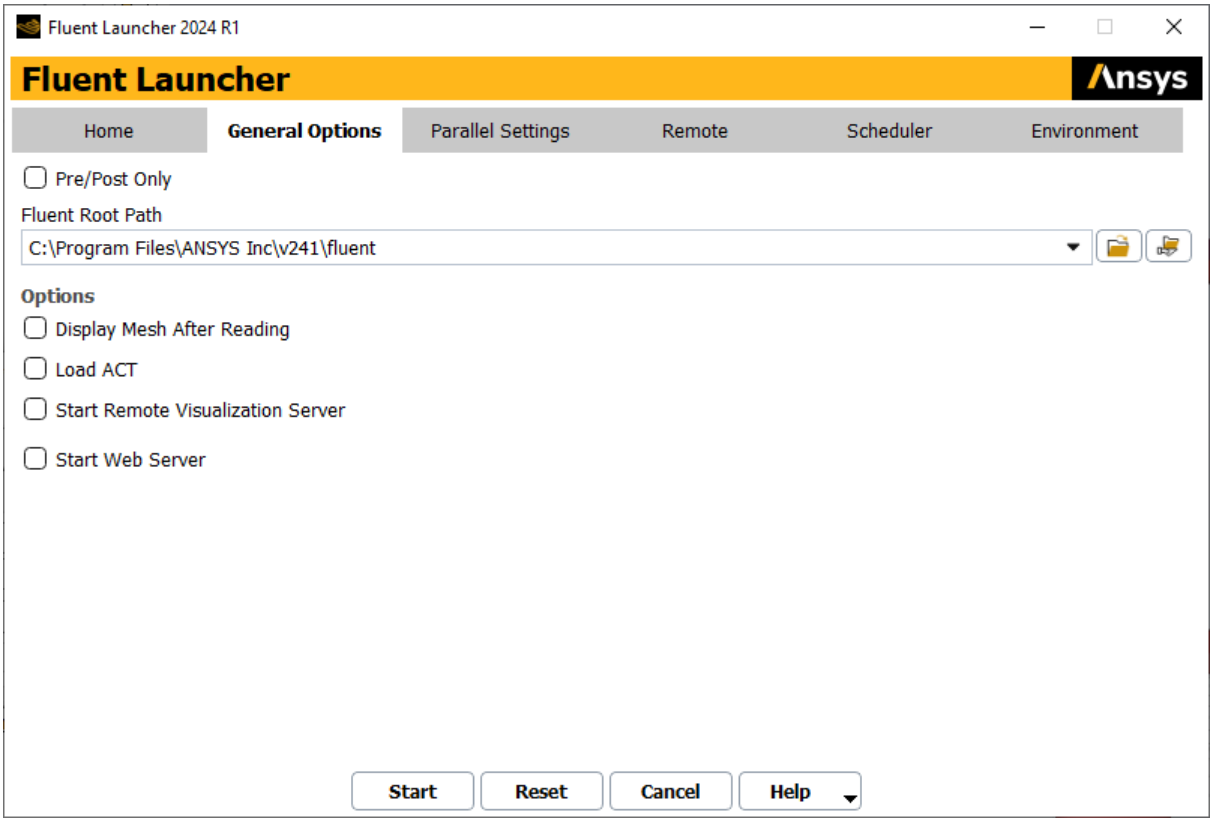
Important:

Fluent Launcher also appears when you start Ansys Fluent within Ansys Workbench. For more information, see the separate Ansys Fluent in Workbench User's Guide.

4.1.2.1. Setting General Options in Fluent Launcher

Set file and path options using the **General Options** tab in Fluent Launcher.

Figure 4.1: The General Options Tab of Fluent Launcher



1. Enable **Pre/Post Only** to run Ansys Fluent with only the setup and postprocessing capabilities available. The default Ansys Fluent full solution mode allows you to set up, solve, and postprocess a problem, while **Pre/Post Only** will not allow you to perform calculations.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Specify the location of the Ansys Fluent installation on your system using the **Fluent Root Path** field, or click to browse through your directory structure. Try to use the UNC path if applicable.

   Note:

   The button automatically converts a local path to a UNC path if any matching shared directory is found (Windows only). Once set, various fields in Fluent Launcher (for example, parallel settings, etc.) are automatically populated with the available options, depending on the Ansys Fluent installations that are available.

3. Choose to **Display Mesh After Reading** (disabled by default). This option is applicable only to volume meshes and not surface meshes. All of the boundary zones will be displayed except for the interior zones of 3D geometries.

   Note:

   You can override this option on a file-by-file basis using the **Display Mesh After Reading** option in the **Select File** dialog box that opens when you are reading in a file.

4. Enable the **Load ACT** option to load Ansys ACT. For additional information on ACT, see Customizing Fluent (p. 4681).

5. Enable **Start Remote Visualization Server** to launch the Fluent workspace as a server so that you can launch the Fluent Remote Visualization Client and connect to this Fluent session. Refer to Remote Visualization and Accessing Fluent Remotely in the *Fluent Workspaces User's Guide* to learn more about this option.

6. Enable **Start Web Server** to access settings for the Fluent web service so that you can use the Fluent Web Server and remotely connect to a Fluent session. Refer to Remotely Accessing Your Simulations Using Ansys Fluent's Web Interface (p. 4445) to learn more about the web interface.

4.1.2.2. Single-Precision and Double-Precision Solvers

Both single-precision and double-precision versions of Ansys Fluent are available on all computer platforms. For most cases, the single-precision solver will be sufficiently accurate, but certain types of problems may benefit from the use of a double-precision version. Several examples are listed below:

- If your geometry has features of very disparate length scales (for example, a very long, thin pipe), single-precision calculations may not be adequate. Note that nodal coordinates are always stored in double precision (even for the single-precision version of Ansys Fluent), so they are not a concern in this regard.
- If your geometry involves multiple enclosures connected via small-diameter pipes (for example, automotive manifolds), mean pressure levels in all but one of the zones can be quite large (since you can set only one global reference pressure location). Double-precision calculations may

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

therefore be necessary to resolve the pressure differences that drive the flow, since these will typically be much smaller than the pressure levels.

- For conjugate problems involving high thermal-conductivity ratios and/or high-aspect-ratio meshes, convergence and/or accuracy may be impaired with the single-precision solver, due to inefficient transfer of boundary information.
- For multiphase problems where the population balance model is used to resolve particle size distributions, which could have statistical moments whose values span many orders of magnitude.

Note:

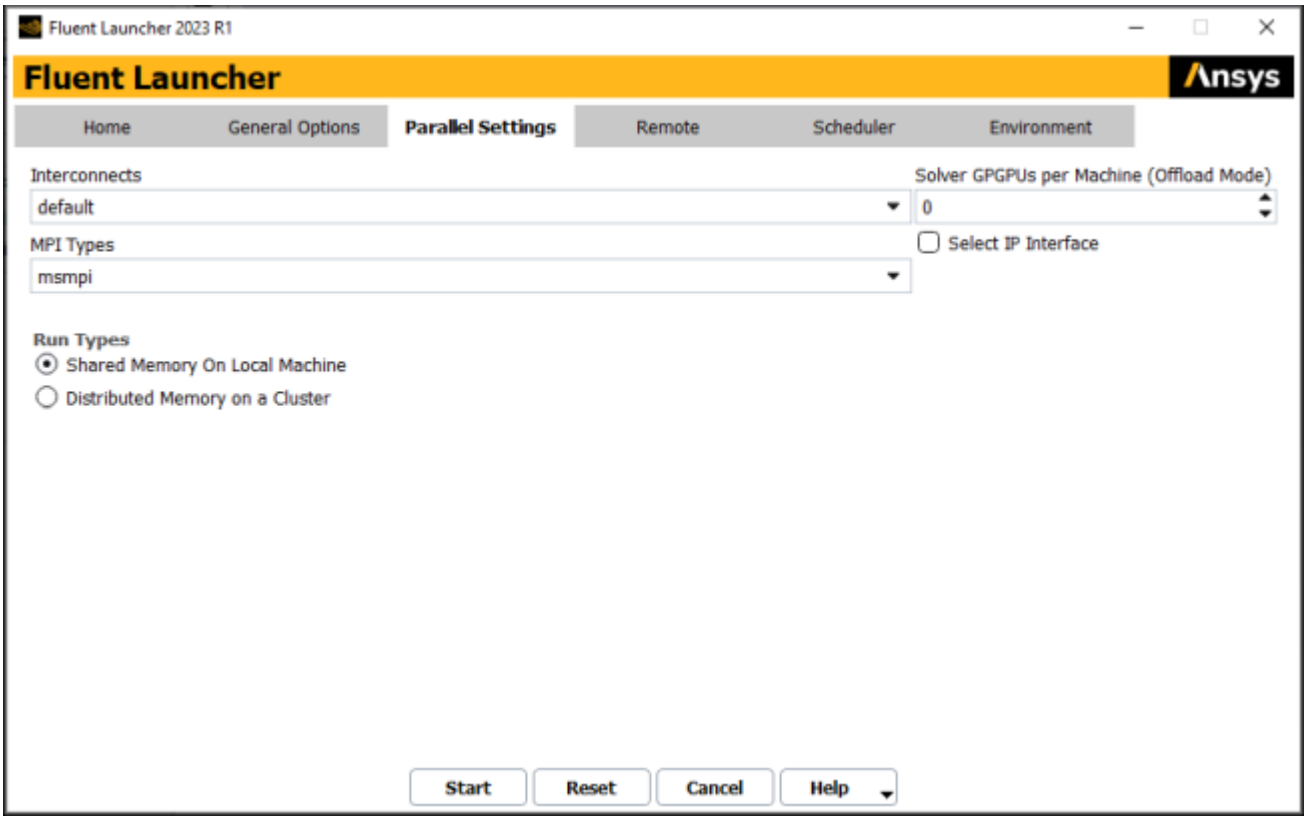
Ansys Fluent allows only a period to be used as a decimal separator. If your system is set to a European locale that uses a comma separator (for example, Germany), fields that accept numeric input may accept a comma, but may ignore everything after the comma. If your system is set to a non-European locale, numeric fields will not accept a comma at all.

Ansys Workbench accepts commas as decimal delimiters. These are translated into periods when data is passed to Ansys Fluent.

4.1.2.3. Setting Parallel Options in Fluent Launcher

The **Parallel Settings** tab allows you to specify settings for running Ansys Fluent in parallel.

Figure 4.2: The Parallel Settings Tab of Fluent Launcher



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. (**Meshing** workspace only) Enter the number of processes to be used for meshing under **Meshing Processes**.

2. (**Meshing** and **Solution** workspaces only) Enter the number of processes to be used for solution under **Solver Processes**. On Linux with the Intel MPI, additional processes will be spawned as necessary when you change to solution mode, in order to bring the total number of processes to this value; this must be set to a value greater than or equal to **Meshing Processes**. For details on this dynamic spawning, see Dynamically Spawning Processes Between Fluent Meshing and Fluent Solution Modes (p. 220).

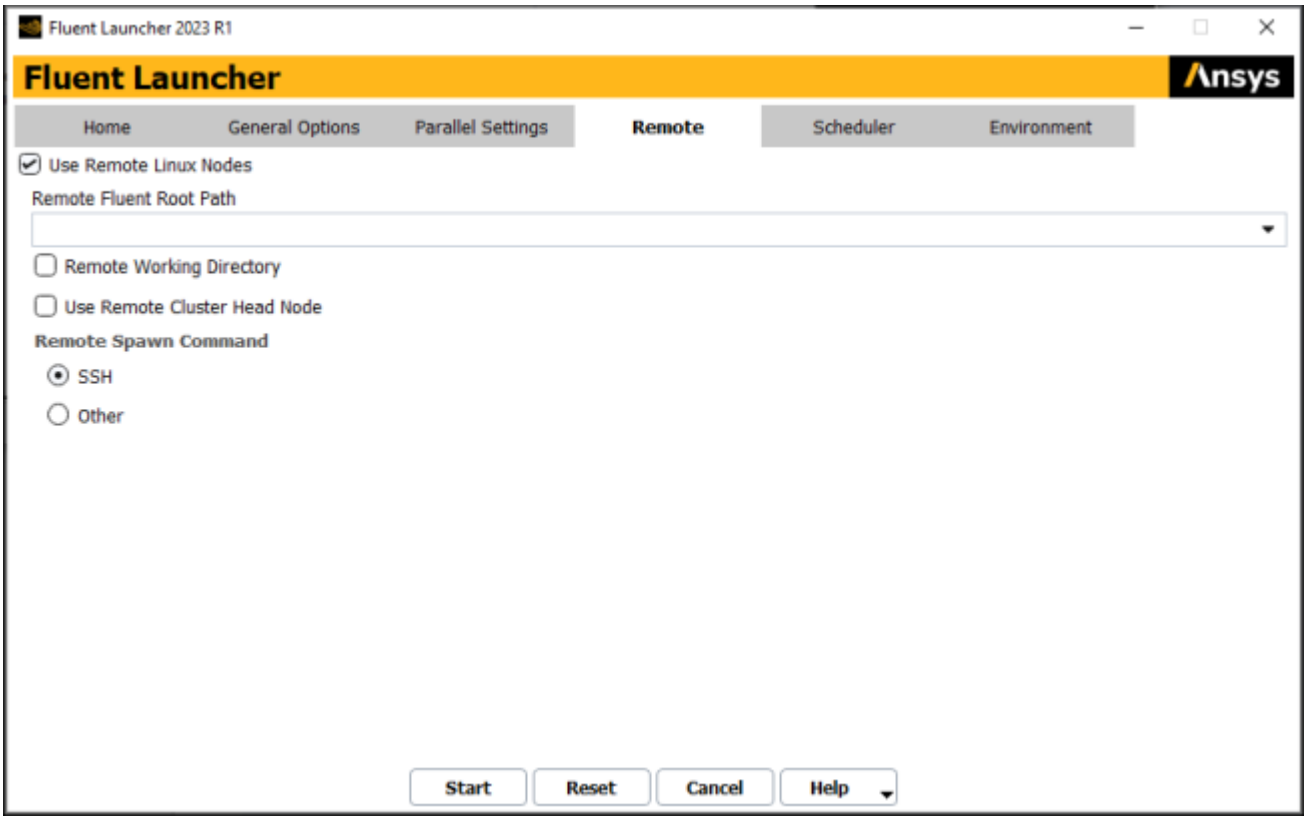
3. (**Meshing** and **Solution** workspaces only) If your machine is equipped with General Purpose Graphics Processing Units you can also specify **Solver GPGPUs per Machine (Offload Mode)**. This setting is useful if you plan to use the accelerated Algebraic Multigrid (AMG) solver and/or the accelerated Discrete Ordinates (DO) solver (for details, see Using General Purpose Graphics Processing Units (GPGPUs) With the Algebraic Multigrid (AMG) Solver (p. 4343) and Accelerating Discrete Ordinates (DO) Radiation Calculations (p. 4352), respectively).

Refer to Parallel Processing (p. 4293) for details on parallel processing using Fluent and Starting Parallel Ansys Fluent Using Fluent Launcher (p. 4296) for additional details parallel process configuration options on this tab.

4.1.2.4. Setting Remote Options in Fluent Launcher

The **Remote** tab (Figure 4.3: The Remote Tab of Fluent Launcher (p. 187)) allows you to specify settings for running Ansys Fluent parallel simulations on Linux clusters, via the Windows interface.

Figure 4.3: The Remote Tab of Fluent Launcher

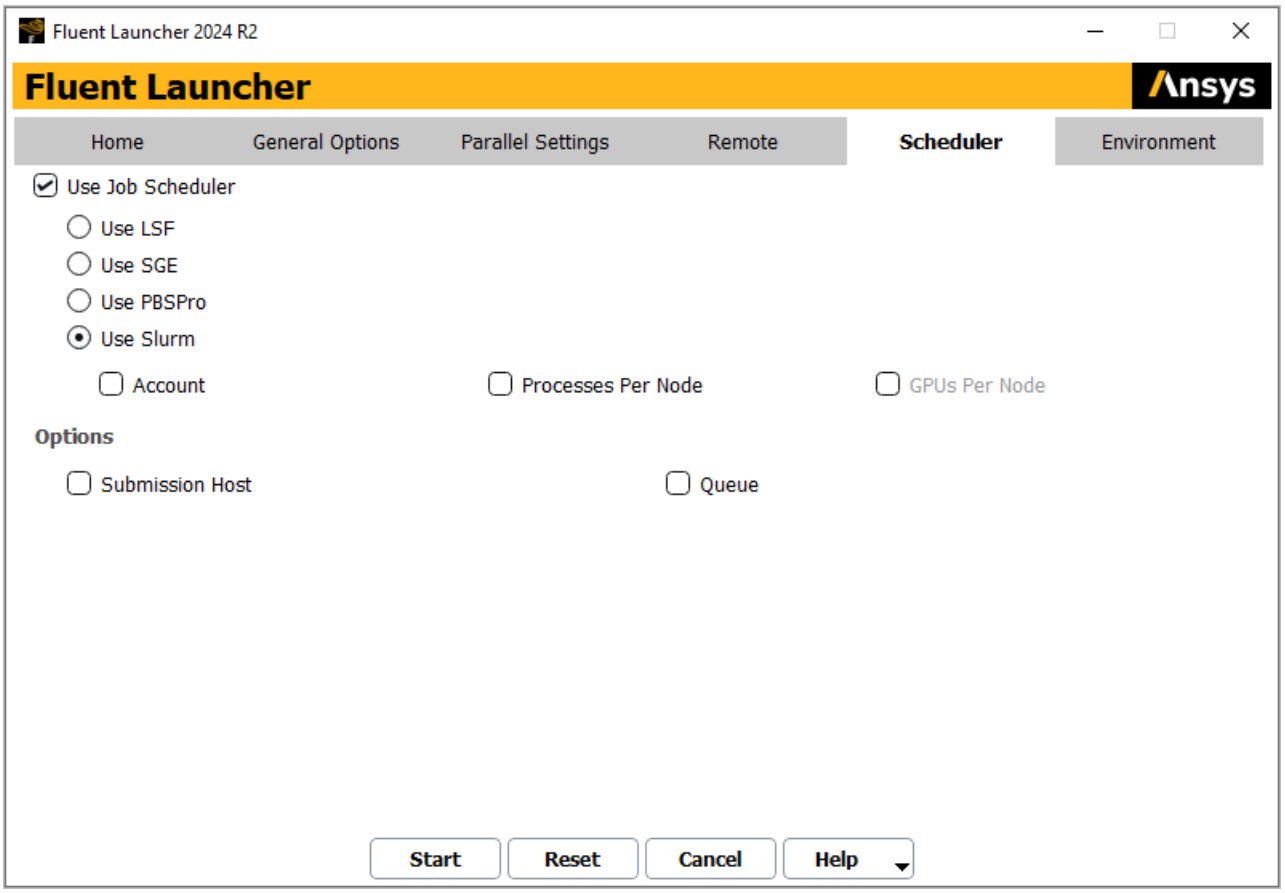


For additional information about this tab, see Setting Additional Options When Running on Remote Linux Machines (p. 4301).

4.1.2.5. Setting Scheduler Options in Fluent Launcher

Enable **Use Job Scheduler** in the **Scheduler** tab (Figure 4.4: The Scheduler Tab of Fluent Launcher (Windows Version) (p. 188)) to specify settings for running Ansys Fluent with various job schedulers (for example, the Microsoft Job Scheduler for Windows, or LSF, SGE, PBS Pro, and Slurm on Linux).

Figure 4.4: The Scheduler Tab of Fluent Launcher (Windows Version)



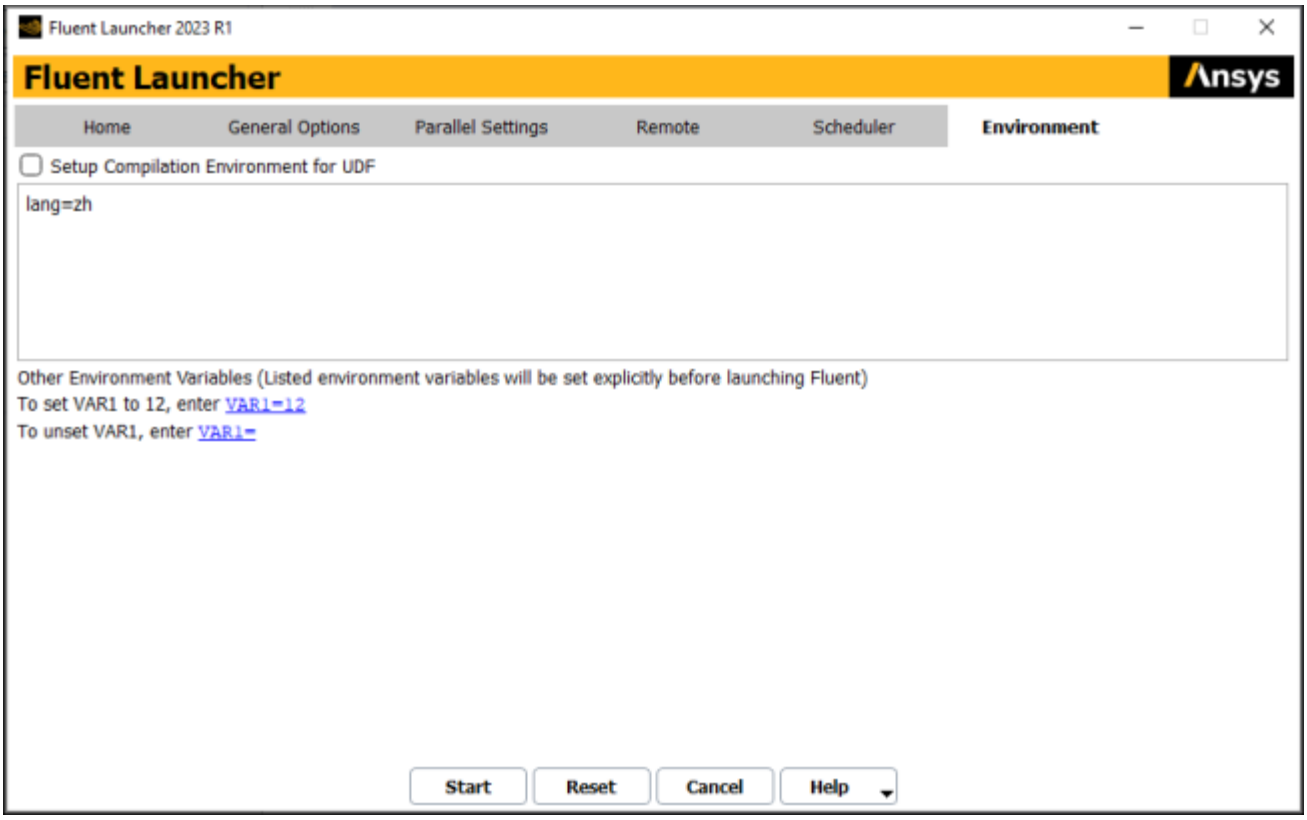
For additional information about this tab, see Setting Parallel Scheduler Options in Fluent Launcher (p. 4298).

4.1.2.6. Setting Environment Options in Fluent Launcher

The **Environment** tab (Figure 4.5: The Environment Tab of Fluent Launcher (p. 189)) allows you to specify compiler settings for compiling user-defined functions (UDFs) with Ansys Fluent (Windows only). The **Environment** tab also allows you to specify environment variable settings for running Ansys Fluent.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 4.5: The Environment Tab of Fluent Launcher



Specify a batch file that contains UDF compilation environment settings by selecting the **Set up Compilation Environment for UDF** check box (enabled by default). Once selected, you can then enter a batch file name in the text field. By default, Fluent Launcher uses the `udf.bat` file that is located in the directory where Ansys Fluent is installed. It is recommended that you keep the default batch file, which is tested with MS Visual Studio C++ and Clang compilers (for supported versions, see Compiler Requirements for Windows Systems), as well as the built-in compiler (Clang) included with the Fluent installation. For more information about compiling UDFs, see the separate Fluent Customization Manual.

Under **Other Environment Variables**, enter or edit license file or environment variable information in the text field. For example, `FLUENT_AFFINITY=<x>` specifies the process binding (affinity) setting, in the same manner as the `-affinity=<x>` command line option (see Parallel Options (p. 198) for details). Using the **Default** button resets the default value(s).

4.1.3. Starting Ansys Fluent on a Windows System

There are two ways to start Ansys Fluent on a Windows system:

- From the Windows **Start** menu, click **Start > Ansys 2024 R2 > Fluid Dynamics > Fluent 2024 R2**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This option starts Fluent Launcher (see Starting Ansys Fluent Using Fluent Launcher (p. 180)). The Fluent Launcher may also be accessed via an icon on your desktop or in the Quick Launch bar.

Note:

If the default "ANSYS 2024 R2" program group name was changed when Ansys Fluent was installed, you will find the **Fluent** menu item in the program group with the new name that was assigned, rather than in the **ANSYS 2024 R2** program group.

- From a Command Prompt window, type `fluent version`, where `version` is replaced with one of the four options specifying the dimension and precision of the solver.
  - `2d` for the 2D, single-precision solver.
  - `3d` for the 3D, single-precision solver.
  - `2ddp` for the 2D, double-precision solver.
  - `3ddp` for the 3D double-precision solver.

For additional information on starting Fluent from the command prompt, see Command Line Startup Options (p. 191).

Important:

To be able to start Ansys Fluent from the command prompt, be sure the path to your Ansys Fluent home directory is in your command search path environment variable by executing the `setenv.exe` program located in the Ansys Fluent directory (for example, `C:\Program Files\ANSYS Inc\v242\fluent\ntbin\win64`).

Tip:

You can also specify the number of processors or start Ansys Fluent in meshing mode from the Command Prompt.

- To specify the number of processors, type `fluent version -tx`, replacing `version` with the desired solver version and `x` with the number of processors. For example, `fluent 3d -t4` to run the 3D version on 4 processors.
- To start in meshing mode, add the command line option `-meshing`. For example, `fluent 3d -meshing` to start in meshing mode.
- Both parallel and meshing mode may be combined. You must specify the number of meshing processes using `-tmy`. For example, `fluent 3ddp -meshing -tm4` will start Ansys Fluent in meshing mode with 4 meshing processes. When switched to solution mode, the solver will be 3D, double precision and run 4 processes. It is not possible to switch from meshing mode to solution mode with a different number of processes on Windows, so if you need to run the calculation with a higher number of processes you must start a new session.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.1.4. Starting Ansys Fluent on a Linux System

There are two ways to start Ansys Fluent on a Linux system:

- Start Fluent from the command line without specifying a version, and then use Fluent Launcher to choose the appropriate version along with other options. See Starting Ansys Fluent Using Fluent Launcher (p. 180) for details.
- Start the appropriate version from the command line by typing `fluent version`, where `version` is replaced with one of the four options specifying the dimension and precision of the solver.
  - `2d` for the 2D, single-precision solver.
  - `3d` for the 3D, single-precision solver.
  - `2ddp` for the 2D, double-precision solver.
  - `3ddp` for the 3D double-precision solver.

Tip:

You can also specify the number of parallel processors or start Ansys Fluent in meshing mode from the command line.

- To specify the number of processors, type `fluent version -tx`, replacing `version` with the desired solver version and `x` with the number of processors. For example, `fluent 3d -t4` to run the 3D version on 4 processors.
- To start in meshing mode, add the command line option `-meshing`. For example, `fluent 3d -meshing` to start in meshing mode.
- Both parallel and meshing mode may be combined. You must specify the number of meshing processes using `-tmy`. For example, `fluent 3ddp -meshing -tm4 -t8` will start Ansys Fluent in meshing mode with 4 meshing processes. When switched to solution mode, the solver will be 3D, double precision and run 8 processes; note that dynamically spawning additional processes in solution mode is only available on Linux with the default MPI.

Note:

Ansys Fluent automatically selects the best graphics driver and defaults to the `X11` driver when it does not detect the required graphics support. You can use the `HOOPS_PICTURE` environment variable to force a particular graphics driver, if you feel it is necessary to use an alternate driver.

4.1.5. Command Line Startup Options

Table 4.1: Available Command Line Options for Linux and Windows Platforms (p. 192) lists the available command line arguments for Linux and Windows. More detailed descriptions of these options can be found in the following sections.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To obtain information about available startup options, you can type `fluent -help` before starting up Fluent.

Table 4.1: Available Command Line Options for Linux and Windows Platforms

Option	Platform	Description
`-act`	all	Load ACT on Fluent startup.
`-affinity=<x>`	all	Specifies the process binding (affinity) setting, as described in Parallel Options (p. 198).
`-app=flremote`	all	Launches the Remote Visualization Client.
`-appscript=<scriptfile>`	all	Runs the specified script in the specified application (must be used with the `-app=<x>` startup option.
`-case <file path> [-data]`	all	Reads the specified case file after Fluent launches; if you include `-data` in the command line, the data file with the same name as the case file is read as well. For example: `fluent 3ddp -case c:\example_location\example.cas.h5 -data`
`-ccp <x>`	Windows only	Uses the Microsoft Job Scheduler, where <x> is the head node name.
`-cflush`	Linux only	Ensures that the file cache buffers are flushed.
`-cnf=<x>`	all	Specifies that <x> is the hosts file or (for Linux) machine list.
`-command="<TUI command>"`	all	Executes the specified text command (`<TUI command>`) at Fluent startup.
`-driver <name>`	all	Sets the graphics driver (available drivers vary by platform, and include `opengl`, `opengl2`, `x11`, and `null` for Linux and `opengl`, `opengl2`, `dx11`, `msw`, and `null` for Windows).
`-env`	all	Show environment variables.
`-g`	all	Run without the GUI or graphics.
`-gpgpu=<n>`	Linux and Win64 only	Specifies the number of GPGPUs per machine that should be used for AMG acceleration and/or DO acceleration. Only available in parallel.
`-gpu[=<n>}`	all	Run with the GPU Solver. You can optionally specify which devices to use; for example, `-gpu=1,2,4`.
`-gr`	all	Run without graphics.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Option	Platform	Description
-gu	all	Run without the GUI but with graphics. You cannot interact with the displayed graphics objects.
-gui_machine=<hostname>	Linux only	Specifies that <hostname> is used for running Cortex (the process that manages the GUI and graphics).
-h<heap size>	all	Specifies the heap space for Cortex (the process that manages the GUI and graphics)
-help	all	Display command line options.
-hidden	Windows only	Run in minimized mode.
-host_ip=<host:ip>	all	Specifies that the IP interface <host:ip> is to be used by the host.
-i <journal>	all	Reads the specified journal file(s). Read multiple journals at once as follows: -i example1.jou -i example2.jou -i example3.jou ...
-license=<x>	all	Specify the capability level for when Fluent launches; <x>={enterprise | premium}.
-meshing	all	Start Fluent in meshing mode (you must specify Fluent as either 3d or 3ddp).
-mpi=<mpi>	all	Specifies that the MPI implementation is <mpi> (for example, intel).
-mpitest	all	Launches an MPI program to collect network performance data and prints to console (Linux) or to the working directory (Windows).
-nm	all	Do not display mesh after reading.
-p<ic>	all	Specify interconnect; <ic>={default | eth | ib}
-pcheck	Linux only	Check the network connections before spawning compute nodes.
-platform=<x>	Linux only	Loads a binary that is specially ported for a particular platform, as described in Performance Options (p. 197).
-post	all	Run the Ansys Fluent postprocessing-only executable.
-r	all	List all releases installed in the current directory.
-r<x>	all	Run release <x> of Ansys Fluent.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Option	Platform	Description
-remote_node=<hostname>	Linux only	Specify the machine to be used for executing mpirun to launch the node processes; if =<hostname> is omitted, the first node in the hosts file will be used.
-scheduler=<scheduler>	Linux only	Run Ansys Fluent under a scheduler; <scheduler> can be set to lsf (LSF), pbs (PBS Professional), sge (Altair Grid Engine (formerly UGE and formerly SGE), or slurm (Slurm).
-scheduler_account=<account>	Linux only	Specifies that the account is set to <account> when running under Slurm.
-scheduler_custom_script	Linux only	Ensures the use of environment variables when using custom scheduler scripts.
-scheduler_gpn=<x>	Linux only	Sets the number of graphics processing units (GPUs) per cluster node to <x> when running under Slurm with the Fluent native GPU solver.
-scheduler_headnode=<head-node>	Linux only	Specifies the scheduler job submission machine name or (for SGE) the SGE qmaster.
-scheduler_list_queues	Linux only	Lists all available queues.
-scheduler_nodeonly	Linux only	Specifies that Cortex and host processes are launched before the job submission and only the parallel node processes are submitted to the scheduler.
-scheduler_opt=<opt>	Linux only	Enables an additional option <opt> that is relevant for the selected scheduler; this command line option can be included multiple times.
-scheduler_pe=<pe>	Linux only	Sets the parallel environment to <pe> when running under SGE.
-scheduler_ppn=<x>	Linux only	Sets the number of node processes per cluster node to <x> when running under Slurm.
-scheduler_queue=<queue>	Linux only	Sets the scheduler queue or partition to <queue>.
-scheduler_stderr=<err-file>	Linux only	Sets the scheduler standard error file to <err-file>.
-scheduler_stdout=<out-file>	Linux only	Sets the scheduler standard output file to <out-file>.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Option	Platform	Description
`-scheduler_tight_coupling`	Linux only	Enables a job-scheduler-supported native remote node access mechanism when running under SGE or PBS Professional.
`-scheduler_workdir=<working-directory>`	Linux only	Sets the working directory for the scheduler job to `<working-directory>`.
`-setenv="<var>=<value>"`	all	Sets the environment variable `<var>` to `<value>`.
`-sifile=<name>.txt`	all	Run Ansys Fluent and start the remote visualization server. *You can provide a path before the server info filename to specify where the file is created.*
`-stream`	Linux only	Prints the memory bandwidth.
`-t<x>`	all	Specifies that the number of processors is <x>.
`-tm<x>`	all	Specifies that the number of processors for meshing is <x>.
`-ws`	all	Starts the web service with default settings.
`-ws=<x>`	all	Starts the web service with a session name <x>.
`-ws-port=<x>`	all	Specifies the port number <x> for starting the web server.
`-ws-portspan=<x>`	all	Specifies the port span <x> for starting the web server.
`-ws-js-url=<x>`	all	Specifies the job service URL <x> for registering the web server

4.1.5.1. ACT Option

`fluent -act` loads Ansys ACT at Fluent startup. For additional information about ACT in Fluent, see Customizing Fluent (p. 4681).

4.1.5.2. Application Option

`fluent -app=flremote` launches either the Fluent Launcher or the specified dimension of Fluent (for example, 3ddp), along with the Fluent Remote Visualization Client. For additional information about the Fluent Remote Visualization Client, refer to Remote Visualization and Accessing Fluent Remotely in the *Fluent Workspaces User's Guide*.

4.1.5.3. Application Script Option

`fluent -appscript`, allows you to specify a script that will run in the specified application (`-appscript` must be used in conjunction with `-app`).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.1.5.4. Graphics and Files Options

Note:

Fluent automatically selects the best graphics driver for the given runtime environment, unless you choose a specific graphics driver with the `fluent -driver` command line option.

(Windows only) The `OpenGL` graphics driver is deprecated and in some instances may cause Ansys Fluent to close unexpectedly. It is recommended that you have a good supported graphics card to ensure the best performance.

`fluent -driver` allows you to specify the graphics driver to be used in the session. When enabling graphics display, you have various options: on Linux, the available drivers include `fluent -driver opengl2`, `fluent -driver opengl`, and `fluent -driver x11`; on Windows, the available drivers include `fluent -driver opengl2`, `fluent -driver opengl`, `fluent -driver dx11`, and `fluent -driver msw` (the latter instructs Ansys Fluent to use the Operating Systems Windows driver). For both Linux and Windows, you can disable graphics display using `fluent -driver null`. For a comprehensive list of the drivers available to you, open a Fluent session, enter the `display/set/rendering-options/driver` text command, and then press the **Enter** key at the `driver>` prompt. For more details about using the driver options, see Hiding the Graphics Window Display (p. 3939).

Note:

For any session that displays graphics in a graphics window and/or saves picture files, having the driver set to `x11`, `msw`, or `null` will cause the rendering / saving speed to be significantly slower.

`fluent -gui_machine=<hostname>` will run Cortex on a specified machine (`<hostname>`) rather than automatically on the same machine as that used for compute node 0. If you just include `-gui_machine` (without `=<hostname>`), Cortex is run on the same machine used to submit the `fluent` command. This option is only available when running on Linux, and may be needed to ensure optimal graphics performance when running Fluent under a scheduler / load manager (using the `-scheduler=<scheduler>` option, as described in Scheduler Options (p. 200)).

Important:

(Exceed onDemand Only) When you are using the `-gui_machine` flag you must also use `-setenv="CORTEX_PRE=ssrun"` to specify the server side rendering to ensure accelerated graphics performance.

For example: `fluent 3ddp -t2 -setenv="CORTEX_PRE=/opt/Exceed_connection_server_13.8_64/bin/ssrun" -scheduler=<scheduler> -scheduler_queue=<queue> -gui_machine=<hostname>`

Note that the path to `ssrun` may be different for your specific environment.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

`fluent -g` will run Cortex without graphics and without the graphical user interface. This option is useful if want to submit a batch job.

`fluent -gr` will run Cortex without graphics. This option can be used in conjunction with the `-i <journal>` option to run a job in "background" mode.

`fluent -gu` will run Cortex without the graphical user interface but will open graphics windows and display graphics objects. You cannot interact with the displayed graphics objects.

To start Fluent and immediately read a journal file, type `fluent -i <journal>`, replacing `<journal>` with the name of the journal file you want to read.

To start Fluent and immediately read a case file, type `fluent <dimensions and precision> -case <path to case>`, replacing `<path to case>` with the path to and name of the case file you want to read.

To start Fluent and immediately read a case and data file, type `fluent <dimensions and precision> -case <path to case> -data`, replacing `<path to case>` with the path to and name of the case file you want to read. The data file must be in the same directory as the case file and its name must exactly match that of the case file (except for the `.dat` file extension).

To start Fluent at a specific capability level (licensing level), type `fluent <dimensions and precision> -license=<enterprise or premium>`.

`fluent -h<heap size>` will update the heap space for Cortex processes to the specified size. For example, `fluent 3ddp -h50000000`. The default heap size is `18000000`. *Increasing the heap size will also increase Fluent's memory usage, which could negatively impact machine performance. Note that you cannot decrease the heap size below the default value of `18000000`.*

`fluent -nm` will run Cortex without displaying the mesh in the graphics window.

Important:

Download graphics card driver updates directly from the graphics card vendor's website, for example www.nvidia.com. Do not use the "Update Driver" feature offered by the operating system as these can sometimes update to an older version of the driver.

4.1.5.5. Meshing Mode Option

`fluent -meshing` specifies that Fluent opens in meshing mode rather than the default solution mode. See Introduction to Meshing Mode in Fluent (p. 217) for further details about the meshing mode.

4.1.5.6. Performance Options

`-cflush` specifies that memory is allocated in such a way as to ensure that all of the associated file cache buffers are flushed. This may resolve processing performance issues. For more details, see Clearing the Linux File Cache Buffers (p. 4353).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

`-platform=<x>` loads a binary that is specially ported for a particular platform. When `<x>=intel`, an AVX2 optimized binary is used that enhances performance when running on processors that support the AVX2 instruction set (available only on Linux).

`-stream` prints the memory bandwidth, using a variant of the STREAM benchmark. This information can be helpful in determining if your memory is set up in an optimal manner.

4.1.5.7. Parallel Options

These options are used in association with the parallel solver.

`-affinity=<x>` specifies the process binding (affinity) settings. The default behavior is optimized for the respective run scenarios. The available choices for this option depends upon the platform on which you are running:

- When running on Linux, you have the following choices:
  - If `<x>` = `off`, Fluent allows the message passing interface (MPI) to manage process binding. This corresponds to the default behavior.
  - If `<x>` = `core`, Fluent assigns each process to an individual core in an optimized manner.
  - If `<x>` = `sock`, Fluent assigns processes to all cores in sockets rather than the individual cores.
- When running on Windows, you have the following choices:
  - If `<x>` = `1`, Fluent manages the affinity, assigning each process to an individual core. This corresponds to the default behavior with shared memory on a local machine.
  - If `<x>` = `off`, Fluent allows the message passing interface (MPI) or the scheduler to manage process binding. This corresponds to the default behavior with distributed memory on a cluster or when running under the Microsoft Job Scheduler.
  - If `<x>` = `numa`, each process is assigned to all cores in a non-uniform memory access (NUMA) node while spreading the processes over the most possible NUMA nodes.

`-ccp <x>` (where `<x>` is the name of the head node) runs the parallel job through the Microsoft Job Scheduler as described in Starting Parallel Ansys Fluent with the Microsoft Job Scheduler (p. 4308).

`-cnf=<x>` (where `<x>` is the name of a hosts file or a list of Linux machines) spawns a compute node on each of the specified machines. For details, see Starting Parallel Ansys Fluent on a Windows System Using Command Line Options (p. 4306) or Starting Parallel Ansys Fluent on a Linux System Using Command Line Options (p. 4311).

`-gpgpu=<n>` specifies the number of general purpose graphics processing units (GPGPUs) per machine to be used in offload mode for AMG acceleration and/or DO acceleration. For more information, see Using General Purpose Graphics Processing Units (GPGPUs) With the Algebraic Multigrid (AMG) Solver (p. 4343) and Accelerating Discrete Ordinates (DO) Radiation Calculations (p. 4352), respectively.

`-gpu[=<n>}` specifies that the native GPU Solver is run, and you can specify which graphics processing units to use as needed; for example, `-gpu=1,2,4`. For further details, see Starting the Fluent GPU Solver from the Command Line (p. 3758).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

`-host_ip=<host:ip>` specifies the IP interface to be used by the host process.

`-mpi=<mpi>` specifies that `<mpi>` is to be used for the MPI. You can skip this flag if you choose to use the default MPI.

`-mpiopt=<x>` allows you to specify any additional MPI flags (`<x>`) to be included in the run (Linux only).

`-mpitest` runs the mpitest program instead of Ansys Fluent to test the network.

`-p<ic>` specifies the use of parallel interconnect `<ic>`, where `<ic>` can be any of the interconnects listed in Starting Parallel Ansys Fluent on a Windows System Using Command Line Options (p. 4306) or Starting Parallel Ansys Fluent on a Linux System Using Command Line Options (p. 4311).

`-pcheck` checks the network connections before spawning compute nodes (Linux only).

By default, the mpirun command (which launches the node processes) is executed on the compute node where the host process is spawned. You can use `-remote_node=<hostname>` to specify a different machine for the execution of this command. If `=<hostname>` is omitted, the first node in the hosts file will be used. (Linux only)

`-ssh` specifies that SSH should be used to spawn remote processes. (Beginning with Ansys Fluent R16.0, SSH is used by default. This option is included primarily for backward compatibility with existing launch scripts, etc.)

`-t<x>` specifies that `<x>` processors are to be used. For more information about starting the parallel version of Ansys Fluent, see Starting Parallel Ansys Fluent on a Windows System (p. 4306) or Starting Parallel Ansys Fluent on a Linux System (p. 4311).

`-tm<x>` specifies that `<x>` processors are to be used for meshing. This value must be less than or equal to the number of processes specified with `-t<x>`.

4.1.5.8. Postprocessing Option

`fluent -post` will run a version of Fluent that allows you to set up a problem or perform postprocessing, but will not allow you to perform calculations. Running Ansys Fluent for pre- and postprocessing requires you to use the `-post` flag on startup. To use this option on Linux, launch Ansys Fluent by adding the `-post` flag after the version number, for example,

```
fluent 3d -post
```

To use this same feature from the graphical interface on Windows or Linux, select the **Pre/Post Only** option in the **General Options** tab of Fluent Launcher, as described in Starting Ansys Fluent Using Fluent Launcher (p. 180).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.1.5.9. Remote Visualization Options

The `-sifile=<name>.txt` option starts Ansys Fluent and the server that is necessary for running the remote visualization client. For additional information on remote visualization, refer to Remote Visualization and Accessing Fluent Remotely.

Note:

You can specify the location for the server info file prior to the filename, for example `-sifile=D:/example_folder/server_info_example_name.txt`. If you do not provide a file path before the file name and you do not provide a path using the `SERVER_INFO_DIR` environment variable, then the file is saved in your working directory.

4.1.5.10. Scheduler Options

The `-scheduler=<scheduler>` option allows you to specify that your Linux session is run under a scheduler / load manager, where `<scheduler>` can be one of the following:

- `lsf`: this allows you to run Ansys Fluent under IBM Spectrum LSF software, and thereby take advantage of the checkpointing features of that load management tool. For further details, see Part 1: Running Fluent Under LSF.
- `pbs`: this runs Ansys Fluent under Altair PBS Professional, and allows you to use the features of this software to manage your distributed computing resources. For further details, see Part 2: Running Fluent Under PBS Professional.
- `sge`: this runs Ansys Fluent under Altair Grid Engine (formerly UGE, formerly SGE) software, and allows you to use the features of this software to manage your distributed computing resources. For further details, see Part 3: Running Fluent Under SGE.
- `slurm`: this runs Ansys Fluent under Slurm, and allows you to use the features of this software to manage your distributed computing resources. For further details, see Part 4: Running Fluent Under Slurm.

Note:

You can use the `-scheduler=<scheduler>` option along with `-gui_machine=<hostname>` or `-gui_machine` (described in Graphics and Files Options (p. 196)), in order to ensure optimal graphics performance. When running under Slurm, `-gui_machine=<hostname>` or `-gui_machine` is also needed to allow dynamic spawning (which is described in Dynamically Spawning Processes Between Fluent Meshing and Fluent Solution Modes (p. 220)), as well as the combination of Slurm + Open MPI + distributed memory on a cluster.

If you use custom scheduler scripts instead of relying on the standard Fluent option (`-scheduler=<scheduler>`), your environment variables related to the job scheduler will not be used unless you include the `-scheduler_custom_script` option with the Fluent options in your script.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Other options are available when you use a scheduler:

- You can specify the scheduler job submission machine name or (for SGE) the SGE qmaster using `-scheduler_headnode=<head-node>` (by default it is `localhost`).
- You can list all available queues using `-scheduler_list_queues`, Note that Fluent will not launch when this option is used.
- You can specify a queue / partition using `-scheduler_queue=<queue>`.
- You can use `-scheduler_nodeonly` to specify that Cortex and host processes are launched before the job submission and that only the parallel node processes are submitted to the scheduler.
- You can enable an additional option for the scheduler using `-scheduler_opt=<opt>`. Note that you can include multiple instances of this option when you want to use more than one scheduler option.
- You can specify the name and directory of the scheduler standard error file using `-scheduler_stderr=<err-file>` (by default it is saved as `fluent.<PID>.e` in the working directory, where <PID> is the process ID of the top-level Fluent startup script).
- You can specify the name and directory of the scheduler standard output file using `-scheduler_stdout=<out-file>` (by default it is saved as `fluent.<PID>.o` in the working directory).
- You can specify the working directory for the scheduler job using `-scheduler_workdir=<working-directory>`, so that scheduler output is written to a directory of your choice rather than the home directory or the directory used to launch Fluent.
- When running under Altair Grid Engine software, you can set the parallel environment using `-scheduler_pe=<pe>`.
- When running under Slurm, you can set the account using `-scheduler_account=<account>` and specify the number of node processes per cluster node (rather than leaving it to the cluster configuration) using `-scheduler_ppn=<x>`. When using the Fluent native GPU solver, you can specify the number of graphics processing units (GPUs) per cluster node using `-scheduler_gpn=<x>` (by default, <x>=0).
- You can enable a job-scheduler-supported native remote node access mechanism using `-scheduler_tight_coupling` in Linux when running under SGE or PBS Professional. Note that if you enable this option, it is not used if the Cortex process is launched after the job submission (which is the default when not using `-scheduler_nodeonly`) and is run outside of the scheduler environment by using the `-gui_machine` or `-gui_machine=`*hostname* option. For details about the MPI / job scheduler combinations that are supported for this tight coupling, see Running Fluent Using a Load Manager.

4.1.5.11. Text Command Option

The `-command="<TUI command>"` option allows you to specify that a single text command (`<TUI command>`) is executed at Fluent startup. The text command must be complete, that is, it cannot rely on further user input in the console after launching. Up to 10 instances of this option can be included, and the text commands will be executed in the order they are entered. When this

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

option is used along with the `-i <journal>` option, the text commands are executed before the journal.

For example, you could enter the following to read a case and start a calculation: `fluent 3d -command="file/read-case file_name.cas" -command="solve/initialize/initialize-flow" -command="sol iter 10"`.

4.1.5.12. Version, Release Options, and Environment Variables

Typing `fluent <version> -r`, replacing `<version>` with the desired version (`2d` or `3d`, or for double precision, `fluent` or `3ddp`), will list all releases of the specified version.

`fluent -r<x>` will run release `<x>` of Ansys Fluent.

Typing `fluent <version> -env`, replacing `<version>` with the desired version, will list all environment variables before running Ansys Fluent.

Including the `-setenv="<var>=<value>"` option sets the environment variable `<var>` explicitly to `<value>` before launching Ansys Fluent. Note that you can include as many instances of this option as you need to set all of the relevant environment variables. You can also unset an environment variable by entering `-setenv="<var>="`.

4.1.5.13. System Coupling Options

The following command line options (in either Windows or Linux) can be used when Ansys Fluent is involved in a system coupling simulation.

`-schost="<x>"` (where `<x>` is the name of the host machine, in quotes) specifies the host machine on which the coupling service is running (to which the co-simulation participant/solver must connect).

`-scport=<y>` (where `<y>` is the port number) specifies the port on the host machine upon which the coupling service is listening for connections from co-simulation participants.

`-scname="<z>"` (where `<z>` is the name of the participant, in quotes) specifies the unique name used by the co-simulation participant to identify itself to the coupling service (see Server File in the *System Coupling User's Guide* for more information).

The general syntax for invoking Ansys Fluent for system coupling is:

`fluent 3d –schost=`*host name in quotes* `–scport=`*port number* `–scname=`*name of the solver in quotes*

For instance:

```
fluent 3d –schost="machine1.domain.com" –scport=1234 –scname="Solution1"
```

Once Ansys Fluent loads the case, initialize the solution using the following command:

```
s i i
```

Once your case is initialized, start the system coupling by typing the following command in the Ansys Fluent text user interface (TUI):

```
(sc-solve)
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For more information, see Performing System Coupling Simulations Using Fluent (p. 4647), as well as the System Coupling User's Guide.

4.1.5.14. Other Startup Options

There are other startup options that are not listed when you type the `fluent -help` command. These options can be used to customize your graphical user interface. For example, to change the Ansys Fluent window size and position you can either modify the .Xdefaults file described in Customizing the Graphical User Interface (p. 935), or you can simply type the following command at startup:

```
fluent <version> -geometry <XX>x<YY>+<xx>-<yy>
```

where <XX> and <YY> are the width and height in pixels, respectively, and +<xx>-<yy> is the position of the window.

Therefore, typing `fluent 3d -geometry 700x500+20-400` will start the 3D version of Ansys Fluent, sizing the Ansys Fluent console to 700x500 pixels and positioning it on your monitor screen at +20-400.

There are additional Qt command line startup options for modifying the graphical stylesheet and more, which can be found in Qt documentation.

4.1.6. Aborting During Startup

By default, Fluent will abort during startup if it is not completed within 10 minutes, in order to avoid wasting resources. Note that this aborting will not happen in the following rare circumstances:

- when running in `-node_only` mode while submitting to a job scheduler
- when running under Microsoft job scheduler in non-batch mode
- when running with mixed Windows / Linux mode and the nodes are submitted to a job scheduler

If your startup is aborted, you should check your networks and computers to make sure they are available and operating appropriately. If you believe the systems are good, you can extend the time allowed before aborting by setting <*x*> to a number of seconds greater than 600 by performing the following:

- launch with the following environment variable: `FLUENT_START_COMPUTE_NODE_TIME_OUT=`<*x*>
- launch with the following command line option: `-setenv="FLUENT_START_COMPUTE_NODE_TIME_OUT=`<*x*>`"`

4.2. Running Ansys Fluent in Batch Mode

Ansys Fluent can be used interactively, with input from and display to your computer screen, or it can be used in a batch or background mode in which inputs are obtained from and outputs are stored in files. Generally you will perform problem setup, initial calculations, and postprocessing of results in an interactive mode. However, when you are ready to perform a large number of iterative calculations, you may want to run Ansys Fluent in batch or background mode. This allows the computer resources

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

to be prioritized, enables you to control the process from a file (eliminating the need for you to be present during the calculation), and also provides a record of the calculation history (residuals) in an output file. While the procedures for running Ansys Fluent in a batch mode differ depending on your computer operating system, Background Execution on Linux Systems (p. 204) provides guidance for running in batch/background on Linux systems, and Background Execution on Windows Systems (p. 205) provides guidance for running in batch/background on Windows systems.

For additional information, see the following sections:

4.2.1. Background Execution on Linux Systems

4.2.2. Background Execution on Windows Systems

4.2.3. Batch Execution Options

4.2.1. Background Execution on Linux Systems

To run Ansys Fluent in the background in a C-shell (csh) on a Linux system, type a command of the following form at the system-level prompt:

```
fluent 2d -g < inputfile > & outputfile &
```

or in a Bourne/Korn-shell, type:

```
fluent 2d -g < inputfile > outputfile 2>&1 &
```

In these examples,

- `fluent` is the command you type to run Ansys Fluent interactively.
- `-g` indicates that the program is to be run without the GUI or graphics (see Starting Ansys Fluent (p. 179)).
- `inputfile` is a file of Ansys Fluent commands that are identical to those that you would type interactively.
- `outputfile` is a file that the background job will create, which will contain the output that Ansys Fluent would normally print to the screen (for example, the menu prompts and residual reports).
- & tells the Linux system to perform this task in background and to send all standard system errors (if any) to `outputfile`.

The file `inputfile` can be a journal file created in an earlier Ansys Fluent session, or it can be a file that you have created using a text editor. In either case, the file must consist only of text interface commands (since the GUI is disabled during batch execution). A typical `inputfile` is shown below:

```
; Read case file
rc example.cas
; Initialize the solution
/solve/initialize/initialize-flow
; Calculate 50 iterations
it 50
; Write data file
wd example50.dat
; Calculate another 50 iterations
it 50
; Write another data file
wd example100.dat
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
; Exit Fluent
exit
yes
```

This example file reads a case file `example.cas`, initializes the solution, and performs 100 iterations in two groups of 50, saving a new data file after each 50 iterations. The final line of the file terminates the session. Note that the example input file makes use of the standard aliases for reading and writing case and data files and for iterating. (`it` is the alias for `/solve/iterate`, `rc` is the alias for `/file/read-case`, `wd` is the alias for `/file/write-data`, etc.) These predefined aliases allow you to execute commonly used commands without entering the text menu in which they are found. In general, Ansys Fluent assumes that input beginning with a `/` starts in the top-level text menu, so if you use any text commands for which aliases do not exist, you must be sure to type in the complete name of the command (for example, `/solve/initialize/initialize-flow`). Note also that you can include comments in the file. As in the example above, comment lines must begin with a `;` (semicolon).

An alternate strategy for submitting your batch run, as follows, has the advantage that the `output-file` will contain a record of the commands in the `inputfile`. In this approach, you would submit the batch job in a C-shell using:

```
fluent 2d -g -i inputfile  >& outputfile &
```

or in a Bourne/Korn-shell using:

```
fluent 2d -g -i inputfile > outputfile 2>&1 &
```

4.2.2. Background Execution on Windows Systems

To run Ansys Fluent in the background on a Windows system, the following commands can be used:

```
fluent 3d -g -i journal

fluent 3d -g -wait -i journal

fluent 3d -hidden -i journal
```

In these examples,

- `fluent` is the command you type to run Ansys Fluent interactively.
- `-g` indicates that the program is to be run minimized in the task bar.
- `-i journal` reads the specified journal file.
- `-wait` is the command you type in a DOS batch file or some other script in a situation where the script must wait until Ansys Fluent has completed its run.
- `-hidden` is similar to the `-wait` command, but also runs Ansys Fluent completely hidden and non-interactively.

To get an output (or transcript) file while running Ansys Fluent in the background on a Windows system, the journal file must contain the following command to write a transcript file:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
; start transcript file
 /file/start-transcript outputfile.trn
```

where the `outputfile` is a file that the background job will create, which will contain the output that Ansys Fluent would normally print to the screen (for example, the menu prompts and residual reports).

See Creating and Reading Journal Files (p. 972) for details about journal files. See Creating Transcript Files (p. 976) for details about transcript files.

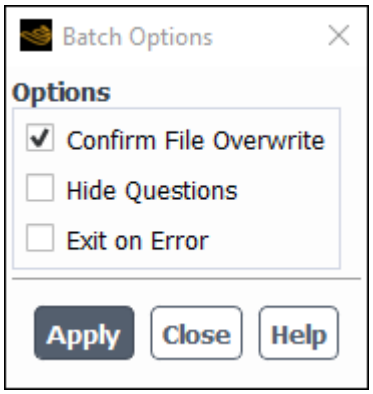
4.2.3. Batch Execution Options

During a typical session, Ansys Fluent may require feedback from you in the event of a problem it encounters. Ansys Fluent usually communicates problems or questions through the use of **Error** dialog boxes, **Warning** dialog boxes, or **Question** dialog boxes. While executing Ansys Fluent in batch mode, you may want to suppress this type of interaction in order to, for example, create journal files more easily.

There are three common batch configuration options available to you when running Ansys Fluent in batch mode. You can access these options using the **Batch Options** dialog box (Figure 4.6: The Batch Options Dialog Box (p. 206)).

File → Batch Options...

Figure 4.6: The Batch Options Dialog Box



The **Batch Options** dialog box contains the following items:

Confirm File Overwrite

determines whether Ansys Fluent confirms a file overwrite. This option is turned on by default.

Hide Questions

allows you to hide **Question** dialog boxes. This option is turned off by default.

Exit on Error

allows you to automatically exit from batch mode when an error occurs. This option is disabled by default.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When run in batch mode through the command prompt or a journal file with **Exit on Error** enabled, Fluent will exit under the following circumstances:

- Normal run termination upon reaching the end of a journal (return value `0`)
- Error returned during scripted text command execution (return value `1`)
- Unexpected input (wrong type) to text command (return value `1`)
- Licensing error (return value 2)

If an invalid text command is entered, Fluent will not exit, but proceed to the next text input.

Note that in Windows you must start Fluent with the `-wait` command line option.

`file` → `set-batch-options`

Any combination of these options can be turned on or off at any given time prior to running in batch mode.

Important:

Batch option settings are *not* saved with case files. They are meant to apply for the duration of the current Ansys Fluent session only. If you read in additional mesh or case files during this session, the batch option settings will not be altered. As batch options are not saved with case files, journal files developed for use in batch mode should begin by enabling the desired batch option settings (if different from the default settings).

4.3. Switching Between Meshing and Solution Modes

You can switch from the meshing mode of Fluent to the solution mode by clicking the **Switch to Solution** button, located by default in the top left corner of the application window. The mesh from your meshing mode session will be transferred and read in the new solution mode session.

You can switch from the solution mode of Fluent to the meshing mode by using the `switch-to-meshing-mode` text command. Note that this text command is only available for 3D sessions, before you have read a mesh or case file.

Note:

When you read a non-conformal interface case file into meshing mode and later switched to the solution mode, note the following limitations:

- Zone IDs may match, however, the corresponding zone names may be inconsistent.
- Boundary conditions on intersected threads are not preserved.
- Unassociated (dangling) non-conformal interface (NCI) surfaces remain present.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.4. Checkpointing an Ansys Fluent Simulation

The checkpointing feature of Ansys Fluent allows you to save case and data files while your simulation is running. While similar to the autosave feature of Ansys Fluent (Automatic Saving of Case and Data Files (p. 958)), which allows you to save files throughout a simulation, checkpointing allows you slightly more control in that you can save an Ansys Fluent job even after you have started the job and did not set the autosave option. Checkpointing also allows you to save case and data files and then exit out of Ansys Fluent. This feature is especially useful when you need to stop an Ansys Fluent job abruptly and save its data.

There are two different ways to checkpoint an Ansys Fluent simulation, depending upon how the simulation has been started.

1. Ansys Fluent running under LSF or SGE

   Ansys Fluent is integrated with load management tools like LSF and SGE. These two tools allow you to checkpoint any job running under them. You can use the standard method provided by these tools to checkpoint the Ansys Fluent job.

   For more information on using Ansys Fluent and SGE or LSF, see Part 3: Running Fluent Under SGE or Part 1: Running Fluent Under LSF, respectively.

2. Independently running Ansys Fluent

   When not using tools such as LSF or SGE, a different checkpointing mechanism can be used when running an Ansys Fluent simulation. You can checkpoint an Ansys Fluent simulation while iterating / time stepping, so that Ansys Fluent saves the case and data files and then continues the calculation, or so that Ansys Fluent saves the case and data files and then exits.

   - Saving case and data files and continuing the calculation:

     On Linux, create a file named `check-fluent` in the working directory; on Windows, create a file named `check-fluent.txt` in the working directory. After the case and data files are saved, the `check-fluent` / `check-fluent.txt` file will be automatically deleted. Note that if multiple Fluent sessions are running with the same working directory, case and data files will be saved for all of them.

   - Saving case and data files and exiting Ansys Fluent:

     On Linux, create a file named `exit-fluent` in the working directory; on Windows, create a file named `exit-fluent.txt` in the working directory. After the case and data files are saved and Fluent exits, the `exit-fluent` / `exit-fluent.txt` file will be automatically deleted.

     Note that if you are running multiple Fluent sessions with the same working directory, case and data files will be saved for all of them, but only a single session will exit. To ensure that your intended session exits, you must type the process ID of the Cortex process for that session as the first line of text in the `exit-fluent` / `exit-fluent.txt` file. You can find this process ID by generating a transcript file, as the process ID is included as the last string of digits in the name of the transcript file (for details, see Creating Transcript Files (p. 976)). Alternatively, the process ID can be found by using operating system tools from a command prompt; for example, you could use the `ps` command in Linux and look for the `PID` of the `CMD` named `cortex.24.2.0`, or you could use the `tasklist` command in Windows and look for the `PID` of the `Image Name` named `cx2420.exe`.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The saved case and data files will have the current iteration number appended to their file names.

Ansys Fluent offers an alternate way to checkpoint an unsteady simulation. While the default behavior is to checkpoint the simulation at the end of the current iteration, for unsteady simulations you have the option of completing all of the iterations in the current time step before checkpointing. This can be set by entering the following Scheme command prior to running the unsteady simulation:

```
(ckpt/time-step? #t)
```

Now when you save the checkpoint file (as described previously), the case and data file will be saved at the end of the current time step and named accordingly. To switch back to the default checkpointing mechanism at the end of the current iteration, use the following Scheme command:

```
(ckpt/time-step? #f)
```

Important:

Note that the `(ckpt/time-step? #t)` command will have the effect only in the case of an unsteady simulation.

Note:

It is recommended that you do *not* use checkpointing when using Ansys Fluent in Workbench. However, if checkpointing is necessary, the `exit-fluent` / `exit-fluent.txt` file can be used and the file will be checked in its default location (the FFF/FLU system directory containing the `*.set` file). If Ansys Fluent is calculating, then the existence of the file is equivalent to an **interrupt** command. Similarly, the `check-fluent` / `check-fluent.txt` file can be used to save the project on demand when Ansys Fluent is calculating.

4.5. Cleaning Up Processes From an Ansys Fluent Simulation

Ansys Fluent lets you easily remove extraneous processes in the event that an Ansys Fluent simulation must be stopped.

When a session is started, Ansys Fluent creates a `cleanup-fluent` script file. The script can be used to clean up all Ansys Fluent-related processes. Ansys Fluent creates the cleanup-script file in the current working directory with a filename that includes the machine name and the process identification number (PID) (for example, `cleanup-fluent-mymachine-1234`).

If the current directory does not possess the proper write permissions, then Ansys Fluent will write the cleanup-script file to your home directory.

If, for example, Ansys Fluent is started on a machine called `thor` and the process identification number is `32895`, Ansys Fluent will create a cleanup-script called `cleanup-fluent-thor-32895` in the current directory. To run the cleanup-script, and clean up all Ansys Fluent processes related to your session, on Linux platforms, type the following command in the console window:

```
sh cleanup-fluent-thor-32895
```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Or, if the shell script already has executable permissions, simply type:

```
cleanup-fluent-thor-32895
```

To clean up extraneous Ansys Fluent processes on Windows (serial or parallel), double-click the corresponding batch file (for example, `cleanup-fluent-thor-32895.bat`) that Ansys Fluent generates at the beginning of each session.

Important:

During a normal run, this file will be deleted automatically after exiting Ansys Fluent. In abnormal situations, you may use this batch file to clean up the Ansys Fluent processes. Once an Ansys Fluent session has been closed, you can safely delete any left over cleanup scripts from your working directory.

Important:

If an Ansys Fluent session hangs or freezes on Windows, and you want to view the complete contents of the Ansys Fluent console output in a transcript file, you should use the `taskkill` command through the DOS command prompt, rather than terminating the Ansys Fluent application through the **Windows Task Manager**.

4.6. Exiting Ansys Fluent

You can exit Ansys Fluent by selecting **Exit** in the **File** ribbon tab. If the present state of the program has not been written to a file, a **Question** dialog box will open to confirm if you want to proceed. You can cancel the exit and write the appropriate file(s) or you can continue to exit without saving the case or data.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Glossary of Terms

This glossary contains a listing of terms commonly used throughout the documentation.

- adaption (p. 211)
- case files (p. 211)
- cell types (p. 212)
- computational fluid dynamics (CFD) (p. 212)
- console (p. 213)
- convergence (p. 213)
- cortex (p. 213)
- data files (p. 213)
- dialog boxes (p. 213)
- discretization (p. 213)
- GUI (p. 213)
- mesh (p. 213)
- models (p. 213)
- node (p. 213)
- postprocessing (p. 213)
- residuals (p. 213)
- skewness (p. 214)
- solvers (p. 214)
- terminal emulator (p. 214)
- TUI (p. 214)

adaption

A technique useful in improving overall mesh quality. The solution-adaptive mesh refinement feature of Ansys Fluent allows you to refine and/or coarsen your mesh based on geometric and numerical solution data. In addition, Ansys Fluent provides tools for creating and viewing adaption fields customized to particular applications.

case files

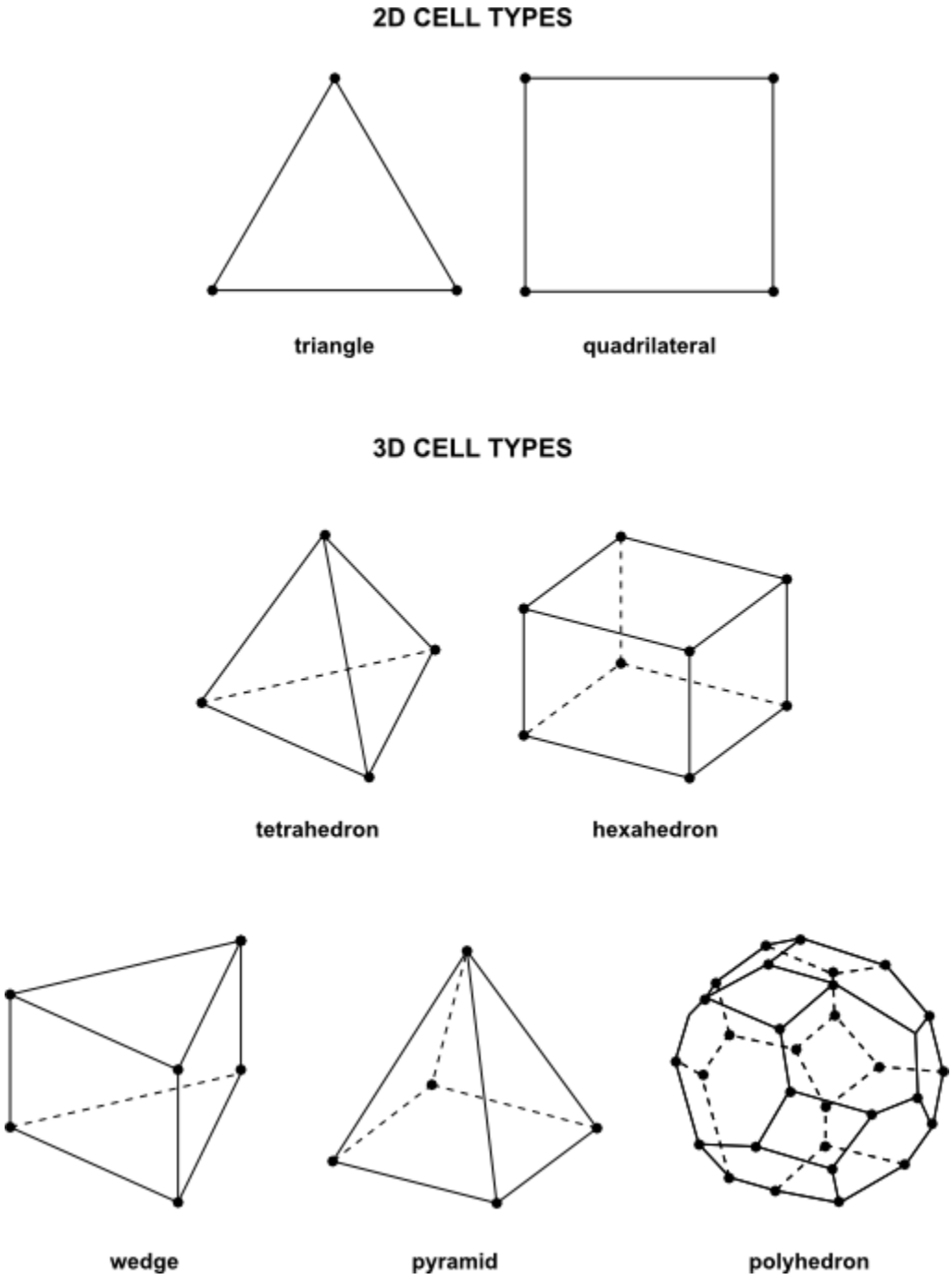
Files that contain the mesh, boundary conditions, and solution parameters for a problem. A case file also contains the information about the user interface and graphics environment.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

cell types

The various shapes or units that constitute the base elements of a mesh. Ansys Fluent can use meshes composed of tetrahedral, hexahedral, pyramid, wedge, or polyhedral cells (or a combination of these).

Figure 8: Cell Types



computational fluid dynamics (CFD)

The science of predicting fluid flow, heat transfer, mass transfer (as in perspiration or dissolution), phase change (as in freezing or boiling), chemical reaction (for example, combustion), mechanical movement (for example, fan rotation), stress or deformation of re-

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	lated solid structures (such as a mast bending in the wind), and related phenomena by solving the mathematical equations that govern these processes using a numerical algorithm on a computer.
console	The console is part of the Ansys Fluent application window that allows for text command input and the display of information.
convergence	The point at which the solution is no longer changing with each successive iteration. Convergence criteria, along with a reduction in residuals, also help in determining when a solution is complete. Convergence criteria are pre-set conditions on the residuals that indicate that a certain level of convergence has been achieved. If the residuals for all problem variables fall below the convergence criteria but are still in decline, the solution is still changing to a greater or lesser degree. A better indicator occurs when the residuals flatten in a traditional residual plot (of residual value vs. iteration). This point, sometimes referred to as convergence at the level of machine accuracy, takes time to reach, however, and may be beyond your needs. For this reason, alternative tools such as reports of forces, heat balances, or mass balances can be used instead.
cortex	A utility that manages Ansys Fluent's user interface and basic graphical functions.
data files	Files that contain the values of the flow field in each grid element and the convergence history (residuals) for that flow field.
dialog boxes	The separate windows that are used like forms to perform input tasks. Each dialog box is unique and employs various types of input controls that make up the form.
discretization	The act of replacing the differential equations that govern fluid flow with a set of algebraic equations that are solved at distinct points.
GUI	The graphical user interface, which consists of the main Ansys Fluent application window, dialog boxes, graphics windows, etc.
mesh	A collection of points representing the flow field, where the equations of fluid motion (and temperature, if relevant) are calculated.
models	Numerical algorithms that approximate physical phenomenon (for example, turbulence).
node	The distinct points of a mesh (p. 213) at which the equations of fluid motion are solved.
postprocessing	The act of analyzing the numerical results of your CFD simulation using reports, integrals, and graphical analysis tools such as contour plots, animations, etc.
residuals	The small imbalance that is created during the course of the iterative solution algorithm. This imbalance in each cell is a small, non-zero

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	value that, under normal circumstances, decreases as the solution progresses.
skewness	The difference between the shape of the cell and the shape of an equilateral cell of equivalent volume. Highly skewed cells can decrease accuracy and destabilize the solution.
solvers	Ansys Fluent has two distinct solvers, based on numerical precision (single-precision vs. double-precision). Within each of these categories, there are solver formulations: pressure based; density based explicit; and density based implicit.
terminal emulator	See console (p. 213).
TUI	The text user interface, which consists of textual commands that can be entered into the terminal emulator.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Part II: Meshing Mode

The section describes how to use Ansys Fluent in meshing mode.

- Introduction to Meshing Mode in Fluent (p. 217) introduces the meshing mode in Fluent, gives an overview of its capabilities, an describes how to get started, along with introducing the graphicsl and textual user interface..
- Getting Started with the Fluent Guided Workflows (p. 277) introduces the task-based guided meshing workflows that make it easier to generate a volume mesh starting from CAD geometries.
- Using the Watertight Geometry Meshing Guided Workflows (p. 293) describes the Watertight Geometry guided workflow that can be applied to most CAD geometries.
- Using the Fault-tolerant Meshing Workflow (p. 389) describes the Fault-tolerant Meshing guided workflow that can be applied to more complicated and/or problematic CAD geometries.
- Improving and Examining the Mesh and its Quality (p. 511) describes the options available for examining the mesh graphically, improving the quality of a volume mesh, and checking the mesh diagnostically.
- Advanced Meshing Topics (p. 577) describes more advanced meshing tools and techniques.
- Appendix A: Importing Boundary and Volume Meshes (p. 853) describes filters that you can use to convert data from various software packages to a form that can be read.
- Appendix B: Mesh File Format (p. 861) describes the format of the mesh file.
- Appendix C: Shortcut Keys (p. 877) lists all the hot-keys (shortcut keys) available.
- Bibliography (p. 891) presents the bibliography for the previous chapters.

Chapter 1: Introduction to Meshing Mode in Fluent

When in meshing mode, Fluent functions as a robust, unstructured mesh generation program that can handle meshes of virtually unlimited size and complexity. Meshes may consist of tetrahedral, hexahedral, polyhedral, prismatic, or pyramidal cells. Unstructured mesh generation techniques couple basic geometric building blocks with extensive geometric data to automate the mesh generation process.

A number of tools are available for checking and repairing the boundary mesh to ensure a good starting point for generating the volume mesh. The volume mesh can be generated from the boundary mesh using one of the approaches described.

The user interface is written in the Scheme language, which is a dialect of LISP. Most features are accessible through the graphical interface or the interactive menu interface. The advanced user can customize and enhance the interface by adding or changing the Scheme functions.

1.1. Meshing Approach

1.2. Meshing Mode Capabilities

1.3. Starting Fluent in Meshing Mode

1.4. Graphical User Interface

1.5. Text User Interface

1.6. Reading and Writing Files

1.1. Meshing Approach

There are two principal approaches to creating meshes in Ansys Fluent Meshing:

- Generate a tetrahedral, hexcore, or hybrid volume mesh from an existing boundary mesh. In this case, you can import a boundary mesh from Ansys Meshing or a third-party mesh generation package. You can import boundary meshes created in CAD/CAE packages by using the appropriate menu item in the **File** → **Import** submenu (or the associated text commands), or you can convert them using the appropriate stand-alone grid filter.

- Generate a tetrahedral, hexcore, or hybrid volume mesh based on meshing objects from a faceted geometry (from CAD or the `.tgf` format from Ansys Meshing). In this case, you need to create a conformally connected surface mesh using the object wrapping and sewing operations before generating the volume mesh. You can alternatively use the CutCell mesher to directly create a hex-dominant volume mesh for the geometry (imported from CAD or the `.tgf` format from Ansys Meshing) based on meshing objects.

When the mesh generation is complete, you can transfer the mesh to solution mode using the Mode toolbar or the command `switch-to-solution-mode`. The remaining operations—such as setting boundary conditions, defining fluid properties, executing the solution, and viewing and postprocessing the results—are performed in solution mode (see the User's Guide (p. 1) for details).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.2. Meshing Mode Capabilities

When in meshing mode, Fluent:

- Functions as a robust, unstructured volume mesh generator
- Generates volume meshes that can be transferred to solution mode in Fluent
- Uses the Delaunay triangulation method for tetrahedra
- Uses the advancing layer method for prisms
- Generates hexcore mesh
- Has a robust surface wrapper tool
- Includes size functions that can produce ideal size distributions for many CFD calculations
- Can directly create a hex-dominant mesh on faceted geometry (using the CutCell mesher)
- Can export polyhedral cells
- Has tools for checking, repairing, and improving boundary mesh to ensure a good starting point for the volume mesh
- Can manipulate face/cell zones
- Is flexible—it allows the most appropriate cell type to be used to generate the volume mesh:
  - Tet meshes are suitable for complex geometries.
  - Hexcore meshes can combine the flexibility of tet, hex, and prism meshes with a smaller cell count and higher hex-to-tet ratio.
  - CutCell (hex-dominant) meshes can be directly created from faceted geometry and can also be combined with prism layers.
  - The Rapid Octree mesher directly operates on faceted geometry and offers a fast and robust meshing capability on high core counts.
- Has hybrid meshes:
  - Prism layers near walls allow proper boundary layer resolution.
  - Allows flow alignments with mesh lines.
  - Generates smaller volume mesh with highly stretched prismatic elements.
- Has non-conformal meshes:
  - Suitable for studies involving selective replacement of parts.
  - Meshes generated separately can be glued together.

1.3. Starting Fluent in Meshing Mode

Starting Fluent in meshing mode is accomplished by selecting the **Meshing** option in the Fluent Launcher, or by adding the directive `-meshing` when using the command line interface.

See Starting and Executing Ansys Fluent (p. 179) for full details on setting dimension and other options for starting in meshing mode.

The .tgrid File

When starting up in meshing mode, Fluent looks in your home directory for an optional file called `.tgrid`. This file is then loaded using the Scheme function `load`. You can use the `.tgrid` file to customize the operation of the code in meshing mode.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For example, the Scheme function `ti-menu-load-string` is used to include text commands in the **.tgrid** file. If the `.tgrid` file contains `(ti-menu-load-string "file read-case test.msh")`, then the mesh file `test.msh` will be read in. For more details about the function `ti-menu-load-string`, see Text Menu Input from Character Strings in the *Fluent Text Command List*.

Important:

Another optional file, `.fluent`, if present, is also loaded at start up. This file may contain Scheme functions that customize the operation of the code in solution mode. When both the `.tgrid` and `.fluent` files are present, the `.fluent` file will be loaded first, followed by the `.tgrid` file, when the meshing mode is launched. Hence, the functions in the `.tgrid` file will take precedence over those in the `.fluent` file for the meshing mode.

The `.fluent` file is not loaded again automatically when switching to solution mode from meshing mode. You will need to load the file separately using the Scheme `load` function, if needed.

1.3.1. Starting the Dual Process Build

The dual process build allows you to run Cortex on your local machine (host) and Fluent on a remote machine. The advantage of using the dual process build is faster response to graphics actions (such as zoom-in, zoom-out, opening a dialog box, and so on) when you use Fluent remotely. If the network connectivity is slow, then graphics actions may appear slow and jerky. By controlling the graphics actions locally, the slow response of the graphics actions can be avoided. For example, if you are handling a big mesh (such as the underhood mesh), you can start a dual process build to run Fluent remotely with only the display set to your local machine.

To start the dual process build of Fluent in meshing mode, do the following:

1. Start Fluent on your local machine using the command `fluent -serv -meshing`.

   The Fluent window will appear with the version prompt in the console.

2. Type `listen` and press **Enter**.

   You will be prompted for a timeout (the period of time to wait for a connection from remote Fluent). The default value is `300` seconds. You can also specify the timeout value based on your requirement. Utilize this time to log into the remote machine and to start Fluent.

3. Press **Enter** again.

   A message will prompt you to start Fluent on the remote machine with the following arguments:

   ```
   -cx host:p1:p2
   ```

   where,

   `host` is the name of the host (local) machine on which Cortex is running.

   `p1` and `p2` are the two integers indicating the connecting port numbers that are used to communicate information between Cortex on the host machine and Fluent on the remote machine.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Login to the remote machine and set the display to the host machine.

5. Start Fluent from the remote machine using the following command: `fluent 3d -cx host:p1:p2`

   The host and port numbers are displayed in the message window.

Note:

The user interface commands related to the **File** menu (such as reading files, importing files) and other **Select File** dialog boxes do not work for the dual process build. You need to use the TUI commands instead (for example, `/file/read-mesh`).

Important:

- The host cannot be detached and reattached; once the connection is broken the data is lost. You need to save the data if the machine needs to be shut down in between.
- All graphics information will be sent over the network, so initially it could take a long time to assemble graphical information (especially if the host and remote server are across continents) but after that the graphics manipulation is fast.

1.3.2. Dynamically Spawning Processes Between Fluent Meshing and Fluent Solution Modes

For parallel simulations started in meshing mode with more than one process, the Fluent session will be started by using the number of processes requested for meshing. When switching to solution mode, Fluent will automatically spawn the remaining parallel node processes needed to achieve the requested number of total solution processes. Even though automatic spawning is used by default at start up for runs using more than one process, you can always change the number of additional processes to be spawned before switching to solution mode using the `/parallel/spawn-solver-process` text user interface (TUI) command.

This text command prompts you for:

- Total number of desired processes (must be greater than or equal to the number of meshing processes). Fluent will spawn additional processes as necessary.
- Interconnect type to be used for the distributed parallel simulation. You can choose from `infiniband`, `ethernet`, `shared memory`, or you can retain the default value.
- Machine list or host file. If you decide to run in parallel using more than one machine, then you should provide the machine list or host file, otherwise, you can skip this option by pressing **Enter**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Option to use `ssh` for distributed simulations, otherwise, you can skip this option by pressing **Enter** to retain the default option.

Important:

Note the following:

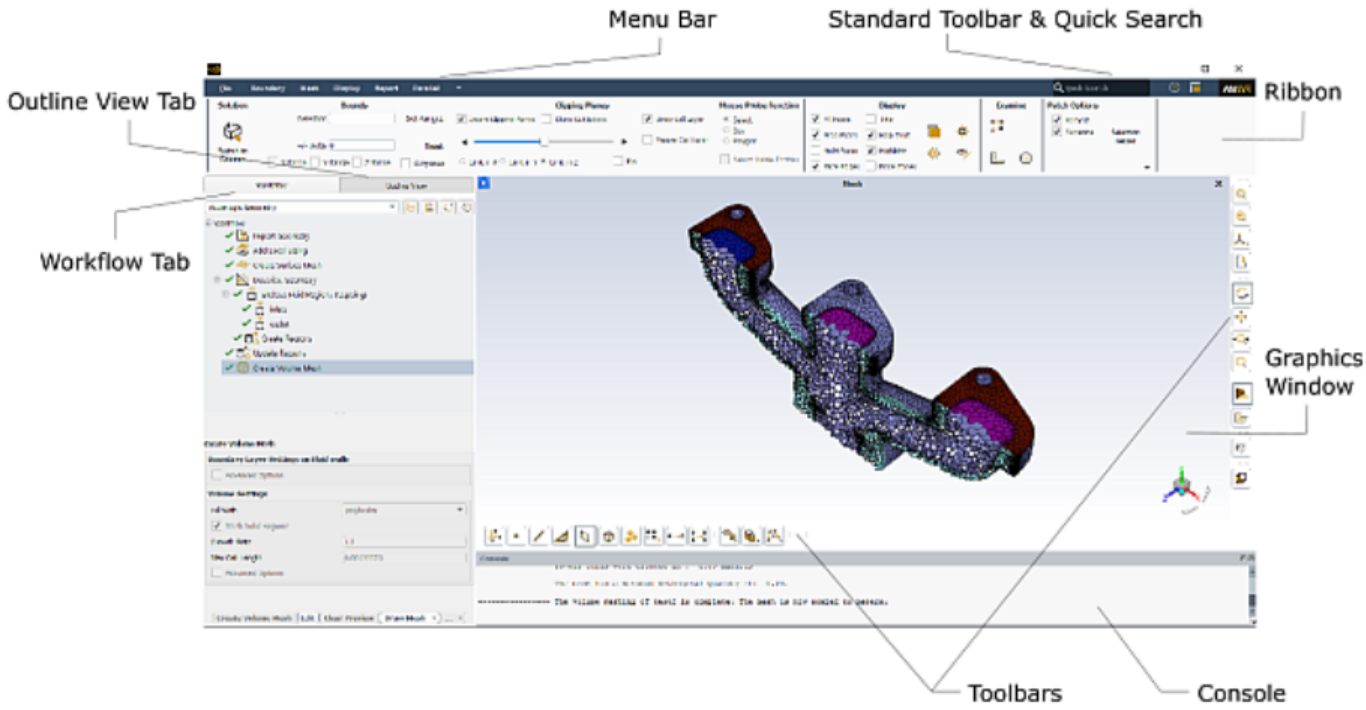
- Dynamic process spawning is not supported on Windows. While you can keep the same number of cores for meshing mode as for solver mode, it is recommended that you continue the solver iterations using another separate solver session.
- Dynamic process spawning is only supported on Linux, with the following MPIs:
  - `default`
  - `intel`
- While running under the Fluent–supported load managers (for example, SGE/LSF/PBS Pro/Slurm), the total number of required parallel node processes must be requested at the start of Fluent session, and Fluent will initially start with the specified number of processes for meshing mode and will automatically spawn the remaining node processes while switching to solution mode. Note that to allow such dynamic spawning when running under Slurm, you must include the `-gui_machine=<host-name>` command line option or select either **Current Machine** or **Specify Machine** from the **Graphics Display Machine** list in the **Scheduler** tab of Fluent Launcher.
- Using a single machine host file with different numbers of meshing and solver cores at the start of an Ansys Fluent session may result in an unintended host allocation. Instead, you should specify a machine host file only for the initial (meshing) mode. Then before switching from the meshing to the solution mode, you should use the `parallel/spawn-solver-processes` text command to request additional cores and specify a separate machine host file for the additional cores.

1.4. Graphical User Interface

The graphical user interface (GUI) components are illustrated in Figure 1.1: The User Interface Components (p. 222). The interface will change depending on whether you are in meshing mode (as described in this guide) or solution mode (as described in the Fluent User's Guide (p. 1)). For details on switching between the meshing and solution mode, see Solution (p. 223).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.1: The User Interface Components



Object-based meshing is a context-driven, visual workflow, accessible using the major interface components. A complete description of the components is found in the User Interface Components (p. 222) section.

Menu bar commands are appropriate for zone-based meshing or advanced display and report options. Full descriptions of the menu commands are in their related chapters in this manual.

Some of the user interface elements can be moved or tabbed together to suit your preferences. You can also modify attributes of the interface (including colors and text fonts) to better match your platform environment. These are described in Customizing the User Interface (p. 246).

The help button ( ) accesses a drop down list for quick access to the integrated help system, including the Fluent User's Guide. The Fluent integrated help system is described in detail in Using the Help System (p. 249).

1.4.1. User Interface Components

The components are described in detail in the subsequent sections.

1.4.1.1. The Ribbon
1.4.1.2. The Workflow Tab
1.4.1.3. The Outline View Tab
1.4.1.4. The Graphics Window
1.4.1.5. Quick Search
1.4.1.6. The Console
1.4.1.7. The Toolbars

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.4.1.8. ACT Start Page

1.4.1.1. The Ribbon

In **Mesh Generation** mode, the ribbon contains options to help with managing the graphical display, selecting objects or zones, and patching options.

Note:

When working with CAD Assemblies, certain meshing ribbon tools are disabled.

The hide ribbon button ( ) is used to minimize the ribbon, allowing more area for the graphics window. Click a second time to maximize the ribbon to restore the graphics window area.

Solution

The **Switch to Solution** option enables you to switch from meshing mode to solution mode. It transfers all of the volume mesh data from meshing mode to solution mode in Ansys Fluent. You will be asked to confirm the mesh is valid and that you want to switch to solution mode.

Important:

- Only volume meshes can be transferred to solution mode; surface meshes cannot be transferred.

  Face zones which are not connected to volume mesh (geometry objects or unreferenced zones in case of mesh object-based workflow) will be transferred as imported surfaces when the volume mesh is transferred from meshing to solution mode. Also, any unmeshed face zones connected to volume mesh (mesh object with some regions filled or unreferenced zones), will be disconnected and transferred as imported surfaces in solution mode.

- You should check that the mesh quality is adequate before transferring the mesh data to solution mode. See Checking the Mesh (p. 530) and Checking the Mesh Quality (p. 534) for details. When you are satisfied with the quality of the generated mesh, you can proceed to solution mode.

- Hanging-node meshes are converted to polyhedra during mesh transfer.

Important:

In object-based workflows, merging cell zones requires that they be in the same volumetric region.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To merge cell zones that cannot be in the same volumetric region because they are not contiguous, you will need to first delete the object(s) only, and then use the **Manage Cell Zones** dialog box.

Note:

- The command `switch-to-solution-mode` corresponds to the **Switch to Solution** button.
- You cannot switch back from solution mode to meshing mode after the mesh data has been transferred.

  However, if no file has been read in solution mode, you can use the command `switch-to-meshing-mode` to switch to meshing mode to generate a mesh to be transferred, if you desire.
- The meshing and solution modes in Fluent have different options available for some user configuration settings. Thus, these configuration settings may be changed when switching from meshing to solution mode and may not be the same when returning to one mode after using the other.

Bounds

Use the **Bounds** group to limit the display region based on proximity to a selected entity in your model.

- **Selection** is used to specify the entity on which the Bounds are centered.

  You can set a selection filter and then click to select the entity in the graphics window.
- Set symmetrical upper and lower distance limits in the **+/- Delta** text box.

  Limit the bounds directionally with the X-, Y-, and Z-Range checkboxes.
- Use **Set Ranges** to apply the display limits. **Reset** disables the bounds display. You will have to redraw to see the effect.
- If **Cutplanes** is checked, the display region is linked to the Bounds tab in the **Display Grid** dialog box. You may insert up to six cutplanes (two in each of the x-, y-, and z-direction) and asymmetrically control their location. See Generating the Mesh Display Using the Display Grid Dialog Box (p. 537).

Clipping Planes

Crops the display region along either a line drawn in the graphics window or along the coordinate system axes when **Insert Clipping Planes** is enabled. The slider allows interactive position of the clipping plane.

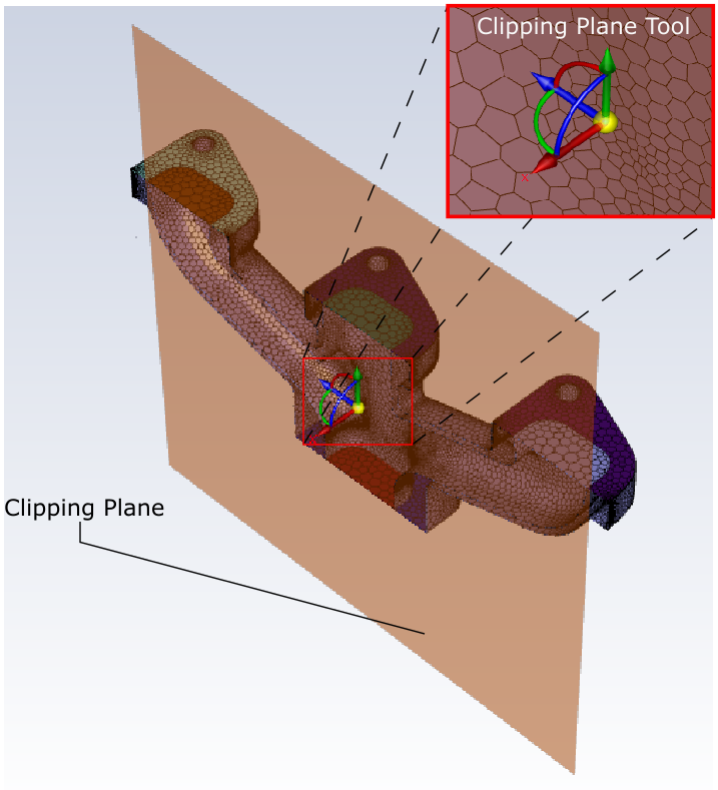
To create a custom clipping plane you can click the **Insert Drag Line** button ( ) which allows you to define an arbitrary plane by drawing in the graphics window. After clicking the **Insert Drag Line** button you can then click once in the graphics window to initiate

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the drag line for the plane. After initiating the drag line, move your cursor across the model, then click again to end the drag line and create the custom plane.

The clipping plane is visible in the graphics window and includes its own clipping tool triad, allowing you to manipulate the plane directly in the graphics window. Select an element of the clipping tool triad, hold down the mouse and then drag, to move or rotate the clipping plane in the graphics window.

Figure 1.2: Clipping Plane and Clipping Plane Tool



Note:

The normal of the clipping plane is always oriented along the x-axis of the local coordinate system, hence differing in orientation at times from the graphics window triad and its global coordinate system orientation.

Note:

To hide the clipping plane, use the command `"(cx-hide-plane-tool "clipping-plane")"`. To toggle the display of the clipping plane's triad tool, use either the **F10** hotkey or the `display/show-hide-clipping-plane-triad` text command.

You can control the size of the clipping plane triad using the **Graphics triad tool size** field in the **Graphics** branch of **Preferences** (**File**>**Preferences...**). *Note that the updated tool size only appears after restarting the Fluent application.*

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Use the **Flip** checkbox to reverse the direction of the clipping plane.

Use the **Show Cut Edges** option to display the cut edges of the model exposed by the clipping plane. This option is disabled by default.

Use the **Draw Cell Layer** checkbox to visualize a layer of cells of the volume mesh on the clipping plane. This option is disabled by default. Once the **Draw Cell Layer** option is enabled, use the **Freeze Cell Layer** checkbox to keep the displayed layer of cells in place while you perform additional mesh display operations.

To draw a layer of cells on the clipping plane for specific cell zones, select the cell zone(s) under Mesh Objects in the Tree and use the **Draw Cell Layer** option in the context menu.

Selection Helper

Use the **Selection Helper** group to assist in selecting face zones, edge zones, objects, object face zones, or object edge zones by a **Name Pattern** and **Geometry Recovery** level. Select the type of zone or object using the **Filter** drop down list, then use the **Name Pattern** field to refine your selection. When selecting **Face Zones** in the **Filter** list, the **Geometry Recovery** option is available to further refine your selection.

The **Advanced...** button opens the **Zone Selection Helper** dialog box. Use this dialog box to expand the zone selection criteria to include the number of entities present in them, or using the minimum or maximum face zone area.

The **Selection Helper** options and the **Zone Selection Helper** dialog box may be used with *all* dialog boxes that contain zone or object lists (for example, **Cell Zones** and **Boundary Zones** dialog boxes).

Mouse Probe Function

Use the **Mouse Probe Function** group to set the behavior of the mouse probe button.

Select

enables the selection of a single entity based on the filter selected and also adds the selected entity to a list that can be used in most dialog boxes.

Box

enables the selection of a group of entities within a box. To define the selection box, click the mouse probe button at one corner of the region to be selected, drag the mouse to the opposite corner, and release the mouse probe button.

Polygon

enables the selection of a group of entities within a polygonal region. To define the selection polygon, click the mouse probe button at one vertex of the polygonal region to be selected, and use the left mouse button to successively select each of the remaining vertices.

Click the mouse probe button again (anywhere in the graphics window) to complete the polygon definition.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select Visible Entities

enables selection of only visible entities (nodes, faces, zones, objects) when the mouse probe function is set to **Box** or **Polygon**. The selection includes only entities visible to the eye, and not those hidden behind other entities in the display. This option is disabled by default. When enabled, ensure that the model is zoomed to an appropriate level for correct selection.

Note:

- If the mesh is not connected, all entities (nodes, faces, zones, objects) will be selected irrespective of whether they are visible or not.
- This visual selection behavior works only on local displays and may generate warning messages when attempting selection on a remote system.

Note:

For additional mouse probe function options, see Controlling the Mouse Probe Function (p. 551)

Display

contains options to control the display in the graphics window.

All Faces

enable or disable the display of all faces in the visible zones or objects, colored by their zone type.

Free Faces

enable or disable the display and highlighting of free faces on the visible zones or objects. A free face is one having at least one edge not shared with a neighboring face.

Multi Faces

enable or disable the display of multi-connected faces on the visible zones or objects, along with their nodes. A multi-connected face is a boundary face that shares an edge with more than one other face, while a multi-connected node is a node that is on a multi-connected edge (that is, an edge that is shared by more than two boundary faces).

Face Edges

enable or disable the display of face edges in the visible zones or objects. This option works in conjunction with the **All Faces** option.

Title

enable or disable the display of the caption block area below the graphic containing date, product, and contents of the display.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Help Text

enable or disable the display of help text for tool buttons or hot-keys. Detailed help is displayed whenever a command is selected by clicking a button (or pressing a hotkey on the keyboard), and remains visible until the command is complete or another command selected.

Highlight

enable or disable the highlighting of objects, face zone labels, volumetric regions, or cell zones selected in the tree.

Edge Zones

enable or disable the display of edge zones comprising the objects drawn in the graphics window.

Marked Faces

enable or disable the display of marked faces in the graphics window.

Transparency

toggle the transparency of the selected objects/zones depending on the mode of selection set. If no object/zone is selected then the entire geometry is made transparent so that internal objects/zones are visible.

Explode

toggle between a normal view and an exploded view of the objects in the geometry.

Edge Zone Selection Mode

enable or disable the edge zone selection mode. This restricts selection to edge zones entities only.

Edges

show or hide edges on selected zones and objects independent of the mode of selection. If no object/zone is selected, then the edges on the displayed zones/objects are shown/hidden.

Examine

contains options for obtaining additional information about the selected entities.

Centroid

prints the coordinates of the centroid of the selected face to the console.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This also works for edges and nodes.

Distance

calculates and displays the distance between two selected locations or nodes. The tool prompts you when more selections are needed, otherwise, it uses the two most recent selections (nodes or coordinates) in the selections list.

Distances can be measured between points, edges, faces, and zones. The distance between zones is computed using the global average centroid of each zone facet to the relative zone distance. So, for calculating the distance between two planar zones, the distance would be based on the starting and ending points being located directly on the zone surfaces; whereas for calculating the distance between two curved surfaces, the distance would be based on the starting and ending points being located offset from the zone surface(s).

Entity Information

prints detailed information about the selected entities in the message window. For more details, see Entity Information (p. 888).

In addition, if a selected zone or object has been set as a **target**, this will toggle the identifying color.

Patch Options

contains additional options applicable to the patching tools.

Remesh

enables automatic remeshing of the patched area.

Separate

enables you to create a separate face zone/object for the new faces created. Additional options for object/zone granularity and type are available in the **Patch Options** dialog box (see Using the Patch Options Dialog Box (p. 620) for details).

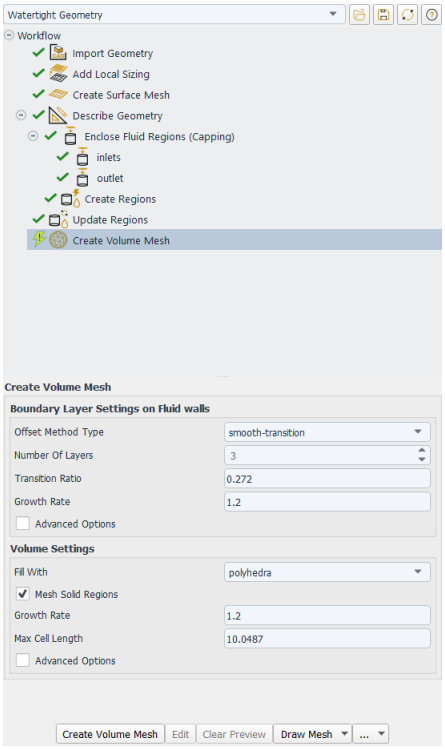
The **Remesh** and **Separate** options are enabled by default.

1.4.1.2. The Workflow Tab

Use the **Workflow** tab to access guided workflow templates that can be used to streamline the use of Fluent in meshing mode. Common tasks and property settings are available to walk you through the process of generating a volume mesh for use in the Fluent solver. For instance, the Watertight Geometry guided workflow can be used to quickly generate a volume mesh from an imported watertight CAD geometry. You can also create your own workflow using the available tasks (such as, adding local sizing controls, creating a surface mesh, capping inlets and outlets, creating regions, and so on). See Getting Started with the Fluent Guided Workflows (p. 277) for more information.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

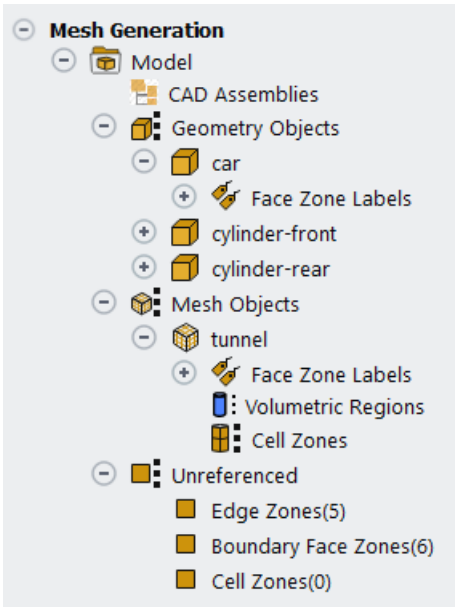
Figure 1.3: The Watertight Geometry Workflow



1.4.1.3. The Outline View Tab

In **Mesh Generation** mode, you use the Outline View tree for object-focused management of the meshing workflow and display. Branches can be expanded and collapsed as required. At each level, right-click and select from context-sensitive menus to manage the mesh generation process.

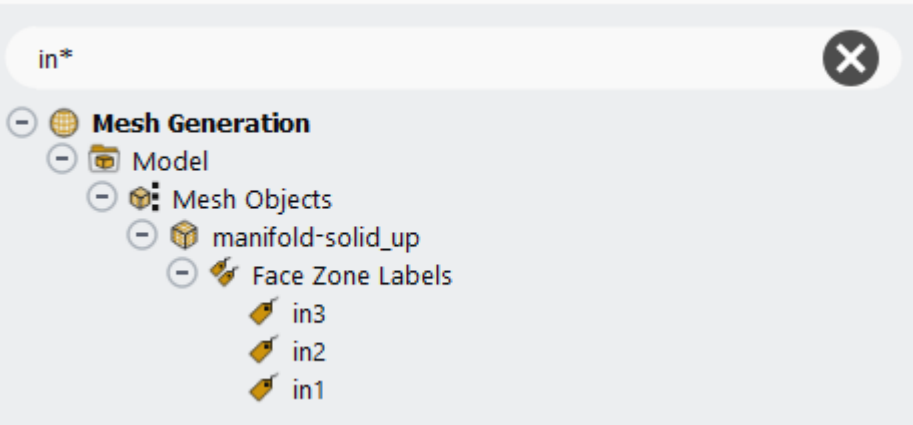
Figure 1.4: The Outline View Tree



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

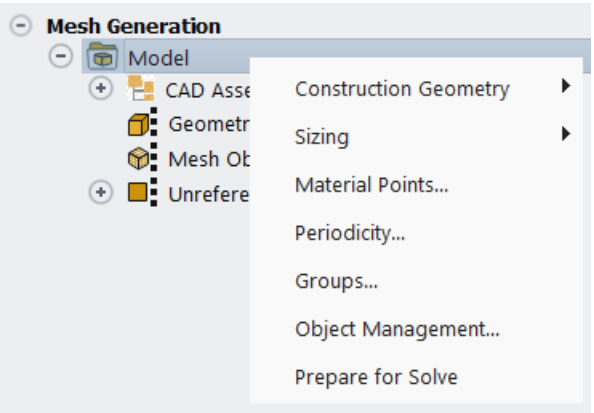
Filtering and the support of wildcards, such as *, ?, [], Boolean NOT (^), Boolean AND (&), as well as Boolean OR (|), is available at the top of the Outline View.

Figure 1.5: Filtering Field



The filter text entry box allows you to search and organize a list using a text string. The text string can include wildcards and regular expressions that allow you to perform pattern matching. For example, searching *let* finds surfaces such as inlets and outlets, including those with longer names, such as "upper-inlet-5". If you have walls separated by a number, you can search example-wall-?-23, which would show example-wall-1-23, example-wall-2-23, and so on. This feature is also available in some list selection controls in various dialog boxes in Fluent Meshing. For more information, see Filtering Lists and Using Wildcards (p. 287).

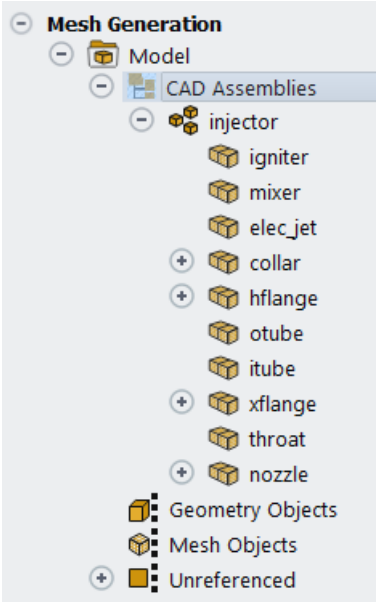
Figure 1.6: Model Level Menu



At the top of the tree, right-click **Model** to access controls not specific to any entity. For example, you can access dialog boxes to create new construction geometry or objects; set mesh size parameters; and manage material points, periodicity, and user-defined groups. You can also prepare the mesh for solving in solution mode.

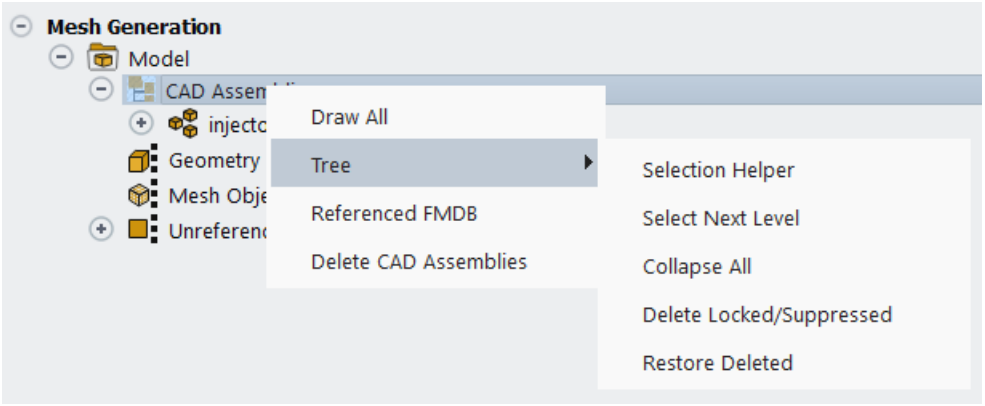
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.7: CAD Assemblies Tree



The **CAD Assemblies** tree is created when the **Create CAD Assemblies** option is selected for CAD import. It represents the CAD tree as it is presented in the CAD package in which it was created. The CAD entities in the tree are categorized as components and bodies. Components represent an assembly, sub-assembly, or part in the original CAD package, while bodies are the basic entities which include CAD zones. You can also set up labels for the CAD zones, if required.

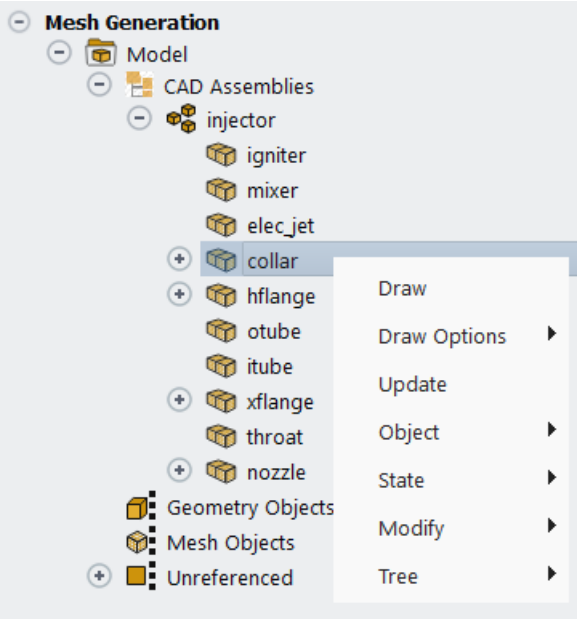
Figure 1.8: CAD Assemblies Menu



Right-click **CAD Assemblies** to draw or delete all imported assemblies, and obtain the referenced FMDB file locations. The **Tree** sub-menu contains options that control the appearance of the CAD Assemblies tree. These options can be used to select or deselect the CAD objects and zones in the tree, expand or collapse the tree branches, and also delete suppressed or locked CAD objects.

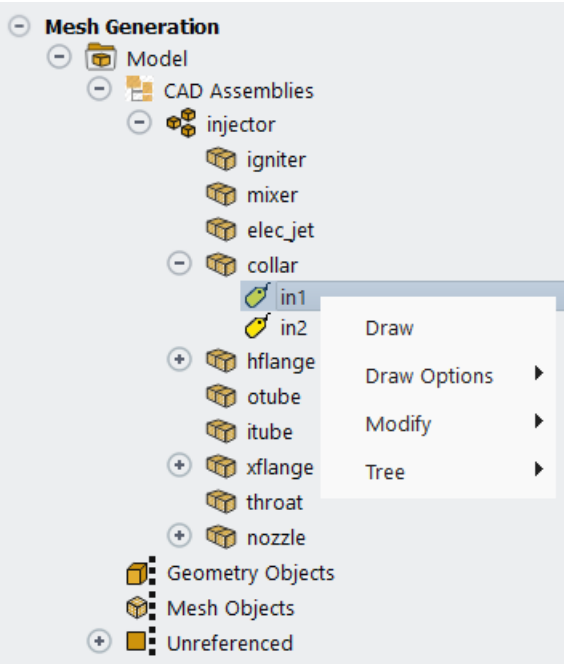
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.9: CAD Component/Body Level Menu



The tree menus at CAD component and body levels contain display options and options for updating the CAD entities, creating and manipulating geometry/mesh objects, setting the state and modifying the CAD entities, and tree selection options.

Figure 1.10: CAD Label Level Menu



The CAD label menu contains options for deleting and renaming the labels.

For a full description of the CAD Assemblies tree menus, see CAD Assemblies (p. 577).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

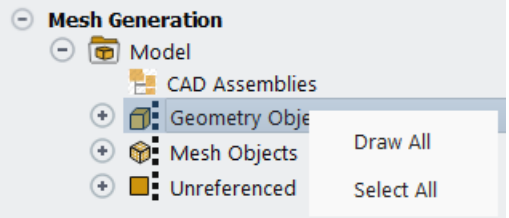
Geometry and Mesh Objects

When reading a mesh file, the tree is populated with **Geometry Objects** and **Mesh Objects**, if already defined. Importing a mesh from other formats will result in the edge zones, boundary face zones, and cell zones available in the **Unreferenced** branch. These zones are not included in any object.

Geometry objects are created when CAD files are imported using the **CAD Faceting** option. The geometry objects may be non-conformal. Mesh objects are created when CAD files are imported using the **CFD Surface Mesh** option. The mesh objects are conformal. See Importing CAD Files (p. 264) for details on the CAD import options.

You can also create geometry/mesh objects from the CAD entities in the CAD Assemblies tree. CAD entities are locked when corresponding geometry or mesh objects are created. See Creating and Modifying Geometry/Mesh Objects (p. 582) for details.

Figure 1.11: Global Object Level Menu

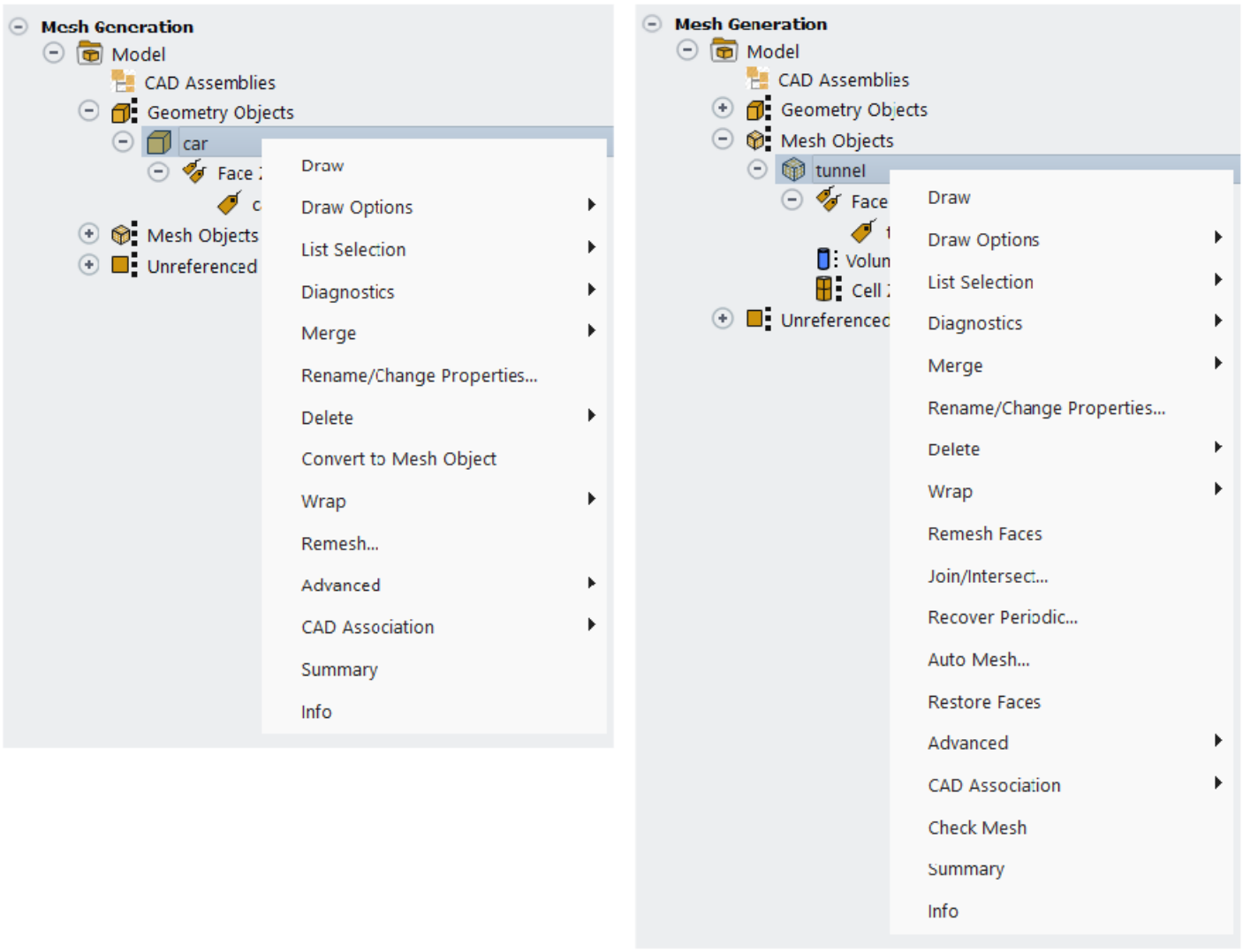


You use the **Geometry Objects** or **Mesh Objects** context-sensitive menu to perform actions on all objects in your model. At the global object level, right-click (**Geometry Objects** or **Mesh Objects**) to draw or select all the objects.

If the objects are associated with CAD entities, you can also update all or detach all objects.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

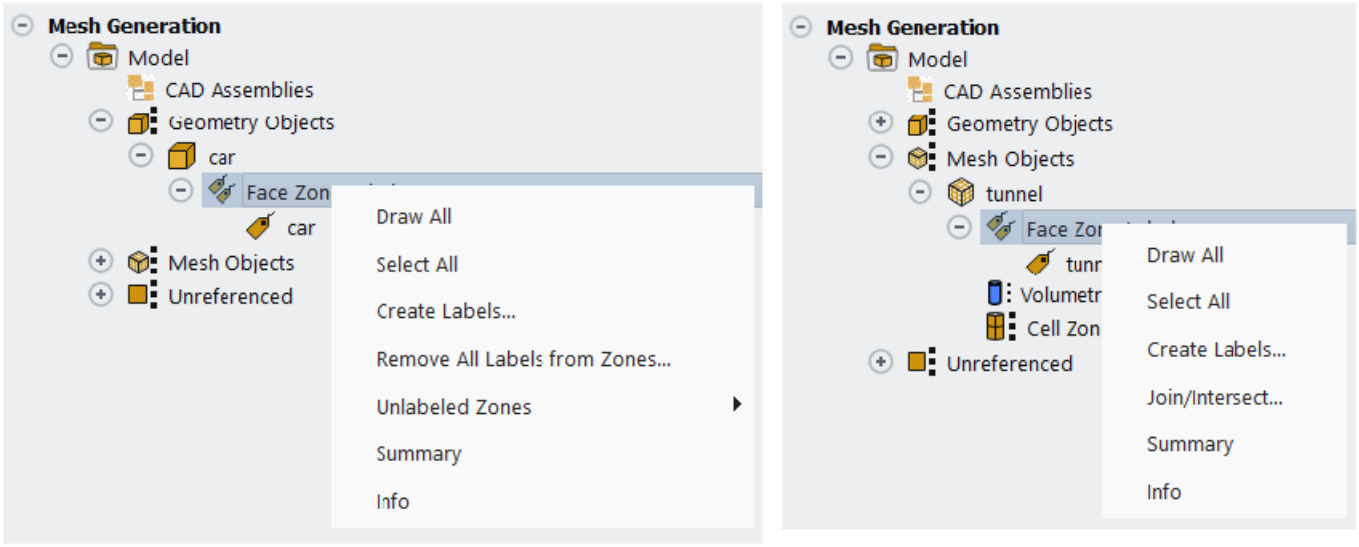
Figure 1.12: Individual Object Level Menu



Right-clicking on an individual **Object** (Geometry or Mesh) name in the Outline View opens a context-sensitive menu to access object level refinement and control tasks. You can also select objects graphically, but the menus are available only when selecting in the Outline View.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.13: Face Zone Labels Level Menu

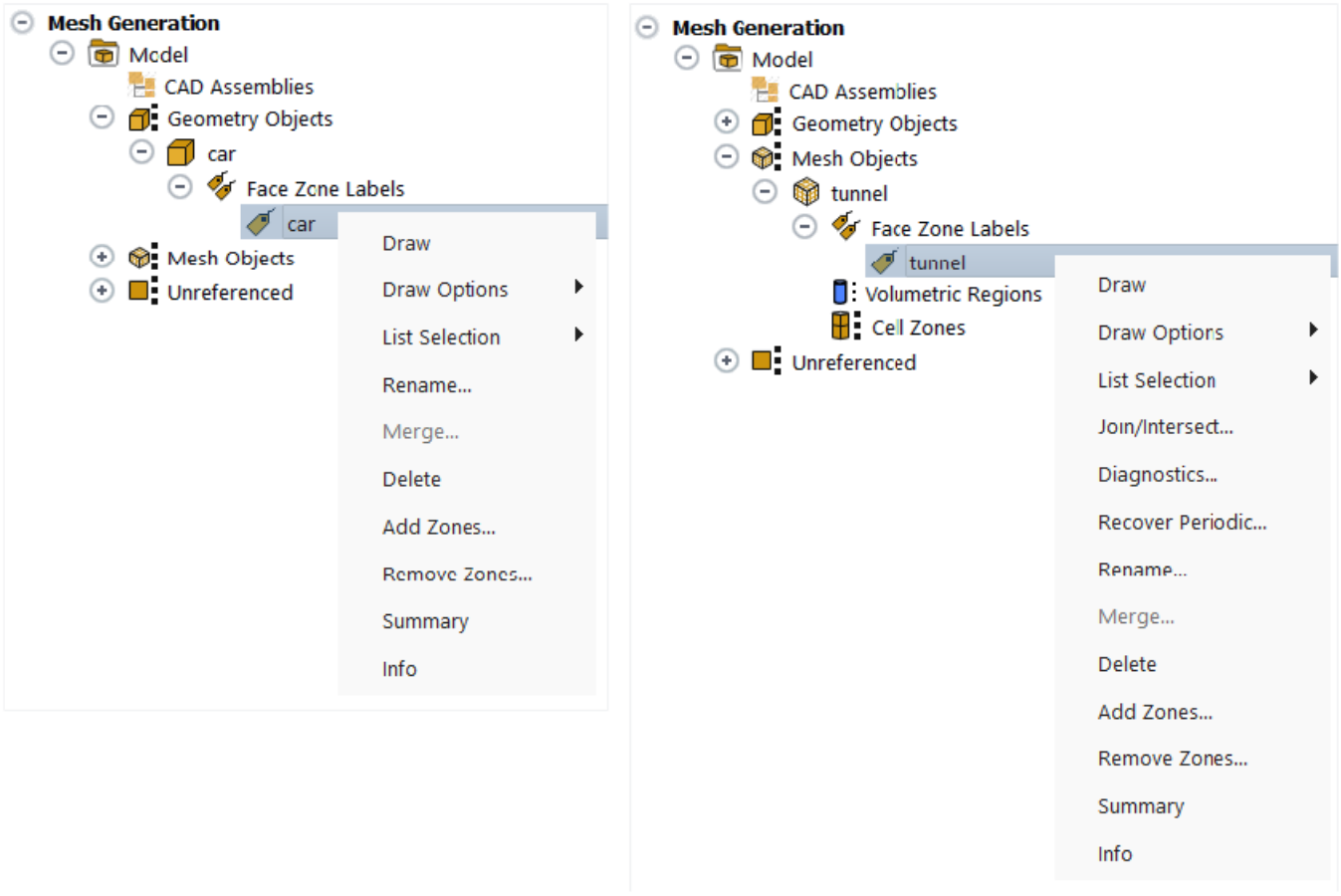


For geometry objects, **Face Zone Labels** are groups of face zones comprising the object. For mesh objects, these are original CAD zones or bodies, or face zones comprising the mesh object. If the mesh object is created by merging multiple mesh objects, the face zone labels represent the objects that were merged. They provide the link to the original geometry. Under Mesh Objects, **Face Zone Labels** form boundaries enclosing the **Volumetric Regions**—separate, closed, water-tight volumes. **Cell Zones** are regions of volume mesh.

The context-sensitive menu for **Face Zone Labels** contains options for drawing and selecting all labels, creating new labels, and obtaining an overall summary or detailed information about the face zone labels. For geometry objects, there are additional options to remove labels from zones and options for displaying and selecting unlabeled zones. For mesh objects, the **Join/Intersect...** option contains options for creating a conformal surface mesh.

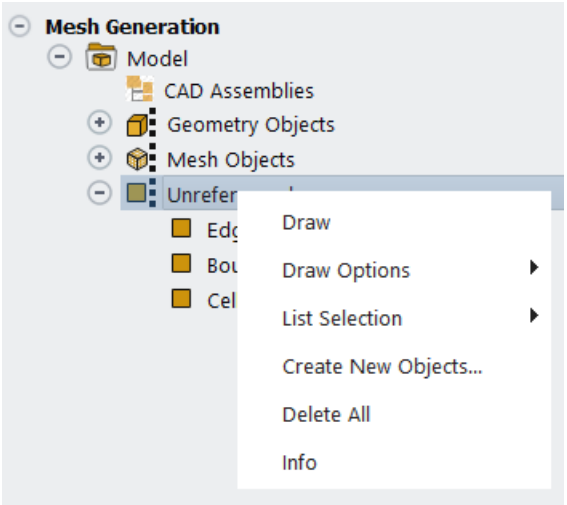
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.14: Individual Label Menu



Right-clicking on a **Zone Label** name under geometry/mesh objects allows for zone level display and selection options, and meshing tasks.

Figure 1.15: Unreferenced Zones Menu



The **Unreferenced** menu includes the common **Draw** and **List Selection** menus as well as options to manage the unreferenced zones.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For a full description of object based meshing using the tree and context menus, refer to Surface Mesh Processes (p. 617).

Note:

Any dialog box opened via right-clicking in the Outline View becomes *modal*. That is, to preserve the active selection, the action must be completed or the dialog box closed before the tree selection can be changed.

1.4.1.4. The Graphics Window

The graphics window displays the current state of your model according to your selected **Rendering**, **Color Scheme** and **Display** options. See Controlling Display Options (p. 541) for details.

The hide ribbon button ( ) is used to expand the area for the graphics window by minimizing the ribbon. Click a second time to maximize the ribbon to restore the graphics window area.

The axis triad indicates the orientation of the model and also provides options for manipulating the orientation in the graphics window.

To change the orientation of the model using the triad, you can:

- Click an axis/semi-sphere to orient the model in the positive/negative direction.
- Right-click an axis/semi-sphere to orient the model in the negative/positive direction.
- Click the cyan iso-ball to set the isometric view.
- Click the white rotational arrows to perform in-plane clockwise or counterclockwise 90 degree rotations.
- Left-click and hold—in the vicinity of the triad—and use the mouse to perform free rotations in any direction. Release the left mouse button to stop rotating.

You can choose how the mouse buttons interact with the graphics display using the **Mouse Buttons** dialog box. See Controlling the Mouse Buttons (p. 548).

1.4.1.5. Quick Search

The search bar (upper right of the Fluent window) allows you to quickly locate the commands or controls that you are looking for. Clicking the search results is equivalent to clicking the same control in the ribbon. Hovering over a search result highlights the location of the control in the ribbon. Clicking a text command search result automatically enters the text of the command, but it does not execute the command; you still have to press Enter in the console to execute the text command.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.4.1.6. The Console

The console is usually located below the Graphics Window, as shown in Figure 1.1: The User Interface Components (p. 222). It is used to provide a text based interface to Fluent meshing.

- The console will display messages relating to meshing or solution procedures. All console information is saved to memory, so you can review it at anytime by using the scroll bar on the right side of the console. The console visually indicates whether the text is an error message (red), user input (blue), or program generated output (black).
- The console behaves like an "xterm" or other Linux command shell tool, or to the MS-DOS Command Prompt window. It enables you to interact with the TUI menu. For more information on the TUI, see Text User Interface in the *Fluent Text Command List*.
- You may interrupt the program by issuing a "break" command (press **Ctrl**+**C**) while data is being processed. You cannot cancel an operation after it is complete and the program has started drawing in the graphics window.
- You may perform text copy and paste operations between the console and other X Window (or Windows) applications that support copy and paste.

Note:

On a Linux system, follow the steps below to copy text to the clipboard:

1. Drag the pointer across the text to be copied.
2. Move the pointer to the target window.
3. Press the middle mouse button to “paste” the text.

The **Auto-scroll** check box enables automatic scrolling of the console when new content is printed. When the **Auto-scroll** is disabled, the cursor stays at the last selected position. Messages will continue to be appended in the console, but the cursor will not automatically scroll to the bottom until the **Auto-scroll** is enabled.

Warning and Error Messages

In addition to being printed in the console, warning and error messages are also available for easier review in separate **Warning** and **Error** dialog boxes. These messaging dialog boxes can be opened by clicking the warning ( ) and error ( ) icons in the upper right of the console. When new messages are printed in the console, the icons are highlighted ( ), indicating that there are new messages for review. *Note that clearing the messages from the **Warning** and **Error** dialog boxes does not remove them from the Console or transcript.*

1.4.1.7. The Toolbars

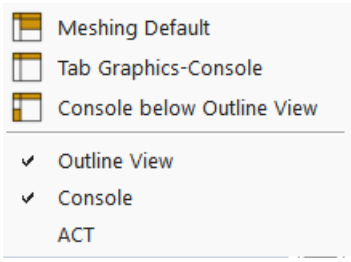
The user interface includes several toolbars to provide shortcuts to performing common tasks. You can enable or disable the visibility of the toolbars that appear in the graphical user interface. You

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

can dock the toolbars around the graphics window or position them "floating" at any convenient location in the user interface.

The **Standard Toolbar** contains options to control the layout and to find additional resources.

- Click the Ansys logo to open the Ansys home page in your default browser.
- Use the options in the **Arrange the workspace** ( ) menu to control the application window layout. You can also enable or disable the visibility of each interface component in this menu.



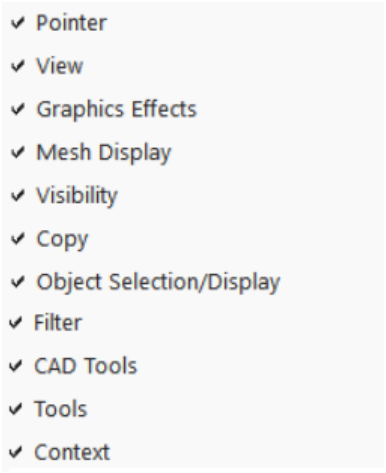
  The most recent arrangement of the user interface components will be saved to a `.cxlayout` file in your home folder. The layout will be restored the next time Ansys Fluent is opened.

  Note:

  Do not use multiple graphics windows in meshing mode.

- **Help** ( ) contains options for accessing the User Documentation and obtaining license usage and product version information. See Using the Help System (p. 249) for details.

On each side of the graphics window are context-sensitive toolbars for quick access to commonly used operations. Right-click any toolbar to access the **Toolbar Options** to set the visibility of the respective toolbars.



1.4.1.7.1. Pointer Tools

You can quickly change the role of the left mouse button using the options in the **Pointer Tools** toolbar. See Controlling the Mouse Buttons (p. 548) for a description of the options.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Box Select** lets you select surfaces by box-selecting, that is clicking and dragging the mouse from left-to-right or right-to-left over your model. For more information, see Controlling the Mouse Button Functions (p. 4020).

- **Rotate View** assigns the **mouse-rotate** function to the left mouse button.

- **Pan** assigns the **mouse-dolly** function to the left mouse button.

- **Zoom In/Out** assigns the **mouse-roll-zoom** function to the left mouse button.

- **Zoom to Area** assigns the **mouse-zoom** function to the left mouse button.

1.4.1.7.2. View Tools

You can quickly change the model display using the **View Tools** toolbar.

- **Perspective or Orthographic View** shows the 3D object on your 2D screen approximately how your eye would see it in real life. This button is only available when running Fluent in 3D.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

shows 3D objects to scale, ignoring perspective view. *When enabled, it makes it so the ruler is accurate*. This button is only available when running Fluent in 3D.

Note:

When **Orthographic** is selected for **Graphics view** in **Preferences** (**Appearance** branch), an additional **Ruler** option appears. Enabling **Ruler** makes it so that the ruler is displayed whenever the view is changed to orthographic. *You can still disable the ruler by toggling the ruler button* *).*

- **Fit to Window** adjusts the overall size of your model to take maximum advantage of the graphics window's width and height.
- **Last View** restores the display to the previous view.
- **Next View** allows you to revert to the displayed objects previous location and orientation in the graphics window.
- **Set view** contains a drop-down of views, allowing you to display the model in isometric or one of six orthographic views.

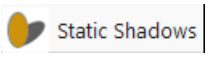
  You can also click on the display axes triad in the graphics window to change to one of the standard views. Use the right mouse button to reverse the orthographic view.

1.4.1.7.3. Graphics Effects Tools

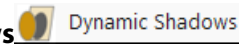
The graphics effects tools allow you to control which graphics effects are shown in the graphics window.

Figure 1.16: The Graphics Effects Tools

The following is a brief description of each of the graphics effects tools:

- **Reflections** redisplays all hidden surfaces.
- **Shadows** controls which shadowing effects are shown:
  - **Static Shadows**  is the shadow of the model on the ground plane.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - **Dynamic Shadows** 

 is the shadow of the model on itself. These shadows are updated as the model is rotated. *This graphics effect may result in reduced graphics performance.*

- **Grid Plane** shows/hides the ground plane grid that is displayed beneath the model.

1.4.1.7.4. Mesh Display Tools

The mesh display configuration options contains predefined settings for how a mesh and objects are displayed.

Figure 1.17: The Mesh Display Tools

The following is a brief description of the mesh display tools:

- **Mesh Display** opens the Mesh Display Dialog Box (p. 4688).

- **Front Faces Transparent** makes the front faces of the displayed object transparent, allowing you to see inside.

1.4.1.7.5. Visibility Tools

There are visibility options available when working in the graphics window:

- **Axes Visibility** turns the axes display on and off.

- **Ruler Visibility** turns the ruler on and off. When enabled, it switches the view to *orthographic* (if it was previously set to *perspective*).

- **Titles Visibility** turns the titles display on and off.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Boundary Markers** enables/disables boundary markers at inlets and outlets when the mesh is displayed.

> **Note:**
>
> This feature is not available in Meshing mode.

1.4.1.7.6. Copy Tools

The copy tools allow you to copy an image of the graphics window to the clipboard and capture an image of the graphics window.

Figure 1.18: The Copy Tools

The following is a brief description of each of the copy tools:

- **Copy to Clipboard** copies the current picture displayed in the selected graphics window tab onto your clipboard.
- **Save Picture** captures an image of the active graphics window. For more information, see Saving Picture Files (p. 271).

1.4.1.7.7. Object Selection/Display Tools

The object selection/display tools allow you to modify the selection and display of objects in the graphics window.

Figure 1.19: The Object Selection/Display Tools

The following is a brief description of each of the object selection/display tools:

- **Deselect the Currently Selected Surfaces** deselects the currently-selected surfaces in the graphics window.

Other object selection/display tools are available within Fluent Meshing, such as the following options within the **Isolate** toolbar:

- **Show All** redisplays all hidden surfaces.
- **Hide Selected Surfaces** hides the selected surfaces from the display.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.4.1.7.8. Filter Toolbar

Use the **Filter** toolbar to set the entity type that the mouse probe filter will recognize. See Controlling the Mouse Probe Function (p. 551).

The selection filters available in the toolbar are **Position**, **Node**, **Edge**, **Zone**, and **Object**. You can also set the mouse probe to **Draw Sizes** and examine the mesh size at the probe point.

The toolbar also contains options for selecting visible entities based on the filter set, deselecting the last entity, and clearing all selections.

1.4.1.7.9. CAD Tools

Use the **CAD Toolbar** to access tools for manipulating CAD entities and creating/managing labels and geometry/mesh objects associated with the CAD entities. This toolbar is available only when the CAD entities are displayed in the graphics window.

See CAD Assemblies (p. 577) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.4.1.7.10. Tools

Use the **Tools** toolbar to access options for constructing geometry primitives or loop selection tools. You can also use the **Isolate** tools to isolate selected zones/objects in the display or limit the display to zones/objects based on area/curvature of the selected zones/objects.

1.4.1.7.11. Context Toolbar

Use the **Context** toolbar to see context-sensitive options based on the entities selected. When entities are selected with the mouse probe, this toolbar changes to represent the tasks and processes applicable to the selected entities, as a subset of all tasks and processes used to generate a mesh.

An example is shown below. Descriptions of available tools are compiled in Shortcut Key Actions (p. 877).

1.4.1.8. ACT Start Page

The **ACT Start Page** is available when ACT functionality is enabled in Fluent. To enable ACT functionality, use one of these options:

- Enable **Load ACT** in the Fluent Launcher.
- Select **ACT** from the **Arrange the workspace** ( ) menu.
- Use the command: `/file/load-act-tool`.

The **ACT Start Page** provides an access point for Ansys ACT functionality in Fluent. From this page, you can access tools that can be used in the development and execution of extensions. For more information, see ACT Tools in the *ANSYS ACT Developer's Guide*.

For information on creating target product wizards for Fluent, see the *ACT Customization Guide for Fluent*.

1.4.2. Customizing the User Interface

You may want to customize the graphical user interface by changing the way that the various elements are arranged. This can be achieved by "dragging" elements and "dropping" them at a new location. For example, the graphics window can be tabbed on top of the console, or the console can be moved below the tree. In meshing mode, you can move the console and the toolbars.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To restore items that you intentionally or unintentionally closed, right-click the top toolbar to restore those items. You can also click to select one of the predefined layouts and restore missing items.

You may also want to customize the graphical user interface by changing attributes such as text color, background color, and text fonts. The program will try to provide default text fonts that are satisfactory for your platform's display size, but in some cases customization may be necessary if the default text fonts make the GUI too small or too large on your display, or if the default colors are undesirable.

The GUI in Fluent is based on the **Qt Toolkit**. If you are unfamiliar with the **Qt Toolkit**, refer to any documentation you may have that describes how to use the **Qt Toolkit** or application. The graphical attributes can be modified in a Qt stylesheet file named `cxdisplay.qss` and placed in your home directory.

1.4.3. Setting User Preferences/Options

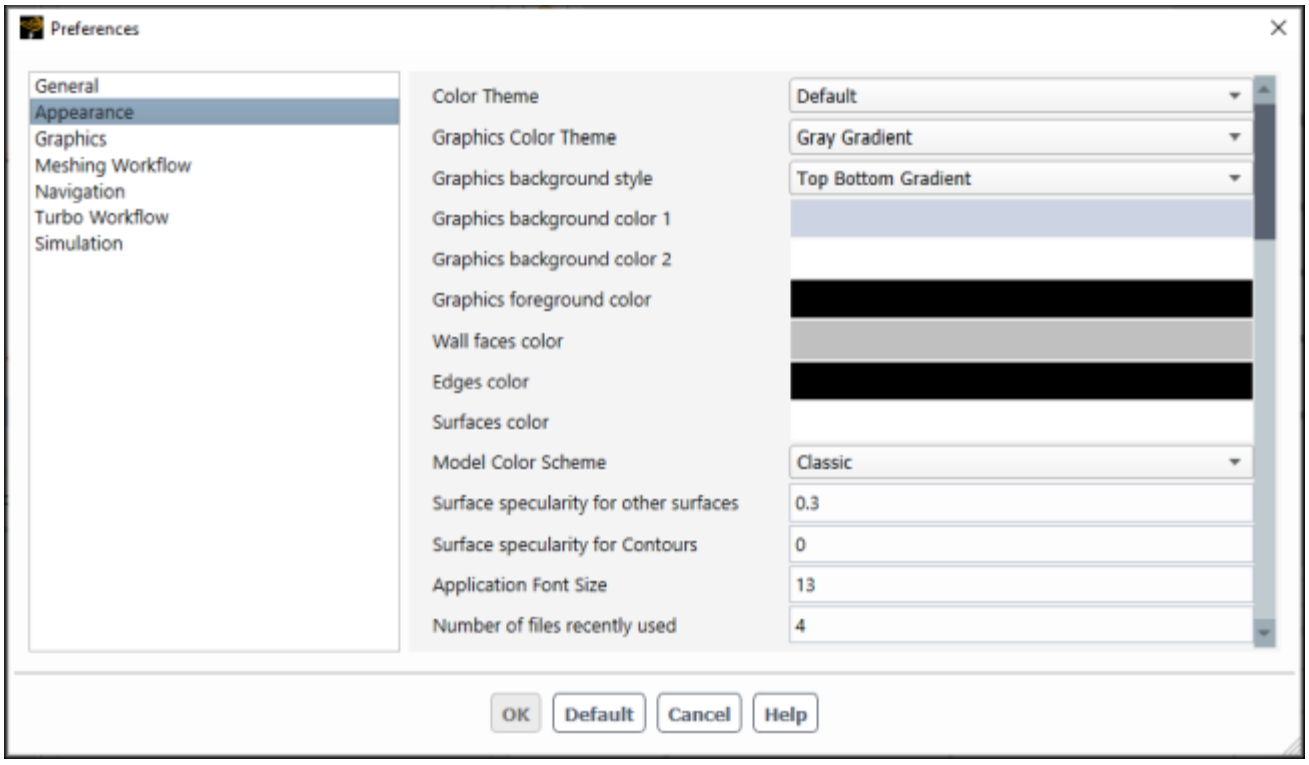
You can specify global settings that are applied whenever you are operating in Ansys Fluent. These settings are case-independent and are controlled using the **Preferences** dialog box.

To review and modify your preferences, open the **Preferences** dialog box by selecting **Preferences...** from the **File** menu.

File → **Preferences...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.20: Preferences Dialog Box



Note:

- Some settings, such as ruler and axis visibility can also be controlled locally within a Fluent session (that is, outside of the **Preferences** dialog box). Settings that deviate from the global settings specified in the **Preferences** dialog box will not be retained beyond the current session.
- If you change a setting locally that is also controlled by **Preferences**, the local change will be shown for this session only. If you open the **Preferences** dialog box and click **Apply** in an attempt to overwrite the local settings change, the global setting will only appear if it is a change from how the setting is already specified globally. For example, if you have **Ansys Logo** set as **Black** in **Preferences**, then you change the logo to **White** in the **Display Options** dialog box, the logo will change from black to white. If you then re-open the **Preferences** dialog box and click **Apply**, the logo will remain white, even though it is specified as **Black** in the preferences dialog box. The reason being that globally, the logo is already specified as black, so clicking **Apply** does not change the current global color of the logo, which is being overruled by your local change to white.
- Not all preference settings apply in all workspaces. For example, the **Flow Model** option in the **General** branch only applies to the **Solution** workspace and **Meshing Workflow** settings only apply to the meshing workflows in the **Meshing** workspace.

The controlling preferences file location is:

- Windows: `%HOMEDRIVE%\%HOMEPATH%\.fluentconf\24.2.0\preferences`

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Linux: `%HOMEPATH%/.fluentconf/24.2.0/preferences`

If no preference file is found (for instance, after installation of a new version or deletion of the preference file), Fluent will attempt to load user preferences from a previous version. If no previous version's preferences are found, Fluent will create a new file using the default settings.

At any time, you can revert your preferences to the default settings by clicking the ***Default*** *button in the* ***Preferences*** *dialog box. Or you can delete the preferences file and Fluent will follow the above rule to create a new one.*

1.4.4. Using the Help System

Fluent includes an integrated help system that provides an easy access to the documentation. Using the graphical user interface, you can access the entire User's Guide and other documentation. The User's Guide and other manuals are displayed in the Ansys Help, which enables you to use the hypertext links and the browser's search and navigation tools to find the information you need.

There are many ways to access the information contained in the online help:

- You can get reference information from the main window or any dialog box by clicking **Help**.
- You can go to the contents page for the Meshing section of the User's Guide and use the hypertext links there to find the information you are looking for. Select **Help** → **User's Guide Contents**.

1.4.4.1. Help for Text Interface Commands

To find information about text interface commands, you can either go to the Meshing section of the **Fluent Text Command List** in the Ansys Help, or use the text interface help system described in Text User Interface in the *Fluent Text Command List*.

1.4.4.2. Obtaining a Listing of Other License Users

If you are running with an existing Fluent license (FluentLM), you can obtain a listing of current Fluent users in the console by selecting **Help** → **License Usage**.

If your installation of Fluent is managed by the Ansys License Manager (ANSLIC_ADMIN), you will see a message that will indicate that licensing is managed by ANSLIC_ADMIN. For additional information on licensing information, refer to the **Installation and Licensing Documentation** in the ANSYS Help.

1. Scroll down to the **Installation and Licensing Documentation** item in the left pane of the viewer.
2. Expand this document by clicking on the icon to the left of the document title.
3. Use the hyperlinks in the main viewer window to find the desired information, or, expand the items in the left pane of the viewer and scroll to the topic of interest.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.5. Text User Interface

The text user interface (TUI) in Fluent, also referred to as The Console (p. 921), is written in a dialect of Lisp called Scheme. Users familiar with Scheme will be able to use the interpretive capabilities of the interface to create customized commands. The text-based menu system provides a hierarchical interface to the underlying procedural interface of the program.

- You can easily manipulate its operation with standard text-based tools—input can be saved in files, modified using text editors, and read back in to be executed.
- The text menu system is tightly integrated with the Scheme extension language, so it can easily be programmed to provide sophisticated control and customized functionality.

A more complete description of the text-based interface, including a full list of commands is available in Fluent Text Command List.

1.6. Reading and Writing Files

During a Fluent session, you may need to:

- Read mesh, case, CAD, journal, Scheme, domain, and size-field files.
- Write mesh, case, journal, transcript, and domain files.
- Save pictures of graphics windows.

These files and operations are described in the following sections.

1.6.1. Shortcuts for Reading and Writing Files
1.6.2. Mesh Files
1.6.3. Case Files
1.6.4. Reading and Writing Size-Field Files
1.6.5. Reading Scheme Source Files
1.6.6. Creating and Reading Journal Files
1.6.7. Creating Transcript Files
1.6.8. Reading and Writing Domain Files
1.6.9. Importing Files
1.6.10. Saving Picture Files

1.6.1. Shortcuts for Reading and Writing Files

The following features make reading and writing files convenient:

1.6.1.1. Binary Files
1.6.1.2. Reading and Writing Compressed Files
1.6.1.3. Tilde Expansion (LINUX Systems Only)
1.6.1.4. Disabling the Overwrite Confirmation Prompt

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.6.1.1. Binary Files

When you write a mesh, case, or size-field file, a binary file is saved by default. Binary files take up less memory than text files and can be read and written more quickly by Fluent.

To save a text file, disable the **Write Binary Files** option in the **Select File** dialog box when you are writing the file.

1.6.1.2. Reading and Writing Compressed Files

Fluent enables you to read and write compressed files. Use the **Select File** dialog box to read or write the files that have been compressed using `compress` or `gzip`.

1.6.1.2.1. Reading Compressed Files

1.6.1.2.2. Writing Compressed Files

1.6.1.2.1. Reading Compressed Files

If you select a compressed file with a `.z` extension, Fluent will automatically invoke `zcat` to import the file. If you select a compressed file with a `.gz` extension, Fluent will invoke `gunzip` to import the file. For example, if you select a file named `flow.msh.gz`, the following message will be reported, indicating that the result of the `gunzip` is imported into Fluent via an operating system pipe.

```
Reading "\" | gunzip -c \"Z:\flow.msh.gz\"\""...
```

When reading a compressed file using the text interface, you only need to enter the file name. Fluent first looks for a file to open using just the input name.

Note:

For Windows systems, only files that were compressed with `gzip` (that is, files with a `.gz` extension) can be read. Files that were compressed using `compress` cannot be read into Fluent on a Windows machine.

1.6.1.2.2. Writing Compressed Files

You can use the **Select File** dialog box to write a compressed file by appending a `.z` or `.gz` extension onto the file name. For example, if you are prompted for a file name and you enter a file name with a `.gz` extension, a compressed file will be written. For example, if you enter **`flow.gz`** as the name for a mesh file, Fluent reports the following message:

```
Writing "| gzip -cfv > Z:\flow.msh.gz"...
```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The status message indicates that the mesh file information is being piped into the `gzip` command, and that the output of the compression command is being redirected to the file with the specified name. In this particular example, the `.msh` extension was added automatically.

Note:

For case files, such compression is not applied for the default Common Fluids Format (CFF), but instead the `.z` or `.gz` extension is simply incorporated into the name (such as `.gz.cas.h5`). This compression is only used for the legacy format (such as when you have entered the following text command: `file/cff-files? no`).

Note:

For Windows systems, compression can be performed only with `gzip`. That is, you can write a compressed file by appending `.gz` to the name, but appending `.z` does not compress the file.

1.6.1.3. Tilde Expansion (LINUX Systems Only)

On LINUX systems, if you specify `~/` as the first two characters of a file name, the ~ is expanded as your home directory. Similarly, you can start a file name with `~username/`, and the `~username` is expanded to the home directory of "username". If you specify `~/file` as the mesh file to be written, Fluent saves the file `file.msh.h5` in your home directory. You can specify a subdirectory of your home directory as well: if you enter `~/examples/file`, Fluent will save the file `file.msh.h5` in the `examples` subdirectory.

1.6.1.4. Disabling the Overwrite Confirmation Prompt

By default, if you ask Ansys Fluent to write a file with the same name as an existing file in that folder, it will ask you to confirm that it is "OK to overwrite" the existing file. If you do not want Ansys Fluent to ask you for confirmation before it overwrites existing files, you can enter the `file/confirm-overwrite?` text command and answer `no`.

1.6.2. Mesh Files

Mesh files are created using the mesh generators (Ansys Meshing, the meshing mode in Fluent, GAMBIT, GeoMesh, and PreBFC), or by several third-party CAD packages. From the point of view of Fluent, a mesh file is a subset of a case file (described in Case Files (p. 257)). The mesh file includes a list of the node coordinates, connectivity information that tells how the nodes are connected to one another to form faces and cells, and the zone types and numbers of all the faces (for example, **wall-1**, **pressure-inlet-5**, **symmetry-2**). The mesh file does not contain any information on boundary conditions, flow parameters. For information about the format of the CAD package files, see Ap-

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

pendix A: Importing Boundary and Volume Meshes (p. 853), and for details on the mesh file format for Fluent, see Appendix B: Mesh File Format (p. 861).

Note:

You can also use the **File** → **Read** → **Case...** menu item to read a mesh file (described in Case Files (p. 257)) because a mesh file is a subset of a case file.

Important:

If the mesh information is contained in two or more separate files generated by one of the CAD packages, you can read them one-by-one by selecting **Append File(s)** in the **Select File** dialog box. You can also read them together and assemble the complete mesh in the meshing mode.

Fluent can save the mesh files with the suffix `.msh.h5`. You need not type the suffix while saving the mesh file, it will be added automatically.

When using the `file/read-mesh` text command to read a mesh file, Fluent first searches for a file with the exact name you typed. If a file with that name is not found, by default it will search for a file with `.msh.h5` appended to the name, and if a file with that name is still not found, it will search for a file with `.msh` appended to the name.

1.6.2.1. Reading Mesh Files

To read a mesh, select **File** → **Read** → **Mesh...** to open the **Select File** dialog box and select the mesh file to be read.

Note:

Reading mesh files also includes reading any associated boundary layer flags during the reading process (in order to identify prism cells when later appending any subsequent mesh files).

You can also use this option to read a Fluent mesh file created with GAMBIT, or to read the mesh contained in a case file.

In addition, you can read a mesh file by 'dragging and dropping' the file directly into an existing Fluent session.

By default, when reading mesh files, Fluent uses the Common Fluids Format (CFF) which is built on the Hierarchical Data Format (HDF5). To read CFF files, use the menu options or TUI commands and simply append `.msh.h5` to the file name.

CFF and HDF files are always binary and make use of built-in compression. Thus, they cannot be viewed in a text editor. However, third-party tools are available that enable you to open and explore the contents of files saved in HDF format.

For reading legacy mesh files (`.msh`) in the current session, you can use the `file/cff-files?` text command, answering `no` to read legacy mesh files.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For reading legacy mesh files in future sessions, you can use the **General** category of the **Preferences** dialog, choosing **Legacy** for the **Default Format for I/O** option.

File → **Preferences...**

Note:

Reading a case file as a mesh file will result in loss of boundary condition data as the mesh file does not contain any information on boundary conditions.

Case files containing polyhedral cells can also be read in the meshing mode of Fluent. You can display the polyhedral mesh, perform certain mesh manipulation operations, check the mesh quality, and so on.

Important:

You cannot read meshes from solvers that have been adapted using hanging nodes. To read one of these meshes in the meshing mode in Fluent, coarsen the mesh within the solver until you have recovered the original unadapted grid.

The `/file/read-options` command enables you to set the following options for reading mesh files:

- Enforce mesh topology: This option is disabled by default. Enabling this option will orient the face zones consistently when the mesh file is read. If necessary, the zones being read will be separated, such that each boundary face zone has at most two cell zones as neighbors, one on either side. Also, internal face zones are inserted between neighboring cell zones that are connected by interior faces.
- Check read data: This option enables additional checks for the validity of the mesh. Enabling this option will check the mesh topology during file read. In case incorrect mesh topology is encountered, warning messages will be displayed and the erroneous entities will be deleted. Note that in case of mesh topology errors, no automatic mesh repair is done, and that parts of the mesh may be non-conformal, contain voids, or be erroneous in other ways. The purpose of the `check-read-data` option is to enable access to corrupt files. This option is disabled by default with the assumption that correct data will be read, and to shorten file read times.

1.6.2.1.1. Reading Multiple Mesh Files

If the mesh is contained in two or more separate files, you can read them together in Fluent and assemble the complete mesh. For example, if you are creating a hybrid mesh by reading in a triangular boundary mesh and a volume mesh consisting of hexahedral cells, read both files at the same time using **File** → **Read** → **Mesh...**.

1.6.2.1.2. Reading 2D Mesh Files in the 3D Version of Fluent

You can read 2D meshes from Fluent into the 3D version of Fluent by using **File** → **Import** → **Fluent 2D Mesh...**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.6.2.2. Reading Boundary Mesh Files

To read a Fluent boundary mesh (contained in a mesh file created with GAMBIT or in a Fluent case file) into Fluent, select **File** → **Read** → **Boundary Mesh...** to open the **Select File** dialog box and select the boundary mesh file to be read.

This option is convenient if you want to reuse the boundary mesh from a file containing a large volume mesh.

If the boundary mesh is contained in two or more separate files, you can read them in together and assemble the complete boundary mesh in Fluent.

1.6.2.3. Reading Faceted Geometry Files from Ansys Workbench in Fluent

You can read faceted geometry files (`*.tgf`) exported from Ansys Workbench in Fluent. To read the faceted geometry file, use **File** → **Read** → **Mesh...** or **File** → **Read** → **Boundary Mesh...**.

The naming of face zones can be controlled by Named Selections defined in Ansys Workbench. For details on exporting faceted geometry from Ansys Workbench, refer to the Ansys Workbench Help.

1.6.2.4. Appending Mesh Files

You can read multiple mesh files one by one instead of reading all of them at once. This process is called as appending the mesh files. To append files, read in the first mesh file using the **Select File** dialog box. Reopen the dialog box and enable **Append File(s)** and read the remaining files one by one.

Note:

Append File(s) is not accessible while reading the first mesh file.

You can also append files using the command `/file/append-meshes-by-tmerge`, which uses the **tmerge** utility in Ansys Fluent. There is no graphical interface equivalent for this text command.

Append Rules:

- If zone names and IDs are duplicated, they will be modified and the changes will be reported in the console.
- Domain information will be retained during the file append operation. If domain names are duplicated, they will be modified and the changes will be reported in the console.
- Refinement region information will be retained during the file append operation. If region names are duplicated, they will be modified and the changes will be reported in the console.
- You can append files comprising only edge zones (without face zones).
- Edge-face zone associations will be retained during the file append operation.
- Zone-specific prism parameter information will be retained during the file append operation.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Boundary layer flag data is retained during mesh file reading and writing so as to identify prism cells during the append operation. In addition, boundary layer cell and face counts are reported during the append operation, and more generally, poly-prism counts are therefore available in the mesh statistics summary.
- Nodal curvature data is read during the append operation.

1.6.2.5. Writing Mesh Files

To write a mesh file in the format that can be read by Fluent, select **File** → **Write** → **Mesh...** to open the **Select File** dialog box and specify the name of the mesh file to be written.

Note:

Writing mesh files also includes writing any associated boundary layer flags during the writing process (in order to identify prism cells when later appending any subsequent mesh files).

By default, when writing mesh files, Fluent uses the Common Fluids Format (CFF) which is built on the Hierarchical Data Format (HDF5). To write mesh files using CFF, use the menu options or TUI commands and simply append `msh.h5` to the file name.

CFF and HDF files are always binary and make use of built-in compression. Thus, they cannot be viewed in a text editor. However, third-party tools are available that enable you to open and explore the contents of files saved in HDF format.

For writing legacy mesh files in the current session, you can use the `file/cff-files?` text command, answering `no` to write legacy mesh files.

For writing legacy mesh files in future sessions, you can use the **General** category of the **Preferences** dialog, choosing **Legacy** for the **Default Format for I/O** option

File → **Preferences...**

Depending on your preference settings, the **Select File** dialog box will have a default value for the **Files of type** field, either CFF files (`*.msh.h5`) or legacy compressed files (`*.msh.gz`), whereby you can just specify a file name (without the extension) and Fluent will append the correct file extension accordingly.

See Binary Files (p. 251) for information about the file format.

The `/file/write-options` command enables you to set the `enforce mesh topology` option for writing mesh files. This option is disabled by default. Enabling this option will orient the face zones consistently when the mesh file is written. If necessary, the zones will be separated, such that each boundary face zone has at most two cell zones as neighbors, one on either side. Also, internal face zones will be inserted between neighboring cell zones that are connected by interior faces.

Note:

You should delete dead zones in the mesh before writing the mesh or case file for Fluent.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.6.2.6. Writing Boundary Mesh Files

Fluent enables you to write a mesh file comprising specific boundary zones. This is useful for large cases where you may want to mesh different parts of the mesh separately and then merge them together. This enables you to avoid frequent switching between domains for such cases. You can write out selected boundaries to a mesh file and then create the volume mesh for the part in a separate session. You can then read the saved mesh into the previous session using the **Append File(s)** option and merge the part with the rest of the mesh.

To write a mesh file comprising selected boundaries, select **File** → **Write** → **Boundaries...** menu item to invoke the **Write Boundaries** dialog box and select the boundaries to be written.

1.6.3. Case Files

Case files contain the mesh, boundary and cell zone conditions, and solution parameters for a problem. They also contain the information about the user interface and graphics environment. Fluent allows you to read and write either text or binary files, in compressed or uncompressed formats (For details, see Binary Files (p. 251) and Reading and Writing Compressed Files (p. 251)). Fluent automatically detects the file type when reading.

Important:

Changing the ID of a thread in the meshing mode may affect the case set up. In such cases, you will be prompted to confirm that you want to proceed with the ID changing operation.

The commands used for reading case files can also be used to read native-format mesh files (as described in Mesh Files (p. 252)) because the mesh information is a subset of the case information. The commands for reading and writing case files are described in the following sections.

1.6.3.1. Reading Case Files

1.6.3.2. Writing Case Files

1.6.3.1. Reading Case Files

To read a case file, select **File** → **Read** → **Case...** to open the **Select File** dialog box and select the case file to be read.

In addition, you can read a case file by 'dragging and dropping' the file directly into an existing Fluent session.

By default, when reading case files, Fluent uses the Common Fluids Format (CFF) which is built on the Hierarchical Data Format (HDF5). To read case files using CFF, use the menu options or TUI commands and simply append `cas.h5` to the file name. Alternatively, use the **CFF Case Files** option from the **Files of type** drop-down list in the **Select File** dialog box.

For reading CFF files in the current session, you can use the `file/cff-files?` text command, answering `yes` to read case files in the CFF format.

For reading CFF files in future sessions, you can use the **General** category of the **Preferences** dialog

File → **Preferences...**

CFF and HDF files are always binary and make use of built-in compression. Thus, they cannot be viewed in a text editor. However, third-party tools are available that enable you to open and explore the contents of files saved in HDF format.

Note:

Cell hierarchy in case files adapted in the solution mode will be lost when they are read in the meshing mode.

Case files containing polyhedral cells can also be read in the meshing mode of Fluent. You can display the polyhedral mesh, perform certain mesh manipulation operations, check the mesh quality, and so on.

Note:

When you read a non-conformal interface case file into meshing mode and later switched to the solution mode, note the following limitations:

- Zone ids may match, however, the corresponding zone names may be inconsistent.
- Boundary conditions on intersected threads are not preserved.
- Unassociated (dangling) non-conformal interface (NCI) surfaces remain present.

1.6.3.1.1. Reading Files Using the Legacy Format

When reading case files, you can still use the legacy case file format. To read legacy case files, you can use the same menu options or TUI commands and simply append `cas` to the file name. Alternatively, select **All Case Files** from the **Files of type** drop-down list in the **Select File** dialog box.

For reading legacy files in the current session, you can use the `file/cff-files?` text command, answering `no` to read case files in the legacy format.

For reading legacy files in future sessions, you can use the **General** category of the **Preferences** dialog

File → **Preferences...**

For the **Default Format for I/O** option, select **Legacy** to use the legacy file format.

1.6.3.2. Writing Case Files

To write a case file in the format that can be read by Fluent, select **File** → **Write** → **Case**.

By default, when writing case files, Fluent uses the Common Fluids Format (CFF) which is built on the Hierarchical Data Format (HDF5). To write case files using CFF, use the menu options or TUI commands and simply append `cas.h5` to the file name. Alternatively, use the default **CFF Case Files** option from the **Files of type** drop-down list in the **Select File** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For writing CFF files in the current session, you can use the `file/cff-files?` text command, answering `yes` to write case files in the CFF format.

For writing CFF files in future sessions, you can use the **General** category of the **Preferences** dialog

File → **Preferences...**

CFF and HDF files are always binary and make use of built-in compression. Thus, they cannot be viewed in a text editor. However, third-party tools are available that enable you to open and explore the contents of files saved in HDF format.

Note:

You should delete dead zones in the mesh before writing the mesh or case file for Fluent.

See Binary Files (p. 251) for information about the save file format.

If you are writing a hexcore or CutCell mesh, enable the **Write As Polyhedra** check button in the **Select File** dialog box. This enables hex cells that are either part of a hanging-node sub-division or are at the boundary of the hex-tet interface, to be converted to polyhedral cells. Enabling this option permits the export of these cells instead of non-conformal meshes.

Note:

Further manipulation of the mesh is restricted after conversion to polyhedra. Only limited operations like displaying the polyhedral mesh, certain mesh manipulation operations, checking the mesh quality are available for polyhedral meshes.

Important:

- Case files that have been read and re-written in the meshing mode are incompatible with previously saved data files. Do not read previously saved data files with the case file when such case files are transferred or read in the solution mode.
- If the zone topology changes due to operations performed in the meshing mode, you should verify the case setup after transferring or reading the case in the solution mode.

1.6.3.2.1. Writing Files Using the Legacy Format

When writing case files, you can still use the legacy case file format. To write legacy case files, you can use the same menu options or TUI commands and simply append `cas` to the file name. Alternatively, select **Case Files** from the **Files of type** drop-down list in the **Select File** dialog box.

For writing legacy files in the current session, you can use the `file/cff-files?` text command, answering `no` to write case files in the legacy format.

For writing legacy files in future sessions, you can use the **General** category of the **Preferences** dialog

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

File → **Preferences...**

For the **Default Format for I/O** option, select **Legacy** to use the legacy file format.

1.6.4. Reading and Writing Size-Field Files

Size-field files contain the size function definitions based on the parameters specified.

Select **File** → **Read** → **Size Field...** to read a size-field file. This will invoke the **Select File** dialog box, where you can specify the name of the size-field file to be read.

Note:

If you read a size-field file after scaling the model, ensure that the size-field file is appropriate for the scaled model (size-field vertices should match the scaled model).

Select **File** → **Write** → **Size Field...** to write a size-field file. This invokes the **Select File** dialog box, where you can specify the name of the size-field file to be written.

See Binary Files (p. 251) for information about the save file format.

1.6.5. Reading Scheme Source Files

A Scheme source file can be loaded in three ways: through the menu system as a scheme file, through the menu system as a journal file, or through Scheme itself.

For large source files, use the **Select File** dialog box invoked by selecting the **File** → **Read** → **Scheme...** menu item or the Scheme `load` function.

```
> (load "file.scm")
```

Shorter files can also be loaded with **File** → **Read** → **Journal...** or the `file/read-journal` command in the text interface (or its `.` or `source` alias).

```
> . file.scm

> source file.scm
```

In this case, each character of the file is echoed to the console as it is read in the same way as if you were typing the contents of the file.

1.6.6. Creating and Reading Journal Files

A journal file contains a sequence of Fluent commands, arranged as they would be typed interactively into the program or entered through the user interface. The user interface commands are recorded as Scheme code lines in journal files. You can also create journal files manually with a text editor. If you want to include comments in your file, put a semicolon (`;`) at the beginning of each comment line.

The purpose of a journal file is to automate a series of commands instead of entering them repeatedly on the command line. It can also be used to produce a record of the input to a program session for

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

later reference, although transcript files are often more useful for this purpose (see Creating Transcript Files (p. 262)).

Command input is taken from the specified journal file until its end is reached, at which time control is returned to the standard input (usually the keyboard). Each line from the journal file is echoed to the standard output (usually the screen) as it is read and processed.

Important:

A journal file by design is a simple record/playback facility. It contains no information about the state in which it was recorded or the state in which it is being played back.

- Be careful not to change the folder while recording a journal file. Also, try to recreate the state in which the journal was written before you read it into the program.

  For example, if the journal file includes an instruction for Fluent to save a new file with a specified name, check that no file with that name exists in your directory before you read in your journal file. If a file with that name exists and you read in your journal file, it will prompt for a confirmation to overwrite the old file when the program reaches the write instruction. Because the journal file contains no response to the confirmation request, Fluent will not be able to continue following the instructions of the journal file.

- Other conditions that may affect the program's ability to perform the instructions contained in a journal file can be created by modifications or manipulations that you make within the program.

  For example, if your journal file displays certain surfaces, you must read in the appropriate mesh file before reading the journal file.

  Important:

  At a given time, only one journal file can be open for recording. But you can read a journal file at any time. You can also write a journal and a transcript file simultaneously.

- You can read multiple journal files in one go.

  Select the files in the order you want them read in Fluent in the **Select File** dialog box.

- You can also create a journal file that makes calls to other journal files.

  The following is an example of such a nested journal file:

```
/file/read-journal "E:/Example_journals_example1.jou" ""
/file/read-journal "E:/Example_journals_example2.jou" ""
/file/read-journal "E:/Example_journals_example3.jou" ""
```

- Whenever you start recording a journal file, the text command `/file/set-tui-version "XX.X"` is added at the top of the file (where `XX.X` corresponds to the version of Ansys Fluent that is recording the journal file). This text command can help journals created in an older version to work properly when used in a newer version, as it will hide the new text user interface (TUI) prompts and restore the deleted TUI prompts in that newer version.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you are writing a journal file in a text editor, it is recommended that you add `/file/set-tui-version "XX.X"` at the top of the file.

Note:

– The specified version must be within two full releases of the version that is running the journal.

– To specify version 2024 R2, enter `"24.2"` for `"XX.X"`.

- Whether you choose to type the text command in full or use partial strings, complete commands are recorded in the journal files.

Important:

– Only successfully completed commands are recorded. For example, if you stopped an execution of a command using **Ctrl**+**C**, it will not be recorded in the journal file.

– If a user interface event happens while a text command is in progress, the user interface event is recorded first.

– All default values are recorded.

To start the journaling process, select **File** → **Write** → **Start Journal...**. Enter a name for the file in the **Select File** dialog box. The journal recording begins and the **Start Journal** menu item becomes **Stop Journal** menu item. You can end journal recording by selecting **Stop Journal**, or by exiting the program.

You can read a journal file into the program using the **Select File** dialog box invoked by selecting **File** → **Read** → **Journal...**.

Journal files are always loaded in the main (top-level) text menu, regardless of where you are in the text menu hierarchy when you invoke the read command.

1.6.7. Creating Transcript Files

A transcript file contains a complete record of all standard input to and output from Fluent (usually all keyboard and user interface input and all screen output) including the start, end, and total time of the session.

The user interface commands are recorded as Scheme code lines in transcript files. Fluent creates a transcript file by recording everything typed as input or entered through the user interface, and everything printed as output in the text window.

The purpose of a transcript file is to produce a record of the program session for later reference. The transcript file cannot be read back into the program because they contain messages and other output transcript files.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can set the **Automatic Transcript** option in the **General** branch of Preferences to automatically record a transcript file starting at the beginning of each new Ansys Fluent session. Automatically created transcripts are named as follows: `fluent-YYYYMMDD-HHMMSS-<processID>.trn`, where `YMD` stand for "year" "month" "day" and `HMS` stand for "hour" "minute" "second".

File → Preferences... → General → Automatic Transcript

The transcript is saved to the working directory if it is writable, otherwise it is saved in the temporary directory. *The absolute path of the temporary directory varies for Windows and Linux, but in all cases, the path where the transcript file is saved is printed in the Console.*

Important:

Only one transcript file can be open for recording at a time (except if you have **Automatic Transcript** enabled, then you can still create an additional transcript file), but you can write a transcript and a journal file simultaneously. You can also read a journal file while a transcript recording is in progress.

To start the transcription process, select **File → Write → Start Transcript...**. Enter a name for the file in the **Select File** dialog box. The transcript recording begins and the **Start Transcript...** menu item becomes the **Stop Transcript** menu item. You can end transcript recording by selecting **Stop Transcript**, or by exiting the program.

1.6.8. Reading and Writing Domain Files

A complete mesh may have multiple domains, each having its lists of nodes, faces, and cell zone IDs. A domain file is all of the domain information written as a separate file. (A mesh file includes the domain information as one section in the file.)

By convention, domain file names are composed of a root with the suffix `.dom`. If you conform to this convention, you do not have to type the suffix when prompted for a filename; it will be added automatically. When Fluent reads a domain file, it first searches for a file with the exact name you typed. If a file with that name is not found, it will search for a file with `.dom` appended to the name. When Fluent writes a domain file, `.dom` will be added to the name you type unless the name already ends with `.dom`.

To read the domain files into Fluent, select **File → Read → Domains...** to invoke the **Select File** dialog box and specify the name of the domain file to be read.

If a domain that is being read already exists in the mesh, a warning message is displayed. Fluent verifies if the zones defining the domains exist in the mesh. If not, it will display a warning message.

To write domain files in Fluent, select **File → Write → Domains...** to invoke the **Select File** dialog box and specify the name of the domain file to be written.

1.6.9. Importing Files

You can import the following file formats using the menu items in the **Import** submenu, or using the associated text commands:

- ANSYS Prep7/cdb files

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- CGNS files
- GAMBIT neutral files
- HYPERMESH ASCII files
- NASTRAN files

For information about the format of these files and details about importing them (if the import commands are not available on your computer), see Appendix A: Importing Boundary and Volume Meshes (p. 853). For information about changing the options related to mesh import see Reading and Writing Files in the *Fluent User's Guide* (p. 250).

Note:

As in solution mode, when importing ANSYS Prep7/cdb files or CGNS files, Ansys Fluent in meshing mode uses the VKI libraries, requiring a VKI license. In addition, older versions of ANSYS Prep7/cdb files (for example version 5.5) are not supported by the VKI libraries and must be regenerated with the latest release.

Importing Multiple Files

You can also import multiple files using the **File** → **Import** menu. Select the file format (for example, ANSYS `prep7/cdb`) and the mesh type (surface or volume) to open the **Select File** dialog box. Select the appropriate files from the **Files** selection list and click **OK**.

Appending Multiple External Files

You can also add files of any external format to an existing mesh. This is known as appending files. To append external files, read or import the first file. Use the **File** → **Import** menu and select the appropriate file format (for example, ANSYS `prep7/cdb`) and the mesh type (surface or volume). Enable **Append File(s)** in the **Select File** dialog box and import the necessary files.

1.6.9.1. Importing CAD Files

You can import CAD models using the CAD readers or associative geometry interfaces (via plug-ins). Refer to CAD Integration in the Ansys Help for detailed CAD-related information.

For information about past, present, and future platform support, see the Platform Support section of the Ansys Website.

Use the **File** → **Import** → **CAD...** menu item to open the **Import CAD Geometry** dialog box, where you can set the basic options for importing CAD files.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- By default, a single file will be imported. Disable **Import Single File** to import multiple files and specify the **Directory** and **Pattern** for the files to be imported.

  Note:

  Ensure that the file path contains the appropriate platform-specific separators (for example, `C:\Tutorials` on Windows systems, `Home/Tutorials` on Linux systems).

  The following special characters are supported in the file name:

  On Windows systems– **+ $ ^ ( ) [ ] { } @ # % _ - = , . ; ' ~ ` !**

  On Linux systems– **+ $ ^ ( ) [ ] { } @ # % _ - = , . : ; ' > " ~ ` !**

- Enable **Append** to append CAD model data to the existing model/mesh.

  Note:

  The **Append** option is available only when a model/mesh has already been read.

- You can select the length unit to scale the mesh on import; models created in other units will be scaled accordingly. The default selection is **mm**.

  Important:

  The imported CAD models are scaled based on the length unit selected for the meshing mode session only. When the model is transferred to solution mode, the model units are reverted to the original CAD units. Refer to Scaling the Mesh (p. 1285) in the Fluent User's Guide (p. 1) for details on scaling the mesh in solution mode.

- The following **Tessellation** options are available:

  – If you select the **CAD Faceting** option, you can choose to refine the CAD faceting. Enable **Refine Faceting** and specify the **Tolerance** for refinement and the **Max Size** in the **CAD Faceting Controls** group box. The default value for **Tolerance** is `0`, which implies no tessellation (faceting) refinement during import. The **Max Size** enables you to specify a maximum facet size for the imported model to avoid very large facets during the file import.

  The **Merge Nodes** option in the **CAD Options** dialog box enables the merging of geometry object nodes during CAD import. This option is enabled by default when the **CAD Faceting** option is selected.

  Note:

  → Use the default value of `0` for an initial (diagnostic) import. You can then determine the minimum size you intend to use for the mesh and import the file(s) again using a **Tolerance** value 1/10th the intended minimum size.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

→ Due to merging of nodes on geometry objects, the sizing computed at the facet nodes may not represent the desired sizing. In this case, disable **Merge Nodes** in the **CAD Options** dialog box and re-import the geometry objects.

– If you select the **CFD Surface Mesh** option, you need to specify the minimum and maximum facet sizes (**Min Size**, **Max Size**), and the **Growth Rate**.

Choose the type of **Size Functions** to be used for creating the surface mesh.

→ By default, a **Curvature** size function will be used for refining the surface mesh based on the underlying curve and surface curvature. Specify the **Curvature Normal Angle** to be used.

→ You can also use **Proximity** size functions for creating the surface mesh, based on the number of cells per gap specified. The proximity size functions can be scoped to edges, faces, or both faces and edges. Note that for proximity size functions, the number of cells per gap can be a real value, with a minimum of 0.1 (see Proximity (p. 587) for more information).

→ The **Auto-Create Scoped Sizing** enables you to create scoped sizing controls based on the defined parameters: the scope is defined for new objects created during import. This option is enabled by default.

→ You can also choose to save a size-field file based on the defined parameters (**Min Size**, **Max Size**, **Growth Rate**, **Curvature Normal Angle**, and **Cells Per Gap**). The size-field will be read on CAD import.

Alternatively, you can use a previously saved size-field file to create the surface mesh by enabling **Use Size Field File**. Specify the **Size Field File** to be used.

Note:

→ Mesh object nodes will always be merged when the **CFD Surface Mesh** is selected for CAD import.

→ The **Max Size** value is limited to 1/10th the bounding box diagonal.

→ If you select a size-field file during CAD import, ensure that size-field file selected is appropriate for the length units selected.

→ Surfaces with failed surface meshes will be separated into distinct face zones and will have the "**failed**" identifier in the face zone name.

→ CFD Surface Mesh options are not supported for Ansys ICEM CFD (*.tin) files.

Selecting **CAD Faceting** will result in **Geometry Objects** on import. Selecting **CFD Surface Mesh** will result in **Mesh Objects** on import.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **CAD Options** dialog box contains additional options that can be set for importing CAD files. The following options are available:

- You can choose to read all CAD files in the subdirectories of the selected directory.

  Note:

  CAD files with only line bodies/wires (edge zones) cannot be imported.

- You can save an intermediary PMDB (*.pmdb) file in the directory containing the CAD files imported. You can use this file to import the same CAD file(s) again with different options set, for a quicker import than the full import. A PMDB file will be saved per CAD file selected.
- You can choose to process **Named Selections** from the CAD files, including Named Selections from Ansys SpaceClaim, Ansys DesignModeler, publications from CATIA, and so on. Additional options to ignore import of Named Selections based on pattern or by wild card are available using the `/file/import/cad-options/named-selections` text command (see the Text Command List for details).

  Note:

  Named Selections defined in Ansys Meshing cannot be imported.

  Important:

  In general, names from the CAD file are retained on import. Valid characters for object/zone names include all alphanumeric characters as well as the following special characters:

  `_ + - : .`

  All other characters, including spaces and slashes (/), are invalid. If an invalid character is specified, it is replaced by a hyphen (-) upon import.

  – A prefix **zone** will be added to the object name and face zone name if the body, part, file, or Named Selection name begins with a digit or a special character other than "**_ :**". For example, importing a file `.test.agdb` with the option **One object per file**, will create an object named **zone.test** comprising the face zone **zone.test**.

  – A suffix **-sheet** will be added to the object name for surface bodies imported, except for surface bodies imported using the **One Object per part** option.

- By default, the curvature data from the nodes of the CAD facets is imported. You can choose to disable this, if desired.
- The **Merge Nodes** option enables the merging of geometry object nodes during CAD import.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This option is enabled by default when the **CAD Faceting** option is selected. However, due to merging of nodes on geometry objects, the sizing computed at the facet nodes may not represent the desired sizing. In this case, disable **Merge Nodes** and re-import the geometry objects.

> **Note:**
>
> Mesh object nodes will always be merged when the **CFD Surface Mesh** is selected for CAD import.

- By default, features will be extracted from the CAD model on import. You can choose to disable this, if desired.

  Specify an appropriate value for **Feature Angle**. The default value is 40.

- The **Object Creation** options enable you to set up object and zone granularity on import.

  – Enable **Create CAD Assemblies** to create the **CAD Assemblies** tree on CAD import. It represents the CAD tree as it is presented in the CAD package in which it was created. All sub-assembly levels from the CAD are maintained on import in Fluent Meshing. See CAD Assemblies (p. 577) for details.

  You can specify CAD object and zone granularity using the options in the drop-down lists in the **Object Creation** group box. You can choose to create one CAD object per **body**, **part**, CAD **file** or **selection** imported, whereas the **program-controlled** option (the default) allows the software to make the appropriate choice. This option makes a choice between per body and per part based on whether shared topology is off or on, respectively. Similarly, you can choose to create one CAD zone per **body** (default), **face**, or **object** imported.

  – You can specify object and zone granularity using the options in the drop-down lists in the **Object Creation** group box. You can choose to create one object per **body**, **part**, CAD **file** or **selection** imported, whereas the **program-controlled** option (the default) allows the software to make the appropriate choice. This option makes a choice between per body and per part based on whether shared topology is off or on, respectively. Similarly, you can choose to create one face zone per **body** (default), **face**, or **object** imported.

  > **Note:**
  >
  > For Ansys ICEM CFD files (*.tin), set the object granularity to one object per **selection**.

  > **Important:**
  >
  > When importing multi-body parts, it is recommended that the **Object Creation** option be set to **one object per part** or **one object per file**. In this case, by default, the part names from the CAD file will not be included in the zone names or added as a prefix to the face zone labels or region names.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Also, the **Tessellation** option should be set to **CFD Surface Mesh** to retain the same topology as in the CAD model.

By definition, objects do not share face and/or edge zones. If the **Object Creation** option is set to one object per body or one object per selection, common face/edge zones are duplicated on import to make the objects independent.

Additional import options are available via text commands, see the Text Command List for details.

- You can continue to import the CAD files, despite errors or problems creating the faceting on certain surfaces, or other issues.
- You can import part names and body names from the CAD files. You can also import enclosure and symmetry named selections from Ansys DesignModeler (*.agdb) files.
- You can separate feature edges based on angle, connectivity, and named selections on import. Edge zone names will have suitable suffixes depending on separation criteria, order of zones, existing zone names and other import options selected.
- You can choose to add the component (assembly or part) name to the object/zone name. By default, the component name will be added to the object/zone name.
- You can choose whether to add the part names from the CAD file to the object and zone names on import. The default setting is `auto` which adds the part names to both object and zone names when object creation granularity is set to **body**. When the object creation granularity is set to **part** or **file**, the part names are not added to the zone names, face zone labels, or the region names, by default. You can also explicitly select `yes` or `no`.
- For zones without Named Selections, you can choose to inherit the object name on import. This option is disabled by default.
- When importing Named Selections, the Named Selection will be used as the object/zone name by default, according the object creation granularity.

  You can also choose to modify zone names by using part or body names as suffixes to the Named Selections spanning multiple parts/bodies. This option is enabled by default.
- By default, Named Selections are not considered when the object creation granularity is set to one object per file. This behavior can be modified with the `use-collection-names?` text command.
- You can also import the Named Selections as face zone labels when the **CFD Surface Mesh** option is selected for **Tessellation**.
- You can also add labels to edges connected to a single face, edges connected to multiple faces, faces shared by bodies (double connected faces).
- You can choose to remove the extension of the CAD files from the object/face zone names on import. This option is disabled by default.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- You can choose to remove the path prefix from the object/face zone names on import. The default setting is `auto` which removes the path prefix from object/face zone names when the object creation granularity is set to one object per file. You can also explicitly select `yes` or `no`.
- You can also choose to modify all duplicate object/zone names by adding incremental integers as suffix. This option is disabled by default.

Note:

- Compressed CAD files (for example, *.stl.zip, *.stl.gz, *.stl.bz2) cannot be imported.
- Filenames with DOS style 8.3 path (shorter path) cannot be imported. Ensure that you give the path name in full while importing the CAD files.
- Virtual topology, suppressed parts/bodies, renamed parts/bodies defined in Ansys Mechanical/Ansys Meshing will be ignored during CAD import.
- To import Ansys DesignModeler files saved with blade geometry created using Ansys BladeModeler (plug-in for Ansys DesignModeler), ensure that the Geometry license preference is set to Ansys BladeModeler as follows:

  1. Select **Tools > License Preferences** in the Ansys Workbench menu.
  2. Click the **Geometry** tab in the **License Preferences** dialog box.
  3. If Ansys BladeModeler is not the first license listed, then select it and click **Move up** as required to move it to the top of the list.

- When using the CATIA V5 Reader on Linux systems, body hierarchy information will not be available. Only the lowest "child" labels will appear in the tree. For more information, refer to CAD Integration in the Ansys Help.
- Tables of platform-specific supported CAD packages can be found at Linux in the *CAD Integration* or Windows in the *CAD Integration*.

Meshing Problematic Imported CAD Geometries

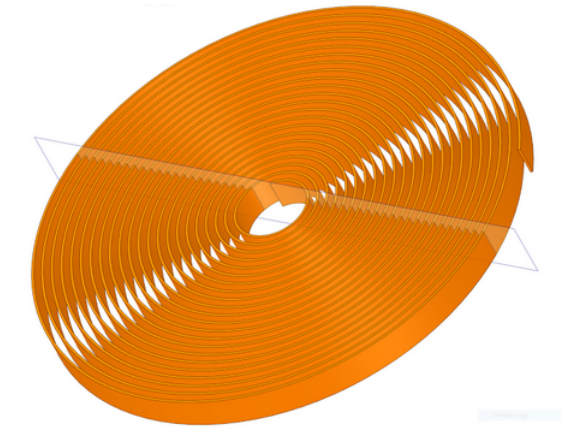
Certain problematic imported CAD geometries (such as multiple coils wound around a centerpoint) may not be easily meshed using Fluent Meshing due to two uncharacteristic properties:

- The ratio of the surface area and the square of the bounding box diagonal is much greater than 1.
- Projecting a point located between the coils can lead to ambiguous results due to spatial proximity. This is especially problematic for surface meshing because the parameter-space-free approach is the most robust approach for the majority of models in Fluent Meshing.

To avoid issues with ambiguous point projection in such geometries, use a plane to split the face of the geometry into an upper half and a lower half:

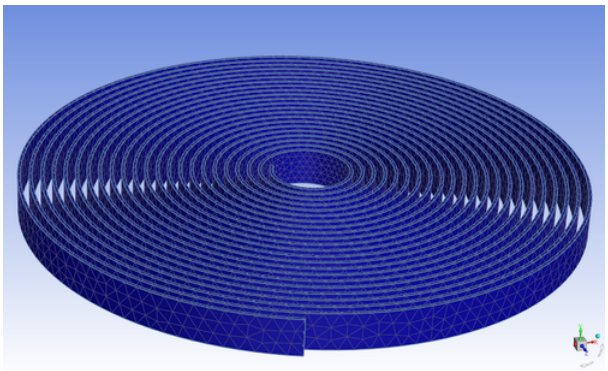
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.21: Splitting the Face of a Coiled Geometry



When the split model is then imported into the meshing mode of Fluent, the single volume is retained, however the face of the volume is split and you can obtain a proper mesh because the point projection ambiguity no longer exists,

Figure 1.22: Imported Coiled Geometry



1.6.10. Saving Picture Files

Graphic window displays can be saved in various formats including: EPS, JPEG, PPM, PostScript, TIFF, PNG, HSF, GLB, and VRML. There can be slight differences between pictures and the displayed graphics windows depending on your settings and hardware, as the pictures may be generated using the internal software renderer while the graphics windows may use specialized graphics hardware for optimum performance. To eliminate such differences and save these files at the fastest rate possible, you must follow all of the following best practices:

- Run Cortex on a suitable machine with an appropriate graphics card and the latest drivers (for details, see the Ansys website). Note that you can assign Cortex to a particular machine using the `-gui_machine=<hostname>` command line option, or by selecting **Specify Machine** from the **Graphics Display Machine** list in the **Scheduler** tab of Fluent Launcher.

- Ensure that Cortex / the host process is run on a separate machine than that used for compute node 0. For example, do not include the machine assigned using the `-gui_machine` option as the first machine in the hosts file / machine list (specified using the `-cnf=x` command line option).

- Do not set the graphics driver to `null`, `x11` (for Linux), or `msw` (for Windows).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- When saving picture files, enable the **Fast hardcopy** option in the **Preferences** dialog box (under **Graphics**).

Many systems provide a utility to "dump" the contents of a graphics window into a raster file. This is generally the fastest method of generating a picture (as the scene is already rendered in the graphics window) and guarantees that the picture will be identical to the window.

For additional information, see the following sections:

1.6.10.1. Using the Save Picture Dialog Box

1.6.10.1. Using the Save Picture Dialog Box

You can use the **Save Picture** dialog box to set the parameters and to save the picture files.

For your convenience, the **Save Picture** dialog box can also be opened using the **Save Picture** button ( ) in the standard toolbar. The procedure for saving a picture file is as follows:

1. Select the appropriate file format.
2. Set the coloring.
3. Specify the file type, if applicable (optional).
4. Define the resolution, if applicable (optional).
5. Set the appropriate options (landscape orientation, white background).
6. If you are generating a window dump, specify the command to be used for the dump.
7. Preview the result (optional).
8. Click the **Save...** button and enter the filename in the resulting **Select File** dialog box.

Tip:

Click **Apply** instead of **Save...** to save the current settings without saving a picture. The applied settings will become the defaults for subsequent pictures.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Choosing the File Format

To choose the file format, select one of the following options in the **Format** list: **EPS**[1][2], **JPEG**, **PPM**, **PostScript**[2], , **TIFF**[3]**VRML**[4], **PNG**, **HSF**, **GLB**, **VRML**, and Window Dumps (LINUX Systems) (p. 274).

Specifying the Color Mode

You can specify which type of **Coloring** you want to use for all formats except VRML and the Window Dump.

Note:

Most monochrome PostScript devices will render **Color** images in shades of gray. Select **Gray Scale** to ensure that the color ramp is rendered as a linearly-increasing gray ramp.

Choosing the File Type

When you save a PostScript or EPS file, you can choose either of the following file types:

- **Raster**: A raster file defines the color of each individual pixel in the image. Raster files have a fixed resolution.
- **Vector**: A vector file defines the graphics image as a combination of geometric primitives like lines, polygons, and text. Vector files are usually scalable to any resolution.

Defining the Resolution

For raster files, you can control the resolution of the image by specifying the size in pixels. Set the desired **Width** and **Height** under **Resolution**. If the values of **Width** and **Height** are both zero, the picture is generated at the same resolution as the active graphics window.

Note:

For PostScript and EPS files, specify the resolution in dots per inch (**DPI**) instead of setting the width and height.

---

[1] (Encapsulated PostScript) output is the same as PostScript output, with the addition of Adobe Document Structuring Conventions (v2) statements. Currently, no preview bitmap is included in EPS output. Often, programs that import EPS files use the preview bitmap to display on-screen, although the actual vector PostScript information is used for printing (on a PostScript device). You can save EPS files in raster or vector format.

[2] Raster is an optional format.

[3] The TIFF driver may not be available on all platforms.

[4] VRML is a graphics interchange format that enables export of 3D geometrical entities that you can display in the graphics window. This format can commonly be used by VR systems and in particular the 3D geometry can be viewed and manipulated in a web-browser graphics window. Non-geometric entities such as text, titles, color bars, and orientation axis are not exported. In addition, most display or visibility characteristics set in Fluent, such as lighting, shading method, transparency, face and edge visibility, and outer face culling, are not explicitly exported but are controlled by the software used to view the VRML file.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Picture Options

You can set two additional options for all picture formats except VRML and the Window Dump.

- Specify **Landscape Orientation** (enabled, default) option, or disable for Portrait orientation.
- Specify a **White Background** (enabled, default), or disable to use the same background as the graphics window.

Fluent also provides options that enable you to save PostScript files that can be printed more quickly. These options are available in the `display/set/picture/driver/post-format` text menu.

Window Dumps (LINUX Systems)

If you select the **Window Dump** format, the program will use the specified **Window Dump Command** to save the picture file. For example, if you want to use `xwd` to capture a window, set the **Window Dump Command** to

```
xwd -id %w >
```

Fluent will automatically interpret `%w` to be the ID number of the active window when the dump occurs.

- When you click **Save...**, the Select File dialog box appears. Enter the filename for the output from the window dump (for example, `myfile.xwd`).
- To make an animation, save the window dumps into numbered files, using the `%n` variable. To do this, use the Window Dump Command `xwd -id %w` and type **`myfile%n.xwd`** as the filename in the **Select File** dialog box.

Note:

Each time you create a new window dump, the value of `%n` increases by one, so you need not track numbers to the filenames manually.

If you use the **ImageMagick** animation program, saving the files in MIFF format (the native **ImageMagick** format) is more efficient. In such cases, use the **ImageMagick** tool `import`. For the **Window Dump Command** enter the default command:

```
import -window %w
```

Specify the output format to be MIFF by using the `.miff` suffix at the end of the filename.

The window-dump feature is both, system and graphics-driver-specific. The commands available for dumping windows depends on your system configuration.

Important:

The window dump will capture the window exactly as it is displayed, including the resolution, colors, transparency, for example. For this reason, all of the inputs that control

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

these characteristics are disabled in the **Save Picture** dialog box when you enable the **Window Dump** format.

Previewing the Image

Before saving a picture file, you can preview the image to be saved. Click **Preview** to apply the current settings to the active graphics window so that you can see the effects of different options interactively before saving the image.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 2: Getting Started with the Fluent Guided Workflows

The Fluent Guided workflows are designed to make it easier to generate a volume mesh starting from CAD geometries. Using the workflows, you can use the predefined steps, and add steps of your own, that walk you through common settings in Fluent in meshing mode to quickly create a valid volume mesh for use in the Fluent solution mode.

The guided workflows are available in the Workflow tab in the Fluent graphical user interface, and can be used to create a conformal, connected volume mesh from an imported CAD geometry.

Select the appropriate workflow from the Workflow tab. Available choices are:

- Use the **Watertight Geometry** workflow for water-tight CAD geometries that do not require much in the way of clean-up and modifications. See Using the Watertight Geometry Meshing Guided Workflows (p. 293) for a description of the workflow and it's tasks.
- Use the **Fault-tolerant Meshing** workflow for more complicated non-water-tight CAD geometries that may require some form of clean-up and modifications (for example, defects such as overlaps, intersections, holes, duplicates, etc.) This workflow can be used for both internal and external flows simulations. See Using the Fault-tolerant Meshing Workflow (p. 389) for a description of the workflow and it's tasks.

Once selected, the workflow is opened and you can proceed with working through the existing tasks, or even creating custom workflows by adding, grouping, or removing tasks.

2.1. Prerequisites for the Fluent Guided Workflows
2.2. Limitations of the Fluent Guided Workflows
2.3. Customizing Workflows
2.4. Understanding Task States
2.5. Operating on Tasks
2.6. Grouping Tasks
2.7. Editing Tasks
2.8. Monitoring Task Updates
2.9. Accessing Advanced Options
2.10. Filtering Lists and Using Wildcards
2.11. Saving and Loading Workflows
2.12. Setting Preferences for Workflows
2.13. Getting Help for Workflow Tasks

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.1. Prerequisites for the Fluent Guided Workflows

The workflows rely on starting with a properly defined CAD geometry, and can support geometries that contains multiple bodies, sharing common faces, or even a single, fully enclosed body. To verify that an assembly of bodies are fully connected, use the **Force Share** option in SCDM, and share any faces that are found. If **Force Share** is not successful, the workflow will try to connect the remaining unconnected faces. The workflow supports named selections from Ansys DesignModeler (DM) as well as groups from Ansys SpaceClaim Direct Modeler (SCDM) if these are applied on faces (not bodies). These entities will appear as labels in the workflow.

Important:

The use of named selections (DM) and groups (SCDM), for faces surrounding openings (for solid models) and for areas where local sizing is needed, is recommended to easily identify these locations in the workflow. Using named selections and groups will also improve persistence during design changes and improve the performance for larger models.

The workflow recognizes typical naming conventions on boundaries, such as "inlet," "outlet," "far-field," "symmetry," "wall," "interior," or "internal." More precisely, the changing of Fluent boundary types based on name patterns is controlled by an advanced option, which is enabled by default. When enabled, the following patterns and corresponding zone types are applied on all zones:

- "*inlet*" maps to a **velocity-inlet** zone
- "*outlet*" maps to a **pressure-outlet** zone
- "*symmetry*" maps to a **symmetry** zone
- "*far*field*" maps to a **pressure-far-field** zone
- "*mass*inlet*" maps to a **mass-flow-inlet** zone
- "*press*inlet*" maps to a **pressure-inlet** zone
- "*mass*outlet*" maps to a **mass-flow-outlet** zone
- "*outflow*" maps to an **outflow** zone
- "*inlet*vent*" maps to an **inlet-vent** zone
- "*outlet*vent*" maps to an **outlet-vent** zone
- "*intake*fan*" maps to an **intake-fan** zone
- "*exh*fan*" maps to an **exhaust-fan** zone
- "*fan*" maps to a **fan** zone
- "*porous*jump*" maps to a **porous-jump** zone
- "*radiator*" maps to a **radiator** zone

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- "*overset*" maps to an **overset** zone

Note:

The boundary type "multiple-types" is automatically assigned to labels containing zones with conflicting boundary types.

Note:

Only shrink-wrapped bodies from SCDM/Discovery geometries should include the string "wrap." Normal selections should not include this pattern.

For internal zones (that is, internal zones shared by multiple regions), the following patterns and corresponding zone types are applied:

- "*internal*" or '*interior*' maps to an **internal** zone
- "wall" maps to a **wall** zone
- "fluid" or prefixes using "enclosure" or "air" map to a fluid region

Using these naming conventions on boundaries will, by default, cause them to be assigned to corresponding Fluent zone types. Consequently, it is *not* recommended to use any of these naming conventions within file names or within bodies or parts in the model. Similarly the use of "fluid" or "enclosure" is, by default, used to identify fluid regions on bodies. Consequently, it is *not* recommended to use these naming conventions within boundary names. In addition, the use of "unknown" should also be avoided.

Note:

The following should be noted with respect to geometry zones and labels:

- Geometry zones will no longer use the `"geom-"` prefix. Instead, geometry zones will now use the `"origin-"` prefix.
- Geometry labels will no longer use the `"object"` prefix.

2.2. Limitations of the Fluent Guided Workflows

Note:

Zone names and their listings in various tasks will likely differ from previous releases, impacting journal files and workflow template (`.wft`) files that reference specific zone names. In such

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

cases, journal files and template files created before the 2020 R1 release will not function, and you will need to replace the old zone names with the new version of the zone names.

Note:

Field-level user assistance (available by hovering next to a field and clicking on the field's exposed help icon) is available for many fields of the workflows, however, it is not available on Linux, or when using ACT.

For more information, see:

- Watertight Geometry Workflow Limitations (p. 294).
- Fault-tolerant Meshing Workflow Limitations (p. 390).

2.3. Customizing Workflows

By default, the guided workflows consist of several existing tasks that represent common operations for mesh preparation. In addition, you can add and remove certain tasks as needed, depending on where you are in the workflow, and the specific level of complexity for your own desired workflow. Each task requires certain inputs and provides certain outputs that other tasks may depend upon, so only certain task may be available to add to the workflow, depending on previous tasks.

The tasks available for the **Watertight Geometry** workflow are described in Using the Watertight Geometry Meshing Guided Workflows (p. 293), whereas the tasks available for the **Fault-tolerant Meshing** workflow are described in Using the Fault-tolerant Meshing Workflow (p. 389).

The following tasks can be added to a workflow multiple times:

- Update Regions
- Add Boundary Type
- Update Boundaries
- Run Custom Journal

Customized workflows can be saved and reused later. For more information, see Saving and Loading Workflows (p. 288).

Standard journals (Creating and Reading Journal Files (p. 260)) are supported in the workflows as well. While using the workflows, only the values that you change are recorded in the journal, making them cleaner and easier to read. Any workflow-related commands use the `workflow` prefix. For example:

```
...
(%py-exec "workflow.InitializeWorkflow(WorkflowType=r'Watertight Geometry')")
(%py-exec "workflow.TaskObject['Import Geometry'].Arguments.set_state(
{r'FileName': r'D:/sample/elbow.scdoc',})")
(%py-exec "workflow.TaskObject['Import Geometry'].Execute()")
...
(%py-exec "workflow.TaskObject['Describe Geometry'].Execute()")
(cx-use-window-id 1)
(cx-set-camera-relative '(38.4549 14.9899 -8.62383) '(0 0 0)
'(-0.0370945 -0.0108113 -0.141883) 0 0)
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
(cx-use-window-id 1)
(cx-set-camera-relative '(24.2404 9.77983 -16.4746) '(0 0 0)
'(-0.0533986 -0.0213353 -0.0927949) 0 0)
(%py-exec "workflow.TaskObject['Create Regions'].Arguments.set_state(
{r'NumberOfFlowVolumes': 6,})")
...
```

Customized journal files can be created and added to either workflow (Running Custom Journal Commands (p. 388)), and must be created from scratch (that is, in a fresh, new Fluent Meshing session). The following example demonstrates a customized scoped prism control file for boundary layers. The example creates prism controls and sets various properties such as the name, type, first height, etc.

```
(
("fuselage-prisms" "last-ratio" 0.2 6 40 1.3 "fluid-1" "fluid-regions" ""
"selected-face-zones" "*fuselage*")
("rest-prisms" "last-ratio" 0.1 8 40 1.1 "fluid-1" "fluid-regions" "" "selected-face-zones"
"*fin* *pylon* *pod* *wing* ")
)
```

Custom journal scripts (or even another upstream task) may, occasionally, lead to zone name changes. These zone name changes make it potentially difficult to later correctly identify zones, and leads to potential problems with other zone-based tasks. Fluent alleviates this problem by using a zone's bounding box to identify a zone based on its location, rather than by its name.

2.3.1. Customizing Your Journal Files Using Python

More advanced customizations are available using a Python interface that can provide easier access to all meshing workflow commands and their values.

- To access the Python console, enter the following in the Fluent Meshing console window:

  ```
  Meshing> (%py-console)
  ```

  This provides you with access to a Python interface where you can interact with Fluent Meshing workflow commands.

- You can obtain specific information about a workflow task using the `help()` method for a specific `TaskObject` in the workflow, as in

  ```
  workflow.TaskObject['<task-name>'].help()
  ```

  where **<task-name>** refers to any named task in the workflows. For instance, the following is an example of the **Import Geometry** task:

  ```
  >>> workflow.TaskObject['Import Geometry'].help()
  NamedObject: TaskObject (Import Geometry)
    Currently active: True
    Members:
    String: TaskType
      Currently active: True
      Allowed values: [Simple, Composite, Compound, Conditional, Compound Child]
    String: ObjectPath
      Currently active: True
    String: CommandName
      Currently active: True
    ListString: TaskList
      Currently active: False
      Default value: []
  ```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
  ListString: InactiveTaskList
    Currently active: False
    Default value: []
  String: State
    Currently active: True
    Allowed values: [Out-of-date, Attention-required, Up-to-date, Forced-up-to-date]
  ListString: Errors
    Currently active: True
  ListString: Warnings
    Currently active: True
  Dict: Arguments
    Currently active: True
    Default value: {}

    A valid state template, derived from the corresponding ImportGeometry command,
    which can be assigned to this Arguments object:
      {r'AppendMesh': False,r'CadImportOptions': {r'CreateCADAssemblies': False,
r'ExtractFeatures': True,r'FeatureAngle': 40,r'ImportCurvatureDataFromCAD': True,
r'ImportNamedSelections': True,r'ImportPartNames': False,r'OneObjectPer': r'Program-controlled',
r'OneZonePer': r'Body',r'OpenAllCadInSubdirectories': False,r'SavePMDBIntermediateFile': False,
r'UsePartOrBodyAsSuffix': False,},r'Directory': None,r'FileFormat': r'CAD',r'FileName': None,
r'ImportCadPreferences': {r'CIRefaceting': False,r'CISeparation': r'none',r'CISeparationAngle': 40,
r'CITolerence': 0,r'EdgeLabel': r'no',r'MaxFacetLength': 0,r'ShowImportCadPreferences': False,},
r'ImportType': r'Single File',r'LengthUnit': r'mm',r'MeshFileName': None,r'MeshUnit': r'm',
r'NumParts': None,r'Pattern': None,}

    Valid dict elements, derived from the corresponding ImportGeometry command:
      String: FileFormat
        Display text: File Format
        Currently active: True
        Default value: CAD
        Allowed values: [CAD, Mesh]
      String: LengthUnit
        Display text: Units
        Currently active: True
        Default value: mm
        Allowed values: [m, cm, mm, in, ft, um, nm]
      String: MeshUnit
        Display text: File Units
        Currently active: False
        Default value: m
        Allowed values: [m, cm, mm, in, ft, um, nm]
      String: FileName
        Display text: File Name
        Currently active: True
      String: MeshFileName
        Display text: File Name
        Currently active: False
      Real: NumParts
        Display text: Number of Parts
        Currently active: True
      String: ImportType
        Display text: Import Type
        Currently active: True
        Default value: Single File
        Allowed values: [Single File, Directory Pattern]
      Bool: AppendMesh
        Display text: Append Mesh
        Currently active: True
        Default value: False
      String: Directory
        Display text: Directory
        Currently active: True
      String: Pattern
        Display text: Pattern
        Currently active: True
      Struct: ImportCadPreferences
        Display text: Import Geometry Preferences
        Currently active: True
        Members:
        Bool: ShowImportCadPreferences
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
          Display text: Advanced Options
          Currently active: True
          Default value: False
        String: CISeparation
          Display text: Separate Zone By
          Currently active: False
          Default value: none
          Allowed values: [none, region, region & angle]
        Real: CISeparationAngle
          Display text: Separation Angle [deg]
          Currently active: False
          Default value: 40
          Minumum value: 0
        Real: CITolerence
          Display text: Tolerance
          Currently active: False
          Default value: 0
          Minumum value: 0
        Real: MaxFacetLength
          Display text: Max Facet Length
          Currently active: False
          Default value: 0
          Minumum value: 0
        Bool: CIRefaceting
          Display text: Use custom faceting
          Currently active: False
          Default value: False
      Struct: CadImportOptions
        Currently active: True
        Members:
        String: OneObjectPer
          Display text: One Object Per
          Currently active: True
          Default value: Program-controlled
          Allowed values: [Body, Part, File, Selection, Program-controlled]
```

- To print all available commands, use the following:

```
>>> dir(meshing)
```

  Note that commands use a similar naming convention that reflects the names of the actual tasks in the workflow (for example, `CreateRegions` or `ModifyMeshRefinement`).

- To print all arguments for a command, use its `help()` method, for instance:

```
>>>meshing.ImportGeometry.help()
```

  When viewing the listing of a command's arguments, you will notice that the command's arguments are named similarly to the actual fields for the corresponding task in the workflow. In addition, for each argument, there are allowed values, default values, display text, and so forth.

  Each object also contains its own help text, containing usage examples that can be easily copied and pasted into your journal files.

  The following is an example using the `help()` method for the `ImportGeometry` object:

```
>>> meshing.ImportGeometry.help()
Command: ImportGeometry
Supported keyword arguments:
  String: FileFormat
    Display text: File Format
    Currently active: True
```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
    Default value: CAD
    Allowed values: [CAD, Mesh]
  String: LengthUnit
    Display text: Units
    Currently active: True
    Default value: mm
    Allowed values: [m, cm, mm, in, ft, um, nm]
  String: MeshUnit
    Display text: File Units
    Currently active: False
    Default value: m
    Allowed values: [m, cm, mm, in, ft, um, nm]
  String: FileName
    Display text: File Name
    Currently active: True
  String: MeshFileName
    Display text: File Name
    Currently active: False
  Struct: ImportCadPreferences
    Display text: Import Geometry Preferences
    Currently active: True
    Members:
    Bool: ShowImportCadPreferences
      Display text: Advanced Options
      Currently active: True
      Default value: False
    String: CISeparation
      Display text: Separate Zone By
      Currently active: False
      Default value: none
      Allowed values: [none, region, region & angle]
    Real: CISeparationAngle
      Display text: Separation Angle [deg]
      Currently active: False
      Default value: 40
      Minumum value: 0
    Real: CITolerence
      Display text: Tolerance
      Currently active: False
      Default value: 0
      Minumum value: 0
    Real: MaxFacetLength
      Display text: Max Facet Length
      Currently active: False
      Default value: 0
      Minumum value: 0
    Bool: CIRefaceting
      Display text: Use custom faceting
      Currently active: False
      Default value: False
  Struct: CadImportOptions
    Currently active: True
    Members:
    String: OneObjectPer
      Display text: One Object Per
      Currently active: True
      Default value: Program-controlled
      Allowed values: [Body, Part, File, Selection, Program-controlled]

Example usage:
  ImportGeometry(CadImportOptions={r'OneObjectPer': r'Program-controlled',},
FileFormat=r'CAD', FileName=None, ImportCadPreferences={r'ShowImportCadPreferences': False,},
LengthUnit=r'mm')
```

2.4. Understanding Task States

Tasks in the workflow tree can have four states:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Tasks that are complete are indicated by a green check mark icon (✔).
- Tasks that are complete but contain warnings are indicated by a green check mark icon with an asterisk (✔*). Tasks with this icon have one or more problematic property settings, however Fluent will still allow you to proceed with defining other tasks in the workflow. The console window may contain information about the details surrounding the problem. Right-click the task while in this state and select **Show Errors and Warnings** to display a dialog the will outline the problems identified with this task.
- Tasks that are incomplete are indicated by a lightning bolt icon (⚡).
- Tasks that require attention are indicated by an exclamation point icon (⚠). Tasks with this icon have one or more problematic property settings that require your attention before proceeding further in the workflow. The console window may contain information about the details surrounding the problem. Right-click the task while in this state and select **Show Errors and Warnings** to display a dialog the will outline the problems identified with this task.

2.5. Operating on Tasks

You can perform various operations on a task in the workflow tree by right-clicking the task in the tree and using the context menu to:

- Add new tasks to the workflow by choosing **Insert Next Task**. The available tasks that can be inserted are arranged by what is recommended.
- Group tasks together by selecting them and choosing **Create Group**.
- Complete, or execute, a task by choosing **Update**.
- Delete a task by choosing **Delete**.
- Rename a task by choosing **Rename**.

The ability to automatically display a task's objects in the graphics window (such as mesh elements, facets, etc.) is disabled for larger models when using the **Fault-tolerant Meshing** workflow, depending on the **Task based display** preference setting (see Setting Preferences for Workflows (p. 289)). If a task has a particular display available, you can display objects for that task in the graphics window by hovering over the task in the workflow tree and selecting the icon that appears. This is useful to quickly visualize a task's objects without having to explicitly visit the task details.

2.6. Grouping Tasks

In addition to using the provided tasks, you can create your own custom group tasks using the **Create Group** command on the context menu in the workflow. A group task is a collection of standard tasks that you have collected and grouped together.

When you update a group task, the system updates all of the tasks within the group sequentially, while standard tasks can only be updated once (without being edited), a group task can be updated at any time.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.7. Editing Tasks

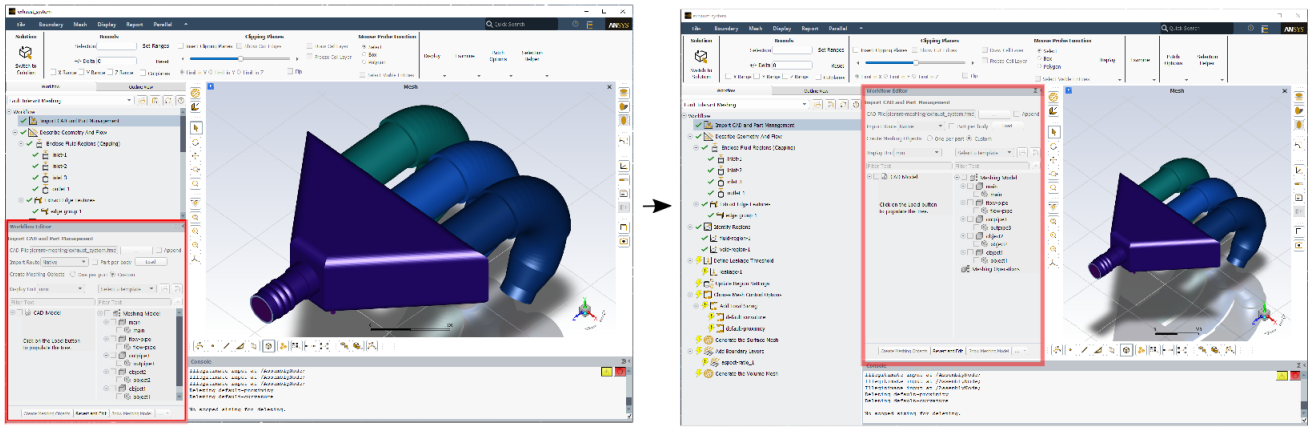
When a task is complete and up-to-date, all controls within that task are disabled. If you need to make any changes to any of your settings within the task, enable the task's controls by clicking the **Edit** button within the task (or click **Edit** in the context menu from the workflow tree). Once you have made your changes, clicking **Update** restores the task's state (causing all subsequent up-to-date tasks to then become out-of-date) and updates the current task with its new changes. Downstream tasks will remain out-of-date until you update them manually. If you decide to select **Cancel**, then all changes are discarded and the task's state will remain up-to-date.

Some tasks include zone or label selection lists where you have made a selection. To edit such tasks, enable the task's controls by clicking the **Revert and Edit** button (or click **Revert and Edit** in the context menu), thereby reverting to the task's previous state, and causing all subsequent up-to-date tasks to then become out-of-date. Once you have made your changes, clicking **Update** then updates the current task with its new values. If you decide to select **Cancel**, then all changes are discarded and the task's state is not restored. The current task and all downstream tasks will remain out-of-date until you update them manually.

Some tasks include one or more fields whose label includes unit information, such that, from an already completed workflow session, you can return to the beginning of the workflow, change the unit designation and update all subsequent tasks, and the numerical values will be automatically updated corresponding to the new unit settings.

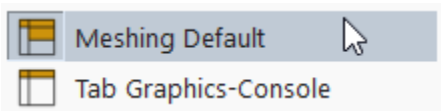
When the **Dockable Workflow Editor** preference is enabled, and a new session started (see Setting Preferences for Workflows (p. 289)), a task's properties can be separated into the **Workflow Editor**, that can then be relocated (undocked) and resized elsewhere in the interface. This is especially useful for workflows with many tasks and/or an abundance of properties in one or more tasks.

Figure 2.1: The Dockable Workflow Editor



You can relocate the Workflow Editor from its default location to any location in the Fluent interface. To return the editor to its default location, use the **Meshing Default** view from the list of view options.

Figure 2.2: Setting the Default Workflow Editor View

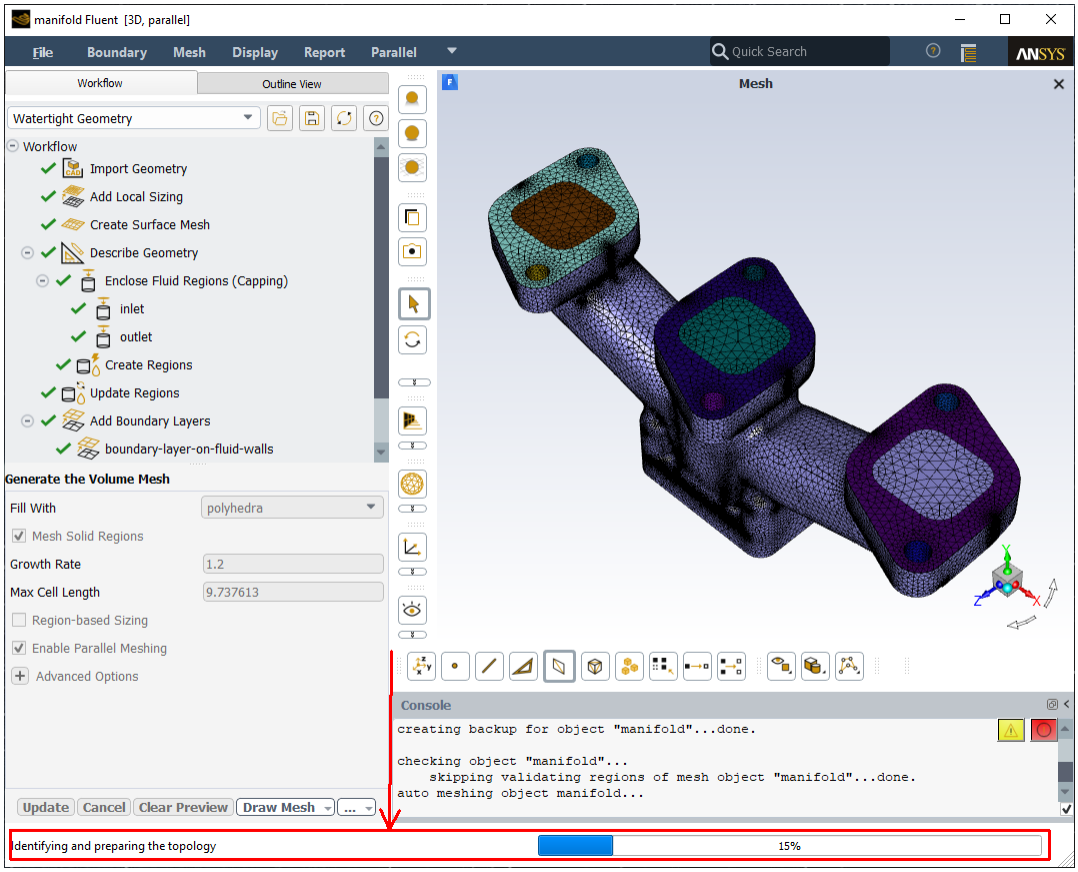


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

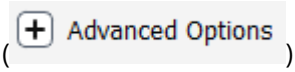
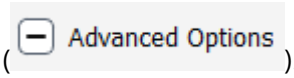
2.8. Monitoring Task Updates

As tasks progress and are updated throughout your workflow, you can follow the progress of such activity at the bottom of the Fluent window where relevant messages and a progress bar appear.

Figure 2.3: Messages and Progress Bar During Meshing Tasks



2.9. Accessing Advanced Options

For several task in the workflows, settings appropriate for advanced users or features are available under **Advanced Options**. Properties listed under advanced options are utilized by the task, though hidden under **Advanced Options** since they do not often need to be changed for most applications. When advanced options are exposed, the '+' icon () becomes a '-' icon ().

2.10. Filtering Lists and Using Wildcards

Several tasks in the workflows employ the use of lists (for example, when selecting multiple zones or labels) and can often contain a large number of items to choose from. By default, you can use the **Filter Text** drop-down to provide text and/or expressions in filtering the list (for example, using *, ?, and []). You can also choose the **Use Wildcard** option in the drop-down to provide wildcard expressions in fil-

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

tering the list. When you use either ? or * in your expression, the matching list item(s) are automatically selected in the list.

Note that with the **Use Wildcard** option, any journal files will capture any wildcards that are used, such that upon replay, the journals recorded with wildcards will dynamically consider all the labels and zones matching the wildcard string found in the journal (for example, `in*`, etc.). By contrast, journals recorded using the **Filter Text** option will use only the explicitly selected labels and zones (for example, `in1`, `in2`, etc.), thereby making them more static and less flexible.

- * - indicates zero or more occurrences of the preceding element. For example, `in*` will list only items starting with "in" such as `in1` and `in2`, whereas `*in*` will list only items that have the string "in" within the name.
- ? - substitutes for a single unknown character. For example, `gr?y` would list "grey" and "gray".
- [] - indicates a range of numbers or characters at the beginning of a string. For example, `[ot]` would match anything starting with "o" and anything starting with "t" in the name. Using `[a-z]` would match anything starting with a character between "a" and "z" inclusively, or using `[0-9]` would match the initial character with any number between "0" and "9" inclusively.
- ^ - indicates a boolean NOT function, or negation. For example, `^*in*` would list anything *not* containing "in".
- | - indicates a boolean OR function. For example, `*part*|*solid*` would list anything containing *either* "part" or "solid" such as `part2-solid-1`, `part2-solid-2`, `part-3`, `solid`, and `solid-1`.
- & - indicates a boolean AND function. For example, `*part*&*solid*` would list anything containing *both* "part" and "solid" such as `part2-solid-1` and `part2-solid-2`.

2.11. Saving and Loading Workflows

As you progress through using the workflows, you can save them at any time using the **Save Workflow** button ( ). When you save the workflow itself, you save the tasks, sub-tasks, and their settings, as your own custom workflow template for future use. You can also load any saved workflow using the **Load Workflow** button ( ). If you make a mistake in setting up your workflow, you can delete the task (using the **Delete** command in the context menu for the selected task), or even reset the entire workflow (using the **Reset Workflow** button ( )).

It should be noted that when you write the mesh file (**File > Write > Mesh...**), Fluent saves the current workflow along with any mesh information within the saved mesh file.

While saving the mesh file, Fluent also locally copies relevant task editing data from the current workflow session's temporary folder into a separate workflow-specific folder (called, for example, *my_mesh_file* `_workflow_files`). This workflow-specific folder contains task editing data, making it easier to use the workflow when reverting and editing tasks, in another Fluent session. Use the corresponding **Save**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

task editing data with mesh file preference to enable or disable this behaviour (see Setting Preferences for Workflows (p. 289) for more information).

Note:

Workflows are saved as workflow template files (using the `*.wft` file extension). The `*.wft` files include the version number of the current release (for example, for the 2024 R2, release, the files are internally designated with `"version": "24.2"`.

Additionally, all default settings in the workflow files or journals are model-based, therefore, default settings will change when the workflow file(s) are applied to a new model. If, however, you have explicitly changed any setting values, then those changes will be preserved as user settings across various models.

2.12. Setting Preferences for Workflows

You can access global preferences when using guided workflows in Fluent through the **Preferences** dialog.

File → **Preferences...**

Through the **General** and the **Meshing Workflow** categories, the **Preferences** dialog contains global settings that are useful when working with guided workflows:

- In the **General** category, use the **Dockable Workflow Editor** option to be able to separate the guided meshing workflow task steps from their properties (that is, enable the detachable **Workflow Editor**). This allows you to relocate and resize the **Workflow Editor** for easier and convenient access, especially when there is limited space available and several properties in a task. Upon changing this option, you will need to restart your Fluent session for it to take effect. See Editing Tasks (p. 286) for more information.
- In the **Meshing Workflow** category, use the **Save task editing data with mesh file** option to ensure that workflow-specific task editing data is saved when you save the mesh file. This data will be saved in a corresponding workflow-specific folder alongside the mesh file (see Saving and Loading Workflows (p. 288) for more information). When this option is enabled, the **Saving data for editing tasks** preference is automatically set to **Write mesh files**.
- For **Saving data for editing tasks**, you can determine how mesh files are managed when you edit your workflow tasks. Choices include:
  - Use the **Write mesh files** option to save a mesh file into a temporary folder while editing mesh-related tasks within the workflow.
    - → When this option is enabled (the default), Fluent creates a mesh file for use when a completed mesh-related task must be edited.
    - → When this option is disabled, Fluent will not create a mesh file, and when the completed mesh-related task must be edited, Fluent automatically updates each previous task in order to generate the required mesh file.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Disabling this preference is recommended if you do not need to edit your workflow tasks once they have been defined.

– Use the **Write into memory** option to save mesh file information to memory while editing mesh-related tasks within the workflow.

Note:

The **Fault-tolerant Meshing** workflow does not fully support the **Write into memory** option, so if selected as a preference, the workflow will use **Write mesh files** instead.

– Use **Do not save data** to not save a mesh file while editing a task.

Note:

If the **Saving data for editing tasks** option is set to **Do not save data**, and you are reverting any changes to a particular task, then updating that task subsequently updates all previous tasks.

Whether the **Saving data for editing tasks** option is set to **Write mesh files** or **Write into memory**, mesh information is saved for a particular task when the task is completed. When you choose to edit any settings to a completed task, the saved mesh information corresponding to that task is retrieved and the task's settings are restored accordingly.

See Editing Tasks (p. 286) for more information about reverting and updating tasks.

- Use the **Save workflow template (.wft) with mesh file** field to control whether or not Fluent will automatically save a workflow template file (for later re-use as needed) whenever you save the mesh. Enabled by default, whenever you write a mesh file, Fluent automatically saves the current workflow to a workflow template file (`*.wft`) using the same name as the mesh into the corresponding `workflow_files` folder.

Note:

The workflow template file is not created when automatically writing any intermediate files such as for reverting/editing tasks.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Use the **Temporary files folder** field to control where Fluent will write the temporary mesh files generated while using the workflows. You can specify a valid location, or by default, the location is set to the current working directory.

  Note:

  If you specify a "." here, Fluent will expand it to include the current working directory. In addition, you can use environment variable syntax in the path such as `$env-var` (Linux) and `%env-var%` (Windows), as well as ~ on Linux that denotes your home.

- Use the **Workflow templates folder** field to specify the location of any custom workflow template (`.wft`) files that you want to use in Fluent meshing mode. Once started, Fluent will load the workflow template files located in this folder so that they are available in the **Workflow** drop-down list.

  Any accompanying customized Scheme (`.scm`) file(s) that correspond to a customized workflow template can also be placed along side the workflow template files. As long as they share the same base file name (for example `"my_custom_template.wft"` and `"my_custom_template.scm"`), selecting the workflow in Fluent automatically loads the corresponding Scheme file(s) as well.

  Note:

  If you specify a "." here, Fluent will expand it to include the current working directory. In addition, you can use environment variable syntax in the path such as `$env-var` (Linux) and `%env-var%` (Windows), as well as ~ on Linux that denotes your home.

- Use the **Verbosity** option to determine whether or not you see task-related messages in the console window that might assist you with your workflow. The default is **off** which limits the output to the console.

- Use the **Task based display** option under **Graphics Settings** to control how Fluent automatically displays items in the graphics window within the tasks of the workflow.

  – A value of **auto** will automatically display objects (or refinement regions, edge zones, size boxes, etc.) in the graphics window within the task, based on the threshold values defined for the **Automatic display face zone limit** and the **Automatic display facet limit**.

  – A value of **yes** will always automatically change displayed objects in the graphics window within the task, according to the task's display requirements.

  – A value of **no** will never automatically change displayed objects in the graphics window, throughout the entire workflow.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2.13. Getting Help for Workflow Tasks

You can access general and task-specific documentation for the workflows and the associated tasks by selecting the **Workflow Help** button ( ). In addition, the workflows include user assistance for most fields without leaving the workflow, available by hovering next to a field and clicking on the field's exposed help icon.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 3: Using the Watertight Geometry Meshing Guided Workflows

This chapter describes the various tasks that are provided when using the **Watertight Geometry** meshing guided workflow:

3.1. Watertight Geometry Workflow Limitations
3.2. Importing Geometries
3.3. Importing Body of Influence Geometries
3.4. Adding Local Sizing
3.5. Generating the Surface Mesh
3.6. Setting Up Periodic Boundaries
3.7. Describing the Geometry
3.8. Applying Share Topology
3.9. Enclosing Fluid Regions
3.10. Creating Regions
3.11. Updating Regions
3.12. Adding Thin Volume Meshing Controls
3.13. Adding Boundary Layers
3.14. Adding Multizone Controls
3.15. Generating the Multizone Mesh
3.16. Generating the Volume Mesh
3.17. Updating Boundaries
3.18. Improving the Surface Mesh
3.19. Adding Boundary Types
3.20. Improving the Volume Mesh
3.21. Transforming the Volume Mesh
3.22. Extruding the Volume Mesh
3.23. Adding Linear Mesh Patterns
3.24. Managing Zones
3.25. Modifying Mesh Refinement
3.26. Creating Local Refinement Regions
3.27. Running Custom Journal Commands

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.1. Watertight Geometry Workflow Limitations

The following limitations have been observed in the Fluent **Watertight Geometry** workflow:

- Extracting a fluid volume in the workflow requires a conformal mesh for all bounding solids.
- When importing an SCDM file into the workflow, any surface body that is inside another body is directly associated to that body. Use named selections in SCDM or use zone selection in the **Add Local Sizing** task as a workaround for this issue.
- The following are known issues with respect to object names and syntax within the workflow:
  - When importing geometries from SCDM, all colon characters (:) in names (such as solids, named selections, etc.) are converted into hyphens.
  - Names of bodies, components, and named selections should **not** start with numbers,special characters, capital letters, or symbols.
- All bodies of influence must have unique names.
- In order to revert and edit any up-to-date task in another session or on another computer system (such as generating a volume mesh), you will need to have the CAD geometry file (`.pmdb`) and the corresponding workflow directory (*`filename`*`_workflow_files`) available, as well as the correct CAD configuration, and the task will be updated from the last position in the workflow tree.
- Multiple periodic systems are not supported in the workflow.
- Using labels in the capping, local sizing, update boundaries, and the modify mesh refinement tasks will be persistent during design changes. Using zones in these tasks will only be persistent for the same design.
- The workflow only supports a single mesh object. Non-conformal mesh setup is not supported.
- If you need to perform changes to the topology, geometry, or the mesh, then you should incorporate them into the workflow using the custom journal task in order to make the changes persistent in the Fluent workflow. See Customizing Workflows (p. 280) for more information.
- The following tasks and categories have their own corresponding limitations:
  - **Import Geometry**
    - → *Before* importing SpaceClaim files (`.scdoc`) into Fluent Meshing, you need to first enable the **Set as the alternative license preference** license preference option in SpaceClaim in order to uphold licensing preferences.
    - → SpaceClaim (`.scdoc`) files that include faceted geometries should use Named Selections on all "faces" of the faceted geometry, ensuring subsequent remeshing and preserving the faceted faces.
    - → The import of 2D models is not supported.
    - → The import of faceted geometries such as `.stl` and `.tgf` is not supported.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

→ The workflow only fully supports the appending of bodies used for a body of influence using a custom journal.

→ You cannot import a mesh into the workflow that was previously generated by the workflow without first removing all geometry objects and faces and meshing objects using the Outline View.

→ When importing a surface or volume mesh (using the **Mesh** option under **File Format** in the **Import Geometry** task):

- Do *not* directly import into this task any surface meshes that were generated from CAD using the **Watertight Geometry** workflow (you should use **File > Read > Mesh** instead).
- Importing meshes will always result in a separation by region.
- Imported meshes using the `.stl` or `.h5` file formats are not supported.
- Only fully connected meshes are supported. Meshes with hanging node adaption, Cutcell meshes and unconnected faceted geometries (like `.stl`, `.tgf`, `.fmd`) are not supported.
- Surface meshes derived from faceted geometries that are not 100% conformal may not successfully remesh into a conformal mesh.
- Volume meshes with non-conformal refinements are not supported.
- Voids cannot be detected, so the default region assignment may not be the same as when importing the corresponding CAD file.

– **Generate the Surface Mesh**

→ You cannot read a surface mesh into the workflow that was previously generated by the workflow without first removing all geometry objects and faces and meshing objects using the Outline View.

→ When the **Shared Topology** task is used, the following advanced options for the **Generate the Surface Mesh** task are actually performed at the end of the **Shared Topology** task:

- Automatic zone separation
- Automatic remesh to remove clustering
- Final surface mesh improvement

→ After surface meshing, some regions may be improperly designated as dead regions if you have assigned both face labels and body labels on the same body and/or using body labels on multiple bodies.

→ You cannot use the **Size Field File** option to create a surface mesh if the size field contains a body of influence or a face of Influence. In such cases, you should provide the corresponding size control file.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

→ When you create Body of Influence (BOI) bodies using copies of the actual models, named selections may also be copied in the process. Note that BOI bodies containing *faces* with named selections may create errors during the **Generate Surface Mesh** task, and should be avoided.

→ For an imported CAD geometry, where no local sizing is employed, and when **Auto Remesh to Remove Clustering?** is set to **Auto** or **No**, a curvature angle of 18 degrees is present in the **Generate Surface Mesh** task, even when **Proximity** is selected. As a work around, simply select **Curvature** as well and use a larger angle (such as 60 degrees).

– **Add Linear Mesh Pattern**

→ You cannot create patterns using parts/objects that contain voids.

→ You cannot make a pattern using a part that has been used in another pattern.

– **Set Up Periodic Boundaries**

→ For cases when the imported body is not a body, but actually just faces, you should use the manual method of setting the periodicity.

→ This task is not available when the **Capability Level** is set to **Pro** in the Fluent Launcher.

→ There are two scenarios when the **Automatic - pick both sides** method is not able to derive the periodicity parameters:

1. when there are no planar faces, and
2. when the normal of the planar faces do not align with the tangent of the arc of the circle sector.

For rotational periodicity, the **Automatic - pick both sides** method may not accurately derive the correct periodicity parameters when the periodicity angle is small (for example, smaller than ~10 degrees), in which case, the **Manual - pick reference side** method is recommended. Additionally, the periodicity angle is best measured (in SCDM for instance) using the geometry's arc angle and arc radius rather than using the angle between objects and their minimum distance.

→ The **Setup Periodic Boundary** task has to be executed *after* Share Topology

– **Apply Share Topology**

→ The gap between two bodies should not exceed 1/2 of the local mesh size on the faces of the gap.

→ The share topology success rate is increased if faces across gaps are parallel. If share topology is across curved faces, then the success rate will improve if the curvature angle is 12 degrees or less.

– **Add Boundary Layers**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

→ Advanced options (such as acute angles, invalid normals, etc.) are not supported if you choose to read a prism control file into this task. In addition, boundary layer controls can be added to the task, however, any conflicts that may occur between controls that are read in from a file and controls that are added to the task are not checked by the system.

– **Generate the MultiZone Mesh**

→ For the occasional complex geometry where cell variation between boundaries is more uniform, multizone meshing results in overly refined meshes. This is due to the workflow applying a uniform-based mesh rather than a pave-based mesh in proximity-based regions (such as between holes and/or between edges and holes). A slight modification in sizing assignments, leading to a less uniform sizing on boundaries, might resolve the issue.

→ See Strategies for Using Multizone Meshing (p. 349) for more details and caveats.

– **Boundary Layer Settings**

→ Global boundary layers settings are accessible in the **Generate the Volume Mesh** task

→ The workflow no longer supports the reading of external `.pzmcontrol` files using custom journals.

– **Volume Mesh Settings**

→ Generating a volume mesh may be problematic when the number of invalid normals is excessive (that is, greater than 10000) and when invoking the **Modify Surface Mesh at Invalid Normals?** option in the **Add Boundary Layers** task. In such cases, you should set **Modify Surface Mesh at Invalid Normals?** option to **no** to improve volume mesh performance.

→ Volume meshing of geometries with non-manifold edge connections is not supported (non-manifold connection for internal baffles is supported, however). The workaround for this is to use face connections or disconnect at the edge connection.

→ You cannot use different volume mesh types on different regions.

→ Persistent renaming only ensures persistence if unique body names are used.

→ The location of the worst volume cells will be automatically displayed if the worst quality is below the **Quality Warning Limit** available under the **Advanced Options**.

– **Improve Volume Mesh**

→ The location of the worst volume cells will be automatically displayed if the worst quality is below the **Quality Warning Limit**.

– **Extrude Volume Mesh**

→ The preview box is available only for selected faces whose normal is aligned with the Cartesian coordinate.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

→ Extruding cylindrical surfaces too far may create invalid meshes. If your cylindrical surface extrusion exhibits undesirable skewness or otherwise is invalid, reduce the total height of the extrusion and perform the operation again to see if results improve.

– **Modify Mesh Refinement**

→ Refinement uses the existing mesh (and not the geometry) as the geometry on which the mesh is refined. Hence, features that are not well captured (from a geometry standpoint) cannot be improved upon.

– **Named Selections on Multiple Bodies**

→ When a Named Selection is defined on multiple bodies, the workflow recognizes the name but not that the group should be merged into a single region. The separation of the bodies into individual cell zones also separates boundary zones which can increase the number of zones considerably for complex models.

3.2. Importing Geometries

Use the **Import Geometry** task to designate a geometry or surface mesh for your simulation.

1. Use the **File Format** field to choose either a **CAD** geometry, or a surface or volume **Mesh** format.

   Note:

   Do not use the workflow to import a surface mesh that was previously generated and saved from another workflow using a CAD geometry. You should only load previously generated meshes using the **File > Read > Mesh** option.

   If you must use such a model in the Watertight Geometry workflow, you should first read the previously generated mesh using the **File > Read > Mesh** option, use the Outline View to delete the geometry object that corresponds to the mesh object, and save the mesh before reading it into the Watertight Geometry workflow.

2. When the **File Format** is set to **CAD**, use the **Import Type** field to import a **Single File** (the default), or **Multiple Files**. When importing multiple files, the **Select File** dialog allows you to make multiple selections, as long as the files are in the same directory and are of the same CAD format.

3. If a **Mesh** format is chosen, specify the **File Unit** in which the mesh was created.

   The workflow will display the units in the graphics window next to the ruler and while calculating distances (for example, when using the **Distance** tool in the Ribbon).

4. Choose a suitable option for the **Units**. You should work in units where the minimum size of the mesh is of the order of one. The mesh will automatically be scaled to meters while transferring the mesh to the Fluent solver

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. If the **CAD** format is chosen, set the **Use Body Labels** field to **Yes** when the CAD geometry contains body labels that you want to use. As the geometry is read in, if the CAD geometry does not contain any body labels, then this field will automatically be set to **No**.

Note:

- When named selections in the CAD geometry are assigned to only a single body, the workflow supports such body labels in lists and tables.
- When the **Use Body Labels** option is set to **Yes**, then the workflow additionally supports "composite body labels" that occur when named selections in the CAD geometry are assigned to *multiple* bodies.
- The workflow also lists bodies for which no (or a single) body label has been defined as well as any newly created regions.
- A named selection added to a surface body in CAD might not always be recognized as a label in the **Update Boundaries** task.

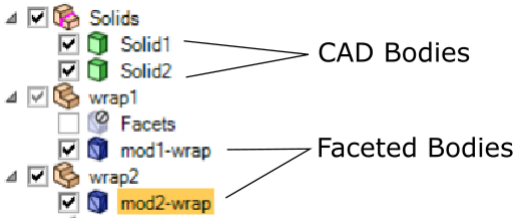
---

6. Click **Advanced Options** to access additional controls prior to performing this task. Options include:

- In some CAD environments (for example SpaceClaim, DesignModeler, or Discovery), faceting and wrapping operations lead to non-CAD geometries that cannot be directly used in the workflow. In such cases, you can set the **Model includes shrink-wrapped bodies?** prompt to **yes** to properly manage such geometries. In such cases, all faceted bodies should only have names that end with "`*wrap`" (or the default of "`shrinkwrap`"), and other bodies should not use such a naming convention.

Note:

You should shrink wrap any faceted bodies to ensure a relatively good quality surface mesh. In addition, each shrink-wrapped faceted body should be defined in its own unique component within the CAD environment.



- If a **Mesh** format is chosen, use the **Automatic Object and Label Creation?** prompt to determine whether or not mesh objects and labels are automatically created upon import, potentially and dramatically increasing the mesh import speed for very large cases. By default,

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

this is set to **yes** however, if it is set to **no**, then no labels are created and a single mesh object is created employing all zones.

Note:

Zones that contain the string "boi" are excluded from object creation.

- Use the **Separate Zone By** option to determine how zones are separated, either by region alone, by region as well as a specified separation angle, by face/edge topology, or none. Zone separation is necessary if local sizing must be applied. Region separation is sufficient for body sizing and/or body of influence, while separation by angle is needed for local face sizing. Separation by face will preserve the face/edge topology so that individual faces and edges can be selected when setting up a multizone mesh.

  When **Separate Zone By** is set to **region & angle**, set the **Separation Angle** to a value for which CAD zones are separated. The lower the value for the angle, the higher the number of faces that will be separated. The default value is **40** degrees.

  When **Separate Zone By** is set to **face**, edge labels will be created automatically on all edges, preserving the face/edge topology so that these individual entities can be selected while setting up a multizone mesh (see Adding Multizone Controls (p. 344)). Zones are automatically merged back on a body/region level after volume meshing. If **Separate Zone By** is set to **none** (the default), the face and edge topology will be merged together while importing the CAD geometry.

  Note:

  For models where only a small portion is to be used for multizone meshing, this option is not recommended since creating and sharing edge labels can be computationally expensive.

  In addition, when **Separate Zone By** is set to **face**, note the following:

  – You will no longer are able to separate zones based on angle (see Generating the Surface Mesh (p. 307)).

  – Multizone meshing will not work if you have edges contained within a face with no angle between them and adjacent faces

  – All named selections on edges defined within the CAD model will be ignored.

  – You can no longer insert or use the **Manage Zones** task prior to generating the volume mesh (see Managing Zones (p. 379)).

- Enable **Use custom faceting** in order to set the following faceting options for your imported geometry:

  – Set the **Tolerance (10% of min-size)** value, however, using the default value of 0 results in the coarsest possible faceting.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– Set the **Max Facet Length** value to avoid very large facets during file import.

7. Browse for a specific **File Name**.

   Supported file types are SpaceClaim (`.scdoc`), Discovery (`.dsco`), and Workbench (`.agdb`) files and also `.pmdb` files.

   Note:

   When a SpaceClaim (`.scdoc`) file is imported into Fluent while in meshing mode, Fluent also creates an intermediary `.pmdb` file that can be imported. The `.pmdb` file should always reside alongside the `.scdoc` file. When changes are made to the geometry in SpaceClaim, and the file is reimported into Fluent, the original `.pmdb` file is overwritten. The `.pmdb` file can be more easily and quickly read into Fluent for additional processing and should be used as long as the `.scdoc` file has not changed.

   Note:

   SpaceClaim (`.scdoc`) files are only supported on Windows. When working on Linux systems, however, you can use the intermediary `.pmdb` file as your geometry file for the workflows.

   On Windows, use the **Import CAD Geometry** dialog to import the CAD file into Fluent, and enable the **Save PMDB (Intermediary File)** option in the **Import Options** dialog. After the file is imported, you can move the generated `.pmdb` file over to your Linux system to use in your workflow.

8. If you have manually entered a CAD geometry file name, click **Import Geometry**, otherwise the task is automatically performed if you browse and select a CAD file.

   If you need to make adjustments to any of your settings in this task, click **Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

9. Proceed to the next step in the workflow.

3.3. Importing Body of Influence Geometries

Bodies of influence are often defined in separate CAD geometry or mesh files. Once you have imported your geometry/mesh into your workflow, you can use the **Import Body of Influence Geometry** task to import any bodies of influence you may have defined in separate files.

Note:

This task must be added *after* you have completed the **Import Geometry** task in your **Watertight Geometry** workflow.

1. Use the **File Type** field to choose either a **CAD** geometry, or a surface or volume **Mesh** file type. The units for length will be the same as those specified in the **Import Geometry** task.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Browse for a specific **File Name**. You can specify multiple CAD and mesh files, since bodies of influence can be specified using several different files, and each CAD file may contain multiple bodies.

   Important:

   Importing geometries with connected bodies and/or regions (for example, share topology) is not permitted.

   Supported file types are SpaceClaim (`.scdoc`), Discovery (`.dsco`), and Workbench (`.agdb`) files and also `.pmdb` files.

   Note:

   When a SpaceClaim (`.scdoc`) file is imported into Fluent while in meshing mode, Fluent also creates an intermediary `.pmdb` file that can be imported. The `.pmdb` file should always reside alongside the `.scdoc` file. When changes are made to the geometry in SpaceClaim, and the file is reimported into Fluent, the original `.pmdb` file is overwritten. The `.pmdb` file can be more easily and quickly read into Fluent for additional processing and should be used as long as the `.scdoc` file has not changed.

   Note:

   SpaceClaim (`.scdoc`) files are only supported on Windows. When working on Linux systems, however, you can use the intermediary `.pmdb` file as your geometry file for the workflows.

   On Windows, use the **Import Body of Influence Geometry** task to import the CAD file into Fluent, and enable the **Save PMDB (Intermediary File)** option in the **Import Options** dialog. After the file is imported, you can move the generated `.pmdb` file over to your Linux system to use in your workflow.

3. Once your selections are made, click **Import Body of Influence Geometry** and proceed onto the next task.

   Note:

   Once this task is completed, the **Add Local Sizing** task is automatically enabled in your workflow (see Adding Local Sizing (p. 302)).

   If you need to make adjustments to any of your settings in this task, click **Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

3.4. Adding Local Sizing

You can gain better control over the mesh size distribution by using the **Add Local Sizing** task. Using this task, you can define specific mesh size controls that operate on specific, localized, portions of the geometry and mesh. Using this task, you can add as many localized size controls to the workflow as

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

you need, depending on the requirements and details of your geometry. Note that this task can only be added to the workflow prior to the **Generate the Surface Mesh** task.

For the **Would you like to add local sizing?** field, select **yes** if you need to define local sizing parameters using the following steps. Otherwise, if you do not need to define local sizing controls, keep the default of **no**, click **Update** and proceed to the next task.

1. Provide a **Name** for the new size control, or use the default name. The default name changes depending on the assigned **Size Control Type**.
2. Provide a **Growth Rate**.
3. Choose the **Size Control Type**. Choices include:
   - Use the **Edge Size** setting for refining the local sizing based on the edge size. This option is only available if the imported geometry contains one or more *named* edges (for example, an ANSYS SpaceClaim Design Modeler CAD geometry with one or more edges explicitly using named selections).
   - Use the **Face Size** setting for refining the local sizing based on the face size.

     Note:

     For imported CAD geometries that include body labels (and if you have set the **Use Body Labels** field to **Yes** in the **Import Geometry** task, see Importing Geometries (p. 298)), available body labels are listed in the task for you to select (they are not selectable, however, from the graphics window).

   - Use the **Body Size** setting for refining the local sizing based on the body size.

     Note:

     For imported CAD geometries that include body labels (and if you have set the **Use Body Labels** field to **Yes** in the **Import Geometry** task, see Importing Geometries (p. 298)), available body labels are listed in the task for you to select (they are not selectable, however, from the graphics window).

   - Use the **Body of Influence** setting, or *BOI*, to assign a maximum size on all parts of your geometry that falls within the boundaries of the body of influence.

     Note:

     For a body that is associated with a Body of Influence (BOI body), keep the following in mind:

     – The BOI body must not share topology with the CAD model itself within SCDM or DM.

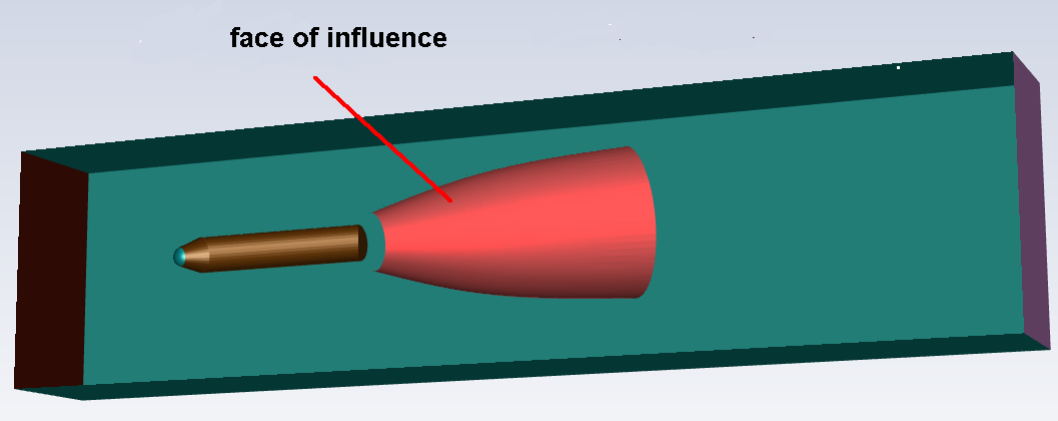
     – The BOI body must have a unique name within SCDM or DM.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

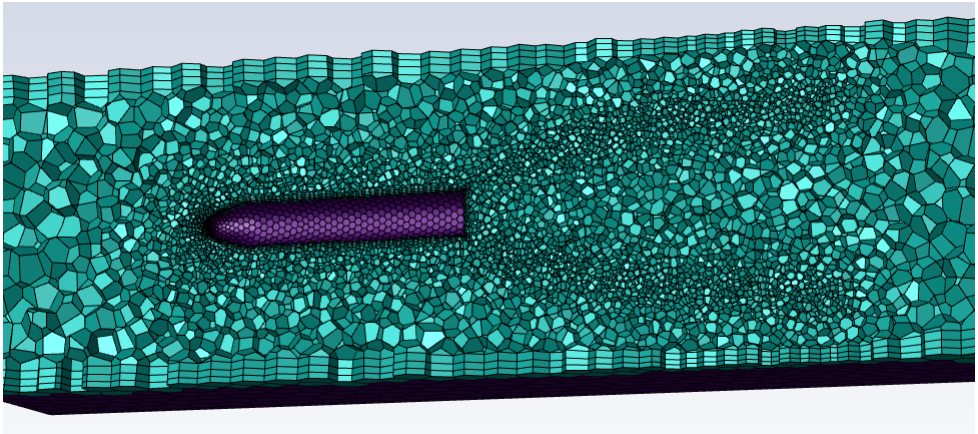
– All imported BOI bodies must be assigned a local sizing control.

You can enable the **Repair Body of Influence** option in the rare case that the initial body of influence is not fully closed.

- Use the **Face of Influence** setting, to assign a maximum size on all parts of your geometry that falls within the boundaries of the face of influence. A *face* of influence is essentially a body of influence on a non-closed, non-connected face (for example, a surface representing the wake behind a moving object).



The face is deleted during surface meshing, however the mesh refinement is visible during volume meshing



- Use the **Curvature** setting for refining the local sizing based on the underlying curve and surface curvature. This size control type is useful, for instance, in models with a combination of large and small scales.

- Use the **Proximity** setting for refining the local sizing based on the number of cells per gap specified.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. If you are using the **Face Size**, **Body Size**, **Body of Influence**, or **Face of Influence** size control type, provide a **Target Mesh Size**.

   Note:

   Clicking in this field displays red boxes in the graphics window, providing a visual representation of the field value. Use the **Clear Preview** button to hide the visualization display.

   Note:

   Ensure that the assigned size of the BOI is smaller than the BOI itself, and not perfectly aligned with the CAD model (or other BOI bodies), to avoid mistakenly invoking shared topology and associated warnings due to thin regions and soft size functions.

   Keep in mind that even if you do not include the BOI bodies in the share topology, the share topology will still affect overlapping BOI bodies if the parent to the bodies have the Share Topology property enabled in SCDM.

5. If you are using the **Curvature** size control type, you have the following options:

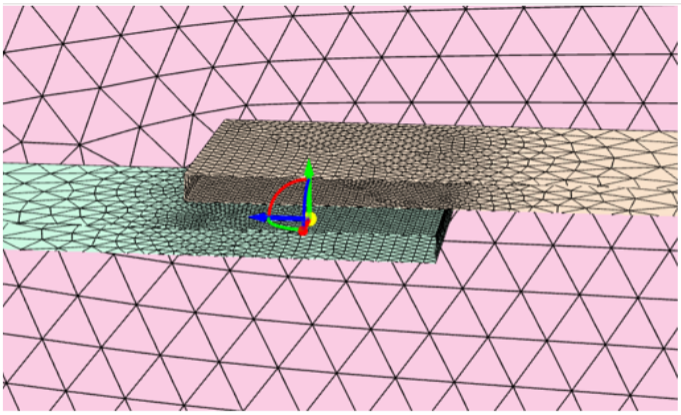
   - Specify a value for the **Local Min Size** for the curvature size control.
   - Specify a value for the **Max Size** for the curvature size control.
   - Specify a value for the **Curvature Normal Angle** for the curvature.
   - Specify a value for the **Scope to** field, where you can localize the size control to faces, edges, faces and edges, or edge labels. Edge selection selects the edges of a particular face.

6. If you are using the **Proximity** size control type, you have the following options:

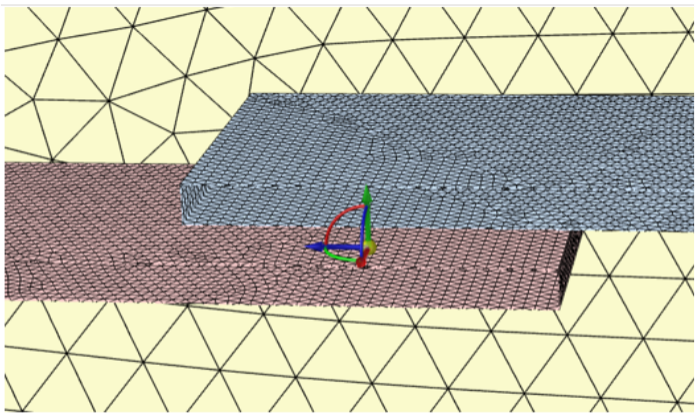
   - Specify a value for the **Local Min Size** for the proximity size control.
   - Specify a value for the **Max Size** for the proximity size control.
   - Specify a value for the **Cells Per Gap**. This value is the number of element layers to be generated in a gap for the edge proximity size function. Note that for proximity size functions, the number of cells per gap can be a real value, with a minimum value of 0.01. See Proximity (p. 587) for more information.
   - Specify a value for the **Scope to** field, where you can localize the size control to faces, edges, faces and edges, or edge labels.

7. If you are using the **Proximity** size control type, and you **Scope to** either **faces** or **faces and edges**, use the **Refine Thin Regions (Ignore Orientation)?** field to determine if you need to additionally refine the mesh in and around thin areas such as between plates, without providing over-refinement. This ignores face proximity within voids and will not allow you to refine in thin voids, but will allow refinement in gaps. This should be used in predominantly fluid regions with no thin solid regions.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Refine Thin Regions? = No:
Refinement Restricted Only to Intersection of Plates



Refine Thin Regions? = Yes:
Refinement Along Both Plates and Their Intersection

8. You can select an available zone or label to apply your local sizing changes. Choose whether to **Select By** the zone name, or the label name in the list below.

   Note:

   Labels originate from the CAD geometry, such as from group names in SpaceClaim geometries, or from named selections in DesignModeler geometries.

   Select items in the list, or use the **Filter Text** option in the drop-down to provide text and/or regular expressions in filtering the list (for example, using *, ?, and []). You can also choose the **Use Wildcard** option in the drop-down to provide wildcard expressions in filtering the list. When you use either ? or * in your expression, the matching list item(s) are automatically selected in the list. Use ^, |, and & in your expression to indicate boolean operations for NOT, OR, and AND, respectively. See Filtering Lists and Using Wildcards (p. 287) for more information.

9. Select the **Draw Size Box** field to visualize the size control's minimum and maximum sizes in the graphics window (in the form of red cubes).

10. Once your selections are made, click **Add Local Sizing** and proceed onto the next task.

    You can add as many local sizing controls as you require for your workflow, each operating on different zones and/or with different sizing parameters. The size controls will appear as sub-tasks under the parent task.

    If you need to make adjustments to any of your settings in this task, select the specific size control sub-task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

    Note:

    If you need to add another local sizing control after you have already created a surface mesh, you need to revert and edit at least one existing local size control in order to properly see and use the available geometry objects in the graphics window and the object listing. Alternatively, you can also re-import your geometry and update the **Add Local Sizing** task.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11. Once you are satisfied with your changes, proceed to the next step in the workflow.

3.5. Generating the Surface Mesh

Use the **Create Surface Mesh** task to create a conformal, connected surface on all of the objects of an imported geometry, or remesh all or parts of an imported surface mesh. Use this task to identify regions that will later be filled with the volume mesh. In many cases, the default values will be sufficient for a useful CFD surface mesh.

1. You can generate your surface mesh based on settings defined in separate size control files or size field files. If you would like to use your own size field or size control files, provide an answer for the **Use Custom Size Field/Control Files?** prompt:

   - Choose **No** (the default) if want to proceed with setting surface mesh details in this task.
   - Choose **Yes, size field file** if you have a size field file to apply to the workflow. Select a **Size Field File** name when prompted. Enable the **Preview Size Field** check box to visualize the size field in the graphics window.

     If the size field contains a body of influence or a face of Influence, you cannot use the **Size Field File** option to create a surface mesh. In such cases, you should provide the corresponding size control file.
   - Choose **Yes, size control file** if you have a size control file to apply to the workflow. Select a **Size Control File** name when prompted.
   - Choose **Yes, both files** if you have both a size field and a size control file to apply to the workflow. Select a **Size Field File** name and a **Size Control File** name when prompted. Enable the **Preview Size Field** check box to visualize the size field in the graphics window.

   Size control files are described in Size Control Files (p. 595). Size field files are described in Size Field Files (p. 595).

2. If you chose to import a surface mesh in the **Import Geometry** task (Importing Geometries (p. 298)), you are prompted to indicate which portions of the surface mesh you want to remesh in this task.

   - If you *have not* assigned local sizing in the **Add Local Sizing** task (Adding Local Sizing (p. 302))::
     - Choose **All** (the default) so that all surfaces will be re-meshed with the settings from this task.
     - Choose **Selected** where you can select specific faces or zones from a list for remeshing. Using this option, you cannot select all surfaces. In such cases, use the **All** option.
     - Choose **None** so that no faces or zones re-meshed in this task.

     Note:

     Changing the mesh size on an imported mesh will always require a local size control.

   - If you *have* assigned local sizing in the **Add Local Sizing** task (Adding Local Sizing (p. 302)):

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– Choose **All** (the default) so that all surfaces will be re-meshed with the settings from this task.

– Choose **Assigned in Local Sizing** to remesh only the boundaries that were assigned in the **Add Local Sizing** task. Enable the **Preview Assigned Local Sizing** check box to see a preview of those previously assigned boundaries.

– Choose **Assigned in Local Sizing and Selected** to remesh those assigned boundaries from the **Add Local Sizing** task, as well as any other boundaries selected in this task. Enable the **Preview Assigned Local Sizing** check box to see a preview of those previously assigned boundaries.

Note:

Adjacent zones are always included with assigned local sizing zones, such that **Assigned in Local Sizing and selected** includes boundaries assigned in the **Add Local Sizing** task, their adjacent zones, as well as the selected boundaries in this task.

3. Set the **Minimum Size** of the facets for the surface mesh.

Note:

Clicking in this field displays red boxes in the graphics window, providing a visual representation of the field value. Use the **Clear Preview** button to hide the visualization display.

4. Set the **Maximum Size** of the facets for the surface mesh.

Note:

Clicking in this field displays red boxes in the graphics window, providing a visual representation of the field value. Use the **Clear Preview** button to hide the visualization display.

5. Specify the **Growth Rate**.

6. Choose a **Size Function**.

- The **Curvature** size function can be used for refining the surface mesh based on the underlying curve and surface curvature.

- The **Proximity** size function can be used for creating the surface mesh, based on the number of cells per gap specified.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- By default, a **Curvature and Proximity** size function is assigned based on both curvature and proximity.

Note:

If local sizing has been added, these curvature and proximity size controls are appended to the local sizes and the resulting size field is used to dictate the sizes during surface meshing.

For additional information, see Size Functions and Scoped Sizing (p. 584).

7. For the **Ignore Proximity Across Objects?** option, determine if you would like to ignore any small artificial gaps that may exist between two objects/bodies; especially applicable to assemblies that typically have duplicate faces and edges in between each body. These duplicate edges and faces may lead to additional artificial refinement and additional expense. Setting this option to **Yes** will effectively create a single proximity control for each body, eliminate artificial gaps, and improve performance.

Note:

Using this option is strongly recommended if you intend to use the **Interface Connect** method in the **Share Topology** task (Applying Share Topology (p. 321)).

---

8. For the **Curvature** or the **Curvature and Proximity** size functions, specify the **Curvature Normal Angle** for the curvature size function. The default value of 18 degrees should approximately produce 20 facets in the circumferential direction of a cylinder.

9. For the **Proximity** or the **Curvature and Proximity** size functions, specify the **Cells Per Gap** for the proximity size function. This value is the number of element layers to be generated in a gap for the edge proximity size function. Note that for proximity size functions, the number of cells per gap can be a real value, with a minimum value of 0.01. See Proximity (p. 587) for more information.

10. For the **Proximity** or the **Curvature and Proximity** size functions, choose the **Scope Proximity To** value for the proximity size function, which can be scoped to edges, faces, or both faces and edges.

11. Select the **Draw Size Box** field to visualize the size control's minimum and maximum sizes in the graphics window (in the form of red cubes).

12. Use the **Separate Out Boundary Zones by Angle?** option to determine whether or not to separate face zones. By default, this is set to **No**.

    If set to **Yes**, separation is based on a specified separation angle. Separation is needed in case named selections for inlets, outlets, capping, local boundary layers, etc. have not been defined within the CAD model in advance. You should only select **Yes** if you need to separate faces for capping, boundary conditions, or inflation on specific faces. Use the **Separation Angle** option to specify a

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

desired angle for determining separation. Assigning a smaller separation angle will produce more zones.

Note:

You will have the option to re-merge zones in the **Generate the Volume Mesh** task (see Generating the Volume Mesh (p. 351)).

---

13. If you want to remesh a portion of the imported surface mesh (by choosing **selected** for the **Which boundaries do you want to remesh?** prompt), you can select an available zone or label to apply your remeshing changes.

    Choose whether to **Select By** the label or zone name in the list below.

    Note:

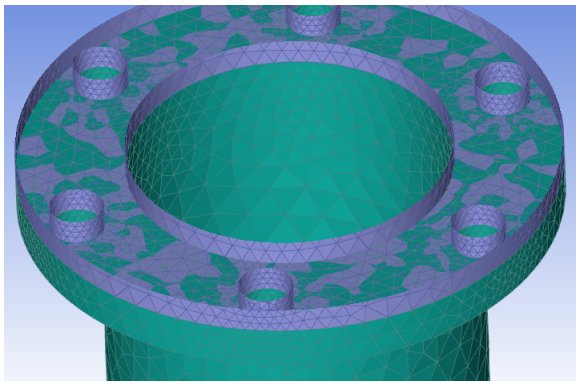
    Labels originate from the CAD geometry, such as from group names in SpaceClaim geometries, or from named selections in DesignModeler geometries.

    Select items in the list, or use the **Filter Text** option in the drop-down to provide text and/or regular expressions in filtering the list (for example, using *, ?, and []). You can also choose the **Use Wildcard** option in the drop-down to provide wildcard expressions in filtering the list. When you use either ? or * in your expression, the matching list item(s) are automatically selected in the list. Use ^, |, and & in your expression to indicate boolean operations for NOT, OR, and AND, respectively. See Filtering Lists and Using Wildcards (p. 287) for more information.

14. Click **Advanced Options** to access additional controls prior to performing this task. For additional information, see Face Connectivity Issues (p. 626) and Quality Checking (p. 628). Options include:

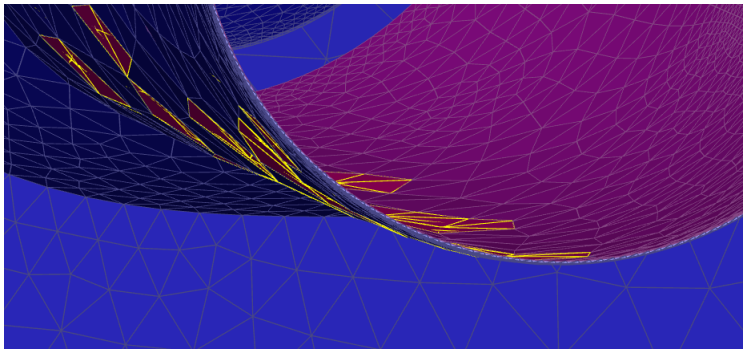
    - Use the **Check Self-Intersection?** field to determine whether or not the system will check for self-intersecting faces as part of the surface mesh creation. This check will, for example, detect if share topology has been omitted. For larger models, this check, however, can be time consuming, so if a model initially passes this test the first time, it can be safely disabled.

      Figure 3.1: Example of a Self-Intersection: Double Faces Appear When Share Topology is Not Enabled

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 3.2: Example of a Self-Intersection: Local Mesh Size is Significantly Larger Than the Pipe Thickness



Note:

Importing a surface mesh naturally involves a limited amount of topological information (as opposed to directly importing a CAD geometry), thereby making it difficult to determine when you can, or when you want to, use share topology. Note that while share topology can be detected using the **Check Self-Intersection?** field. This option, however, tends to be computationally expensive for large meshes. When trying to determine when to, or when not to, use share topology, consider the following scenarios:

– If you do not know if share topology is required: enable the **Check Self-Intersection?** option (with default settings), and share topology will be automatically enabled.

– If you know that share topology is required, but you would like more control of the settings: disable the **Check Self-Intersection?** option, and manually enable the **Apply Share Topology** task (Applying Share Topology (p. 321)).

– If you know that share topology is not required: disable the **Check Self-Intersection?** option, and do not enable the **Apply Share Topology** task. This is useful to consider when, for example, you have a large mesh and do not want to incur the computational expense of using the **Check Self-Intersection?** option.

Note that these same scenarios apply even if you are using a CAD plug-in instead of a CAD reader.

- Use the **Smooth Folded Faces/Repair Free Nodes Limit** option to provide a value limiting when folded faces are resolved during surface mesh creation.
- Use the **Auto Assign Zone Types?** option to automatically assign boundary types to zones during surface mesh creation. For example:

If the Boundary Name Contains...	**...Assign Boundary Type to...**
*mass*inlet*	mass flow inlet

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the Boundary Name Contains...	...Assign Boundary Type to...
*inlet*vent*	inlet vent
*press*inlet*	pressure inlet
inlet	velocity inlet
*mass*outlet*	mass flow outlet
*outlet*vent*	outlet vent
outlet	pressure outlet
*intake*fan	intake fan
exh*fan*	exhaust fan
*fan or *fan.* or *fan:* or *fan_	fan
symmetry	symmetry
*far*field*	pressure far field
outflow	outflow
*porous*jump*	porous jump
radiator	radiator
overset	overset
internal or *interior*	internal
interface	interface

Note:

In addition, only exterior boundary zones are assigned to corresponding external boundary types (that is, an interior zone named "inlet" will not be assigned to a **velocity-inlet** boundary type).

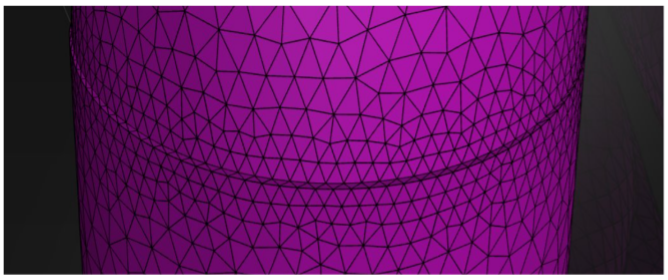
- Use the **Invoke Quality Improve?** option to choose whether or not quality measures are applied during surface mesh creation. Surface mesh quality improvements should be invoked after importing the CAD model.
  - When **Invoke Quality Improve?** is enabled, an attempt will be made to improve the quality on those triangles where the skewness is above the **Quality Improve Skewness Limit**.
  - When **Invoke Quality Improve?** is enabled, four consecutive attempts are made to improve the quality, each using a larger adjacent angle, until the **Quality Improve Max Angle**.
  - When **Invoke Quality Improve?** is enabled, and after quality improvements, using the **Quality Improve Skewness Limit** and **Quality Improve Max Angle** are complete, if any remaining triangles are above the **Quality Improve Collapse Skewness Limit** value, these triangles will be aggressively removed using a fixed maximum angle of 120 degrees.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

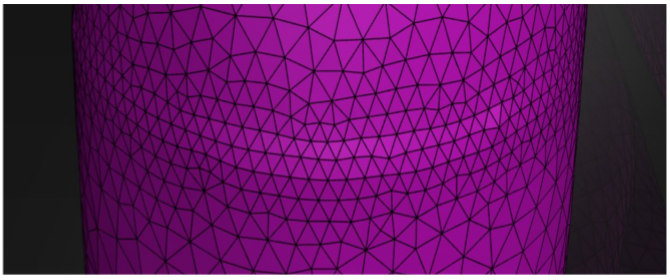
- When using bodies of influence (for example, body sizing, local refinement regions, etc.), you can use the **Set Volume Mesh Max Size?** prompt to determine whether or not you need to assign a maximum size for the volume mesh. By default, this is set to **no**, however, when set to **yes**, you can then specify a value for the **Volume Mesh Max Size**, and eliminate the need for Fluent to perform additional size function calculations when generating the volume mesh.

- Use the **Remove Steps?** prompt if there are any small imperfections (ledge or step artifacts from the original CAD geometry) that can be removed as part of the surface meshing task. If you specify **yes**, then you need to indicate the **Max Step Width**.

  For example, a typical 90 degree step artifact can be removed and replaced with a flattened mesh, removing any artificial pressure loss. Since this is performed before auto-remeshing, some coarsening may occur locally.



Remove Step? = No:
Step is Retained



Remove Step? = Yes:
Step is Removed

  Step removal only occurs if the step(s) and the adjacent triangles belong to the same face zone or the same label, therefore, there are ways to protect steps from being removed:

  – Adding a label to the step itself (or to a face adjacent to the step) will protect the step from being removed.

  – Zone separation will protect all steps.

  Note:

  All steps within a zone (or within a label) are checked and treated together, therefore it is enough that one step in a zone has a label adjacent to it in order to protect all steps.

  This can also be performed as part of improving the surface mesh (see Improving the Surface Mesh (p. 359)).

  In other words, steps can only be removed under the following conditions:

  – The step, and adjacent faces, does not belong to any named selection

  – The step, and adjacent faces, belongs to the same named selection

  – In some cases, you should consider disabling the quality improvement option in the **Generate the Surface Mesh** task so that you obtain a "cleaner" step removal.

- Use the **Auto Remesh to Remove Clustering?** option to automatically remesh the surface mesh to remove excessive clustering of nodes.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– For imported mesh files, the **auto** option will not remesh by default. Remeshing will only occur if you are remeshing all faces *and* have set the **Auto Remesh to Remove Clustering?** option to **yes**.

– For imported size field files, the **auto** option will not remesh by default, and you should set the **Auto Remesh to Remove Clustering?** option to **yes**.

– For all other cases, using the default of **auto** will only invoke automatic remeshing if the **Add Local Sizing** task has been added to the workflow.

Alternatively, you can use the much faster refaceting technique rather than the automatic remeshing technique for the surface mesh by selecting **no, use re-faceting** for the **Auto Remesh to Remove Clustering?** option. This is particularly useful for thin volume meshing driven by face-face proximity where the original faceting is not fine enough to provide an accurate mesh size.

15. Click **Generate the Surface Mesh** to generate a CFD surface mesh for the imported CAD geometry.

During the process of creating the surface mesh for an imported CAD assembly geometry containing multiple parts, Fluent can detect whether or not the geometry has shared topology enabled. If the imported geometry does *not* have shared topology enabled, Fluent will provide your workflow with a **Shared Topology** task (within the **Describe the Geometry** task, see Applying Share Topology (p. 321)) where you can identify and close any problematic gaps and choose whether to join and/or intersect the problematic faces.

When the **Shared Topology** task is used, the following advanced options for the **Generate the Surface Mesh** task are actually performed at the end of the **Shared Topology** task:

- Automatic zone separation
- Automatic remesh to remove clustering
- Final surface mesh improvement

Note:

An assortment of information and messages appear in the console when the task is complete. For instance, for imported CAD geometries that include body labels (and if you have set the **Use Body Labels** field to **Yes** in the **Import Geometry** task, see Importing Geometries (p. 298)), statistics such as skewed cells, average and maximum skewness, and face count for available body labels are listed for you to review.

If you need to make adjustments to any of your settings in this task, click **Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

16. Proceed to the next step in the workflow.

Additional quality improvements can be made to surface meshes using the **Improve Surface Mesh** task (see Improving the Surface Mesh (p. 359)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.6. Setting Up Periodic Boundaries

You can use the **Set Up Periodic Boundaries** task to define rotational and/or translational periodicity in your simulation. This task essentially re-meshes a periodic face to exactly match the reference side of the face, as well as define the corresponding Fluent boundary types (periodic and shadow), in preparation for the Fluent solver. If your periodic simulation also involves shared topology (Applying Share Topology (p. 321)), you should first invoke the **Apply Share Topology** task prior to the **Set Up Periodic Boundaries** task.

Note:

This task is not available when the **Capability Level** is set to **Pro** in the Fluent Launcher.

Note:

This task can usually be added to your workflow *after* the **Create Surface Mesh** task, however, you are able to add this task *before* generating your surface mesh, albeit with some limitations (noted below). Specifically, you are limited to only being able to manually pick the periodic faces and only via labels. Benefits can include, for example, that for large models, where the size function calculation is very time consuming, and when the mesh density is asymmetric between the periodic sides, inserting the periodicity setup task before generating the surface mesh will avoid an additional size function calculation.

In addition, you cannot use the **Manual - pick reference side** option to apply periodicity when you have zones that consist of multiple, detached regions. Using the **Automatic - pick both sides** option (along with using labels) separates these zones into individual regions on both sides, allowing for a successful set up and recovery of the periodic boundary condition.

Multiple instances of the **Set Up Periodic Boundaries** task can be inserted into the workflow for defining multiple periodic conditions. Additionally, automatic multiple periodic setup is provided for setting up a translational periodic boundary with multiple periodic pairs.

Note:

The following limitations apply when defining multiple periodic conditions or when using automatic multiple periodic setup:

- You cannot have two different periodic boundaries sharing an edge.
- The combination of multiple periodicity and multizone meshing within a single region is not supported.

Automatic multiple periodic setup can only be used to define **Translational** periodic boundaries and **Automatic - pick both sides** must be selected for the **Method**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Use the **Type** field to choose whether you are creating **Rotational** or **Translational** periodic boundaries.

   Note:

   In order to ensure the recovery of translational periodicity, you should also apply a local proximity size control on periodic surfaces (with a **Cells Per Gap** setting of 2).

2. Use the **Method** field to choose how you are going to define your periodic boundary. This option is not available when this task is added to the workflow *before* the **Generate Surface Mesh** task: the manual method is all that is available.

   - Choose **Automatic - pick both sides** to define the periodic boundary based on your zone/label selections

     Note:

     When using this option, you must select two zones or labels.

   - Choose **Manual - pick reference side** to manually assign properties of the periodic boundary.

     Note:

     When using this option, you only need to select a single zone or label.

     You would use this option in the following situations:

     – You already know the periodicity angle, or translational shift, the components of the origin, and the components of the vector, and which side is the reference side.

     – All your periodic faces are non-planar.

     – The mesh is asymmetric on the two periodic faces, and you need to use the side with the refined mesh as a reference.

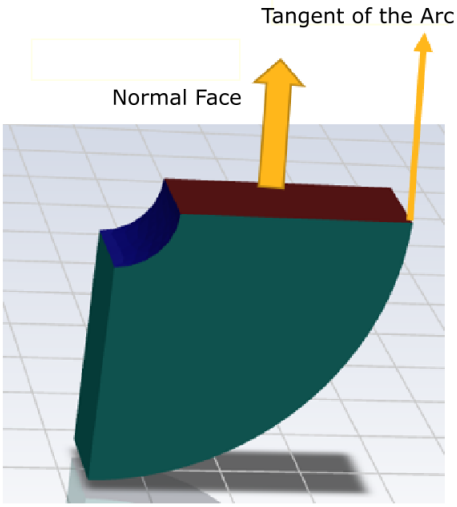
     Note:

     There are two scenarios when the **Automatic - pick both sides** method is not able to derive the periodicity parameters:

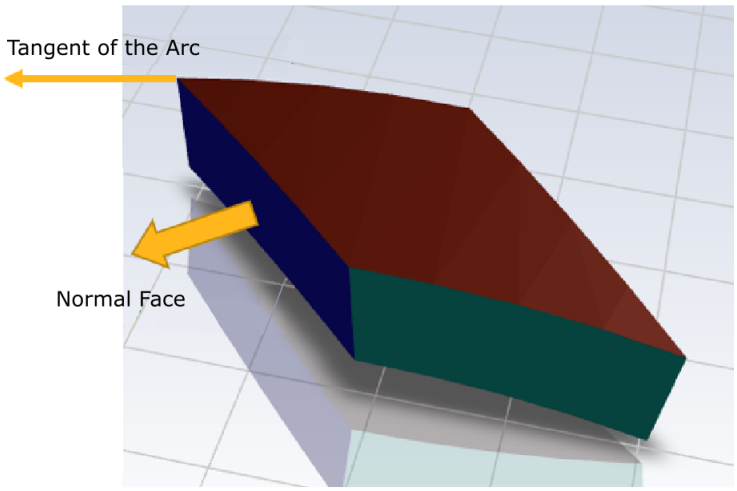
     a. when there are no planar faces, and

     b. when the normal of the planar faces do not align with the tangent of the arc of the circle sector.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Automatic method works since the angle can be derived from periodic faces.



Automatic method does not work since the angle cannot be derived from periodic faces.

For rotational periodicity, the **Automatic - pick both sides** method may not accurately derive the correct periodicity parameters when the periodicity angle is small (for example, smaller than ~10 degrees), in which case, the **Manual - pick reference side** method is recommended. Additionally, the periodicity angle is best measured (in SCDM for instance) using the geometry's arc angle and arc radius rather than using the angle between objects and their minimum distance.

For rotational periodicity:

a. Specify a value for the **Periodicity Angle**.

b. For the **Origin Components**, specify a value for **X**, **Y**, and **Z**.

c. For the **Vector Components**, specify a value for **X**, **Y**, and **Z**.

For translational periodicity, under **Translational Shift**, specify a value for **X**, **Y**, and **Z**.

3. Choose whether to **Select By** the zone name or the label name in the list below.

Note:

Labels originate from the CAD geometry, such as from group names in SpaceClaim geometries, or from named selections in DesignModeler geometries.

Select items in the list, or use the **Filter Text** option to provide text and/or regular expressions in filtering the list (for example, using *, ?, and []). See Filtering Lists and Using Wildcards (p. 287) for more information.

4. If your model consists of multiple translational periodic pairs, you can select **yes** for the **Are you using the Automatic Multiple Periodic Setup?** option to automatically define the periodicity for multiple periodic pairs.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- When set to **yes**, the **Tagging method for Automatic Multiple setup** option becomes available for specifying how the periodic pairs will be identified. If your model has one named selection for each periodic pair, you should select the **Paired** option, which is the default selection. If your model contains either of the following, you should select the **Alphabetical** option:

  a. If your model contains one named selection at each side of the model in alphabetical order.

  b. If your model contains one named selection for all periodic surfaces and one named selection for all shadow surfaces.

5. (optional) Use the **Remesh Asymmetric Mesh Boundaries** field to determine whether to remesh boundaries when there is an asymmetric mesh on the periodic faces: **auto** will let the system determine the best course of action, **yes** will permit the remeshing of the boundaries when the task is updated, and **no** will not remesh the boundaries.

6. (optional) By default, the table only lists external wall boundaries. When using labels, you can select the **List All Labels** check box to see additional boundary labels, such as fluid-fluid internal boundaries.

7. Once your selections are made, click the **Set Up Periodic Boundaries** button and proceed to the next task.

   Once the task is completed, you will see an indicator in the graphics window showing the origin and vector, as well as updated mesh, where you can verify that the periodic sides have identical meshes. When the automatic option is used, the console displays the calculated periodicity angle, origin components, and vector components for future reference.

   If you need to make adjustments to any of your settings in this task, click **Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

   Note:

   If you return to the **Set Up Periodic Boundaries** task after creating a volume mesh, you can restore the visibility of the mesh by using the **F2** hot key and either the **Fit to Window** icon ( ) or the **Draw Mesh** button at the bottom of the task.

3.7. Describing the Geometry

The **Watertight Geometry** workflow uses the **Describe Geometry** task to let you specify the type of geometry you are importing: whether it is a solid model, a fluid model, or both. Settings in this task determine what sub-tasks are available for the overall task.

Note:

This task involves zone assignments based on naming conventions of various boundaries. See Prerequisites for the Fluent Guided Workflows (p. 278) for more detailed information.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. For the **Geometry Type**, choose from the following:

   - **The geometry consists of only solid regions**: for solid models, you are provided with sub-tasks related to creating capping surfaces over any openings in the geometry, and calculating regions.
   - **The geometry consists of only fluid regions with no voids**: for fluid models with no pockets, you are provided with a sub-task for updating boundaries.
   - **The geometry consists of both fluid and solid regions and/or voids**: for geometries with both a solid and a fluid, and potential pockets, you are provided with sub-tasks related to creating capping surfaces over any openings in the geometry, updating boundaries, and calculating regions.

2. Indicate whether or not your goal is to cover any openings in your geometry and to extract a fluid region.

3. Indicate whether or not you want to have Fluent convert all fluid-fluid boundary types from 'wall' to 'internal'. The default is **No**, however, if you select **Yes**, Fluent will convert all interior boundary types from walls to internal boundaries. Named selections that include the string "wall" are excluded from this conversion. Note that only internal boundaries bounded by two fluid regions are converted into internal zone types. If new fluid regions are assigned, this task is executed after the **Update Regions** task. Internal boundaries that are designated as "baffles" are retained as walls.

4. Indicate whether or not you want to apply the geometry's Share Topology settings. The default is **No**, however, if you select **Yes**, Fluent will add an **Apply Share Topology** task to the workflow (see Applying Share Topology (p. 321)).

   Note:

   In situations where you want to use overlapping non-conformal interfaces, you must use the non-conformal option. In all other situations, such as when you have totally disconnected bodies (that is, with no overlap), you should instead elect to choose the Share Topology option even if there is nothing to share.

5. Indicate whether or not you need to consider multiple zone meshing. The default is **No**, however, if you select **Yes**, Fluent will add an **Add Multizone Controls** task (see Adding Multizone Controls (p. 344)) and a **Generate Multizone Mesh** task to the workflow (see Generating the Multizone Mesh (p. 350)).

6. Indicate whether or not you want to create non-conformal meshes between objects.

   Note:

   In situations where you want to use overlapping non-conformal interfaces, you must use the non-conformal option. In all other situations, such as when you have totally

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

disconnected bodies (that is, with no overlap), you should instead elect to choose the Share Topology option even if there is nothing to share.

Note:

This option is not available if your geometry is already conformal (that is, when Share Topology is already employed.

In addition, this option is not supported when using the **Modify Surface Mesh at Invalid Normals?** advanced option in the **Add Boundary Layer** task (Adding Boundary Layers (p. 340)).

For more information, see Best Practices for Using Non-conformal Interfaces (p. 320)

7. Once your selections are made, click the **Describe Geometry** button and proceed to the next task.

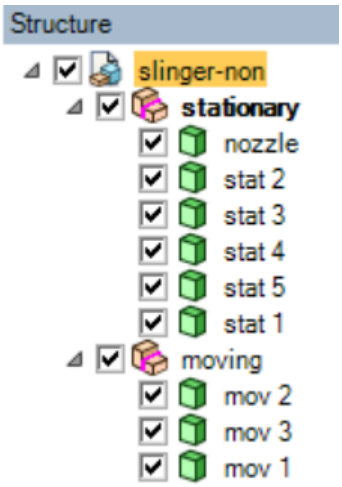
   If you need to make adjustments to any of your settings in this task, click **Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

3.7.1. Best Practices for Using Non-conformal Interfaces

To properly take advantage of using non-conformal meshes, you should be aware of the following limitations and caveats:

- When using body labels, ensure that all model bodies have unique names.
- In the **Describe Geometry** task, you cannot use the **Non-Conformal Mesh between objects?** option in conjunction with the **Apply Share Topology** task.
- The final volume mesh is performed sequentially for each conformal part.
- In order to extract a flow volume, all parts (and their caps) are required to be conformal, therefore, you cannot create flow volumes that surround non-conformal parts.
- Edges from two different non-conformal parts should not be combined to form a single edge label (that is, an edge label should only be defined within its own non-conformal part).
- When using force-share in SpaceClaim/DesignModeler/Discovery, if the **Only Create Contact Groups** option is invoked, contact group named selections are created between non-conformal parts. If the non-conformal option is invoked in workflow, these contact groups are converted to "interface" zone boundary types. This option will **not** have an impact on shared body-body interfaces, only across non-conformal interfaces.
- The following figure illustrates the proper setup of a model in SpaceClaim/DesignModeler/Discovery to be meshed using the non-conformal option in the Watertight Geometry meshing workflow:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Under each sub-component, the bodies are shared. This sharing is indicated as such by the red line in the component icon. Note however, that the top component's icon does not have any such line. When a line appears in the icon at that level, the setup is incorrect and the **Generate Surface Mesh** task cannot be completed.

3.8. Applying Share Topology

During the process of creating the surface mesh for an imported CAD assembly geometry containing multiple parts, Fluent can detect whether or not the geometry has shared topology enabled. If the imported geometry does not have shared topology enabled, Fluent will provide your workflow with an **Apply Share Topology** task (within the **Describe the Geometry** task) where you can identify and close any problematic gaps and choose whether to join and/or intersect the problematic faces.

In an assembly, each individual body contains all of its faces, none of which are shared with another body, therefore making the model "non-conformal" to Fluent, where special handling is required for the solver. By performing the **Apply Share Topology** task, all bodies in contact will share faces across bodies, and the model is made "conformal" therefore making the model compatible with the Fluent solver.

Note:

When ANSYS SpaceClaim DesignModeler cannot perform a **Force Share** operation, the problematic faces are identified as specifically named selections (for example, `Connect Topology 25` or `Connect Topology 100`, and so on). You can use such partially shared topologies within the workflow, however, you must retain those specifically named selections - you should *not* remove or rename them.

Note:

You must resolve acute face-face angles using fillets before attempting any share topology operation, since these could otherwise be unintentionally collapsed.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Specify a value for the maximum gap distance (either **Max Gap Distance for Join-Intersect** or **Max Gap Distance for Connect** depending on the **Share Topology Method**).

2. Click the **Show Marked Gaps** button to identify gaps that are less than the gap distance.

   Note:

   The maximum gap distance should be equal to or less than half of the value specified for the **Minimum Size** in the **Generate the Surface Mesh** task. Also, the maximum gap distance should never exceed the thickness of a solid or fluid body, otherwise the solid or fluid bodies may potentially collapse.

   Note:

   You should be cautious when setting an appropriate **Minimum Size** for the surface mesh. Using an unnecessarily small minimum size value, relative to the gap distance, may cause problems generating a successful **Share Topology** task.

   See Troubleshooting Gap Marking (p. 324) for more information.

3. For the **Share Topology Method**, keep the default value of **Join-Intersect** in order to join and/or intersect the problematic faces, or choose **Interface Connect** in order to connect edges of overlapping face pairs, providing potential increases in speed and robustness.

   Note:

   When using the **Interface Connect** method, it is strongly recommended that you also use the **Ignore Proximity Across Objects?** option in the **Create Surface Mesh** task (Generating the Surface Mesh (p. 307)).

   Note:

   Since the **Interface Connect** method uses both edge stitching and the join operations to resolve overlapping faces, the **Interface Connect** method will still locate the overlapping faces and attempt to join them even if labels have not been assigned to some overlapping face pairs.

   In addition, note that imprinting of overlapping faces in SCDM/Discovery is highly recommended for the **Interface Connect** method.

4. Click **Advanced Options** to access additional controls prior to performing this task. Options include:

   - For the **Interface Connect** share topology method, use the **Interface Labels Selection Method** to determine how interface labels are to be selected when using the **Interface Connect** share topology method. Options include:

     – **Automatic - Using Connect Topology** (available when force share connect topology is utilized in the SCDM geometry) will automatically use the labels where

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

connect topology is used. In SCDM, these labels are of the form `Connect To-pology1`, and so on. Fluent translates them into the workflow and processes the labels as `connect-topology1`, and so on, before finally being removed once the **Share Topology** task is complete.

  - **Manual** allows you to manually select the interface labels from the list of available labels. This option lets you select both the connect topology labels as well as any specific labels that you need to retain (for example, if you have defined connect topology labels in SCDM but you want to retain some interfaces and keep specific labels for later re-use in the solver or post processing).

  - **Automatic** will automatically separate face zones, identify overlapping faces, and assign the interface connect labels. This option is useful when connect topology has not been utilized in Ansys SpaceClaim Direct Modeler (SCDM) or if the mesh was obtained from another source, such as Ansys DesignModeler (DM). Performance, however, may be impacted.

    Note:

    Zones are separated during the automatic interface connect, however, they will be merged back after the volume mesh is generated.

- When using the **Interface Connect** share topology method, for the **Will You Set Up Periodic Boundaries?** prompt, select **yes** if periodic boundaries are present. Select the appropriate periodic boundaries from the list since these boundaries need to be treated differently in this task.
- For the **Join-Intersect** share topology method, use the **Join Pair Normal Angle** option to specify the feature angle to identify features in the overlap region.
- Use the **Initial Relative Tolerance** option to specify the tolerance value (relative to face edges) for locating the overlapping faces. This is the relative tolerance for joining or intersecting face pairs, or the relative tolerance for connecting edge and face pairs.
- For the **Join-Intersect** share topology method, use the **Number of Join Attempts** option to control the number of times the task attempts to join faces within the gap.
- Use the **Join Tolerance Increment** option to control the change in tolerance with each join attempt.
- For the **Join-Intersect** share topology method, use the **Join and Intersect Method** option to control whether the gap faces are joined, whether they are intersected, or both.
- Use the **Rename Internals Based on Body Names?** prompt to determine whether you will apply body names from the imported geometry and use them in naming internal

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

boundaries. For instance, the internal face between two regions "solid" and "body" would be renamed to "solid-body".

Note:

The **Initial Relative Join Tolerance** can be aggressive, in relation to the **Number of Join Attempts**, so increasing the number of attempts may yield better results for certain circumstances. In addition, improvements are possible if you cross-imprint overlapping faces within SpaceClaim so that you have one-to-one face matching.

---

5. Once your selections are made, click the **Apply Share Topology** button and proceed to the next task.

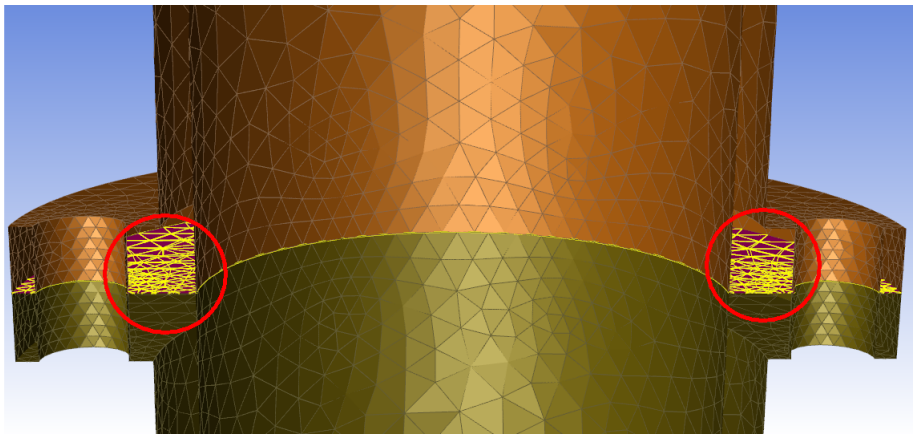
   The task locates and joins problematic face gaps and intersections through a series of incremental attempts until the geometry exhibits the equivalent of shared topology.

   If you need to make adjustments to any of your settings in this task, click **Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

3.8.1. Troubleshooting Gap Marking

In the **Shared Topology**, use the **Show Marked Gaps** button to display any gaps in the geometry less than the maximum gap distance size. Gaps between object pairs are identified and highlighted in the graphics window.

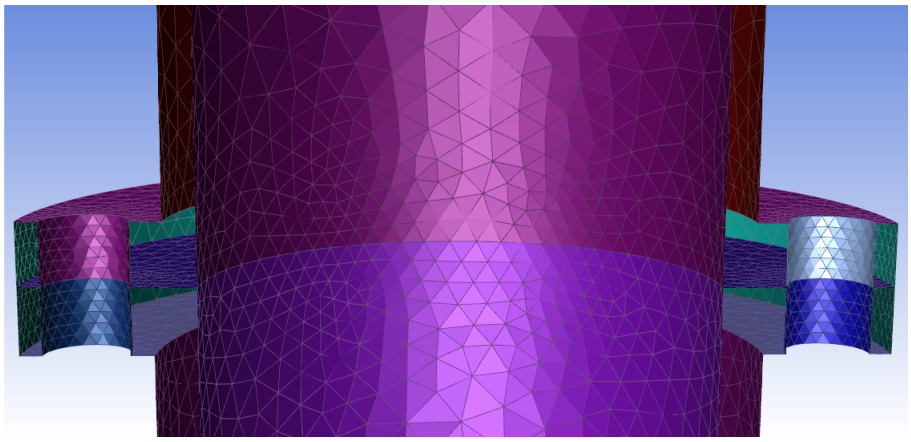
Figure 3.3: Showing Marked Gaps



Once marked, you can select the **Apply Share Topology** button to update the task and view the final results. Based on the task's default settings, pairs are processed and joined and/or intersected according to various tolerances over a number of incremental steps, until all identified pairs are successfully processed and all gaps between parts are closed.

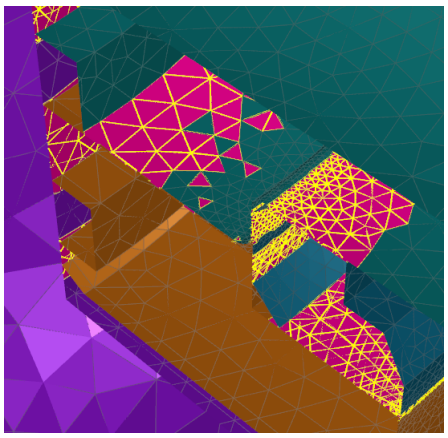
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 3.4: Applying Share Topology to Marked Gaps



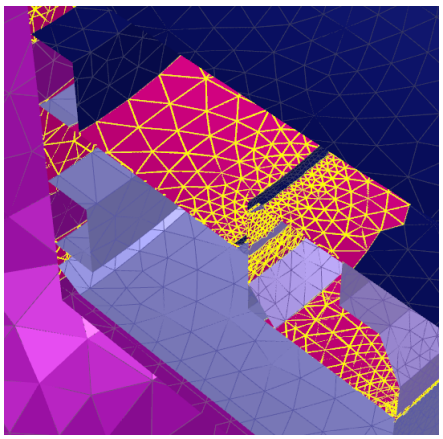
In some cases, sporadic gap markings are possible when the maximum gap distance is not sufficient:

Figure 3.5: Incomplete Gap Marking (Maximum Gap Distance = 0.12)



Improvements can be made by adjusting the maximum gap distance accordingly.

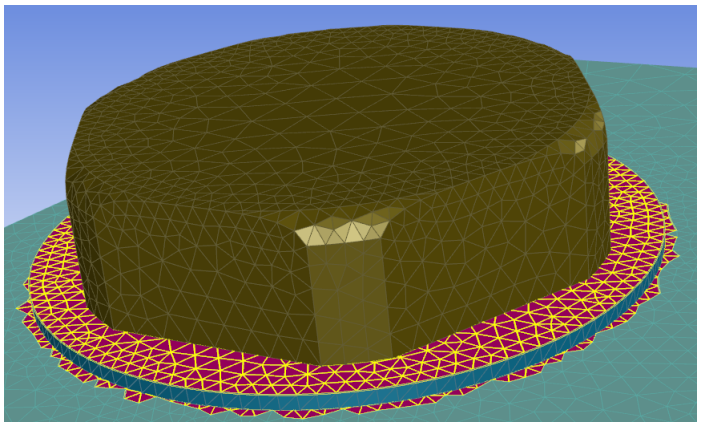
Figure 3.6: Complete Gap Marking (Maximum Gap Distance = 0.15)



Additionally, using a value for the gap distance that is too aggressive may lead to a collapse of the geometry entirely once the task is updated.

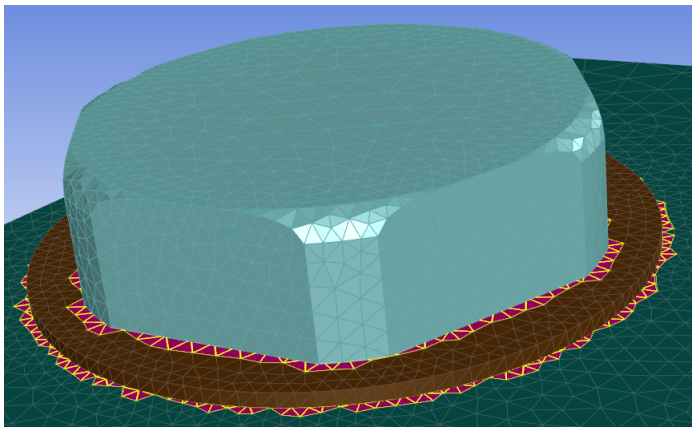
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 3.7: Excessive Gap Marking Around a Washer (Maximum Gap Distance = 1.2)



Slight readjustments of the maximum gap distance in accordance with the minimum size can be successful:

Figure 3.8: Proper Gap Marking Around a Washer (Maximum Gap Distance = 0.8)



In some case, you may be required to adjust and re-adjust the value of the maximum gap distance field so that all gaps are properly identified and closed.

3.9. Enclosing Fluid Regions

For solid model geometries, you can use the **Enclose Fluid Regions (Capping)** task to cover, or cap, any openings in your geometry in order to later calculate your fluid region(s).

1. Specify a **Name** for the capping surface, or use the default name. The default name changes depending on the assigned **Zone Type**.

2. Choose the **Zone Type** for the new cap. Choices include:

   - velocity-inlet
   - pressure-outlet
   - pressure-inlet

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

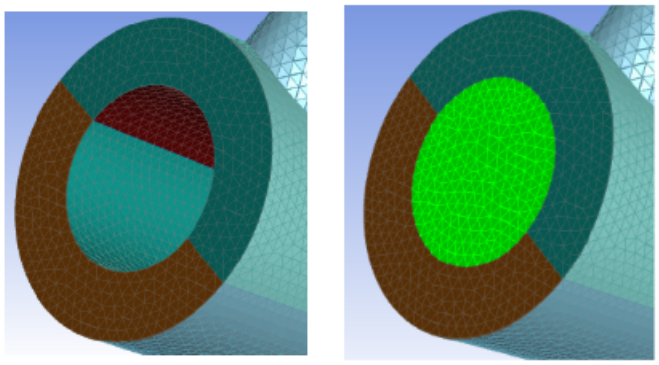
- pressure-far-field
- mass-flow-inlet
- outflow
- symmetry
- wall

3. Choose whether the **Cap Type** will be based on a single surface opening, or if it is an annular opening with two surfaces.

Note:

A single surface opening can potentially have more than one face comprising the single surface:

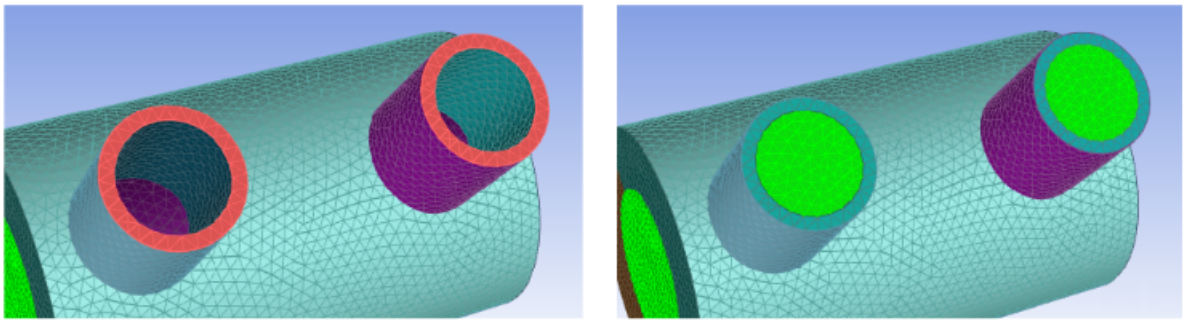
Figure 3.9: Example of a Single Surface Cap with Multiple Faces



A single surface opening can also mean selecting multiple, distinctly separate, surfaces to create a single cap:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

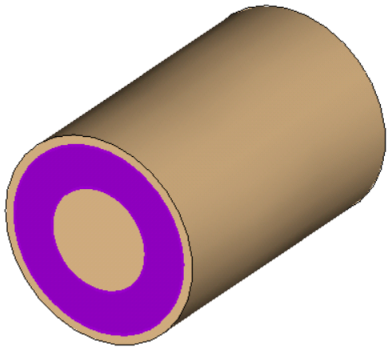
Figure 3.10: Example of a Single Surface Cap with Multiple Faces



Note:

A single named selection using both the inner and outer face is the only valid selection for annular capping using labels.

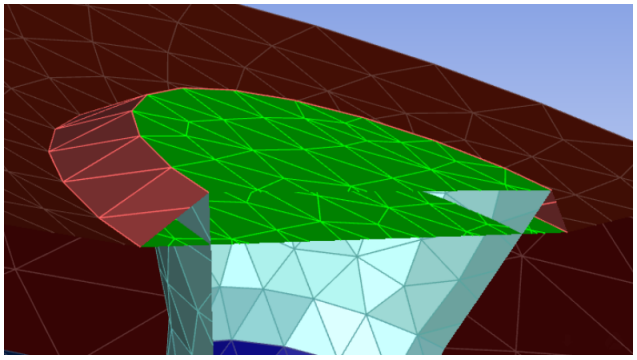
Figure 3.11: Example of an Annular Cap Type



Note:

The workflow does not support tilted annular faces for capping (highlighted in red, below). Currently, two caps are created (highlighted in green, below) and the resulting intersecting cap must be manually removed,

Figure 3.12: Example of a Problematic Tilted Annular Opening



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Choose whether to **Select By** the label or zone name in the list below.

   Note:

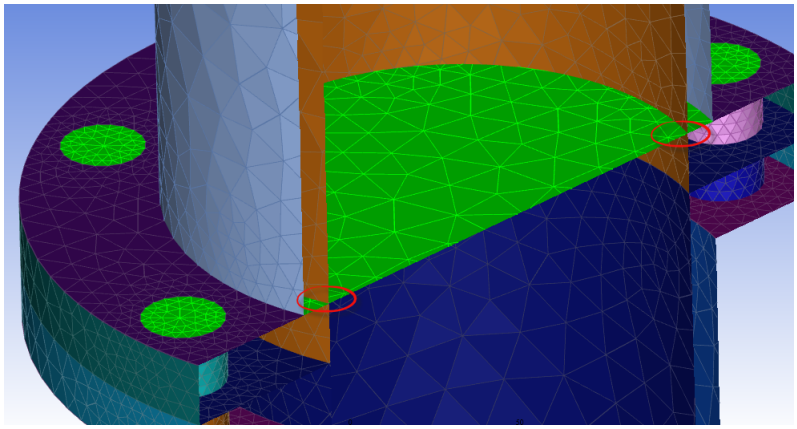
   Labels originate from the CAD geometry, such as from group names in SpaceClaim geometries, or from named selections in DesignModeler geometries.

   Select items in the list, or use the **Filter Text** option in the drop-down to provide text and/or regular expressions in filtering the list (for example, using *, ?, and []). You can also choose the **Use Wildcard** option in the drop-down to provide wildcard expressions in filtering the list. When you use either ? or * in your expression, the matching list item(s) are automatically selected in the list. Use ^, |, and & in your expression to indicate boolean operations for NOT, OR, and AND, respectively. See Filtering Lists and Using Wildcards (p. 287) for more information.

5. Click **Advanced Options** to access additional controls prior to performing this task. Options include:

   - Use the **Check Cap Self-Intersection?** option to control whether or not the system will detect if the capping surface intersects with any other face in the model. If an intersected face is found, it is automatically deleted. To increase the efficiency of the capping task, this option should be set to **no**.

     Figure 3.13: Example of a Self-Intersection: Additional Cap Intersects With Other Surfaces

   - Use the **Max Cap Edge Count Limit** option to control the number of edges that can be present on the capping surface.

6. Click **Create Cap(s)**. The new capping object will appear in the workflow and in the graphics window.

   In cases where a face has multiple holes, and only a portion of them will be used for fluid extraction, Fluent will cap all holes on the face, and will only use the capped openings that are required for extracting the fluid region.

   If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

7. Repeat as needed for additional inlets, outlets, etc., until all openings have been assigned a type and have been created.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. Once all openings have been covered, proceed to the next step in the workflow.

3.10. Creating Regions

You can specify the number of fluid regions to be included in your simulation using the **Create Regions** task.

1. Choose the number of fluid volumes you wish to create in the **Estimated Number of Fluid Regions** field, or use the default value.

   Note:

   The system will detect additional regions if they exist, however, it will detect fluid regions only where they are connected to capping surfaces.

   By default, all bodies are solid regions and all voids are considered "dead" regions. The conversion to a fluid region can happen in many ways:

   - There is a named selection matching `*fluid*` or `*enclosure*` or `air*`.
   - Any region sharing a boundary with any of the inlet or outlet boundary types will automatically changed to fluid region.
   - Any region sharing "internal" boundaries will also change to "fluid" region.

   When compute regions occurs the estimated value are based on existing fluid regions.

2. If any dead regions are present, you can use the **Do you want to retain dead region names?** prompt to determine how such regions are named. Voids or dead regions are usually named `dead0`, `dead1`, `dead2`, and so on, and can remain so when this prompt is set to **no**. When this prompt is set to **yes**, however, the dead region names will also be prefixed with the original dead region name (usually derived from an adjacent region), such as `dead0-fluid:1`, `dead1-fluid:2`, and so on.

3. Click **Create Regions**.

   If you need to make adjustments to any of your settings in this task, click **Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

   Note:

   Once this task is completed and updated, if fluid extraction is required, Fluent will display the newly created fluid regions.

4. Use the **Draw Regions** button to display the available regions in the graphics window.

5. When the flow volume is generated, proceed to the next step in the workflow.

   For solid models, you may proceed to the **Generate the Volume Mesh** task.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Additional enhancements can be made to regions using the **Update Regions** task (see Updating Regions (p. 331)).

3.11. Updating Regions

You can update the properties of any defined region using the **Update Regions** task. This task can be added to the workflow as many times as you require.

The table contains a list of all of the defined regions, and their assigned types.

1. (optional) Use the **Filter** button to filter the table contents based on a particular column.

2. Assign a **Region Name** as needed by double-clicking the current name.

   You can also rename one or more regions by selecting them in the table, right-click, and select **Set Region Name** in the context menu, and provide a new name directly in the menu.

   For imported CAD geometries that include body labels (and if you have set the **Use Body Labels** field to **Yes** in the **Import Geometry** task, see Importing Geometries (p. 298)), body label names are also listed alongside region names in the table (though their names cannot be edited).

3. Assign a **Region Type** as needed using the corresponding drop-down menu. Available region types include:

   - fluid
   - solid
   - dead

   Note:

   Dead regions are the same as a void or a pocket in the domain, and are *not* transferred to the Fluent solver.

   Multiple regions can be assigned a specific type all at once by selecting them in the table, right-click, and select **Set Region Type** in the context menu, then designate a type for the selected regions directly in the menu.

4. (optional) Click the **Print Summary** button to display (in the console window) a list of the number and type of regions. For example:

   ```
   ---------------- Model consists of 1 Fluid 2 Solid and 0 Dead(void) regions.

   ---------------- The total number of regions is 3.
   ```

5. Use the **Draw Regions** button to display the available regions in the graphics window.

   Multiple regions can be visualized all at once by selecting them in the table, right-click, and select **Draw Selections** in the context menu.

6. When you are satisfied with the region assignments, click **Update Regions**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

7. Once all regions have been updated, proceed to the next step in the workflow.

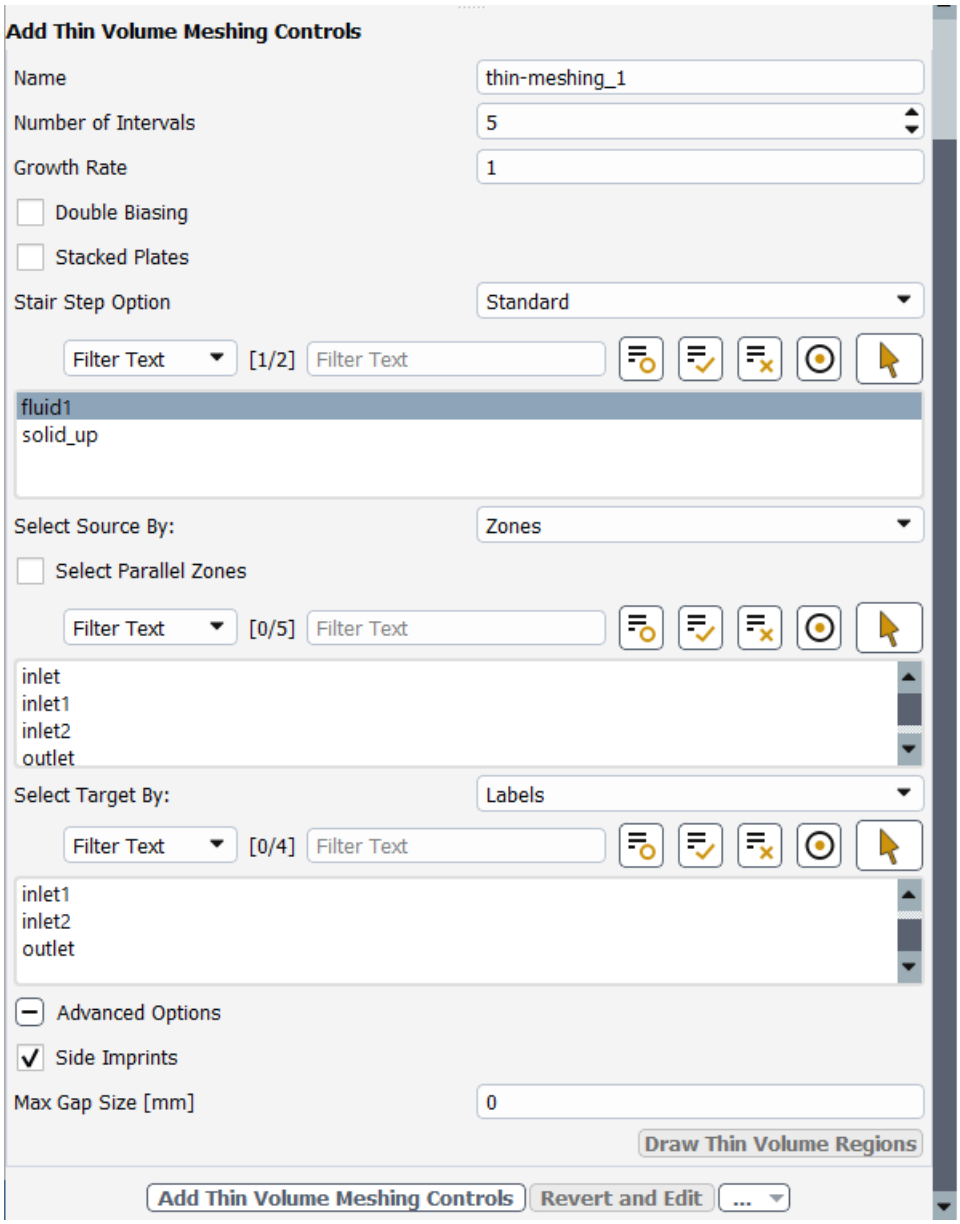
3.12. Adding Thin Volume Meshing Controls

You can use the **Add Thin Volume Meshing Controls** task to create a layered mesh within thin structures that can be combined with fill methods for volume mesh generation. Thin volume meshing controls can also be utilized for meshing stacked planar plates as described in Thin Volume Meshing of Stacked Plates (p. 338). Prior to using this task, note the following assumptions and limitations:

- Other than for stacked plates, the structure where thin meshing is applied should be "thin" and have a height/base ratio less than 5.
- Boundary layers are not supported within thin volume regions. All thin volume regions will be filtered out from the list of available regions in the **Add Boundary Layers** task.
- The **Add Thin Volume Meshing Controls** task must be inserted before the **Add Boundary Layers** task.
- Thin volume meshing is not compatible with periodic boundaries.
- If you create both a multizone mesh and a thin volume mesh (in different regions), the **Non-Conformal Mesh between Multizone and the rest?** option will automatically be enabled within the **Generate the Multizone Mesh** task.
- When the **Add Thin Volume Meshing Controls** task is added, all regions (with or without thin volume mesh controls) will be meshed in serial.
- For some configurations, robustness issues may arise which can usually be resolved by using a finer mesh. Additionally, it is recommended that all gaps have a minimum resolution of 1 cell per gap.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 3.14: The Add Thin Volume Meshing Controls Task Page



1. Specify the thin volume mesh parameters:

   a. The **Growth Rate** defines the expansion rate of the extrusion for each thin volume mesh layer and is set to 1 by default. A growth rate of 1.2 for example will expand each layer of the extrusion by 20 percent of the previous length.

   b. Doubling biasing on edges of the thin volume mesh layers can be invoked by enabling the **Double Biasing** option. Enabling double biasing will automatically set the **Growth Rate** to 1.3.

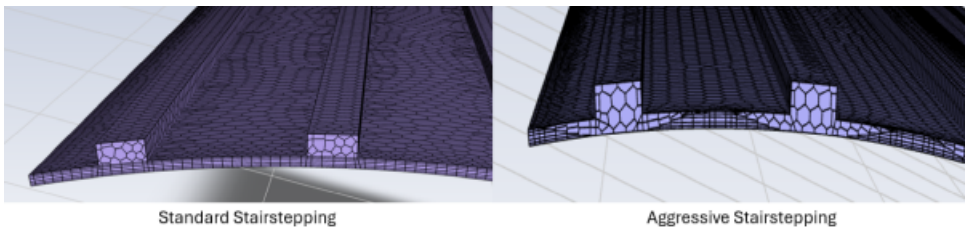
Note:

When **Double Biasing** is disabled, the thin volume mesh can only be graded from the **Source** to the **Target**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. Choose whether to have **Standard** or **Aggressive** stairstepping for the **Stairstep** option.

Figure 3.15: Comparison of Standard and Aggressive Starstepping



2. Select the **Region** where thin volume meshing is to be applied.

3. Choose how you want to select the **Source** and **Target**, by **Labels** or by **Zones**. Note that the thin volume mesh will be created in the region between the selected source and target surfaces. For example, the source and target surfaces for an electromagnetic air-gap model can be visualized in Figure 3.16: Target and Source Zones for Electromagnetic Air-Gap Model (p. 335) below.

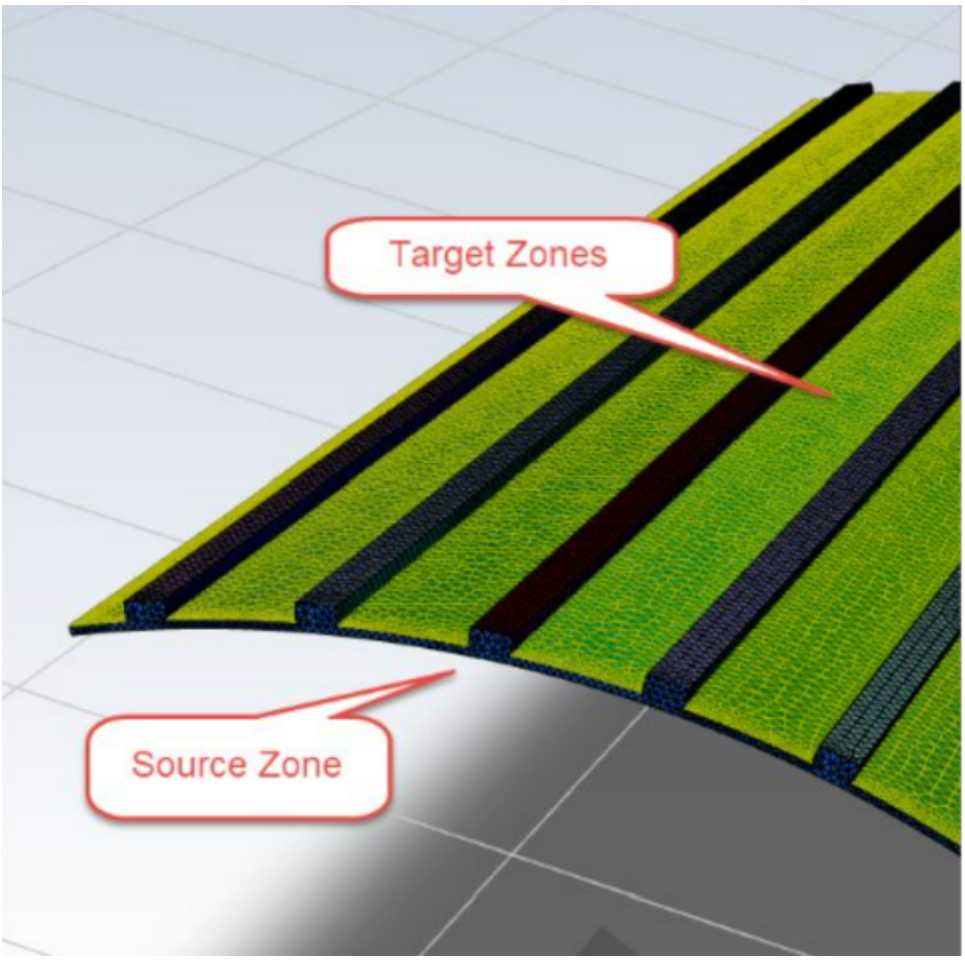
Note:

You can manually select regions and source/target surfaces within the lists, or you can select them using the graphics window by first clicking the button, then selecting the regions/surfaces in the graphics window which will automatically update the selected regions/surfaces in the list. Note that after clicking the button, the button will become highlighted ( ), indicating that graphical selection is enabled.

When manually selecting items in this lists, you can use the **Filter Text** option in the drop-down to provide text and/or regular expressions in filtering the list (for example, using *, ?, and []). You can also choose the **Use Wildcard** option in the drop-down to provide wildcard expressions in filtering the lists. When you use either ? or * in your expression, the matching list item(s) are automatically selected in the list. Use ^, |, and & in your expression to indicate boolean operations for NOT, OR, and AND, respectively. See Filtering Lists and Using Wildcards (p. 287) for more information.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 3.16: Target and Source Zones for Electromagnetic Air-Gap Model



In the above model, a series of non-thin structures are placed along the top surface of the thin air-gap, whereas the bottom surface of the air-gap is smooth throughout. For this example, and for most cases, the smooth (bottom) surface of the thin structure should be selected as the **Source** zone. For the **Target** zone(s), the surface(s) attached to any non-thin structures should be selected.

Note:

- The surface mesh size on source and target faces should be between 1 and 2 times the overall thin-region thickness.
- If the thin volume region is non-conformally connected to another volumetric region, the source and target region can only be selected by zone.

4. Click **Advanced Options** to access additional controls prior to performing this task.

   a. **Side Imprints** are enabled by default which specifies the mesher to project the outer nodes of the thin volume mesh onto adjacent boundary face zones. This ensures that geometric details of the thin volume are accurately captured at the boundary.

   b. The maximum gap distance (**Max Gap Size**) between thin mesh layers is computed automatically and should not be changed for most cases.

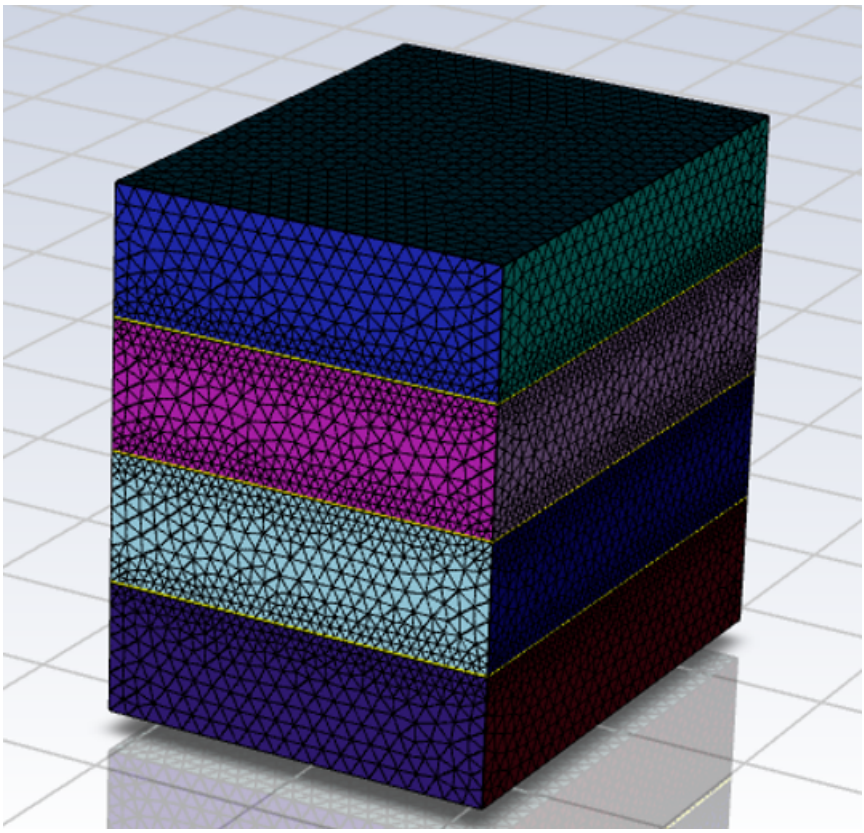
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Click the **Add Thin Volume Meshing Controls** button to save the changes.

3.12.1. Thin Volume Meshing of Parallel Zones

If you have multiple zones in parallel that you want to select for thin meshing, you can enable the **Select Parallel Zones** option. In Figure 3.17: Stack of Thin and Thick Plates (p. 336) a stack of plates is shown which consists of both thick and thin plates.

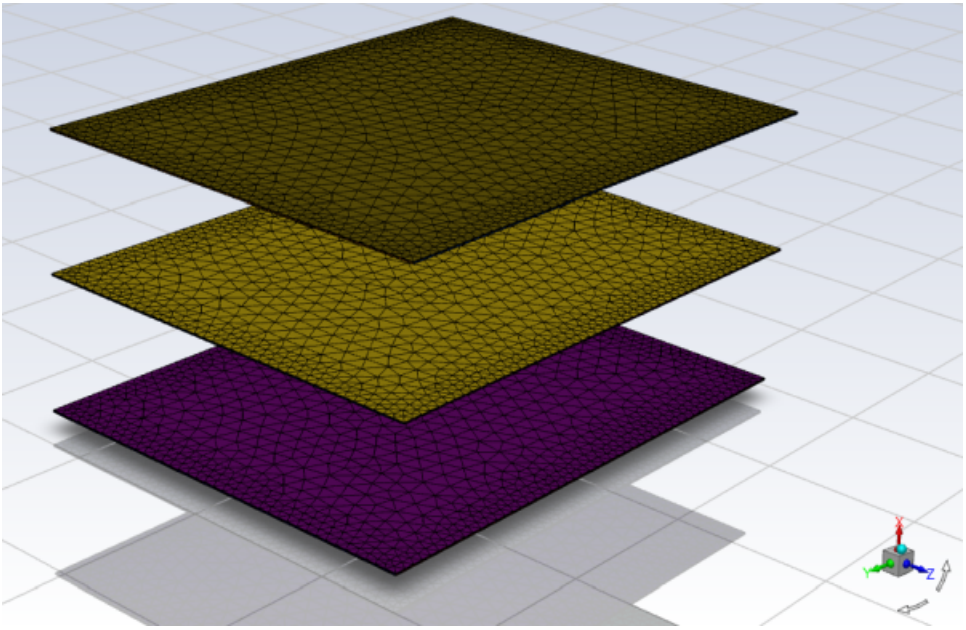
Figure 3.17: Stack of Thin and Thick Plates



In this example, the **Select Parallel Zones** option can be utilized to create a thin volume mesh for all thin plates within the stack, which are shown in Figure 3.18: Thin Plates (p. 337).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 3.18: Thin Plates

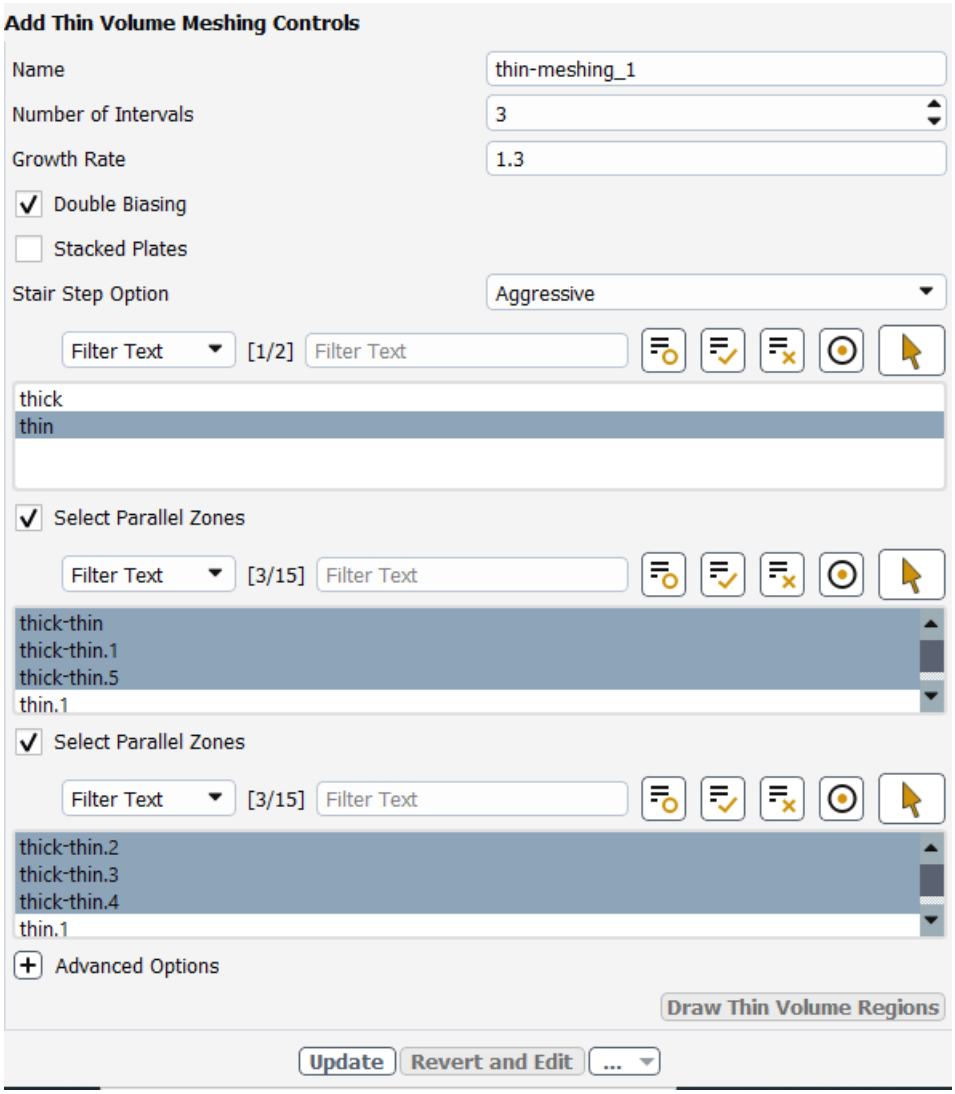


To mesh the thin plates, the **thin** region is first selected, then the **Select Parallel Zones** option is enabled for the **Source**. For the source surfaces, the top surface (facing the positive x-axis) of one of the three plates is selected, which will automatically select the top surface of all three thin plates.

After selecting the source surfaces, the **Target** surfaces are enabled by following a similar procedure. First the **Select Parallel Zones** option is enabled, then the bottom surface (facing the negative x-axis) of one of the three plates is selected, which will automatically select the bottom surface of all three thin plates as the target.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 3.19: Thin Volume Meshing Controls for Parallel Zones



3.12.2. Thin Volume Meshing of Stacked Plates

For models consisting of stacked planar plates, you have the ability to select all source-target zones that are aligned with the global x-y-z plane by enabling the **Stacked Plates** option. Note that for thin meshing of stacked plates, only the region(s) and the source face(s) are selected which will automatically specify the source-target zones.

When stacked plates is enabled, thin volume meshing controls can be defined as follows:

1. Specify the sizing of the mesh layers to be based on **Intervals** or based on the **Size** of the plates using the **Assign Size Using** option.

   a. When **Intervals** is selected as the **Assign Size Using** option, specify the number of mesh layers to be created within the thin volume mesh for **Number of Intervals**.

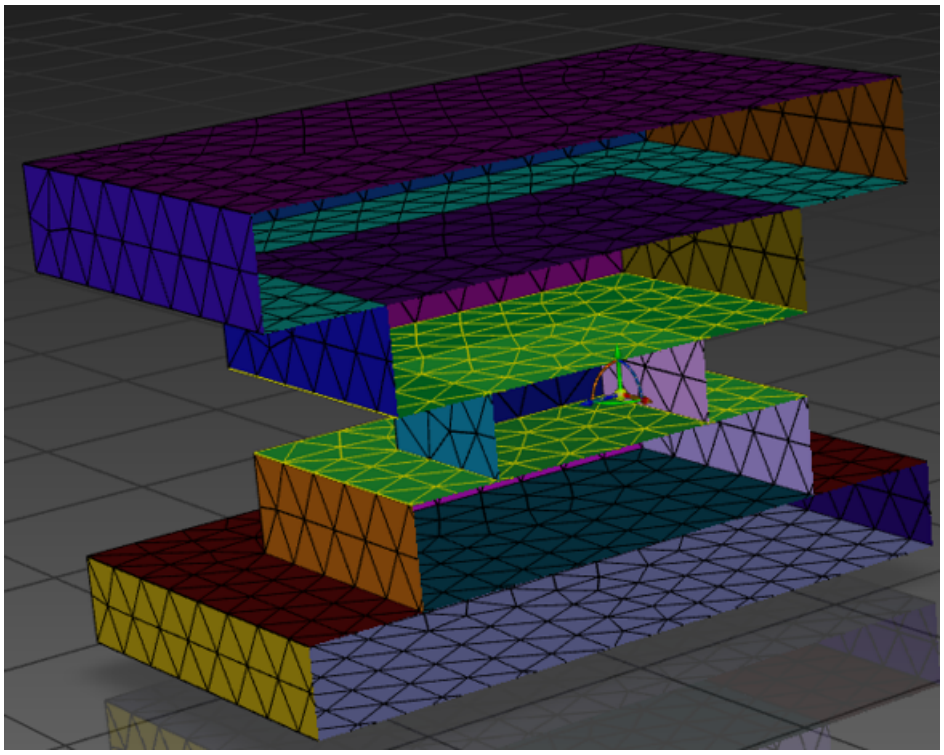
   b. When **Size** is selected for the **Assign Size Using** option, specify the **Minimum Number of Intervals** desired for the thin mesh and enter the **Size** of each layer.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Select the stacked plates region(s) where the thin volume mesh will be created.

3. When selecting the source-target zones, you must find the key source face(s) on regions that are imprinted into all selected regions. In general, the key source face will be the plane(s) located within the center of the stack. Figure 3.20: Key Source Faces (p. 339) below shows the 4 key source faces for a stacked plates model.

   Figure 3.20: Key Source Faces

   In the above figure, there are 4 possible key source faces enabled for demonstration purposes. However, only one of the 4 key source faces needs to be specified in order to properly define the source-target zones.

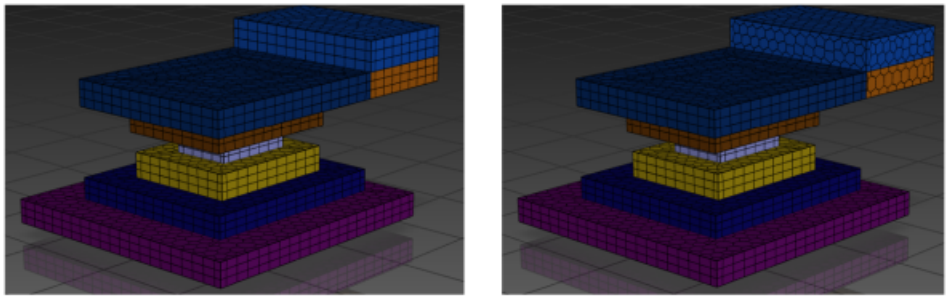
4. After selecting the key source face for your case, it's recommended to click the **Draw Thin Volume Regions** button to ensure that the key source selection will properly select the desired source-target zones.

5. Under **Advanced Options**:

   a. You can use the **Include Co-Planar Regions on All Levels** option to specify whether or not adjacent stacked plate(s) (as shown in Figure 3.21: Stacked Plates with Co-planar (p. 340)) are included in the thin mesh. This option is set to **Yes** by default.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 3.21: Stacked Plates with Co-planar



3.13. Adding Boundary Layers

You can define boundary layers along various regions using the **Add Boundary Layers** task. This task can be added to the workflow as many times as you require. See Prism Meshing Options for Scoped Prisms (p. 775) for more information.

For the **Add Boundary Layers?** field:

- Select **yes** if you need to specify boundary layers using this task.
- Select **no** if you do not need to account for boundary layers. Click **Update** and proceed to the next task.
- Select **yes, using prism control file** if you have a prism control file that contains the definition of the boundary layer, and there are no other boundary layer settings already added to this task. You can use the **Read Prism Control File** field to specify (or browse for) a `.pzmcontrol` file. The corresponding fields in this task will be populated based on the contents of the prism control file.

Note:

– While you can only read a single boundary layer/prism control file into the task, afterwords, you can add any number of boundary layer controls to the task as required.

– Advanced options (such as acute angles, invalid normals, etc.) are not supported if you choose to read a prism control file into this task. In addition, boundary layer controls can be added to the task, however, any conflicts that may occur between controls that are read in from a file and controls that are added to the task are not checked by the system.

– Prism control files can also be created in the workflow as part of the **Generate the Volume Mesh** task (see Generating the Volume Mesh (p. 351)).

---

1. Specify a **Name** for the boundary layer, or use the default value. Note that the default name is dependent on the value of the **Offset Method Type**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Choose an **Offset Method Type**. The offset method that you choose determines how the mesh cells closest to the boundary are generated. See Offset Distances (p. 765) for more information. Choices include:

   - **aspect-ratio**: allows you to control the aspect ratio of the boundary layer cells (or prism cells) that are extruded from the base boundary zone. The aspect ratio is defined as the ratio of the prism base length to the prism layer height.
   - **uniform**: allows you to generate every new node (child) to be initially the same distance away from its parent node (that is, the corresponding node on the previous layer, from which the direction vector is pointing).
   - **last-ratio**: allows you to control the aspect ratio of the boundary layer cells (or prism cells) that are extruded from the base boundary zone.
   - **smooth-transition**: allows you to use the local tetrahedral element size to compute each local initial height and total height so that the rate of volume change is smooth. Each triangle that is being inflated will have an initial height that is computed with respect to its area, averaged at the nodes. This means that for a uniform mesh, the initial heights will be roughly the same, while for a varying mesh, the initial heights will vary

   The following are additional fields that can be set, depending on the value of the **Offset Method Type**.

   i. If the **Offset Method Type** is set to **smooth-transition** or **last-ratio**, specify the **Transition Ratio**. This value determines the ratio of the height of the last layer in the inflation and the first cell in the volume fill.

   ii. If the **Offset Method Type** is set to **last-ratio**, specify **First Height** for the height of the first layer of cells in the boundary layer.

   iii. If the **Offset Method Type** is set to **aspect-ratio**, specify the **First Aspect Ratio**. You can control the heights of the inflation layers by defining the aspect ratio of the inflations that are extruded from the inflation base. The aspect ratio is defined as the ratio of the local inflation base size to the inflation layer height. The value for the **First Aspect Ratio** allows you to specify the first aspect ratio to be used.

   iv. If the **Offset Method Type** is *not* set to **last-ratio**, provide the **Growth Rate**. This value determines the relative thickness of adjacent inflation layers. As you move away from the face to which the inflation control is applied, each successive layer is approximately one growth rate factor thicker than the previous one.

3. Specify the **Number of Layers**. This value determines the maximum number of boundary layers to be created in the mesh.

4. For the **Add in** field, specify what regions you would like to add the boundary layers.

   - Use the **fluid-regions** option to add the boundary layer to just the fluid regions.
   - Use the **solid-regions** option to add the boundary layer to just the solid regions.
   - Use the **named-regions** option to select from a list of available named region(s) in your simulation.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For imported CAD geometries that include body labels (and if you have set the **Use Body Labels** field to **Yes** in the **Import Geometry** task, see Importing Geometries (p. 298)), body label names are also listed alongside named regions in the selection list.

5. For the **Grow on** field, specify where you would like to develop the boundary layers.

   - Use the **only-walls** option to grow the boundary layer along just wall surfaces.
   - Use the **all-zones** option to grow the boundary layer along all surfaces or zones.
   - Use the **solid-fluid-interface** option to grow the boundary layer along just interface of the fluid and solid region.
   - Use the **selected-zones** option to select from the **Zones** list of available named zone(s) in your geometry.
   - Use the **selected-labels** option to select from the **Labels** list of available named label(s) in your geometry.
   - For the **selected-zones** and **selected-labels** options, select items in the list, or use the **Filter Text** option in the drop-down to provide text and/or regular expressions in filtering the list (for example, using *, ?, and []). You can also choose the **Use Wildcard** option in the drop-down to provide wildcard expressions in filtering the list. When you use either ? or * in your expression, the matching list item(s) are automatically selected in the list. Use ^, |, and & in your expression to indicate boolean operations for NOT, OR, and AND, respectively. See Filtering Lists and Using Wildcards (p. 287) for more information.

6. Click **Advanced Options** to access additional controls prior to performing this task.

   - Use the **Ignore Boundary Layers at Acute Angles?** option to choose whether to automatically ignore boundary layers where there are acute angles.

     Note that if there are sharp angles adjacent to other regions with boundary layers, some boundary layer removal may occur in those adjacent regions.

     If you indicated **yes** to the **Ignore Boundary Layers at Acute Angles?** prompt, you can use the **Acute Angle Buffer Layers** option to indicate the number of buffer layers that can be placed around ignored boundary layer faces, extending the ignored regions around sharp angles. Increasing the **Acute Angle Buffer Layers** value increases the number of faces for which the boundary layer will be ignored at acute angles.

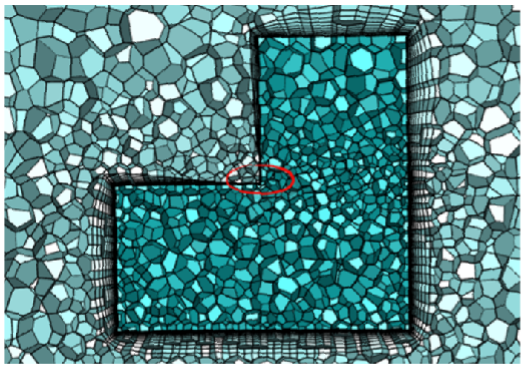
   - Use the **Modify Surface Mesh at Invalid Normals?** option to choose whether to automatically change the surface mesh where invalid normal faces are detected.

     Note:

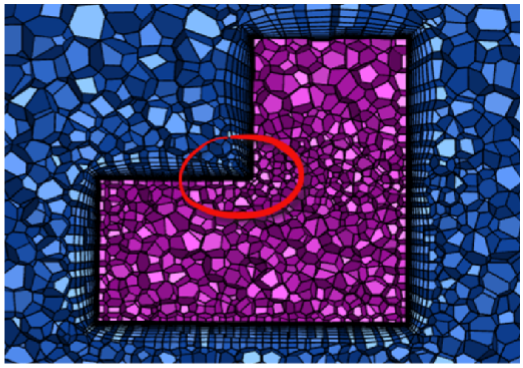
     This option is not supported when working with non-conformal meshes, that is, when the non-conformal prompt is enabled in the **Describe Geometry** task (Describing the Geometry (p. 318)).

     To grow the boundary layer mesh in the proper direction (away from the boundary), normal vectors (valid) are required at the boundary face nodes of the surface mesh.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

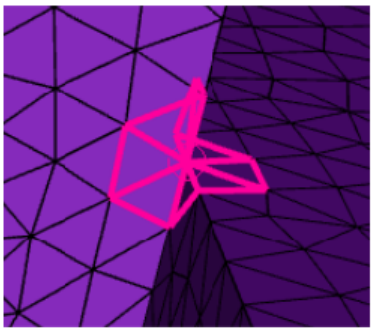


Improper Boundary Layer Growth
(Invalid Normals)

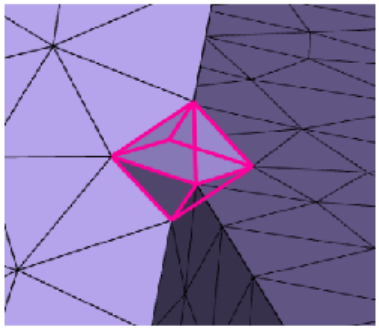


Proper Boundary Layer Growth
(Valid Normals)

In some cases, the normal vector cannot be determined using the normals of all surrounding face normals (invalid), and the surface mesh must be modified by introducing an octahedral dead region where a normal vector can then be evaluated.



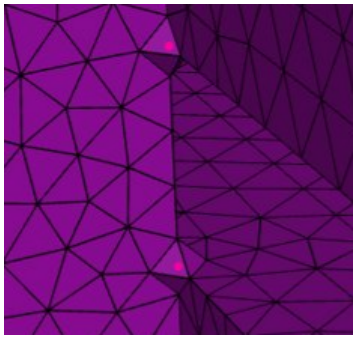
Unmodified Surface Mesh
(Invalid Normal)



Modified Surface Mesh
(Valid Normal)

Such dead regions will subsequently be visible in the **Update Regions** task upon generation of the volume mesh.

Invalid normals are indicated in the graphics window using spheres.



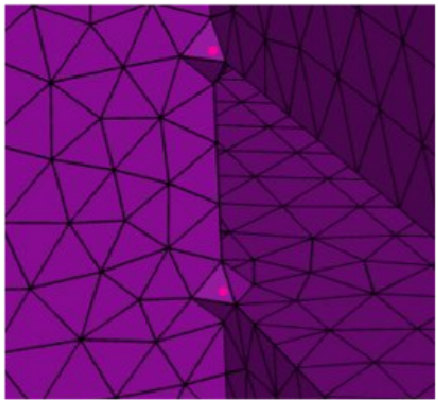
When you specify **yes** for the **Modify Surface Mesh at Invalid Normals?** prompt, the following options are available to you when working with invalid normals:

– Use the **Sphere Radius Factor at Invalid Normals** option to determine the size of the spheres created at the invalid normal location. The spheres have a radius of one half of the average

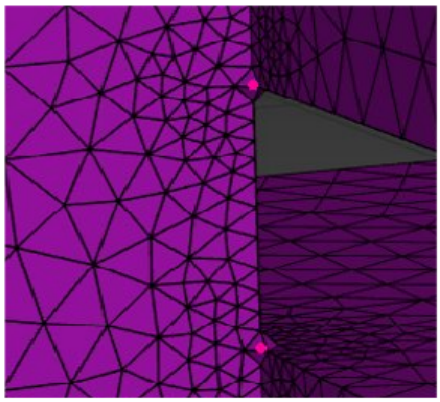
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

mesh edge length adjacent to the invalid location. The default factor of 0.8 ensures a sphere size that is slightly smaller than that.

- Use the **Allowed Tangency at Invalid Normals** option to control the tangency of the invalid normal faces. An "invalid normal" location with all 90 degree angles has a tangency of 1, so faces are still treated as an invalid normal even if the angle deviates slightly from 90 degrees (resulting in a tangency of 0.98).
- Use the **Remesh at Invalid Normals?** option to determine whether or not to remesh around the area of the invalid normal(s). When enabled, a local (non-size function-based) remeshing procedure is performed to obtain similar sizes around the sphere with a reasonable transition to the surrounding mesh.



Remesh at Invalid Normal? = no



Remesh at Invalid Normal? = yes

Note:

When verbosity is enabled, the x, y, and z locations of each invalid normal are printed in the console

Additional, global boundary layer settings are available in the **Generate the Volume Mesh** task (Generating the Volume Mesh (p. 351)).

7. (optional) Use the **Draw Regions** button to visually display all regions, or just the fluid region(s).
8. Click **Add Boundary Layers** to generate the appropriate boundary layers for the imported CAD geometry.

   If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

3.14. Adding Multizone Controls

In order to create a pure (hexagonal) mesh for complicated geometric configurations, you might at times be forced to manually de-construct, or slice, the geometry into separate bodies. A multi-zonal meshing approach ("multizone"), however, can let you create a purely hexagonal mesh throughout all regions of a geometry, automatically deconstructing the geometry into separate bodies. See Understanding Multizone Meshing (p. 348) for more information and details about using multizone meshing.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Use the **Add Multizone Controls** task in your workflow to set up your own multi-zonal-based controls for the specific regions in your geometric model.

1. For the **Multizone Controls and Sizing for** option, select either **Regions** and **Edges** so that the multizone controls are generated based on region or edge selection. You can also choose **Body Labels** for imported CAD geometries that include body labels (and if you have set the **Use Body Labels** field to **Yes** in the **Import Geometry** task, see Importing Geometries (p. 298)).

   Note:

   Edges can be defined in either of the following:

   - In the CAD model, such that the imported geometry should contain one or more *named* edges (for example, an ANSYS SpaceClaim Design Modeler CAD geometry with one or more edges explicitly using named selections).
   - Using either the **Extracted Edges** field or the **Split Cylinders** field in the **Manage Zones** task (see Managing Zones (p. 379)). Note that this option is not available when **Separate Zone By** is set to **face** in the **Import Geometry** task (see Importing Geometries (p. 298)).

   Before assigning any edge zone-based multizone controls, you need to first define at least one region-based multizone control.

   In addition, since selecting the correct face zones is necessary for proper extraction, if you select multiple zones that share at least one edge, you are prompted to select a minimal set of edges in order to avoid the improper extraction at the location of the shared edges (particularly at T-connections).

   When **Body Labels** is selected for the multizone control, body label names are also listed in the selection list.

2. For region-based multizone controls, perform the following steps:

   a. Specify the **Name** for the control, or keep the default value.

   b. Select a choice for the **Mesh Method** field, or keep the default value. Choices include:

      - **Standard** technique.
      - **Thin** volume technique (for only a single layer).

      Both of these methods support the same region input (fixed sizing or size control functions, etc.)

   c. Select a choice for the **Fill With** field, or keep the default value. Choices include

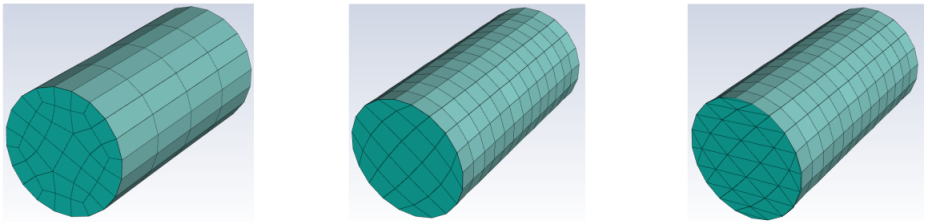
      - **Hex-Pave**: fills the selected region(s) with hexahedra elements. All connected regions must use the same volume fill type. If they are set differently, the region fill types will be automatically set to use this fill type.
      - **Hex-Map**: fills the selected region(s) with hexahedra elements.
      - **Prism**: fills the selected region(s) with prism elements

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Mixed**: fills the selected region(s) with hexahedra elements in addition to some prism elements.

Note that **Hex-Pave** can be mixed with **Hex-Map** within connected regions.

Figure 3.22: Examples of Fill Types



d. Select a choice for the **Use Fixed Sweep Size?** field, or keep the default value of **no** where a variable sweep size (based on size functions) is applied to the multizone mesh. Setting the **Use Fixed Sweep Size?** field to **yes** allows you to specify a fixed value for the multizone's sweep size using the **Max Sweep Size** field.

Note:

Any connected multizone control-enabled regions must use the same setting for the **Use Fixed Sweep Size?** field

---

e. Select a region to apply the multizone mesh controls.

f. Make a **Source-Target Selection** using the corresponding list (of either **Labels** or **Zones**). This allows you to choose your sources and targets manually for the multizone meshing. You can also provide the ability to select all source-target zones that are parallel to a global plane by choosing **Zones parallel to XY plane**, **Zones parallel to XZ plane**, or **Zones parallel to YZ plane**.

When selecting **Zones**, and when your desired zones are aligned with the global x,y, or z plane, use the **Select Parallel Zones** checkbox to automatically select all parallel zones in the selected region(s).

Note:

You can manually select the regions or source-target selection within the corresponding list, or you can select them using the graphics window by first clicking the button, then selecting the regions or surfaces in the graphics window which automatically selects the corresponding regions/surfaces in the list. After clicking the button, the button will become highlighted ( ), indicating that graphical selection is enabled. Note that if multiple regions/surfaces have been selected, only the last region/surface selected in the list can be de-selected using the graphics window. Additionally,

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

when the **Select Parallel Zones** option is enabled and multiple edges have been selected, graphical de-selection is not supported.

When manually selecting items in the list, you can use the **Filter Text** option in the drop-down to provide text and/or regular expressions in filtering the list (for example, using *, ?, and []). You can also choose the **Use Wildcard** option in the drop-down to provide wildcard expressions in filtering the lists. When you use either ? or * in your expression, the matching list item(s) are automatically selected in the list. Use ^, |, and & in your expression to indicate boolean operations for NOT, OR, and AND, respectively. See Filtering Lists and Using Wildcards (p. 287) for more information.

3. For edge-based multizone controls, perform the following steps:

   a. Specify the **Name** for the control, or keep the default value.

   b. For the **Assign Size Using** field, indicate whether you want to assign the edge size control based on **Interval**, **Size** (the default), or **Smallest Height**.

      Depending on your selection, you will need to provide a value for the **Number of Intervals**, the minimum **Size**, or the **Smallest Height** for the edge-based multizone control, accordingly.

   c. Select a **Growth Pattern**, where the choices provide a visual indication of the pattern of growth for the volume mesh extrusion.

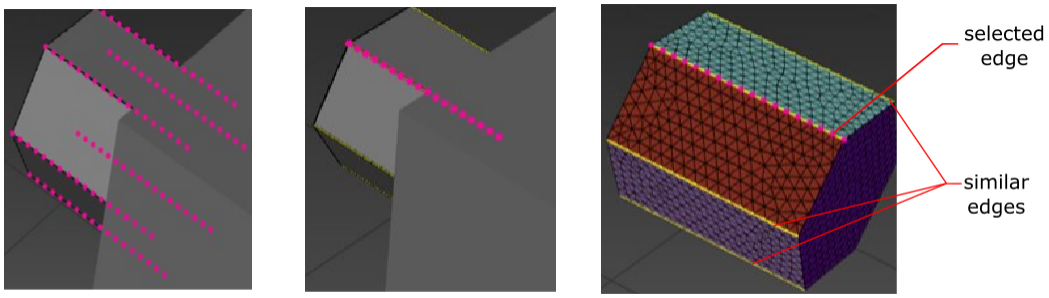
      For edge size controls, and when using the variable **Growth Pattern**s, you can also determine how the growth is to be determined using the **Assign Growth Using** field, where you can choose to provide a **Growth Rate** or a **Bias Factor**. The **Bias Factor** is the ratio of the largest to the smallest segment on the edge.

   d. When your desired edges are aligned with the global x,y, or z plane, use the **Select Parallel Edges** checkbox to automatically select all parallel edges in the selected region(s).

   e. If edge labels are automatically created on all edges, preserving the face/edge topology, you can use the **Show edge biasing?** field to determine if you want to save time and preview any edge biasing, since when many edges are selected, there can be many nodes and biases that can take additional time.

      - Select **yes, selected** (the default in new cases) to limit the preview to only the explicitly selected edges. The selection can be either from the list of edges, or directly from the graphics window. Any remaining edges that are similar to those selected are highlighted in yellow.
      - Select **yes, all** (the default for existing models) to have the preview contain all edges.
      - Select **no** to not preview any edge biasing.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



all edges selected | selected edge | selected edge with similar edges

Note:

- If you are defining edge-based named selection in your CAD model in order to use edge-based multizone meshing, you must separate the face zones by an angle that results in a face zone boundary along each of the defined edges in the **Generate Surface Mesh** task (see Generating the Surface Mesh (p. 307)).
- If a connected multizone control-enabled region contains both sweep size and edge-based sizing, the sizing that results in the highest cell count will be honored.
- In some cases, particularly using edge size controls based on the **Interval** or the **Smallest Height**, there will be some differences between the mesh preview and the mesh itself. This is due to Fluent's use of simplified algorithms in meshing mode to ensure optimal performance.

4. Click **Add Multizone Controls** to create and add the specified multizone control to your workflow. Your can re-use the task to create as many separate multizone controls as you require.

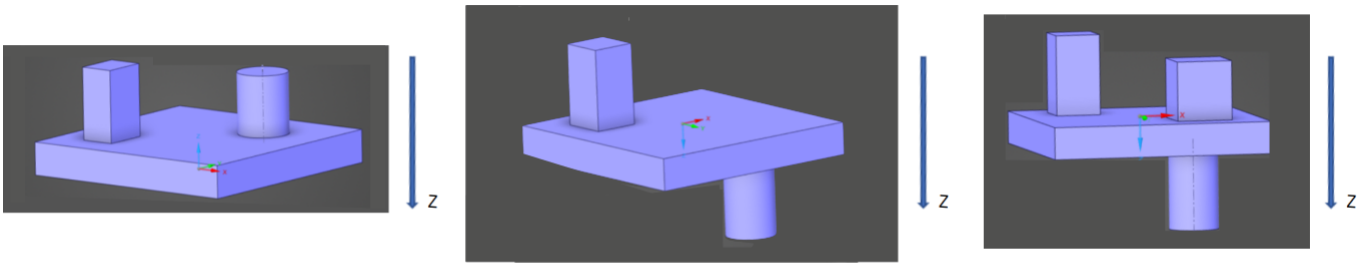
   If you need to make adjustments to any of your settings in this task, click **Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

3.14.1. Understanding Multizone Meshing

Essentially, the concept of "multizone meshing" is equivalent to generating a mesh by performing a “sweep” along a particular direction, often involving multiple bodies. The technique requires a number of source and target faces that are normal to the sweep direction and uses them to create hexahedra (or hexahedra-dominant) meshes.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 3.23: Sample Geometries



(1) (2) (3)

If you examine Figure 3.23: Sample Geometries (p. 349), the first illustration (1) represents a single body, where the "sweep-able" direction (Z) is from top to bottom. You would need to provide all faces normal to that direction (that is, the top two faces, and both large faces on each side of the block) as the source/ target selections. The multizone technique would also apply even if the geometry represented three connected bodies (using share topology), where, in that case, additional shared faces would need to be added to the source-target selection. Likewise, the multizone technique can also be applied to illustration (2) using essentially the same selections.

Illustration (3) however, represents a scenario where the multizone technique would not be easily applicable. This is due to the fact that the upper right-hand block and the lower right-hand cylinder overlap in the same plane (as you view the model from above or below in the sweep direction). A non-conformal connection between the cylinder and the plate are required for the multizone method to work in this situation.

3.14.2. Strategies for Using Multizone Meshing

There are some additional requirements or recommendations that need to be met for a successful and high quality mesh using the multizone technique.

1. All zones need to be separated by angle. There are two options available to do this:

   a. In the **Generate Surface Mesh** task, set **Separate Out Boundary Zones by Angle?** to **Yes**. The zones would be re-merged back together once you generate the volume mesh.

   b. Insert a **Manage Zones** task before the **Add Multizone Control** task, and use the default separation to separate out the bodies that will be meshed for multizone meshing. These bodies will also be merged back together once you generate the volume mesh. Note that this option is not available when **Separate Zone By** is set to **face** in the **Import Geometry** task (see Importing Geometries (p. 298)).

2. When you are using multizone meshing on very thin plates (for example, thinner than the minimum size), the improve surface mesh operations in the **Generate Surface Mesh** task can inadvertently damage the geometry. There are two options available to avoid this:

   a. In the **Generate Surface Mesh** task under **Advanced Options**, set the **Invoke Quality Improve?** prompt to **No**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

   b. If you have regions that are not being meshed using multizone, and for which surface quality improvements are necessary, insert an **Improve Surface Mesh** task directly after the **Add Multizone Control** task.

3. If you experience multizone meshes where the mesh on cylinders appears to be "twisted", insert a **Manage Zones** task before the **Add Multizone Control** task and use the **Split Cylinder** operation to split the cylinders, which will reduce the twist. Note that this option is not available when **Separate Zone By** is set to **face** in the **Import Geometry** task (see Importing Geometries (p. 298)).

4. If you are applying multizone meshing to thin plates, the **Thin** mesh method option may be more stable and faster than the **Standard** option.

5. With nonconformal multizone meshing, **interface** zone types are automatically created on the non-conformal interface boundaries. This can be an issue when adding mesh boundary layers along only walls (that is, when you set the **Grow on** option to **only-walls** in the **Add Boundary Layer** task).

3.14.3. Working With Boundary Layers in Multizone Meshing

Boundary layers in multizone meshing are not as "flexible" as the boundary layers with unstructured meshes since they do not include stair-stepping and gap factors and local shrinkage is not predictable. When working with boundary layers, try the following strategies:

1. Start by testing without boundary layers to ensure that it works

2. Start with uniform boundary layers, as boundary layer shrinkage is easily predictable

3. If the default boundary layer option (**only-walls**) proves unsatisfactory, try the **all-zones** option instead.

3.15. Generating the Multizone Mesh

Once you have set up and added your multizone controls (Adding Multizone Controls (p. 344)), you can generate a computational mesh for your multizone(s) using the **Generate the Multizone Mesh** task.

1. Specify the **Quality Improve Limit**, or keep the default value. This sets the threshold for when mesh quality improvements are automatically invoked that employ the orthogonal quality limit, and is recommended to be around 0.04. Cells below this limit will automatically be displayed.

2. Select your region(s) for which multizone controls have been created and where you want to generate the multizone mesh.

   For imported CAD geometries that include body labels (and if you have set the **Use Body Labels** field to **Yes** in the **Import Geometry** task, see Importing Geometries (p. 298)), body label names are also listed alongside named regions in the selection list.

   Select items in the list, or use the **Filter Text** option in the drop-down to provide text and/or regular expressions in filtering the list (for example, using *, ?, and []). You can also choose the **Use Wildcard** option in the drop-down to provide wildcard expressions in filtering the list. When you use either ? or * in your expression, the matching list item(s) are automatically selected in the list. Use ^, |, and & in your expression to indicate boolean operations for NOT, OR, and AND, respectively. See Filtering Lists and Using Wildcards (p. 287) for more information.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Use the **Non-Conformal Mesh between Multizone and the rest?** prompt to optionally specify that multizone regions are non-conformally connected to other volumetric regions. If you want to have a conformal mesh but, because of meshing constraints, that is not possible, then you can switch to non-conformal here and avoid doing so in the CAD model.

   This option is not available when your geometry consists of a body inside another body where the bodies are not touching each other.

4. Use the **Size Function Scale Factor** option to enable the scaling of the multizone mesh. In some cases when the multizone region is too coarse when compared to the adjacent surface mesh, a connection is not possible. You can specify a size function scaling factor here to improve the sizing match between the multizone and the non-multizone regions and avoid any free faces. Typically, a value between 0.7 and 0.8 is recommended. This field is not available when using non-conformal meshes.

5. Click **Generate the Multizone Mesh** to generate a volume mesh for the specified multizone(s). Connected regions will be meshed simultaneously. If all regions have multizone controls, and all are selected, then this task removes and takes the place of the **Generate the Volume Mesh** task in the workflow.

   If you need to make adjustments to any of your settings in this task, click **Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

6. Use the **Draw Mesh** button to display the fluid and/or solid meshes.

Note:

For more information and some helpful suggestions when using multizone meshing, see Strategies for Using Multizone Meshing (p. 349) and Working With Boundary Layers in Multizone Meshing (p. 350).

3.16. Generating the Volume Mesh

You can generate a computational mesh for your fluid volume(s) using the **Generate the Volume Mesh** task. In many cases, the default values will be sufficient.

1. Choose the **Solver** for which you wish to generate the volume mesh. The default is **Fluent**, however you can also choose **CFX**.

Note:

When **CFX** is chosen as the target **Solver**, the following settings in this task are changed to ensure a compatible volume mesh:

- For **Volume Fill**, you can only choose **tetrahedral** and **hexcore**.
- When using **hexcore**, then **Avoid 1/8 octree transition** is set to **yes** by default.
- Sets the **Use default stair-step handling?** prompt to **No, Exclude both checks**, essentially keeping stair-step handling to a minimum.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Choose the type of **Fill With** that you require. Available options are:
   - tetrahedral
   - hexcore
   - polyhedra
   - poly-hexcore
3. Indicate whether to **Mesh Fluid Regions** or not. This is enabled by default, and can be enabled along with the **Mesh Solid Regions** option, however, both options cannot be disabled at the same time.
4. Indicate whether to **Mesh Solid Regions** or not. This is enabled by default, and can be enabled along with the **Mesh Fluid Regions** option, however, both options cannot be disabled at the same time.
5. If the **Fill With** method is set to **tetrahedral** or **polyhedra**, specify the **Growth Rate**. This value determines the relative length-based size change of cells from the boundary (or the boundary layer cap) towards the interior of the domain.
6. If the **Fill With** method is set to **hexcore** or **poly-hexcore**, specify the number of **Buffer Layers** and **Peel Layers**. The buffer layers are additional layers of cells to alleviate a rapid transition from finer cells to coarser cells (see Buffer Layers (p. 799) for details). The peel layers are additional layers that control the gap between the hexahedra core and the geometry (see Peel Layers (p. 799) for details).
7. If the **Fill With** method is set to **hexcore** or **poly-hexcore**, specify the **Min Cell Length** field to determine the minimum length of the volume mesh cell.

   Note:

   Clicking in this field displays red boxes in the graphics window, providing a visual representation of the field value. Use the **Clear Preview** button to hide the visualization display.

8. Specify the **Max Cell Length** field to determine the maximum length of the volume mesh cell.

   Note:

   Clicking in this field displays red boxes in the graphics window, providing a visual representation of the field value. Use the **Clear Preview** button to hide the visualization display.

9. If, when generating your surface mesh, you elected to separate zones based on angle (see Generating the Surface Mesh (p. 307)), you will see the **Merge Back the Separated Boundary Zones?** prompt.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Here, you can elect to re-merge the zones prior to creating the volume mesh. By default, this option is set to **No**.

> **Note:**
>
> You should *not* invoke this option if you want to use body labels on multiple bodies to merge cell zones, or if you plan on using the **Manage Zones** task (see Managing Zones (p. 379)).

10. For the **Sizing Method**, choose how the cell sizing will be evaluated.

    - Select **Global** (the default) to assign global sizing controls such as **Growth Rate** and the **Max Cell Length**.
    - Select **Region-based Sizing** to display a table of available regions where you can assign local sizing controls on a per-region basis.

      For imported CAD geometries that include body labels (and if you have set the **Use Body Labels** field to **Yes** in the **Import Geometry** task, see Importing Geometries (p. 298)), body label names are also listed alongside region names in the table (though their names cannot be edited).

      When **Fill With** is set to **tetrahedral** or **polyhedra**, you can specify the **Max Cell Length** and **Growth Rate** for specific region(s), whereas when **Fill With** is set to **hexcore** or **poly-hexcore**, you can specify the **Max Cell Length** for specific region(s).

      > **Note:**
      >
      > A body's target mesh size in the context of local sizing and a region's maximum cell length in the context of volume meshing are not identical. When performing local sizing, the body's target mesh size sets the size on both the boundaries and the interior of the region and should be used to set small sizes within a region. Performing local sizing impacts the surface mesh, and it therefore is performed prior to surface meshing. Setting the region's maximum cell length size only controls the size on the interior of the body and should be used to control the maximum cell length of a region, and has no impact on the surface mesh.
      >
      > A recommended approach would be to provide a face size control on a body (rather than a body size control) and, for the target mesh size, provide the same value for the region's maximum cell length, therefore providing the same result as if you were to perform body sizing.

11. Apply parallel processing of the volume mesh using the **Enable Parallel Meshing** option (enabled by default). This option is available when the **Number of Layers** for the boundary layer is greater than 1, and when the number of parallel processors is greater than 1. The option is applicable for any **Fill With** method, such as **polyhedra**, **poly-hexcore**, **hexcore**, or **tetrahedral**. For information about parallel processing and automatically partitioning the mesh, see Auto Partitioning (p. 741). Disable this option if you are interested in only generating the volume mesh in serial mode.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12. Click **Advanced Options** to access additional controls prior to performing this task. Options include:

- Use the **Quality Method** option to choose from several different types of mesh quality controls (skewness, aspect ratio, change in size, and so on). Choices include **Orthogonal** (the default for the workflows), **Enhanced Orthogonal**, and **Skewness**. For more information, see Quality Measure (p. 566). The quality method chosen here will also be used in the **Improve Volume Mesh** task if you later add that task to the workflow, however, you can still choose a different quality method for that task if desired.

- Use the **Invoke Persistent Renaming** option to allow the volume mesh components to use persistent and unique names for the solver. This will make zone names equivalent to region names, and will make cell and face zone names unique. Using this field is highly recommended for any parametric study.

  Note:

  Persistent renaming only works if all body names are unique.

- If multiple body labels (named selections) are defined in your imported CAD geometry, you can use the **Merge Body Label Bodies** field to determine if you would like to merge such bodies or not upon meshing the volume (the default is **yes**).

  Important:

  Body names between each Body Label should be unique, while inside a Body Label they can be the same.

  If set to **yes**, after volume meshing, the following occurs:

  – All cell zones within a body label will be merged into a single cell zone and take the name of the body label.

  – All face zones adjacent to the merged cell zones will also be merged.

  – Interior face zones between cells will be renamed. For instance, those between "`cell-a`" and "`cell-b`" will be renamed to "`cell-a-cell-b`".

  – All face labels will always be preserved, and automatic renaming does not change the face labels. Face labels that have been separated are also automatically remerged.

  If set to **no**, you can insert the **Manage Zone** task and perform the same merging operations by selecting **Body Labels** as the **Type** and select all body labels in the list. See Managing Zones (p. 379) for more information.

- Use the **Use Size Field?** option to determine whether or not to use size fields as part of generating the volume mesh.

- Use the **Polyhedral Mesh Feature Angle** option to set the angle to preserve features when using a polyhedral-based mesh.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Use the **Avoid 1/8 Octree Transition?** option to determine whether or not you want to avoid any potential 1:8 cell transition in the hexcore region of the volume mesh, replacing any abrupt change in the cell size with polyhedral cells.
- Use the **Fill Polyhedra in Solids?** option to fill only polyhedra cells in all solid regions during volume meshing using the poly-hexcore volume fill method. This option is available when there is at least one solid region and one fluid region. The workflow assumes that the largest volume represents a fluid region such that, when using this option, you achieve a higher quality volume mesh by filling the largest (fluid) region with poly-hexcore cells.
- When using the poly-hexcore volume fill method, use the **Use Size Field in Solids?** option to use size-field-based sizing for polyhedra regions during volume meshing. This setting is available when the **Use Size Field?** and **Fill Polyhedra in Solids?** options are enabled and is recommended when BOI(s) are defined that include polyhedra regions. Size-field based sizing is required for BOI(s) to be respected inside all polyhedra regions.
- Use the **Solid Region Growth Rate** field to control the growth rate of all solid regions in the volume mesh. Note that this field is only available for poly-hexcore volume meshes and when the **Fill Polyhedra in Solids?** option is set to **yes**. By default, the growth rate is set to a value of 1.5.
- Use the **Quality Improve Limit** option to set the threshold for when mesh quality improvements are automatically invoked that employ the orthogonal quality limit.
- Use the **Check Self Proximity** option to determine whether or not you are going to check for any proximity issues (such as overlapping surfaces, very small gaps between surfaces, very sharp angles, etc.) while generating the volume mesh. If set to **yes**, problematic areas will be highlighted in the graphics window.
- If you have added (or read in) any boundary layers to your workflow, you can write out a boundary layer (prism) control file to save their settings to use later. Use the **Write Prism Control File** option to specify whether or not you want to save a boundary layer control file that contains your boundary layer specifications. If you select **yes**, a `.pzmcontrol` file (based on the CAD file name) will be written to your working directory during volume mesh creation. If the file already exists, you will be notified and you can choose to overwrite the file or not.

Note:

Advanced options (such as acute angles, invalid normals, etc.) are not supported and cannot be saved to the prism control file.

---

13. Click **Global Boundary Layer Settings** to access additional boundary layer controls prior to performing this task. These options are only available when you have defined a boundary layer using the **Add Boundary Layer** task (Adding Boundary Layers (p. 340)). Options include:
    - Use the **Merge Boundary Layer Cells Within Regions?** to determine whether or not you want to have the boundary layer mesh merged into the bulk mesh.
    - Use the **Gap Factor** option to specify the relative gap between two boundary layer caps in a narrow channel. A value of 1 indicates a gap that is of the same order as the boundary layer cap triangle size in the inflation layer.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

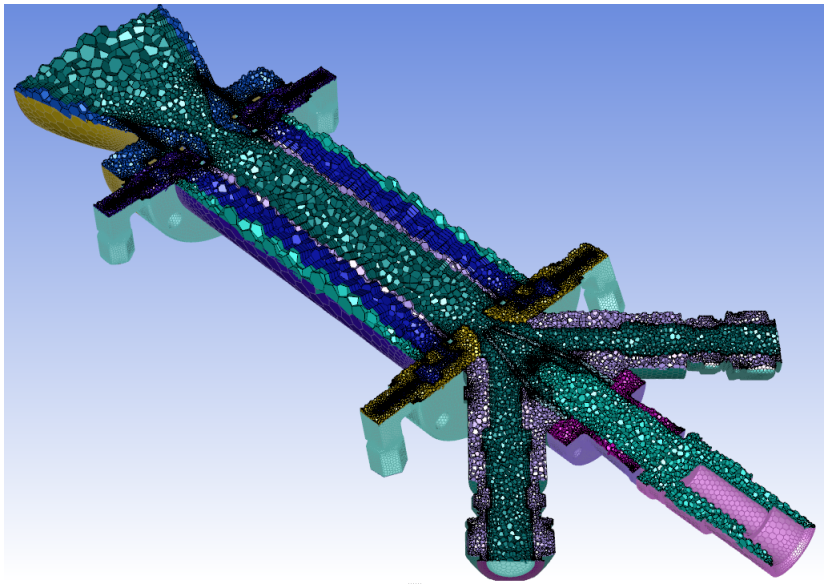
- Use the **Max Aspect Ratio** option to specify the maximum aspect ratio for the boundary layer when proximity compression is applied.
- Use the **Min Aspect Ratio** option to specify the minimum aspect ratio for the boundary layer.
- Use the **Keep First Boundary Layer Height?** option to retain the initial boundary layer's height.
- Use the **Adjacent Attach Angle** option to set the angle for which the boundary layer would imprint on an adjacent boundary.
- Use the **Use default stair-step handling?** option to reduce the stair-stepping at certain locations based on quality or proximity criteria. By default, **Yes** allows you to retain the default stair-step handling, otherwise you can also choose **No, Exclude proximity check**, **No, Exclude quality check** and **No, Exclude both checks**.
- **Stairstep Exposed Quads** can be used when generating a tetrahedral mesh with prism cells and is set to **No** by default. Selecting **Yes** for this option will enable stair-stepping for exposed quadrilateral faces (exposed quads) on prism cells. Stair-stepping these exposed quads can prevent the mesh generation from "hanging" or becoming unresponsive during pyramid creation and can enhance pyramid quality.

14. Click **Generate the Volume Mesh** to generate a volume mesh for the imported CAD geometry.

    If you need to make adjustments to any of your settings in this task, click **Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

15. Use the **Draw Mesh** button to display the fluid and/or solid meshes.

Figure 3.24: Example of a Fluid and a Solid Volume Mesh



Additional enhancements can be made to the volume mesh using the **Improve Volume Mesh** task (see Improving the Volume Mesh (p. 361)). For more information about volume meshing, see Object-Based Volume Meshing (p. 642).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When you are satisfied with the volume mesh, you can proceed to setting up your CFD simulation in Fluent solver mode.

Note:

When using the meshing workflows, after generating the volume mesh, the default quality measure is set to Orthogonal Quality, and will be reported as such when querying the mesh quality.

3.17. Updating Boundaries

You can update the properties of any defined boundary using the **Update Boundaries** task. This task can be added to the workflow as many times as you require.

Note:

This task involves zone assignments based on naming conventions of various boundaries. See Prerequisites for the Fluent Guided Workflows (p. 278) for more detailed information.

1. Choose a **Selection Type** as either by **label** or by **zone**.
2. (optional) Use the **Filter** button to filter the table contents based on a particular column.
3. (optional) Rename any **Boundary Name** by double-clicking the label in the table and entering a new name.

   You can also rename a boundary by selecting it in the table, right-click, and select **Set Boundary Name** in the context menu, and provide a new name directly in the menu.
4. (optional) Re-assign any **Boundary Type** to another value by selecting a type in the table and using the corresponding drop-down menu. Choices include:

   - velocity-inlet
   - pressure-outlet
   - pressure-inlet
   - pressure-far-field
   - mass-flow-inlet
   - mass-flow-outlet
   - outflow
   - symmetry
   - wall
   - internal

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- interface
- overset
- outlet-vent
- intake-fan
- inlet-vent
- exhaust-fan
- fan
- porous-jump
- radiator

Multiple boundaries can be assigned a specific type all at once by selecting them in the table, right-click, and select **Set Boundary Type** in the context menu, then designate a type for the selected boundaries directly in the menu.

Note:

If the **Update Boundaries** task is inserted into the workflow after an **Extrude Volume Mesh** task or a **Transform Volume Mesh** task, then you will need to manually ensure that any internal boundaries are not assigned on external zones (and vice versa).

---

5. (optional) By default, the table only lists external wall boundaries. Select the **List All Boundaries** to see additional boundaries, such as fluid-fluid internal boundaries.

Note:

Toggling the **List All Boundaries** option will discard any changes you may have recently made in the table.

---

6. When you are satisfied with the boundary assignments, click **Update Boundaries**.

   If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

7. Multiple boundaries can be visualized all at once by selecting them in the table, right-click, and select **Draw Selections** in the context menu.

   Use the **Draw Boundaries** button to visualize all boundaries or just wall boundaries.

8. Once all boundaries have been updated, proceed to the next step in the workflow.

Additional boundary types can be created using the **Add Boundary Type** task (see Adding Boundary Types (p. 360)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.18. Improving the Surface Mesh

You can make improvements to your surface mesh using the **Improve Surface Mesh** task. For additional information, see Face Connectivity Issues (p. 626) and Quality Checking (p. 628).

1. Provide a value for the **Face Quality Limit**.

2. Click **Advanced Options** to access additional controls prior to performing this task. Options include:

   - Use the **Quality Method** option to choose from several different types of mesh quality controls (skewness, aspect ratio, change in size, and so on). Choices include **Orthogonal** (the default for the workflows), **Enhanced Orthogonal**, and **Skewness**. For more information, see Quality Measure (p. 566). The quality method specified here is derived from the value set in the **Generate the Volume Mesh** task, however, you are able to make a different selection here if desired.

   - Use the **Quality Improve Max Angle** option to set the maximum angle between the normals of adjacent faces during mesh improvements.

   - Use the **Quality Improve Iterations** option to specify the number of iterations that will be performed to improve the mesh.

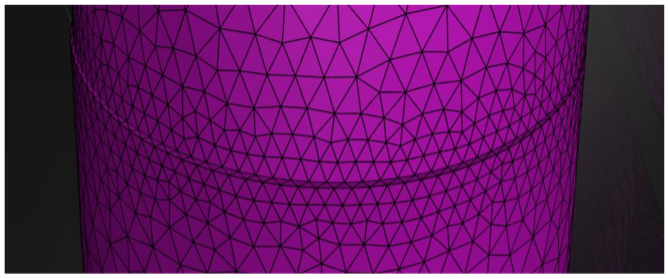
   - Use the **Quality Improve Collapse Skewness Limit** option to specify the limiting skewness value when improving the mesh. After quality improvements using the **Quality Improve Max Angle** are performed, if any remaining triangles are above the **Quality Improve Collapse Skewness Limit**, these will be aggressively removed using a fixed maximum angle of 120 degrees.

     Note:

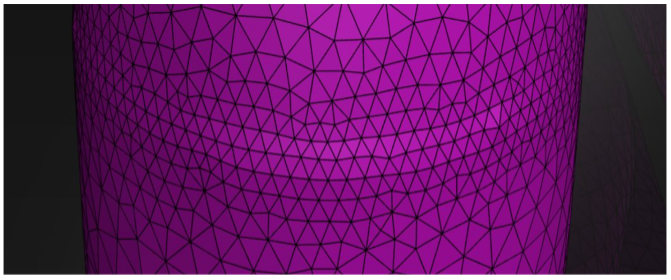
     The default value for the **Quality Improve Collapse Skewness Limit** changes automatically when you increase the value for the **Face Quality Limit**.

3. Use the **Remove Steps?** prompt if there are any small imperfections (ledge or step artifacts from the original CAD geometry) that can be removed as part of the surface meshing improvement task. If you specify **yes**, then you need to indicate the **Max Step Width** and a **Step Skewness Quality Limit** (provided so as not to remove well-refined steps).

   For example, a typical 90 degree step artifact can be removed and replaced with a flattened mesh, removing any artificial pressure loss.



Remove Step? = No:
Step is Retained



Remove Step? = Yes:
Step is Removed

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Step removal only occurs if the step(s) and the adjacent triangles belong to the same face zone or the same label, therefore, there are ways to protect steps from being removed:

- Adding a label to the step itself (or to a face adjacent to the step) will protect the step from being removed.
- Zone separation will protect all steps.

Note:

All steps within a zone (or within a label) are checked and treated together, therefore it is enough that one step in a zone has a label adjacent to it in order to protect all steps.

In other words, steps can only be removed under the following conditions:

- The step, and adjacent faces does not belong to any named selection
- The step, and adjacent faces belongs to the same named selection
- In some cases, you should consider disabling the quality improvement option in the **Generate the Surface Mesh** task so that you obtain a "cleaner" step removal.

4. Click **Improve Surface Mesh**.

   If you need to make adjustments to any of your settings in this task, click **Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

5. Once you are satisfied with your changes, proceed to the next step in the workflow.

Note:

Changing the **Face Quality Limit** too extensively can result in aggressive changes to your surface mesh.

3.19. Adding Boundary Types

You can create additional boundaries for your simulation using the **Add Boundary Type** task. This task can be added to the workflow as many times as you require.

1. Provide a **Name** for the new boundary, or use the default name. The default name changes depending on the assigned **Boundary Type**.
2. Choose the **Boundary Type**. Choices include:
   - velocity-inlet
   - pressure-outlet
   - pressure-inlet

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- pressure-far-field
- mass-flow-inlet
- mass-flow-outlet
- outflow
- symmetry
- wall
- internal

3. Use the list to select an available zone to apply to the boundary.

   Select items in the list, or use the **Filter Text** option in the drop-down to provide text and/or regular expressions in filtering the list (for example, using *, ?, and []). You can also choose the **Use Wildcard** option in the drop-down to provide wildcard expressions in filtering the list. When you use either ? or * in your expression, the matching list item(s) are automatically selected in the list. Use ^, |, and & in your expression to indicate boolean operations for NOT, OR, and AND, respectively. See Filtering Lists and Using Wildcards (p. 287) for more information.

4. Use the **Merge the zones?** field to determine whether the selected zones are to be merged (the default) or not.

5. Click **Add Boundary Type**. The new boundary will appear in the workflow.

   If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

6. Once you are satisfied with your changes, proceed to the next step in the workflow.

3.20. Improving the Volume Mesh

You can make improvements to your volume mesh by using the **Improve Volume Mesh** task.

1. Use the **Quality Method** option to choose from several different types of mesh quality controls (skewness, aspect ratio, change in size, and so on). Choices include **Orthogonal** (the default for the workflows), **Enhanced Orthogonal**, and **Skewness**. For more information, see Quality Measure (p. 566). The quality method selected here by default is the same as that set in the **Generate the Volume Mesh** task.

2. Provide a value for the **Cell Quality Limit**.

3. Click **Advanced Options** to access additional controls prior to performing this task. Options include:

   - Use the **Quality Improve Min Angle** option to set the minimum angle between the normals of adjacent faces during mesh improvements.
   - Use the **Quality Improve Iterations** option to specify the number of iterations that will be performed to improve the mesh.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Use the **Allow Smoothing of Features for Remaining Bad Cells?** option to choose whether or not to ignore geometric features that are related to any bad cells that remain in the volume mesh. This allows you to improve your volume mesh by ignoring any geometric features in the vicinity of bad cells.

4. Click **Improve Volume Mesh**.

   If you need to make adjustments to any of your settings in this task, click **Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

   For more information, see Improving the Mesh (p. 511).

5. Once you are satisfied with your changes, proceed to the next step in the workflow.

3.21. Transforming the Volume Mesh

After you have created a volume mesh, you can add the **Transform Volume Mesh** task to your workflow. This task allows you to create and apply translational or rotational transformations to your volumetric mesh.

Figure 3.25: Example of a Copying and Rotating a Volume Mesh (Periodic)

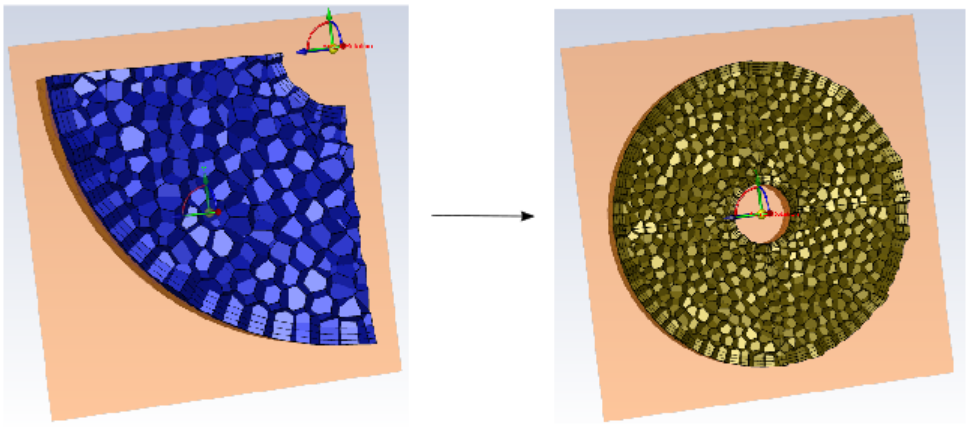
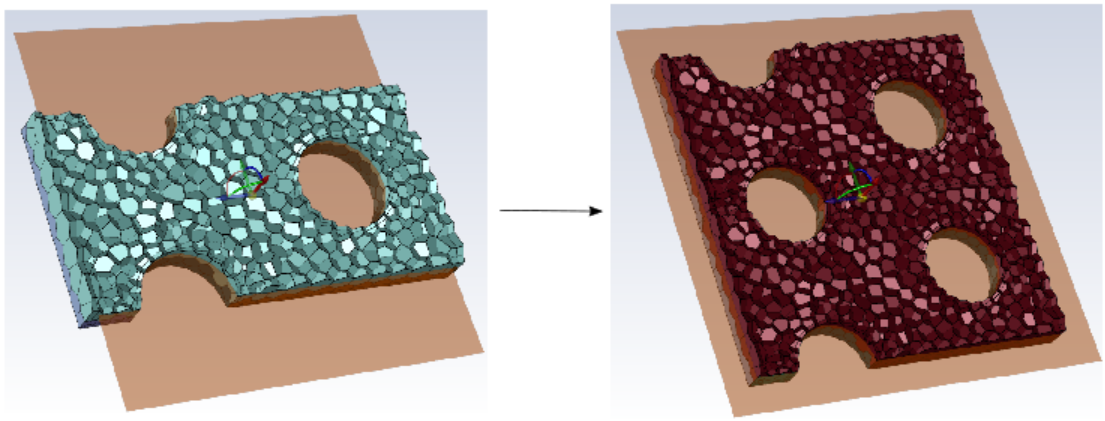


Figure 3.26: Example of Copying and Translating a Volume Mesh (Periodic)



1. Provide a **Name**, or use the default name.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Select the **Type** of transformation: either **Translational** or **Rotational**. The default name changes upon the selected type (`translate_copy_1` for translational transformations, or `rotate_copy_1` for rotational transformations).

3. When periodicity is detected in the model, the **Method** option is available to help you provide additional settings based on periodicity. By default, the **Method** is set to **Automatic - using existing periodicity**. In the case of rotational periodicity, the angle, origin, and vector components are automatically extracted from the stored periodicity. Additionally, the number of copies is automatically set to produce a full 360 degree model. In the case of translational periodicity, the translational shift vector and distance are automatically assigned.

   Note:

   The **Automatic - using existing periodicity** option is not available if there are multiple periodic zones.

   Otherwise, use the **Manual** option to provide your own values to these settings for your rotational or translational transformations.

4. Select a mesh object from the list.

   Use the **Filter Text** option in the drop-down to provide text and/or regular expressions in filtering the list (for example, using *, ?, and []). You can also choose the **Use Wildcard** option in the drop-down to provide wildcard expressions in filtering the list. When you use either ? or * in your expression, the matching list item(s) are automatically selected in the list. Use ^, |, and & in your expression to indicate boolean operations for NOT, OR, and AND, respectively. See Filtering Lists and Using Wildcards (p. 287) for more information.

5. For **Translational** transformations, specify values for the **Translational Shift** in the **X**, **Y**, and **Z** directions.

6. For **Rotational** transformations, specify values for the **Angle** as well as the **Rotation Axis Origin** and **Rotation Axis Direction** in the **X**, **Y**, and **Z** directions.

7. For the **Create a Copy and Translate?** prompt:

   - Choose **yes** if you want to make a copy of the volume mesh and apply the translation to the copy. If this option is selected, you can make the appropriate selection for the **Number of Copies?** prompt.

   - Choose **no, only translate** to apply the volume mesh translation directly to the original volume mesh.

8. For the **Create a Copy and Rotate?** prompt:

   - Choose **yes** if you want to make a copy of the volume mesh and apply the rotation to the copy. If this option is selected, you can make the appropriate selection for the **Number of Copies?** prompt.

   - Choose **no, only rotate** to apply the volume mesh rotation directly to the original volume mesh.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9. When periodicity is detected, and your are automatically using the model's periodic settings, use the **Merge the cell and face zones?** prompt to choose whether or not to merge cell and face zones during the volume mesh transformation to avoid any zone duplication.

   Note:

   The **Merge the cell and face zones?** option is not available when the **Manual** option is selected for the **Method**.

10. Use the **Rename cell and face zones?** prompt to choose whether or not to rename the cell and face zones during the volume mesh transformation.

    When set to **yes**, the new cell zone name is created with an underscore ("_") and its copy number appended to the original zone name (e.g., `my_zone_1`, `my_zone_2`, etc.).

    When set to **no**, any copied cell or face zones (e.g., `my_zone`, `my_zone_1`, `my_zone_2`, etc.) will be created with names that use colons (":") and the copy's ID (e.g., `solid`, `solid:<id-1>`, `solid:<id-1>:<id-2>`, etc.).

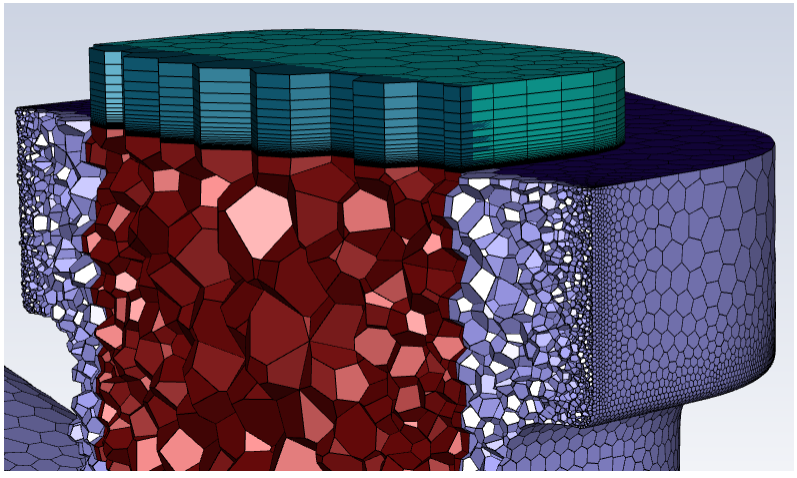
11. Click **Preview Transformation** to see a preview of what the volume mesh transformation will look like in the graphics window.

12. Click **Transform Mesh**. The transformed mesh will appear in the graphics window.

    If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

3.22. Extruding the Volume Mesh

After you have created a volume mesh, you can add the **Extrude Volume Mesh** task to your workflow. This task allows you to extend the volume mesh out beyond the original domain using one or more co-planar boundaries.



1. Provide a **Name**, or use the default name.

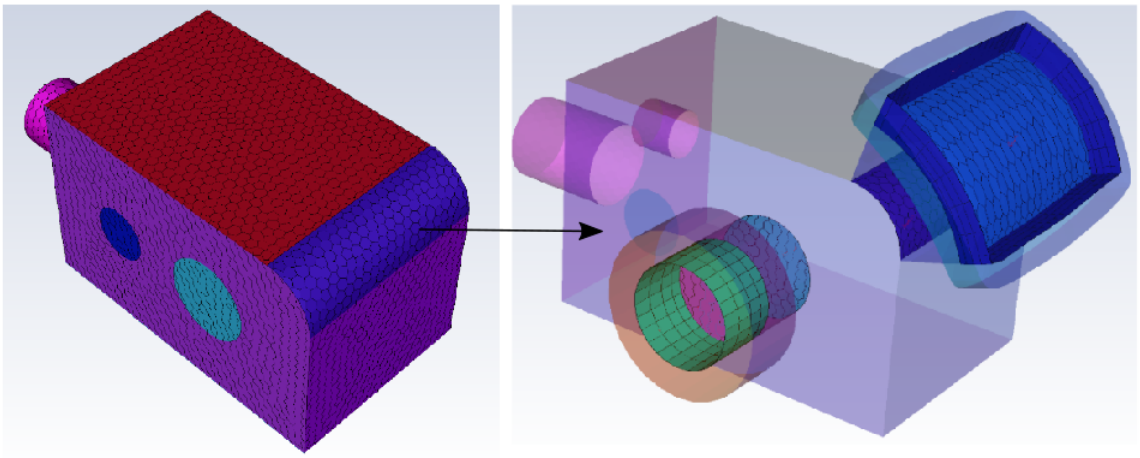
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Select a boundary face from the list. The extrusion will be performed perpendicularly to the selected boundary, so if you select multiple boundaries, they all must be co-planar. A preview box appears in the graphics window, the size of which is based on the values you provide in the task.

   Note:

   The preview box is available only for selected faces whose normal is aligned with the Cartesian coordinate.

   Extrusions can be applied to both planar and non-planar surfaces. For non-planar, curved, surfaces, the extrusion process is performed normal to the selected surfaces.



   Note:

   Periodic or shadow face zones cannot be extruded. If you select such zones for extrusion, Fluent provides a warning and automatically assigns such zones to type `'wall'` so that you can complete the extrusion task.

   Use the **Filter Text** option in the drop-down to provide text and/or regular expressions in filtering the list (for example, using *, ?, and []). You can also choose the **Use Wildcard** option in the drop-down to provide wildcard expressions in filtering the list. When you use either ? or * in your expression, the matching list item(s) are automatically selected in the list. Use ^, |, and & in your expression to indicate boolean operations for NOT, OR, and AND, respectively. See Filtering Lists and Using Wildcards (p. 287) for more information.

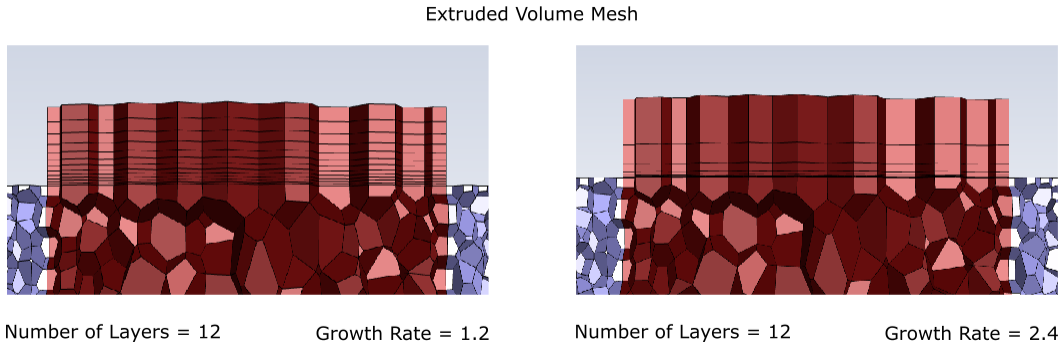
3. Choose an extrusion **Method** based on the specified **Total Height** value (the overall height of the extrusion) or based on the specified **First Height** value (the height of the first layer of the extrusion).

4. Specify whether the volume extrusion is derived from normal-based faceting, or direction-based faceting. When **Normal Based** is enabled (the default), the volume extrusion is based on normal-based faceting, such that for each layer, the normal is calculated and smoothing occurs, and is suitable for non-planar surfaces. For planar surfaces, disable this option to use a direction-based approach where the direction is chosen based on the average normal of the entire surface, and is used to extrude all layers.

5. Specify the **Number of Layers** for the extrusion.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Specify the **Growth Rate** for the extrusion. For example, a growth rate of 1.2 will expand each layer of the extrusion by 20 percent of the previous length.

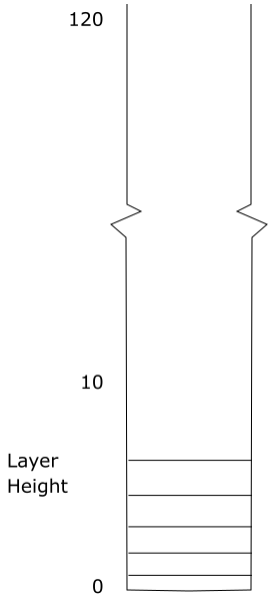


The extrusion process uses these parameter values and geometrically grows the volume mesh according to the following relation:

$$X_N = \sum_{N=2}^{N} x_0 R^{N-1} \tag{3.1}$$

where X_N is the **Total Height** for N layers, x_0 is the **First Height**, R is the **Growth Rate**, and N is the **Number of Layers**.

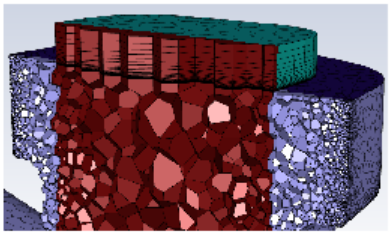
The layer heights for the extrusion are calculated in a geometric fashion, as in:



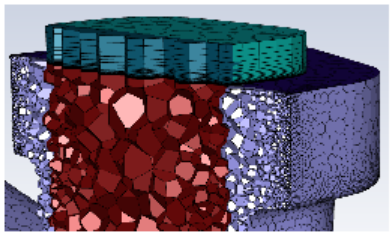
7. Click **Advanced Options** to access additional controls prior to performing this task. Options include:

   - Select a **Growth Pattern**, where the choices provide a visual indication of the pattern of growth for the volume mesh extrusion.
   - Keep the default selection for the **Merge With Adjacent Region** prompt: as needed.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



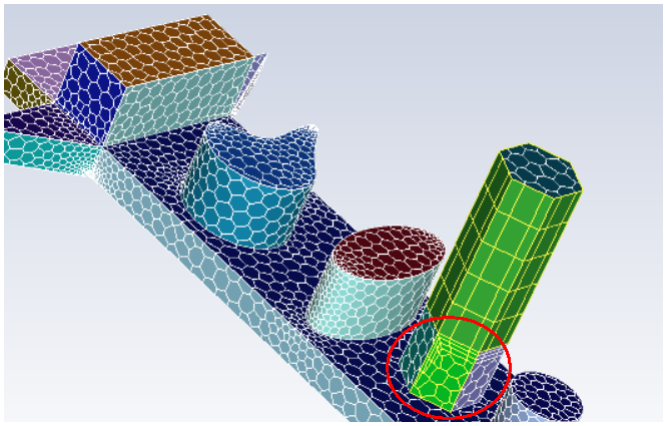
Merge With Adjacent Zones = Yes



Merge With Adjacent Zones = No

Note:

– Volume mesh extrusions performed on shapes other than rectangles and squares (such as triangles, pentagons, hexagons, etc.) will automatically merge all side faces irrespective of the original faces, and irrespective of this property setting.



Fluent splits the side adjacent surfaces only by 90 degrees and merges with the adjacent zones accordingly. For other shapes, however, side faces are not split. As a workaround, you can create a custom task to split the side surfaces for specific cases.

Using the task based display ( ) for this task displays the boundary zones of the mesh along with the adjacent cell zones of the extruded face.

– Extruding cylindrical surfaces too far may create invalid meshes. If your cylindrical surface extrusion exhibits undesirable skewness or otherwise is invalid, reduce the total height of the extrusion and perform the operation again to see if results improve.

– You cannot extrude the volume mesh on zones that are already connected to cells, or when the adjacent face zone is not a structured quadrilateral face zone.

– When **Merge With Adjacent Region** is turned off, and the volume mesh is extruded, Fluent creates the additional cell zones with names based on your extrusion task **Name**. For example, if you create an extrusion task named "my_extrusion_task" on two selected boundaries, then the newly

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

created cell zones will be named "my_extrusion_task-1" and "my_extrusion_task-2" and so forth. A summary of the cell zones is provided upon volume extrusion.

8. Click **Extrude Mesh**. The extruded mesh will appear in the graphics window.

   If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

3.23. Adding Linear Mesh Patterns

Using the **Add Linear Mesh Pattern** task, you can create linear patterns of objects based on one or more CAD parts. This capability can greatly simplify meshing for CAD geometries that require multiple, linearly spaced parts such as in modeling batteries, for example.

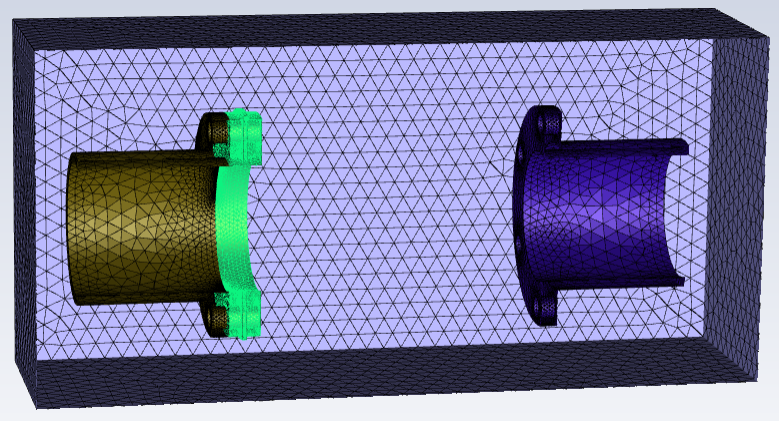
Note:

This task must be added *after* the **Generate the Surface Mesh** task and *before* the **Describe Geometry** task. In certain situations, however, objects that need to be selected to perform mesh pattern operations are merged at the end of surface meshing. To avoid this, the **Add Linear Mesh Pattern** task needs to be inserted into the workflow *prior to executing* the **Generate the Surface Mesh** task.

1. Provide a **Name** for the new mesh pattern, or use the default name.

2. Select the object(s) in the list from which you want to create a mesh pattern. The selected part(s) will be highlighted in the graphics window.

Figure 3.27: Example of a Geometry With Part Selected for Adding a Mesh Pattern



Use the **Filter Text** option in the drop-down to provide text and/or regular expressions in filtering the list (for example, using *, ?, and []). You can also choose the **Use Wildcard** option in the drop-down to provide wildcard expressions in filtering the list. When you use either ? or * in your expression, the matching list item(s) are automatically selected in the list. Use ^, |, and & in your expression to indicate boolean operations for NOT, OR, and AND, respectively. See Filtering Lists and Using Wildcards (p. 287) for more information.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. For the **Estimate Both Axis and Pitch** field, indicate whether or not you want Fluent to provide a best approximation of the pattern's axis direction and the displacement of the pattern's objects (pitch), or whether you want to estimate the **Pitch Only** (default).

   Note:

   This estimation only takes place once, either when the object is selected, or when the option is changed.

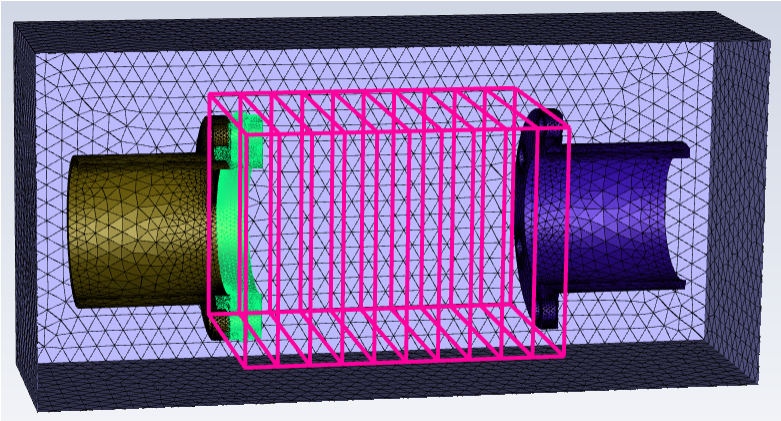
4. For the **Axis Direction**, specify a normalized value for **X-Component**, **Y-Component**, and **Z-Component** of the direction of the pattern's axis.

5. Specify the **Pitch**, or displacement factor, that defines the spacing between the pieces, or *units*, of the pattern.

6. Specify the **Total Number of Units** to indicate the overall number of units in the pattern you wish to create.

7. Click **Preview Pattern** to see a preview of the pattern in the graphics window, based on the specifications provided. The preview will only consist of an outline of the proposed number of units, their axis direction and their spacing. If necessary, make adjustments to the pattern axis definition, pitch, etc., and click the **Preview Pattern** button again as needed.

   Figure 3.28: Example of a Geometry With a Preview of the Mesh Pattern

8. Use the **Highlight Unit Alignment?** field to add graphical highlights to the interfaces between the pattern units so that you can more easily inspect the final pattern for proper alignment. The highlights are visible once the mesh pattern is added. Misaligned units can cause a failure in the share topology of the battery cells.

9. For the **Invoke Custom Numbering or Pattern?** field, select **yes** to expose additional fields regarding how repeated cells are numbered and labelled as meshing objects. For instance, repeated cells in battery models follow a specific naming process, and these fields ensure proper naming of the repeated units.

   - Specify a value for the **First Unit First Number** field. This value will be appended to the end of the name of the first unit's meshing object.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Specify a value for the **First Unit Second Number** field. This value will also be appended to the end of the name of the first unit's meshing object (and be incremented accordingly), and dictates the labelling for the pattern's meshing objects.

  Note:

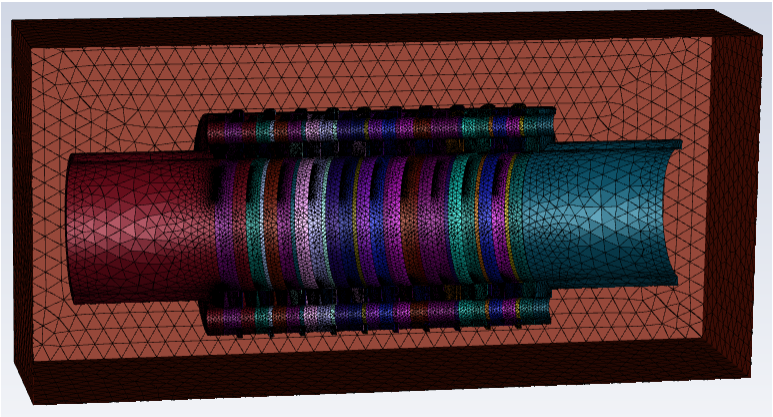
  The names generated for the meshing objects that are created using the designated pattern are dependent on what you assign for the **First Unit First Number** and the **First Unit Second Number** fields (if greater than 1). For instance, the naming convention is useful when modeling batteries with various groupings of battery cells. Fluent internally calculates the number of repeating cells based on these two numbers.

  For example, if a selected part named `cellpart` has the **First Unit First Number** value set to `1`, and the **First Unit Second Number** is set to `1`, then that means there is a single battery cell in the unit the name of the first unit in the pattern will be `cellpart_1`. Subsequent unit names are incremented by one (the **First Unit Second Number**) to be `cellpart_2`, `cellpart_3`, etc.). If, however, the **First Unit Second Number** is set to something other than `1`, such as `4` for example, then that means there are four battery cells in the unit and the name of the first unit in the pattern will *start* with `cellpart_1` and another suffix will be appended and incremented by the **First Unit Second Number** to be `cellpart_1-4`. Subsequent units in the pattern will then have names incremented accordingly, and be called `cellpart_5-8`, `cellpart_9-12`, etc.).

- For **Use Custom Pattern?** field, select **yes** if you want to apply your own custom pattern in the entry field below, The syntax for describing a pattern is discussed in Creating Custom Patterns Using Scripts (p. 371).

10. Click **Add Linear Mesh Pattern**. The mesh pattern will appear in the graphics window.

Figure 3.29: Example of a Geometry With a Mesh Pattern



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

Note:

To ensure that the units of the mesh pattern are properly connected, you must also use a **Shared Topology** task in your workflow (see Applying Share Topology (p. 321)).

Note:

Additional linear mesh patterns cannot be created based on objects that have already been used in an existing mesh pattern.

11. Once you are satisfied with your changes, proceed to the next step in the workflow.

3.23.1. Creating Custom Patterns Using Scripts

Custom patterns can be generated in the **Add Linear Mesh Pattern** task by using a specialized scripting syntax. A single pattern can easily be constructed with a few commands. Multiple patterns can also be combined together easily and performed at one time. In either case, the commands can be copied and pasted into the text window in the **Add Linear Mesh Pattern** task.

There are two main approaches for creating a custom pattern:

- **Singular Pattern** - where a single naming convention is applied to all objects. This assumes that the unit already has a pattern indicative of a "word-number" pattern (such as a single word followed by a single number, for example, `Cell1`). With each copy, the number is incremented and a '0" is inserted (for example, `Cell01`, `Cell02`, `Cell03`, etc.). Within this pattern type, there are two rules governing naming: **regular** (default) and **explicit**.

  Use the **explicit** approach where only a single copy is created with a designated prefix. The command simply copies zones with the `name1` prefix to the `name2` prefix. For example, `one-`, `two-`, `three-`, etc. You can combine explicit patterns with regular patterns. You can also have multiple explicit pattern commands before you start using regular commands.

- **Dual Pattern** - where a dual naming convention is applied to two sets of objects. This assumes that the units already have a pattern indicative of a "word-number-word-number" prefix (for example `C1P1`, `C1P2`, etc.). With each copy, the first number is incremented (for example, `C1P1-C1P2`, `C2P1-C2P2`,`C3P1-C3P2`, etc.). Within this pattern type, there are two rules governing naming: **regular** (default) and **flip**.

  Use the **flip** name rule to flip prefixes names between each objects at each copy. This is often needed if the copy includes a 180 degree rotation.

The syntax for a custom pattern is as follows. See Examples of Creating Custom Patterns Using Scripts (p. 374) for more information.

```
<number-of-operations> (<name1> <name1-value>)
                               (<name2> <name1-value>)
                               (<number1> <number1-value>)
                               (<number2> <number1-value>)
```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
                                (rotation-axis (x-value y-value z-value))
                                (rotation-angle (x-value y-value z-value))
                                (translation-vector (x-value y-value z-value))
                                (pitch <pitch-value>)
                                (namerule <namerule-value>)
                                (type <type-value>)
                                (copies <number-of-copies>)
end
```

where

number-of-operations

The number of operations that make up the pattern.

name1

The first name designation.

number1

The first number designation, set to `1` by default.

name2

The second name designation, depending on the context. When using a singular pattern (with the `explicit` name rule) this is the prefix of the copy. When using a dual pattern, this is the second name of the prefix.

number2

The second number designation, depending on the context. When using a singular pattern, this is the increment of `number1`. When using a dual pattern, this is the second number in the prefix. By default, this is set to `1`.

rotation-axis

The rotational axis setting (if any), set to `(0 1 0)` by default. For example, use "`rotation-axis (0 0 1)`" to indicate that the pattern's copies will rotate along the Z-axis.

rotation-angle

The rotational angle setting (if any), set to `0` by default, in degrees. For example, use "`rotation-angle 45`" to indicate that the pattern's copies will rotate 45 degrees.

translation-vector

The translational vector setting (if any), set to `(0 0 1)` by default. For example, use "`translation-vector (1 0 0)`" to indicate that the pattern's copies will be translated in the (relative) X direction. When (`pitch 0`), then you would specify the *absolute* translation in X, Y, or Z.

pitch

The pitch setting (if any), set to `0` by default. For example, use "`pitch 20`" to indicate that the displacement factor between a pattern's copies will be 20.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

namerule

The name rule change (if any), set to `regular` by default.

For a singular pattern, you can choose from `regular` (default) or `explicit`. Use the `explicit` name rule to use explicit naming.

For a dual pattern, you can choose from `regular` (default) or `flip` pattern. Use the `flip` name rule to flip prefixes names between each objects at each copy. This is often needed if the copy includes a 180 degree rotation.

number-of-operations

The number of operational elements contained in the first pattern.

type

The pattern type designation (if any), set to `single` by default. For example, use "`type dual`" to indicate that the pattern will use the dual pattern approach (where a dual naming convention is applied to two sets of objects). The type can also be `single` (the default value) indicating a single naming convention is applied to all objects.

copies

The number of copies created. For example, use "`copies 5`" to indicate that the pattern will be composed of 5 copies.

Important:

You should keep the following rules in mind when using this syntax:

- The number of operations *must* be the first number specified in the pattern definition. The order of the remaining keyname-value pairs is not important.
- Each pattern definition (or copy) must be on its own individual line.
- The pattern definition only needs to include keyname-value pairs that are changing.
- In most cases, the most recent copy is used to create the next copy.

  Using the most recent copy as the starting point, you can perform a sequential series of copy operation combinations (e.g., first-second, second-third, third-fourth, etc.), however, you *cannot* make the second copy in the sequence based on the first copy (e.g., first-second, first-third, first-fourth, etc.).

  The exception is when the `explicit` name rule is used in combination with the `regular` name rule). Note that you can also have more than one explicit name rule pattern commands before you start using regular name rule commands.

See Examples of Creating Custom Patterns Using Scripts (p. 374) for more information.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.23.1.1. Examples of Creating Custom Patterns Using Scripts

Multiple patterns can be included at once and copied and pasted into the text field of the task. For instance, the following is an example of three custom patterns you can define at one time:

```
5 (name1 c) (number1 1) (translation-vector (1 0 0)) (pitch 15.3) (copies 8)
2 (pitch 30) (copies 1)
2 (rotation-angle 180) (copies 3)
end
```

In this case, the first pattern has a total of 5 commands designating `name1`, `number1`, the `translation-vector` value, the `pitch` value, and the number of `copies`.

```
5 (name1 c) (number1 1) (translation-vector (1 0 0)) (pitch 15.3) (copies 8)
```

The second pattern has a total of two commands designating the `pitch` value and providing a singular copy.

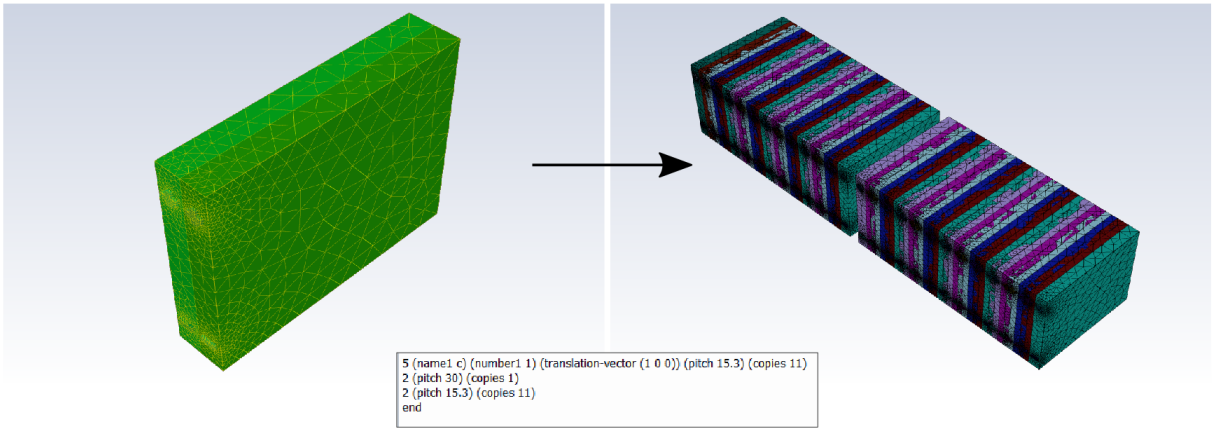
```
2 (pitch 30) (copies 1)
```

The third pattern has a total of two commands designating the `rotation-angle` as 180 degrees and creating three `copies`.

```
2 (rotation-angle 180) (copies 3)
```

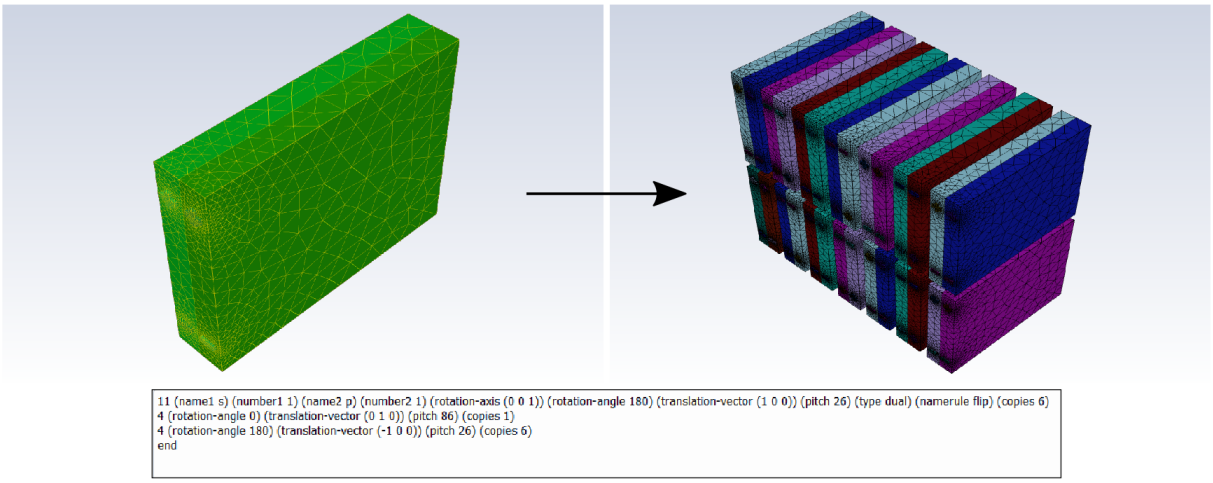
The series of patterns is completed using the `end` statement.

Figure 3.30: An Example of a Custom Singular Pattern



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

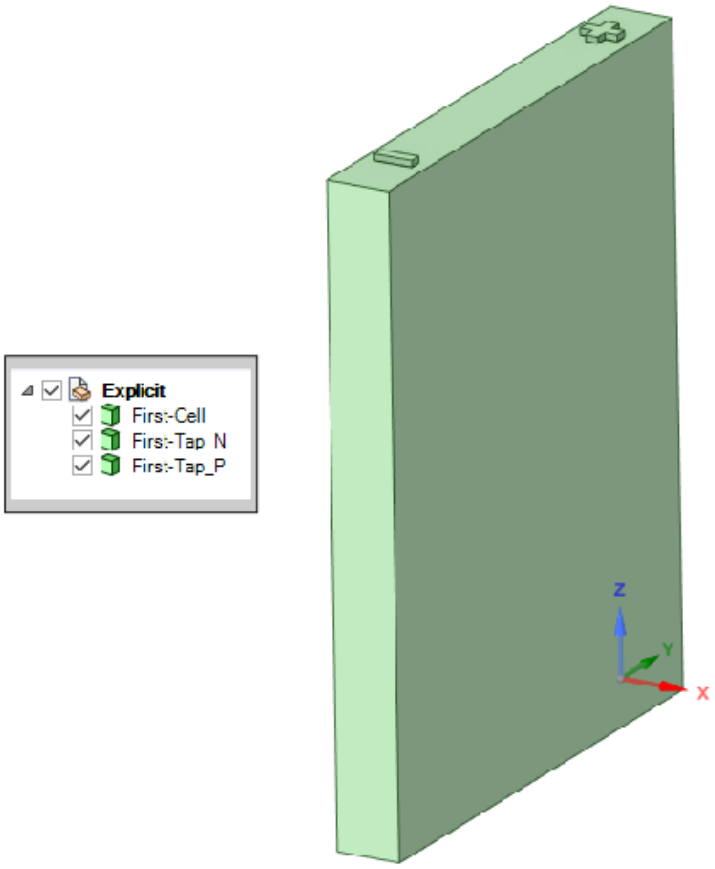
Figure 3.31: An Example of a Custom Dual Pattern



3.23.1.1.1. Pattern Example Using Explicit Name Rule

In the case where you are only interested in creating a series of sequential copies, use the `explicit` name rule. In this case, only sequential copying is allowed and automatically handled, such that `name1 = name2` from the previous custom pattern.

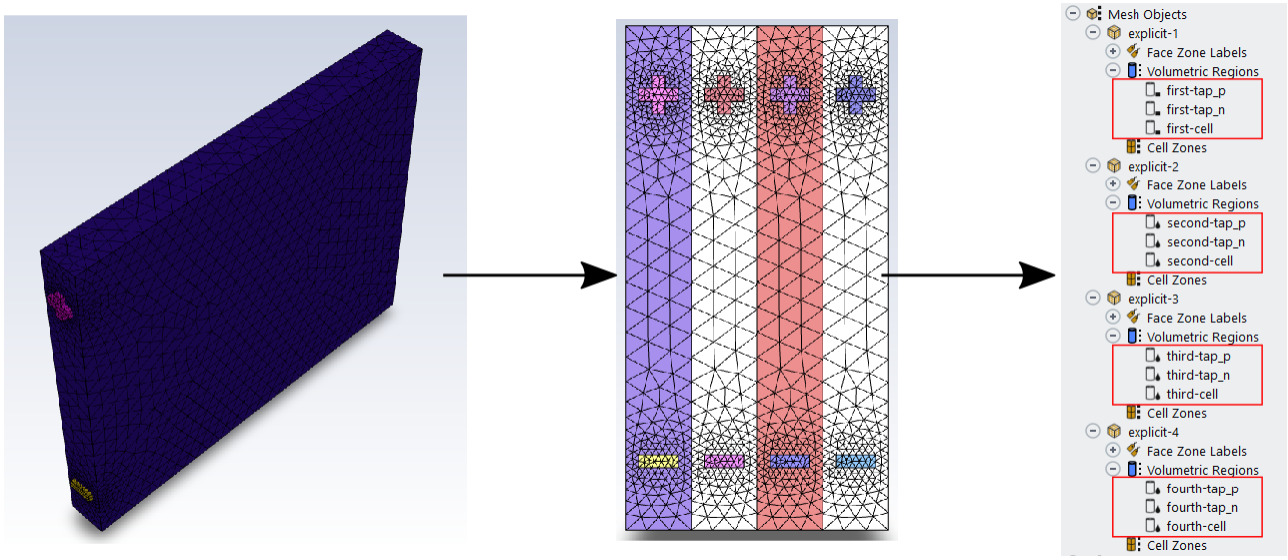
For instance, if you originally have a geometry of a battery cell with three components:



If you wanted to make a series of sequential battery cells using a custom mesh pattern, for example, you could use something similar to the following using explicit naming:

```
6 (name1 first)  (name2 second) (translation-vector (1 0 0)) (pitch 10)  (namerule explicit) (copies 1)
2 (name2 third) (copies 1)
2 (name2 fourth) (copies 1)
end
```

The first line calls for making a copy of the three sub-components, replacing the prefix "first" with "second" in their names, and applying a translation and a pitch. The second line creates another copy based on the first copy, replacing the prefix "first" with "third". The third line creates yet another copy of the first pattern, replacing the prefix "first" with "fourth" in the name. When the entire pattern is applied within the workflow task, you get four sequential copies, each with their unique prefixes based on the naming of the first set of sub-components.

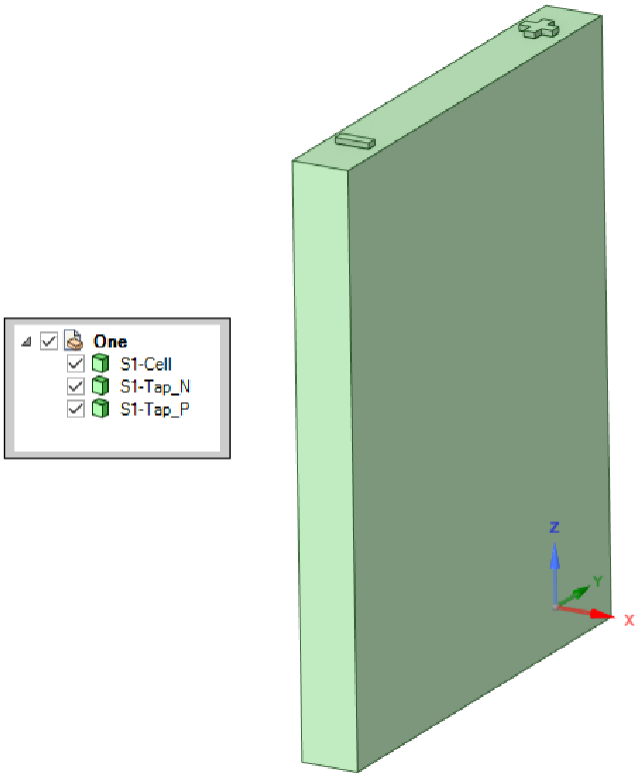


3.23.1.1.2. Pattern Example of Using Both Explicit and Regular Name Rule

The following is an example of using explicit naming with a regular singular pattern approach.

Similar to the previous example, if you originally have a geometry of a battery cell with three components:

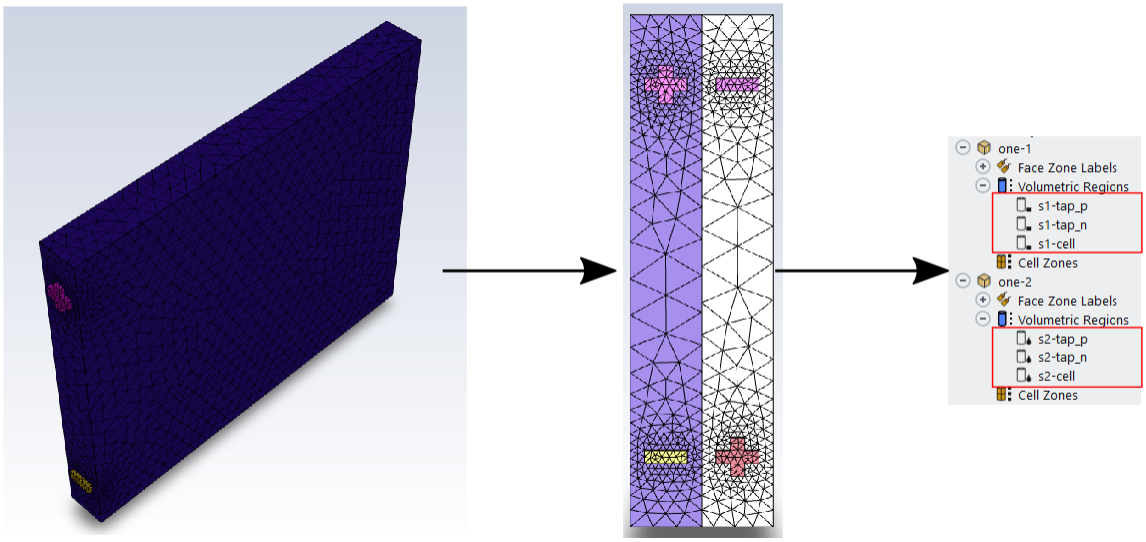
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If you wanted to make a series of sequential, alternating battery cells using a custom mesh pattern, for example, you could use something similar to the following using explicit and regular naming pattern:

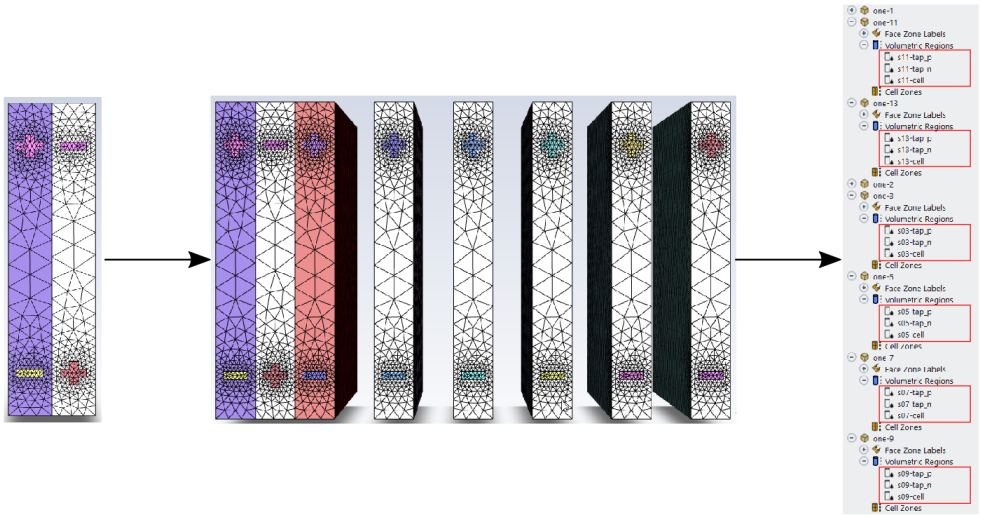
```
8 (name1 s1) (name2 s2) (translation-vector (1 0 0)) (pitch 10) (rotation-angle 180) (rotation-axis (0 0 1))
7 (name1 s) (number1 1) (number2 2) (rotation-angle 0) (pitch 20)  (namerule regular) (copies 6)
2 (number1 2) (copies 6)
end
```

The first line calls for making a copy of the three sub-components, replacing the prefix "s1" with "s2" in their names, and applying a translation, a rotation, and a pitch.

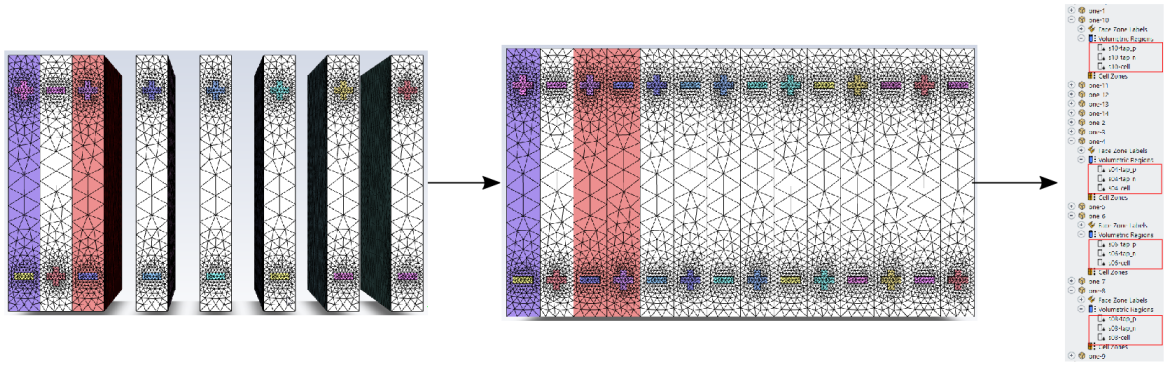


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The second line creates six additional copies based on the first pattern's translation and incrementing the prefix by two (e.g., from `s01` to `s03`, `s05`, `s07`, etc.).

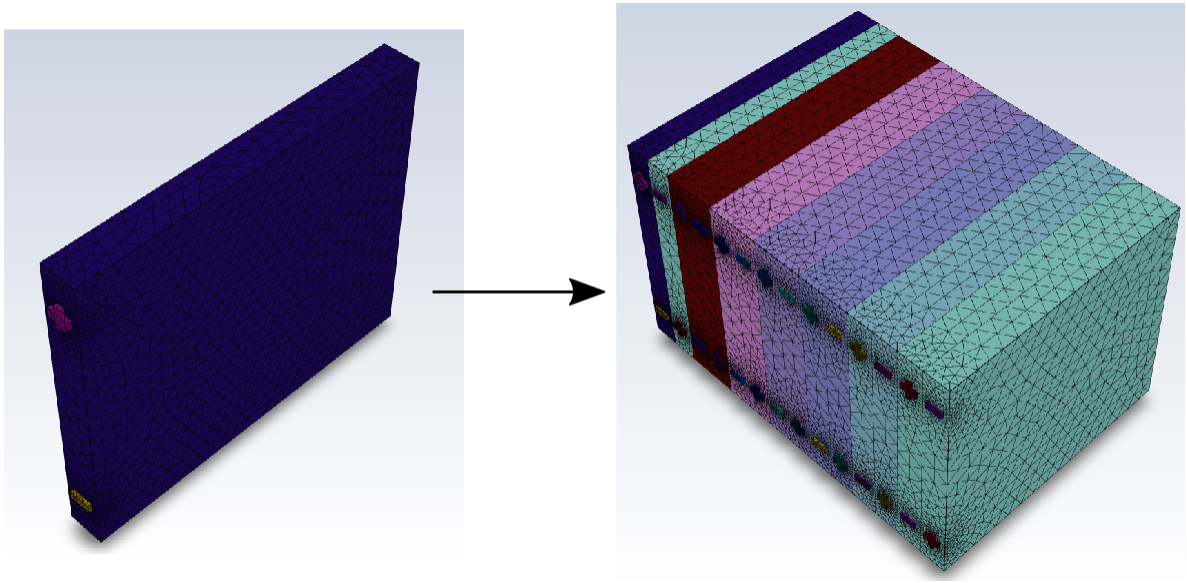


The third line creates another six copies of the first pattern's second cell translation and increments the prefix by two (e.g., from `s02` to `s04`, `s06`, `s08`, etc.).



When the entire pattern is applied within the workflow task, you get a collection of seven sequential, alternating sets of battery cells, each with their unique prefixes based on the naming of the first set of sub-components.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3.24. Managing Zones

In some cases, such as when using multizone meshing, you can use the **Manage Zones** task prior to generating the volume mesh in order to separate zones, split cylinders and create edge labels in preparation for other tasks in the workflow.

Note:

Managing zones before volume meshing is not available when **Separate Zone By** is set to **face** in the **Import Geometry** task (see Importing Geometries (p. 298)).

In other cases, after you have generated your volume mesh, you can use the **Manage Zones** task to make additional changes to your cell and/or face zones (such as renaming or merging) prior to proceeding to the Fluent solver. This can especially useful for large models with numerous zones.

Note:

This task allows the support of models with "multiple" body labels that might originate as named spaces from SpaceClaim or DesignModeler (such as the numerous parts of an electric motor, for instance). Body names between each body label should be unique, however, while inside a body label, the names can be the same.

Note:

Labels and bodies should **not** have the same name, since doing so would lead to ambiguities. Bodies whose name contains the string 'fluid' will automatically be assigned as fluid regions, however, the same is not true for solids, since that is usually the default name for any body in both SpaceClaim and DesignModeler.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Managing Zones Before Volume Meshing

For cases when you have added the **Manage Zones** task *before* the **Generate the Volume Mesh** task, you can perform the following steps.

1. Choose whether to **Select By** the label or zone name in the list below.

   Note:

   Labels originate from the CAD geometry, such as from group names in SpaceClaim geometries, or from named selections in DesignModeler geometries.

   When selecting by labels, for imported CAD geometries that include body labels (and if you have set the **Use Body Labels** field to **Yes** in the **Import Geometry** task, see Importing Geometries (p. 298)), body label names are also listed in the selection list.

   Use the **Filter Text** option in the drop-down to provide text and/or regular expressions in filtering the list (for example, using *, ?, and []). You can also choose the **Use Wildcard** option in the drop-down to provide wildcard expressions in filtering the list. When you use either ? or * in your expression, the matching list item(s) are automatically selected in the list. Use ^, |, and & in your expression to indicate boolean operations for NOT, OR, and AND, respectively. See Filtering Lists and Using Wildcards (p. 287) for more information.

2. For the selected region(s), choose a zone management **Operation** from the drop-down menu. You can choose to:

   - **Separate Zones** and provide a value for the **Separation Angle** for which zones are separated. The lower the value for the angle, the higher the number of faces that will be separated. The default value is **40** degrees. Separated faces are merged back after you generate the volume mesh.

     Note:

     Bodies for multizone meshing will need a separation by zones, such that it will be enforced if it is not performed in this task or in the **Generate the Surface Mesh** task.

   - **Split Cylinders** for selected cylindrical regions. The splitting operation also creates edge labels around split faces.
   - **Split normal to X**, **Split normal to Y**, or **Split normal to Z** for selected non-cylindrical regions (such as heat exchanger tubes), to split bodies in a selected direction normal to a global direction.
   - **Extract Edges** in preparation for using edge sizing.

3. Regardless of the operation chosen, you can also provide a **Control Name** (or keep the default).

Managing Zones After Volume Meshing

For cases when you have added the **Manage Zones** task *after* the **Generate the Volume Mesh** task, you can perform the following steps.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Set the **Type** of zone to **Cell Zone** or **Face Zone**.

   You also have the choice of **Body Labels** as the **Type**, if your imported CAD geometry contains bodies with multiple body labels. You can use this task to merge such bodies, noting the **Merge Body Label Bodies** field in the **Generate Volume Mesh** task (Generating the Volume Mesh (p. 351))

   Note:

   When using multizone meshing and body labels, body label cell zones are always merged.

2. Apply type filtering for the zones:

   - For cell zones, use the **Type Filter** to filter the list of available objects as **Fluid** or **Solid** or **All**.
   - For face zones,use the **Type Filter** to filter the list of available objects as **Internal**, **Fluid-Fluid**, **Solid-Fluid**, **Fluid-Solid**, **External-Solid**, **External-Fluid**, or **External**.

3. Apply value filtering for the zones:

   - For cell zones, use the **Volume Filter** to only display a list of cell zones within a certain volume.
   - For face zones: use the **Area Filter** to only display a list of face zones within a certain area.
   - You can set either filter to **All**, **Less than**, **More than**, or **Equal to** the corresponding **Area** value (for face zones) or **Volume** (for cell zones). When using the **Equal to** option, for the **Equal within a range of (%)** field, specify a percentage range for the volume/area.

4. Additional filtering options and wildcards are available in the list as well to help in selecting cell zones.

   Use the **Filter Text** option in the drop-down to provide text and/or regular expressions in filtering the list (for example, using *, ?, and []). You can also choose the **Use Wildcard** option in the drop-down to provide wildcard expressions in filtering the list. When you use either ? or * in your expression, the matching list item(s) are automatically selected in the list. Use ^, |, and & in your expression to indicate boolean operations for NOT, OR, and AND, respectively. See Filtering Lists and Using Wildcards (p. 287) for more information.

5. To merge selected cell/face zones:

   i. Select **Merge** as the **Operation**.

   ii. Specify the portion of the **Name** of the selected zones that you want to merge, or keep the default name. The task attempts to create an appropriate name. If the target name already exists, then a numerical value will be appended to the name (such as `my_label_1`, `my_label_2`, etc.).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

iii. For **Do you want to merge adjacent faces?**, select either **Yes** or **No**.

> **Note:**
>
> The merging of cell zones also allows you to merge the underlying face zone(s). Zones with face labels (named selections), however, are not supported during the face merge.

6. To change the prefix of the names of selected cell/face zones:

   i. Select **Change Prefix** as the **Operation**.

   ii. Use the **From** field to indicate the prefix that you want to change, and use the **To** field to indicate what you would like to change the prefix to. The task attempts to change the prefix accordingly. If the target name already exists, then a numerical value will be appended to the name (such as `my_label_1`, `my_label_2`, etc.).

7. To rename selected cell/face zones:

   i. Select **Rename** as the **Operation**.

   ii. Specify a new **Name** for the selected zone. The task attempts to create an appropriate name. If the target name already exists, then a numerical value will be appended to the name (such as `my_label_1`, `my_label_2`, etc.).

8. To merge selected cell/face zones of bodies with multiple body labels:

   i. Select **Body Labels** as the **Type**.

   ii. Select the body labels in the list.

   iii. Select **Merge cells within each body label** as the **Operation**.

   iv. Specify a new **Name** for the selected zone, or use the default name. The task attempts to create an appropriate name. If the target name already exists, then a numerical value will be appended to the name (such as `my_label_1`, `my_label_2`, etc.).

   v. For **Merge Adjacent Faces?**, select either **Yes** or **No**.

9. Click **Manage Zone** to apply your changes for this task. Use the **Draw Mesh** button to display the surface mesh to inspect any zone changes.

   If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

3.25. Modifying Mesh Refinement

You can perform additional enhancements to your surface mesh using the **Modify Mesh Refinement** task. It is not recommended to use this task to coarsen the mesh locally, particularly on curved surfaces, instead, perform any coarsening operations using the **Add Local Sizing** task (see Adding Local Sizing (p. 302) for more information).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Choose a means for adding or controlling the remeshing using the **Operation(s)** drop-down menu.

   You can choose to simply **Add** a mesh refinement control, and remesh later when the task is completed and updated, or you can choose to **Add & Remesh** a mesh refinement control that will add the control and perform a remeshing operation at the same time.

   When **Add & Remesh** is used, and you select only one label, no additional *labels* are created. In all other cases, a new label is created, representing the selected zones/labels.

2. Provide a **Name**, or use the default name (`labelsize_1`).

3. Specify a **Target Mesh Size**.

4. Choose whether to **Select By** the label or zone name in the list below.

   Note:

   Labels originate from the CAD geometry, such as from group names in SpaceClaim geometries, or from named selections in DesignModeler geometries.

   Use the **Filter Text** option in the drop-down to provide text and/or regular expressions in filtering the list (for example, using *, ?, and []). You can also choose the **Use Wildcard** option in the drop-down to provide wildcard expressions in filtering the list. When you use either ? or * in your expression, the matching list item(s) are automatically selected in the list. Use ^, |, and & in your expression to indicate boolean operations for NOT, OR, and AND, respectively. See Filtering Lists and Using Wildcards (p. 287) for more information.

5. Click **Advanced Options** to access additional controls prior to performing this task. Options include:

   - Use the **Remesh Feature Min Angle** option to specify the minimum angle for which features will be preserved upon remeshing.
   - Use the **Remesh Feature Max Angle** option to specify the maximum angle for which features will be preserved upon remeshing.
   - Use the **Remesh Corner Angle** option to specify the corner angle for remeshing.

6. Click **Modify Mesh Refinement**. The additional surface mesh settings will be applied and visible. For more information about surface meshes, see Object-Based Surface Meshing (p. 616).

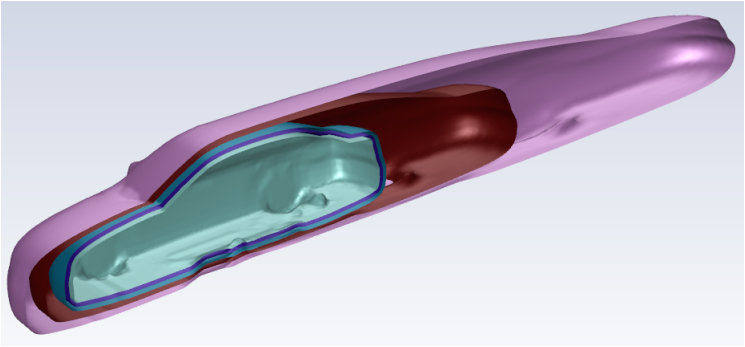
   If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

3.26. Creating Local Refinement Regions

When simulating flow within and around an object, your geometry may not contain a fully defined refinement region, or body of influence (BOI). If a volume region needs mesh refinement, that region can be identified by watertight surface. You can manually create such a surface in the **Create Local Refinement Regions** task by: defining a bounding box around one or more selected objects; or creating an offset surface. For example, this volume region, which surrounds the relevant aspects of your flow physics, can be used to as an alternative to refine the mesh in a particular area, such as the wake region behind a vehicle.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 3.32: An Example of a Refinement Region Around a Car



1. For **Type**, choose from the following:

   - Select **Box** to define a bounding box for the refinement region.
   - Select **Offset Surface** if the refinement region will be offset from the surface of an existing, selected object or zone, based on the boundary layer. You should make sure that you select a closed body for the offset surface.

2. Specify a **Name** for the body of influence, or use the default name (`local-refinement-1`).

3. For the **Coordinate Specification Method** prompt (when you select **Box** as the **Type**), you can choose from the following options:

   - Use the **Ratio relative to geometry size** option so that you can define the refinement region based on the relative size of selected object(s) or zone(s).
   - Use the **Directly specify coordinates** option so that you can explicitly define the location and dimension of the refinement region without having to select object(s) or zone(s).

4. Specify a value for the **Mesh Size** for the refinement region.

5. Choose whether to **Select By** the label or zone name in the list below.

   Note:

   Labels originate from the CAD geometry, such as from group names in SpaceClaim geometries, or from named selections in DesignModeler geometries.

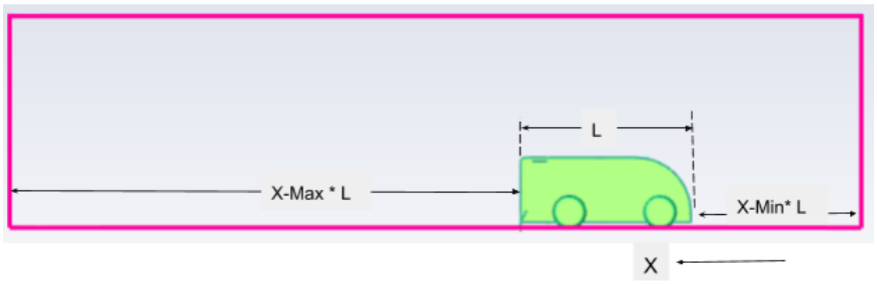
   Select items in the list, or use the **Filter Text** option in the drop-down to provide text and/or regular expressions in filtering the list (for example, using *, ?, and []). You can also choose the **Use Wildcard** option in the drop-down to provide wildcard expressions in filtering the list. When you use either ? or * in your expression, the matching list item(s) are automatically selected in the list. Use ^, |, and & in your expression to indicate boolean operations for NOT, OR, and AND, respectively. See Filtering Lists and Using Wildcards (p. 287) for more information.

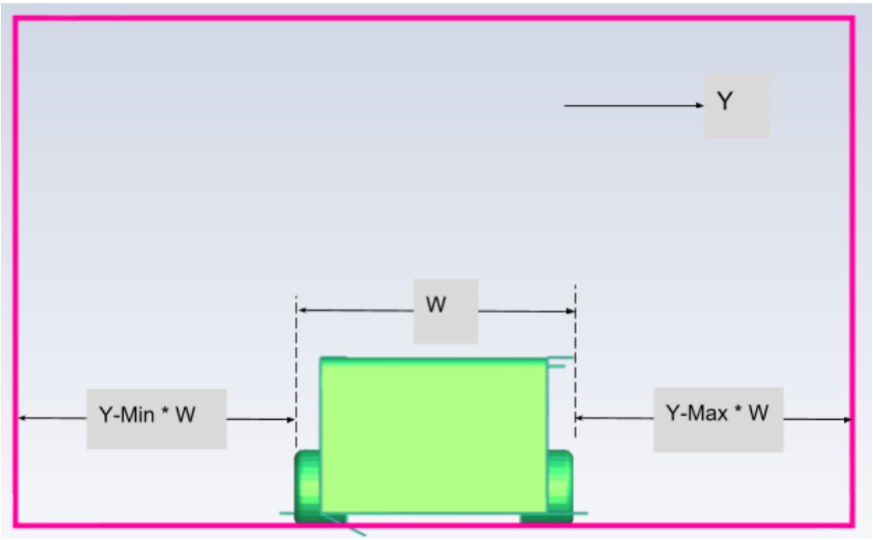
6. Once one or more object(s) or zone(s) is selected, you can define the extents of the refinement region around the selection by defining a bounding box.

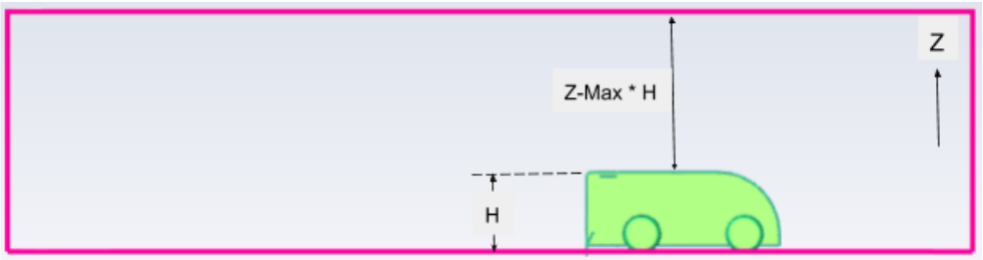
7. Depending on the **Type** of refinement region, additional parameters are required.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For a **Box** refinement region:
  - When you select **Ratio relative to geometry size**, the **Box Parameters** hold the minimum and maximum extension ratios for the X, Y, and Z dimensions. By default, they are set to extend the dimensions of a bounding box around your selected object(s) or zone(s) out by a factor equivalent to the total singular length of the geometry in the X, Y, and Z directions. You can change the values according to your needs, and the display in the graphics window will change accordingly.







  - When you select **Directly specify coordinates**, the **Box Parameters** hold the minimum and maximum extension *distance* for the X, Y, and Z dimensions. By default, they are set to values that correspond to the extension ratios, but you can set them to more suitable values as required, and the display in the graphics window will change accordingly.

- For an **Offset Surface** refinement region:
  - Specify the **Flow Direction**. It is used to determine the wake. The wake is downstream of the selected geometry object(s).
  - Use the **Flip** option if the flow direction is in opposite direction.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– Specify a value for the **Defeaturing Size**, or use the default value. This will obtain the rough shape of the selected object(s), which are wrapped using this size.

– Specify a value for the **Total Thickness**, or use the default value. This is how far from the selected object(s) the rough shape is offset.

– Specify a value for the **Boundary Layer Levels**, or use the default value, to determine the number of offset surfaces to be created.

– Specify a value for the **Wake Growth Factor**, or use the default value. Each scaled offset surface is grown in size in the wake direction by this factor from the previous scaled offset.

– Specify a value for the **Cross Wake Growth Factor**, or use the default value. Each scaled offset surface is grown in size in the cross-wake direction by this factor from the previous scaled offset.

– Specify a value for the **Wake Levels**, or use the default value. The rough shape is scaled in the wake direction as well as in the cross-wake direction. This value determines how many scaled offsets are to be created.

The workflow uses an assigned size to create a wrapper that in turn is used to create the refinement region bodies around the selected object(s).

For example, given an automobile geometry, the base is the surface on the vehicle, which has maximum size of 8 mm. The following size control settings were automatically generated using **Boundary Layer Levels** set to **2** and **Wake Levels** set to **4**:

Name	Size Function	Max Size
name	soft	8
name-1	boi	16
name-2	boi	32
name-wake-1	boi	64
name-wake-2	boi	128
name-wake-3	boi	256
name-wake-4	boi	512

The refinement regions and the various settings can be visualized using Figure 4.22: An Example of Multiple Refinement Regions Around a Car (p. 435) and Figure 4.23: An Example of Multiple Refinement Regions Around a Vehicle (p. 435).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 3.33: An Example of Multiple Refinement Regions Around a Car

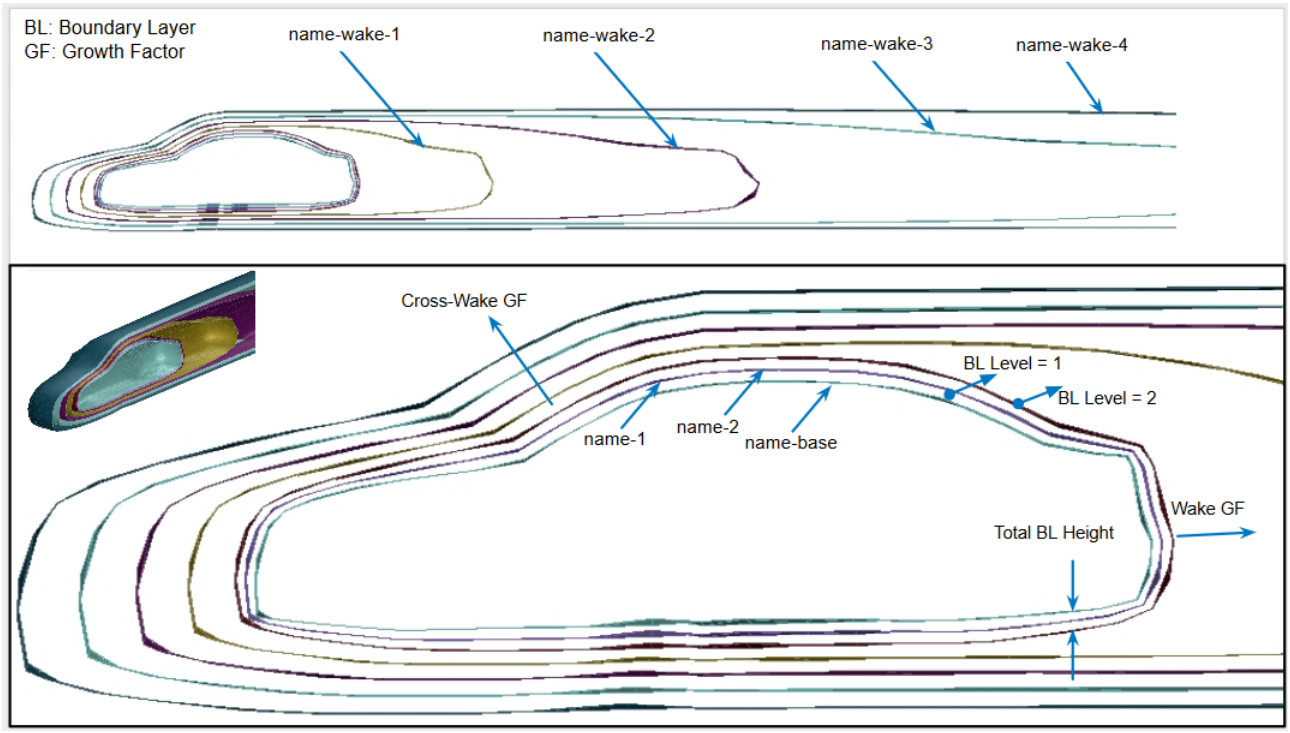
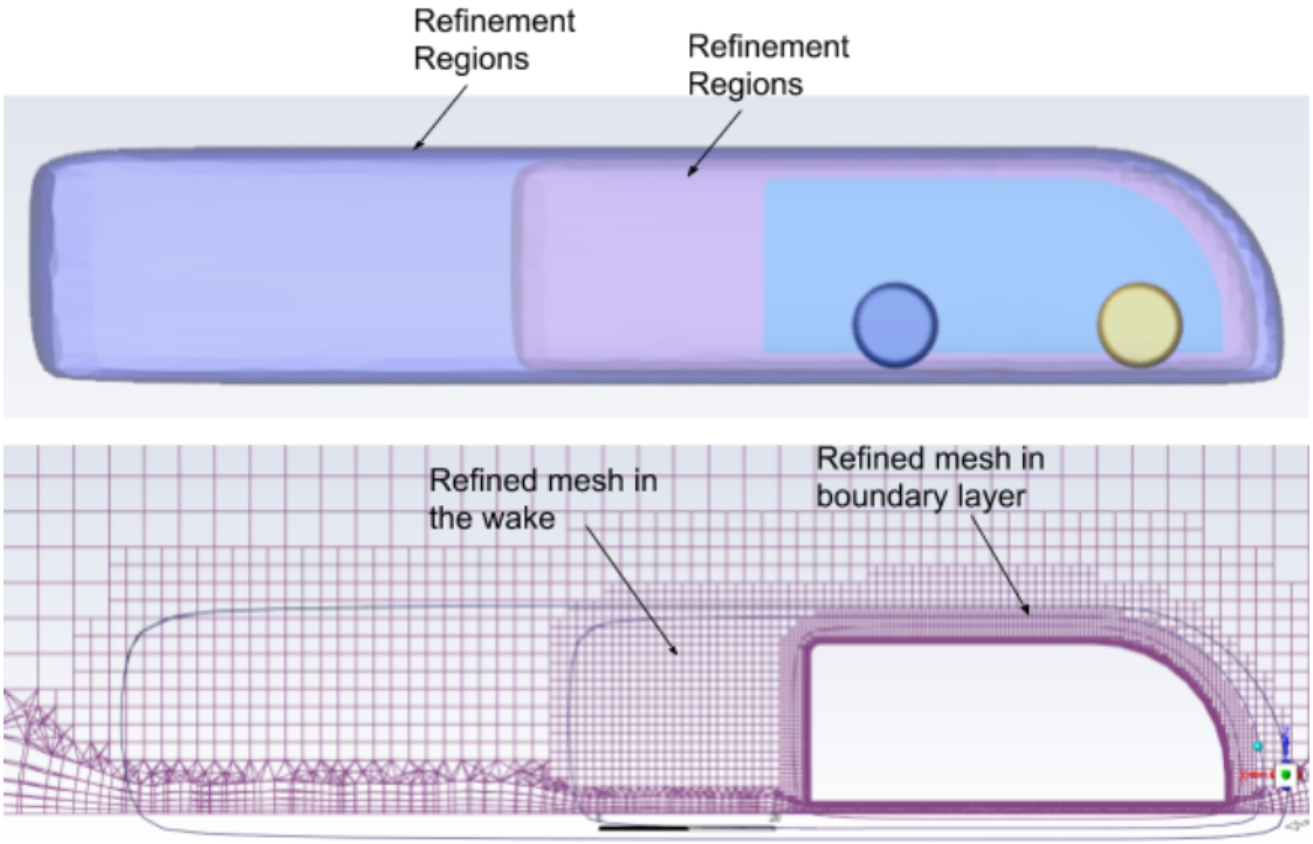


Figure 3.34: An Example of Multiple Refinement Regions Around a Vehicle



8. Once your selections are made, click **Create Local Refinement Regions** and proceed to the next task. For more information about bodies of influence, see Body of Influence (p. 592).

   If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

3.27. Running Custom Journal Commands

Customize your meshing workflow using the **Run Custom Journal** task.

Use a text editor to copy portions of any of your own journal files, and paste them into this task to perform additional meshing refinement. See Customizing Workflows (p. 280) for more information.

Once you are satisfied with your changes, click **Run Custom Journal**, and proceed to the next step in the workflow.

If you need to make adjustments to any of your settings in this task, click **Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

Note:

This task can be added to the workflow as many times as you require, however, this task cannot be the initial task in a custom workflow. In addition, this task assumes that you are familiar with scripting and the available standard Fluent commands in meshing mode. See Query and Utility Functions in the *Fluent Text Command List* - Wrap Functions in the *Fluent Text Command List* for more information.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 4: Using the Fault-tolerant Meshing Workflow

This chapter describes the various tasks that are provided when using the **Fault-tolerant Meshing** guided workflow:

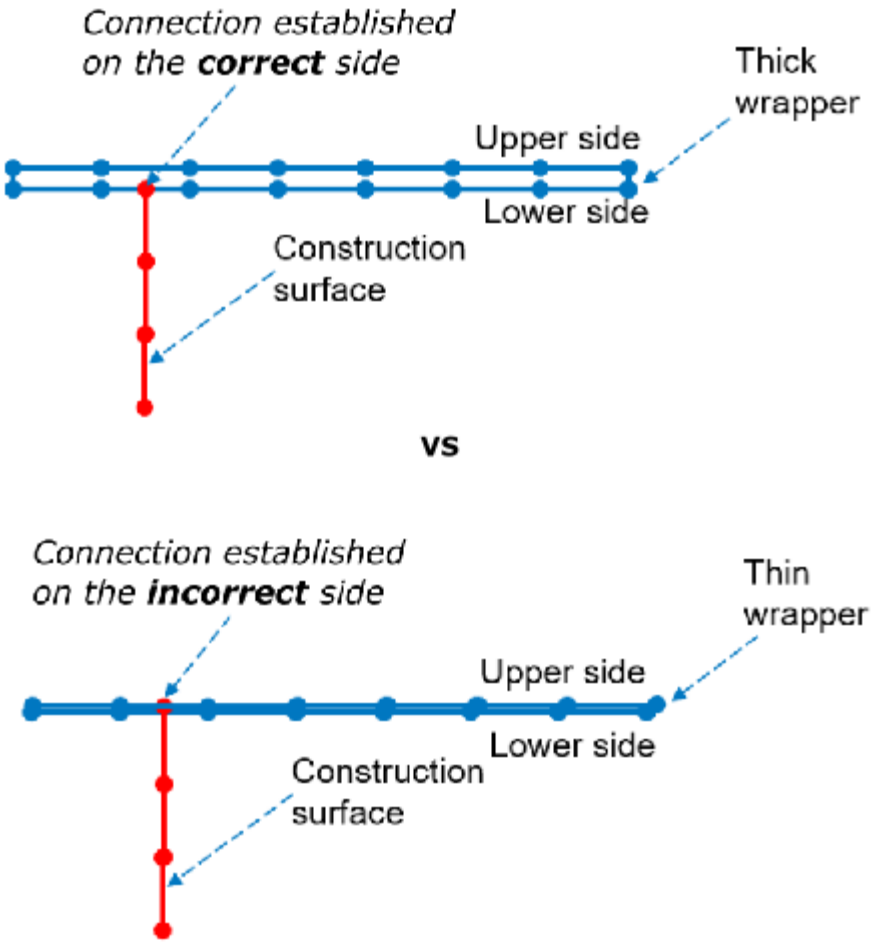
4.1. Fault-tolerant Meshing Workflow Limitations
4.2. Importing CAD Geometries and Managing CAD Parts
4.3. Describing the Geometry and the Flow
4.4. Enclosing Fluid Regions
4.5. Creating External Flow Boundaries
4.6. Creating Local Refinement Regions
4.7. Identifying Construction Surfaces
4.8. Extracting Edge Features
4.9. Adding Thickness to Your Geometry
4.10. Creating Porous Regions
4.11. Identifying Regions
4.12. Defining Leakage Thresholds
4.13. Updating Your Region Settings
4.14. Choosing Mesh Control Options
4.15. Adding Local Size Controls
4.16. Creating Contact Patches
4.17. Creating Gap Covers
4.18. Generating the Surface Mesh
4.19. Updating Boundaries
4.20. Describing Overset Features
4.21. Adding Boundary Layers
4.22. Identifying Deviated Faces
4.23. Generating the Volume Mesh
4.24. Creating Overset Mesh Interfaces
4.25. Identifying Orphans
4.26. Transforming the Volume Mesh
4.27. Extruding the Volume Mesh
4.28. Managing Zones
4.29. Separating Contacts
4.30. Choosing Part Replacement Options

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.1. Fault-tolerant Meshing Workflow Limitations

The following limitations have been observed in the Fluent **Fault-tolerant Meshing** workflow:

- The workflow cannot properly connect construction surfaces to wrapped regions that are too thin.



  In such cases, the wrapper faces between the upper and lower side are too close, and the construction surface connection may occur on the wrong side (the upper side rather than the lower side). For 3D cases, the improper connections can occur on the upper and/or lower sides, creating an incomplete intersected edge loop.

- To ensure a smooth mesh everywhere in the domain, a growth rate applied locally in the domain will extend to the full domain, especially when the localized growth rate value is smaller than the global growth rate value.

- **Reverting and Editing Tasks That Use Existing Size Controls:** When size controls are defined in the workflow using an existing size control file, if any of the intermediate checkpoint files are not available when reverting and editing any of the size-control tasks, the execution of the overall workflow may lead to additional unwanted task updates. To avoid this issue, you can use default size-controls when possible, or you can create the intermediate checkpoint files by saving the workflow template, loading the template back into a new session, and then updating the tasks.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Import CAD and Part Management**
  - On Windows and Linux, you cannot import multiple CAD files that contain different units into the same workflow.
  - *Before* importing SpaceClaim files (`.scdoc` and `.scdocx`) into Fluent Meshing, you need to first enable the **Set as the alternative license preference** license preference option in SpaceClaim in order to uphold licensing preferences.
  - When importing CAD files, the naming convention for CAD parts can not contain the `\` character.
  - The import of 2D models is not supported.
  - When importing JT CAD data, the **JTOpen** option is available on more recent Linux OS variants (requiring a `glibc` supplying `'memcpy@GLIBC_2.14'`), that is, it is not supported on Red Hat / CentOS 6 series.
  - When importing CAD data via the **Workbench** route on Linux, the SLES 15 platform is not supported.
  - Importing polyhedra and poly-hexcore mesh files is not supported.
  - Importing CAD files is not supported when using remote Linux nodes from Windows.
- **Generate the Surface Mesh**
  - The workflow does not support adding zone-specific settings (such as boundary layers) outside the workflow. You can, however, set up zone-specific boundary layers within the workflow using the **Add Boundary Layers** task.
- **Add Boundary Layers**
  - The workflow does not support adding boundary layers in solid regions.
- **Generate the Volume Mesh**
  - You should avoid scenarios where, for a fixed total initial boundary layer height, a specified growth rate will never be respected for a large number (~!00) of splits. Such a scenario results in the first heights of the boundary layer being too small (~1e-16) and introduces system limitations that should be avoided.
- **Extrude Volume Mesh**
  - The preview box is available only for selected faces whose normal is aligned with the Cartesian coordinate.
  - Extruding cylindrical surfaces too far may create invalid meshes. If your cylindrical surface extrusion exhibits undesirable skewness or otherwise is invalid, reduce the total height of the extrusion and perform the operation again to see if results improve.
- **Create Collar Mesh**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - When creating the collar mesh, you should only use the **Automatic** option to preview the location of the collar mesh, but you should use the **Numerical Inputs** option to define the actual location.

- **Preferences**

  - The workflow does not fully support the **Write into memory** option, so if selected as a preference, the workflow will use **Write mesh files** instead. See Setting Preferences for Workflows (p. 289) for more information about preferences in saving data for editing tasks.

- **Limitations for Using the Poly Hexcore Volume Fill Method in Parallel**

  - The mesh quality may not be perfect, due to a few invalid cells, and the change in size could be large for a few cells in the poly transition area.

4.2. Importing CAD Geometries and Managing CAD Parts

Use the **Import CAD and Part Management** task to import one or more CAD files into your simulation and to perform selective management of which parts you may want to include in your simulation, if needed. Selected parts are then designated as potential meshing objects, where selective settings can be adjusted as desired. Upon completing the task, meshing objects are created for later use in your Fluent simulation.

You can import one or more CAD geometry files into your workflow, as well as append additional files as needed. Once specified, you can determine how the portions of your CAD geometry will be converted into meshing objects for your simulation: automatically *by part*, or by manually choosing your *custom* parts or assemblies.

Upon loading the CAD geometry, you can view the components of the CAD geometry in the CAD Model tree. You can also view potential meshing objects in the Meshing Model tree. Various properties for your meshing objects (such as feature extraction angle, or custom refaceting settings) can be viewed and adjusted accordingly. When you are satisfied with your meshing model objects and their properties, you can complete the task, thereby creating meshing objects for later use in your Fluent simulation.

Note:

You can work more easily in this especially complicated task by detaching, relocating, and resizing the editor portion of this workflow task. See Editing Tasks (p. 286) for more information.

Note:

A modified version of this task (designed only for appending CAD files) is also used when replacing CAD parts (see Choosing Part Replacement Options (p. 494) for more information).

---

1. For the **CAD File** field, provide a path and a name for the CAD file that you want to load into the task, or use the (**...**) button to browse for a specific file.

   Standard ANSYS file types are supported, including `.scdoc`, `.scdocx`, `.dsco`, `.agdb`, `.fmd`, `.fmdb`, `.pmdb`, `.tgf`, and `.msh*`.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Other CAD formats are also supported, including `.jt`, `.plmxml`, `.stl`, `.CATPart`, `.CATProduct`, `.prt`, `.x_t`, `.sat`, `.step`, `.iges`, and `.igs`. See, for example, File Format Support for additional formats.

To quickly import multiple CAD files, you can use basic wildcard expression patterns such as the * or ? wildcards. For instance, to import multiple STL files, you can use `*.stl` in the **CAD File** field.

2. Select an appropriate **Import Route** option, or keep the default value, depending on your requirements and platform. In most cases, the default value is recommended and should lead to the desired data being imported, however, you should be aware of and verify the CAD configurations being made because they may be considered during import. See CAD Integration for details.

   The workflow uses an `.fmd` file to extract and persist CAD information for this task. To generate an appropriate `.fmd` file, the available options for the **Import Route** depend on the selected **CAD File**, and your particular platform (Windows or Linux). The various **Import Route** options include:

   - The **Native** option loads the CAD file natively into FM, and internally generates an FMD file that is then loaded into the workflow. This option is available for `.fmd`, `.fmdb`, and `.stl` file formats.

     Note:

     Using this import route, the CAD data should not differ across Windows and Linux for default settings.

   - The **SCDM** option uses ANSYS SpaceClaim Direct Modeler to read the CAD file and internally generates an `.fmd` file that is then loaded into the workflow. You may also want to note any SpaceClaim file options that you may have configured.

     Note:

     This route is only available on Windows, and is not available on Linux.

   - The **DSCO** option, available only when Discovery is installed, allows you to import Discovery (`.dsco`) files.

   - The **Workbench** option uses Ansys Workbench CAD readers or plug-ins to attach or load the given CAD file and generates an `.fmd` file that is then loaded into the workflow.

     Note:

     This route is available on Windows and Linux., though not all formats are supported on Linux when compared to Windows. Also, third-party add-on modules can lead to slightly different CAD data upon import (for example, names, assembly hierarchy, etc.).

   - The **ACIS** option is the default import route for `.sab` and `.sat` file types.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The **MSH** option is the default import route for `.tgf`, `.msh.h5`, and `.msh` file types.

  Note:

  While importing `.msh*` files as faceted CAD geometries, zone naming will be influenced by pre-existing labels in the `.msh*` files. If this is not desired, then, before importing the files, remove the labels from the `.msh*` files. Otherwise, if certain zone names are desired, then labels can be used to control the naming.

- The **JTOpen** option is the default import route for `.jt` and `.plmxml` file types, and only faceted data is imported.

  Note:

  The **JTOpen** option is available on more recent Linux OS variants (requiring a `glibc` supplying `'memcpy@GLIBC_2.14'`), that is, it is not supported on Red Hat / CentOS 6 series.

3. For selected `.stl` files, use the **File Unit** field to indicate the units in which the file was created.

   Note:

   Imported `.stl` files can have different sets of units.

4. For selected `.jt` files using the **JTOPEN** import route, you can use the **Jt LOD** field to choose the level of CAD tessellation accuracy (level of detail) exposed after import. For example, typically, you might have three levels of detail (such as, 1 for 'fine', 2 for 'medium', and 3 for 'coarse') but up to ten might be contained in the `.jt` file. If a certain level of detail does not exist in the file, the next finer level of detail is taken automatically.

5. Enable the **Part per body** option to import the CAD geometry such that all bodies are available as individual parts when the geometry is loaded into the CAD Model tree.

6. (optional) You can choose to apply a special part naming convention through the **Settings** drop-down button by selecting the **Prefix component name** option. Once selected, and when the file is loaded into the CAD Model tree, the names of the geometry's individual parts are prefixed with the names of their corresponding components.

   Figure 4.1: Example of Using a Component Name Prefix

   The naming convention is applied to the names of the objects in the Meshing Model tree as well, whether they are created on a per-part basis or using a custom approach.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In addition, the **Remove empty components** option, enabled by default, removes any empty CAD components while loading the CAD file.

7. Select an option from the **Create Meshing Object Per** field.

   - Choose **One per part** so that Fluent automatically creates meshing objects for all the parts within the CAD model.
   - Choose **Custom** to be able to manually assign which parts of your CAD model are to be included as meshing objects in your simulation.

8. Use the **Display Unit** field to specify the units that you want to work with when the geometry and mesh is displayed in the graphics window. This field also specifies the dimensions of the generated mesh.

   The workflow will display the units in the graphics window next to the ruler and while calculating distances (for example, when using the **Distance** tool in the Ribbon).

9. When the **Import Route** is set to **Workbench**, you can use the **Settings** drop-down button to select additional CAD import settings, such as:

   - **Ignore Solid Names**: (for `.stl` files) allows you to import your CAD geometry while ignoring the names assigned to solids.

     Note that binary STL files contain a single solid and may have an associated solid name, whereas ASCII STL files contain one or more solids and each can have a solid name. The **Ignore Solid Names** option allows to control whether or not to use the name contained in the STL file for naming mesh objects and components.

   - **Import Solid Bodies**: allows you to import solid bodies along with your CAD geometry.
   - **Import Surface Bodies**: allows you to import surface bodies along with your CAD geometry.
   - **Import Line Bodies**: allows you to import line bodies along with your CAD geometry.

10. Use the **Load** button to load the CAD file into the task.

    Note:

    You can append additional CAD file(s) to your currently loaded CAD file. See Appending CAD Files (p. 396) for details.

11. Use the CAD Model tree to visualize the hierarchy of your CAD model. By default, the CAD Model tree is automatically populated with the components of the imported and/or appended CAD file(s). For more information, see Working with the CAD Model Tree (p. 397)

    Note:

    If you have read in a previously generated mesh created from another fault-tolerant workflow session, you need to click the **Load** button to reload the CAD file into the task and to see its components in the CAD Model tree.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12. Use the Meshing Model tree to visualize the hierarchy of your meshing objects.

    - If **Create Meshing Objects Per** is set to **One per part**, the Meshing Model tree is automatically populated with all of the corresponding parts from the CAD model, providing you with a preview of the number of meshing objects that will be generated.
    - If **Create Meshing Objects Per** is set to **Custom**, the Meshing Model tree is initially empty, and you can select individual parts and manually add them to the Meshing Model tree, providing you with a more customized approach to the number of meshing objects that will be generated.

    For more information, see Working with the Meshing Model Tree (p. 401).

13. Set various properties, such as feature extraction angle, or custom refaceting settings, for selected meshing objects ( ). See Setting Properties for Meshing Model Objects (p. 404) for details.

14. Once your selections are made, click **Create Meshing Objects** to generate a collection of meshing objects based on the imported CAD geometry.

    If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

15. Proceed to the next step in the workflow.

4.2.1. Appending CAD Files

Once a CAD file is loaded into the **Import CAD and Part Management** task, you can append additional files to the current CAD model.

a. Select the **Append** check box.

b. Specify additional CAD files in the **Append File** field, provide a path and a name for the CAD file that you want to import, or use the (**...**) button to browse for a specific file.

   To quickly append multiple CAD files, you can use basic wildcard expression patterns such as the * or ? wildcards. For instance, to append multiple STL files, you can use `*.stl` in the **Append File** field.

c. For selected `.stl` files, use the **File Unit** field to indicate the units in which the file was created.

   Note:

   Appended `.stl` files can have different sets of units.

d. For selected `.jt` files using the **JTOPEN** import route, you can use the **Jt LOD** field to choose the level of CAD tessellation accuracy (level of detail) exposed after import.

e. Repeat as required, and when finished, select the **Append** button.

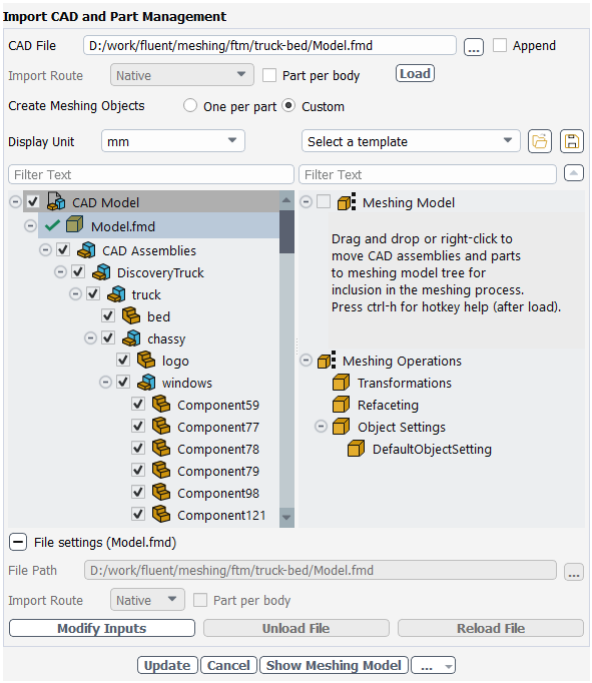
   The contents of the appended file(s) are added to the tree, after the original CAD objects

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.2.2. Working with the CAD Model Tree

Many CAD models contain tens and hundreds of separate components, and you might only be interested in performing a CFD simulation on a small portion of those components. In the CAD Model tree, you can take portions of the CAD model and isolate them for your simulation by moving them from the CAD Model tree and placing them into the Meshing Model tree.

Figure 4.2: A CAD File Loaded into the CAD Model Tree



- Objects in the CAD Model tree are listed using the CAD file name ( ), and can consist of assemblies ( ) or parts ( ). You can have any number of levels in the tree.

  When the top node of the CAD Model tree is selected, use the context menu to **Collapse**/**Expand** and/or **Collapse All**/**Expand All** of the nodes of the tree, as well as to **Sort by Name**.

  Selecting a top level file name in the CAD Model tree allows you to view and edit various import options related to the selected CAD file (under **File settings**), such as:

  – **File Path**: contains the original location of the imported CAD file.

  – **Import Route**: indicates the specific nature of how the CAD file was imported into the workflow, depending on your geometry import requirements and platform. In most cases, the default value is recommended and should lead to the desired data being imported, however, you should be aware of and verify the CAD configurations being made because they may be considered during import. See CAD Integration for details.

  – **Prefix component name**: (optional) allows you to have the names of the geometry's individual parts to be prefixed with the names of their corresponding components.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 4.3: Example of Using a Component Name Prefix

The naming convention is applied to the names of the objects in the Meshing Model tree as well, whether they are created on a per-part basis or using a custom approach.

– **Part Per Body**: use this option to import the CAD geometry such that all bodies are available as individual parts when the geometry is loaded into the CAD Model tree.

– Use the **Modify Inputs** button if you want to make any changes to the exposed settings for the currently selected CAD file.

– Use the **Unload File** button to remove the selected CAD file from the CAD Model tree. You can also perform the same operation using the **Unload** option from the context menu for a selected CAD file in the CAD Model tree.

– Use the **Reload File** button to reload the selected CAD file into the CAD Model tree. You can also perform the same operation using the **Reload** option from the context menu for a selected CAD file in the CAD Model tree.

CAD files listed in the CAD Model tree can indicate their status as either 'update-required' with a lightning bolt icon ( ) or as 'up-to-date' with a green check mark icon ( ). When you later update the task and create and update your meshing objects, the status of any out-of-date file(s) changes to an 'up-to-date' status in the tree. For geometries with many imported CAD files, access to a more granular file status allows Fluent to save time and resources by only having to update selective CAD file(s).

Note:

As you select your CAD file(s), they are listed in the CAD Model tree and their initial state is 'update-required' ( ). Prior to using the **Load** button, you can view and edit selected CAD file properties before they are fully loaded into the task using the **Modify Inputs** button.

- Select CAD model objects directly in the tree in order to perform various tasks. Use **Ctrl**+**left-click** to select multiple non-sequential objects in the tree. Use **Shift**+**left-click** to select multiple sequential objects in the tree. See Using Hot Key Shortcuts in the Model Trees and the Graphics Window (p. 424) for additional selection and display shortcuts for this task.

Note:

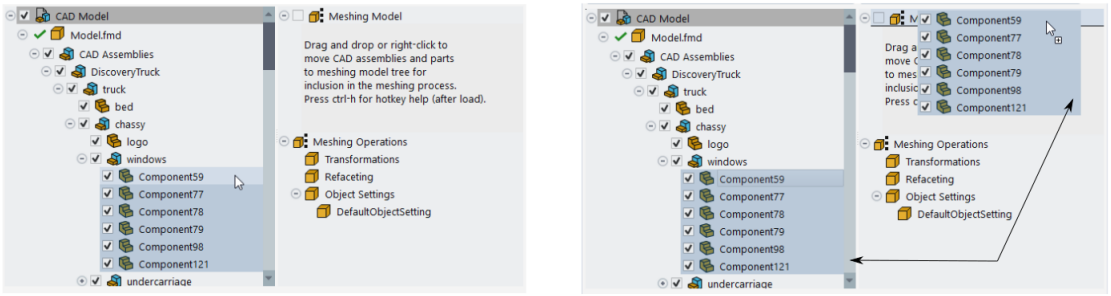
An object is highlighted when it is selected in the tree. The check box for an object is used for displaying that object in the graphics window.

Alternatively, you can right-click the object directly in the graphics window, and it will become selected in the corresponding tree. Use **Ctrl**+**right-click** to select multiple objects in the graphics window (and thereby selecting multiple objects in the tree).

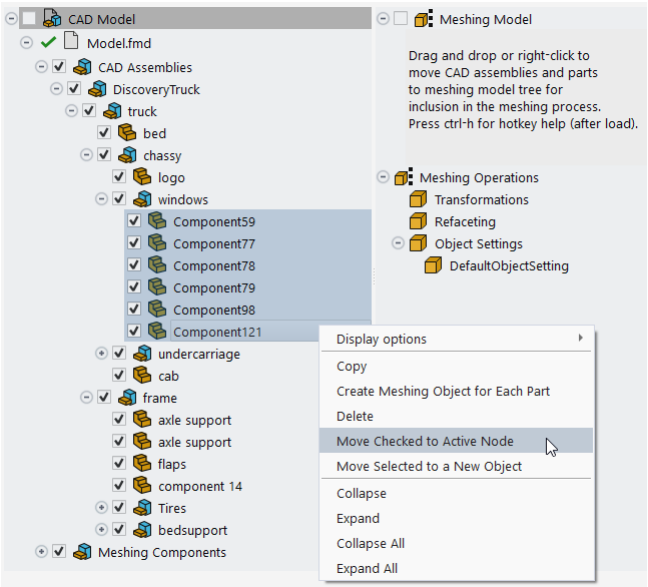
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Display a CAD model object in the graphics window by enabling the check box ( ) to the left of object in the tree. Likewise, to hide an object from view in the graphics window, disable the check box. Selecting/deselecting a parent assembly node in the tree to show/hide the geometry in the graphics window allows for all of its component parts to also be shown/hidden.
- Use the **Filter Text** field to perform name-based searching and filtering on models containing a large number of components.

  Enter text into the **Filter Text** field to see the results instantly. Click the 'x' icon ( ) to clear the filer text and return the list to its original state.
- Select and move objects from the CAD Model tree and place them in the Meshing Model tree:
  - Drag and drop components from the CAD Model tree onto the Meshing Model tree on the right.



  - Alternatively, identify one or more part(s) - by enabling the corresponding check box(es) - in the CAD Model tree, right-click and select **Move Checked to Active Node** in the context menu, so that all of the graphically visible objects are moved to active node



    Alternatively, you can use the use the tree's filter and/or wildcard expressions along with the **Move** button to selectively choose items from the CAD Model tree and move them to the active node in the Meshing Model tree.

  - Select the object(s) directly in the graphics window, and use the **Ctrl**+**middle-click** mouse button combination to move them from the CAD Model tree to the Meshing Model tree.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Selecting an object directly in the graphics window also selects the corresponding object in the tree. You need to right-click an object to select it in the graphics window. Use **Ctrl**+**right-click** to select multiple objects in the graphics window, thereby selecting the corresponding objects in the tree.

– As items are moved from the CAD Model tree, they are hidden in the CAD Model tree and made visible in the Meshing Model tree. You can toggle the visibility of moved objects altogether by using the **Hide Transferred Objects** (or the **Show Transferred Objects**) CAD Model context menu option. If you mistakenly add a component to the Meshing Model tree, you can easily remove it using the **Restore to CAD Model** option from the context menu, thereby adding it back to the CAD Model tree.

- If the CAD Model tree is active, you can visualize the checked CAD objects by clicking the **Show CAD Model** button.
- Various features are available from the context menu of the CAD Model tree, depending on what is selected, including:

  Use the **Display Options** menu to access related sub-options, such as

  → Use the **Display Options > Add to Display** option to visualize the selected items from the CAD Model tree in the graphics window. You can select one or more items in the tree and display them. Using a similar **Add to Display** option in the Meshing Model tree, you can also simultaneously visualize Meshing model objects along side CAD Model objects.

  → Use the **Display Options > Display** option to display only the selected items in the graphics window.

  Note:

  You can only display objects using either the CAD Model tree or the Meshing Model tree, but not both.

  – At the top-level node of the CAD tree, use the **Hide Transferred Objects** option to hide the visibility of CAD Model tree objects that have been transferred to the Meshing Model tree. Likewise, use the **Show Transferred Objects** option to expose the visibility of CAD Model tree objects that have been transferred to the Meshing Model tree.

  – Use the **Copy** option to copy the selected object to be pasted to another object in the Meshing Model tree.

  – Use the **Create Meshing Object for Each Part** option to automatically create a part for the selected object and move it to the Meshing Model tree.

  – Use the **Delete** option to remove the selected items from the CAD Model tree.

  – Use the **Move Checked to Active Node** option to automatically move the checked object to the active node in the Meshing Model tree.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Alternatively, you can use the use the tree's filter and/or wildcard expressions along with the **Move** button to selectively choose items from the CAD Model tree and move them to the active node in the Meshing Model tree.

– Use the **Move Selected to New Object** option to be able to easily transfer the selected CAD model object(s) onto another object in the Meshing Model tree.

Note that if only a selected child object is moved, then the moved object is renamed according to the body name.

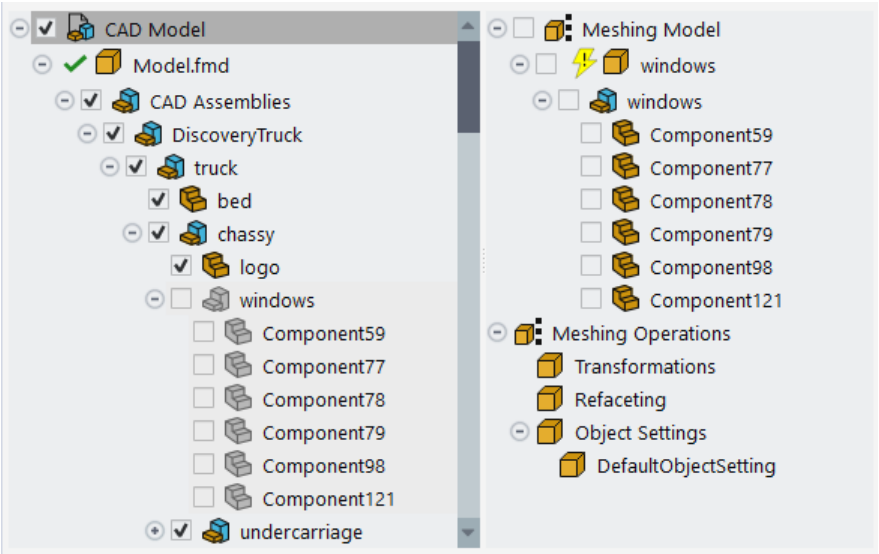
– Use the **Collapse All** or **Collapse** options to close all or a portion of the tree.

– Use the **Expand All** or **Expand** options to open all or a portion of the tree.

4.2.3. Working with the Meshing Model Tree

The Meshing Model tree is primarily populated with selective parts from your CAD model. The components in the Meshing Model tree will be the pieces of your CAD model that you want to include in your Fluent simulation. They will ultimately become meshing objects, and visible as such, in the **Outline View** in your Fluent session.

Figure 4.4: Selected Portions of a CAD File Loaded into the Meshing Model Tree



- Objects in the Meshing Model tree can be top-level meshing objects ( ), assemblies ( ), or parts ( ). You can only have three levels in the tree.

  When the top node of the Meshing Model tree is selected, use the context menu to **Collapse**/**Expand** and/or **Collapse All**/**Expand All** of the nodes of the tree, as well as to **Sort by Name**.

- New objects in the Meshing Model tree can indicate their status as either 'update-required' with a lightning bolt icon ( ) or as 'up-to-date' with a green check mark icon ( ). When you later update the task and create and update your meshing objects, the status of any out-of-date object(s) changes to an 'up-to-date' status in the tree. For geometries with many parts and assemblies, access to a

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

more granular object status allows Fluent to save time and resources by only having to update selective portions of the geometry.

- Select meshing objects directly in the tree in order to perform various tasks. Use **Ctrl**+**left-click** to select multiple non-sequential objects in the tree. Use **Shift**+**left-click** to select multiple sequential objects in the tree. See Using Hot Key Shortcuts in the Model Trees and the Graphics Window (p. 424) for additional selection and display shortcuts for this task.

  Note:

  An object is highlighted when it is selected in the tree. The check box for an object is used for displaying that object in the graphics window.

  Alternatively, you can right-click the object directly in the graphics window, and it will become selected in the corresponding tree. Use **Ctrl**+**right-click** to select multiple objects in the graphics window, thereby selecting multiple objects in the tree.

- Display a meshing object in the graphics window by enabling the check box ( ) to the left of object in the tree. Likewise, to hide an object from view in the graphics window, disable the check box. Selecting/deselecting a parent assembly node in the tree to show/hide the geometry in the graphics window allows for all of its component parts to also be shown/hidden.

- Use the **Filter Text** field to perform name-based searching and filtering on models containing a large number of components.

  Enter text into the **Filter Text** field to see the results instantly. Click the 'x' icon ( ) to clear the filer text and return the list to its original state.

- Set various properties for selected meshing objects (, , , or). See Setting Properties for Meshing Model Objects (p. 404) for details.

- If the Meshing Model tree is active, you can visualize the checked meshing objects by using the **Show Meshing Model** button at the bottom of the task.

- Note that you can manually organize your Meshing Model tree by dragging portions of the tree and dropping them onto other portions of the tree. For instance, you can drag Meshing Model objects and drop them into various Meshing Operations (transformations, refacets and object settings) in order to more easily define your meshing operations.

- Various features are available from the context menu of the Meshing Model tree, depending on where in the hierarchy you are located, including:

  – Use the **Create** option to access related sub-options, such as

  → Use the **Create > Object** option to add a generic object to the tree.

  By default the Meshing Model tree is empty, however, using the **Create > Object** option from the context menu, you can create your own custom collection of object categories, where you can add components from the CAD Model tree. Once created, you can save

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

that collection of simulation meshing objects using the **Save Meshing Objects** icon ( ) where you can save the object collection in a `.mmt` file. If you need to use that collection of objects again, you can use the **Load Meshing Objects** icon ( ) and locate a previously saved `.mmt` file.

The `*.mmt` object collection files include the version number of the current release (for example, for the 2024 R2, release, the files are internally designated with `"version": "24.2"`.

You can also use the **Select a template** drop-down field above the tree to choose an available template of objects, such as **External Flow**, which contains a template of object categories specific to typical external flow objects.

→ Use the **Create > Refacet** option to define refaceting operations for the selected component(s). A refacet operation appears in the Meshing Operations tree (with a default name such as `Refacet-1`). Expand the **Refaceting** properties (below the tree) to assign specific properties to the refaceting operation. See Performing Refaceting Operations on Meshing Model Objects (p. 417) for more details.

→ Use the **Create > Object Setting** option to define object setting operations for the selected component(s). A object setting operation appears in the Meshing Operations tree (with a default name such as `ObjectSetting-1`). Expand the **Object Settings** properties (below the tree) to assign specific properties to the objects setting operation. See Performing Object Setting Operations on Meshing Model Objects (p. 419) for more details.

→ Use the **Create > Transform** option to specify transformational operations (such as translation, rotation, scaling, etc.) for the selected component(s). A corresponding transform operation appears in the Meshing Operations tree (with a default name such as `Translate-1`) where you can expand the transformation properties (below the tree) to assign specific properties to the particular transformation. See Performing Transformation Operations on Meshing Model Objects (p. 411) for more details.

– Use the **Move** option to access related sub-options, such as

→ Use the **Move > Move to a New Object** option to relocate the selected component(s). to a new meshing object in the Meshing Model tree where you can move it and rename it accordingly.

→ Use the **Move > Move to a New Sub-object** option to relocate the selected component(s). to a new meshing object in the Meshing Model tree where it is a new sub-object of a new parent of the same name.

→ Use the **Cut** option to remove an object from the Meshing Model tree where you can then paste it back into the same tree at a different location.

– Use the **Display Options** menu to access related sub-options, such as

→ Use the **Display Options > Add to Display** option to visualize the selected items from the Meshing Model tree in the graphics window. You can select one or more items in the

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

tree and display them. Using a similar **Add to Display** option in the CAD Model tree, you can also simultaneously visualize CAD model objects along side Meshing Model objects.

→ Use the **Display Options > Display** option to display only the selected items in the graphics window.

Note:

You can only display objects using either the CAD Model tree or the Meshing Model tree, but not both.

– If an object has any meshing operations associated with it, use the **List Meshing Operations** option to list those meshing operations in the console window. See Listing Meshing Operations (p. 422) for more information.

– Use the **Rename** option to assign a new name for the selected object in the tree.

– Use the **Restore to CAD Model** option to remove the selected item from the tree and restore it in the CAD Model tree.

– Use the **Set as Active Node** option to specify the current node as the active node. The active node's label is indicated in **bold**. When a node is selected as the active node, you are able to transfer checked CAD model objects more easily to the active node using the context menu.

– Use the **Collapse All** or **Collapse** options to close all or a portion of the tree.

– Use the **Expand All** or **Expand** options to open all or a portion of the tree.

4.2.4. Setting Properties for Meshing Model Objects

Meshing model objects listed in the tree have various properties that you can set, if necessary. Options vary depending on whether you are creating meshing objects automatically based on parts, or whether you are performing a custom meshing object creation.

4.2.4.1. Creating Automatic Meshing Objects on a Per Part Basis

4.2.4.2. Creating Customized Meshing Objects

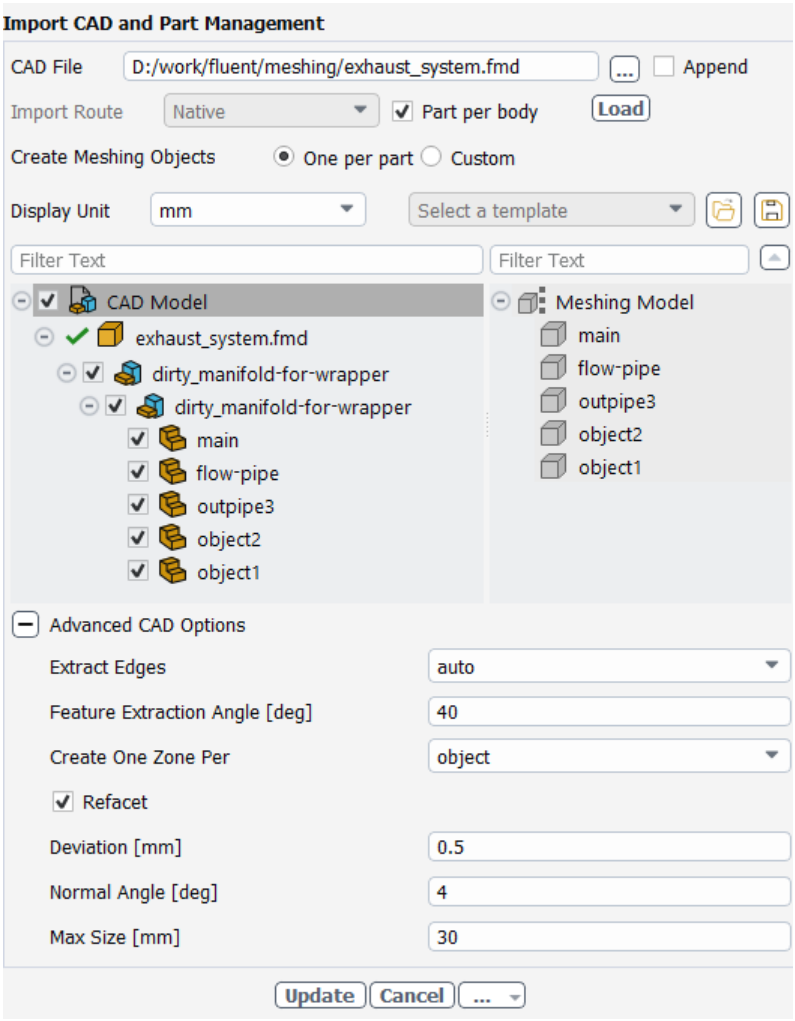
4.2.4.1. Creating Automatic Meshing Objects on a Per Part Basis

By setting the **Create Meshing Objects** field to **One per part**, you can automatically create meshing objects based on the individual parts of your imported CAD geometry.

Prior to creating the per-part meshing objects, you can access various CAD geometry properties under **Advanced CAD Options**.

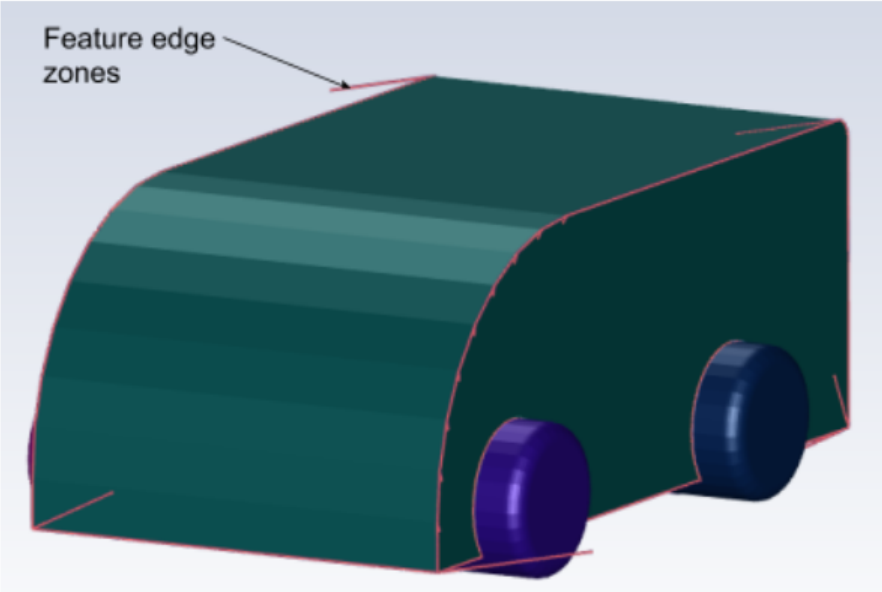
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 4.5: CAD Properties of a Selected Per-Part Meshing Model Object

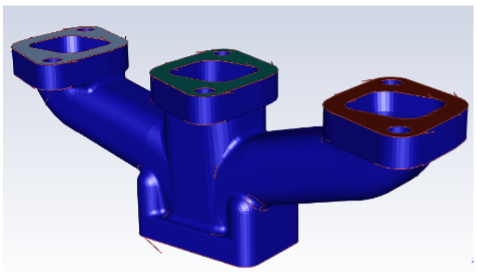
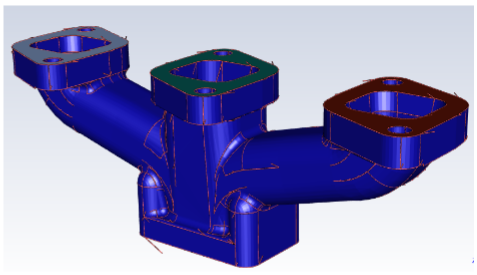


- Use the **Extract Edges** field to determine how edges will be extracted from the CAD geometry (**auto**, **yes**, or **no**). Setting this option to **auto** will extract edges from the CAD geometry when the number of meshing objects is less than 10,000. If the number of meshing objects exceeds this limit, then no edges are extracted. When this option is set to **yes**, then edges are extracted regardless of the number of meshing objects. No edges are extracted when this option is set to **no**.

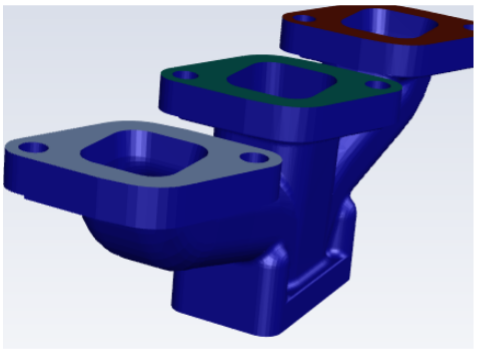
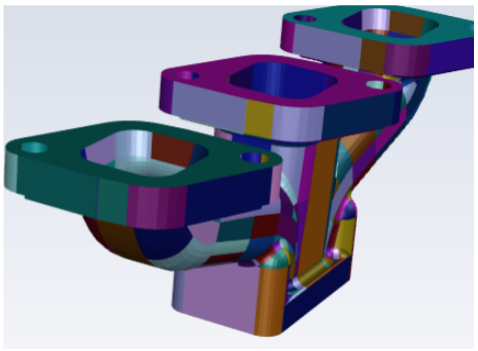
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- When extracting edges, specify a value for the **Feature Extraction Angle** field, or use the default value of 40 degrees, to specify the angle at which features will be extracted from the CAD model on import. See Faceting Considerations (p. 422) for details.

	
Angle = 40 degrees	Angle = 4 degrees

- Set the **Create One Zone Per** option to **object**, **part**, **body**, or **face**, accordingly. For instance, if **face** is selected, each CAD surface or face will be assigned to a face zone.

	
Without creating zones by face	With creating zones by face

- When creating meshing objects by part, use the **Refacet** check box to enable or disable additional faceting refinement in this task. When enabled, you can change the faceting of the selected object. Only the faceted CAD model is used during the meshing process. In

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

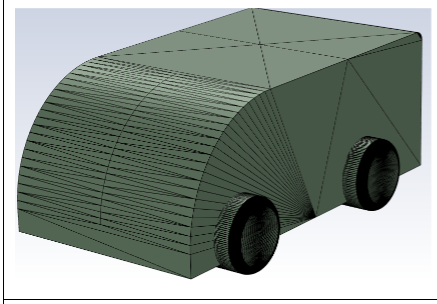
either case, when disabled, normal CAD faceting is performed as defined by the imported CAD file(s).

– The **Deviation** controls how far facet edges are away from the model.

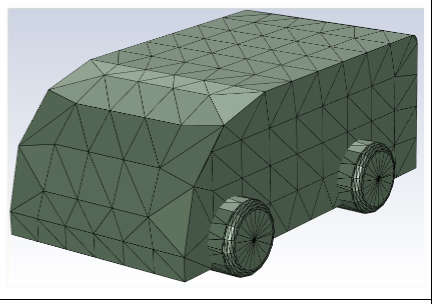
– The **Normal Angle** also controls how far facet edges are away from the model. Decreasing the normal angle will result in more facets along the curved edges.

– The **Max Size** controls the maximum size of the facets after the deviation and the normal angle are respected.

In the following example, the left-hand model uses small values for the deviation and the normal angle in order to capture the curved surface, and a large value for the maximum size in order to get larger facets on the flat surface. The right-hand model uses larger values for the deviation and the normal angle in order to capture the curved surface coarsely, and a smaller value for the maximum size in order to capture the flat surface evenly.



Deviation=0.5, Normal Angle = 4, Max Size=30



Deviation=2, Normal Angle = 30, Max Size=8

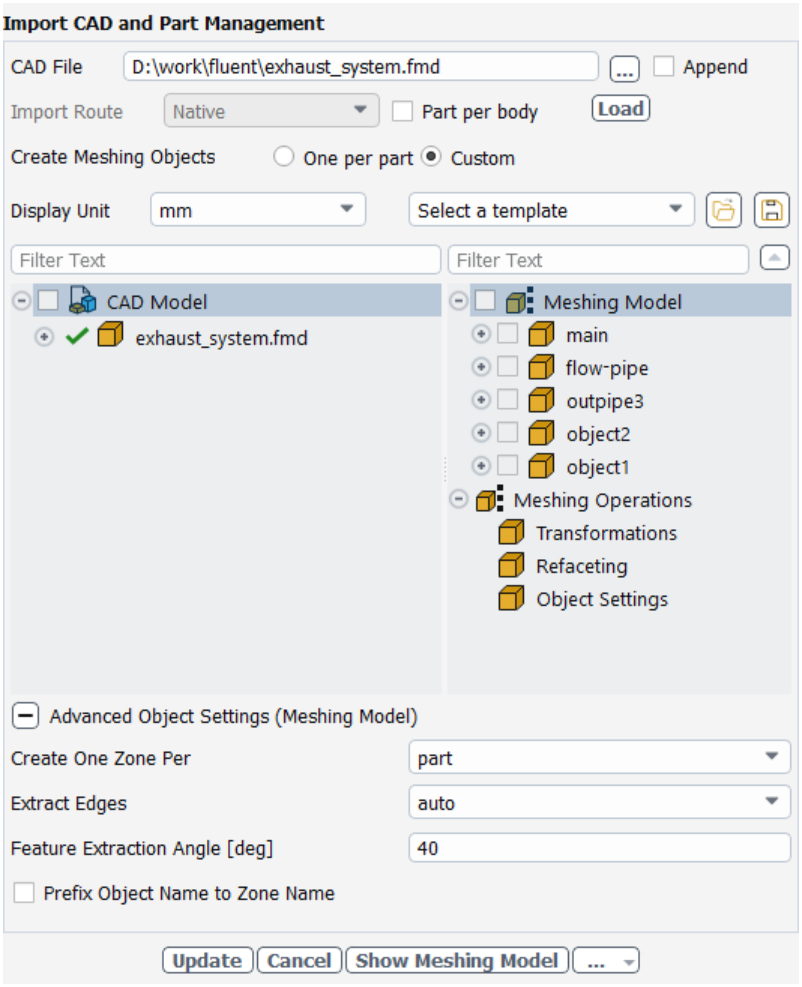
4.2.4.2. Creating Customized Meshing Objects

By setting the **Create Meshing Objects** field to **Custom**, you can customize the creation of your meshing objects, and selectively choose various properties, transformations, and refaceting options, for any portion of your imported CAD geometry.

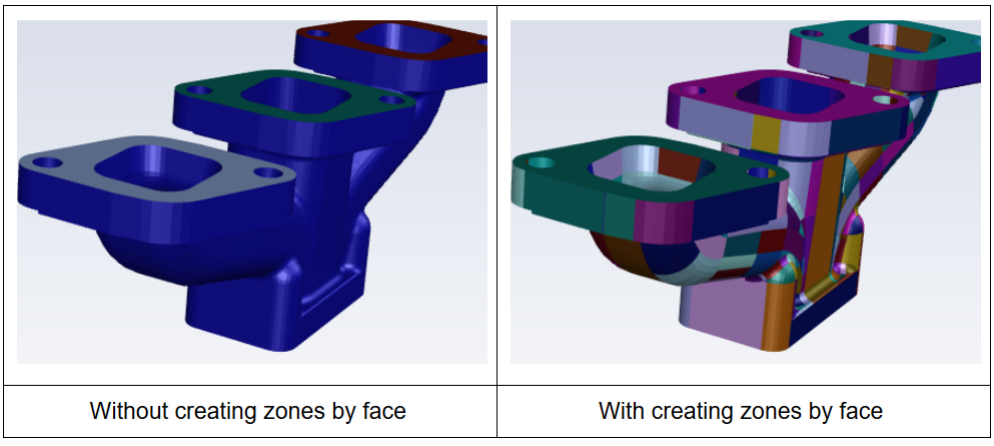
Properties for custom meshing objects are available under **Advanced Object Settings**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 4.6: Properties of a Selected Custom Meshing Model Object



- Set the **Create One Zone Per** option to **object**, **part**, **body**, or **face**, accordingly. For instance, if **face** is selected, each CAD surface or face will be assigned to a face zone.

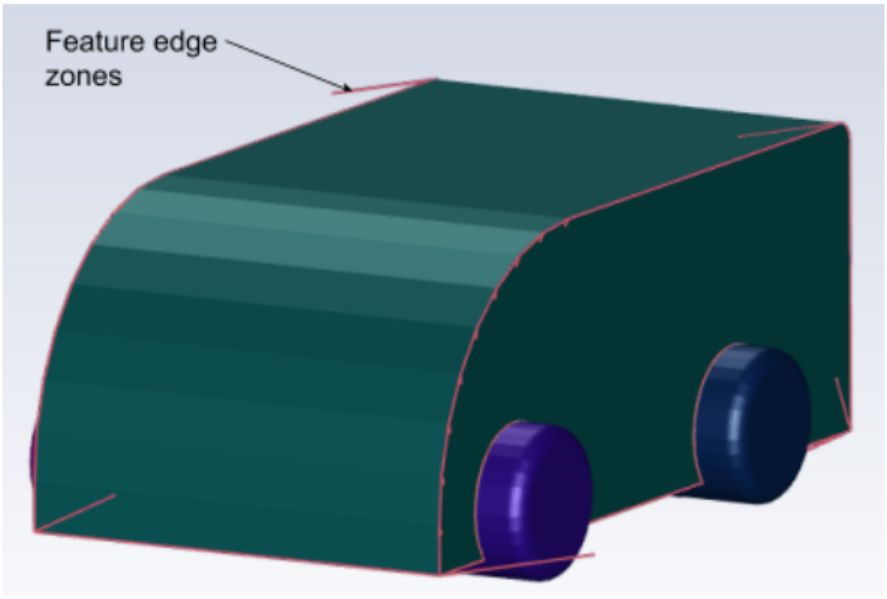


Without creating zones by face	With creating zones by face

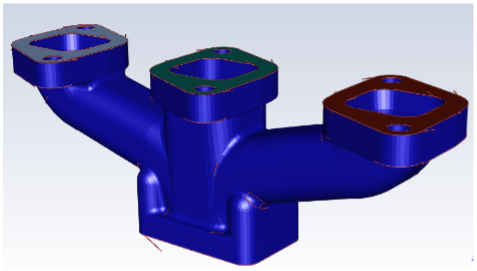
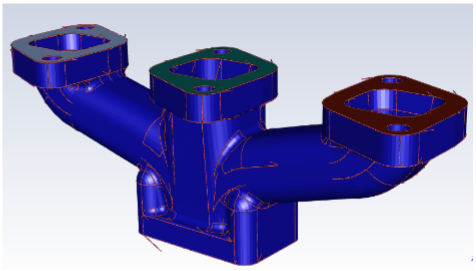
- Use the **Extract Edges** field to determine how edges will be extracted from the CAD geometry (**auto**, **yes**, or **no**). Setting this option to **auto** will extract edges from the CAD geometry when the number of meshing objects is less than 10,000. If the number of meshing objects exceeds this limit, then no edges are extracted. When this option is set to **yes**, then edges

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

are extracted regardless of the number of meshing objects. No edges are extracted when this option is set to **no**.



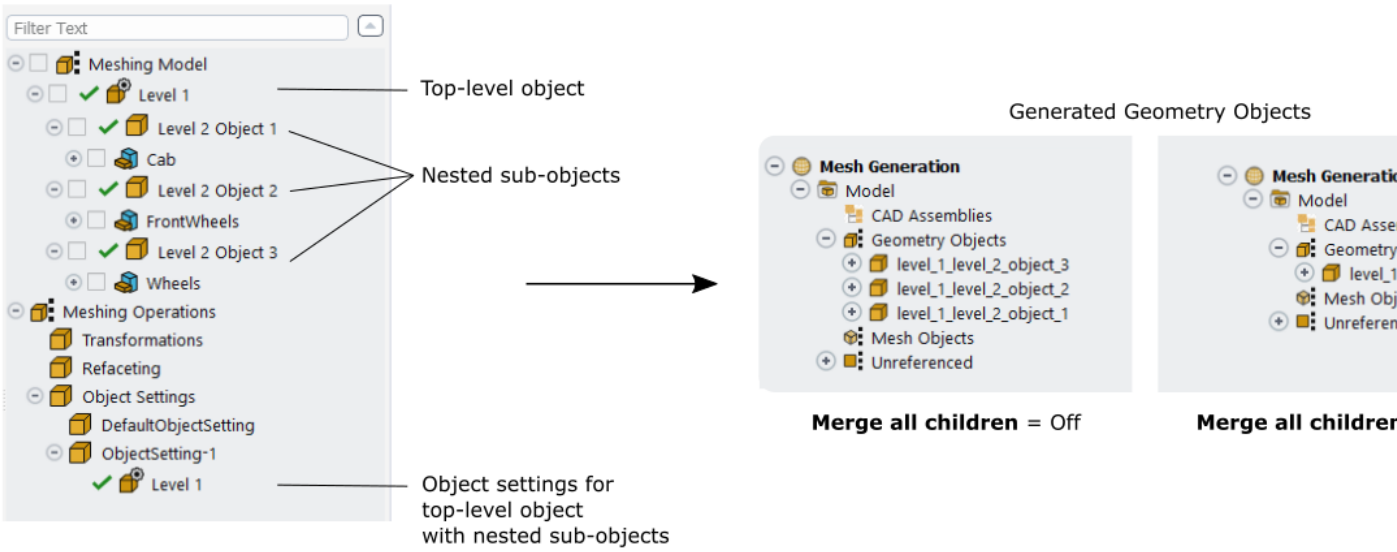
- When extracting edges, specify a value for the **Feature Extraction Angle** field, or use the default value of 40 degrees, to specify the angle at which features will be extracted from the CAD model on import. See Faceting Considerations (p. 422) for details.

	
Angle = 40 degrees	Angle = 4 degrees

- Enable or disable the **Prefix Object Name to Zone Name** accordingly. Enabling this allows for naming the corresponding face zone using the object name as a prefix. This option is available only for custom meshing object creation.

- Enable or disable the **Merge all children** accordingly. This option is only available when there are nested objects in the Meshing Model tree, and you want to assign object property settings to the top-level object. Upon updating the task, disabling this option ensures that each nested sub-object becomes its own geometry object, whereas enabling this option merges all child sub-objects into a single geometry object.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- Enable or disable the **Use default settings for sub-objects** accordingly. This option is only available when there are nested objects in the Meshing Model tree, and you want to assign object property settings to the top-level object. Upon updating the task, enabling this option applies the top-level (or the default) object settings to all child sub-objects, whereas disabling this option lets you define separate object settings for your individual sub-objects.

Note:

Use the **...** menu button at the bottom of the task, and select the **Save FMD** option to save your changes in a new `.fmd` file for future use. This is especially useful for particularly large, complicated models in which you have made many specialized changes to the properties (such as refaceting), and allows for a faster load time. Note that mirrored components cannot be saved to the `.fmd` format.

4.2.5. Performing Operations on Meshing Model Objects

You can perform specific operations on one or more meshing model object(s). The operations are grouped together at the bottom of the Meshing Model tree.

Note:

To remove all meshing operations at once, right-click the **Meshing Operations** node of the Meshing Model tree and select **Delete All** from the context menu.

Likewise, to update all meshing operations at once, right-click the **Meshing Operations** node of the Meshing Model tree and select **Update All** from the context menu.

4.2.5.1. Performing Transformation Operations on Meshing Model Objects
4.2.5.2. Performing Refaceting Operations on Meshing Model Objects
4.2.5.3. Performing Object Setting Operations on Meshing Model Objects
4.2.5.4. Listing Meshing Operations

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

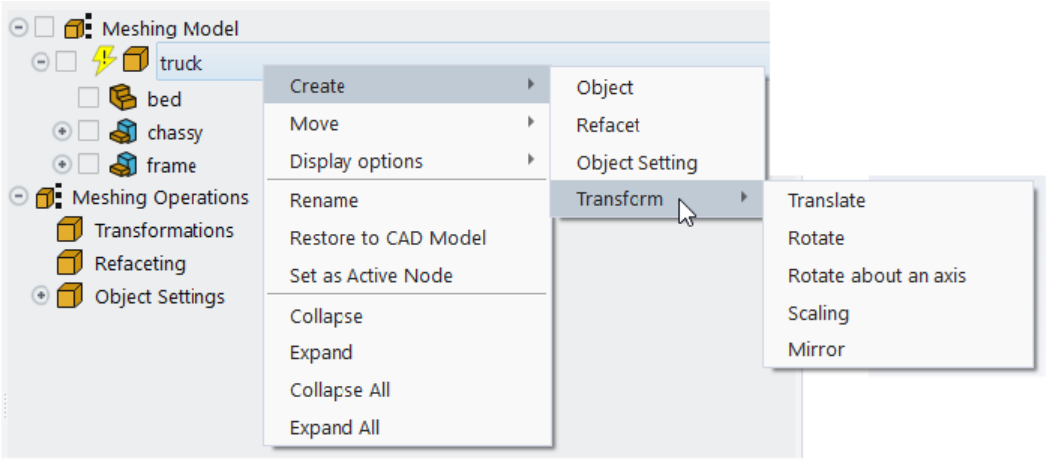
4.2.5.1. Performing Transformation Operations on Meshing Model Objects

After you have imported the original CAD model, there may be times when you need to slightly adjust or alter the geometry. You can make such modifications (such as simple Cartesian-based rotational and translational operations) on selected portions of your meshing model, prior to creating the meshing objects based on your imported CAD geometry.

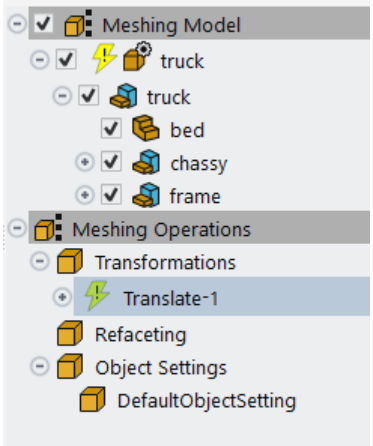
Note:

When capturing transformation operations in journal files, be aware that any unused translation and/or rotation input(s) will be recorded even if the transform type is changed prior to applying the transformation operation.

To create a transformation for a meshing model component, select one or more components from the Meshing Model tree,



and use the **Create > Transform** context menu option to select a transformational operation (such as translation, rotation, scaling, etc.) for the selected component(s). A corresponding transform operation appears in the Meshing Operations tree (under **Transformations**) with a corresponding default name (`Translate-1` for example). Any transformations you define are collected under **Meshing Operations** and can be performed alone, or combined with each others in a particular sequence if desired. Transformations are updated when the meshing objects are created with this task.



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In addition, you can use the context menu to **Delete**, **Add to Display**, **Display**, or **Rename** a transformation operation.

You can identify objects in the tree that have operations (such as transformations) applied to them since they will have a distinct icon (or , or).

You can create as many different transformations as you need. For a selected transformation object, use the context menu to perform other general operations such as renaming, deleting, etc.

Note:

To remove all meshing transformation operations at once, right-click on the **Transformations** node under the **Meshing Operations** node of the Meshing Model tree and select **Delete All** from the context menu.

Likewise, to update all meshing transformation operations at once, right-click on the **Transformations** node under the **Meshing Operations** node of the Meshing Model tree and select **Update All** from the context menu.

4.2.5.1.1. Applying a Translation Transformation

4.2.5.1.2. Applying a Rotation Transformation

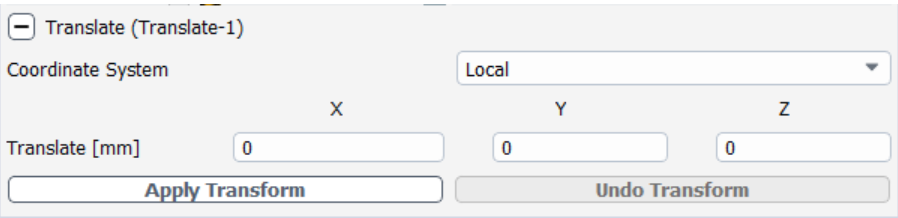
4.2.5.1.3. Applying a Rotation About an Axis Transformation

4.2.5.1.4. Applying a Scaling Transformation

4.2.5.1.5. Applying a Mirror Transformation

4.2.5.1.1. Applying a Translation Transformation

To create a translational transformation for a meshing model component, select one or more components from the Meshing Model tree, and choose **Translate** in the **Create Transform** context menu where you can then change the following properties:



1.

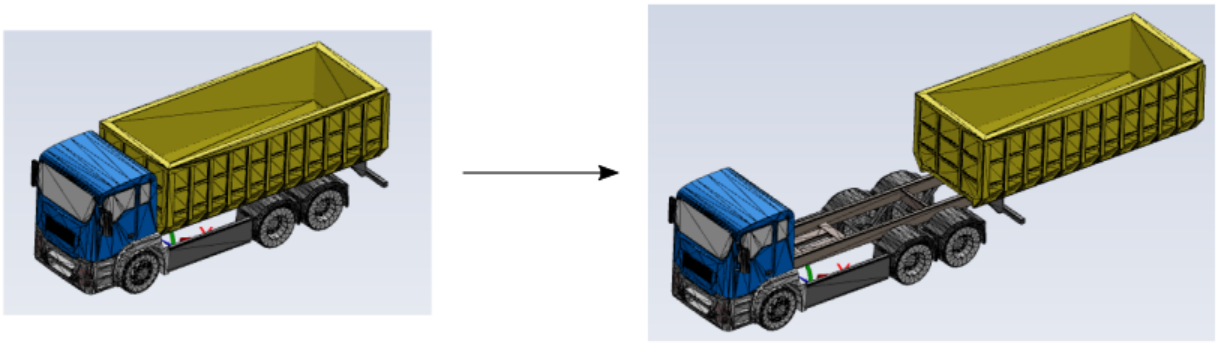
2. Assign the **Coordinate System** as either **Local** or **Global**.

   The local coordinate system is one that is associated with the individual CAD geometry as it is being created, where one chooses to re-orient that CAD sketch and create the model. Its orientation can vary based on its placement in an assembly of the entire model (for example, tires for a car). The global coordinate system has a fixed orientation, whereas the local coordinate system will have a different orientation. If, however, the CAD part was created without changing the coordinate systems, then there will be no difference between the local and global coordinate systems.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Use the **Translate** field to specify a translational transformation. Enter an appropriate value (in the display units) in the **X**, **Y**, and/or **Z** direction, based on the specified coordinate system.

Figure 4.7: Transformation Example: Truck Bed Translation Along the Global X Axis for 5 Meters



4. Click **Apply Transform** to see the results of your settings. Likewise, click **Undo Transform** to return to the original orientation.

4.2.5.1.2. Applying a Rotation Transformation

To create a rotational transformation for a meshing model component, select one or more components from the Meshing Model tree, and choose **Rotate** in the **Create Transform** context menu where you can then change the following properties:

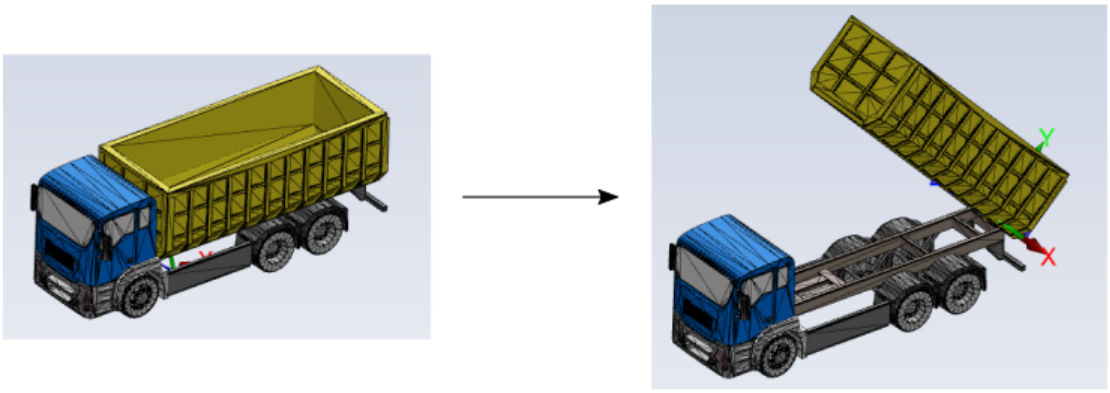
1. Assign the **Coordinate System** as either **Local** or **Global**.

   The local coordinate system is one that is associated with the individual CAD geometry as it is being created, where one chooses to re-orient that CAD sketch and create the model. Its orientation can vary based on its placement in an assembly of the entire model (for example, tires for a car). The global coordinate system has a fixed orientation, whereas the local coordinate system will have a different orientation. If, however, the CAD part was created without changing the coordinate systems, then there will be no difference between the local and global coordinate systems.

2. Use the **Rotate** field to specify a rotational transformation. Enter an appropriate value (in degrees) around the **X**, **Y**, and/or **Z** axis, based on the specified coordinate system.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

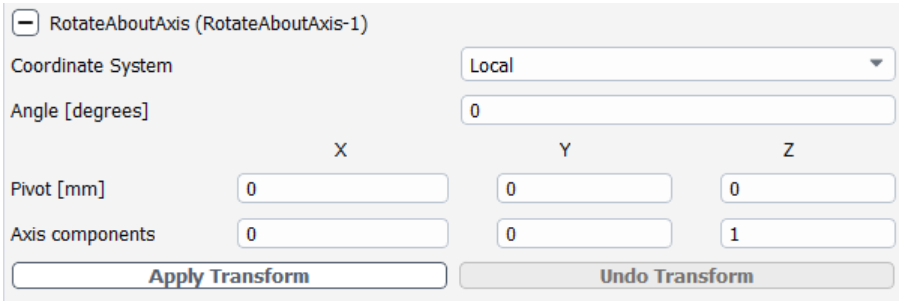
Figure 4.8: Transformation Example: Truck Bed Rotation of -45 Degrees About Local X Axis



3. Click **Apply Transform** to see the results of your settings. Likewise, click **Undo Transform** to return to the original orientation.

4.2.5.1.3. Applying a Rotation About an Axis Transformation

To create an axial rotational transformation for a meshing model component, select one or more components from the Meshing Model tree, and choose **Rotate About Axis** in the **Create Transform** context menu where you can then change the following properties:



1. Assign the **Coordinate System** as either **Local** or **Global**.

   The local coordinate system is one that is associated with the individual CAD geometry as it is being created, where one chooses to re-orient that CAD sketch and create the model. Its orientation can vary based on its placement in an assembly of the entire model (for example, tires for a car). The global coordinate system has a fixed orientation, whereas the local coordinate system will have a different orientation. If, however, the CAD part was created without changing the coordinate systems, then there will be no difference between the local and global coordinate systems.

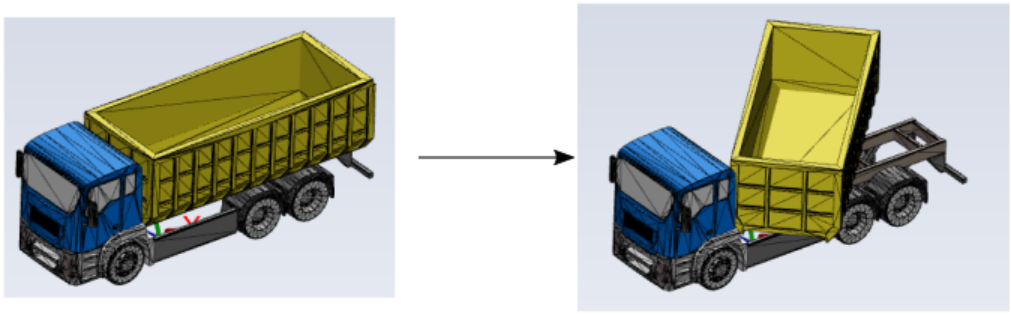
2. Use the **Angle** field to specify a rotational angle (in degrees) of transformation.

3. Specify the **Pivot** point of the rotation using an appropriate value for **X**, **Y**, and/or **Z**, based on the specified coordinate system.

4. Specify the **Axis components** values for the **X**, **Y**, and/or **Z**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 4.9: Transformation Example: Truck Bed Rotation of 45 Degrees With a Global Pivot in X About the Z Axis



5. Click **Apply Transform** to see the results of your settings. Likewise, click **Undo Transform** to return to the original orientation.

4.2.5.1.4. Applying a Scaling Transformation

To create a scaling transformation for a meshing model component, select one or more components from the Meshing Model tree, and choose **Scaling** in the **Create Transform** context menu where you can then change the following properties:

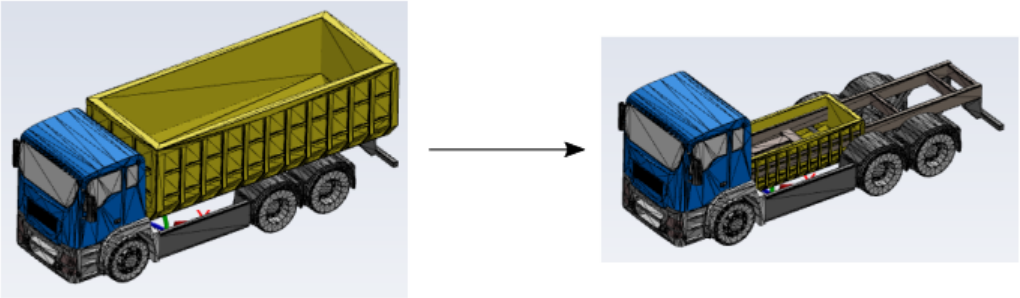
1. Assign the **Coordinate System** as either **Local** or **Global**.

   The local coordinate system is one that is associated with the individual CAD geometry as it is being created, where one chooses to re-orient that CAD sketch and create the model. Its orientation can vary based on its placement in an assembly of the entire model (for example, tires for a car). The global coordinate system has a fixed orientation, whereas the local coordinate system will have a different orientation. If, however, the CAD part was created without changing the coordinate systems, then there will be no difference between the local and global coordinate systems.

2. Use the **Scale factor** field to specify a factor to scale the part size. Enter an appropriate value around the **X**, **Y**, and/or **Z** axis, based on the specified coordinate system.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

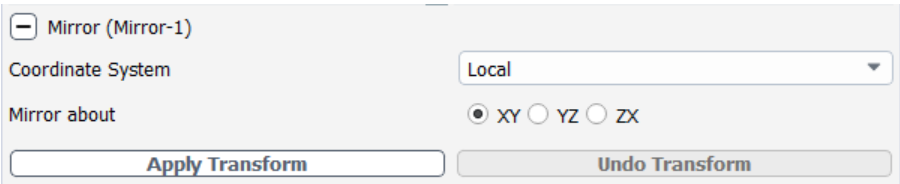
Figure 4.10: Transformation Example: Truck Bed Scaled by Half for Global X, Y, and Z



3. Click **Apply Transform** to see the results of your settings. Likewise, click **Undo Transform** to return to the original orientation.

4.2.5.1.5. Applying a Mirror Transformation

To create a mirroring transformation for a meshing model component, select one or more components from the Meshing Model tree, and choose **Mirror** in the **Create Transform** context menu where you can then change the following properties:



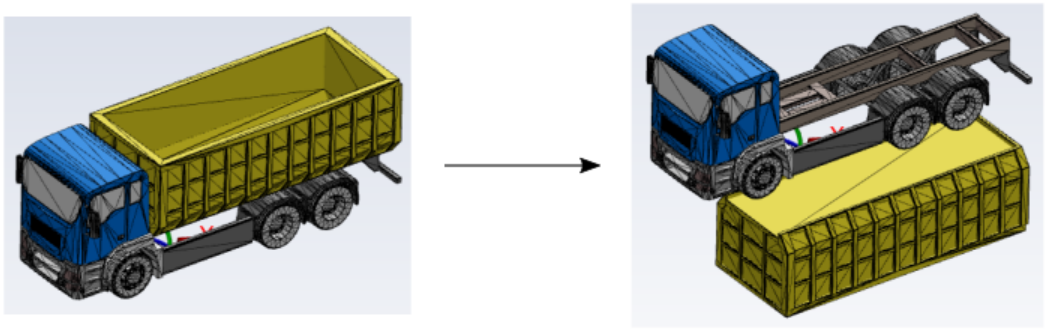
1. Assign the **Coordinate System** as either **Local** or **Global**.

   The local coordinate system is one that is associated with the individual CAD geometry as it is being created, where one chooses to re-orient that CAD sketch and create the model. Its orientation can vary based on its placement in an assembly of the entire model (for example, tires for a car). The global coordinate system has a fixed orientation, whereas the local coordinate system will have a different orientation. If, however, the CAD part was created without changing the coordinate systems, then there will be no difference between the local and global coordinate systems.

2. Use the **Mirror about** field to specify the mirroring plane as either the **XY** plane, the **YZ** plane, or the **XZ** plane, based on the specified coordinate system.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 4.11: Transformation Example: Truck Bed Mirrored About the Global XY Plane



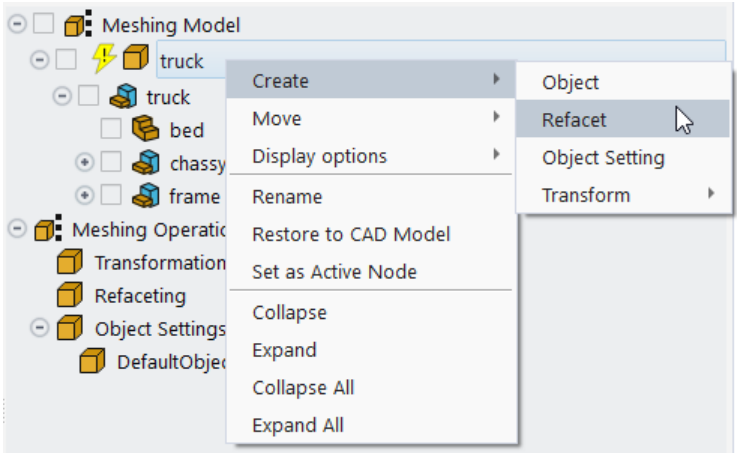
3. Click **Apply Transform** to see the results of your settings. Likewise, click **Undo Transform** to return to the original orientation.

4.2.5.2. Performing Refaceting Operations on Meshing Model Objects

You can perform additional faceting refinement for selected customized meshing object(s) by applying individual refaceting operations. Refaceting will refacet the original CAD geometry. Only the faceted CAD model is used during the meshing process. When refaceting operations are not used, normal CAD faceting is performed as defined by the imported CAD file(s). See Faceting Considerations (p. 422) for details.

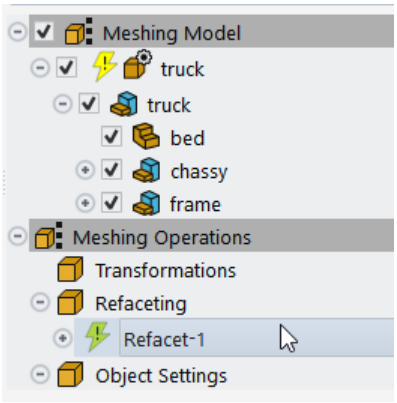
To create a refaceting operation for a meshing model component:

1. Select one or more components from the Meshing Model tree.



2. Use the **Create > Refacet** context menu option to create a refacet operation for the selected component(s). A refacet operation appears in the Meshing Operations tree (under **Refaceting**) with the default name of `Refacet-1`.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

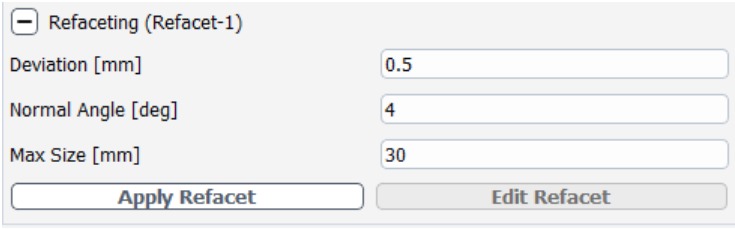


You can use the context menu to **Delete**, **Add to Display**, **Display**, or **Rename** a refacet operation.

You can identify objects in the tree that have operations (such as refaceting) applied to them since they will have a distinct icon (or , or).

You can create as many different refaceting operations as you need. For a selected object, use the context menu to perform other general operations such as renaming, deleting, etc.

3. Select `Refacet-1` and expand the **Refaceting** properties (the +/- icon below the tree) to assign specific properties to the refaceting operation.



4. The **Deviation** controls how far facet edges are away from the model.

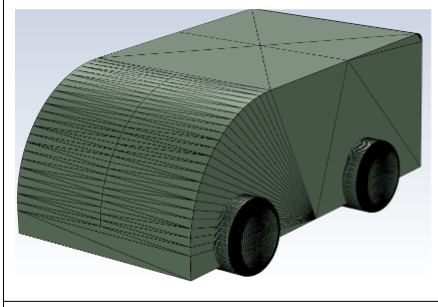
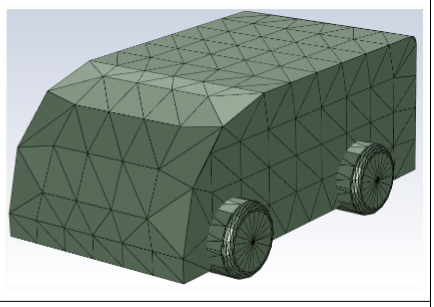
5. The **Normal Angle** also controls how far facet edges are away from the model. Decreasing the normal angle will result in more facets along the curved edges.

6. The **Max Size** controls the maximum size of the facets after the deviation and the normal angle are respected.

7. Click **Apply Refacet** to see the results of your settings and to preview changes to the refaceting settings in the graphics window while you are still within the task.

   In the following example, the left-hand model uses small values for the deviation and the normal angle in order to capture the curved surface, and a large value for the maximum size in order to get larger facets on the flat surface. The right-hand model uses larger values for the deviation and the normal angle in order to capture the curved surface coarsely, and a smaller value for the maximum size in order to capture the flat surface evenly.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	
Deviation=0.5, Normal Angle = 4, Max Size=30	Deviation=2, Normal Angle = 30, Max Size=8

Refaceting will refacet the original CAD geometry. Only the faceted CAD model is used during the meshing process. In either case, when disabled, normal CAD faceting is performed as defined by the imported CAD file(s).

Click **Edit Refacet** to return to a state where you can change values and repeat as needed for as many objects in the tree as required.

Note:

To remove all meshing refaceting operations at once, right-click on the **Refacet-ing** node under the **Meshing Operations** node of the Meshing Model tree and select **Delete All** from the context menu.

Likewise, to update all meshing refaceting operations at once, right-click on the **Refaceting** node under the **Meshing Operations** node of the Meshing Model tree and select **Update All** from the context menu.

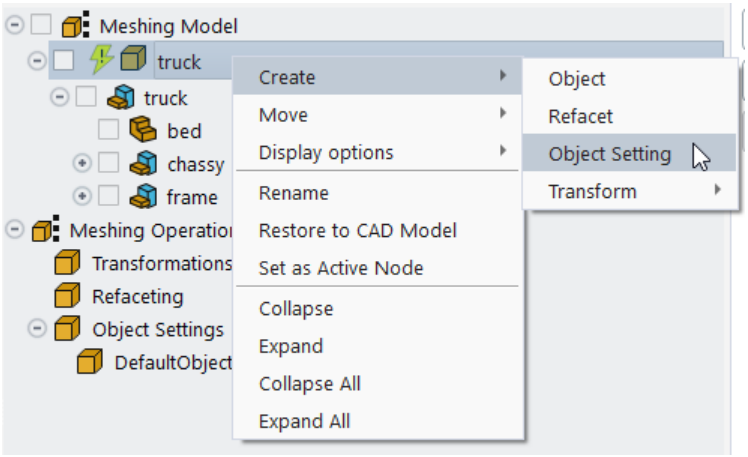
4.2.5.3. Performing Object Setting Operations on Meshing Model Objects

You can perform basic object property settings on multiple meshing model objects all at once (properties such as zone assignment, edge extraction, feature extraction angle, etc. as described in Creating Customized Meshing Objects (p. 407)).

To create a object setting operation for a meshing model component:

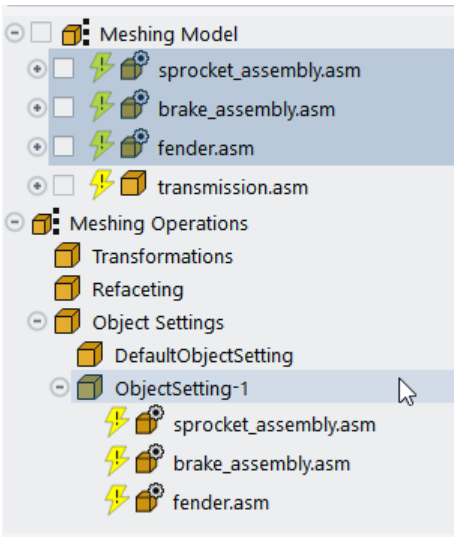
1. Select one or more components from the Meshing Model tree.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Alternatively, you can select the assembly's parent node in the tree if you want to apply the same object settings to all of its children.

2. Use the **Create Object Setting** context menu option to create a object setting operation for the selected component(s). An object setting operation appears in the Meshing Operations tree (under **Object Settings**) with the default name of `ObjectSetting-1`.



You can use the context menu to **Delete**, **Add to Display**, **Display**, or **Rename** one or more selected object settings. The **Add to Display** option allows you to visualize one or more selected object settings from the Meshing Model tree in the graphics window.

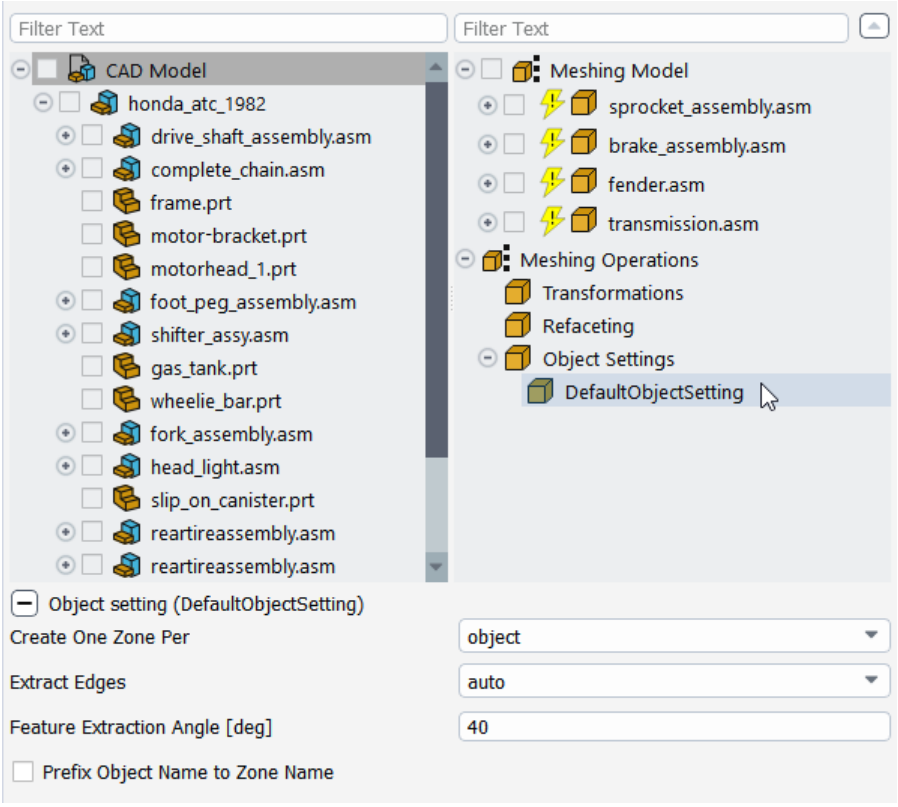
You can identify objects in the tree that have operations (such as object settings) applied to them since they will have a distinct icon (or , or).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can create as many different object setting operations as you need. For a selected object, use the context menu to perform other general operations such as renaming, deleting, etc.

Note:

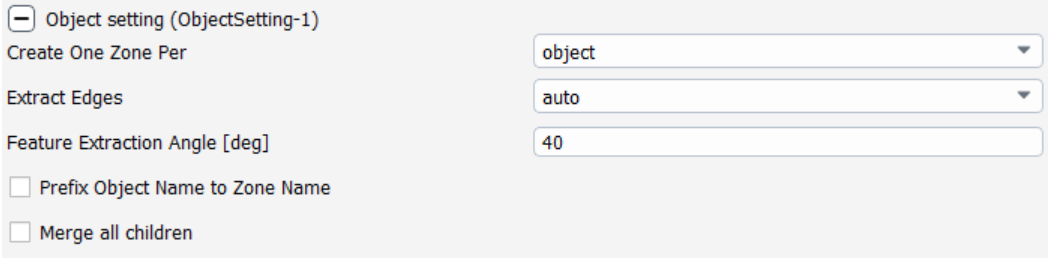
As part of the **Object Settings**, the **DefaultObjectSetting** node is available if you want to specify and apply a common set of object properties to several meshing objects at the same time.



The **DefaultObjectSetting** node is only for editing its properties to apply them to one or more other meshing objects, and cannot be renamed or removed.

---

3. Select `ObjectSetting-1` and expand the **Object Settings** properties (the +/- icon below the tree) to assign specific properties to the object setting operation.



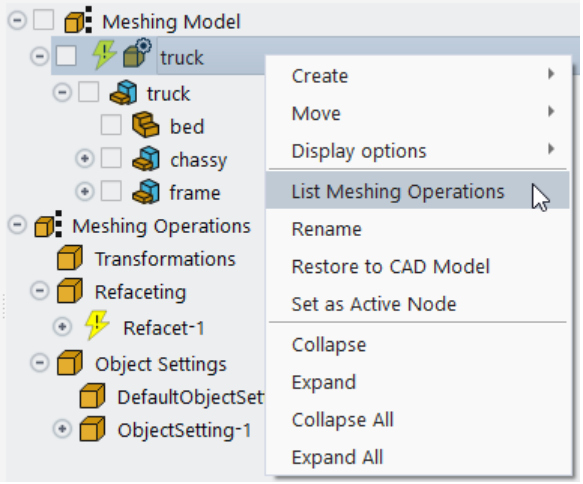
4. Any of the provided properties can be set and applied to all of the selected parts in the Meshing Model tree using a single object setting. Properties of meshing objects from pre-

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

vious mesh files read into the workflow can still be edited under **Advanced Object Settings**. The available properties are described in Creating Customized Meshing Objects (p. 407).

4.2.5.4. Listing Meshing Operations

If an object has any meshing operations associated with it, use the **List Meshing Operations** option to list those meshing operations in the console window.



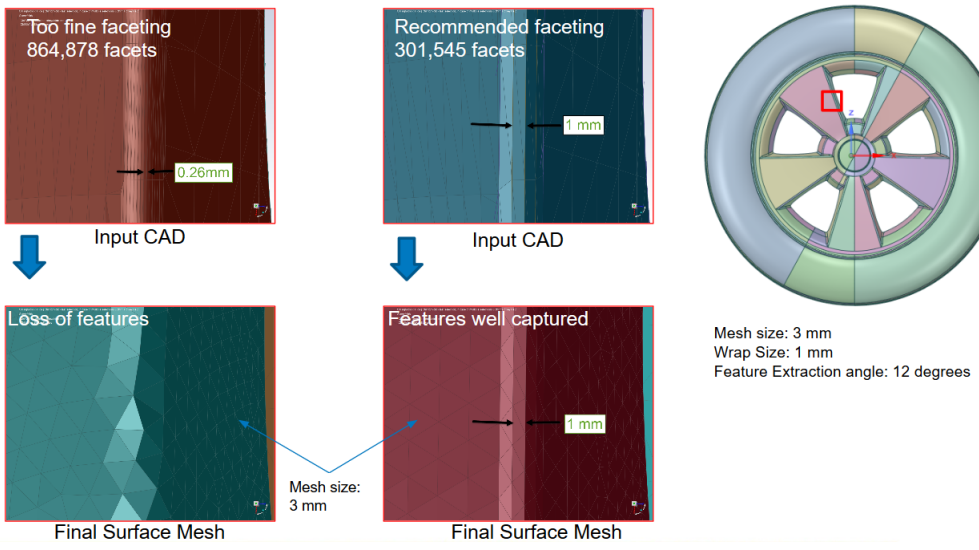
The following is an example of output to the console window:

```
Meshing operations on meshing object: bed are as follows:
Transformations: (Transform-1)
Refaceting: Refacet-2
```

4.2.6. Faceting Considerations

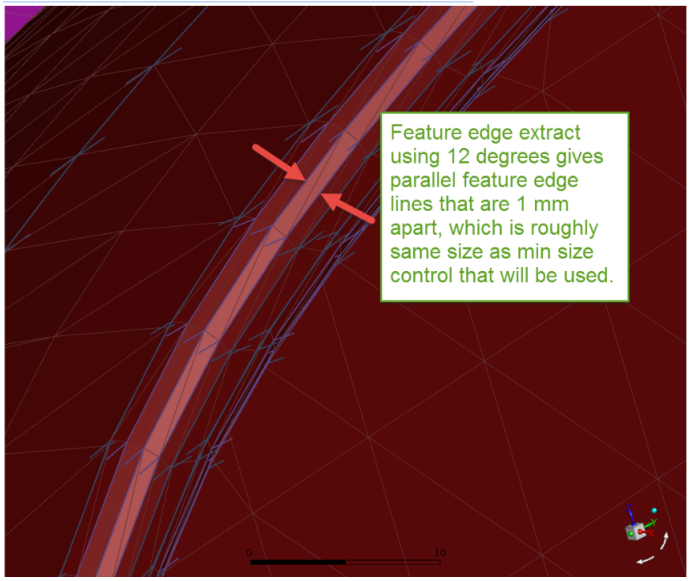
Faceting is very important for many reasons: if the faceting is too coarse, you will lose fidelity; if the faceting is too fine, then you will not capture important features in rounds or fillets.

Figure 4.12: Faceting Example



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

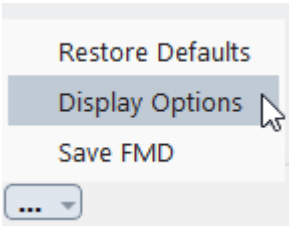
Here the minimum facet edge in the fillet is roughly the same size as the minimum local size control to be used. Once feature edges are extracted using 12 degrees, you will have:



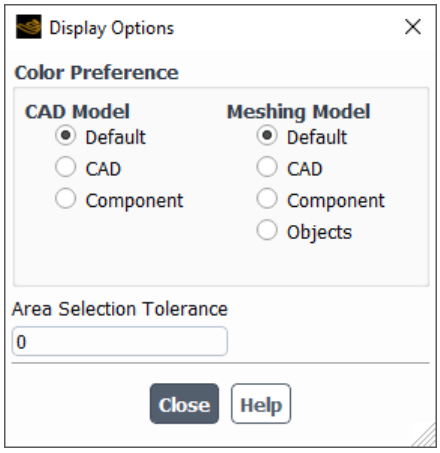
This will lead to faster graphics operation (due to lighter model), faster meshing, and better feature capturing. Even if the target size becomes larger (not more than 2 times), it will still respect those feature lines, leading to anisotropic mesh, and reduced cell count.

4.2.7. Setting Display Options for CAD Model and Meshing Objects

To change how different components are colored and displayed in the graphics window, use the **...** menu button at the bottom of the task, and select **Display Options**.



This displays the **Display Options** dialog:



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Display Options** dialog allow you to select color preferences for the displayed components of the CAD Model tree or the Meshing Model tree. Options include to color by **Default** (automatic), by **CAD**, by **Component**, or by **Objects**. You can also provide an **Area Selection Tolerance**.

4.2.8. Using Hot Key Shortcuts in the Model Trees and the Graphics Window

In addition to the hot key shortcuts that are available in Fluent in meshing mode (Appendix C: Shortcut Keys (p. 877)), the following selection and display hot key shortcuts are available in the **Import CAD and Part Management** task:

	Action	**Keys**
Selection Tools	Select component instances	**Ctrl**+**I**
	Select parent assembly	**Ctrl**+**P**
	Clear selections	**f2**
	Clear CAD Model tree selections	**Ctrl**+**Y**
Display Mode Tools	Preview the other display mode	**Ctrl**+**J**
	Disable the preview of the other display mode	**Ctrl**+**K**
	Toggle between the preview of the other display mode	**Ctrl**+**L**
Display Tools	Show all	**Ctrl**+**Shift**+**A**
	Show/hide edges	**Ctrl**+**Shift**+**E**
	Hide	**Ctrl**+**Shift**+**H**
	Isolate	**Ctrl**+**Shift**+**I**
Miscellaneous Tools	Compute distance	**Ctrl**+**D**
	Print this help	**Ctrl**+**H**
	Position filter	**Ctrl**+**X**
	Toggle right mouse button	**f3**
	Toggle selection method	**f4**
	Auto scale	**Ctrl**+**A**
	CAD object filter	**Ctrl**+**B**
	Move selected component(s) into a new object in the Meshing Model tree,	**Alt**+**M**

4.3. Describing the Geometry and the Flow

The **Describe Geometry and Flow** task lets you specify the type of geometry you have and the type of flow you are trying to simulate. Settings in this task determine what sub-tasks are available for the overall task.

1. For the **Flow Type**, choose from the following:

   - If you are going to be simulating external flow over or around an object (such as a truck, car, bicycle, etc.), select **External flow around object** and a **Create External Flow Boundaries** task is added to the workflow (see Creating External Flow Boundaries (p. 429)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you are going to be simulating flow within an object (such as a manifold, or catalytic converter, etc.), select **Internal flow through the object** and an **Enclose Fluid Regions (Capping)** task is added to the workflow (see Enclosing Fluid Regions (p. 426)).

- If you are going to be simulating flow over and around an object, as well as within an object, select **Both external and internal flow** and a **Create External Flow Boundaries** task and an **Enclose Fluid Regions (Capping)** task will be added to the workflow (see Enclosing Fluid Regions (p. 426) and Creating External Flow Boundaries (p. 429)).

2. Under **Standard Options**, you can access typical enhancements to your workflow. Options include:

   - For internal flows, for **Create large caps?**, indicate whether or not you will need to cover, or cap, any openings in your geometry (inlets and outlets, for example). If you select **Yes**, then an **Enclose Fluid Regions (Capping)** task is added to the workflow (see Enclosing Fluid Regions (p. 426)).

   - For external flows, for **Add an enclosure?**, indicate whether or not you will require any enclosures to capture external flow regions. If you select **Yes**, then a **Create External Flow Boundaries** task is added to the workflow (see Creating External Flow Boundaries (p. 429)).

   - For **Add local refinement?**, indicate whether or not your geometry will require any local refinement regions. If you select **Yes**, then a **Create Local Refinement Regions** task is added to the workflow (see Creating Local Refinement Regions (p. 431)), and when, in that task you select **Offset Surface** as the refinement region type, local refinement face zones are retained. If you select **No**, any existing unreferenced face zones will be removed.

   - For **Identify construction surfaces?**, indicate whether or not your geometry contains any predefined construction surfaces (such as cylinders for moving reference frames, or capping surfaces, etc.). If you select **Yes**, then an **Identify Construction Surface** task is added to the workflow (see Identifying Construction Surfaces (p. 436)).

3. Under **Advanced Options**, you can access more advanced tasks that can be added to the workflow. Options include:

   - For **Identify regions?**, indicate whether or not you need to specifically identify and analyze particular regions. If you select **Yes**, then an **Identify Regions** task is added to the workflow (see Identifying Regions (p. 450)).

   - For **Close leakages?**, indicate whether or not your geometry contains any problems (such as gaps or overlapping/intersecting surfaces) that may create leakages that need to be closed. If you select **Yes**, then a **Define Leakage Threshold** task is added to the workflow (see Defining Leakage Thresholds (p. 452)).

   - For **Add thickness to any walls?**, indicate whether or not your geometry contains any zones with zero thickness. If you select **Yes**, then an **Add Thickness** task is added to the workflow (see Adding Thickness to Your Geometry (p. 443)).

   - For **Extract edge features?**, indicate whether or not your geometry will require any form of edge or feature extraction. If you select **Yes**, then an **Extract Edge Features** task is added to the workflow (see Extracting Edge Features (p. 439)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For **Include porous regions?**, indicate whether or not your geometry contains any porous regions. If you select **Yes**, then a **Create Porous Regions** task is added to the workflow (see Creating Porous Regions (p. 446)).
- For **Enable overset options?**, indicate whether or not your utilize any overset meshing options. If you select **Yes**, then a **Describe Overset Features** task (Describing Overset Features (p. 471)), a **Create Overset Mesh** task (Creating Overset Mesh Interfaces (p. 485)), and an **Identify Orphans** task (Identifying Orphans (p. 485)) are added to the workflow.

  Note:

  For more information about overset meshes and using them with Fluent in solution mode, see Overset Meshes (p. 1206)

4. Once your selections are made, click **Describe Geometry and Flow** and proceed onto the next task.

   If you need to make adjustments to any of your settings in this task, click **Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

4.4. Enclosing Fluid Regions

For internal flow simulations with solid model geometries, you can use the **Enclose Fluid Regions (Capping)** task to cover, or cap, any openings in your geometry in order to later calculate your fluid region(s).

1. Specify a **Name** for the capping surface.
2. Choose the **Zone Type** for the new cap. Choices include:
   - velocity-inlet
   - pressure-outlet
   - pressure-inlet
   - pressure-far-field
   - mass-flow-inlet
   - outflow
   - symmetry
   - wall

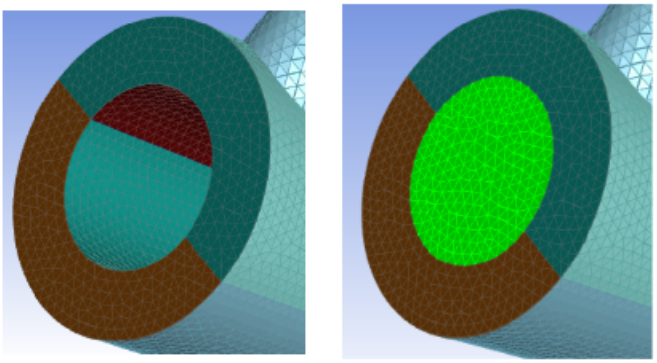
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Choose whether the **Cap Type** will be based on a single surface opening, or if it is an annular opening with two surfaces.

Note:

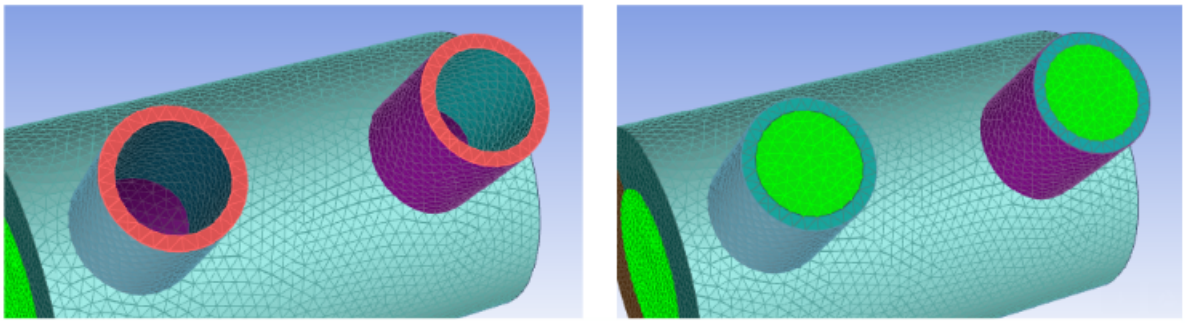
A single surface opening can potentially have more than one face comprising the single surface:

Figure 4.13: Example of a Single Surface Cap with Multiple Faces



A single surface opening can also mean selecting multiple, distinctly separate, surfaces to create a single cap:

Figure 4.14: Example of a Single Surface Cap with Multiple Faces

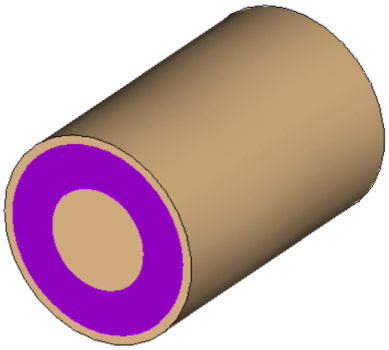


Note:

A single named selection using both the inner and outer face is the only valid selection for annular capping using labels.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

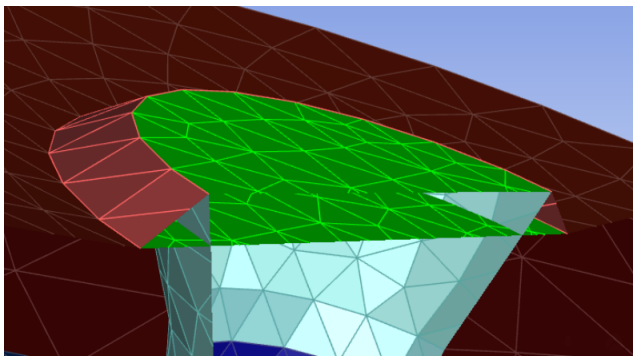
Figure 4.15: Example of an Annular Cap Type



Note:

The workflow does not support tilted annular faces for capping (highlighted in red, below). Currently, two caps are created (highlighted in green, below) and the resulting intersecting cap must be manually removed,

Figure 4.16: Example of a Problematic Tilted Annular Opening



4. Choose whether to **Select By** the label or zone name in the list below.

Note:

Labels originate from the CAD geometry, such as from group names in SpaceClaim geometries, or from named selections in DesignModeler geometries.

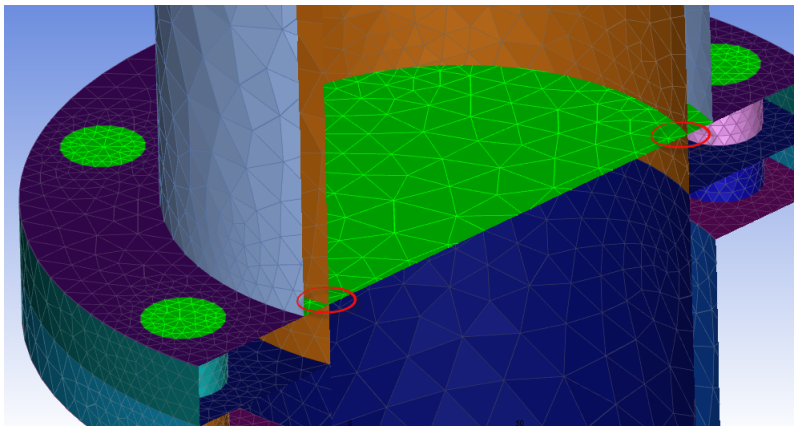
Select items in the list, or use the **Filter Text** option in the drop-down to provide text and/or regular expressions in filtering the list (for example, using *, ?, and []). You can also choose the **Use Wildcard** option in the drop-down to provide wildcard expressions in filtering the list. When you use either ? or * in your expression, the matching list item(s) are automatically selected in the list. Use ^, |, and & in your expression to indicate boolean operations for NOT, OR, and AND, respectively. See Filtering Lists and Using Wildcards (p. 287) for more information.

5. Click **Advanced Options** to access additional controls prior to performing this task. Options include:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Use the **Check Cap Self-Intersection?** option to control whether or not the system will detect if the capping surface intersects with any other face in the model. If an intersected face is found, it is automatically deleted. To increase the efficiency of the capping task, this option should be set to **no**.

  Figure 4.17: Example of a Self-Intersection: Additional Cap Intersects With Other Surfaces

- Use the **Max Cap Edge Count Limit** option to control the number of edges that can be present on the capping surface.

6. Click **Add Cap(s)**. The new capping object will appear in the workflow and in the graphics window.

   In cases where a face has multiple holes, and only a portion of them will be used for fluid extraction, Fluent will cap all holes on the face, and will only use the capped openings that are required for extracting the fluid region.

   If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

7. Repeat as needed for additional inlets, outlets, etc., until all openings have been assigned a type and have been created.

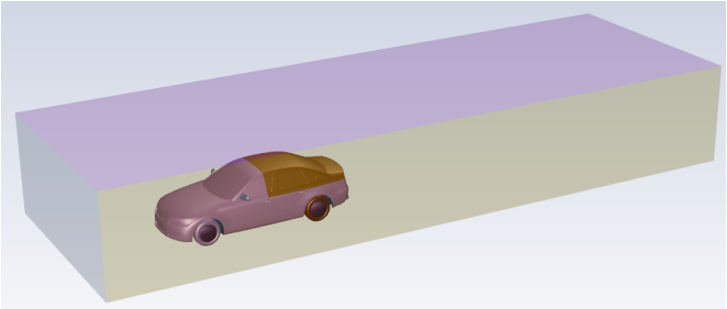
8. Once all openings have been covered, proceed to the next step in the workflow.

4.5. Creating External Flow Boundaries

When simulating external flow around an object, your geometry may not contain a fully defined external flow boundary. You can manually create an external flow boundary in this task, by defining a bounding box that surrounds the relevant aspects of your geometry, such as around a vehicle.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 4.18: An Example of an External Flow Boundary



1. Specify a **Name** for the external boundary, or use the default name (`tunnel`).

2. For **Method**, select **Create new boundary** if you are creating a new external flow boundary, or select **Use existing boundary** if the external flow boundary is already defined as part of your imported geometry.

3. For the **Coordinate Specification Method** prompt (when you select **Create new boundary** as the **Method**), you can choose from the following options:

   - Use the **Ratio relative to geometry size** option so that you can define the external flow boundary based on the relative size of selected object(s) or zone(s).

   - Use the **Directly specify coordinates** option so that you can explicitly define the location and dimension of the external flow boundary without having to select object(s) or zone(s).

4. For the **Extraction Method**, make a selection based on whether you would like to extract the external flow region either as a **surface mesh** object (a direct surface remesh of the object) or a **wrap** object. The object setting is applied later when generating the surface mesh.

5. For new external flow regions, additional settings are required.

   - When you select **Ratio relative to geometry size**, choose whether to **Select By** the object, label, or zone name in the list below.

     Select items in the list, or use the **Filter Text** option in the drop-down to provide text and/or regular expressions in filtering the list (for example, using *, ?, and []). You can also choose the **Use Wildcard** option in the drop-down to provide wildcard expressions in filtering the list. When you use either ? or * in your expression, the matching list item(s) are automatically selected in the list. Use ^, |, and & in your expression to indicate boolean operations for NOT, OR, and AND, respectively. See Filtering Lists and Using Wildcards (p. 287) for more information.

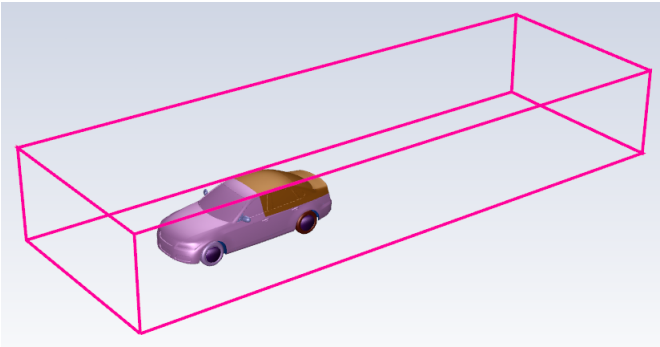
     Once one or more object(s) or zone(s) is selected, you can define the extents of the external boundaries around the selection by defining a bounding box.

     The **Box Parameters** hold the minimum and maximum extension ratios for the X, Y, and Z dimensions. By default, they are set to extend the dimensions of a bounding box around your selected object(s) or zone(s) out by a factor equivalent to the total singular length of the geometry in the X, Y, and Z directions. You can change the values according to your needs, and the display in the graphics window will change accordingly.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

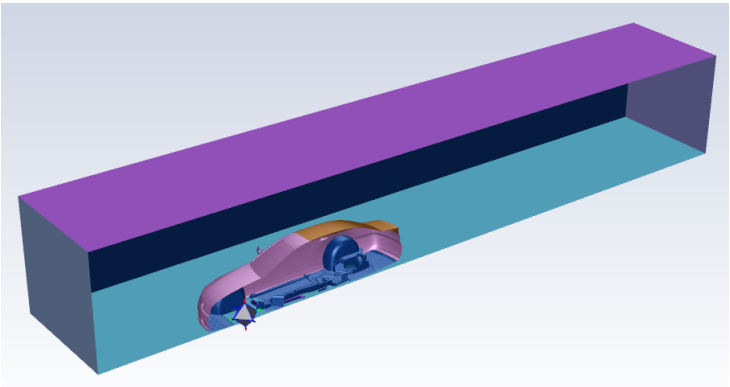
- When you select **Directly specify coordinates**, the **Box Parameters** hold the minimum and maximum extension *distance* for the X, Y, and Z dimensions. By default, they are set to values that correspond to the extension ratios, but you can set them to more suitable values as required, and the display in the graphics window will change accordingly.

Figure 4.19: An External Flow Bounding Box



6. Once your selections are made, click **Create External Flow Boundaries** and proceed onto the next task.

Figure 4.20: An External Flow Boundary Around a Car



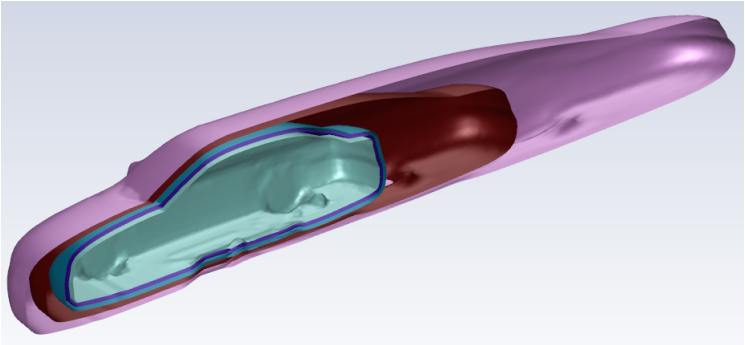
If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

4.6. Creating Local Refinement Regions

When simulating flow within and around an object, your geometry may not contain a fully defined refinement region, or body of influence (BOI). If a volume region needs mesh refinement, that region can be identified by watertight surface. You can manually create such a surface in this task by: defining a bounding box around one or more selected objects; using one or more existing objects that have already been imported; or creating an offset surface or box. For example, this volume region, which surrounds the relevant aspects of your flow physics, can be used to as an alternative to refine the mesh in a particular area, such as the wake region behind a vehicle.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 4.21: An Example of a Refinement Region Around a Car



1. For **Type**, choose from the following:

   - Select **Box** to define a bounding box for the refinement region.
   - Select **Existing** if the refinement region is already defined as part of your imported geometry.
   - Select **Offset Surface** if the refinement region will be offset from the surface of an existing, selected object or zone, based on the boundary layer.
   - Select **Offset Box** if the refinement region will be offset from the surface of an existing, selected object or zone, based on a scaled bounding box.

2. Specify a **Name** for the body of influence, or use the default name (`local-refinement-1`).

3. For the **Coordinate Specification Method** prompt (when you select **Box** as the **Type**), you can choose from the following options:

   - Use the **Ratio relative to geometry size** option so that you can define the refinement region based on the relative size of selected object(s) or zone(s).
   - Use the **Directly specify coordinates** option so that you can explicitly define the location and dimension of the refinement region without having to select object(s) or zone(s).

4. Specify a value for the **Mesh Size** for the refinement region.

5. Choose whether to **Select By** the object, label, or zone name in the list below.

   Select items in the list, or use the **Filter Text** option in the drop-down to provide text and/or regular expressions in filtering the list (for example, using *, ?, and []). You can also choose the **Use Wildcard** option in the drop-down to provide wildcard expressions in filtering the list. When you use either ? or * in your expression, the matching list item(s) are automatically selected in the list. Use ^, |, and & in your expression to indicate boolean operations for NOT, OR, and AND, respectively. See Filtering Lists and Using Wildcards (p. 287) for more information.

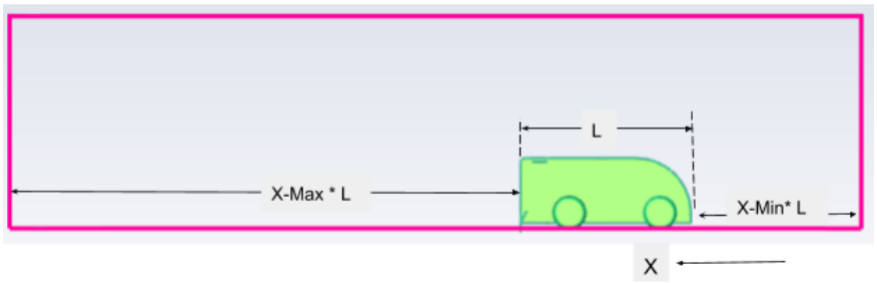
6. Once one or more object(s) or zone(s) is selected, you can define the extents of the refinement region around the selection by defining a bounding box.

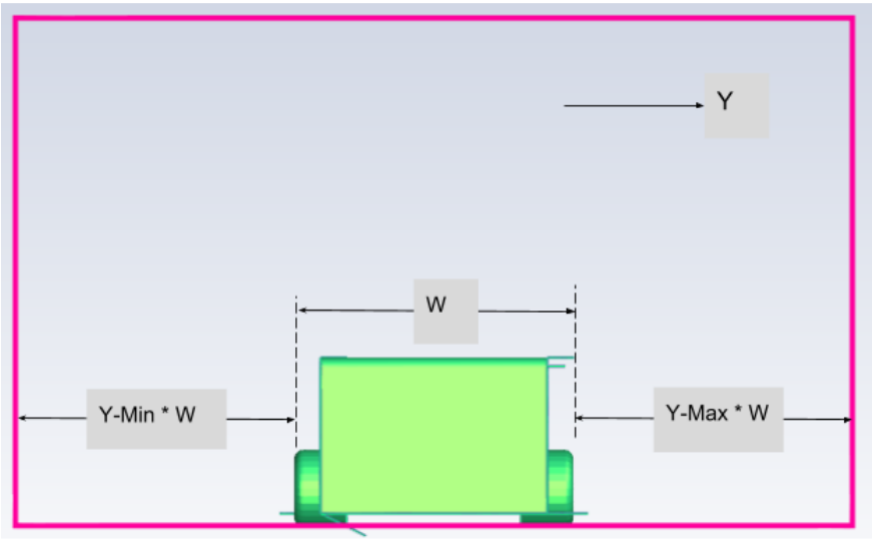
7. Depending on the **Type** of refinement region, additional parameters are required.

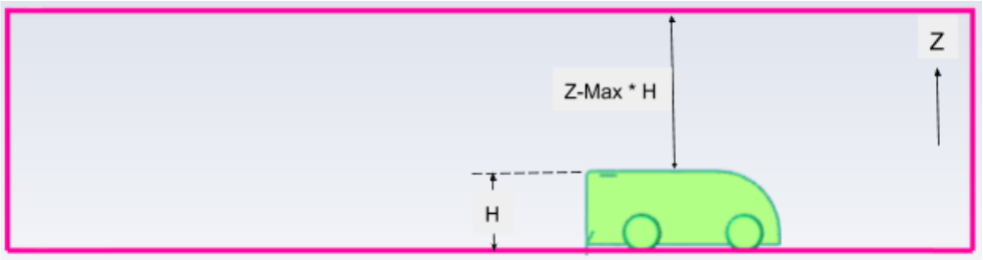
   - For a **Box** refinement region:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– When you select **Ratio relative to geometry size**, the **Box Parameters** hold the minimum and maximum extension ratios for the X, Y, and Z dimensions. By default, they are set to extend the dimensions of a bounding box around your selected object(s) or zone(s) out by a factor equivalent to the total singular length of the geometry in the X, Y, and Z directions. You can change the values according to your needs, and the display in the graphics window will change accordingly.







– When you select **Directly specify coordinates**, the **Box Parameters** hold the minimum and maximum extension *distance* for the X, Y, and Z dimensions. By default, they are set to values that correspond to the extension ratios, but you can set them to more suitable values as required, and the display in the graphics window will change accordingly.

- For an **Offset Surface** refinement region:

  – Specify the **Flow Direction**. It is used to determine the wake. The wake is downstream of the selected geometry object(s).

  – Use the **Flip** option if the flow direction is in opposite direction.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– Specify a value for the **Defeaturing Size**, or use the default value. This will obtain the rough shape of the selected object(s), which are wrapped using this size.

– Specify a value for the **Total Thickness**, or use the default value. This is how far from the selected object(s) the rough shape is offset.

– Specify a value for the **Boundary Layer Levels**, or use the default value, to determine the number of offset surfaces to be created.

– Specify a value for the **Wake Growth Factor**, or use the default value. Each scaled offset surface is grown in size in the wake direction by this factor from the previous scaled offset.

– Specify a value for the **Cross Wake Growth Factor**, or use the default value. Each scaled offset surface is grown in size in the cross-wake direction by this factor from the previous scaled offset.

– Specify a value for the **Wake Levels**, or use the default value. The rough shape is scaled in the wake direction as well as in the cross-wake direction. This value determines how many scaled offsets are to be created.

For example, given an automobile geometry, the base is the surface on the vehicle, which has maximum size of 8 mm. The following size control settings were automatically generated using **Boundary Layer Levels** set to **2** and **Wake Levels** set to **4**:

Name	Size Function	Max Size
name	soft	8
name-1	boi	16
name-2	boi	32
name-wake-1	boi	64
name-wake-2	boi	128
name-wake-3	boi	256
name-wake-4	boi	512

The refinement regions and the various settings can be visualized using Figure 4.22: An Example of Multiple Refinement Regions Around a Car (p. 435) and Figure 4.23: An Example of Multiple Refinement Regions Around a Vehicle (p. 435).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 4.22: An Example of Multiple Refinement Regions Around a Car

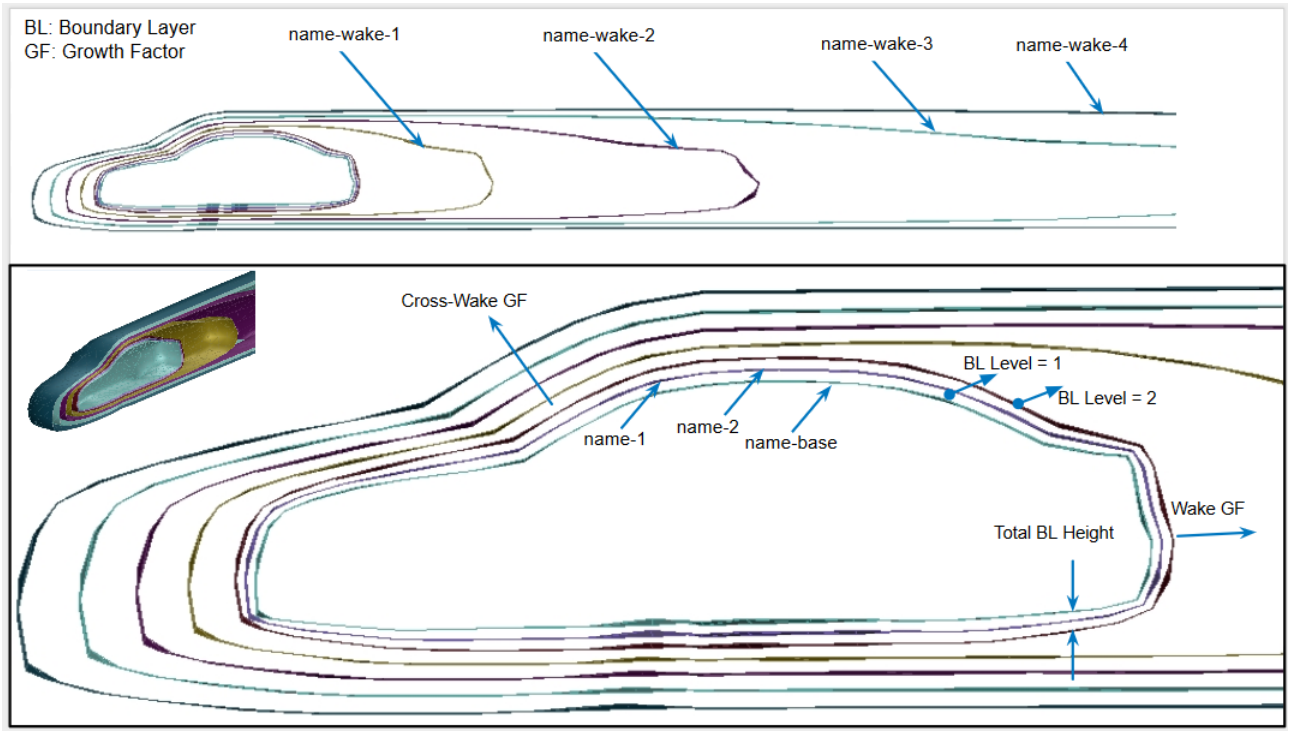
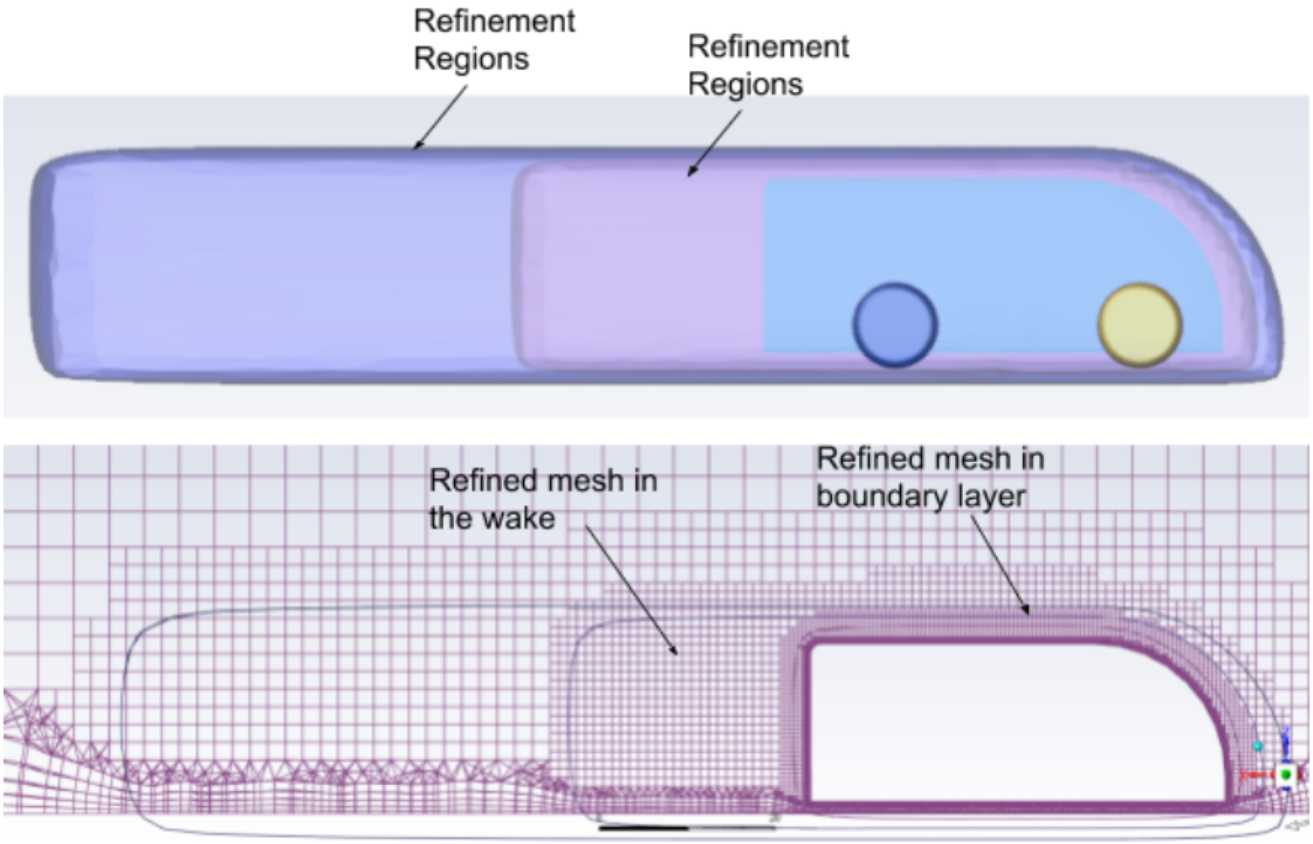


Figure 4.23: An Example of Multiple Refinement Regions Around a Vehicle



- For an **Offset Box** refinement region:
  - Specify the **Flow Direction**. It is used to determine the wake. The wake is downstream of the selected geometry object(s).
  - Use the **Flip** option if the flow direction is in opposite direction.
  - Specify a value for the **Wake Growth Levels**, or use the default value. The rough shape is scaled in the wake direction as well as in the cross-wake direction. This value determines how many scaled offsets are to be created.
  - Specify a value for the **Wake Growth Factor**, or use the default value. Each scaled offset box is grown in size in the wake direction by this factor from the previous scaled offset.
  - Specify a value for the **Cross Wake Growth Factor**, or use the default value. Each scaled offset box is grown in size in the cross-wake direction by this factor from the previous scaled offset.

8. Once your selections are made, click **Create Local Refinement Regions** and proceed to the next task. For more information about bodies of influence, see Body of Influence (p. 592).

   Note:

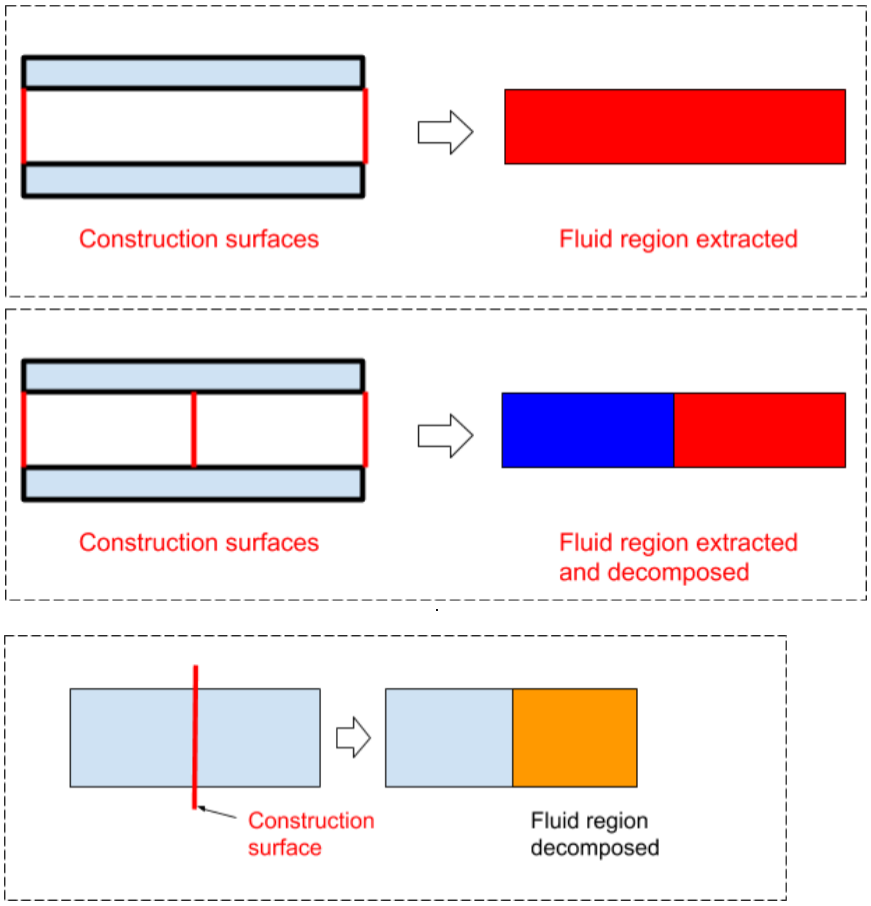
   You should avoid creating intersecting refinement regions, as they can result in unwanted sizes locally, or coarsened meshes, during volume mesh refinement. You should instead create multiple refinement regions to avoid intersections.

   If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

4.7. Identifying Construction Surfaces

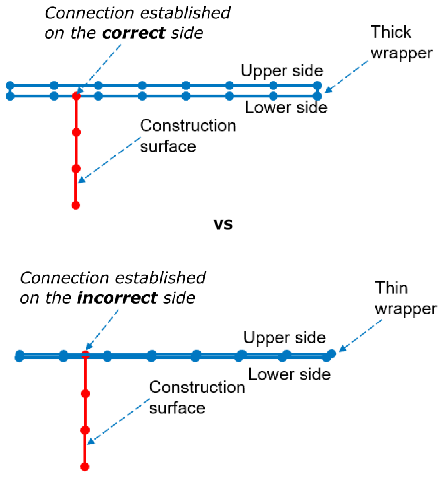
For geometries that have primitive construction surfaces (for example, capping surfaces, or cylindrical surfaces for moving reference frames) already defined as part of the CAD model, you can use the **Identify Construction Surface** task to locate those portions of the geometry and identify them as such so that Fluent can handle them accordingly during the meshing process. These surfaces are used to further decompose the main fluid into multiple regions, such as regions around a rotating fan for a moving reference frame (MRF) simulation.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

The workflow cannot properly connect construction surfaces to wrapped regions that are too thin.



1. Specify a **Name** for the construction surface, or use the default name (`construction-surface-1`).

2. For **Method**, choose from the following:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Select **Existing** if the construction surface is already defined as part of your imported geometry.
- Select **Box** if the construction surface is to be defined using a bounding box.
- Select **Offset Surface** if the construction surface will be offset from the surface of an existing, selected object or zone.

3. When the **Method** is set to **Existing**, you can then select a single label, object, or zone in the corresponding list.
4. When the **Method** is set to **Box**, you can set the **Coordinate Specification Method** to the following options:

   - Use the **Ratio relative to geometry size** option so that you can define the construction surface based on the relative size of selected object(s) or zone(s). Choose whether to **Select By** the object, label, or zone name in the corresponding list.
   - Use the **Directly specify coordinates** option so that you can explicitly define the location and dimension of the construction surface without having to select object(s) or zone(s).

   In addition, under **Box Parameters**, additional settings are required.

   - When you select **Ratio relative to geometry size** as the specification method, the **Box Parameters** hold the minimum and maximum extension ratios for the X, Y, and Z dimensions. By default, they are set to extend the dimensions of a bounding box around your selected object(s) or zone(s) out by a factor equivalent to the total singular length of the geometry in the X, Y, and Z directions. You can change the values according to your needs, and the display in the graphics window will change accordingly.
   - When you select **Directly specify coordinates** as the specification method, the **Box Parameters** hold the minimum and maximum extension *distance* for the X, Y, and Z dimensions. By default, they are set to values that correspond to the extension ratios, but you can set them to more suitable values as required, and the display in the graphics window will change accordingly.

5. When the **Method** is set to **Offset Surface**:

   - Select one or more items in the list of pre-existing objects.
   - Specify a value for the **Defeaturing Size**, or use the default setting. This will obtain the rough shape of the selected object(s), which are wrapped using this size.
   - Specify a value for the **Height** of the offset construction surface, or keep the default setting. This is how far from the selected object(s) the rough shape is offset.

6. Once your selections are made, click **Identify Construction Surface** and proceed onto the next task.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

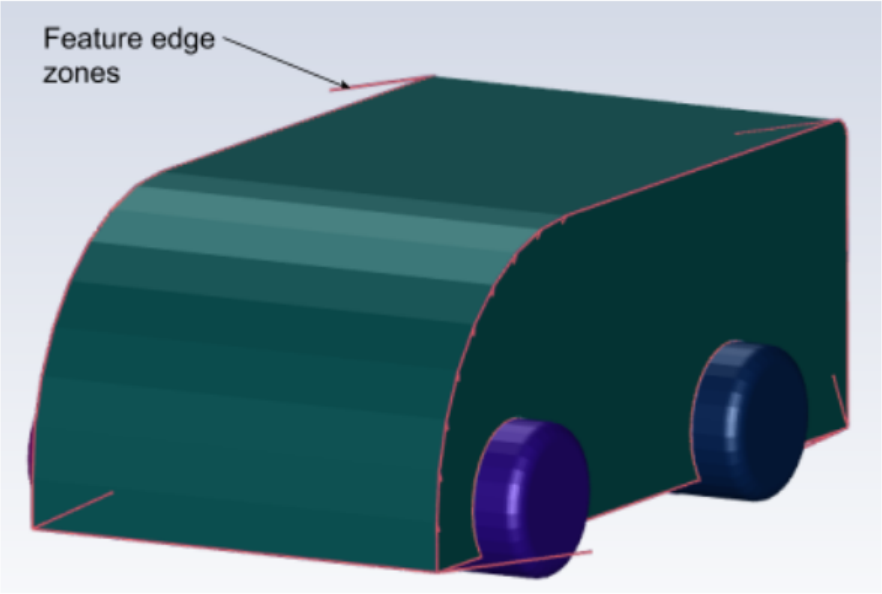
If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

Note:

You can directly select items in the lists or you can use the **Filter Text** option in the drop-down to provide text and/or regular expressions in filtering the list (for example, using *, ?, and []). When applicable, you can also choose the **Use Wildcard** option in the drop-down to provide wildcard expressions in filtering the list. When you use either ? or * in your expression, the matching list item(s) are automatically selected in the list. Use ^, |, and & in your expression to indicate boolean operations for NOT, OR, and AND, respectively. See Filtering Lists and Using Wildcards (p. 287) for more information.

4.8. Extracting Edge Features

You can use the **Extract Edge Features** task to identify important geometric features in order to retain their fidelity for your simulation and to improve the integrity of the final surface mesh. Once these important feature are identified, additional operations are applied to imprint them onto the final surface mesh. Size controls can also be applied to them to refine the mesh on those features.



1. Specify a **Name** for the specific edge feature object, or use the default name (`edge-group-1`).
2. For the **Extraction Method Using** field, select the technique to be used for the feature extraction. Choices are **Feature Angle**, **Intersection Loops**, or **Sharp Angle**.
3. In the list below, you can choose an item in the list, or enter text to filter out the available items in the list before selecting it.

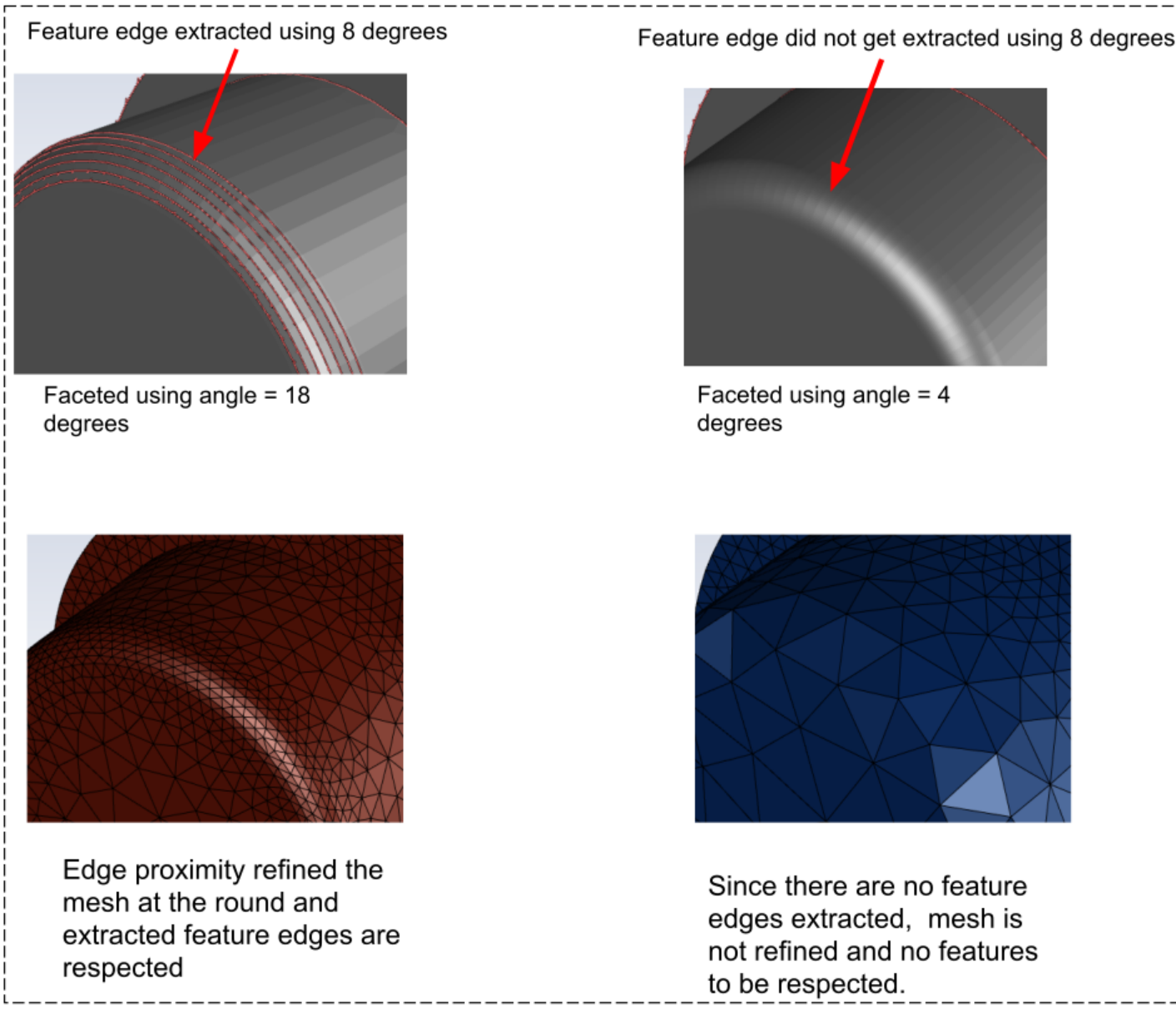
   When using the **Intersection Loops** option, you can choose whether to **Select By** the object, label, or zone name in the list below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select items in the list, or use the **Filter Text** option in the drop-down to provide text and/or regular expressions in filtering the list (for example, using *, ?, and []). You can also choose the **Use Wildcard** option in the drop-down to provide wildcard expressions in filtering the list. When you use either ? or * in your expression, the matching list item(s) are automatically selected in the list. Use ^, |, and & in your expression to indicate boolean operations for NOT, OR, and AND, respectively. See Filtering Lists and Using Wildcards (p. 287) for more information.

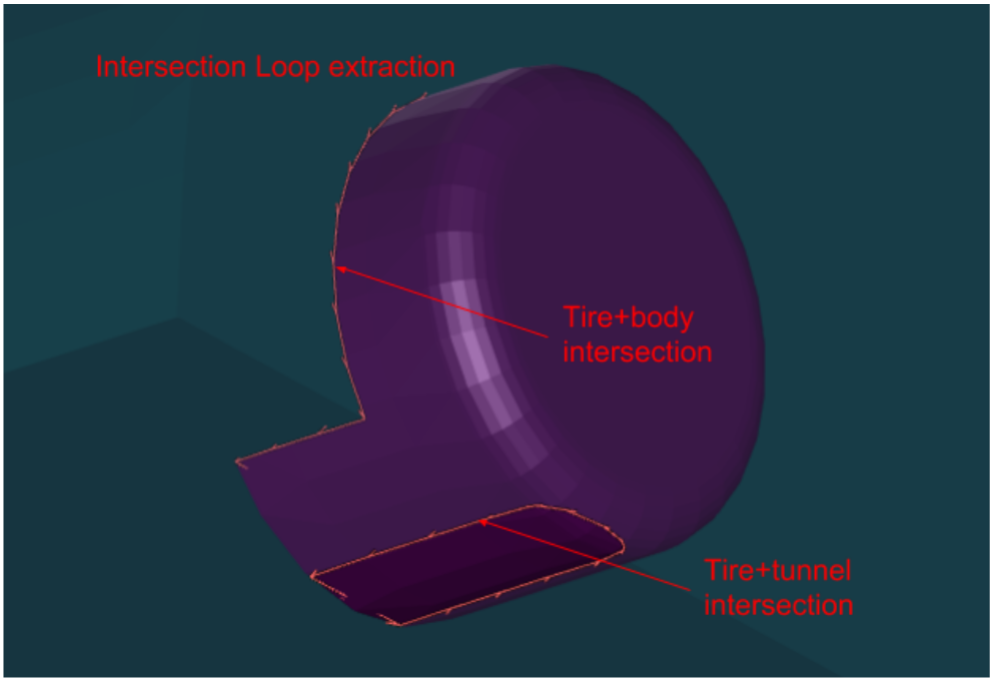
4. Depending on the extraction method, you will have to provide the following:

   - For the **Feature Angle** extraction method, specify the **Feature Angle** that you want to use as the threshold for feature extraction. If the angle between two adjacent geometry facet normals is more than this feature angle value, then a feature is extracted at the sharing edge.

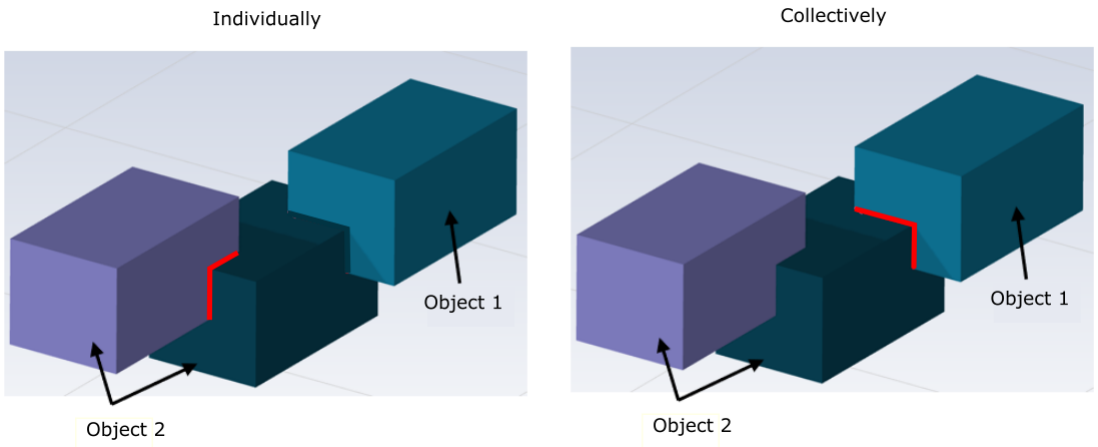


   - For the **Intersection Loops** extraction method (when selecting objects), specify the **Intersected By** option that you want to use as the threshold for feature extraction: either **individually** or **collectively**. This is a feature located at the intersection of two geometry facets. For **collectively**, only facets across objects are considered. For **individually**, all facets are considered.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- The following figure also demonstrates the difference between **individually** versus **collectively** settings, where **individually** is indicative only between zones within a single object.



- For the **Sharp Angle** extraction method, specify the **Sharp Angle** that you want to use as the threshold for feature extraction. If the angle between two adjacent geometry facet normals is more than this sharp angle value, then a feature is extracted at the sharing edge. The default angle is 110 degrees, indicating that these surfaces have a very sharp angle between them. You can define more refined size controls on them and also apply special operations on these objects to force sharing nodes of the final surface mesh onto the geometry.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 4.24: An Example of Addressing Sharp Angles - CAD Geometry

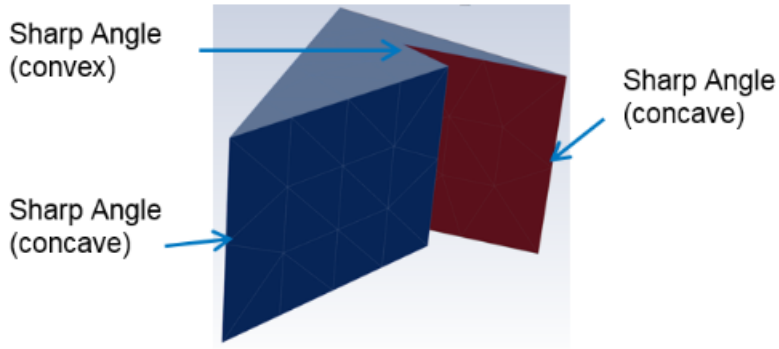
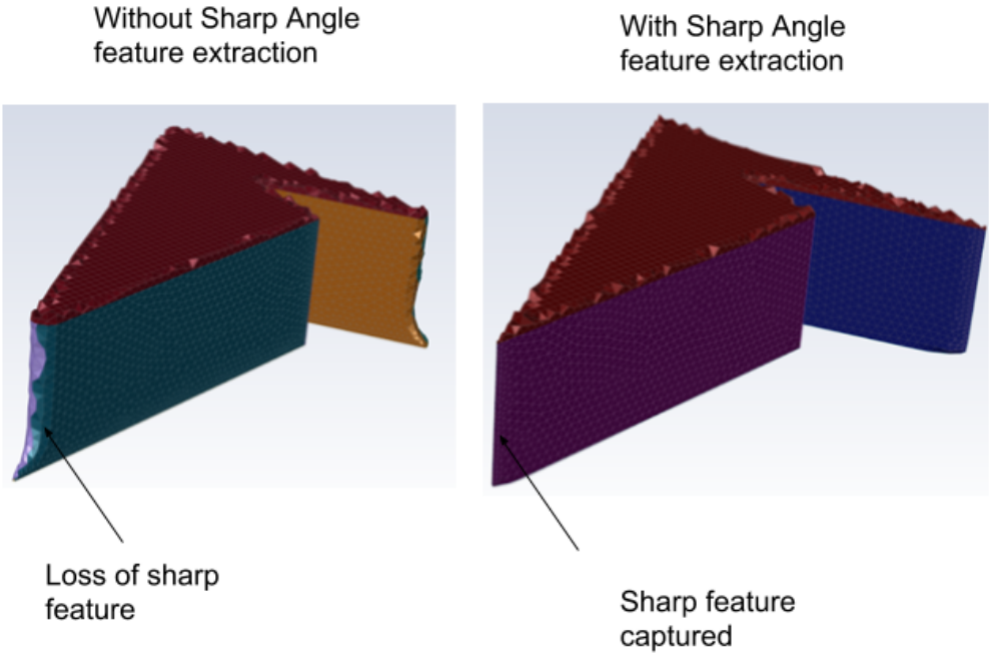
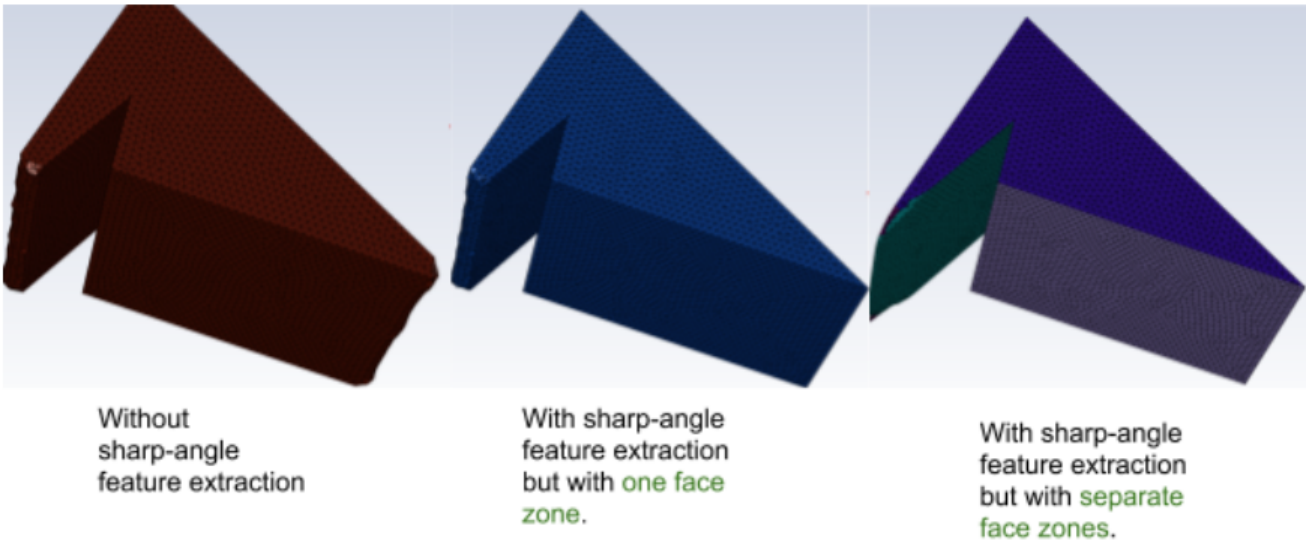


Figure 4.25: An Example of Addressing Sharp Angles - Final Mesh

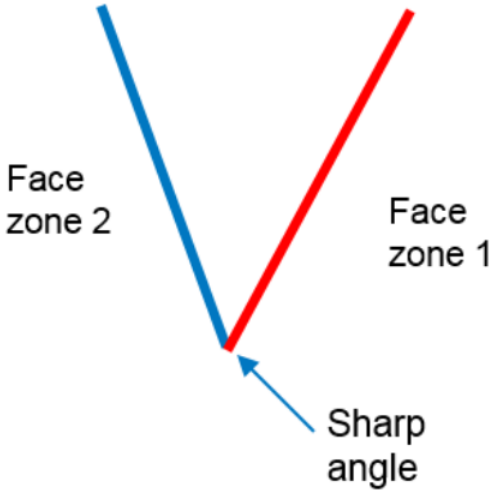


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 4.26: Sharp Angles With and Without the Zones Separated By Face



To force the sharing nodes, the geometry face zones need to be separated at the sharp angle edge. Select face zones per part for that object.



5. Once your selections are made, click **Extract Edge Features** and proceed onto the next task.

   If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

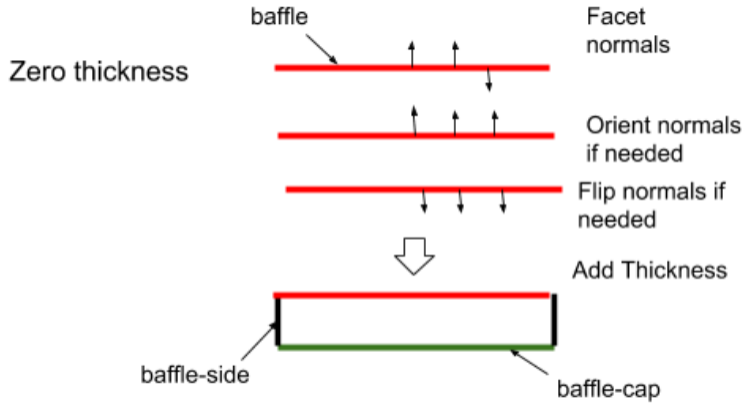
   When visualizing the results of this workflow task in the graphics window (using the icon), Fluent Meshing displays the edge zones and applies transparency to all visible zones.

4.9. Adding Thickness to Your Geometry

Some portions of an imported geometry may not have a thickness (such as a baffle, or other interior wall, for instance), and may be important or relevant to your simulation. A more refined surface mesh can be generated if all important and relevant aspects of the geometry have a certain thickness. Not all portions of a geometry require a thickness. However, if it has no thickness, and is important for your

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

simulation, and contains an adequate number of quality geometry facets, then you can use the **Add Thickness** task to define a thickness to that portion of the geometry. The geometry is thickened in the normal direction of the facets. You may want to reverse the facet normal direction (see Manipulating Boundary Zones (p. 688)) before applying this operation.



1. Specify a **Name** for the zero thickness geometry, or use the default name (`face_group`).

2. Choose whether to **Select By** the object, label, or zone name in the list below.

   Select items in the list, or use the **Filter Text** option in the drop-down to provide text and/or regular expressions in filtering the list (for example, using *, ?, and []). You can also choose the **Use Wildcard** option in the drop-down to provide wildcard expressions in filtering the list. When you use either ? or * in your expression, the matching list item(s) are automatically selected in the list. Use ^, |, and & in your expression to indicate boolean operations for NOT, OR, and AND, respectively. See Filtering Lists and Using Wildcards (p. 287) for more information.

3. Specify the **Offset Distance** that you want to use as the distance for the zero thickness geometry. The geometry will be thickened by this value. Negative values are allowed to preview the opposite/flipped direction. The original face normal will be kept, but you can add thickness in either direction based on a positive or negative value.

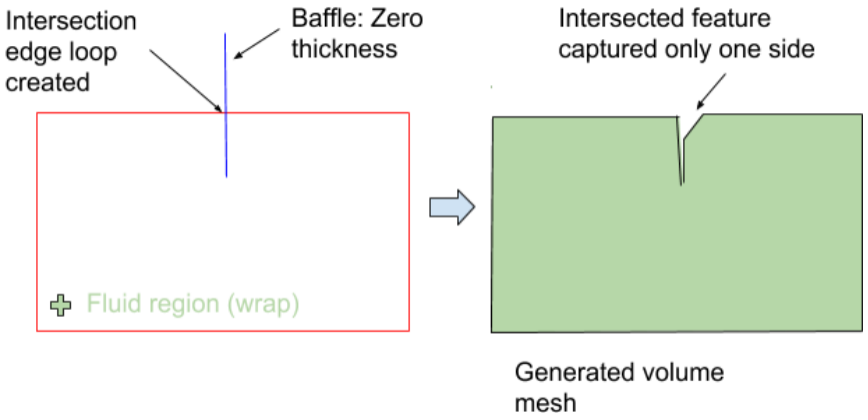
4. Once your selections are made, click **Add Thickness** and proceed onto the next task.

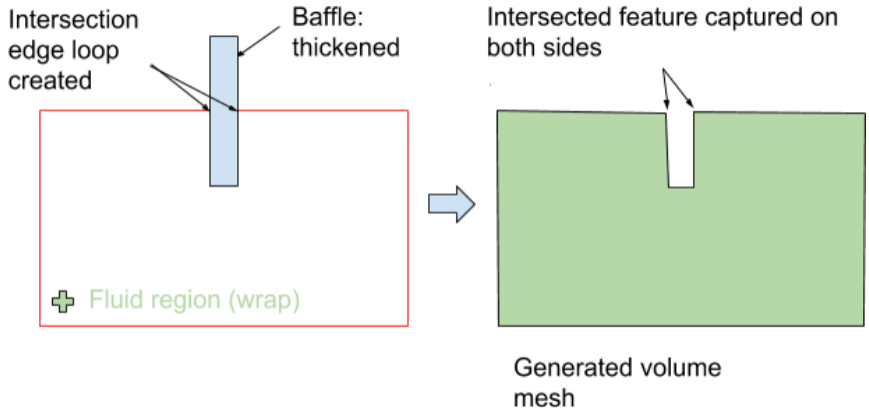
   If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

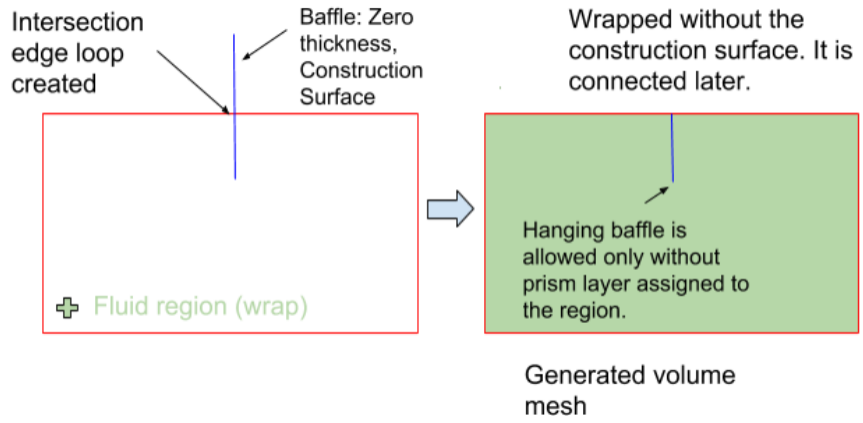
   When visualizing the results of this workflow task in the graphics window (using the icon), Fluent Meshing only displays the object(s) selected for the zero thickness geometry.

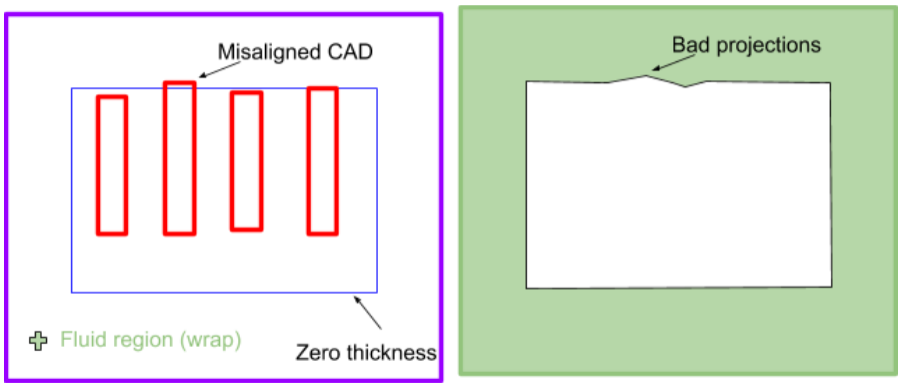
   The following illustrations demonstrate some examples of requiring thickness:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

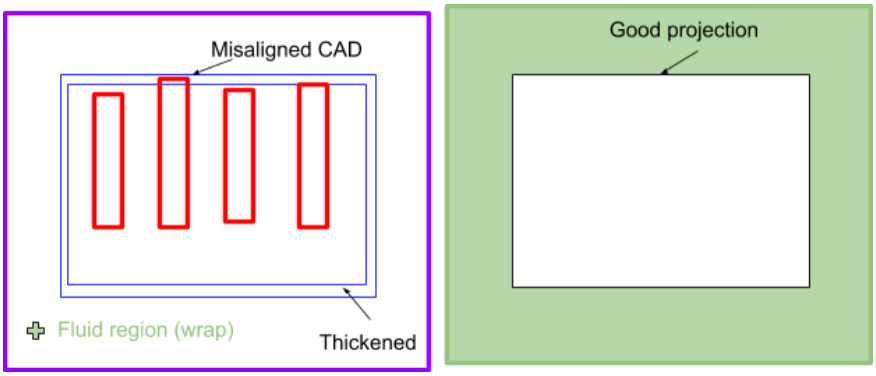








Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4.10. Creating Porous Regions

If your geometry contains some regions where porous media is present and you need to simulate flow through those porous regions, you can use the **Create Porous Regions** task to define portions of your geometry that will exhibit porous behavior in the flow field.

Figure 4.27: An Example of a Porous Region

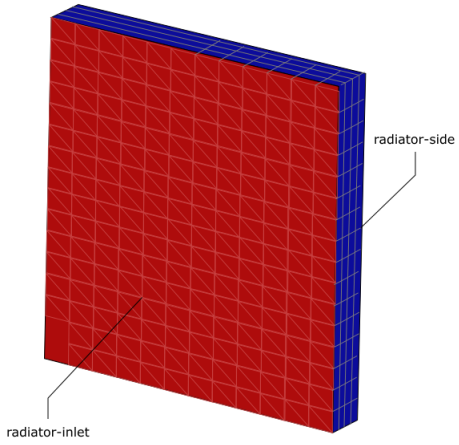
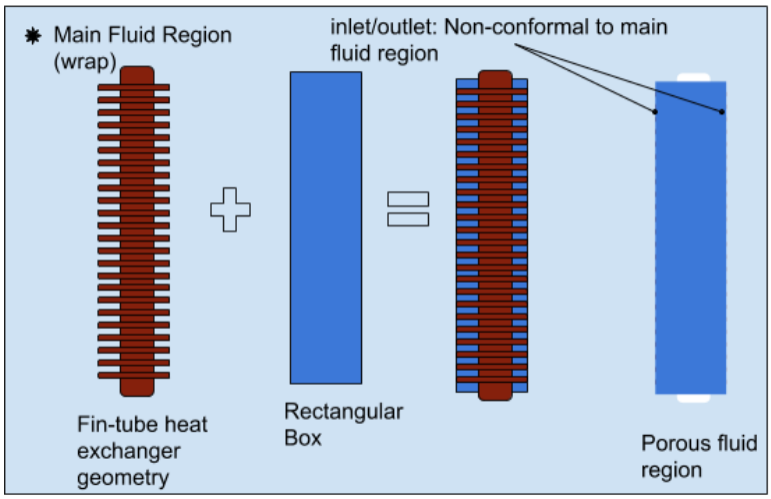


Figure 4.28: An Example of a Porous Region: Fins and Tubes in a Heat Exchanger



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Simulating the pressure drop of the fluid passing through a porous media requires a separate region. This separate porous region requires a very coarse and structured mesh. The **Create Porous Regions** task can be used to generate such a mesh, which is used for rectangular-shaped regions.

For the **Creation Method**, you can choose from the following:

- Select **Text file** to read in a text file containing the details of the porous region.

  1. Specify a **Name** for the porous region, or use the default name (`porous-region`)

  2. For the **File Name**, specify the location and name of the text file containing the definition of the porous region.

     The format of the file should contain information related to the porous zone, such as the name, the coordinates of the corner points of the porous region (P1, P2, P3, and P4, described below), the cell size for each, as well as the buffer size and/or ratio: For example:

     ```
     (
     (name "my_porous_region")
     (p1 (-10 -5 -5))
     (p2 (-10 5 5))
     (p3 (0 -5 -5))
     (p4 (-10 -5 5))
     (size-p1-p2 4)
     (size-p1-p3 4)
     (size-p1-p4 4)
     (buffer-size-ratio 0.05)
     )
     ```

  3. Once you have selected a file, click **Create Porous Regions** and proceed onto the next task. The named cell zone for the porous region will be created as a child of the task. Fluent will also create a geometry object for all triangulated bounding faces.

     If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

  .

- Select **Direct coordinates** to directly enter the location and the dimensions of the porous region.

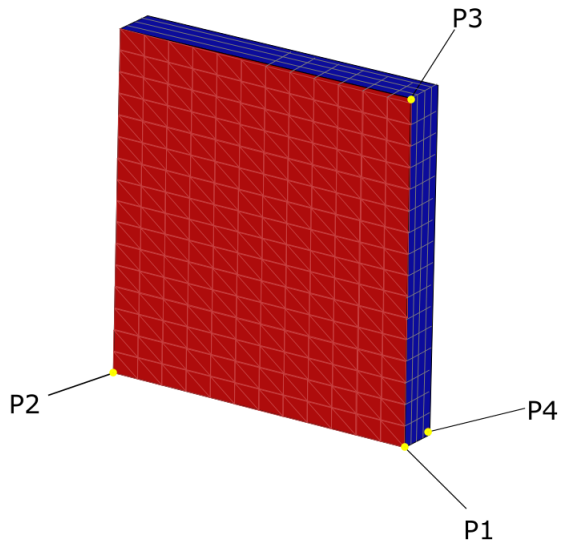
  1. Specify a **Name** for the porous region, or use the default name (`porous-region`).

  2. Choose whether the **Specify location by** field is set to **Node Selection** or **Position Selection**.

  3. Use the right mouse button to select four points in the graphics window to define the porous region (P1, P2, P3, and P4). Their coordinates will be listed in the table in the task.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

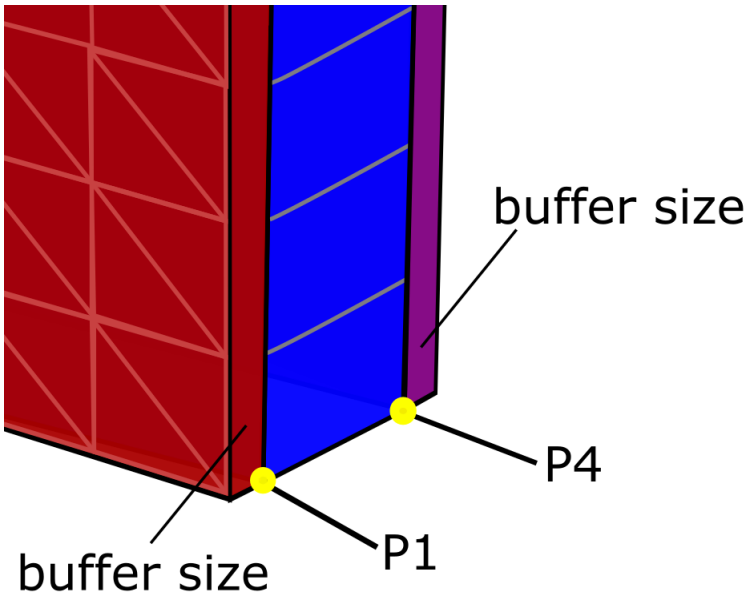
Figure 4.29: The Points of a Porous Region



4. Specify a value for:
   - Either the **Cell Size Between P1 and P2** or the **Number of Cells**.
   - Either the **Cell Size Between P1 and P3** or the **Number of Cells**.
   - Either the **Cell Size Between P1 and P4** or the **Number of Cells**.
5. Specify a value for the **Buffer Size Ratio** along with the **Buffer Size**. There are two advantages for having a buffer zone:
   - The wrapper will not detect the inlet or outlet of the core, which will avoid zone recovery issues, making for a much cleaner rezoning of the wrapper surfaces.
   - The macro-based heat exchanger model in the Fluent solver does not allow mesh interfaces at the inlet/outlet of the core. With the buffer layer, the inlet and outlet are conformal, and the buffer can be very thin (for example, 1 to 2 mm).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 4.30: The Buffer Size for a Porous Region



6. Once your selections are made, click **Create Porous Regions** and proceed onto the next task. The final cell zone for the porous region will be created. Fluent will also create a geometry object for all triangulated bounding faces.

   If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

- Select **Nonrectangular** to provide information for a porous region that is not rectangular in shape.

  1. Specify a **Name** for the porous region, or use the default name (`porous-region`).

  2. Specify a value for the **Mesh Size** of the porous region, or use the default value of 8 millimeters.

  3. Specify a value for the **Thickness** of the porous region, or use the default value of 8 millimeters.

  4. Specify the **Number of Layers**, or divisions, along the thickness of the porous region, or use the default value of 3 layers.

  5. Specify a value for the **Feature Extraction Angle** field, or use the default value of 40 degrees, for the angle at which features will be extracted for the porous region.

  6. Specify a value for the **Buffer Size** for the porous region, or use the default value of 8 millimeters (see Figure 4.30: The Buffer Size for a Porous Region (p. 449)).

  7. Choose whether to **Select By** the object, label, or zone name in the list below.

     Select items in the list, or use the **Filter Text** option in the drop-down to provide text and/or regular expressions in filtering the list (for example, using *, ?, and []). You can also choose the **Use Wildcard** option in the drop-down to provide wildcard expressions in filtering the

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

list. When you use either ? or * in your expression, the matching list item(s) are automatically selected in the list. Use ^, |, and & in your expression to indicate boolean operations for NOT, OR, and AND, respectively. See Filtering Lists and Using Wildcards (p. 287) for more information.

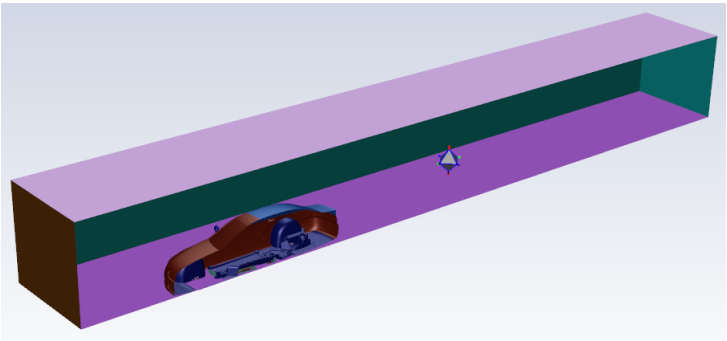
8. Use the **Flip** option to flip the orientation of the clipping plane to suit your requirements.

4.11. Identifying Regions

In complex geometries, there could be hundreds or thousands of regions ranging in various sizes, however, much of the time, you are only interested in only a few regions. Use the **Identify Regions** task to pinpoint specific regions in your imported geometry, such as a vehicle in an external flow simulation. Here, you are positioning specific points in the domain where certain objects of interest can be identified and classified for later use in your simulation.

In most cases, there is not a single object that defines a region. Such regions, such as most fluids and some voids, can only be identified using a specific point (also referred to as a *material* point).

Figure 4.31: Identifying a Fluid Region in the Wake Behind a Car



Note:

Material points can also be placed inside any additional regions that will be created as a result of the decomposition of the main fluid region by construction surfaces (see Identifying Construction Surfaces (p. 436)). If material points are not placed inside any of the decomposed regions, these regions will be automatically identified and named according to the name of the construction surface. For example, if you have identified a construction surface in your workflow, named `fan_mrf`, any decomposed regions created from this construction surface and which neighbors any user-defined regions, will be automatically named as `fan_mrf-1`, `fan_mrf-2`, etc.

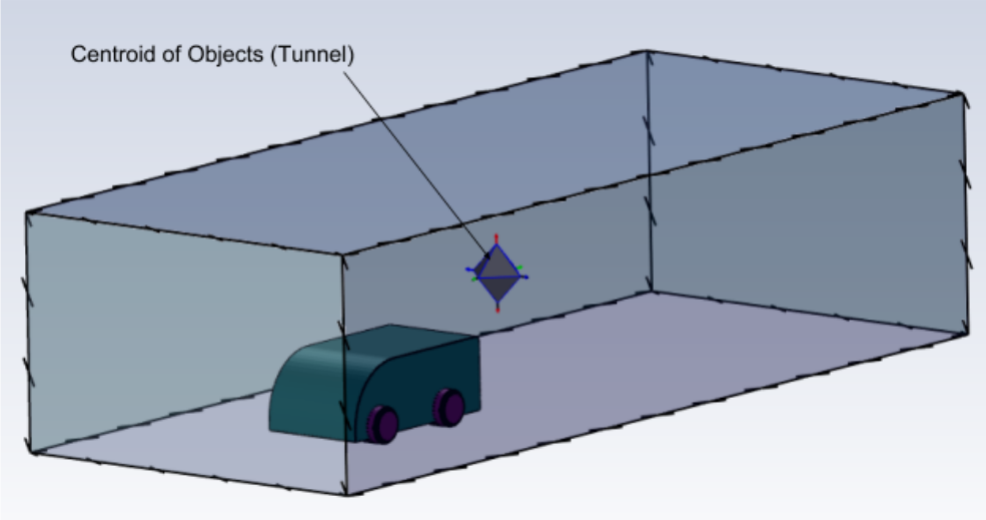
For the **Identify Any Fluid or Void Region(s)?** field, keep the default value of **yes** if your model requires you to specify any fluid or void regions using the following steps. Otherwise, select **no**, click **Update** and proceed to the next task.

1. Specify a **Name** for the region, or use the default name (`fluid-region-1`). The default name changes depending on the assigned **Region Type**.

2. For **Define Location Using** field, choose a method to specify the location of the region:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Select **Numerical Inputs** to assign specific coordinates to a particular region. You can use the **Compute from Graphical Selection** field to determine the region's location based on selecting a point in the graphics window. Additionally, you can use the **View Exact Coordinates** field to see and edit the specific **X**, **Y**, and **Z** coordinate values for the region's identified location.
- Select **Centroid of Objects** to calculate the position of the region based on the centroid of the selected object(s). Additionally, you can use the **View Exact Coordinates** field to view the specific **X**, **Y**, and **Z** coordinate values for the region's identified location.



- Select **Offset Method** to calculate the region based on an offset distance relative to the centroid of a selected object/zone. Select one or more objects or zones in the corresponding **Select By** list, and then specify the offset distance using the **Offset X**, **Offset Y**, and **Offset Z** fields.

3. Choose a **Region Type** for the region (either **fluid**, **solid**, or **void**).
4. For the **Link to Construction Surface(s)** field, keep the default value of **no** for most cases involving a singular fluid region. If, however, you mean to identify an additional fluid region that is either inside or adjacent to a construction surfaces, then set this value to **yes** in order to properly mesh the additional fluid region accordingly (that is, using a surface mesh).
5. Choose whether to **Select By** the object, label, or zone name in the list below.

   Select items in the list, or use the **Filter Text** option in the drop-down to provide text and/or regular expressions in filtering the list (for example, using *, ?, and []). You can also choose the **Use Wildcard** option in the drop-down to provide wildcard expressions in filtering the list. When you use either ? or * in your expression, the matching list item(s) are automatically selected in the list. Use ^, |, and & in your expression to indicate boolean operations for NOT, OR, and AND, respectively. See Filtering Lists and Using Wildcards (p. 287) for more information.
6. Once your selections are made, click **Identify Regions** and proceed onto the next task.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

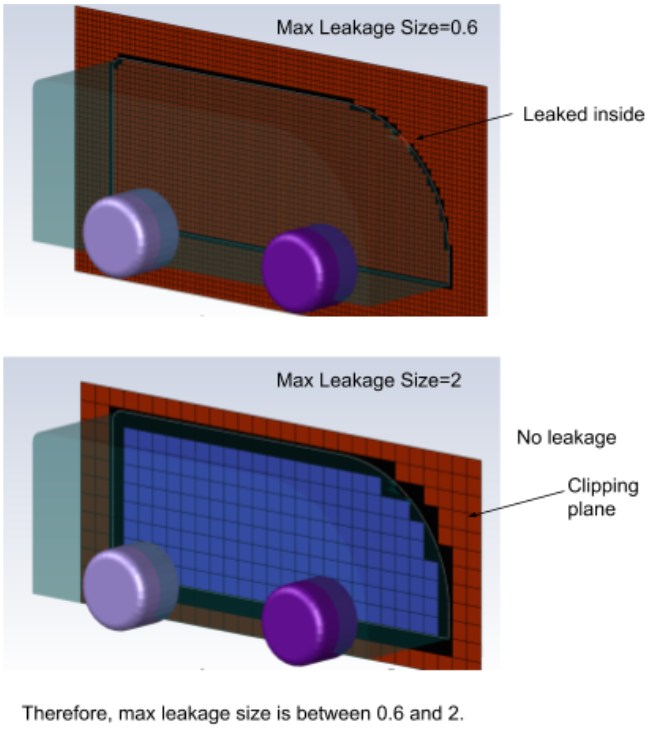
Note:

If you have previously defined regions based on specific meshing objects that you subsequently remove or rename (for example, in the **Import CAD and Part Management** task), you may need to revert this task in order to refresh the list of meshing objects available for your region definition. Fluent will prompt you if the list must be refreshed.

When visualizing the results of this workflow task in the graphics window (using the icon), Fluent Meshing applies transparency to the fluid region(s).

4.12. Defining Leakage Thresholds

Once you have defined all of your objects in the **Import Geometry and Part Management** task, it's always recommended to make sure that there are no large parts missing and that all the parts fit together well. For instance, an imported car geometry may be missing a large part (such as a door), resulting in a large hole and causing potential flow leakage into the cabin. There may also be potential imperfections in the geometry (for example, missing bolts, missing sealants, etc.), resulting in other, medium sized holes. These holes may be larger than any of your initial local size controls, resulting in leaks that need to be closed.



Any missing, misaligned parts, or small imperfections will lead to leakages that need to be identified and closed using this task.

For the **Would you like to close any leakages for your region(s)?** field, select **yes** if your model requires you to locate and address any leakage issues using the following steps. Otherwise, if your model does

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

not have any leakage issues to address, keep the default of **no**, click **Update** and proceed to the next task.

1. Specify a **Name** for the leakage threshold control, or use the default name (`leakage-1`).

2. Use the **Select By** field to indicate how your regions will be displayed in the table. You can select **object** or **identified region** (such as previously identified fluid, solid, or void regions). This can be helpful when you have a large number of regions.

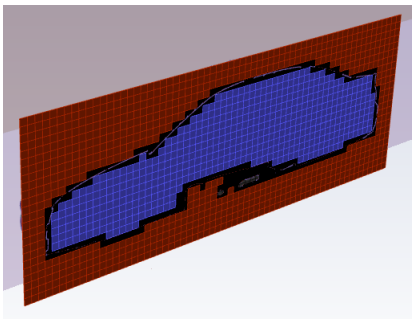
3. Use the list to select the region that requires a leakage threshold control.

   Select items in the list, or use the **Filter Text** option in the drop-down to provide text and/or regular expressions in filtering the list (for example, using *, ?, and []). You can also choose the **Use Wildcard** option in the drop-down to provide wildcard expressions in filtering the list. When you use either ? or * in your expression, the matching list item(s) are automatically selected in the list. Use ^, |, and & in your expression to indicate boolean operations for NOT, OR, and AND, respectively. See Filtering Lists and Using Wildcards (p. 287) for more information.

4. Specify a **Maximum Leakage Size**.

5. Select the **Preview Leakages** button to see if there are any leakages present or not. If you can see leakages occurring between regions, adjust the **Maximum Leakage Size** and click the **Preview Leakages** button again to reconstruct the generalized mesh. Repeat as needed until all leaks have been resolved.

   Figure 4.32: Identifying Potential Leakages Within a Car

6. Use a **Leakage Plane** controls to assess the interior of your geometry to help you to preview and identify leakages.

   - Use the slider to adjust the **Location** accordingly to visualize any leakages.
   - Specify the **Orientation** as either in the **X**, **Y**, or **Z** direction.
   - Use the **Flip** option to flip the orientation of the clipping plane to suit your requirements.

7. Once your selections are made, click **Define Leakage Threshold** and proceed onto the next task.

   If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

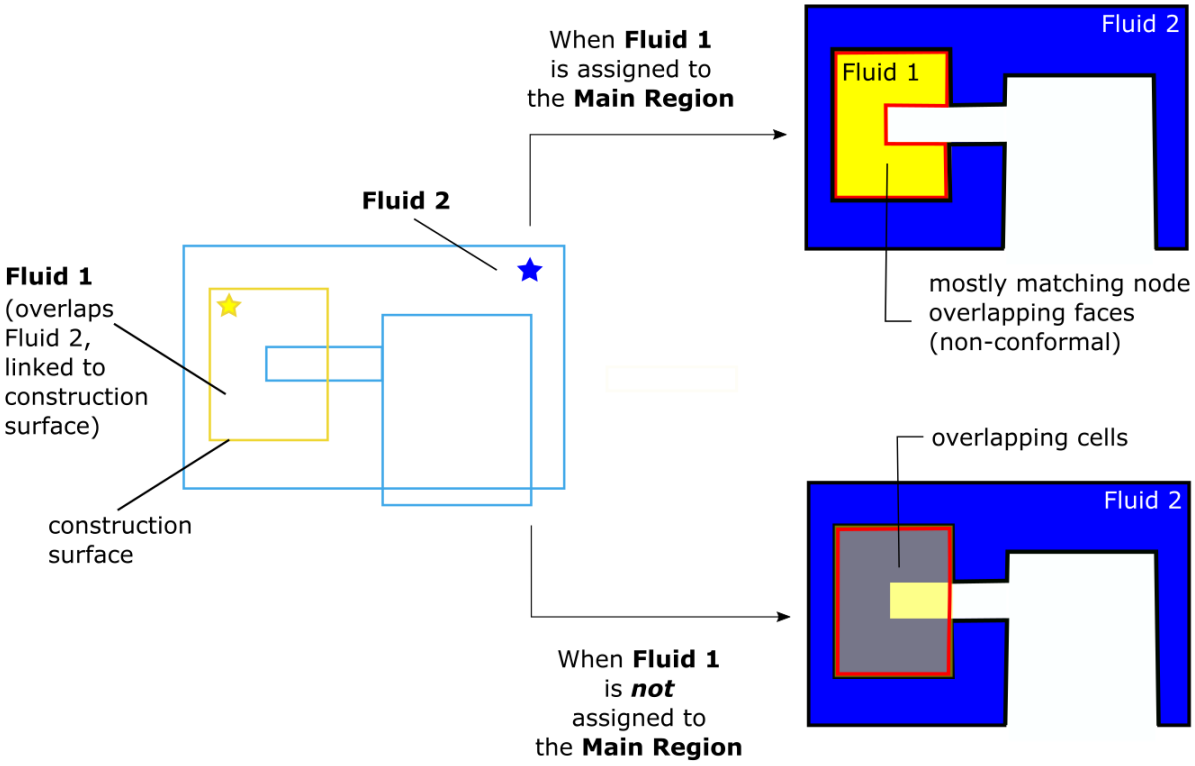
When visualizing the results of this workflow task in the graphics window (using the icon), Fluent Meshing only displays the object(s) selected for the leakage threshold.

4.13. Updating Your Region Settings

You can review and update the properties of any defined regions using the **Update Region Settings** task. Any automatically detected regions will be filled with hexcore cells, designated as fluid, and given a single layer of prism cells for the boundary layer.

1. The **Main Region** field indicates the name of the primary fluid region for your simulation.

   In cases where there are multiple regions to consider, ensure that you choose the correct fluid region, especially in cases where there are construction surfaces, since construction surfaces are only linked to the main region.



2. Use the **Displayed Regions** field to indicate how your regions will be displayed in the table. You can select **All Regions**, **Object Based Regions**, or **Identified Regions**. This can be helpful when you have a large number of regions.

3. (optional) Use the **Filter** button to filter the table contents based on a particular column.

4. For regions that are going to be a part of an overset mesh, set the corresponding **Overset Component** assignment to **yes**, where you are then restricted to either **fluid** or **void** as the region **Type**.These overset components will be meshed independently.

5. Assign the **Extraction Method** to either **wrap**, **surface mesh**, or **cfd surface mesh**, for standard and overset regions. In addition, for regions that are going to be a part of an overset mesh, you can also use an **existing mesh**.

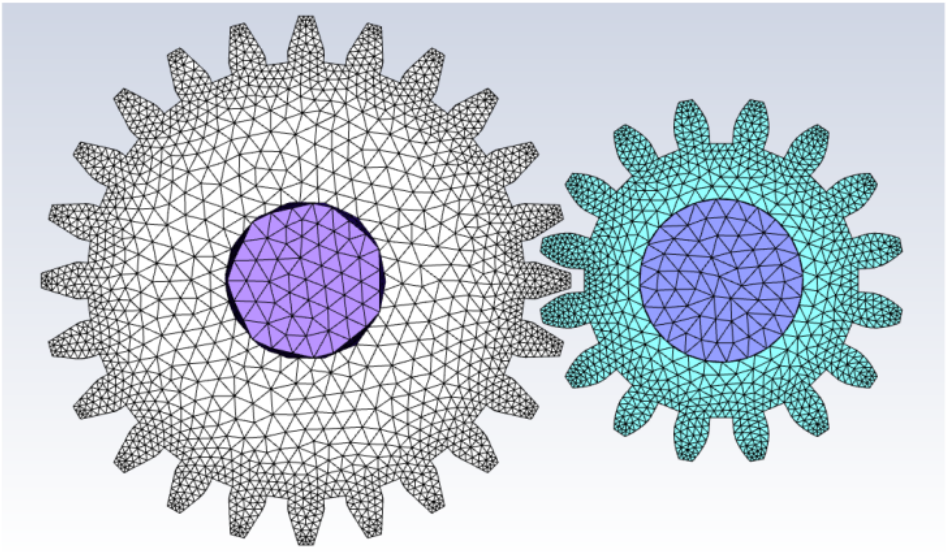
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Multiple regions can be assigned a specific extraction method all at once by selecting them in the table, right-click, and select **Set Extraction Method** in the context menu, then designate an extraction method for the selected regions directly in the menu.

Fluid regions that are within or adjacent to construction surfaces (using the **Link to Construction Surface(s)** field) will automatically use the **surface mesh** extraction method.

Note:

- The **cfd surface mesh** option allows you to mesh the selected part(s) based on the original CAD model, rather than by using the initial faceting, providing a higher quality surface mesh. Using this option also negates the need to perform any refaceting operations on the imported CAD model in the **Import CAD and Part Management** task.



  To use this feature with SpaceClaim files (`.scdoc` and `.scdocx`) or SCDM files, you must use the **Workbench** option for the **Import Route** in the **Import CAD and Part Management** task.

- If the **Extraction Method** is set to either **surface mesh** or **cfd surface mesh**, there should be no intersection with other regions in the same object.
- If the **Extraction Method** is set to **cfd surface mesh** for a mesh object that contains multiple volumetric regions, after volume meshing:
  - All cell zone names will be based on the name of the corresponding region.
  - If the **Type** is set to **fluid** for the meshing object, only volumetric region names containing "fluid" in the name will retain **fluid** for their type and the rest will be set to **solid**.
  - A volume meshing summary will be printed in the console for the meshing object.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Choose a region **Type** as either **fluid**, **solid**, or **void**.

   Multiple regions can be assigned a specific type all at once by selecting them in the table, right-click, and select **Set Type** in the context menu, then designate a type for the selected regions directly in the menu.

7. Choose a **Volume Fill**, or keep the default setting. For both fluids and solids, you can choose from **tet**, **hexcore**, **poly**, or **poly-hexcore**.

   Multiple regions can be assigned a specific volume fill type all at once by selecting them in the table, right-click, and select **Set Volume Fill** in the context menu, then designate a volume fill type for the selected regions directly in the menu.

8. Specify a value for the **Leakage Size**, or keep the default value.

   Multiple regions can be assigned a specific leakage size value all at once by selecting them in the table, right-click, and select **Set Leakage Size** in the context menu, then specify a value for the selected regions directly in the menu.

   You can enter values for the **Leakage Size** directly in the table (for **fluid**, **solid**, and **void** regions that employ the **wrap** extraction method) rather than by using the **Define Leakage Threshold** task.

9. When you are satisfied with the region assignments, click **Update Region**.

   If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

10. One or more regions can be visualized all at once by selecting them in the table, right-click, and select **Draw Selections** in the context menu.

11. Once all regions have been updated, proceed to the next step in the workflow.

4.14. Choosing Mesh Control Options

The **Choose Mesh Control Options** task allows you to pick and choose various means of generating and refining the mesh in your simulation.

1. Specify how you would like to manage your size controls.

   - Select **Create new** to start with a new collection of size controls.
   - If you are creating new size controls, you need to specify the nature of the size controls you are going to use.
     - Select **Default** to create curvature and proximity size controls with default settings, as well as local size controls based on the number of objects in your model (such as external flow regions, or local refinement regions). Using this option and updating the task will add an **Add Local Sizing** task to the workflow and as many size controls as there are objects, each using their own default settings. You can also choose to create your own custom size controls using the **Default** option, and they will be retained until you remove them, even when the task has been reverted.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the event that you remove any local sizing controls when using the **Default** option, you can quickly restore them by updating the **Choose Mesh Control Options** task. In addition, in the event that you make changes to the default values and/or remove any altered local sing controls, Fluent will provide information and guidance as to how to restore any missing sizing controls, if desired, when you update the task.

– Select **Custom** to populate as many size controls as you need using your own customized settings. Using this option and updating the task will add a single **Add Local Sizing** task to your workflow, where you create as many size controls as you require.

Note:

For either default or custom size controls, you can choose to use the **Reset** button's **Re-Create Local Refinement Regions Sizing** option (located at the bottom of the task). This option removes any existing local refinement settings and adds new local refinement settings based on the default size controls created in earlier tasks.

For more information, see (see Adding Local Size Controls (p. 459))

- Select **Read existing size control file** to add your size controls based on those already defined in a size control file (`*.szcontrol`).

  Use the **Compute Size Field For** field to determine which type of existing size field file you are going to use. Choose from:

  – **Both Wrap and Target** - use this option if you already have the wrap and target size fields computed and saved in separate files. Supply the file name and location for both the **Wrap Size Control File** and the **Target Size Control File** fields.

  – **Target Only** (the default) - use this option if you only have the target size field computed and saved in a separate file. Supply the file name and location using the **Target Size Control File** field.

  – Select **Wrap Only** - use this option if you only have the wrap size field computed and saved in a separate file. Supply the file name and location using the **Wrap Size Control File** field.

  – If you have chosen **Target Only** or **Wrap Only**, you will need to specify a value for the **Wrap/Target Size Field Ratio** field, or keep the default value. This is the ratio of the wrap size and the target size controls. For example, if this ratio is 0.67 and the initial minimum target size is 1 and initial maximum target size is 16, then the initial minimum wrap size will be 0.67 and the initial maximum wrap size will be 10.72 .This value does not affect the BOI or soft size controls, such that the **Wrap/Target Size Field Ratio** is set to 1 for BOI or soft size controls. See Working With Wrap and Target Sizes (p. 458) for more information.

- Select **Read existing size field file** in order to use a size field file (`*.sf`) that is already computed.

  Use the **Existing Size Field For** field to determine which type of existing size field file you are going to use. You can select from:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Both Wrap and Target** (the default) - use this option if you already have the wrap and target size fields computed and saved in separate files. Supply the file name and location for both the **Wrap Size Field File** and the **Target Size Field File** fields.
- **Target Only** - use this option if you only have the target size field computed and saved in a separate file. Supply the file name and location using the **Target Size Field File** field.
- Select **Wrap Only**, use this option if you only have the wrap size field computed and saved in a separate file. Supply the file name and location using the **Wrap Size Field File** field.
- If you choose **Target Only** or **Wrap Only**, you will need to specify a value for the **Wrap/Target Size Field Ratio** field, or keep the default value. This is the ratio of the wrap size and the target size controls. For example, if this ratio is 0.67 and the initial minimum target size is 1 and initial maximum target size is 16, then the initial minimum wrap size will be 0.67 and the initial maximum wrap size will be 10.72 .This value does not affect the BOI or soft size controls, such that the **Wrap/Target Size Field Ratio** is set to 1 for BOI or soft size controls. See Working With Wrap and Target Sizes (p. 458) for more information.

2. Click **Advanced Options** to access additional controls prior to performing this task. Options include:
   - For the **Apply Quick Edge Proximity?** prompt, select **Yes** to speed up the calculation and reduce memory requirements when using a proximity size function with geometries having a very large number of small feature edges.
   - Specify a value for the **Solid/Fluid Size Ratio** field, or keep the default. This is the ratio of the actual size of the solid and the actual size of the fluid, used for solid meshing. For example, if this ratio is 1.5 and the fluid size is 1, then the solid size will be 1.5
3. Once your selections are made, click **Choose Options** and proceed onto the next task.

   If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

   Use the **Restore Local Sizings** button to return to your original local sizing objects and their default settings.

4.14.1. Working With Wrap and Target Sizes

Fluent Meshing uses both target and wrap size fields. First, the geometry is wrapped using finer wrap sizes. Then, the wrapper nodes are snapped to the feature edges. Finally, the wrapper surfaces are re-meshed using the desired, coarser, target sizes. The wrapper size determines what features will be captured and what features will be ignored. Since the wrapper size produces surface meshes that are too fine (in order to capture features), the remeshing operation significantly reduces the size while respecting those features.

You can use the **Compute Size Field For** field to have more flexibility and control when calculating the size field for very complex models. By default, the size field is calculated for **Both Wrap and Target**. If you choose **Target Only** or **Wrap Only**, however, only one size field is calculated. The other size field is determined using the **Wrap/Target Size Control Ratio**, rather than being recalculated, potentially saving computational resources for very complicated models.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These settings will determine the size control settings in the subsequent **Add Local Sizing** task (Adding Local Size Controls (p. 459)), and how the size controls are calculated. If you select **Both Wrap and Target**, then you will have to set size control properties (minimum size, maximum size, growth rate, and curvature normal angle) for both the target and the wrap objects. However, if you select **Target Only**, you only need to provide values for the target's size control properties. Likewise, if you select **Wrap Only**, you must provide values for just the wrap's size control properties.

Note:

As stated earlier, in order to properly capture wrapped regions and boundaries, the **Fault-tolerant Meshing** workflow uses two types of size controls:

- ***Initial (Wrap)*** - a finer size control used for more refined mesh resolution to ensure geometry capture and region identification. This value is calculated using the wrap/target size ratio and the target size value.
- ***Target*** - the coarser size control used in re-meshing to the required mesh resolution for the solution.

The wrap/target size ratio default value is 2/3 and is recommended in most cases, however, you may occasionally need to decrease this value to better capture geometric features.

Note:

When using the **Both Wrap and Target** option, the surface or volume mesh generated will be the same quality as one that is read into the workflow. However, when using either the **Target Only** or the **Wrap Only** options, you will have two different meshes because the target size fields of these two options are different.

With the **Target Only** option, the size defined in the **Add Local Sizing** task are going to be used in both surface and volume meshing and will directly decide the target size field, Also, the **Wrap/Target Size Field Ratio** is going to affect the feature extraction such that a smaller value will increase computational costs but yield a more accurate result.

With the **Wrap Only** option, however, the target size field is based on the **Wrap/Target Size Field Ratio**, where everything is scaled by the inverse of the ratio. The wrap sizes defined in the **Add Local Sizing** task are used to help you control how exactly you want to capture the original geometry, however, the **Wrap/Target Size Field Ratio** will be used to calculate the target size field which is equal to the inverse of the product of the **Wrap/Target Size Field Ratio** and the wrap size field (including the boi and soft sizing types). It should also be noted that you can use size boxes (available using the **Show Size Box** or the **Show Wrap Size Box** check boxes in the **Add Local Sizing** task) to preview the size boxes if necessary. When using the **Wrap Only** option, you should use the **Show Size Box** check box because it will have a coarser mesh when compared to the defined wrap sizes.

4.15. Adding Local Size Controls

Use the **Add Local Sizing** task to create individual sizing controls for your mesh. For every size control that you create, it is added to the workflow as a subtask of the **Add Local Sizing** task.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Specify a **Name** for the mesh sizing control, or use the default name. The default name changes depending on the **Size Functions** setting.

2. Choose an option for the **Size Functions** field. Available options are:

   - Choose the **curvature** size function that computes edge and face sizes using their size and normal angle parameters.
   - Choose the **proximity** size function that computes edge and face sizes in 'gaps' using the specified minimum number of element layers.
   - Choose the **soft** size function that enables you to set the maximum size on the selected zone, while the specified growth rate from the defined size influences the size on adjacent zones.
   - Choose the **boi** size function enables you to specify a body of influence (that is, a refinement region for the sizing control).

   For additional information, see Size Functions and Scoped Sizing (p. 584).

3. Specify a value for the **Minimum Size** for the size control when using the **curvature** or **proximity** size function types.

4. Specify a value for the **Maximum Size** for the size control.

5. Specify a value for the **Growth Rate** for the size control.

6. Specify a value for the **Cells Per Gap** when using the **proximity** size function type. Note that for proximity size functions, the number of cells per gap can be a real value, with a minimum value of 0.01. See Proximity (p. 587) for more information.

7. Specify a value for the **Curvature Normal Angle** when using the **curvature** size function type.

8. If applicable, deselect **Ignore Self** when using the **proximity** size function type. By default, **Ignore Self** is enabled, such that the proximity between two facets belonging in a single face zone will be ignored, avoiding unnecessary over-refinement.

9. Choose an option in the **Scope Proximity To** field, where the size controls can be scoped to **edges**, **faces**, or to both **faces and edges**. Note that if you select **faces-and-edges**, you can only select objects.

10. Choose whether to **Select By** the object, label, or zone name in the list below.

    Select items in the list, or use the **Filter Text** option in the drop-down to provide text and/or regular expressions in filtering the list (for example, using *, ?, and []). You can also choose the **Use Wildcard** option in the drop-down to provide wildcard expressions in filtering the list. When you use either ? or * in your expression, the matching list item(s) are automatically selected in the list. Use ^, |, and & in your expression to indicate boolean operations for NOT, OR, and AND, respectively. See Filtering Lists and Using Wildcards (p. 287) for more information.

11. Select the **Show Size Box** field to visualize the size control's minimum and maximum sizes in the graphics window (in the form of red cubes).

    To make zone selection easier when the **Scope Proximity To** field is set to **edges**, the size boxes are displayed on the edge zones in the graphics window.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12. Select the **Show Wrap Size Box** field to visualize the wrap size control's minimum and maximum sizes in the graphics window (in the form of red cubes).

13. Click **Show Wrap Settings** to display additional wrap-specific controls prior to performing this task Options include:

    - When **Size Functions** is set to **curvature** or **proximity**, specify a value for the **Wrap Min Size** field.
    - Specify a value for the **Wrap Max Size** field.
    - Specify a value for the **Wrap Growth Rate** field.
    - When **Size Functions** is set to **curvature**, specify a value for the **Wrap Curvature Normal Angle** field.
    - When **Size Functions** is set to **proximity**, specify a value for the **Wrap Cells Per Gap** field.
    - Select the **Show Wrap Size Box** field to visualize the wrap's minimum and maximum sizes in the graphics window (in the form of red cubes).

    Note:

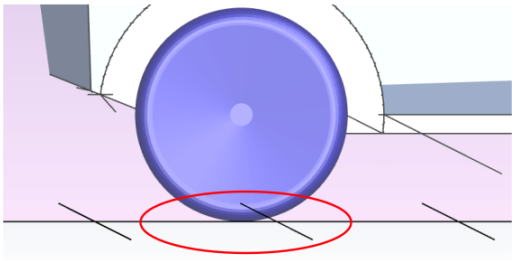
    See Choosing Mesh Control Options (p. 456) for more information about target and wrap settings.

14. Once your selections are made, click **Add Local Sizing** and proceed onto the next task.

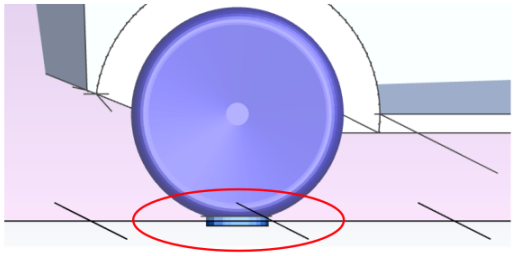
    If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

4.16. Creating Contact Patches

Occasionally, contact between two objects (such as the contact between an automobile tire and the pavement surface) contains sharp angles that cannot easily be captured during the meshing process. In such cases, you can use the **Create Contact Patch** task to create a localized area around the contact surface or a 'patch' that you can use to circumvent the meshing process and avoid meshing issues that could potentially occur in such close contact areas.



Area of Contact



Contact Patch Applied to Area of Contact

1. Specify a **Name** for the contact patch object, or keep the default value (`contact-patch-1`).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

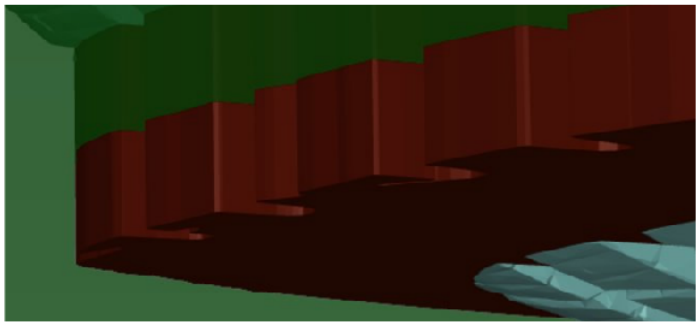
2. Choose whether to **Select By** the object, label, or zone name in the list below, of items that contain the gap that you want to cover.

   Select items in the list, or use the **Filter Text** option in the drop-down to provide text and/or regular expressions in filtering the list (for example, using *, ?, and []). You can also choose the **Use Wildcard** option in the drop-down to provide wildcard expressions in filtering the list. When you use either ? or * in your expression, the matching list item(s) are automatically selected in the list. Use ^, |, and & in your expression to indicate boolean operations for NOT, OR, and AND, respectively. See Filtering Lists and Using Wildcards (p. 287) for more information.

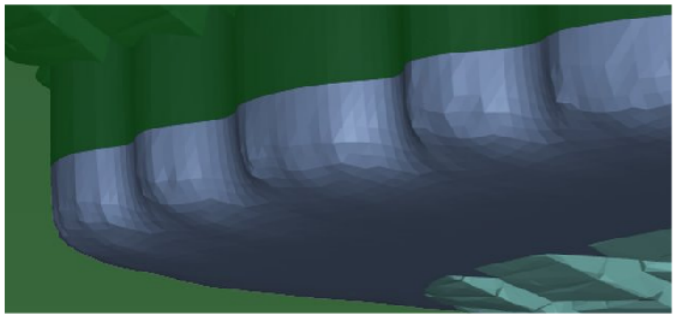
   The first list contains potential contact *sources* (such as a tire). The second list (the **Ground Zone** list) contains potential contact *targets* (that is, where the contact is being applied to, such as the ground for a tire).

3. Specify the **Offset Distance** that you want to use as the distance of the contact patch geometry from the ground zone. The patch geometry will be thickened by twice this value. Negative values are allowed to apply to opposite/flipped direction, however, contact and intersection needs to occur based on this distance.

4. Specify a value for the **Defeaturing Size**, or keep the default value. This setting controls the smoothness of the contact patch. With the default value of 0, no smoothing takes place. With a value greater than 0, the patch is defeatured to create a smooth patch. This will lead to better quality volume mesh at the contact, for instance, between the tire and the ground.



Defeaturing Size = 0



Defeaturing Size = 4

5. Specify the **Feature Angle** (from 0 to 90 degrees) that you want to use to extract features on the contact patch object.

6. Use the **Patch Hole** option if you want to create a filled contact patch, as opposed to a hollow contact patch.

7. Use the **Flip** option to create the contact patch in the opposite direction.

8. Once your selection is made, click **Create Contact Patch** and proceed onto the next task.

   If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

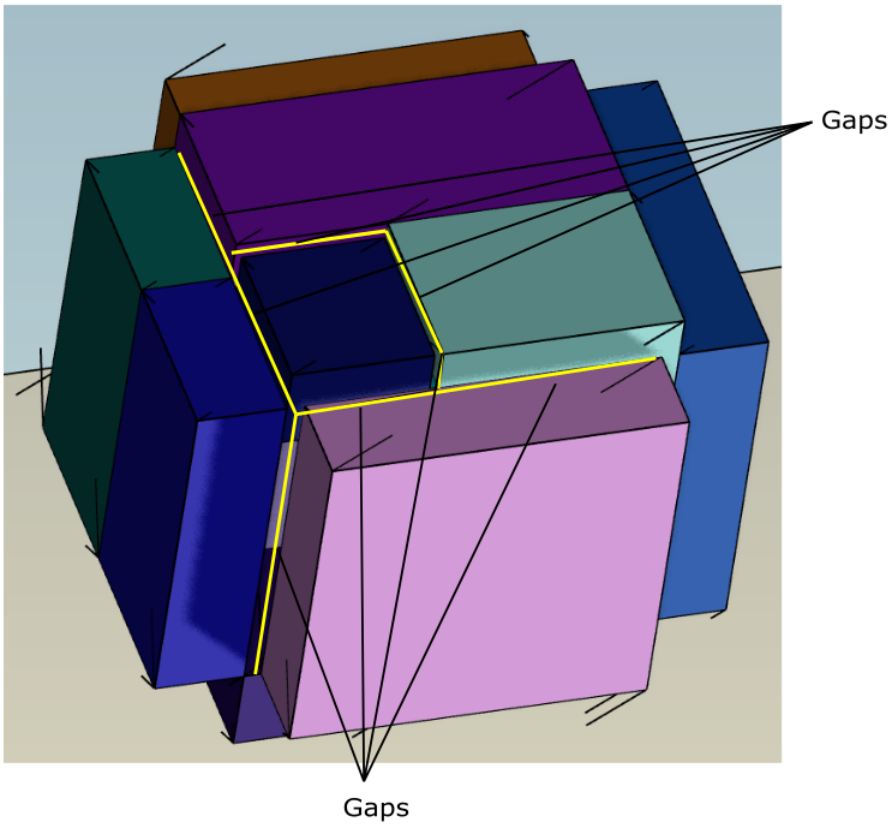
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.17. Creating Gap Covers

If your geometry contains any additional gaps or openings, use the **Create Gap Cover** task to cover such gaps before you generate the surface mesh.

Note:

This task is available to add to your workflow after the **Describe Geometry and Flow** task, and before the **Generate the Surface Mesh** task. Additionally, only a single **Create Gap Cover** task is allowed in your workflow.



1. Specify a **Name** for the gap covering object, or keep the default value (`gap-cover-1`).
2. Select the **Method** to determine how the gap cover is created.

   The default setting of **Uniform Wrapper** requires you to provide a value for the maximum width of the gap (the **Max Gap Size**). Alternatively, you can use the **Wrapper Based on Size Field** option to use the size controls defined in the **Choose Mesh Control Options** task.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The setting of **Wrapper Based on Size Field** requires you to specify the **Max Gap Size Factor**, or keep the default value. The size of the gap(s) that are to be closed is calculated as the product of the gap size factor and the local initial size field.

Note:

If this task is located at a point in the workflow prior to the **Choose Mesh Control Options** task, then you can only use the **Uniform Wrapper** setting for the **Method**.

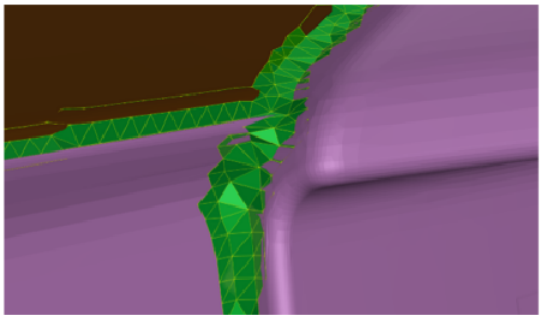
3. Choose whether to **Select By** the object, label, or zone name in the list below, of items that contain the gap that you want to cover.

   Select items in the list, or use the **Filter Text** option in the drop-down to provide text and/or regular expressions in filtering the list (for example, using *, ?, and []). You can also choose the **Use Wildcard** option in the drop-down to provide wildcard expressions in filtering the list. When you use either ? or * in your expression, the matching list item(s) are automatically selected in the list. Use ^, |, and & in your expression to indicate boolean operations for NOT, OR, and AND, respectively. See Filtering Lists and Using Wildcards (p. 287) for more information.

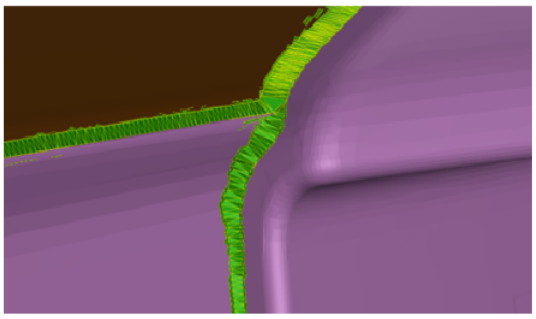
4. For the **Only Between Zones** field, select **Yes** if you want to only cover gaps that exist between different boundary zones, otherwise select **No** to cover gaps both within and between boundary zones.

5. Specify a value for the **Resolution Factor**, or keep the default value. This allows you to control the resolution of the gap cover size based on a scaling of the **Max Gap Size** (or **Max Gap Size Factor**). It ranges from 0.0625 to 1 with a default value of 1.0). The higher the **Resolution Factor**, the more likely that some gaps may not be fully covered. Depending on the gap in question, lowering the **Resolution Factor** reduces the wrapper to sufficiently cover the gap in most cases.

6. Specify a value for the **Refine Factor at the Gap**, or keep the default value. This setting allows you to specify the level of refinemet for the gap-cover (patch). Descreasing the value increases the refinement of the patch.



Refine Factor at the Gap = 1

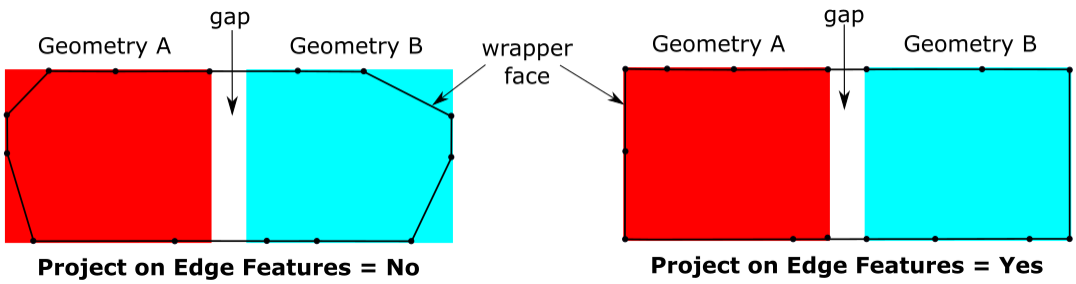


Refine Factor at the Gap = 0.25

7. Specify a value for the **Maximum Number of Island Faces**, or keep the default value. This controls the maximum face count required for isolated areas (islands) to be created during gap cover generation. Any islands that have a face count smaller than this value will be removed, and only larger islands will remain. This allows you to control the maximum number of island faces your gaps may contain, reducing the number of any unwanted artifacts.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

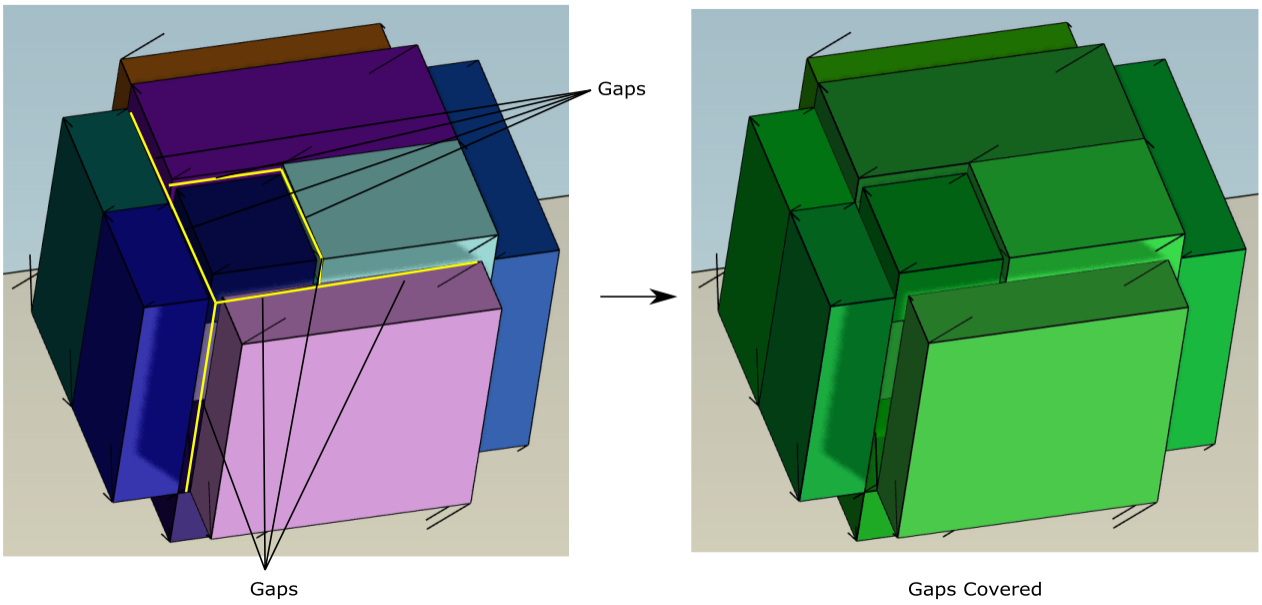
8. Choose whether or not to **Project Nodes on Edge Features?**. When this option is set to **Yes**, the gap covers are more accurate. Once the coarse wrap closes any gaps, this option also snaps the nodes of the wrapper onto all previously defined edge features to more closely cover the gaps. Setting this option to **Yes**, however, can be computationally expensive when modeling large vehicles (such as in aerospace), thus, the default is **No**.



   Here, when set to **No**, wrapper faces at the corners are not on the geometry and are incorrectly marked as a gap. When set to **Yes**, only wrap faces at the gap are marked.

9. Once your selection is made, click **Create Gap Cover** and proceed onto the next task.

   When you are done, any gaps in the geometry will be closed and you will be able to create a surface mesh.



   If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

   When visualizing the results of this task using the icon in the workflow,Fluent Meshing applies transparency to the objects selected for the gap cover, and displays the subsequent gap cover group itself as a solid.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4.18. Generating the Surface Mesh

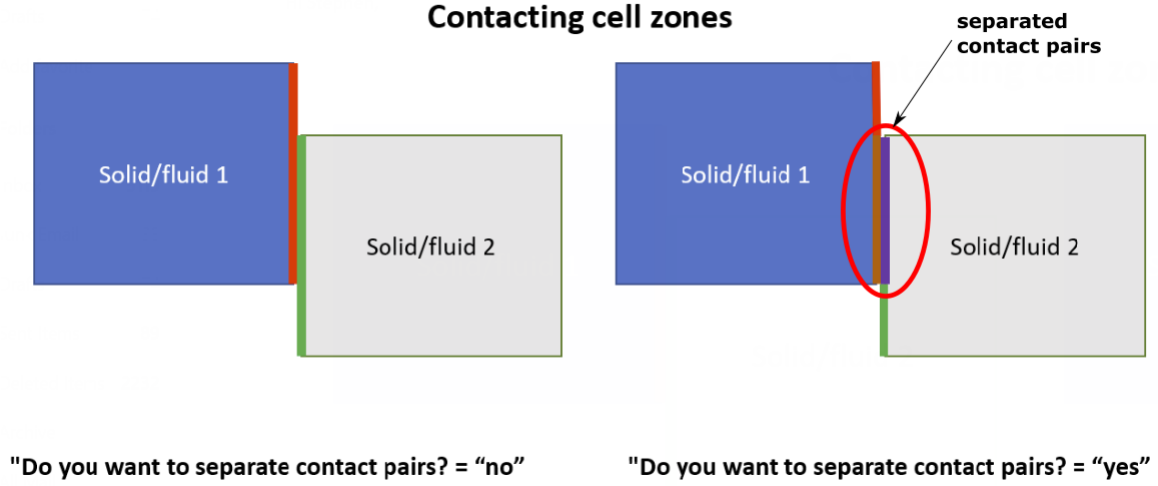
The **Generate the Surface Mesh** task will close all the leakages to objects and void regions and then generate only the surface mesh.

If required, specify the **Surface Mesh Target Skewness**, or keep the default setting. This value is the target maximum surface mesh quality, and is recommended to be between 0.7 and 0.85.

Select **Save Mesh** if you would like to save the surface mesh with your simulation. Additional related options are available under **Advanced Options**.

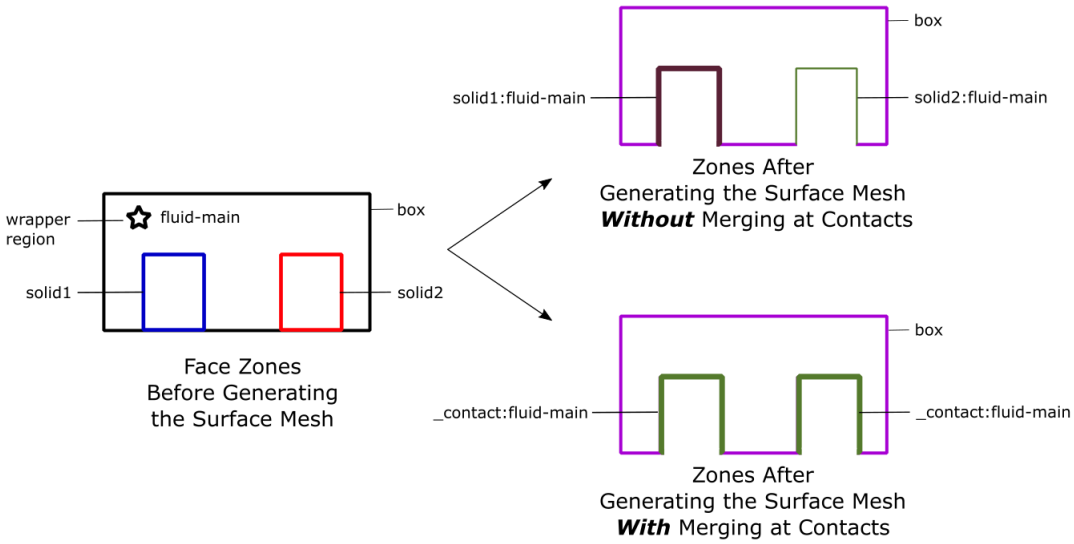
For **Advanced Options**, you can:

- Indicate whether or not you want to save intermediate files. Select **Yes** if you want to save files after each major operation such as computing size field, wrapping, surface remesh, etc. This is useful when meshing large models so that you can load those files of interest during the meshing process. Specify the **Directory** as needed.
- Use the **Separate Contact Pairs?** prompt to indicate whether or not you want to separate contact pairs between fluids and solids in the surface mesh. Contact pairs will be separated during volume meshing and are given the prefix "`i-*`" as in, for example, `i-1-fs-flureg1:brakepad-brake_pad.1:flu-reg-1`, `i-2-fs-flureg1:brakepad-brake_pad.2:flu-reg-1`, and so on.



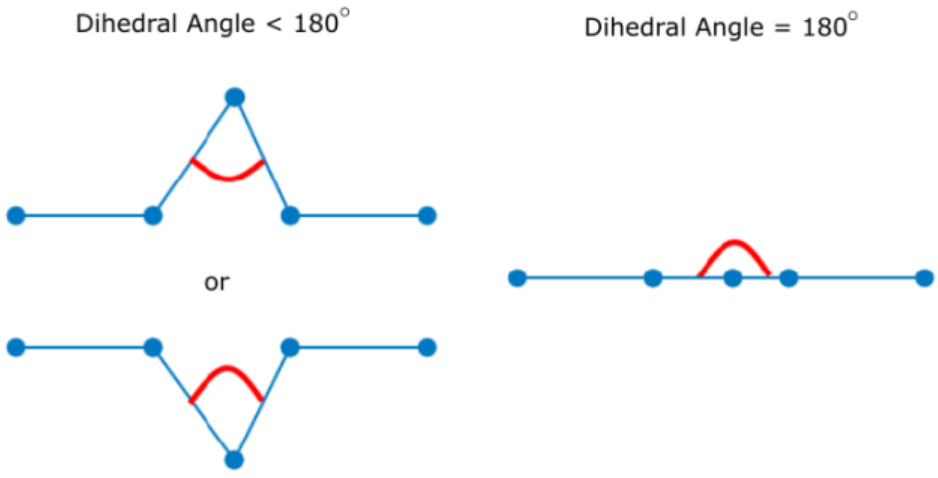
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Specify a value for the **Maximum Number of Island Faces**. Any islands that have a face count smaller than this value will be removed, and only larger islands will remain upon creation of the surface mesh.
- Use the **Merge Wrapper at Solid Contacts** prompt to indicate whether or not you want to allow contacts between solid and fluid regions to be merged into the surface mesh wrapper. When enabled, all bounding faces of a fluid region wrap that come into contact with solid regions will be merged into a single zone (using the prefix `"_contact"`). Each respective wrapped fluid region will have one `"_contact"` zone associated with it.



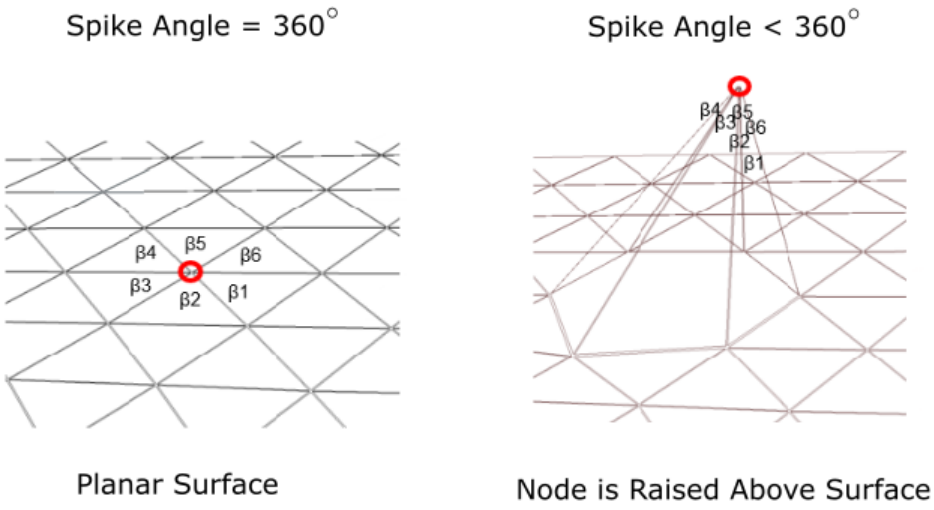
- Use the default values, or specify your own values for the **Min Spike Angle** and the **Min Dihedral Angle**. These correspond to the minimum spike and dihedral angles of the surface mesh before generating the cells. Additional improvements will be applied to the surfaces based on these quality measures.

  Dihedral angles are angles around an edge, such that, for a flat surface, the dihedral angle is 180 degrees, as illustrated below:



  The spike angle is the sum of the angles around a node, such that, for a planar surface, the spike angle is 360 degrees, as illustrated below:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

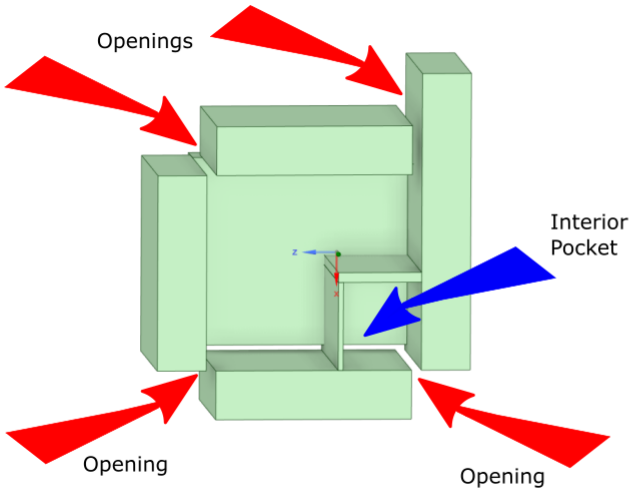
When generating the surface mesh, all spike angles less than 200 degrees and dihedral angle less than 16 degrees are removed. The default values are 200 and 16 degrees respectively. A spike angle of 250 degrees and a dihedral angle of 30 degrees are recommended, however, you should not exceed 260 degrees for the spike angle, or exceed 30 degrees for the dihedral angle.

- Use the **Project on Geometry of the CFD Surface Mesh Objects?** prompt to determine whether, after surface meshing, Fluent will project the mesh nodes back onto to the original CAD model.
- Use the **Auto Assign Zone Types?** option to automatically assign boundary types to zones during surface mesh creation. For example:

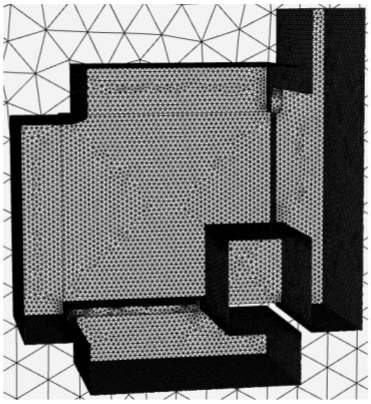
If the Boundary Name Contains...	**...Assign Boundary Type to...**
*mass*inlet*	mass flow inlet
*press*inlet*	pressure inlet
inlet	velocity inlet
*mass*outlet*	mass flow outlet
outlet	pressure outlet
symmetry	symmetry
*far*field*	pressure far field
outflow	outflow
wall	wall
overset	overset
internal or *interior*	internal
interface	interface

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

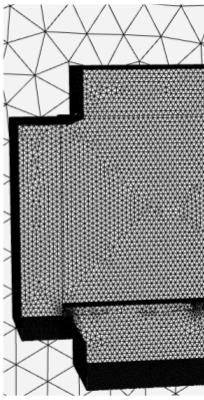
- Use the **Merge Interior Pockets?** field to determine whether or not to extend or expand the surface mesh into any interior cavities or pockets.



Interior View of Geometry with an Interior Pocket

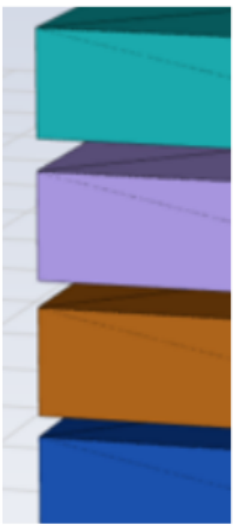


Surface Mesh (Merge Interior Pockets? = No)

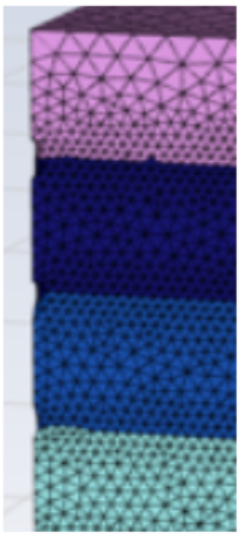


Surface Mesh (Merge Int

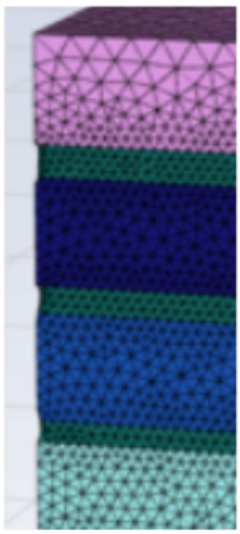
- When gap covers are present, use the **Gap-Cover Zone Recovery** field to determine whether or not to keep or remove the zones representing the cap covers. When set to **Yes**, the zones representing the gap covers are retained, whereas when set to **No** (the default), the zones for the gap covers are removed.



Geometry With Gaps



Gap-Cover Zone Recovery set to **No**



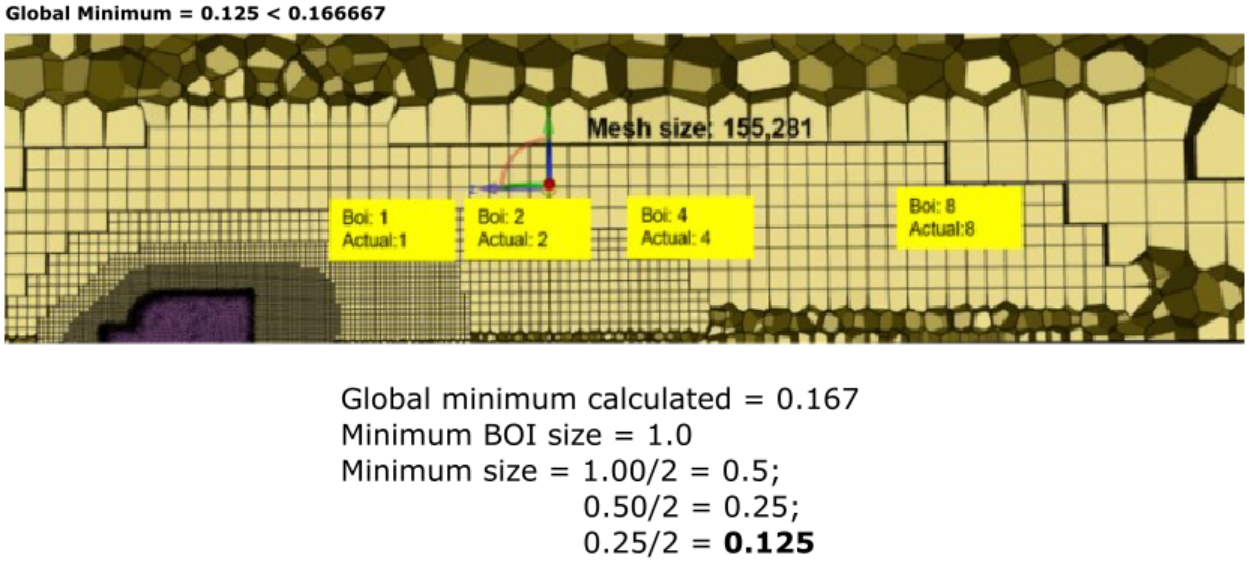
Gap-Cover Zone Recovery set to **Yes**

- Specify a value for the **Global Minimum** for the surface mesh. The default value for the **Global Minimum** is determined by considering your available size controls and bodies of influence.

  The workflow first determines the lowest value of all minimum values out of all the size controls, considering both wrap and target size control types (for example, 0.167). Then, the workflow searches for the smallest value for the body of influence (for example, 1). The workflow will then divide the smallest BOI value in half repeatedly (for example, 1/2 = 0.5, 0.5/2 = 0.25, 0.25/2 =

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0.125, etc.), until the result is smaller than the smallest minimum value (for example, 0.167). Therefore, in this example, the global minimum is then set to a default value of 0.125.



Global minimum calculated = 0.167
Minimum BOI size = 1.0
Minimum size = 1.00/2 = 0.5;
0.50/2 = 0.25;
0.25/2 = **0.125**

This leads to an improvement when considering polyhedra and hexahedra cells, where previously, the adjusted minimum size was not utilized and resulted in an increased number of cells.

Once your selections are made, click **Generate the Surface Mesh** and proceed onto the next task.

The generated surface mesh is applied to all fluid and solid regions (if applicable). By default, the final surface mesh is viewed as separated zones.

If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

4.19. Updating Boundaries

Use the **Update Boundaries** task to update the properties of any defined boundary. This task is automatically part of the workflow, prior to generating the surface mesh, and can be added to the workflow as many times as you require.

1. (optional) Use the **Filter** button to filter the table contents based on a particular column.
2. (optional) Rename any **Boundary Name** by double-clicking the label in the table and entering a new name.

   You can also rename a boundary by selecting it in the table, right-click, and select **Set Boundary Name** in the context menu, and provide a new name directly in the menu.
3. (optional) Re-assign any **Boundary Type** to another value by selecting a type in the table and using the corresponding drop-down menu. Choices include:

   - velocity-inlet
   - pressure-outlet

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- pressure-inlet
- pressure-far-field
- mass-flow-inlet
- mass-flow-outlet
- outflow
- symmetry
- wall
- internal
- interface
- overset

Multiple boundaries can be assigned a specific type all at once by selecting them in the table, right-click, and select **Set Boundary Type** in the context menu, then designate a type for the selected boundaries directly in the menu.

4. (optional) By default, the table only lists external wall boundaries. Select the **List All Boundaries** option to see additional boundaries, such as fluid-fluid internal boundaries.
5. Use the **Draw Boundaries** button to visualize boundaries on **All Zones**, **Inlet Zones**, **Outlet Zones**, or **Wall Zones**.

   Multiple boundaries can be visualized all at once by selecting them in the table, right-click, and select **Draw Selections** in the context menu.
6. When you are satisfied with the boundary assignments, click **Update Boundaries**.

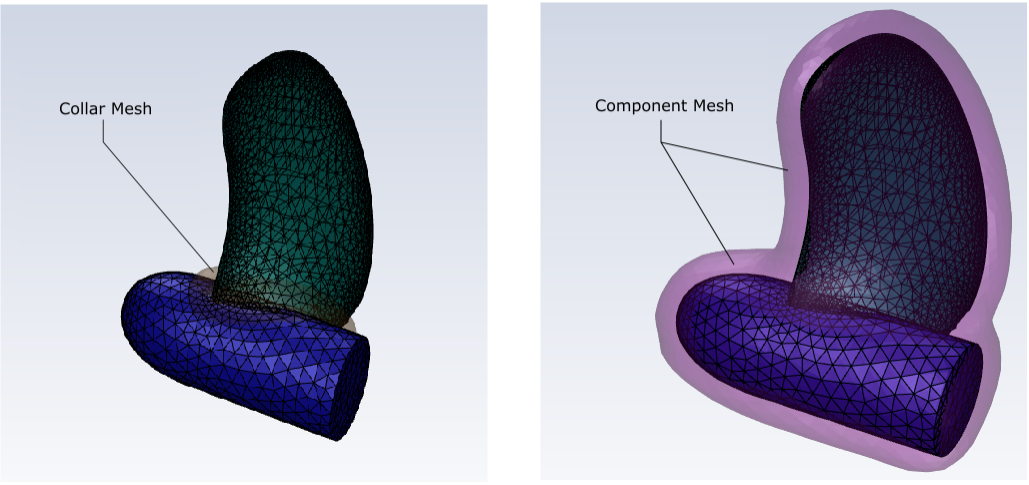
   If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.
7. Once all boundaries have been updated, proceed to the next step in the workflow.

4.20. Describing Overset Features

Use the **Describe Overset Features** task to add additional overset meshing-specific tasks to the workflow. Two common types of meshes are available when using overset meshing: collar meshes and component meshes.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 4.33: Collar and Component Meshes for a Propeller and Hub Geometry



A *collar* mesh is a one that surrounds an intersecting point of a geometry. To add a collar mesh to your overset mesh, select **Yes** for the **Do you want to add a collar mesh?** prompt. This will add a **Create Collar Mesh** to your workflow. See Creating Collar Meshes (p. 472) for more information.

A *component* mesh is a one that closely surrounds a specific component of interest of a geometry. To add a component mesh to your overset mesh, select **Yes** for the **Do you want to add a component mesh?** prompt. This will add a **Create Component Mesh** to your workflow. See Creating Component Meshes (p. 474) for more information.

Note:

For more information about overset meshes and using them with Fluent in solution mode, see Overset Meshes (p. 1206)

4.20.1. Creating Collar Meshes

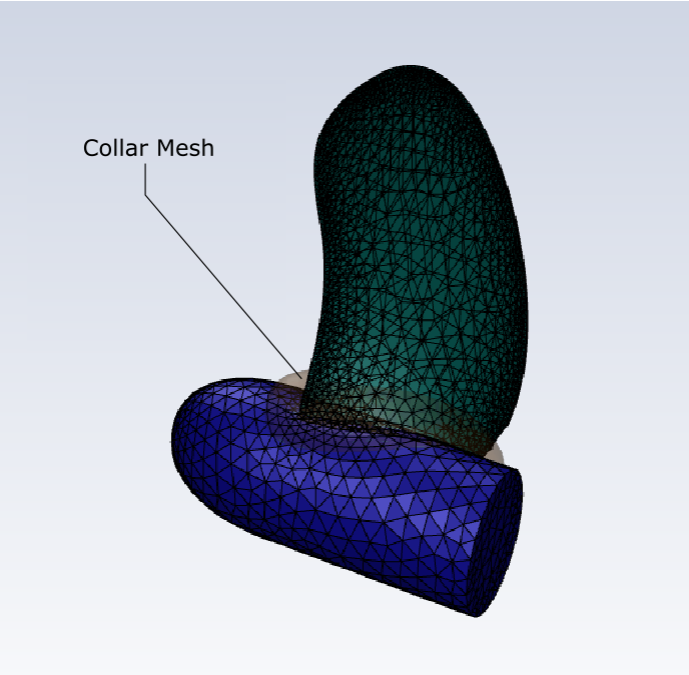
Use this task to create a collar mesh for use with overset meshing in your workflow.

Note:

For more information about collar meshes, see Overset Topologies (p. 1208)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 4.34: A Collar Mesh for a Propeller and Hub Geometry



A collar mesh is often necessary when you have two intersecting bodies and you are considering overset meshing.

1. Specify a **Name** for the collar mesh, or use the default name (`collar-mesh-1`).
2. Choose the **Creation Method** that you will use to create the collar mesh.
   - Use the **Intersected Objects** option to create the collar mesh based on two selected intersecting bodies.
   - Use the **Intersected Zones** option to create the collar mesh based on two intersecting zones, for example, if you have one zone per face or if you use named selections in SpaceClaim to create separate zones.
   - Use the **Edge Based Collar** option to create the collar mesh using selected edges.
   - Use the **Existing** option if you have an existing object that can represent the collar mesh that already is part of your imported geometry.
3. Specify a value for the **Max Cell Size**.
4. For **Fill With**, choose the type of mesh cell to use to fill the collar mesh. Available options are **tetrahedral**, **hexcore**, **poly**, or **poly-hexcore**.
5. Depending on your choice of **Creation Method**, make an appropriate selection from the list.

   Choose whether to **Select By** the object, label, or zone name in the list below.

   Select items in the list, or use the **Filter Text** option in the drop-down to provide text and/or regular expressions in filtering the list (for example, using *, ?, and []). You can also choose the **Use Wildcard** option in the drop-down to provide wildcard expressions in filtering the

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

list. When you use either ? or * in your expression, the matching list item(s) are automatically selected in the list. Use ^, |, and & in your expression to indicate boolean operations for NOT, OR, and AND, respectively. See Filtering Lists and Using Wildcards (p. 287) for more information.

6. If you have selected **Intersected Objects** or **Edge based Collar** as the **Creation Method**, you can specify a value for the **Total Thickness** that you want to use as the thickness of the collar mesh. By default, this value corresponds to four times the value of the **Max Cell Size**.

7. Use the **Define Location Using** field to choose how you want to assign the location of the offset collar mesh. You can choose **Automatic** to have the system determine the best place for the collar mesh based on the intersected bodies or zones, or you can choose to use distinct **Numerical Inputs** for **X**, **Y**, and **Z** to have more control over the collar mesh location.

   Note:

   When creating the collar mesh, you should only use the **Automatic** option to preview the location of the collar mesh, but you should use the **Numerical Inputs** option to define the actual location.

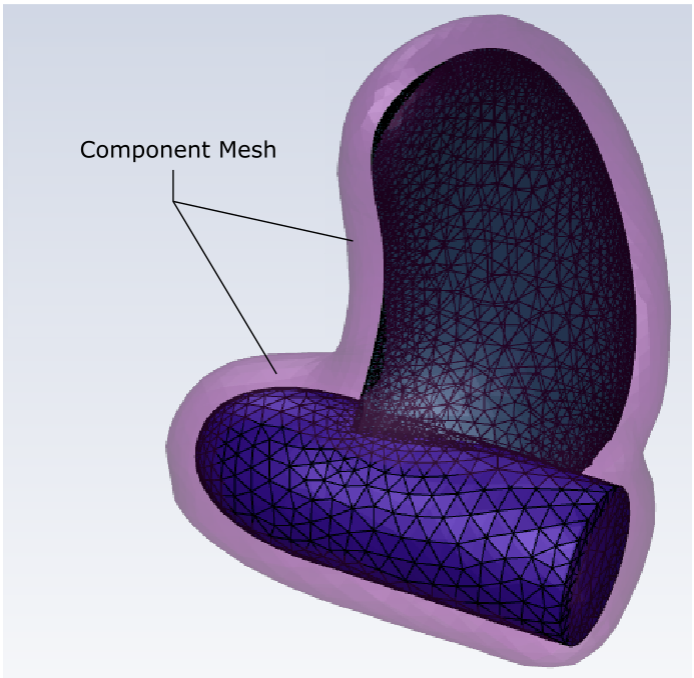
8. Click **Create Collar Mesh**. The collar mesh will appear in the graphics window.

   If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

4.20.2. Creating Component Meshes

Use this task to create a component mesh for use with overset meshing in your workflow.

Figure 4.35: A Component Mesh for a Propeller and Hub Geometry



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A component mesh is often necessary when you want to have a localized mesh around one or more specific portions of your geometry while considering overset meshing.

Note:

For more information about component meshes, see Overset Topologies (p. 1208)

1. Specify a **Name** for the component mesh, or use the default name (`component-mesh-1`).
2. Choose the **Creation Method** that you will use to create the component mesh.
   - Use the **Overset Offset Surface** option to create the component mesh based on a surface that is offset from one or more selected bodies or zones.
   - Use the **Background Box** option to create the component mesh using a bounding box around the imported geometry that represents the background (for example, an enclosing box around a fan blade assembly, representing the area around the fan geometry).
   - Use the **Existing** option if you have an existing object that can represent the component mesh that already is part of your imported geometry.
3. Specify a value for the **Max Cell Size**.
4. For **Fill With**, choose the type of mesh cell to use to fill the component mesh. Available options are **tetrahedral**, **hexcore**, **poly**, or **poly-hexcore**.
5. Choose whether to **Select By** the object, label, or zone name in the list below.

   Select items in the list, or use the **Filter Text** option in the drop-down to provide text and/or regular expressions in filtering the list (for example, using *, ?, and []). You can also choose the **Use Wildcard** option in the drop-down to provide wildcard expressions in filtering the list. When you use either ? or * in your expression, the matching list item(s) are automatically selected in the list. Use ^, |, and & in your expression to indicate boolean operations for NOT, OR, and AND, respectively. See Filtering Lists and Using Wildcards (p. 287) for more information.
6. When the **Creation Method** is set to **Overset Offset Surface**, under **Offset Properties**, you can also specify a value for the **Total Thickness** that you want to use as the thickness of the component mesh.
7. When the **Creation Method** is set to **Background Box**, under **Box Parameters**, additional settings are required.
   - When you select **Ratio relative to geometry size**, the **Box Parameters** hold the minimum and maximum extension ratios for the X, Y, and Z dimensions. By default, they are set to extend the dimensions of a bounding box around your selected object(s) or zone(s) out by a factor equivalent to the total singular length of the geometry in the X, Y, and Z directions. You can change the values according to your needs, and the display in the graphics window will change accordingly.
   - When you select **Directly specify coordinates**, the **Box Parameters** hold the minimum and maximum extension *distance* for the X, Y, and Z dimensions. By default, they are set to values that correspond to the extension ratios, but you can set them to more

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

suitable values as required, and the display in the graphics window will change accordingly.

8. Click **Create Component Mesh**. The component mesh will appear in the graphics window.

   If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

4.21. Adding Boundary Layers

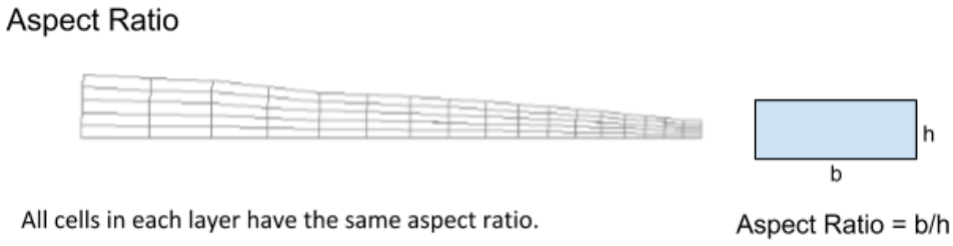
For most fluid regions, the boundary layer flow along the walls of the geometry is best captured using specialized boundary layer elements within the volume mesh (also called *prisms* or *inflation layers*). Prism elements are generated on the walls of the geometry that face the fluid region (typically referred to as "wetted" walls). You can use the **Add Boundary Layers** task to assign different regions to have their own boundary layer controls. This task can be added to the workflow as many times as you require. When a boundary layer is assigned to a region, all face zones associated with that region will be assigned those settings. Any zone-specific settings (except the number of layers) defined in the **Prisms** dialog are respected. For more information about prisms, see Generating Prisms (p. 750) and Growth Options for Zone-Specific Prisms (p. 757) and Prism Meshing Options for Scoped Prisms (p. 775).

For the **Add Boundary Layers?** field, select **yes** if your model requires you to locate and address any boundary layers using the following steps. Otherwise, if your model does not require you to have boundary layers, keep the default of **no**, click **Update** and proceed to the next task.

1. Specify a **Name** for the boundary layer, or use the default value. Note that the default name is dependent on the value of the **Offset Method Type**.

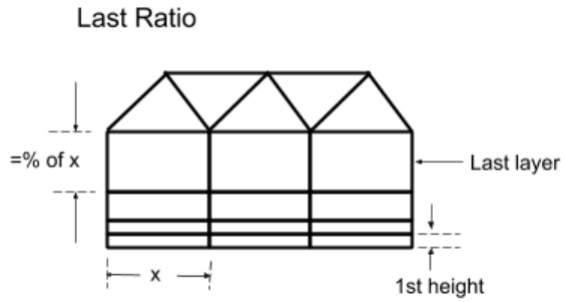
2. Choose an **Offset Method Type**. The offset method that you choose determines how the mesh cells closest to the boundary are generated. See Offset Distances (p. 765) for more information. Choices include:

   - **aspect-ratio**: allows you to control the aspect ratio of the boundary layer cells (or prism cells) that are extruded from the base boundary zone. The aspect ratio is defined as the ratio of the prism base length to the prism layer height.

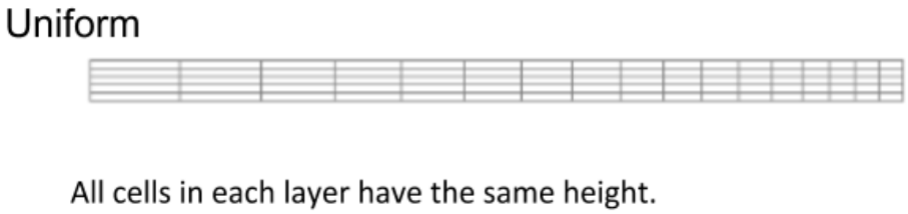
     

   - **last-ratio**: allows you to control the aspect ratio of the boundary layer cells (or prism cells) that are extruded from the base boundary zone. You can specify **First Height** for the first prism layer.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- **uniform**: allows you to generate every new node (child) to be initially the same distance away from its parent node (that is, the corresponding node on the previous layer, from which the direction vector is pointing).



i. If the **Offset Method Type** is set to **last-ratio**, specify the **Transition Ratio**. This value determines the ratio of the height of the last layer in the inflation and the first cell in the volume fill.

ii. If the **Offset Method Type** is set to **aspect-ratio**, specify the **First Aspect Ratio**. You can control the heights of the inflation layers by defining the aspect ratio of the inflations that are extruded from the inflation base. The aspect ratio is defined as the ratio of the local inflation base size to the inflation layer height. The value for the **First Aspect Ratio** allows you to specify the first aspect ratio to be used.

In addition, you can specify the **First Height**. This value is the height of the first layer of cells in the boundary layer.

iii. If the **Offset Method Type** is *not* set to **last-ratio**, provide the **Growth Rate**. This value determines the relative thickness of adjacent inflation layers. As you move away from the face to which the inflation control is applied, each successive layer is approximately one growth rate factor thicker than the previous one.

3. Specify the **Number of Layers**. This value determines the maximum number of boundary layers to be created in the mesh.

4. For the **Add in** field, specify what regions you would like to add the boundary layers.

   - Use the **fluid-regions** option to add the boundary layer to just the fluid regions.

   - Use the **named-regions** option to select from a list of available named region(s) in your simulation.

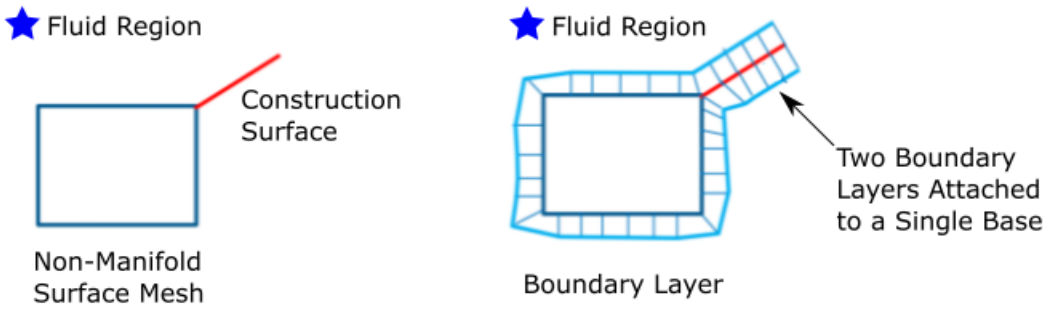
5. For the **Post Improvement Method** option, specify how you would like to improve the boundary layer after its been generated in the specified area(s): as a **Continuous** or a staggered **Stair Step** boundary layer.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Continuous** method does not support baffles, and you should avoid using it if your specified region(s) contains any such structures; you should instead use the **Stair Step** method, where baffles are supported, however, they cannot be split.

Note:

Since continuous boundary layers are only applicable for manifold regions, the task will not create boundary layers if it detects a non-manifold surface mesh (for example, when two boundary layers are attached to a single base, such as when considering a construction surface around a non-manifold surface mesh).



See Prism Meshing Options for Scoped Prisms (p. 775) for more information.

6. For the **Grow on** field, specify where you would like to develop the boundary layers.

   - Use the **only-walls** option to grow the boundary layer along just wall surfaces.
   - Use the **all-zones** option to grow the boundary layer along all surfaces or zones.
   - Use the **selected-zones** option to select from a list of available **Zones** along which to grow the boundary layer. This option is available when **Add In** is set to **named-regions** and when **Stair Step** is selected as the **Post Improvement Method**.
   - Use the **selected-labels** option to select from a list of available **Labels** along which to grow the boundary layer.
   - For the **selected-zones** and **selected-labels** options, select items in the list, or use the **Filter Text** option in the drop-down to provide text and/or regular expressions in filtering the list (for example, using *, ?, and []). You can also choose the **Use Wildcard** option in the drop-down to provide wildcard expressions in filtering the list. When you use either ? or * in your expression, the matching list item(s) are automatically selected in the list. Use ^, |, and & in your expression to indicate boolean operations for NOT, OR, and AND, respectively. See Filtering Lists and Using Wildcards (p. 287) for more information.

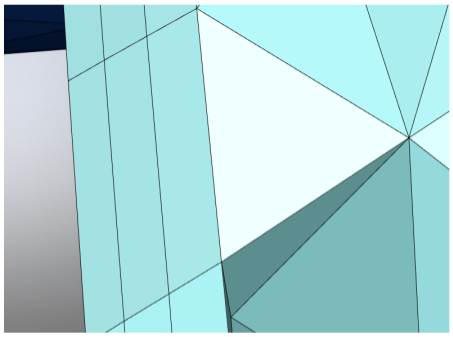
7. For the **Split Prisms?** field, choose whether or not to provide split prism layers within the individual boundary layers. This option is not available when the **Offset Method Type** is set to **last-ratio**.

   When set to **Yes**, specify the **Number of Split Prisms Per Layer**. This can provide additional refinement in the flow volume's boundary layer.

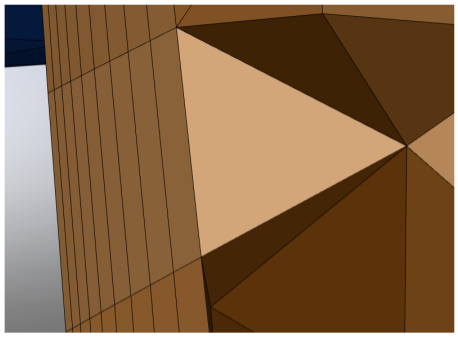
   For instance, for a boundary layer definition where the **Number of Layers** is set to **3**, then, when setting **Split Prisms?** to **Yes**, and setting the **Number of Split Prisms Per Layer** to **3** will, once the

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

volume mesh is generated, provide three layers of split prisms for each of the defined layers, for a total of nine layers along the boundary.



Number of Layers = 3
Split Prism? = No



Number of Layers = 3
Split Prism? = Yes
Number of Split Prisms Per Layer = 3

Note:

- When using the **Stair Step** method, if you choose to split prisms (setting **Split Prisms?** to **Yes**), you must also ensure that the **Number of Layers** is the same for all stair-step tasks, however, the **Number of Split Prisms Per Layer** can be different. There are no restrictions on the **Number of Layers** when you choose not to split prisms.
- When using the **Continuous** method (where generally, "continuous" means that prisms have the same number of layers from all zones), if you choose to split prisms (setting **Split Prisms?** to **Yes**),the **Number of Layers** can only be defined in the first continuous prism task. Subsequent continuous prism layer tasks can be added by setting **Grow on** to **selected-zones** where you cannot change the number of prism layers, however, you can change other parameters (such as the aspect ratio, number of splits, etc.). In other words, the first continuous layer task sets parameters for the region and any additional, subsequent prism layer tasks overwrite all parameters except for the number of layers on the **selected-zones**.

  The "continuous" exception is from inlets and outlets when you choose to grow the prism layers along **only-walls**. If the first continuous task is set to **only-walls**, Fluent does not allow for split prisms on any additional subsequent continuous task because an inlet or outlet may have been selected for this task (where inlets and outlets do not have any layers to split because they were **only-walls** be begin with).
- Just as you can define a broad variety of boundary layer specifications, you can also split prisms along one or more boundaries with various combinations of growth mechanisms and techniques.

8. (optional) Use the **Draw Regions** button to visually display all regions, or just the fluid region(s).
9. Click **Add Boundary Layers** to generate the appropriate boundary layers for the imported CAD geometry.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you need to make adjustments to any of your settings in this task, click **Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

4.22. Identifying Deviated Faces

After generating a surface mesh and setting up boundary layers, the **Identify Deviated Faces** task is available to help you identify how the wrapped surface mesh differs from the original geometry. This task can be useful for identifying deviations in the surface mesh in, for example, geometries with sharp angles.

1. Specify a **Name** for the deviation control, or use the default value.

2. Use the **Select By** field to choose whether to select the object name or the zone name in the list below.

   Select items in the list, or use the **Filter Text** option in the drop-down to provide text and/or regular expressions in filtering the list (for example, using *, ?, and []). You can also choose the **Use Wildcard** option in the drop-down to provide wildcard expressions in filtering the list. When you use either ? or * in your expression, the matching list item(s) are automatically selected in the list. Use ^, |, and & in your expression to indicate boolean operations for NOT, OR, and AND, respectively. See Filtering Lists and Using Wildcards (p. 287) for more information.

3. Keep the **Auto Compute** field enabled so that the **Minimum Deviation** and the **Maximum Deviation** are automatically calculated. Otherwise, disable it and specify your own values for the **Minimum Deviation** and the **Maximum Deviation**.

4. Use the **Overlay with** options to determine how you want the deviated faces to be displayed (with or without the **Mesh** or **Geometry**).

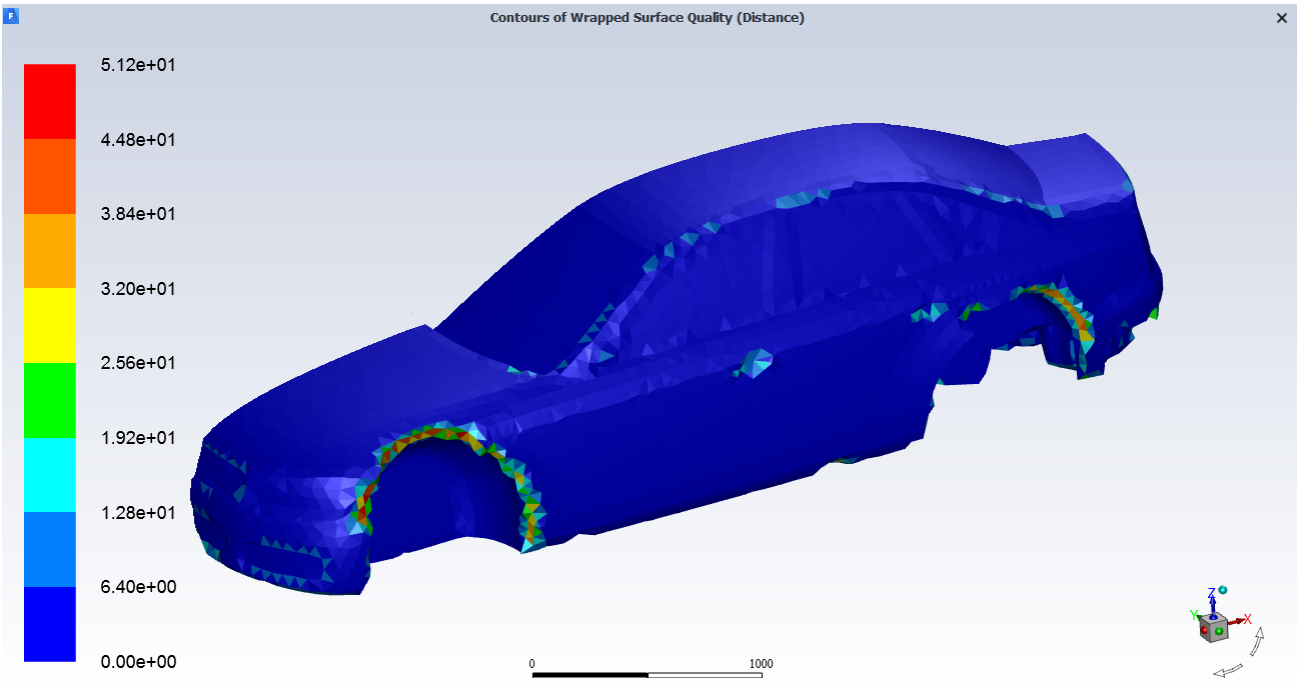
5. Use the **Include Gap Cover Geometry** option to determine if you want to include any gap covers (Creating Gap Covers (p. 463)) in the check for deviated faces. If so, the default minimum and maximum deviation range is automatically calculated.

6. Use the **Deviation** drop-down button to see visualization options for identifying any irregularities in the surface mesh.

   - Select the **Draw Contours** option to see any irregular faces in the graphics window, displayed as contours of wrapped surface quality where you can identify areas of the surface mesh that do not exactly match the original geometry.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 4.36: Example of Surface Mesh Quality Contours



- Select the **Compute** option to calculate values for the **Minimum Deviation** and the **Maximum Deviation**.

7. Once your selections are made, click **Identify Deviated Faces** and proceed onto the next task.

   If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

4.23. Generating the Volume Mesh

The **Generate the Volume Mesh** task will perform various operations and generate the volume mesh.

If required, specify the **Quality Improve Limit**, or keep the default setting. This sets the threshold for when mesh quality improvements are automatically invoked that employ the orthogonal quality limit, and is recommended to be around 0.04.

Select **Enable Parallel Meshing for Fluids** if you would like to perform parallel volume meshing on your fluid region(s) for poly, hexcore and poly-hexcore volume fill types. When parallel is enabled, both boundary layer generation methods are parallelized: continuous and stair-step (also called "scoped").

Select **Save Mesh** if you would like to save the volume mesh file with your simulation.

If you would like to review and change the volume meshing technique for your region(s), you can select the **Enable Region Settings** option. This displays a tabular overview of the simulation's volume region(s) (including overset component regions) and their mesh settings (formerly set in the **Update Region Settings** task), that you can change accordingly before finally generating the volume mesh.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Choose a **Volume Fill**, or keep the default setting. For both fluids and solids, you can choose from **tet**, **hexcore**, **poly**, or **poly-hexcore**.

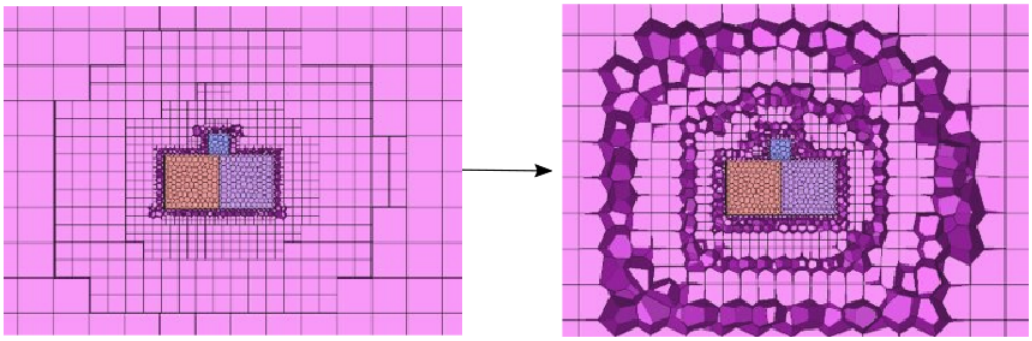
  Multiple regions can be assigned a specific volume fill type all at once by selecting them in the table, right-click, and select **Set Volume Fill** in the context menu, then designate a volume fill type for the selected regions directly in the menu.

- The **Max Cell Length** can be changed on a per-region basis as well, if desired.

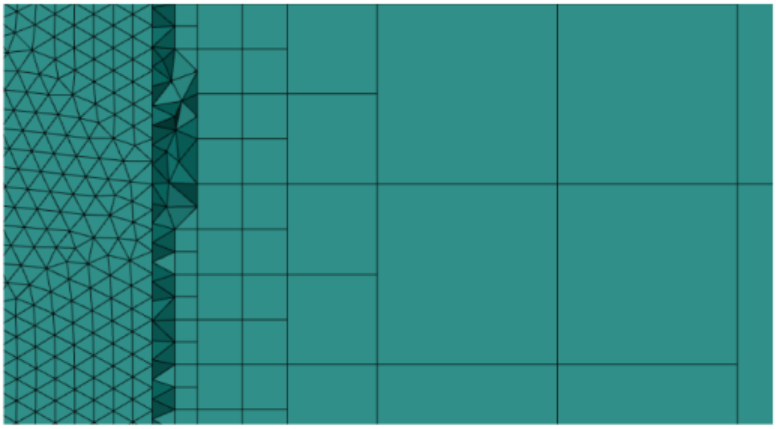
For **Advanced Options**,

- Use the **Quality Method** option to choose from different types of mesh quality controls (aspect ratio, change in size, and so on). Choices include **Orthogonal** (the default for the workflows) and **Enhanced Orthogonal**. For more information, see Quality Measure (p. 566). The quality method chosen here will also be used in the **Improve Volume Mesh** task if you later add that task to the workflow, however, you can still choose a different quality method for that task if desired.

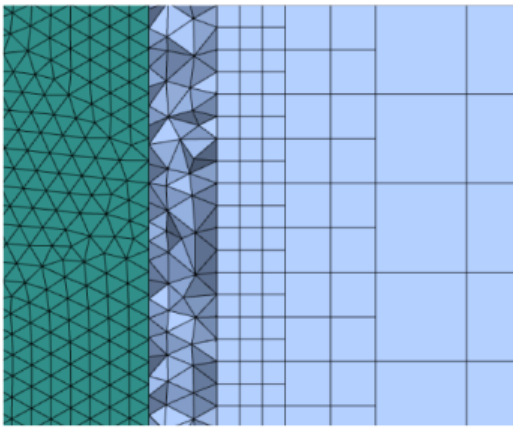
- Use the **Avoid 1/8 Octree Transition in Hexcore Region?** option to determine whether or not you want to avoid any potential 1:8 cell transition in the hexcore or polyhexcore region of the volume mesh, replacing any abrupt change in the cell size with tetrahedral or polyhedral cells.



- Use the **Octree Peel Layers** option to define the number of octree layers to be removed between the boundary and the core (see Peel Layers (p. 799) for details). The resulting cavity will be filled with tet cells for hexcore meshes and with poly cells for polyhexcore meshes.



Peel Layer = 0

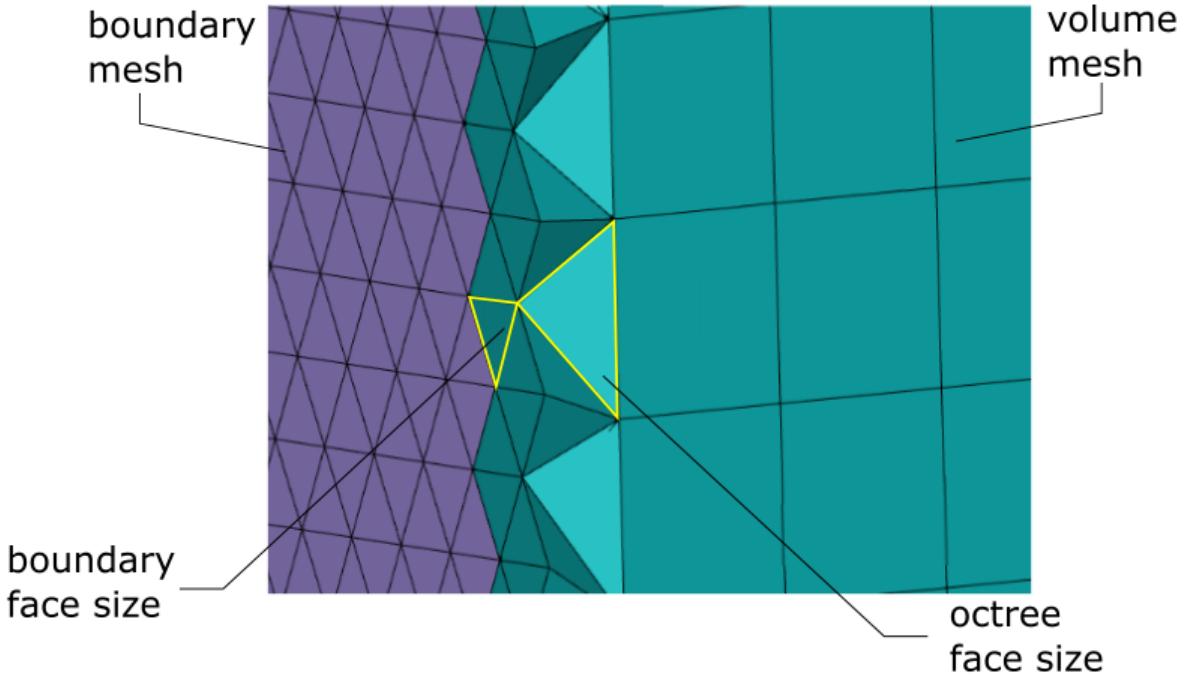


Peel Layer = 3

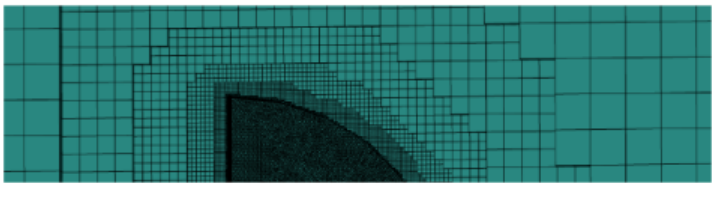
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Use the **Use Size Field** option to determine whether or not you want to use size fields when generating the volume mesh. Generating the volume mesh using size fields can require additional memory as you increase the number of processing cores. This is because the size field is replicated for each core as the size field is not properly distributed. When using size fields, you are limited by the size of the machine. When not using size fields, however, you require less memory and you can use a higher number of cores with limited RAM, leading to a faster mesh generation. When a size field is not used, then you must also make adjustments, if any, to the following fields:

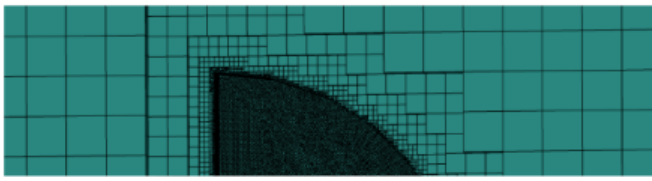
  – Use the **Octree/Boundary Size Ratio** option to set the ratio between the octree face size and the boundary face size. The default is 2.5 such that the octree mesh near the boundary is 2.5 times larger than the boundary mesh.



  – Use the **Buffer Layers** option to set the number of buffer layers for the octree volume mesh. If size controls have not been defined previously, then the default is 2, otherwise the default is calculated based on the maximum growth size. The buffer layers are additional layers of cells to alleviate a rapid transition from finer cells to coarser cells (see Buffer Layers (p. 799) for details).



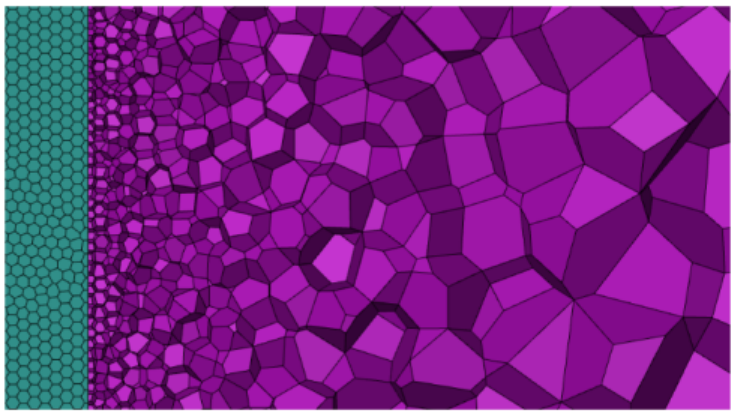
Buffer Layer = 4



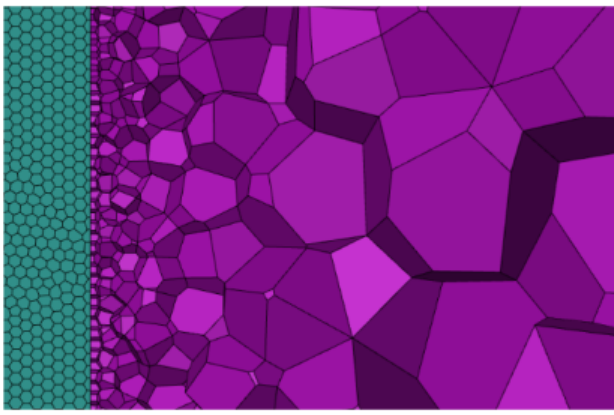
Buffer Layer = 1

  – Use the **Tet/Poly Growth Rate** option to specify the maximum growth rate for tet and poly cells. By default, this corresponds to a growth rate of 1.2.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

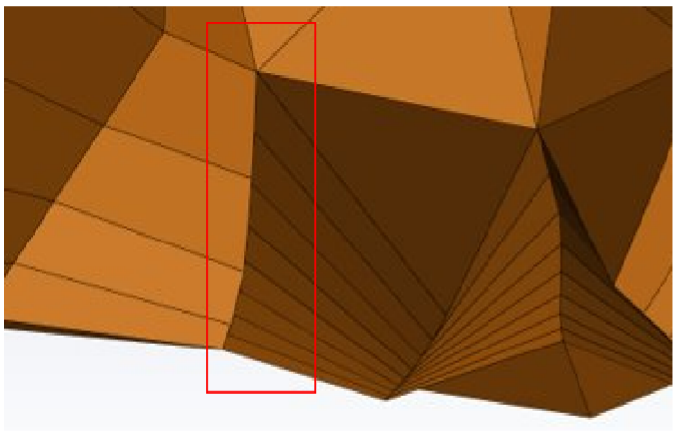


Growth Rate = 1.2

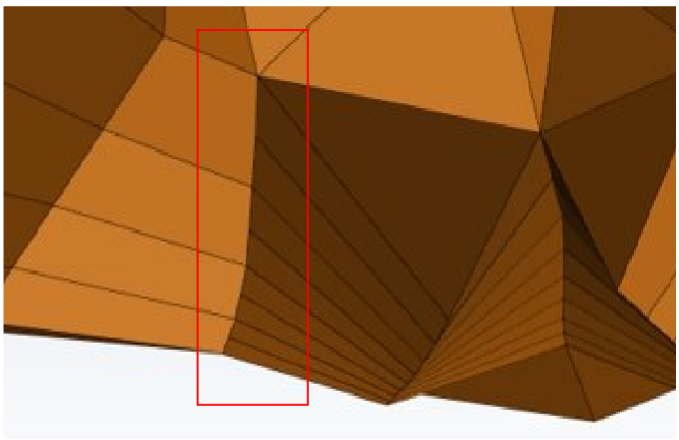


Growth Rate = 1.6

- Neighboring zones with different numbers of layers will lead to conformal prism layers between them. Use the **Conformal Prism Split** option to determine whether you want to split the boundary layer cells conformally or not. When this option is set to **Yes**, the prism sides of the two zones will share nodes. This option is only available when stair-stepping is invoked. Note that adjacent regions should have an even ratio of prism layers when using this option.



Conformal Prism Split = No



Conformal Prism Split = Yes

- The **Tet-Prism Stairstep Exposed Quads?** option can be used when generating a tetrahedral mesh with prism cells and is set to **No** by default. Selecting **Yes** for this option will enable stair-stepping for exposed quadrilateral faces (exposed quads) on prism cells. Stair-stepping will prevent pyramids from being created on these exposed quads, which generally would lead to poor quality in the exposed quad location.

Note:

For all volume fill methods other than **tetrahedral**, the **Tet-Prism Stairstep Exposed Quads?** option is set to **No** and can not be changed.

Once your selection is made, click **Generate the Volume Mesh** and proceed onto the next task.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

From this task, if you need to address any surface contact issues, you can add a **Separating Contacts** task to your workflow (see Separating Contacts (p. 494)).

Note:

When using the meshing workflows, after generating the volume mesh, the default quality measure is set to Orthogonal Quality, and will be reported as such when querying the mesh quality

---

4.24. Creating Overset Mesh Interfaces

Use the **Create Overset Interface** task to create an interface between selected overset mesh objects.

1. Specify a **Name** for the overset interface, or use the default (`overset-interface-1`).

2. Select two or more mesh objects from the list.

   Select items in the list, or use the **Filter Text** option in the drop-down to provide text and/or regular expressions in filtering the list (for example, using *, ?, and []). You can also choose the **Use Wildcard** option in the drop-down to provide wildcard expressions in filtering the list. When you use either ? or * in your expression, the matching list item(s) are automatically selected in the list. Use ^, |, and & in your expression to indicate boolean operations for NOT, OR, and AND, respectively. See Filtering Lists and Using Wildcards (p. 287) for more information.

3. Once your selection is made, click **Create Overset Interface** and proceed onto the next task.

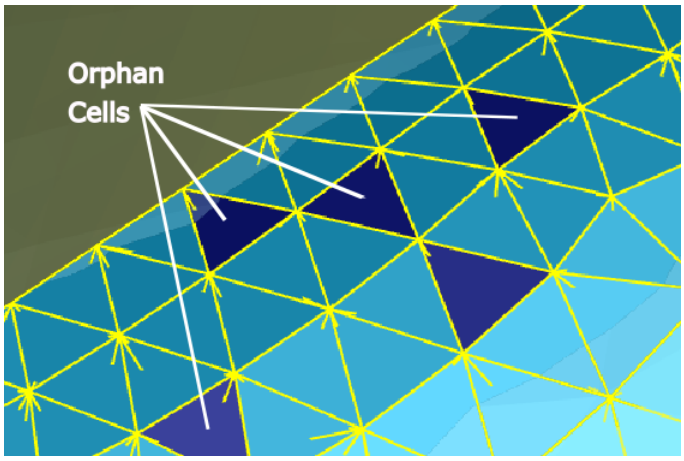
   If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

4.25. Identifying Orphans

Occasionally, creating overlapping overset meshes creates mismatched cells, or orphan cells. The presence of orphan cells generally indicates that there is insufficient overlap between meshes or that the mesh resolutions do not match well. See Overset Meshes (p. 1206) for more information.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 4.37: Example of Orphan Cells in an Offset Mesh



Use the **Identify Orphans** task to identify any orphan cells In selected overset meshed.

1. Select one or more mesh objects from the list.

   Select items in the list, or use the **Filter Text** option in the drop-down to provide text and/or regular expressions in filtering the list (for example, using *, ?, and []). You can also choose the **Use Wildcard** option in the drop-down to provide wildcard expressions in filtering the list. When you use either ? or * in your expression, the matching list item(s) are automatically selected in the list. Use ^, |, and & in your expression to indicate boolean operations for NOT, OR, and AND, respectively. See Filtering Lists and Using Wildcards (p. 287) for more information.

2. Choose a **Donor Priority Method**, that will control the location of the overset mesh. See Overset Domain Connectivity (p. 1212) for more details about orphans, donors, and receptors in overset meshes.

   - Select **Cell Volume Based** to ensure the donor priority is based on the cell size (proportional to the inverse of the cell volume)

   - Select **Boundary Distance Based** to ensure the donor priority is based on the boundary distance (proportional to the inverse of the distance to the closest boundary)

3. For the **Overlap Boundaries?** option, determine if you need to account for any overlapping boundaries that may be present in your overset mesh (due to overlapping geometry and boundaries or those sometimes generated by collar meshes). You can improve the overset performance by setting this option to **no**.

4. Use the **Enable Grid Priorities** check box to turn on/off the ability to prioritize your overset meshes, or grids. When this option is turned on, a table of overset grids are presented where you can view their **Names** and edit their **Grid Priority** settings directly in the table. By default, all component meshes are given the same priority of 0. You can assign a higher value to increase the prioritization of a particular component mesh. The overset grid priorities are then carried over into the Fluent solver, and can be confirmed using the `define/overset-interface/grid-priorities` command.

5. Use the **Check Overset Interface Intersection** option, enabled by default, to check for any intersections in your overset interfaces while identifying orphans. If you want to skip the check for intersections, then disabling this option increases the speed of identifying orphans.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Indicate the **Number of Orphans**.

7. Once your selection is made, click **Identify Orphans** to visualize orphan cells in the graphics window.

8. You can visualize orphan cells, as well as solve and receptor cells using the **Show Orphan Cells**, **Show Solve Cells**, or **Show Receptor Cells** buttons respectively. These options can also be used in conjunction with the visualization of the clipping plane via the ribbon.

9. If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

4.26. Transforming the Volume Mesh

After you have created a volume mesh, you can add the **Transform Volume Mesh** task to your workflow. This task allows you to create and apply translational or rotational transformations to your volumetric mesh.

1. Provide a **Name**, or use the default name.

2. Select the **Type** of transformation: either **Translational** or **Rotational**. The default name changes upon the selected type (`translate_copy_1` for translational transformations, or `rotate_copy_1` for rotational transformations).

3. Select a mesh object from the list.

   Select items in the list, or use the **Filter Text** option in the drop-down to provide text and/or regular expressions in filtering the list (for example, using *, ?, and []). You can also choose the **Use Wildcard** option in the drop-down to provide wildcard expressions in filtering the list. When you use either ? or * in your expression, the matching list item(s) are automatically selected in the list. Use ^, |, and & in your expression to indicate boolean operations for NOT, OR, and AND, respectively. See Filtering Lists and Using Wildcards (p. 287) for more information.

4. For **Translational** transformations, specify values for the **Translational Shift** in the **X**, **Y**, and **Z** directions.

5. For **Rotational** transformations, specify values for the **Angle** as well as the **Rotation Axis Origin** and **Rotation Axis Direction** in the **X**, **Y**, and **Z** directions.

6. For the **Create a Copy and Translate?** prompt:

   - Choose **yes** if you want to make a copy of the volume mesh and apply the transformation to the copy. If this option is selected, you can make the appropriate selection for the **Number of Copies?** prompt.

   - Choose **no, only translate** to apply the volume mesh transformation directly to the original volume mesh.

7. Click **Preview Transformation** to see a preview of what the volume mesh transformation will look like in the graphics window.

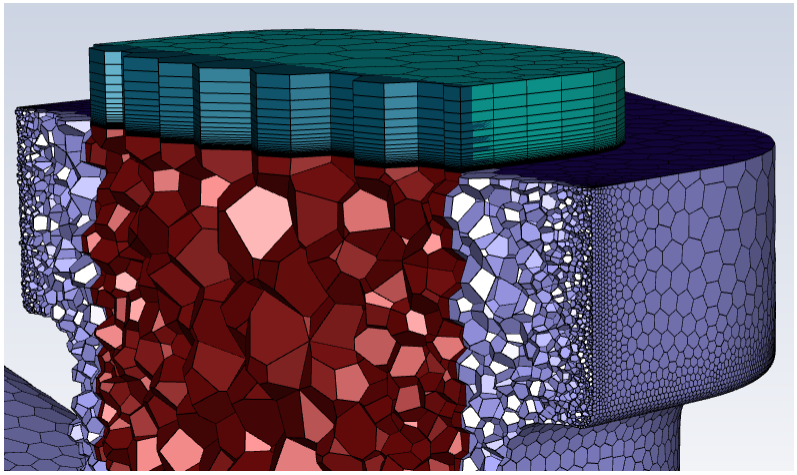
8. Click **Transform Mesh**. The transformed mesh will appear in the graphics window.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

4.27. Extruding the Volume Mesh

After you have created a volume mesh, you can add the **Extrude Volume Mesh** task to your workflow. This task allows you to extend the volume mesh out beyond the original domain using one or more co-planar boundaries.



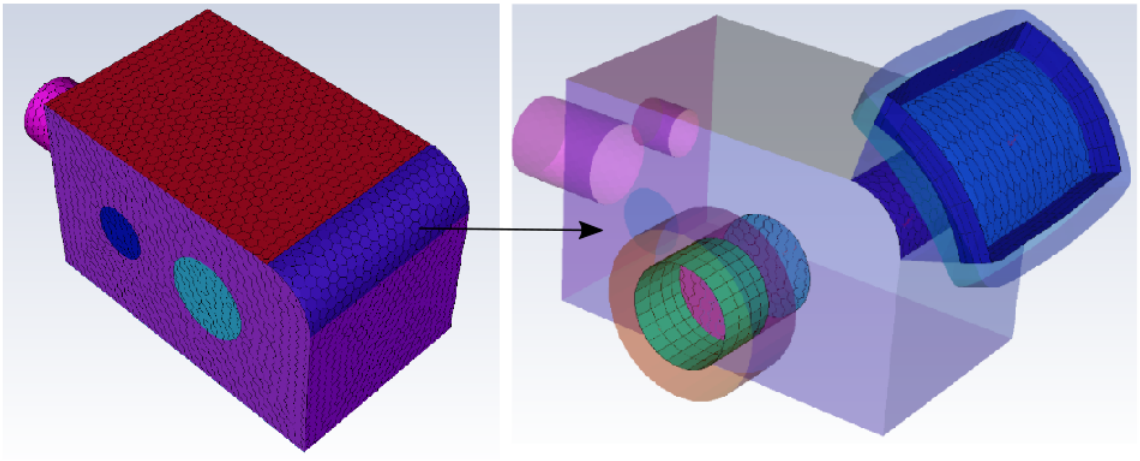
1. Provide a **Name**, or use the default name.
2. Select a boundary face from the list. The extrusion will be performed perpendicularly to the selected boundary, so if you select multiple boundaries, they all must be co-planar. A preview box appears in the graphics window; the size of which is based on the values you provide in the task.

Note:

The preview box is available only for selected faces whose normal is aligned with the Cartesian coordinate.

Extrusions can be applied to both planar and non-planar surfaces. For non-planar, curved, surfaces, the extrusion process is performed normal to the selected surfaces.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

Periodic or shadow face zones cannot be extruded. If you select such zones for extrusion, Fluent provides a warning and automatically assigns such zones to type 'wall' so that you can complete the extrusion task.

Use the **Filter Text** option in the drop-down to provide text and/or regular expressions in filtering the list (for example, using *, ?, and []). You can also choose the **Use Wildcard** option in the drop-down to provide wildcard expressions in filtering the list. When you use either ? or * in your expression, the matching list item(s) are automatically selected in the list. Use ^, |, and & in your expression to indicate boolean operations for NOT, OR, and AND, respectively. See Filtering Lists and Using Wildcards (p. 287) for more information.

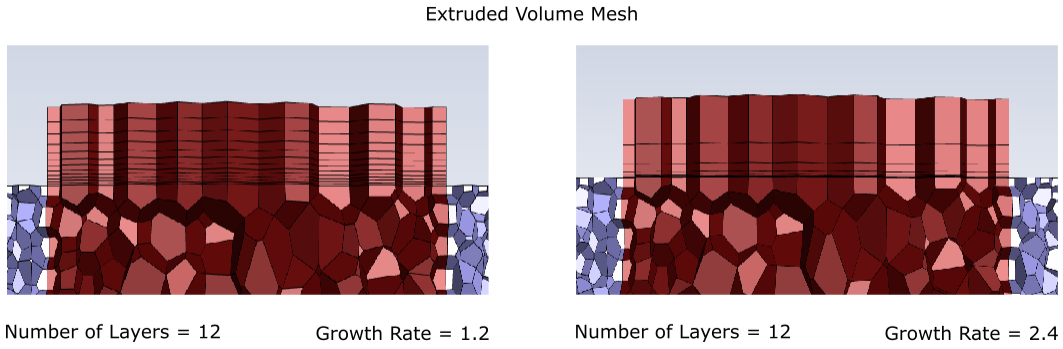
3. Choose an extrusion **Method** based on the specified **Total Height** value (the overall height of the extrusion) or based on the specified **First Height** value (the height of the first layer of the extrusion).

4. Specify whether the volume extrusion is derived from normal-based faceting, or direction-based faceting. When **Normal Based** is enabled (the default), the volume extrusion is based on normal-based faceting, such that for each layer, the normal is calculated and smoothing occurs, and is suitable for non-planar surfaces. For planar surfaces, disable this option to use a direction-based approach where the direction is chosen based on the average normal of the entire surface, and is used to extrude all layers.

5. Specify the **Number of Layers** for the extrusion.

6. Specify the **Growth Rate** for the extrusion. For example, a growth rate of 1.2 will expand each layer of the extrusion by 20 percent of the previous length.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

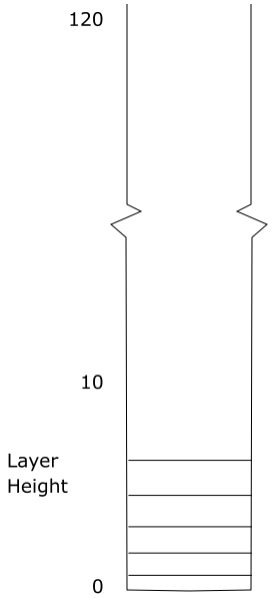


The extrusion process uses these parameter values and geometrically grows the volume mesh according to the following relation:

$$X_N = \sum_{N=2}^{N} x_0 R^{N-1} \tag{4.1}$$

where X_N is the **Total Height** for *N* layers, x_0 is the **First Height**, *R* is the **Growth Rate**, and *N* is the **Number of Layers**.

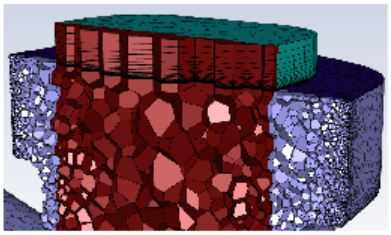
The layer heights for the extrusion are calculated in a geometric fashion, as in:



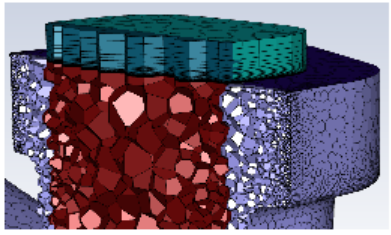
7. Click **Advanced Options** to access additional controls prior to performing this task. Options include:

   - Select a **Growth Pattern**, where the choices provide a visual indication of the volume mesh extrusion pattern.
   - Keep the default selection for the **Merge With Adjacent Region** prompt: as needed.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



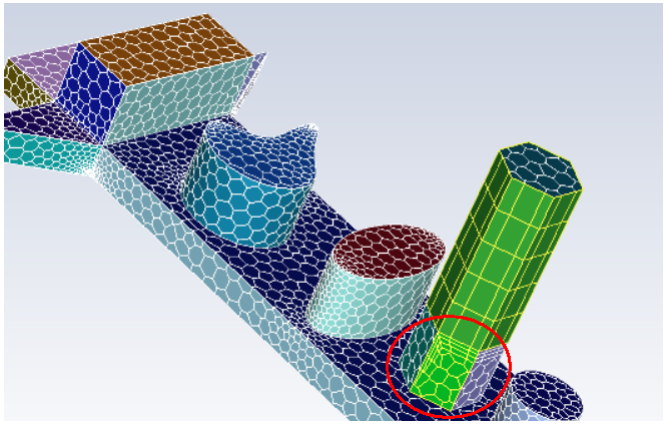
Merge With Adjacent Zones = Yes



Merge With Adjacent Zones = No

Note:

Volume mesh extrusions performed on shapes other than rectangles and squares (such as triangles, pentagons, hexagons, etc.) will automatically merge all side faces irrespective of the original faces, and irrespective of this property setting.



Fluent splits the side adjacent surfaces only by 90 degrees and merges with the adjacent zones accordingly. For other shapes, however, side faces are not split. As a workaround, you can create a custom task to split the side surfaces for specific cases.

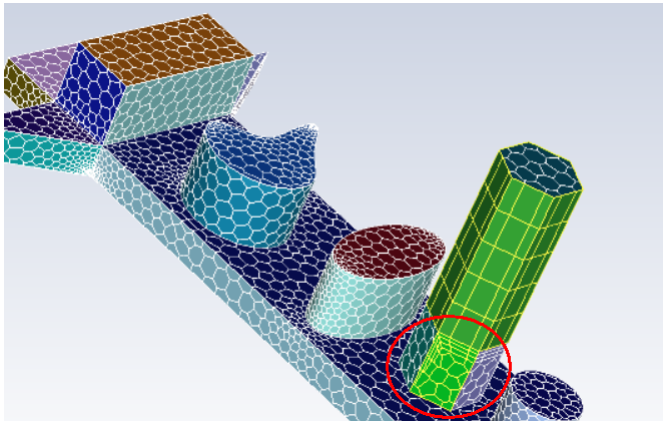
Note:

Extruding cylindrical surfaces too far may create invalid meshes. If your cylindrical surface extrusion exhibits undesirable skewness or otherwise is invalid, reduce the total height of the extrusion and perform the operation again to see if results improve.

Note:

– Volume mesh extrusions performed on shapes other than rectangles and squares (such as triangles, pentagons, hexagons, etc.) will automatically merge all side faces irrespective of the original faces, and irrespective of this property setting.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Fluent splits the side adjacent surfaces only by 90 degrees and merges with the adjacent zones accordingly. For other shapes, however, side faces are not split. As a workaround, you can create a custom task to split the side surfaces for specific cases.

Using the task based display ( ) for this task displays the boundary zones of the mesh along with the adjacent cell zones of the extruded face.

- Extruding cylindrical surfaces too far may create invalid meshes. If your cylindrical surface extrusion exhibits undesirable skewness or otherwise is invalid, reduce the total height of the extrusion and perform the operation again to see if results improve.
- You cannot extrude the volume mesh on zones when the adjacent face zone (in the extrusion direction) is not a structured quadrilateral face zone and it is already connected to cells.

8. Click **Extrude Mesh**. The extruded mesh will appear in the graphics window.

   If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

4.28. Managing Zones

After you have generated your volume mesh, you can use the **Manage Zones** task make additional changes to your cell and/or face zones (such as renaming or merging) prior to proceeding to the Fluent solver. This can especially useful for large models with numerous zones.

Note:

Labels and bodies should **not** have the same name, since doing so would lead to ambiguities. Bodies whose name contains the string 'fluid' will automatically be assigned as fluid regions, however, the same is not true for solids, since that is usually the default name for any body in both SpaceClaim and DesignModeler.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Set the **Type** of zone to **Cell Zone** or **Face Zone**.
2. Apply type filtering for the zones:
   - For cell zones, use the **Type Filter** to filter the list of available objects as **Fluid** or **Solid** or **All**.
   - For face zones,use the **Type Filter** to filter the list of available objects as **Internal**, **Fluid-Fluid**, **Solid-Fluid**, **Fluid-Solid**, **External-Solid**, **External-Fluid**, or **External**.
3. Apply value filtering for the zones:
   - For cell zones, use the **Volume Filter** to only display a list of cell zones within a certain volume.
   - For face zones: use the **Area Filter** to only display a list of face zones within a certain area.
   - You can set either filter to **All**, **Less than**, **More than**, or **Equal to** the corresponding **Area** value (for face zones) or **Volume** (for cell zones). When using the **Equal to** option, for the **Equal within a range of (%)** field, specify a percentage range for the volume/area.
4. Additional filtering options and wildcards are available in the list as well to help in selecting cell zones.
5. To merge selected cell/face zones:
   i. Select **Merge** as the **Operation**.
   ii. Specify the portion of the **Name** of the selected zones that you want to merge, or keep the default name. The task attempts to create an appropriate name. If the target name already exists, then a numerical value will be appended to the name (such as `my_label_1`, `my_label_2`, etc.).
   iii. For **Do you want to merge adjacent faces?**, select either **Yes** or **No**.

   Note:

   The merging of cell zones also allows you to merge the underlying face zone(s). Zones with face labels (named selections), however, are not supported during the face merge.

6. To change the prefix of the names of selected cell/face zones:
   i. Select **Change Prefix** as the **Operation**.
   ii. Use the **From** field to indicate the prefix that you want to change, and use the **To** field to indicate what you would like to change the prefix to. The task attempts to change the prefix accordingly. If the target name already exists, then a numerical value will be appended to the name (such as `my_label_1`, `my_label_2`, etc.).
7. To rename selected cell/face zones:
   i. Select **Rename** as the **Operation**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ii. Specify a new **Name** for the selected zone. The task attempts to create an appropriate name. If the target name already exists, then a numerical value will be appended to the name (such as `my_label_1`, `my_label_2`, etc.).

8. Click **Manage Zone** to apply your changes for this task. Use the **Draw Mesh** button to display the surface mesh to inspect any zone changes.

   If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

4.29. Separating Contacts

Once you have completed the volume mesh, you can add the **Separate Contacts** task to address any issues with contact surfaces and separate any contacts between fluid cell zones (fluid regions).

1. Use the **Separate Contacts** field to enable or disable the ability to separate any existing contacts between surfaces.

2. Once your selections are made, click **Separate Contacts** to update the task.

   Upon completion, the task will generate a Scheme file called `out_solver_f2f_set_nci.scm`. This Scheme file can be loaded into the Fluent solver after mesh scaling (**File > Read > Scheme**), which will automatically set the mesh interfaces.

   If you need to make adjustments to any of your settings in this task, click **Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

4.30. Choosing Part Replacement Options

After you have generated a volume mesh, you can add the **Choose Part Replacement Options** task to your workflow to replace, add, or remove portions of you original imported CAD geometry. This can be useful in studying mesh differences (or flow field changes) due to geometric differences introduced to your CAD model (for example, replacing mirrors, fenders, or spoilers on an automobile). Part replacement can also be combined with introducing translations, rotations, etc for your CAD parts.

You can use this task to avoid re-importing multiple, subtly different CAD geometries and performing the individual workflow tasks on separate CAD models. Using this task in the workflow, once any parts are replaced, you can very easily and quickly update the volume mesh to see the results of your changes.

1. Use the **Do you need to append parts?** prompt to determine whether or not you are going to add any parts to your original geometry. Selecting **Yes** will create a new **Append CAD and Part Management** task that you can use to append new CAD part(s) to the existing CAD model. This new task is a replica of the original task in the workflow where you can import any additional parts that you want to use as replacement parts. See Appending Replacement Parts (p. 495) for details.

2. Use the **Do you need to add local sizing?** prompt to determine whether or not you are going to add local sizing objects to your part replacement task. Selecting **Yes** will create a new **Add Local Sizing for Part Replacement** task. This task is where you can define local size controls for your part replacement regions. See Adding Local Sizing for Replacement Parts (p. 497) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Use the **Do you need to add boundary layers?** prompt to determine whether or not you are going to add additional boundary layers to your part replacement task, for newly appended objects. Selecting **Yes** will create a new **Add Boundary Layers for Part Replacement** task. This task is where you can define local boundary layer and prism controls for your part replacement regions. See Adding Boundary Layers for Replacement Parts (p. 497) for details.

4. Click **Choose Part Replacement Options** to apply your changes for this task before proceeding to the next task.

   If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

Once the task is added, there are additional subtasks that you need to complete, regardless of your selections in the **Choose Part Replacement Options** task:

- **Part Replacement Settings** - a task where you can define specific details surrounding your overall part replacement options. See Applying Part Replacement Settings (p. 495) for details.
- **Update the Volume Mesh** - a task where you can easily regenerate the volume mesh around the specific areas surrounding the replaced parts. See Updating the Volume Mesh (p. 497) for details.

4.30.1. Appending Replacement Parts

This task is a similar to the workflow's initial CAD import and part management task, with many of the same settings, however, it is designed to just accommodate *appending* new, additional CAD parts that you will want to add to the workflow as replacement parts. For any new CAD files you want to use as part replacements, append them to the workflow's original geometry using this part replacement-specific task. See Importing CAD Geometries and Managing CAD Parts (p. 392) (and Appending CAD Files (p. 396)) for more information about the various settings in this task.

In addition using this task to import new CAD parts, you can also use this task to define special transformations to apply to your part replacements if necessary, such as rotations and translations. See Performing Operations on Meshing Model Objects (p. 410) for more information about object transformations.

Note:

This task is not available if you have any objects that contain solids.

4.30.2. Applying Part Replacement Settings

Use this task to define a particular part replacement strategy: whether you are adding, removing, or replacing a part; and identifying the part(s) you are using as a replacement and the part(s) you are going to replace.

1. Provide a **Name**, or use the default name.

2. Choose an **Operation** to perform for this part replacement task. You can choose from **Replacement**, **Addition**, or **Removal**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Choose a particular **Method** of replacement. You can choose from **Surface Mesh Based** or **Volume Mesh Based**.

   - The volume mesh based approach defines a separate region for the area of interest surrounding the part replacement. Volume meshing is performed only in this region and thus is much faster than regenerating the entire volume mesh.

     Note:

     To use the **Volume Mesh Based** approach, you must create construction surfaces to surround the area of the part to be replaced. See Identifying Construction Surfaces (p. 436) for more information. This method will be selected by default if Ansys Fluent detects any existing construction surfaces in the workflow.

   - While the surface mesh approach does not require construction surfaces, the approach does require you to remesh all of your volume regions, even though the surface mesh is updated only for the replacement part(s). This approach can be useful for smaller, more subtle part replacements while still maintaining performance.

4. For part replacement or removal, use the **Select a Source Object** list to pick the original object(s) that you wish to replace or remove. By default, you can use the **Filter Text** drop-down to provide text and/or regular expressions in filtering the list (for example, using *, ?, and []). See Filtering Lists and Using Wildcards in the *Fluent User's Guide* (p. 287) for more information.

   For part replacement or addition, use the **Select a Target Object** list to pick the new object(s) that you wish to replace or add. By default, you can use the **Filter Text** drop-down to provide text and/or regular expressions in filtering the list (for example, using *, ?, and []). See Filtering Lists and Using Wildcards in the *Fluent User's Guide* (p. 287) for more information.

   Note:

   *Part replacement does not support regions with boundary layers defined on **selected-zones** or **selected-labels**; only boundary layers defined on **all-zones** or **only-walls** are supported.*

5. Under **Advanced Options**, you have the following settings:

6. Specify a value for the **Scaling Factor for New Objects** option. This will determine the size of the bounding box surrounding the selected object where the part replacement is to take place.

7. For **Define Location Using** field, choose a method to specify the location of the region:

   - Select **Numerical Inputs** to assign specific coordinates to a particular region. You can use the **View Exact Coordinates** field to see and edit the specific **X**, **Y**, and **Z** coordinate values for the region's identified location.

     Note:

     This technique generally uses the selected object's centroid to define the initial location point and may require further refinement. More accurate location re-

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

finement is achieved more specifically using the x, y, and z inputs and occasionally previewing the location.

- Select **Region Around Source Objects** to calculate the position of the region based on the selected object(s).

8. Click **Part Replacement** to update the task with your selections.

   If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

4.30.3. Adding Local Sizing for Replacement Parts

Use this task to add local sizing to your part replacements. This task is similar to the existing **Add Local Sizing** task. See Adding Local Size Controls (p. 459) for more information.

When processing the **Choose Part Replacement Options** task, the workflow will notify you if there is no existing **Add Local Sizing** task, and if so, the workflow checks the settings in the **Add Local Sizing for Part Replacement** task and compares them to the existing **Add Local Sizing** task.

4.30.4. Adding Boundary Layers for Replacement Parts

Use this task to add local boundary layer and prism settings to your part replacements. This task is similar to the existing **Add Boundary Layers** task. See Adding Boundary Layers (p. 476) for more information.

When working in the **Add Boundary Layers for Part Replacement** task, the workflow checks the settings in this task and compares them to the existing **Add Boundary Layers** task. For instance, any zones previously defined in the **Add Boundary Layers** task are compared to those specified in this task.

Note:

If you have already defined your fluid region's boundary layer along **all-walls** or **all-zones** in the existing **Add Boundary Layers** task, then since the new part is associated with the same fluid region, the new part will also have the same boundary layer settings associated with it.

4.30.5. Updating the Volume Mesh

When you have completed your part replacement settings, you can proceed to updating the volume mesh.

1. Select **Enable Parallel Meshing for Fluids** if you would like to perform parallel volume meshing on your fluid region(s) for poly, hexcore and poly-hexcore volume fill types. When parallel is enabled, both boundary layer generation methods are parallelized: continuous and stair-step (also called "scoped").

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Click the **Generate the Volume Mesh** button to update the volume mesh.

Note:

When the volume is filled for part replacement, the mesh is distributed for parallel meshing, then agglomerated and migrated to a single active node. Therefore, parallel checks will only show one active node even when parallel meshing is used successfully.

If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 5: Using the 2D Meshing Guided Workflow

This chapter describes the various tasks that are provided when using the **Watertight Geometry** meshing guided workflow:

5.1. 2D Meshing Workflow

5.1. 2D Meshing Workflow

The **2D Meshing** workflow is available from the list of meshing workflows for perform common meshing operations on 2D geometries.

Note:

When edges are drawn as part of the updating a task within the 2D workflow, they are colored based on their connectivity according to the following rules:

- Edges adjacent to one face: red
- Edges having the same face on both sides: cyan
- Edges having two different faces on both sides: black
- Edges connected to three faces: magenta
- Edges connected to four faces: yellow

5.1.1. Loading the CAD Geometry
5.1.2. Updating Regions
5.1.3. Updating Boundaries
5.1.4. Defining Global Sizing
5.1.5. Adding Local Sizing
5.1.6. Adding 2D Boundary Layers
5.1.7. Generating the Surface Mesh
5.1.8. Exporting the Fluent 2D Mesh

5.1.1. Loading the CAD Geometry

Use the **Load CAD Geometry** task to designate a geometry or surface mesh for your simulation.

1. Browse for a specific **File Name**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Supported file types are SpaceClaim (`.scdoc`), Discovery (`.dsco`), and Workbench (`.agdb`) files and also `.fmd` and `.pmdb` files.

Note:

When a SpaceClaim (`.scdoc`) file is imported into Fluent while in meshing mode, Fluent also creates an intermediary `.pmdb` file that can be imported. The `.pmdb` file should always reside alongside the `.scdoc` file. When changes are made to the geometry in SpaceClaim, and the file is reimported into Fluent, the original `.pmdb` file is overwritten. The `.pmdb` file can be more easily and quickly read into Fluent for additional processing and should be used as long as the `.scdoc` file has not changed.

Note:

SpaceClaim (`.scdoc`) files are only supported on Windows. When working on Linux systems, however, you can use the intermediary `.pmdb` file as your geometry file for the workflows.

On Windows, use the **Import CAD Geometry** dialog to import the CAD file into Fluent, and enable the **Save PMDB (Intermediary File)** option in the **Import Options** dialog. After the file is imported, you can move the generated `.pmdb` file over to your Linux system to use in your workflow.

2. Choose a suitable option for the **Units**. You should work in units where the minimum size of the mesh is of the order of one. The mesh will automatically be scaled to meters while transferring the mesh to the Fluent solver

3. Select an appropriate **Import Route** option, or keep the default value, depending on your requirements and platform. In most cases, the default value is recommended and should lead to the desired data being imported, however, you should be aware of and verify the CAD configurations being made because they may be considered during import. See CAD Integration for details.

   The workflow uses an `.fmd` file to extract and persist CAD information for this task. To generate an appropriate `.fmd` file, the available options for the **Import Route** depend on the selected **CAD File**, and your particular platform (Windows or Linux). The various **Import Route** options include:

   - The **Native** option loads the CAD file natively into FM, and internally generates an FMD file that is then loaded into the workflow. This option is available for `.fmd`, `.fmdb`, and `.stl` file formats.

     Note:

     Using this import route, the CAD data should not differ across Windows and Linux for default settings.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The **SCDM** option uses ANSYS SpaceClaim Direct Modeler to read the CAD file and internally generates an `.fmd` file that is then loaded into the workflow. You may also want to note any SpaceClaim file options that you may have configured.

  Note:

  This route is only available on Windows, and is not available on Linux.

- The **DSCO** option, available only when Discovery is installed, allows you to import Discovery (`.dsco`) files.
- The **Workbench** option uses Ansys Workbench CAD readers or plug-ins to attach or load the given CAD file and generates an `.fmd` file that is then loaded into the workflow.

  Note:

  This route is available on Windows and Linux., though not all formats are supported on Linux when compared to Windows. Also, third-party add-on modules can lead to slightly different CAD data upon import (for example, names, assembly hierarchy, etc.).

4. Use the **Refacet** check box to enable or disable additional faceting refinement in this task. When enabled, you can change the faceting of the CAD geometry. Only the faceted CAD model is used during the meshing process. In either case, when disabled, normal CAD faceting is performed as defined by the imported CAD file(s).
   - The **Deviation** controls how far facet edges are away from the model.
   - The **Normal Angle** also controls how far facet edges are away from the model. Decreasing the normal angle will result in more facets along the curved edges.
   - The **Max Size** controls the maximum size of the facets after the deviation and the normal angle are respected.
5. Once all selections have been made, click **Load CAD Geometry**.

   If you need to make adjustments to any of your settings in this task, click **Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.
6. Proceed to the next step in the workflow.

5.1.2. Updating Regions

You can update the properties of any defined region using the **Update Regions** task. This task can be added to the workflow as many times as you require.

The table contains a list of all of the defined regions, and their assigned types.

1. (optional) Use the **Filter** button to filter the table contents based on a particular column.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Assign a **Region Type** as needed using the corresponding drop-down menu. Available region types include:

   - fluid
   - solid
   - dead

   Multiple regions can be assigned a specific type all at once by selecting them in the table, right-click, and select **Set Region Type** in the context menu, then designate a type for the selected regions directly in the menu.

3. Use the **Draw Regions** button to display the available regions in the graphics window.

   Multiple regions can be visualized all at once by selecting them in the table, right-click, and select **Draw Selections** in the context menu.

4. When you are satisfied with the region assignments, click **Update Regions**.

   If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

5. Once all regions have been updated, proceed to the next step in the workflow.

5.1.3. Updating Boundaries

You can update the properties of any defined boundary using the **Update Boundaries** task. This task can be added to the workflow as many times as you require.

1. Choose a **Selection Type** as either by **label** or by **zone**.

2. (optional) Use the **Filter** button to filter the table contents based on a particular column.

3. (optional) Rename any **Boundary Name** by double-clicking the label in the table and entering a new name.

4. (optional) Re-assign any **Boundary Type** to another value by selecting a type in the table and using the corresponding drop-down menu. Choices include:

   - velocity-inlet
   - pressure-outlet
   - pressure-inlet
   - pressure-far-field
   - mass-flow-inlet
   - mass-flow-outlet
   - outflow

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- symmetry
- wall
- internal
- interface
- overset
- outlet-vent
- intake-fan
- inlet-vent
- exhaust-fan
- fan
- porous-jump
- radiator
- axis

Multiple boundaries can be assigned a specific type all at once by selecting them in the table, right-click, and select **Set Boundary Type** in the context menu, then designate a type for the selected boundaries directly in the menu.

5. When you are satisfied with the boundary assignments, click **Update Boundaries**.

   If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

6. Multiple boundaries can be visualized all at once by selecting them in the table, right-click, and select **Draw Selections** in the context menu.

   Use the **Draw Boundaries** button to visualize all boundaries or just wall boundaries.

7. Once all boundaries have been updated, proceed to the next step in the workflow.

5.1.4. Defining Global Sizing

Use this task to specify various global mesh sizing features for the imported geometry.

1. Specify a value for the **Minimum Size** for the global size control.
2. Specify a value for the **Maximum Size** for the global size control.
3. Specify a value for the **Growth Rate** for the global size control.
4. Choose a **Size Function** for the new global size control.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The **Curvature** size function can be used for refining the surface mesh based on the underlying curve and surface curvature.

  For this option, you can also specify the **Curvature Normal Angle** for the curvature size function. The default value of 18 degrees should approximately produce 20 facets in the circumferential direction of a cylinder.

- The **Proximity** size function can be used for creating the surface mesh, based on the number of cells per gap specified.

  For this option, you can also specify the **Cells Per Gap** for the proximity size function. This value is the number of element layers to be generated in a gap for the edge proximity size function. Note that for proximity size functions, the number of cells per gap can be a real value, with a minimum value of 0.01. See Proximity (p. 587) for more information.

- By default, a **Curvature and Proximity** size function is assigned based on both curvature and proximity.

  For this option, you can also specify the **Curvature Normal Angle** for the curvature size function. The default value of 18 degrees should approximately produce 20 facets in the circumferential direction of a cylinder.

  In addition, you can also specify the **Cells Per Gap** for the proximity size function. This value is the number of element layers to be generated in a gap for the edge proximity size function. Note that for proximity size functions, the number of cells per gap can be a real value, with a minimum value of 0.01. See Proximity (p. 587) for more information.

Note:

If local sizing has been added, these curvature and proximity size controls are appended to the local sizes and the resulting size field is used to dictate the sizes during surface meshing.

For additional information, see Size Functions and Scoped Sizing (p. 584).

5. Once your selections are made, click **Define Global Sizing** and proceed onto the next task.

   If you need to make adjustments to any of your settings in this task, click **Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

6. Once you are satisfied with your changes, proceed to the next step in the workflow.

5.1.5. Adding Local Sizing

You can gain better control over the mesh size distribution by using the **Add Local Sizing** task. Using this task, you can define specific mesh size controls that operate on specific, localized, portions of the geometry and mesh. Using this task, you can add as many localized size controls to the workflow as you need, depending on the requirements and details of your geometry. Note that this task can only be added to the workflow prior to the **Generate Initial Surface Mesh** task.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For the **Would you like to add local sizing?** field, select **yes** if you need to define local sizing parameters using the following steps. Otherwise, if you do not need to define local sizing controls, keep the default of **no**, click **Update** and proceed to the next task.

1. Provide a **Name** for the new size control, or use the default name. The default name changes depending on the assigned **Size Control Type**.

2. Provide a **Growth Rate**.

3. Choose the **Size Control Type**. Choices include:

   - Use the **Edge Size** setting for refining the local sizing based on the edge size. This option is only available if the imported geometry contains one or more *named* edges (for example, an ANSYS SpaceClaim Design Modeler CAD geometry with one or more edges explicitly using named selections). For this setting, you can also provide a **Target Mesh Size**.

   - Use the **Face Size** setting for refining the local sizing based on the face size. For this setting, you can also provide a **Target Mesh Size**.

   - Use the **Body of Influence** setting, or *BOI*, to assign a maximum size on all parts of your geometry that falls within the boundaries of the body of influence. For this setting, you can also provide a **Target Mesh Size**.

   - Use the **Curvature** setting for refining the local sizing based on the underlying curve and surface curvature. This size control type is useful, for instance, in models with a combination of large and small scales. For this option, you have the following additional settings:

     – Specify a value for the **Local Min Size** for the curvature size control.

     – Specify a value for the **Max Size** for the curvature size control.

     – Specify a value for the **Curvature Normal Angle** for the curvature.

     – Specify a value for the **Scope to** field, where you can localize the size control to faces, edges, faces and edges, or edge labels. Edge selection selects the edges of a particular face.

   - Use the **Proximity** setting for refining the local sizing based on the number of cells per gap specified. For this option, you have the following additional settings:

     – Specify a value for the **Local Min Size** for the proximity size control.

     – Specify a value for the **Max Size** for the proximity size control.

     – Specify a value for the **Cells Per Gap**. This value is the number of element layers to be generated in a gap for the edge proximity size function. Note that for proximity size functions, the number of cells per gap can be a real value, with a minimum value of 0.01.

     – Specify a value for the **Scope to** field, where you can localize the size control to faces, edges, faces and edges, or edge labels. Edge selection selects the edges of a particular face.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. You can select an available zone or label to apply your local sizing changes. Choose whether to **Select By** the zone name, or the label name in the list below.

5. Select the **Draw Size Boxes** field to visualize the size control's minimum and maximum sizes in the graphics window (in the form of red cubes).

6. Once your selections are made, click **Add Local Sizing** and proceed onto the next task.

   You can add as many local sizing controls as you require for your workflow, each operating on different zones and/or with different sizing parameters. The size controls will appear as sub-tasks under the parent task.

   If you need to make adjustments to any of your settings in this task, select the specific size control sub-task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

   Note:

   If you need to add another local sizing control after you have already created a surface mesh, you need to revert and edit at least one existing local size control in order to properly see and use the available geometry objects in the graphics window and the object listing. Alternatively, you can also re-import your geometry and update the **Add Local Sizing** task.

7. Once you are satisfied with your changes, proceed to the next step in the workflow.

5.1.6. Adding 2D Boundary Layers

You can define 2D shell boundary layers along various regions using the **Add 2D Boundary Layers** task. This task can be added to the workflow as many times as you require.

Use this task to create a layered mesh on the surface mesh from certain edges. This, together with boundary layer mesh, can create an anisotropic mesh.

For the **Add 2D Boundary Layers?** field:

- Select **yes** if you need to specify boundary layers using this task.
- Select **no** if you do not need to account for boundary layers. Click **Update** and proceed to the next task.

1. Specify a **Name** for the shell boundary layer, or use the default value. Note that the default name is dependent on the value of the **Offset Method Type**.

2. Choose an **Offset Method Type**. The offset method that you choose determines how the mesh cells closest to the boundary are generated. See Offset Distances (p. 765) for more information. Choices include:

   - **aspect-ratio**: allows you to control the aspect ratio of the boundary layer cells (or prism cells) that are extruded from the base boundary zone. The aspect ratio is defined as the ratio of the prism base length to the prism layer height.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In this case, you can also specify the **First Aspect Ratio**. You can control the heights of the inflation layers by defining the aspect ratio of the inflations that are extruded from the inflation base. The aspect ratio is defined as the ratio of the local inflation base size to the inflation layer height. The value for the **First Aspect Ratio** allows you to specify the first aspect ratio to be used.

- **uniform**: allows you to generate every new node (child) to be initially the same distance away from its parent node (that is, the corresponding node on the previous layer, from which the direction vector is pointing).

  In this case, you can also specify the **First Height** for the height of the first layer of cells in the boundary layer.

- **smooth-transition**: allows you to use the local tetrahedral element size to compute each local initial height and total height so that the rate of volume change is smooth. Each triangle that is being inflated will have an initial height that is computed with respect to its area, averaged at the nodes. This means that for a uniform mesh, the initial heights will be roughly the same, while for a varying mesh, the initial heights will vary

3. Specify the **Number of Layers**. This value determines the maximum number of boundary layers to be created in the mesh.

4. Specify the **Growth Rate** for the boundary layer. This value determines the relative thickness of adjacent inflation layers. As you move away from the face to which the inflation control is applied, each successive layer is approximately one growth rate factor thicker than the previous one. For example, a growth rate of 1.2 will expand each layer of the extrusion by 20 percent of the previous length.

5. For the **Edge Selection Type** field, specify whether to select edges by **label** or **zone**. Select items in the list, or use the **Filter Text** option in the drop-down to provide text and/or regular expressions in filtering the list (for example, using *, ?, and []). You can also choose the **Use Wildcard** option in the drop-down to provide wildcard expressions in filtering the list. When you use either ? or * in your expression, the matching list item(s) are automatically selected in the list. Use ^, |, and & in your expression to indicate boolean operations for NOT, OR, and AND, respectively.

6. For the **Grow on** field, specify where you would like to develop the boundary layers.

   - Use the **all adjacent faces** option to grow the boundary layer along all adjacent surfaces.

   - Use the **selected-zones** option to select from the **Zones** list of available named zone(s) in your geometry.

   - Use the **selected-labels** option to select from the **Labels** list of available named label(s) in your geometry.

7. Click **Advanced Options** to access additional controls prior to performing this task.

   - Use the **Gap Factor** option to specify the relative gap between two boundary layer caps in a narrow channel. A value of 1 indicates a gap that is of the same order as the boundary layer cap triangle size in the inflation layer.

   - Use the **Max Aspect Ratio** option to specify the maximum aspect ratio for the boundary layer when proximity compression is applied.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Use the **Min Aspect Ratio** option to specify the minimum aspect ratio for the boundary layer.
- Use the **Local Remesh?** prompt to specify whether or not you want to apply remeshing to the adjacent triangles after the shell boundary layer.
- Use the **Remesh Growth Rate** field to specify the growth rate for the local remeshing.
- Use the **Refine Stretched Quads?** prompt to specify whether or not you want to refine any stretched quadrilateral cells.
- Use the **Max Projection Angle** to control the projection of sides on adjacent edges. If the angle is above this threshold, it does not imprint.
- Use the **Max Face Skewness** field to specify the target skewness that you want to achieve.

8. Click **Add 2D Boundary Layers** to generate the appropriate boundary layers for the imported CAD geometry.

   If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

5.1.7. Generating the Surface Mesh

Use the **Generate the Surface Mesh** task to create a conformal, connected surface on all of the objects of an imported geometry, or remesh all or parts of an imported surface mesh. Use this task to identify regions that will later be filled with the volume mesh. In many cases, the default values will be sufficient for a useful CFD surface mesh.

1. Use the **Generate Quads?** option to determine whether or not to generate quadrilateral cells in the initial surface mesh.

   Note:

   After enabling the **Generate Quads?** option, the **Split Quads?** option will no longer be available.

2. Use the **Project on Geometry** check box to determine whether, after surface meshing, Fluent will project the mesh nodes back onto to the original CAD model. This option is not available when multi-threading.
3. Use the **Enable Multi Threading** option to determine whether or not to use more than one thread in generating the initial surface mesh (enabled by default). If enabled, you can set the **Number of Multi Threads**, or use the default value of 4. This option is not available when projecting onto the geometry.
4. Click **Generate Initial Surface Mesh** to generate a CFD surface mesh for the imported CAD geometry.

   If you need to make adjustments to any of your settings in this task, click **Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

5. Proceed to the next step in the workflow.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.1.8. Exporting the Fluent 2D Mesh

Since this 2D meshing workflow exists in the normal (3D) Fluent meshing environment, it is not possible to directly switch to the 2D Fluent solver. Instead, you can create a special 2D mesh file that can only be loaded into the 2D Fluent solver.

When you have completed setting up the 2D meshing workflow details, specify the **File Name** (or use the default name that is based on the imported CAD file) and click the **Write 2D Mesh** button to create the 2D mesh file.

Note:

You cannot read this 2D mesh back into the 2D meshing workflow.

From here, you can go on to add more tasks to the workflow, if required, and/or proceed to performing any mesh checking or diagnostics.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 6: Improving and Examining the Mesh and its Quality

Once you have generated a mesh, you may need to examine its features to determine if there are areas for improvement and improve its quality. This chapter describes the various ways of performing these operations.

6.1. Improving the Mesh

6.2. Examining the Mesh

6.3. Determining Mesh Statistics and Quality

6.1. Improving the Mesh

A volume mesh created from a high-quality surface mesh may contain some high-skewness cells. The poor cells may result from unsuitable mesh size distribution over the domain or, more often, are caused by constraints imposed by the boundaries.

After creating a tetrahedral or hybrid mesh, you can improve the quality of the mesh by smoothing nodes and swapping faces. Smoothing and face swapping are tools that help to improve the quality of the final numerical mesh. You can also use the improve command which combines operations like collapsing cells, node smoothing, face swapping, and inserting nodes. This chapter describes the options available for improving the mesh quality by removing highly skewed cells.

6.1.1. Smoothing Nodes

6.1.2. Swapping

6.1.3. Improving the Mesh

6.1.4. Removing Slivers from a Tetrahedral Mesh

6.1.5. Modifying Cells

6.1.6. Moving Nodes

6.1.7. Cavity Remeshing

6.1.8. Manipulating Cell Zones

6.1.9. Separating Cell Zones

6.1.10. Manipulating Cell Zone Conditions

6.1.11. Using Domains to Group and Mesh Boundary Faces

6.1.12. Checking the Mesh

6.1.13. Selectively Checking the Volume Mesh

6.1.14. Checking the Mesh Quality

6.1.15. Clearing the Mesh

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.1.1. Smoothing Nodes

Smoothing repositions the nodes to improve the mesh quality. The smoothing methods available are:

6.1.1.1. Laplace Smoothing
6.1.1.2. Variational Smoothing of Tetrahedral Meshes
6.1.1.3. Skewness-Based Smoothing of Tetrahedral Meshes

6.1.1.1. Laplace Smoothing

Laplace smoothing is used to improve (reduce) the average skewness of the mesh. In this method, a Laplacian smoothing operator is applied to the unstructured grid to reposition nodes. The new node position is the average of the positions of its node neighbors.

The relaxation factor (a number between 0.0 and 1.0) multiplies the computed node position increment. A value of zero results in no movement of the node and a value of unity results in movement equivalent to the entire computed increment.

This node repositioning strategy improves the skewness of the mesh, but usually relaxes the clustering of node points. In extreme circumstances, the unchecked operator may create grid lines that cross over the boundary, creating negative cell volumes. To prevent such crossovers, the skewness of the resulting cells is checked before the node is repositioned. This makes the smoothing operation time-consuming.

The smoothing operator can also be applied repeatedly, but as the number of smoothing iterations increases, the node points have a tendency to pull away from boundaries and the mesh tends to lose any clustering characteristics.

6.1.1.2. Variational Smoothing of Tetrahedral Meshes

Variational smoothing is available only for tetrahedral meshes. It can be considered as a variant of Laplace smoothing. The new node position is computed as a weighted average of the circumcenters of the cells containing the node. The variational smoothing method is provided as a complement to Laplace smoothing.

6.1.1.3. Skewness-Based Smoothing of Tetrahedral Meshes

Skewness-based smoothing is available only for tetrahedral meshes. When you use skewness-based smoothing, a smoothing operator is applied to the mesh, repositioning interior nodes to lower the maximum skewness of the mesh. Interior nodes will be moved to improve the skewness of cells with skewness greater than the specified minimum skewness. You can also specify an appropriate value for the minimum improvement, if required. This allows you to stop performing the smoothing iterations when the maximum change in cell skewness is less than or equal to the value specified for minimum improvement.

The maximum change in cell skewness will be compared with the specified value for minimum improvement. When the maximum change in cell skewness is less than or equal to the value specified for minimum improvement, further smoothing iterations will no longer yield appreciable improvement in the mesh. The smoothing will be stopped at this point even if the requested number of iterations has not been completed.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This skewness-based smoothing process can be very time-consuming, so it is advisable to perform smoothing only on cells with high skewness. Improved results can be obtained by smoothing the nodes several times. There are internal checks that will prevent a node from being moved if moving it causes the maximum skewness to increase, but it is common for the skewness of some cells to increase when a cell with a higher skewness is being improved. Hence, you may see the average skewness increase while the maximum skewness is decreasing.

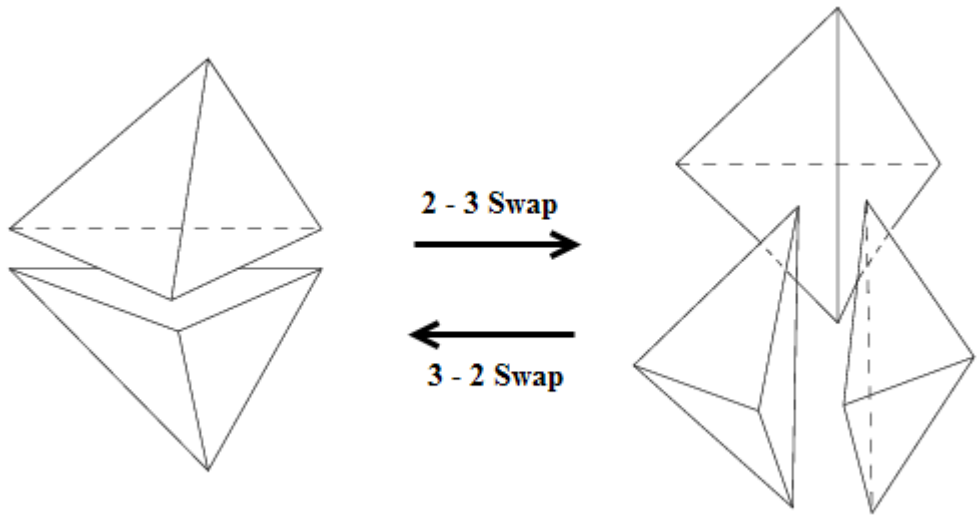
You should consider whether the improvements to the mesh due to a decrease in the maximum skewness are worth the potential increase in the average skewness. Performing smoothing only on cells with very high skewness (for example, 0.8 or 0.9) may decrease the adverse effects on the average skewness.

6.1.2. Swapping

For tetrahedral meshes, swapping involves searching for a specific configuration of cells and replacing it by an alternative configuration. The default option is a 3–2 swap configuration where three tetrahedra are replaced by two tetrahedra after swapping. The other possible combinations are the 2–3 swap configuration (replacing two tetrahedra by three) and the 4–4 swap configuration (replacing four existing tetrahedra with four alternate tetrahedra).

Figure 6.1: 2–3 and 3–2 Swap Configurations (p. 513) shows the 2–3 and 3–2 swap configurations. The two tetrahedra (on the left) having a common interior face can be replaced by three tetrahedra having a common interior edge. The common face will be replaced by three interior faces and an interior edge during swapping. Conversely, the three tetrahedra (on the right) have two interior faces each and share a common interior edge. During swapping for a 3–2 configuration, three interior faces and the common interior edge will be replaced by a single face. This results in two tetrahedra having a common interior face.

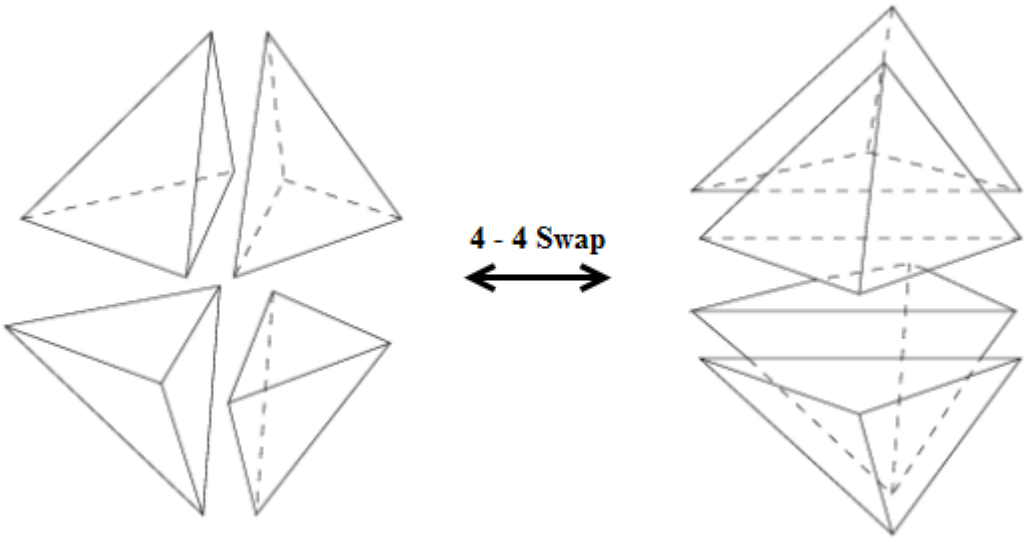
Figure 6.1: 2–3 and 3–2 Swap Configurations



Another possible swap configuration is the 4–4 swap configuration where four tetrahedra will be replaced by four alternate tetrahedra. In Figure 6.2: 4–4 Swap Configuration (p. 514), either the common interior edge or two common faces can be replaced, resulting in four alternate tetrahedra.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.2: 4–4 Swap Configuration



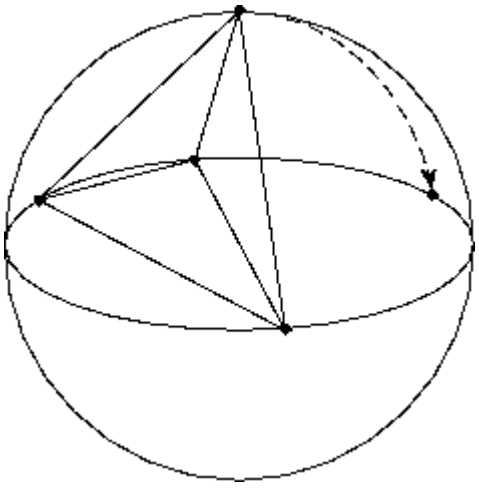
6.1.3. Improving the Mesh

The **Improve** operation is an automated procedure for sliver removal or for reducing the maximum skewness in the mesh. The improvement is carried out by removing cells above the specified skewness threshold by collapsing cells, swapping faces, smoothing nodes, and inserting new nodes iteratively. Each operation is specialized, and the mesh will be modified only if the mesh is noticeably improved. The skewness before and after an operation is taken into account to determine the improvement. Hence, the lower the skewness of the cells involved, the larger the improvement will have to be in order for the mesh to be modified. The improve operation is a more elaborate version of the **Remove Slivers** option invoked from the **Refinement** tab in the **Tet** dialog box.

6.1.4. Removing Slivers from a Tetrahedral Mesh

A sliver typically denotes a flat tetrahedral cell. Figure 6.3: Sliver Formation (p. 514) shows an acceptable tetrahedron.

Figure 6.3: Sliver Formation



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the top node of the tetrahedron were to travel along the path of the dotted line in the direction of the arrow, as it approached the end of the line the resulting cell would be a degenerate tetrahedron, or a sliver.

In the following sections, the term sliver is used to denote all types of poorly shaped cells. There are several commands for removing slivers or to reduce the maximum skewness of the mesh.

6.1.4.1. Automatic Sliver Removal

6.1.4.2. Removing Slivers Manually

6.1.4.1. Automatic Sliver Removal

Sliver removal operations can be invoked during the tetrahedral mesh refinement process. The **Remove Slivers** option in the **Refinement** tab of the **Tet** dialog box controls the removal of slivers during the meshing process. The **Remove Slivers** option includes operations such as collapsing cells (to remove nodes), face swapping, smoothing, and point insertion, which are invoked iteratively.

Usually the sliver cells will be removed during refinement, but occasionally a few might be left behind. In such cases, you can use the options in the **Slivers** tab in the **Tet Improve** dialog box to remove the slivers manually. The **Improve** command uses an automated procedure to remove slivers or to improve the mesh quality in general.

6.1.4.2. Removing Slivers Manually

The **Tet Improve** dialog box contains options for removing slivers manually. The operations available for sliver removal are as follows:

Smoothing Boundary Slivers

This operation involves smoothing nodes on sliver cells having at least one node on the boundary. During smoothing, the nodes will be repositioned so long as the skewness of the surrounding cells is improved. The nodes on features will also be smoothed, but will not be projected on to the original geometry. However, nodes at branch points (more than two feature edges at the node) and end points (one feature edge at the node) will be fixed. The nodes will be smoothed until the skewness value is less than the specified value. The default values for the skewness threshold, minimum dihedral angle between boundary faces, and feature angle are `0.985`, `10`, and `30`, respectively.

Smoothing Interior Slivers

This operation involves smoothing non-boundary nodes on sliver cells having skewness greater than the specified threshold value. The default value for the skewness threshold is `0.985`.

Swapping Boundary Slivers

A flat boundary cell containing two boundary faces can be removed by moving the boundary to exclude the cell from the zone in which it is located, effecting a minor change in the geometry. However, if there is another live zone on the other side of the boundary, this operation will result in the cell being moved to the other zone. In such cases (for example, conjugate heat transfer problems), you can decide which live zone is least critical, and then move the boundary sliver to that zone.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The default values for the skewness threshold and the minimum dihedral angle between faces are `0.95` and `10`, respectively.

Refining Boundary Slivers

This operation attempts to increase the volume of boundary slivers to create a valid tetrahedral cell. Tetrahedra having one or two faces on the boundary are identified and then the edge opposite the boundary faces are split. The edge opposite the face pair with the largest dihedral angle will be split for a tetrahedron with one boundary face, while the edge opposite the boundary faces will be split for a tetrahedron having two boundary faces. The split node is then smoothed such that the volume of the tetrahedron increases, thereby creating a valid tetrahedral cell.

Refining Interior Slivers

This operation attempts to remove the sliver by placing a node at or near the centroid of the sliver cell. Swapping and smoothing are then performed to improve the skewness.

Collapsing Slivers

This operation attempts to collapse the edge of a skewed sliver cell on any one of its neighbors. The default skewness threshold is `0.985`.

Note:

- If you are not using a two-sided wall condition for the boundary, you can slit the face zone containing the sliver (using the `/boundary/slit-boundary-face` command) and then perform the sliver-removal operation.
- Multiple slivers may exist on top of each other thus, requiring multiple operations to remove them all.

6.1.5. Modifying Cells

Additional tools are available for you to perform primitive operations on the cells such as smoothing nodes, swapping cells, splitting cells, and so on. This section describes the generic procedure for modifying the cells using the **Modify Cells** dialog box. You will also use the **Display Grid** dialog box during the modification of the cells.

6.1.5.1. Using the Modify Cells Dialog Box

6.1.5.1. Using the Modify Cells Dialog Box

1. Display the cells or cell zones to be modified using the options in the **Display Grid** dialog box.
2. Select the type of entity (**cell**, **face**, **node**, and so on) you want in the **Filter** list in the **Modify Cells** dialog box.
3. Select the entity to be modified in the graphics window.
4. Click the appropriate button in the **Operation** group box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The display will be automatically updated to reflect the change made by the operation.

5. Repeat the procedure to perform different operations on the cells.

> **Warning:**
>
> Save the mesh periodically as not all operations are reversible.

The options in the **Draw Cells** group box enable drawing of cells using nodes or faces selected in the graphics window.

- Use the **Using Nodes** option to display cells which contain the selected nodes.
- Use the **Using Faces** option to display cells which contain the selected faces.

The operations available in the **Modify Cells** dialog box for modifying cells are as follows:

Smoothing Nodes

The selected nodes will be repositioned based on the average of the surrounding nodes during node smoothing. Select the nodes to be smoothed and click **Smooth** in the **Operation** group box to smooth nodes.

Splitting Cells

The selected cell will be refined by the addition of a node at the centroid of the cell during splitting. Each cell will be split into four cells.

Moving Nodes

You can move the selected node either to a specified position or by a specified magnitude.

Do the following to move a node to a particular position:

1. Select **node** in the **Filter** list and select the node to be moved.
2. Select **position** in the **Filter** list and select the appropriate position.
3. Click **Move To** in the **Operation** group box.

Do the following to move a node by a specified magnitude:

1. Select **node** in the **Filter** list and select the node to be moved.
2. Enter the magnitude by which you want to move the node in the **Enter Selection** field and press **Enter**.

   The increment will now be selected in the **Selections** list.
3. Click **Move By** in the **Operation** group box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Swapping Cells

You can perform either a 3-2 configuration swap or a 2-3 configuration swap. Refer to Swapping (p. 513) for details on the swap configurations. Do the following to swap cells:

1. Select the appropriate option in the **Filter** list and select the entities to be swapped.
   - Select 3 cells or the common edge in order to perform a 3-2 swap.
   - Select 2 cells or the common face in order to perform a 2-3 swap.
2. Click **Swap** in the **Operation** group box.

Determining the Coordinates of the Centroid

Do the following to determine the centroid:

1. Select the appropriate option in the **Filter** list (**cell**, **face**, **edge**, or **node**) and select the required entity.
2. Click **Centroid** in the **Operation** group box.

The centroid coordinates will be printed in the console.

Determining the Distance Between Entities

Do the following to determine the distance between entities:

1. Select the appropriate option in the **Filter** list and select the two entities between which the distance is to be determined.
2. Click **Distance** in the **Operation** group box.

The distance between the selected entities will be printed in the console.

Projecting Nodes

You can project nodes onto a specified projection line or plane. Do the following to project nodes:

1. Define the projection line or projection plane, as appropriate.
   a. Select the appropriate option in the **Filter** list.
   b. Select two entities to define the projection line or three entities to define the projection plane.
   c. Click **Set** in the **Operation** group box to define the projection line or plane.

      The line coordinates or plane position will be reported in the console.
2. Select the nodes to be projected.

   The projection line/plane will be highlighted in the display window.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click **Project** in the **Operation** group box.

6.1.6. Moving Nodes

Highly skewed meshes can be improved by moving the nodes of the cells. Moving nodes manually is a time consuming process. The node movement process for improving the mesh quality is automated. You can specify quality improvement based on the quality measure specified or based on the warp (quadrilateral elements). You can also choose between the automatic correction procedure and the semiautomatic correction procedure for the quality-based correction. You can also repair negative volume cells by moving nodes using the automatic correction procedure.

6.1.6.1. Automatic Correction

6.1.6.2. Semi-Automatic Correction

6.1.6.3. Repairing Negative Volume Cells

6.1.6.1. Automatic Correction

The automatic correction allows you to improve all the cells in the selected cell zone based on the specified criteria. You can also improve the cells based on the warp values.

You can access the **Auto Node Move** dialog box using the **Mesh** → **Tools** menu; or by right clicking on **Cell Zones** or any individual cell zone in the tree. Selecting from the tree will automatically select the individual cell zone or all zones.

Quality-Based Improvement

For the quality-based correction, you can specify the quality limit based on the quality measure selected, dihedral angle, and the number of iterations per node to be moved. The cells which have a quality above the specified threshold limit will be selected.

For boundary nodes, you can restrict the node movement in the plane containing each of the boundary faces sharing the node being moved. Nodes on sharp features and free edges will not be considered for movement.

The node to be moved for a particular cell will be selected based on the selection of zones in the **Boundary Zones** selection list and an alternative position for the node will be determined. The node will be moved to the new position only if the quality of the cell and its neighbors is improved by the change in node position. The procedure is repeated for the specified iterations per node. You can also set the number of repetitions through the automatic correction procedure as required. By default, the correction procedure is performed only once.

Warp-Based Improvement

For the warp-based correction, you can specify the maximum warp and the number of iterations per face to be improved. The quadrilateral faces which have a warp value greater than the specified maximum warp will be selected. The ideal position for the node to be moved will be determined based on the remaining three nodes. The node will be moved to the new position only if the warp of the face decreases and the cell quality does not deteriorate by the change in node position. The procedure is repeated for the specified iterations per face. You can also set the number of repetitions through the automatic correction procedure as required. By default, the correction procedure is performed four times.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.1.6.2. Semi-Automatic Correction

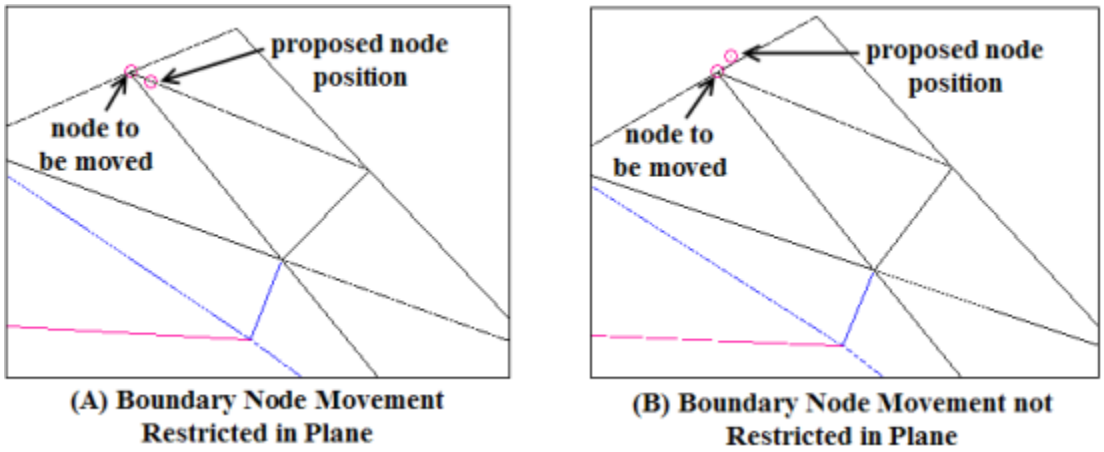
The semi-automatic correction is available only for quality-based improvement. The generic procedure for using the semiautomatic correction is as follows:

1. Select the appropriate zones in the **Cell Zones** drop-down list and the **Boundary Zones** selection list.

2. Select the quality measure and specify values for **Quality Limit**, **Iterations/Node**, and **Dihedral Angle** as appropriate.

3. Enable **Restrict Boundary Nodes Along Surface** if required. When this option is enabled, the movement of the boundary node will be limited to the plane containing the boundary faces sharing the boundary node (see Figure 6.4: Movement of Boundary Nodes (p. 520)).

   Figure 6.4: Movement of Boundary Nodes



4. Click **Skew** to display the cell with the worst quality depending on the quality limit specified. The cell having the worst quality and cells/faces within a pre-defined range of the cell will be displayed.

5. Click **Propose**. The node to be moved and the alternative position will be highlighted. The improvement in the skewness will also be reported in the console.

6. If the proposed position is appropriate, click **Accept**; otherwise, click **Refuse** and then **Propose** to obtain the next suggestion.

7. Click **Next Skew** to proceed with the node correction for the next cell.

6.1.6.3. Repairing Negative Volume Cells

You can also repair negative volume cells by moving nodes. Specify the appropriate boundary zones, dihedral angle, the number of iterations per node to be moved and the number of iterations of the automatic node movement procedure (default, 1). You can also choose to restrict the movement of boundary nodes along the surface.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.1.7. Cavity Remeshing

Cavity remeshing is useful in parametric studies because it enables you add, remove, and replace different parts of the existing mesh. You can compare alternative designs by creating a cavity around the object to be replaced and then, inserting the new object and connecting it to the existing mesh. Prisms can be grown using appropriate parameters and the cavity can be filled with cells. You can also improve the quality of the volume mesh by creating an appropriate cavity around the skewed cells and remeshing it.

You can create a cavity in an existing mesh by removing cells in a defined bounded region. The cells intersecting the bounded region will be marked and the cavity boundaries will be extracted from the marked cells. The marked cells will then be deleted to create the cavity.

The various options available are:

- Removing zones: This option allows you to specify zones to be removed from the existing volume mesh.
- Adding zones: This option allows you to add new zones to the existing volume mesh.
- Replacing zones: This option allows you to remove a set of zones and replace them with a new set of zones.
- Improving a region: This option allows you to define a cavity around skewed cells in the existing mesh. You can modify the mesh as appropriate and then remesh the cavity.

The following sections describe the cavity remeshing options:

6.1.7.1. Tetrahedral Cavity Remeshing

6.1.7.2. Hexcore Cavity Remeshing

6.1.7.1. Tetrahedral Cavity Remeshing

The generic procedure for remeshing a cavity with tetrahedra using the **Cavity Remesh** dialog box is as follows:

1. Select the appropriate zones from the **Remove Boundary Zones** and **Add Boundary Zones** selection lists.
   - **Removing Zones**: Select the zones to be removed in the **Remove Boundary Zones** selection list. Make sure that no zones have been selected in the **Add Boundary Zones** list.
   - **Adding Zones**: Select the zones to be added in the **Add Boundary Zones** selection list. Make sure that no zones have been selected in the **Remove Boundary Zones** list.
   - **Replacing Zones**: Select the zones to be removed in the **Remove Boundary Zones** selection list and the zones to be added in the **Add Boundary Zones** selection list.
   - **Improving a Region**: Make sure that no zones have been selected in the **Remove Boundary Zones** and **Add Boundary Zones** selection lists.
2. Enable **Create Face Group** if you want to create a user-defined group (UDG) comprising the zones defining the cavity domain.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **cavity** UDG and the corresponding **cavity** domain will be created, but the **global** domain will be retained as active when the **Create Face Group** option is enabled. When this option is disabled, the **cavity** domain will be activated when the cavity is created.

Important:

The **Create Face Group** option is useful when using the cavity remeshing feature for large cases. For such cases, you can avoid frequent switching between domains by enabling the **Create Face Group** option when creating the cavity. The basic procedure is as follows:

a. Create a UDG for the cavity and save the boundary mesh for the cavity group defined using the **File/Write/Boundaries...** option (see Writing Boundary Mesh Files (p. 257)).

b. Read this boundary mesh in a separate session and create the volume mesh in the cavity as appropriate.

c. Save the volume mesh and read it back into the previous session using the **Append File(s)** option.

d. Connect the meshed cavity to the parent mesh and merge the cavity domain with the parent domain.

---

3. Enter an appropriate value for **Scale** and click **Compute**. The extents of the bounding box will be computed (based on the zones selected in the **Remove Boundary Zones** and **Add Boundary Zones** selection lists, and the scale factor specified) and reported in the **Cavity Remesh** dialog box. Alternatively, you can specify the extents of the bounding box as required.

   Warning:

   You need to manually specify the extents of the bounding box when using the cavity remeshing feature to improve a region of skewed cells.

4. Specify the orientation of the bounding box in the **Orient** group box.

5. Click **Draw** to preview the cavity domain to be created.

6. Click **Create** to create the cavity domain. The boundary zones touching the cavity bounding box will be separated and included in the cavity domain along with the new zones to be added. Any zones to be removed will not be included in the cavity domain. The existing volume mesh in the cavity will be removed and boundary zones extracted from the interior zones (if any) will be changed to **internal** type and included in the cavity domain.

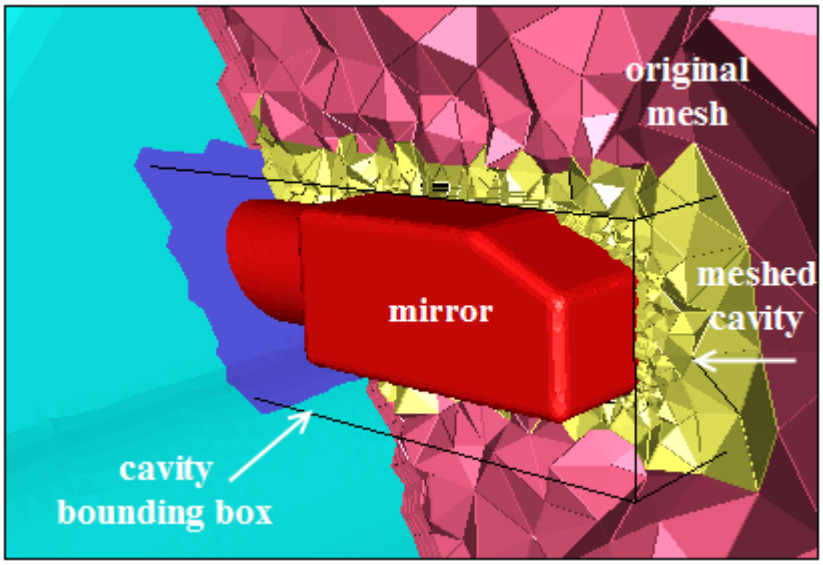
7. Connect the new zones with the boundaries of the cavity domain using node merging or intersect operations when removing, adding, or replacing zones in the volume mesh. Refer to Manipulating Boundary Nodes (p. 653) and Intersecting Boundary Zones (p. 654) for details. Modify the boundary mesh (if required) when improving a region in the volume mesh. Refer to Manipulating the Boundary Mesh (p. 652) for details on the various mesh improvement options.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. Activate the cavity domain and create the volume mesh as appropriate.

   Figure 6.5: Cavity Around a Mirror Remeshed With Tetrahedra (p. 523) shows a cavity created around the rear-view mirror of a car. You can see the original mesh and the cavity created around the mirror and remeshed with tetrahedra. The bounding box defined for the cavity is also shown.

   Figure 6.5: Cavity Around a Mirror Remeshed With Tetrahedra



9. Activate the global domain and delete the boundary zones which are no longer required. Merge the cavity domain with the parent domain using the command `/mesh/cavity/merge-cavity`.

   During the merging operation, the cavity cell zones will be merged with the cell zones in the parent domain. The boundaries extracted from the interior zones will be converted to **interior** type and merged with the corresponding zones in the parent domain. Other boundary zones included in the cavity domain will be merged with the parent face zones.

6.1.7.2. Hexcore Cavity Remeshing

The generic procedure for remeshing a cavity with a hexcore mesh using the **Cavity Remesh** dialog box is as follows:

1. Select the appropriate zones from the **Remove Boundary Zones** and **Add Boundary Zones** selection lists.

   - **Removing Zones**: Select the zones to be removed in the **Remove Boundary Zones** selection list. Make sure that no zones have been selected in the **Add Boundary Zones** list.
   - **Adding Zones**: Select the zones to be added in the **Add Boundary Zones** selection list. Make sure that no zones have been selected in the **Remove Boundary Zones** list.
   - **Replacing Zones**: Select the zones to be removed in the **Remove Boundary Zones** selection list and the zones to be added in the **Add Boundary Zones** selection list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Improving a Region**: Make sure that no zones have been selected in the **Remove Boundary Zones** and **Add Boundary Zones** selection lists.

Note:

- Hexcore cavity remeshing is not supported for baffles/dangling walls.
- Hexcore cavity remeshing is not supported if the mesh contains any dead cell zones.

2. Enable **Hexcore Cavity** to remesh the cavity with hexcore mesh.

Note:

If this option is disabled, only the tetrahedral cells in the cavity will be replaced during the cavity remeshing.

Note:

The **Hexcore Cavity** option is no longer supported and will be removed in a future release.

3. Enter an appropriate value for **Scale** and click **Compute**. The extents of the bounding box will be computed (based on the zones selected in the **Remove Boundary Zones** and **Add Boundary Zones** selection lists, and the scale factor specified) and reported in the **Cavity Remesh** dialog box. Alternatively, you can specify the extents of the bounding box as required.

Warning:

You need to manually specify the extents of the bounding box when using the cavity remeshing feature to improve a region of skewed cells.

Note:

The **Orient** group box will be disabled when the **Hexcore Cavity** option is enabled.

4. Click **Draw** to preview the cavity domain to be created.

Note:

Any dead cells which intersect the cavity bounding box should be converted to fluid or solid type.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Click **Create** to create the cavity domain. The boundary zones touching the cavity bounding box will be separated and included in the cavity domain along with the new zones to be added. Any zones to be removed will be deleted from the global domain. The existing volume mesh in the cavity will be removed and boundary zones extracted from the interior zones (if any) will be changed to **internal** type and included in the cavity domain.

6. Connect the new zones with the boundaries of the cavity domain using node merging or intersect operations when removing, adding, or replacing zones in the volume mesh. Ensure that the new boundaries are properly connected with the existing boundaries. Refer to Manipulating Boundary Nodes (p. 653) and Intersecting Boundary Zones (p. 654) for details. Modify the boundary mesh (if required) when improving a region in the volume mesh. Refer to Manipulating the Boundary Mesh (p. 652) for details on the various mesh improvement options.

7. If required, create the prism layers as appropriate.

   Important:

   To facilitate easier scripting of design changes in Hexcore meshes with prism layers using the Cavity Remeshing utility, the prism base boundary zones separated during the cavity creation are suffixed by **_cavity-prism:#**. This allows you to easily identify the zones to be assigned prism growth settings while remeshing.

8. Click **Remesh** to create the volume mesh in the cavity.

   During the remeshing operation, the cavity cell zones will be merged with the cell zones in the parent domain. The boundaries extracted from the interior zones will be converted to **interior** type and merged with the corresponding zones in the parent domain. If the remeshing involves hexcore mesh generated to selected boundaries, the old boundaries will be removed and replaced during the remeshing operation.

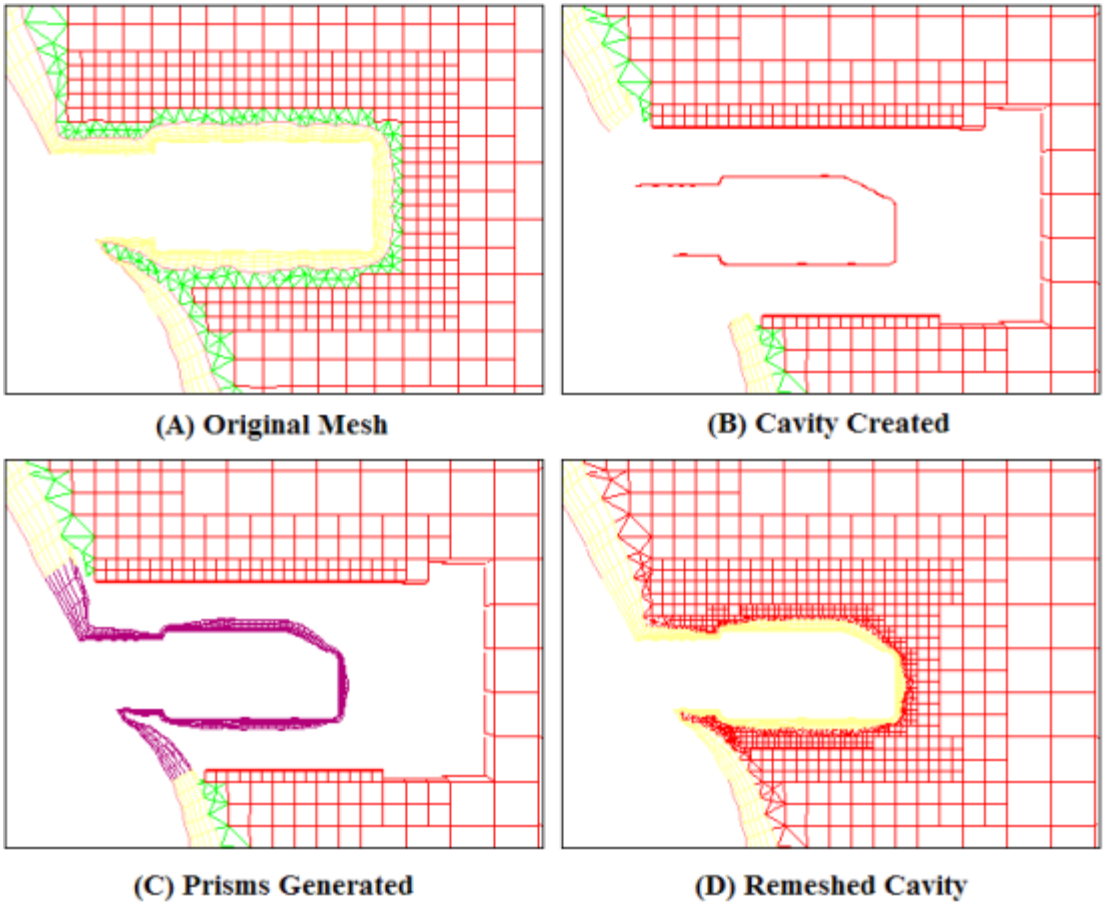
   Note:

   Do not change domains between the **Create** and **Remesh** operations.

Figure 6.6: Cavity Around a Mirror Remeshed With Hexcore Mesh (p. 526) shows the procedure for creating a cavity to replace the rear-view mirror of a car. You can see the original mesh and the cavity created around the mirror. Prisms are then generated and the cavity is remeshed with hexcore mesh.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.6: Cavity Around a Mirror Remeshed With Hexcore Mesh



6.1.8. Manipulating Cell Zones

When a volume mesh is created (of any cell shape) from a boundary mesh, these cells will be grouped into cell zones (contiguous zones separated by boundaries). You can manipulate these zones to control further mesh generation or to duplicate an existing volume mesh to model a repeated geometry.

6.1.8.1. Active Zones and Cell Types

6.1.8.2. Copying and Moving Cell Zones

6.1.8.1. Active Zones and Cell Types

After the initial mesh is generated, all the cells are grouped into contiguous zones separated by boundaries. An artifact of the meshing algorithm is that a virtual zone is created outside the outer boundary, and it is always given a cell type of dead. This zone is automatically deleted upon completion of the initial mesh generation. If the initial mesh generation is interrupted for some reason, this zone will remain in the mesh until the initialization is completed. Other zone types available are fluid and solid.

The zone just inside the outer boundary is automatically set to be active and labeled a fluid zone, although you can change this type later. When refining the mesh, only the active zones are refined.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

By toggling the zones between active and inactive, you can refine different groups of zones independently, using different mesh parameters for the different groups.

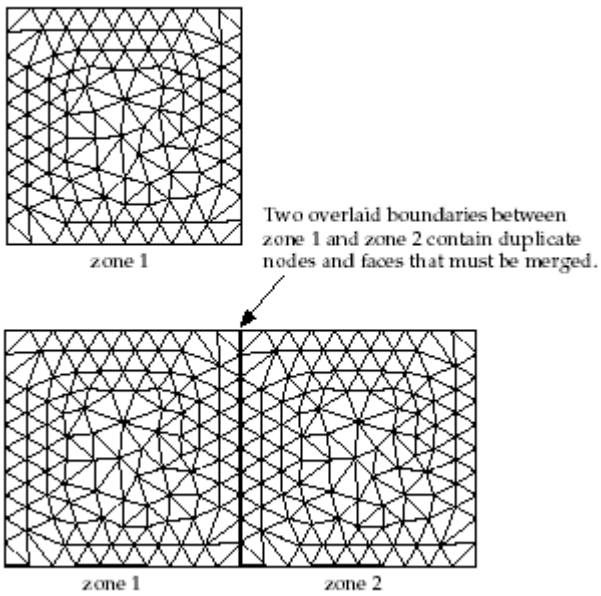
If you plan to refine all cell zones using the same refinement parameters, change the **Non-Fluid Type** in the **Tet** dialog box before initializing the mesh. If you change the **Non-Fluid Type** to any type other than **dead**, all zones will be set active automatically after the initialization occurs. This eliminates the need for you to set all zones to be active in the **Cell Zones** dialog box.

6.1.8.2. Copying and Moving Cell Zones

If you are creating a mesh for a geometry that repeats periodically, you can simplify the meshing tasks. To do this, create the boundary and volume mesh for just one of the repeated sections. Copy the appropriate cell zones to the required locations. If the copy shares a boundary with the original zone (that is, if the two zones are connected), ensure that the distribution of nodes is the same on the two overlaid boundaries.

A simplified case is illustrated in Figure 6.7: Copying and Translating a Cell Zone (p. 527). Here, the volume mesh was created for zone 1, and then copied and translated to create zone 2. The node distribution on the left boundary of zones 1 and 2 is the same as the distribution on the right boundary. Because the left boundary of zone 2 is overlaid on the right boundary of zone 1, there will be duplicate nodes. It is important that you merge these duplicate nodes.

Figure 6.7: Copying and Translating a Cell Zone



The procedure for doing this is as follows:

1. Open the **Merge Boundary Nodes** dialog box.
2. Compare all nodes on both boundaries. To do this, select the two zones in both the **Compare...** and **With...** group boxes.
3. Disable **Only Free Nodes** for both the zones.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Click the **Merge** button to merge the duplicate nodes.

   After the duplicate nodes on the two boundaries have been merged, one of the two boundary face zones will be deleted automatically. Because duplicate faces are merged when the duplicate nodes are merged, one zone will no longer have any faces.

6.1.9. Separating Cell Zones

Use the **Manage Cell Zone** dialog box to separate out cell zones based on a specific type. For instance, you can separate out designated cell zones based on wedge-prism, hex-prism, or poly-prism type.

The procedure for doing this is as follows:

1. Open the **Manage Cell Zone** dialog box.
2. Select the **Separate** option.
3. Select a **Type**. You can choose from **Wedge-Prism**, **Hex-Prism**, or **Poly-Prism**.
4. Select one or more **Cell Zones** or **Cell Zone Groups**.
5. Click the **Apply** button to perform the separation.

   Select **Draw** to visualize the cell zone(s), or select **List** to print out valid cell zones.

You can also separate the prism cells (wedge prisms, hex-prisms, and poly-prisms) from the cell zone using the `separate-wedge-prisms`, `separate-prism-from-hex` or the `separate-prism-from-poly` text commands from the `mesh/separate/` text command menu.

6.1.10. Manipulating Cell Zone Conditions

Case files read in the meshing mode also contain the boundary and cell zone conditions along with the mesh information. The **Cell Zone Conditions** dialog box enables you to copy or clear cell zone conditions when a case file is read.

- You can copy the cell zone conditions from the zone selected in the **With** list to those selected in the **Without** list using the **Copy** option.
- You can clear the cell zone conditions assigned to the zones selected in the **With** list using the **Clear** option.

6.1.11. Using Domains to Group and Mesh Boundary Faces

Domains allow you to group different boundary zones together so that you can create tetrahedral meshes in the region they enclose, or you can limit the zones available for a display or report to only those zones in a selected subset of the domain, rather than the entire domain.

6.1.11.1. Using Domains

6.1.11.2. Defining Domains

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.1.11.1. Using Domains

If you are generating a hybrid mesh containing hexahedra, tetrahedra, and pyramids, you can identify a domain of the global mesh as the region in which you want to generate tetrahedral cells. You can also use domains to group boundary zones so that you can perform diagnostics on them or display them. When you display the grid, the zones available for display will be only those zones that are included in the active domain. Similarly, diagnostic reports will report information about only those zones.

- If you want to check a subset of the global domain, you can create and activate a domain that includes the desired zones, and then proceed with the display or report.
- If you want your grid display or report to include all zones in the mesh, activate the **global** domain in the **Domains** dialog box.

6.1.11.2. Defining Domains

The procedure for defining a new domain is as follows:

1. Deselect all zones in the **Boundary Zones** list and click **Create**. It is quicker to create an empty domain and then add the zones you want, instead of creating a domain with many zones and then removing those you do not want.
2. In the **Boundary Zones** list, select the zones you want to include in the new domain. If you are not sure about the zones, click **Draw** to display the zones that are currently selected in the **Boundary Zones** list.

   It is possible to select all triangular or quadrilateral boundary face zones by choosing **tri** or **quad** in the **Boundary Zone Groups** list.

   - If you are creating a domain within a hybrid mesh to create tetrahedra, make sure that the domain contains all zones required to enclose the region that is to be meshed with tetrahedra, and *only* those zones.
   - If the zones you select do not completely enclose the region, or if you include additional zones that do not bound this region, the tetrahedral meshing is likely to fail or be incorrect.
3. Click **Change**. The dialog box will be updated so that the node, face, and cell zones highlighted in their respective lists are those that are affiliated with the boundary zones in the domain.
4. Select the domain that you want to mesh (or display or report on) in the **Domains** list, and then click the **Activate** button. This domain is then considered to be the active domain, and the **Activate** button is disabled until you select another domain.

   By default, the most recently created domain is automatically set to be the active domain, so you only need to explicitly set the active domain if it is not the one you just created.

Note:

If you are dissatisfied with the domain definition, delete it using the **Delete** button and start over, or modify it by selecting it and using the **Change** button.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.1.12. Checking the Mesh

When you complete the mesh generation process, you need to check the mesh before saving it.

The mesh checking capability will check the mesh connectivity and the orientation of the faces (face handedness, which should be right-handed for all faces because the solvers use a right-handed system). The domain extents and statistics for volume and face area will be reported along with the results of other checks on the mesh.

Mesh → **Check**.

The mesh check information will be printed in the console. The sample output is as follows:

```
 Domain extents.
  x-coordinate: min = -2.500000e+00, max = 2.500000e+00.
  y-coordinate: min = -4.357625e-15, max = 2.000000e+00.
  z-coordinate: min = -1.111022e-04, max = 2.000000e+00.
 Volume statistics.
  minimum volume: 2.297312e-09.
  maximum volume: 7.856795e-03.
  total volume: 1.953600e+01.
 Face area statistics.
 minimum face area: 1.258676e-06.
 maximum face area: 4.944555e-02.
 average face area: 1.939640e-04.
Checking number of nodes per edge.
Checking number of nodes per face.
Checking number of nodes per cell.
Checking number of faces/neighbors per cell.
Checking cell faces/neighbors.
Checking isolated cells.
Checking face handedness.
Checking periodic face pairs.
Checking face children.
Checking face zone boundary conditions.
Checking for invalid node coordinates.
Checking poly cells.
Checking zones.
Checking neighborhood.
Checking modified centroid.
Checking non-positive or too small area.
Checking face zones thread type.
 Done.
```

The domain extents list the x, y, and z coordinates in meters.

The volume statistics include the maximum, minimum, and total cell volume in m^3. A negative value for the minimum volume indicates that one or more cells have improper connectivity. The negative volume cells should be eliminated before the computing the flow solution.

The face area statistics include the maximum, minimum, and average face area in m^2. A value of zero for minimum face area indicates that one or more cells have degenerate faces. The mesh check may also fail if cells have faces with very small non-zero face areas. Such cells should also be corrected before computing the flow solution.

The topological information verified includes the number of nodes and faces per cell. A tetrahedral cell should have 4 faces and 4 nodes while a hexahedral cell should have 6 faces and 8 nodes.

Next, the face-handedness and face node order for each zone will be checked. The zones should contain all right-handed faces and all faces should have the correct node order. The modifed centroid

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

will be checked as well which are cells that contain left-handed faces without any node order issues. Additional diagnostics can be performed for any modified centroids using the **Diagnostics Tools** dialog box. Besides this, the mesh check report will also display warnings based on the results of the checks previously described. If any problems are reported, you need to repair the mesh.

Note:

When checking the mesh, Fluent Meshing checks *all* mesh data, including edges, faces, cells, and nodes that belong to geometry and mesh objects as well as unreferenced zones.

To have more control over the mesh check, use the **Selective Mesh Check** dialog (Selectively Checking the Volume Mesh (p. 531)).

If you want to check the mesh only for those parts of the mesh connected to cell zones (for the Fluent solver), use the following text commands:

```
/mesh/prepare-for-solve yes
/mesh/check
```

Note:

The `prepare-for-solve` command removes the geometry objects. For more information about the command, see `mesh/`.

Note:

The mesh check will issue a warning if multiple cell zones are maintained across an `interior` boundary. The boundary type in such cases should be set to `internal` instead.

The mesh check additionally detects interface zone validity (since an interface zone is not allowed for the face zone shared by a fluid-fluid or a solid-solid cell zone).

Note:

While checking the mesh, Fluent in meshing mode may detect very small face areas (for instance, either non-positive or very small values, such as less than 1e-17 m^2). These very small faces may not be detected by Fluent in solution mode since there, checking the mesh uses a scaled version of the original geometry. As a workaround, in Fluent in meshing mode, you can change the mesh and its units accordingly. Alternatively, within Fluent in solution mode, you can scale the geometry using the original units of the imported geometry.

6.1.13. Selectively Checking the Volume Mesh

After creating a volume mesh, it is important to perform a thorough check of the mesh in order to ensure that the mesh is valid. In some cases, you might prefer to have more control over what aspects of the mesh are checked, or you may wish to perform the mesh check on specific zones, rather than all zones.

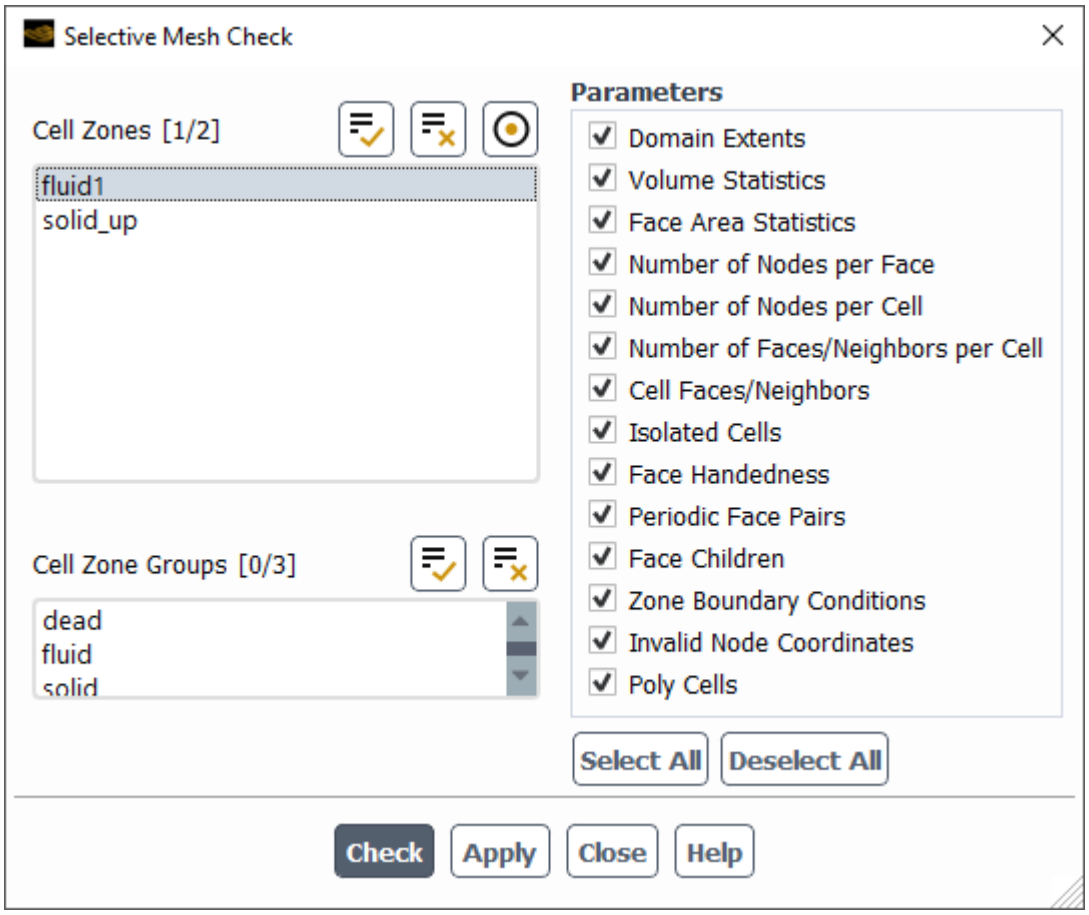
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To selectively check the volume mesh, use the corresponding option available under the **Mesh** menu.

Mesh → Selective Check

This displays the **Selective Mesh Check** dialog where you can have more flexibility and more granular control over the mesh check procedure and scope.

Figure 6.8: The Selective Mesh Check Dialog



1. Select one or more cell zones from the **Cell Zone** list.
2. Alternatively, you can select one or more groups of cell zones from the **Cell Zone Groups** list.
3. From the **Parameters** group, select specific items from the list of available mesh checking categories, use the **Select All** or **Deselect All** buttons accordingly, or keep the default selection of all the items.
   - **Domain Extents**
   - **Volume Statistics**
   - **Face Area Statistics**
   - **Number of Nodes per Face**
   - **Number of Nodes per Cell**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Number of Faces/Neighbors per Cell**
- **Cell Faces/Neighbors**
- **Isolated Cells**
- **Face Handedness**
- **Periodic Face Pairs**
- **Face Children**
- **Zone Boundary Conditions**
- **Invalid Node Coordinates**
- **Poly Cells**

4. Once the cell zone(s) and the parameters are selected, click the **Check** button to perform the selective mesh check, and view the output in the console window. The check is performed for the selected zone and the corresponding adjacent faces, including the interiors of the selected zone, as well as any corresponding edges.

You can perform a selective mesh check using the text user interface (TUI).

`mesh/selective-mesh-check`

The command will prompt you for one or more cell zones, as well as whether to include any of the mesh check parameters. For example:

```
Meshing/mesh> selective-mesh-check
Select one or more cell zones for the mesh check.
Available cell zone(s): (solid_up:232 fluid1)
()
Cell Zones(1) [()] fluid1
Cell Zones(2) [()]
Specify selective mesh check parameters:
Domain Extents? [yes] y
Volume Statistics? [yes] y
Face Area Statistics? [no] n
Number of Nodes per Face? [yes] n
Number of Nodes per Cell? [yes] n
Number of Faces/Neighbors per Cell? [no] n
Cell Faces/Neighbors? [no] n
Isolated Cells? [yes] n
Face Handedness? [yes] n
Periodic Face Pairs? [yes] n
Face Children? [yes] n
Zone Boundary Conditions? [yes] n
Invalid Node Coordinates? [yes] n
Poly Cells? [yes] n
```

Once the prompts are complete, the mesh check will proceed and display the results in the console window. For example, given the previous input example, the output would look like:

```
Mesh check for cell zone(s): (fluid1)
Domain extents.
  x-coordinate: min = -3.627150e+02, max = 1.345894e+01.
  y-coordinate: min = -1.250188e+02, max = 2.475194e-09.
  z-coordinate: min = -1.018144e+02, max = 1.343869e+01.
Volume statistics.
```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
  minimum volume: 1.420005e-03.
  maximum volume: 2.548519e+02.
    total volume: 8.211535e+05.
Done.
```

6.1.14. Checking the Mesh Quality

It is important to check the quality of the volume mesh to evaluate whether it is sufficient for the problem you are modeling. It is recommended that you check the mesh quality before transferring the mesh data to solution mode or writing out the mesh/case file.

You can obtain information about the volume mesh quality by selecting the **Mesh/Check Quality** menu item.

Mesh → Check Quality

When you select **Mesh → Check Quality**, the quality information will be printed in the console. The sample output is as follows:

```
Mesh Quality:

Minimum Orthogonal Quality = 4.37436e-01

Maximum Aspect Ratio = 1.66634e+02
```

> **Note:**
>
> The quality reported corresponds to that reported using the **Report Quality** button in the **General** task page in solution mode (refer to Mesh Quality (p. 1146) for details).

For information about additional quality metrics, set the `/mesh/check-quality-level` to **1** prior to using the **Mesh → Check Quality** option. In addition to the orthogonal quality and Fluent aspect ratio, additional metrics such as cell squish and skewness will be reported when the `check-quality-level` is set to **1**.

6.1.15. Clearing the Mesh

If you are dissatisfied with the volume mesh generated, you can choose to clear the mesh and start again from the boundary mesh. When the mesh is cleared, all interior nodes and faces, and all cells both live and dead are deleted. Only the boundary nodes and faces will be left. After the mesh is cleared, you can generate a new mesh.

This feature is available via **Mesh → Clear**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also use the text command `/mesh/clear-mesh`. To delete the boundary mesh, use the text command `mesh/reset-mesh`. When you use either of these commands you will be asked to confirm that you want to clear or reset the mesh.

Important:

If you have used domains to generate the mesh or grouped zones for reporting (as described in Using Domains to Group and Mesh Boundary Faces (p. 528)), only the mesh in the active domain will be cleared.

6.2. Examining the Mesh

The following sections explain how to examine the mesh visually as well as manipulate the graphics display and mouse function. You can specify various parameters that affect how the mesh appears in the graphics window, manipulate the lighting and create composite views, control the display environment, modify the mouse button functions, and set program control variables.

6.2.1. Displaying the Mesh
6.2.2. Controlling Display Options
6.2.3. Modifying and Saving the View
6.2.4. Composing a Scene
6.2.5. Controlling the Mouse Buttons
6.2.6. Controlling the Mouse Probe Function
6.2.7. Annotating the Display
6.2.8. Setting Default Controls

6.2.1. Displaying the Mesh

You can manipulate the display of mesh entities and modify the way they appear in the graphics window. Options exist for drawing and selecting the entities at your choice of complexity (object, zone, face, edge, node). Tools exist for setting colors or highlighting based on specific parameters such as free or multiply-connected, size field, or geometry recovery. You can also manipulate the display to show a cross-section, adjust lighting, or overlay multiple images.

6.2.1.1. Generating the Mesh Display using Onscreen Tools
6.2.1.2. Generating the Mesh Display Using the Display Grid Dialog Box

6.2.1.1. Generating the Mesh Display using Onscreen Tools

The most commonly used mesh display tools are easily accessible in the graphical user interface ribbon. For more flexibility and advanced display controls, see Generating the Mesh Display Using the Display Grid Dialog Box (p. 537).

In the Model tree, select the objects to be displayed using the right mouse button. At the global **Geometry Objects** or **Mesh Objects** level, you can choose to **Draw All**. At the individual object level, the **Draw** menu item allows you to draw the selected object exclusively, or use the **Draw Options** menu to **Add** to or **Remove** from the existing display, as well as **Highlight** or **Select** the object.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the meshing mode ribbon, there are groups of tools to interactively crop the display of the mesh, to assist in selecting zones or objects, or to modify how the model is displayed in the graphics window.

- You can use the **Bounds** group to limit the display to within a specified distance of a selected entity.

  Select an entity (node, edge, face, zone) and specify a **Delta** value. The **Set Ranges** and **Reset** buttons affect all directions simultaneously, or enable and disable the directions individually. Enable **Cutplanes** to limit the graphics display to within delta value of the selected entity in the selected ranges.

  Tip:

  The settings in the **Bounds** group are also available when examining the volume mesh. The **Display** → **Grid** dialog box has a **Bounded** check box on the **Cells** tab, and the **Cell Zones** context menu includes an option to display the cells in the specified range.

- You can use the **Clipping Planes** group to interactively crop the display in one direction and pan the cropped view as necessary.

  Enable **Insert Clipping Planes** and use the slider to adjust the viewing limits. You can use the **Flip** and **Limit in** options to further manipulate the position and direction of the clipping plane.

  If you want to select and view only certain types of cells, use the **Display Grid** dialog box and select the desired cell **Types** that you want to view and click **Apply**. Any subsequent update of the clip plane view would be based on the selections made in the **Display Grid** dialog box.

  You can also use the **Show Cut Edges** option to display the cut edges of the model exposed by the clipping plane. This option is disabled by default.

  Enable the **Draw Cell Layer** option in order to quickly visualize a layer of cells of the volume mesh on the clipping plane. Once the **Draw Cell Layer** option is enabled, you can enable the **Freeze Cell Layer** option in order to keep the displayed cell layer in place as you continue to study the mesh using other tools.

  To draw a layer of cells on the clipping plane for specific cell zones, select the cell zone(s) under Mesh Objects in the Tree and use the **Draw Cell Layer** option in the context menu.

  Note:

  Your settings in the **Clipping Planes** ribbon group also work in conjunction with specified ranges in the **Bounds** ribbon group, such that your bounds selections and range settings are respected when interacting with the clipping plane.

- You can use the **Selection Helper** group to assist in selecting zones or objects by a **Name Pattern** (wild cards accepted) and **Geometry Recovery** level.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Use the **Filter** drop down to specify face zones, edge zones, objects, object face zones or object edge zones for selection. The **Advanced** button opens a dialog box offering more options for zone selection.

- You can use the **Selection** group to choose how to select entities using the mouse in the graphics display.
- The **Global Display** group contains several check boxes to enable or disable certain elements of the graphics window (Title, Help Text) or model display (Faces, Edges, Highlights).
- The **Local Display** group contains several tools to help with visualizing the current model. These include Transparency, Exploded view, Centroid, Edge Zone selection mode, Face edge display and Measure Distance.
- Several on-screen tool buttons have associated hot-keys. A complete list is available in Shortcut Key Actions (p. 877).

6.2.1.2. Generating the Mesh Display Using the Display Grid Dialog Box

The **Display Grid** dialog box includes a full set of tools to control the display of items using several criteria.

To generate the mesh display:

1. Select **Display** → **Grid...**.
2. Select the entities to be displayed in the **Zones** lists in the **Nodes**, **Edges**, **Faces**, or **Cells** tabs. You can also select several entities of the same group using the **Zone Groups** lists in these tabs.
3. Select the display **Options** as appropriate from the group box in the **Nodes**, **Edges**, **Faces**, or **Cells** tabs.

   You may select intrinsic parameters such as skewness or size. When you specify a range for a particular parameter, the entities that satisfy that condition are displayed. If more than one range limitation is specified, then only those entities that match all specified range limitations are drawn.
4. Select the display **Types** as appropriate from the group box in the **Faces** or **Cells** tabs.

   You can select **Tri**, **Quad**, and/or **Poly** face types. In addition, you can select **Tet**, **Hex**, **Pyramid**, **Wedge**, **Hex-Prism**, and/or **Poly-Prism** cell types.
5. The **Bounds** tab offers enhanced functionality relative to the ribbon group.

   You may insert up to six cutplanes (two in each of the x-, y-, and z-direction) and asymmetrically control their location based on specific coordinates. Alternatively, you can compute ranges based on the proximity to a selected entity.
6. Set the display options as appropriate in the **Attributes** tab. See Mesh Display Attributes (p. 538)

   You can specify information regarding color, display of solid (filled) faces, and shrinkage of faces and cells. These features can help you visualize your mesh effectively and quickly determine the cause of any problems in the mesh.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Click **Display** to draw the mesh in the active graphics window.

   To cancel a display operation, press **Ctrl**+**C** while the data is being processed in preparation for graphical display. You cannot cancel the operation after the program begins to draw in the graphics window.

6.2.1.2.1. Mesh Display Attributes

The mesh display options include modifying the mesh colors, adding the outline of important features to a display, displaying normals, shrinking the faces and/or cells in the display, adding lights for filled mesh displays, and so on.

Modifying the Mesh Colors

You can control the colors used to render the mesh for each zone type or entity. You can modify the colors using the **Grid Colors** dialog box.

1. To open the **Grid Colors** dialog box, go to the **Attributes** tab in the **Display Grid** dialog box, and click **Colors...**.

   By default, the **By Type** coloring option is selected, enabling you to assign colors based on zone type.

2. To change the color assigned to a particular zone type, select the zone type in the **Types** list.

3. Select the required color in the **Colors** list.

4. If you prefer to assign colors based on the zone ID, select the **By Surface** option.

5. Select **By Normal** to color one side of the faces in grey and the other side in yellow, depending on the normal direction. The side colored grey indicates normal direction (normals pointing toward you), while the side colored yellow has normals pointing away.

Important:

You can set colors individually for the mesh displayed on each of the zones using the **Scene Description** dialog box.

Adding Features to an Outline Display

For closed 3D geometries, the standard outline display may not show sufficient details to accurately depict the shape. This is because for each boundary, only the edges on the outside of the geometry (that is, those used by only one face on the boundary) are drawn. You can capture additional features using the **Feature** option in the **Display Grid** dialog box. Specify an appropriate value for the **Feature Angle** to obtain the outline display required.

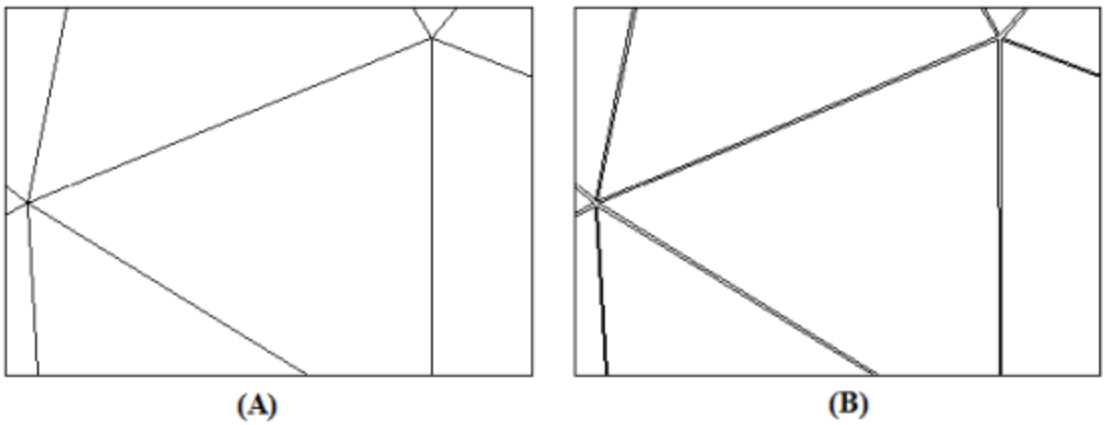
Shrinking Faces and Cells in the Display

To distinguish individual faces or cells in the display, you may want to enlarge the space between adjacent faces or cells by increasing the **Shrink Factor** in the **Display Grid** dialog box. The default value of zero produces a display in which the adjacent faces or cells overlap. A value of 1 creates the opposite extreme, where each face or cell is represented by a point and there is a considerable

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

space between entities. A small value such as 0.01 may be sufficient to enable you to distinguish a face or cell from its neighbor. Figure 6.9: Mesh Display (A) With Shrink Factor = 0 (B) With Shrink Factor = 0.01 (p. 539) shows displays with different **Shrink Factor** values.

Figure 6.9: Mesh Display (A) With Shrink Factor = 0 (B) With Shrink Factor = 0.01



Adding Lights

You can add lights with a specified color and direction to your display. These lights can enhance the appearance of the display, especially for 3D geometries.

- Use the **Display Options** dialog box to enable lighting effects. **Lights On** is enabled by default with the **Automatic** lighting method. The **Automatic** lighting method automatically selects the best lighting method for the current display in the graphics window. You may disable **Lights On** or select an alternate method from the **Lighting** drop-down list. **Flat** is the most basic method: there is no interpolation within the individual polygonal facets. **Gouraud** and **Phong** have smoother gradations of color because they interpolate on each facet.

- The **Lights** dialog box contains options for creating lights and then enabling/disabling individual lights as needed. In this way, you can retain lights that you have defined previously but do not want to use at present.

  The default light, **light 0** is defined to be dark gray with a direction of (1,1,1). To define additional lights, do the following:

  1. Increase **Light ID** to a new value (for example, 1).

  2. Enable **Light On**.

  3. Define the light color by entering a descriptive string (for example, `lavender`) in the **Color** field, or by moving the **Red**, **Green**, and **Blue** sliders to obtain the desired color. The default color for all lights is dark gray.

  4. Specify the light direction by doing one of the following:

     – Enter the (X, Y, Z) Cartesian components under **Direction**.

     – Click the middle mouse button in the desired location on the sphere under **Active Lights**. (You can also move the light along the circles on the surface of the sphere by

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

dragging the mouse while holding down the middle button.) You can rotate the sphere by pressing the left mouse button and moving the mouse (like a trackball).

– Use your mouse to change the view in the graphics window so that your position in reference to the geometry is the position from which you would like a light to shine. Then click **Use View Vector** to update the X,Y,Z fields with the appropriate values for your current position and update the graphics display with the new light direction. This method is convenient if you know where you want a light to be, but you are not sure of the exact direction vector.

5. Repeat steps 1–4 to add more lights.

6. When the lights have been defined, click **Apply** in the **Lights** dialog box to save the definitions.

To remove a light, enter the ID number of the light to be removed in the **Light ID** field and then clear the **Light On** check box. When a light is disabled, its definition is retained, so you can easily add it to the display again at a later time by selecting the **Light On** check box.

If you have made changes to the light definitions, but you have not yet clicked **Apply**, you can reset the lights by clicking **Reset**. All lighting characteristics will revert to the last saved state (that is, the lighting that was in effect the last time you opened the dialog box or clicked on **Apply**).

Using Styles

The **Style Attributes** dialog box is opened by clicking on the **Styles...** button in the **Attributes** tab frame of the **Display Grid** dialog box. It controls the appearance of faces, edges, and nodes that are displayed using the **Display Grid** dialog box. To modify the attributes of a certain type of face, edge, or node (for example, unmeshed), select the appropriate item in the **Styles** list, change the parameters, and click **Apply**. You will see the effect of these changes the next time you display the grid.

Table 6.1: Default Style Attributes

Styles	Faces Visible	Edges Color	Nodes Visible	Color	Weight	Visible	Color	Symbol	Size
left-handed	yes	red	yes	red	2	yes	red	o	0.5
refine	yes	magenta	yes	yellow	2	yes	magenta	o	0.5
free	yes	cyan	yes	foreground	2	yes	orange	o	0.5
multi	yes	yellow	yes	foreground	2	yes	magenta	o	0.5
unmeshed	no	–	yes	orange	2	yes	red	o	0.5
unused	no	–	yes	green	2	yes	green	x	0.5
mark	yes	magenta	yes	yellow	2	yes	magenta	o	0.5
tag	yes	magenta	yes	yellow	2	yes	magenta	o	0.5
face-size	yes	blue	yes	foreground	1	no	–	–	–
cell-size	no	–	yes	red	1	no	–	–	–
face-quality	yes	red	yes	foreground	1	no	–	–	–
cell-quality	no	–	yes	red	1	no	–	–	–

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.2.2. Controlling Display Options

The **Display Options** dialog box contains options for changing some of the rendering parameters for the mesh displayed in the graphics window. You can also control the lighting attributes in a scene or modify parameters based on the graphics hardware and software you are using or change the graphics window layout or color scheme. After making a change to any of the parameters, click **Apply** to update the active graphics window with the new attributes.

Display → Options...

Rendering

The following **Rendering** options are available:

Line Width

Use this control to increase the width from the default value of 1 pixel.

Point Symbol

The default symbol is a + sign inside a circle. Selecting a different symbol in the **Point Symbol** drop-down list may help visibility in some situations.

Animation Options

There are two animation options that you can choose from:

All

uses a solid-tone shading representation of all geometry during mouse manipulation.

Wireframe

uses a wireframe representation of all geometry during mouse manipulation. If your computer has a graphics accelerator, you may not want to use this option; otherwise, the mouse manipulation may be very slow.

Double Buffering

Enabling the **Double Buffering** option can dramatically reduce screen flicker during graphics updates. Note, however, that if your display hardware does not support double buffering and you turn this option on, double buffering will be done in software. Software double buffering uses extra memory.

Front Faces Transparent

This option enables you to turn off the display of outward pointing faces of shells or meshes. This is sometimes useful for displaying both sides of a slit wall. By default, when you display a slit wall, one side will "bleed" through to the other. When you enable the **Front Faces Transparent** option, the display of a slit wall will show each side distinctly as you rotate the display. This option can also be useful for displaying two-sided walls (that is, walls with fluid or solid cells on both sides). Please note, however, that enabling it can also hides some unintended surfaces on certain viewing

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

angles, e.g. while displaying Contours, it might be visible only from one side of the surface and not from the other

Hidden Surface Removal

If you use hidden surface removal, ANSYS Fluent will try to determine which surfaces in the display are behind others. If you do not use hidden surface removal, all surfaces will be displayed, and a cluttered display will result for most 3D mesh displays.

You can choose one of the following methods for performing hidden surface removal in the **Hidden Surface Method** drop-down list. These options vary in speed and quality, depending on the device you are using.

Hardware Z-buffer

is the fastest method if supported by your hardware. The accuracy and speed of this method is hardware dependent.

Painters

will show fewer edge-aliasing effects than hardware-z-buffer. This method is often used instead of software-z-buffer when memory is limited.

Software Z-buffer

is the fastest of the accurate software methods available (especially for complex scenes), but it is memory intensive.

Z-sort only

is a fast software method, but it is not as accurate as software-z-buffer.

Color Scheme

Choose the background color — gradient (Workbench) or solid black (Classic).

Layout

Choose to enable or disable additional display status information:

- **Titles**: caption block area below the graphic containing date, product, and contents of the display. The caption block supports limited text editing. Position the cursor at the end of a line to add or delete text. Text in the caption block will not be deleted when you clear **Annotations**.
- **Axes**: X-Y-Z triad showing orientation. Click on the tip of one axis to rotate the model to the selected orthogonal view, or the cyan-colored dot for isometric view.
- **Scale Ruler**: Visual indicator showing the size in the plane of the graphics display. The indicator automatically resizes as the model is zoomed in the graphics display.
- **Logo**: Ansys and version number.
- **Colormap**: when displaying simulation results, the scale may be shown as function of the visible spectrum.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Graphics Device Information

If you need to know which graphics driver you are using and what graphics hardware it recognizes, you can click **Info** in the **Display Options** dialog box. The graphics device information will be printed in the text window.

Note:

When working remotely, you can optimize the graphics window settings using the text command `remote-display-defaults`. Restore the settings for your local display using the text command `native-display-defaults`.

6.2.3. Modifying and Saving the View

You can use the **Views** dialog box to control the model orientation and zoom level in the graphics window.

Display → Views

- You can choose isometric or one of the standard orthographic views from the list. The displayed model will be centered and zoomed to fit the available graphics window.

  You can also use the **Set view** shortcut ( ) in the View Tools (p. 241) to set one of the standard views.

- You can modify the view by scaling, centering, or rotating the model. You can **Save** a modified view or **Delete** any view from the list.

- You can use **Write...** to open the **Write Views** dialog box to save selected views to a file, which you can **Read...** and use with other mesh files.

6.2.3.1. Mirroring a Non-symmetric Domain

You can use the **Mirror Planes** dialog box to define a symmetry plane for a non-symmetric domain for use with graphics. Click **Define Plane...** to open the dialog box.

1. In the **Plane Equation** group box, specify the coefficients of A, B, C, and the distance from the origin.

   The equation for the mirror plane will be Ax+By+Cz=distance.

2. Click **Add**.

   The **Mirror Planes** list contains a list of all mirror planes you have defined (but not the mirror planes that exist in the domain due to symmetry as they cannot be modified).

3. Select a mirror plane from the list. Click **Apply** to see both the original and the mirrored image in the graphics window.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

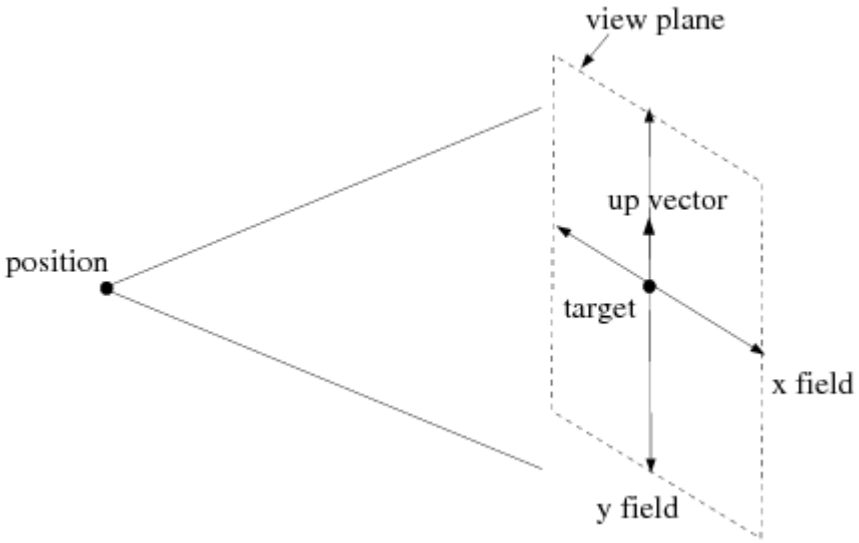
6.2.3.2. Controlling Perspective and Camera Parameters

You can use the **Camera Parameters** dialog box to control perspective and modify the "camera" through which you are viewing the graphics display instead of translating, rotating, and zooming the display.

You can choose to display a perspective view of the graphics (default) or an orthographic view. Select the appropriate option in the **Projection** drop-down list in the **Camera Parameters** dialog box..

The **Camera** is defined by four parameters (see Figure 6.10: Camera Definition (p. 544)):

Figure 6.10: Camera Definition



- **Position** is the camera's location.
- **Target** is the location of the point the camera is looking at.
- **Up Vector** indicates to the camera which way is up.
- **Field** indicates the field of view (width and height) of the display.

Select the parameter to be modified and specify the coordinates or field distances in the **X**, **Y**, and **Z** fields. Click **Apply** after you change each camera parameter.

Important:

When using the sliders and dial to manipulate the view, select **All** in the **Display Options** dialog box so that you can watch the display move interactively while you move the slider or the dial indicator.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.2.4. Composing a Scene

After displaying the mesh or parts of the mesh in your graphics window, you may want to overlay an additional display or move entities around and change their characteristics to increase the effectiveness of the scene displayed.

Display → **Scene...**

You can use the **Scene Description** dialog box—and the **Display Properties** dialog box and the **Transformations** dialog box that are opened from it—to rotate, translate, and scale each entity individually, as well as to change the color and visibility of each entity. You can make geometric entities visible and invisible, thereby adding or deleting entities from the scene one at a time.

6.2.4.1. Changing the Display Properties

6.2.4.2. Transforming Geometric Entities in a Scene

6.2.4.3. Adding a Bounding Frame

6.2.4.4. Using the Scene Description Dialog Box

6.2.4.1. Changing the Display Properties

To enhance the scene in the graphics window, you can change the color, visibility, and other display properties of each geometric entity in the scene using the options available in the **Display Properties** dialog box.

- You can specify different colors for displaying the edges and faces of a zone to show the underlying mesh (edges) when the faces are filled and shaded.
- You can also make a selected entity temporarily invisible. If, for example, you are displaying the entire mesh for a complicated problem, you can make entities visible or invisible to display only certain boundary zones of the grid without regenerating the grid display using the **Display Grid** dialog box.
- You can also use the visibility controls to manipulate geometric entities for efficient graphics display.

For details, refer to Using the Scene Description Dialog Box (p. 546).

6.2.4.2. Transforming Geometric Entities in a Scene

When composing a scene in your graphics window, it is helpful to move a particular entity from its original position or to increase or decrease its size. For example, you may want to temporarily move an interior portion of the mesh outside the mesh boundaries where it can be seen and interpreted more easily.

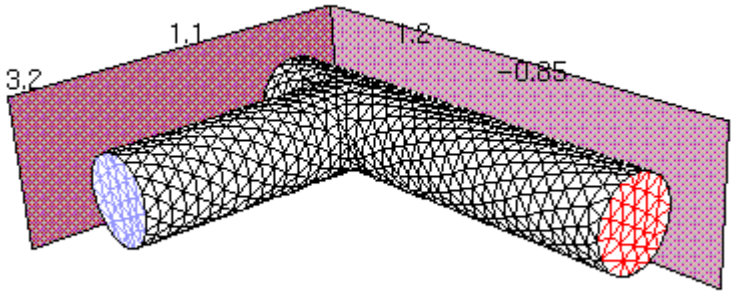
You can also move an entity by rotating it about a specified point. If you want to display one entity more prominently than the others, you can scale its size. All these capabilities are available in the **Transformations** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.2.4.3. Adding a Bounding Frame

You can optionally add a bounding frame to the display of the domain and include measure markings to indicate the length, height, and/or width of the domain using the **Bounding Frame** dialog box.

Figure 6.11: Graphics Display with Bounding Frame



To add a bounding frame to the display:

1. Click **Frame Options...** in the **Scene Description** dialog box.
2. Select **Domain** or **Display** in the **Frame Extents** list to indicate whether the bounding frame should encompass the domain extents or only the portion of the domain that is shown in the display.
3. Indicate the bounding plane(s) to be displayed by clicking on the white square on the appropriate plane of the box shown under the **Axes** heading. You can use any of the mouse buttons. The square will turn red to indicate that the associated bounding plane will be displayed in the graphics window.
4. Specify where you would like to see the measurement annotations by clicking on the appropriate edge of the box. The edge will turn red to indicate that the markings will be displayed along that edge of the displayed geometry
5. Click **Display** to update the display with the current settings.

If you want to include the bounding frame in all subsequent displays, enable the **Draw Frame** option in the **Scene Description Dialog Box** and click **Apply**. If this option is not enabled, the bounding box will appear only in the current display; it will not be redisplayed when you generate a new display (unless you have overlays enabled).

6.2.4.4. Using the Scene Description Dialog Box

You can use the **Scene Description** dialog box and its associated dialog boxes as follows:

1. Select the mesh entities in the **Names** selection list in the **Scene Description** dialog box.

   The **Names** list is a list of the mesh entities that currently exist in the scene (including those that are presently invisible). If you select more than one entity, any operation (color specification,

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

transformation, and so on) will apply to all the selected entities. You can also select the entities by clicking them in the graphics window using the mouse probe button (right mouse button, by default).

2. Set the color, visibility, and other display properties for the mesh entities selected using the **Display Properties** dialog box.

   - You can specify different colors for individual mesh entities. You can specify different colors for displaying the edges and faces of a zone to show the underlying mesh (edges) when the faces of the grid are filled and shaded.

     To modify the color of faces, edges, or lines, choose **face-color**, **edge-color**, **line-color**, or **node-color** in the **Color** drop-down list. The **Red**, **Green**, and **Blue** color scales will show the RGB components of the face, edge or line color, which you can modify by moving the sliders on the color scales. When you are satisfied with the color specification, click **Apply** to save it and update the display.

   - You can control the visibility of individual mesh entities using the **Visible** option. Hence, you can make an entity visible or invisible to display only certain boundary zones, without regenerating the entire mesh display.

   - To enable the effect of lighting for the selected entities on or off, select **Lighting**. You can choose to have lighting affect only certain entities instead of all of them.

   - To toggle the filled display of faces for the selected zones, use the **Faces** option. Enabling **Faces** on here has the same effect as turning it on for the entire mesh in the **Display Grid** dialog box.

   - To enable the display of outer edges, use the **Outer Faces** option. This option is useful for displaying both sides of a slit wall. By default, when you display a slit wall, one side will "bleed" through to the other. When you disable the **Outer Faces** option, the display of a slit wall will show each side distinctly as you rotate the display. This option can also be useful for displaying two-sided walls (that is, walls with fluid or solid cells on both sides).

   - To enable/disable the display of interior and exterior edges of the zones, select **Edges**.

   - To enable/disable the display of the outline of the zones, select **Perimeter Edges**.

   - To enable/disable the display of feature lines, select **Feature Edges**.

   - To enable/disable the display of the lines, select **Lines**.

   - To enable/disable the display of nodes, select **Nodes**.

3. To overlay one display over another, select the mesh entities in the **Names** selection list, enable **Overlays** in the **Scene Composition** group box and click **Apply**. Once overlaying is enabled, subsequent graphics that you generate will be displayed on top of the existing display in the

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

active graphics window. To generate a plot without overlays, you must disable the **Overlays** option and click **Apply**.

Note:

To overlay bounded cell zones over face zones in the display, do the following:

1. Display only the bounded cell zones (face zones should not be selected).
2. Select the cell zones in the **Names** selection list in the **Scene Description** dialog box, enable **Overlays** and click **Apply**.
3. Display the face zones with the appropriate bounds specified.

If you select the bounded face zones first and enable **Overlays**, you will need to deselect the face zones in the **Display Grid** dialog box before displaying the bounded cell zones.

---

4. To transform the entities displayed, select the mesh entities in the **Names** selection list in the **Scene Description** dialog box and click **Transform...** to open the **Transformations** dialog box for the selected entities.

   - To translate the selected entities, enter the translation distance in each direction in the X, Y, and Z real number fields under **Translate**.
   - To rotate the selected entities, enter the number of degrees by which to rotate about each axis in the X, Y, and Z integer number fields under **Rotate By**. You can enter any value between -360 and 360. By default, the rotation origin will be (0,0,0). If you want to spin an entity about its own origin, or about some other point, specify the X, Y, and Z coordinates of that point under **Rotate About**.
   - To scale the selected entities, enter the amount by which to scale in each direction in the X, Y, and Z real number fields under **Scale**. To avoid distortion of the shape, be sure to specify the same value for all three entries.

6.2.5. Controlling the Mouse Buttons

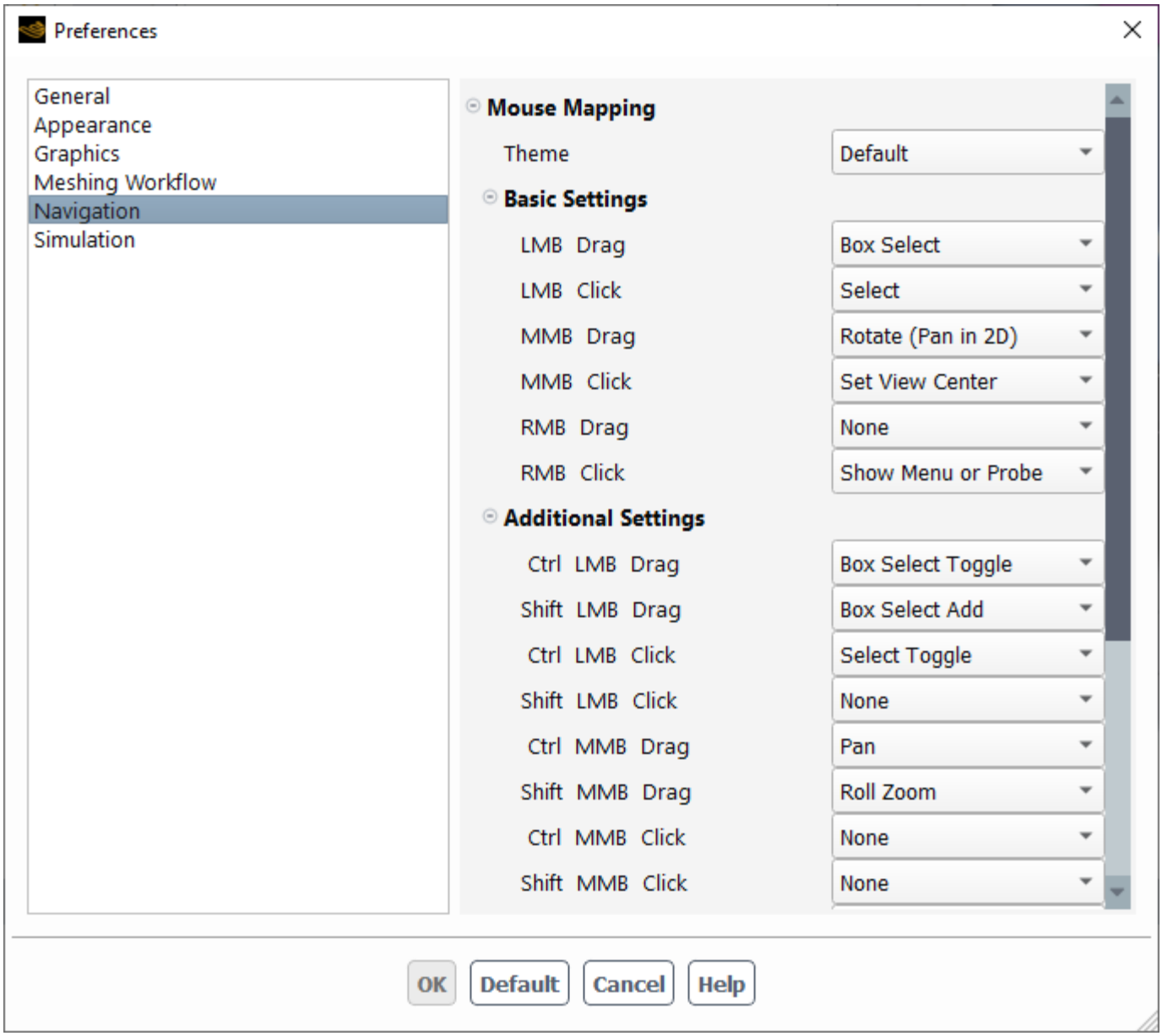
A convenient feature of Ansys Fluent is that it lets you assign a specific function to each of the mouse buttons. According to your specifications, clicking a mouse button in the graphics window will cause the appropriate action to be taken. These functions apply only to the graphics windows; they behave differently when an XY plot or histogram is displayed. For information about the use of mouse buttons in these plots, see Plot Types (p. 4057). Clicking any mouse button in a graphics window will make that window the active window.

You can control mouse button mappings using the **Navigation** branch in Preferences, accessed by clicking **Preferences...** under the **File** ribbon tab.

File → **Preferences...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.12: The Navigation Branch of Preferences



The **Theme** category allows you to choose from multiple predefined mouse assignments. It will change to the **Custom** theme once you make changes to the mouse actions.

The available button functions available are listed below:

Box Select

Dragging from left to right selects the surfaces and volumes that are completely enclosed within the box. Dragging from right to left selects any surface or volume that is in contact with the box.

Box Select Add

Dragging from left to right selects the surfaces and volumes that are completely enclosed within the box. Dragging from right to left selects any surface or volume that is in contact with the box. Each selection is added to the previous, not replaced.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Box Select Toggle

Dragging from left to right selects the surfaces and volumes that are completely enclosed within the box. Dragging from right to left selects any surface or volume that is in contact with the box. If you box select over a surface that is already selected, it will be de-selected.

Pan

Enables you to translate the view by dragging the mouse while holding down the button. The function completes when the mouse button is released or the cursor leaves the graphics window.

Probe

Enables you to select items from the graphics windows and request information about displayed scenes. If you have selected **short description** from the **Probe** drop-down list and then click the mouse-probe button in the graphics window, only the identity of the item you clicked will be printed out in the console; if you have selected **long description** instead, more detailed information about a selected item will be displayed.

For additional information on probe functionality, refer to Controlling the Mouse Probe Function (p. 551).

Roll-Zoom

Enables you to rotate or zoom the view, depending on which direction you drag the mouse. If you drag the mouse horizontally, the display will rotate about the axis normal to the screen. If you drag it vertically, the display will be magnified (if you drag it down) or shrunk (if you drag it up). The function completes when the mouse button is released or the cursor leaves the graphics window.

Rotate (Pan in 2D)

Enables you to rotate the view by dragging the mouse across the screen. Dragging horizontally rotates the object about the screen's y-axis; vertical mouse movement rotates the object about the screen's x-axis. The function completes when the mouse button is released or the cursor leaves the graphics window.

Set View Center

Moves the selected point to the center of the graphics window.

Zoom

Enables you to draw a zoom box, anchored at the point at which the button was pressed, by dragging the mouse with the button held down. When you release the button, if the dragging was from left to right, a magnified view of the area within the zoom box will fill the window. If

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the dragging was from right to left, the area of the window is shrunk to fit into the zoom box, resulting in a "zoomed out" view.

Note:

Use **Esc** to deselect all surfaces in the graphics window.

Important:

3Dconnexion Space products (Mouse, Pilot, and Navigator) are not supported with Ansys Fluent.

Note:

The following are known issues concerning Fluent mouse button behaviors in meshing mode that differ from those found in solution mode:

- Context menus are not available in the graphics window, so the corresponding options (**Show Menu** and **Show Menu or Probe**) options only apply to solution mode.
- While the **Probe** function provides surface information in the console window, it also acts as a **Select Toggle** where objects can be selected and deselected. In addition, while the **Select** function allows you to select objects, it also acts as a **Probe** and provides surface information in the console window.

6.2.6. Controlling the Mouse Probe Function

You can assign a specific mouse probe function and filter using the **Mouse Probe** dialog box. Once set, clicking the mouse probe button in the graphics window causes the action to occur.

Important:

The **box** and **polygon** probe functions will only work if **Probe** is assigned to a "drag" operation, such as **RMB Drag**, and the default probe setting is a "click" operation. You can change the mouse mappings in the **Navigation** branch of **Preferences**, which is accessed via **File** > **Preferences...**.

Display → **Mouse Probe...**

Use the **Function** list to choose an action for the mouse probe button.

off

disables the mouse probe.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

label

displays the label for the selected entity in the graphics window.

select

enables the selection of an entity by clicking. The selected entity is highlighted in the graphics window and, if open, in a list in most dialog boxes.

print

prints information about the selected entity in the console. See Reporting Mesh Information (p. 575) for a description of the output.

box

enables the selection of a group of entities within a box. The selected entities are highlighted in the graphics window and, if open, in a list in most dialog boxes. To define the selection box, click the mouse probe button at one corner of the region to be selected, drag the mouse to the opposite corner, and release the mouse probe button.

polygon

enables the selection of a group of entities within a polygonal region. The selected entities are highlighted in the graphics window and, if open, in a list in most dialog boxes. To define the selection polygon, click the mouse probe button at one vertex of the polygonal region to be selected, and use the left mouse button to successively select each of the remaining vertices.

Click the mouse probe button again (anywhere in the graphics window) to complete the polygon definition.

Use the **Filter** selection list to specify the type of entity selected. The available filters are **off**, **cell**, **face**, **edge**, **node**, **zone**, **position**, **object**, and **size**.

Note:

- If **off** is chosen and a selection is made, it is first checked to see if it is a cell, then a face, an edge, and so on.
- When the **node** filter is used, if a cell or face is selected the node closest to the selection point is picked. Thus nodes do not have to be displayed to be picked.

When using **box** select or **polygon** select, by default, all entities will be selected. To select only visible entities (nodes, faces, zones, objects) instead, enable **Select Visible Entities** in the **Mouse Probe Function** group in the ribbon. The selection will then include only entities visible to the eye, and not

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

those hidden behind other entities in the display. When enabled, ensure that the model is zoomed to an appropriate level for correct selection.

Note:

- If the mesh is not connected, all entities (nodes, edges, faces, zones, objects) will be selected irrespective of whether they are visible or not.
- This visual selection behavior works only on local displays and may generate warning messages when attempting selection on a remote system.

6.2.7. Annotating the Display

Text annotations with optional attachment lines may be added to the graphics windows.

Display → **Annotate...**

The text is added to the window at a location chosen with the mouse, using the mouse-probe button (see Controlling the Mouse Buttons (p. 548) for information on setting the mouse buttons). Dragging with the mouse-probe button pressed will draw an attachment line from the point where the mouse was first clicked to the point where it was released. The annotation text will be placed at the point where the mouse button was released.

The annotation text is associated with the active graphics window; it is removed only when the annotations are explicitly cleared. You can edit the text in the graphics window's caption block by the left mouse button in the desired location. When a cursor appears, you can type the new text or delete the existing text. Text in the caption block will *not* be deleted when you clear annotations.

6.2.8. Setting Default Controls

The **Display** → **Controls** menu item opens the **Controls** dialog box to set file-specific variables called `tgvars`.

The dialog box has two components: the **Categories** drop-down list and the actual variables associated with that category.

Select the desired category, and then enter the desired default values. Click **Apply** to update your file with the new values.

6.3. Determining Mesh Statistics and Quality

The best way to determine the quality of a mesh is by looking at statistics such as maximum skewness, rather than just performing a visual inspection. Unlike structured meshes, unstructured meshes are nearly impossible to comprehend with only a graphical plot. The Mesh category of the ribbon (Performing Meshing Diagnostics (p. 554)) and the **Report** menu has a variety of reporting capabilities. The types of mesh information that can be reported include:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The size, quality, and statistics of a mesh, minimum, maximum, and average values of cell/face size and quality along with summary reporting and investigative tools for studying and trouble-shooting the mesh in fine detail. See Performing Meshing Diagnostics (p. 554) for details.
- The size of the mesh (that is, the number of nodes, faces, and cells in it): use the **Report Mesh Size** dialog box.

  Report → **Mesh Size...**
- Minimum, maximum, and average values of face size and quality: use the **Report Face Limits** dialog box.

  Report → **Face Limits...**
- Minimum, maximum, and average values of cell size and quality in a volume mesh: use the **Report Cell Limits** dialog box. The default quality measure is skewness, but you can also report the limits based on other quality measures or the range of change in cell size.

  Report → **Cell Limits...**
- Boundary cell quality limits: use the **Report Boundary Cell Limits** dialog box.

  Report → **Boundary Cell Limits...**
- Information about individual components of the mesh: use the `print-info` command.

Important:

If you used domains to generate the mesh or group zones for reporting (as described in Using Domains to Group and Mesh Boundary Faces (p. 528)), the report will apply only to the active domain.

These diagnostics and reporting operations are described in the following sections.

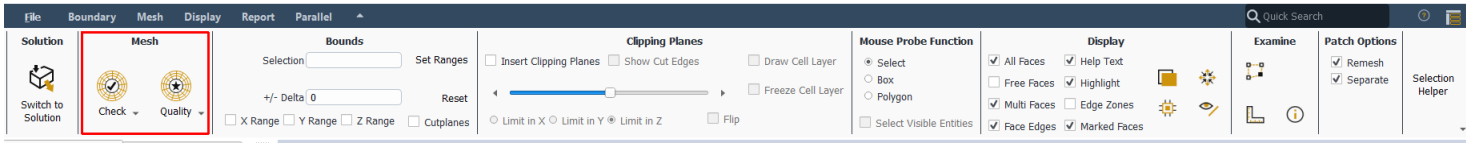
6.3.1. Performing Meshing Diagnostics
6.3.2. Determining Mesh Statistics
6.3.3. Determining Mesh Quality
6.3.4. Reporting Mesh Information

6.3.1. Performing Meshing Diagnostics

While Fluent Meshing already contains a number of separate mesh diagnostic tools, the **Mesh** category in the Ribbon includes many useful mesh checking and quality diagnostics from a single access point.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.13: The Mesh Diagnostic Tools in the Ribbon



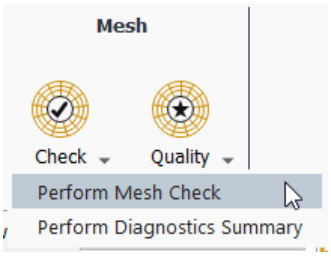
Note:

You can also perform mesh diagnostics using the text user interface. See `diagnostics/` for more information.

6.3.1.1. Performing a Mesh Check and Diagnostics

Use the **Mesh** > **Check** Ribbon menu to access common mesh checking and diagnostic commands.

Figure 6.14: The Mesh Check Diagnostic Tools in the Ribbon



- **Perform Mesh Check**: checks the mesh statistics and prints details to the console window. For example:

```
----------------------------------------
Mesh Check Scope   : 1 Objects, 14 Face Zones, 2 Cell Zones
Objects            : (manifold-solid_up)
----------------------------------------
Domain extents.
  x-coordinate: min = -3.627150e+02, max = 1.345893e+01.
  y-coordinate: min = -1.250188e+02, max = 2.475194e-09.
  z-coordinate: min = -1.018232e+02, max = 1.343616e+01.
Volume statistics.
  minimum volume: 3.402263e-05.
  maximum volume: 2.673741e+02.
    total volume: 1.512242e+06.
Face area statistics.
   minimum face area: 7.036576e-03.
   maximum face area: 3.314637e+01.
   average face area: 2.873890e+00.
Checking number of nodes per edge.
Checking number of nodes per face.
Checking number of nodes per cell.
Checking number of faces/neighbors per cell.
Checking cell faces/neighbors.
Checking isolated cells.
Checking face handedness.
Checking periodic face pairs.
Checking face children.
Checking face zone boundary conditions.
Checking for invalid node coordinates.
Checking poly cells.
Checking zones.
Checking neighborhood.
Checking interfaces.
```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
Checking modified centroid.
Checking non-positive or too small area.
Done.
---------------------------------------
```

By default, the extent, or scope, of the mesh check depends on the current state of the workflow. The scope can include all or some of the existing objects (for example, geometry objects, meshing objects, and unreferenced objects). Use the **Diagnostics Scope** dialog box to change the scope of the analysis (see Setting the Scope of Your Diagnostics (p. 561)).

- **Perform Diagnostics Summary**: diagnoses the mesh and prints a summary to the console window. The summary can contain surface and volume statistics, as well as recommendations for further analysis. For instance:

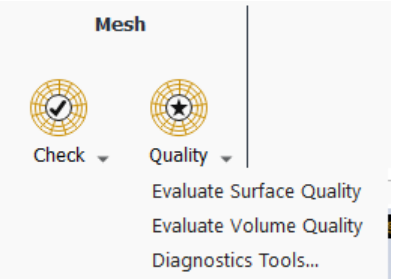
```
---------------------------------------
Diagnostics Scope  : 1 Objects, 22 Face Zones, 5 Cell Zones
Objects            : (hemisphere_mb)
---------------------------------------
Surface Diagnostics :

Total Number of Faces          =     4040
Maximum Skewness               =     0.28
Maximum Aspect Ratio           =    15.53
Number of Free Faces           =        0
Number of Multi Faces          =      982 --> found on 22 face zone(s)
Number of Self Intersections   =        0
Number of Cross Intersections  =     1290 --> found on 8 face zone(s)
Number of Self Proximity       =        0
Number of Point Contacts       =        0
Number of Invalid Normals      =        0
---------------------------------------
Volume Diagnostics :

Total Number of Cells          =    18374
Minimum Orthogonal Quality     =     0.19
Maximum Aspect Ratio           =    21.79
---------------------------------------
```

By default, the extent, or scope, of the summary depends on the current state of the workflow. Along with surface and volume statistics, the summary can include any unreferenced cell zone ids and their associated face zone ids. Use the **Diagnostics Scope** dialog box to change the scope of the analysis (see Setting the Scope of Your Diagnostics (p. 561)). .

6.3.1.2. Diagnosing the Mesh Quality

Use the **Mesh** > **Quality** Ribbon menu to access common mesh quality evaluations and tools.

Figure 6.15: The Mesh Quality Diagnostic Tools in the Ribbon



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Evaluate Surface Quality**: performs a surface quality evaluation and prints a summary to the console window. For instance:

```
----------------------------------------
Surface Quality [Measure : Skewness]
Minimum = 2.2357664e-06, Maximum = 0.56028138, Average = 0.036612933.

Marking Face in Range (0.51028138 0.56028138)...
-------------- ----------
 Cluster Number Entity Count
-------------- ----------
              1          1
              2          1
```

  This command also automatically displays poor-quality marked faces in the graphics window.

  Note:

  Certain surface diagnostics, such as checks for free surfaces, multiple faces, self intersections, and so on, are *not* supported in parallel, when the mesh is distributed. Surface mesh diagnostics are only necessary prior to generating a volume mesh (where the mesh is distributed). Once you generate a volume mesh, surface diagnostics are not required (and the **Evaluate Surface Quality** option will be grayed out and unavailable).. If a surface mesh is subsequently required, you can always agglomerate the mesh, or save the mesh and read it back into Fluent Meshing.

- **Evaluate Volume Quality**: performs a volume quality evaluation and prints a summary to the console window. For instance:

```
----------------------------------------
Cell Quality [Measure : Orthogonal]
Minimum = 0.25625551, Maximum = 0.97745704, Average = 0.73252959.

Marking Cells in Range (0.25625551 0.30625551)...
-------------- ----------
 Cluster Number Entity Count
-------------- ----------
              1          2
              2          1
```

  This command also automatically displays poor-quality marked cells and the specific region in the graphics window.

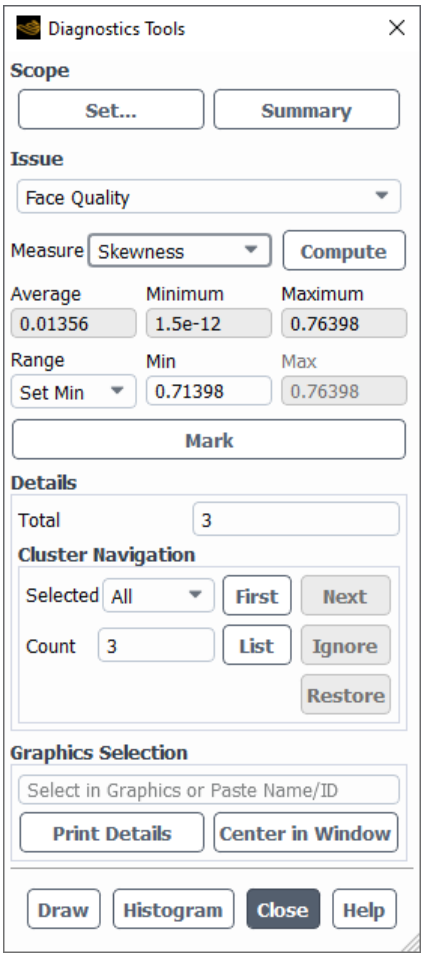
- **Diagnostic Tools...**: opens the **Diagnostics Tools** dialog box (see Using the Diagnostics Tools Dialog Box (p. 557)).

6.3.1.3. Using the Diagnostics Tools Dialog Box

The **Diagnostics Tools** dialog box provides access to several diagnostic resources in a single tool.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.16: The Diagnostics Tools Dialog Box



From this tool, you have convenient access to the following:

1. Use the **Scope** fields to control the scope and breadth and focus of the diagnosis for the current mesh.

   - Click the **Set...** button to open the **Diagnostics Scope** dialog box where you can change the scope of the analysis (see Setting the Scope of Your Diagnostics (p. 561)).

   - Click the **Summary** button to print a general diagnosis summary of the loaded mesh to the console window. The summary includes information about regions (names and number of face/cell zones), face/cell zone summaries, surface and volume checking and diagnostics reporting, suggestions for further diagnostics, and so on. The summary is based on whatever specific scope settings have been applied using the **Diagnostics Scope** dialog box. For example:

```
----------------------------------------
Diagnostics Scope  : 1 Objects, 13 Face Zones, 1 Cell Zones
Objects            : (multizone_prism_3zones)
----------------------------------------
Surface Diagnostics :

Total Number of Faces          =     4468
Maximum Skewness               =     0.56
Maximum Aspect Ratio           =     1.88
Number of Free Faces           =       52 --> found on 4 face zone(s)
Number of Multi Faces          =      312 --> found on 8 face zone(s)
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
Number of Self Intersections   =     1431 --> found on 6 face zone(s)
Number of Cross Intersections  =     1290 --> found on 8 face zone(s)
Number of Self Proximity       =     1349 --> found on 2 face zone(s)
Number of Point Contacts       =        0
Number of Invalid Normals      =        0
---------------------------------------
Volume Diagnostics :

Total Number of Cells          =     3851
Minimum Orthogonal Quality     =     0.26
Maximum Aspect Ratio           =     2.72
Number of Isolated Cells       =        0
---------------------------------------
```

2. Use the **Issues** fields to investigate problematic areas of your mesh, based on the summary. Issues can be categorized by **Face Quality**, **Free Faces**, **Multi Faces**, **Self Intersections**, **Cross Intersections**, **Self Proximity**, **Point Contacts**, **Invalid Normals**, **Cell Quality**, **Stair-Step Locations**, **Isolated Cells**,**Exposed Quads**, **Modified Centroid**, and **Domain Extents**.

   Note:

   Any identified isolated cells will be listed in the diagnostic summary and when marked, can be removed using the **Delete** button.

   Note:

   Based on the results of using the **Summary** button, only the problematic **Issues** and the **Domain Extents** will be available to diagnose. So if the overall summary lists no free faces or self intersections in the entire mesh, for example, then there is no need to diagnose them.

   - When the **Issue** is set to **Face Quality** or **Cell Quality** (when applicable), use the **Measure** field to further investigate mesh issues based on mesh quality measures.

     – Face quality measures include **Skewness**, **Area**, **Size Change**, **Aspect Ratio**, and **Dihedral Angle**.

     – Cell quality measures include **Orthogonal**, **Skewness**, **Volume**, **Size Change**, **Aspect Ratio**, and **ICEMCFD**.

     – For a selected cell/face quality measure, click the **Compute** button to calculate the **Average**, **Minimum**, **Maximum** cell/face values. A histogram is generated for the whole range by default. Vary the **Min** and **Max** as needed using the **Range** drop-down. You can change the **Range** to be set manually in a specific range of values (**In Range**), or to set the upper limit (**Set Max**), or to set the lower limit (**Set Min**). The default settings are adjusted accordingly based on the selected **Measure**.

   - When the **Issue** is set to **Domain Extents**, you are presented with the x, y, and z minimum and maximum values for a bounding box that defines the extents of the mesh domain (**Xmin**, **Ymin**, **Zmin**, **Xmax**, **Ymax**, and **Zmax**).

     To inspect a particular portion of your mesh, enable the **Update Bounds on Selection** check box and select a face or zone in the graphics window (or enter specific values for

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

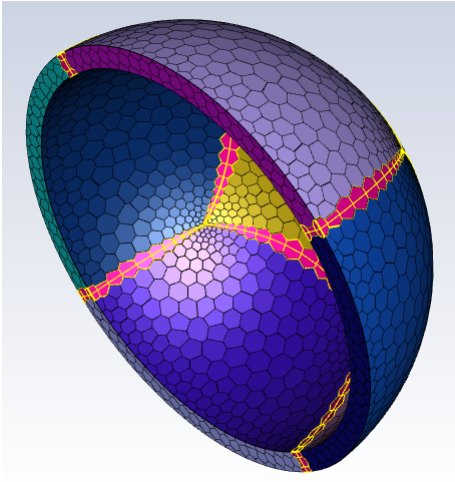
the x, y, or z min/max fields). This will change the extents of the domain according to what you have selected in the graphics window or the fields, and display a specific portion of your mesh. To return to the original bounding box values, disable the check box and click the **Compute** button.

Note:

The **Domain Extents** controls can also be used in conjunction with the **Graphical Selection** fields, where you can print out details of the selected objects in the console window, and center the selected object in the graphics window.

3. When applicable, use the **Mark** button to visibly mark problematic facets and isolate and automatically visualize all or portions of your mesh in the graphics window, based on the selected **Issue** (excluding **Domain Extents**). Collections of problematic facets are grouped into clusters that you can examine separately.

Figure 6.17: Example of Marked Mesh Cells (Multi Face)



Note:

Though generally performed automatically when using the meshing diagnostics tools, keep in mind that you can always use the **Mark Faces** check box in the **Display** category in the Ribbon to show/hide marked faces.

4. Use the **Details** fields to investigate your marked clusters and navigate through them to further identify any problems in your mesh. The **Total** field indicates the total number of clusters of marked cells. The **Cluster Navigation** fields let you view some or all of the identified clusters.

   - Use the **Selected** drop down list to select a particular cluster, or to select **All** of them (the graphics window will display the current selection). Alternatively, you can use the **First** and **Next**/**Previous** buttons to navigate through the collection of clusters as well. As you navigate your clusters, they are individually displayed in the graphics window for

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

closer examination. If an identified cluster is acceptable and presents no problems, use the **Ignore** button to remove it from the list of problematic clusters (and use the **Restore** button to re-add any ignored clusters).

- Use the **Count** to identify the number of problematic cells in the selected cluster.
- Use the **List** button to print details about the problematic cells to the console. for example:

```
Meshing>
------------- ------------------------------------------ ------------------ ------------------------ ------------------
   Entity Name                          Location (x y z)           Skewness               Face zone      Cell Neighbours
------------- ------------------------------------------ ------------------ ------------------------ ------------------
        bf2125                    (4773.8914 1606.9968 0)         0.20018987                   body             (#f #f)
        bf2126          (4773.8914 1606.9968 7.1054274e-15)         0.26276672                   body             (#f #f)
        bf2427               (2724.5304 424.6461 1045.8975)         0.85559527                   body             (#f #f)
        bf2118              (2724.5304 424.6461 -1045.8975)         0.85559527                   body             (#f #f)
        bf2423              (2726.5849 345.36676 813.40325)         0.86004384                   body             (#f #f)
        bf2421             (2726.5849 345.36676 -813.40325)         0.86004384                   body             (#f #f)
        bf2426               (2726.5849 424.6461 1017.0355)         0.89095915                   body             (#f #f)
        bf2117              (2726.5849 424.6461 -1017.0355)         0.89095915                   body             (#f #f)
        bf2137             (4704.9039 1626.7572 -1749.2579)         0.89563344                   body             (#f #f)
        bf2143              (4704.9039 1626.7572 1749.2579)         0.89563344                   body             (#f #f)
        bf2145               (3110.3876 1789.998 13840.279)         0.99905892                   body             (#f #f)
        bf2139              (3110.3876 1789.998 -13840.279)         0.99905892                   body             (#f #f)
```

> **Note:**
>
> Useful keyboard shortcuts are provided in the graphics window, however, some particularly useful ones are: the up/down arrow keys to increase/decrease the display bounds, and **Ctrl-g** combination to draw face zones connected to selected zones.

5. Use the **Graphics Selection** fields to examine details for items that you select in the graphics window. By selecting an object (face, cell etc.) in the graphics window (or pasting an ID using output from the console window), click the **Center in Window** button to bring your graphics selection(s) more into focus. You can also print out a summary of the statistics for the selected object(s) by clicking the **Print Details** button. For example:

```
Meshing> selected hemisphere_mb.12
selected hemisphere_mb.19
selected hemisphere_mb.16

------------------- ------------------------ -------------------- -------------------- --------------------
       Face Zone ID                     Name      maximum-skewness      max-skew Entity               Region
------------------- ------------------------ -------------------- -------------------- --------------------
                 13          hemisphere_mb.12           0.22660859              bf30207      (hemisphere_mb)
                 20          hemisphere_mb.19           0.11830494              bf31612             (fluid)
                 17          hemisphere_mb.16           0.25266775              bf30939           (fluid_1)
```

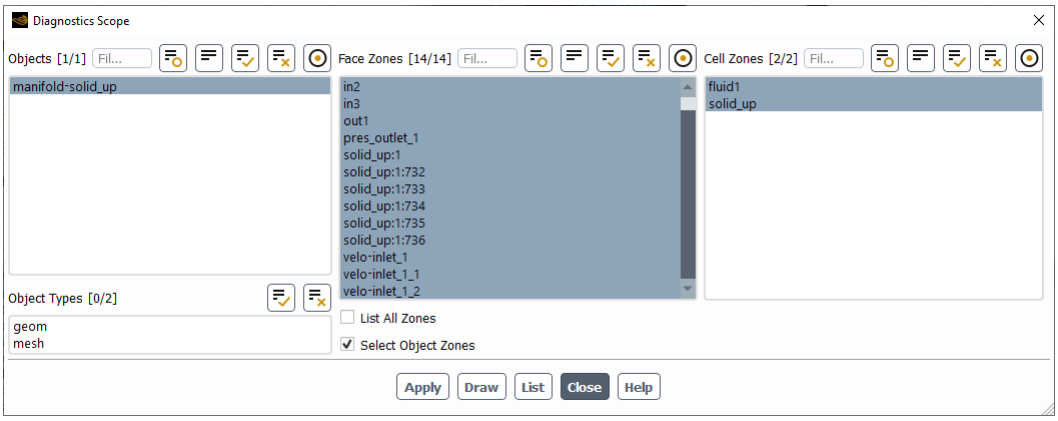
Use the **Histogram** button to generate a histogram of data for the selected issue in the chosen range, if applicable. Use the **Draw** button to draw all zones using the selected scope and the marked faces and cell in the chosen range, if applicable. Use the available shortcut keys (Appendix C: Shortcut Keys (p. 877)) to better interact with the display (for example, **Ctrl**+**/** inserts a clipped plane at the centroid of a selected entity/zone.)

6.3.1.4. Setting the Scope of Your Diagnostics

The **Diagnostics Scope** dialog box lets you determine the breadth of the mesh investigation. From here, you can define the scope of your diagnostic analysis to one or more selected mesh objects, face zones, or cell zones.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.18: The Diagnostics Scope Dialog Box



- Use the **Objects** list to choose from available objects. Use the **Object Types** list to choose from geometry objects or mesh objects. Filtering and the support of wildcards, such as *, ?, [], Boolean NOT (^), Boolean AND (&), as well as Boolean OR (|), is available for the **Objects** list.
- Use the **Face Zones** list to choose from available face zones. Use the **List All Zones** check box to show or hide all available zones. Use the **Select Object Zones** in order to quickly pick all zones associated with the selected object. Filtering and the support of wildcards, such as *, ?, [], Boolean NOT (^), Boolean AND (&), as well as Boolean OR (|), is available for the **Face Zones** list.
- Use the **Cell Zones** list to choose from available cell zones. Filtering and the support of wildcards, such as *, ?, [], Boolean NOT (^), Boolean AND (&), as well as Boolean OR (|), is available for the **Cell Zones** list.
- Use the **Draw** button to visualize your selections in the graphics window.
- Use the **List** button to generate a detailed listing of your zone information. For example:

```
------------------------------
Number of Objects Selected : 1
------------------------------
Object Name : fluid-region-1
Number of Edge Zones  : 0
Number of Face Zones  : 7
Number of Cell Zones  : 0

Regions Info:

 Region Name : fluid-region-1
 Region Type : fluid
 Number of Face Zones  : 7

Total Number of Regions    : 1
Number of Fluid Regions    : 1, (fluid-region-1)
Number of Unfilled Regions : 1, (fluid-region-1)
------------------------------
Face Zones Summary :

      id                            name                 type    count      tri      quad     poly
  _______   ____________________________   ____________________   _____   ______   ______   _____
  169034                      lotuscar.1                   wall   126636   126636        0       0
  169325                     tunnel-xmax                   wall      134      134        0       0
  169322                     tunnel-xmin                   wall      130      130        0       0
  169323                     tunnel-ymax                   wall      549      549        0       0
  169321                     tunnel-ymin                   wall      586      586        0       0
  169326                     tunnel-zmax                   wall      970      970        0       0
  169324                     tunnel-zmin                   wall    13552    13552        0       0
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.2. Determining Mesh Statistics

The **Report Mesh Size** dialog box reports the number of nodes, faces, and cells. Nodes and faces are grouped into those defining the boundaries and those used inside cell zones. Click **Update** to view the latest mesh statistics.

If the generation of the initial mesh fails, you can determine the number of meshed boundary nodes and faces by enabling **Report Number Meshed** in the **Report Mesh Size** dialog box. The headings **Boundary** and **Interior** will be replaced by **Total** and **Meshed**, respectively, and the total number of boundary nodes and faces will be reported along with the number that were meshed.

Note:

Additional mesh statistics diagnostics and reporting tools are available in the ribbon. See Performing Meshing Diagnostics (p. 554) for details.

6.3.3. Determining Mesh Quality

A good quality mesh guarantees the best analysis results for the problem, minimizes the need for additional analysis runs, and improves your predictive capabilities. The mesh should be fine enough to resolve the primary features of the problem being analyzed. The mesh resolution depends on the boundary mesh from which you start and the parameters controlling the generation of the interior mesh. In a high-quality mesh, the change in size from one face or cell to the next should be minimized. Large differences in size between adjacent faces or cells will result in a poor computational mesh because the differential equations being solved assume that the cells shrink or grow smoothly.

Note:

Additional mesh quality diagnostic and reporting tools are available in the ribbon. See Performing Meshing Diagnostics (p. 554) for details.

6.3.3.1. Determining Surface Mesh Quality

6.3.3.2. Determining Volume Mesh Quality

6.3.3.3. Determining Boundary Cell Quality

6.3.3.4. Quality Measure

6.3.3.1. Determining Surface Mesh Quality

Before generating a volume mesh, check the quality of the faces to get an indication of the overall mesh quality. To check face size and quality limits, use the **Report > Face Limits...** dialog box. For 3D meshes, a maximum of less than 0.9 and an average of 0.4 are good. The lower the maximum skewness, the better the mesh. See Quality Measure (p. 566) for information on face and cell quality measures available.

The default quality measure is skewness, but you can also report other quality measures such as the aspect ratio limits or the range in size change for a face zone. Before creating a layer of pyramid cells, check the aspect ratio of the quadrilateral faces that will form the base of the pyramids. This aspect ratio should be less than 8, otherwise the triangular faces of the pyramids will be highly skewed. If the aspect ratio is greater than 8, regenerate the quadrilateral faces:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If the quadrilateral faces were created in a different preprocessor, return to that application and try to reduce the aspect ratio of the faces in question.
- If the quadrilateral faces were created during the building of prism layers, rebuild the prisms using a more gradual growth rate.

Check the skewness of triangular faces on a boundary from which you are going to build prisms to ensure that the quality of the prism cells will be good. After you create the prisms, check the skewness of the triangular faces that were created during the prism generation.

Face Distribution histogram

When checking the quality of the mesh you may find it useful to look at a histogram of boundary face quality or size. You can generate such a plot or report using the **Face Distribution** dialog box.

Display → Plot → > Face Distribution

The number of faces on the selected zones with a size or quality measure value (for example, skewness) in the specified range in each of the regularly spaced partitions is displayed in a histogram format. The x axis shows the size or quality and the y axis gives the number of faces or the percentage of the total number of faces.

You can modify the appearance of the histogram using the options available in the **Axes** dialog box. The following parameters can be modified:

- Labels for the axes.
- Scaling of the axes: decimal (default) or logarithmic.
- Range of values: enable **Auto Range** or specify minimum and maximum values.
- Major and minor rules: horizontal or vertical lines marking the primary and secondary data divisions, respectively. You can select the line color and thickness to be used.
- Number format.

You can also modify the appearance of the histogram using the options available in the **Curves** dialog box. The following parameters can be modified:

- Line style patterns, colors, and weight.
- Marker symbols, colors, and styles.

When the histogram plot is displayed in the graphics window, you can use any of the mouse buttons to add text annotations to the plot. See Controlling the Mouse Buttons (p. 548) and Annotating the Display (p. 553) for more information about using the mouse buttons and annotation features.

6.3.3.2. Determining Volume Mesh Quality

After generating the volume mesh, check the quality of the cells to get an indication of overall mesh quality. The quality of the mesh plays a significant role in the accuracy and stability of the numerical computation. To check cell size and quality limits, use the **Report > Cell Limits...** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The default quality measure is skewness, but you can also report other quality measures such as the aspect ratio limits or the range in size change for a cell zone. See Quality Measure (p. 566) for information on face and cell quality measures available.

Cell Distribution histogram

You can generate a histogram plot to graphically display cell size or quality by using the controls in the **Cell Distribution** dialog box.

Display → Plot → > Cell Distribution

The number of cells in the selected zones with a size or quality value (for example, skewness) in the specified range in each of the regularly spaced partitions is displayed in a histogram format. The x axis shows either the size or the quality and the y axis gives the number of cells or the percentage of the total.

You can modify the appearance of the histogram using the options available in the **Axes** dialog box. The following parameters can be modified:

- Labels for the axes.
- Scaling of the axes: decimal (default) or logarithmic.
- Range of values: enable **Auto Range** or specify minimum and maximum values.
- Major and minor rules: horizontal or vertical lines marking the primary and secondary data divisions, respectively. You can select the line color and thickness to be used.
- Number format.

You can also modify the appearance of the histogram using the options available in the **Curves** dialog box. The following parameters can be modified:

- Line style patterns, colors, and weight.
- Marker symbols, colors, and styles.

When the histogram plot is displayed in the graphics window, you can use any of the mouse buttons to add text annotations to the plot. For more information about the annotation features, see Controlling the Mouse Buttons (p. 548) and Annotating the Display (p. 553).

6.3.3.3. Determining Boundary Cell Quality

For boundary layer flows, boundary cells with low skewness are very important. Use the **Report Boundary Cell Limits** dialog box to report the quality of cells containing a specified number of boundary faces or nodes.

For tetrahedral meshes, the maximum skewness will not be lower than the maximum face skewness, because the boundary cell skewness is limited by the boundary face skewness.

You can remove highly skewed cells with two boundary faces by using the **Tet Improve** dialog box.

Mesh → > Tools > → Tet Improve...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.3.4. Quality Measure

To specify the quality measure (skewness, aspect ratio, change in size, and so on), use the **Quality Measure** dialog box. The default quality measure is skewness.

Report → Quality Measure...

Mesh quality is determined by the following quality measures:

Orthogonal Quality

The orthogonal quality can simply be defined as:

Orthogonal Quality = 1 - **Inverse Orthogonal Quality**

Orthogonal Quality can be reported using the **Quality** option in the ribbon or the **Report Quality** button in the **General** task page in solution mode (refer to Mesh Quality (p. 1146) for details).

Note:

Use the `report/enhanced-orthogonal-quality?` text command to use an enhanced definition of the orthogonal quality measure that combines a variety of quality measures, including: the orthogonality of a face relative to vectors from the cell centroid to the face centroid and to the neighboring cell centroid; a metric that detects poor cell shape at a local edge (such as twisting and/or concavity); and the variation of normals between the faces that can be constructed from the cell face. This definition is optimal for evaluating thin prism cells.

In addition, make sure you have enabled the enhanced orthogonal quality metric when generating and writing your file data.

Enhanced Othogonal Quality

Employs an enhanced definition of the orthogonal quality measure that combines a variety of quality measures, including: the orthogonality of a face relative to a vector between the face and cell centroids; a metric that detects poor cell shape at a local edge (such as twisting and/or concavity); and the variation of normals between the faces that can be constructed from the cell face. This definition is optimal for evaluating thin prism cells.

The range is from -1 to 1. Under ideal conditions, the orthogonal quality is close to 1, whereas valid cells require the quality to be more than 0. The following table lists the range of orthogonal quality values and the corresponding cell quality.

Table 6.2: Orthogonal Quality Ranges and Cell Quality

Orthogonal Quality	Cell Quality
1	orthogonal
0.9–<1	excellent
0.75–0.9	good

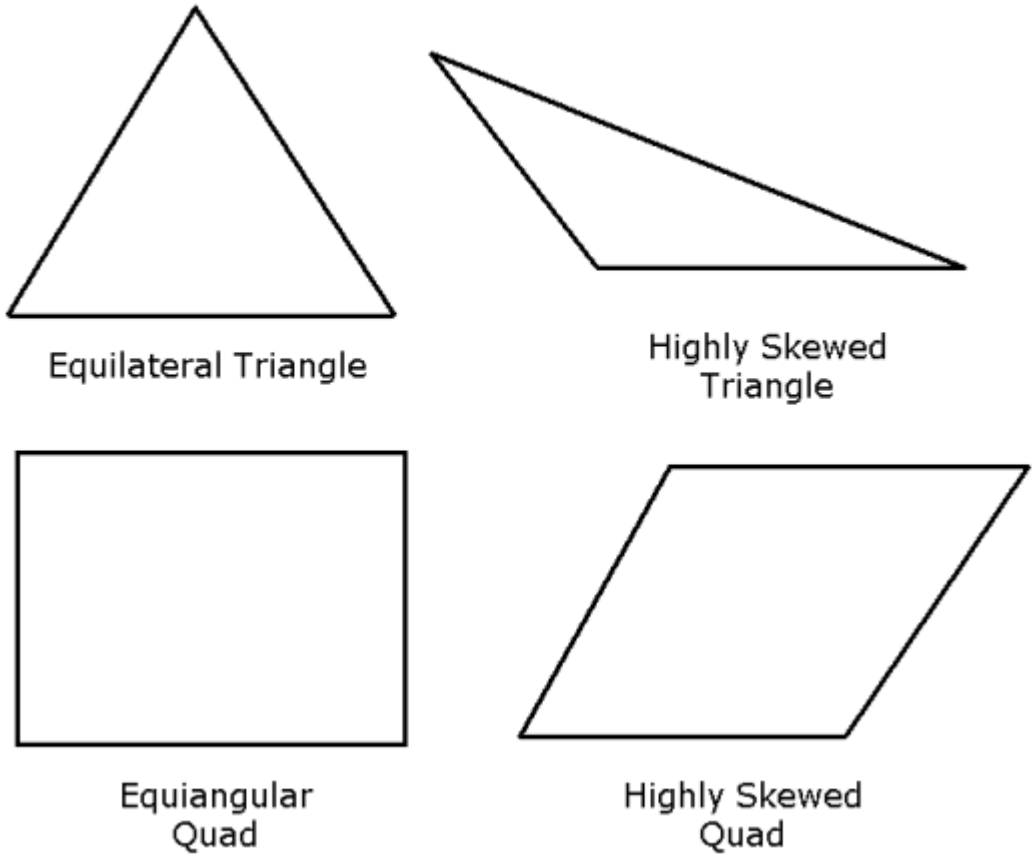
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Orthogonal Quality	Cell Quality
0.5–0.75	fair
0.25–0.5	poor
>0–0.25	bad (sliver)
0	degenerate

Skewness

Skewness is one of the primary quality measures for a mesh. Skewness determines how close to ideal (that is, equilateral or equiangular) a face or cell is:

Figure 6.19: Ideal and Skewed Triangles and Quadrilaterals



The following table lists the range of skewness values and the corresponding cell quality.

Table 6.3: Skewness Ranges and Cell Quality

Skewness	Cell Quality
1	degenerate
0.9–<1	bad (sliver)
0.75–0.9	poor
0.5–0.75	fair

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Skewness	Cell Quality
0.25–0.5	good
>0–0.25	excellent
0	equilateral

According to the definition of skewness, a value of 0 indicates an equilateral cell (best quality) and a value of 1 indicates a completely degenerate cell. Degenerate cells (slivers) are characterized by nodes that are nearly coplanar. Cells with a skewness value above 1 are invalid.

Highly skewed faces and cells should be avoided because they can lead to less accurate results than when relatively equilateral/equiangular faces and cells are used.

Two methods for measuring skewness are:

- Based on the equilateral volume (applies only to tetrahedra).
- Based on the deviation from a normalized equilateral angle. This method applies to all cell and face shapes, such as pyramids and prisms.

The default skewness method for tetrahedra is the equilateral volume method, but you can change to the angle deviation method using the **Quality Measure** dialog box.

Equilateral-Volume-Based Skewness

In the equilateral volume deviation method, skewness is defined as

$$\text{Skewness}=\frac{\text{Optimal Cell Size}-\text{Cell Size}}{\text{Optimal Cell Size}} \tag{6.1}$$

where, the optimal cell size is the size of an equilateral cell with the same circumradius.

In 3D meshes, most cells should be rated good or better, but a small percentage will generally be in the fair range and there are usually even a few poor cells. The presence of poor cells can indicate poor boundary node placement. You should try to improve your boundary mesh as much as possible because the quality of the overall mesh can be no better than that of the boundary mesh.

Normalized Equiangular Skewness

In the normalized angle deviation method, skewness is defined (in general) as

$$\max\left[\frac{\theta_{max}-\theta_e}{180-\theta_e},\frac{\theta_e-\theta_{min}}{\theta_e}\right] \tag{6.2}$$

where

θ_{max} = largest angle in the face or cell

θ_{min} = smallest angle in the face or cell

θ_e = angle for an equiangular face/cell (such as 60 for a triangle, 90 for a quad, and so on)

The cell skewness will be the maximum skewness computed for any face. For example, an ideal pyramid (skewness = 0) is one in which the 4 triangular faces are equilateral (and equiangular)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and the quadrilateral base face is a square. The guidelines in Table 6.3: Skewness Ranges and Cell Quality (p. 567) apply to the normalized equiangular skewness as well.

Size Change

Size change is the ratio of the area (or volume) of a face (or cell) in the geometry to the area (or volume) of each neighboring face (or cell). This ratio is calculated for every face (or cell) in the domain. The minimum and maximum values are reported for the selected zones.

Edge Ratio

The edge ratio is defined as the ratio of maximum length of the edge of the element to the minimum length of the edge of the element.

By definition, edge ratio is always greater than or equal to 1. The higher the value of the edge ratio, the less regularly shaped is its associated element. For equilateral element shapes, the edge ratio is always equal to 1.

Aspect Ratio

The aspect ratio of a face or cell is the ratio of the longest edge length to the shortest edge length. The aspect ratio applies to triangular, tetrahedral, quadrilateral, and hexahedral elements and is defined differently for each element type.

The aspect ratio can also be used to determine how close to ideal a face or cell is.

- For an equilateral face or cell (such as an equilateral triangle, a square, and so on), the aspect ratio is 1.
- For less regularly-shaped faces or cells, the aspect ratio will be greater than 1 as the edges differ in length.
- For triangular faces and tetrahedral cells and for pyramids, you can usually focus on improving the skewness, and the smoothness and aspect ratio will consequently be improved as well.
- For prisms, it is important to check the aspect ratio and/or the change in size in addition to the skewness, because it is possible to have a large jump in cell size between two cells with low skewness or a high-aspect-ratio low-skew cell.

Squish

Squish is a measure used to quantify the non-orthogonality of a cell with respect to its faces. It is defined as follows:

$$1-\left(A\cdot r_c\right)/\left|r_c\right| \tag{6.3}$$

where

A = face unit area vector

r_c = the vector connecting the adjacent cell centroids (for face squish) or the cell and face centroids (for cell squish)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Warp (Quadrilateral Elements only)

Face warp applies only to quadrilateral elements and is defined as the variation of normals between the two triangular faces that can be constructed from the quadrilateral face. The actual value is the maximum of the two possible ways triangles can be created.

Mathematically, it is expressed as follows:

$$\frac{Z}{min[a,b]} \tag{6.4}$$

where

Z = the deviation from a best-fit plane that contains the element

a, b = the lengths of the line segments that bisect the edges of the element

The value of face warp ranges between 0 and 1. A value of 0 specifies an equilateral element and a value of 1 specifies a highly skewed element.

Dihedral Angle

Dihedral angle applies only to faces and not cells. The dihedral angle is defined as the angle between the normals of adjacent faces. This quality measure is useful in locating sharp corners in complicated geometries. The dihedral angle value ranges from 0–180.

ICEM CFD Quality

The ICEM CFD quality measure calculates element quality based on the quality values in ANSYS ICEM CFD. This measure is only available for cells, not faces.

- Tetrahedra: The quality is calculated as the skewness of the tetrahedral element.

  The remaining quality values are based on various quality metrics computed in ANSYS ICEM CFD.

  Quality = 1 - Ansys ICEM CFD Quality

- Hexahedra: The quality is based on the Determinant, Max Orthogls, and Max Warpgls metrics in ANSYS ICEM CFD.

  – The determinant is the ratio of the smallest and the largest determinant of the Jacobian matrices, where a Jacobian matrix is computed at each node of the element.

  – The max orthogls metric calculates the maximum deviation of the internal angles of the element from 90 degrees. Angles between 180 and 360 degrees (deviation up to 270 degrees) will also be considered.

  – The warp for each face is calculated as the maximum angle between the triangles connected at the diagonals of the face. The max warpgls metric is calculated as the maximum warp of the faces comprising the element.

  These values are normalized and the minimum value of the three normalized diagnostics will be used.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Pyramids: The quality is based on the Determinant computed in ANSYS ICEM CFD.

  The determinant is the ratio of the smallest and the largest determinant of the Jacobian matrices, where a Jacobian matrix is computed at each node of the element.

- Prisms: The quality is based on the Determinant and Warp computed in ANSYS ICEM CFD.

  – The determinant is the ratio of the smallest and the largest determinant of the Jacobian matrices, where a Jacobian matrix is computed at each node of the element.

  – The warp for each face is calculated as the maximum angle between the respective edges and a plane containing the mid-points of the edges comprising the face. The warp for the element is then calculated as the maximum warp of the faces comprising the element.

  The quality is calculated as the minimum of the determinant and warp.

For further details on the quality measures available in ANSYS ICEM CFD, refer to the ANSYS ICEM CFD Help Manual.

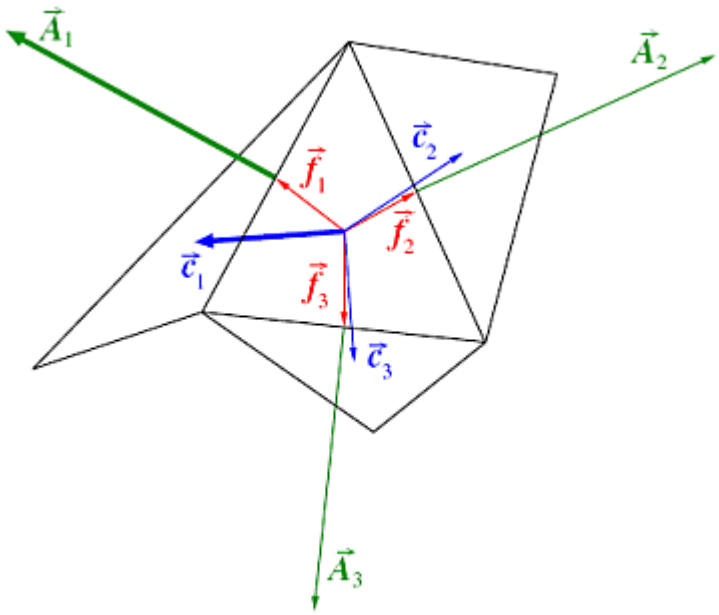
The range of quality values reported is between 0-2 (the range in ANSYS ICEM CFD is -1 to 1, and a value of 2 corresponds to -1 in ANSYS ICEM CFD, 1 to 0, and 0 to 1, respectively). A value of 0 indicates a perfect, non-distorted element, 1 indicates a degenerate element, and values above 1 indicate an invalid element (such as concave, dihedral angle > 180 degree, or negative volume elements).

Ortho Skew

The ortho skew quality for a cell is computed using the face normal vector, $\vec{A}_i$ for each face; the vector from the cell centroid to the centroid of each of the adjacent cells, $\vec{c}_i$; and the vector from the cell centroid to the centroid of each face, $\vec{f}_i$. Figure 6.20: Vectors Used to Compute Ortho Skew/Inverse Orthogonal Quality for a Cell (p. 572) illustrates the vectors used to determine the ortho skew quality for a cell.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.20: Vectors Used to Compute Ortho Skew/Inverse Orthogonal Quality for a Cell



For each face i, the calculation first finds the value of (1 - the cosine of the angle between the face normal vector and the corresponding vector from the centroid of the cell to the centroid of that face):

$$1-\frac{\vec{A}_i \cdot \vec{f}_i}{|\vec{A}_i||\vec{f}_i|} \tag{6.5}$$

Next, for each face i, the calculation finds the value of (1 - the cosine of the angle between the face normal vector and the corresponding vector from the centroid of the cell to the centroid of the adjacent cell that shares that face):

$$1-\frac{\vec{A}_i \cdot \vec{c}_i}{|\vec{A}_i||\vec{c}_i|} \tag{6.6}$$

The reported Ortho Skew quality for a cell then depends on the cell type:

- For most cells, Ortho Skew is the maximum of the above quantities computed for each face.
- For pyramid cells, Ortho Skew is the maximum of the cell skewness and the above quantities computed for each face.

Note:

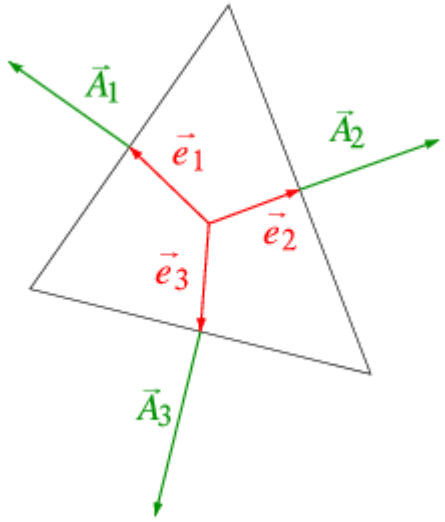
- When the cell is located on the boundary, the vector $\vec{c}_i$ across the boundary face is ignored during the quality computation.
- When the cell is separated from the adjacent cell by an internal wall, the vector $\vec{c}_i$ across the internal boundary face is ignored during the quality computation.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- When the adjacent cells share a parent-child relation, the vector $\vec{f}_i$ is the vector from the cell centroid to the centroid of the child face while the vector $\vec{c}_i$ is the vector from the cell centroid to the centroid of the adjacent child cell sharing the child face.

The ortho skew quality for faces is similarly computed using the edge normal vector, $\vec{A}_i$, and the vector from the face centroid to the centroid of each edge, $\vec{e}_i$. Figure 6.21: Vectors Used to Compute Ortho Skew Quality for a Face (p. 573) illustrates the vectors used to determine the ortho skew quality for a face.

Figure 6.21: Vectors Used to Compute Ortho Skew Quality for a Face



The ortho skew for a face is the maximum of the computed values for (1 - the cosine of the angle between the edge normal vector and the vector from the centroid of the face to the centroid of the edge), for each edge i:

$$1-\frac{\vec{A}_i \cdot \vec{e}_i}{|\vec{A}_i||\vec{e}_i|} \tag{6.7}$$

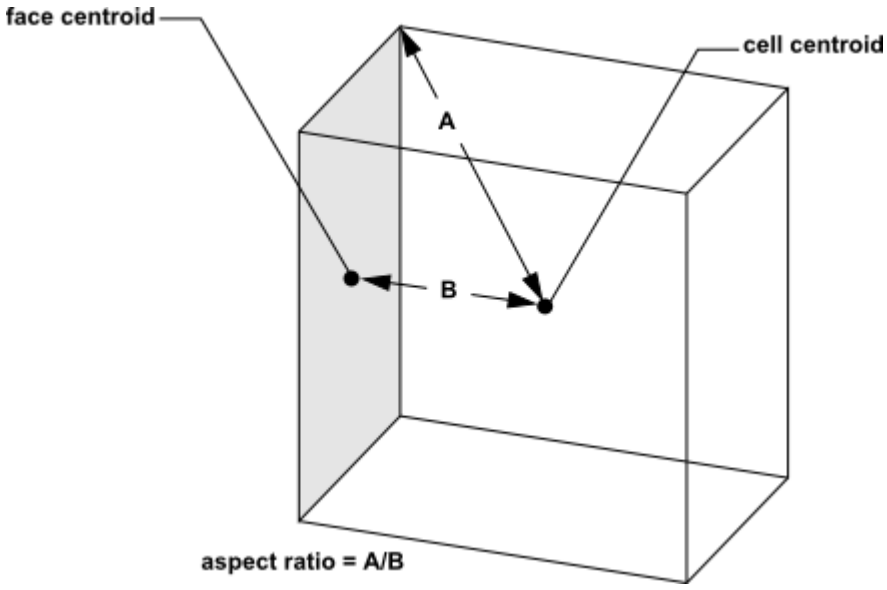
When the Rapid Octree mesher is used, the quality measure is changed to ortho skew by default.

Fluent Aspect Ratio

The Fluent aspect ratio is a measure of the stretching of a cell. It is computed as the ratio of the maximum value to the minimum value of any of the following distances: the normal distances between the cell centroid and face centroids, and the distances between the cell centroid and nodes. For a unit cube (see Figure 6.22: Calculating the Fluent Aspect Ratio for a Unit Cube (p. 574)), the maximum distance is 0.866, and the minimum distance is 0.5, so the aspect ratio is 1.732. This type of definition can be applied on any type of mesh, including polyhedral.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.22: Calculating the Fluent Aspect Ratio for a Unit Cube



Inverse Orthogonal Quality

The inverse orthogonal quality for a cell is computed using the face normal vector, $\vec{A}_i$ for each face; the vector from the cell centroid to the centroid of each of the adjacent cells, $\vec{c}_i$; and the vector from the cell centroid to the centroid of each face, $\vec{f}_i$ (see Figure 6.20: Vectors Used to Compute Ortho Skew/Inverse Orthogonal Quality for a Cell (p. 572)). For each face, the cosines of the angle between $\vec{A}_i$ and $\vec{c}_i$, and between $\vec{A}_i$ and $\vec{f}_i$ are calculated. The smallest calculated cosine value is the orthogonality of the cell. Then, inverse orthogonality is found by subtracting this cosine value from 1.

Finally, **Inverse Orthogonal Quality** depends on cell type:

- For tetrahedral, prism, and pyramid cells, Inverse Orthogonal Quality is the maximum of the cell skewness and the inverse orthogonality.
- For hexahedral and polyhedral cells, Inverse Orthogonal Quality is the same as the inverse orthogonality.

The worst cells will have an Inverse Orthogonal Quality closer to 1 and the best cells will have an Inverse Orthogonal Quality closer to 0.

Note:

Inverse Orthogonal Quality = 1 - **Orthogonal Quality**

Orthogonal Quality can be reported using the **Quality** option in the ribbon or the **Report Quality** button in the **General** task page in solution mode (refer to Mesh Quality (p. 1146) for details).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.4. Reporting Mesh Information

In addition to the mesh diagnostic and reporting tools available in the ribbon (see Performing Meshing Diagnostics (p. 554) for details), the mouse probe **print** function displays information about individual components of the mesh. The type of information is determined by the selected mouse probe filter.

The `print-info` command reports information about an individual mesh entity (that is, a node, face, or cell). In some circumstances, the information includes prefixes or punctuation to identify the information type. An entity is identified by its prefix with an index value. Valid prefixes are: `bn` (boundary node), `n` (node), `bf` (boundary face), `f` (face), and `c` (cell). Hence, the first boundary node would be `bn1`.

The output for some of the entity types is as explained in this section.

cell

The following information is listed for a cell:

- zone ID
- the nodes forming the cell
- the faces forming the cell
- the size
- the coordinates of the circumcenter (relevant for tetrahedral cells only)
- square of the circumcenter radius (relevant for tetrahedral cells only)
- the quality of the cell

```
c26 = (1 (n262 n34 bn204 bn205) (f4743 f5372 f1822 f3426)
0.00020961983 (2.7032523 0.32941867 0.072823988)
0.0081779587 0.44769606)
```

The default measure of quality is skewness, but you can use the **Quality Measure** dialog box to specify aspect ratio or change in size instead.

face

The following information is listed for a face:

- associated zone ID
- the nodes forming the face
- the neighboring cells

```
f32 = (4 (n95 bn197 n90) (c4599 c1279))
```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Due to the manner in which the algorithm maintains memory, not all indices will have values—that is, an empty slot can be caused by a delete operation. Empty slots are reused in subsequent operations, so the actual entity at a particular index may change as the mesh is generated.

boundary face

The following information is listed for a boundary face:

- associated zone ID
- the nodes forming the face
- the neighboring cells
- the periodic shadow (null if not periodic)
- the quality of the face

```
bf17 = (2 (bn1308 bn1314 bn1272) (() c1533) () 0.014393554)
```

The default measure of quality is skewness, but you can use the **Quality Measure** dialog box to specify aspect ratio or change in size instead.

node

The following information is listed for a node:

- associated zone ID
- Cartesian coordinates of the node
- the node radius

```
n30 = (5 (2.433799 0.078610075 0.52846689) 0.12702106)
```

For interior nodes, the node radius is the distance-weighted average to the surrounding nodes.

boundary node

The following information is listed for the boundary node:

- associated zone ID
- Cartesian coordinates of the node
- node radius
- faces using the node.

```
bn1 = (3 (0 0 0) 0.1 (bf2099 bf1093 bf193))
```

For boundary nodes, the node radius is the average distance to neighboring boundary nodes.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 7: Advanced Meshing Topics

While the Fluent Meshing Guided workflows are adequate for most meshing applications, Fluent in meshing mode still contains powerful tools and techniques that more advanced users will find useful.

7.1. CAD Assemblies

7.2. Size Functions and Scoped Sizing

7.3. Objects and Material Points

7.4. Object-Based Surface Meshing

7.5. Object-Based Volume Meshing

7.6. Manipulating the Boundary Mesh

7.7. Wrapping Objects

7.8. Creating a Mesh

7.9. Generating Prisms

7.10. Generating Tetrahedral Meshes

7.11. Generating the Hexcore Mesh

7.12. Generating Polyhedral Meshes

7.13. Generating Poly-Hexcore Meshes

7.14. Generating the CutCell Mesh

7.15. Generating Rapid Octree Meshes

7.1. CAD Assemblies

The **CAD Assemblies** mode offers additional tools for imported CAD data in Fluent Meshing. The **CAD Assemblies** tree represents the CAD tree as it is presented in the CAD package in which it was created. All sub-assembly levels from the CAD are maintained on import in Fluent Meshing.

Advantages of using the CAD Assemblies tree include the following:

- The CAD assemblies tree enables faster visualization which saves time in part management for large, complex models comprising multiple assemblies.
- It enables re-importing or updating of selected parts or bodies using different faceting qualities and topology representations from a neutral database file.
- Geometry and mesh objects can be easily created for only those assemblies, parts, or bodies that are needed for the analysis. Other assemblies, parts, or bodies can be simply suppressed.
- The geometry and mesh objects are linked to the corresponding CAD objects (which are then locked), enabling quick design changes and updates.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Labels can be assigned to entities, and these are preserved throughout the object-based meshing workflow in Fluent Meshing.

The use of CAD assemblies for CAD import in Fluent Meshing is described in the following sections.

7.1.1. CAD Assemblies Tree
7.1.2. Visualizing CAD Entities
7.1.3. Updating CAD Entities
7.1.4. Manipulating CAD Entities
7.1.5. CAD Association

7.1.1. CAD Assemblies Tree

The **CAD Assemblies** tree is created when the **Create CAD Assemblies** option is selected for CAD import. It represents the CAD tree as it is presented in the CAD package in which it was created. All sub-assembly levels from the CAD are maintained on import in Fluent Meshing.

To create the CAD Assemblies tree on import, enable **Create CAD Assemblies** in the **Object Creation** group box in the **CAD Options** dialog box. The CAD object and zone granularity can be specified in the **Object Creation** group box. You can choose to create one CAD object per **part**, **body**, CAD **file** or **selection** imported. Similarly, you can choose to create one CAD zone per **body**, **face**, or **object** imported.

Tip:

Importing CAD objects by body gives an ideal CAD assemblies tree, with the original sub-assemblies preserved.

The CAD assemblies tree includes individual CAD assemblies or sub-assemblies imported in Fluent Meshing. The entity at the highest level is also referred to as the **root node**.

The CAD entities in the tree are categorized as **components** and **bodies**. Components represent an assembly, sub-assembly, or part in the original CAD package, while bodies are the basic entities in the CAD assemblies tree which include CAD zones. You can also set up labels for the CAD zones, if required. Named Selections are also imported as labels. These labels are preserved throughout the object-based meshing workflow in Fluent Meshing.

In addition to the context-sensitive menus, you can manage the CAD entities using hotkeys or onscreen tool buttons. Tools exist for visualizing the CAD entities, for operations such as updating and modifying the CAD entities, creating and/or modifying geometry or mesh objects, and tree selection options. See Appendix C: Shortcut Keys (p. 877) for more information.

Note:

When working with CAD Assemblies, certain meshing ribbon tools are disabled.

At the global **CAD Assemblies** level, you can use the menu options to draw or delete all the assemblies imported, and obtain the locations of the referenced FMDB files. The **Tree** sub-menu contains options for navigation and selections in the tree.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.1.1.1. FMDB File
7.1.1.2. CAD Entity Path
7.1.1.3. CAD Assemblies Tree Options

7.1.1.1. FMDB File

The FMDB file (*.fmdb) is a CAD neutral file that is created when the CAD Assemblies are created during CAD import. This file enables quick re-import of the CAD data with changes to the faceting qualities and/or topology representations.

- When importing a single file, the FMDB file is created in the folder containing the CAD file and has the same name as the file imported.
- When importing multiple files, the FMDB file is created in the folder containing the CAD files and is named `multiple.fmdb` by default. You can specify an appropriate name in the **FMDB Location** dialog box when prompted.

The menu at the global CAD Assemblies level enables you to obtain a list of the referenced FMDB files. Select **Referenced FMDB** in the menu to open the **Referenced FMDB files** dialog box containing the list of FMDB files.

7.1.1.2. CAD Entity Path

All CAD entities and labels in the CAD Assemblies tree are referred to by their path. The path is used in commands in the `cad-assemblies` menu (see `cad-assemblies/` for details).

To see the path, right-click the CAD entity in the tree and select **Tree** → **Show Path**.

Examples of the CAD entity path:

```
|assembly.agdb|component_1
|assembly.agdb|component_1|body_1
|assembly.agdb|body_1|label
```

7.1.1.3. CAD Assemblies Tree Options

The **Tree** menu contains options that control the appearance of the CAD Assemblies tree. These options can be used to select or deselect the CAD objects and zones in the tree, expand or collapse the tree branches, and also delete suppressed or locked CAD objects.

At the global **CAD Assemblies** level:

- The **Selection Helper** enables you to select or deselect CAD objects or labels in the tree based on the specified **Name Pattern**. You can choose **cad-objects**, **leaf-cad-objects**, **non-leaf-cad-objects**, or **label** in the **Filter** list.
- The **Select Next Level** option selects the CAD assemblies or objects at the next lower level.
- The **Collapse All** option collapses the tree to the level selected.
- The **Delete Locked/Suppressed** option deletes all locked or suppressed CAD assemblies or objects.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The **Restore Deleted** option restores previously deleted CAD assemblies or objects.

At the component level:

- The **Selection Helper** enables you to select CAD objects or labels in the tree based on the specified **Name Pattern**. You can choose **leaf-cad-objects**, **non-leaf-cad-objects**, or **label** in the **Filter** list.
- The **Select Next Level** option selects the CAD assemblies or objects at the next lower level.
- The **Expand All** option expands the tree to show all subsequent levels (CAD objects and labels, if applicable). Similarly, the **Collapse All** option collapses the tree to the level selected.
- The **Show Path** option prints the CAD entity path in the console.
- The **Suppress Other** option suppresses all other CAD entities at the same level.
- The **Select Associated Objects** selects geometry/mesh objects associated with the selected component.

At the body level:

- The **Show Path** option prints the CAD entity path in the console.
- The **Suppress Other** option suppresses all other CAD entities at the same level.
- The **Select Associated Objects** selects geometry/mesh objects associated with the selected body.

At the label level:

- The **Select All Parent** option selects all the parent CAD entities in the tree.
- The **Show Path** option prints the CAD entity path in the console.

7.1.2. Visualizing CAD Entities

CAD entities can be displayed using the options available for the CAD Assemblies tree:

- The global menu for the **CAD Assemblies** contains an option to draw all the CAD assemblies imported.
- The menu for individual components and bodies contains an option to draw the selected entity. You can also use the additional options in the **Draw Options** menu to add, remove, or highlight the selected entity in the display. You can also draw the unlabeled zones for the entity selected.
- The menu for labels contains an option to draw the selected label. You can also use the additional options in the **Draw Options** menu to draw all labels and add, remove, or highlight the selected label in the display. You can also draw the overlapping zones for the label selected.

The CAD entities are displayed using a different color palette to that used for the geometry/mesh objects. The zone and object selection filters also apply to the CAD zones and objects.

Other visualization tools and shortcuts are also applicable to the CAD entities. You can use the shortcut keys or onscreen tools to select visible entities, deselect all or the last selected entity,

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

hide/show entities. You can also use the **Isolate** tools to isolate selected entities in the display or limit the display to entities based on area/curvature of the selected entities.

7.1.3. Updating CAD Entities

The **Update** option enables reimporting the CAD entities using new parameters. Select the CAD objects in the tree and select **Update**. Alternatively, select the CAD objects in the graphics display and click to open the **Update** dialog box.

Tip:

Be sure to clear the tree (using **Ctrl**+**Y**) before selecting items to update.

- You can change the CAD zone granularity, if required.
- Select the **Tessellation** option and specify the tessellation controls:
  - If you select the **CAD Faceting** option, you need to specify the **Tolerance** for refinement and the **Max Size** in the **CAD Faceting Controls** group box. The default value for **Tolerance** is **0**, which implies no tessellation (faceting) refinement during import. The **Max Size** enables you to specify a maximum facet size for the imported model to avoid very large facets during the file import.
  - If you select the **CFD Surface Mesh** option, you need to specify the minimum and maximum facet sizes (**Min Size**, **Max Size**), and the curvature normal angle to be used for refining the surface mesh based on the underlying curve and surface curvature. You can optionally use the edge proximity size function for creating the surface mesh, based on the number of cells per gap specified. You can also choose to save a size-field file based on these defined parameters (that is, **Min Size**, **Max Size**, **Curvature Normal Angle**, **Cells Per Gap**). Note that for proximity size functions, the number of cells per gap can be a real value, with a minimum of 0.1 (see Proximity (p. 587) for more information).

    Alternatively, you can use a previously saved size-field file to create the surface mesh by enabling **Use Size Field File**.
- You can choose to import edge zones from the CAD entities. Specify an appropriate value for **Angle**.

7.1.4. Manipulating CAD Entities

CAD management operations are available from the tree menus as well as through the graphics icons and hot-keys.

7.1.4.1. Creating and Modifying Geometry/Mesh Objects
7.1.4.2. Managing Labels
7.1.4.3. Setting CAD Entity States
7.1.4.4. Modifying CAD Entities

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.1.4.1. Creating and Modifying Geometry/Mesh Objects

The **Object** menu contains options for creating geometry/mesh objects from the CAD entities and modifying existing objects. Select the CAD entities in the tree and then choose the appropriate menu option. Alternatively, select the CAD objects in the graphics display and select the option from the **CAD Tools**.

- Use the **Create** ( ) option to create a new geometry/mesh object. The **Create Object** dialog box contains options for creating a geometry or mesh object based on the tree selections. Specify the **Object Name**, **Object Type** (geom or mesh), and the **Cell Zone Type** (solid, fluid, or dead). In case of multiple selections, you can choose to create **One Object per CAD Object Selection**. You can also choose to retain the CAD zone granularity by enabling **Keep CAD Zones Granularity**.

  Note:

  The CAD association will be transferred to the objects when you choose to retain the CAD zone granularity for object creation.

- Use the **Add to** ( ) option to add the selected CAD entities to an existing object. The **Add to Object** dialog box contains a list of the current objects. Select the object to be modified and click **Add**.

  When CAD entities are added to an object, the zones are merged to create a single zone. If the CAD entities have labels defined:

  – Adding the entities to an object with no labels defined will result in the labeled zones retained as separate zones.

  – Adding the entities to an object with the same labels defined will result in the labels being merged, but retain the zones retained as separate zones.

- Use the **Replace** option ( ) to replace an existing object with the selected CAD entities. The **Replace Object** dialog box contains a list of the current objects. Select the object to be replaced and click **Replace**.

7.1.4.2. Managing Labels

You can set up labels for the CAD zones, if required. Named Selections are also imported as labels. These labels are preserved throughout the object-based meshing workflow in Fluent Meshing.

You can manage labels for CAD entities using the **Manage Labels** dialog box. Select the CAD zones in the display using the zone selection filter ( ) and then click **Manage Labels** ( ).

- Use the **Create/Add label** option to create a new label for the CAD zones selected and click **Add**. If **Remove CAD Zones from Graphics** is enabled (default), the selected CAD zones will be removed from the display after the label is created.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Use the **Create/Add label** option to add an existing label to the CAD zones selected. Select the labels from the list and click **Add**. If **Remove CAD Zones from Graphics** is enabled (default), the selected CAD zones will be removed from the display after the zones are added to the label.
- Use the **Remove label** option to remove an existing label from the CAD zones selected. Select the labels from the list and click **Remove**.

The **Modify** menu for labels contains options for deleting and renaming labels.

- Use the **Delete** option to delete the selected labels.
- Use the **Rename** option to rename the labels. Specify the name in the **Rename Labels** dialog box and click **OK**. In case of multiple selections, the specified name will be used, with a suitable index as suffix. For example, specifying a new label name **wall** will result in labels **wall.1**, **wall.2**, etc.

7.1.4.3. Setting CAD Entity States

The **State** menu for the CAD entities (component or body) contains options for setting the CAD entity state.

Locked

CAD entities are locked when corresponding geometry or mesh objects are created. Locked entities cannot be modified or used for creating objects. This prevents the use of the same CAD entity in multiple objects.

To unlock an entity, use the **State** → **Unlock** option.

Suppressed

CAD entities can be suppressed if they are not required for the analysis. You can transfer only necessary entities to geometry or mesh objects for meshing using the object-based workflow and suppress the remaining entities.

To suppress an entity, use the **State** → **Suppress** option. Alternatively, select the CAD objects and click .

To unsuppress an entity, use the **State** → **Unsuppress** option.

7.1.4.4. Modifying CAD Entities

The options in the **Modify** menu at the CAD entity (component or body) level enable modifying the CAD entities. Select the CAD entities in the tree and then choose the appropriate menu option. Alternatively, select the CAD objects in the graphics display and select the option from the **CAD Tools**.

- Use the **Extract Edge Zones** option ( ) to extract the feature edge zone for the selected entities. Specify the **Angle** in the **Extract Edge Zones** dialog box and click **Create**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Use the **Rename** ( ) option to rename the selected entities. Specify the name in the **Rename CAD Assemblies** dialog box (or the **Rename Entities** dialog box) and click **OK**. For multiple entities, the specified name will be used, with a suitable index as suffix. For example, specifying a new name **wall** will result in entities **wall.1**, **wall.2**, etc.

- Use the **Add Prefix** option to add a prefix to the selected entities. Specify the prefix in the **Prefix name** dialog box and click **OK**.

7.1.5. CAD Association

The **CAD Association** menu for geometry/mesh objects contains options for modifying the selected objects based on the associated CAD entities. You can also attach or detach the CAD entities from the objects.

- Use the **Update from CAD** option to update the geometry/mesh objects based on changes to the associated CAD objects.

- Use the **Unlock CAD** option to unlock the CAD objects associated with the selected geometry/mesh objects.

- Use the **Select CAD** option to select the CAD objects associated with the selected geometry/mesh objects.

- Use the **Detach CAD** option to detach the CAD objects associated with the selected geometry/mesh objects. All association will be removed and the geometry/mesh objects will be independent of changes to the CAD entities.

- Use the **Attach CAD** option to attach CAD objects to the selected geometry/mesh objects. Select CAD objects to be associated with the geometry/mesh objects in the tree and click **Confirm** in the **Attach CAD assemblies** dialog box. The selected geometry/mesh objects will be associated with the CAD objects which will then be locked.

- Use the **Restore CAD** option to restore the geometry/mesh object from the associated CAD objects.

7.2. Size Functions and Scoped Sizing

Size Functions and **Scoped Sizing** provide control over how the mesh size is distributed on a surface or within the volume. They provide accurate sizing information for the mesh distribution and precise refinement control.

Scoped sizing differs from size functions in how the sizing can be associated with objects or zones, respectively. Scoped sizing may be applied to model features such as faces, edges, face zone labels or unreferenced face or edge zones. You can optionally select the type of object (geom, mesh) while applying scoped sizing. Scoped sizing can be defined on individual zone or object entities by selecting from a list or using wildcards (*). For convenience, your scoped sizing definitions can also be saved to a file (`*.szcontrol`) which can be read in and reused for similar models having the same naming conventions.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The size field is computed based on the size functions and/or scoped sizing defined. You can remesh surfaces and edges based on the size field. The CutCell mesher also uses the size field to refine the initial Cartesian mesh.

Important:

Size functions can be computed only for triangulated zones. For zones comprising non-triangular elements, you can triangulate the zones manually before computing the size functions. Alternatively, you can use the command `triangulate-quad-faces?` before computing the size functions. This command identifies the zones comprising non-triangular elements and uses a triangulated copy of these zones for computing the size functions.

When the size functions or scoped sizing is used, the mesh distribution is influenced by

- The minimum and maximum size values
- The growth rate
- The size source which can be any one of the following:
  - Edge and face curvature, based on the normal angle variation between adjacent edges or faces.
  - Edge and face proximity, based on the number of element layers created in a gap between edges or faces.
  - The body of influence defined.
  - Constant user-defined sizes through hard and soft behaviors. The curvature, proximity, body of influence, and soft size functions have soft behavior. The meshed and hard size functions have hard behavior.

This chapter contains the following sections:

7.2.1. Types of Size Functions or Scoped Sizing Controls
7.2.2. Defining Size Functions
7.2.3. Defining Scoped Sizing Controls
7.2.4. Computing the Size Field
7.2.5. Using the Size Field

7.2.1. Types of Size Functions or Scoped Sizing Controls

The following size functions or scoped sizing controls are available:

7.2.1.1. Curvature
7.2.1.2. Proximity
7.2.1.3. Meshed
7.2.1.4. Hard
7.2.1.5. Soft
7.2.1.6. Body of Influence

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.2.1.1. Curvature

The curvature size function/scoped control computes edge and face sizes using their size and normal angle parameters, which are either automatically computed or defined.

The curvature sizing is defined by the following parameters:

- Min, Max size
- Growth rate
- Normal angle

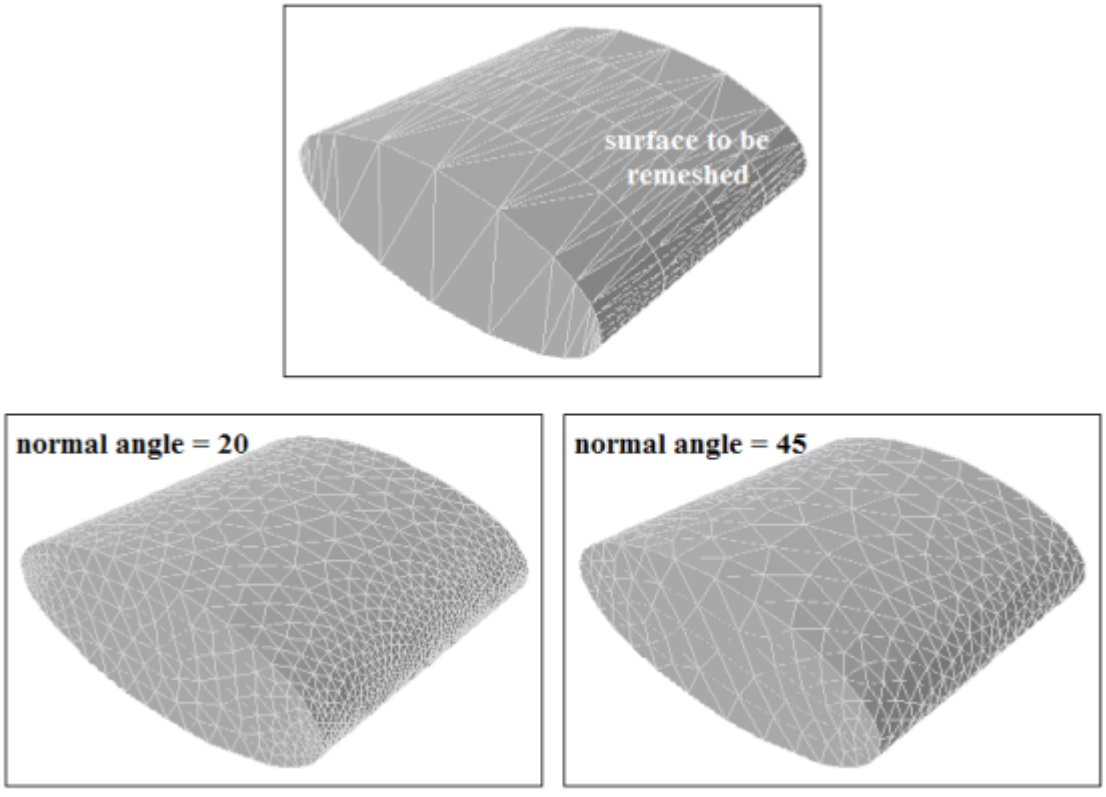
The curvature size function/scoped control uses the normal angle parameter as the maximum allowable angle that one element edge may span. For example a value of 5 implies that a division will be made when the angle change along the curve is 5 degrees; hence, a 90 degree arc will be divided into approximately 18 segments.

Note:

As the curvature values are computed approximately using edges and face facets, there may be some numerical errors, especially when face facets are excessively stretched.

Figure 7.1: Use of Curvature Sizing (p. 586) shows an example where the surface has been remeshed based on a curvature size function. The change in normal angle and growth rate controls the size distribution.

Figure 7.1: Use of Curvature Sizing



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.2.1.2. Proximity

The proximity size function/scoped control computes edge and face sizes in 'gaps' using the specified minimum number of element layers. For the purposes of specifying proximity sizing, a 'gap' is defined in one of two ways:

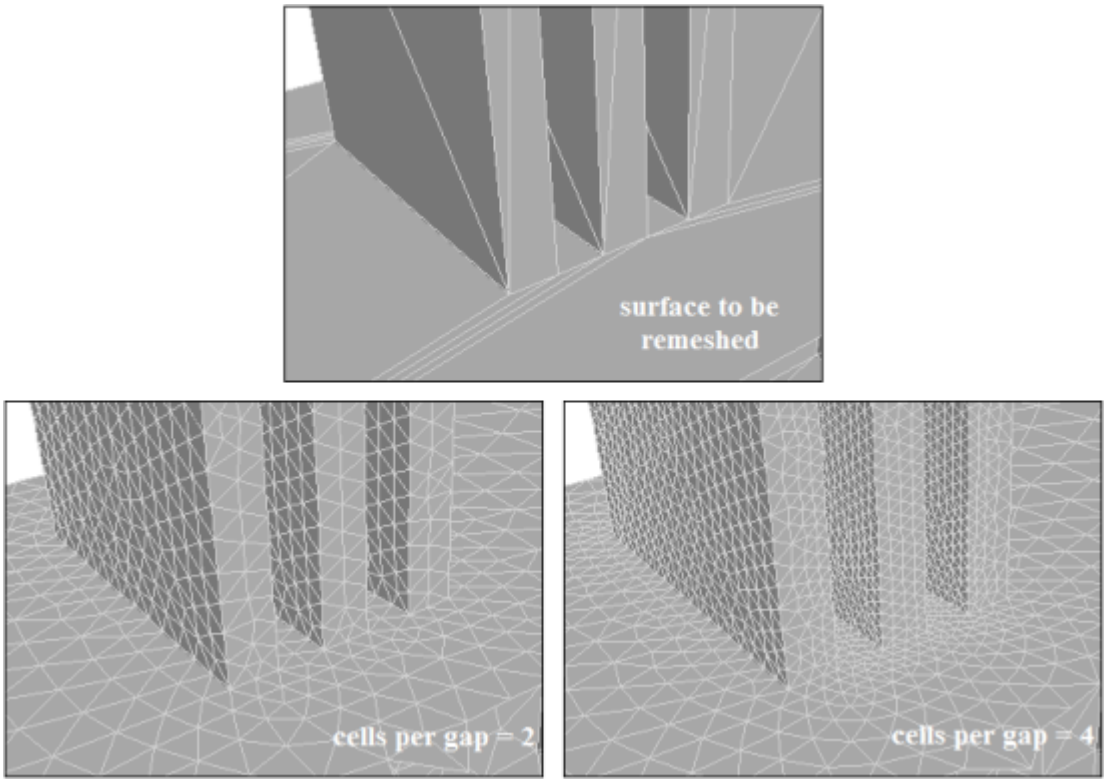
- The area between two opposing boundary edges of a face
- The internal volumetric region between two faces

The proximity sizing is defined by the following parameters:

- Min, Max size
- Growth rate
- Cells per gap

Figure 7.2: Use of Proximity Sizing (p. 587) shows an example where the surface has been remeshed based on a proximity size function. The change in the cells per gap and growth rate parameters control the size distribution.

Figure 7.2: Use of Proximity Sizing



Note:

For proximity size functions, the number of cells per gap can be a real value, with a minimum of 0.01. This has the effect of increasing the size of face elements located

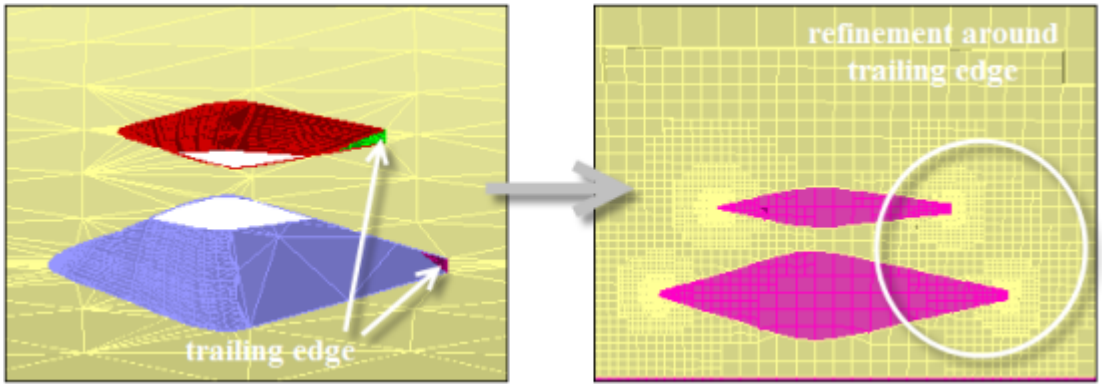
farther away from edges, and stretching face elements with larger sizes along side faces, or gaps, thereby reducing the overall face count, and ultimately the cell count.

Additional options for defining the face proximity sizing are as follows:

- The **Face Boundary** option enables you to compute the shell proximity (edge-edge proximity within each face). The proximity between feature edges on the face zones selected is computed. This option is particularly useful for resolving trailing edges and thin plates without using the hard size function.

  The example in Figure 7.3: Use of the Face Boundary Option for Face Proximity (p. 588) shows the use of this option for a blade configuration. Though the normals on the blade surface point outward, the cells across the trailing edges will be refined based on the proximity size function defined for the trailing surfaces when the **Face Boundary** option is enabled.

  Figure 7.3: Use of the Face Boundary Option for Face Proximity

  Note:

  The **Face Boundary** option works on the internally extracted boundary edge zones of the face zones. Edge zones extracted in the meshing mode/CAD imported edges will not be considered for the proximity calculation.

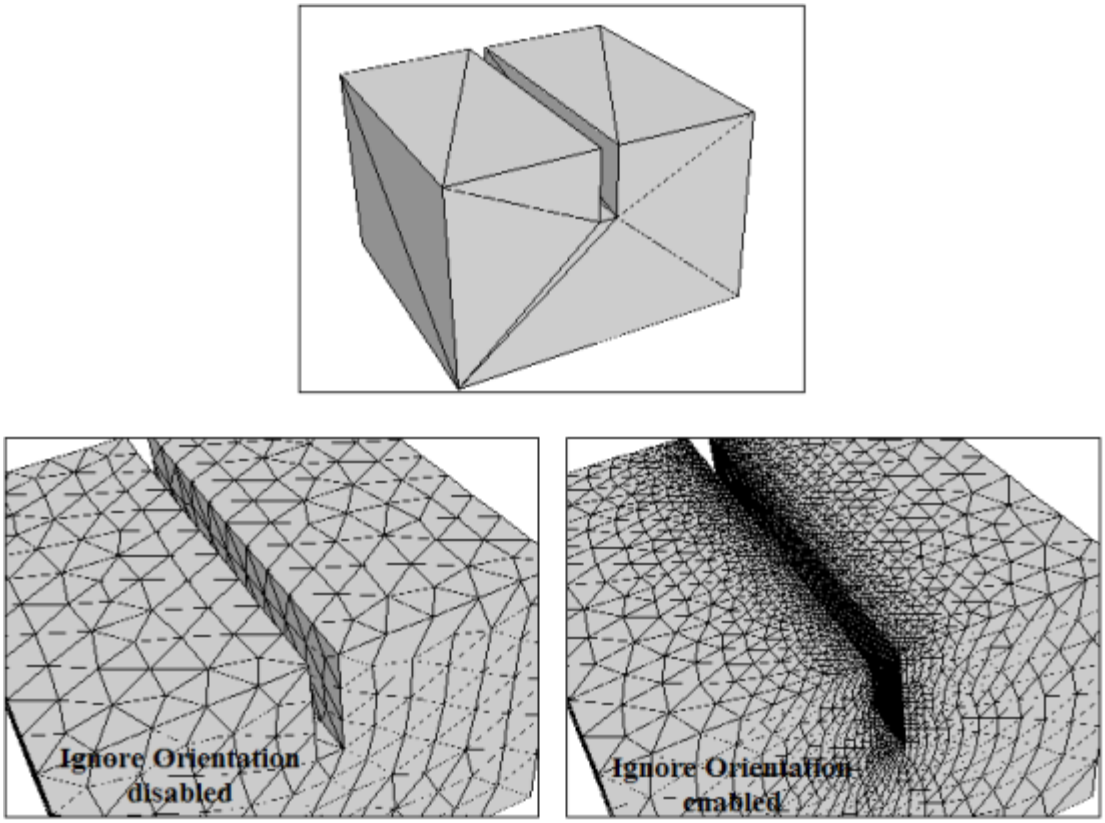
- The **Face - Face** option enables you to compute the proximity between two faces in the face zones selected. When the **Face - Face** option is enabled, additional options for ignoring self proximity (**Ignore Self**) and ignoring the face normal orientation (**Ignore Orientation**) are also available.

  The **Ignore Self** option can be used with the **Face - Face** option in cases where self proximity (proximity between faces in the same face zone) is to be ignored. This option is disabled by default.

  The **Ignore Orientation** option can be used to ignore the face normal orientation during the proximity calculation. This option is enabled by default. In general, the proximity depends on the direction of face normals. An example is shown in Figure 7.4: Use of the Ignore Orientation Option for Face Proximity (p. 589). The normals on the grooved box point inward. With only the **Face - Face** option, the proximity size function does not refine the surface along the entire groove length. When the **Ignore Orientation** option is enabled along with the **Face - Face** option, the surface will be refined along the groove length.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.4: Use of the Ignore Orientation Option for Face Proximity



Note:

You must select at least one of the **Face Boundary** and **Face - Face** options; otherwise, an error will be reported.

The edge proximity size function depends only on the distance between the edges, irrespective of their association with a face zone or the orientation of the face zones associated with the edge zones.

Important:

When using a proximity sizing in certain geometries (with angle > 30 degrees or comprising extended region of non-intersecting faces), the proximal face zones may not be detected and may result in a warning message. In such cases, split the proximity sizing into multiple proximity scoped sizing controls.

Tip:

When using a proximity size function with geometries having a very large number of small feature edges, you can speed up the calculation and reduce memory requirements by enabling **Quick Edge Proximity**. Go to **Display** → **Controls**, select **Size Functions**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

from the **Categories** drop down list, and check **Quick Edge Proximity**. Accuracy will be reduced when **Quick Edge Proximity** is enabled.

7.2.1.3. Meshed

The meshed size function/scoped control enables you to set the size based on existing sizes. This provides gradation between the minimum and maximum size based on the specified growth rate.

The default behavior for the meshed size function/scoped control is soft, allowing other hard sizes or locally smaller sizes to override it. You can enable **Hard Meshed Size Functions** by going to the **Display > Controls** menu, and selecting **Size Functions** in the drop down list.

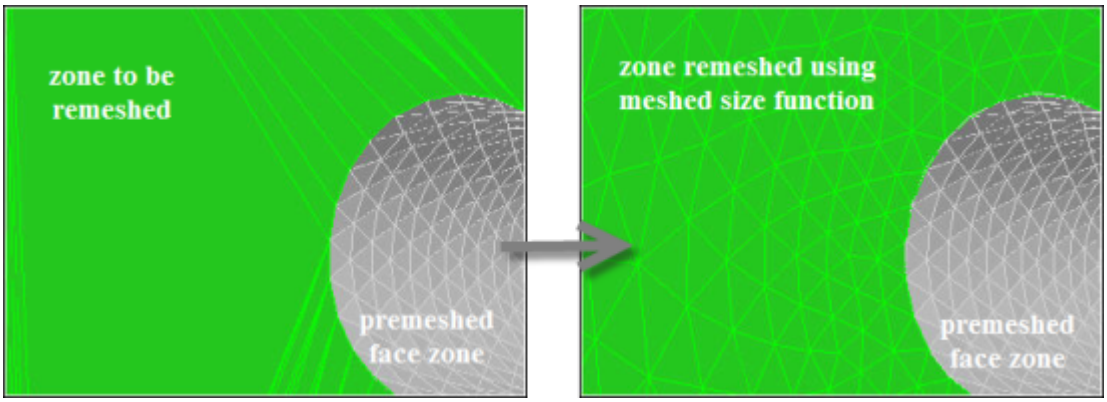
Note:

In Release 15.0 and earlier, the behavior for the meshed size function/scoped control was hard. This behavior is saved with the mesh file. Therefore, when reading a mesh saved using such a release, the **Hard Meshed Size Functions** control may be enabled.

The meshed sizing is defined by the growth rate.

In Figure 7.5: Use of Meshed Sizing (p. 590), the face zone is remeshed based on the premeshed face zone indicated.

Figure 7.5: Use of Meshed Sizing



7.2.1.4. Hard

The hard size function/scoped control enables you to maintain a uniform size based on the size specified, while the growth rate from the defined size influences the size on adjacent zones. The hard sizing will override any other size function specified.

The hard sizing is defined by the following parameters:

- Min Size

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Growth rate

Important:

- It is recommended to not have two hard sizes next to each other as the mesh size transition between the two will not be smooth.
- If two hard sizes are applied at the same location, the latter will be honored. The smaller size rule does not apply in this case.

7.2.1.5. Soft

The soft size function/scoped control enables you to set the maximum size on the selected zone, while the specified growth rate from the defined size influences the size on adjacent zones. When the **soft** sizing is selected for edges and/or faces, the size will be affected by other size functions/scoped controls. The minimum size on the zone will be determined based on the influence of other size functions/scoped controls, else a uniform size will be maintained. In other words, a soft sizing is ignored in a region where other sizing controls specify smaller sizes.

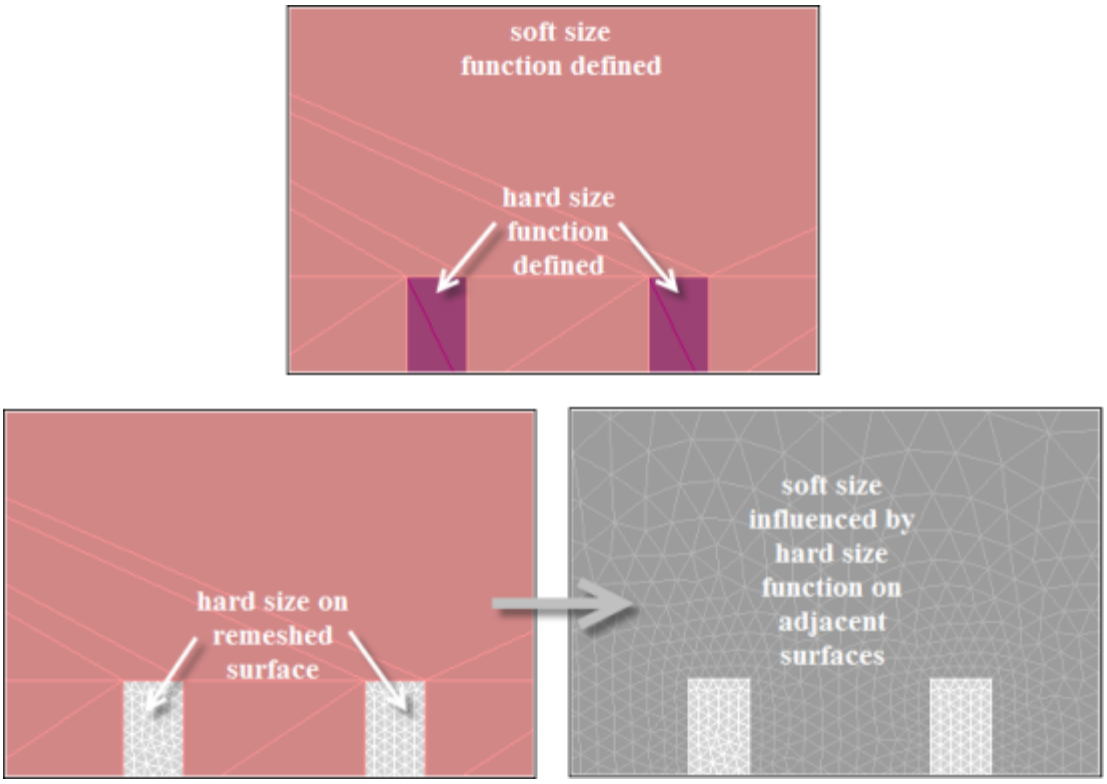
The soft sizing is defined by the following parameters:

- Max size
- Growth rate

In the example in Figure 7.6: Use of Soft Sizing (p. 592), the minimum size is determined by the hard sizing applied on the smaller face zones indicated, and maximum size is limited by the soft sizing applied.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.6: Use of Soft Sizing



7.2.1.6. Body of Influence

The body of influence size function/scoped control enables you to specify a body of influence (that is, a region for sizing control). The maximum mesh size will be equal to the specified size within the body of influence. The minimum size will be determined based on the influence of other size functions/scoped controls. An example is shown in Figure 7.7: Use of Body of Influence Sizing (p. 593).

Note:

The set of face zones selected to define the body of influence should constitute a geometrically closed region. If an open region is used as a body of influence, the sizing will be processed as a soft sizing.

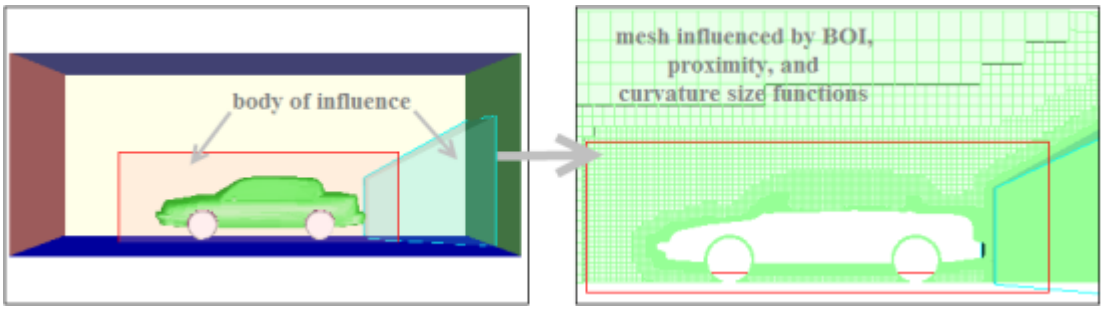
The body of influence sizing is defined by the following parameters:

- Max Size
- Growth rate

In Figure 7.7: Use of Body of Influence Sizing (p. 593), the mesh is generated based on the body of influence sizing defined. The finer mesh size is obtained due to other size functions (for example, curvature, proximity) defined in addition to the body of influence size functions.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.7: Use of Body of Influence Sizing



Important:

- In case of multiple non-intersecting closed bodies, a single BOI size function/scoped sizing control can be set up. The sizing can be scoped to the set of face zones comprising the respective bodies.
- In case of multiple intersecting closed bodies for scoping the same BOI sizing, you need to create separate BOI size functions/scoped sizing controls for each geometric body.

Note:

When using BOIs with periodic boundaries, if the BOI extends outside the domain, it may cause unnecessary refinement.

7.2.2. Defining Size Functions

Size functions can be defined using the **Size Functions** dialog box. Right-click on **Model** in the tree and select **Functions...** from the **Sizing** menu. The generic procedure to define size functions is as follows:

1. Ensure that the global controls are set as required. The relevant size function parameter values (minimum and maximum size, growth rate) will be updated based on the global controls specified.
2. Enable **Face Zones** (or **Edge Zones**) as appropriate.
3. Select the boundary zones (or the edge zones) for which the size function is to be defined in the **Boundary Zones** (or **Edge Zones**) selection list.

   Note:

   All boundary face zones and edge zones included in the global domain are available for defining size functions, even if a local domain has been activated.

4. Select the appropriate size function type in the **Size Function Type** drop-down list in the **Define Size Function** group box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Enter an appropriate size function name in the **Name** field or leave the field blank if you want to have the name generated automatically. In this case, the **Name** will be assigned according to the zone type (face or edge) and the size function type. (For example, **face-curvature-sf-5** indicates that the curvature size function is defined for face zones. The size function ID is 5.)

6. Specify the size function parameters applicable for the selected size function as appropriate and click **Create**.

   The defined size function will be available in the **Size Functions** list.

7.2.2.1. Creating Default Size Functions

You can create default size functions based on face and edge curvature and proximity using the **Create Defaults** option in the **Size Functions** dialog box. Alternatively, you can use the command `/size-functions/create-defaults` to create the default size functions.

The following size functions will be defined:

- Curvature size function on all edge zones, with the global minimum and maximum sizes and growth rate, and a normal angle of **18**.
- Curvature size function on all face zones, with the global minimum and maximum sizes and growth rate, and a normal angle of **18**.
- Proximity size function on all edge zones, with the global minimum and maximum sizes and growth rate, and the cells per gap set to **3**.
- Proximity size function on all face zones, with the global minimum and maximum sizes and growth rate, and the cells per gap set to **3**.

When the **Create Defaults** option is used after the default size functions have been created, the previous definitions will be updated based on any changes to the global minimum and maximum sizes and growth rate.

7.2.3. Defining Scoped Sizing Controls

Scoped sizing controls can be defined using the **Scoped Sizing** dialog box. Right-click on **Model** in the tree and select **Scoped...** from the **Sizing** menu. The generic procedure for defining scoped sizing controls is as follows:

1. Set the global scoped sizing controls as required. The relevant local control parameter values (minimum and maximum size, growth rate) will be updated based on the global values specified.

2. Select the appropriate scoped control type in the **Type** drop-down list in the **Local Scoped Sizing** group box.

3. Enter an appropriate name in the **Name** field.

4. Specify the local sizing parameters applicable for the selected scoped control type as appropriate.

5. Select the scope for the scoped control defined from the **Scope To** drop-down list. Additionally, select the **Object Type** (**Geom** and/or **Mesh**) and specify a suitable pattern in the **Selections** field.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also click to open the **Scope** dialog box and select the objects, face zone labels, face zones, or edge zones for defining the size control scope.

6. Click **Create New**.

   The defined size control will be available in the **Controls** list.

You can validate the defined scoped sizing controls using the command `/scoped-sizing/validate`. An error will be reported if the scoped sizing controls do not exist or the scope for one (or more) controls is invalid.

7.2.3.1. Size Control Files

The size controls file (`*.szcontrol`) contains the scoped sizing control definitions. The control name and type, and the scope of the control will be included in the size control file along with global size parameters.

To read a size controls file, click **Read...** in the **Scoped Sizing** dialog box to invoke the **Select File** dialog box and specify the name of the file to be read. Alternatively, you can use the `/scoped-sizing/read` command and specify the name of the file to be read.

To write a size controls file, click **Write...** in the **Scoped Sizing** dialog box to invoke the **Select File** dialog box and specify the name of the file to be written. Alternatively, you can use the `/scoped-sizing/write` command and specify the name of the file to be written.

7.2.4. Computing the Size Field

The size field can be computed based on the defined size functions and scoped sizing controls by clicking **Compute** in the **Size Functions** dialog box or the **Scoped Sizing** dialog box. Alternatively, use the command `/size-functions/compute` or `/scoped-sizing/compute` to compute the size field.

Important:

If the size field has been computed in the current session, sizes will be based on the computed size field. You cannot define additional size functions or scoped sizing controls, or modify the current sizes without deleting the size field.

7.2.4.1. Size Field Files

Size field files contain the size function definitions based on the parameters specified.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select the **File/Read/Size Field...** menu to read a size field file. This will invoke the **Select File** dialog box, where you can specify the name of the size field file to be read. Alternatively, you can use the `/file/read-size-field` command and specify the name of the file to be read.

Important:

If a size field file has been read in the current session, sizes will be based on the size field read. You cannot define additional size functions or scoped sizing controls, or modify the current sizes without deleting the size field.

Select the **File/Write/Size Field...** menu to save a size field file based on the parameters set. This will invoke the **Select File** dialog box, where you can specify the name of the size field file to be written. Alternatively, you can use the `/file/write-size-field` command and specify the name of the file to be written.

7.2.4.2. Using Size Field Filters

Additional size field filtering options are available after the size field is computed/read. These options are available in the **Size Field Filters** dialog box.

Click **Filters...** in the **Size Functions** or **Scoped Sizing** dialog box to open the **Size Field Filters** dialog box.

- You can specify a scale factor to filter the size output from the size field, without deleting and recomputing the size field. The scaling filter can be applied as follows:

  1. Specify an appropriate value for **Factor**, **Min** and **Max** (for **Scale**).
  2. Click **Apply**.

  Alternatively, you can use the command `/size-functions/set-scaling-filter`, and specify the scale factor, minimum and maximum size values.

- You can apply periodicity to the size field, without deleting and recomputing the size field as follows:

  1. Click **Set...** to open the **Periodicity** dialog box.
  2. Select the type of periodicity (**Rotational** or **Translational**).
  3. For translational periodicity, specify the shift vector components You can also click **Define** and select 2 nodes to set up the shift vector for translational periodicity.
  4. For rotational periodicity, enter an appropriate value for **Angle**.
  5. For rotational periodicity, specify the pivot point and axis of rotational periodicity.

     You can also select 1–6 nodes and click **Define** to set up the pivot point and axis of rotation for rotational periodicity as follows:

     – If only 1 node is selected, the pivot point is at the node location and the axis of rotation is the global z-axis.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For 2 nodes, the pivot point is at the midpoint of the nodes selected and the axis of rotation is the global z-axis.
- For 3 nodes, the pivot point is at the first node selected. The axis of rotation is the local z-axis normal to the plane defined by the three points, the positive direction is determined by the right-hand rule.
- For 4, 5 or 6 nodes, the first 3 points define a circle. The pivot point is at the center of the circle. The axis of rotation is the local z-axis normal to the circular plane, the positive direction is determined by the right-hand rule.

6. Click **Apply** to enable **Periodicity** in the **Size Field Filters** dialog box.

Note:

- If periodicity is set up prior to computing the size-field, the **Periodicity** filter will be enabled when the size-field is computed.

Alternatively, you can set up the periodic boundary using the **Make Periodic Boundaries** dialog box. See Creating Periodic Boundaries (p. 700) for details.

Boundary → Create → Periodic...

You can also use the command `/size-functions/enable-periodicity-filter` and specify the angle, pivot, and axis of rotation. If periodicity has been previously defined, the existing settings will be applied.

7.2.4.3. Visualizing Sizes

Before computing the size field, you can check if the global minimum and maximum sizes are suitable locally. After computing the size field, you can set the selection filter to **size** and use the probe to determine local size or make a contour plot of the sizing on any surface

You can display the contours of size using the options in the **Size Functions** dialog box as follows:

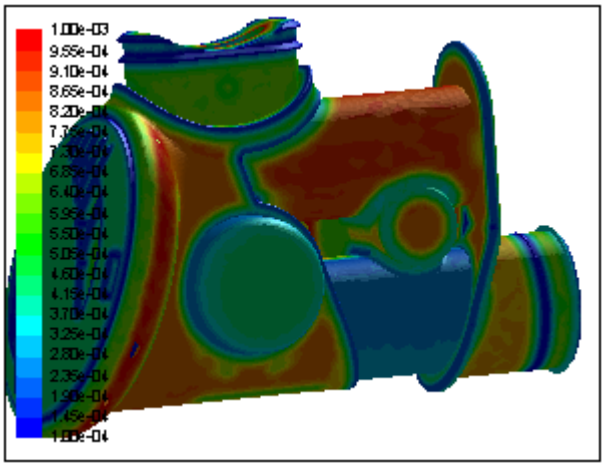
1. Select the face zones in the **Boundary Zones** selection list in the **Size Functions** dialog box.
2. Specify appropriate values for **Min** and **Max** in the **Contours** group box.
3. Click **Draw** (in the **Contours** group box).

Alternatively, after selecting the face zones, use the hot key **Ctrl**+**T** (Miscellaneous Tools), then **Ctrl**+**C** to draw size contours.

Figure 7.8: Contours of Size (p. 598) shows the display of contours of size on the selected face zones after the size field has been computed or read

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.8: Contours of Size

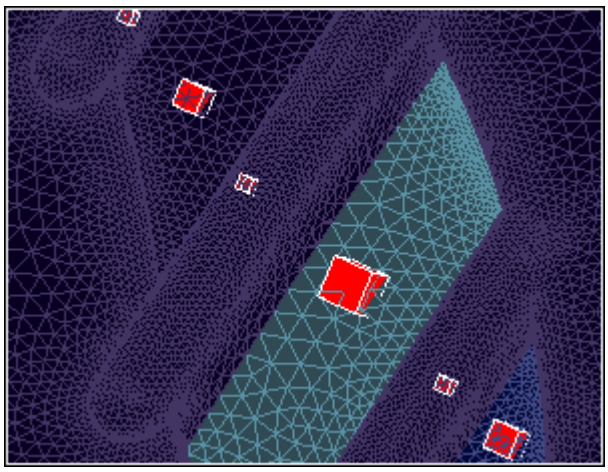


A visual indication of mesh size is also available using the mouse probe.

Size selection filter

If the selection filter is set to **size** (hot key **Ctrl**+**Y**), right-click at the required locations to see the size boxes indicating the mesh size. See Figure 7.9: Display of Mesh Size Based on Size Field (p. 598).

Figure 7.9: Display of Mesh Size Based on Size Field



Preview sizes

Alternatively, after selecting the face zones, use the hot key **Ctrl**+**T** (Miscellaneous Tools), then **Ctrl**+**P** to preview sizes on the selected zones. You can set the minimum and maximum size values in the **Preview Sizes** dialog box.

7.2.5. Using the Size Field

The size field can be used to remesh surfaces and edges. The CutCell mesher also uses the size field to refine the initial Cartesian mesh.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Remeshing Surfaces

The generic procedure for remeshing surfaces is as follows:

1. Select the surface(s) to be remeshed. You can use the graphics select tools or select from a list in a dialog box.
2. Use the hotkey **Ctrl**+**Shift**+**R** or click the Remesh button to open the **Zone Remesh** dialog box.
3. Choose the appropriate **Sizing** from the drop down list and set other parameters as necessary.
4. Click **Remesh**.

For more control of surface remeshing parameters, use the **Surface Retriangulation** dialog box as follows:

Boundary → **Mesh** → **Remesh**

1. Select the surfaces to be remeshed in the **Face Zones** list.
2. Enable **Size Function** in the **Face Remesh Options** group box and click the **Specify** button to open the **Size Functions** dialog box and check that the size functions are appropriate. See Defining Size Functions (p. 593).

   Alternatively, ensure that the size field has been computed or read in.
3. Set the other options for face remeshing as appropriate.
4. Click **Remesh**.

Important:

Edge zones associated with face zones are not remeshed implicitly. If you have feature edge zones associated with the surface being remeshed, you need to remesh them before remeshing the face zones.

Remeshing Edges

The generic procedure for remeshing edges is similar to the above procedure for surfaces. Use the hotkey **Ctrl**+**Shift**+**Z**, or click the Edge Zone restriction button, to constrain the selection to edge zones only.

For more control of edge remeshing parameters, use the **Feature Modify** dialog box as follows:

Boundary → **Mesh** → **Feature**

1. Ensure that the edge zones are extracted as required.
2. Select the edges to be remeshed in the **Edge Zones** list in the **Feature Modify** dialog box.
3. Select **Remesh** in the **Options** list and select **Size Function** in the **Method** drop-down list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Make sure the size functions are defined as appropriate in the **Size Functions** dialog box (see Defining Size Functions (p. 593)). Alternatively, ensure that the size field has been computed or read in.

5. Click **Apply**.

Refining the CutCell Mesh

The CutCell mesher uses the size functions/size field to refine the initial Cartesian mesh as described in The CutCell Meshing Process (p. 811).

7.3. Objects and Material Points

This chapter describes the use of objects and material points for identifying the mesh region.

7.3.1. Objects

7.3.2. Material Points

7.3.1. Objects

An object—generally a set of face zones and edge zones—is used to identify the domain to be meshed. By including edge zones in the object definition, you are able to capture the object features even when wrapping.

Objects are generally closed solid volumes, closed fluid (wetted) volumes, capping surfaces, or individual face zones that can be used for meshing. For example, using capping surfaces in conjunction with a material point and a closed solid volume enables you to extract the flow volume using wrapping.

Objects (defined or imported) are independent of each other; that is, objects do not share face and/or edge zones. In cases where objects are defined using a common face/edge zone, the common face/edge zones are duplicated to make the objects independent.

Object Based Meshing

Objects can be used for mesh generation as described:

Surface Meshing

Object wrapping or join and intersect operations are used to create a conformal, connected surface mesh. This is the first of a two-step process for creating a tetrahedral, hexcore, polyhedral, or hybrid mesh in meshing objects. This process is described in Object-Based Surface Meshing (p. 616).

Volume Meshing

The **Volumetric Regions** are calculated from the conformal, connected surface mesh, and then filled with tetrahedral, hexcore, or polyhedral mesh, with or without inflation layers. This is the second of a two-step process for meshing objects and is described in Object-Based Volume Meshing (p. 642)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Auto Mesh

An object-based workflow to generate a volume mesh. You can select the desired mesh object in the Model tree and select **Auto Mesh...** from the menu available. Alternatively, use the **Mesh** → **AutoMesh...** menu to open the **Auto Mesh** dialog box, then select the mesh object in the **Object** drop-down list.

See Using the Auto Mesh Dialog Box (p. 731) or Meshing All Regions Collectively Using Auto Mesh (p. 648) for details on using the **Auto Mesh** dialog box.

Note:

The **No Fill** option is not available when a mesh object is selected for volume meshing.

Auto Fill Volume

A selective workflow to generate a volume mesh on selected volumetric regions. You can select the desired **Volumetric Regions** in the Model tree and select **Auto Fill Volume** from the menu available. See Meshing Regions Selectively Using Auto Fill Volume (p. 650).

CutCell Meshing

A general purpose, hex-dominant meshing technique using a direct surface and volume approach. Objects can be used to determine the inclusion of Cartesian grid entities for CutCell meshing. The Cartesian grid will be refined and snapped to any face zone included in the objects selected for CutCell meshing. See Generating the CutCell Mesh (p. 810).

Objects descriptions include **attributes** (type, cell zone type and priority) and **entities** (face zone labels, volumetric regions and cell zones).

You can manage your objects using several tools accessible with hotkeys, onscreen tool buttons, text commands, or by using tools in the **Manage Objects** dialog box. Tools exist for operations such as object creation, modification, changing properties, alignment, remeshing, deleting, merging, and moving.

7.3.1.1. Object Attributes

7.3.1.2. Object Entities

7.3.1.3. Managing Objects

7.3.1.1. Object Attributes

Each object has attributes such as object type, cell zone type, and priority.

Object Type

The following object types are available:

Geometry

Objects imported through CAD using the CAD Faceting option (see Importing CAD Files (p. 264)) or created for a given geometry. The geometry objects may be non-conformal.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mesh

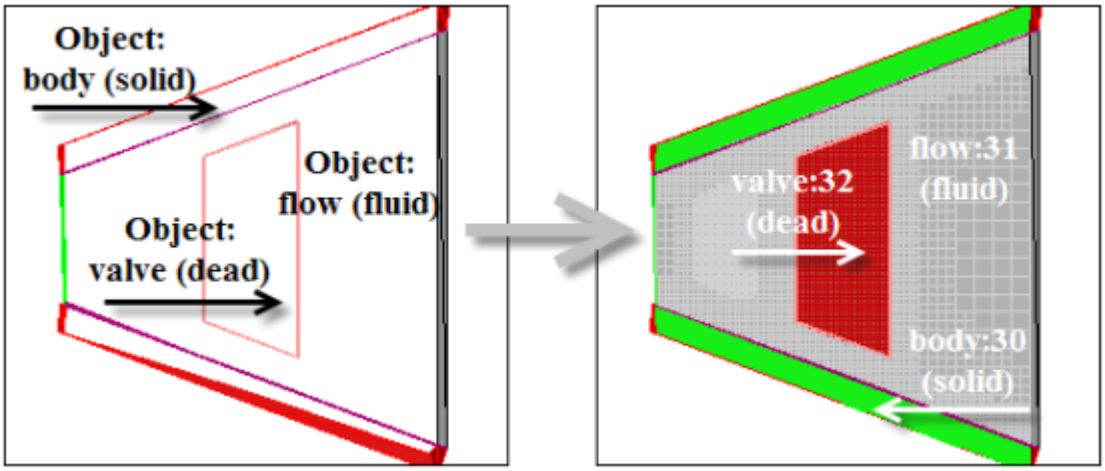
Objects that are good quality surface mesh representations of the geometry. Mesh objects may contain multiple volumes with shared faces. They may be imported using the **CFD Surface Mesh** option for CAD import or created using the object wrapping options. Mesh objects can also be created using the Sew operation.

Cell Zone Type

The object cell zone type indicates the type of cell zone created when the mesh is generated based on objects.

Figure 7.10: Mesh With Different Cell Zone Types (p. 602) shows a CutCell mesh with different cell zone type assigned to respective objects.

Figure 7.10: Mesh With Different Cell Zone Types



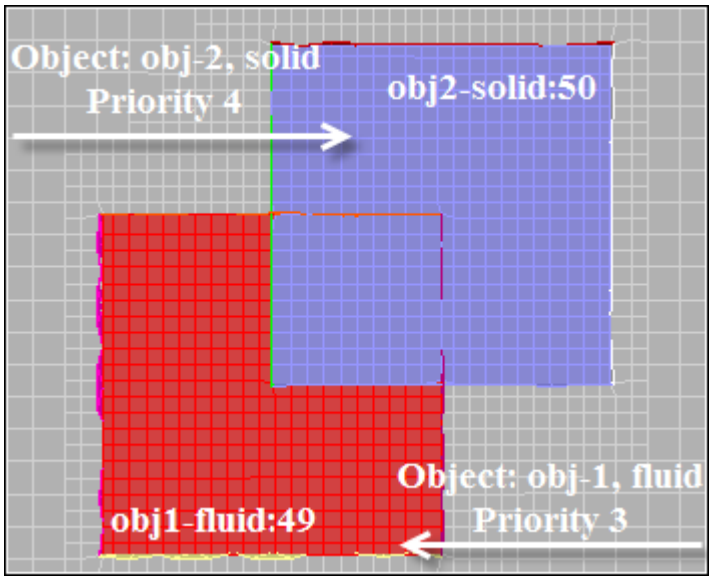
Priority

The object priority controls the inclusion of the mesh entities. In case of overlapping objects, the entities in the overlapped region will be included with the object having a higher priority value.

Figure 7.11: Use of the Object Priority for Overlapping Objects (p. 603) shows an example with overlapping objects. The overlapped region is included with the zone corresponding to the object having the higher priority value (in this case, 4).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.11: Use of the Object Priority for Overlapping Objects



Note:

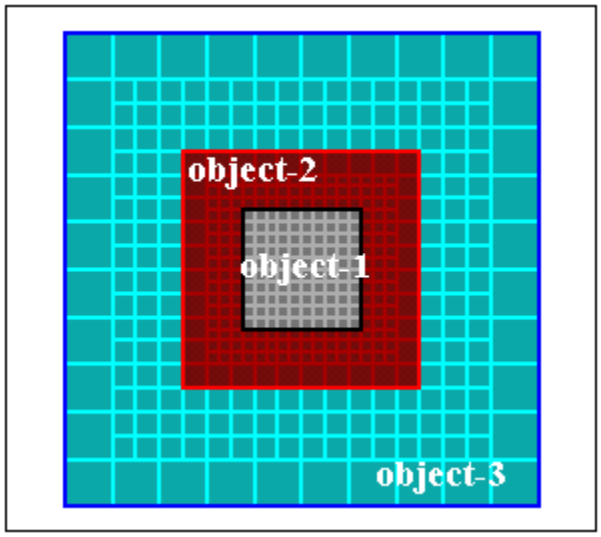
Multiple objects having the same priority assigned will be merged into a single cell zone, irrespective of cell zone type.

Priority is also important when objects are created for bounding boxes or wind tunnels, and so on. In such cases, the object created for a bounding box or wind tunnel must be assigned the lowest priority.

7.3.1.1.1. Creating Objects

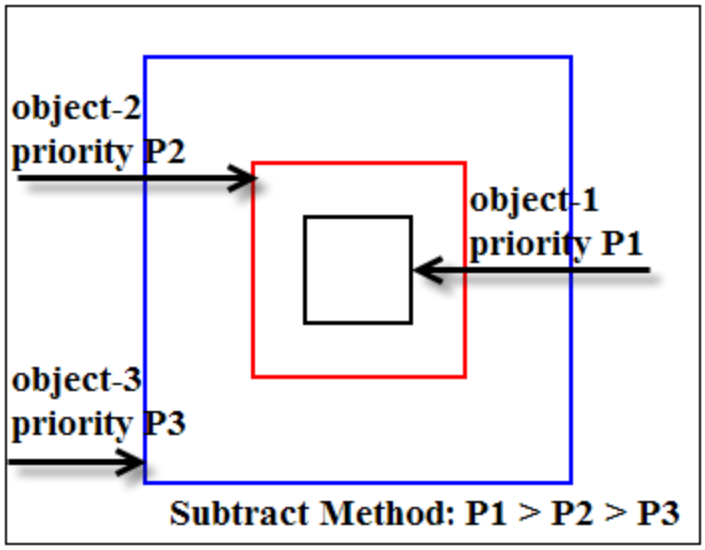
Figure 7.12: Creating Objects—Example (p. 603) shows the Subtract method for creating objects, using an example with three non-intersecting bodies.

Figure 7.12: Creating Objects—Example



The order of priority assigned to individual objects is important when using the Subtract method. Zones in higher priority objects will be ignored when defining lower priority objects. In this case the order of priority is object-1 > object-2 > object-3.

Figure 7.13: Objects Defined Using the Subtract Method



1. Select the face and edge zones comprising the innermost body, set the **Cell Zone** type, and set the highest **Priority** value.
2. Select the face and edge zones comprising the inner body, set the **Cell Zone** type, and set an intermediate **Priority** value.
3. Select the face and edge zones comprising the outer body, set the **Cell Zone** type, and set the lowest **Priority** value.

Note:

Objects (defined or imported) are independent of each other; that is, objects do not share face and/or edge zones. In cases where objects are defined using a common face/edge zone, the common face/edge zones are duplicated to make the objects independent.

7.3.1.2. Object Entities

Object entities include face zone labels, volumetric regions, and cell zones. Volumetric regions and cell zones are available only for mesh objects.

- Face zone labels: For geometry objects, these are groups of face zones comprising the object. For mesh objects, these are original CAD zones or bodies, or face zones comprising the mesh object. If the mesh object is created by merging multiple mesh objects, the face zone labels represent the objects that were merged. They provide the link to the original geometry.

  When the **CAD Assemblies** option is selected for CAD import, you can set up labels for the CAD zones, if required. Named Selections are also imported as labels. These labels are preserved throughout the object-based meshing workflow in Fluent Meshing.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Several tools for managing Face Zone Labels are available via the context-sensitive menus.

Note:

Geometry objects can contain unlabeled zones. Mesh objects always contain face zone labels which are derived from the associated geometry objects. In case of unlabeled zones in the geometry object, the mesh object face zone label is the same as the associated geometry object.

- Volumetric regions: These are finite, contiguous domains that are ready for volume meshing.
- Cell zones: These are created when the volume mesh is generated.

7.3.1.2.1. Using Face Zone Labels

Various options are available for face zone labels via the context-sensitive menus in the tree:

- The context-sensitive menu for **Face Zone Labels** contains options for drawing and selecting all labels and obtaining an overall summary or detailed information about the face zone labels.

  For **Geometry Objects**, you have these additional options:

  – Unlabeled face zones can be labeled using the **Create Labels...** option.

    The **Create Labels** dialog box opens. Select the face zones from the **Face Zones** selection list and specify an appropriate **Label Name**. For geometry objects, you can choose to list all available object face zones or only unlabeled object face zones for selection. Enable the **Create a label for each face zone** checkbox to generate an individual label for each face zone.

  – Use the **Remove All Labels from Zones...** to remove labeled face zones from an object.

    The **Remove All Labels** dialog box opens. Select the Object and Labeled Face Zones, and then click **Remove All**.

    Note:

    Unlabeled face zones are not supported for Mesh objects.

  – Unlabeled zones can be drawn and/or selected without having to create a new label using the **Unlabeled Zones...** options.

  For **Mesh Objects**, you have the additional **Join/Intersect...** option which opens a dialog box to assist with creating a conformal surface mesh.

- The context-sensitive menu for **individual labels** contains options for renaming, merging, and deleting face zone labels in addition to the standard options for drawing and selecting all labels, and obtaining an overall summary or detailed information.

  You can also add or remove zones from an existing face zone label.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– Select **Add Zones...** from the context sensitive menu to open the **Add Zones to Label** dialog box. Select the face zones and click **Add**. For geometry objects, you can add unlabeled zones or those already included in other labels within the object selected.

– Select **Remove Zones...** from the context sensitive menu to open the **Remove Zones from Label** dialog box. Select the face zones and click **Remove**.

For **Mesh Objects**, the context-sensitive menu for individual labels contains additional options for:

– connecting the face zones using **Join/Intersect...**.

– finding and repairing face connectivity and quality problems using **Diagnostics...**.

– creating or recovering periodic boundaries using **Recover Periodic...**.

7.3.1.3. Managing Objects

You can manage your objects using several tools accessible with hotkeys, onscreen tool buttons, text commands, or by using tools in the **Manage Objects** dialog box.

Tools exist for operations such as object creation, modification, changing properties, alignment, remeshing, deleting, merging, and moving.

7.3.1.3.1. Using hotkeys and onscreen tools

7.3.1.3.2. Using the Manage Objects Dialog Box

7.3.1.3.1. Using hotkeys and onscreen tools

Tools for the most common object management operations are accessible via hotkeys, icons, or text commands.

7.3.1.3.1.1. Creating Objects for CAD Entities

7.3.1.3.1.2. Creating Objects for Unreferenced Zones

7.3.1.3.1.3. Creating Multiple Objects

7.3.1.3.1.4. Easy Object Creation and Modification

7.3.1.3.1.5. Changing Object Properties

7.3.1.3.1.6. Automatic Alignment of Objects

7.3.1.3.1.7. Remeshing Geometry Objects

7.3.1.3.1.8. Creating Edge Zones

7.3.1.3.1.1. Creating Objects for CAD Entities

When the **CAD Assemblies** option is selected for CAD import, the CAD entities can be used to create or modify geometry/mesh objects. The objects are associated with the CAD entities (which are then locked), enabling re-importing or updating of selected parts or bodies using different faceting qualities and topology representations for quick design changes and updates.

- Use the options in the **Object** menu for CAD entities for creating geometry/mesh objects and modifying existing objects. Select the CAD entities in the tree and then choose the appropriate menu option. Alternatively, select the CAD objects in the graphics display and select

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the option from the **CAD Tools**. See Creating and Modifying Geometry/Mesh Objects (p. 582) for details.

- Use the options in the **CAD Association** menu for geometry/mesh objects for modifying the selected objects based on the associated CAD entities. You can also attach or detach the CAD entities from the objects. See CAD Association (p. 584) for details.

7.3.1.3.1.2. Creating Objects for Unreferenced Zones

Use the options in the **Create Objects** dialog box to define objects for unreferenced boundary face zones and edge zones. Right click **Boundary Face Zones** in the **Unreferenced** branch of the tree and select **Create New Objects...** from the context-sensitive menu.

1. Enter an appropriate name in the **Object Name** field.

   You can also have the object name generated automatically by leaving the **Object Name** field blank. In this case, the object name will be assigned based on the **Prefix**, cell zone type, and priority specified (for example, an object named **object-fluid:3-20** has prefix **object-**, cell zone type **fluid**, priority **3**, and object ID **20**).

2. Select the appropriate option from the **Cell Zone Type** drop-down list.

3. Set the priority.

4. Select the appropriate type from the **Object Type** drop-down list (default, **geom**).

5. Enable the **Create a label for each face zone** checkbox to generate an individual label for each face zone.

6. Click **Create**.

7.3.1.3.1.3. Creating Multiple Objects

You can also create one object per selected zone using the command `/objects/create-multiple`. This is particularly useful for CAD exported models, as you can define an object per part. An object will be created for each selected zone and will be named automatically based on the specified prefix and priority. The name assigned is **prefix face zone name-priority:object ID**. (For example, an object named **object-wall-3:20** will be created for the face zone **wall**, with the specified prefix **object-** and priority **3**. The object ID is 20.)

1. Specify the first and last zone for which objects are to be created. You can also use wildcards for specifying the face zones to be considered.

   Note:

   You need to specify valid zone names or IDs.

2. Specify the prefix to be used for the object name and the cell zone type.

3. Specify the priority for the first object (for the first zone selected) and the increment in priority.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When the increment is set to a value greater than zero, the priority will be assigned in the order of face zone ID. If the increment is set to zero, all objects will have the same priority.

A geometry object will be created for the first and last zone (as specified) and for all valid face zones having IDs between the first and last zone.

7.3.1.3.1.4. Easy Object Creation and Modification

You can easily move a zone from one object to another or create a new object using the **Transfer Zones** tool ( ) or **Ctrl**+**Shift**+**Y** hotkey combination.

1. Select the zone(s) to be moved or to form a new object.
2. Press the hotkey combination or click .
3. If a target object is set, the selected zone(s) are moved to the target. If no target is set, enter the new object name in the **Create Geometry Object** dialog box.

To set a target object,

1. Select the object using the mouse probe.
2. Press the hotkey combination **Ctrl**+**S** or .

Note:

Empty objects will be automatically deleted.

7.3.1.3.1.5. Changing Object Properties

Use the **Change Object Properties** dialog box to change the properties of objects based on selections in the graphics window. Select one or more objects and then use the hot-key **Ctrl**+**Shift**+**N** to open the **Change Object Properties** dialog box.

Alternatively, select the objects in the Model tree and then select **Rename/Change Properties...** from the menu available to access the **Change Object Properties** dialog box.

If a single object is selected, you can specify a new object name. If multiple objects are selected, you can specify a common prefix for the objects selected. This enables you to easily view the objects with the same prefix using the tree view button for selection lists. You can also choose to rename object zones and set the geometry recovery option for the object zones (high or low).

7.3.1.3.1.6. Automatic Alignment of Objects

You may also fit objects together in precise alignment, for example to position a flange on a body by aligning bolt holes.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The procedure is initiated via the hotkey **Ctrl**+**Shift**+**G**, and uses a temporary local coordinate systems (LCS) to achieve the alignment.

Defining a local coordinate system by selecting 1-6 nodes works as follows:

- If only 1 node is selected, the LCS origin is at the node location and axes are aligned to global coordinate system.
- For 2 nodes, the LCS origin is at the midpoint of nodes and axes are aligned to global coordinate system.
- For 3 nodes, the origin is at the first point, the LCS x-axis is along a vector from the first to the second point, and the LCS y-axis is along a vector from the first point to the 3rd point.
- For 4, 5 or 6 nodes, the first 3 points define a circle. The LCS origin is at the center of the circle and the z-axis is normal to the circular plane (positive direction is determined by the right-hand rule).
  - For 4 nodes, the x-axis is defined by a vector from the center of the circle to the projection of the 4th point on the circular plane.
  - For 5 nodes, the x-axis is defined by a vector from the center of the circle to the projection of the mid-point of 4th and 5th points on the circular plane.
  - For 6 nodes, the x-axis is defined by a vector from the center of the circle to the projection of the circumcenter of 4th, 5th and 6th points on the circular plane.

7.3.1.3.1.7. Remeshing Geometry Objects

You can create mesh objects from the geometry objects without wrapping by using the **Remesh** dialog box.

1. Select **Remesh...** in the context-sensitive menu for geometry objects to open the **Remesh** dialog box.
2. Select **Individually** or **Collectively** from the **Target** list.

   Enter the object name when using the collective option.
3. Click **OK**.

   The size field based on currently defined sizing controls or size functions will be used to remesh the geometry objects.

7.3.1.3.1.8. Creating Edge Zones

You can create edge zones on selected face zones or selected surfaces using the onscreen tools or context menus. This feature is described in Extract Edge Zones (p. 707).

7.3.1.3.2. Using the Manage Objects Dialog Box

The **Manage Objects** dialog box contains options that enable you to define objects and perform certain object manipulation operations.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Right-click on **Model** in the tree and select **Object Management...** from the menu to access the **Manage Objects** dialog box.

- Select the **Objects**, **Face Zones**, or **Edge Zones** from their respective lists as required. Filtering and the support of wildcards, such as *, ?, [], Boolean NOT (^), Boolean AND (&), as well as Boolean OR (|), is available for each.

  Note:

  If edge zones are to be included in the object definition, they must have been created and visible in the lists.

- **List all zones** controls how face and edge zones are listed.
  - If enabled, all available zones are listed.
  - If disabled, the lists will contain only zones that are not included in existing objects. You can use this to identify zones that are not associated with objects.
- **Select Object Zones** controls highlighting of face and edge zones in the lists. If enabled, when an object is selected, face and edge zones in the object will be highlighted.

Choose one of the tabs to access the object management tools.

7.3.1.3.2.1. Defining Objects

7.3.1.3.2.2. Object Manipulation Operations

7.3.1.3.2.3. Object Transformation Operations

7.3.1.3.2.1. Defining Objects

You can define the objects using the options in the **Definition** tab of the **Manage Objects** dialog box.

1. Enter an appropriate name in the **Object Name** field.

   You can also have the object name generated automatically by leaving the **Object Name** field blank. In this case, the object name will be assigned based on the **Prefix**, cell zone type, and priority specified (for example, an object named **object-fluid:3-20** has prefix **object-**, cell zone type **fluid**, priority **3**, and object ID **20**).

2. Select the appropriate option from the **Cell Zone Type** drop-down list.
3. Set the priority.
4. Select the appropriate type from the **Object Type** drop-down list (default, **geom**).
5. Click **Create**.

You can modify the object definition using the **Change** option in the **Manage Objects** dialog box. Select the object to be modified, make the necessary changes in the **Definition** tab, and click **Change**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also use the **Change Type to Mesh** option in the menu available for geometry objects.

Objects can be deleted using the **Delete** option in the **Definition** tab. You can also enable **Include Faces and Edges** to delete the faces and edges comprising the object, when the object is deleted.

Note:

When an object is deleted along with the face and edge zones comprising the object, any corresponding face/edge groups will also be deleted.

7.3.1.3.2.2. Object Manipulation Operations

The following object manipulation operations can be performed using the options in the **Operations** tab of the **Manage Objects** dialog box or using the options from the menu available on any geometry or mesh object selected in the Outline View:

- Objects of the same type (geometry or mesh) can be merged using the **Merge Objects...** option. Specify the name for the merged object in the **Merge Objects** dialog box.

  When multiple mesh objects are merged, the face zone labels represent the objects that were merged.

  Important:

  Merging zones that have different face zone labels will result in a merged zone with the original labels appended.

- Wall face zones comprising objects can be merged using the **Merge Walls** option.
- The edge zones comprising an object can be merged into a single edge zone using the **Merge Edges** option.

  Note:

  If the object contains edge zones of different types (boundary and interior), the edge zones of the same type (boundary or interior) will be merged into a single edge zone.

- Intersection loops can be created within an object or between objects using the options in the **Intersection Loops** group box.
- Edge zones can be extracted from the face zones included in the specified objects, based on the feature angle value specified using the options in the **Edge Zones** group box. You can specify whether only feature edges or all edges are to be extracted for the objects selec-

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ted. Previously created edges will automatically be disassociated from the object and added to the **Unreferenced** branch of the Outline View tree.

Note:

– This functionality is also available in the **Extract Edges** dialog box. See Extract Edge Zones (p. 707)

– Similar functionality is also available in the Feature Modify dialog box and Surface Retriangularization dialog box. See Creating and Modifying Features (p. 672) and Remeshing Boundary Zones (p. 678).

- A face group and an edge group comprising the face zones and edge zones included in the specified objects can be created using the options in the **Zone Group** group box.
- The face zones comprising the object can be separated based on the angle or seed specified using the options in the **Separate Faces** group box.
- When the face zones and/or edge zones comprising an object are deleted, you will be prompted to update the object definition. You can use the command `/objects/update` to update the defined objects per the changes.
- You can rename the face and edge zones comprising the object based on the object name and also specify the separator to be used.

7.3.1.3.2.3. Object Transformation Operations

The following object transformation operations can be performed using the options available in the **Transformations** tab of the **Manage Objects** dialog box:

- Objects can be rotated using the **Rotate** option. Specify the angle of rotation and the pivot point and the axis of rotation by selecting 1-6 nodes in the graphics window.

  The pivot point and the axis of rotation can be defined by selecting 1-6 nodes as follows:

  – If only 1 node is selected, the pivot point is at the node location and the axis of rotation is the global z-axis.

  – For 2 nodes, the pivot point is at the midpoint of the nodes selected and the axis of rotation is the global z-axis.

  – For 3 nodes, the pivot point is at the first node selected. The axis of rotation is the local z-axis normal to the plane defined by the three points, the positive direction is determined by the right-hand rule.

  – For 4, 5 or 6 nodes, the first 3 points define a circle. The pivot point is at the center of the circle. The axis of rotation is the local z-axis normal to the circular plane, the positive direction is determined by the right-hand rule.

- Objects can be scaled using the **Scale** option. Specify the scale factors (X, Y, Z) for the scaling operation.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Objects can be translated using the **Translate** option. Specify the vector components to define the translation, or click **Define** and select two screen locations to determine the translation.

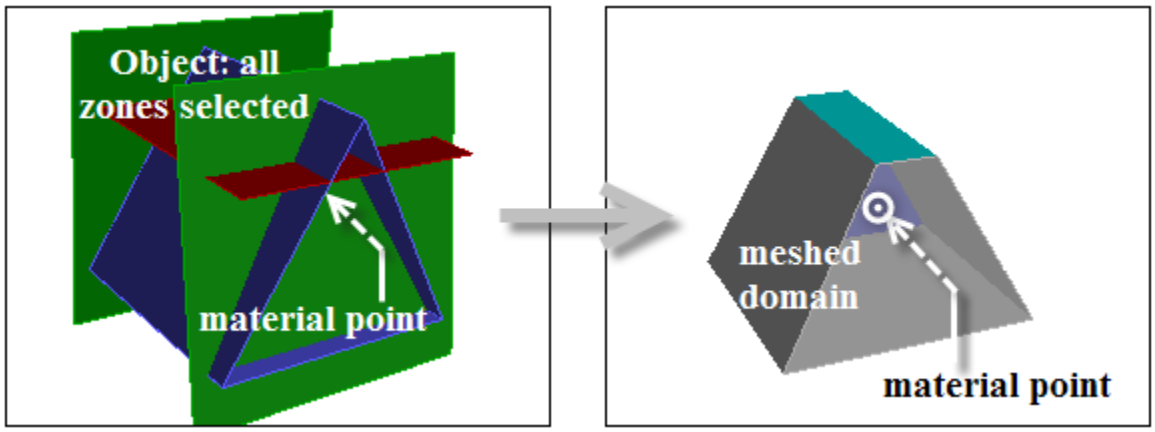
7.3.2. Material Points

In addition to objects, material points can be defined to allow the mesher to separate the cell zone. Typically, a material point can be defined to retrieve a cell zone for which an object cannot be defined, or the object definition alone is not sufficient to retrieve the cell zone. Material points can also be used to retrieve regions from a non-contiguous cell zone. Contiguous regions will be separated based on the respective material points defined. The cell zone retrieved based on the defined material point will be of the type fluid and have the specified name.

The following examples demonstrate the use of a material point in addition to objects:

- Some cases involving "dirty" geometry may result in multiple voids. In this case, the volume to be meshed can be recovered by defining an object comprising the zones enclosing the domain to be meshed, combined with a material point within the expected meshed domain (see Figure 7.14: Using Material Points—Example (p. 613)).

Figure 7.14: Using Material Points—Example



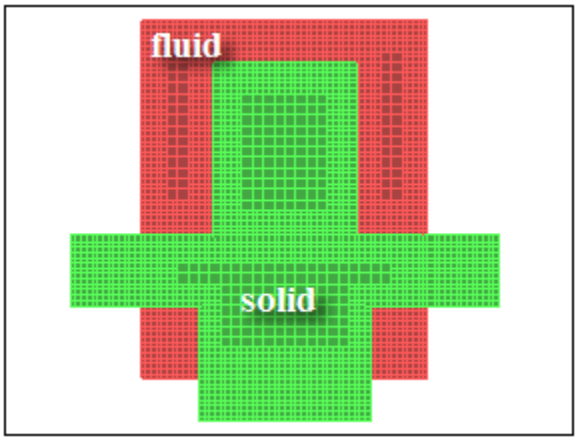
Note:

The intersection loops can be created (see Object Manipulation Operations (p. 611)) to recover the intersecting features accurately.

- In Figure 7.15: Example—CutCell Mesh, Only Objects Defined (p. 614), the use of only objects to define the meshed domain results in a mesh with two cell zones, **solid** and **fluid**.

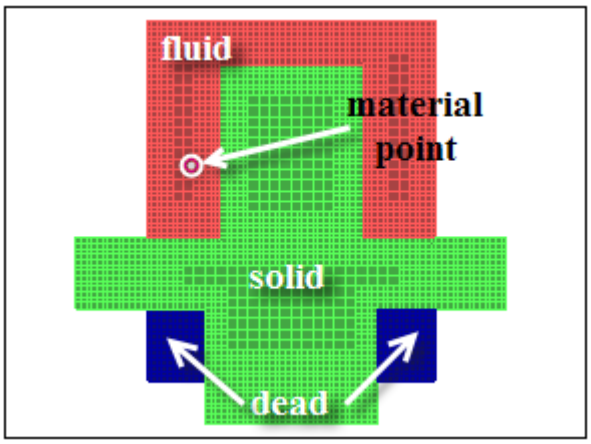
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.15: Example—CutCell Mesh, Only Objects Defined



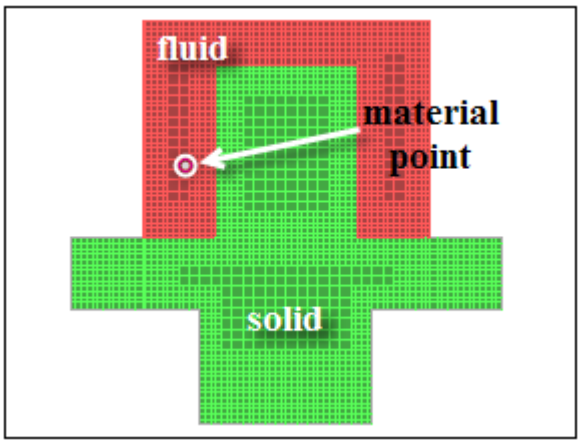
By specifying a material point in addition to the object definition (fluid or dead), the **fluid** zone in Figure 7.15: Example—CutCell Mesh, Only Objects Defined (p. 614) will be further separated into a fluid zone (containing the material point) and a dead zone (see Figure (A) (p. 614)). If **auto-delete-dead-zones?** is enabled (default), the separated dead zones will be deleted automatically (Figure (B) (p. 614)).

(A) CutCell Mesh—Material Point and Objects Defined, **auto-delete-dead-zones?** Disabled



(B) CutCell Mesh—Material Point and Objects Defined, **auto-delete-dead-zones?** Enabled

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



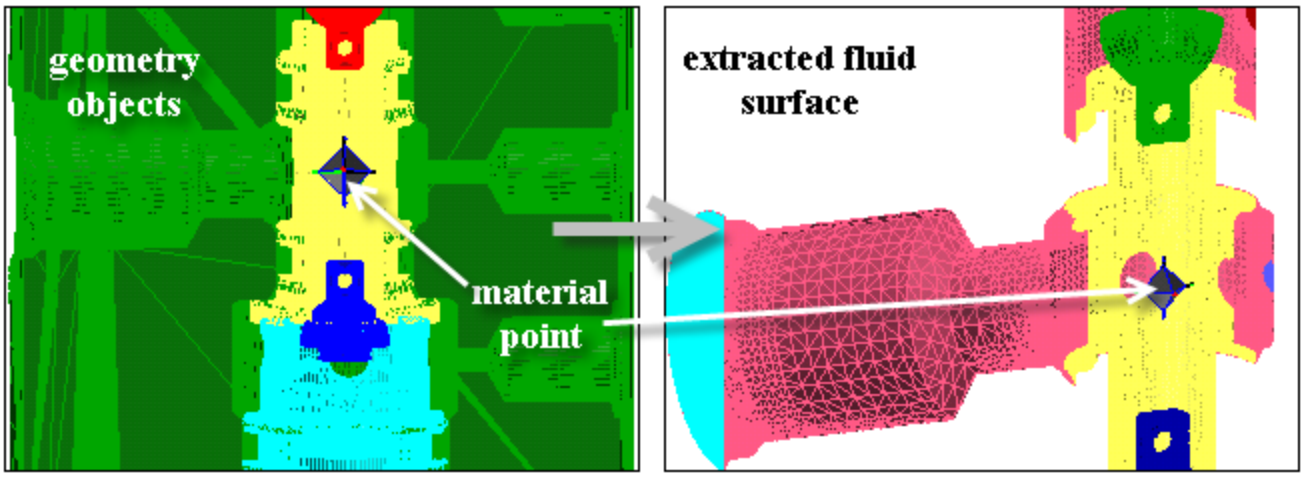
Note:

For cases where a single region is separated by a double-sided surface (**fan**, **radiator**, or **porous-jump**), you need to define a material point for each of the regions to be recovered (that is, both upstream and downstream of the double-sided surface). Separate cell zones will be recovered for each region on either side of the double-sided surface.

You can merge the cell zones manually after the mesh has been generated.

- Figure 7.16: Example—Fluid Surface Extracted From Geometry Objects and Material Point (p. 615) shows the use of a material point in addition to objects defined to extract the internal fluid surface, using the object wrapping operation.

Figure 7.16: Example—Fluid Surface Extracted From Geometry Objects and Material Point



7.3.2.1. Creating Material Points

Right-click **Model** in the tree and select **Material Points...** from the menu to access the **Material Points** dialog box.

Click **Create...** to open the **Create Material Point** dialog box. Follow the process described.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the appropriate zones or objects in the graphics window. The selections should be such that the material point created will lie at a central point in the fluid domain.
2. Click **Compute** to obtain the material point coordinates based on the selections.

   You can also specify the coordinates manually if the material point location is known.
3. Enable **Preview** to verify that the location is appropriate.
4. Enter an appropriate fluid zone name in the **Name** field.
5. Click **Create**.

Use the **List** button to display the x-, y-, and z- coordinates of the selected material point(s) in the console.

Use the **Delete** button to remove the selected material point(s).

Use the **Draw** button to display the selected material point(s) in the graphics window.

7.4. Object-Based Surface Meshing

Object-based meshing is the recommended meshing approach with which you can generate a tetrahedral, hexcore, or polyhedral volume mesh, with or without inflation layers. In this approach, you first create a conformal, connected surface mesh on all the objects to be meshed. The surface mesh, and material points if required, are then used to identify the regions to be filled with the volume mesh.

This chapter describes the processes used to create a conformal, connected surface mesh. Steps to create the volume mesh are described in Object-Based Volume Meshing (p. 642). Refer to Objects and Material Points (p. 600) for more information on meshing objects.

Important:

Ensure that the model is suitably scaled during import and the global minimum size is at least 0.01 to avoid numerical problems during mesh generation.

The tools to complete the object-based surface meshing steps are found in the context-sensitive menus in the **Outline View** or in the onscreen tools and hotkeys. Instructions are described in the following sections.

7.4.1. Surface Mesh Processes
7.4.2. Preparing the Geometry
7.4.3. Diagnostic Tools
7.4.4. Connecting Objects
7.4.5. Advanced Options

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.1. Surface Mesh Processes

In principal, there are two basic workflows to create a conformal, connected surface mesh from an unconnected assembly.

Note:

It is assumed that objects are already created upon CAD import. Object management, if necessary, is described in Objects (p. 600).

Single fluid volume using Wrapper based workflow

The basic Wrapper based workflow for a single fluid volume simulation follows these steps:

1. Import the CAD model using the **CAD Faceting** tessellation option.

   Geometry objects are created. See Importing CAD Files (p. 264) for additional information.

2. Define sizing and compute the size field.

   Scoped Sizing is recommended to apply individual size controls on specific objects as required. See Defining Scoped Sizing Controls (p. 594) for additional information.

3. Define material points to assist in identifying the fluid volume.

   See Material Points (p. 613) for additional information.

4. Check and fix the geometry, particularly gaps.

   You can use the context menu **Diagnostics** → **Geometry**, accessible from any Geometry object, to assist in finding any geometry issues. See Diagnostic Tools (p. 625) for additional information.

   Missing geometry objects may be constructed as described in Preparing the Geometry (p. 619).

5. Define periodic boundaries, if applicable.

   See Creating Periodic Boundaries (p. 700) for additional information.

6. Wrap the model.

   Step through the options available in the **Wrap** context menu. See The Wrapping Process (p. 705) for details.

7. Check the mesh and improve the quality, if necessary.

   You can use the context menu **Diagnostics** → **Connectivity and Quality**, accessible from any mesh object.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Complex topology using Join and Intersect based workflow

The basic Join/intersect based workflow for complex topology such as CHT simulation, follow these steps:

1. Import the CAD model using the **CFD Surface Mesh** tesselation option.

   Mesh objects are created. See Importing CAD Files (p. 264) for additional information.

   > **Note:**
   >
   > If the assembly has already been connected (for example, using Share Topology operations in Ansys DesignModeler or SpaceClaim), it is sufficient to just import the CAD using the **One object per file** and **CFD Surface Mesh** options.

2. Check the mesh and fix the connectivity, if necessary.

   You can use the context menu **Diagnostics** → **Connectivity and Quality**, accessible from any mesh object.

3. Check and fix the geometry, particularly gaps.

   You can use the context menu **Diagnostics** → **Geometry**, accessible from any Geometry object, to assist in finding any geometry issues. See Diagnostic Tools (p. 625) for additional information.

   Missing geometry objects may be constructed as described in Preparing the Geometry (p. 619).

4. Merge all objects to a single object using the **Merge** option in the context menu.

5. Connect all bodies.

   You can use the **Join/Intersect** context menu or the onscreen tools for **Join** or **Intersect**. See Connecting Objects (p. 629) for additional information.

6. Optionally define sizing, compute the size field and remesh if necessary.

   Remesh is recommended as it often reduces mesh count and improves quality while maintaining sufficient mesh density. You can use the **Remesh Faces** context menu or the onscreen tools.

   Computing a size field is necessary only if you choose to remesh. Scoped Sizing is recommended to apply individual size controls on specific objects as required. See Defining Scoped Sizing Controls (p. 594) for additional information.

7. Define periodic boundaries, if applicable.

   See Creating Periodic Boundaries (p. 700) for additional information.

8. Check the mesh and improve the quality, if necessary.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can use the context menu **Diagnostics** → **Connectivity and Quality**, accessible from any mesh object.

When the surface mesh is conformally connected and of sufficient quality, proceed to generate the volume mesh as described in Object-Based Volume Meshing (p. 642).

7.4.2. Preparing the Geometry

When the geometry is imported from CAD, there may be a number of gaps and the faceted geometry may be disconnected. Though operations such as merging nodes and faceted stitching can be used to partially connect the model, some gaps may remain and features may be lost.

You may want to perform tasks such as create a wind tunnel or far-field domain, close annular gaps, create capping surfaces for inlets or outlets, or create groups of zones for models with a large number of zones. Right-click on **Model** in the tree and select the appropriate option from the menu to access the options to perform such operations.

7.4.2.1. Using a Bounding Box

7.4.2.2. Closing Annular Gaps in the Geometry

7.4.2.3. Patching Tools

7.4.2.4. Using User-Defined Groups

7.4.2.1. Using a Bounding Box

The bounding box tool can be used to create a wind tunnel or far-field domain for the imported geometry. A geometry object can be created for the bounding box surface created. You can also use the bounding box tool to create a body of influence to be used for defining size functions. See Using the Bounding Box Dialog Box (p. 690) for details on the options for creating a bounding box. Right-click on **Model** in the tree and select **Construction Geometry** → **Bounding Box...** to open the **Bounding Box** dialog box.

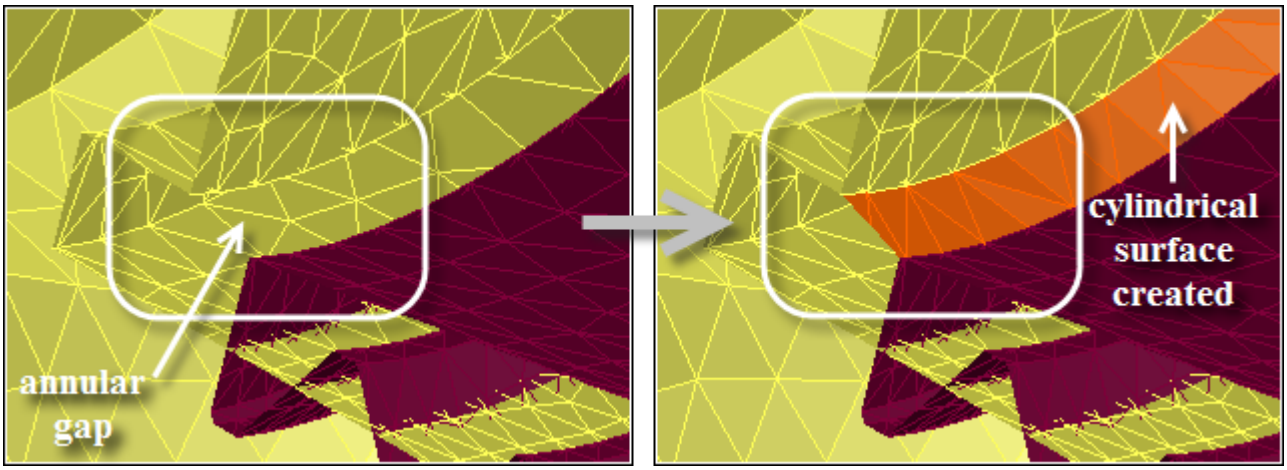
7.4.2.2. Closing Annular Gaps in the Geometry

See Creating a Cylinder/Frustum (p. 693) to create a cylindrical or annular surface to close radial gaps in the geometry. A geometry object can be created for the surface created. Right-click on **Model** in the tree and select **Construction Geometry** → **Cylinder...** to open the **Cylinder** dialog box.

Figure 7.17: Closing a Radial Gap (p. 620) shows an example where a radial gap is closed using a cylindrical surface.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.17: Closing a Radial Gap



1. Select **3 Arc, 1 Height Node** in the **Options** list.
2. Click **Select Nodes...** and select 3 nodes on one circle and one (height node) on the other across the radial gap.
3. Enter an appropriate value for **Edge Length**.
4. Enable **Create Object** and disable **Caps**.
5. Click **Create**.

7.4.2.3. Patching Tools

The patching tools enable filling of unwanted holes in the geometry. A hole may be bounded by either free edges or feature edges. Use the patching tools to:

- patch holes associated with free faces in multiple zones
- fill punched holes in a single zone
- cap inlets/outlets and assign the appropriate zone type
- patch other complex shapes to close gaps including sharp angles and small pockets in the geometry using the loop selection tools

Use the patching tools to locate and fix the major holes in the geometry. If you miss any holes, you can fix them later using the hole detection feature. See Fixing Holes in Objects (p. 711) for details.

7.4.2.3.1. Using the Patch Options Dialog Box

7.4.2.3.2. Using the Loop Selection Tool

7.4.2.3.1. Using the Patch Options Dialog Box

When the patching tools are used, by default, the new faces are added to a new face zone and are remeshed. These options are available in the **Patch Options** group in the ribbon.

- **Remesh** enables the automatic remeshing of the patched area.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Separate** enables the creation of a separate face zone/object for the new faces created. Additional options for object/zone granularity and type are available in the **Patch Options** dialog box.

The **Patch Options** dialog box contains options for object/zone granularity and type and is available when the **Separate** option is selected. Specify the object/zone granularity as follows:

- Select **New Object** to create a new object for the face zones. Specify the object name and a label name. If the label name is not specified, the object name will be used as the label name. Face zone names will be the same as the label names. Select the **Object Type** and **Zone Type** from the lists.
- Select **Add to Object** to add the face zones to an existing object. Select the object and specify a label name. If the label name is not specified, the default name **patch:#** (# indicates the ID) will be used. Face zone names will be the same as the label names. Select the **Zone Type** from the list.
- Select **Add to Unreferenced** to create unreferenced face zones. Select the **Zone Type** from the list. The default name **patch:#** (# indicates the ID) will be used for the zone name. These zones will be available in the **Unreferenced** branch of the tree.

Using Edge/Node Selections

Select edges/nodes in the graphics window to fix the holes in the geometry.

- Fixing holes by selecting edges.

  1. Set the patching options in the **Patch Options** group in the ribbon.
  2. Select the edge filter ( ) or hot-key **Ctrl**+**E**) and select any edge on the hole boundary. You can select either free edges or feature edges.
  3. Click or press **F5** to create the surface that closes the hole.

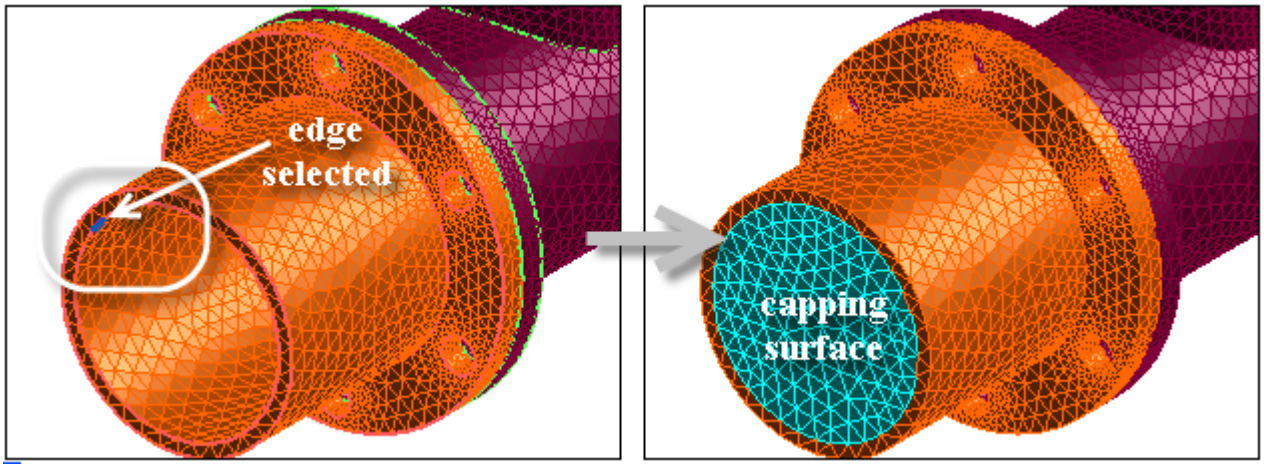
     a. If the **Separate** option is enabled, set the object/zone granularity and type in the **Patch Options** dialog box (Using the Patch Options Dialog Box (p. 620)). If the **Remesh** option is enabled, the face zones will be remeshed.

     b. Click **Create** in the **Patch Options** dialog box.

  Figure 7.18: Creating a Surface Using an Edge (p. 622) shows an example where the capping surface is created by selecting an edge on the existing object.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.18: Creating a Surface Using an Edge



- Fixing holes by selecting nodes.

  1. Set the patching options in the **Patch Options** group in the ribbon.

  2. Select the node filter (or hot-key **Ctrl**+**N**) and select the nodes around the hole.

  3. Press or **F5** to create the surface that closes the hole.

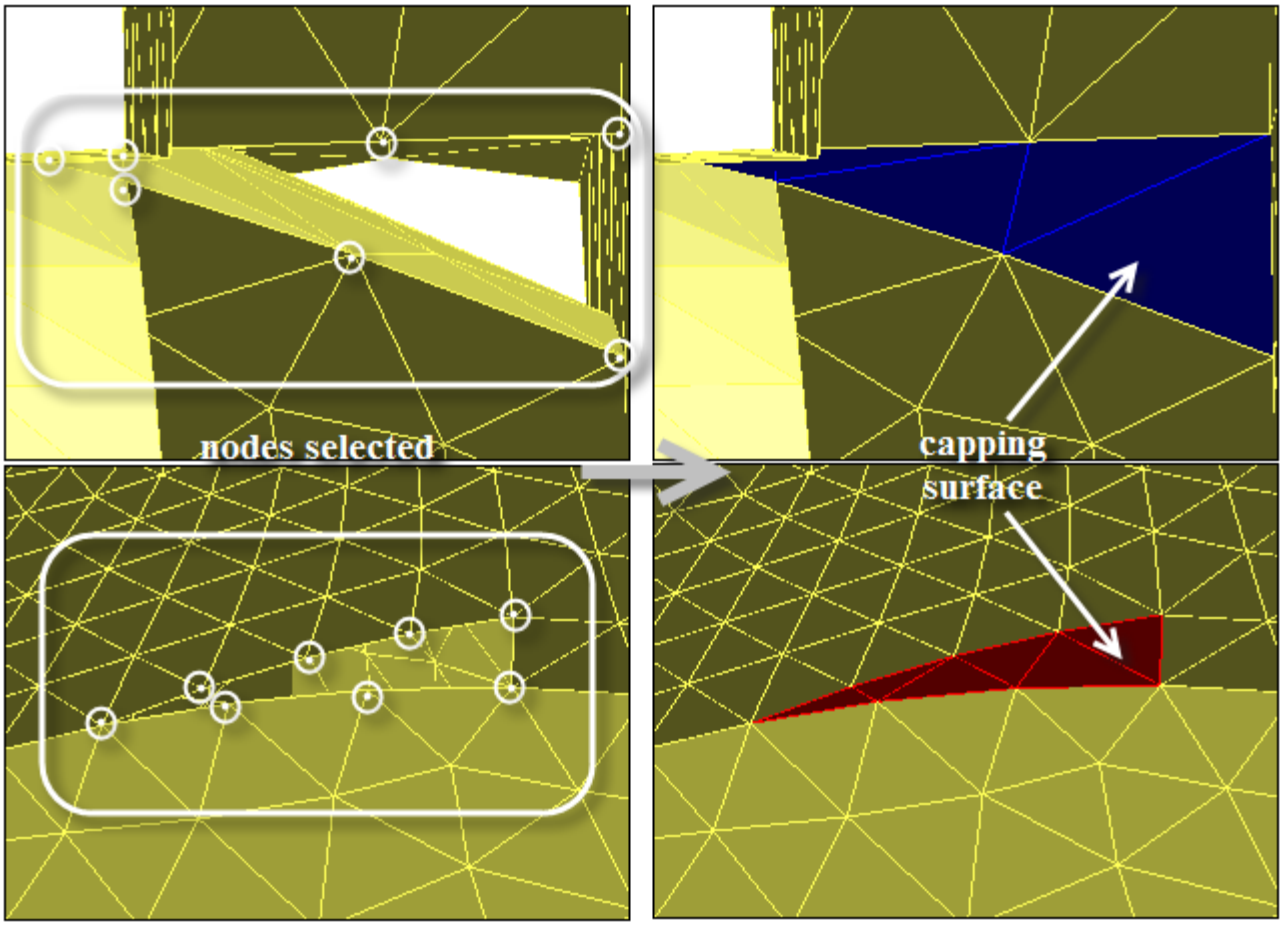
     a. If the **Separate** option is enabled, set the object/zone granularity and type in the **Patch Options** dialog box (Using the Patch Options Dialog Box (p. 620)). If the **Remesh** option is enabled, the face zones will be remeshed.

     b. Click **Create** in the **Patch Options** dialog box.

Figure 7.19: Creating a Surface Using Nodes (p. 623) shows an example where the capping surfaces are created by selecting nodes on the existing object.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.19: Creating a Surface Using Nodes



Patching Multiple Holes in a Single Zone

All holes associated with free edges or punched holes in a single zone can be closed in a single patching operation using the hot-key **Ctrl**+**R** ( ).

1. Set the patching options in the **Patch Options** group in the ribbon.
2. Select the appropriate entity for the operation:
   - For patching holes with free faces, select the appropriate face zones in the graphics window.
   - For patching punched holes, select a face adjacent to a hole in the face zone.

> **Note:**
>
> Punched holes can be closed only in a single face zone having finite thickness. There should be no free faces in the face zone being patched.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click or use the hot-key **Ctrl**+**R**.

   a. If the **Separate** option is enabled, set the object/zone granularity and type in the **Patch Options** dialog box (Using the Patch Options Dialog Box (p. 620)). If the **Remesh** option is enabled, the face zones will be remeshed.

   b. Click **Create** in the **Patch Options** dialog box.

7.4.2.3.2. Using the Loop Selection Tool

The loop selection tool can be accessed by clicking in the graphics window or using the hot-key **Ctrl**+**Shift**+**L**. This tool provides additional options, such as using fewer node selections, for creating the capping surface. You can also select positions to define the loop.

The following selection options are available:

- In the first group of tools, choose how the path between selected nodes/positions is defined - by edges, feature, boundary, or direct path. Click to switch between selecting nodes or positions to define the loop.
- The second group of icons is used to select open or closed loop. Then, for closed loop mode, you can choose how the path between the first and last nodes is defined - by edges, feature, boundary, or direct path.

After making the necessary selections, click or use the hot-key **Ctrl**+**K** in the Loop Selection mode to open the **Create Cap** dialog box.

The **Create Cap** dialog box contains options for object/zone granularity and type and for remeshing the capping surface.

1. Specify the object/zone granularity.

   - Select **New Object** to create a new object for the face zones. Specify the object name and a label name. If the label name is not specified, the object name will be used as the label name. Face zone names will be the same as the label names. Select the **Object Type** and **Zone Type** from the lists.
   - Select **Add to Object** to add the face zones to an existing object. Select the object and specify a label name. If the label name is not specified, the default name **patch:#** (# indicates the ID) will be used. Face zone names will be the same as the label names. Select the **Zone Type** from the list.
   - Select **Add to Unreferenced** to create unreferenced face zones. Select the **Zone Type** from the list. The default name **patch:#** (# indicates the ID) will be used for the zone name. These zones will be available in the **Unreferenced** branch of the tree.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Enable **Remesh** to remesh the capping surface created.

3. Click **Create** in the **Create Cap** dialog box.

For the list of hot-keys associated with the options in the Loop Selection toolbar, refer to Appendix C: Shortcut Keys (p. 877).

7.4.2.4. Using User-Defined Groups

You can define user-defined groups to better handle large models. The options for manipulating user-defined groups are available in the **User Defined Groups** dialog box. Right-click on **Model** in the tree and select **Groups...** to open the **User Defined Groups** dialog box.

You can create a face group and an edge group comprising the face zones and edge zones included in the specified objects using the options in the **Zone Group** group box in the **Operations** tab in the **Manage Objects** dialog box. You can activate a particular group using the **Activate** option in the **User Defined Groups** dialog box.

Additionally, a face zone group is automatically created when a mesh object is created using the **Sew** operation. This face zone group is prefixed by **_mesh_group**, and enables easy selection of mesh object face zones for various operations (improve, smooth, and so on).

Note:

When an object is deleted along with the face and edge zones comprising the object, the corresponding groups will also be deleted.

Important:

You cannot create a new group having the name **global**, or having the same name as one of the default groups.

7.4.3. Diagnostic Tools

The options in the **Diagnostic Tools** dialog box enable you to find and fix problems in boundary meshes, or display boundary mesh statistics, of selected objects. The **Diagnostic Tools** dialog box is accessed by selecting the appropriate option from the **Diagnostics** sub-menu available by right-clicking on any geometry or mesh object selected in the Outline View.

Diagnostic tools are available for the following:

- finding and fixing **Geometry** problems, such as gaps or intersections between objects.
- finding and fixing **Connectivity** problems in the surface mesh, such as free or multi-connected edges, and overlapping or intersecting faces.
- finding and fixing **Quality** problems in the surface mesh.

A **Summary** table of mesh statistics can be obtained using the **Summary** button at the bottom of the **Diagnostic Tools** dialog box, or from the context-sensitive menu on any geometry or mesh object selected in the tree.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.3.1. Geometry Issues
7.4.3.2. Face Connectivity Issues
7.4.3.3. Quality Checking
7.4.3.4. Summary

Note:

When selecting **Diagnostics...** from under a mesh object's **Face Zone Label**, the **Geometry** options are not available. The dialog box contains only **Face Connectivity** and **Quality** options for the boundary zones included in the **Face Zone Label**.

7.4.3.1. Geometry Issues

Problems in the geometry, such as gaps or intersections can be located and fixed using the diagnostic tools. Select the objects (geometry or mesh) in the Model tree, and then select **Geometry...** from the **Diagnostics** sub-menu available from the context-sensitive menu.

1. Select the desired **Issue** and then set the relevant marking options. The issues that can be diagnosed are as follows:

   - Self Intersections
   - Cross Intersections
   - Self Face Proximity
   - Cross Face Proximity
   - Self Edge Proximity

2. Click **Mark** to identify and obtain a count of the **Unvisited** problems in your boundary mesh.
3. Click **First** (**Next**) to step through the problems individually. At each step, the identified problem region will be highlighted in the graphics window.
4. To correct the identified problem (if necessary), click the appropriate button in the **Operations** group box. You can choose to fix all identified problem areas or the area that is currently displayed.

Note:

Many other geometry modification tools are available. The buttons presented in the **Operations** group box represent the most likely tools for the type of issue.

7.4.3.2. Face Connectivity Issues

Problems in the surface mesh such as free or multi-connected faces, self-intersecting faces, or other problematic configurations can be located and fixed using the diagnostic tools. Select the objects

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(geometry or mesh) in the Model tree, and then select **Connectivity and Quality...** from the **Diagnostics** sub-menu available from the context-sensitive menu.

Note:

If a **Face Zone Label** is selected under a mesh object, the **Diagnostics...** menu will not have any submenu choices, but will open directly with **Face Connectivity** and **Quality** options available.

1. Select the boundary zones to be examined.
   - Select **All** to select the boundary zones from a list including all boundary face zones available.
   - Select **Unmeshed** to select the boundary zones from a list of the unmeshed tri zones available. The unmeshed zones are those that are not connected to a volume mesh.
2. On the **Face Connectivity** tab, select the desired **Issue** and then set the relevant marking options. The issues that can be diagnosed are as follows:
   - Free
   - Multi
   - Self Intersections
   - Self Proximity
   - Duplicate
   - Spikes
   - Islands
   - Steps
   - Slivers
   - Point Contacts
   - Invalid Normals

     Note:

     Zone-specific or scoped prism settings should be applied prior to using this option.

   - Leaks
   - Deviation
3. Click **Mark** to identify and obtain a count of the **Unvisited** problems in your boundary mesh.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Click **First** (**Next**) to step through the problems individually. At each step, the identified problem region will be highlighted in the graphics window.

5. To correct the identified problem (if necessary), click the appropriate button in the **Operations** group box. You can choose to fix all identified problem areas or the area that is currently displayed.

Note:

Many other mesh modification tools are available. For example, you can do **Local Remesh** (use the hotkey **Ctrl**+**Shift**+**R**) or Smooth (**F6**) instead of Collapse (**Ctrl**+**J**) to remove slivers. The buttons presented in the **Operations** group box represent the most likely tools for the type of issue.

7.4.3.3. Quality Checking

Problems with the surface mesh quality can be located and fixed using the diagnostic tools. Select the objects (geometry or mesh) in the Model tree, and then select **Connectivity and Quality...** from the **Diagnostics** sub-menu available from the context-sensitive menu.

Note:

If a **Face Zone Label** is selected under a mesh object, the **Diagnostics...** menu will not have any submenu choices, but will open directly with **Face Connectivity** and **Quality** tabs available.

---

1. Select the boundary zones to be examined.

   - Select **All** to select the boundary zones from a list including all boundary face zones available.
   - Select **Unmeshed** to select the boundary zones from a list of the unmeshed tri zones available. The unmeshed zones are those that are not connected to a volume mesh.

2. On the **Quality** tab, set the number of quality measures and specify up to three quality measures to be used. Then select from the following quality measure options.

   - Skewness
   - Size Change
   - Edge Ratio
   - Area
   - Aspect Ratio
   - Warp
   - Dihedral Angle
   - Ortho Skew

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click **Mark** to identify and obtain a count of the **Unvisited** problems in your boundary mesh.

4. Click **First** (**Next**) to step through the problems individually. At each step, the identified problem region will be highlighted in the graphics window.

5. To correct the identified problem (if necessary), select the appropriate option in the **Operations** group box and click **Apply for All**. Alternatively, you can choose to collapse or smooth all marked faces that are currently displayed.

For more information on how Fluent calculates the quality and adjusts the mesh, see the Quality Measures (p. 563) page.

Note:

Many other mesh modification tools are available. For example, you can do **Local Remesh** (use the hotkey **Ctrl**+**Shift**+**R**) instead of Smooth (**F6**) or Collapse (**Ctrl**+**J**) to remove a skewed face. The buttons presented in the **Operations** group box represent the most likely tools for the type of issue.

7.4.3.4. Summary

A tabular summary of mesh statistics can be displayed in the console by selecting **Summary** from the context sensitive menu for a geometry or mesh object in the Outline View tree.

The displayed data includes the number of free-, multi-, and duplicate faces; face quality (skewness) statistics, and total number of faces and face zones. This is the same information as is displayed by clicking the **Summary** button in the **Diagnostic Tools** dialog box.

7.4.4. Connecting Objects

The join and intersect operations are used to connect face zone labels within a mesh object by joining overlapping faces or intersecting face zones. In case of multiple mesh objects, merge the objects into a single mesh object and then proceed to join/intersect them. For best results, it is recommended that faces be of similar size where the join or intersect operation is occurring. The process is interactive, fast, scriptable, and enables direct control over local shape and quality, and volumetric and surface overlaps.

Join/Intersect may be employed as a bottom-up strategy that enables you to build multiple sub-assemblies individually, and then connect the sub-assemblies to create the final assembly. It can be used to connect all the face zone labels within a mesh object into the final assembly in one operation. Support for part replacement without global remeshing is inherent in the process.

An example of the join operation is shown in Figure 7.20: Overlapping Surfaces (p. 630) and Figure 7.21: Connected Surfaces After Join (p. 630). The overlapping area of the face zones is separated based on the parameters specified and merged into a single separated face zone after the join operation. The joined surfaces will be locally remeshed. The multi-connected faces indicate that the zones are now connected.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.20: Overlapping Surfaces

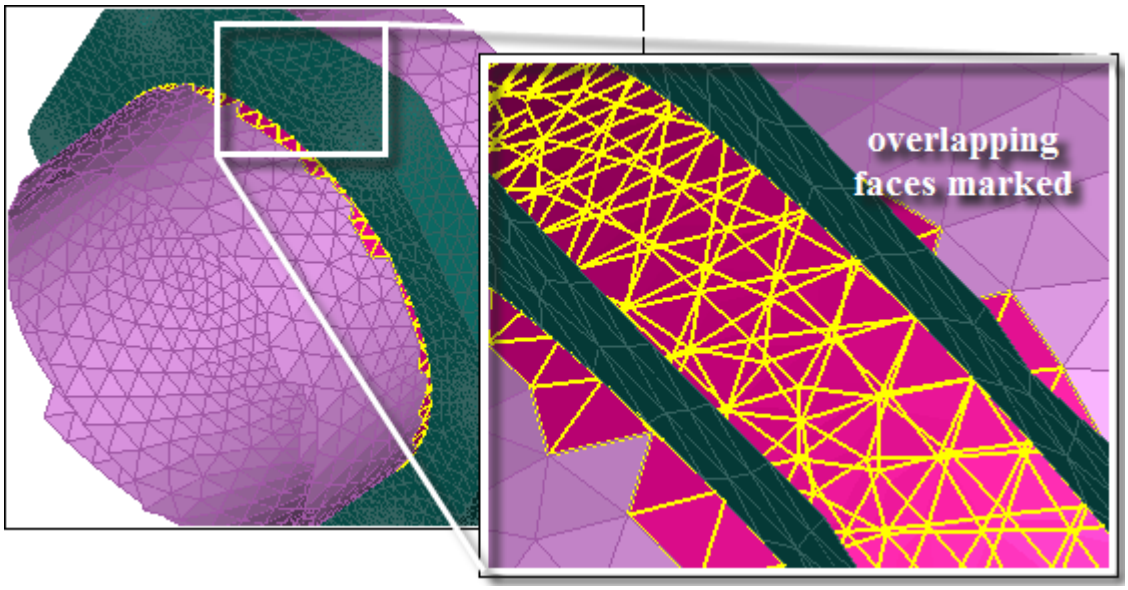
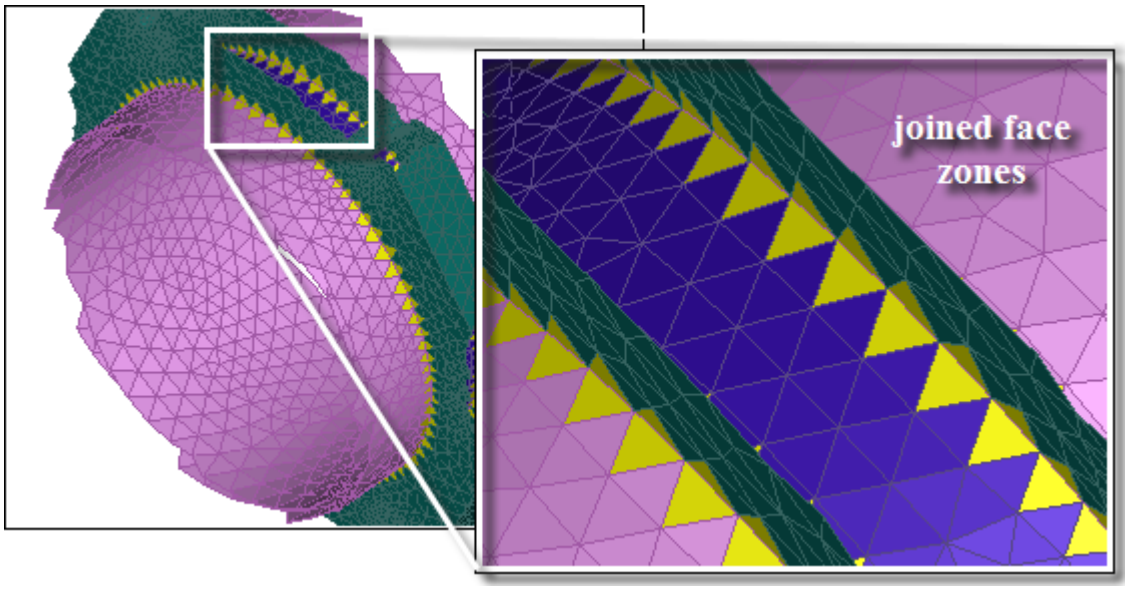


Figure 7.21: Connected Surfaces After Join



An example of the intersect operation is shown in Figure 7.22: Intersecting Surfaces (p. 631) and Figure 7.23: Connected Surfaces After Intersect (p. 631). The intersecting faces are marked based on the parameters specified. The intersecting face zones are connected and locally remeshed.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.22: Intersecting Surfaces

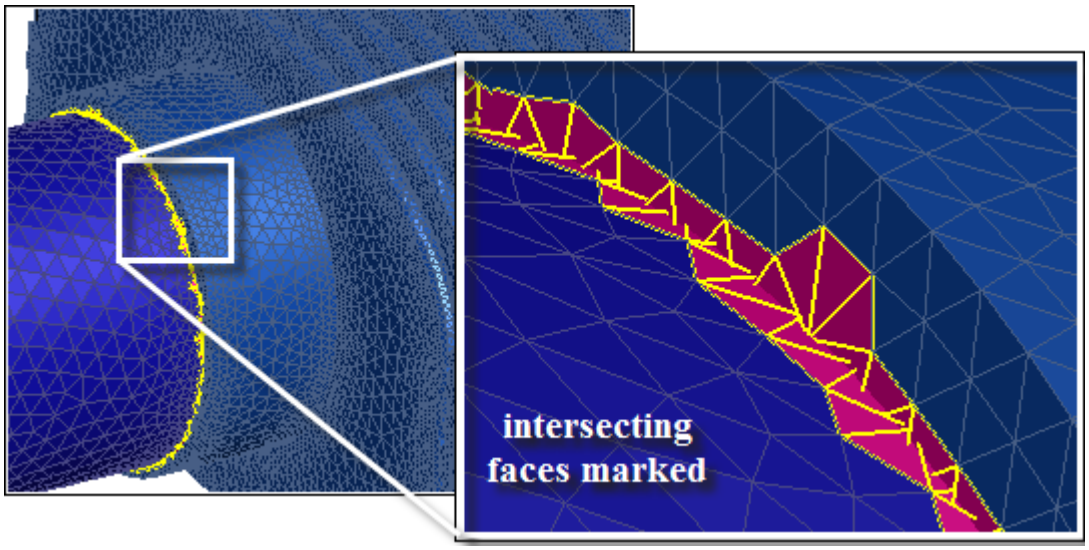
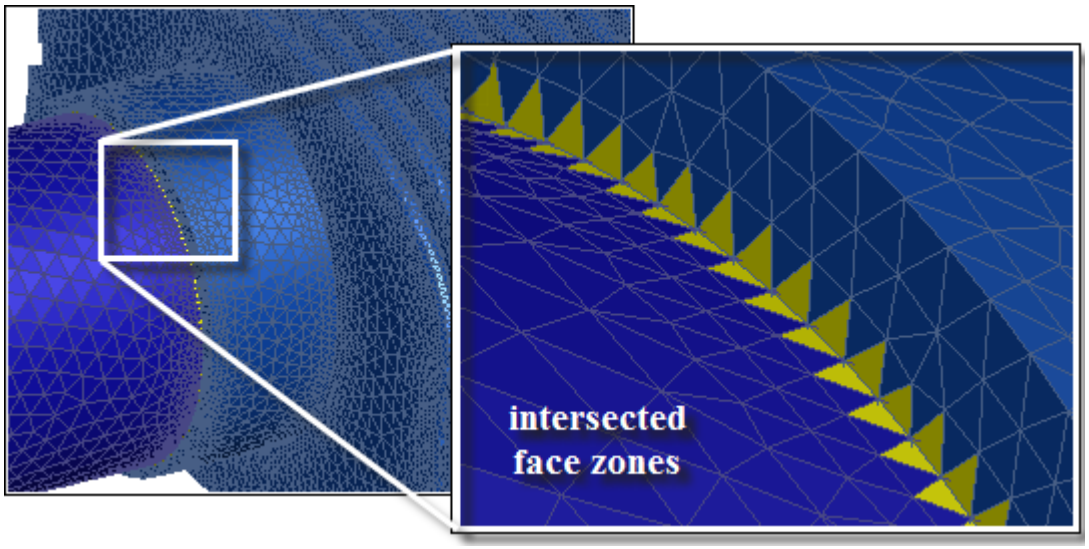


Figure 7.23: Connected Surfaces After Intersect



Important:

After any **Join** or **Intersect** operation, remesh is called automatically. To disable the post-remesh operation, use one of the text commands:

```
/objects/join-intersect/controls/remesh-post-intersection? no
/boundary/remesh/controls/intersect/remesh-post-intersection? no
```

The following options are available for connecting object zones:

7.4.4.1. Using the Join/Intersect Dialog Box

7.4.4.2. Using the Join Dialog Box

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.4.3. Using the Intersect Dialog Box

7.4.4.1. Using the Join/Intersect Dialog Box

The **Join/Intersect** dialog box is accessed from the context-sensitive menu under any **Mesh Object**, or any **Face Zone Labels** under a **Mesh Object**. For best results, the following practices are highly recommended:

- Prepare clean input prior to using **Join/Intersect**:
  - Resolve free, duplicate and sliver faces using the **Diagnostics** tools. See Diagnostic Tools (p. 625).
  - Identify self-proximity locations and separate it into different zones using the **Diagnostics** tools.
  - Remove gaps between the face zones to be joined to get clean contacts (overlaps). Alternatively, choose **Absolute Tolerance** with a value greater than or equal to the known gap if joining over smaller gaps.
  - Have similar mesh sizes (density) on face zones to be joined.
- If wrapped meshes are used with **Join**, use the **High** option together with a scaled **Size Function** to produce a very fine mesh. After join, all the surfaces may be remeshed with the default size function.

The **Join/Intersect** dialog box can be used as follows:

1. Select **Join/Intersect...** from the context-sensitive menu available for the **Mesh Object** selected in the tree to open the **Join/Intersect** dialog box. The corresponding **Face Zone Labels** and **Face Zones** are listed.

   Note:

   Face Zone Label represents either the collective name of the face zones in the mesh object or, in the case of conformal faceted import with one object per part, the bodies of the part.

2. Choose **Join** or **Intersect** in the **Operation** group box.

   Tip:

   Always use the join operation first, then intersect.

   a. For the **Join** operation, **Skip Bad Joins** is enabled by default (in the **Controls** group box).

   b. Specify an appropriate value for**Min. Dihedral Angle**.

      The **Skip Bad Joins** option enables joined pairs to be undone locally if a self intersection is found or if the smallest dihedral angle is less than the specified value.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. In the **Parameters** group box, specify the decision thresholds for **Angle** (default is 40 degrees) and **Tolerance** (default is relative tolerance of 0.05, or 5% of local triangle size).

   Check **Absolute Tolerance** to specify tolerance is in the same dimensional units as the geometry.

4. Use the global **Join** (**Intersect**), under the **Face Zones** list, to perform the selected **Operation** on selected **Face Zones** using the specified **Parameters**.

   Alternatively, use **Local** controls to perform the operations on *pairs* of overlapping or intersecting face zones.

   Local is recommended for geometry where you do not know the tolerances.

   Note:

   Length scales of geometric features should be smaller than the local triangle size.

   a. Click **Find Pairs** in the **Local** group box.

   b. Click **First** (and subsequently **Next**) to step through each pair of surfaces individually. The display is automatically limited to the area of overlap/intersection since **Bounded View** is enabled by default.

   c. Click **Mark Faces** to view the overlapping/intersecting faces.

   d. Click **Join** (or **Intersect**) in the **Local** group box to perform the selected **Operation** on the selected pair of surfaces.

   Important:

   - Since the join operation separates the two overlapping face zones and keeps only one separated face zone, it is possible that the separated zone may overlap with a third zone, so you should repeat the operation iteratively until no overlaps are found. The intersection operation does not separate the face zones; still, it is a good practice to repeat the operation iteratively until no intersections are found.
   - The surface should be inspected for self-intersections, duplicates, and so on after **Join** or **Intersect** operations using the **Diagnostics** tools.

Important:

Quads are not supported by **Join** or **Intersect**.

7.4.4.2. Using the Join Dialog Box

Use the **Join** dialog box to join overlapping face zones based on selections in the graphics window.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the overlapping face zones in the graphics window and click . Alternatively, use the hot-key **Ctrl**+**T** to invoke the miscellaneous tools and then **Ctrl**+**J** to open the **Join** dialog box.
2. Specify an appropriate **Tolerance** (default is relative tolerance of 0.05, or 5% of local size). Alternatively, enable **Absolute Tolerance** and specify a value greater than or equal to the known gap.
3. Click **Mark**. After verifying the marked faces, select the face zones to be joined again.
4. Click **Join**.

   Click **Undo** if the results of the join operation are unsatisfactory. The operation can be undone until the next mesh operation or until the **Join** dialog box is closed.

7.4.4.3. Using the Intersect Dialog Box

Use the **Intersect** dialog box to intersect face zones based on selections in the graphics window.

1. Select the intersecting face zones in the graphics window and click . Alternatively, use the hot-key **Ctrl**+**T** and then **Ctrl**+**I** to open the **Intersect** dialog box.
2. Specify an appropriate **Tolerance** (default is relative tolerance of 0.05, or 5% of local size). Alternatively, enable **Absolute Tolerance** and specify the absolute value.
3. Enable **Separate** if you need to separate the intersecting zones at the edge loop of intersection.
4. Retain the default selection of **Ignore Parallel Faces** and enter an appropriate value for **Parallel Angle**. The default value is 5.
5. Click **Mark**. After verifying the marked faces, select the face zones to be intersected again.
6. Click **Intersect**.

   Click **Undo** if the results of the intersect operation are unsatisfactory. The operation can be undone until the next mesh operation or until the **Intersect** dialog box is closed.

7.4.5. Advanced Options

Objects define the domain to be meshed. The following advanced options are available:

7.4.5.1. Object Management
7.4.5.2. Removing Gaps Between Mesh Objects
7.4.5.3. Removing Thickness in Mesh Objects
7.4.5.4. Sewing Objects

7.4.5.1. Object Management

The **Manage Objects** dialog box contains the following tabs:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Definition**: Used to create, modify, or delete objects.
- **Operations**: Used for manipulations such as merge, extract edges, create groups, and separate faces.
- **Transformations**: Used to rotate, scale or translate objects.

Right-click on **Model** in the tree and select **Object Management...** from the menu to access the **Manage Objects** dialog box.See Using the Manage Objects Dialog Box (p. 609).

Note:

When the face zones and/or edge zones comprising an object are deleted, the object definition will be updated. If all the zones associated with an object are deleted, the empty object will be deleted as well.

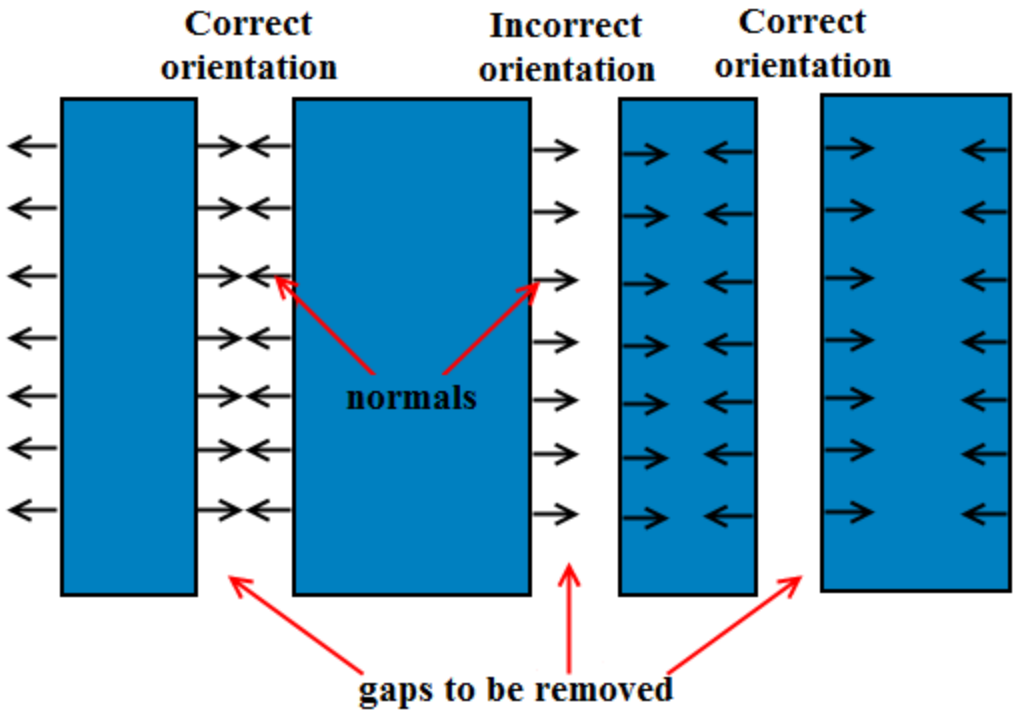
7.4.5.2. Removing Gaps Between Mesh Objects

Gaps between mesh objects can be removed using the options in the **Remove Gaps** dialog box. The gaps can be between surfaces (face-face) or between a surface and an edge (face-edge).

1. Right click on the object in the Outline View. Select **Advanced** → **Remove Gaps...** from the context sensitive menu.
2. Select **Remove Gaps Between Objects** in the **Operation** list.
3. Specify an appropriate value for the **Min. Gap Distance**, **Max. Gap Distance**, and **Percentage Margin**.
4. Specify an appropriate value for **Critical Angle**. The critical angle is the maximum angle between the faces constituting the gap to be removed.
5. **Ignore Orientation** is enabled by default. If the thickness of any of the object selected is less than the **Max. Gap Distance**, then you can disable **Ignore Orientation**. In this case the orientations of the normals will be considered. Ensure that, the normals in the gaps to be removed are appropriately oriented (Figure 7.24: Orientation of Normals in Gap (p. 636)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.24: Orientation of Normals in Gap



6. Select the appropriate option for feature edge extraction (**none**, **feature**, or **all**) and specify the **Extract Angle** to be used.

7. Select the type of gap in the **Gap Type** list (**Face-Face** or **Edge-Face**).

   - For face-face gap removal, select the appropriate option for projection in the **Order** list. You can choose to project the faces to the object of higher priority (**Low To High Priority**) or to the object of lower priority (**High to Low Priority**).

8. Click **Mark** to see the faces marked for projection.

9. Click **Remove** to remove the gaps between the objects selected.

Figure 7.25: Removing Gaps Between Objects—Face-Face Option (p. 637) shows an example where a face-face gap between mesh objects has been removed.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.25: Removing Gaps Between Objects—Face-Face Option

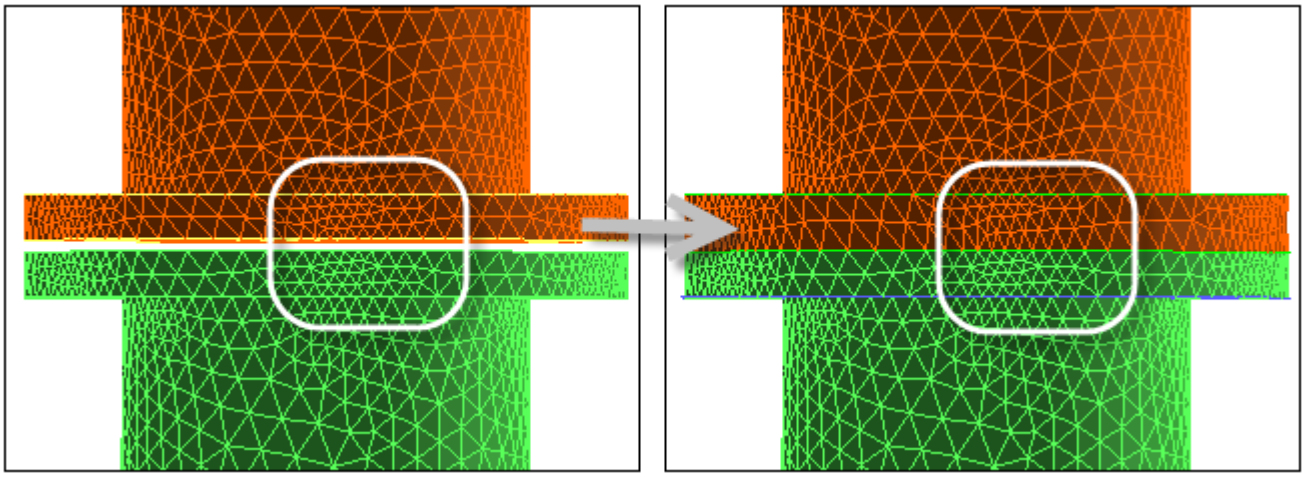
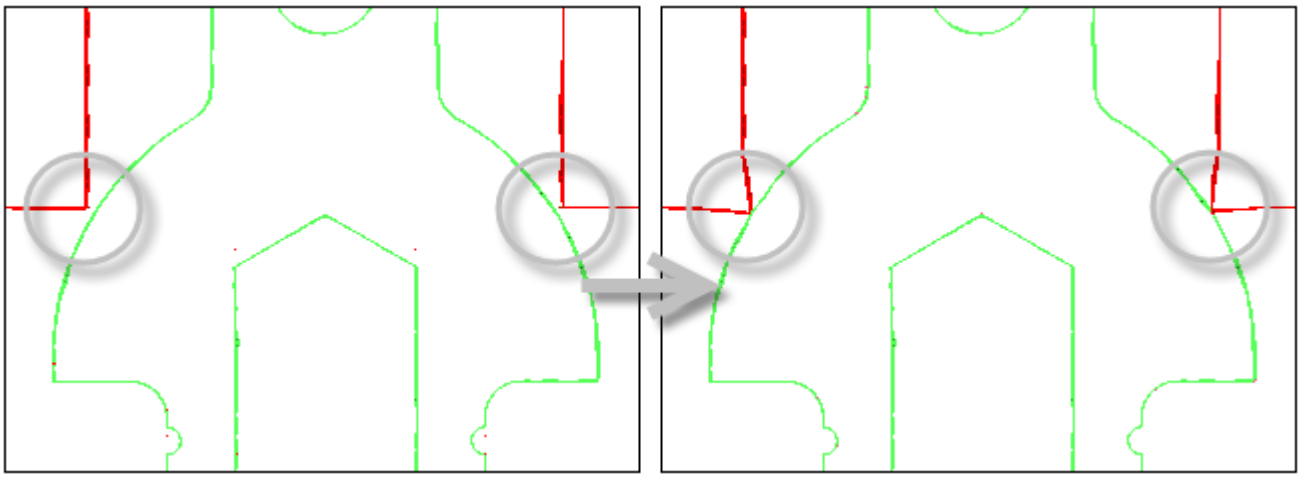


Figure 7.26: Removing Gaps Between Objects—Face-Edge Option (p. 637) shows an example where a face-edge gap between mesh objects has been removed.

Figure 7.26: Removing Gaps Between Objects—Face-Edge Option



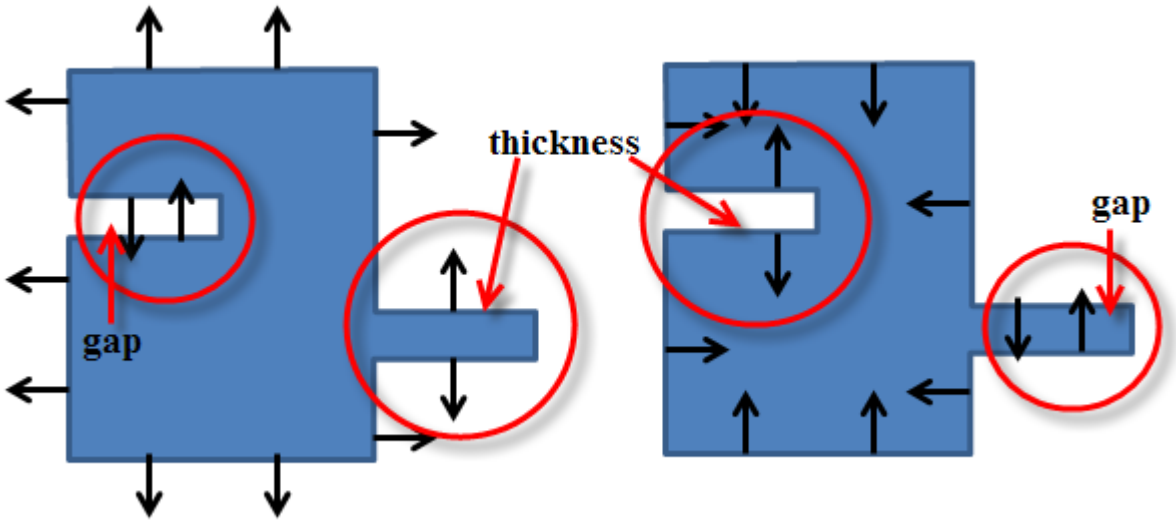
7.4.5.3. Removing Thickness in Mesh Objects

The thickness across a mesh object can be removed by projecting the close surfaces to a mid-surface. During the thickness removal operation, the object face zones will be separated in order to project the close surfaces to the mid-surface and the separated zones will be merged back after the projection. The options for thickness removal are available in the **Remove Gaps** dialog box.

Configurations can be distinguished as gaps or thicknesses based on the orientation of the normals on the mesh object (Figure 7.27: Gap and Thickness Configurations (p. 638)). The normals across a gap configuration point toward each other, while for a thickness configuration, the normals point away from each other.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.27: Gap and Thickness Configurations



When using the option for thickness removal, ensure that the object normals are appropriately oriented depending on the configurations to be removed (see Figure 7.27: Gap and Thickness Configurations (p. 638)).

The generic procedure for removing thickness in objects using the **Remove Gaps** dialog box is as follows:

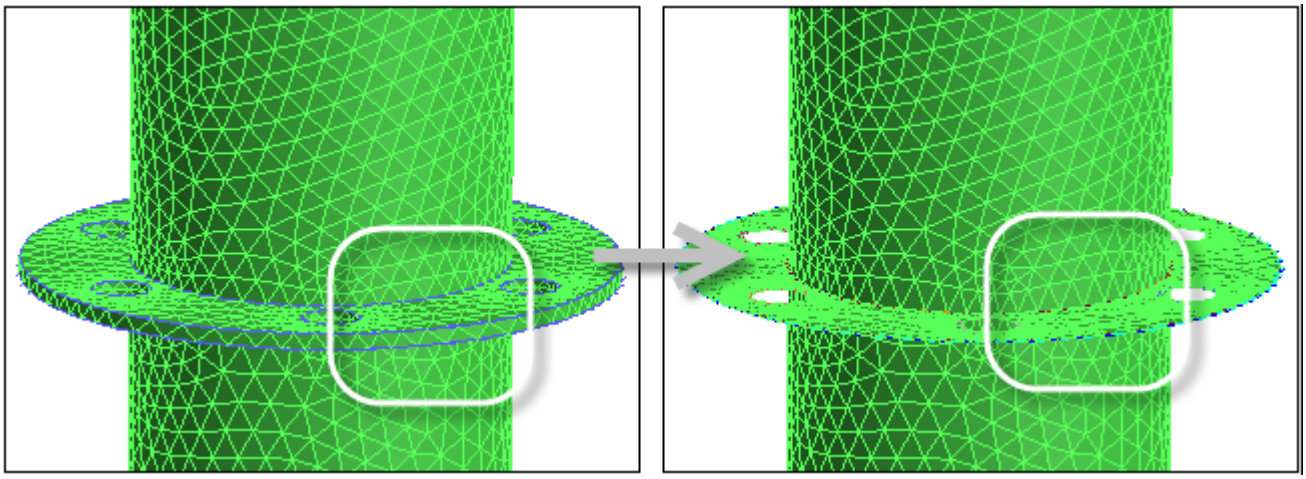
1. Right click on the object in the Outline View. Select **Advanced** → **Remove Gaps...** from the context sensitive menu.
2. Select **Remove Thickness In Objects** in the **Operation** list.
3. Specify an appropriate value for the **Max. Gap Distance** and **Percentage Margin**.
4. Specify an appropriate value for **Critical Angle**. The critical angle is the maximum angle between the faces constituting the thickness to be removed.
5. Select the appropriate option for feature edge extraction (**none**, **feature**, or **all**) and specify the **Extract Angle** to be used.
6. Click **Remove** to remove the thickness in the objects selected.

> **Note:**
>
> The thickness removal operation involves separation and merging back of face zones, which may affect the mesh quality. It is recommended that you save the mesh before proceeding.

Figure 7.28: Removing Thickness in Objects (p. 639) shows an example where the thickness in the mesh object has been removed.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.28: Removing Thickness in Objects



7.4.5.4. Sewing Objects

The sewing operation is a face connecting operation applicable to mesh objects. Disconnected assemblies can be connected to create the conformal, triangular surface mesh on the specified objects. This operation creates conformal mesh between bodies and produces a topology-verified model. Normals are also reoriented suitably for further prism meshing.

The procedure to Sew objects is:

1. Select the mesh objects in the Outline View. Right-click and select **Advanced** → **Sew...** from the context sensitive menu.
2. Ensure that the mesh objects for the sew operation are selected in the **Objects** list in the **Sew** dialog box.
3. Specify the name for the mesh object to be created for the selected mesh objects.
4. If desired, disable the **Improve** option.

   If disabled, you may need to improve the surface mesh quality of the mesh object created using the options in the **Diagnostic Tools** dialog box and the **Improve** dialog box.

Figure 7.29: Mesh Objects to be Connected (p. 640) shows an example with disconnected mesh objects. The sewing operation creates the conformal surface mesh by connecting the individual objects into a single mesh object (Figure 7.30: Mesh Object Created by Sewing (p. 640)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.29: Mesh Objects to be Connected

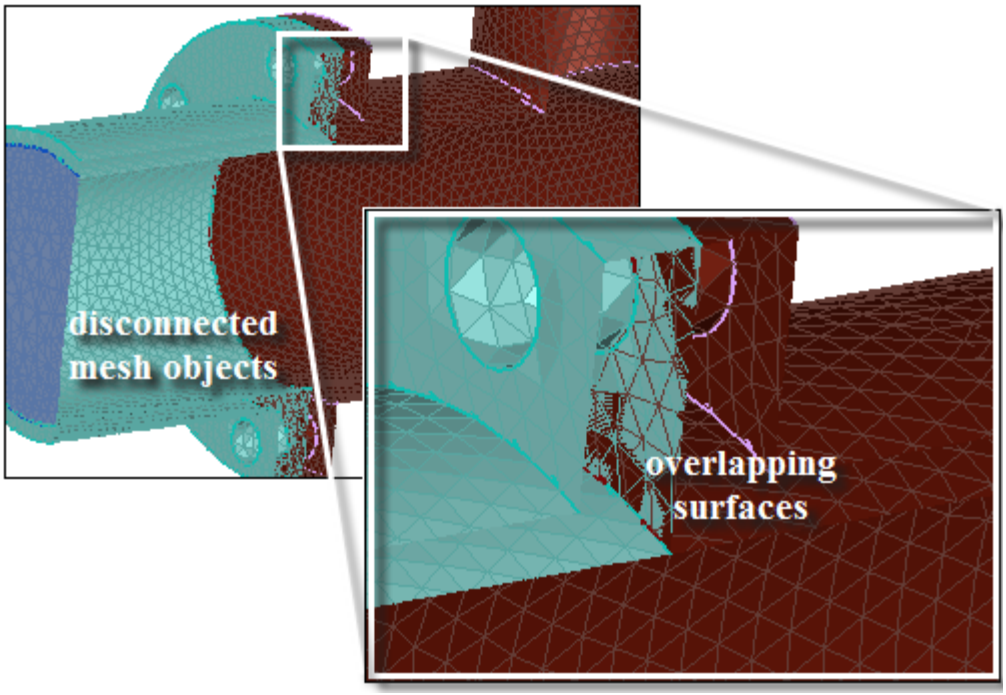
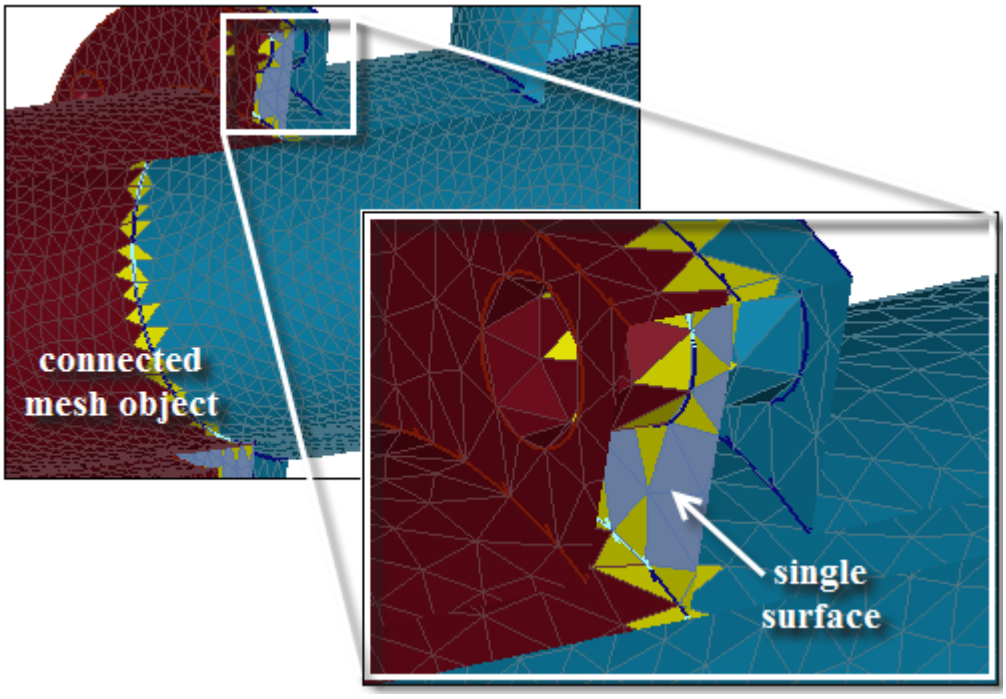


Figure 7.30: Mesh Object Created by Sewing



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

A face zone group is automatically created for the mesh object created by the **Sew** operation. This face zone group is prefixed by **_mesh_group**, and enables easy selection of mesh object face zones for various operations (improve, smooth, and so on).

Note:

The sew operation is not needed for a single mesh object, or flow volume (only) extraction type problems. You can use the options in the **Diagnostic Tools** dialog box and then use the **Improve** dialog box or the command `/objects/improve-object-quality` to improve the surface mesh (see Improving the Mesh Objects (p. 718)).

You can use the command `/objects/sew/sew` to connect the mesh objects. Specify the objects to be connected and the name for the mesh object to be created.

7.4.5.4.1. Resolving Thin Regions

Surfaces in close proximity constitute thin regions in the mesh. Examples of thin regions include sharp corners, trailing edge configurations, and so on, which may not be recovered accurately enough during the sewing operation and surface elements may span between nodes on the proximal surfaces.

You can use the command `/objects/sew/set/include-thin-cut-edges-and-faces` to allow better recovery of such configurations during the sewing operation.

7.4.5.4.2. Processing Slits

In cases containing baffles, when the **shrink-wrap** method is used for the object wrapping operation, the mesh object is created with nearly overlapping surfaces representing the baffle. Though the surfaces are nearly overlapping, there is a numerically a small angle between them (parallel face angle). Such configurations constitute slits in the mesh object.

The command `/objects/sew/set/process-slits-as-baffles?` enables you to collapse the nearly overlapping surfaces corresponding to the baffle when the sew operation is performed to create the mesh object. Specify the maximum slit thickness relative to the minimum size specified and the parallel face angle between the faces comprising the slit when `process-slits-as-baffles` is enabled.

Note:

When `process-slits-as-baffles` is enabled for the **Sew** operation, it is recommended that you check the mesh object created for voids or pockets. Use the command `/objects/merge-voids` to remove any voids or pockets created in the mesh object (see Removing Voids (p. 641) for details).

7.4.5.4.3. Removing Voids

The command `/objects/merge-voids` enables you to remove voids or pockets created in the mesh object.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.5. Object-Based Volume Meshing

This chapter describes the volume meshing approach with which you can generate a tetrahedral, hexcore, polyhedral, or hybrid volume mesh, with or without inflation layers, for mesh objects created from the imported geometry. You need to create a conformally connected surface mesh using the object wrapping or join/intersect operations before generating the volume mesh. See Object-Based Surface Meshing (p. 616) for details.

The tools to complete these steps can be found in the context-sensitive menus in the **Outline View** tree. Instructions are described in the following sections.

7.5.1. Volume Mesh Process

7.5.2. Volumetric Region Management

7.5.3. Generating the Volume Mesh

7.5.4. Cell Zone Options

Note the following alternatives:

- You can use the CutCell mesher to directly create a hex-dominant volume mesh for the geometry based on meshing objects created. See Generating the CutCell Mesh (p. 810) for details.
- With the Rapid Octree mesher, you can use meshing objects to directly create a mostly isotropic volume mesh that is designed to scale on clusters that include thousands of cores, as described in Generating Rapid Octree Meshes (p. 828).

7.5.1. Volume Mesh Process

When the conformal, connected surface mesh is ready, use the Outline View tree to navigate the volume meshing workflow as follows:

1. Set up **Volumetric Regions**.

   Volumetric regions are finite, contiguous domains ready to receive a volume mesh. You use the **Compute** menu option to initialize the volumetric regions within a mesh object. If you perform operations such as merging or deleting, you should **Validate** the regions before computing the volume mesh. The **Update** option will recompute existing volumetric regions while preserving names and types.

   Tools to compute, update and validate the volumetric regions, as well as tools to modify selected regions, are accessible via the context-sensitive menus. For a full description of available operations, see Volumetric Region Management (p. 643).

2. Fill the volumetric regions collectively or individually, as appropriate.

   a. Right-click on any individual volumetric region and you can access menus to setup **Scoped Prism**, **Tet**, or **Hexcore** volume meshing parameters, as appropriate. Descriptions of meshing parameters are found in Prism Meshing Options for Scoped Prisms (p. 775), Generating Tetrahedral Meshes (p. 780), and Generating the Hexcore Mesh (p. 795).

      After setting mesh parameters, you can compute the mesh in an individual volumetric region using the **Auto Fill Volume** option. See Meshing Regions Selectively Using Auto Fill Volume (p. 650).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. To apply meshing parameters and compute the volume mesh collectively for all volumetric regions of the selected object(s), use the **Auto Mesh** option in the **Volumetric Regions** context menu. See Meshing All Regions Collectively Using Auto Mesh (p. 648).

   You can also access the **Auto Mesh** option from the object level context menu, and access the scoped prism option directly using **Setup Scoped Prisms...** from the **Volumetric Regions** menu.

3. Examine the volume mesh and verify the mesh quality.

   a. The computed volume mesh is placed into **Cell Zones**. You can use the context menus to draw the cell zones individually or collectively to examine the mesh. Cell zones may be merged or deleted if necessary. Volumetric regions are not affected if cell zones are deleted. Refer to Cell Zone Options (p. 651).

   b. Improve the mesh quality using the **Auto Node Move** tool, if needed. See Moving Nodes (p. 519).

When you are satisfied with the quality of the volume mesh, prepare the mesh for transfer to solution mode. Right click on **Model** at the top of the tree and select **Prepare for Solve** from the context menu. Operations such as deleting dead zones, deleting geometry objects, deleting edge zones, removing face/cell zone name prefixes and/or suffixes, and deleting unused faces and nodes are performed during this operation.

Important:

In object-based workflows, merging cell zones requires that they be in the same volumetric region.

To merge cell zones that cannot be in the same volumetric region because they are not contiguous, you will need to first delete the object(s) only, and then use the **Manage Cell Zones** dialog box.

Note:

When you generate a poly mesh, node weights for node-based gradients are enabled by default. For postprocessing, this setting can improve the accuracy of the displayed results near wall edges when you are displaying contours on the poly mesh. When you transfer the poly mesh to solution mode, a message will notify you that this interpolation is enabled. You can disable it by setting the `/display/set/nodewt-based-interp?` command to **no**.

7.5.2. Volumetric Region Management

Volumetric regions are finite, contiguous domains that are ready for volume meshing. Volumetric regions are computed from the conformal surface mesh and material points. If a volumetric region is changed, you should validate or update the regions prior to filling with the volume mesh.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can use the context-sensitive menu available for **Volumetric Regions** for managing the regions collectively:

- Use **Compute** to initialize the regions; **Validate** to confirm the finite, contiguous domains after changes; and **Update** to preserve names and types while recalculating regions.
- You can use the **Select Regions** and **External Baffles** sub-menus to assist in selecting or visualizing regions based on type.
- Object-based meshing parameters are set and the volumetric regions filled using the **Auto Mesh** tool. The **Setup Scoped Prisms...** tool is a direct link to the **Scoped Prisms** dialog box.
- You can get surface mesh statistics using the **Summary** option, and a listing of region composition using the **Info** option.

Individually, you can use the context-sensitive menu for a named volumetric region to modify and examine the region, as well as set up regional mesh parameters.

- Use **Draw**, **Draw Options**, and **List Selection** to assist in selecting and visualizing the regions.
- The tools available in the **Diagnostics...**, **Change Type**, **Manage**, and **Remesh Faces** menus are used to modify the volumetric region.
- You can set up regional mesh parameters using the **Scoped Prisms**, **Tet**, and **Hexcore** menus. The **Auto Fill Volume...** tool is used to generate the volume mesh for the individual region.
- **Delete Cells** will remove any existing volume mesh while preserving the volumetric region information.
- You can get surface mesh statistics using the **Summary** option and the region's composition using the **Info** option.

The available options are described in detail in the following sections.

7.5.2.1. Computing and Verifying Regions

7.5.2.2. Volumetric Region Operations

7.5.2.1. Computing and Verifying Regions

The controls described here are found in the context-sensitive menu accessed by right-clicking on **Volumetric Regions**.

Compute

Volumetric regions need to be computed prior to object based volume meshing. The **Compute Regions** dialog box contains options for computing volumetric regions to produce ready to mesh domains. The computing of regions includes topology checks, re-orienting of normals, and baffle identification and handling before generating the volume mesh.

The volumetric region computation can also handle overlapping zones, so long as they are not multi-connected.

1. Select **Compute...** to open the **Compute Regions** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Select the material points in the **Material Points** selection list and click **OK**.

Note:

When regions are computed, region names and types will be based on the face zone labels of the mesh object selected. If regions are recomputed, all previous region names and types will be over written.

Validate

Volumetric regions must be closed, water-tight domains. If individual regions are modified after initially computing volumetric regions, the modified regions need to be validated before proceeding to volume meshing. Regions can be modified using operations such as merging, deleting, or renaming them or changing the region type. Also, the region may also be modified if scoped prism settings are applied.

To manually validate the volumetric regions, select **Validate**. Also, the regions will be validated when the volume mesh is generated.

Update

Some of your modifications may involve changing the name or type for the volumetric region. You can recalculate the volumetric region without affecting the region name or type using the **Update** control.

Select Regions

Connects to a sub-menu that allows you to select multiple regions based on type (fluid, solid, dead, or all).

External Baffles

Connects to a sub-menu that allows you to display, select, or remove the baffle(s) from the volumetric regions.

Setup Scoped Prisms

Provides direct access to **Scoped Prisms** dialog box in which you can define inflation layers to specific entities in the model.

Auto Mesh

Use this menu item for setting up volume fill meshing parameters and computing the volume mesh in the full object.

Summary

From the **Volumetric Regions** menu, you can use **Summary** to obtain an overall summary with counts of face zones, all, free, multi, and duplicate faces, the maximum skewness and number of faces with skewness > 0.85.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Info

From the **Volumetric Regions** menu, you can use **Info** to obtain a listing of all regions with type, volume, material point, and face zones.

7.5.2.2. Volumetric Region Operations

The context-sensitive menus for individual volumetric regions offer several options to select or draw regions. With one or more regions selected in the tree:

- Use the **Draw** menu item to draw the highlighted region(s), replacing what was previously displayed.
- Use the **Draw Options** submenu to add, remove, highlight, or select the region(s) to (from) the display.

  This submenu also provides options to draw, add, remove, highlight, or select regions based on boundary type (**Walls** or **Baffles**), interface type (**Fluid-Fluid** interface, **Fluid-Solid** interface, or **Solid-Solid** interface), or volume type (**Filled Volume** or **Empty Volume**).
- Use the **List Selection** submenu to add, remove, or select the region(s) to (from) a selection list in a dialog box.

  This submenu also provides options to select, add, or remove regions based on boundary type (**Walls** or **Baffles**), interface type (**Fluid-Fluid** interface, **Fluid-Solid** interface, or **Solid-Solid** interface), or volume type (**Filled Volume** or **Empty Volume**).

Operations to modify the selected volumetric region(s) include the following:

- Face connectivity and quality based diagnostic tools are available for fluid volumetric regions. Select **Diagnostics...** in the menu available for the region selected to open the **Diagnostic Tools** dialog box.
- Use the **Change Type** option to change the region type in the corresponding drop down list.
- Merge multiple regions using the **Manage** → **Merge...** option in the menu. Enter the **New Region Name** and select the type for the merged region in the **New Region Type** drop-down list in the **Merge Regions** dialog box.

  Note:

  If there are shared face zones, merging regions will delete the shared face zones. However, if there are cell zones associated with the regions, then merging the regions will not delete the shared face zones. In this case, the shared face zones will be deleted when the cell zones are deleted.

- Rename individual regions using the **Manage** → **Rename** option in the menu. Enter the **New Region Name** in the **Rename Region** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Delete regions using the **Manage** → **Delete** option in the menu.

  Tip:

  Deleting regions may cause face zones to be deleted. It is recommended that the region type be changed to dead instead of deleting the region.

- Use the **Remesh Faces** option in the menu to remesh the face zones based on the existing size field.

Volume meshing operations include options to set up meshing parameters, fill the volume based on the parameters set, and manipulate the cell zones in the regions.

- Use the **Scoped Prisms** → **Set...** option to open the **Scoped Prisms** dialog box for setting scoped prism controls.
- Use the **Tets** → **Set...** option to open the **Tet** dialog box for setting tetrahedral mesh controls.
- Use the **Hexcore** → **Set...** option to open the **Hexcore** dialog box for setting hexcore mesh controls.
- Use the **Auto Fill Volume...** option to open the **Auto Fill Volume** dialog box to create the volume mesh for the selected volumetric regions based on the meshing parameters set.
- Use the **Delete Cells** option to delete the cell zones of the volumetric regions.

The **Summary** and **Info** options print information to the console about the surface mesh and the region's composition, respectively, for the selected volumetric region(s).

7.5.3. Generating the Volume Mesh

When regions have been computed and verified, you can proceed to generate the volume mesh. Use the process described below that is best suited to your workflow or problem:

7.5.3.1. Meshing All Regions Collectively Using Auto Mesh

7.5.3.2. Meshing Regions Selectively Using Auto Fill Volume

Note:

The workflow for object-based volume meshing does not support partial meshing within a region. Operations such as prisms generation only, Tet-initialization only, Hexcore only, etc., require a domain-based workflow.

Important:

In the object-based volume meshing procedure, by default a backup of the surface mesh is created before volume meshing starts. To restore the surface mesh at any point, select **Restore Faces** in the context-sensitive menu for the mesh object. When you select the **Restore Faces** option, the current object face zones and cell zones will be deleted.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To disable the backup, set the `/mesh/auto-mesh-controls/backup-object` command to **no**. In that case, you will not be able to restore the surface mesh using the **Restore Faces** option.

There may be a difference in the initial volume mesh generated for an object and that generated after restoring the object surface mesh due to differences in the order of zones/entities processed during volume meshing.

7.5.3.1. Meshing All Regions Collectively Using Auto Mesh

The **Auto Mesh** dialog box contains options for generating the volume mesh for all computed volumetric regions of the mesh object.

The volume mesh can be generated as follows:

1. Open the **Auto Mesh** dialog box from the context-sensitive menu available by right-clicking on any mesh object or its **Volumetric Regions** or **Cell Zones** branch in the tree.

   You can also use the **Mesh** → **Auto Mesh** menu item to open the **Auto Mesh** dialog box.

2. Ensure that the mesh object is selected in the **Object** drop-down list.

   Note:

   If you open the **Auto Mesh** dialog box from the context-sensitive menu in the tree, the mesh object to which the cell zones, or volumetric regions, belong is automatically selected.

3. Enable/disable **Keep Solid Cell Zones** and **Keep Dead Cell Zones**, as required.

4. Select the appropriate option in the **Grow Prisms** drop-down list in the **Boundary Layer Mesh** group box.

   a. Retain the default selection of **none** if you do not need to grow prism layers for the current meshing approach.

   b. Select **scoped** if you want to specify object-based prism controls. Click **Set...** to open the **Scoped Prisms** dialog box and define the prism controls for the mesh object. Refer to Prism Meshing Options for Scoped Prisms (p. 775) for details.

      Tip:

      You can save your scoped prism controls to a file (*.pzmcontrol) for use in batch mode, or read in a previously saved scoped prism file.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. Select **zone-specific** if you want to specify zone-specific prism parameters. Click **Set...** to open the **Prisms** dialog box and specify the zone-specific prism parameters. Refer to Procedure for Creating Zone-based Prisms (p. 753) for details.

Note:

Poly meshing does not support zone-specific prisms.

5. Select the appropriate quad-tet transition elements from the **Quad Tet Transition** list. Click the **Set...** button to open the **Pyramids** dialog box or the **Non Conformals** dialog box (depending on the selection) and specify the appropriate parameters. Refer to Creating Pyramids (p. 735) and Creating a Non-Conformal Interface (p. 739) for details.

Note:

The **Quad Tet Transition** options are not applicable to poly mesh.

6. Select the appropriate option from the **Volume Fill** list. Click the **Set...** button to open the **Tet**, **Hexcore**, or **Poly** dialog box (depending on the selection). Specify the appropriate parameters. Refer to Initializing the Tetrahedral Mesh (p. 788), Refining the Tetrahedral Mesh (p. 790), Controlling Hexcore Parameters (p. 798), and Generating Polyhedral Meshes (p. 804) for details.

Note:

- The **No Fill** option is not available when a mesh object is selected for volume meshing.
- Some operations such as prisms generation only, Tet-initialization only, Hexcore only, are not available for object-based volume meshing. Similarly, options under the **Zones** group in the **Tet** and **Hexcore** dialog boxes, that require a zone-based workflow, have no effect for object-based volume meshing.

7. Specify the appropriate **Volume Fill Options**.

a. For the **Tet** or **Poly** methods, set the following:

- Select the appropriate option for **Cell Sizing**.
  - **Size Field** specifies that the cell size is determined based on the current size-field.
  - **Geometric** specifies that the cell size in the interior of the domain is obtained by a geometric growth from the closest boundary according to the growth rate specified.

    Set the **Growth Rate** required.
- Specify the **Max Cell Length**. Click **Compute** to compute the maximum cell size based on the mesh object.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. For the **Hexcore** method, set the following:

- Select the appropriate option for **Type**.

  Retain the default selection of **Cartesian** or select the **Octree** type.

- Set the number of **Buffer Layers** and **Peel Layers**.

- Specify the **Max Cell Length** for the Cartesian approach. Click **Compute** to compute the maximum cell size based on the mesh object.

  Specify the **Min Cell Length** for the Octree approach.

8. Enable or disable additional **Options** as desired.

   - Enable **Merge Cell Zones within Regions** to create a single cell zone within a region, or disable to keep the cell zones separate.

9. Click **Mesh** in the **Auto Mesh** dialog box.

Alternatively, you can use the command `/mesh/auto-mesh` to generate the mesh automatically. Specify a mesh object name for object-based auto mesh; if no name is given, face zone based auto mesh is performed. Specify the mesh elements to be used when prompted. Specify whether to merge the cells into a single zone or keep the cell zones separate. For face zone based meshing, specify whether to automatically identify the domain to be meshed based on the topology information.

Note:

You can specify the meshing parameters for the mesh elements (prisms, pyramids or non-conformals, tet, or hex) using either the respective dialog boxes or the associated text commands prior to using the `auto-mesh` command. Commands for poly meshes are located under the `mesh/poly` menu.

7.5.3.2. Meshing Regions Selectively Using Auto Fill Volume

The **Auto Fill Volume** dialog box contains options for generating the volume mesh in selected volumetric regions for mesh objects.

The volume mesh can be generated as follows:

1. In the tree, expand **Volumetric Regions** and select the regions to be meshed.

2. Set up the regional meshing parameters for the selected regions.

   a. Specify scoped prism controls for the boundary layer mesh, if applicable. Select **Scoped Prisms** → **Set...** in the context-sensitive menu to open the **Scoped Prisms** dialog box. Refer to Prism Meshing Options for Scoped Prisms (p. 775) for details.

   b. Specify tetrahedral mesh or hexcore mesh parameters, as applicable.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select **Tets** → **Set...** to open the **Tet** dialog box. Refer to Initializing the Tetrahedral Mesh (p. 788), and Refining the Tetrahedral Mesh (p. 790) for details.

Select **Hexcore** → **Set...** to open the **Hexcore** dialog box. Refer to Controlling Hexcore Parameters (p. 798) for details.

Note:

When you click the **Compute** button in the **Tet** or **Hexcore** dialog box, you will be asked if the maximum cell size is to be computed based on the mesh object selected. Click **Yes** to recompute the cell sizes based on the mesh object.

Alternatively, open the **Auto Fill Volume** dialog box from the menu for the selected regions. Click the **Set...** button in the **Boundary Layer Mesh** and **Volume Fill** group box to set the scoped prism and tetrahedral/hexcore mesh parameters, respectively.

3. Set the boundary layer mesh and volume fill options in the **Auto Fill Volume** dialog box.

   a. Enable **Grow Scoped Prisms** if scoped prism controls have been set up.

   b. Select **Tet** or **Hexcore** in the **Volume Fill** list.

4. Click **Mesh** in the **Auto Fill Volume** dialog box.

Repeat this procedure for any empty volumetric regions until all regions are filled.

7.5.4. Cell Zone Options

The context-sensitive menu for **Cell Zones** contains options for visualizing and manipulating all cell zones collectively:

- Use **Draw All Boundaries** to draw all the cell zone boundaries or **Draw All Cells in Range** to draw all the cells within a specified range (for either unreferenced or referenced zones). The range can be set in the **Bounds** group in the ribbon.
- Use **Auto Mesh...** to access the **Auto Mesh** tool for volume meshing.
- Use **Auto Node Move...** to access the **Auto Node Move** tool for improving the mesh quality by moving nodes.
- Use the **Select Cell Zones** sub-menu for selection of all cell zones as well as selection by type (fluid, solid, or dead).
- Use **Delete All** to delete all cell zones, if needed.
- Use **Summary** to obtain an overall summary of cell count and quality or **Info** to obtain individual cell zone type and cell zone count by type.

The context-sensitive menu for individual cell zones offers several options to select or draw cell zones.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Use **Draw Boundaries** to draw the boundaries of the selected zones, or **Draw Cells in Range** to draw the cells within a specified range.
- Use the **Draw Options** submenu to add, remove, highlight, or select the cell zone(s) to (from) the display.
- Use the **List Selection** submenu to add, remove, or select the cell zone(s) to (from) a selection list in a dialog box.
- Use **Auto Node Move...** to access the **Auto Node Move** tool for improving the mesh quality of the selected zone by moving nodes.
- Use **Merge** to combine two or more selected cell zones into a single zone.

  Important:

  In object-based workflows, merging cell zones requires that they be in the same volumetric region.

  To merge cell zones that cannot be in the same volumetric region because they are not contiguous, you will need to first delete the object(s) only, and then use the **Manage Cell Zones** dialog box.

- Use **Delete** to delete the cell zones, if needed. Volumetric regions are not affected if cell zones are deleted.
- Use **Summary** to obtain a summary of cell count and quality or **Info** to obtain individual cell zone type and cell zone count by type.

7.6. Manipulating the Boundary Mesh

The first step in producing an unstructured mesh is to define the shape of the domain boundaries. You can create a boundary mesh in which the boundaries are defined by triangular or quadrilateral facets using a preprocessor (GAMBIT or a third-party CAD package) and then create a mesh in the meshing mode in Fluent. You can also modify the boundary mesh to improve its quality and create surface meshes on certain primitive shapes. The following sections discuss mesh quality requirements and various techniques for generating an adequate boundary mesh for numerical analysis.

7.6.1. Manipulating Boundary Nodes
7.6.2. Intersecting Boundary Zones
7.6.3. Modifying the Boundary Mesh
7.6.4. Improving Boundary Surfaces
7.6.5. Refining the Boundary Mesh
7.6.6. Creating and Modifying Features
7.6.7. Remeshing Boundary Zones
7.6.8. Faceted Stitching of Boundary Zones
7.6.9. Triangulating Boundary Zones
7.6.10. Separating Boundary Zones

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.6.11. Projecting Boundary Zones
7.6.12. Creating Groups
7.6.13. Manipulating Boundary Zones
7.6.14. Manipulating Boundary Conditions
7.6.15. Creating Surfaces
7.6.16. Removing Gaps Between Boundary Zones
7.6.17. Using the Loop Selection Tool

7.6.1. Manipulating Boundary Nodes

Manipulation of boundary nodes is an effective way to influence the boundary mesh quality. Operations for deleting unwanted boundary nodes can be performed in the **Merge Boundary Nodes** dialog box or with the associated text commands.

7.6.1.1. Free and Isolated Nodes

7.6.1.1. Free and Isolated Nodes

The mesh generation algorithm does not permit duplicate nodes; that is, two nodes that have the same Cartesian coordinates. Duplicate nodes may be created by mesh generators that preserve the node locations at adjoining edges of adjacent surfaces, but give different labels to the two sets of nodes. The nodes and edges at which these surfaces meet are termed free nodes and free edges.

Figure 7.31: Free Nodes

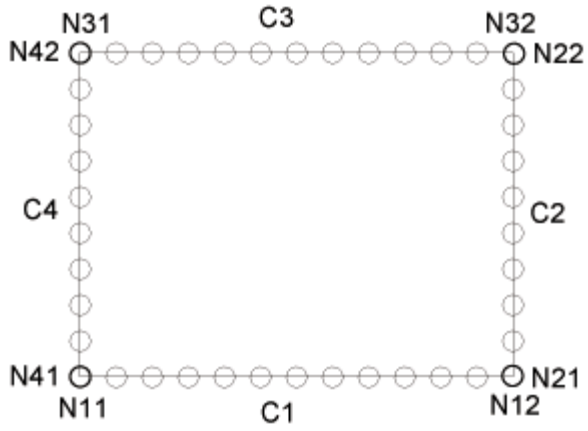


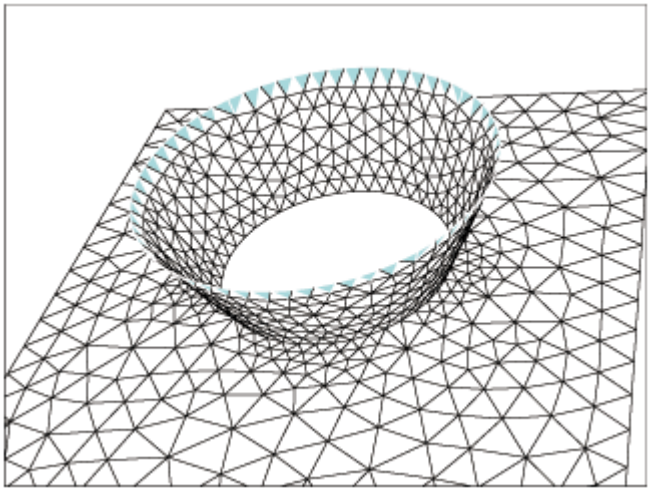
Figure 7.31: Free Nodes (p. 653) shows a simple geometry in which the free nodes are marked. Although the node at the end of curve C1 (N12) is located in the same position as the node at the beginning of curve C2 (N21), each is a free node because it is not connected in any way to the adjoining curve.

Though the nodes have the same location, the mesher knows only that they have different names, and not that the curves meet at this location. Similarly, a free edge is a surface edge that is used by only one boundary face. To check the location of free nodes, use the **Display Grid** dialog box. Free edges are acceptable when modeling a zero-thickness wall ("thin wall") in the geometry (for example, Figure 7.32: Example of a Thin Wall (p. 654)). Isolated nodes are nodes that are not used

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

by any boundary faces. You can either retain these nodes to influence the generation of the interior mesh (see Inserting Isolated Nodes into a Tet Mesh (p. 728)), or delete them.

Figure 7.32: Example of a Thin Wall



7.6.2. Intersecting Boundary Zones

You can connect triangular boundary zones in the geometry using the set of intersection operations available. These can be used to resolve intersections, overlaps, and for connecting zones along the free boundaries.

7.6.2.1. Intersecting Zones

7.6.2.2. Joining Zones

7.6.2.3. Stitching Zones

7.6.2.4. Using the Intersect Boundary Zones Dialog Box

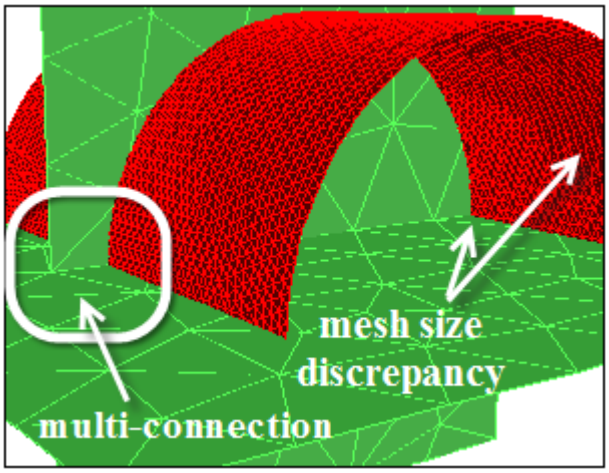
7.6.2.5. Using Shortcut Keys/Icons

7.6.2.1. Intersecting Zones

The intersect option is used to connect intersecting tri boundary zones. Figure 7.33: Intersection of Boundary Zones (p. 655) shows an example where the intersect option can be used. The connection is made along the curve (or line) of intersection of the boundary zones. You can use the intersection operation on multi-connected faces as well as in regions of mesh size discrepancy.

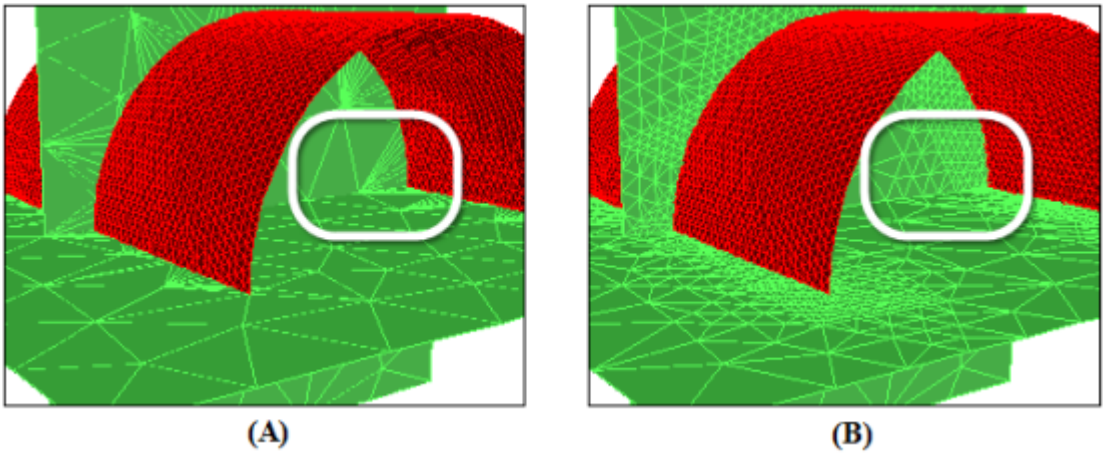
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.33: Intersection of Boundary Zones



To intersect boundary zones with a gap between them, specify an appropriate **Tolerance** value. All zones with the distance between them less the specified tolerance value will be intersected. The tolerance can be either relative or absolute. When intersecting zones having different mesh sizes, you can use the **Refine** option to obtain a better graded mesh around the intersecting faces (see Figure 7.34: Intersection (A) Without and (B) With the Refine Option (p. 655)).

Figure 7.34: Intersection (A) Without and (B) With the Refine Option



7.6.2.2. Joining Zones

The join option is used to connect two overlapping tri boundary zones (Figure 7.35: Partially Overlapping Faces (p. 656)). The overlapping areas of both the boundary zones are merged and the mesh at the boundary of the region of overlap is made conformal. To join surfaces that are on top of each other but not connected (with a small gap), specify an appropriate **Tolerance** value. The portion of the surfaces within the tolerance value will be joined. The boundary zone selected in the **Intersect Tri Zone** defines the shape of the combined surface in the overlap region. The shape in the **With Tri Zone** may be changed to perform the join operation.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.35: Partially Overlapping Faces

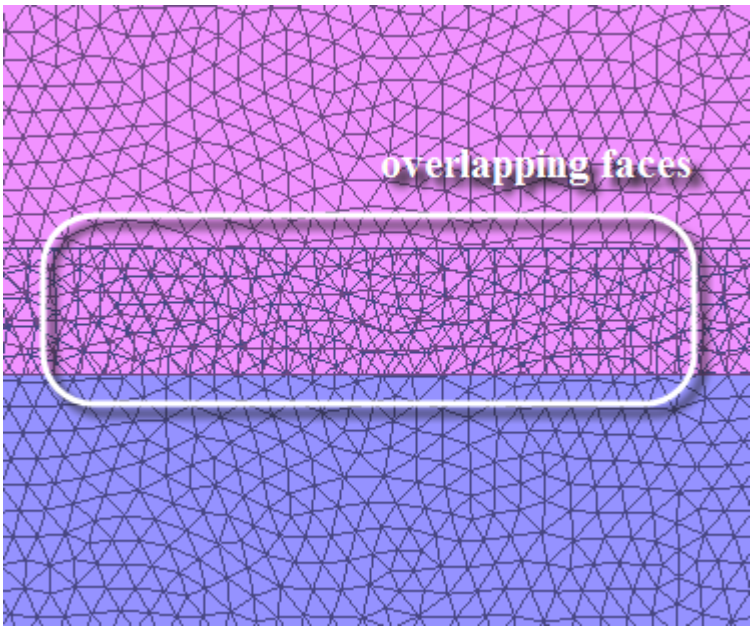
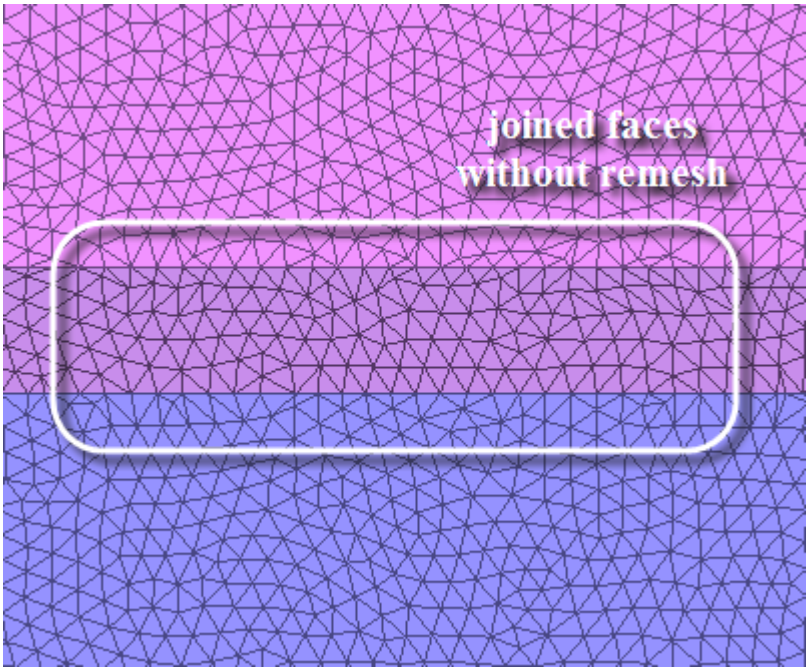


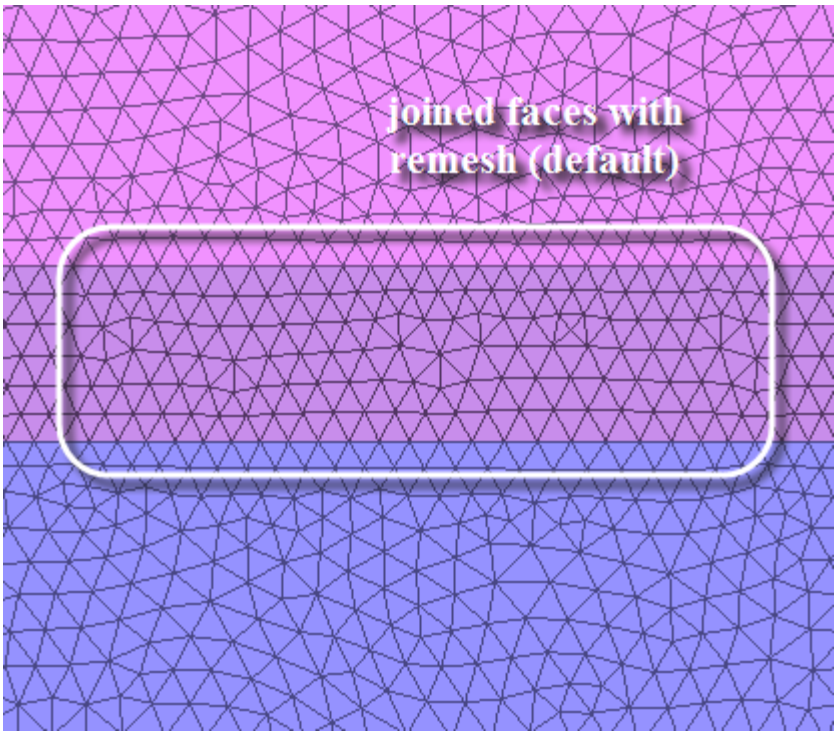
Figure 7.36: Joining of Overlapping Faces (p. 656) and Figure 7.37: Remeshing of Joined Faces (p. 657) show the overlapped faces after joining and after remeshing the joined faces.

Figure 7.36: Joining of Overlapping Faces



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.37: Remeshing of Joined Faces



Tip:

In case of completely overlapping face zones, you may need to separate the zones and then join individual pairs. In such cases, you may use the `/boundary/check-duplicate-geom` command to delete the duplicate face zone instead.

7.6.2.3. Stitching Zones

The stitch option is used to connect two tri boundary zones along their free edges. You cannot use this option to connect the surfaces at a location other than the free edges in the mesh. Gaps within the given tolerance are closed using nearest point projection.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.38: Nearest Point Projection for Stitching

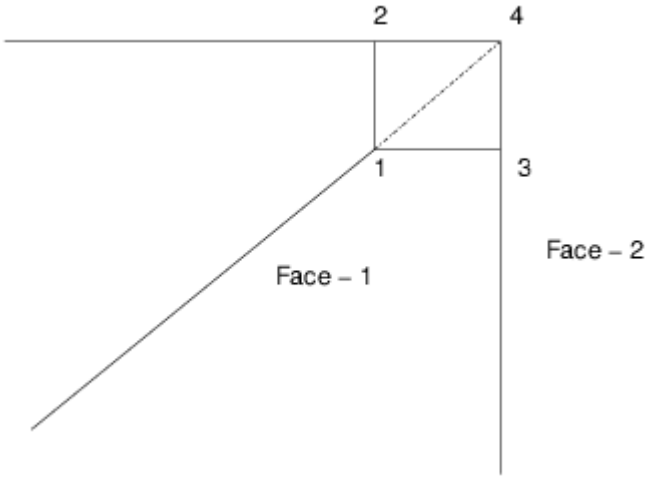
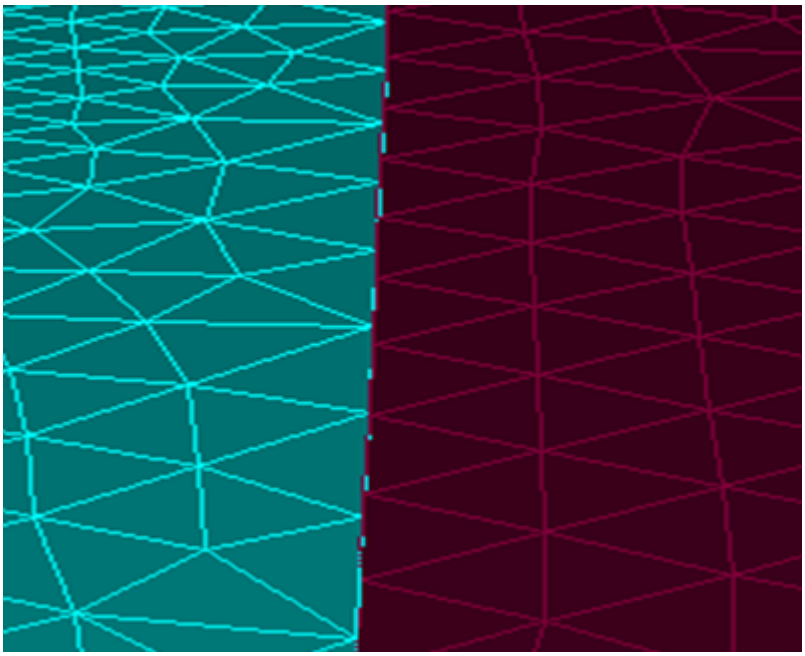


Figure 7.38: Nearest Point Projection for Stitching (p. 658) shows a cut through the two surfaces, `Face-1` and `Face-2`, that are separated by a gap. The points of nearest projection will determine the location of the intersection curve. Therefore, `point-1` will be connected to `point-2` or `point-3`. All three connect operations allow a small gap (within the tolerance specified) between the intersecting boundary zones; however, the gap should not distort the shape of the geometry.

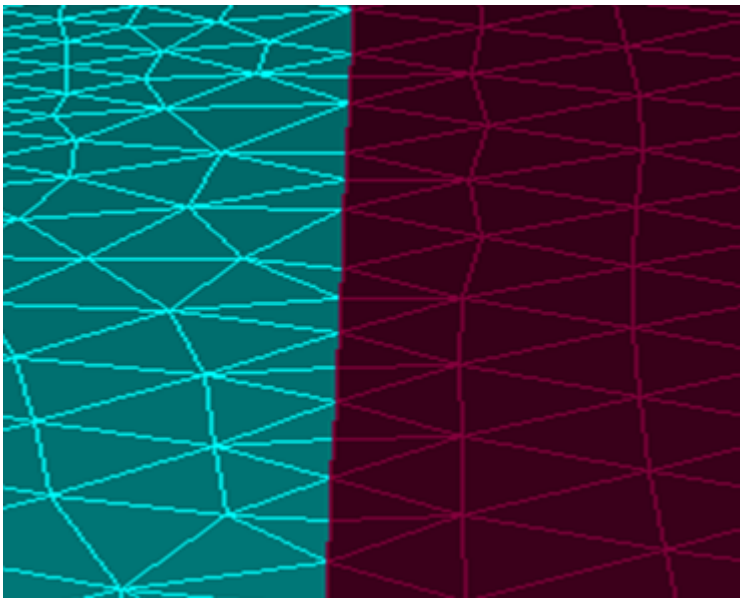
Figure 7.39: Surfaces Before Stitch (p. 658) and Figure 7.40: Surfaces After Stitch (p. 659) show the surfaces before and after the stitch operation, respectively

Figure 7.39: Surfaces Before Stitch



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.40: Surfaces After Stitch



7.6.2.4. Using the Intersect Boundary Zones Dialog Box

In general, all three connect operations calculate the intersection curve (or line) between the two surfaces to be connected. The intersection curve is constructed as follows:

- **Intersect** constructs the curve as the intersection of two zones.
- **Join** constructs the curve as the outer boundary of the overlapping region within the specified tolerance of the two surfaces.
- **Stitch** constructs the curve along the free boundaries and within the specified tolerance.

The intersection curve is remeshed with a local spacing calculated from the intersecting surfaces. The intersection curve is inserted into the surfaces and will result in a retriangulation of the surfaces along the intersection curve.

The `/boundary/remesh/remesh-overlapping-zones` command extracts the boundary edge zones from the zone to imprint. The intersecting curve is inserted into the zones. During the insertion, the zones are retriangulated.

After any connect operation, remesh is called automatically. To disable the post-remesh operation, use the text command:

`/boundary/remesh/controls/intersect/remesh-post-intersection?` **no**

To perform any of the intersection operations, do the following:

1. Select the boundary zones you want to intersect in the **Intersect Tri Zone** list.
2. Select the boundary zones with which you want to intersect the selected boundary zone in the **With Tri Zone** list.
3. Select the appropriate operation from the **Operation** list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Specify the appropriate **Tolerance** value (if the surfaces have a gap between them).
5. Enable **Absolute Tolerance**, **Refine**, or **Separate** as appropriate.
6. Click **Mark**.

   The faces that will be affected by the intersection operation are highlighted. This also helps you decide whether the specified tolerance is appropriate.
7. Click **Apply**.

7.6.2.5. Using Shortcut Keys/Icons

You can use the **Join** dialog box to join overlapping face zones based on selections in the graphics window.

Select the overlapping face zones in the graphics window and click . Alternatively, use the hot-key **Ctrl**+**T** to invoke the miscellaneous tools and then **Ctrl**+**J** to open the **Join** dialog box.

The detailed procedure is described in Using the **Join** Dialog Box (p. 633).

You can use the **Intersect** dialog box to join overlapping face zones based on selections in the graphics window.

Select the overlapping face zones in the graphics window and click . Alternatively, use the hot-key **Ctrl**+**T** to invoke the miscellaneous tools and then **Ctrl**+**I** to open the **Intersect** dialog box.

The detailed procedure is described in Using the **Intersect** Dialog Box (p. 634).

7.6.3. Modifying the Boundary Mesh

Tools are available for making boundary repairs, enabling you to perform primitive operations on the boundary mesh, such as creating and deleting nodes and faces, moving nodes, swapping edges, merging and smoothing nodes, collapsing nodes, edges, and faces, splitting faces, and moving faces to another boundary zone.

7.6.3.1. Using the Modify Boundary Dialog Box
7.6.3.2. Operations Performed: Modify Boundary Dialog Box
7.6.3.3. Locally Remeshing a Boundary Zone or Faces
7.6.3.4. Moving Nodes

7.6.3.1. Using the Modify Boundary Dialog Box

This section describes the generic procedure for modifying the boundary mesh using the **Modify Boundary** dialog box. In addition to the **Modify Boundary** dialog box, you may also use the **Display Grid** dialog box during the modification process.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Display the boundary zones that you want to modify, using the **Display Grid** dialog box. If you need to modify many zones, display them one at a time to make the graphics display less cluttered.

2. Select the type of entity you want to select with the mouse: **edge**, **node**, **position**, and so on in the **Filter** list in the **Modify Boundary** dialog box.

3. Select the entities you want to operate on using the mouse-probe button (a right-click, by default) in the graphics window.

   You can select individual entities one at a time, or select a group of them by defining a selection region. See Controlling the Mouse Probe Function (p. 551) for details. The selected entities will appear in the **Selections** list in the **Modify Boundary** dialog box.

4. Click the appropriate **Operation** button to perform the boundary modification.

   The mesh is automatically re-displayed after the operation is performed, enabling you to immediately see the effect of your change.

5. Repeat the process to perform different operations on different entities.

> **Warning:**
>
> Save the mesh periodically as it is not always possible to undo an operation.

7.6.3.2. Operations Performed: Modify Boundary Dialog Box

You can perform the following operations using the **Modify Boundary** dialog box:

Creating Nodes

To create nodes, do the following:

1. Select the required positions (or enter node coordinates explicitly in the **Enter Selection** box).

2. Select **node** in the **Filter** list or press **Ctrl**+**N**.

3. Click **Create** or press **F5**.

Creating Faces

To create a face, do the following:

1. Select 3 or 4 nodes and the optional zone.

   Use the hot keys **Ctrl**+**N** and **Ctrl**+**F** to select node and face as **Filter**, respectively.

2. Click **Create** or press **F5**.

While creating a face:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you do not select a zone, the new face will be in the same zone as an existing face that uses one of the specified nodes.
- If the nodes you use to create a face are used by faces in different zones, make sure that the new face is in the right zone.
- If you create a face and it is in the wrong zone, use the rezoning feature.

Creating a Zone

To create a new zone, do the following:

1. Select **zone** in the **Filter** list (or **Ctrl**+**Z**).
2. Click **Create** or press **F5**. The **Create Boundary Zone** dialog box will open, prompting you for the zone name and type.
3. Specify the name and zone type as appropriate in the **Create Boundary Zone** dialog box.
4. Click **OK**. The new zone will automatically be added to the **Selections** list in the **Modify Boundary** dialog box.

Deleting a Node/Face/Zone

To delete the nodes or faces, do the following:

1. Select the nodes or faces or zones to be deleted.
2. Click **Delete** or press **Ctrl**+**W** on the keyboard.

Merging Nodes

To merge nodes, do the following:

1. Select the two nodes to be merged.
2. Click **Merge** or press **F9**.

Important:

The first node selected is retained; the second node is merged onto the first node.

Tip:

You can merge multiple pairs of nodes by selecting an even number of nodes, in the correct order, before clicking **Merge** (or pressing **F9**). The first and second nodes will be merged, then the third and fourth, and so on.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Moving Nodes

To move the node to any position in the domain, do the following:

1. Select **node** in the filter list (or **Ctrl**+**N**).
2. Select the node you want to move.
3. Choose **position** in the filter list (or **Ctrl**+**X**).
4. Select the position coordinates or click the position in the graphics window to which you want to move the selected node.
5. Click **Move To**.

To move the node by specifying the magnitude of the movement, do the following:

1. Select **node** in the filter list (or **Ctrl**+**N**).
2. Select the node you want to move.
3. Enter the magnitude by which you want to move the selected node.
4. Click **Move By**.

Rezoning Faces

To rezone one or more faces, do the following:

1. Select the faces you want to move.
2. Select the zone to which you want the selected faces to move.
3. Click **Rezone** (or **Ctrl**+**O**). You can create a zone if you need to move faces to a new zone.

Collapsing Nodes/Edges/Faces

To collapse nodes, edges, or faces, do the following:

1. Select the appropriate **Filter**.
2. Select the two nodes (or edges/faces) you want to collapse.
3. Click **Collapse** (or **Ctrl**+**^**).

While collapsing:

- If a pair of nodes is selected, both the nodes are moved towards each other (at the midpoint) and collapsed into a single node.
- If an edge is selected, the two nodes of the edge collapse onto the midpoint of the edge and surrounding nodes are connected to the newly created node.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If a triangular face is selected, a new node is created at the centroid of the triangle and the selected triangular face gets deleted.

Note:

You can also collapse multiple pairs of entities by selecting multiple entities before clicking **Collapse**. Ensure that an even number of entities is selected. The first and the second entity will be collapsed, then the third and the fourth, and so on.

Smoothing Nodes

To smooth nodes, do the following:

1. Select the nodes you want to smooth.
2. Click **Smooth** or press **F6**.

   The node will be placed at a position computed from the average of the surrounding nodes.

Splitting Edges

To split edges, do the following:

1. Select the edges you want to split.
2. Click **Split** or press **F7**.

   All faces sharing the edge will be split into two faces.

If you select multiple edges and they share a face, the split operation may not be completed. If the face referenced by the split operation for the second edge has already been split by the operation on the first edge, the second split operation will not be possible because the referenced face that no longer exists. If this happens, redisplay the mesh and reselect the edge that was not split. In such cases it may also be easier to split the face rather than the edge.

Splitting Faces

To split faces, do the following:

1. Select the faces you want to split.
2. Click **Split** or press **F7**.

   Each triangular face will be split into three faces by adding a node at the centroid. Each quadrilateral face will be split into two triangular faces.

Perform edge swapping after this step to improve the quality of the local refinement.

Swapping Edges

To swap an edge of a triangular face, do the following:

1. Select the edges as appropriate.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Click **Swap** or press **F8**. If the triangular boundary face on which you perform edge swapping is the cap face of a prism layer, the swapping will automatically propagate through the prism layers.

   Note:

   Edge swapping is not available for quadrilateral faces.

Finding Coordinates of the Centroid

To find the location of the centroid of a face or cell, do the following:

1. Set **Filter** to **face** or **cell** as appropriate.
2. Select the face or the cell using the mouse probe button.
3. Click the **Centroid** button (or **Ctrl**+**L**).

   The face or cell centroid location will be printed in the console window.

Calculating Distance Between Entities

To compute the distance between two entities, do the following:

1. Set **Filter** to **face**, **edge**, or **cell** as appropriate.
2. Select the two entities.
3. Click **Distance** (or **Ctrl**+**D**).

   For example, if an edge (or face or cell) and a node are selected, the distance between the centroid of the edge (or face or cell) and the node is computed and printed to the console window.

Projecting Nodes

To reconstruct features in the surface mesh that were not captured in the surface mesh generation, project selected nodes onto a specified line or plane.

The **Create Boundary Zone** dialog box will appear automatically when you create a new face zone. You can specify the name and type of the new zone in this dialog box.

To project nodes, do the following:

1. Define the projection line or plane. For a projection line, select two entities and for a projection plane, select three entities. If edges, faces, or cells are selected, their centroidal locations will be used.
2. Click **Set** (or **Ctrl**+**S**) and the projection line or plane will be shown in the graphics display.
3. Select the nodes to be projected.
4. Click **Project** (or **Ctrl**+**P**).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The selected nodes are projected onto the projection line or plane that you defined with the **Set** operation.

Simplifying Boundary Modification

The following functions simplify the boundary modification process:

Finding the Worst/Marked Faces

You can display faces in the descending order of their quality as follows:

1. To find the face having the worst quality in the mesh, select **Quality Limit** and click **First** or press **F11**.

   The worst face will be displayed in the graphics window and its quality and zone ID are reported in the console.

   - The longest edge of the face and the node opposite it are selected, and the display is limited to the neighborhood of the highly skewed face.
   - If the mesh has not been displayed, the worst face, its quality, and the zone in which it lies will be reported (in the console).

2. Click **Next** (or the right-arrow key).

   The face having the next higher quality will be displayed in the graphics window. When you subsequently click **Next**, the face having the next higher quality (after that of the previously displayed or reported face) will be displayed or reported.

3. Click **Reset** (or the left-arrow key) to reset the display to the worst quality element.

You can also find the worst face within a subset of zones by activating a group containing the required zones using the **User Defined Groups** dialog box and then clicking **First**. When you click the **Next** button after activating a particular group, the face having the next higher quality within the active group will be displayed. (Ensure that the **global** group is activated to have all the zones available.)

To display the marked faces in succession, do the following:

1. Select **Mark** and click **First** (or **F11**) to find the first marked face.

   The face will be displayed in the graphics window.

2. Click **Next** (or the right-arrow key).

   The next marked face is displayed in the graphics window.

3. Click **Reset** (or the left-arrow key) to reset the display to the first marked face.

   You can use the `/bounday/unmark-selected-faces` command (or **Ctrl**+**U**) to unmark the faces.

To improve the quality of the face, use the following operations:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Use the **Smooth** operation to smooth the node opposite the longest face.
- Use the **Merge** operation to collapse the shortest edge of the face, merging the other two edges together. The longer of the remaining two edges is retained, while the shorter one is merged with the other edge.
- Use the **Swap** operation to swap the selected edge.
- Use the **Split** operation to refine the face by bisecting the selected edge.

If the selected entities are not appropriate, clear them, choose the appropriate items, and perform the desired operations.

Deselecting a Selected Entity

If you select an inappropriate entity, you can click on it again in the graphics window to deselect it. You can also select it in the **Selections** list in the **Modify Boundary** dialog box and click **Clear**. You can use **F2** to deselect all entities selected.

> **Warning:**
>
> Deselect operations are performed only on the items selected in the **Selections** list.

Undoing an Operation

To undo an operation, click **Undo** or press **F12**. In some cases, a particular sequence of operations cannot be undone. Hence, make sure that you save the mesh periodically between the modifications.

Click **Undo** or press **F12** *n* times to undo the last *n* operations.

> **Warning:**
>
> The **Undo** operation is limited to the operations in the **Modify Boundary** dialog box (or the `/boundary/modify` menu). If other operations/commands are interleaved, the **Undo** operation may cause unexpected results.

Note:

You can also fix holes in the geometry. Refer to Fixing Holes in Objects (p. 711) for details.

7.6.3.3. Locally Remeshing a Boundary Zone or Faces

The **Zone Remesh** dialog box contains options for remeshing face zones selected in the graphics window. To remesh face zones, select them in the graphics window and press **Ctrl**+**Shift**+**R** to open the **Zone Remesh dialog box**.

1. Select the sizing source (**size-field** or **constant-size**).
2. Specify the feature angle to be preserved while remeshing the selected zones.
3. Specify the **Constant Size** value when the **constant-size** method is selected.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Preview** button allows you to display size boxes to visualize the proposed constant size.

4. Click **Remesh**.

The **Local Remesh** dialog box contains options for remeshing marked faces or faces based on selections in the graphics window. Select the faces in the graphics window. Press **Ctrl**+**Shift**+**J** for face marking options. After selecting/marking the faces, press **Ctrl**+**Shift**+**R** to open the **Local Remesh** dialog box.

1. Set the number of radial layers of faces to be remeshed in the **Rings** field.
2. Specify the feature angle to be preserved while remeshing the selected faces.
3. Select the sizing source (**geometric**, **size-field**, or **constant-size**).
4. Specify the **Constant Size** value when the **constant-size** method is selected.

   The **Preview** button allows you to display size boxes to visualize the proposed constant size.

5. Click **Remesh**.

7.6.3.4. Moving Nodes

To specify a distance and direction for moving a node using the **Move Nodes** dialog box, do the following:

1. Select a **Seed Node** from your model.
2. Set the number of nodes to move in the **Node Count** box.
3. Set the move distance in the **Move by** box.
4. Select a direction.

   - **Seed-Normal** moves all the nodes in parallel to the seed node normal.
   - **Local-Normal** moves each node in the direction of its own normal.
   - **Flip** moves the nodes in the opposite direction.

5. Use **Draw** to preview the direction and distance selected.

You may use the **Boundary Zones** selection list and **Boundary Zone Groups** list along with **Draw** to isolate the zone of interest in the display.

7.6.4. Improving Boundary Surfaces

The quality of the volume mesh is dependent on the quality of the boundary mesh from which it is generated. You can improve boundary surfaces to improve the overall mesh quality.

You can improve the boundary mesh by specifying an appropriate quality limit depending on the quality measure considered. You can also smooth and swap faces on the boundary surfaces to improve the mesh quality. You can use the **Boundary Improve** dialog box to improve the surfaces. You can

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

diagnostically determine the boundary mesh quality using the **Check** and **Skew** buttons available when the **Swap** option is selected.

7.6.4.1. Improving the Boundary Surface Quality

7.6.4.2. Smoothing the Boundary Surface

7.6.4.3. Swapping Face Edges

7.6.4.1. Improving the Boundary Surface Quality

You can improve the boundary surface quality using skewness, size change, aspect ratio, or area as the quality measure.

- For improving the boundary surface quality based on skewness, size change, and aspect ratio, specify the quality limit, the angle, and the number of improvement iterations. All the elements above the specified quality limit will be improved.
- For improving based on the area, collapse faces and then either swap the edges or smooth the surface. All faces having area smaller than the specified minimum absolute size will be collapsed.

You can also specify the minimum relative size (size of the neighboring entity) to be considered while using the **Collapse and Swap** option.

7.6.4.2. Smoothing the Boundary Surface

Smoothing of the surface mesh allows you to control the variation in the size of the mesh elements, thereby improving the accuracy of the numerical analysis. Smoothing is critical in regions of proximity or regions where surfaces intersect and the accuracy of the approximations used in numerical analysis techniques deteriorates with rapid fluctuations in the element size. The smoothing procedure involves relocating of the mesh nodes without changing the mesh topology.

7.6.4.3. Swapping Face Edges

Edge swapping can be used to improve the triangular surface mesh. The procedure involves checking each pair of faces that shares an edge and identifying the connecting diagonal that results in the most appropriate configuration of faces within the resulting quadrilateral. For a face considered, if the unshared node on the other face lies within its minimal sphere, the configuration is considered to be a Delaunay violation and the edge is swapped. The procedure makes a single pass through the faces to avoid cyclic swapping of the same set of edges. Thus, the edge swapping process is repeated until no further improvement is possible. At this stage, even if a few Delaunay violations exist, the differences resulting from continual swapping are marginal.

Important:

If the triangular boundary zone selected is the cap face zone of a prism layer, the edge swapping will automatically propagate though the prism layers.

7.6.5. Refining the Boundary Mesh

To use refinement regions for local refinement in some portion of the domain (for example, to obtain a high mesh resolution in the wake of an automobile), you may refine the associated boundary zones

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

as well. When you perform the local refinement, the boundary faces that border the refinement region will not be refined. It is therefore possible that you will have a jump in face size where a small cell touches a large boundary face. To improve the smoothness of the mesh, use the **Refine Boundary Zones** dialog box to appropriately refine the boundary zones that border the refinement region before performing the refinement of the volume mesh. Boundary refinement can be performed only on triangular boundary zones.

7.6.5.1. Procedure for Refining Boundary Zones

7.6.5.1. Procedure for Refining Boundary Zones

To refine boundary zones based on marked faces, do the following:

1. Open the **Refine Boundary Zones** dialog box.

   Boundary → Mesh → Refine...

2. Select **Mark** in the **Options** list and define the refinement region. Click the **Local Regions...** button to open the **Boundary Refinement Region** dialog box. Define the refinement region as appropriate.

3. Select the zones to be refined in the **Tri Boundary Zones** list.

4. Select the region to be refined in the **Regions** list. The **Max Face Area** will be updated based on the value specified in the **Boundary Refinement Region** dialog box.

5. Click **Apply** to mark the faces to be refined.

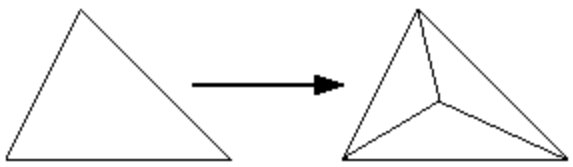
   The faces in the selected zones having face area greater than the **Max Face Area** specified will be marked.

6. Select **Refine** in the **Options** list and **Mark** in the **Refinement** group box.

7. Click **Apply**.

   The marked faces are refined by dividing them into three faces:

   Figure 7.41: Refining a Triangular Boundary Face



To refine boundary zones based on proximity, do the following:

1. Open the **Refine Boundary Zones** dialog box.

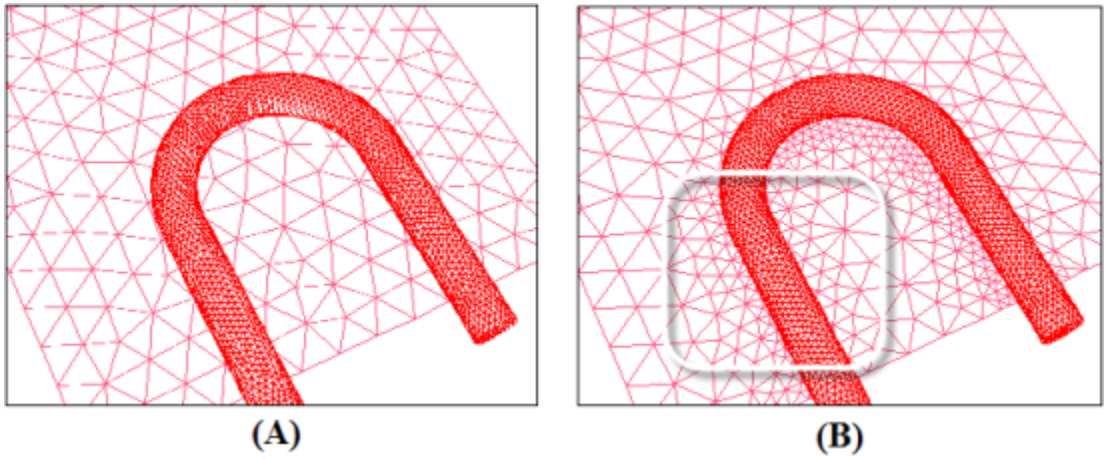
   Boundary → Mesh → Refine...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Select **Refine** in the **Options** list and **Proximity** in the **Refinement** group box.
3. Select the zone from which the proximity is to be determined in the **Tri Boundary Zones** selection list.
4. Specify the **Relative Distance** and number of refinement iterations as appropriate.
5. Click **Apply**.

   The faces in the proximity of the specified zone are refined as shown here:

Figure 7.42: Boundary Mesh (A) Before and (B) After Refining Based on Proximity



To further improve the quality of the refined boundary mesh, do the following:

1. Select **Swap** in the **Options** list and specify the **Max Angle** and **Max Skew** as appropriate (use the **Refine Boundary Zones** dialog box). See Swapping (p. 513) for details about swapping.
2. Click **Apply**.
3. If the geometry of the boundary is close to planar, you can improve the mesh quality further by selecting the **Smooth** option, specifying the **Max Angle**[1] and **Relax**[2] parameters, as appropriate (in the **Refine Boundary Zones** dialog box), and clicking **Apply**.

> **Warning:**
>
> If the geometry is far from planar, smoothing is not recommended as it may modify the shape of the boundary.

[1] Max Angle specifies the maximum angle between two adjacent face normals. When the **Swap** option is active, only faces with an angle below this value will be swapped. This restriction prevents the loss of sharp edges in the geometry. The valid range of entries is 0 to 180° and the default is 10°. The larger the angle, the greater the chance that a face swap will occur that may have an impact on the flow solution.

[2] Relax specifies the relaxation factor by which the computed change in node position should by multiplied before the node is moved. A value of zero results in no node movement, and a value of 1 results in movement equivalent to the entire computed increment.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you want to repeat the process for another refinement region, first select the **Clear** option and click **Apply** to clear all marks.

7.6.6. Creating and Modifying Features

Boundary → Mesh → Feature...

Geometric features, such as ridges, curves, or corners should be preserved while performing various operations (for example, smoothing, remeshing) on the boundary mesh. You can create edge zones for a face zone and if required, you can also modify the node distribution on the edge zone. The **Feature Modify** dialog box contains options available for creating and modifying edge zones. You can also draw the edge zones to determine their direction (that is, determine the start and the end points).

7.6.6.1. Creating Edge Zones

7.6.6.2. Modifying Edge Zones

7.6.6.3. Using the Feature Modify Dialog Box

Important:

- For object-based meshing, you can use the context menus in the tree or onscreen tools to create edge zones on selected face zones or surfaces. These options use the Fixed angle criterion. See Extract Edge Zones (p. 707).
- You can also use the **Surface Retriangulation** dialog box for creating edge zones before remeshing the face zones. The **Surface Retriangulation** dialog box allows you to use the face-zone approach only.

7.6.6.1. Creating Edge Zones

Edge zones can be created according to the specified combination of the edge zone creation approach and the angle criterion.

The angle criteria used for creating edge zones are as follows:

- Fixed angle criterion

  This method considers the feature angle between adjacent faces when creating edge zones. You can specify the minimum feature angle between adjacent faces as a parameter for edge zone creation. The common edge thread between two faces will be created when the feature angle is greater than the value specified.

- Adaptive angle criterion

  This method compares the angle at the edge with the angle at neighboring edges. If the relation between the angles matches the typical patterns of the angles in the neighborhood of the feature edge, the edge in question is considered to be a feature edge. You do not need to specify a value for the feature angle in this case.

The approaches available for edge zone creation are as follows:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Face zone approach

  The edge thread is created on the entire face zone based on the specified angle criteria. The face zone approach is useful when creating edge threads on common edges where two surfaces of the zone intersect each other. The common edge is considered to be a feature edge when the angle value specified (fixed angle criterion) is less than the feature angle. Alternatively, the edge thread at the common edge can be created by detecting the change in the feature angle automatically (adaptive angle criterion).

- Face seed approach

  The edge thread is created surrounding the surface on which the seed face is defined based on the specified angle criteria. The common edge is considered to be a feature edge when the angle value specified (fixed angle criterion) is less than the feature angle. Alternatively, the edge thread at the common edge can be created by detecting the change in the feature angle automatically (adaptive angle criterion).

The **Face Seed** approach is available only when you use the **Feature Modify** dialog box for creating edge zones. If you use the **Surface Retriangulation** dialog box instead, the **Face Zone** approach is used for creating the edge zones.

Figure 7.43: Surface Mesh - Feature Angle = 60

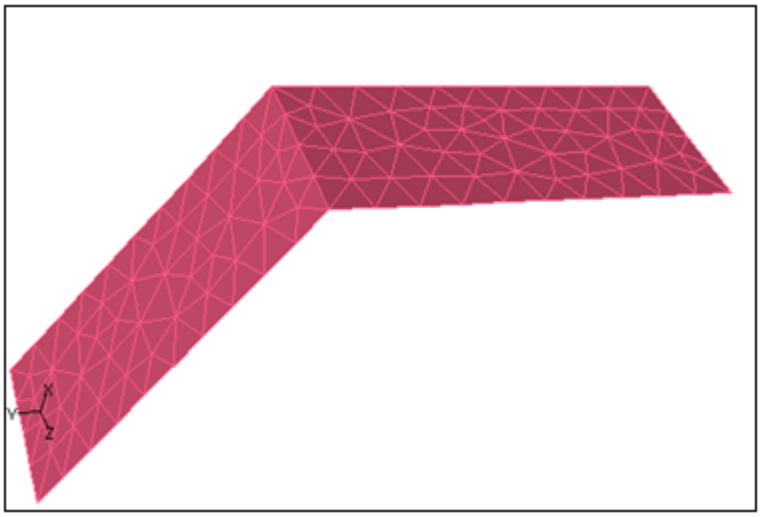


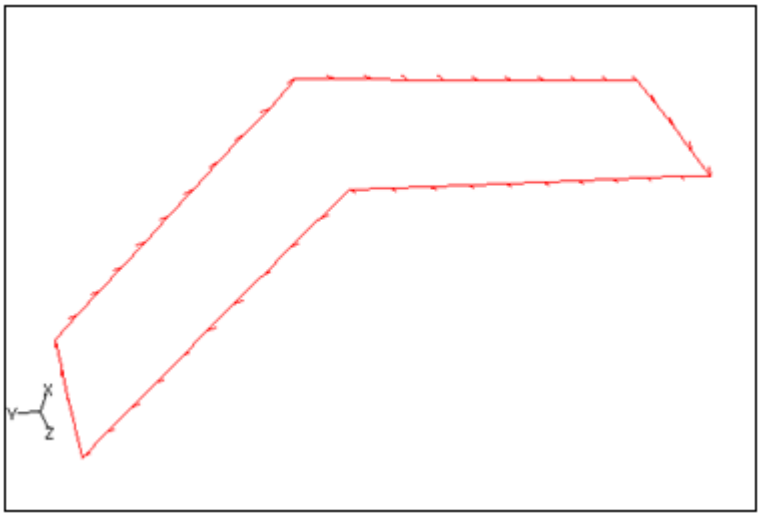
Figure 7.43: Surface Mesh - Feature Angle = 60 (p. 673) shows a surface mesh with two faces connected at a common edge and having a feature angle of 60 degrees. Both faces are in the same face zone.

Figure 7.44: Edge Zone for Face Zone Approach and Fixed Angle = 65 (p. 674) and Figure 7.47: Edge Zones for Face Seed Approach and Fixed Angle = 55 (or Adaptive Angle) (p. 675) show the edge zones created for different combinations of approach and angle criterion.

- Figure 7.44: Edge Zone for Face Zone Approach and Fixed Angle = 65 (p. 674) shows the single edge zone created by using the **Face Zone** approach and **Fixed** angle criterion, with the angle specified as 65 degrees. The edge thread at the common edge is not created as the specified value for **Angle** is greater than the feature angle.

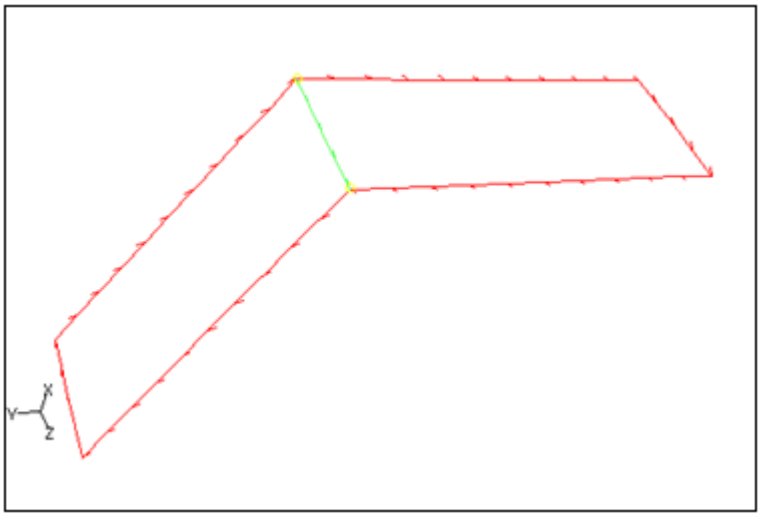
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.44: Edge Zone for Face Zone Approach and Fixed Angle = 65



- Figure 7.45: Edge Zones for Face Zone Approach and Fixed Angle = 55 (or Adaptive Angle) (p. 674) shows the edge zones created by using the **Face Zone** approach and **Fixed** angle criterion, with the angle specified as 55 degrees. The interior edge thread at the common edge is created because the specified value for **Angle** is smaller than the feature angle. Alternatively, if you use the **Adaptive** angle criterion, the change in angle will be detected automatically and the interior edge thread will be created as shown in Figure 7.45: Edge Zones for Face Zone Approach and Fixed Angle = 55 (or Adaptive Angle) (p. 674).

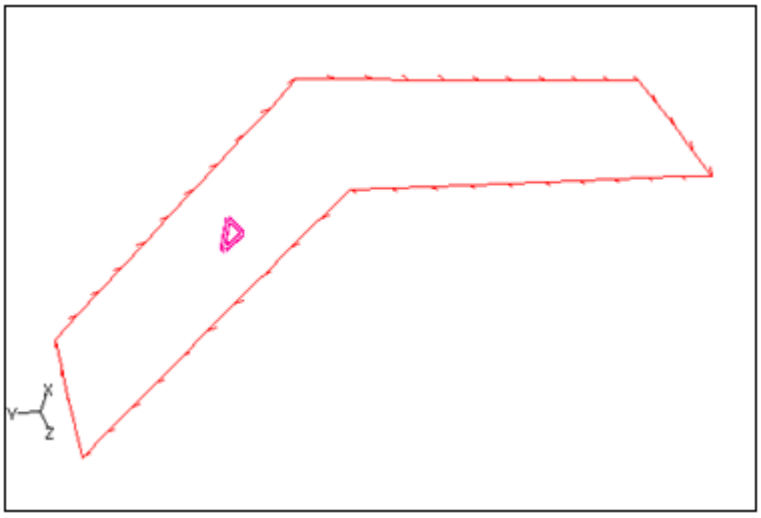
Figure 7.45: Edge Zones for Face Zone Approach and Fixed Angle = 55 (or Adaptive Angle)



- Figure 7.46: Edge Zone for Face Seed Approach and Fixed Angle = 65 (p. 675) shows the single edge zone created by using the **Face Seed** approach and **Fixed** angle criterion, with the angle specified as 65 degrees. The edge thread at the common edge is not created because the specified value for **Angle** is greater than the feature angle.

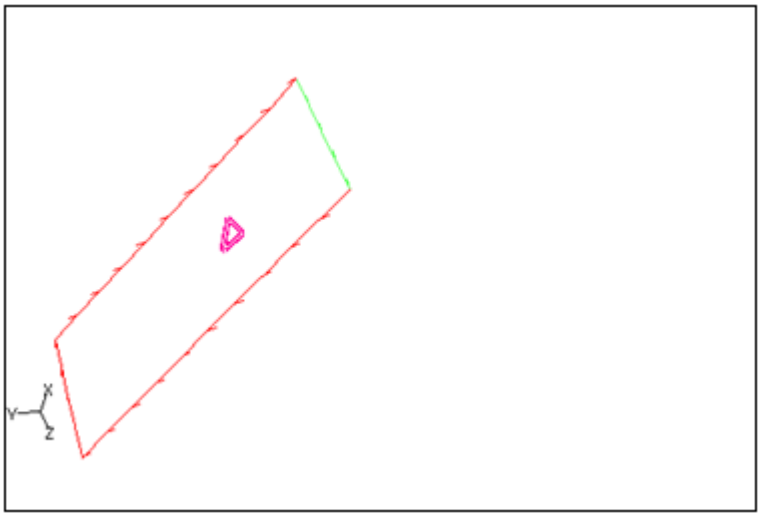
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.46: Edge Zone for Face Seed Approach and Fixed Angle = 65



- Figure 7.47: Edge Zones for Face Seed Approach and Fixed Angle = 55 (or Adaptive Angle) (p. 675) shows the edge zones created by using the **Face Seed** approach and **Fixed** angle criterion, with the angle specified as 55 degrees. The boundary edge thread is created based on the seed face selected. The interior edge thread at the common edge is created because the specified value for **Angle** is smaller than the feature angle. Alternatively, if you use the **Adaptive** angle criterion, the change in angle will be detected automatically and the boundary and interior edge threads will be created as shown here:

Figure 7.47: Edge Zones for Face Seed Approach and Fixed Angle = 55 (or Adaptive Angle)



7.6.6.2. Modifying Edge Zones

The following edge modification options are available:

- Deleting edge zones.
- Copying existing edge zones (including the nodes) to a new edge zone.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Toggling the edge zone type between boundary and interior.
- Grouping and ungrouping edge zones.
- Orienting the edges on the edge zone to point in the same direction.
- Reversing the direction of the edge zone.

  Note:

  The direction of a boundary edge zone determines the side from which new faces are formed. The direction of a boundary edge zone should be right-handed with respect to the average normal of the face zone to be remeshed. However, the direction is not so important in the case of interior edge zones because faces are always formed on both sides of the zone.

- Separating the edge zone based on the connectivity and feature angle specified.
- Merging multiple edge zones into a single zone.

  Note:

  Only edge zones of the same type (boundary or interior) can be merged.

- Remeshing the edge zones to modify the node distribution.
- Projecting the edges of the edge zone onto a face zone.

  You can select the closest point method or specify the direction in which the edge should be projected onto the selected face zone.

- Intersecting edge zones to create a new edge zone comprising the common edges.

7.6.6.3. Using the Feature Modify Dialog Box

The **Feature Modify** dialog box can be used for creating edge zones as follows:

1. Select the required zones from the **Boundary Zones** selection list.
2. Select **Create** from the **Options** list.
3. Select the appropriate option from the **Approach** drop-down list. Select the appropriate **Seed Face** when using the **Face Seed** approach.
4. Select the appropriate option from the **Angle Criterion** drop-down list. Specify an appropriate value for the **Angle** when using the **Fixed** angle criterion.
5. Enable **Add Edges to Object** to add the extracted edges to the object comprising the boundary face zones selected.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Click **Apply** to create the edge zones.

Important:

- For object-based meshing, you can use the context menus in the tree or onscreen tools to create edge zones on selected face zones or surfaces. These options use the Fixed angle criterion. See Extract Edge Zones (p. 707).
- You can also use the **Surface Retriangulation** dialog box for creating edge zones before remeshing the face zones. The **Surface Retriangulation** dialog box allows you to use the face-zone approach only.

The **Feature Modify** dialog box can be used for modifying edge loops as follows:

- Operations such as deleting, copying, grouping/ungrouping, orienting, separating, and merging edge loops, toggling the edge loop type, and reversing the edge loop direction:

  1. Select the appropriate zones in the **Edge Zones** selection list.

     Warning:

     You can select only one edge zone when separating an edge zone.

  2. Click the appropriate button in the **Edge Modify** group box.

- Remeshing edge zones:

  1. Select **Remesh** from the **Options** list.
  2. Select the appropriate zones from the **Edge Zones** selection list.
  3. Select an appropriate method from the **Method** drop-down list. You can specify a constant spacing of nodes or select either the arithmetic or the geometric method for node spacing. You can also select the **Size Field** option to use the size field to remesh the edge zones.

     For the **Constant**, **Arithmetic**, or **Geometric** methods, set the following parameters:

     a. Specify values for **First Spacing** and **Last Spacing** as required.

     Note:

     For the **Constant** method, the value specified for **First Spacing** will be the constant node spacing. Also, the **Last Spacing** option is not relevant for the **Constant** method and will not be available.

     b. Specify an appropriate value for **Feature Angle**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. Enable **Quadratic Reconstruct**, if required. The quadratic reconstruction option allows you to reconstruct the edge by fitting a quadratic polynomial between the original edge nodes.

Alternatively, for remeshing using the size field, make sure the size field is defined as required (see Computing the Size Field (p. 595)).

4. Click **Apply** to remesh the edge zone.

- Projecting edge zones:

1. Select **Project** from the **Options** list.
2. Select the appropriate zones in the **Edge Zones** selection list.
3. Select the appropriate face zone from the **Face Zones** selection list.
4. Select the appropriate projection method from the **Method** drop-down list. The **Closest Point** method specifies that the edge should be projected to the closest point on the face zone selected. The **Specific Direction** method allows you to project the edge on the face zone in a specific direction.
5. Specify the direction in which the edges should be projected when using the **Specific Direction** method.
6. Click **Apply** to project the edge onto the selected face zone.

- Intersecting edge zones:

1. Select **Intersect** from the **Options** list.
2. Select the appropriate zones in the **Edge Zones** selection list.
3. Enable **Delete** in the **Overlapped Edges** group box if you want to automatically delete all the overlapping edges.

   You can use the `delete-overlapped-edges` text command to delete individual overlapping edges.
4. Specify an appropriate value for **Intersection Tolerance**.
5. Click **Apply** to intersect the selected edge zones.

7.6.7. Remeshing Boundary Zones

Boundary → Mesh → Remesh...

In some cases, you may need to regenerate the boundary mesh on a particular boundary face zone. You may find that the mesh resolution on the boundary is not high enough, or that you want to generate triangular faces on a boundary that currently has quadrilateral faces. Remeshing of boundary faces can be accomplished using the **Surface Retriangulation** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can remesh the boundary face zones based on edge angle, curvature, and proximity.

7.6.7.1. Creating Edge Zones

7.6.7.2. Modifying Edge Zones

7.6.7.3. Remeshing Boundary Face Zones

7.6.7.4. Using the Surface Retriangulation Dialog Box

7.6.7.1. Creating Edge Zones

To remesh a face zone, you first need to create edge zones on the borders of the face zones using the parameters available in the **Edge Create** group box in the **Surface Retriangulation** dialog box (see Using the Surface Retriangulation Dialog Box (p. 680)).

You can create the edge zones according to your requirement by specifying an appropriate combination of the edge zone creation approach and angle criteria (refer to Creating Edge Zones (p. 672) for details).

Important:

The **Face Seed** approach is available only when you use the **Feature Modify** dialog box for creating edge zones. Click the **Feature Modify...** button to open the **Feature Modify** dialog box.

Note:

- For object-based meshing, you can create edge zones on selected face zones or surfaces using the context menus in the tree or onscreen tools. These options use the Fixed angle criterion. See Extract Edge Zones (p. 707).
- You can also use the **Feature Modify** dialog box to create new or modify existing edge zones before remeshing the face zones.

You can also draw the edge zones to determine their direction (that is, the start point and the end point).

7.6.7.2. Modifying Edge Zones

You can modify the node distribution on the edge zones using the **Feature Modify** dialog box (opened using the **Feature Modify...** button in the **Surface Retriangulation** dialog box). If you want to assign different node distributions to two or more portions of an edge zone, you can separate the zone based on a specified feature angle between consecutive edges. Separation is performed automatically at multiply-connected nodes.

After creating edge zones using an appropriate combination of the edge zone creation approach and angle criteria, modify the edge zone as required. You can modify the edge zones using the options available in the **Feature Modify** dialog box. Refer to Using the Feature Modify Dialog Box (p. 676) for details on using the various options available in the **Feature Modify** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

It is also possible to modify the edges of the zones using the operations in the **Modify Boundary** dialog box. Any edges you create must have the same direction as the edge zone.

Important:

You cannot remesh a continuous edge zone. You must first separate it into two or more non-continuous edge zones (that is, edge zones with start and end points).

7.6.7.3. Remeshing Boundary Face Zones

If the mesh resolution on the boundary face zone is not enough, or you want to create triangular faces on a boundary face zone that currently has quadrilateral faces, you can remesh that boundary face zone. You can remesh the boundary face zone using the **Surface Retriangulation** dialog box (see Using the Surface Retriangulation Dialog Box (p. 680) for details).

7.6.7.4. Using the Surface Retriangulation Dialog Box

The generalized procedure for remeshing a boundary face zone using the **Surface Retriangulation** dialog box is as follows:

1. Create the edge zones as appropriate.

   a. Select the boundary face zone for which you want to create edge zones in the **Boundary Face Zones** selection list.

   b. Select the appropriate option from the **Angle Criterion** drop-down list.

      By default, the **Face Zone** approach is used to create edge zones. Therefore, you can only specify the required **Angle Criterion** in the **Surface Retriangulation** dialog box. If however, you want to use **Face Seed** approach, you can use the **Feature Modify** dialog box to create the edge zones instead (see Creating Edge Zones (p. 672)).

   c. Click **Create**.

      The edge zones created will now be available in the **Edge Zones** selection list.

   d. Select the appropriate zones in the **Edge Zones** selection list and click **Draw** to display them.

      The selected edge zones will be displayed in the graphics window. If you are not satisfied with the edge zones and you want to modify them, open the **Feature Modify** dialog box.

2. Modify the edge zones as required using the options available in the **Feature Modify** dialog box. Click the **Feature Modify...** button to open the **Feature Modify** dialog box. Refer to Modifying Edge Zones (p. 675) for details.

   When you are satisfied with the edge zones you can proceed to remesh the faces.

3. Select the zone to be remeshed in the **Boundary Face Zones** list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can select only a single boundary face zone for remeshing, unless the **Use Conformal Remesh** option is enabled.

4. Set the appropriate remeshing options in the **Face Remesh Options** group box.

   a. Enable **Size Field** if you want to use the size field to remesh the faces.

   Note:

   Edge zones associated with face zones are not remeshed implicitly. If you have feature edge zones associated with the surface being remeshed, you need to remesh them before remeshing the face zones.

   b. Select the appropriate options from the **Reconstruction (Order)** drop-down list in the **Face Remesh Options** group box.

   c. Enable **Replace Face Zone**, if required.

   Important:

   Remeshing can be performed on both triangular and quadrilateral face zones. However, it will always result in a triangular face zone.

   d. Enable **Use Conformal Remesh** if you want to conformally remesh multiple face zones connected along the shared boundary.

   Note:

   - This option is available only when **Size Field** is enabled and **None** is selected in the **Reconstruction** drop-down list. You will be asked to compute the size field or read a size field file.
   - Periodic face zones cannot be remeshed using this option.

   - Set the minimum **Corner Angle** to specify the minimum angle between feature edges that will be preserved during remeshing.

   Note:

   The shared boundary between different zones will be remeshed only if all the face zones incident to it are selected for conformal remeshing.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Click **Remesh** to remesh the face zones.

Note:

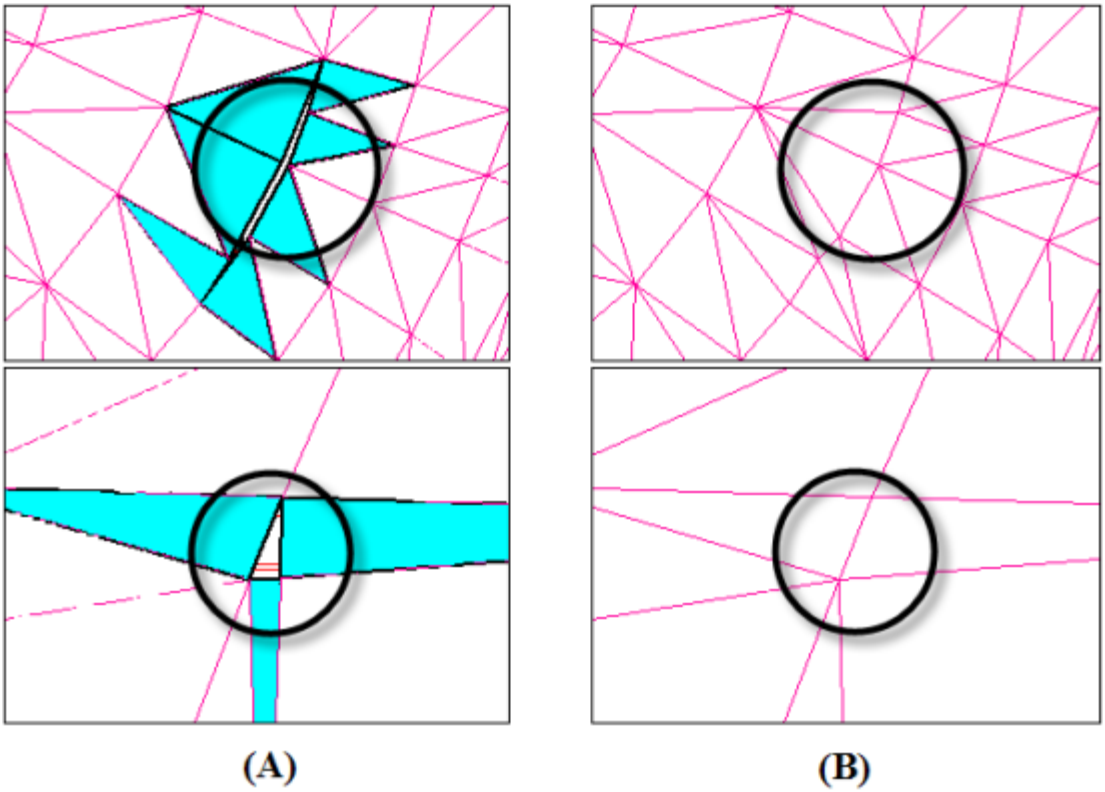
Edge zones are saved when the mesh file is written.

7.6.8. Faceted Stitching of Boundary Zones

You can repair surfaces having internal cracks or free edges using the **Faceted Stitch** option. You can specify an appropriate tolerance value within which the free edges will be stitched. The **Self Stitch only** option allows you to stitch the edges within the same boundary zone. The faceted stitching operation is available only for triangular boundaries.

Figure 7.48: Mesh (A) Before and (B) After Using the Faceted Stitch Option (p. 682) shows the repair of a surface with internal cracks.

Figure 7.48: Mesh (A) Before and (B) After Using the Faceted Stitch Option



The command `/boundary/remesh/faceted-stitch-zones` enables you to perform the faceted stitching of zones.

Note:

Features may not be maintained when using the faceted stitching operation.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

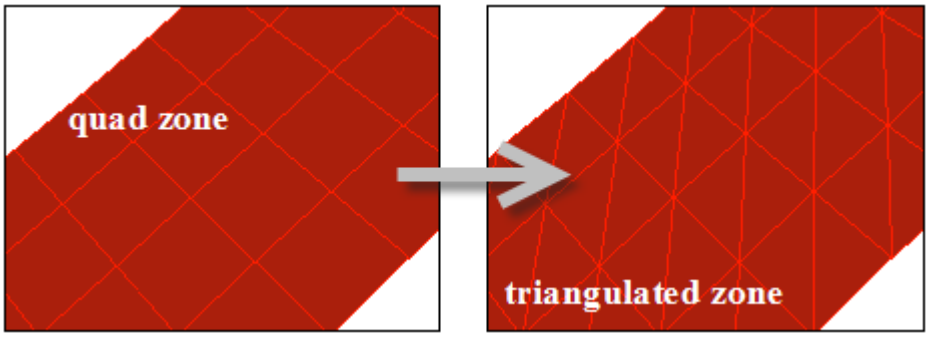
7.6.9. Triangulating Boundary Zones

Some operations like intersection, joining, stitching, and wrapping are available only to triangular boundary zones. You can remesh a quadrilateral face zone with triangular faces as shown in Figure 7.49: Triangulating a Boundary Zone (p. 683).

You can use the **Triangulate Zones** dialog box to perform this operation. The dialog box includes an option to either copy the quad zones and triangulate the copied zones or replace the original quad zones with the triangulated zone.

You can also use the command `/boundary/remesh/triangulate` to perform this operation.

Figure 7.49: Triangulating a Boundary Zone



7.6.10. Separating Boundary Zones

There are several methods available that allow you to separate a single boundary face zone into multiple zones of the same type. If your mesh contains a zone that you want to break up into smaller portions, you can make use of these options. For example, if you created a single wall zone when generating the mesh for a duct, but you want to generate different mesh shapes on specific portions of the wall, you will need to break that wall zone into two or more wall zones.

7.6.10.1. Separating Face Zones using Hotkeys

7.6.10.2. Using the Separate Face Zones dialog box

7.6.10.1. Separating Face Zones using Hotkeys

You can use the hotkey **Ctrl**+**Shift**+**S** to separate faces or zones based on what has been selected. If **help text display** is active, a description of the face zone separation options is displayed.

- If a multi-region face zone is selected, separation will be by region.
- If a single-region face zone is selected, separation will be by angle. The angle may be set using the **Separate Face Zones** dialog box.
- If a face (or edge) is selected, the face zone (edge zone) separation will be by seed.
- If edge zone with face seed selection, then the face zone is separated by edge zone.
- If no other selection, separation will be by marked faces.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Faces are **marked** using the hotkey **Ctrl**+**Shift**+**J** and the mouse probe/face selection filter. If **help text display** is active, a description of the marking options is displayed.

- **Ctrl**+**S** marks individually selected faces.
- **Ctrl**+**D** marks areas by flood-filling.
- **Ctrl**+**R** marks areas by adding rings around the selected face.
- **Ctrl**+**Q** marks faces by quality.
- **Ctrl**+**G** marks faces by angle relative to the selected face, on the entire face zone.
- **Ctrl**+**L** marks unmarked island faces.
- **Ctrl**+**U** unmarks selected/all faces.
- **Ctrl**+**I** opens a dialog box to set options for marking faces.

7.6.10.2. Using the Separate Face Zones dialog box

There are six methods available for separating a boundary face zone using the **Separate Face Zones** dialog box accessed via **Boundary > Zone > Separate...**. They are:

Separating Using Angle

For geometries with sharp corners, it is often easy to separate face zones based on the significant angle. Faces with normal vectors that differ by an angle greater than or equal to the specified angle value will be placed in different zones.

For example, if the mesh consists of a cube, and all 6 sides of the cube are in a single wall zone, you would specify a significant angle of 89°. Because the normal vector for each cube side differs by 90° from the normals of its adjacent sides, each of the 6 sides will be placed in a different wall zone.

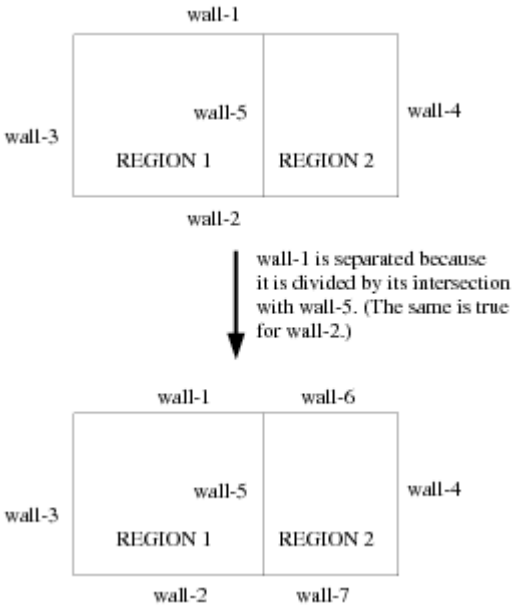
Separating Using Regions

You can also separate face zones based on contiguous regions. For example, if you want to generate the mesh in different regions of the domain using different meshing parameters, you may need to split up a boundary zone that encompasses more than one of these regions. Separating based on region splits non-contiguous boundary face zones (that is, zones that are separated into two or more isolated groups) into multiple zones.

This command will also split zones that are divided by another face zone. An example could be two face zones touching in a “T". Using this command on the top zone (for example, wall-1 in Figure 7.50: Face Separation Based on Region (p. 685)) would split it into two zones. However, individual faces in the corners at the “T" junction may be put in their own zones. To check for this problem, list the new face zones (using the **List** button in the **Boundary Zones** dialog box), looking for zones with a single face in them. You can then merge these faces into the appropriate zone.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

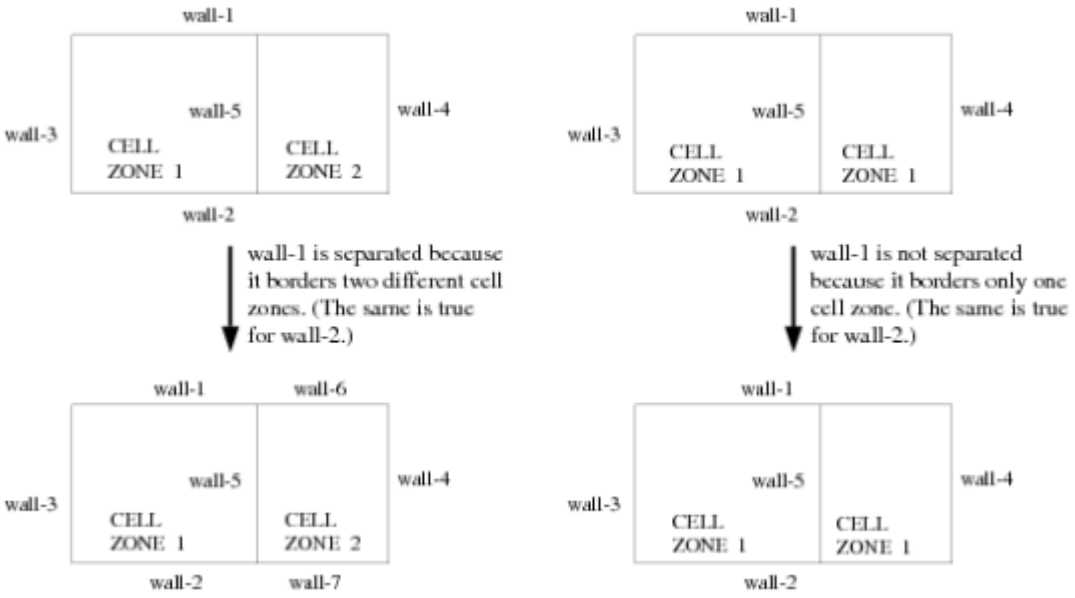
Figure 7.50: Face Separation Based on Region



Separating Based on Neighboring Cell Zones

Region separation will split wall-1 in Figure 7.50: Face Separation Based on Region (p. 685) into two zones regardless of whether the two regions are in the same cell zone. However, neighbor-based separation will yield different results. If both regions are in the same cell zone, wall-1 will not be separated (see Figure 7.51: Face Separation Based on Cell Neighbor (p. 685)). If they are in different cell zones, the zone will be separated. Thus, when neighbor separation is used, wall-1 will be separated only if it is adjacent to more than one cell zone. If the two regions are in two different cell zones, then wall-1 has two different neighboring cell zones and therefore it will be separated into two wall zones.

Figure 7.51: Face Separation Based on Cell Neighbor



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Separating Based on the Face/Element Shape

You can also separate face zones based on the shape of the faces. For example, if a face zone contains both triangular and quadrilateral faces, you can separate the zone into two zones (one containing the triangular faces, and the other containing the quadrilateral faces).

Separating Using a Seed Element

You can separate face zones by specifying a face element (in the face zone) as a seed face. You can also separate different faces of a single face zone using this method. The surface on which you define a seed face gets separated from rest of the face zone. You can separate face zones using the seed face based on the following criteria:

- Feature Angle Criteria

  This method enables you to separate the surface on which you have defined a seed face from the surfaces around it based on the specified value of the feature angle. The feature angle is the angle between the normal vectors of the cells. To separate the face zones based on this criteria, do the following:

  1. Select **Seed** in the **Options** list and **Angle** in the **Flood Fill Options** list.
  2. Specify the seed element in the **Face Seed** text entry field. Right-click the face you want to choose as a seed element in the graphics window. The **Face Seed** field will be updated automatically.
  3. Specify the required feature angle in the **Angle** field.
  4. Click **Separate**.

  The surface on which you defined the seed face will be separated from other surfaces of the zone for which the feature angle change is greater than or equal to the specified value. For example, if the mesh consists of a cube, and all 6 sides of the cube are in a single wall zone, specify a significant angle of 89° and specify a seed face on any one of the walls. Because the normal vector for each cube side differs by 90° from the normals of its adjacent sides, the face on which you have defined a seed cell will be placed in a different wall zone. Therefore, two zones will be created, one zone will have a face on which you defined a seed face and the second zone will have remaining faces.

- Edge Zone Criteria

  This method enables you to separate the surface, on which you have defined a seed face, from the other faces in the zone based on the existing edge zones associated with it. You must create the edge zones for the given mesh to use this method.

  To separate the face zones based on this criteria, do the following:

  1. Select **Seed** in the **Options** list and **Edge Loop** in the **Flood Fill Options** list.
  2. Specify the seed element in the **Face Seed** text entry field.

     For this method, you will only specify the seed element. The **Angle** field will not be available.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click **Separate**.

Important:

Create edge threads on the surface zones again using the **Surface Retriangulation** dialog box after performing above operations.

Separating Based on Marked Faces

You can separate face zones by placing marked faces in a new zone. To use this option in the **Separate Face Zones** dialog box, explicitly define a subregion of the domain (using the **Boundary Refinement Region** dialog box), then separate face zones based on whether or not each face in the specified zone is in the selected local region.

7.6.11. Projecting Boundary Zones

Another mesh refinement method involves projecting the nodes of one face zone onto another (possibly non-planar) face zone to create a new face zone that has the same connectivity as the original face zone. This new face zone is created after the projection, and no cell zones are created. The face zone that is projected is not modified in any way.

Projecting a face zone is used mainly to fill in gaps by extending the domain through the projection. The original connectivity is maintained after the projection, with the effect being that elements on the connected side zones will be stretched to cover the projection distance. Affected side zones should then be remeshed to obtain regular size elements on them. Such a remeshing results in a new side zone, after which you can (and should) delete the original side zone. Finally, you can mesh the domain to get the volume elements.

7.6.12. Creating Groups

You can create groups of faces and edges that will be available in all the dialog boxes along with the default groups (for example, boundary, tri, quad, and so on). The face and edge zones are grouped separately. The **User Defined Groups** dialog box enables you to define new face and/or edge groups, update existing groups, activate or delete a particular group. Although the dialog box is opened from the **Boundary** menu, it can be used with all dialog boxes that contain zone lists.

Note:

When a user-defined group is activated, the wild-cards used for zone selection in all the text commands will return zones contained in the active group. For example, the command `/display/boundary-grid *` will display all the boundary zones contained in the active group.

For object based meshing (see Object-Based Surface Meshing (p. 616)), you can create a face group and an edge group comprising the face zones and edge zones included in the specified objects using the options in the **Zone Group** group box in the **Operations** tab in the **Manage Objects** dialog box. Additionally, a face zone group is automatically created when a mesh object is created using the **Sew** operation. This face zone group is prefixed by **_mesh_group**, and enables easy selection of mesh object face zones for various operations (improve, smooth, and so on).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For CutCell meshing, the mesher separates the face zones by cell neighbor and creates a face zone group for the face zones of each fluid cell zone. See Generating the CutCell Mesh (p. 810) for details.

Note:

When an object is deleted along with the face and edge zones comprising the object, the corresponding groups will also be deleted.

7.6.13. Manipulating Boundary Zones

Boundary zones are groups of boundary faces. Usually the grouping collects boundary faces with the same boundary conditions, but further sub-groupings are often used to preserve a sharp edge in the surface mesh or simply as an artifact of the boundary mesh generation process.

Each zone has a unique ID, which must be a positive integer. You can use the options in the **Manage Face Zones** dialog box to manipulate the face zones. find information about each zone, identify them, merge zones or delete them, change the boundary type of all faces in a zone, rename zones, and rotate, scale, or translate zones.

- Click **List** to obtain information about the selected face zones. The zone ID, name, boundary type, and number of faces by type (tri or quad) will be reported.
- Use the **Change Type** option to change the boundary type of the selected face zones.

  Note:

  When changing the boundary type of any zone to type `interior`, ensure that there is a single cell zone across the `interior` boundary. Retaining multiple cell zones across an `interior` boundary can cause undesirable results with further tet meshing or smoothing operations.

  Also, face zones having no/one neighboring cell zone should not be changed to type `interior`.

  The mesh check will issue a warning if multiple cell zones are maintained across an `interior` boundary. The boundary type in such cases should be set to `internal` instead.

- Use the **Copy** option to copy the nodes and faces of the selected face zones.
- Use the **Delete** option to delete the selected face zones. You can optionally delete the nodes of the face zones as well (enabled by default).
- Use the **Merge** option to merge the selected face zones based on **Alphabetical Order** (default) or **Larger Area**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Use the **Rename** option to rename the selected face zones. You can optionally change the zone name prefix as well.

  Note:

  The zone name can have a maximum of 256 characters.

- Use the **Flip Normals** option to flip the direction of all face normals on the selected face zones.
- Use the **Orient** option to consistently orient the face normals on the selected face zones.
- Use the **Rotate** option to rotate all nodes of the selected face zones through the angle specified. Enter the pivot and axis of rotation or use the **Define** option to select six nodes or positions to define the pivot and axis instead. You can optionally create a copy instead of replacing the original zones.
- Use the **Scale** option to scale all nodes of the selected face zones by the scale factors specified. You can optionally create a copy instead of replacing the original zones.
- Use the **Translate** option to translate all nodes of the selected face zones by the translation offsets specified. Use the **Define** option to select two nodes or positions to define the translation vector instead. You can optionally create a copy instead of replacing the original zones.

The hotkey **Ctrl**+**Shift**+**N** opens the **Change Zone Properties** dialog box which enables you to quickly rename the selected zone, set the boundary type, and set the geometry recovery option (low or high).

Note:

When changing the boundary type of any zone to type `interior`, ensure that there is a single cell zone across the `interior` boundary. Retaining multiple cell zones across an `interior` boundary can cause undesirable results with further tet meshing or smoothing operations.

Also, face zones having no/one neighboring cell zone should not be changed to type `interior`.

The mesh check will issue a warning if multiple cell zones are maintained across an `interior` boundary. The boundary type in such cases should be set to `internal` instead.

7.6.14. Manipulating Boundary Conditions

Case files read in the meshing mode also contain the boundary and cell zone conditions along with the mesh information. The **Boundary Conditions** dialog box enables you to copy or clear boundary conditions assigned to the boundary zones when a case file is read.

- You can copy the boundary conditions from the zone selected in the **With** list to those selected in the **Without** list using the **Copy** option.
- You can clear the boundary conditions assigned to the zones selected in the **With** list using the **Clear** option.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.6.15. Creating Surfaces

You can create specific types of surfaces within the existing geometry using one of the options available in the **Boundary/Create** menu.

The **Construct Geometry** tool ( ) also enables you to create a bounding box or cylinder/frustum for selected or all zones displayed in the graphics window.

The following sections explain how to create surfaces.

7.6.15.1. Creating a Bounding Box

7.6.15.2. Creating a Planar Surface Mesh

7.6.15.3. Creating a Cylinder/Frustum

7.6.15.4. Creating a Swept Surface

7.6.15.5. Creating a Revolved Surface

7.6.15.6. Creating Periodic Boundaries

7.6.15.1. Creating a Bounding Box

In some cases, you may want to create a box that encloses the input geometry (for example, creating a wind tunnel around the geometry). You can create a bounding box around the input geometry or only the selected zones of the geometry using the **Bounding Box** dialog box, or the **Construct Geometry** tool. You can also specify the required clearance values of the bounding box from the boundaries of the geometry.

There are two methods available for creating bounding box:

Using Absolute Values

This method enables you to create the bounding box by specifying the minimum and maximum extents of the bounding box in X, Y, and Z directions.

Using Relative Values

This method enables you to create the bounding box by specifying the relative coordinate values with reference to the selected face zone.

7.6.15.1.1. Using the Bounding Box Dialog Box

The procedure for creating a bounding box is as follows:

1. Select the zones around which you want to create a bounding box in the **Face Zones** list.
2. Select the appropriate method in the **Method** list.
    a. For the **Absolute** method, specify the bounding box extents (**X Min**, **X Max**, **Y Min**, **Y Max**, **Z Min**, and **Z Max**). If you click **Compute**, the extents will be computed such that the bounding box encloses the selected boundary zones.
    b. For the **Relative** method, specify the clearance values in the **Delta** entry fields (**Delta X Min**, **Delta X Max**, **Delta Y Min**, **Delta Y Max**, **Delta Z Min**, and **Delta Z Max**).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Initially, all the **Delta** entry fields will be set to `0`. This implies that the bounding box will touch the boundaries of the selected face zones. Positive delta values indicate that the bounding box will be created outside the initial bounding box while negative values indicate that the bounding box will be created inside the initial bounding box.

3. Specify an appropriate value for **Edge Length**. When you click **Compute** for the **Absolute** method, the value will be automatically set to 1/10th that of the minimum length of the bounding box.

4. Enable **Create Object** if you need to create a geometry object based on the bounding box face zone created.

Note:

Do not use the **Create Object** option if the box is to be used as a body of influence while setting up the size functions.

---

5. Click **Draw** to visualize the bounding box.

6. Click **Create** to create a bounding box based on the specified parameters.

7.6.15.1.2. Using the Construct Geometry Tool

The **Construct Geometry** tool ( ) enables you to create a bounding box for selected or all zones displayed in the graphics window. The bounding box extents are computed based on the entities selected or displayed and are indicated in the graphics window.

1. Select the zones (if required) and click the **Bounding Box** tool ( ) to preview the bounding box extents. The bounding box is always created in the global X-Y-Z axes.

2. The bounding box extents can be altered interactively by selecting the direction and dragging the mouse to change the box dimensions. Click the yellow dot on the bounding box surface to select the direction.

3. Click **Create** ( ) to open the **Create Object** dialog box.

   a. Enter an appropriate **Object Name**.

   b. Specify the mesh size for the surface mesh. By default, the edge length is computed as one fifth of the smallest side. Alternatively, enable **Specify Sizing** and specify the size to be used. Click **Preview** to visualize the size set.

4. Click **Create** to create the bounding box. A geometry object comprising the bounding box face zones will be created.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.6.15.2. Creating a Planar Surface Mesh

In some cases, you may need to create a plane surface mesh in the geometry (for example, creating a baffle-like surface inside a hollow tube). You can create a plane surface and mesh the surface using triangular faces of the required size using the **Plane Surface** dialog box.

> **Warning:**
>
> It is possible to create a planar surface only of rectangular shape; you cannot create a planar surface of any other shape.

There are two methods available for creating planar surface mesh:

- Axis Direction Method:

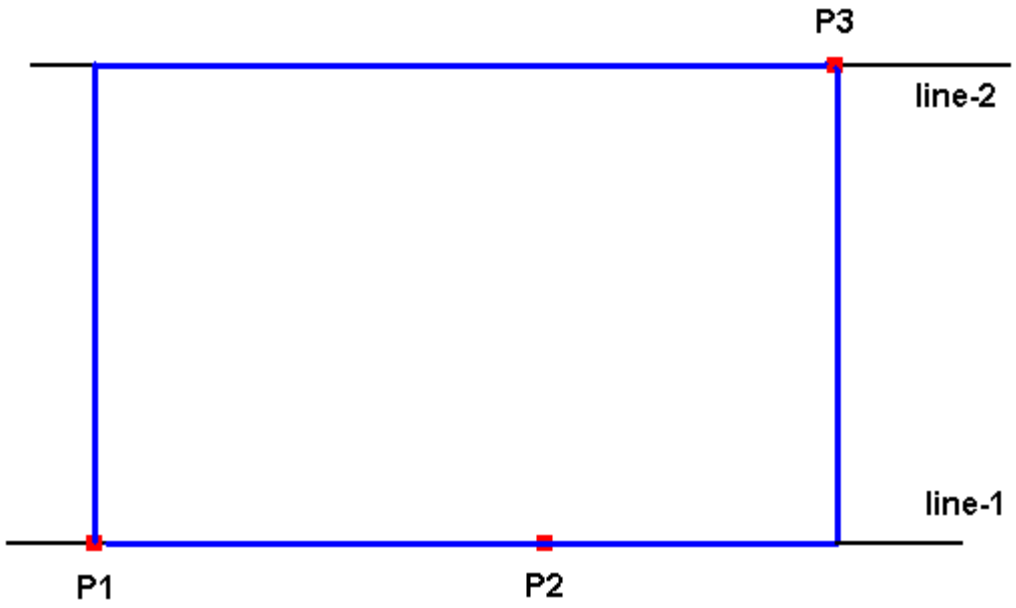
  This method enables you to create the plane surface perpendicular to any of the coordinate axes. Select the axis perpendicular to which you want to create a planar surface mesh and then, specify the coordinates of the points that will form a rectangular surface perpendicular to the axis selected. You can also create a plane surface enclosing the boundaries of the selected face zone using this method.

- Planar Points Method:

  This method enables you to create a plane surface mesh from three points in the geometry selected using the mouse.

  The concept of the planar points method is shown in Figure 7.52: Planar Points Method (p. 692). After specifying the planar points, the first point (**P1**) and second point (**P2**) are connected to each other by a line (**line-1**). Another line (**line-2**) is drawn through the third point (**P3**) parallel to the first line. Perpendiculars are drawn from points **P1** and **P3** on **line-2** and **line-1** respectively.

  Figure 7.52: Planar Points Method



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This creates a rectangular surface that you can mesh as required.

7.6.15.2.1. Using the Plane Surface Dialog Box

The procedure for creating a surface mesh is as follows:

1. Select the appropriate method in the **Options** list.

   a. For the **Axis Direction** method, select the appropriate face zones, direction, and specify the coordinates of the points perpendicular to the axis.

      If you select **X Axis** then the entry box for specifying coordinates in X direction will not be accessible. This applies to the other two axes as well.

   b. For the **Points** method, specify the coordinates for the three points defining the plane. You can click the **Select Points...** button and select the points using the mouse button.

2. Specify an appropriate value for **Edge Length**. If you click **Compute**, the **Edge Length** will be computed as 1/10th of the minimum distance along the coordinate axes.

3. Enable **Create Object** if you need to create a geometry object based on the plane surface face zone created.

4. Click **Draw** to visualize the surface.

5. Click **Create** to create the planar surface.

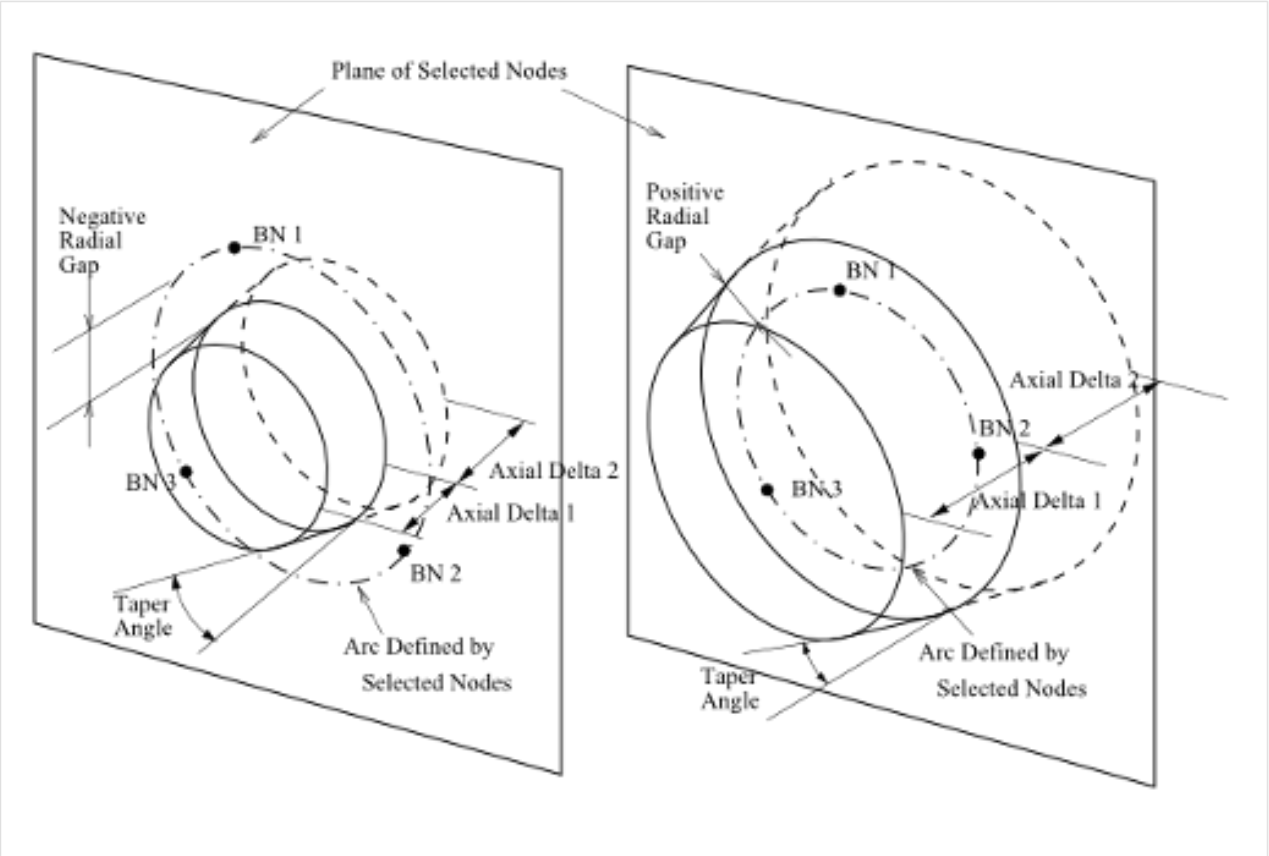
7.6.15.3. Creating a Cylinder/Frustum

In some cases, you may want to create a cylinder or frustum within the existing geometry (for example, creating an MRF zone for problems involving moving parts such as rotating blades or impellers, creating a cylindrical surface to close a gap in the geometry, and so on). You can create a cylindrical surface and mesh it with a triangular surface mesh using the options available in the **Cylinder** dialog box, or the **Construct Geometry** tool.

- **Using 3 Arc Nodes:** You can create a cylindrical surface using three nodes that lie on a circular arc (see Figure 7.53: Cylinder Defined by 3 Arc Nodes, Radial Gap, and Axial Delta (p. 694)). Specify the radial gap and taper angle that will determine the actual radii of the cylinder/frustum to be created. You can specify a positive or negative radial gap value depending on the required size of the cylinder/frustum. A taper angle of zero will result in a cylinder. The axial delta values determine the axial length of the cylinder/frustum. The **Caps** option enables you to create the circular capping surfaces along with the cylindrical surface.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.53: Cylinder Defined by 3 Arc Nodes, Radial Gap, and Axial Delta



- **Using 3 Arc Nodes and a Height Node:** You can create a cylindrical surface using three nodes which lie on a circular arc, and a fourth node to determine the height of the cylinder/frustum (see Figure 7.54: Cylinder Defined by 3 Arc Nodes and a Height Node (p. 695)). The radii, height, and taper angle will be determined based on the nodes selected.

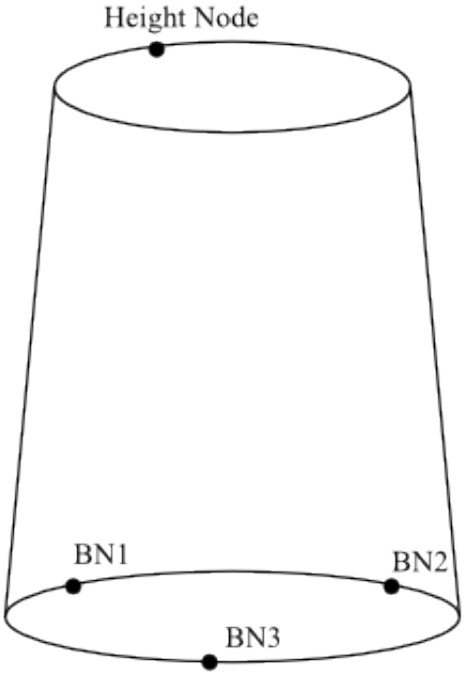
  Note:

  A planar annular surface will be created if the four nodes selected are in the same plane (that is, the height is zero).

  The **Caps** option enables you to create the circular capping surfaces along with the cylindrical surface.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

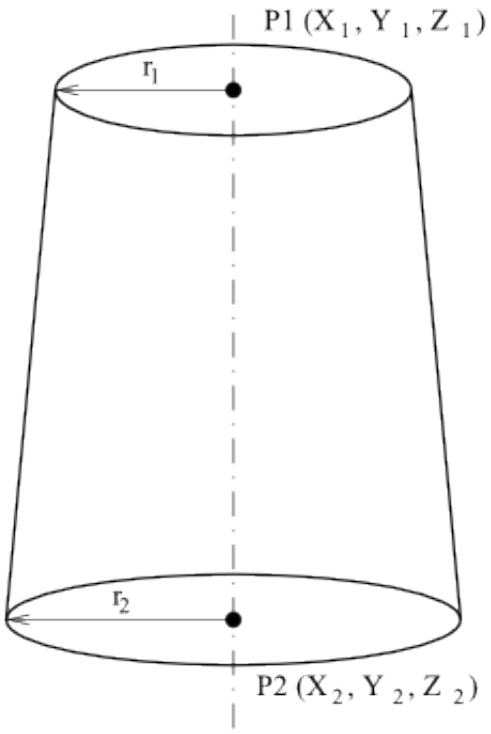
Figure 7.54: Cylinder Defined by 3 Arc Nodes and a Height Node



- **Using 2 Axis Locations or 2 Axis Nodes:** You can also create a cylindrical surface by specifying the radii (r1, r2) of the cylinder/frustum and two points (P1 and P2) defining the axis (see Figure 7.55: Cylinder Defined by Axial Points and Radii (p. 696)). Equal values of r1 and r2 will result in a cylinder. The axis can be defined by specifying the location (X, Y, Z) of the points or by specifying the appropriate boundary nodes corresponding to the axial points P1 and P2. The **Caps** option enables you to create the circular capping surfaces along with the cylindrical surface.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.55: Cylinder Defined by Axial Points and Radii



7.6.15.3.1. Using the Cylinder Dialog Box

The procedure for creating a cylindrical surface is as follows:

1. Select the appropriate option for defining the cylinder.

    a. For the **3 Arc Nodes** method, select the nodes on the circular arc. Enter appropriate values for **Axial Delta 1**, **Axial Delta 2**, **Taper Angle**, and **Radial Gap**.

    b. For the **3 Arc, 1 Height Node** method, select the 3 nodes on the circular arc and the height node.

    c. For the **2 Axis Locations** and **2 Axis Nodes** methods, specify the points defining the axis. You can specify the locations (or node IDs) manually. Alternatively, you can click the **Select Points...** (or the **Select Nodes...**) button and select the points using the mouse. Enter appropriate values for **Radius1** and **Radius2**.

2. Enter an appropriate value for **Edge Length**.

3. Enable **Caps** to create the circular capping surfaces along with the cylindrical surface.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Enable **Create Object** if you need to create a geometry object based on the cylinder/frustum face zones created.

   Note:

   Do not use the **Create Object** option if the cylinder/frustum is to be used as a body of influence while setting up the size functions.

5. Click **Preview** to preview the cylinder to be created.

6. When you are satisfied with the settings, click **Create** to create the cylindrical surface. Enter an appropriate zone name prefix in the **Object/Zone Prefix** dialog box and click **OK**.

7.6.15.3.2. Using the Construct Geometry Tool

The **Construct Geometry** tool ( ) enables you to create a cylinder/frustum based on selections in the graphics window. The cylinder/frustum dimensions are computed based on the entities selected or displayed and are indicated in the graphics window.

1. Select the entities and click the **Cylinder** tool ( ) to preview the cylinder extents. Alternatively, click the **Frustum** tool ( ) to preview the frustum extents.

   - When no selections are made, the cylinder/frustum is aligned along the global Z-axis. The default height and radius are computed based on the bounding box dimensions for the entities displayed.
   - When face zones are selected, the cylinder/frustum is aligned along the global Z- axis. If any two dimensions of the bounding box for the selected zones are the same, the cylinder/frustum will be aligned along the third (remaining) direction. The default height and radius are computed based on the bounding box dimensions for the zones selected.
   - When a single node is selected, it is used as an axis node. The cylinder/frustum will be aligned along the global Z- axis. The default radius and height are equal and computed as one-tenth the length of the diagonal of the bounding box for the displayed zones. If no zones are displayed, a value one-tenth the length of the diagonal of the global bounding box will be used.
   - When two nodes are selected, the mid-point of the line joining the two is used as an axis node and the cylinder/frustum will be aligned along the global Z-axis. If the nodes selected are aligned in the Z-axis, the cylinder/frustum will be aligned along the global Y-axis instead. The default radius and height are equal, and computed as half the distance between the selected nodes.
   - When three nodes are selected, the cylinder/frustum base circle passes through the selected nodes. The axial direction is determined by the right hand thumb rule. The default radius and height are equal. The radius is determined by the nodes selected.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- When four nodes are selected, the first three are used to determine the cylinder/frustum base circle. The fourth node is used to determine the height. The radius is determined by the first three nodes selected.

2. The cylinder dimensions can be altered interactively by selecting the direction and dragging the mouse to change the dimensions. Click the yellow dot on the cylinder surface to change the height.

   The frustum dimensions can be altered interactively by selecting the direction and dragging the mouse to change the dimensions. Click the yellow dot on either surface to change the radii and height.

3. Click **Create** ( ) to open the **Create Object** dialog box.

   a. Enter an appropriate **Object Name**.

   b. Specify the mesh size for the surface mesh. By default, the edge length is computed as one-seventh the average radius or height, whichever is smaller. Alternatively, enable **Specify Sizing** and specify the size to be used. Click **Preview** to visualize the size set.

   c. The **Caps** option enables you to create the circular capping surfaces along with the cylindrical surface. Disable this option to obtain only the cylindrical surface.

4. Click **Create** to create the cylinder/frustum. A geometry object comprising the cylinder/frustum face zones will be created.

7.6.15.4. Creating a Swept Surface

In some cases, you may want to create a swept surface by projecting an edge zone along a specified linear distance in a specified direction. You can create a swept surface using the options available in the **Swept Surface** dialog box.

7.6.15.4.1. Using the Swept Surface Dialog Box

The procedure for creating a swept surface is as follows:

1. Create the edge zone for the swept surface.

   - Use the interactive edge zone creation tool to extract edge zones from existing face zones or surfaces. See Extract Edge Zones (p. 707) for details.

   - Use the Loop Selection tool to create an edge zone from selected nodes or points. See Using the Loop Selection Tool (p. 703) for details.

   - Use edge zone creation and modification options available in the **Manage Objects**, **Feature Modify**, or **Surface Retriangularization** dialog boxes. See Object Manipulation Operations (p. 611), Using the Feature Modify Dialog Box (p. 676), or Using the Surface Retriangulation Dialog Box (p. 680) respectively.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Open the **Swept Surface** dialog box.

   Boundary → **Create** → **Swept Surface...**

3. Select the edge zone to be swept from the **Edge Zones** drop-down list.

4. Select the corresponding faces from the **Face Zones** selection list.

5. Specify the distance along which the edge is to be swept in the **Total Distance** field.

6. Specify the appropriate value in the **No. of Offsets** field.

7. Specify the **Vector** defining the direction in which the edge is to be swept.

   Alternatively, you can click **Define** and select two nodes or positions to specify the vector. The **Total Distance** is also computed based on the nodes/positions selected.

8. Enable **Split Quad Faces**, if required.

9. Enable **Create Object** if you need to create a geometry object based on the swept surface face zone created.

10. Click **Create** to create the swept surface.

7.6.15.5. Creating a Revolved Surface

In some cases, you may want to create a revolved surface from specific edge zones. The revolved surface is created by revolving the selected edge zones through the angle specified using the pivot and axis of rotation defined. You can create a revolved surface using the options available in the **Revolved Surface** dialog box.

7.6.15.5.1. Using the Revolved Surface Dialog Box

The procedure for creating a revolved surface is as follows:

1. Create the edge zones to be used for creating the revolved surface.

   - Use the interactive edge zone creation tool to extract edge zones from existing face zones or surfaces. See Extract Edge Zones (p. 707) for details.
   - Use the Loop Selection tool to create an edge zone from selected nodes or points. See Using the Loop Selection Tool (p. 703) for details.
   - Use edge zone creation and modification options available in the **Manage Objects**, **Feature Modify**, or **Surface Retriangularization** dialog boxes. See Object Manipulation Operations (p. 611), Using the Feature Modify Dialog Box (p. 676), or Using the Surface Retriangulation Dialog Box (p. 680) respectively.

2. Open the **Revolved Surface** dialog box.

   Boundary → **Create** → **Revolved Surface...**

3. Select the edges to be revolved from the **Edge Zones** selection list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Specify the appropriate value in the **Number of Segments** field.
5. Specify the angle through which the edge is to be revolved in the **Angle** field.
6. Specify an appropriate value for **Scale Factor** depending on the radius required for the revolved surface.
7. Specify the pivot point and the axis of revolution. Click **Define** and select 1-6 nodes to define the pivot and axis as follows:
   - If only 1 node is selected, the pivot point is at the node location and the axis of rotation is the global z-axis.
   - For 2 nodes, the pivot point is at the midpoint of the nodes selected and the axis of rotation is the global z-axis.
   - For 3 nodes, the pivot point is at the first node selected. The axis of rotation is the local z-axis normal to the plane defined by the three points, the positive direction is determined by the right-hand rule.
   - For 4, 5 or 6 nodes, the first 3 points define a circle. The pivot point is at the center of the circle. The axis of rotation is the local z-axis normal to the circular plane, the positive direction is determined by the right-hand rule.
8. Enable **Create Object** if you need to create a geometry object based on the revolved surface face zone created.
9. Click **Create** to create the revolved surface.

7.6.15.6. Creating Periodic Boundaries

Use the **Make Periodic Boundaries** dialog box to **Create** or **Recover** the periodic relationship between leader and shadow face zones in a single mesh object. The periodic boundaries are identical and contain either face or node correspondence information.

The **Make Periodic Boundaries** dialog box is accessible using the context sensitive menu under any mesh object or using the **Boundary** → **Create** → **Periodic...** menu.

Create Periodic Boundaries

You can **Create** new periodic boundaries using the following procedure.

1. In the preprocessor, create only *one* of the boundaries which is to be made periodic. The to-be-periodic boundary may have multiple face zones, but should be any non-periodic boundary type.
2. In the **Make Periodic Boundaries** dialog box in Fluent meshing, select the boundary zone(s) from the Boundary Zones list. You can also select the zone(s) graphically; the names will be highlighted in the Boundary Zones list.
3. Enter, or check for accuracy, the periodicity information (angle, pivot and axis for rotational periodicity; shift vector for translational periodicity) in the **Make Periodic Boundaries** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Periodicity information may be defined graphically (see below) or using the context-sensitive menu under **Model** in the Outline View, or read in with the mesh file.

4. Click **Create**.

   A corresponding periodic shadow boundary will be added to the object. A zone type of periodic will be assigned to both the periodic and the periodic-shadow zones, and the face/node correspondence will be generated.

Important:

- This is the only way to create periodic boundaries in the meshing mode; it is not sufficient to simply set a zone type to be periodic.
- Periodicity information will be stored with the mesh data and will be extracted when the mesh file is read. Multiple periodicities may be created, but only the first is read automatically with the mesh data.

To define periodicity information by other means, follow these steps.

- For rotational periodicity, the pivot point and axis of rotation can be defined graphically by selecting 1-6 nodes as follows, and then clicking **Define**.
  - If only 1 node is selected, the pivot point is at the node location and the axis of rotation is the global z-axis.
  - For 2 nodes, the pivot point is at the midpoint of the nodes selected and the axis of rotation is the global z-axis.
  - For 3 nodes, the pivot point is at the first node selected. The axis of rotation is the local z-axis normal to the plane defined by the three points, the positive direction is determined by the right-hand rule.
  - For 4, 5 or 6 nodes, the first 3 points define a circle. The pivot point is at the center of the circle. The axis of rotation is the local z-axis normal to the circular plane. The x-axis (0°) is defined by the 4th, 5th and 6th points and the positive direction is determined by the right-hand rule.
- For translational periodicity, the shift vector can be defined graphically clicking **Define** and then selecting two nodes.
- Rotational periodicity information may be set using the **Periodicity** dialog box accessed by right clicking on **Model** in the Outline View.
- Rotational periodicity information may be set using the text command **`boundary/set-periodicity`**.

When the periodic-shadow boundary is created from the original (periodic) boundary, the nodes around the outer edges of the shadow zone will be duplicates of existing nodes. These duplicates

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

will be marked as free, so they can be verified by counting them and drawing them. Before generating the initial mesh, you must merge these nodes.

Important:

To ensure that the periodic-shadow boundary creation works properly, you must define the node distribution correctly in the preprocessor that generates the boundary mesh.

Ensure that the distribution of nodes on the boundaries that will be shared by the shadow zone and the surfaces adjacent to it is the same as the distribution on the boundaries shared by the original (periodic) zone and its adjacent surfaces.

Note:

Files created prior to Release 15 written in mesher mode are automatically converted when read into mesher mode. These files may not contain sufficient information to properly set up periodic information in case of multiple periodic pairs.

Recover Periodic Boundaries

You can **Recover** periodic boundaries if it exists from a mesh file using the following procedure.

1. Read the mesh file.
2. In the **Make Periodic Boundaries** dialog box in Fluent meshing, select the periodic boundary zone(s) from the Boundary Zones list.

   Tip:

   You can also select the zone(s) graphically, and the names will be highlighted in the Boundary Zones list.

3. If it exists, periodic information (angle, pivot, and origin) will be extracted from the mesh file when it is read and will appear in the **Periodicity** dialog box. Alternatively, you can manually enter periodic information as described above.
4. Click **Recover**.

   The Recover feature will check for any existing face zone(s) at the periodic shadow boundary. If a shadow face zone is more complex than the corresponding leader, the recover operation will fail for that zone and a warning will appear in the console. You may be able to recover the periodic boundary by reversing the angle and selecting the more complex face zone(s).

Note:

- Recover first creates the periodic shadow boundary and then removes any duplicate face zone(s) at the periodic shadow boundary.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Periodic boundary recovery may be initiated using the text command **`boundary/recover-periodic-surfaces`**.

7.6.16. Removing Gaps Between Boundary Zones

Use the **Remove Boundary Gaps** dialog box to remove gaps between boundary zones.

1. Select the target zone in the graphics window. Click to set the target zone.
2. Select the zones for the gap removal operation and click to open the **Remove Boundary Gaps** dialog box. Alternatively, select the zones and use the hot-key **Ctrl**+**K**.
3. Specify an appropriate value for the **Min. Gap Distance**, **Max. Gap Distance**, and **Percentage Margin**.
4. Specify an appropriate value for **Critical Angle**. The critical angle is the maximum angle between the faces constituting the gap to be removed.
5. Click **Mark** to see the faces marked for projection.
6. Click **Remove** to remove the gaps between the objects selected.

7.6.17. Using the Loop Selection Tool

The loop selection tool can be accessed by clicking in the graphics window or using the hot-key **Ctrl**+**Shift**+**L**. This tool provides options for creating an open or closed loop of nodes. You can create an edge zone or capping surface based on the loop selected. You can also select positions instead of nodes to define the loop.

Figure 7.56: Loop Selection Toolbar

The following selection options are available:

- In the first group of tools, choose how the path between selected nodes/positions is defined - by edges, feature, boundary, or direct path. Click to switch between selecting nodes or positions to define the loop.
- The second group of icons is used to select open or closed loop. Then, for closed loop mode, you can choose how the path between the first and last nodes is defined - by edges, feature, boundary, or direct path.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Creating a Capping Surface

After making the necessary selections, click (hot-key **Ctrl**+**K**) in the Loop Selection mode to open the **Create Cap** dialog box. The **Create Cap** dialog box contains options for object/zone granularity and type and for remeshing the capping surface.

1. Specify the object/zone granularity.

   - Select **New Object** to create a new object for the face zones. Specify the object name and a label name. If the label name is not specified, the object name will be used as the label name. Face zone names will be the same as the label names. Select the **Object Type** and **Zone Type** from the lists.

   - Select **Add to Object** to add the face zones to an existing object. Select the object and specify a label name. If the label name is not specified, the default name **patch:#** (# indicates the ID) will be used. Face zone names will be the same as the label names. Select the **Zone Type** from the list.

   - Select **Add to Unreferenced** to create unreferenced face zones. Select the **Zone Type** from the list. The default name **patch:#** (# indicates the ID) will be used for the zone name. These zones will be available in the **Unreferenced** branch of the tree.

2. Enable **Remesh** to remesh the capping surface created.

3. Click **Create** in the **Create Cap** dialog box.

Creating an Edge Zone

After making the necessary selections, click (hot-key **Ctrl**+**L**) in the Loop Selection mode to open the **Create Edge Zones** dialog box. The **Create Edge Zones** dialog box enables you to add the edge zone to an existing object or create an unreferenced edge zone.

Selecting all Nodes

After making the necessary selections, click (hot-key **Ctrl**+**J**) in the Loop Selection mode to select all the nodes on the loop. These selections can then be used for operations such as collapsing, merging, or smoothing nodes.

For the list of hot-keys associated with the options in the Loop Selection toolbar, refer to Appendix C: Shortcut Keys (p. 877).

7.7. Wrapping Objects

Geometries migrated from various CAD packages often contain gaps and overlaps between the surfaces due to algorithm and tolerance differences of the CAD packages. Repairing such geometries manually

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

is a tedious and time-consuming process. Wrapping provides the ability to create reliable meshes for such geometries without extensive manual clean up and reduces the time required for preprocessing.

The object wrapping operation:

- extracts a conformal, well connected mesh object on the relevant surfaces of the objects selected.
- can repair gaps and overlaps in the model at the expense of a user-specified degree of geometry details.
- can handle unclean geometries and does not require a watertight representation of the geometry.
- can be used for defeaturing or when you need to walk over features.

The wrapper is useful in the following industrial applications:

- Automotive
  - Underhood thermal management (engine only, front car, full car)
  - Cabin HVAC
  - External aerodynamics
  - Brake cooling and engine cooling
- Aerospace
  - Engine core compartment
  - Cockpit HVAC, cabin HVAC
  - Landing gear
- Drill bit applications
- Smoke and fire spread
- Biomedical applications
- Other applications with bad input geometries

7.7.1. The Wrapping Process

The wrapping operation uses an appropriate material point to identify the relevant surfaces of the selected objects. A well-connected **mesh** object is created.

The general procedure for creating a wrapper surface is as follows:

1. A coarse Cartesian grid is overlaid on the selected objects (including gaps and overlaps) to create a contiguous region. This Cartesian grid is used to automatically clean the input geometry and to create the water-tight representation.
2. The Cartesian grid is then refined based on the size functions to better represent the selected objects.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. The intersection between the Cartesian grid and the input geometry is calculated and the intersecting cells are identified and marked.

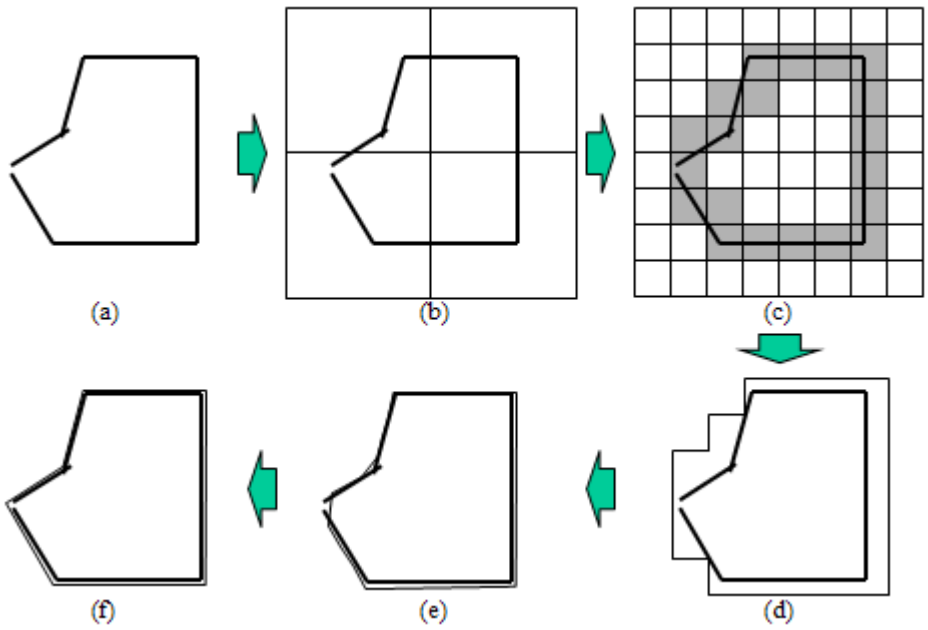
4. The interface is extracted on the boundary of the non-intersecting Cartesian volume region that encloses the material point. A watertight, faceted representation is created along the boundary of the intersecting cells.

5. The nodes on this faceted representation are projected onto the faces and feature edges of the input geometry which then results in a wrapper surface closely representing the input geometry. The edges are imprinted on the wrapped zones, and individual zones are recovered and rezoned based on the original geometry object(s).

6. The wrapper surface quality is improved by post-wrapping operations such as smoothing, swapping, and so on. Degenerate and island edges are deleted, and intersected and remeshed as appropriate. Surfaces are remeshed based on size functions/size field.

Figure 7.57: Schematic Representation of Wrapping Process (p. 706) is a simple illustration of these steps.

Figure 7.57: Schematic Representation of Wrapping Process



Note:

- If the global minimum size specified cannot be resolved, an error will be reported. Set up appropriate sizes and compute the size field before wrapping.

  If the minimum and maximum sizes from the size field cannot be resolved, increase the minimum size and recompute the size field before wrapping.

- The wrapping operation approximates the geometry using a stairstep-like Cartesian grid without projection. It requires finer cells to resolve thin gaps.

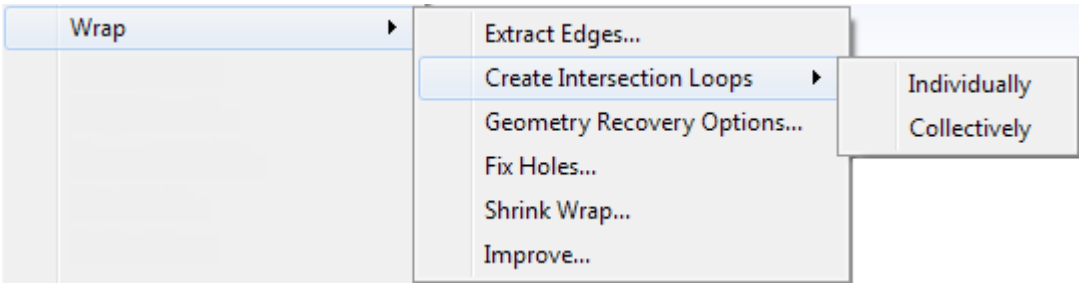
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In cases when a gap area is curved and not aligned to the Cartesian axes, you may need to refine 3–4 times finer than the gap thickness. This should be taken into account while setting the global and local minimum size for size functions and the cells per gap for the proximity size function being used.

You may specify a **Resolution Factor** to allow finer cells without changing the size function.

- The dimensions of the distortion (for example hole, gap, and so on) in the input geometry should be smaller than that of the size of the Cartesian cells created by the wrapper. If there is significant distortion in the input geometry, repair it to the extent that the distortion becomes smaller in size. Large holes, if present in the initial geometry, should be filled. Otherwise such holes will be ignored in the wrapping process.

The object wrapping utility is accessed using the context sensitive menus. Right click on any object and select **Wrap**. The following sections discuss tools and options used in the wrapping process.



Note:

The **Improve...** option is available only for mesh objects.

7.7.1.1. Extract Edge Zones
7.7.1.2. Create Intersection Loops
7.7.1.3. Setting Geometry Recovery Options
7.7.1.4. Fixing Holes in Objects
7.7.1.5. Shrink Wrapping the Objects
7.7.1.6. Improving the Mesh Objects
7.7.1.7. Object Wrapping Options

7.7.1.1. Extract Edge Zones

You can easily extract edge zones from existing face zones or selected surfaces using context menus from the tree or onscreen tools. In either case, edge zones are created using the **Fixed Angle** criterion as described in Creating Edge Zones (p. 672)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Using Onscreen Tools

You can use the onscreen tools to create edge zones interactively on selected face zones or selected surfaces. This method has the advantage of previewing any edges that meet the specified criteria before creating the edge zones.

1. Set the required Selection Filter.
   - Use **Face Selection Filter** to create edge zones on the surface(s) that contain the selected face(s).
   - Use **Zone Selection Filter** to create edge zones on the selected face zone(s).
2. Using the mouse probe, select the face(s) or face zone(s) for which edge zones are to be created.
3. Click the **Create Edge Zones** tool button ( ).
   - If a zone is selected, the **Interactive Edge Zone Creation** dialog box opens.
   - If a face is selected, the **Create Edge Zones By seed** dialog box opens.

   Any edges on the selected surface, or face zone, that meet the dihedral angle criterion will be highlighted.
4. Adjust the **Preview Edge Angle** slider as necessary to select the edge threads to be included in the new edge zone, based on the **Fixed Angle** criterion.
5. Click **Create Edge Zones**.

   The new edge zone will be added to the object that contains the selected face (or face zone).

Using Context Menus

You can use the context menus to extract edge zones for any selected geometry or mesh object in the tree.

1. In the **Outline View**, select the geometry object or mesh object.
2. Right-click on the selected object, and select **Wrap** → **Extract Edges...**.

   Note:

   This option is also available under the **Advanced** menu.

3. Specify the threshold **Angle**.

   From the **Option** list, choose whether **feature** or **all** edges are to be extracted.
4. Click **OK**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The extracted edge zones are placed in the **Unreferenced** branch of the tree.

Note:

- The **Extract Edges** dialog box offers the same functionality as the **Edge Zones** group on the **Operations** tab of the **Manage Objects** dialog box. See Object Manipulation Operations (p. 611).
- The **Feature Modify** and **Surface Retriangularization** dialog boxes also contain groups to modify or create Edge Zones. See Creating and Modifying Features (p. 672) or Remeshing Boundary Zones (p. 678).

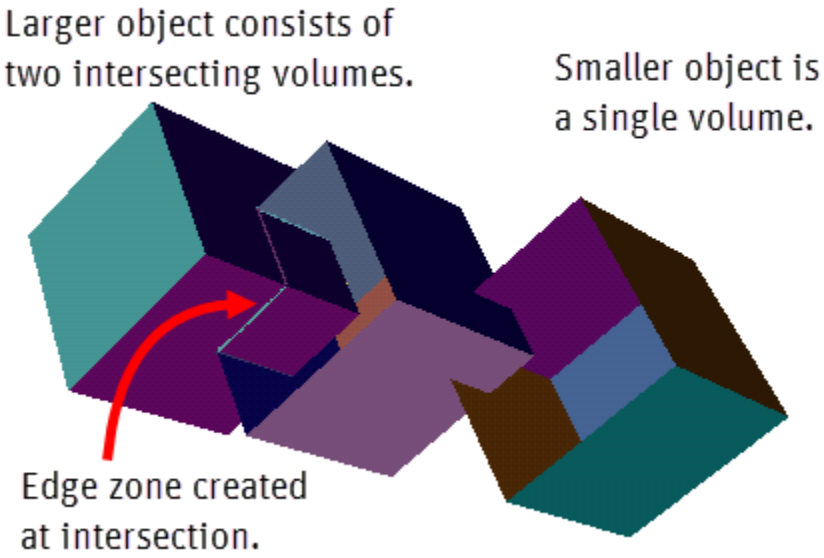
7.7.1.2. Create Intersection Loops

You can use these options to choose how face zones are processed within objects, prior to wrapping.

7.7.1.2.1. Individually

Within each object, edge zones are created on the loops where the face zones overlap. In Figure 7.58: Individual Object Loop (p. 709), after applying **Create Intersection Loops** → **Individually** to all objects, only one such edge zone is created - in the large geometry object at the intersection of the two cubic volumes.

Figure 7.58: Individual Object Loop

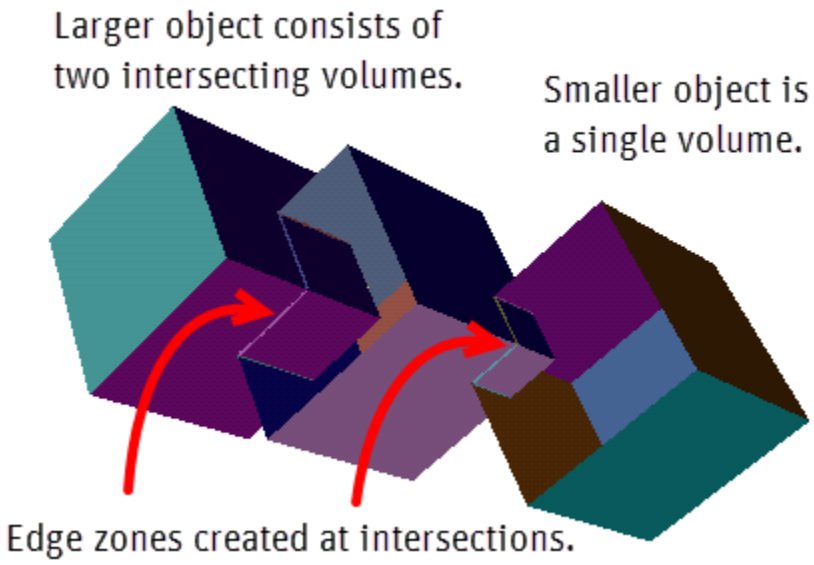


7.7.1.2.2. Collectively

Edge zones are created on the loops where face zones overlap, both within a single object and between multiple objects. In Figure 7.59: Collective Object Loops (p. 710), an additional edge zone is created where the large geometry object intersects the smaller object.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.59: Collective Object Loops



7.7.1.3. Setting Geometry Recovery Options

You can determine the surfaces and features which need to be captured with higher accuracy based on the usage of the model. For example, when using a car model for an external aerodynamic simulation, the recovery of external surfaces requires greater accuracy. Other components like the braking system or underhood components can be recovered with lower accuracy since they are not as significant for the simulation.

The recovery of features and surface mesh quality are controlled by setting the **Geometry Recovery** attribute for objects or face zones in the **Geometry Recovery Options** dialog box. The wrapped surface mesh inherits the geometry recovery attribute from the underlying objects or face zones.

Low

enables you to create a rough wrapped representation of the selected objects. Features are not preserved and the mesh quality is not as good as a CFD mesh.

High

enables better feature capture and high quality surface mesh. By default, the **High** attribute is set for all face zones in the mesh.

To set the geometry recovery attribute,

1. Right-click on any geometry or mesh object, and then select **Wrap** → **Geometry Recovery Options...** to open the **Geometry Recovery Options** dialog box.

2. Select the objects or face zones in the selection list.

3. Click **Apply** for the appropriate option in the **Geometry Recovery** group box.

To visualize the geometry recovery option, use the hot key **Ctrl**+**Shift**+**C** to go to Color Options Mode, then **Ctrl**+**G** to apply color by geometry recovery.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.7.1.4. Fixing Holes in Objects

The region/volume of interest should be well connected before the volume mesh can be generated. Any holes or leaks need to be located and fixed before proceeding with volume meshing. Holes/leaks can be located using a material point. When you select the material point and set the minimum and maximum limits for the hole size, those locations, within the set size limits, at which a path traces back to the material point through the geometry are recognized as holes or leaks.

When you refine an existing region or specify additional sizing functions for better representation of the geometry, the minimum size may be reduced to a size smaller than some existing leakages. These leakages can be detected automatically using the automatic hole detection tool in Ansys Fluent Meshing. The refinement of a region may result in the joining of previously separate regions through cells newly introduced by the refinement (region collision). While refining a region, Ansys Fluent Meshing will automatically detect cells causing region collision and group them. Such groups of cells will be identified as holes in the region. The number of holes exposed by the current refinement will be reported in the console.

The following operations allow you to automatically detect holes:

- Refining a single region.
- Specifying local size function and additional zone-specific sizes after initialization.
- Refining local regions defined according to requirements.

You can use the **Fix Holes** dialog box to locate and fix such holes as described in the following generic procedure:

1. Right-click on the selected geometry or mesh object(s) in the Model tree and then select **Wrap** → **Fix Holes...**.

   You can add or change **Objects** using the selection list in the dialog box. Click **Draw** to display only the selected object(s) in the graphics window.

2. Select an appropriate material point.

   The default material point (**external**) is a suitable point, external to the selected objects. Click **Change** to enable the **Material Point** drop-down list, if necessary.

3. Retain the **Use Size Field** option, if required.

   This will use a size field based on previously defined size controls or size functions to determine the limits for hole sizes.

   a. Set a **Resolution Factor** if desired.

      Setting a value less than 1 can help to find holes not aligned to the Cartesian axes by using finer sampling than the given size field.

      In situations where a hole's minimum size is larger than the size field's minimum size, using a Resolution Factor helps to find holes without modifying the size field. That is, you can use the same size field for both hole fixing and wrapping.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. Set the minimum and maximum limits for the **Hole Sizes** under consideration.

The size values enable you to limit the search for holes to a relevant subset based on the size range. As you locate the respective holes and fix them, the possibility of false holes is reduced.

You can click **Draw Sizes** to check the sizes on the geometry; click **Draw** to turn off size display.

In situations where a hole's minimum size is larger than the size field's minimum size, using the **Resolution Factor** helps to find holes without modifying the size field. That is, you can use the same size field for both hole fixing and wrapping.

You can alternatively choose to use a specific size to find the holes or leaks in the mesh.

To find holes based on a specified size, disable **Use Size Field**. Enter an appropriate value for **Min Size** and click **Apply**.

The **Max Size** can be set to limit the search to a relevant size range.

4. Click **Find Holes** to locate all the holes based on the specified parameters.

The number of holes is reported in the **Count** field in the **Wetted Holes** group box.

Using the Pan Regions Dialog Box

The options in the **Pan Regions** dialog box enable you to observe and analyze the region to be wrapped, overlayed on a plane aligned with the Cartesian grid. The plane can be passed through the selected region (or all available regions) along the X, Y, or Z direction, as required. The interior of the selected region(s) is displayed on the plane, at every position of the plane. Click **Pan Regions...** to open the **Pan Regions** dialog box.

You can also overlay the boundary surfaces while panning, and clip the boundary surfaces on either side of the cutting plane.

To pan through all the regions of your choice, do the following:

1. Display the required region (using the **Draw** button below the **Regions** list).
2. Click the **Pan Regions...** button to open the **Pan Regions** dialog box.
3. Select the appropriate axis along which you want to pan through the selected regions from the **Direction** list.

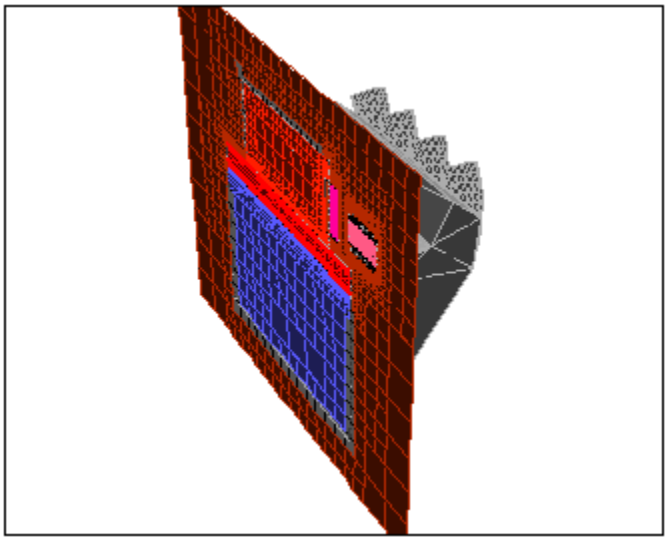
The **Start**, **End**, and **Increment** fields will be updated automatically based on the cell size distribution of the Cartesian grid. You can change these values as appropriate.

4. Enable **Overlay Graphics** if you want to see the geometry along with the pan plane.

Select **Positive** or **Negative** to clip the surfaces on the positive or negative side of the cutting plane. Figure 7.60: Overlaid Geometry Clipped with the Pan Plane (p. 713) shows the geometry on the positive side of the cutting plane.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.60: Overlaid Geometry Clipped with the Pan Plane



5. Select the regions to be analyzed in the **Region** selection list. By default, all regions are selected. In cases comprising a large number of Cartesian regions, you can pan through selected regions instead.

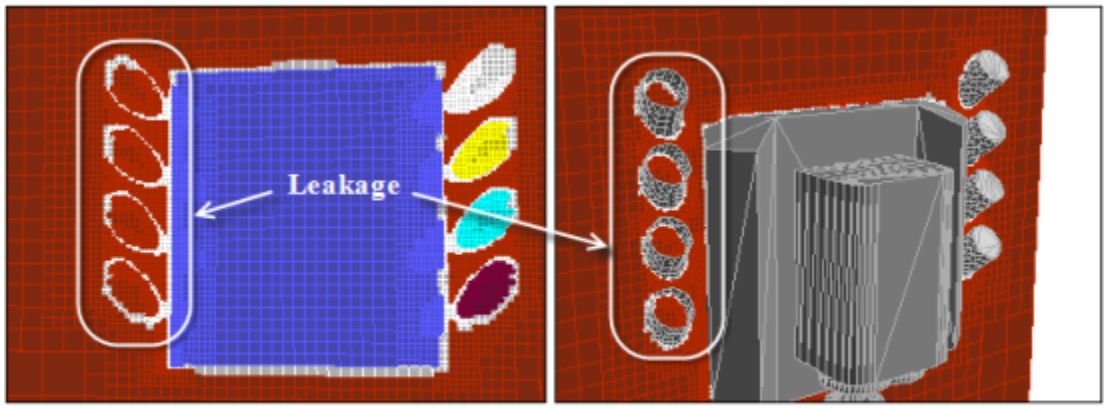
6. Click **Pan** to start the plane movement through the selected regions.

   Alternatively, use the arrow buttons or the slider to manually move the pan plane to a particular location.

   The interior of the Cartesian grid is displayed on every position of the plane during its movement through the region. Increase (or decrease) the **Increment** value to increase (or decrease) the speed of movement of the plane.

   Figure 7.61: Leak Detection Using the Pan Regions Dialog Box (p. 713) shows how the leakage can be detected using the **Pan Regions** dialog box. If, at any position of the plane, the color of the region is seen inside the geometry, there may be a leak or hole in the Cartesian grid.

Figure 7.61: Leak Detection Using the Pan Regions Dialog Box



7. Adjust the plane at the position where the leak is seen on the plane.

> **Warning:**
>
> The **Pan Regions** dialog box only enables you to know whether or not there is a leak or hole in the geometry. If the geometry has a hole or leak, you need to find its exact location and fill it.

Closing holes

Options for closing or opening the holes are in the **Operations** group box:

- You use the **Wetted Holes** group box to traverse the holes detected, or to globally patch or open all the holes.
  - Click **Draw All** to view all the wetted holes detected.
  - Click **First** to view the first hole. The display will be limited to the region of the hole.

    Click **Next** repeatedly to traverse all the wetted holes and examine them individually.
  - Click **Patch All** to automatically patch all the wetted holes detected.
  - Click **Open All** to open all the wetted holes detected (when the holes identified do not represent actual holes).
- You use the **Selected Holes** or **Create Patch** group box to close the displayed hole individually.
  - Click **Patch** to automatically close the currently displayed hole.
  - Click **Open** if the currently displayed hole does not represent an actual hole. Opening a hole enables you to indicate that the approximated wetted region should propagate through the configuration detected as a hole.
  - Click **Ignore** if the currently displayed hole is not relevant for the object wrapping/sewing operation.
  - Click **Cylinder...** to open the **Cylinder** (p. 696) dialog box to create a cylindrical surface to fix the hole.
- You use the **Trace to Points** group box to locate holes or leaks by tracing a path from the **Material Point** to a selected **Target Point** through all the objects. Even after you fix the holes/leaks, you can use the **Trace to Points** options to verify that no tiny leaks remain.

  1. Select a point from the **Target Points** selection list and click **Trace**.

     The path connecting the target point to the material point will be highlighted and will pass through the hole/leak.
  2. Fix the hole using the options in the **Wetted Holes**, **Selected Holes**, or **Create Patch** group box, as appropriate.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click **Update** in the **Wetted Surface** group box to update the wetted surface representation.
4. Click **Update** in the **Trace to Points** group box and then click **Trace** again to locate any remaining holes/leaks.
5. Repeat the hole fixing steps as needed to ensure no holes/leaks remain.

- You use the options in the **Wetted Surface** group box to view the approximate representation of the volume or region of interest based on the objects and material point selected. You can also create a mesh object using the shrink-wrap method.
  - Click **Show** to view the approximate representation of the wetted surface, based on the objects and material point selected. Enable **Overlay Graphics** to view the object(s) along with the wetted surface.

    Click **Hide** to hide the wetted surface in the graphics window.
  - Click **Update** to update the wetted surface representation after any hole fixing operations (patch, open, creating caps, and so on) are performed.

    Examine the updated region for further leaks or holes. Fill any remaining leaks or holes before proceeding.

After all the holes/leaks are fixed you can proceed to wrap the objects. Specify an appropriate **New Object Name** and **New Label Name** (if required). Click **Shrink Wrap** to create a mesh object for the selected objects using the shrink-wrap method.

Important:

The octree refinement used to identify and repair holes occurs in a different order than when used to wrap objects. Thus, using this button may generate a different mesh than if you use the **Wrap** dialog box as described in Shrink Wrapping the Objects (p. 715)

7.7.1.5. Shrink Wrapping the Objects

The **Wrap** dialog box contains options for shrink wrapping the selected objects. Select the objects to be wrapped in the Outline View and then select **Wrap** → **Shrink Wrap...** from the context sensitive menu to open the **Wrap** dialog box.

1. Select the appropriate option, **Individually** or **Collectively**, from the **Target** list. These options are described in Creating Individual Mesh Objects (p. 715) and Creating a Collective Mesh Object (p. 716).

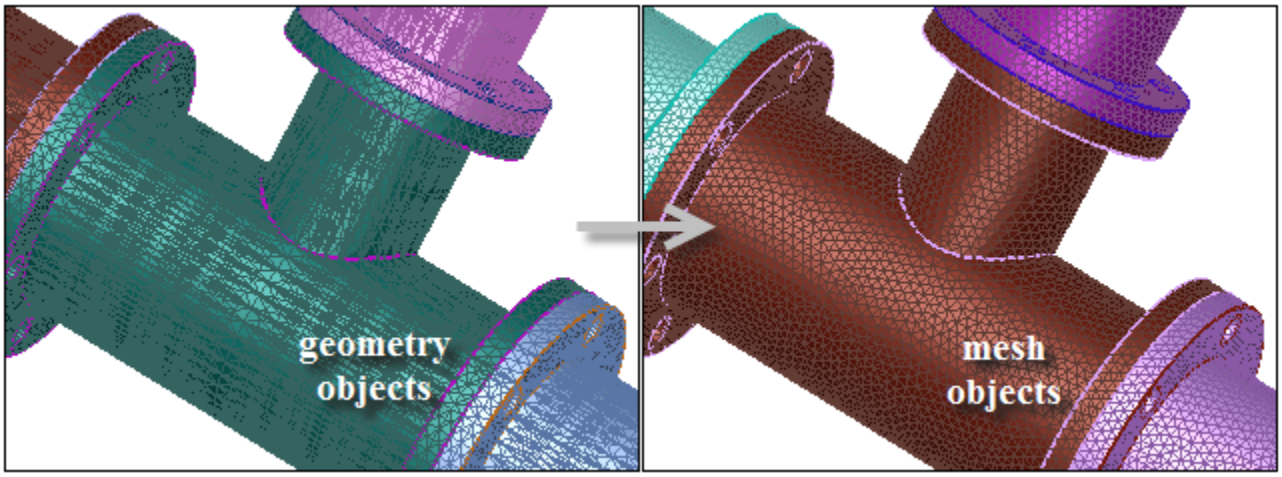
Creating Individual Mesh Objects

The **Individually** option enables you to create a conformal surface mesh for each object selected. A well-connected mesh object and corresponding zones will be created for each object. The mesh object name is the original object name with the suffix **-mesh**. This operation uses a suitable material point that is external to the objects selected. Hence, any internal voids or features will be eliminated. The mesh objects created are suitable for repair operations such as gap or thickness removal, or as the final surface mesh.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.62: Wrapping Individual Objects (p. 716) shows the mesh objects created for geometry objects using the **Individually** option.

Figure 7.62: Wrapping Individual Objects

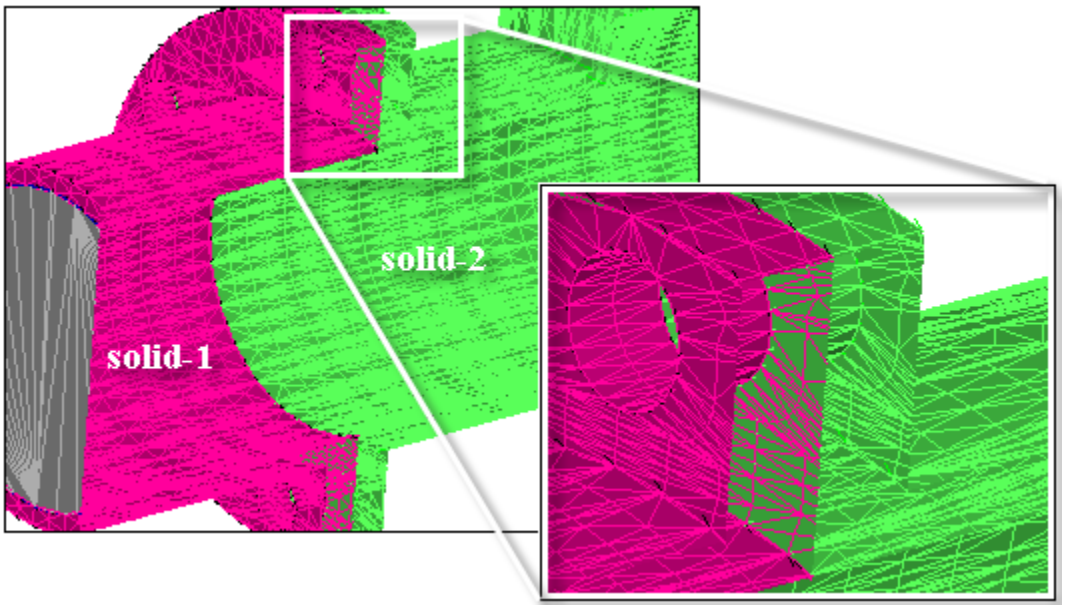


Creating a Collective Mesh Object

The **Collectively** option enables you to create a single, well-connected mesh object and corresponding zones, based on the objects selected. You can specify an appropriate name for the mesh object and a new label name.

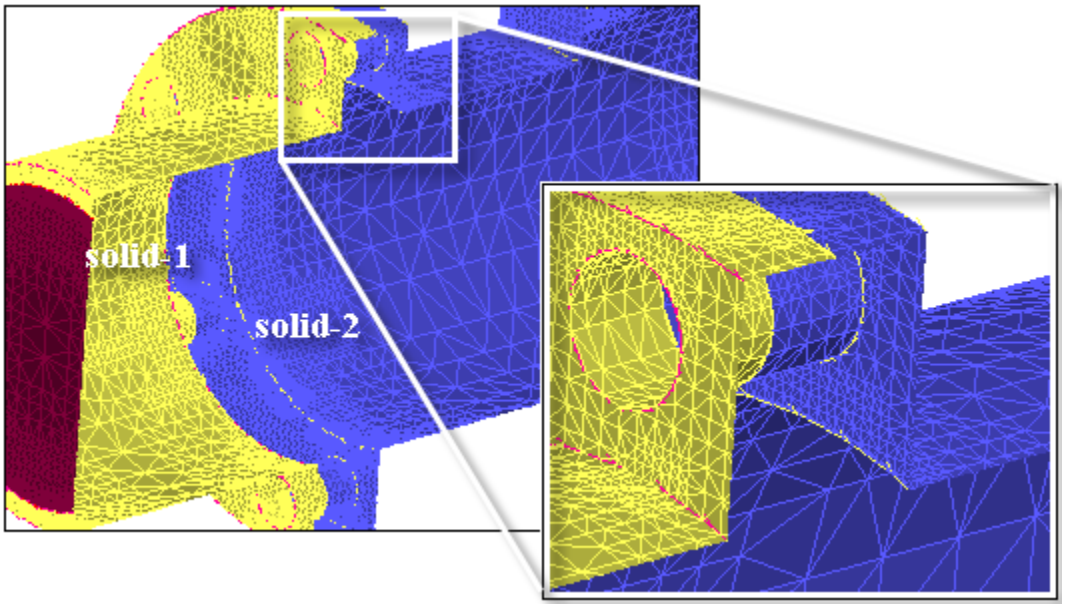
When you select **external** in the **Material Point** drop-down list, a suitable reference point external to the objects is selected for the object wrapping operation. Hence, any internal voids or features will be eliminated. Figure 7.63: Multiple Solids (p. 716) shows an example with multiple solid objects. The aim of the object wrapping operation is to mesh the solids conformally and create a single solid cell zone in the final mesh. Figure 7.64: Single Solid Surface (p. 717) shows the mesh object created, where the multiple solids are unified.

Figure 7.63: Multiple Solids



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

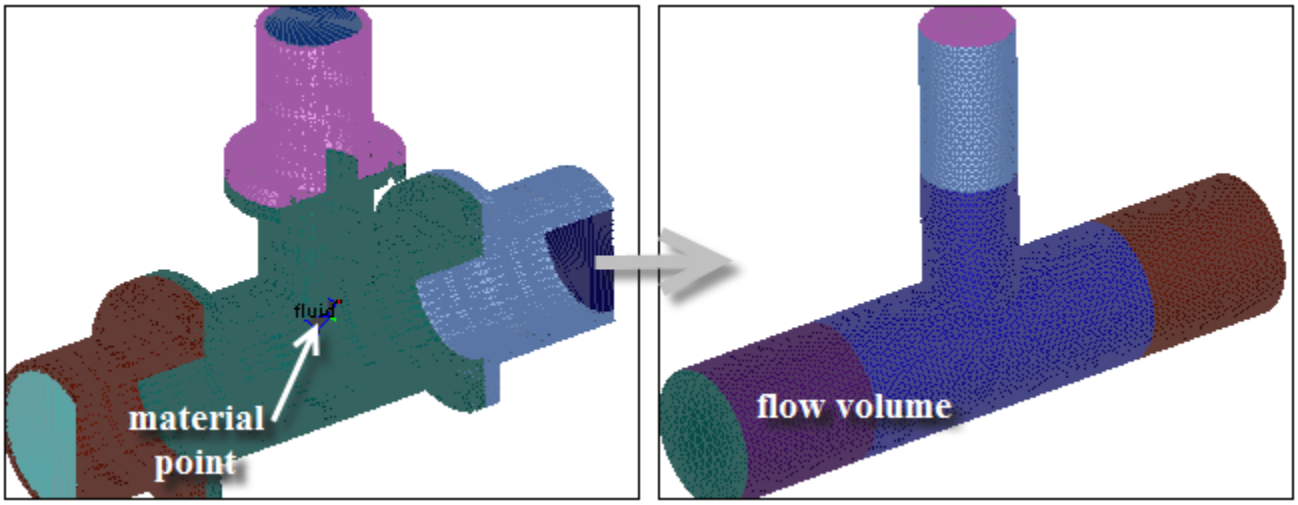
Figure 7.64: Single Solid Surface



Alternatively, you can use this option to create the flow volume, when the surrounding solids are not needed in the final mesh. All objects bounding the flow volume should be selected for the wrapping operation. Select the appropriate material point needed to identify the "wetted" region that is the flow volume. The material point can be defined in the **Material Points** dialog box.

Figure 7.65: Extracting the Flow Volume (p. 717) shows the extraction of the internal flow volume for a T-junction by specifying a material point and using the **Fluid Surface** option.

Figure 7.65: Extracting the Flow Volume



- For the **Collectively** option, specify the **New Object Name** and **New Label Name**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Set the **Resolution Factor**. This field can be used to set sampling coarser or finer than the final surface mesh.

   Tip:

   For industrial size problems where the geometry recovery is set to high, it is recommended that the resolution factor not be smaller than 0.7.

Important:

Wrapping models with a large number of face zones may cause significant slow down in performance. Merging face zones with the same boundary conditions will improve the efficiency of the wrapping operation.

Important:

When wrapping large, complex geometries with small mesh sizes, you may need to increase the memory allocation. From the **Display** → **Controls** menu, select **Shrink Wrap** in the **Categories** drop down list, and specify a new value for **Max Octree Memory**.

Important:

By default, the initial bounding box for shrink wrapping is the **global** bounding box. In multiple session wrapping, a newly created wrapper surface may increase the global bounding box, which affects subsequent wrapper results. To avoid this issue, go to the **Display**+**Controls** menu, select **Shrink Wrap** in the **Categories** drop down list, and choose **selection** for the **Initial Bounding Box**.

You can use the command `/objects/wrap/wrap` to create the mesh object. Specify the objects to be wrapped and other relevant parameters.

The command `/objects/wrap/set/shrink-wrap-rezone-parameters` enables you to set the parameters for improving the mesh object surface quality using rezoning. The geometry object zones will be separated based on the separation angle specified to improve the feature imprinting on the mesh object.

7.7.1.6. Improving the Mesh Objects

The options in the **Improve** dialog box enable you to improve the surface mesh quality of mesh objects.

The wrapper surface created after imprinting is of good quality and it represents the input geometry very well in most regions. However, you can improve it further in some regions of the geometry such as sharp corners and curves. The post wrapping improvement operations allow you to improve the wrapper surface by performing various operations such as smoothing, swapping, inflating thin regions, removing crossovers, and so on.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the mesh objects in the Outline View. Right-click and select **Wrap** → **Improve...** from the context sensitive menu.
2. Choose the appropriate **Method**.
   - The **Smooth and Improve** option improves the mesh by a combination of smoothing, swapping, and surface mesh improvement operations. Object normals are correctly oriented and island faces are also deleted. You can optionally coarsen the surface mesh by specifying a suitable **Coarsening Factor**. Additional **Imprint** operations can be done to improve feature capture on the surface mesh.

     Note:

     Using the **Imprint** option may result in quality deterioration.

   - The **Surface Remesh** option improves the mesh by remeshing based on the current size field. Object normals are correctly oriented and island faces are also deleted.
3. Set **Coarsening Factor** and **Imprint** as desired.

   The wrapper surface created may be finer than you require in some regions. You can coarsen the mesh in such regions or globally for the entire wrapper surface. This operation also reduces the cell count of the mesh, thereby reducing the computation time.
4. Click **Improve**.

Note:

Additional controls can be set in the **Objects** category in the **Display** → **Controls** dialog box for improving the mesh object quality.

Enable the **Improve Quality Aggressively** option to collapse faces without preserving boundary features. This option is disabled by default.

You can alternatively use the command `/objects/improve-object-quality`.

7.7.1.7. Object Wrapping Options

The following object wrapping options are available:

7.7.1.7.1. Resolving Thin Regions During Object Wrapping
7.7.1.7.2. Detecting Holes in the Object
7.7.1.7.3. Improving Feature Capture For Mesh Objects

7.7.1.7.1. Resolving Thin Regions During Object Wrapping

Surfaces in close proximity constitute thin regions in the mesh. Examples of thin regions include sharp corners, trailing edge configurations, and so on, which may not be recovered accurately

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

enough during the object wrapping operation and surface elements may span between nodes on the proximal surfaces.

You can use the command `/objects/wrap/set/include-thin-cut-edges-and-faces` to enable better recovery of such configurations during the object wrapping operation.

7.7.1.7.2. Detecting Holes in the Object

The command `/objects/wrap/set/report-holes?` enables you to check for holes in the object created. Holes, if any will be reported at the end of the object wrapping operation.

The command `/objects/wrap/set/max-free-edges-for-hole-patching` enables you to set the maximum number of free edges in a loop to fill the holes.

The command `/objects/wrap/check-holes` enables you to check for holes in the objects. The number of hole faces marked will be reported.

7.7.1.7.3. Improving Feature Capture For Mesh Objects

The command `/objects/improve-feature-capture` enables you to imprint the edges comprising the mesh object on to the object face zones to improve feature capture for mesh objects. You can specify the number of imprinting iterations to be performed.

Note:

The geometry objects used to create the mesh object should be available when the `improve-feature-capture` command is invoked. Additionally, the face zones comprising the objects should be of type other than geometry.

7.8. Creating a Mesh

After reading the boundary mesh and performing the necessary modifications (such as merging duplicate nodes, edge swapping) you will create the volume mesh.

Depending on the type of mesh you are starting from (boundary mesh only, boundary mesh with hexahedral cells in one or more regions, and so on), you can automatically generate the mesh, manually generate the mesh step by step, or use a combination of manual and automatic commands.

You can create several types of meshes comprising different element types. The meshing strategy and the use of the **Auto Mesh** tool are described in this chapter. Information on how to deal with thin regions, quad-tet transition elements (pyramid and non-conformal meshing), and creation of a heat exchanger zone are also described. Detailed descriptions of meshing techniques such as prism meshing, tetrahedral and hexcore meshing options, and so on are described in subsequent chapters.

7.8.1. Choosing the Meshing Strategy

7.8.2. Using the Auto Mesh Dialog Box

7.8.3. Generating a Thin Volume Mesh

7.8.4. Generating Pyramids

7.8.5. Creating a Non-Conformal Interface

7.8.6. Creating a Heat Exchanger Zone

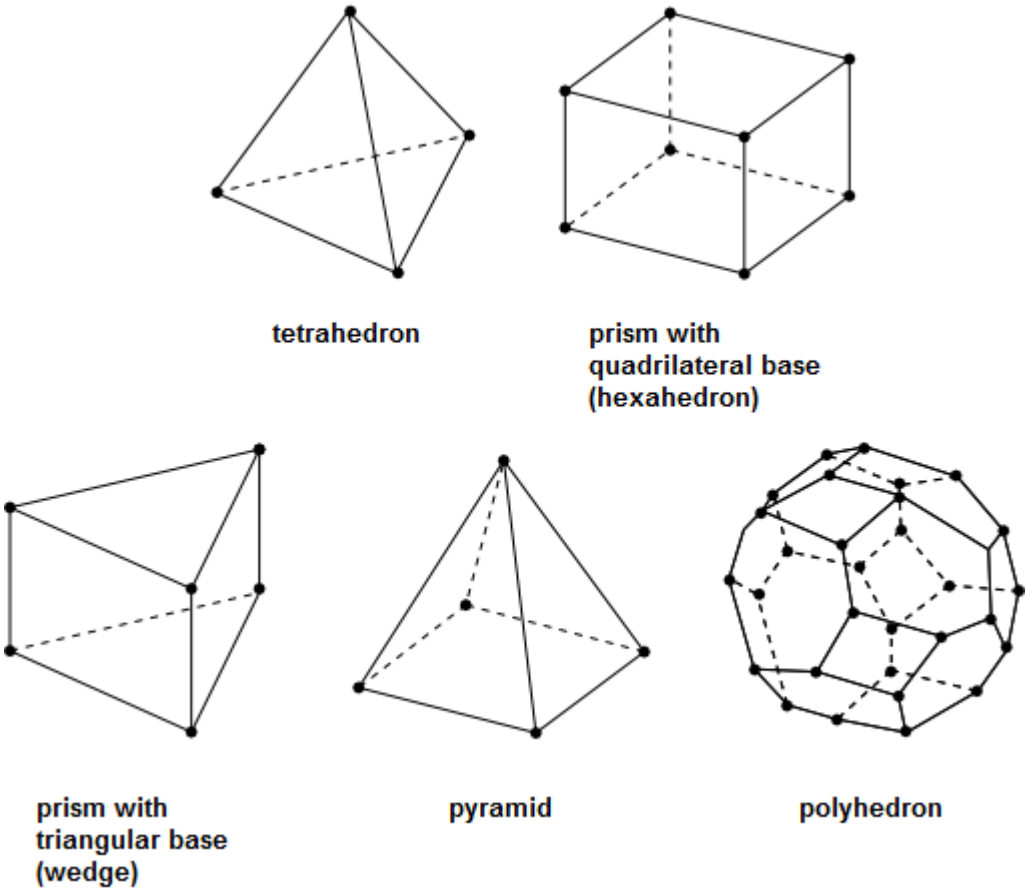
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.8.7. Parallel Meshing

7.8.1. Choosing the Meshing Strategy

You can use a variety of cell shapes to mesh the domain:

Figure 7.66: Possible Mesh Cell Shapes



A mesh consisting entirely of tetrahedral elements is a tetrahedral mesh, and a mesh with any combination of cell shapes is referred to as a hybrid mesh.

Before generating a volume mesh, determine the shapes that are appropriate for the case you are solving, then follow the instructions for creating the required cell types. Most cases will fall into one of the following categories:

7.8.1.1. Boundary Mesh Containing Only Triangular Faces

7.8.1.2. Mixed Boundary Mesh

7.8.1.3. Hexcore Mesh

7.8.1.4. CutCell Mesh

7.8.1.5. Rapid Octree Mesh

7.8.1.6. Additional Meshing Tasks

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.8.1.7. Inserting Isolated Nodes into a Tet Mesh

7.8.1.1. Boundary Mesh Containing Only Triangular Faces

If you require a high mesh resolution in some portion of the domain, such as a boundary layer, you can obtain an efficient and better quality mesh by meshing that portion with prisms (wedges) and then meshing the rest of the domain with tetrahedra (tets). The resulting mesh is referred to as a viscous hybrid mesh.

The procedure is as follows:

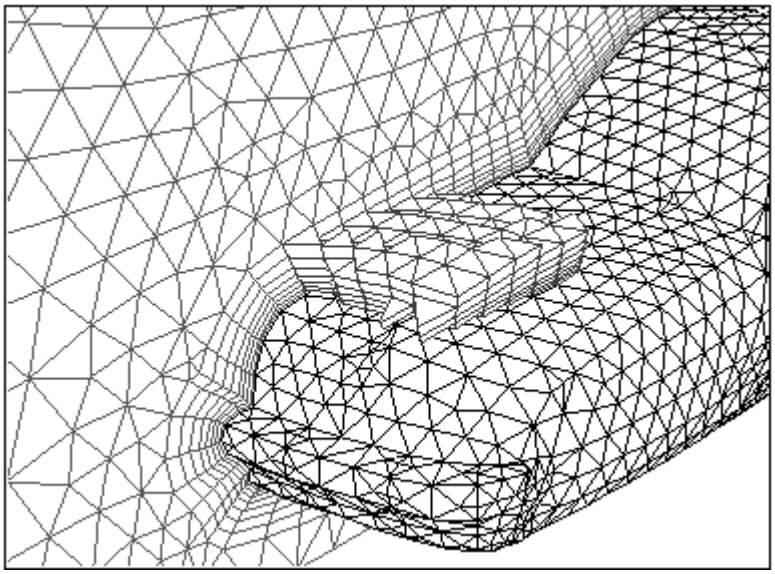
1. Build one or more layers of prisms, starting from the appropriate boundary (or boundaries). Refer to Generating Prisms (p. 750) for details.

2. Create a domain encompassing the region to be meshed with tetrahedra. Refer to Using Domains to Group and Mesh Boundary Faces (p. 528) for details.

3. Generate the tets in the selected domain using either an automatic tet mesh generation, a manual tet mesh generation, or a combination of the two. Refer to Generating Tetrahedral Meshes (p. 780) for details.

Figure 7.67: Mesh with Prisms in a Boundary Layer Region (p. 722) shows several layers of prisms in a portion of a mesh created in this manner. The prisms extend throughout the entire region bounded by the quadrilateral faces, but only a few of them are shown here.

Figure 7.67: Mesh with Prisms in a Boundary Layer Region



The surface mesh originally contained only triangular faces. The quadrilateral faces are created automatically when the prisms are built on the triangular faces.

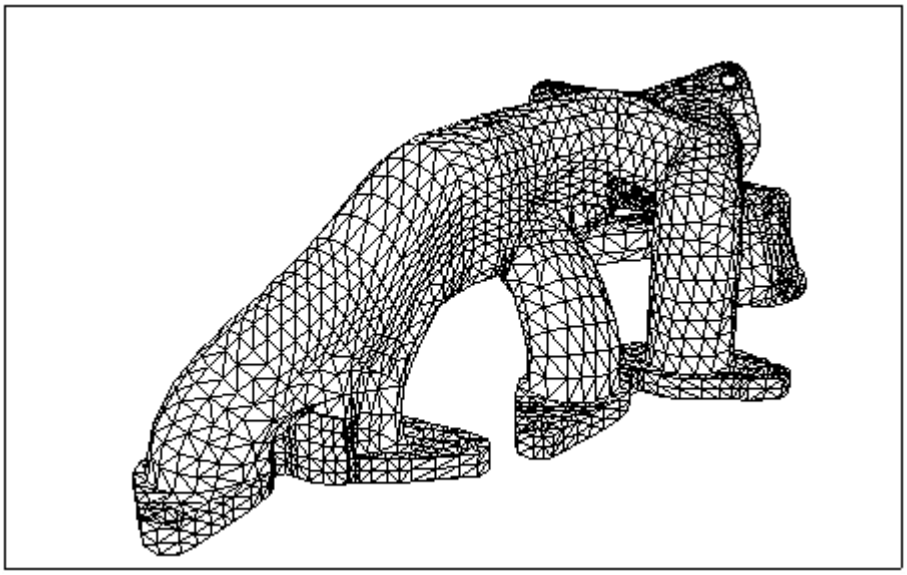
If the quadrilateral faces of the prisms do not lie on the external boundary of the domain (that is, if the prism region begins and/or ends in the interior of the domain), create a layer of transitional pyramids between steps 1 and 2. Refer to Generating Pyramids (p. 735) for details.

If you have no special boundary layer resolution requirements, you can generate a mesh consisting entirely of tetrahedra (see Figure 7.68: Surface Mesh Containing Only Tetrahedra (p. 723)). You can

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

use the automatic tetrahedral mesh generation procedure, the manual procedure, or a combination of both. Refer to Generating Tetrahedral Meshes (p. 780) for details.

Figure 7.68: Surface Mesh Containing Only Tetrahedra



7.8.1.2. Mixed Boundary Mesh

Start from a boundary mesh that contains triangular and quadrilateral faces as well as hexahedral cells in the quadrilateral face regions. The resulting mesh is referred to as a *zonal hybrid mesh*.

1. Add a layer of pyramids to the quadrilateral boundary face zone that lies between the hexahedral region and the adjacent region to be meshed with tetrahedra. This creates the triangular boundary face zone that is required to create tetrahedra in the adjacent region. Refer to Generating Pyramids (p. 735) for details.

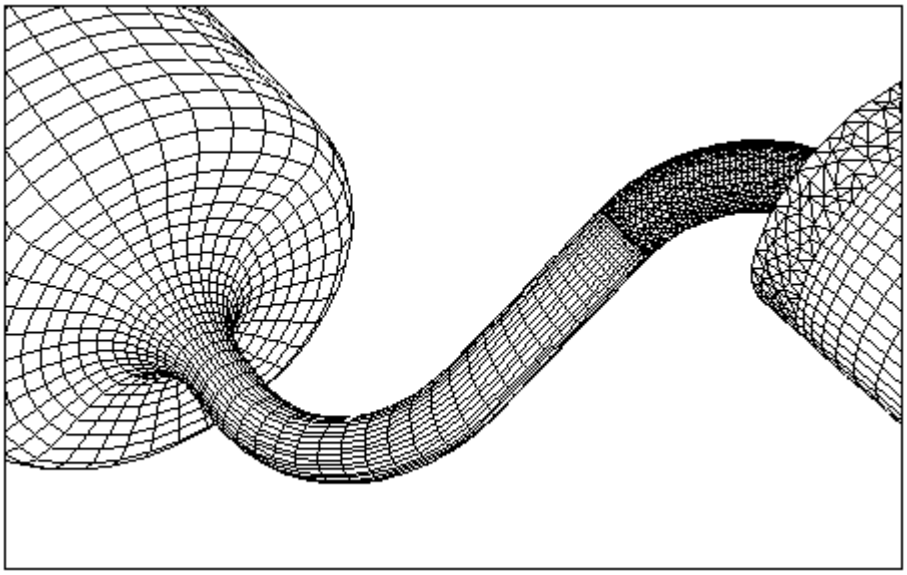
2. Create a domain encompassing the region to be meshed with tetrahedra. Refer to Using Domains to Group and Mesh Boundary Faces (p. 528) for details.

3. Generate the tetrahedra in the selected domain using either automatic or manual tet mesh generation, or a combination of both. Refer to Generating Tetrahedral Meshes (p. 780) for details.

Figure 7.69: Surface Mesh (p. 724) shows the surface mesh for a portion of a grid containing hexahedra, pyramids, tetrahedra, and prisms that was created on a plenum feeding a valve-port cylinder.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.69: Surface Mesh



- The less complicated plenum pipe on the left of the figure is meshed using hexahedral cells.
- The more complex valve port (valve not visible because it is inside the surrounding pipe) is meshed using tetrahedra.
- Pyramids are used as a transition between the hexahedral grid for the plenum and the tetrahedral grid for the valve port. This transition occurs where the triangular and quadrilateral faces meet in the middle of the figure.
- Additionally, the quadrilateral faces produced by extending triangular faces in the cylinder (that is, the quadrilateral sides of the resulting prism wedges) can be seen in the far right of the figure.

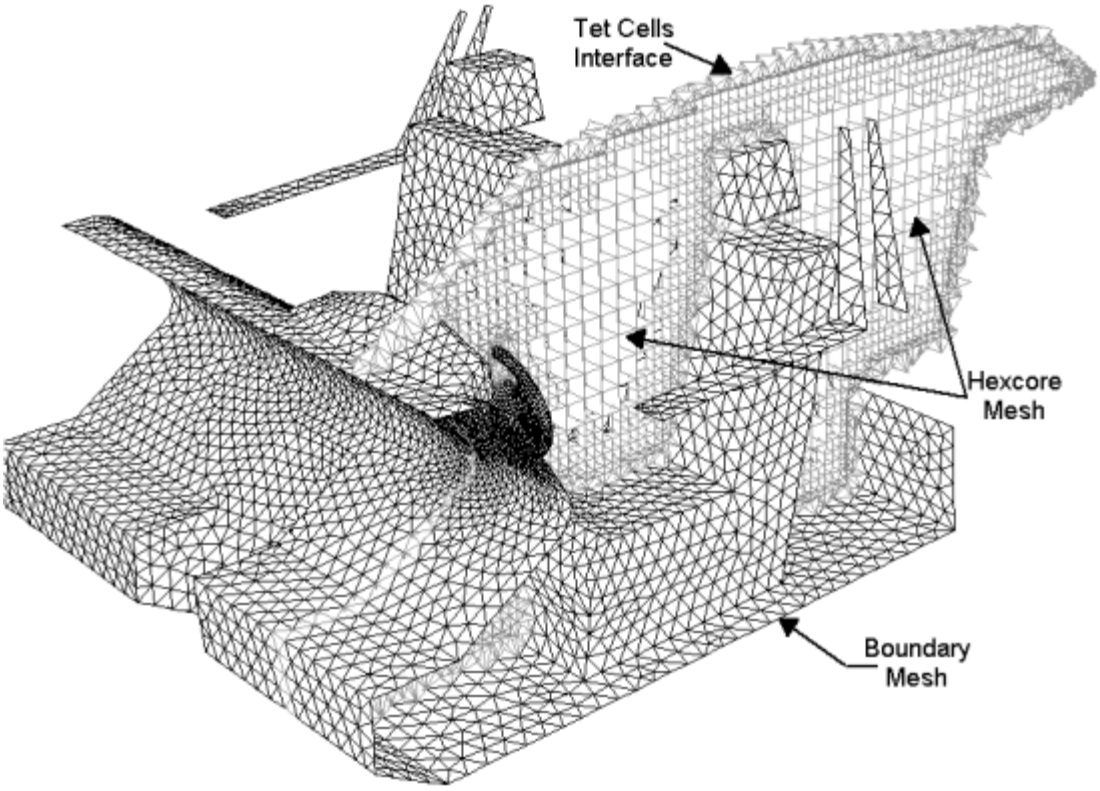
7.8.1.3. Hexcore Mesh

The hexcore mesh features a tetrahedral/hybrid mesh adjacent to walls and a Cartesian mesh in the core flow region. Figure 7.70: Hexcore Mesh (p. 725) shows the typical hexcore mesh. The hexcore meshing scheme creates a mesh consisting of two regions:

- An inner region composed of regular Cartesian cells.
- An outer region consisting of tetrahedral elements.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.70: Hexcore Mesh



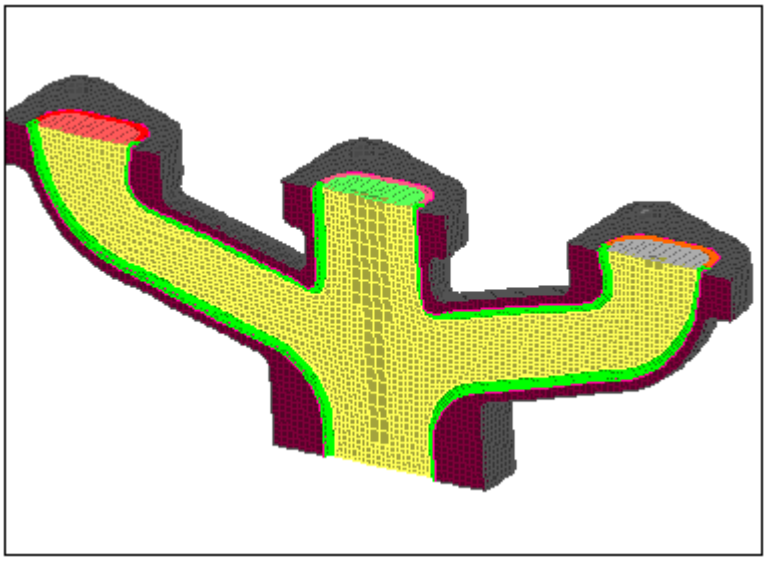
Wedge elements are created only when boundary layers are attached on faces pre-meshed with triangular elements. It combines the automation and geometric flexibility of tetrahedral/hybrid meshes with greatly reduced cell counts in many applications. The hexcore mesh is most beneficial in geometries with large open spaces, as in automotive, aerospace, and HVAC applications. Refer to Generating the Hexcore Mesh (p. 795) for details.

7.8.1.4. CutCell Mesh

CutCell meshing is a general-purpose hex-dominant meshing technique that can be used instead of tetrahedral or hexcore meshing, without requiring a very high-quality surface mesh as a starting point. This method uses a direct surface and volume approach without the need of cleanup or decomposition, thereby reducing the turnaround time required for meshing. A key feature is the large fraction of hex cells in the mesh.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.71: CutCell Mesh



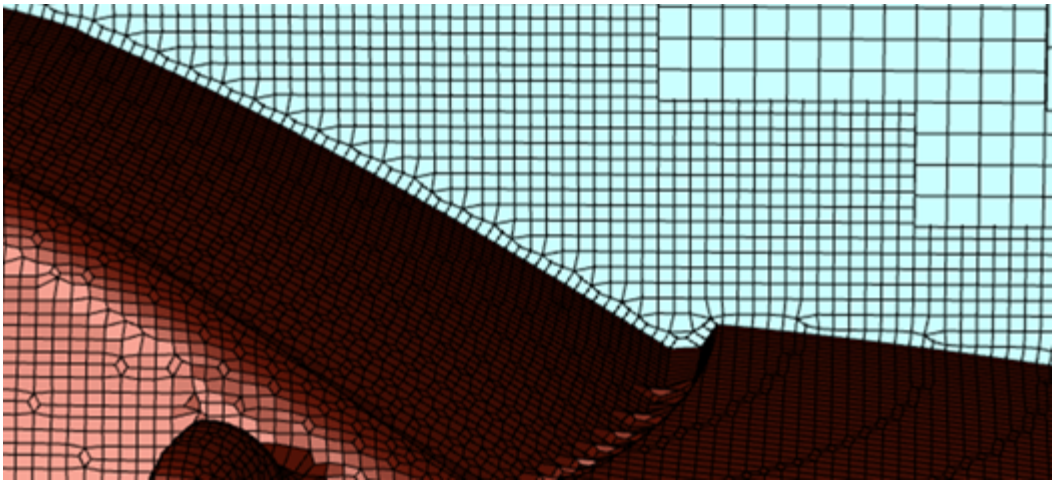
Refer to Generating the CutCell Mesh (p. 810) for details.

7.8.1.5. Rapid Octree Mesh

The Rapid Octree mesher enables a fast hexahedral mesh generation with two choices of boundary treatment methods to resolve the geometry. The two boundary treatment methods are:

- boundary projection to create a single prismatic layer on the surface with optional local refinements
- Cartesian snapping of the closest node onto the boundary for constant-size surface mesh generation

Figure 7.72: Rapid Octree Mesh with Projection Boundary Treatment Method



The generated mesh topology is very similar to the hexcore mesh generation and provides a mixed hexahedral and (for transitional cells) polyhedral cell description. The method used for the evaluation of cell quality is changed to ortho skew by default at the end of the meshing process (for details about quality, see Quality Measure (p. 566)). Since the mesh generation is designed to scale on

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

clusters that include thousands of cores, it is very well suited for the generation of meshes with a very high cell count for scale-resolving simulations (for example, automotive external flows), as well as for any application where geometric defeaturing can be accepted to gain robustness and speed.

As the meshes are close to being uniform in the streamwise, spanwise, and wall normal directions, they are most suitable for Wall-Function LES (WFLES). A minimum mesh resolution for boundary layers would be 5x5x5 cells per boundary layer volume.

Refer to Generating Rapid Octree Meshes (p. 828) for details.

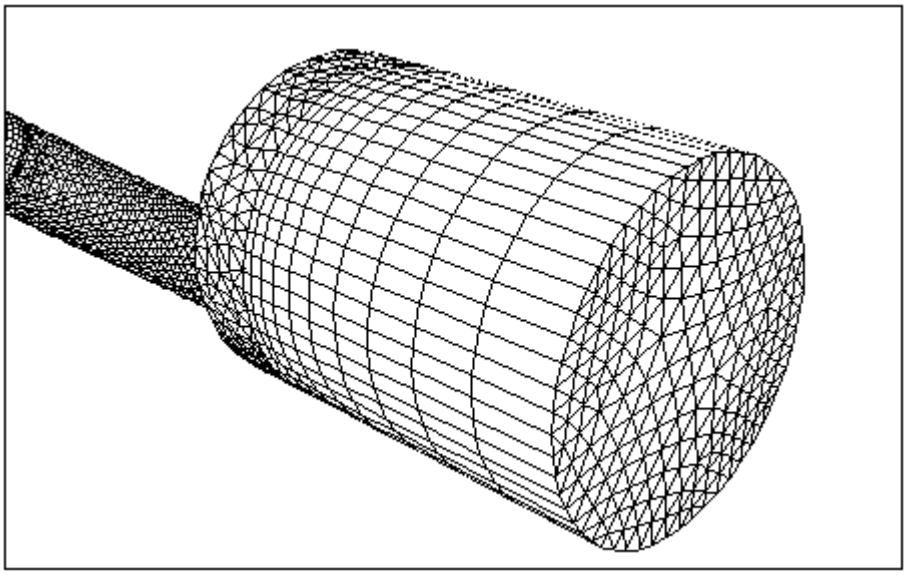
7.8.1.6. Additional Meshing Tasks

Additional meshing tasks that can be handled are:

- If you have a complete volume mesh and want to extend some portion of the domain (for example, increase the length of an inlet pipe), you can grow one or more layers of prisms from the current external (quadrilateral or triangular) boundary.

  Figure 7.73: Extending an Existing Tetrahedral Mesh Using Prisms (p. 727) shows a region of prisms (wedges) extended from the triangular face zone that bounds a tetrahedral region.

  Figure 7.73: Extending an Existing Tetrahedral Mesh Using Prisms

- Unless there is a reason to use hexahedral cells in the quad regions, it is preferable to convert a mixed tri/quad boundary mesh to an all-tri boundary mesh and then create a tetrahedral mesh for a 3D boundary mesh consisting of only triangular faces.

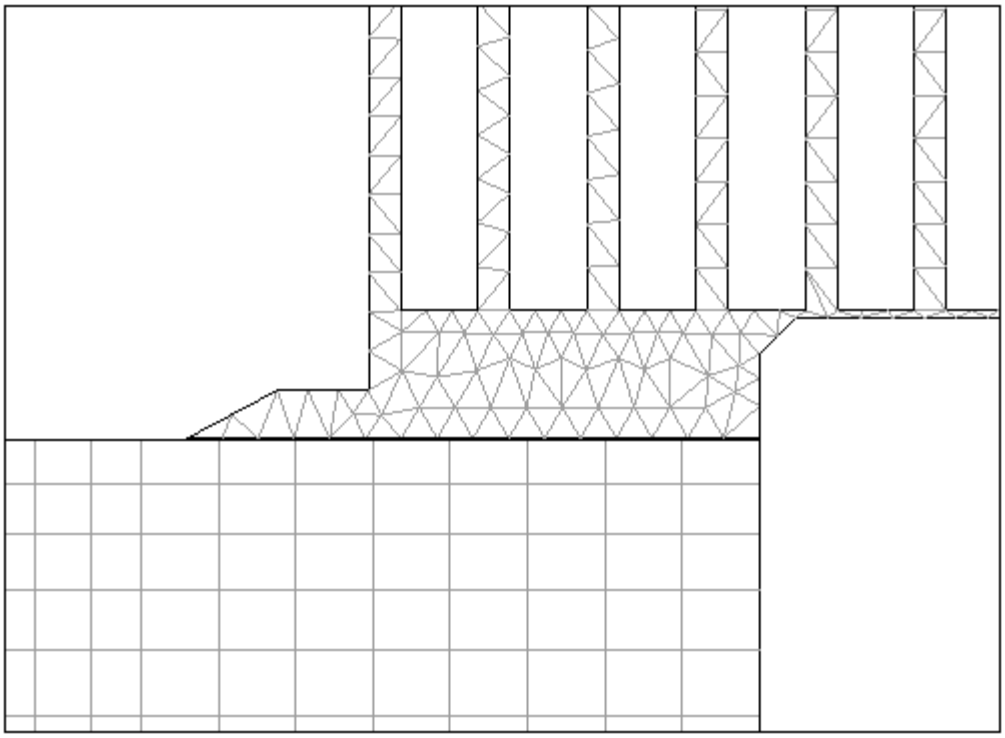
  Use the **Triangulate Zones** dialog box or the command `/boundary/remesh/triangulate` to convert quad face zones to tri face zones.

  You can then use the **Boundary Improve** dialog box to improve the skewness of the triangular boundary zone created.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- You may choose to use a non-conformal interface to define the computational domain. This type of interface enables you to relax the requirement for point-to-point matching at the interface between the meshes, as illustrated in Figure 7.74: Example of a Non-Conformal Interface (p. 728).

Figure 7.74: Example of a Non-Conformal Interface



This feature of relaxing the requirement for point-to-point matching at the interface between the meshes is particularly useful in parametric studies where you want to change an isolated region of the domain without changing the entire mesh. The procedure is as follows:

1. Read the meshes in meshing mode in Fluent. These meshes need not share nodes, edges, faces, or cells.

2. Create the volume mesh using an appropriate meshing strategy. You can also create domains for each independent mesh region and mesh the individual domains separately.

3. Separate the region of non-conformal interface into new face zones. Use **Boundary** → **Manage** to change the face zone type of the two surfaces that will be treated as non-conformal to **interface**.

4. Transfer the mesh to solution mode in Fluent and create the non-conformal interface using the **Mesh Interfaces** dialog box. Refer to Mesh Interfaces Dialog Box (p. 5525) for details.

7.8.1.7. Inserting Isolated Nodes into a Tet Mesh

To add nodes to the mesh without specifying the faces, you can create a boundary zone in the boundary mesh (in the program that created it) just for this purpose, then you can introduce the nodes associated with these faces.

This feature is useful for clustering nodes (and therefore cells) in a controlled manner. Figure 7.75: Mesh Generated Using Isolated Nodes to Concentrate Cells (p. 730) shows a mesh that was

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

generated using this method to cluster the nodes behind the wedge. A mesh that was generated for the same geometry without clustering is shown in Figure 7.76: Mesh Generated Without Using Isolated Nodes (p. 730). You can follow either of the two procedures to insert isolated nodes into a mesh:

- Delete the face zone before meshing.
- Introduce the nodes using face zones after meshing.

Deleting the Face Zone Before Meshing

This procedure is as follows:

1. Delete the face zone, but leave the associated nodes. Disable **Delete Nodes** in the **Manage Face Zones** dialog box to retain these nodes.

   Note:

   By default, the unused nodes are deleted when the faces of the zone are deleted.

2. Disable **Delete Unused Nodes** in the **Tet** dialog box and generate the volume mesh.

   Note:

   By default, unused nodes will be deleted during the automatic meshing.

   The nodes will be introduced when you initialize the mesh. When you use this procedure, *all* nodes must be inserted into the mesh or the initialization will fail.

   Warning:

   Do not place isolated nodes too close to the boundary or to other nodes.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.75: Mesh Generated Using Isolated Nodes to Concentrate Cells

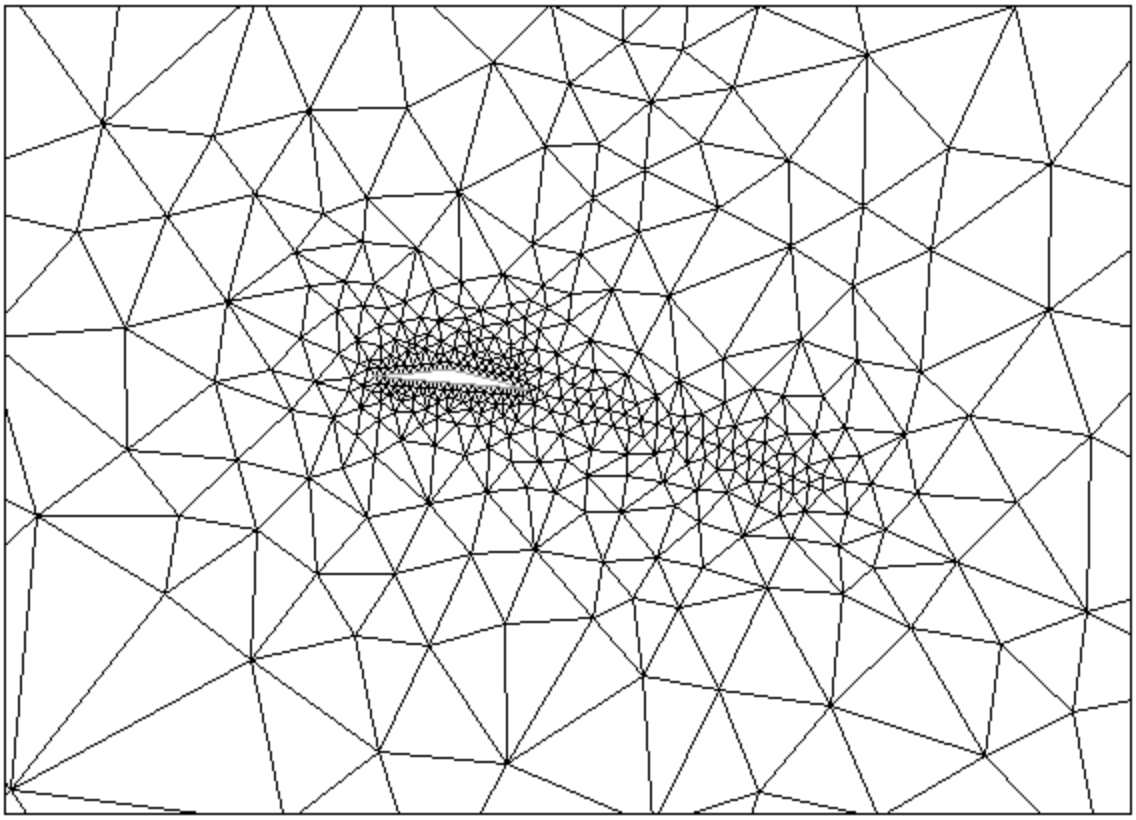
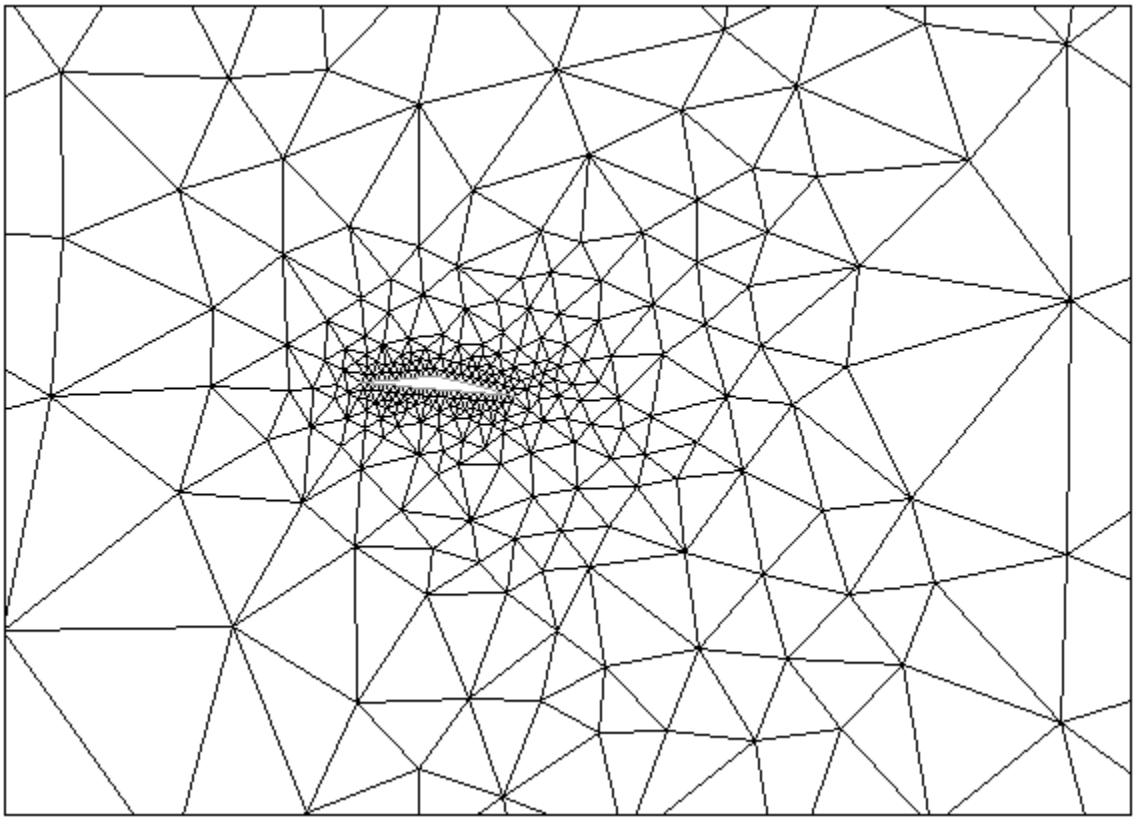


Figure 7.76: Mesh Generated Without Using Isolated Nodes



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Introducing the Nodes Using Face Zones After Meshing

This procedure is as follows:

1. Create a subdomain that does not include the face zones used to control the mesh density.
2. Create the volume mesh using the subdomain.
3. Activate the global domain.
4. Introduce the additional nodes using the text command: `/mesh/modify/mesh-nodes-on-zone`.

> **Important:**
>
> The `/mesh/modify/mesh-nodes-on-zone` command will delete the faces associated with the face zone.

7.8.2. Using the Auto Mesh Dialog Box

The **Auto Mesh** dialog box enables you to automatically create the volume mesh using the different mesh elements available. This dialog box can used for generating the volume mesh based on face zones or based on a mesh object and relevant material points. For more detail about object-based mesh generation, refer to Objects (p. 600).

The generic procedure for using the **Auto Mesh** dialog box for creating the volume mesh is as follows:

1. Determine the meshing approach (face zone based or object based) and the mesh elements required for the particular case.
2. Open the **Auto Mesh** dialog box using the **Mesh** → **Auto Mesh...** menu item or from the context-sensitive menu available by right-clicking on any **Mesh Object**, or its **Cell Zones** or **Volumetric Regions** branch in the Outline View.
3. Select the appropriate option in the **Object** drop-down list in the **Auto Identify Volume** group box.
   a. For the face zone based meshing approach, ensure that **none** is selected in the **Object** drop-down list.

   > **Note:**
   >
   > While performing face zone-based (non-object based) volume meshing, any zones overlapping the domain need to be resolved first.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. For the object based meshing approach, select the appropriate mesh object in the **Object** drop-down list. Enable/disable **Keep Solid Cell Zones** and **Keep Dead Cell Zones** as required.

> **Tip:**
>
> If you open the Auto Mesh dialog box from the context-sensitive menu in the Model tree, the Mesh Object is automatically selected.

4. Select the appropriate option in the **Grow Prisms** drop-down list in the **Boundary Layer Mesh** group box.

   a. Retain the default selection of **none** if you do not need to grow prism layers for the current meshing approach.

   b. Select **scoped** if you want to specify object-specific prism parameters. Click **Set...** to open the **Scoped Prisms** dialog box and define the prism parameters and scope. Refer to Prism Meshing Options for Scoped Prisms (p. 775) for details.

   c. Select **zone-specific** if you want to specify zone-specific prism parameters. Click **Set...** to open the **Prisms** dialog box and specify the zone-specific prism parameters. Refer to Procedure for Creating Zone-based Prisms (p. 753) for details.

   > **Note:**
   >
   > Growing prisms on periodic boundaries is not supported.

5. Select the appropriate quad-tet transition elements from the **Quad Tet Transition** list. Click **Set...** to open the **Pyramids** dialog box or the **Non Conformals** dialog box (depending on the selection) and specify the appropriate parameters. Refer to Creating Pyramids (p. 735) and Creating a Non-Conformal Interface (p. 739) for details.

6. Select the appropriate option from the **Volume Fill** list. Click **Set...** to open the **Tet**, **Hexcore**, **Poly**, or **Poly-Hexcore** dialog box (depending on the selection) and specify the appropriate parameters. Refer to Initializing the Tetrahedral Mesh (p. 788), Refining the Tetrahedral Mesh (p. 790), Controlling Hexcore Parameters (p. 798), Steps for Creating the Polyhedral Mesh (p. 805), and Generating Poly-Hexcore Meshes (p. 808) for details.

   > **Note:**
   >
   > The **No Fill** option is not available for object based volume meshing.

7. Specify the appropriate **Volume Fill Options**.

   a. For the **Tet** or **Poly** methods, set the following:

      - Select the appropriate option for **Cell Sizing**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - **Size Field** specifies that the cell size is determined based on the current size-field.
  - **Geometric** specifies that the cell size in the interior of the domain is obtained by a geometric growth from the closest boundary according to the growth rate specified.

    Set the **Growth Rate** required.

- Specify the **Max Cell Length**. Click **Compute** to compute the maximum cell size based on the mesh object.

b. For the **Hexcore** method, set the following:

- Select the appropriate option for **Type**.

  Retain the default selection of **Octree** or select the **Cartesian** type.

- Set the number of **Buffer Layers** and **Peel Layers**.
- Specify the **Max Cell Length** for the Cartesian approach. Click **Compute** to compute the maximum cell size based on the mesh object.

  Specify the **Min Cell Length** for the Octree approach.

c. For the **Poly-Hexcore** method, set the following:

- Specify the number of **Buffer Layers** and **Peel Layers**.
- Specify the **Min Cell Length** for the poly-hexcore approach.

The hybrid poly-hexcore volume meshing method is a combination of the existing poly and the octree hexcore meshing algorithms, and is available for serial processing only, with object based workflows when the **Grow Prisms** option is set to **none** or **scoped**.

While similar to the **Octree** mesh option, the **Poly-Hexcore** option leverages the use of hexcore volume meshing and uses a polyhedron transitional mesh rather than a tetrahedron transitional mesh.

Note that you can control the sizing (maximum cell length and/or growth rate) for specific regions using the text command: `mesh/tet/controls/set-region-based-sizing` (for tet or poly) and `mesh/hex/controls/set-region-based-sizing` (for hexcore or poly-hexcore).

8. For object-based meshing, enable **Merge Cell Zones within Regions** to create a single cell zone within a region, or disable to keep the cell zones separate.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For zone-based meshing, enable **Merge Cell Zones** to create a single cell zone within a domain, or disable to keep the cell zones separate.

Note:

For object-based meshing and parallel processing, after your mesh is distributed, this option is active automatically.

9. The **Auto Partition (Parallel Meshing)** option enables automatic decomposition of the mesh into suitable partitions for parallel meshing (see Auto Partitioning (p. 741) for more information). Disable this option if you are interested in only generating the volume mesh in serial mode.

10. Click **Mesh** to automatically create the mesh.

Alternatively, you can use the command `/mesh/auto-mesh` to generate the mesh automatically. Specify a mesh object name for object-based auto mesh; if no name is given, face zone based auto mesh is performed. Specify the volume fill type to be used when prompted. Specify whether to merge the cells into a single zone or keep the cell zones separate. For face zone based meshing, specify whether to automatically identify the domain to be meshed based on the topology information.

Note:

You can specify the meshing parameters for the mesh elements (prisms, pyramids or non-conformals, tet, or hex) using either the respective dialog boxes or the associated text commands prior to using the `auto-mesh` command.

7.8.3. Generating a Thin Volume Mesh

Thin Volume Mesh creates sweep-like mesh for a body occupying a thin gap. You define boundary face zones for source and target such that the source face normal should point to the target. The source face mesh may be triangles or quads. Four other controls are then used by Fluent Meshing to grow the volume mesh from the source zone to the target zone.

To generate a Thin Volume Mesh:

1. Check the source boundary mesh to ensure that free nodes or faces with high skewness do not exist. See Manipulating Boundary Nodes (p. 653) and Determining Surface Mesh Quality (p. 563) for details.

2. Open the **Thin Volume Mesh** dialog box.

   Mesh → **Thin Volume Mesh**

3. Select the **Source Boundary Face Zone** from the list.

4. Select the **Target Boundary Face Zone** from the list.

5. Specify the **Gap Thickness**.

   - If set to 0, the mesher will automatically calculate the gap thickness.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

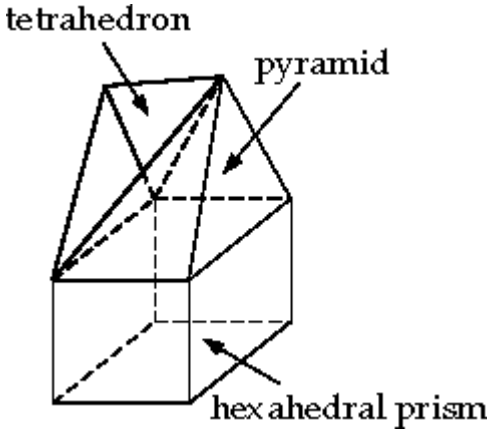
- If non-zero, the Gap Thickness defines the maximum separation between source and target zones in the swept-mesh region.

6. Specify the **Number of Divisions** between source and target faces.
7. Specify the **Growth Rate** between adjacent layers.
8. Choose **Remesh overlap zones** behavior. Remesh overlap zones replaces any overlapped part of the surface mesh on the target and adjacent faces. Original meshes in these areas are replaced.
9. Click **Create** to calculate the thin solid mesh.
10. Click **Draw** to display the mesh in the graphics window of the user interface.

7.8.4. Generating Pyramids

A pyramid has a quadrilateral face as its base and four triangular faces extending from the sides of the quadrilateral up to a single point above the base. See Figure 7.77: Pyramid Cell—Transition from a Hexahedron to a Tetrahedron (p. 735).

Figure 7.77: Pyramid Cell—Transition from a Hexahedron to a Tetrahedron



To generate a conformal mesh with a region of tetrahedral cells adjacent to a region of hexahedral cells, you will first create a layer of pyramids as a transition from quadrilateral faces to triangular faces. After creating a single layer of pyramids, the resulting triangular faces will be used to create tetrahedra.

To create pyramids, you need to specify the boundary from which the pyramids will be built, the method for determining the top vertex of each pyramid, and the pyramid height.

7.8.4.1. Creating Pyramids
7.8.4.2. Zones Created During Pyramid Generation
7.8.4.3. Pyramid Meshing Problems

7.8.4.1. Creating Pyramids

The procedure for creating a layer of pyramids from a quadrilateral boundary zone is as follows:

1. Check the aspect ratio limits of the boundary face zones on which you need to build pyramids.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Report → Size Field

The use of a high-aspect-ratio quadrilateral for the base of a pyramid produces skewed triangular faces that can cause problems during the tetrahedral mesh generation. If the maximum quadrilateral face aspect ratio is much greater than 10, you will need to regenerate them.

- If the faces were created in a different preprocessor, return to that application and try to reduce the aspect ratio of the faces in question.
- If the faces were created during the building of prism layers, rebuild the prisms using a more gradual growth rate.

2. Select the appropriate quadrilateral boundary zones in the **Boundary Zones** selection list in the **Pyramids** dialog box.

   Mesh → Pyramids

   Click **Draw** to view the selected zones.

   If the quadrilateral zone you require does not appear in the list, use the `/boundary/reset-element-type` text command to update the type of the zone. It is possible that the quadrilateral zone may not be recognized the due to changes made to the boundary mesh.

   For example, if you separate a mixed (tri and quad) face zone into a tri face zone and a quad face zone, each of these will be identified as a mixed zone. You need to reset the element type for the quad zone for it to be recognized and included in the **Boundary Zones** selection list.

3. Select the appropriate method for determining the pyramid vertex location in the **Options** list. The **skewness** method is selected by default, and is appropriate for most cases.

4. Specify the height of the pyramids by setting the **Offset Scaling** value.

5. Click **Create**. The new pyramid cell zone and the new face zones created will be reported in the console. You can then use the **Display Grid** dialog box to view these new zones.

   Display → Grid

6. Change the boundary type of the quadrilateral base zone to the appropriate type (if necessary).

   Boundary → Manage

Note:

Because you built cells next to the quadrilateral base zone, its original boundary type may no longer be correct.

Important:

The pyramids should be automatically created on the appropriate side of the specified boundary zones. If the pyramids are on the wrong side, do the following:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Delete the newly created zones related to the pyramids.
2. Reverse the normal direction on the quadrilateral boundary where the pyramids are being built (using the **Flip Normals** option in the **Manage Face Zones** dialog box).
3. Recreate the pyramids.

7.8.4.2. Zones Created During Pyramid Generation

When pyramids are generated, at least two new zones are created: a cell zone containing the pyramid cells and a face zone containing the triangular faces of the pyramids.

The following zones will be created during pyramid generation:

- The cell zone containing the pyramids (`pyramid-cells-n`).
- The face zone containing the triangular faces of the pyramid cells (`base-zone-pyramid-cap-n`).

  For example, if the pyramids were built from the quadrilateral face zone **wall-4**, they will be placed in a new zone called **wall-4-pyramid-cap-9** (where the 9 is the zone number assigned).
- The face zones containing the pyramid sides that use existing faces from the original boundary mesh (`base-zone-pyramid-side:n`), where, `n` is the zone number assigned.

  For example, if triangular boundary faces from the zone **wall-3** are used, they will be placed in a new zone called **wall-3-pyramid-side-6** (where the 6 is the zone number assigned).

Important:

If you include the `pyramid-side` boundary zones when defining the domain in which you are going to generate a tetrahedral mesh, the tetrahedral meshing will fail.

7.8.4.3. Pyramid Meshing Problems

Most problems associated with creating pyramid layers manifest themselves in the subsequent process of generating the tetrahedral mesh.

Rapid Changes in Volume

Rapid changes in the sizes of cells have a negative influence on the convergence and accuracy of the numerical solution. The pyramid layer creation can produce rapid variations in cell volume in the following situations:

- If the quadrilateral surface mesh has faces with rapid changes in size.
- If there is great disparity between the sizes of the quadrilateral faces and the neighboring triangular faces used in the pyramid creation.

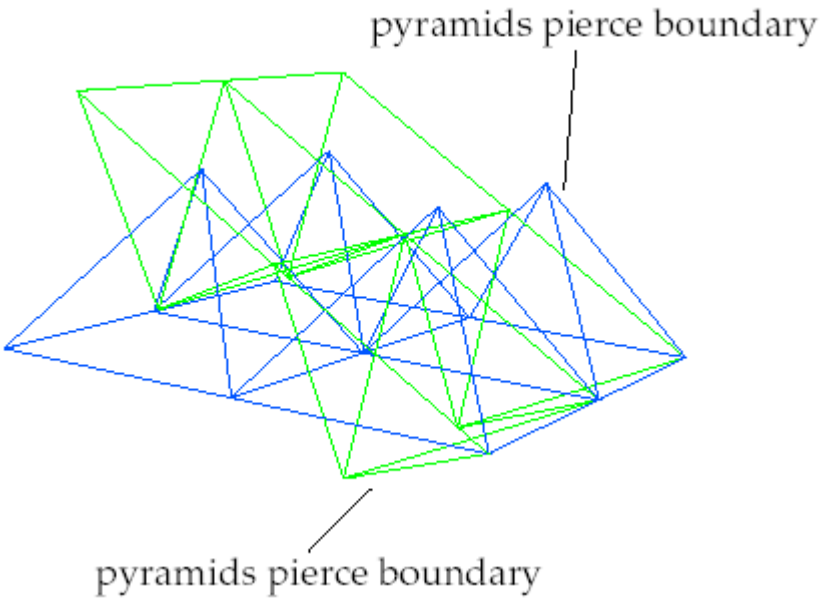
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can avoid the rapid variation in volume by creating quadrilateral and neighboring triangular grids with smooth variations in face size.

Intersecting Faces

If the quadrilateral surface used to create pyramids has highly concave corners, the resulting pyramids may pierce each other and/or neighboring boundary faces:

Figure 7.78: Pyramid Cells Intersecting Each Other and Boundary

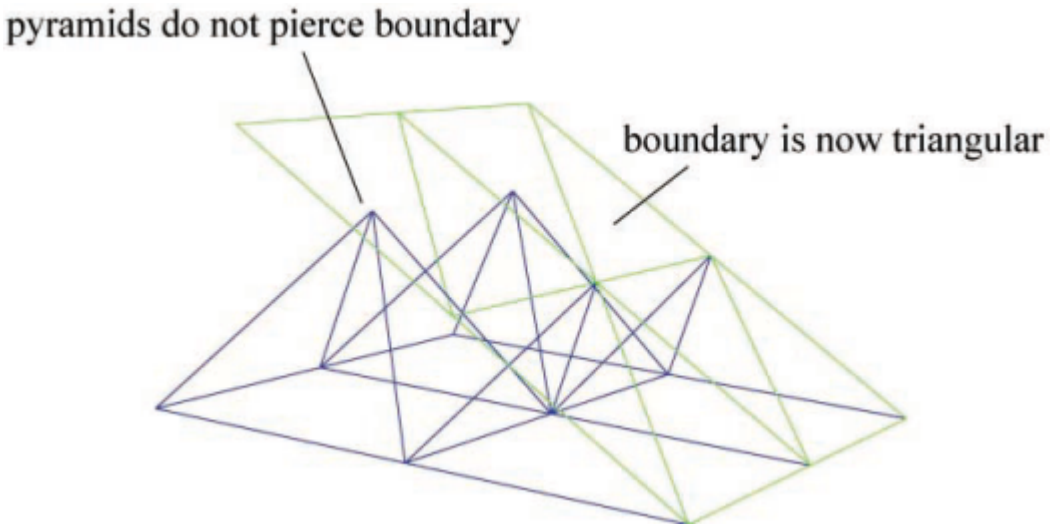


In such cases, you can either increase the resolution to prevent the intersections or alter the meshing strategy. An alternative is to separate the quadrilaterals in the concave corner into another zone (using techniques described in Separating Boundary Zones (p. 683) or Modifying the Boundary Mesh (p. 660)), create triangular faces from the quadrilateral faces using the `/boundary/remesh/triangulate` text command, and then create pyramids.

The skewness-based pyramid creation will use the existing triangular faces and avoid the intersection problem (see Figure 7.79: Fixed Intersecting Pyramid Cells Using Triangular Faces (p. 739)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.79: Fixed Intersecting Pyramid Cells Using Triangular Faces



High Aspect Ratio

Creating pyramids on quadrilateral faces with very high aspect ratios results in highly skewed triangular faces. Subsequent attempts to create a tetrahedral mesh from these elements will produce a poor-quality mesh.

Irrespective of the method used to generate the quadrilaterals, modify the meshing strategy to reduce the aspect ratio using an external grid generation package or the prism layer capability.

7.8.5. Creating a Non-Conformal Interface

For meshes containing both hexahedral and tetrahedral elements, you can generate a non-conformal interface to avoid creating intermediate pyramids as transition elements between the quadrilateral and triangular surfaces. You can also choose to create a non-conformal interface when growing prisms from a boundary on a surface mesh, to avoid quad faces in the domain to be meshed. The surfaces containing quad elements will be copied and then triangulated while keeping the original surfaces intact. The free nodes of the triangulated surface will then be merged with the nodes on the original surface mesh. Both surfaces will then be converted to interface type.

The options in the **Non Conformals** dialog box enable you to create a non-conformal interface.

7.8.5.1. Separating the Non-Conformal Interface Between Cell Zones

7.8.5.1. Separating the Non-Conformal Interface Between Cell Zones

The command `/mesh/non-conformals/separate` enables you to separate the face zones at the non-conformal interface between the cell zones specified. Specify the cell zones where the interface is not conformal, an appropriate gap distance (absolute or relative), and the critical angle to be used for separating the face zones. The gap distance used for the separation is the larger value of the absolute gap distance specified and the relative gap distance times the average local edge length. You can also choose to orient the boundary face zones after separation and additionally write a journal file during the separation operation. This journal file can then be read in solution mode to create the mesh interfaces automatically.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The separated face zones will be named as follows: **cell zone 1:cell zone 2-orig face zone 1** for face zone 1 attached to cell zone 1 and **cell zone 2:cell zone 1-orig face zone 2** for face zone 2 attached to cell zone 2. If the separated interface zone has more than one face zone, the original face zone contributing a larger number of faces will be used for the name.

Invalid contact locations will be skipped during the separation operation. The location of the invalid contact will be reported in the console.

7.8.6. Creating a Heat Exchanger Zone

Many engineering systems, including power plants, climate control, and engine cooling systems typically contain heat exchangers. However, for most engineering problems, it is impractical to model individual fins and tubes of the heat exchanger core.

You can create a heat exchanger volume mesh using the options in the **Heat Exchanger Mesh** dialog box.

The heat exchanger mesh created contains prisms generated from a quad split surface mesh. You need to specify four points (either by selecting the locations or nodes) and the required intervals between the first selected point and the each of the remaining points to create the heat exchanger mesh. A meshed plane is created using the first three specified points and the corresponding intervals. Prisms are created on the meshed plane using the fourth point and the corresponding interval.

Mesh → Create → Heat Exchanger...

1. In the **Location** group box, select **Position** or **Nodes** to specify the source for location coordinates.
2. Click **Select Points...** or **Select Nodes...** and select the four points (nodes) in the correct order.

   Important:

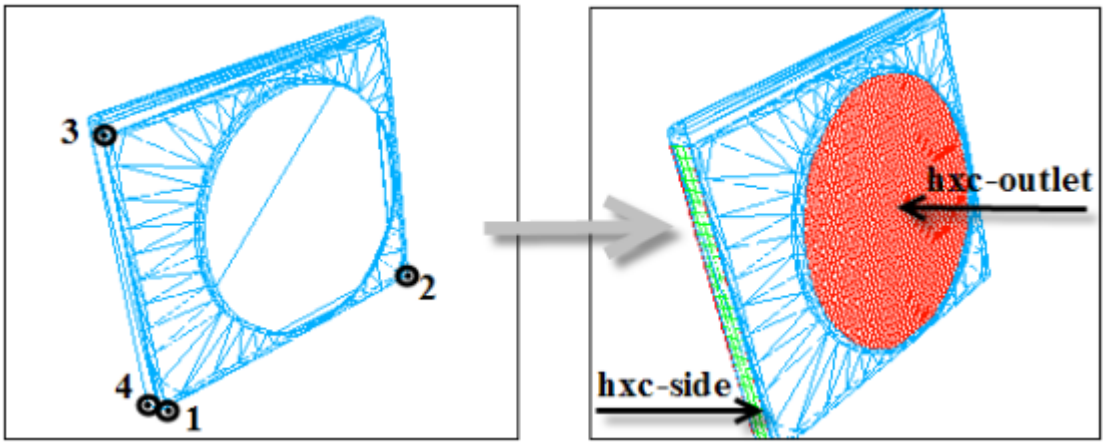
   The order of selection of the points is important because the heat exchanger zone is created based on the intervals specified between the first selected point and each of the remaining points. If the points are not specified in the correct order, you will get a heat exchanger zone that is different from the required one.

3. In the **Create By** group box, select **Interval** or **Size** to setup the mesh density.
4. Specify the number of intervals (or mesh size) between selected points (nodes) 1–2, 1–3, and 1–4, respectively.
5. Enable **Create Object**, if you need to create a mesh object for the heat exchanger mesh zones created.
6. Click **Preview** to preview the heat exchanger zones and modify the parameters if you are dissatisfied with the results.
7. Click **Create**.
8. Specify the prefix for the zones as required in the **Object/Zone Prefix** dialog box and click **OK**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The heat exchanger zones (prefixed by **hxc-**) are created as shown in Figure 7.80: Creating the Heat Exchanger Mesh (p. 741).

Figure 7.80: Creating the Heat Exchanger Mesh



7.8.7. Parallel Meshing

If you launched Fluent with parallel processing for meshing enabled, you will be able to distribute the mesh data across the available compute nodes, or recombine the mesh data as required. For additional information on specifying parallel processing, see Starting Ansys Fluent Using Fluent Launcher (p. 180).

The following options are available for parallel meshing:

7.8.7.1. Auto Partitioning

7.8.7.2. Controlling the Threads

7.8.7.1. Auto Partitioning

Auto partitioning enables automatic decomposition of the mesh into suitable partitions for parallel meshing. Auto partitioning is enabled using the **Auto Partition (Parallel Meshing)** option in the **Auto Mesh** dialog box. This option is available for the **Poly**, **Poly-Hexcore**, and the **Octree**-based **Hexcore** volume fill methods. When using the **Tet** volume fill method with zone-based or scoped prism settings, Fluent generates the prisms in parallel by default and then generates the tetrahedral mesh in serial.

Note:

Auto partitioning is only available when you launch Fluent with more than one parallel processor.

Follow these steps to generate the mesh using auto-partitioning.

1. Prepare the surface mesh for volume meshing. Ensure that the mesh quality and connectivity are appropriate.

2. Enable **Auto Partition (Parallel Meshing)** in the **Auto Mesh** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When **Auto Partition (Parallel Meshing)** is enabled, and *after* volume meshing has occurred, several elements of the graphical user interface are no longer available (Availability of Graphical User Interface Options After Parallel Meshing (p. 742)). Likewise, there are elements of the text user interface that are also no longer available (Availability of Text Interface Options After Parallel Meshing (p. 744)).

3. Create the volume mesh using the options in the **Auto Mesh** dialog box (see Using the Auto Mesh Dialog Box (p. 731)).

   Note:

   - Available volume meshing options include zone-specific or scoped prisms, pyramid and non-conformal quad-tet transition, and tetrahedral fill.

4. Improve the volume mesh using the options available, if required.

5. Save the mesh or case file.

7.8.7.1.1. Availability of Graphical User Interface Options After Parallel Meshing

After auto partitioning is enabled, and *after* parallel meshing has occurred, the following items in the graphical user interface are not available as noted:

Location	Unavailable Items
Display (Ribbon)	**Free Face**
	Multi Faces
Patch Options (Ribbon)	**Remesh**
	Separate
Select File Dialog (**File** > **Read Mesh**)	**Append File(s)**
Boundary Menu	**Merge Nodes**
	Intersect
	Create
Modify Boundary Dialog **(Operations)**	**Create**
	Merge
	Delete
	Collapse
	Split
	Swap
Separate Face Zones Dialog **(Options)**	**Prisms**
	Thin Volume Mesh
	Seed
Mesh Menu	**Prisms**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Location	Unavailable Items
	Thin Volume Mesh
	Pyramids
	Non-Conformals
	Tet
	Hexcore
	CutCell
	Rapid Octree
	Create
Mesh > Tools	**Tet Improve** (not available for the Poly- Hexcore volume fill method, otherwise, available for the Tet volume fill method using prisms)
	Cavity Remesh
Mesh > Tools > Prism	**Improve** (not available for the Poly- Hexcore volume fill method, otherwise, available for the Tet volume fill method using prisms)
	Split
	Post Ignore
	Tet Improve Cavity
Modify Cells Dialog **(Options)**	**Split**
	Swap
Outline View: **Model** context menu options	**Construction Geometry**
	Periodicity
	CAD Assemblies
Outline View: **Model > Geometry Objects** context menu options	**Diagnostics**
	Convert to Mesh Object
	Wrap
	Remesh
	Advanced
	CAD Association
Outline View: **Model > Mesh Objects** context menu options	**Diagnostics**
	Wrap
	Remesh Faces
	Join/Intersect
	Recover Periodic
	Restore Faces
	Advanced
	CAD Association
Outline View:	**Join/Intersect**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Location	Unavailable Items
Model > Mesh Objects > Face Zone Labels sub-level context menu options	**Diagnostics**
	Recover Periodic
Outline View: **Model > Mesh Objects > Volumetric Regions** context menu:	**Compute**
	Validate
	Update
	External Baffles
	Setup Scoped Prisms
Outline View: **Model > Mesh Objects > Volumetric Regions** sub-level context menu options	**Diagnostics**
	Manage
	Remesh Faces
	Scoped Prisms
	Tet
	Hexcore
	Auto Fill Volume
	CAD Association
Graphics Toolbars: **Construct Geometry**	
Graphics Toolbars: **Loop Selection Mode**	
Graphics Toolbars: Miscellaneous Portions of the **Context** Toolbar	

7.8.7.1.2. Availability of Text Interface Options After Parallel Meshing

After auto partitioning is enabled, and *after* parallel meshing has occurred, the following items in the text user interface are not available as noted:

TUI Menu	Command	Status
`objects/`	`create`	available
	`create-and-activate-domain`	not available
	`create-groups`	not available
	`create-intersection-loops`	not available
	`create-multiple`	available
	`create-new-mesh-object/`	not available

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	check-mesh	available
	delete	available
	delete-all	available
	delete-all-geom	available
	delete-unreferenced-faces-and-edges	available
	extract-edges	available
	improve-feature-capture	not available
	improve-object-quality	not available
	join-intersect/	not available
	labels/	not available
	list	available
	merge	not available
	merge-edges	not available
	merge-voids	not available
	merge-walls	available
	remove-gaps/	not available
	rename-cell-zone-boundaries-using-labels	available
	rename-object	available
	rename-object-zones	available
	rotate	available
	scale	available
	separate-faces-by-angle	not available
	separate-faces-by-seed	not available
	set/	not available
	sew/	not available
	summary	available
	translate	available
	update	available
	volumetric-regions/	not available
	wrap/	not available
mesh/		
	auto-mesh-multiple-objects	not available
	auto-prefix-cell-zones	not available
	cavity/	not available
	create-heat-exchanger	not available
	cutcell/	not available
	domains/	available

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	laplace-smooth-nodes	not available
	manage/active-list	available
	manage/adjacent-face-zone	available
	manage/auto-set-active	available
	manage/change-prefix	available
	manage/change-suffix	available
	manage/copy	available
	manage/delete	available
	manage/get-material-point	available
	manage/id	available
	manage/list	available
	manage/merge	available
	manage/merge-dead-zones	available
	manage/name	available
	manage/origin	available
	manage/revolve-face-zone	available
	manage/rotate	available
	manage/rotate-model	available
	manage/scale	available
	manage/scale-model	available
	manage/set-active	available
	manage/translate	available
	manage/translate-model	available
	manage/type	available
	modify/extract-unused-nodes	not available
	modify/list-skewed-cells	not available
	modify/repair-negative-volume-cells	not available
	non-conformals/	not available; available when **Auto Partition** is disabled
	pyramid/	not available
	prepare-for-solve	available
	prism/	available
	prism/create	not available
	prism/mark-ignore-faces	not available
	prism/mark-nonmanifold-nodes	not available
	prism/controls/	not available
	poly/	not available
	repair-face-handedness	not available

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	selective-mesh-check	not available
	scoped-prisms/	not available
	tet/	not available
	thin-volume-mesh/	not available
boundary/		
	auto-slit-faces	not available
	check-boundary-mesh	not available
	check-duplicate-geom	not available
	clear-marked-nodes	not available
	compute-bounding-box	not available
	count-free-nodes	not available
	count-marked-faces	not available
	count-unused-bound-node	not available
	count-unused-faces	not available
	count-unused-nodes	not available
	create-revolved-surface	not available
	create-swept-surface	not available
	delete-all-dup-faces	not available
	delete-duplicate-faces	not available
	delete-free-edge-faces	not available
	delete-island-faces	not available
	delete-unconnected-faces	not available
	delete-unused-nodes	available
	feature/	not available
	fix-mconnected-edges	not available
	improve/	not available
	jiggle-boundary-nodes	not available
	make-periodic	not available
	mark-duplicate-nodes	not available
	mark-face-intersection	not available
	mark-face-proximity	not available
	mark-faces-in-region	not available
	manage/auto-delete-nodes?	available
	manage/change-prefix	available
	manage/change-suffix	available
	manage/copy	available
	manage/create	available
	manage/delete	available

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	manage/flip	available
	manage/id	available
	manage/list	available
	manage/merge	available
	manage/name	available
	manage/origin	available
	manage/orient	available
	manage/remove-suffix	available
	manage/rotate	available
	manage/rotate-model	available
	manage/scale	available
	manage/scale-model	available
	manage/translate	available
	manage/translate-model	available
	manage/type	available
	manage/user-defined-groups/	available
	mark-duplicate-nodes	not available
	mark-face-intersection	not available
	mark-face-proximity	not available
	mark-faces-in-region	not available
	merge-nodes	not available
	merge-small-face-zones	not available
	merge-nodes	not available
	merge-small-face-zones	not available
	orient-faces-by-point	not available
	project-face-zone	not available
	recover-periodic-surfaces	not available
	refine/	not available
	remesh/	not available
	resolve-face-intersection	not available
	scale-nodes	not available
	separate/	not available
	set-periodicity	not available
	slit-boundary-face	not available
	smooth-marked-faces	not available
	unmark-selected-faces	not available
	wrapper/	not available

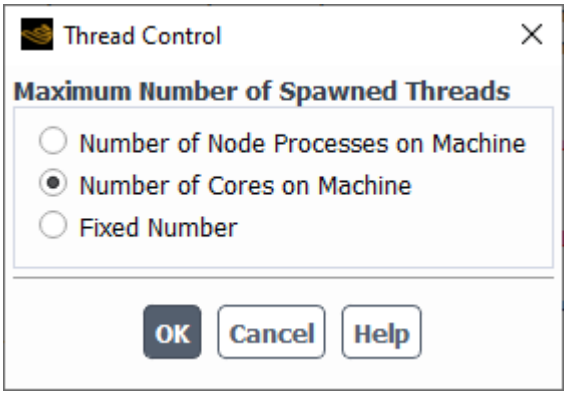
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.8.7.2. Controlling the Threads

You can control the maximum number of threads on each machine by using the **Thread Control** dialog box (Figure 7.81: The Thread Control Dialog Box (p. 749)).

Parallel → Thread Control...

Figure 7.81: The Thread Control Dialog Box



You have the following options when using the **Thread Control** dialog box:

- **Number of Node Processes on Machine**

  When this option is chosen, the maximum number of threads on each machine is equivalent to the number of Ansys Fluent node processes on each machine.

  When the mesh data is distributed across multiple nodes, then each node will spawn a single node process. When the mesh data is *not* distributed (i.e., resides only on a single node), however, then the node will spawn to the total number of node processes. Consider using this option to potentially increase the speed of checking the mesh and its quality when the mesh is not distributed.

- **Number of Cores on Machine**

  This is the default option. When this option is chosen, the maximum number of threads on each machine is equivalent to the number of cores on the machine. Ansys Fluent obtains the number of cores from the operating system. This may be applicable when the multi-threaded part of the calculation is dominating the computation time, and you want to take full advantage of computational resources.

  When the mesh data is distributed across multiple nodes, each node will spawn to the ratio of the maximum number of cores and the number of compute nodes. When the mesh data is *not* distributed (i.e., the mesh data resides only on a single node, e.g. **node-0**), then **node-0** will spawn to the maximum allowed number of open multi-processing threads.

- **Fixed Number**

  When this option is chosen, you may specify the maximum number of threads that can be spawned on each machine in the number-entry box below **Fixed Number**. This may only be applicable when you want to have fine control of the number of threads on each machine; it is not recommended in general.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

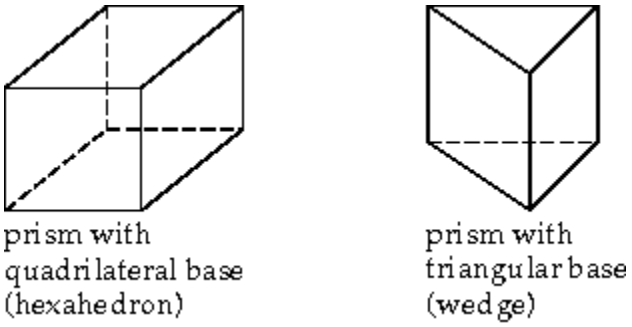
When the mesh data is distributed across multiple nodes, each node will spawn to the quotient of the **Fixed Number** and the number of compute nodes. When the mesh data is *not* distributed (i.e., resides only on a single node), however, then each node will spawn to the **Fixed Number**.

7.9. Generating Prisms

You can create prism cells starting from either quadrilateral or triangular boundary faces, or both. Prisms can be used to resolve a boundary layer region of a tetrahedral mesh. They can be used to extend some portion of a domain for which the volume mesh already exists (for example, increase the length of an inlet pipe), or to create a volume mesh by extrusion.

Figure 7.82: Prism Shapes (p. 750) shows the types of prisms that can be generated.

Figure 7.82: Prism Shapes



To create prisms, you need to specify the surfaces from which the prisms will be built and the parameters that control the prism growth. Details about how the parameters are used to generate the prisms are provided in The Prism Generation Process (p. 751).

For zone-based prism growth, the prisms are grown from selected boundary zones. Growth parameters such as offset and growth methods, prism height, the number of layers, and the direction may be applied only to specific zones. Additional growth options including options for prism proximity detection and splitting can also be specified. Instructions for building zone-based prisms are provided in Procedure for Creating Zone-based Prisms (p. 753). The additional options are described in Prism Meshing Options for Zone-Specific Prisms (p. 756).

For object-based meshing, use the scoped prism process to select regions for prism creation and set up controls for the prism generation. Each control specifies the offset method, prism height, number of layers and growth rate (scoped prism uses the geometric growth method). The scope for prism creation can be set to **fluid-regions**, **solid-regions**, or specifically **named-regions**. Within the selected regions, you can grow prism layers on **only-walls** (default), **all-zones**, or the **solid-fluid-interface**. Alternatively, you can grow prism layers on selected zones (**selected-face-zones**) or face zone labels (**selected-labels**). The scoped prism controls can be read from a previously saved file or can be written to a file for further use. These options are described in Prism Meshing Options for Scoped Prisms (p. 775).

Problems related to prism generation are discussed in Prism Meshing Problems (p. 778).

Note:

Generating prisms on periodic boundaries is not supported.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.9.1. The Prism Generation Process

The following process is used by the mesher to build the prisms and to create a high-quality mesh.

1. Boundary mesh analysis:

   The initial surface mesh is analyzed to determine if any adjacent zones should be projected to adjacent zones and retriangulated (if triangular), or if their faces should be used as the sides of new prism cells (if quadrilateral).

   For details on determining whether to use existing boundary zones or not, see Using Adjacent Zones as the Sides of Prisms (p. 770).

2. Prism layer growth.

   Prism layers are grown one by one, based on the layer height specification. This is a multistep procedure, and hence some of the steps may be modified or skipped:

   a. Determine the initial direction vectors so that the direction in which to build the prisms is determined. The direction methods available are discussed in Direction Vectors (p. 768).

   b. Check for proximity and determine shrinkage of prism layers based on the proximity options specified.

   c. Determine the initial offset distances so that the mesher will know how far from the corresponding nodes on the previous layer to place the new nodes that define the prisms. See Offset Distances (p. 765) for details.

   d. Smooth the initial direction vectors. See Normal Smoothing (p. 769) for details.

   e. Adjust and smooth the offset distances to eliminate spikes and dips in the new prism layer. See Offset Smoothing (p. 767) for details

   f. Project new nodes along the outer edges of the prism layer to adjacent zones, based on the analysis in step 1.

   g. Create prism faces and cells using the new nodes. The new faces at the top of the prism layer are referred to as cap faces.

   h. The new cap faces of skewness value greater than the specified threshold are improved by edge swapping or edge node smoothing.

   i. Apply global smoothing across the new surface. This is also skewness-driven, but is based on the nodes not the faces. See for details.

   j. Check the quality of all new cells and faces.

   k. If quality problems are detected, prism layer creation is stopped. If more layers are to be grown, the process is repeated starting from substep (a).

3. Zone clean up: After all the prism layers have been grown, the new faces along the outer sides of the layers are moved to zones of the same types as the zones to which their nodes are projected. Unprojected faces are collected in a single `prism-side` zone.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Retriangulation: Adjacent triangular zones that have been projected to adjacent zones are automatically retriangulated. Triangles that are overlapped by new quadrilateral faces from the prism layers are removed. Most of the original nodes are used in the new triangulation. The result is a conformal interface between the sides of the prism layers and the adjacent triangular zones.

7.9.1.1. Zones Created During Prism Generation

When prisms are generated, several new cell and face zones are created. You can identify the new zones by their default names.

- The cell zone containing the prisms will be named `prism-cells-#`.
- The face zone created at the end of the last prism layer will be named `prism-cap-#`.
- If the prism layers are bounded by one or more existing triangular face zones, each of these will be retriangulated so that the faces of the zone that are adjacent to the prisms will become quadrilateral faces, while the rest of the faces in the zone remain triangles.

  If an original triangular face zone is called `symmetry`, the portion of it that still contains triangles will be retained as `symmetry`, while the portion containing quadrilateral faces will be named `symmetry-quad:#`.
- Any faces that were not projected to adjacent zones (see Using Adjacent Zones as the Sides of Prisms (p. 770)) are collected into a zone named `prism-side-n`.
- For a zone `wall-#`, the zone created by ignoring prism growth in the region of proximity will be named as `wall-#:ignore`, where, # is the zone number assigned.

Important:

All the ignored threads related to a base thread will be merged into a single thread by default. You can however change this using the command `/mesh/prism/controls/merge-ignored-threads?` which will generate more than one thread per base thread.

To merge one or more prism cell zones with other cell zones (and merge the corresponding face zones), use the **Merge** option in the **Manage Cell Zones** dialog box.

By default, the `prism-cap-#` zones will be wall zones. If you do not want to include these walls in the model, change them to interior zones (or any other type) using the **Manage Face Zones** dialog box.

Note:

If the prism mesh is generated using the **Auto Mesh** tool and **Merge Cell Zones** is enabled, the new face zones will be merged with the original boundary zones and a single cell zone will be created.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.9.2. Procedure for Creating Zone-based Prisms

The procedure for creating prisms by setting up zone-specific parameters is as follows:

1. Check the boundary mesh to ensure that free nodes or faces with high skewness do not exist. See Manipulating Boundary Nodes (p. 653) and Determining Surface Mesh Quality (p. 563) for details.

2. Select the boundary zones from which you want to grow prisms in the **Boundary Zones** selection list in the **Prisms** dialog box.

   Mesh → Prisms...

   Alternatively, when using **Auto Mesh**, select **zone-specific** in the **Grow Prisms** drop-down list and click **Set...** to open the **Prisms** dialog box.

   Important:

   If you read in a mesh file created in a previous version, it is recommended that you reset all the prism parameters using the command `/mesh/prism/reset-parameters` before proceeding with setting the prism parameters.

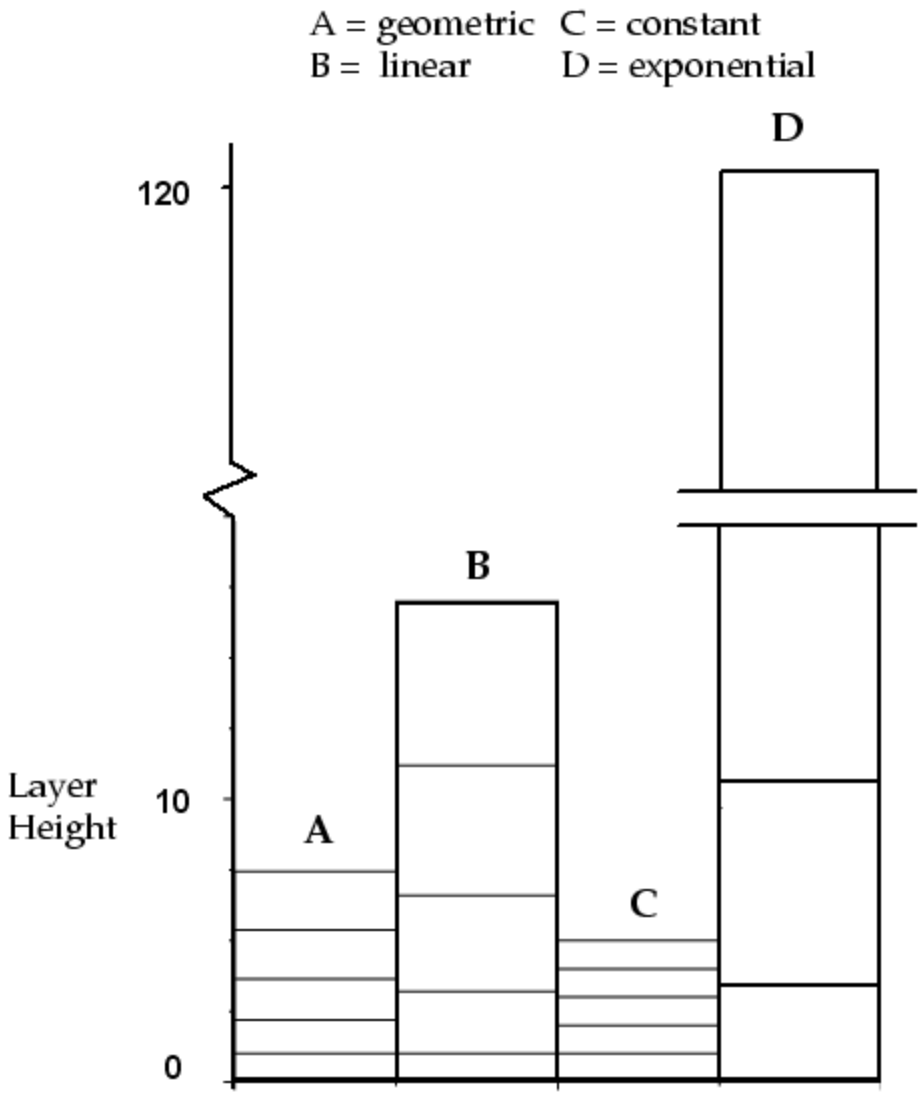
3. Specify prism growth parameters in the **Growth** tab.

   a. Select the appropriate **Offset Method** and **Growth Method** for growing prisms.

      Figure 7.83: Layer Heights Computed Using the Four Growth Methods (p. 754) shows layer heights for all four methods, using a first height of 1 and a slope/rate/exponent of 1.2.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.83: Layer Heights Computed Using the Four Growth Methods



See Offset Distances (p. 765) for details.

b. Define the height of the first prism layer (**First Height**/**First Aspect Ratio**).

c. Set the growth method related parameter (**Slope**, **Rate**, or **Exponent**, as applicable) and the **Number of Layers**.

d. Specify options for prism growth, proximity detection, and splitting of prism layers (if needed) in the **Prisms Growth Options** dialog box (opened using the **Growth Options...** button), if required. Refer to Prism Meshing Options for Zone-Specific Prisms (p. 756) for details.

e. Click the **Plot** button to preview the height distribution.

For information on setting different growth parameters for different zones, see Growing Prisms Simultaneously from Multiple Zones (p. 757).

4. Specify the direction for prism growth (see Direction Vectors (p. 768)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. Click the **Direction** tab in the **Prisms** dialog box to view the direction parameters.

b. Verify the orientation of the normals for the zones on which you want to grow prism layers. Ensure that the normals are pointing in the direction that you want to build the prisms.

   You can use the **Color by Normal** option, accessible using the hotkey **Ctrl**+**Shift**+**C** then **Ctrl**+**N** to verify the normal direction.

   - If the boundary zone normals are incorrectly oriented, click the **Orient Normals...** button to open the **Orient Normals** dialog box. Specify the material point based on which the normals are to be oriented and click **Apply**.
   - For mesh objects, use the options in the **Orient Mesh Object Face Normals** group box to orient the normals.

     a. Select the mesh object in the **Object Name** drop-down list.

     b. Select **Region** or **Material Point** as appropriate, and then select the region or material point in the drop-down list.

     c. Ensure that **Select** is enabled, and enable **Select Walls** and/or **Select Baffles** as needed.

     d. Click **Orient**.

        Face boundary zone groups comprising the prism base zones will be created (prefixed by **_prisms**).

c. Specify the method for determining the direction of the prisms:

   - Select **Normal** in the **Method** list to compute normal direction vectors. This method takes into account the change in direction required for a curved region. Specify the values for the maximum angle change.
   - Select **Uniform** in the **Method** list to use a constant, uniform direction for flat prism regions. Specify the appropriate direction vector.

5. Set the parameters for prism improvement and prism projection available in the **Improve** and **Project** tabs, respectively, as required. See Using Adjacent Zones as the Sides of Prisms (p. 770) for more information.

6. Click **Apply** to save your settings.

7. After applying your prism settings you can use the **Diagnostics Tools** dialog box to identify exposed quadrilateral faces (**Exposed Quads**) prior to generating prisms. as outlined in Using the Diagnostics Tools Dialog Box (p. 557).

8. Save the mesh:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

File → Write → Mesh

Important:

It is a good practice to save the mesh at this point. If you are dissatisfied with the prisms generated, you can read in this mesh and modify the prism parameters to re-generate the prisms.

9. Click **Create** to generate only the prism cell zone.

   Refer to Prism Meshing Problems (p. 778) for suggestions on solving problems occurring during prism generation.

   Alternatively, when using **Auto Mesh**, click **Apply** and close the **Prisms** dialog box. The prism cells will be created first, and then the mesher will proceed to generating the tet/hexcore mesh. Refer to Using the Auto Mesh Dialog Box (p. 731) for details.

10. Check the maximum face skewness value reported in the console when the final prism layer is created, to ensure that the value is acceptable.

    This is especially important if you are going to generate tetrahedral or pyramidal cells using this new boundary zone.

    The new zone should not have highly skewed triangles or quadrilaterals. For more information about the quality of quadrilateral faces used for creating pyramids, refer to Creating Pyramids (p. 735).

    If the reported maximum skewness is too high, read the mesh file you saved after setting the prism parameters and try again with different parameters. Refer to Prism Meshing Problems (p. 778) for details.

11. A number of new zones are created when the prism generation is complete (see Zones Created During Prism Generation (p. 752)). If the prism-cap or prism-side face zones are *not* supposed to be walls (for example, if they are supposed to be simple interior zones), change them to the appropriate boundary type.

    Boundary → Manage

For most cases, performing these steps will result in an acceptable mesh of prismatic cells, but for some complex geometries, you may have to modify the procedure. If you are not satisfied with the prisms generated, you can read in the mesh you saved before generating prisms and modify the prism parameters. Refer to subsequent sections for details on modifying the prism generation parameters to create a better mesh.

The procedure for using zone-specific prism controls for creating prisms during volume meshing using **Auto Mesh** is described in Using the Auto Mesh Dialog Box (p. 731).

7.9.3. Prism Meshing Options for Zone-Specific Prisms

This section describes the prism meshing options available for creating a prism mesh based on zone-specific settings.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.9.3.1. Growth Options for Zone-Specific Prisms
7.9.3.2. Offset Distances
7.9.3.3. Direction Vectors
7.9.3.4. Using Adjacent Zones as the Sides of Prisms
7.9.3.5. Post Prism Mesh Quality Improvement

7.9.3.1. Growth Options for Zone-Specific Prisms

This section describes the additional growth options available for creating a prism mesh based on zone-specific settings.

7.9.3.1.1. Growing Prisms Simultaneously from Multiple Zones
7.9.3.1.2. Growing Prisms on a Two-Sided Wall
7.9.3.1.3. Ignoring Invalid Normals
7.9.3.1.4. Detecting Proximity and Collision
7.9.3.1.5. Splitting Prism Layers
7.9.3.1.6. Preserving Orthogonality

7.9.3.1.1. Growing Prisms Simultaneously from Multiple Zones

You can select multiple face zones for simultaneously growing prism layers. If the face zones are connected, the prisms grown will also be connected. A single zone of prism cells and a single zone of cap faces will be created for each set of simultaneously grown layers.

You can grow prisms from multiple zones with the same or different growth parameters.

- To use the same growth parameters for all zones, follow the steps in Procedure for Creating Zone-based Prisms (p. 753). In this case, the same number of layers are grown from all zones, and all other parameters (growth method, offset method, direction method, and so on) also remain the same.

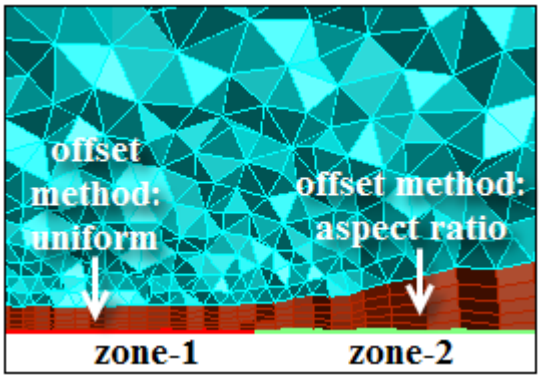
- To use different growth parameters for different zones, you can apply different growth methods, offset methods, first height, and other growth parameters (as required for the growth method selected) individually for each zone and click the **Apply** button in the **Zone Specific Growth** group box in the **Prisms** dialog box.

  If you specify different growth parameters for different zones, all other parameters (direction method, offset method, and so on) can be set separately for each zone. In this case, the offset height of each node that is shared by multiple zones will be the average of the heights applied on the separate zones.

  This produces a continuous transition between the zones (Figure 7.84: Different Growth Parameters on Adjacent Zones (p. 758)). Offset smoothing (see Offset Smoothing (p. 767)) is recommended in these cases to avoid sharp height changes at such edges.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.84: Different Growth Parameters on Adjacent Zones



To assign different growth controls and different offset types to different face zones, replace steps 2 and 3 in Procedure for Creating Zone-based Prisms (p. 753) with the following steps:

1. Select the zones for which you want to specify a set of growth parameters in the **Boundary Zones** list in the **Prisms** dialog box.

2. Set the appropriate **Offset Method**, **Growth Method**, **First Height**, and the related growth parameters (as required) in the **Growth** tab.

3. Specify parameters for proximity/collision detection in the **Prisms Growth Options** dialog box (see Detecting Proximity and Collision (p. 761) for details).

   Important:

   By default, a single zone of prism cells and cap faces is generated for the simultaneously grown prism layers.

4. Click **Apply** in the **Prisms** dialog box.

5. Repeat this procedure for other zones for which you want to apply different growth controls.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Click **Create**.

Warning:

Before you click the **Create** button, ensure that *all* the zones for which you want to grow prisms are selected in the **Boundary Zones** list.

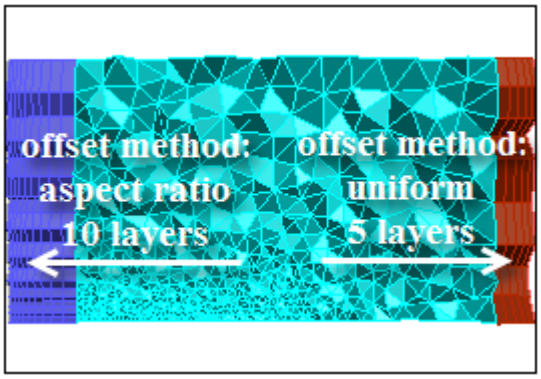
Note:

When the **Prisms** dialog box is used to generate prism layers on multiple zones (adjacent or nonadjacent), the number of layers is determined by the chronologically last selected zone in the **Boundary Zones** selection list.

Important:

- If you specify different number of layers to be grown on adjacent zones, the same number of layers (generally the smaller number of layers) will be grown on both the zones (see Figure 7.84: Different Growth Parameters on Adjacent Zones (p. 758)).
- If you specify different number of layers on multiple (nonadjacent) zones and use the **Prisms** dialog box to generate the prism layers, the same number of layers will be grown on the respective zones. The number of layers is determined by the chronologically last selected zone in the **Boundary Zones** selection list.
- If you specify different number of layers on multiple (nonadjacent) zones and use the **Auto Mesh** dialog box to generate the prism layers, the prism layers will be grown separately from each zone:

Figure 7.85: Different Growth Parameters on Nonadjacent Zones—Using the Auto Mesh Option

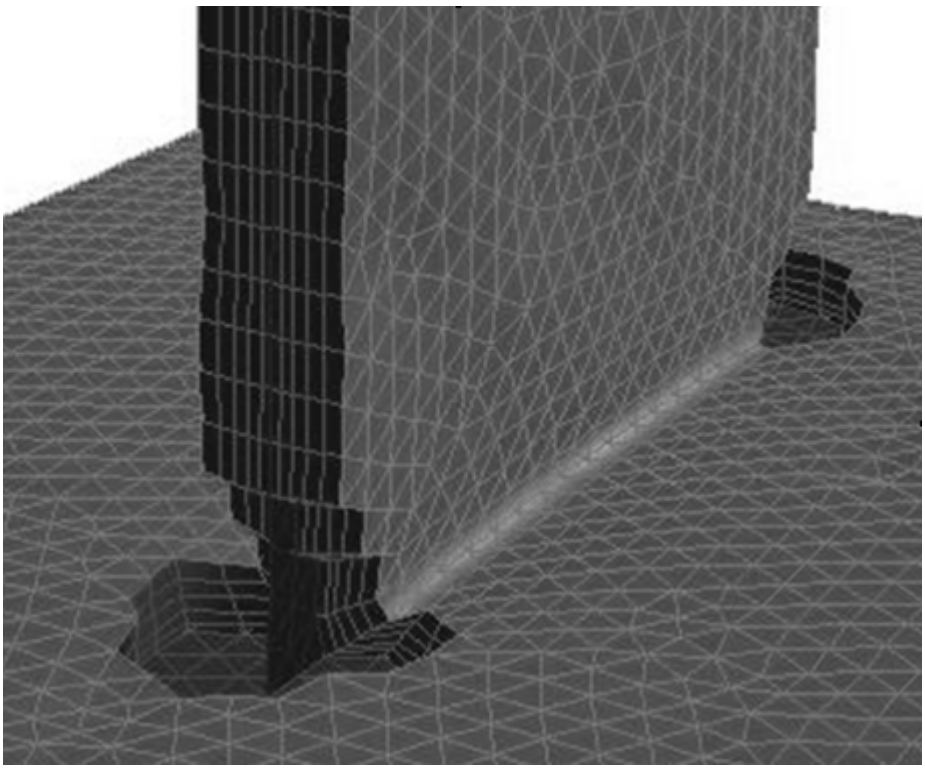


7.9.3.1.2. Growing Prisms on a Two-Sided Wall

Two-sided walls may be present in some models. To grow prisms on both sides of a two-sided wall (for example, growing prisms on dangling walls), do the following:

1. Select the two-sided wall on which you want to grow prisms from the **Boundary Zones** selection list.
2. Specify the prism growth parameters as required.
3. Enable **Grow on Two Sided Wall** in the **Growth** tab of the **Prisms** dialog box.
4. Click **Create**. Figure 7.86: Prism Growth on a Dangling Wall (p. 760) shows an example of prisms grown on a dangling wall.

Figure 7.86: Prism Growth on a Dangling Wall



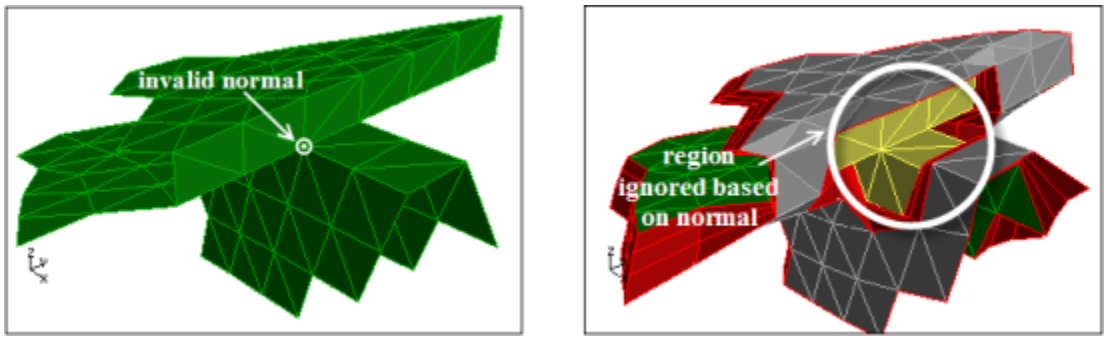
7.9.3.1.3. Ignoring Invalid Normals

Some models may contain regions where the normals considered while growing prisms may be invalid. The normal at a particular node may be nearly tangential to the surrounding faces in the model and as a result, the prism generation may fail. For such cases, you can choose to ignore regions of the model where invalid normals exist, during prism generation. Prism layers will be grown from the remaining regions according to the specified parameters.

Figure 7.87: Ignoring Invalid Normals (p. 761) shows a portion of the model where the normal at the highlighted node will be nearly tangential to some of the surrounding faces. You can see that the region around this node has been ignored while creating prisms.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.87: Ignoring Invalid Normals



You can ignore regions based on normals while creating prisms as follows:

1. Specify the prism growth parameters.
2. Select the **Boundary Zone** containing the invalid normal and click the **Growth Options...** button to open the **Prisms Growth Options** dialog box.
   a. Enable **Ignore Invalid Normals** in the **General Options** group box.
   b. Click **Apply** and close the **Prism Growth Options** dialog box.
3. Grow prisms as required.

7.9.3.1.4. Detecting Proximity and Collision

If the zones on which you want to grow prisms are very close to each other, the prism layers from zones may intersect or collide with each other. This results in bad quality of prism layers.

For example, in Figure 7.88: Collision of Prism Layers (p. 762), the prism layers grown from proximal surfaces intersect each other. The **Allow Shrinkage** option allows you to avoid the intersection of prism layers by adjusting the height of the prism layers in closely placed zones. Figure 7.89: Prism Layers Shrunk to Avoid Collision (p. 762) shows the use of this option to avoid intersection of the prism layers by adjusting the prism layer height.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.88: Collision of Prism Layers

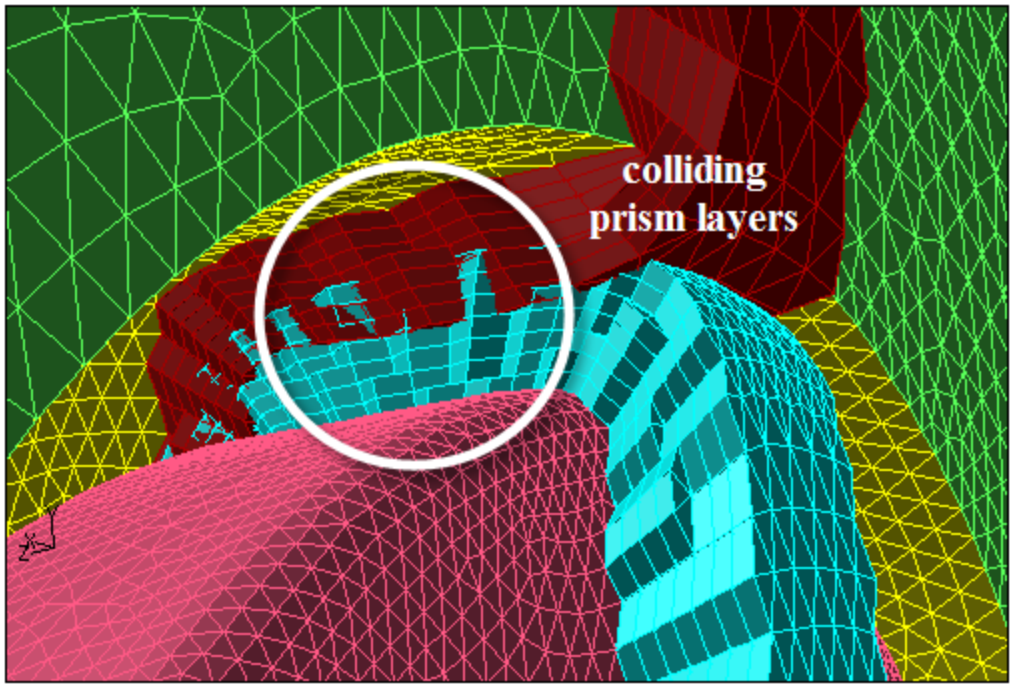
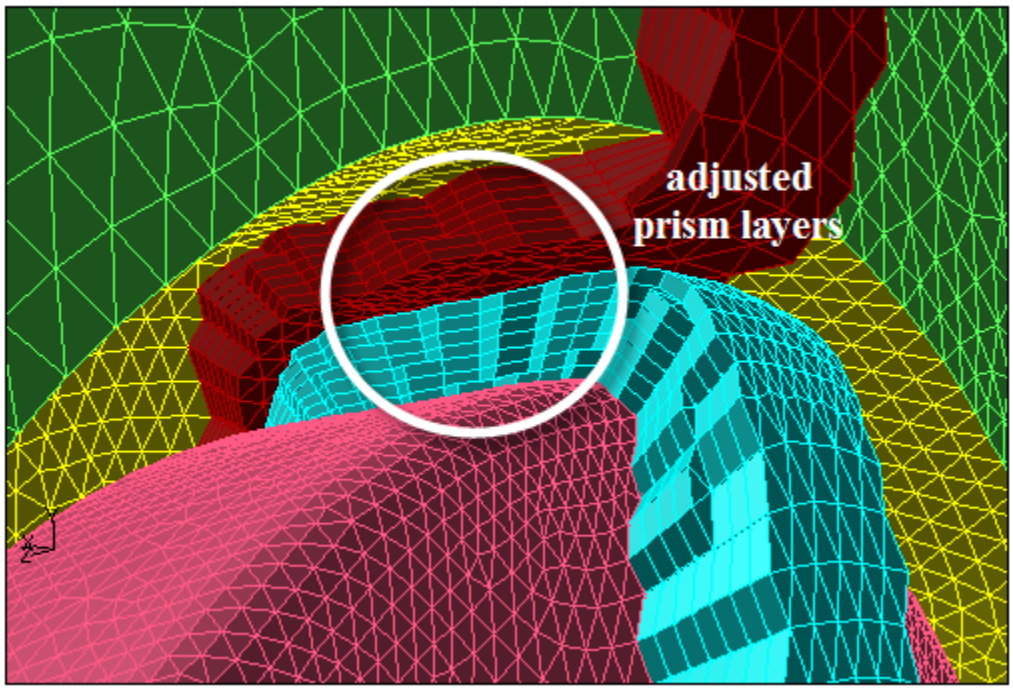


Figure 7.89: Prism Layers Shrunk to Avoid Collision



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

A value of n for **Gap Factor** implies that a gap equal to n times the maximum base edge length at the node in question will be maintained. Hence, a value of 1 implies that the gap maintained is equal to the maximum base edge length at the node considered.

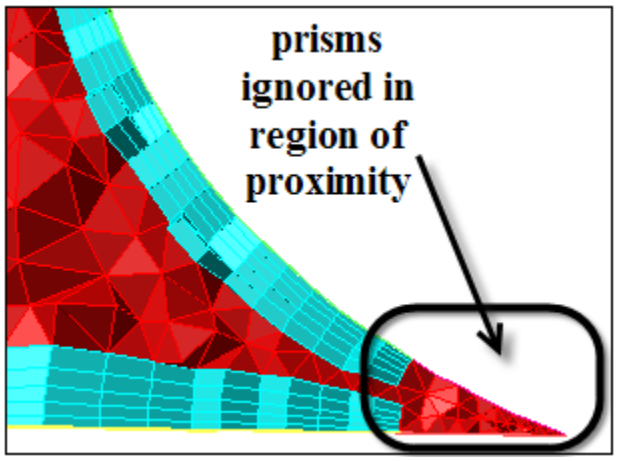
Important:

Offset heights are scaled only in the regions where the intersection of prism layers takes place. For the remaining regions, the prisms are created according to the specified parameters.

You can also ignore prisms in regions of close proximity during prism generation. The area of proximity will be determined based on the shrink factor specified. Prism layers will be grown from the remaining regions according to the specified parameters.

Figure 7.90: Ignoring Areas of Proximity (p. 763) shows a portion of the model where the region of proximity in a sharp corner has been ignored while creating prisms.

Figure 7.90: Ignoring Areas of Proximity



For automatic adjustment of intersecting/colliding prism layers while growing prisms, do the following:

1. Specify the prism growth parameters.
2. Click **Growth Options...** in the **Growth** tab to open the **Prisms Growth Options** dialog box.
3. Enable the appropriate options in the **General Options** group box.
4. Specify the parameters required for the proximity calculation and controlling the prism layer height.
    a. Enable **Allow Shrinkage** to allow shrinkage of prism layers in areas of proximity. This option is enabled by default.
    b. Enable **Keep First Layer Offsets**, if required.

       This option allows you to keep the original first offset height and scale the offset heights for the remaining layers in case of intersecting prism layers.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. Specify an appropriate value for **Gap Factor**.

d. Enable **Allow Ignore** to ignore the prism growth parameters in areas of proximity.

e. Enter an appropriate value for **Max Aspect Ratio** or **Max Shrink Factor** as required.

f. Click **Apply** and close the **Prism Growth Options** dialog box.

5. Click **Create** in the **Prisms** dialog box.

7.9.3.1.5. Splitting Prism Layers

You can generate fewer prism layers and then split them as part of the prism generation process to generate the total number of prism layers required. This option is faster than generating the same total number of prism layers.

You can specify the number of divisions per prism layer as follows:

1. Specify the prism growth parameters.
2. Click **Growth Options...** to open the **Prisms Growth Options** dialog box.
3. Enable **Split** in the **Split Options** group box.
4. Specify the **Divisions Per Layer**.
5. Click **Apply** and close the **Prism Growth Options** dialog box.

Note:

To create split prism layers, there should be a unique and valid prism cap face zone (with the name `*prism-cap*`) associated with the prism cell zone.

7.9.3.1.6. Preserving Orthogonality

In some cases, it may be important for the prismatic mesh to be orthogonal near the original boundary. For such cases, there are two ways to preserve orthogonality:

- Decrease the value specified for the **Max. Angle Change** in the **Direction** tab of the **Prisms** dialog box to a lower value to limit the change in normal direction in each prism layer.

  This parameter is set to 60 degrees by default. This means the normal direction at a node can change up to 60 degrees during normal, edge, and node smoothing. For example, if you reduce this parameter to 10 degrees, the change in normal direction will be more gradual, and the mesh near the original boundary will be nearly orthogonal.

- Specify an explicit number for **Orthogonal Layers** in the **Direction** tab of the **Prisms** dialog box.

  When you specify orthogonal layers, you should decrease the **Max Angle Change** to approximately 10 degrees. This will prevent sudden sharp changes in normal direction at the first non-orthogonal layer.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For a non-orthogonal layer, edge swapping, normal, edge, or node smoothing will be performed so that the layer is orthogonal to the original boundary. For example, if you specify 5 orthogonal layers, no smoothing will be performed on the first 5 layers, resulting in direction vectors that are normal to the original surface mesh faces. Full smoothing is used on the sixth and subsequent layers.

Important:

If you are preserving orthogonality near the original boundary, make sure that the prism layers do not grow too quickly. Use small layer heights, relative to the sizes of the faces on the original boundary. If prism layers grow quickly, the nearly-orthogonal direction vectors are likely to cross at sharp concave corners, causing the prism generation to fail. The prism layer generation will be stopped at this point. See Normal Smoothing (p. 769).

Preserving orthogonality can affect skewness and lead to left-handed faces and/or negative volumes. See Negative Volumes/Left-Handed Faces/High Skewness (p. 779) for details.

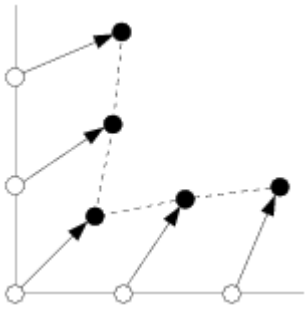
7.9.3.2. Offset Distances

The offset distance for a given node is the distance between adjacent layers at that node. This distance is based on the prism layer height computed from the specified prism growth parameters. The new node for a prism layer is placed at this distance along the direction vector.

There are four methods available for determining the offset distances:

- **Uniform Offset Distance Method**: In this method, every new node (child) is initially the same distance away from its parent node (that is, the corresponding node on the previous layer, from which the direction vector is pointing).

  Figure 7.91: Uniform Offset Distance Method

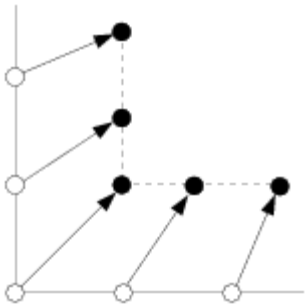
  Uniform distances are fine for planar surface meshes, but they result in crevices at sharp corners, for example, the 90 degree corner in Figure 7.91: Uniform Offset Distance Method (p. 765). The direction vector at the corner node will be at an angle of 45 degrees, while the vectors at the adjacent nodes will be slightly more orthogonal to their faces. For uniform distances, this places the nodes for the first layer as shown in Figure 7.91: Uniform Offset Distance Method (p. 765).

  The dashed line connecting the new nodes begins to pinch inward at the corner. The distances between parent and child nodes are still all the same, but the angles of the direction vectors introduce this pinching effect. Without some sort of correction such crevices can eventually collapse.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Minimum Height Offset Distance Method**: In this method, child nodes are guaranteed to be *at least* as far from their parent nodes as the distance computed for the current layer from the growth inputs. The mesher will also try to retain the shape of the original boundary.

  Figure 7.92: Minimum-Height Offset Distance Method

  For a perfect 90 degree corner, this results in a perfect 90 degree corner for the next layer (see Figure 7.92: Minimum-Height Offset Distance Method (p. 766)). Thus, in this method the shape is better preserved if no additional smoothing is applied.

  The minimum height method have some limitations. If the angle of the direction vector at the corner is not 45 degrees (that is, if it does not exactly bisect the angle between the faces), the minimum-height method can increase skewness.

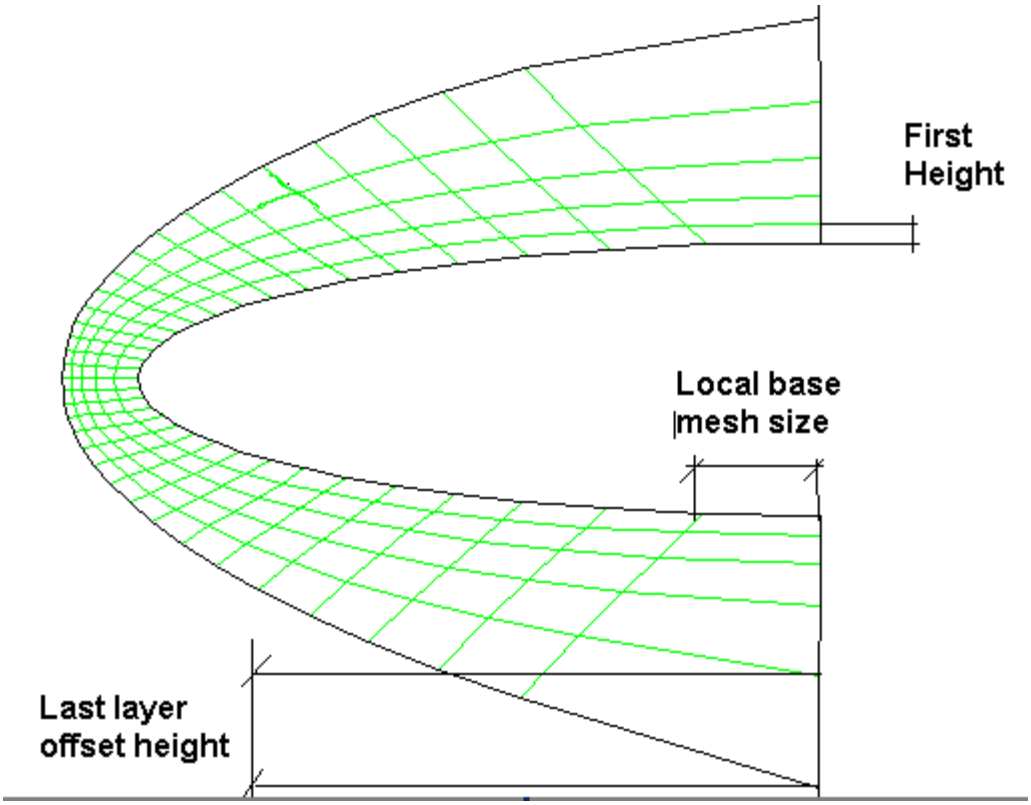
- **Aspect Ratio Method**: This method allows you to control the aspect ratio of the prism cells that are extruded from the base boundary zone. The *aspect ratio* is defined as the ratio of the prism base length to the prism layer height. The various growth methods (constant, linear, geometric, and exponential) can be used to specify the aspect ratio of the first layer.

  When a non-constant growth method is used, the aspect ratio for subsequent layers will change accordingly. The heights of the prism cells will vary according to the local face sizes in the surface mesh, providing a convenient way to automatically vary prism heights across a zone.

- **Last Ratio Method**: This method also allows you to control the aspect ratio of the prism cells that are extruded from the base boundary zone. You can specify **First Height** for the first prism layer. If you select this method, the various growth methods will not be accessible. The last aspect ratio method is explained in Figure 7.93: Last Ratio Method (p. 767).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.93: Last Ratio Method



Local base mesh size is used to find out the offset height for the last layer. For example, if you specify 80 as the **Last Percent** value the offset height of the last layer will be 0.8 times the local base mesh size. Local growth rate is used to calculate the other intermediate offset heights exponentially.

Note:

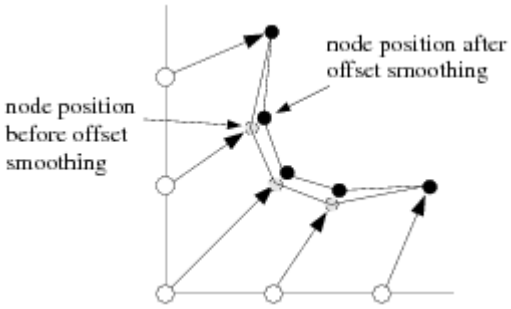
Once calculated, the actual last last layer offset height is not adjusted in cases of concave or convex base, or non-orthogonal layers.

Offset Smoothing

The purpose of offset smoothing is to eliminate spikes and dips in the new surface layer. Smoothing is applied iteratively. Figure 7.94: Effect of Offset Smoothing (p. 768) shows how the nodes are moved during offset smoothing.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.94: Effect of Offset Smoothing



7.9.3.3. Direction Vectors

As each layer of prisms is built, the mesher needs to know the direction in which to build. There are two methods available for determining the direction vectors:

- **Extrusion Method**: For creating straight-sided prism regions without any curvature, you can specify a uniform direction vector for all prism layers. Select **Uniform** in the **Method** list in the **Direction** tab of the **Prisms** dialog box.

  Figure 7.95: Uniform Direction Vector for a Straight-Sided Prism Region

  Specify the uniform direction vector or click **Compute** in the **Vector** group box. This constant vector will be used for all prism layers, instead of computing a new direction for each layer (see Figure 7.95: Uniform Direction Vector for a Straight-Sided Prism Region (p. 768)).

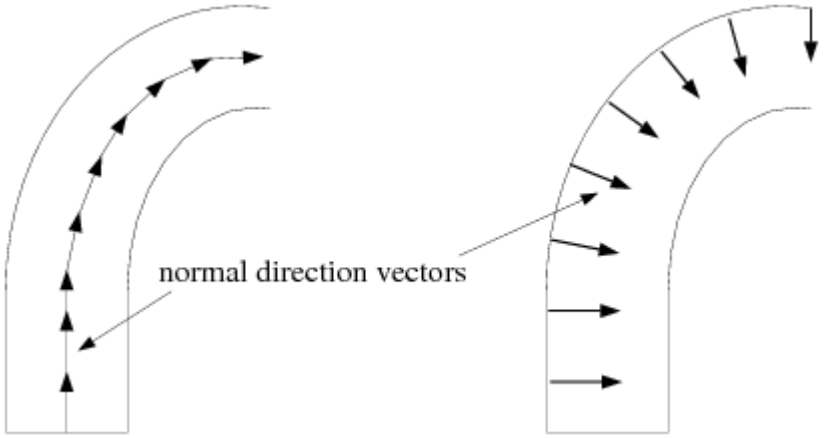
  Note:

  The **Grow On Two Sided Wall** option cannot be used when the **Uniform** method is selected.

- **Normal Method**: For regions with curvature, the appropriate normal direction vector at each node will be determined because it may be different for each node and each layer. See Figure 7.96: Normal Direction Vectors for a Curved Prism Region (p. 769). Select **Normal** in the **Method** group box in the **Direct** tab of the **Prisms** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.96: Normal Direction Vectors for a Curved Prism Region

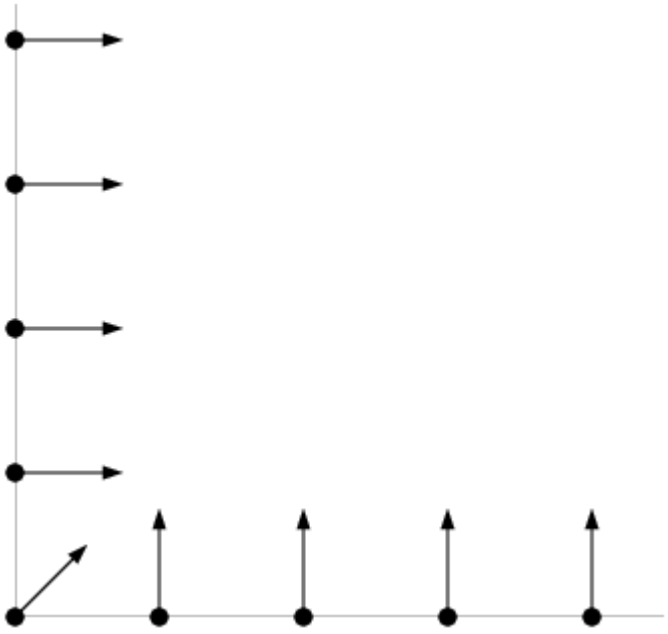


Normal Smoothing

The normal direction vectors obtained using the methods described in Direction Vectors (p. 768) are smoothed so that there is a gradual change in direction from one node to the next. This will reduce the chance of direction vector intersection, which causes the prism generation to fail. This step is not necessary when a uniform direction vector is used because there is no change in direction from one node to the next.

Figure 7.97: Normal Direction Vectors Before Smoothing (p. 769) shows normal direction vectors in the vicinity of a sharp (90 degree) corner.

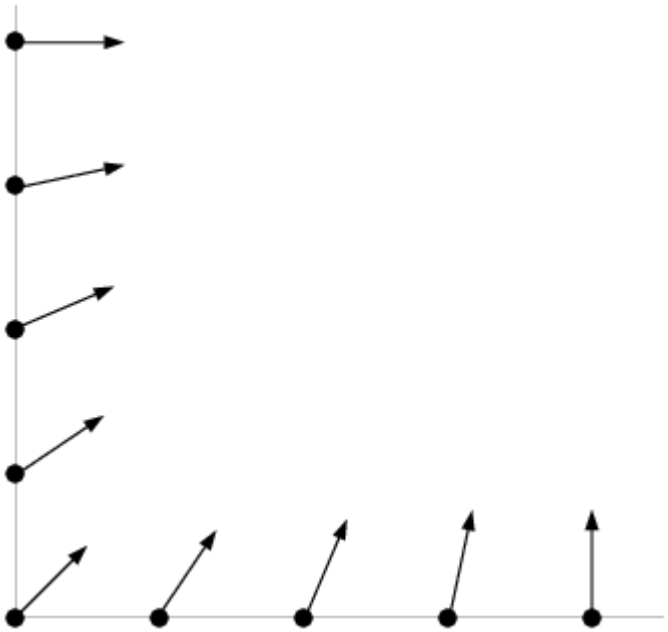
Figure 7.97: Normal Direction Vectors Before Smoothing



The direction vector is at an angle of 45 degree at the corner, while elsewhere it is 0 or 90 degrees. These direction vectors will intersect and prisms cannot be generated. Figure 7.98: Normal Direction Vectors After Smoothing (p. 770) shows the normal direction vectors near the corner after they have been smoothed. After smoothing, the normal direction changes more gradually from node to node.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.98: Normal Direction Vectors After Smoothing



It is possible to create very thin prism layers using the direction vectors shown in Figure 7.97: Normal Direction Vectors Before Smoothing (p. 769), as long as the last layer lies below the point where the vectors cross.

7.9.3.4. Using Adjacent Zones as the Sides of Prisms

For each zone adjacent to the zones from which you grow prisms, the angle between the prism growth direction and the adjacent zone will be checked. Depending on the size of this angle, the mesher will decide whether to use the adjacent zone for the prism sides, or create a new zone.

Using Adjacent Quadrilateral Face Zones

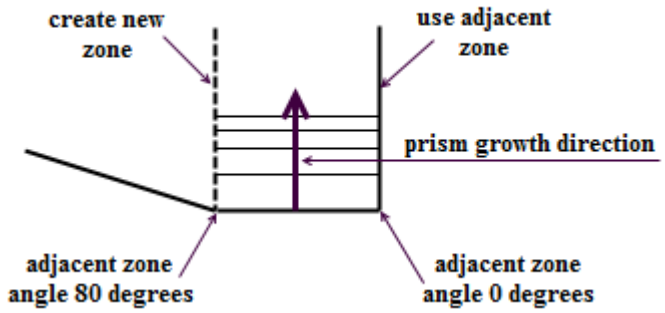
An adjacent zone with quadrilateral faces will be used without modification as the prism-side boundary if it satisfies the following requirements:

- It must share nodes with the boundary zone from which you are building the prisms that is, there must be no free nodes (see Free and Isolated Nodes (p. 653)) where the zones touch.
- The angle between the adjacent zone and the prism growth direction must be less than the specified threshold, **Max. Adjacent Zone Angle** (in the **Project** tab of the **Prisms** dialog box). The default threshold value is 75 degrees.

For example, if you are growing prisms from the bottom surface in Figure 7.99: Effect of Adjacent Zone Angle (p. 771) using the default maximum adjacent zone angle of 75 degrees, the zone on the left will be excluded, but the zone on the right will be used as it is. A new zone will be created for the left sides of the prisms.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.99: Effect of Adjacent Zone Angle



To fill in the gap on the left side of Figure 7.99: Effect of Adjacent Zone Angle (p. 771), create pyramids using the prism side faces, create a domain, and generate tetrahedral cells.

Projecting to Adjacent Triangular Face Zones

If an adjacent zone has triangular faces, two additional steps will be performed to incorporate the zone into the prism layers: projection and retriangulation. The mesher will project the outer nodes of the prisms onto the triangular faces of the adjacent zone, provided it satisfies the following requirements:

- It must share nodes with the boundary zone from which you are building the prisms. That is, there must be no free nodes (see Free and Isolated Nodes (p. 653)) where the zones touch.
- The angle between the adjacent zone and the prism growth direction must be less than the specified threshold, **Max. Adjacent Zone Angle** (in the **Project** tab of the **Prisms** dialog box). The default threshold value is 75 degrees.

In Figure 7.99: Effect of Adjacent Zone Angle (p. 771), the zone on the right will be projected to and retriangulated, while the zone on the left will be excluded.

Retriangulation

As shown in Figure 7.82: Prism Shapes (p. 750), the sides of the prism will always be quadrilateral faces, regardless of the type of face from which the prism is built (triangular or quadrilateral). If the prism-side nodes are projected to an adjacent triangular boundary face zone:

1. The prism-side faces on the shared boundary will be overlaid on the triangular faces of the existing boundary zone.
2. The triangular boundary zone will be retriangulated so that the triangular portion of the zone ends at the border of the portion now filled with quadrilateral faces. This is done to obtain a conformal mesh where all the nodes match up at face edges.
3. The triangles that were overlapping the quadrilateral prism-side faces will be deleted.

Figure 7.100: Symmetry Zone and Car Wall Before Prism Generation (p. 772) shows the initial boundary mesh for a car and the symmetry boundary beside it. If the prisms are generated from the car body *without* retriangulation of the triangular symmetry boundary, the resulting boundary mesh will be as shown in Figure 7.101: Symmetry Zone and Car Wall After Prism Generation Without

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Retriangulation (p. 772). With retriangulation, the boundary mesh will appear as in Figure 7.102: Symmetry Zone and Car Wall After Prism Generation and Retriangulation (p. 773).

Figure 7.100: Symmetry Zone and Car Wall Before Prism Generation

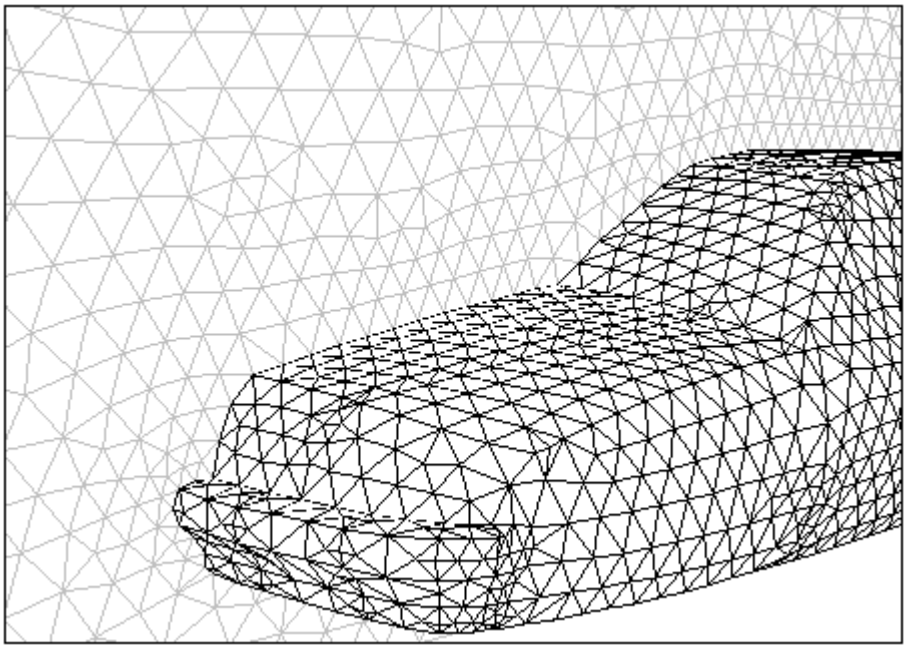
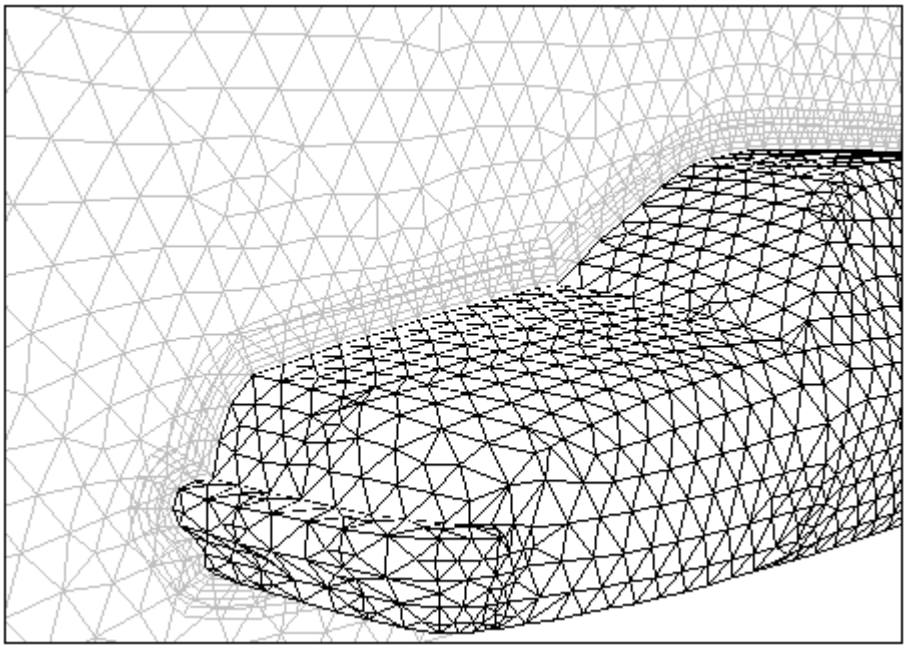
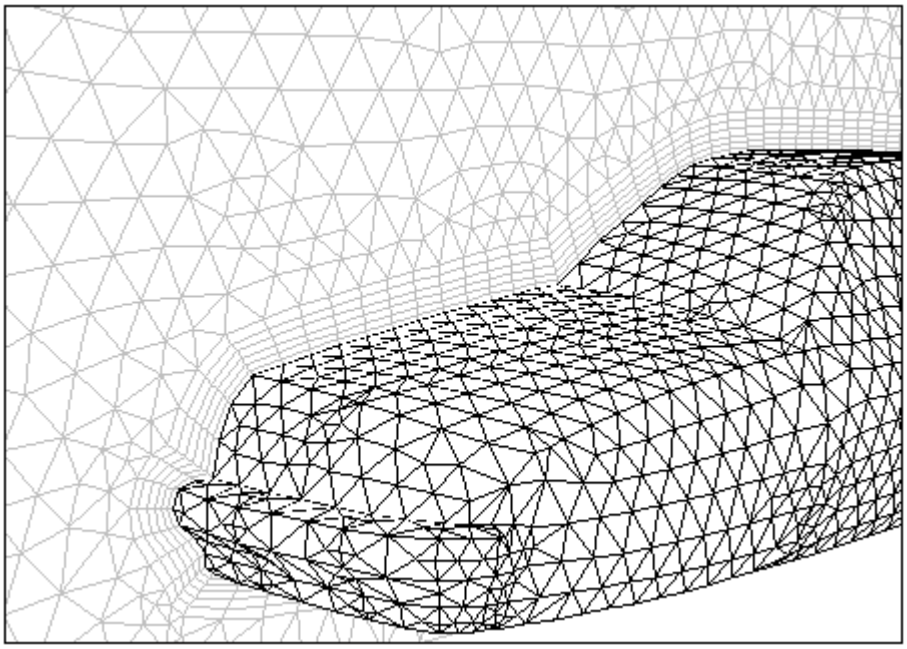


Figure 7.101: Symmetry Zone and Car Wall After Prism Generation Without Retriangulation



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.102: Symmetry Zone and Car Wall After Prism Generation and Retriangulation



7.9.3.5. Post Prism Mesh Quality Improvement

You can improve prism cell quality in a postprocessing step after all the required prism layers are created. The following options are available:

7.9.3.5.1. Improving the Prism Cell Quality

7.9.3.5.2. Removing Poor Quality Cells

7.9.3.5.3. Improving Warp

7.9.3.5.1. Improving the Prism Cell Quality

The **Prism Improve** dialog box contains options that allow you to improve the prism cell quality based on the quality measure selected. The quality measures available are ICEM CFD quality, orthoskew, skewness, or squish. The following options are available:

- The **Smooth** option allows optimization based smoothing of prism cells. Poor quality cells can be identified based on the quality measure selected. The nodes of cells with quality worse than the specified **Max Cell Quality** value will be moved to improve quality. The cell aspect ratio will also be maintained based on the value specified for `max-aspect-ratio`.

- The **Improve** option collects and smooths cells in layers around poor quality cells. Poor quality cells can be identified based on the quality measure selected. Cells with quality worse than the specified **Max Cell Quality** value will be identified, and the nodes of the cells surrounding the poor quality cells will be moved to improve quality. The cell aspect ratio will also be maintained based on the value specified for `max-aspect-ratio`.

- The **Smooth and Improve** option uses a combination of node movement and optimized smoothing to improve the quality. This option is a combination of the **Smooth** and **Improve** options. The cell aspect ratio will also be maintained based on the value specified for `max-aspect-ratio`.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When the **Smooth and Improve** option is enabled in the **Post Operations** group box in the **Improve** tab of the **Prisms** dialog box, the prism cells will be improved based on the options selected in the **Prism Improve** dialog box, after the required prism layers are created.

You can alternatively use the options in the **Prism Improve** dialog box to improve the prism cell quality in a postprocessing step after the mesh is created.

Mesh → Tools → Prism → Improve

7.9.3.5.2. Removing Poor Quality Cells

In cases where the prism mesh is created separately (without using the **Merge Cell Zones** option in the **Auto Mesh** tool) and the **prism-cap** zone exists, you can use additional options to remove layers of poor quality cells in regions of poor quality and sharp corners.

- The **Prism Post Ignore** dialog box contains options that allow you to remove poor quality prism cells based on quality, intersection, interior warp, and feature edges. You can specify the number of cell rings to be removed around the marked cells. Cells will be marked for removal in regions of sharp corners based on the **Ignore Options** selected and then extended based on the number of cell rings specified. Additional cells will be marked for removal in regions of high aspect ratio and feature angle (if selected in the **Expand Ignore Options** group box) around the exposed prism side.

  The **Feature Edges** tab contains options for manipulating feature edges to be used for the post-ignore operation. You can extract edge zones from the prism base zones when the prism cap zone is selected in the **Boundary Zones** list. The edge zones are extracted based on the **Feature Angle** specified. Other operations like merging or separating the edge zones are also available.

  The boundary will be smoothed at feature corners after the prism cells have been removed. The prism-side faces exposed by the removal of the prism cells will be collected in a zone named `prism-side-#`, while for a zone `wall-#`, the faces corresponding to the ignored prism cells will be collected in a zone named `wall-#:ignore`. You can also optionally smooth the prism side nodes from the base node to the cap node to create better triangles for the non-conformal interface.

  When the **Ignore** option is enabled in the **Post Operations** group box on the **Improve** tab of the **Prisms** dialog box, the prism cells will be removed based on the options selected in the **Prism Post Ignore** dialog box, after the required prism layers are created.

  Note:

  The **Feature Edge** option, under **Ignore Options** in the **Prism Post Ignore** dialog box, requires a prism-cap zone to be present so it cannot be used as a **Post Operation** control during prism generation.

  You can alternatively use the options in the **Prism Post Ignore** dialog box to remove layers of poor quality cells in a postprocessing step after the mesh is created.

  Mesh → Tools → Prism → Post Ignore

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The **Prism Tet Improve Cavity** dialog box contains options for creating a cavity in regions where the prism quality is adequate, but the quality of adjacent tetrahedra is poor. The cavity is created based on the tetrahedral cell zone, the quality measure and the corresponding threshold value specified. Additional cells will be removed based on the number of expand cell rings specified. You can create a cavity comprising only tetrahedral cells or optionally include prism cells when the cavity is created. When prism cells are also included in the cavity, you can specify whether the non-conformal interface is to be created.

  Mesh → Tools → Prism → Tet Improve Cavity

7.9.3.5.3. Improving Warp

Improving warp in prisms is a postprocessing step which is carried out after all the required prism layers are created. The face warp in the generated prism faces is improved by moving the nodes of the face to make it planar.

When the **Improve Warp** option is enabled in the **Post Operations** group box in the **Improve** tab of the **Prisms** dialog box, the prism layers having faces with warp $\geq$ 0.5 are identified. The nodes on the identified faces are moved and the new location of the node is updated only when:

1. The overall maximum warp of the faces connected to the node has decreased.
2. The overall maximum skewness of the cells connected to the node has decreased or remained the same.

This is useful in regions having complex sharp corners, where the nodes of a prism cell may have drastic normal change.

7.9.4. Prism Meshing Options for Scoped Prisms

Scoped prism controls allow you to apply prism layer parameters to specific regions of the mesh object. Accessible through the **Auto Mesh** volume-meshing tool, the **scoped** prism process first creates boundary layers based on the defined controls, and then proceeds to generating the tet/hexcore mesh.

Important:

- Volumetric regions need to be computed prior to object based volume meshing. The computing of regions includes topology checks, re-orienting of normals, and baffle identification and handling before generating the volume mesh.
- The direction of prism growth does not need to be set up separately. The prism layers are always grown into the regions selected; normals are re-oriented accordingly when the regions are computed.
- The scoped prisms method does not use all of the settings made in the **Prisms** dialog box or using the commands in the `/mesh/prism` menu. Scoped prisms are used for object based meshing and are widely used in the `mesh/scoped-prism` menu. Additionally, the scoped prism method is just another way to define controls on a group of zones, whereas final prism meshing ultimately takes place at the zone level.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To set up scoped prism controls, select **scoped** from the **Grow Prisms** drop-down list in the **Auto Mesh** dialog box and click **Set...** to open the **Scoped Prisms** dialog box.

- You use the **Scope To** drop down list to select fluid or solid regions.

  You can select **fluid-regions**, **solid-regions**, or **named-regions**. If named-regions is selected, you can enter the region name (wildcards supported) in the **Volume Scope** field or click the **Browse** button ( ) and select the regions in the **Volume Scope** dialog box.

- Within the selected regions, you use the **Grow On** drop-down list to select the boundary type on which to grow prism layers.

  You can use **all-zones** for unrestricted boundary type. Alternatively, use **only-walls** to scope prism growth by wall boundary type. To scope prisms by boundary name, use **selected-labels**, **selected-face-zones**, or **solid-fluid-interface** and enter the name or pattern in the **Boundary Scope** field, or click the **Browse** button ( ) to use the **Boundary Scope** dialog box.

  If interior baffles exist, the **Grow on both sides of baffles** option will be available. You can choose to grow prisms on both sides (default) or on a single side. In case of single side, use the **Color by Normal** option (hot-key combination **Ctrl**+**Shift**+**C** and **Ctrl**+**N**) to see on what side (gray) the prisms will grow.

- You use the **Offset Method** drop down list to select from **uniform**, **aspect-ratio**, or **last-ratio** as the prism offset method. The **geometric** growth method is used; you can specify the growth rate as appropriate.

- A meaningful **Name** is recommended.

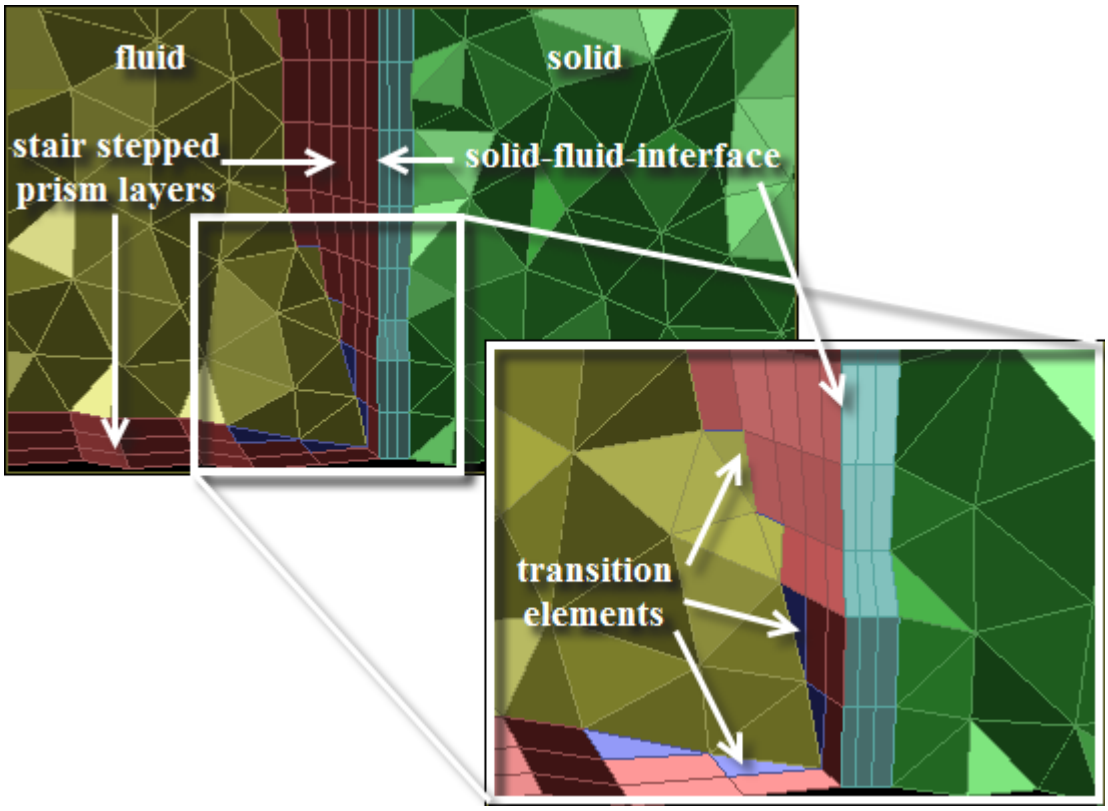
Important:

Fluent Meshing supports multiple scoped prism controls, however only one scoped prism control can be applied to a face zone in a given region.

In Figure 7.103: Use of Multiple Scoped Prism Controls (p. 777), prism layers are grown on adjacent zones within the fluid region, but with multiple scoped controls defined. The prism layers are stair stepped to obtain a gradual transition between the layers. Also, **solid-fluid-interface** is selected from the **Grow On** drop-down list, to enable growth of different number of layers in the solid and fluid regions.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.103: Use of Multiple Scoped Prism Controls



Note:

- The scoped prism controls can be read from a previously saved file (`*.pzmcontrol`) or can be written to a file for further use.
- The command `/mesh/scoped-prisms/set-no-imprint-zones` enables you to specify boundary zones which should not be imprinted during the prism generation.

Proximity Handling

Once the initial prism layers are generated, the prism mesh is improved by a combination of optimization-based smoothing and node movement to smooth the prism cell layers. The scoped prisms approach includes two steps of collision avoidance—layer compression and stair stepping. Use the **Growth Options** button to setup controls to avoid collision of prism layers in proximal regions.

Enable **Keep First Layer Offsets** to preserve the first offset height as other layers are modified to prevent collisions.

In regions of close proximity the prism layers are first compressed to resolve gaps and sharp corners, based on the **Gap Factor** and **Max Aspect Ratio** specified.

After this, cells are identified for stair stepping — a post prism meshing process carried out to avoid prism layer collision and maintain mesh quality. In areas where it is not possible to meet the quality

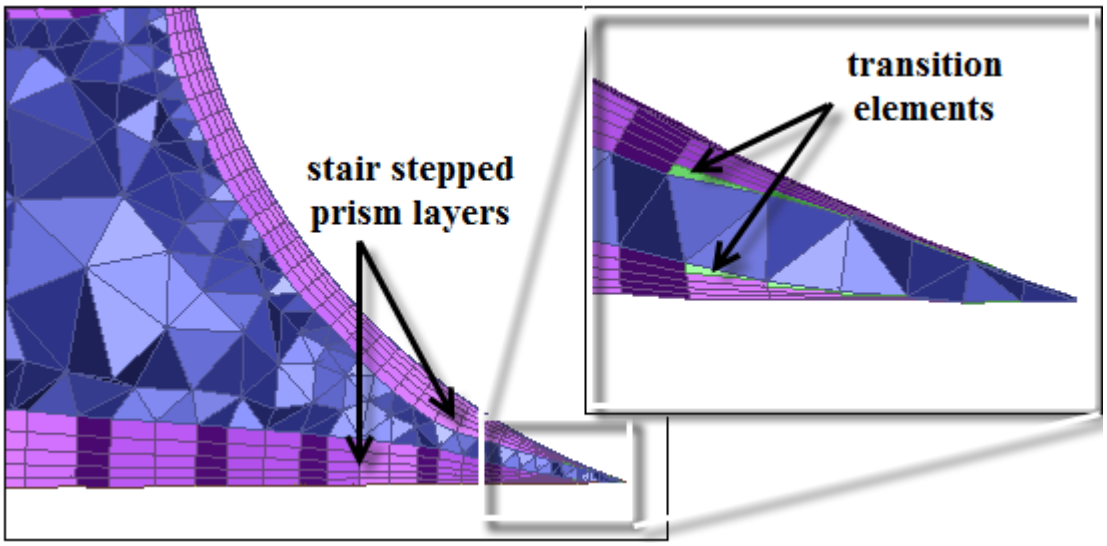
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

limit specified by the **Stair Stepping Threshold**, or where intersections are detected, the higher prism layers will be removed and filled with pyramids and tetrahedra as transition elements between the remaining layers and the volume fill. A list of locations where the prism layers were stair stepped is printed in the console. For prism meshing, it is recommended to use **ICEM CFD Quality** as the quality method since it is more stringent than the **Skewness** metric.

If the final mesh quality is not acceptable, you can use the **Auto Node Move** tool with ICEM CFD quality measure to improve the mesh.

Figure 7.104: Stair Stepped Prism Layers in Sharp Corner (p. 778) shows an example of a sharp corner where the prism layers have been stair stepped.

Figure 7.104: Stair Stepped Prism Layers in Sharp Corner



7.9.5. Prism Meshing Problems

This section discusses a number of common problems that you may encounter when generating prismatic meshes. An appropriate solution is also recommended for each problem. If the prism generation fails or is unsatisfactory, you may read in the mesh you saved before starting the prism generation (in step 6 of Procedure for Creating Zone-based Prisms (p. 753)) and repeat the process, incorporating the appropriate corrections.

The most common prism meshing problems are:

- Orientation
- Retriangulation failures
- Too many or too few nodes, or unknown face combinations
- Negative volumes, left-handed faces, or high skewness
- Growing too far
- Large jumps in prism height at the edges of layers

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Orientation

If the faces of the boundary zone from which you are building prisms are not all oriented in the same direction, you will be alerted that the faces have been reoriented, and the prism generation process will proceed.

Retriangulation Failures

If nodes, where adjacent zones meet the zones (from which prisms are growing) are duplicated, the mesher will be unaware that the zones are connected. It will ignore such zones when considering nodes for projection, and hence retriangulation is likely to fail.

Solution: To verify this problem check the messages printed before the first prism layer. It should contain a list of all the zones connected to the zones from which you are growing the prisms.

1. Make sure there are no free nodes in the surface mesh before growing any prisms.
2. Use the **Count Free Nodes** button in the **Merge Boundary Nodes** dialog box to check for free nodes.
3. Enable the **Free** option in the **Faces** section of the **Display Grid** dialog box to see where the free nodes are located.

Too Many/Few Nodes or Unknown Face Combinations

If you receive the following type of messages, then you are most likely growing prisms into existing cells.

```
Warning: wedge cell c887 has too many nodes
Warning: wedge cell c1928 has too few nodes
Warning: base_mask = 7 (0x7), unknown face combinations
```

Either you have already grown prisms from the selected zones, or you are growing them in the wrong direction into cells on the other sides of the selected zones. In the latter case, reorient the normals and growing the layers. See step 4 in Procedure for Creating Zone-based Prisms (p. 753).

Negative Volumes/Left-Handed Faces/High Skewness

The prism layer creation process will automatically stop if negative cell volumes, left-handed faces, or high skewness are detected.

Left-handed faces are faces that have collapsed into their cells. You can view them by enabling the **Left Handed** option in the **Faces** section of the **Display Grid** dialog box. Very high skewness occurs when the value of skewness is greater than the specified **Max. Allowable Skewness** in the **Improve** section of the **Prisms** dialog box). These problems can occur where prism layer fronts are advancing too quickly in areas of high curvature.

For example, the first layer height may be greater than the minimum edge length of your original surface mesh. You should compare the **First Height** with the results of clicking the **Edge Size Limits** button in the **Growth** section of the **Prisms** dialog box. The **First Height** should usually be smaller than the minimum edge size.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution: If the **First Height** is reasonable but you are unable to successfully generate your requested **Total Height**, try one of the following solutions:

- Reduce the **First Height** or the parameter used for the growth **Method** (that is, **Slope**, **Rate**, or **Exponent**), and increase the **Number of Layers**. Extra layers allow smoothing to compensate better for sharp edges and corners.
- If you have specified a nonzero value for **Orthogonal Layers** (in the **Direct** section of the **Prisms** dialog box), change it to zero. Increasing orthogonality can reduce robustness.
- If you have reduced the **Max. Angle Change** (in the **Direct** section of the **Prisms** dialog box) from its default of 70 degrees to improve orthogonality, reset it to 70.
- If you have disabled edge swapping or normal, offset, or edge smoothing, turn them back on.
- If you are using the minimum-height offset method (enabled with the `mesh/controls/prism/offset-method`text command), switch back to the uniform method.
- Increase the number of **Layers** in the **Improve** section of the **Prisms** dialog box if it is smaller than the number of layers being grown.
- If you are growing simultaneously from multiple zones, use zone-specific growth controls. Use smaller initial heights and/or growth rates for zones that are having problems.

Growing Too Far

The problem here is growing too many layers. Ideally, to provide for a smooth transition to the nearly-equilateral tetrahedra, the volume of the tetrahedra should be the local growth rate times the volume of the last layer prism cells.

Solution: To ensure this does not happen, verify that the prism parameters satisfy the above condition.

Large Jumps in Prism Height

If the prisms use quadrilateral faces of adjacent zones, and the node spacings on these faces are very different from the layer heights, the prism heights jump at the outer edges.

Solution: To fix this problem, replace the adjacent quadrilateral face zones with triangular face zones, using the `/boundary/remesh/triangulate` text command. When you reattempt the prism generation, the zone will be projected to and retriangulated.

Because the outer nodes of the prisms are projected to the adjacent boundary zone, and the original nodes on that portion of the zone are discarded, the different node spacing on the adjacent zone will not affect the prism heights at the outer edges.

7.10. Generating Tetrahedral Meshes

This chapter describes how to create the tetrahedral mesh in the domain. It also describes some of the common problems faced during tetrahedral meshing.

You can use one of the following techniques to generate tetrahedral meshes:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Automatic mesh generation (see Automatically Creating a Tetrahedral Mesh (p. 781))
- Manual mesh generation (see Manually Creating a Tetrahedral Mesh (p. 785))
- A combination of manual and automatic commands (see Initializing the Tetrahedral Mesh (p. 788), Refining the Tetrahedral Mesh (p. 790))

If the cells (for example, prisms or pyramids) have already been created in some portion of the computational domain, you need to create a domain encompassing the region to be meshed with tetrahedral cells before starting with the tetrahedral mesh generation. See Using Domains to Group and Mesh Boundary Faces (p. 528) for details. In such cases, all automatic and/or manual meshing actions will apply only to the active domain.

Alternatively, instead of creating a new domain, you can specify the cell zones to be preserved while generating the tetrahedral mesh, before proceeding with the mesh generation. You can use the command `/mesh/tet/preserve-cell-zone` to specify the cell zones to be preserved.

7.10.1. Automatically Creating a Tetrahedral Mesh

The **Mesh** → **Auto Mesh** feature contains commands that encapsulate the recommended volume mesh generation strategy. The starting point for this procedure is a valid surface mesh.

7.10.1.1. Automatic Meshing Procedure for Tetrahedral Meshes
7.10.1.2. Using the Auto Mesh Tool
7.10.1.3. Automatic Meshing of Multiple Cell Zones
7.10.1.4. Automatic Meshing for Hybrid Meshes
7.10.1.5. Further Mesh Improvements

7.10.1.1. Automatic Meshing Procedure for Tetrahedral Meshes

The automatic tetrahedral mesh generation process is divided into two fundamental tasks: initialization and refinement.

- The automatic initialization procedure includes the following steps:
  1. Merging free nodes.
  2. Deleting unused nodes.
  3. Improving the surface mesh.
  4. Initializing the mesh.
  5. Generating and separating cell regions.

Important:

If the mesh fails to initialize, typically this indicates a problem with the surface mesh. In some rare cases, the sliver size (which is automatically computed) may need to be changed. This may happen for domains in which the minimum boundary face size is very small as compared to the domain extent. When changing

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the sliver size, the rule of thumb is that the specified value should be of the order 1e-12 times the minimum face area.

- The automatic refinement procedure includes the following steps:

  1. Sorting the boundary faces by size.
  2. Refining the boundary cells.
  3. Reverse sorting cells by skewness.
  4. Refining the active cell zones.
  5. Swapping cells based on skewness.
  6. Reverse sorting cells by skewness.
  7. Smoothing the mesh.
  8. Removing boundary slivers.

You can select either the advancing front refinement, or the skewness-based refinement of the tetrahedral mesh. The refinement procedure is repeated a number of times.

You can control the number of repetitions for skewness-based refinement by setting the **Number of Levels** in the **Tet Refine Controls** dialog box. For each subsequent level of refinement, the cell skewness thresholds are lowered. Additional refinement levels increase the mesh resolution (and the number of cells) and decrease the average cell skewness.

Each refinement level actually consists of two sweeps through the refinement procedure:

- One sweep at the appropriate skewness threshold for that particular level of refinement.
- Another sweep with a high skewness threshold and a relaxed **Min Boundary Closeness** set in the **Tet Refine Controls** dialog box.

  This sweep attempts to reduce highly skewed cells in confined regions of the geometry.

Boundary slivers will be removed after the refinement sweeps are complete if the **Remove Slivers** option is enabled in the **Refinement** tab of the **Tet** dialog box.

In some rare cases, it may happen that the first sweep of the first refinement level will never finish because more skewed cells are formed due to refinement, which will be further refined to create more skewed cells, and so on. Such a problem may occur when there is a problem with the boundary mesh, such as rapid mesh transition in small gaps, and so on. You can avoid this problem by using the incremental improvement option for the refinement parameters or by increasing the **Min Node Closeness** value in the **Tet Refine Controls** dialog box.

You can select the fast transition option for the refinement parameters in order to generate a mesh with a smaller number of cells. When this option is selected, appropriate refinement parameters will be used to generate fewer cells during meshing. The rate of change of the cell size is increased

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

in order to reduce the number of cells generated. Alternatively, you can specify an appropriate cell sizing option with a sufficiently large value for the growth rate.

Important:

Various steps in the automatic meshing process can be added or eliminated using the **Tet** dialog box. In addition, most aspects of the automatic meshing process can be changed dynamically through the Scheme interface. Contact your support engineer for details.

7.10.1.2. Using the Auto Mesh Tool

You can use the **Mesh** → **Auto Mesh** tool to automatically:

- Generate a tetrahedral volume mesh starting from a boundary mesh.
- Generate a tetrahedral volume mesh in an unmeshed domain of a mesh that contains other cell shapes.

The **Auto Mesh** tool enables you to generate the tetrahedral volume mesh automatically. When you select **Tet** in the **Volume Fill** list and click **Mesh** in the **Auto Mesh** dialog box:

- If the volume mesh does not exist, all the steps listed in Automatic Meshing Procedure for Tetrahedral Meshes (p. 781) will be performed.
- If a volume mesh already exists, a **Question** dialog box will appear, asking if you want to clear the existing mesh. If you click **Yes**, the volume mesh will be cleared and the active steps in the automatic mesh generation process will be performed. If you click **No**, the operation will be canceled.

Important:

You can use the command `/mesh/tet/preserve-cell-zone` to specify the cell zones to be preserved during mesh generation and click **Mesh** to proceed with the automatic mesh generation.

7.10.1.3. Automatic Meshing of Multiple Cell Zones

The **Auto Mesh** tool enables you to mesh multiple cell zones automatically. This is useful when the mesh has multiple cell zones (for example, the problem requires a fluid zone and one or more solid zones). You can refine the mesh using refinement parameters specific to individual zones, if required. The procedure for meshing multiple cell zones includes the following steps:

1. Initialize the mesh.
2. Activate the appropriate zones using the **Manage Cell Zones** dialog box.
3. Set the zone-specific refinement parameters and refine the active zones.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you want to mesh *all* the zones included in the mesh with the same refinement parameters, modify the **Non-Fluid Type** in the **Tet** dialog box.

Important:

By default, the **Non-Fluid Type** is set to **dead** as the mesh is assumed to consist of a single fluid region and one or more dead regions. The active zone is considered to be the fluid zone and only this fluid zone will be considered for refinement during the automatic meshing process.

When the **Non-Fluid Type** is set to a type other than **dead** (for example, **solid**), all the zones will be active after the initialization is complete. Hence, all the zones will be considered for refinement. If required, you can change the zone type using the **Manage Cell Zones** dialog box.

Select the appropriate type from the **Non-Fluid Type** drop-down list in the **Tet Zones** group box in the **Tet** dialog box before initializing the mesh.

Note:

For zone-based meshing, if any cell zone has at least one boundary zone type as inlet, it will automatically be set to fluid type. For object based meshing, volume region type is used to determine the cell zone type.

7.10.1.4. Automatic Meshing for Hybrid Meshes

The **Auto Mesh** tool can also create prisms and pyramids automatically, allowing automatic generation of hybrid meshes.

- If a mix of surface mesh types (quadrilateral and triangular) is present in the domain, pyramid cell zones will be created on the quadrilateral boundary faces before meshing the tetrahedral domain.
- If prism growth parameters have been attached to a boundary zone (using the **Prisms** dialog box), the prism layers will be automatically extruded before meshing the tetrahedral domain.

  When pyramids or prisms are generated using the automatic mesh generation feature, the intermediate boundary zones will be merged automatically after the tetrahedral mesh generation is complete. This enhancement avoids the creation of additional boundary zones such as **prism-side** and **pyramid-cap** (see Zones Created During Prism Generation (p. 752) and Zones Created During Pyramid Generation (p. 737)).
- You can transition from quad to tri faces using the **Non Conformals** option. This option is useful when you want to avoid pyramids on quad faces when growing prisms from a boundary. For mixed surface mesh types (quadrilateral and triangular), you can select **Non Conformal** in the **Quad Tet Transition** list in the **Auto Mesh** dialog box. The surfaces containing quad faces will be copied and triangulated, keeping the original faces intact. The free nodes of the triangulated surface will then be merged with the nodes of the original surface mesh and both the surfaces will be converted to interface type.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.10.1.5. Further Mesh Improvements

Examine the following after completing the automatic mesh generation process:

- The mesh size with the **Report Mesh Size** dialog box (see also Determining Mesh Statistics (p. 563)).
- The prism layer cell quality distribution (see Determining Boundary Cell Quality (p. 565)), if appropriate.
- The tet cell quality reports (see also Determining Volume Mesh Quality (p. 564)).

It is typically possible to reduce the maximum skewness to the range 0.8–0.9. Skewness values higher than 0.9 are typically obtained due to constraints imposed by the surface meshes. If you still have highly skewed cells apply additional swapping and smoothing to improve the quality (see Smoothing Nodes (p. 512) and Swapping (p. 513)). To increase the density of the mesh locally, use refinement regions (see Using Local Refinement Regions (p. 791)).

> **Warning:**
>
> Do not over-refine the mesh. The volume mesh produced should be of sufficient density to resolve the shape of the geometry. The mesh can be improved more effectively using the solution-adaptive mesh capability provided in the solution mode in Fluent.

7.10.2. Manually Creating a Tetrahedral Mesh

In addition to the **Auto Mesh** tool, you can control the tetrahedral mesh generation process by modifying parameters at each step. The basic operations are described here, and the methods for modifying the associated parameters are described in Initializing the Tetrahedral Mesh (p. 788) and Refining the Tetrahedral Mesh (p. 790).

7.10.2.1. Manual Meshing Procedure for Tetrahedral Meshes

7.10.2.1. Manual Meshing Procedure for Tetrahedral Meshes

The basic components of the manual meshing process are examining and repairing the surface mesh, creating an initial mesh, refining the mesh, and improving the mesh.

The following steps describe the recommended meshing procedure.

Step 1: Examining the Surface Mesh

The first step in the mesh generation process is to examine the validity and quality of the surface mesh. The recommended approach includes the following steps:

1. Checking for free and isolated nodes (see Free and Isolated Nodes (p. 653)).
2. Checking and improving the surface mesh quality (see Improving Boundary Surfaces (p. 668)).
3. Visually examining the surface mesh for free, multiply-connected, skewed, and/or close-proximity faces (see Displaying the Mesh (p. 535)).
4. Making local repairs if required (see Modifying the Boundary Mesh (p. 660)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

After obtaining a valid, (ideally) high-quality surface mesh, you can proceed to create the volume mesh.

Step 2: Creating the Initial Mesh

The first step in generating the volume mesh is creating the initial mesh. This process first creates a pre-meshed box encompassing the entire geometry, and then sequentially introduces each boundary node into the mesh. As the nodes of a boundary face are inserted into the mesh, any necessary mesh modifications needed to insert the face are performed, effectively inserting both boundary nodes and faces into the mesh simultaneously. For more information see Initializing the Tetrahedral Mesh (p. 788).

Activating Multiple Zones (for Multi-zone Meshes Only)

If the mesh has multiple regions (for example, the problem requires a fluid zone and one or more solid zones), you can refine the mesh using refinement parameters specific to individual zones, if required. The procedure for meshing multiple cell zones includes the following steps:

1. Initialize the mesh.
2. Activate the appropriate zones using the **Manage Cell Zones** dialog box.
3. Set the zone-specific refinement parameters and refine the active zones.

   If you want to mesh *all* the zones included in the mesh with the same refinement parameters, modify the **Non-Fluid Type** in the **Tet** dialog box.

   Select the appropriate type from the **Non-Fluid Type** drop-down list in the **Tet Zones** group box in the **Tet** dialog box.

   Important:

   By default, the **Non-Fluid Type** is set to **dead** as the mesh is assumed to consist of a single fluid region and one or more dead regions. The active zone is considered to be the fluid zone and only this fluid zone will be considered for refinement during the automatic meshing process.

   When the **Non-Fluid Type** is set to a type other than **dead** (for example, **solid**), all the zones will be active after the initialization is complete. Hence, all the zones will be considered for refinement. If required, you can change the zone type using the **Manage Cell Zones** dialog box.

   Note:

   For zone-based meshing, if any cell zone has at least one boundary zone type as inlet, it will automatically be set to fluid type. For object based meshing, volume region type is used to determine the cell zone type.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Step 3: Refining the Mesh

You will refine the initial mesh by adding cells at the boundary and in the interior. The refinement methods available are the skewness-based method and the advancing front method. You can also define local refinement regions using the **Tet Refinement Region** dialog box. You can modify the parameters in the **Refinement** tab of the **Tet** dialog box and select additional refinement options for skewness-based refinement in the **Tet Refine Controls** dialog box, if required.

Step 4: Improving the Mesh

You can improve the skewness by smoothing, swapping, and refining the mesh further to improve the quality of the volume mesh. At any point in the mesh generation process, you can compute and plot the cell skewness distribution using the **Cell Distribution** dialog box. The number of cells in the specified range in each of the regularly spaced partitions is displayed in a histogram format. The *x* axis shows either the size or the quality and the *y* axis gives the number of cells or the percentage of the total. This will give you an idea of the improvement required. It is typically possible to reduce skewness to 0.8–0.85 (for simple geometries) or 0.9–0.95 (for more complex geometries).

Note:

If you have used domains to generate the mesh or group zones for reporting (as described in Using Domains to Group and Mesh Boundary Faces (p. 528)), you can report the cell distribution only for those cell zones that are in the active domain.

Step 4a: Swapping and Smoothing without Refining

Swapping and smoothing improve the mesh by manipulating the nodes and faces without increasing the total number of cells. Refinement, on the other hand, improves the mesh by adding nodes, which typically increases the number of cells. To get the best possible mesh with the minimum number of cells, perform only smoothing and swapping before refining the mesh any further.

Smoothing repositions interior nodes to lower the maximum skewness of the mesh. For swapping, given $n+2$ nodes in dimension n, there are at most two triangulations of the nodes depending on the configuration of the nodes. In cases where two triangulations exist, the two alternatives are examined, and the one that has the lowest maximum skewness is selected. Smoothing and swapping are automatically performed to improve the mesh when the **Improve Mesh** option is enabled in the **Refinement** tab of the **Tet** dialog box.

Refer to Smoothing Nodes (p. 512) and Swapping (p. 513) for details on improving the mesh by swapping and smoothing.

Step 4b: Further Refinement

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can further refine the cells in the active zones by changing parameters such as **Max Cell Volume**, and **Max Cell Skew** and **Max Boundary Cell Skew** (available only for skewness-based refinement).

Important:

Changing mesh size controls other than **Max Cell Volume** will have no effect on a mesh that has already been refined. Alternately, use local refinement regions. Refer to Using Local Refinement Regions (p. 791) for details.

Refining with skewness parameters reduced to values less than 0.5 will rarely improve the mesh and will be very time-consuming.

Important:

Do not over-refine the mesh, the volume mesh produced should be of sufficient density to resolve the shape and any intuitive flow features (for example, wall boundary layers). Additional resolution can be more effectively produced using the solution-adaptive mesh capability provided in the solution mode in Fluent.

At this step you will have an acceptable mesh for most geometries.

Step 4c: Boundary Slivers and Other Sources of Lingering High Skewness

When viewing the cell distribution plot, if you find that there are a few cells with very high skewness, check to see where they are located (that is, on the boundary or in the interior). To do so, use the **Report Boundary Cell Limits** dialog box. When using that dialog box, note that if you have used domains to generate the mesh or group zones for reporting (as described in Using Domains to Group and Mesh Boundary Faces (p. 528)), the report will apply only to the active domain.

Refer to Removing Slivers from a Tetrahedral Mesh (p. 514) and Moving Nodes (p. 519) for details on removing slivers and other mesh improvement options.

You can initialize and refine the tetrahedral mesh in a single step by clicking the **Init & Refine** button in the **Tet** dialog box after setting the appropriate parameters in the **Initialization** and **Refinement** tabs. Alternatively, you can set the appropriate parameters in the **Initialization** tab and click the **Init** button to create the initial mesh. You can then refine the initial mesh by setting the appropriate parameters in the **Refinement** tab and clicking the **Refine** button.

If you need to refine a particular region, you can define the region to be refined using the **Tet Refinement Region** dialog box.

7.10.3. Initializing the Tetrahedral Mesh

The first step in the volume mesh generation process is initialization. The initial mesh consists of the nodes and triangles of the boundary surface mesh. For some geometries, it is not possible to create a tetrahedral mesh from the boundary nodes alone. In such cases, a small number of nodes are automatically added in the interior of the domain. Interior nodes may also have to be added to resolve numerical problems associated with the Delaunay criterion.

7.10.3.1. Initializing Using the Tet Dialog Box

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.10.3.1. Initializing Using the Tet Dialog Box

You can use **Mesh** → **Tet** to initialize the mesh using the options available in the **Initialization** tab of the **Tet** dialog box.

1. Select the appropriate option from the **Non-Fluid Type** drop-down list. Enable **Delete Dead Zones**, if required.

   When the initial mesh is generated, all the cells are grouped into contiguous zones separated by boundaries. The mesh is considered to contain a single fluid zone and one or more dead regions. The zone just inside the outer boundary is set to be active and is labeled a fluid zone. All other non-fluid zones will be inactive. Only active zones will be considered for refinement during the mesh generation process.

   Note:

   For zone-based meshing, if any cell zone has at least one boundary zone type as inlet, it will automatically be set to fluid type. For object based meshing, volume region type is used to determine the cell zone type.

   You can refine different groups of zones using different refinement parameters for each group by toggling the zones between active and inactive. If however, you need to use the same refinement parameters for all the zones, you can change the specification of **Non-Fluid Type** to a type other than **dead** (for example, **solid**). When the **Non-Fluid Type** is set to a type other than **dead**, all the zones will be activated after initialization. Hence, you can set the appropriate refinement parameters without setting all the zones to be active.

   Note:

   If you have enabled the **Delete Dead Zones** option and subsequently change the region type from **dead** to either **fluid** or **solid**, then while generating the volume mesh using the **Auto Mesh** tool, the regions marked as dead are not filled with the volume mesh and are disconnected from the regions that are being volume meshed. These regions can later be reconnected after volume meshing by merging the relevant nodes.

2. Specify additional initialization parameters, if required. These parameters are available in the **Tet Init Controls** dialog box. Click the **Controls...** button to open the **Tet Init Controls** dialog box.

3. Click **Init** to initialize the mesh.

   A **Working** dialog box will appear, informing you that the initialization is in progress. Click the **Cancel** button in the **Working** dialog box to abort the mesh initialization process. Canceling the initialization will leave the mesh incomplete.

   - If you try to initialize the mesh after canceling an initial attempt, a dialog box will ask if it is OK to clear the incomplete mesh.

     After you approve, the initialization process will begin again.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you try to initialize the mesh when duplicate nodes exist, the initialization will fail. You must clear the mesh and merge the duplicate nodes before attempting the initialization again.
- If you try to initialize the mesh when a volume mesh already exists, a **Question** dialog box will appear, asking if you want to clear the existing mesh. If you click **Yes**, the volume mesh will be cleared and then the mesh will be initialized. If you click **No**, the operation will be canceled.
- If you need to preserve the existing mesh during the meshing process, use the command `/mesh/tet/preserve-cell-zone` to specify the cell zones to be preserved and click the **Init** button to proceed with the initialization.
- If the initial mesh generation fails, check the validity of the surface mesh. In some rare cases, you may need to change the sliver size parameter (by using the **Tet** dialog box).

Refer to Common Tetrahedral Meshing Problems (p. 793) for more information on meshing problems.

7.10.4. Refining the Tetrahedral Mesh

After initializing the mesh, you can improve the quality and density of the mesh using global and local refinement. Global refinement enables you to refine all cells in the active zones, while local refinement enables you to refine cells within a specified region.

In most applications, you will use only global refinement to create an acceptable discretization of the volume. Local refinement is used to modify the grading away from boundaries or increase the resolution of an interior region of the mesh. You need to define and activate the regions to be refined before proceeding.

The refinement process includes a series of sweeps through a sequence of sorting, refining, reverse sorting, and swapping and smoothing of the cells. You can specify the number of refinement levels to be performed for skewness-based refinement. Each refinement level consists of two iterations through the refinement procedure. For each subsequent level of refinement, the cell skewness thresholds are lowered. Additional refinement levels will increase the mesh resolution (and the number of cells) and decrease the average cell skewness. The preset refinement control parameters are available in the **Tet Refine Controls** dialog box. Alternatively, you can select the advancing front refinement method.

Specify whether the mesh has to be improved or if the slivers have to be removed during refinement.

- The **Improve Mesh** option performs additional smoothing and swapping with lowered skewness thresholds, attempting to improve the average skewness of the mesh.
- The **Remove Slivers** option attempts to lower the maximum skewness by improving highly skewed (sliver) cells. There are two approaches for sliver removal: fast and aggressive. Both methods use the same controls and give similar results for good quality surface meshes. In case of poor surface meshes (meshes with large size difference between neighboring elements, narrow gaps with widely varying mesh sizes on either side of the gap, and so on), the aggressive method will typically succeed in improving the mesh to a greater extent, but it may be slower than the default fast method.

  The aggressive method corresponds to the **Improve** option in the **Tet Improve** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The boundary mesh is fixed during the mesh improvement and sliver removal operations.

Refinement Controls

The refinement controls for the skewness method are available in the **Tet Refine Controls** dialog box. Click the **Controls...** button in the **Refinement** tab of the **Tet** dialog box to open the **Tet Refine Controls** dialog box.

Note:

Skewness-based refinement controls are no longer supported and will be removed in a future release.

The refinement controls for the advancing front method are available in the `/mesh/tet/controls/adv-front-method` menu. The `/mesh/tet/controls/adv-front-method/skew-improve/target?` command is important as it enables you to enable targeted skewness-based refinement for the advancing front method. The refinement process will attempt to reach the targeted skewness (specified by the command `/mesh/tet/controls/adv-front-method/skew-improve/target-skew`). Though the default values work well in most situations, it may be advantageous to increase the `target-skew` value in some cases (for example, cases where a combination of a poor surface mesh and narrow gaps will result in poor cells in the mesh) to increase the meshing speed.

7.10.4.1. Using Local Refinement Regions

7.10.4.2. Refinement Using the Tet Dialog Box

7.10.4.1. Using Local Refinement Regions

A refinement region limits refinement to specific cells inside the domain. When you use the **Refine** option in the **Tet Refinement Region** dialog box, the cells within the activated regions are refined. The primary use of refinement regions is to reduce the cell size in the region to less than it would be normally. Currently, the only possible region shape is a box, which can be oriented as required.

The region is defined by an *x*, *y*, and *z* range and its orientation about the *x*, *y*, and *z* axes. Any cell within the region or that intersects the region will be subjected to the refinement criteria. You can control the concentration of interior nodes by specifying the maximum cell volume in the refinement region. It is possible to save different values for the **Maximum Cell Volume** when you save each region. You can also specify an outer region with the required volume growth to have a smooth transition between the original and the refined cells.

Note:

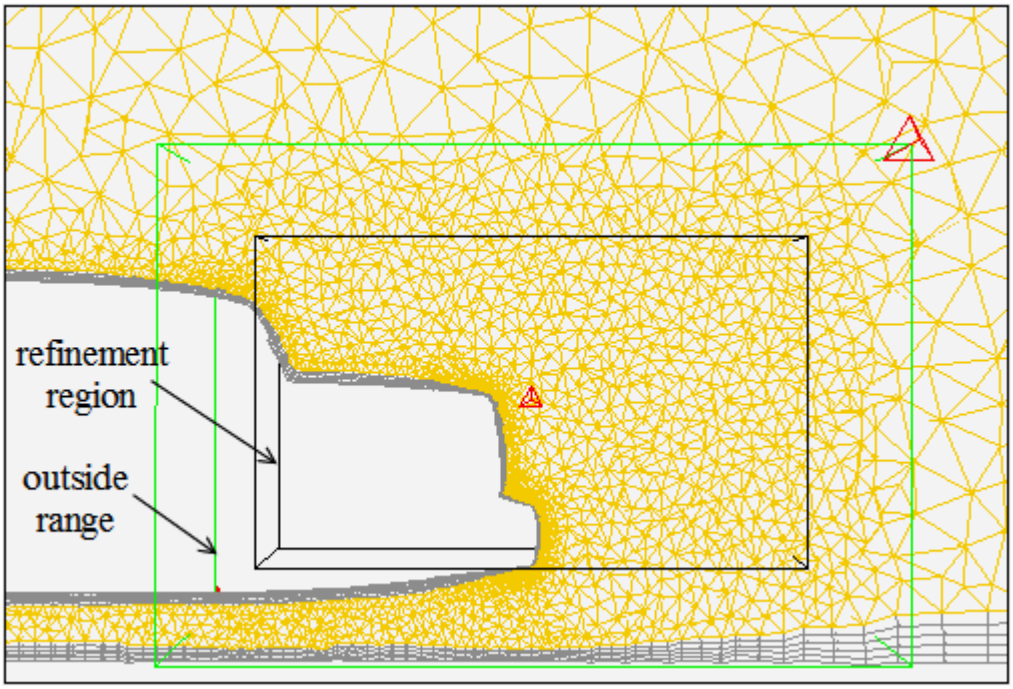
In addition to the use of refinement regions for refinement during the mesh generation process, they can also be used as a postprocessing tool to refine a region of an existing mesh.

You can create multiple regions that overlap each other and the geometry. To create additional regions, you can copy an existing region and then modify the parameters as required. The default region includes the entire geometry.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.105: Local Refinement Region for the Tetrahedral Mesh (p. 792) shows the refinement region and outer region. The maximum cell size for the refinement and outer regions is displayed based on the maximum cell volume and outside volume growth specified. For information on refining triangular boundary faces in anticipation of local refinement, refer to Refining the Boundary Mesh (p. 669).

Figure 7.105: Local Refinement Region for the Tetrahedral Mesh



7.10.4.2. Refinement Using the Tet Dialog Box

You can use **Mesh** → **Tet** to refine the mesh using the options available in the **Refinement** tab of the **Tet** dialog box.

1. Select the appropriate refinement method from the **Refine Method** drop-down list.
2. Enable the appropriate options in the **Options** group box.
3. Specify an appropriate value for **Max Cell Volume**.
4. Select the appropriate option from the **Cell Sizing** drop-down list.

   Ensure that the cells in the interior are not larger than the size required by selecting the appropriate option in the **Cell Sizing** list. The following options are available:

   geometric

   specifies that the cell size in the interior of the domain is obtained by a geometric growth from the closest boundary according to the growth rate specified.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

size-field

specifies that the cell size is determined based on the current size-field.

Note:

Additional cell sizing options (**none** (for skewness-based refinement only) and **linear**) are available only via text commands. See the Text Command List for details.

---

5. Modify the refinement control parameters, if required.

   - For the advancing front method, use the commands available in the `/mesh/tet/controls/adv-front-method` menu. You can also specify the targeted skewness for the refinement process (see Refinement Controls (p. 791) for details).

6. To refine specific regions of the mesh, click the **Local Regions...** button to open the **Tet Refinement Region** dialog box. Define the refinement regions and activate them. The number of activated regions will be reported in the **Message** field in the **Refinement** tab of the **Tet** dialog box.

7. Click **Refine** to refine the mesh.

7.10.5. Common Tetrahedral Meshing Problems

Most problems with the mesh generation process become manifest in the failure to generate an initial mesh. There are two sources of such problems:

- An invalid surface mesh
- Incorrect sliver size specified

Some of the common problems and suggestions for checking and fixing the mesh are described in this section.

To look at the nodes and faces that have not been meshed, enable **Unmeshed** in the **Display Grid** dialog box.

Duplicate Nodes or Faces

Some codes create multiple copies of the same node where two boundary curves or surfaces meet, resulting in a connectivity problem. This problem can be detected by reporting the number of free nodes or by drawing the free edges. If the geometry does not contain infinitely thin walls then there should not be any free nodes or edges. You can remove duplicate nodes using the `/boundary/delete-duplicate-nodes` command.

The presence of duplicate faces can be detected in one of two ways:

- If the nodes of the duplicate faces are not shared by the non-duplicate faces, the duplicate faces may have free edges that can be displayed.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If the nodes are not distinct or have been merged, there will be multiply-connected edges.

Note:

Duplicate faces can be handled, it is not necessary to remove them.

Extra Nodes or Faces

Nodes that are not used by any face are called unused nodes. The unused nodes can easily be found and deleted using the **Merge Boundary Nodes** dialog box, if those nodes are not required. However, there are cases where these additional nodes may be useful (see Inserting Isolated Nodes into a Tet Mesh (p. 728)). Extra faces can be identified and removed using the method described for duplicate faces.

To merge nodes that are not on a free edge, in the **Merge Boundary Nodes** dialog box disable **Only Free Nodes** in the **Compare...** or **With...** group box or in both group boxes. For example, if you read a hybrid mesh containing hexahedral cells (and quadrilateral boundary faces) and triangular boundary faces, there may be some duplicate nodes on adjacent quadrilateral and triangular boundary zones. To merge them, compare free nodes on both zones with *all* nodes on both zones (that is, select both zones in the **Compare...** and **With...** group boxes, and disable **Only Free Nodes** in one of them).

Intersecting Faces

If there are intersecting faces in the mesh, you will not be able to generate a mesh until the problem faces are removed. Most intersecting faces can be located using the `/boundary/mark-face-intersection` command. If intersecting boundary faces are encountered during initialization, a message will be displayed.

After the meshing fails, the unmeshed faces can be drawn to see the problem area. Usually, because the intersecting face will have been meshed, it will not be drawn and some additional sleuthing will be required. Draw all the faces near the unmeshed faces by setting display bounds (by using the **Display Grid** dialog box). Click the **Mark** button in the **Intersect Boundary Zones** dialog box to highlight the face intersection. Alternatively, you can use the command `/boundary/mark-face-intersection` to highlight the face intersection. When you set the **Feature Angle**, note that this specifies the minimum angle between the feature edges that should be preserved during retriangulation. All the edges in the zone having feature angle greater than the specified **Feature Angle** are retained. This option is useful for preserving the shape of the intersecting boundary zones. A value in the range of 10–50° is recommended; a large value may distort the shape of the intersecting boundary zones.

Poor Boundary Node Distribution

Some meshing problems can be caused by a poor quality surface mesh. The surface mesh may have highly skewed faces, or two or more boundaries in close proximity may have very different face sizes. In the latter case, the boundaries can be connected (for example, in a corner) or completely separate. Either way, the boundary mesh cannot be created without considering the impact of other nearby boundaries.

If the gap between two boundaries is **D**, the largest edge on a face should not exceed **D**. The mesh quality is extremely important when a coarse mesh is required in a small gap.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Non-Closed Boundaries

You can generate an initial mesh even if there is a hole in a boundary either due to missing faces or a gap between two zones. In this case, instead of having two separate zones on opposite sides of the boundary, you will have just one combined zone. This situation is evident because you will have very few zones.

- If the hole is in the outer boundary, the cells outside the boundary will be combined with the cells inside, resulting in an error. The edges around the hole will be marked as free edges and therefore can be displayed.
- If the problem is caused by a small gap between boundaries and the duplicate nodes have already been merged, increase the tolerance and perform additional merging.

The following command can be used to detect holes in the geometry by tracing the path between the two specified cells:

```
/mesh/tet/trace-path-between-cells
```

A path between the selected cells will be highlighted.

Interior Node Near Boundary

During refinement, a node may be placed too close to a boundary face, resulting in a highly skewed cell. You can detect this situation by displaying the highly skewed cells. You can then smooth the mesh to eliminate the problem.

If the interior node is constrained by the boundary faces, increase the **Min Boundary Closeness** (in the **Tet Refine Controls** dialog box) and regenerate the mesh.

7.11. Generating the Hexcore Mesh

Hexcore meshing is a hybrid meshing scheme that generates axis-aligned Cartesian cells inside the core of the domain and tetrahedral cells close to the boundaries. Hanging-node (or H-) refinements on the Cartesian cells enable efficient cell size transition from boundary to interior of the domain. This results in fewer cells with full automation and can handle complex geometries, internal walls and gaps.

The hexcore meshing scheme is applicable to all volumes but is useful mainly for volumes with large internal regions and few internal boundaries such as intrusions or holes. The **Hexcore** submenu contains options to control the hexcore mesh generation. The starting point is a valid surface mesh.

7.11.1. Hexcore Meshing Procedure

7.11.2. Using the Hexcore Dialog Box

7.11.3. Controlling Hexcore Parameters

7.11.1. Hexcore Meshing Procedure

You can control the general shape, size, and density of the core, as well as the size of elements created at the outer boundary of the core using the parameters in the **Hexcore** dialog box and the commands in the `/mesh/hexcore/` menu.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

There are two approaches available for generating the hexcore mesh.

- The default **Cartesian** type.
- The **Octree** type which supports the size controls defined, or the computed size field. This results in faster hexcore mesh generation. However, it does not support generation of hexcore mesh up to boundaries.

When the **Cartesian** type is used, internally the following steps are taken to generate the hexcore mesh:

1. Generating initial Cartesian cells inside a bounding box (or the region specified) around the volume to be meshed.
2. Marking both, the Cartesian cells that intersect the boundary mesh and those cells having sizes larger than the average size of the faces they intersect.
3. Marking additional buffer-layer cells adjacent to the cells marked in Step 2 (if specified).
4. Marking additional cells to enforce one-level refinement difference between adjacent cells.
5. Subdividing all marked (in steps 2–4) cells and repeating steps 2–5 until the local face size criteria is met.
6. Deleting cells that intersect or are within some distance to the closest face on the surface mesh.
7. Triangulating the external surface of the hexcore by converting quads into tri interface faces. The quad faces become parents (with type `parent-face`) of the interface faces.

   Note:

   Pyramids are not used for the transition from the hexcore to the tetrahedral cell regions.

8. Smoothing the interface faces (if interface smoothing is enabled).
9. Initializing and refining the tetrahedral cells between the interface faces and the boundary mesh.

   Note:

   The maximum skewness reported at the end of the refinement process is not necessarily the final maximum skewness. Sliver cells on the interfaces can be removed at a later stage.

10. Removing sliver cells on the interface faces.
11. Merging Cartesian cells with tetrahedral (and wedge, if present) cells to form contiguous cell zones.

When the **Octree** type is used, internally the following steps are taken to generate the hexcore mesh.

1. A uniform Octree domain is initialized for the region specified by the volume to be meshed.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. The octants are refined to local surface mesh sizes and to respect BOI size controls, if defined.

   If the size-field exists, the Octree refinement will be driven by the size-field instead.

3. Peel layers are enforced. Cells that intersect the surface mesh or are within some distance to the surface mesh are deleted.

4. Buffer layers are enforced as applicable.

5. The external surface of the hexcore mesh is triangulated, the quads are converted to tri interface faces. The quad faces become parents (with type `parent-face`) of the interface faces.

   Note:

   Pyramids are not used for the transition from the hexcore to the tetrahedral cell regions.

6. The interface faces are smoothed (if interface smoothing is enabled).

7. The tetrahedral cells between the interface faces and the surface mesh are generated.

8. The sliver cells on the interface faces are removed.

9. The hex cells are merged with tetrahedral cells to form contiguous cell zones.

Important:

If prism parameters have been attached to boundary zones, the prism layers will first be generated. The hexcore meshing procedure will then be applied to the resulting prism caps, along with the other boundary zones that were not involved in the prism mesh generation.

7.11.2. Using the Hexcore Dialog Box

To create a hexcore mesh using the **Hexcore** dialog box, do the following:

1. Read in a boundary mesh and check and improve its quality, if necessary.

2. (optional) Split any quad faces in the boundary mesh into triangular faces using the `/boundary/remesh/triangulate` command.

   By default, quad faces are not allowed while initializing the hexcore mesh.

   Alternatively, use the command `/mesh/non-conformals/controls/enable?` to enable the creation of a non-conformal interface.

   If this option is enabled, all the surfaces having quad elements will be copied and remeshed with triangular faces. The free nodes of the triangular mesh will be merged with the original surface mesh. The method to be used for retriangulation can be specified using the command `/mesh/non-conformals/controls/retri-method`.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The `quad-split` method is the default method used for retriangulation. You can select `prism`, `quad-split`, or `remesh` as appropriate.

3. Select the appropriate option in the **Type** list.

   Retain the default selection of **Cartesian** or select the **Octree** type.

4. Specify the hexcore parameters.

   Some parameters (maximum cell length or minimum cell length) required during the hexcore meshing are automatically calculated. The hexcore meshing parameters can be changed using the **Hexcore** dialog box or using the commands in the `/mesh/hexcore` text menu.

   Note:

   Some operations, such as Hexcore only, are not available for object-based volume meshing. Similarly, options under the **Zones** group in the **Hexcore** dialog box, that require a zone-based workflow, have no effect for object-based volume meshing.

5. Click **Create** in the **Hexcore** dialog box to generate the hexcore mesh.

7.11.3. Controlling Hexcore Parameters

The parameters that control the hexcore mesh generation can be changed using the **Hexcore** dialog box. The following options are available:

7.11.3.1. Maximum or Minimum Cell Length
7.11.3.2. Buffer Layers
7.11.3.3. Peel Layers
7.11.3.4. Defining Hexcore Extents
7.11.3.5. Local Refinement Regions

7.11.3.1. Maximum or Minimum Cell Length

The use of Maximum or Minimum cell length is determined by your choice of **Type**.

- When using the **Cartesian** type, the **Max Cell Length** is used for generating the initial Cartesian cells.

  An optimal maximum cell length can be automatically calculated for a particular mesh. Click the **Compute** button next to the **Max Cell Length** field to obtain the optimal maximum cell length for the mesh. Alternatively, specify the maximum cell length as required.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- When using the **Octree** type, the **Min Cell Length** as set by the global minimum size for scoped sizing is used for generating the hexcore mesh.

  Note:

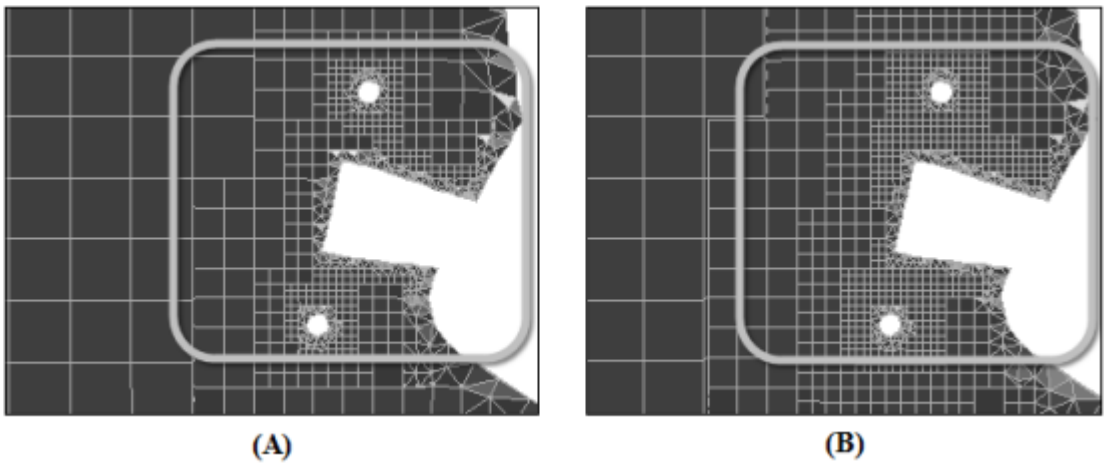
  Changing the default **Min Cell Length** value will change the global minimum size for scoped sizing as well.

  If a size-field has been computed, the minimum cell length will be the global minimum size specified for the size-field.

7.11.3.2. Buffer Layers

The Cartesian cells are marked (and subsequently subdivided) to satisfy the size requirement on the boundary mesh. When there is large disparity in size distribution between the boundary mesh and the initial Cartesian cells, there will be a rapid transition from fine to coarser cells. To avoid this, additional layers of cells are marked adjacent to those marked by the size requirement (see Figure 7.106: Hexcore Mesh Using (A) Buffer Layers = 1 (B) Buffer Layers = 2 (p. 799)). You can control the number of additional layers by setting the **Buffer Layers** in the **Hexcore** dialog box. The default number of buffer layers is set to 1. Setting the number of buffer layers to zero may result in a poor quality mesh.

Figure 7.106: Hexcore Mesh Using (A) Buffer Layers = 1 (B) Buffer Layers = 2



Note:

If the size-field has been computed and the Octree type is used to generate the mesh, the hexcore mesh size is driven by the size-field. In this case, the number of buffer layers is not required.

7.11.3.3. Peel Layers

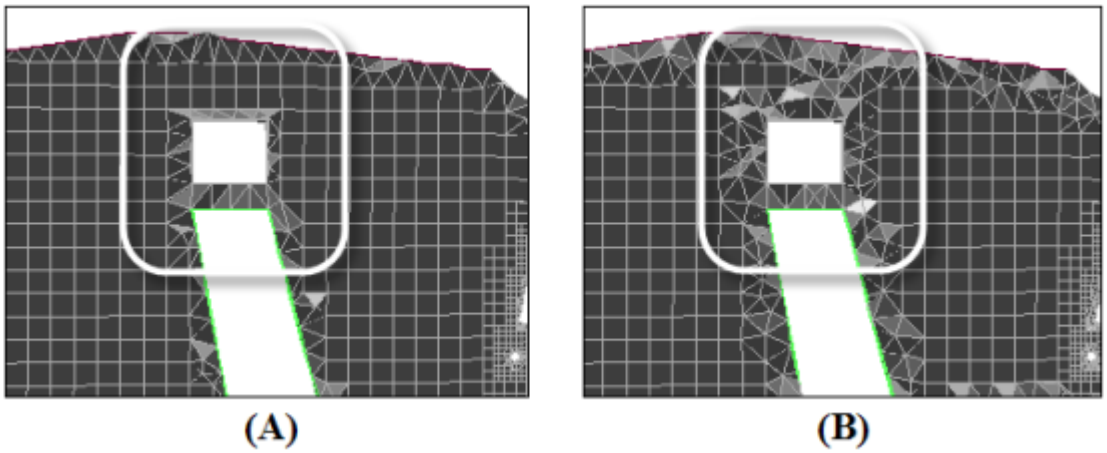
The peel layers control the gap between the hexahedra core and the geometry. After the Cartesian cells are subdivided to meet the size requirement, the cells intersected by boundary mesh and those within some distance to the closest face on the boundary mesh are deleted. The default value

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

for **Peel Layers** is 1, hence this distance is assumed to be the height of an ideal tetrahedral cell on the boundary face. If **Peel Layers** is set to 0, the gap size can be smaller than the ideal height. The resulting hexcore mesh will contain the maximum number of Cartesian cells possible for the chosen parameters.

Figure 7.107: Hexcore Mesh Using (A) Peel Layers = 0 (B) Peel Layers = 2 (p. 800) shows the hexcore mesh generated for different values of **Peel Layers**.

Figure 7.107: Hexcore Mesh Using (A) Peel Layers = 0 (B) Peel Layers = 2



7.11.3.4. Defining Hexcore Extents

When using the **Cartesian** type, you can choose to create the hexcore mesh up to the boundary of the domain instead of creating a tetrahedral mesh at the boundary. The hexcore mesh extents can be defined by specifying a box in which the hexcore mesh is to be generated and/or a set of axis-aligned planar boundaries to which the hexahedral core is to be extended. The extents can be defined as follows:

1. Enable **Define Hexcore Extents** in the **Hexcore** dialog box.
2. Click the **Specify...** button to open the **Outer Domain Parameters** dialog box.
3. Enable **Coordinate Extents** in the **Outer Domain Parameters** dialog box. The minimum and maximum domain extents will be automatically updated with values slightly greater than the bounding box of the surface mesh in the domain. The coordinate extents of the hexcore outer box can also be specified as required.

   Important:

   The hexcore mesh will extend to the defined domain extents. A warning will be displayed if the gap between the user-defined domain extents and geometric boundaries is less than 20% of size of the bounding box which contains the given geometry.

4. Enable **Draw Outer Box** and click **Draw** to verify that the domain extents are correctly defined.
5. Click **Apply** in the **Outer Domain Parameters** dialog box.

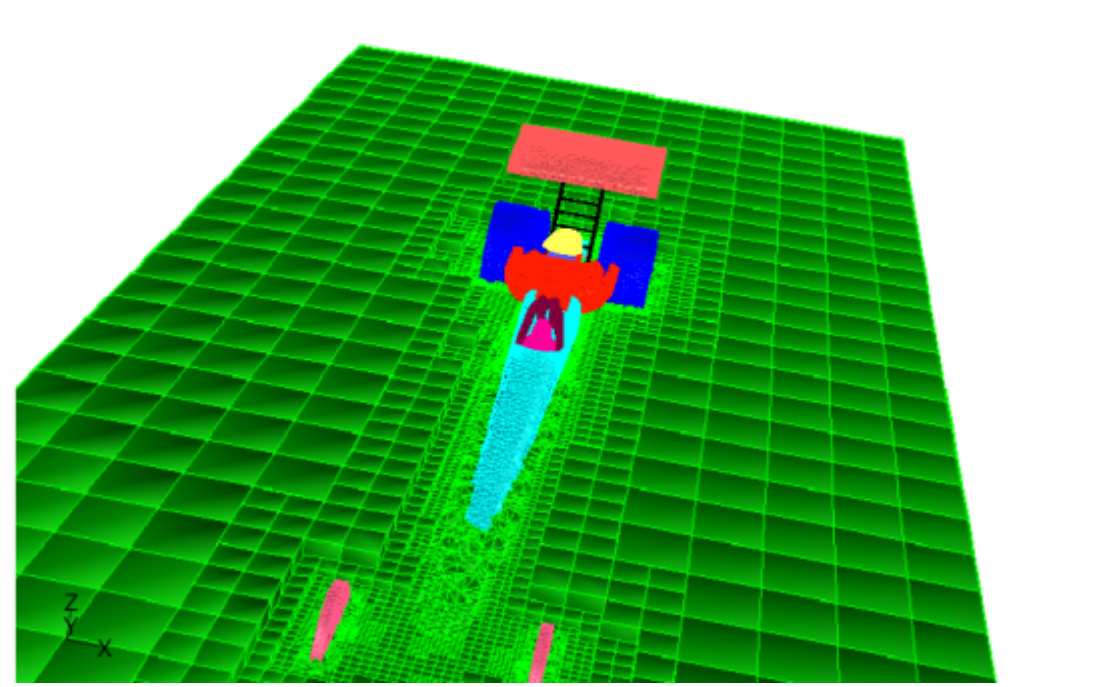
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Create the hexcore mesh.

This feature is useful for the hexcore mesh generation for external flow domains (for example, external aerodynamics cases where the boundary conformity is not needed for far-field boundaries). This also helps increase the count of hexahedral cells in the mesh.

In Figure 7.108: Hexcore to the Far-Field Boundary (p. 801), the hexcore mesh has been extended until the domain boundary and does not have a tetrahedral mesh at the far-field boundary.

Figure 7.108: Hexcore to the Far-Field Boundary



7.11.3.4.1. Hexcore to Selected Boundaries

You can use the **Boundary Extents** option in the **Outer Domain Parameters** dialog box to imprint quad faces on selected axis-aligned planar boundaries. The selected boundary will be replaced with a mix of quad- and tri-face zones. You can generate the hexcore mesh to selected boundaries as follows:

1. Enable **Define Hexcore Extents** in the **Hexcore** dialog box.
2. Click the **Specify...** button to open the **Outer Domain Parameters** dialog box.
3. Enable **Boundary Extents** in the **Outer Domain Parameters** dialog box and select the appropriate planar, axis-aligned boundaries to which the hexcore mesh is required.

   Note:

   The planar boundaries must be split into separated boundary zones.

   Click **Apply** in the **Outer Domain Parameters** dialog box.

   The hexcore box coordinates will automatically snap to the selected boundaries.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Enable **Draw Outer Box**. Draw the outer box along with the selected boundaries to verify that the box snaps to the selected boundaries.

If the box does not snap to a selected boundary, it indicates that the hexcore mesh cannot be grown to the boundary. If the selected boundary zone is a plane which is misaligned with the axis, you can align it using the **Auto Align** option as follows:

a. Select the zones to be aligned in the **Boundary Zones** selection list.

   The **Auto Align** group box will now be activated.

b. Enable **Auto Align**.

c. The recommended auto align tolerance value can be computed for the selected set of boundary zones. Click **Compute** to determine the recommended tolerance or enter an appropriate tolerance value.

d. Click **Apply** in the **Auto Align** group box to align the selected zones which are within the tolerance specified.

> **Warning:**
>
> The **Auto Align** option may deform the geometry permanently. Take care while selecting the boundary zones.

e. Draw the outer box along with the selected boundaries to verify that the box has snapped to the selected boundaries.

4. Enable **Delete Old Face Zones**, if required and click **Apply** in the **Outer Domain Extents** dialog box.

5. Create the hexcore mesh.

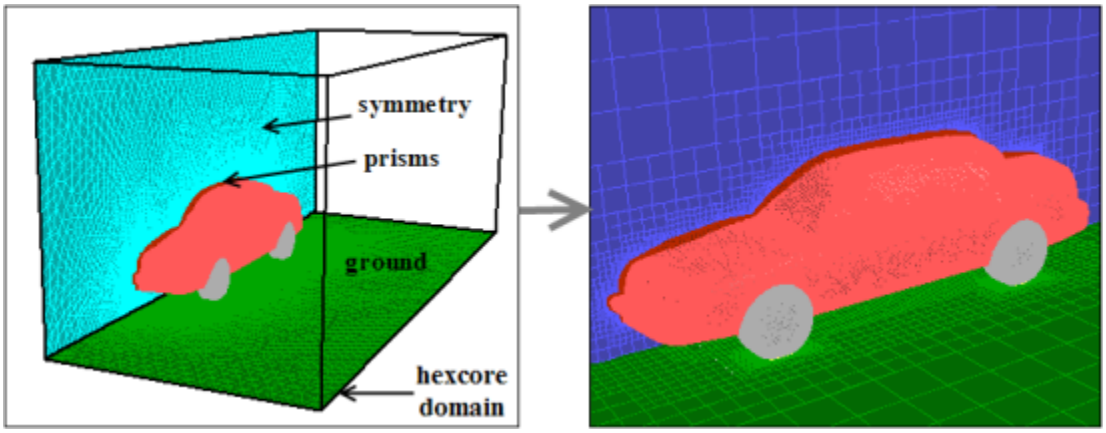
   The newly created quad face zones will be named with the original zone names.

The hexcore mesh will extend to the selected boundaries, aligned with the *X*, *Y*, and *Z* axes. An example is shown in Figure 7.109: Hexcore to Boundaries (p. 803).

> **Warning:**
>
> If you attempt to use this option when the geometry is completely within the domain, an error will be reported.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.109: Hexcore to Boundaries



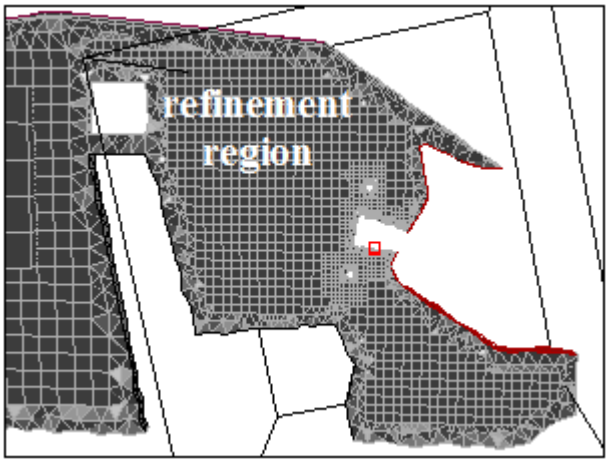
7.11.3.5. Local Refinement Regions

When using the **Cartesian** type, you can use local refinement regions to refine specific cells within the domain. During the hexcore meshing procedure, the cells within the activated local refinement regions will be refined. Currently, the only possible region shape is a box, either aligned with the coordinate axes, or oriented as required.

The region extents are defined by the center and the length of the region. The cell size within the region can be manipulated by setting the level of refinement relative to the maximum cell length in the hexcore domain. You can also orient the region as required. You can create multiple regions that overlap each other and the geometry. To create additional regions, you can copy an existing region and then modify the parameters as required. The default region includes the entire geometry.

You can use the **Draw** button in the **Hexcore Refinement Region** dialog box to display the defined region. Figure 7.110: Local Refinement Region for the Hexcore Mesh (p. 803) shows a refinement region defined. A sample Cartesian cell with the specified **Max Length** is also displayed at the center of the refinement box.

Figure 7.110: Local Refinement Region for the Hexcore Mesh



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.12. Generating Polyhedral Meshes

Polyhedral meshes contain polyhedral cells. An advantage that polyhedral meshes have shown relative to tetrahedral or hybrid meshes is the lower overall cell count, almost 3-5 times lower for unstructured meshes than the original cell count. Since the polyhedral mesh has a lower cell count than the equivalent original tetrahedral mesh, solution convergence will generally be faster, possibly saving some computational expense.

Polyhedral meshes can be generated using the object-based volume meshing approach. The starting point is a valid surface mesh (see Surface Mesh Processes (p. 617)).

Prerequisites for generating a polyhedral mesh include:

- A valid surface mesh must be a triangular mesh of good quality.
- Surface triangulation size should not exceed material thickness.
- The aspect ratio of prisms should be less than 50.

Note:

The polyhedral meshing process changes triangle faces in the surface mesh to hexagonal faces. Once the polyhedral mesh exists, you cannot regenerate the volume mesh unless you first select the **Restore Faces** option. When you select **Restore Faces**, the current object face zones and cell zones are deleted. You can then begin the volume meshing process again using the original valid surface mesh as the starting point. See Generating the Volume Mesh (p. 647) for more information about **Restore Faces**.

The process followed and steps required to generate the polyhedral mesh are explained.

7.12.1. Meshing Process for Polyhedral Meshes

7.12.2. Steps for Creating the Polyhedral Mesh

7.12.1. Meshing Process for Polyhedral Meshes

The **Auto Mesh** dialog box contains options for setting up and generating a polyhedral mesh from a valid surface mesh for all computed volumetric regions of the mesh object.

- If enabled, the prism mesh is generated based on the **scoped** prism settings specified. Polyhedral meshing does not support zone-specific prisms.
- The polyhedral meshing process starts by generating a tetrahedral mesh in two stages: initialization and refinement.
  - The automatic initialization procedure includes the following steps:
    1. Merging free nodes.
    2. Deleting unused nodes.
    3. Improving the surface mesh based on polyhedra-driven criteria.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Initializing the mesh.
5. Generating and separating cell regions.

   The cell zones are then associated with the regions and the cell zone type will be applied. Dead cell zones will be deleted.

– The refinement procedure includes the following steps:

1. Inserting nodes for refining the mesh.
2. Two stages of tet mesh improvement.

- The polyhedral mesh is created as a dual of the tetrahedral mesh, such that the vertex of the mesh-element (tetrahedron) is close to the center of the solver-element (polyhedron). Tessellations are created for the boundary and cell zones. When prism mesh is generated, the dual is used directly, meaning the layering structure of the mesh is maintained.
- The polyhedral mesh is improved by operations like merging short edges and splitting cells. Concave cells, concave boundary faces, and stair-stepped poly cells are split to improve quality. The quality is further improved by smoothing.

  The best quality measures for polyhedral meshing are squish and ortho skew. If you select squish or ortho skew, skewness evaluations during smoothing will be supplemented with additional internal measures. If you select a quality measure other than squish or ortho skew, mesh generation will use an internal variant of squish because it provides some direct control over the convexity of the cell.

7.12.2. Steps for Creating the Polyhedral Mesh

The **Auto Mesh** dialog box contains options to control the polyhedral volume mesh generation.

You can generate the poly mesh as follows:

1. Open the **Auto Mesh** dialog box from the context-sensitive menu available by right-clicking on any mesh object or its **Volumetric Regions** or **Cell Zones** branch in the tree.

   You can also use the **Mesh** → **Auto Mesh** menu item to open the **Auto Mesh** dialog box.

2. Ensure that the mesh object is selected in the **Object** drop-down list.

   Note:

   If you open the **Auto Mesh** dialog box from the context-sensitive menu in the tree, the Mesh Object to which the cell zones or volumetric regions belong is automatically selected.

3. Enable/disable the **Keep Solid Cell Zones** option, as appropriate.
4. Select the appropriate option in the **Grow Prisms** drop-down list in the **Boundary Layer Mesh** group box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. Retain the default selection of **none** if you do not need to grow prism layers for the current meshing approach.

b. Select **scoped** if you want to specify object-based prism controls. Click **Set...** to open the **Scoped Prisms** dialog box and define the prism controls for the mesh object. Refer to Prism Meshing Options for Scoped Prisms (p. 775) for details.

Tip:

You can save your scoped prism controls to a file (*.pzmcontrol) for use in batch mode, or read in a previously saved scoped prism file.

Note:

- Poly meshing does not support zone-specific prisms.
- Stair-stepping will occur in regions of transition between the prism layers and the adjacent tets.

5. Set the **Poly** mesh parameters.

Select the **Poly** option from the **Volume Fill** list and set the following **Volume Fill Options**.

- Select the appropriate option for **Cell Sizing**.
  - **Size Field** specifies that the cell size is determined based on the current size-field.
  - **Geometric** specifies that the cell size in the interior of the domain is obtained by a geometric growth from the closest boundary according to the growth rate specified.

    Set the **Growth Rate** required.
- Specify the **Max Cell Length**. Click **Compute** to compute the maximum cell size based on the mesh object.

Note:

You can also set these parameters in the **Poly** dialog box.

Click the **Set...** button to open the **Poly** dialog box.

a. Select the options in the **Options** group box.

The **Merge Free Nodes**, **Delete Unused Nodes**, and **Improve Poly Mesh** options are enabled by default. You can also include improving the surface mesh by enabling the **Improve Surface Mesh** option.

b. Specify an appropriate value for **Feature Angle**. This sets the threshold for preserving features. The default setting is 30 degrees.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. Select the appropriate cell zone type from the **Non-Fluid Type** list in the **Poly Zones** group box.

Note:

Dead cell zones will be automatically deleted after the mesh is initialized.

When the initial mesh is generated, all the cells are grouped into contiguous zones separated by boundaries. The mesh is considered to contain a single fluid zone and one or more dead regions. The zone just inside the outer boundary is set to be active and is labeled a fluid zone. All other non-fluid zones will be inactive. Only active zones will be considered for refinement during the mesh generation process.

Note:

Volume region type is used to determine the cell zone type.

You can refine different groups of zones using different refinement parameters for each group by toggling the zones between active and inactive. If however, you need to use the same refinement parameters for all the zones, you can change the specification of **Non-Fluid Type** to a type other than **dead** (for example, **solid**). When the **Non-Fluid Type** is set to a type other than **dead**, all the zones will be activated after initialization. Hence, you can set the appropriate refinement parameters without setting all the zones to be active.

d. In most cases, the default node spacing threshold should be acceptable. If you want to change it, click **Controls** to open the **Poly Init Controls** dialog box where you can modify the value.

e. Set poly cell growth parameters using the options in the **Cell Size** group box. Specify appropriate values for **Max Cell Volume** and **Growth Rate**. You can use the **Compute** button to determine the maximum cell size based on the size field.

Ensure that cells in the interior are not larger than the size required by selecting the appropriate option in the **Cell Sizing** list:

geometric

specifies that the cell size in the interior of the domain is obtained by a geometric growth from the closest boundary according to the growth rate specified.

size-field

specifies that the cell size is determined based on the current size-field.

f. Click the **Local Regions** button to access the **Tet Refinement Region** dialog box, where you can setup and activate local tetrahedral refinement regions. These regions are used during the initial tetrahedral mesh generation; they are not directly applicable to the poly mesh. The number of activated regions will be reported in the **Message** field.

6. Return to the **Auto Mesh** dialog box and enable or disable additional **Options** as desired.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Enable **Merge Cell Zones within Regions** to create a single cell zone within a region, or disable to keep the cell zones separate.

7. Click **Mesh** in the **Auto Mesh** dialog box.

7.12.2.1. Further Mesh Improvements

Examine the following after completing the automatic mesh generation process:

- The mesh size with the **Report Mesh Size** dialog box (see also Determining Mesh Statistics (p. 563)).
- The prism layer cell quality distribution (see Determining Boundary Cell Quality (p. 565)).
- The polyhedral cell quality reports (see Determining Volume Mesh Quality (p. 564)).

Note:

The quality measure will be set to **Inverse Orthogonal Quality** after the polyhedral mesh is generated.

7.12.2.2. Transferring the Poly Mesh to Solution Mode

Node weights for node-based gradients are enabled by default for poly meshes generated in Fluent Meshing. This setting can improve the accuracy of the displayed results near wall edges when you are displaying contours on a native poly mesh.

When you transfer the poly mesh to solution mode, a message will notify you that this interpolation is enabled. You can disable it by setting the `/display/set/nodewt-based-interp?` command to **no**.

7.13. Generating Poly-Hexcore Meshes

Poly-hexcore meshes contain polyhedral and hexcore cells. Therefore, the poly-hexcore mesh characteristics contains elements of both types of meshes. For more information about polyhedra meshes, see Generating Polyhedral Meshes (p. 804). Likewise, for more information about hexcore meshes, see Generating the Hexcore Mesh (p. 795).

7.13.1. Steps for Creating the Poly-Hexcore Mesh

7.13.1. Steps for Creating the Poly-Hexcore Mesh

The **Auto Mesh** dialog box contains options to control the poly-hexcore volume mesh generation.

You can generate the poly-hexcore mesh as follows:

1. Open the **Auto Mesh** dialog box from the context-sensitive menu available by right-clicking on any mesh object or its **Volumetric Regions** or **Cell Zones** branch in the tree.

   You can also use the **Mesh** → **Auto Mesh** menu item to open the **Auto Mesh** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Ensure that the mesh object is selected in the **Object** drop-down list.

   Note:

   If you open the **Auto Mesh** dialog box from the context-sensitive menu in the tree, the Mesh Object to which the cell zones or volumetric regions belong is automatically selected.

3. Enable/disable the **Keep Solid Cell Zones** option, as appropriate.

4. Select the appropriate option in the **Grow Prisms** drop-down list in the **Boundary Layer Mesh** group box.

   a. Retain the default selection of **none** if you do not need to grow prism layers for the current meshing approach.

   b. Select **scoped** if you want to specify object-based prism controls. Click **Set...** to open the **Scoped Prisms** dialog box and define the prism controls for the mesh object. Refer to Prism Meshing Options for Scoped Prisms (p. 775) for details.

      Tip:

      You can save your scoped prism controls to a file (*.pzmcontrol) for use in batch mode, or read in a previously saved scoped prism file.

5. Set the **Poly-Hexcore** mesh parameters.

   Select the **Poly-Hexcore** option from the **Volume Fill** list and set the following **Volume Fill Options**.

   - Specify the number of **Buffer Layers** and **Peel Layers** for the poly-hexcore mesh.
   - Specify the **Min Cell Length**. Click **Compute** to compute the maximum cell size based on the mesh object.

   Click the **Set...** button to open the **Poly-Hexcore Controls** dialog box.

   a. Select the **Avoid 1:8 Cell Jump in Hexcore** option to eliminate the 1:8 cell transition that can occur in the hexcore region. When enabled, use the **Replace (With Polyhedra)** drop-down menu to select whether just large cells, just small cells, or if both types of cells are to be replaced with polyhedra cells in the hexcore transition.

      You can also use the corresponding text command: `mesh/poly-hexcore/controls/avoid-1:8-cell-jump-in-hexcore`.

   b. Select the **Mark Core Region Cell Type as Hex** to have only hexcore cells within the core of the volume mesh.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also use the corresponding text command: `mesh/poly-hexcore/controls/mark-core-region-cell-type-as-hex?`. The default value is `yes`.

c. Select the **Only Polyhedra for Selected Regions** to assign polyhedra only to selective regions. This option is available only if you have first computed your region(s).

Once this option is enabled, you need to choose a specific mesh object and your regions. For the specified **Mesh Object**, identify the **Largest Region** or click **Compute** to calculate the largest region. Once determined, the selected region becomes the active region.

For **Select Regions (Largest Region Excluded)**, you can choose **All fluid regions**, **All solid regions**, or **other**. Fluent automatically excludes the largest region from the listing of available regions to choose.

When you choose **other**, the **Selected Regions** field is available so that you can specify one or more regions. Enter the region name(s), or use the **...** button to open the **Volume Scope Poly** dialog.

In the **Volume Scope Poly** dialog, you can select one or more regions, using the **Draw** button to visualize the regions prior to selection. Once you have selected your region(s), click **OK**, and those selections will be displayed as **Selected Regions** in the **Poly Hexcore Controls** dialog.

You can also use the corresponding text command: `mesh/poly-hexcore/controls/only-polyhedra-for-selected-regions`. For example:

```
Meshing/mesh/poly-hexcore/controls> only-polyhedra-for-selected-regions
Mesh object [""] part-1
only-polyhedra-for-selected-regions? [yes] yes
Do you want to specify the largest volume region or compute? [yes] yes

Largest volume region name [""] "region_2"

Select regions: [other]>--> other
Use the following list and enter a valid region name for the mesh object "part-1" (Largest Region Exclu
(region_1)
Region name [""] "region_1"
Region name [""]

Meshing/mesh/poly-hexcore/controls>
```

6. Return to the **Auto Mesh** dialog box and enable or disable additional **Options** as desired.

   - Enable **Merge Cell Zones within Regions** to create a single cell zone within a region, or disable to keep the cell zones separate.

7. Click **Mesh** in the **Auto Mesh** dialog box.

7.14. Generating the CutCell Mesh

CutCell meshing is a general purpose hex-dominant meshing technique. The CutCell meshing algorithm is suitable for a large range of applications, and due to the large fraction of hex cells in the mesh, often produces better results than regular tetrahedral meshes. This method can be used instead of tetrahedral or hexcore meshing, without requiring a very high quality surface mesh as a starting point. Also, this

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

method uses a direct surface and volume approach without the need of cleanup or decomposition, thereby reducing the turnaround time required for meshing.

The following sections are described in this chapter:

7.14.1. The CutCell Meshing Process
7.14.2. Using the CutCell Dialog Box
7.14.3. Improving the CutCell Mesh
7.14.4. Post CutCell Mesh Generation Cleanup
7.14.5. Generating Prisms for the CutCell Mesh
7.14.6. The Cut-Tet Workflow

7.14.1. The CutCell Meshing Process

The CutCell meshing process involves the following approach:

1. Objects, material points (optional), and size functions are defined.
2. The initial size of the Cartesian grid is computed based on the minimum and maximum size set for the size functions.
3. A uniform Cartesian grid is created within the bounding box for the geometry.

   The base size for the Cartesian grid is computed from the minimum and maximum size specified in the **Size Functions** dialog box as follows:

   $$\textbf{Base Size} = 2^{n} \times \textbf{Min Size}$$

   such that

   $$\textbf{Base Size} \leq \textbf{Max Size}$$

   where **n** is the number of refinement levels.

   Note:

   You should maintain the ratio between the base size and the global minimum size such that $\textbf{Base Size} = 2^{n} \times \textbf{Min Size}$. This ensures that the correct minimum size is used during the CutCell meshing.

   Warning:

   During initialization, the Cartesian grid created will contain the maximum number of Cartesian cells possible for the computed base size. If the cell count of the initial Cartesian grid exceeds the limit, use the command `/mesh/cutcell/set/max-initial-cells` to set a more appropriate number.

4. The size function values are computed and the grid is then adaptively refined based on the local size function values.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.111: Schematic Representation of the Cartesian Grid Refinement Using Size Functions (p. 812) shows a schematic representation of the refinement using size functions. The size source specified via the size functions will be assimilated into the size function octree. The final mesh at respective locations will reflect an interpolated size based on the size functions.

Figure 7.111: Schematic Representation of the Cartesian Grid Refinement Using Size Functions

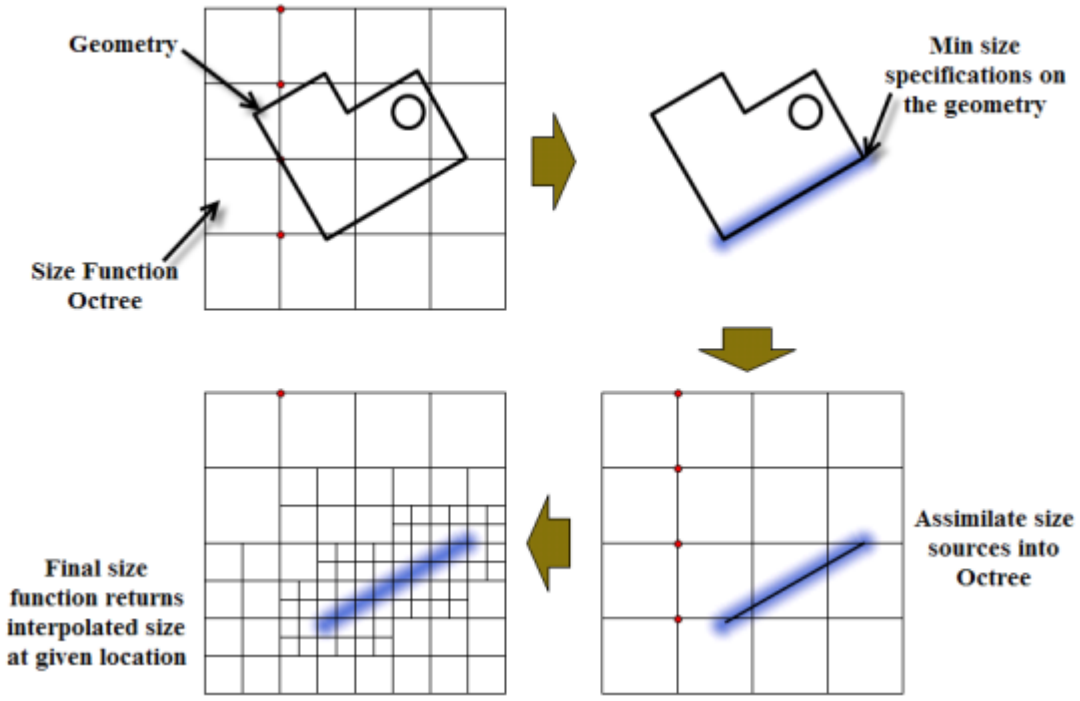
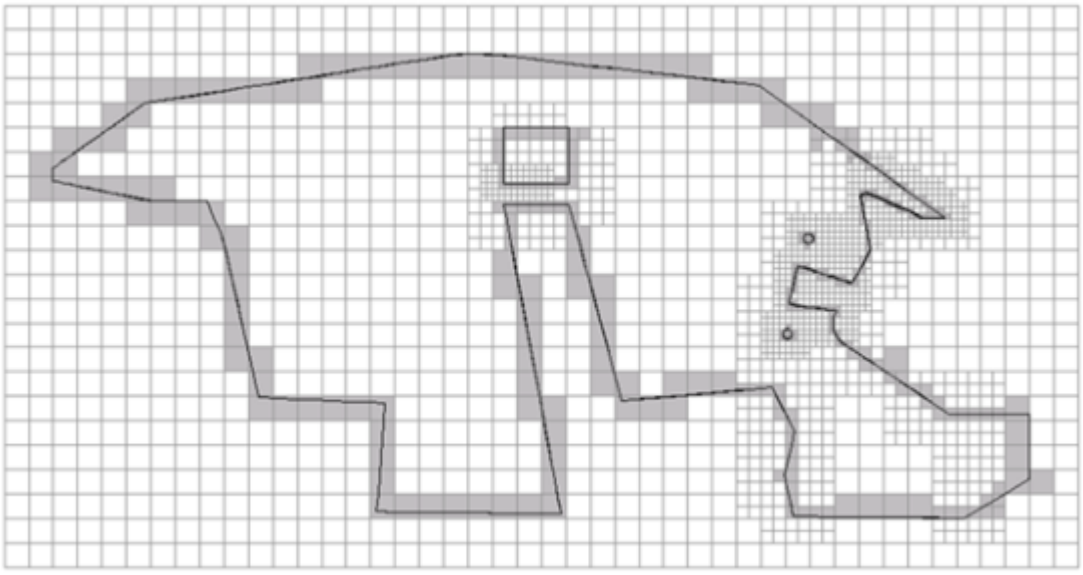


Figure 7.112: Mesh After Refinement (p. 812) shows the mesh after refining the initial grid based on size functions.

Figure 7.112: Mesh After Refinement

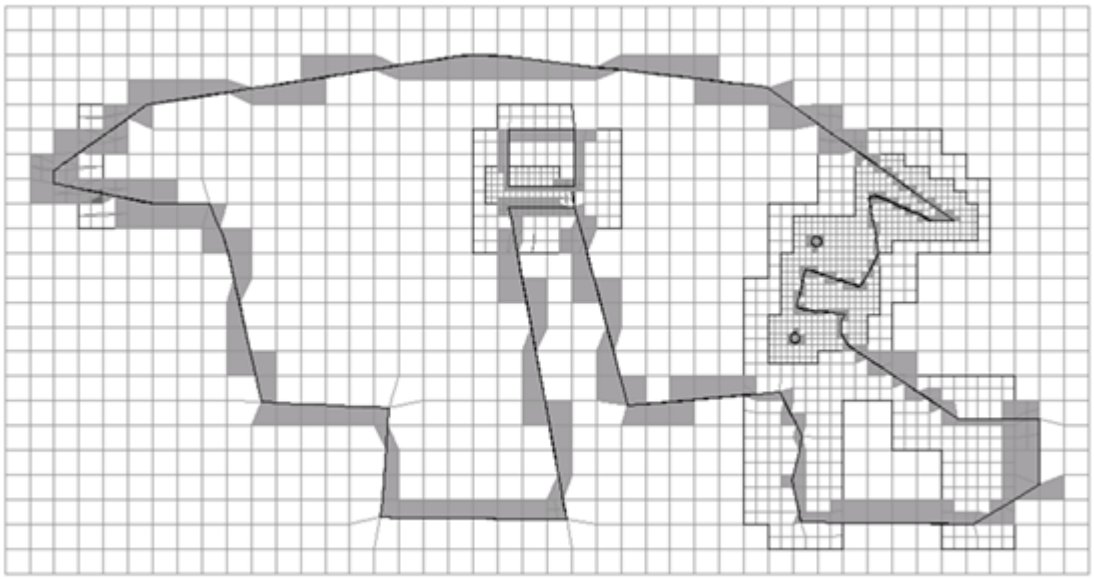


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. The cells intersected by the geometry are marked. Only nodes on marked cells are considered for projection. The nodes are projected to the geometry (corner, edge, and face in order of reducing priority).

   Figure 7.113: Mesh After Projection (p. 813) shows the mesh after node projection.

   Figure 7.113: Mesh After Projection



6. The edges intersected by the geometry are identified. Mesh edges to be preserved/recovered are determined, and are used to construct mesh faces. Once the mesh faces are identified, cells are decomposed to recover these faces. The cells are decomposed based on a number of templates.

   Note:

   The CutCell mesher may have problems capturing features like acute internal and external face angles (for example, trailing edges of fins, wheel-ground intersections). If such features are not recovered properly, the prisms generated at such locations are most likely to have bad quality.

   In such cases, you can use the `set-thin-cut-edge-zones` command (see Resolving Thin Regions (p. 819)) and specify the edges where the feature capturing fails, and then regenerate the mesh.

   The quality of the cells thus generated is improved.

7. You can set the default parameters to be used for improving the CutCell mesh using the command `/mesh/cutcell/set/set-post-snap-parameters`. This command sets the quality limits and other parameters relevant to the node movement and cavity remeshing that are performed to improve quality.

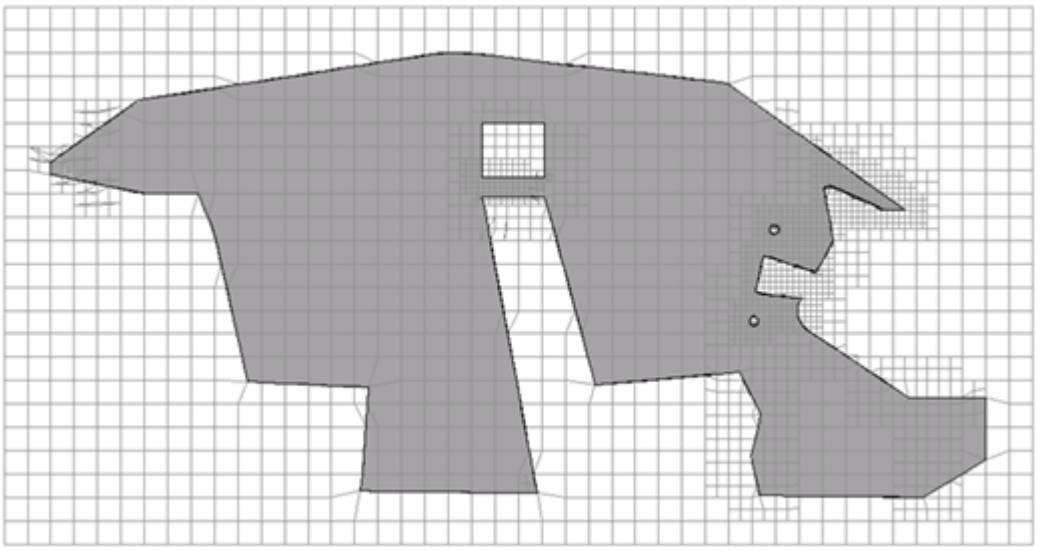
8. Cells are separated into cell zones based on the respective objects and material points (if any). When a cell has a vertex that lies out of the object while the other lies within the object, the cell

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

will be decomposed further to represent the boundary crossing the cell. A cell included in multiple objects will be included with the object having the highest priority.

Figure 7.114: Cells Separated After Decomposition (p. 814) shows the cells separated into respective cell zones after decomposition.

Figure 7.114: Cells Separated After Decomposition



9. Dead zones are generally deleted (**`auto-delete-dead-zones?`** is enabled by default). Solid zones will be retained or deleted, depending on the setting of **`auto-delete-solid-zones?`** (disabled by default).

10. The boundary mesh is recovered and separated based on the underlying geometry.

    - Faces whose adjacent neighboring cells are in different cell zones automatically constitute the boundary mesh.
    - The neighboring cells of a face on an internal baffle are in the same cell zone. In such cases, faces close to and nearly parallel to the baffle surface are recovered to represent the baffle surface.
    - As each cell zone is a closed region, the mesh boundary is conformal.

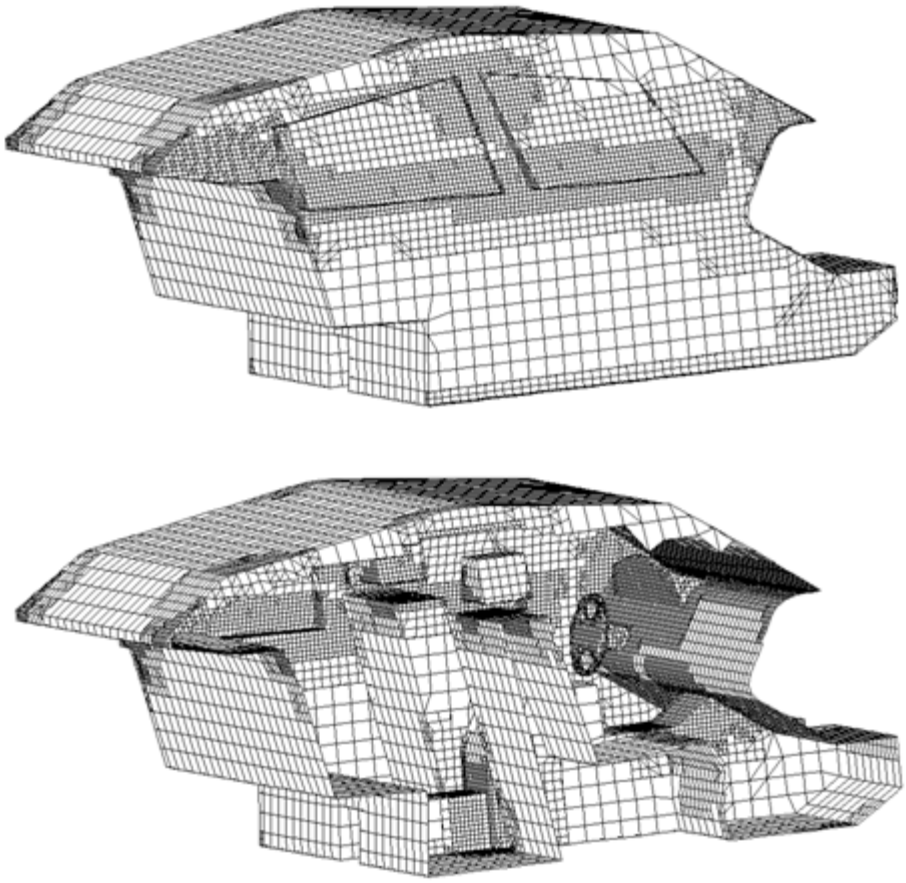
    The boundary zone types are assigned based on the underlying geometry zone type. Figure 7.115: CutCell Mesh After Boundary Recovery (p. 815) shows the CutCell mesh after the boundary mesh is recovered.

    Note:

    The CutCell mesher will assign the type **wall** on the surface recovered over a geometry zone of the type **internal** (for example, baffles) and the type **geometry**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.115: CutCell Mesh After Boundary Recovery



7.14.2. Using the CutCell Dialog Box

The **CutCell** dialog box and the commands in the `/mesh/cutcell` menu enable you to perform various tasks related to generating the CutCell mesh.

You can use **Create** in the **CutCell** dialog box to generate the CutCell mesh based on the objects and material points selected.

The generic procedure for generating the CutCell mesh is as follows:

1. Define the objects.

    a. Right click on **Model** in the Outline View and select **Object Management...** to open the **Manage Objects** dialog box. See Using the Manage Objects Dialog Box (p. 609).

    b. Make sure the objects are appropriately defined (see Object Attributes (p. 601) for details).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. Create capping surfaces if required (see Patching Tools (p. 620) for details).

Important:

The objects defined include the corresponding edges that are used for capturing features during the CutCell mesh generation. If you are starting from an earlier setup, you may need to add the appropriate edge zones to the object before proceeding. Several options for creating the **Edge Zones** are described in Extract Edge Zones (p. 707).

2. Define the material points, if needed.

   a. Right click on **Model** in the Outline View and select **Material Points...** to open the **Material Points** dialog box.

   b. Make sure the material points are appropriately defined (see Creating Material Points (p. 615) for details).

3. Define the size functions as appropriate (see Defining Size Functions (p. 593) for details).

   a. Right click on **Model** in the Outline View and select **Functions** from the **Sizing** context menu to open the **Size Functions** dialog box.

   b. Make sure the size functions are appropriately defined.

4. Select the objects and material points to be used for the CutCell mesh generation in the **CutCell** dialog box.

   a. From the **Mesh** menu, select **Cutcell...** to open the **CutCell** dialog box.

   b. Select the appropriate objects in the **Objects** selection list.

   c. Select the appropriate material points in the **Material points** selection list.

   d. Enable **Keep Solids Cell Zones** in the **Options** group box, if required.

5. Click **Create** in the **CutCell** dialog box.

   The face zones are separated by cell neighbor and normals on face zones connected to the fluid cell zones are oriented into the fluid zone. A face zone group is created for the face zones of each fluid cell zone. Additionally, the defaults for post volume mesh prism generation will be set (see Generating Prisms for the CutCell Mesh (p. 821) for details).

6. Right click on **Model** in the Outline View and select **Prepare for Solve**.

   Operations such as deleting dead zones, deleting geometry objects, deleting edge zones, removing face/cell zone name prefixes and/or suffixes, deleting unused faces and nodes are performed during the cleanup operation.

7. Generate prism layers, if required.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. Click **Create Prisms...** to open the **Prisms** dialog box.

b. Examine the face zone group created for the face zones of the fluid cell zones and determine the face zones for which prism meshing parameters are to be specified.

c. Specify the prism meshing parameters as appropriate and click **Apply.**

d. Click **Create** in the **Prisms** dialog box.

8. Verify the quality of the CutCell mesh and perform quality improvement operations, if required.

Note:

Use the **File** → **Write** → **Case...** menu item with the **Write As Polyhedra** option enabled to write the case file in the format that can be read in solution mode in Fluent.

7.14.2.1. Handling Zero-Thickness Walls

Certain geometries may have components that have zero-thickness. Such configurations can be handled during the CutCell meshing process. There are two types of zero-thickness walls:

- Baffles (zero-thickness walls having the same fluid/solid zone on either side)
- Interior walls (zero-thickness walls having different fluid/solid zones on either side)

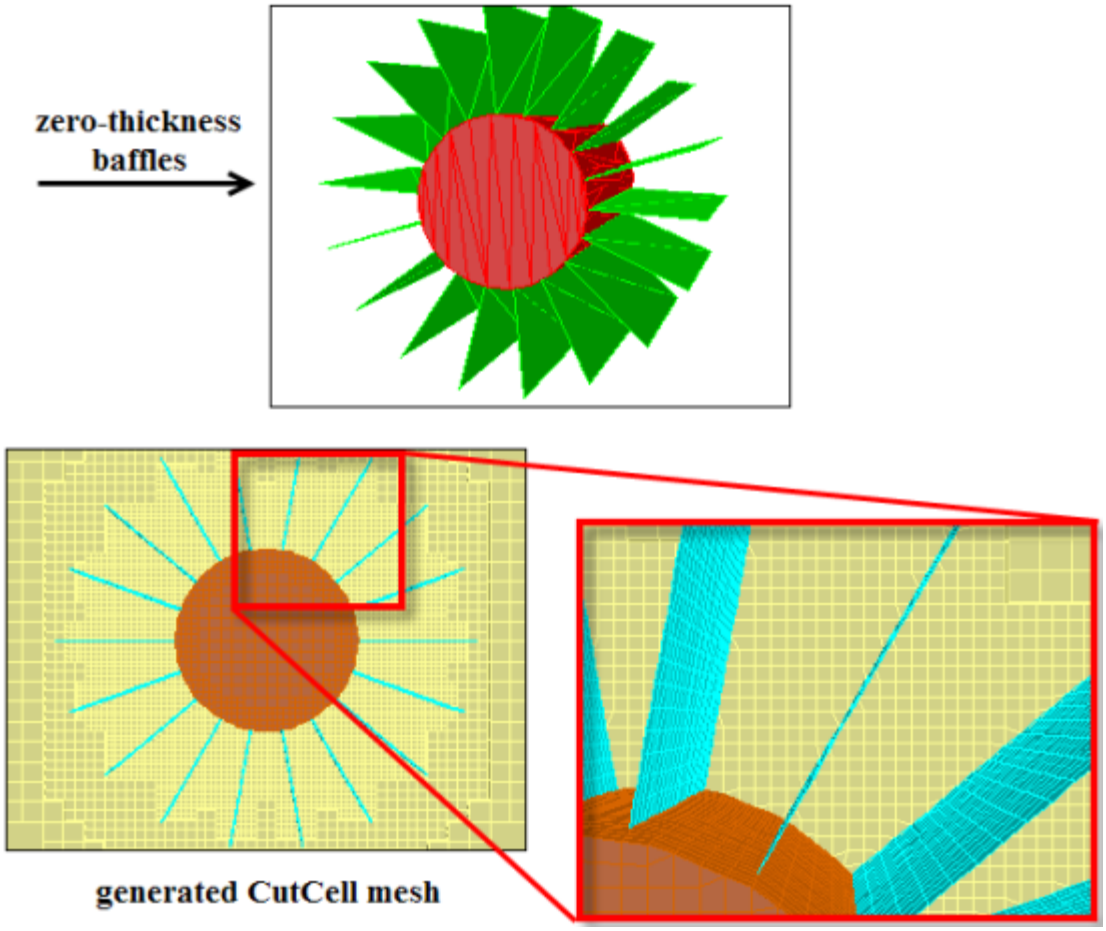
To allow the recovery of baffles, any such surface must be of the type **internal**, which will be recovered as a **wall**. For jump conditions, the surface must be of one of the types **fan**, **radiator**, or **porous-jump**, which will be recovered based on the type defined. All such surfaces should be included in the object defined. If not, the surface will not be recovered.

The recovered surface for the zero-thickness baffles will be prefixed by **cutcell-two-sided** in the generated CutCell mesh.

Figure 7.116: Mesh Generated for Geometry Having Zero-Thickness Baffles (p. 818) shows an example where the CutCell mesh has been generated for a stirrer geometry having zero-thickness baffles.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.116: Mesh Generated for Geometry Having Zero-Thickness Baffles



For interior walls, you can define two objects, each including the zones comprising the respective domains. Hence, the interior wall will appear in both objects defined, and will be recovered properly.

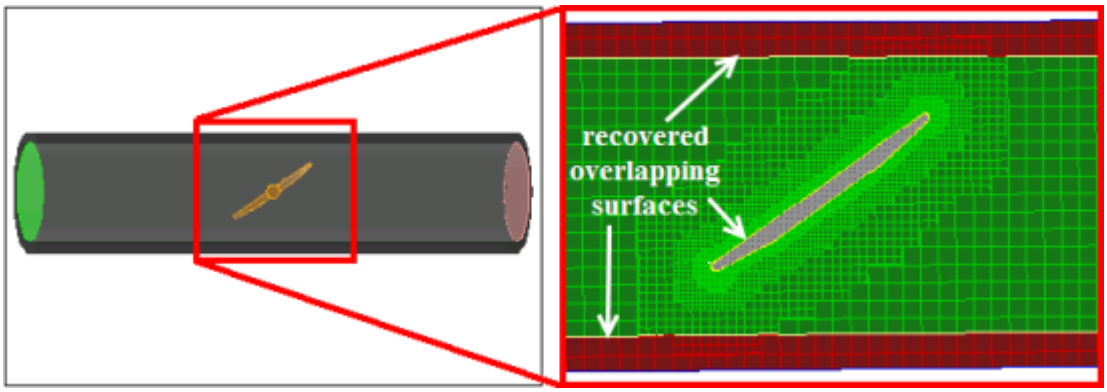
7.14.2.2. Handling Overlapping Surfaces

Overlapping surfaces are surfaces from independently defined objects which partially or fully overlap. To allow the recovery of overlapping surfaces as a separate surface, the overlapping walls must be included in both defined objects. The distance between such surfaces must be at least ten times smaller than the minimum size set to avoid “trapped" cell zones. The overlapping surface will be included with the recovered boundary from the object having a higher priority value.

Figure 7.117: Recovering Overlapping Surfaces (p. 819) shows an example where the CutCell mesh has been generated for a butterfly valve. The overlapping surfaces between the valve and flow region as well as the pipe walls and flow region are recovered.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.117: Recovering Overlapping Surfaces



7.14.2.3. Resolving Thin Regions

Surfaces in close proximity constitute thin regions in the mesh. Examples of thin regions include sharp corners, trailing edge configurations, and so on. Such configurations may not be recovered accurately enough by the CutCell mesher, and surface elements may span between nodes on the proximal surfaces.

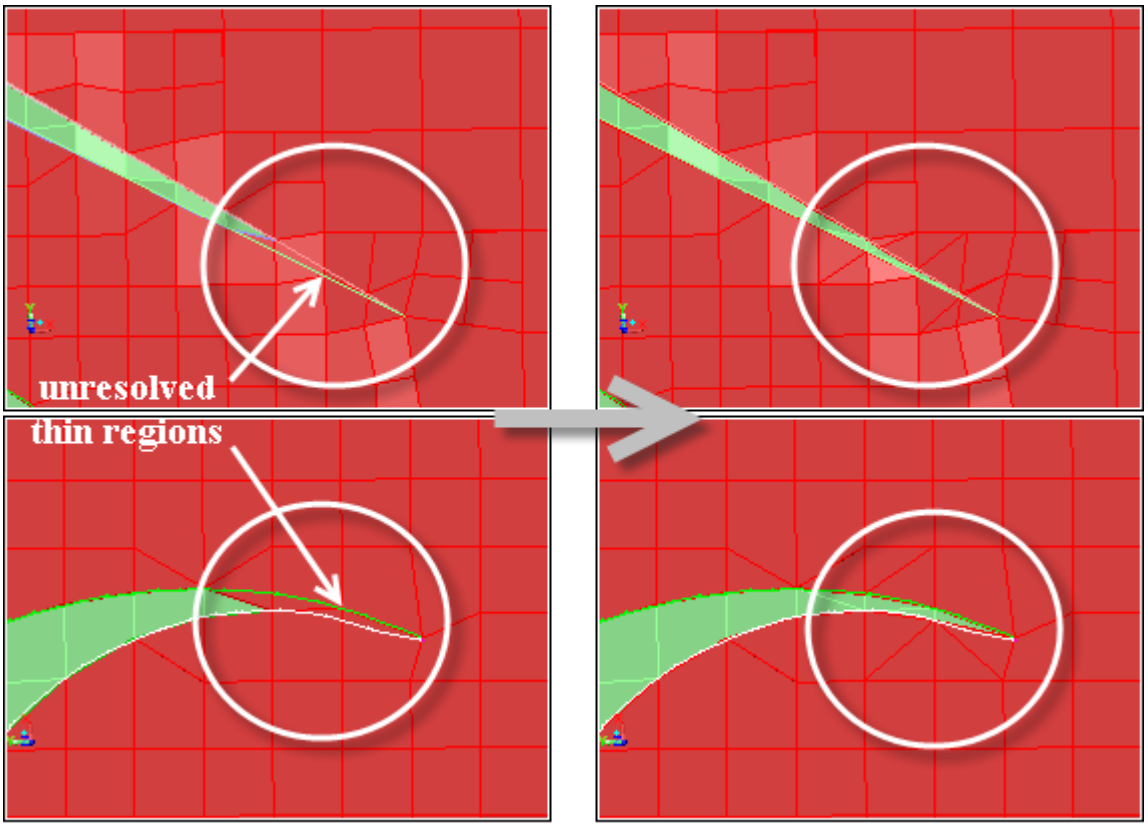
You can explicitly define thin regions that need to be resolved when the CutCell mesh is generated.

- The command `/mesh/cutcell/set/set-thin-cut-face-zones` enables you to specify the face zones constituting the thin regions to be recovered.
- The command `/mesh/cutcell/set/set-thin-cut-edge-zones` enables you to specify edge zones defining the features to be recovered in thin regions.

Figure 7.118: Resolving Thin Regions (p. 820) shows an example where the thin regions have been resolved during the CutCell meshing process.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.118: Resolving Thin Regions



7.14.3. Improving the CutCell Mesh

The **Mesh/Tools/Auto Node Move...** menu item opens the **Auto Node Move** dialog box, which you can use to improve the CutCell mesh quality.

The following text commands also enable you to improve the CutCell mesh:

`/mesh/cutcell/modify/cavity-remeshing`

enables you to use the **Cavity Remeshing** utility to improve the CutCell mesh quality near the boundary. Specify the cell zones to be improved and the quality limit as appropriate. For details on the options available, refer to Cavity Remeshing (p. 521).

Note:

Face zones of type **internal** are recovered as type **wall** in the cutcell mesher. These should be reset to type **internal** before using the cavity remesher.

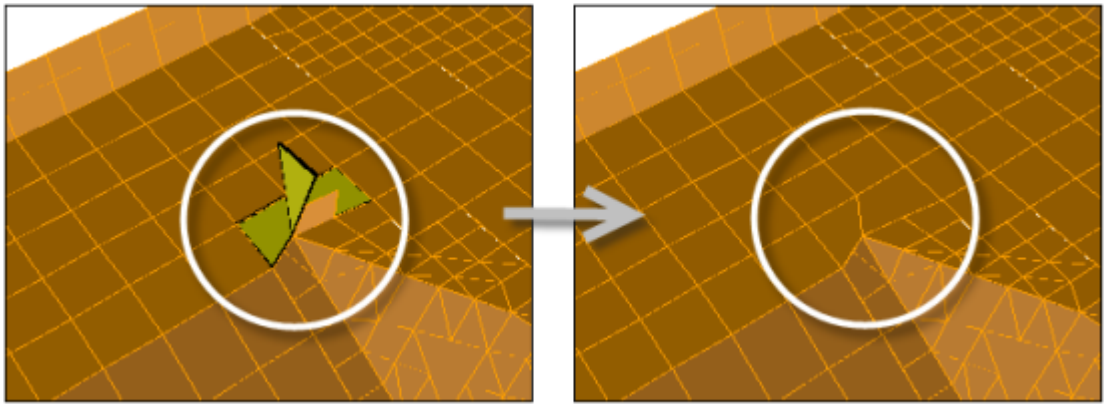
`/mesh/cutcell/modify/rezone-multi-connected-faces`

enables you to resolve multi-connected configurations on the CutCell boundary. Specify an appropriate value for the critical count for contiguous manifold faces.

An example is shown in Figure 7.119: Rezoning Multiply Connected Faces (p. 821) where the multiply connected faces around the surface are removed.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.119: Rezoning Multiply Connected Faces



7.14.4. Post CutCell Mesh Generation Cleanup

After generating the CutCell mesh, you can perform cleanup operations such as deleting dead zones, deleting geometry objects, deleting edge zones, removing face/cell zone name prefixes and/or suffixes, deleting unused faces and nodes. These operations can be performed by selecting **Prepare for Solve** in the context menu under **Model** in the Outline View.

Note:

It is recommended that you use the **Delete** option in the **Manage Cell Zones** dialog box or the command `/mesh/manage/delete` to delete cell zones in the CutCell mesh, instead of the **Mesh/Clear** option or the `/mesh/clear-mesh` command.

7.14.5. Generating Prisms for the CutCell Mesh

After generating the CutCell mesh, the face zones are separated by cell neighbor and normals on face zones connected to the fluid cell zones are oriented into the fluid zone. A face zone group is created for the face zones of each fluid cell zone. Additionally, the defaults for post volume mesh prism generation will be set. These include the following prism controls to reduce stair-stepping of prism layers:

- Disable edge swapping.

  ```
  /mesh/prism/controls/improve/edge-swap? no
  ```

- Disable face smoothing.

  ```
  /mesh/prism/controls/face-smooth? no
  ```

- Set the skewness threshold for edge swapping and edge and node smoothing to `0.95`.

  ```
  /mesh/prism/controls/improve/swap-smooth-skew 0.95
  ```

- Ensure that shrinkage for prism layers is enabled.

  ```
  /mesh/prism/controls/proximity/allow-shrinkage? yes
  ```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Set the smoothing rate (rate at which shrinkage is propagated laterally) to `1.2`.

  ```
  /mesh/prism/controls/proximity/smoothing-rate 1.2
  ```

- Disable ignoring of nodes that have poor normals.

  ```
  /mesh/prism/controls/normal/ignore-invalid-normals? no
  ```

- Set the cell quality criterion for smoothing and quality checking to `0.999`.

  ```
  /mesh/prism/controls/improve/max-allowable-cell-skew 0.999
  ```

- Enable smoothing of normals along the feature lines of the base face zones.

  ```
  /mesh/prism/controls/improve/identify-feature-line? yes
  ```

- Set the maximum allowable skewness for cap faces after smoothing to `0.999`.

  ```
  /mesh/prism/controls/improve/max-allowable-cap-skew 0.999
  ```

- Set the quality method to **Orthoskew**.

  ```
  /mesh/prism/quality-method orthoskew
  ```

- Enable the improvement of cell quality for every prism layer. This will involve smoothing of normals in the current layer and perturbation smoothing to improve cell quality in the lower layer.

  ```
  /mesh/prism/controls/improve/cell-quality-improve? yes
  ```

- Enable the adjustment of prism heights at prism cap corners to improve cell quality.

  ```
  /mesh/prism/controls/improve/corner-height-weight? yes
  ```

- Disable forcible smoothing of cells if cell quality remains bad after regular smoothing.

  ```
  /mesh/prism/improve/smooth-brute-force? no
  ```

You can generate prism layers on the appropriate face zones as follows:

1. Click **Create Prisms...** in the **CutCell** dialog box to open the **Prisms** dialog box.
2. Examine the face zone group created for the face zones of the fluid cell zones and determine the face zones for which prism meshing parameters are to be specified.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Specify the prism meshing parameters as appropriate and click **Apply.**

   Note:

   Hanging-node cells on a boundary for which prism generation has been assigned, will be triangulated before the prism generation starts.

   Important:

   Attempting to grow thicker prism layers in areas where the aspect ratio of the base to the prism cap is very large may result in an invalid mesh. In such cases, (for example, external flow problems) it is recommended that aspect ratio based growth be used to avoid problems with invalid meshes.

4. Click **Create** in the **Prisms** dialog box.

   As the volume mesh already exists, a **Question** dialog box will appear, asking if you want to morph the existing volume mesh. Click **Yes** to generate the prism layers.

   Alternatively, use the command `/mesh/cutcell/create-prism` to create the prism layers. Specify the cell zones into which the prism layers are to be grown. The gap factor controls the number of elements in regions of proximity.

   Note:

   If the cell aspect ratio exceeds the specified maximum aspect ratio, a message will appear during the prism meshing process, indicating that shrinkage was limited by the maximum aspect ratio specified.

   You could also reduce the gap factor to avoid the cell aspect ratio exceeding the specified maximum aspect ratio. Reducing the gap factor (for example, a value of 0.5) may improve the quality, but could have a negative impact on the robustness of the morphing. A higher value (for example, 1.5) is generally more robust, but may not result in the best mesh quality.

   Note:

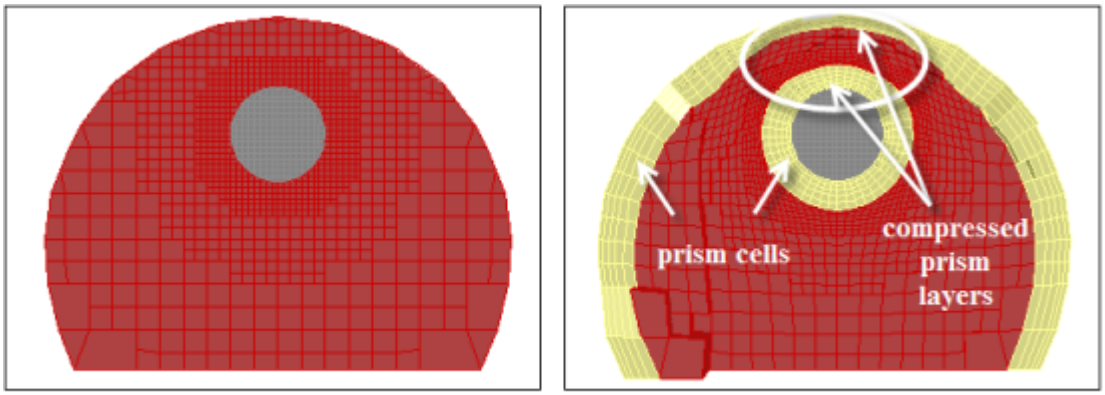
   During the surface mesh morphing, only the boundaries of face threads will be treated as features.

   When prism layers are grown into cell zones sharing a face, the prisms will be imprinted on the shared face.

   Figure 7.120: Generating Prisms for the CutCell Mesh (p. 824) shows the prism layers created for the CutCell mesh. The prism layers will be compressed in regions of proximity and bad normals. Note that local stair-stepping may occur in areas of poor quality.

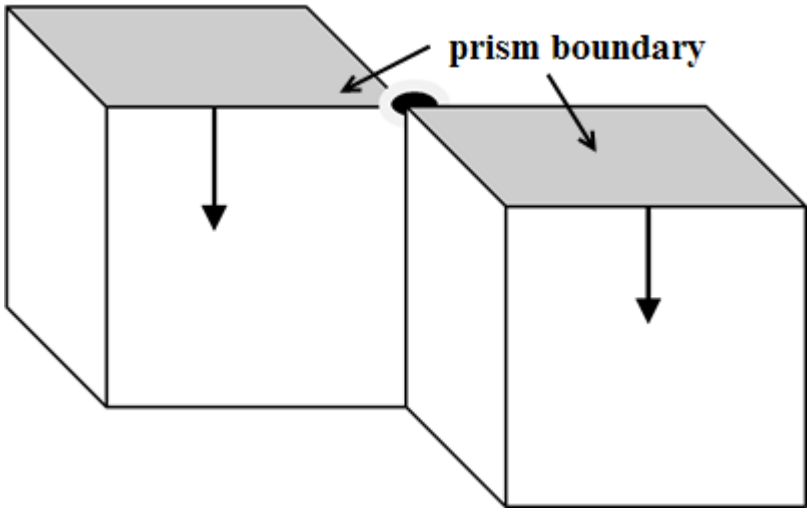
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.120: Generating Prisms for the CutCell Mesh



- When prism layers are grown into two volumes sharing an edge, stair-stepping will occur at the common vertex between the volumes (Figure 7.121: Prism Growth Limitations—Volumes Sharing an Edge (p. 824)).

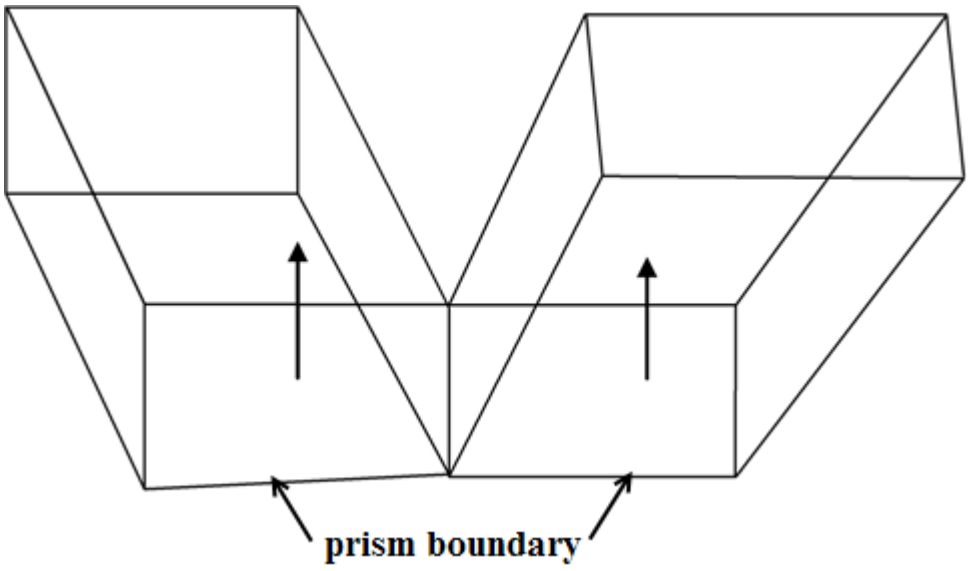
  Figure 7.121: Prism Growth Limitations—Volumes Sharing an Edge



- When prism layers are grown into two volumes sharing an edge, stair-stepping will occur at the nodes on the common boundary (Figure 7.122: Prism Growth Limitations—Volumes Sharing an Edge (p. 825)).

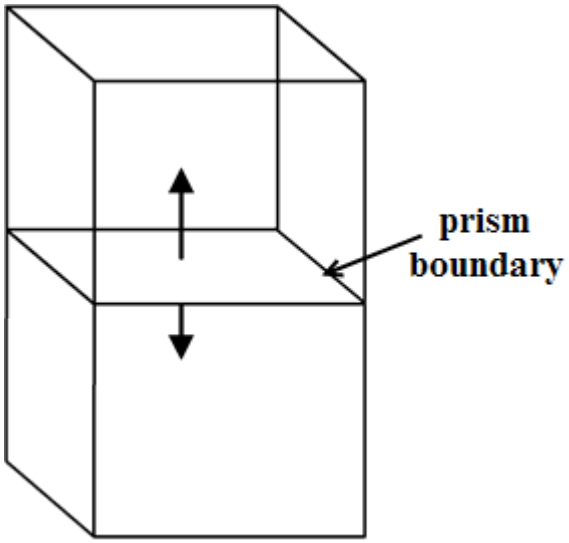
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.122: Prism Growth Limitations—Volumes Sharing an Edge



- Prism layers cannot be grown on both sides of a surface shared by adjacent volumes (Figure 7.123: Prism Growth Limitations—Volumes Sharing the Prism Base (p. 825)).

Figure 7.123: Prism Growth Limitations—Volumes Sharing the Prism Base



A combination of node movement and cavity remeshing is carried out to improve the CutCell mesh quality after the prism layers have been generated. The parameters for improving the CutCell mesh quality can be specified using the command `/mesh/cutcell/set/set-post-morph-parameters` prior to creating the prism layers. The quality method considered is that set by the `/mesh/cutcell/set/set-cutcell-quality-method` command.

5. Ensure that the quality of the prisms created is appropriate. If the quality of the prism cells is low, you can use post-prism smoothing to improve the quality. Use the options in the **Prism Improve** dialog box or text commands such as `/mesh/prism/improve/smooth-improve-prism-cells` to improve the prism cell quality.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. To further improve the CutCell mesh using the command `/mesh/cutcell/modify/post-morph-improve`, modify the relevant parameters using the command `/mesh/cut-cell/set/set-post-morph-parameters`. The quality considered is that set by the `/mesh/cutcell/set/set-cutcell-quality-method` command.

   Note:

   This operation uses a combination of node movement and cavity remeshing to improve the CutCell mesh quality. For details on the relevant parameters, refer to Moving Nodes (p. 519) and Cavity Remeshing (p. 521).

   Note:

   It is recommended that you use the `/mesh/cutcell/modify/post-morph-improve` command for cell zones other than the prism cells.

7.14.6. The Cut-Tet Workflow

The Cut-Tet workflow enables you to create a tetrahedral, hexcore, or prism mesh based on a triangulated and improved CutCell surface mesh. The initial requirement is the generated CutCell mesh.

The generic workflow is as follows:

1. Make the CutCell boundaries conformal to remove hanging-nodes.

   ```
   /mesh/cutcell/modify/split-boundary cutcell-* ,
   ```

   The command `/mesh/cutcell/modify/split-boundary` creates a copy of the specified CutCell boundary zones and makes the boundary mesh conformal at the hanging-nodes on the copied zones. The new zones will be named based on the original zone names prefixed by **split-**.

2. Clear the volume mesh.

   ```
   /mesh/clear-mesh
   ```

3. Triangulate the split CutCell boundary zones.

   ```
   /boundary/remesh/triangulate split-* , yes
   ```

   The split CutCell boundary zones will be replaced by the corresponding triangulated boundary zones.

4. Improve the boundary mesh by swapping edges based on a node degree value other than 6. The node degree is defined as the number of edges connected to the node.

   ```
   /boundary/improve/degree-swap
   ```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Use boundary smoothing operations to further improve the boundary mesh quality (see subsequent step).

Note:

Do not use the `/boundary/improve/swap` operation immediately after the `/boundary/improve/degree-swap` operation, as this will restore the original degree configurations.

5. Improve the triangulated boundary mesh further using any of the following options available:

   a. Use wrapper smoothing operations to improve the boundary mesh quality.

      i. Change the type of the triangulated boundary mesh to **wrapper**.

      ii. Use the wrapper post improve operations to improve the boundary mesh quality. Refer to Diagnostic Tools (p. 625) or Improving the Mesh Objects (p. 718)for detailed descriptions of the options available.

      Note:

      Make sure the original geometry is retained when the CutCell mesh is generated, this required for reprojection when using the post improve operations.

   b. Use operations like improving based on boundary mesh quality, smoothing, and swapping to improve the boundary mesh quality. Refer to Improving Boundary Surfaces (p. 668) for the detailed descriptions of the options available.

   c. Use surface remeshing with the size functions defined for the CutCell mesh.

      i. Extract edge zones from the triangulated boundary mesh.

      This enables you to maintain the boundary node locations on the remeshed faces and facilitates the connection between the remeshed faces using simple node merge operations.

      Split the edges having sizes bigger than the size function defined using the Scheme command `(ti-refine-edge-threads-by-sf (get-edge-zones-of-filter 'split-*))`

      ii. Remesh the triangulated boundary mesh using the defined size functions using the Scheme command `(ti-remesh-multiple-threads (get-face-zones-of-filter 'split-*) #f #t "none" #t)`

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This enables you to remesh multiple surfaces in a single operation.

Tip:

You can also add a meshed size function on the edge zones extracted from the triangulated boundary mesh to improve the quality.

iii. Merge duplicate boundary nodes on the remeshed boundary zones.

6. Generate the tetrahedral/hexcore/prism mesh. Refer to Generating Tetrahedral Meshes (p. 780), Generating the Hexcore Mesh (p. 795), and Generating Prisms (p. 750) for the detailed descriptions of the meshing options available. You can also use the **Auto Mesh** option (see Using the Auto Mesh Dialog Box (p. 731)) for generating the mesh.

   Note:

   To generate the tetrahedral mesh using the defined size functions, use `(tgsetvar! 'impose/cell-size-method 4)`. This will also respect the body of influence size functions defined for the CutCell meshing.

   You can also use additional operations during the volume meshing process, as appropriate. For example, `/mesh/manage/merge-dead-zones` can be used to merge dead zones having a cell count lower than the specified threshold value, with the adjacent cell zone.

7. Improve the mesh quality using the various improvement options available. Refer to Improving the Mesh (p. 511) for detailed descriptions of the options available.

7.15. Generating Rapid Octree Meshes

The trend in modern CFD goes towards large-scale, unsteady simulations. For Scale Resolving Simulations (SRS, for example, LES) in industrial applications, meshes with billions of cells are required, and simulations are run on thousands of CPUs. To facilitate such simulations, massive parallel mesh generation is needed as well, where meshing must be robust with limited user interaction. Furthermore, SRS requires isotropic cells in order to resolve transported eddies, both in the bulk and also close to the wall.

The Rapid Octree mesher is based on massively parallel algorithms and data structures. It allows you to use an existing boundary geometry to generate a new volume mesh that is based on cube cells, such that the resulting meshes are mostly isotropic. At boundaries, the geometry is approximated by snapping or projecting vertices onto the surface if possible, but defeatured if under-resolved by the prescribed mesh size. Rather than capturing every single geometric feature, the paradigm is to maintain robust cell quality to enable stable simulations.

A typical target application for the Rapid Octree mesher is external aerodynamics for which a faceted geometry of the object and the dimensions of the outer bounding box (for example, a wind tunnel) are provided. Meshes for internal flows can be easily generated as well.

The following sections are described in this chapter:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.15.1. Using the Rapid Octree Mesher
7.15.2. Limitations of the Rapid Octree Mesher

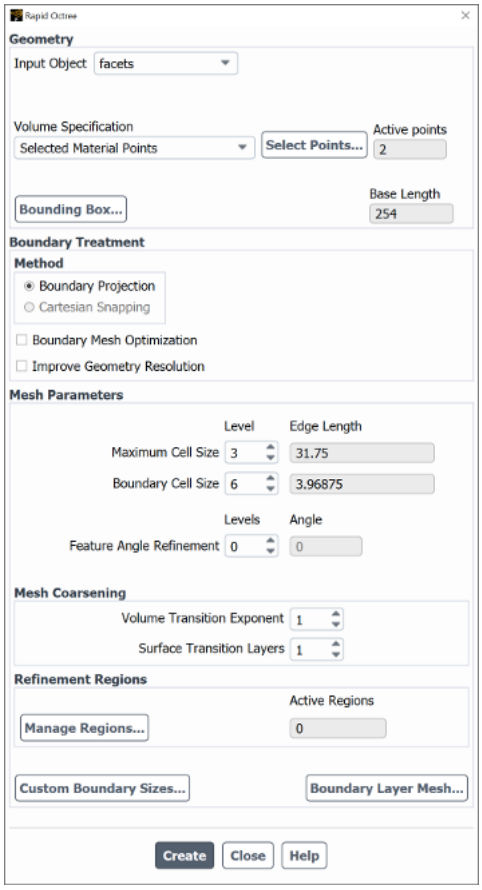
7.15.1. Using the Rapid Octree Mesher

To begin, you must read in a boundary geometry that provides a triangulated surface; this can be an enclosed surface mesh or a volume mesh. (Note that this original mesh data will be cleared during this process.) Then you can access the Rapid Octree meshing algorithm through the **Mesh** menu:

Mesh → Rapid Octree...

This opens the **Rapid Octree** dialog box.

Figure 7.124: The Rapid Octree Dialog Box



To generate the octree mesh, you must define settings in the three top-level group boxes (which are described in the following sections), and click the **Create** button at the bottom of the **Rapid Octree** dialog box. If the resulting mesh is not satisfactory, you can read the mesh again or (if the mesh was generated from a geometry object or mesh object) you can restore the object state (including its surfaces) as it was prior to the meshing operation by entering the following text command: `/mesh/rapid-octree/undo-last-meshing-operation`; then you can regenerate the mesh with revised settings.

7.15.1.1. Geometry
7.15.1.2. Boundary Treatment

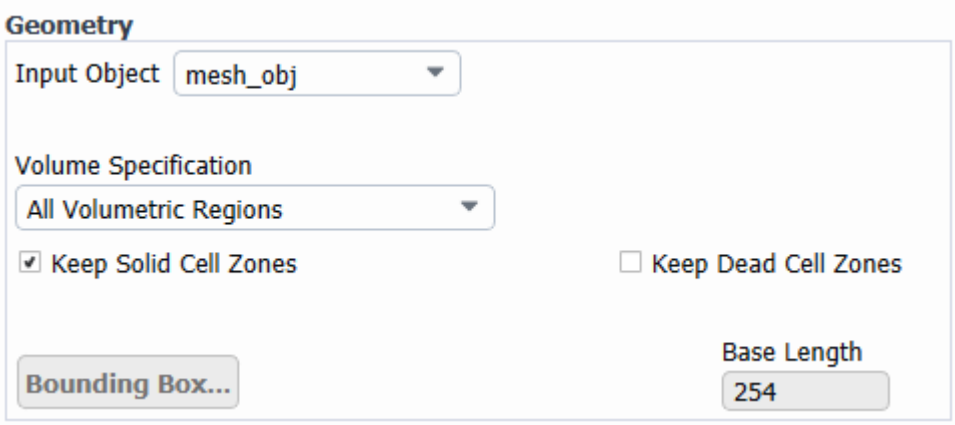
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.15.1.3. Mesh Parameters

7.15.1.1. Geometry

The **Geometry** group box allows you to specify the input object, specify the volume to be meshed, define the bounding box for the geometry, and display the base length (which is the main size for all subsequent mesh size specifications).

Figure 7.125: The Geometry Group Box



See the following for details:

7.15.1.1.1. Specifying the Input Object

7.15.1.1.2. Specifying the Volume

7.15.1.1.3. Defining the Bounding Box

7.15.1.1.4. Reporting the Base Length

7.15.1.1.1. Specifying the Input Object

You can make a selection from the **Input Object** drop-down list to specify that the geometry is a particular geometry object or a mesh object; alternatively, you can select **none** so that all available surface zones are used. Note that when a geometric object is selected and meshed, a corresponding mesh object is created.

7.15.1.1.2. Specifying the Volume

You must indicate the region to be meshed by making a selection from the **Volume Specification** drop-down list. These selections will result in the creation of a single volume / cell zone, unless otherwise noted.

Note:

If you selected a mesh object for the **Input Object** and want to use volumetric regions as part of the **Volume Specification**, you must compute the volumetric regions, as described in Volumetric Region Management (p. 643).

The following selections are available from the **Volume Specification** drop-down list:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Select **Bound by Geometry** for internal flows, so that the volume inside the geometry is meshed.

- Select **Bound by Bounding Box** for external flows, so that the volume between the bounding box and the geometry is meshed. Note that the face zones that are created for the sides of this bounding box are named **tunnel-**<*dom_ext*>, where <*dom_ext*> represents the domain extent (**tunnel-xmax**, **tunnel-xmin**, **tunnel-ymax**, and so on).

- Select **Filled from Point** if you want to define the coordinates of a point (in the **X**, **Y**, and **Z** fields) located in the region to be meshed. The point can be either within the geometry for internal flows, or between the bounding box and the geometry for external flows.

- Select **Selected Material Points** if you want to generate disconnected cell zones (based on the existing volumetric regions) by defining and selecting material points within the various regions. To connect the resulting cell zones, you will need to create non-conformal mesh interfaces in the solution mode (as described in Using a Non-Conformal Mesh in Ansys Fluent (p. 1185)). Note that the **Selected Material Points** selection does not require a watertight geometry. It is available regardless of what you selected from the **Input Object** drop-down list:

  – If you selected a geometry object or a mesh object for the **Input Object**, a mesh object will be created for each zone, since the zones will not be conformally connected.

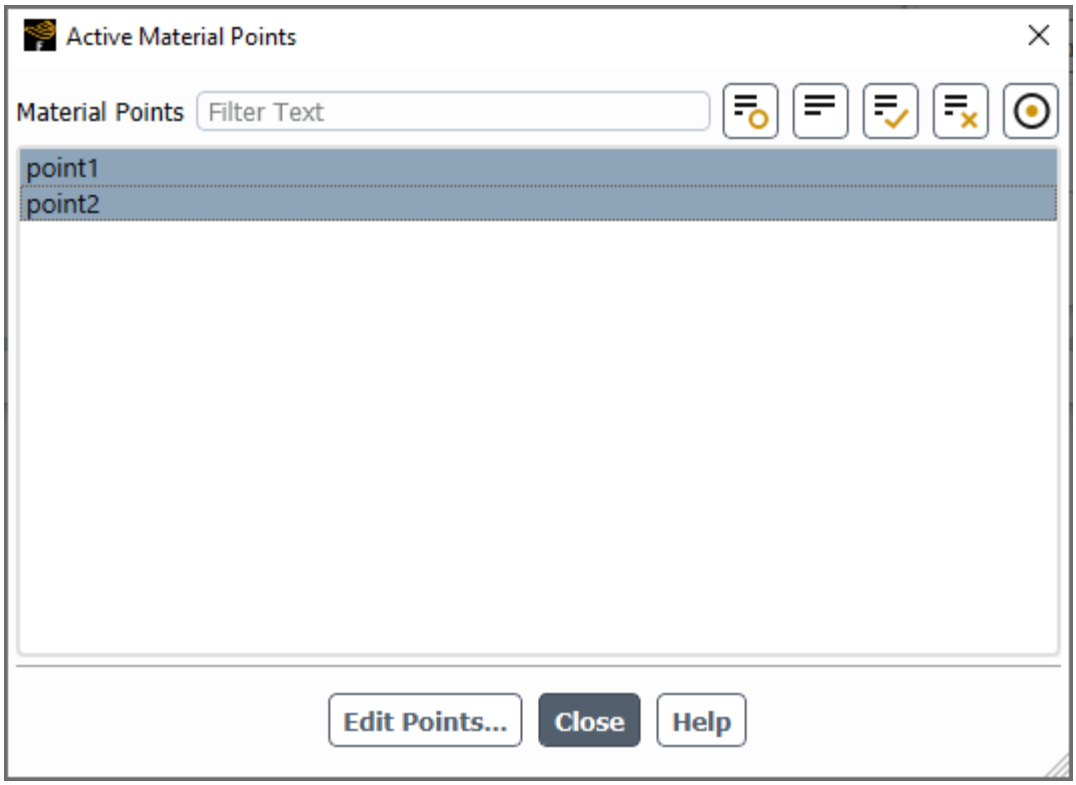
  – If you selected **none** for the **Input Object**, separate mesh objects will not be created for each zone; instead, all generated cell and face zones will be stored as unreferenced. All other entities (objects, edges, cell zones, and so on) in the session will be deleted in the process.

  – If the regions are separated by face zones of type interface, each face zone will be split into two zones that have **-nci:<ID>** appended to the original name.

  To complete the setup for the **Selected Material Points** selection, you must click the **Select Points...** button and select material points in the **Active Material Points** dialog box that opens. These material points should be located in the separate regions you want to mesh. If you have not already created any material points, you can do so by clicking the **Edit Points...** button in the **Active Material Points** dialog box, and then using the **Material Points** dialog box that opens (as described in Creating Material Points (p. 615)). As you select material points, the total number is updated in the **Active points** field in the **Rapid Octree** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.126: Selecting Active Material Points



Tip:

– It is recommended that you use one level of angular refinement for interfaces with corners, to promote better matching between the resulting cell zones.

– If the geometry is not watertight, there can be cases in which the defined material points are not separated completely by the geometry. This can result in the creation of fewer cell zones than you intended. You can consider closing the geometry by creating patches (for example, using SpaceClaim). Such patches do not need to be stitched to the rest of the geometry.

– You may want to enable the **Improve Geometry Resolution** option along with the **Selected Material Points** selection, as it can improve the interface matching between the cell zones. For further details, see Improve Geometry Resolution (p. 838).

- If you have selected a mesh object from the **Input Object** drop-down list, you can select the name of a particular volumetric region. The **X**, **Y**, and **Z** coordinates provide information about the location of the point that is used to generate the mesh.
- Select **All Volumetric Regions** if you want to generate disconnected cell zones based on all of the existing volumetric regions. Note that this selection should only be used if you have a watertight geometry. It is available if you have selected a mesh object from the **Input Object** drop-down list. A mesh object will be created for each cell zone, since the zones will not be conformally connected. If mesh objects are separated by face zones of type interface, each

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

face zone will be split into two zones that have **-nci:<ID>** appended to the original name; these zones can be used to create a non-conformal mesh interface in the solution mode (as described in Using a Non-Conformal Mesh in Ansys Fluent (p. 1185)).

With this selection, you can enable options in order to **Keep Solid Cell Zones** and/or **Keep Dead Cell Zones** in the resulting mesh.

Tip:

It is recommended that you use one level of angular refinement for interfaces with corners, to promote better matching between the zones.

Tip:

Regardless of what you select from the **Volume Specification** drop-down list, if the meshing fails, you can try to change the settings so that the coordinates of the point used to generate the mesh is further away from any given boundary than the cell size defined for that boundary. If you can redefine the coordinates of the point, you can increase the distance from the boundaries; otherwise, you can decrease the cell size on the nearest boundaries. If the latter change is still unsuccessful or undesirable, you can instead use the **Selected Material Points** method described previously.

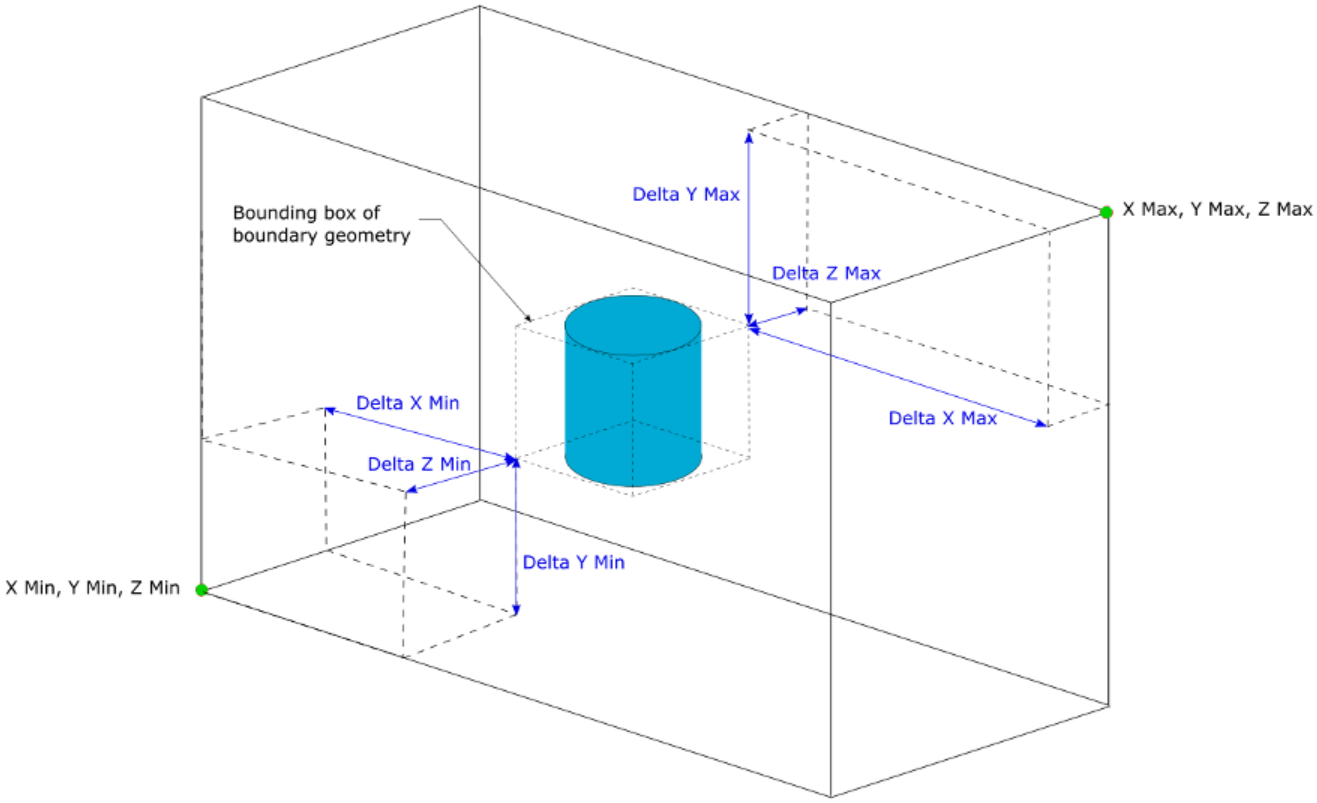
7.15.1.1.3. Defining the Bounding Box

A bounding box is used to define the Rapid Octree meshing region. You can define the box by specifying the absolute extents (min / max x, min / max y, min / max z), or by specifying the offset relative to a box around one or more face zones (delta min / max x, delta min / max y, delta min / max z). The input geometry you want to mesh must be within this bounding box. For external flows, you will size the box to represent the extents of your flow domain (for example, the dimensions of the wind tunnel), with your geometry located appropriately; for internal flows, you will ensure that the box merely encompasses the geometry.

Note:

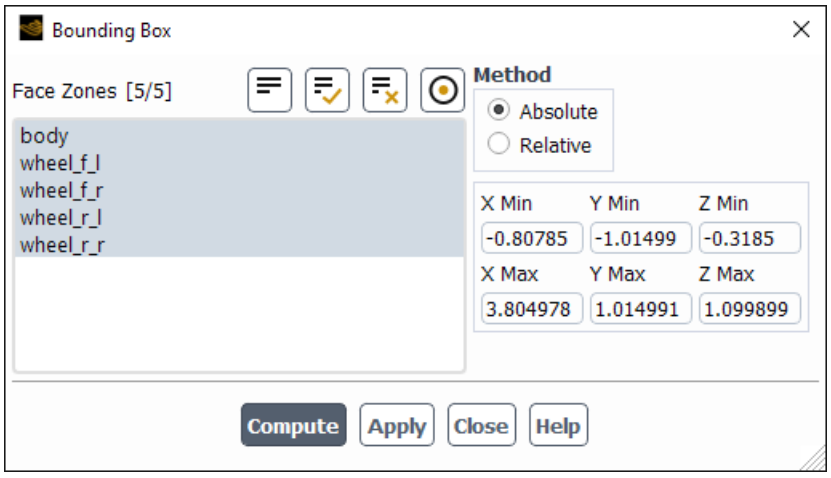
The bounding box is not available when you select **All Volumetric Regions** or a particular volumetric region from the **Volume Specification** drop-down list, as the meshing region is automatically defined and does not require your input.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In the **Rapid Octree** dialog box, click the **Bounding Box...** button to open the **Bounding Box** dialog box.

Figure 7.127: Bounding Box Dialog Box



Perform the following steps:

1. Select the **Face Zones** you want within the bounding box.
2. Select the **Method** by which you will define the box:
   - **Absolute**: after selecting this method, click **Compute** to calculate the extents of a bounding box based on the selected face zones and display the relevant coordinates as a minimum

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

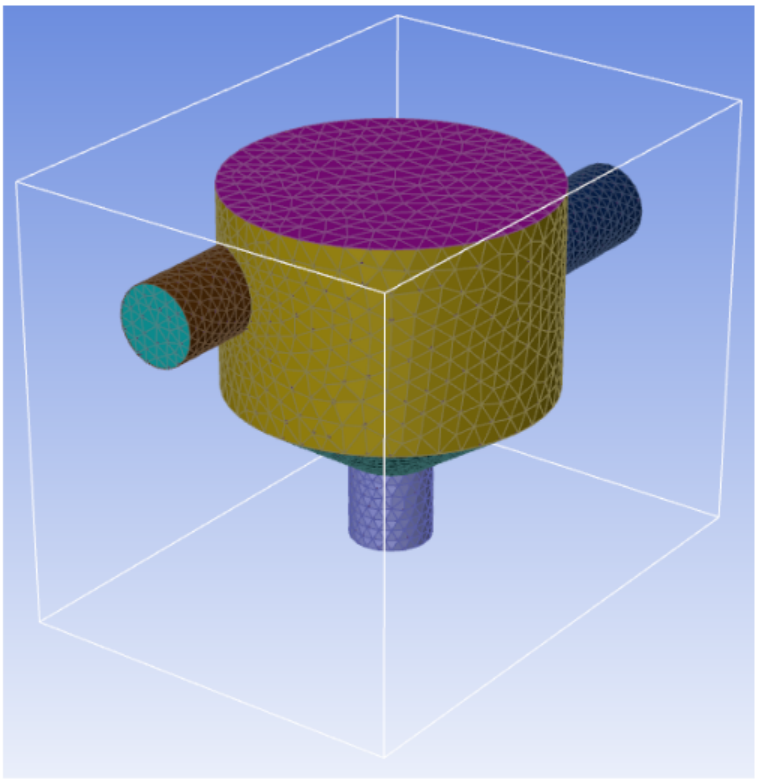
(**X Min**, **Y Min**, **Z Min**) and maximum (**X Max**, **Y Max**, **Z Max**). You can then adjust these values to make the box larger or smaller.

- **Relative**: after selecting this method, adjust the minimum offset (**Delta X Min**, **Delta Y Min**, **Delta Z Min**) and the maximum offset (**Delta X Max**, **Delta Y Max**, **Delta Z Max**) from the selected face zones.

3. Click **Apply** to create the bounding box and display it with the selected face zones in the graphics window.

4. Click **Close** to return to the **Rapid Octree** dialog box.

Figure 7.128: Bounding Box Surrounding Geometry



7.15.1.1.4. Reporting the Base Length

The **Base Length** is a non-editable field that reports the maximum extent of the bounding box and serves as the base length scale in the mesh generation process. All subsequent size specifications are based on it. The level specifications correspond to a division of the base size by the power of two to the given level.

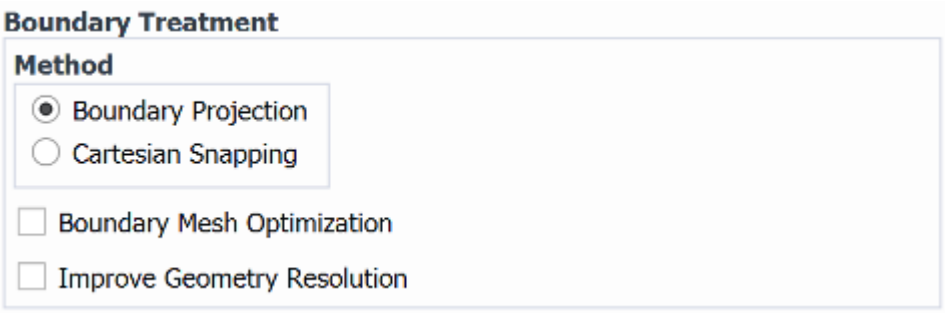
7.15.1.2. Boundary Treatment

In the **Boundary Treatment** group box, the **Method** list allows you to choose between the **Boundary Projection** and **Cartesian Snapping** methods. **Cartesian Snapping** can only handle a constant size boundary mesh, so there are no other related controls in the dialog box; it is not available if you have selected **All Volumetric Regions** from the **Volume Specification** drop-down list. For **Boundary Projection**, it is possible to refine the surface mesh according to feature angles

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

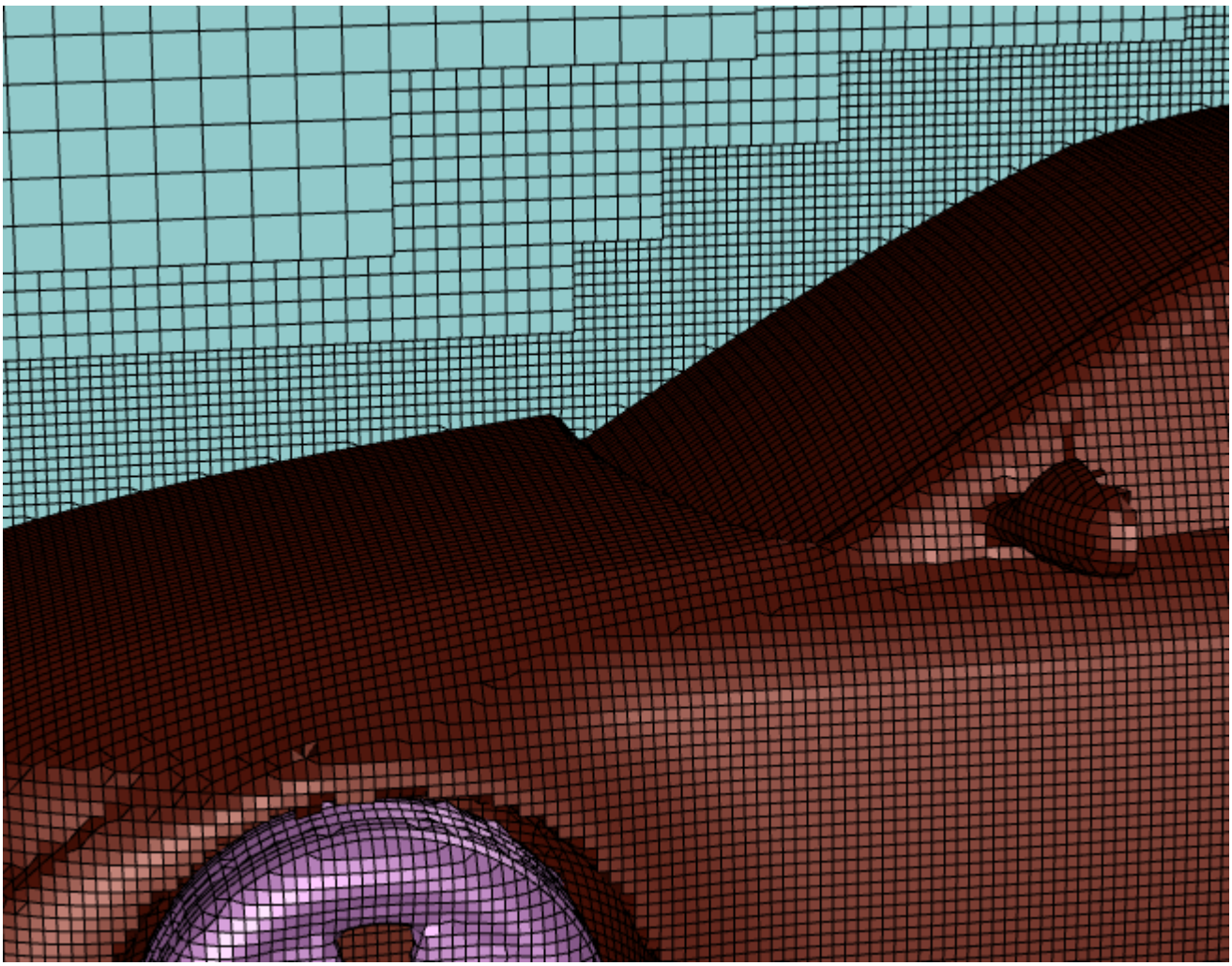
and/or local size by using the **Custom Boundary Sizes...** button, as described in Mesh Parameters (p. 840).

Figure 7.129: The Boundary Treatment Group Box



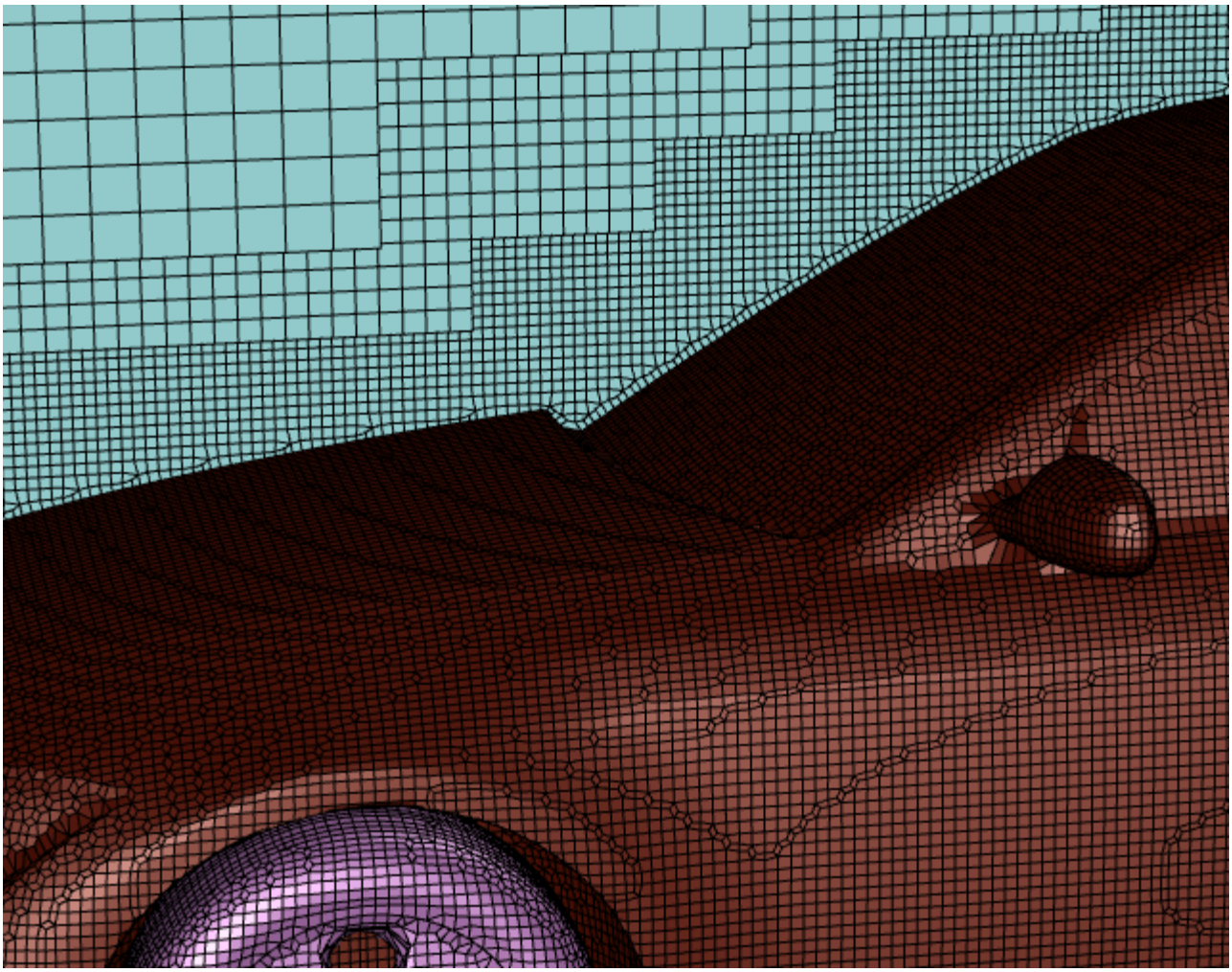
Neither approach will capture feature lines in the model. This defeaturing will always be in the length scale of the boundary mesh size. The **Cartesian Snapping** approach is slightly faster than the **Boundary Projection** approach, but is not as good at representing thin structures.

Figure 7.130: Mesh Generated by Cartesian Snapping



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.131: Mesh Generated by Boundary Projection



7.15.1.2.1. Boundary Mesh Optimization

When using the **Boundary Projection** treatment method, you can enable the **Boundary Mesh Optimization** option in order to specify that the cells in the layer adjacent to boundary zones are optimized in terms of orthogonal quality at the time of meshing. When applied along with multiple prismatic layers (see Boundary Layer Mesh (p. 845)), note that the optimization is performed on the cells prior to their being split into the prismatic layers.

Important:

This boundary mesh optimization is a very expensive process and will increase the mesh generation time significantly. It is not generally recommended, but may be necessary in some cases (for example, complex cases with multiple prismatic layers).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.132: The Boundary Treatment Group Box

7.15.1.2.2. Improve Geometry Resolution

A known limitation of the Rapid Octree mesher is that defeaturing can occur at inlets, outlets, interior boundaries, and so on, such that the surface area of a generated boundary does not exactly match that of the original geometry. You can attempt to minimize such defeaturing by assigning boundary types, refining the mesh, and/or defining boundary conditions (as described in Limitations of the Rapid Octree Mesher (p. 850)). If such attempts are not sufficient, as part of the **Boundary Projection** treatment method you can enable the **Improve Geometry Resolution** option. This option applies an iterative projection of boundary faces onto the geometry, with the goal of minimizing the distance of the boundary face centroid to the geometry. This "face projection" is then internally iterated with mesh smoothing steps to improve the cell quality.

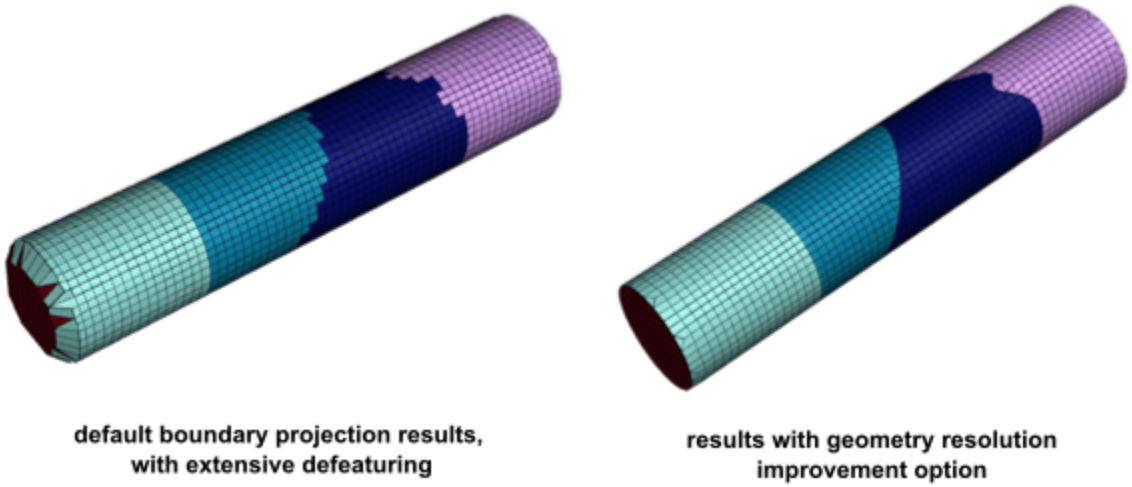
Note:

The **Improve Geometry Resolution** option is an expensive process and will increase the mesh generation time significantly (for example, by a factor of 4).

Figure 7.133: The Boundary Treatment Group Box

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.134: Improving the Geometry Resolution



Note:

- Although the capturing of corners and face zone boundaries is improved significantly by using the **Improve Geometry Resolution** option, there can still be defeaturing. This can especially affect the capturing of face zone boundaries, most noticeably when the adjacent cells have nonuniform sizes. It is recommended that you check the mesh for such defeaturing, in order to avoid errors due to inaccurate boundary conditions.

- For complex geometries, the **Improve Geometry Resolution** option can lead to significant deterioration of cell quality. It is therefore augmented with an automatic mesh repair step at the end of the mesh generation. Since this step can significantly increase the mesh generation time when there are large numbers of bad cells, it is only executed if the number of bad cells is below a threshold of 500. If this threshold is exceeded, you will receive a warning in the console, and you can try to address the issue by performing the following:

  – You can manually initiate a mesh repair. It is recommended that you first change the quality method to `inverse-ortho-quality` through the `report/quality-method` text command; then you can perform the mesh repair on selected cells zones by using the `mesh/modify/auto-node-move` text command. For further details, see `mesh/` in the *Fluent Text Command List*.

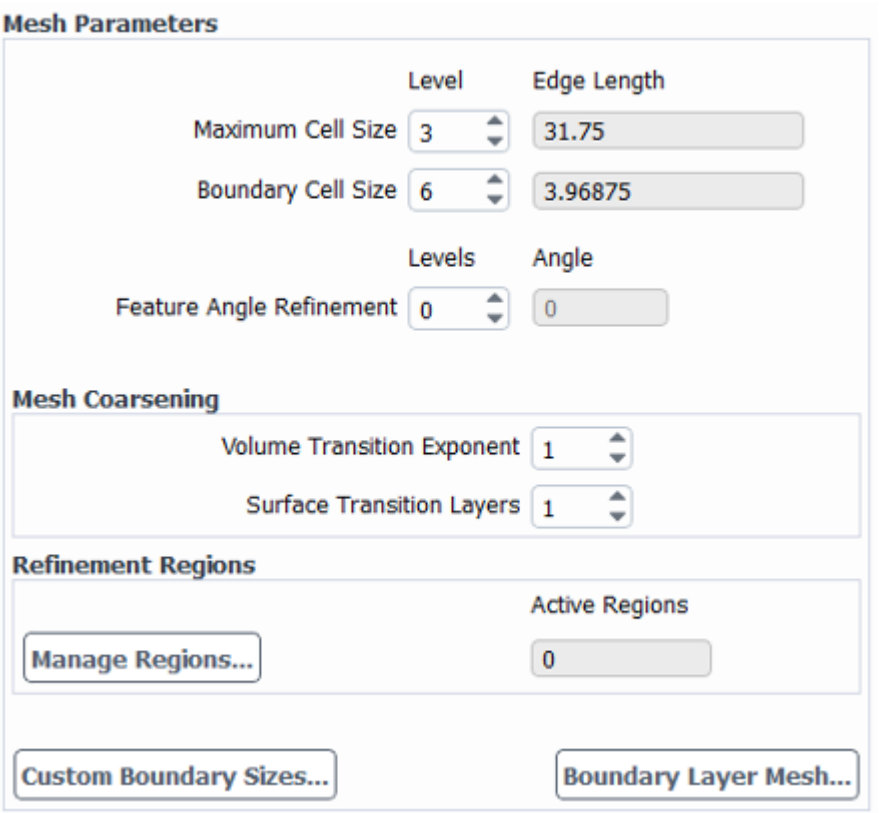
  – You can delete the bad cells by using the following text command for specified cell zones: `mesh/rapid-octree/delete-poor-quality-cells`. When a cell is deleted, new face zones of type symmetry are created for the adjacent cells on the faces that were shared with the deleted cell. This text command should be used with care in order to avoid unwanted effects during the solution (such as solution errors due to bad geometry representation, or even solver divergence).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.15.1.3. Mesh Parameters

The **Mesh Parameters** group box allows you to define global mesh sizes by adjusting the **Level** for the following: the **Maximum Cell Size** specifies a maximum size for the mesh; and the **Boundary Cell Size** specifies the sizing on the geometry. Based on the values you enter, the **Edge Length** values will be updated for your review. Note that due to the nature of the octree method, only certain cell size values can be represented by the mesh; the available sizes are obtained by dividing the base size by the power of two to the given number of levels.

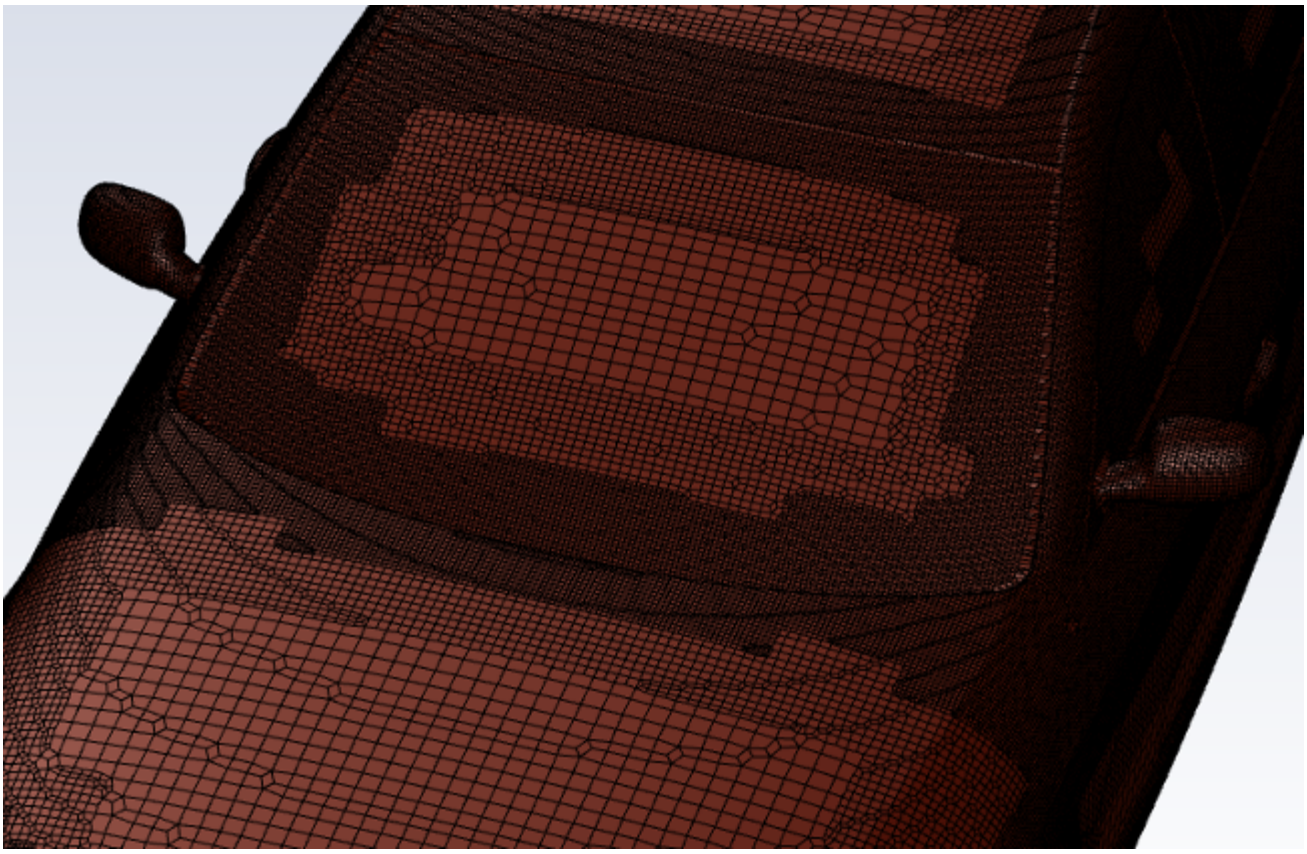
Figure 7.135: The Mesh Parameters Group Box



When using the **Boundary Projection** treatment method, you can define the **Feature Angle Refinement** fields as a global setting, so that the surface cell size is refined automatically wherever faces meet at an angle that is equal to or less than the specified **Angle**. At those locations, the boundary mesh is refined according to the given number of refinement **Levels**. Note that the **Levels** value is not absolute (like the **Boundary Cell Size** value), but is relative to the **Max Size** in the **Surface Sizing** dialog box. The **Surface Sizing** dialog box also allows you to set a unique local **Angular Refinement** for individual or groups of zones. For further details, see Custom Boundary Sizes (p. 844)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

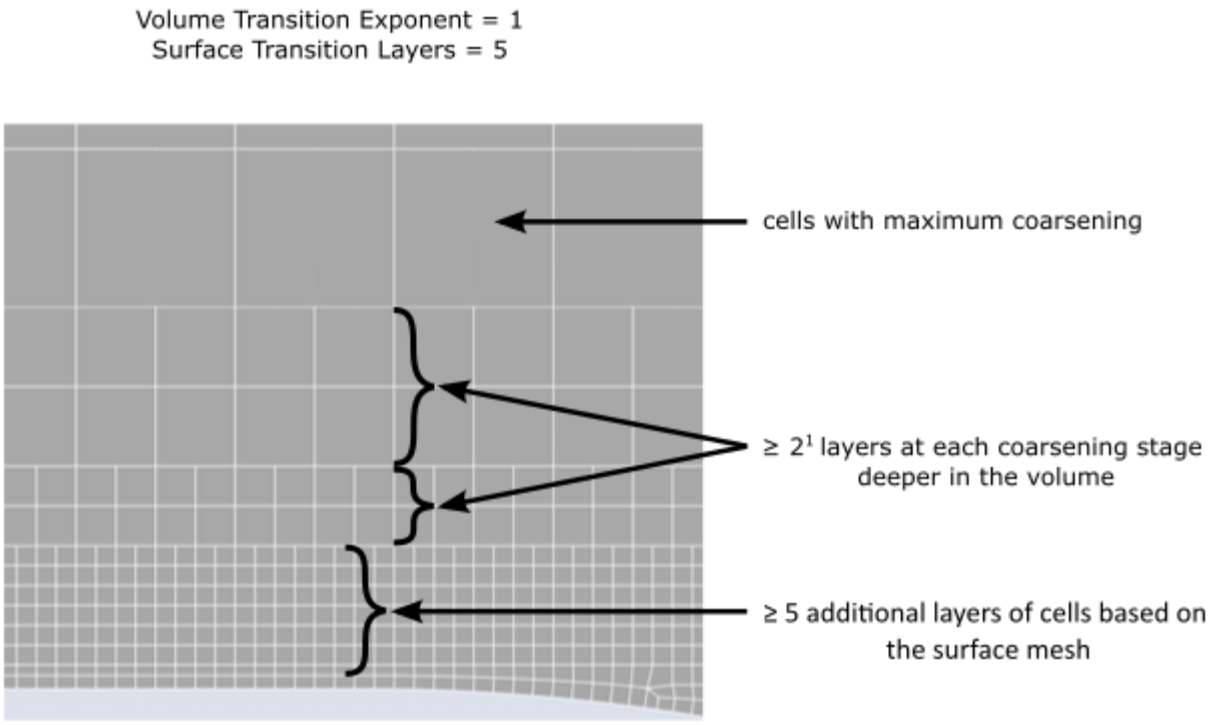
Figure 7.136: Surface Mesh with Two Levels of Feature Angle Refinement and 10° Threshold



The **Mesh Coarsening** group box provides fields that control how the mesh coarsens as it proceeds from the geometry to deeper within the fluid volume. You can adjust the **Volume Transition Exponent** to control the transition from the fine cells near the surfaces to the coarse cells in the volume: enter 0 for the fastest possible transition, or higher numbers for a slower transition; as the value increases, so too will the thickness of cell layers of the same size at each stage of the transition. The number of layers at each intermediate stage of coarsening will be at least 2 to the power of the number you enter here. You can also set the **Surface Transition Layers** to define the minimum number of layers of constant-size cells that are added to the layer adjacent to the surfaces of the geometry; these cells are generally isotropic, with the dimensions determined by the surface mesh.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.137: Mesh Coarsening Controls



Finally, you can use the buttons at the bottom of the **Mesh Parameters** group box to set the following:

7.15.1.3.1. Refinement Regions

7.15.1.3.2. Custom Boundary Sizes

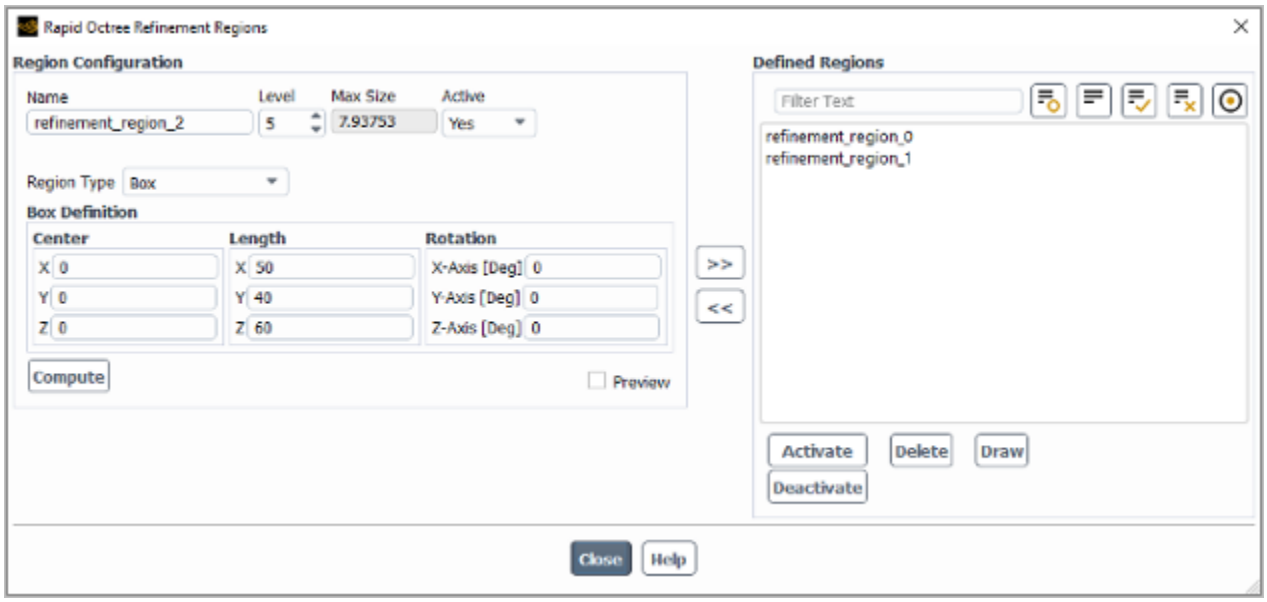
7.15.1.3.3. Boundary Layer Mesh

7.15.1.3.1. Refinement Regions

To refine specific regions of the mesh, click the **Manage Regions...** button in the **Refinement Regions** group box to open the **Rapid Octree Refinement Regions** dialog box. There you can define the refinement regions and manage them.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.138: The Rapid Octree Refinement Regions Dialog Box



To begin creating a region, you can provide a **Name** for the definition. Then adjust the **Level**, to produce your desired **Max Size** for the cells in this region.

Next, select the **Region Type** and define it's extents and orientation. You can choose from the following:

- **Box**

  For a box, you define the **Center** coordinates and the **Length** in each of the coordinate axes. You can also define **Rotation** values about each of the coordinate axes, using the right-hand rule; note that the **X-Axis** rotation is applied first, then the **Y-Axis** rotation, and finally the **Z-Axis** rotation.

- **Frustum**

  This shape is a frustum of a cone, that is, a tapered cylinder. You define the coordinates of the centers of the circular faces at the ends (**Center 0** and **Center 1**), as well as the **Radii** of the frustum at these points (**Radius 0** and **Radius 1**, respectively).

Tip:

- To get good initial values for your box or frustum definition, you can click on a face zone (or multiple face zones, if you hold the **Ctrl** key) in the graphics display, and then click the **Compute** button so that the region is sized to encompass the selected zone(s).

- Enable the **Preview** option to automatically display your region in the graphics display. The shape will be translucent and gray, with a red cube in the center that represents the **Max Size** of the cells.

Note that when clicking on face zones and/or previewing the refinement region, you may need to adjust what face zones are displayed by using the **Display Grid** dialog

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

box (as described in Generating the Mesh Display Using the Display Grid Dialog Box (p. 537)).

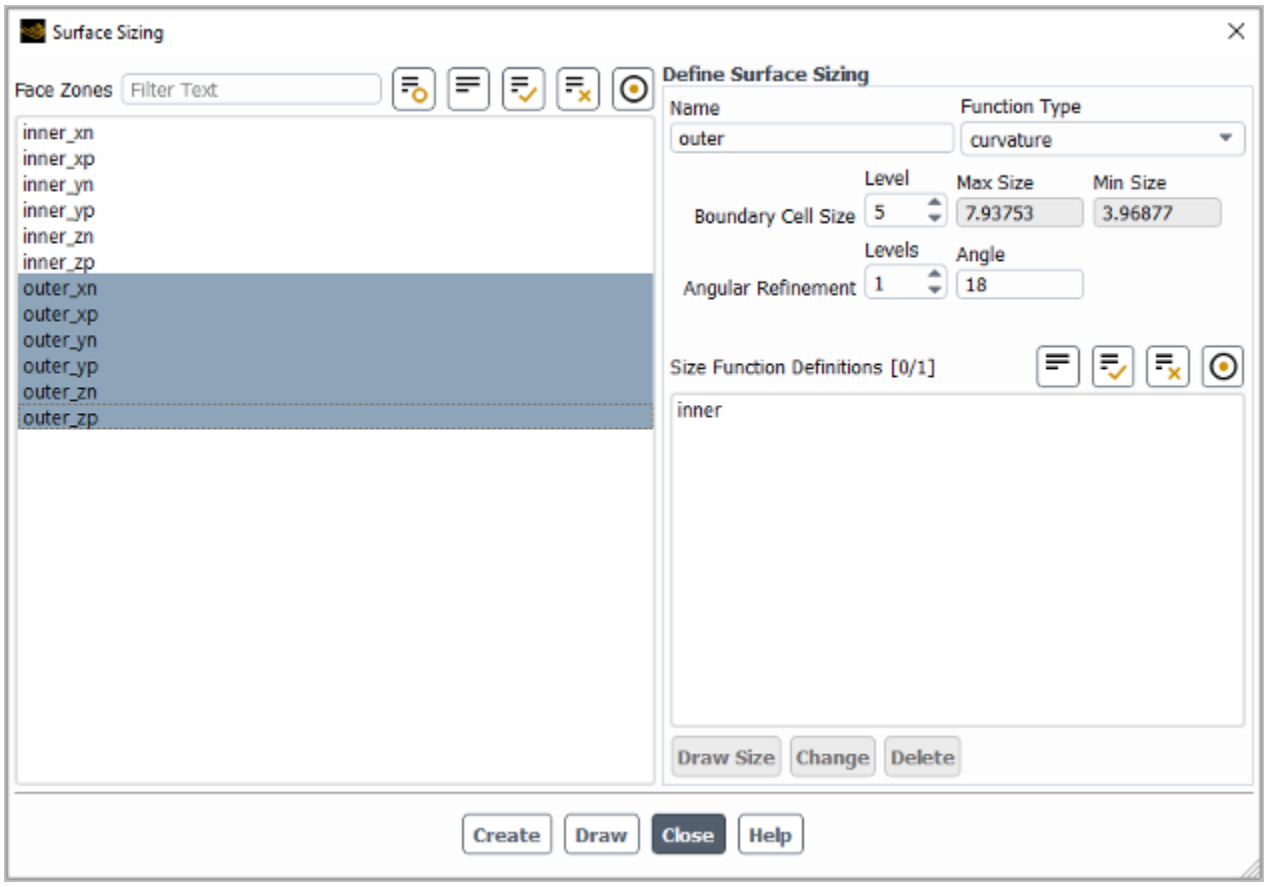
Click the **>>** button to save your settings. The name will be listed under **Defined Regions**, as well as whether it is inactive (in brackets).

You can repeat the previous steps to create additional definitions for refinement. At any point you can make a single selection from the **Defined Regions** list to display the settings in the fields of the **Region Configuration** group box. After reviewing the settings, you can: choose to allow it to remain as is; edit the settings as necessary and click the **>>** button to save it again; or click the **<<** button or the **Delete** button to delete it. You can also make selections from the **Defined Regions** list and use the **Activate** or **Deactivate** buttons to determine whether they are active, and/or **Draw** them in the graphics display. Note that the **Active Regions** field in the **Rapid Octree** dialog box displays the number of active refinement regions defined in the **Defined Regions** list.

7.15.1.3.2. Custom Boundary Sizes

When using the **Boundary Projection** treatment method, you can click the **Custom Boundary Sizes...** button to open the **Surface Sizing** dialog box, and there define size functions for the geometry, as well as local **Angular Refinement** settings.

Figure 7.139: The Surface Sizing Dialog Box



You can use the **Surface Sizing** dialog box to do the following:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.15.1.3.2.1. Creating Size Functions
7.15.1.3.2.2. Draw, Change, and Delete Functions

7.15.1.3.2.1. Creating Size Functions

To create a function, make selections in the **Face Zones** list or by clicking the mouse button on zones in the graphics window. After selection, enter a **Name** for the sizing function, and define a refinement **Level** for the **Boundary Cell Size**. The default selection of **soft** for the **Function Type** specifies that the global **Feature Angle Refinement** settings in the **Rapid Octree** dialog box are overwritten, so that no refinement is applied based on the angles at which faces meet; as an alternative, you have the option of selecting **curvature** for the **Function Type** and then defining local **Angular Refinement** settings for individual or groups of zones, so that additional **Levels** of refinement are applied relative to the **Max Size**. By clicking **Create**, the function is defined and added to the **Size Functions** list.

Each size function can include multiple face zones. The default level (for selected or unselected face zones) is the value specified for the **Boundary Cell Size** in the **Rapid Octree** dialog box. A size function can thereby also specify a coarser grid than the default, but not coarser than the **Maximum Cell Size**.

If a face zone is assigned to more than one size function, a warning is issued during mesh creation and the minimum specified level is used.

7.15.1.3.2.2. Draw, Change, and Delete Functions

After a size function is created, it can be selected from the **Size Functions** list to display the settings, such as the **Max Size**. You can then perform the following:

- Click the **Draw Size** button to display small cubes on the face zones of the size function in the graphics window. These cubes indicate the size of the mesh. If a feature angle refinement is also specified, smaller cubes indicate the minimum cell size on that boundary zone.
- Adjust the settings (zones, name, level and/or feature angle refinement) and click **Change** to revise the size function.
- Click **Delete** to delete the selected size function.

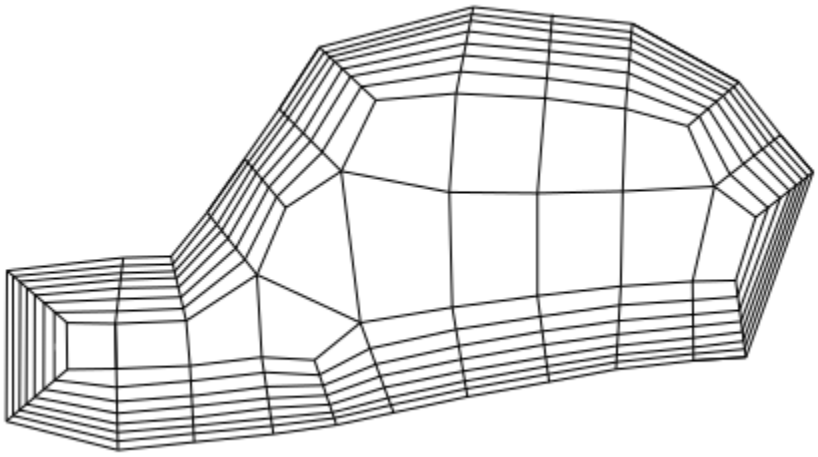
7.15.1.3.3. Boundary Layer Mesh

When using the **Boundary Projection** treatment method, by default a single layer of prismatic cells is generated at boundary zones, and these cells are generally isotropic. You have the option of defining multiple prismatic layers for specific boundary zones, so that this original isotropic layer is instead divided into many anisotropic layers at the time of meshing. This allows you to increase the mesh resolution perpendicular to the anticipated streamwise direction without overly increasing the cell count. It is an alternative to creating such layers after meshing by using other tools in the meshing mode or by using mesh adaption in the solution mode; it also allows you to combine this splitting with boundary mesh optimization (see Boundary Mesh Optimization (p. 837)), which can be helpful for complex cases.

When using this feature, you will define the total number of anisotropic prism layers that you want for a given boundary zone or group of boundary zones, along with other settings that determine the heights of the various layers.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.140: Multiple Prism Layers on Exterior Boundary Zones



Important:

It can be helpful to keep in the mind that with the Rapid Octree mesher, the anisotropic prism layers are not created at the boundaries as a first step before the remainder of the interior is meshed (as is common with other meshing methods), but instead a preliminary interior mesh is created first and then the cell layers near the boundaries are divided and/or adjusted to produce the anisotropic layers.

To define multiple prismatic layers at the boundaries, perform the following steps:

1. Select **Boundary Projection** from the **Method** list in the **Boundary Treatment** group box.
2. (Optional) You can enable the **Boundary Mesh Optimization** option for complex cases. For further details, see Boundary Mesh Optimization (p. 837).
3. Define the other settings in the **Rapid Octree** dialog box, such as the **Boundary Cell Size** and **Feature Angle Refinement**, so that you get appropriate values when you initialize the prism layer settings (as described in a later step).
4. Click the **Boundary Layer Mesh...** button to open the **Boundary Layer Mesh Definition** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.141: The Boundary Layer Mesh Definition Dialog Box

Here you can repeatedly perform the following steps to define the prism layer settings for individual or groups of boundary zones:

a. Select one or more **Face Zones**.

b. (Optional) Click the **Draw** button in the **Surface Zones** group box to display the selected zones in the graphics window, so that you can confirm they are the intended zones.

c. In the **Layer Settings** group box, enter a **Name** for this layer settings definition.

d. Enter the total **Number of Layers** that you want produced from the original isotropic layer adjacent to the selected **Face Zones**.

e. Make a selection from the **Define First Layer By** list to specify how you want to define the first anisotropic layer:

   - Select **Height** if you know the absolute height you would like for the first prism layer.
   - Select **Aspect Ratio** if you know the aspect ratio (that is, the base length of the cell divided by the height) you would like for cells in the first prism layer.

f. Click the **Initialize** button. This populates the settings in the **Options** group box with reasonable values based on the settings you have already defined, which you can then evaluate and change as needed. You should click this button again any time you change the settings described previously.

g. If you selected **Height** from the **Define First Layer By** drop-down list, enter the **First Layer Height** that you would like. When defining this field, it may be useful to review the **Edge Length** displayed for the **Boundary Cell Size** in the **Rapid Octree** dialog box. Note that you must enter a number that is greater than 0 for this field, otherwise Fluent will revise the value.

h. If you selected **Aspect Ratio** from the **Define First Layer By** drop-down list, enter the **First Layer Aspect Ratio** that you would like. This ratio is defined as the base length of the cell (that is, the minimum cell dimension on the surface mesh) divided by the height. Note that you must enter a number that is equal to or greater than 1 for this field, otherwise Fluent will revise the value.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

i. Make a selection from the **Automatic Growth Factor** drop-down list to indicate if you want the growth rate of the prism layers after the first layer to be determined automatically by Fluent. If **No** is selected for this drop-down list, you must specify an appropriate value in the **Growth Factor** field. When possible, this factor is multiplied by the height of a prism layer (starting with the first layer) to determine the height of each subsequent prism layer. The goal is to provide a suitable transition for the refined prism layer cells to the coarser cells deeper within the mesh, so that there is not a rapid change in cell volume. You must enter a number that is equal to or greater than 1 for this field, otherwise Fluent will revise the value.

Note:

When **Height** is selected from the **Define First Layer By** drop-down list, it is not possible to select **Yes** for **Automatic Growth Factor** if **Yes** is also selected for **Automatic Expansion Factor** (a drop-down list that is described in a later step); only one of these settings can be determined automatically.

j. Make a selection from the **Automatic Expansion Factor** drop-down list to indicate if you want the expansion factor allowed for the original isometric layer to be determined automatically by Fluent. If you select **No**, you must enter an appropriate value in the **Expansion Factor** field. When needed, the height of the original isometric layer adjacent to the geometry will be expanded by this factor prior to being subdivided into the prism layers. Such expansion allows the resulting prism layers to more closely adhere to your other specifications. Note that you must enter a number that is equal to or greater than 1 for this field, otherwise Fluent will revise the value.

k. Click the **>>** button to save your settings. The name will be listed under **Active Definitions**, as well as the number of face zones on which it is applied (in brackets).

l. You can repeat the previous steps to create additional definitions for prism layers. At any point you can make a single selection from the **Active Definitions** list to display the settings in the fields of the **Boundary Layer Mesh Definition** dialog box; after reviewing the settings, you can: choose to allow it to remain as is; edit the settings as necessary and click the **>>** button to save it again; or click the **<<** button or the **Delete** button to delete it. If you know that you want to delete all of the definitions, you can click the **Delete All** button.

Note:

When setting up the prism layer definitions, note the following:

- The prism layers that result from a given definition may not perfectly reflect the settings you defined, as Fluent may need to automatically adjust them during the meshing process to account for the nature of the domain / geometric constraints, as well as to provide reasonable values for the settings. The **Number of Layers** that you specify for a definition will always be respected, but the height, ratio, and/or factors you request in the **Options** group box may be changed, with priority given to the expansion factor if possible.

- If you try to create a new prism layer definition with a face zone that is already used in a previous definition, the previous definition will be changed to either

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

not include the common face zone (if the definition includes a face zone that is unique) or completely overwritten by the new definition (if all of the face zones are shared by the new definition).

- Where two boundary zones with different numbers of prismatic layers meet, the transition is handled at the intersection by the creation of a polyhedral cell. In other words, stair-stepping (which is used for scoped prism controls, as shown in Figure 7.103: Use of Multiple Scoped Prism Controls (p. 777)) is *not* used for the prismatic layers generated by the Rapid Octree mesher to gradually reduce the number of layers for one of the zones. When the two zones are both walls, it is recommended that you define the settings so that the height of the first prism layer is bigger on the boundary zone with a smaller number of layers, in order to mitigate jumps in the cell volumes at the transition. When the two zones are a wall and a non-wall (that is, a symmetry zone, an inlet, or an outlet), the polyhedral transition cell at the interface might affect the solution; you can try to mitigate this by using the bounding box to "cut" the mesh at the non-wall boundary.
- It is recommended that your specified **Expansion Factor** not exceed 2.5, as higher values may result in heavy distortions to the octree cells deeper within the mesh volume.

When all of the prism layer definitions are complete, close the **Boundary Layer Mesh Definition** dialog box.

5. Use the **Rapid Octree** dialog box to generate the mesh.
6. Review the resulting mesh and rerun the Rapid Octree mesher with different settings as needed.

Note:

For complex geometries, the creation of prism layers can lead to significant deterioration of cell quality. It is therefore augmented with an automatic mesh repair step at the end of the mesh generation. Since this step can significantly increase the mesh generation time when there are large numbers of bad cells, it is only executed if the number of bad cells is below a threshold of 500. If this threshold is exceeded, you will receive a warning in the console, and you can try to address the issue by performing the following:

- You can manually initiate the mesh repair. It is recommended that you first change the quality method to `inverse-ortho-quality` through the `report/quality-method` text command; then you can perform the mesh repair on selected cells zones by using the `mesh/modify/auto-node-move` text command. For further details, see `mesh/` in the *Fluent Text Command List*.
- You can delete the bad cells by using the following text command for specified cell zones: `mesh/rapid-octree/delete-poor-quality-cells`. When a cell is deleted, new face zones of type symmetry are created for the adjacent cells on the faces that were shared with the deleted cell. This text command should be used with care in order to avoid unwanted effects during the solution

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(such as solution errors due to bad geometry representation, or even solver divergence).

- You can rerun the Rapid Octree mesher with revised prism layer definitions so that the **Expansion Factor** is set to 1, and then manually initiate mesh repair and/or delete poor quality cells as needed.

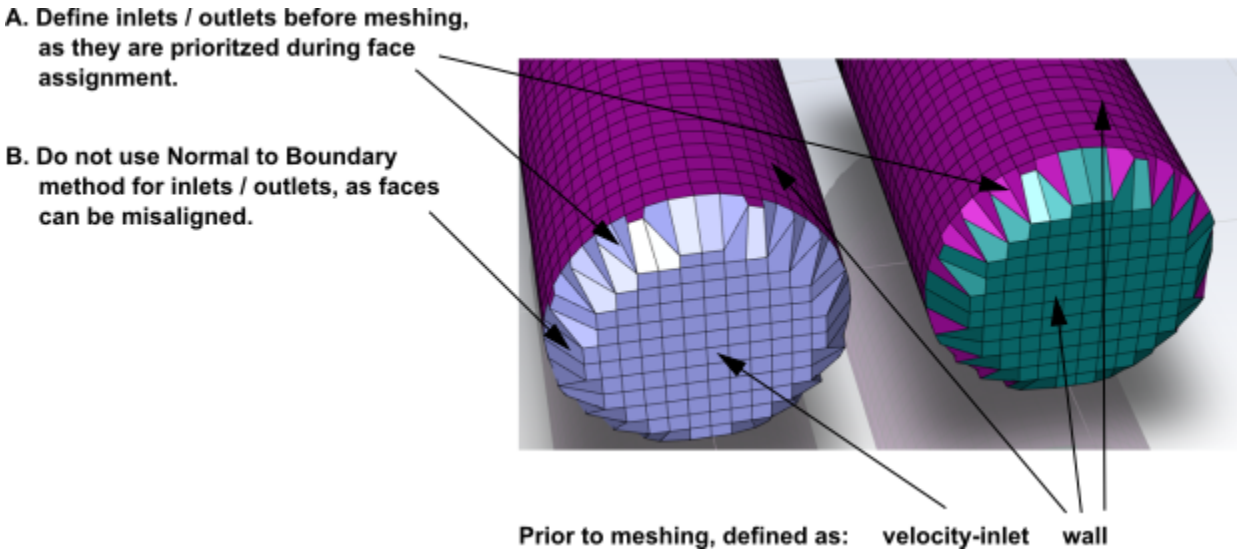
If ultimately you cannot avoid creating invalid or low quality cells, you will have to delete the prism layer definitions on those particular boundary zones and instead refine the mesh after meshing by using other tools in the meshing mode or by using mesh adaption in the solution mode.

7.15.2. Limitations of the Rapid Octree Mesher

The following limitations exist for the Rapid Octree mesher:

- Defeaturing can impact the resulting inlets and outlets:
  - The surface area of a boundary generated by the Rapid Octree mesher may not exactly match that of the original geometry. You can minimize this discrepancy by assigning boundary types to the surfaces prior to using the Rapid Octree mesher, as inlets and outlets will be prioritized when assigning individual cell faces to the boundary zones in order to best preserve the projected area (see note A. in the following figure). It is recommended that you evaluate the surface areas of inlets and outlets after mesh generation, and if they are not appropriate, perform the mesh generation from the geometry again using settings that will produce a more refined mesh.
  - Cell faces at the edges of inlets and outlets may not be oriented with the original geometry surface, and so when defining the boundary conditions for such zones in the solution mode it is recommended that you do not use the **Normal to Boundary** specification method for the velocity or direction. See note B. in the following figure.

Figure 7.142: Defeaturing Considerations



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you cannot avoid the defeaturing by these steps, then you could instead try enabling the **Improve Geometry Resolution** option, as described in Improve Geometry Resolution (p. 838).

- You can only have constant boundary cell sizes when using the **Cartesian Snapping** boundary treatment method.
- With the **Cartesian Snapping** boundary treatment method, the boundary geometry must be a consistently closed triangulated surface (that is, there should be no gaps, no overlaps, no self-intersection, no interior boundary zones bounding mesh volumes on both sides) with consistent normals. With the **Boundary Projection** boundary treatment method, it may be possible for small faults to be tolerated.
- Multiple cell zones can only be created using the **Selected Material Points** or **All Volumetric Regions** selection from the **Volume Specification** drop-down list.
- To ensure an acceptable mesh quality for the projection boundary handling, nodes may be moved away from the boundary, which results in small bumps in the surface mesh. This occurs mostly with very coarse meshes, at sharp corners, and at mesh size changes. Using a higher volume transition exponent and/or mesh refinement can mitigate the effect.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Appendix A. Importing Boundary and Volume Meshes

The volume-mesh generation scheme requires sets of triangular and/or quadrilateral elements defining the boundaries of the computational domain. In addition to the basic capability provided in GAMBIT, Fluent can use other popular CAD/CAE software packages.

A.1. GAMBIT Meshes

A.2. TetraMesher Volume Mesh

A.3. Meshes from Third-Party CAD Packages

A.1. GAMBIT Meshes

GAMBIT can create both surface and volume meshes. See the GAMBIT Modeling Guide for details.

A.2. TetraMesher Volume Mesh

ICEM CFD Engineering writes a RAMPANT file from TetraMesher. TetraMesher generates tetrahedral volume grids using a recursive subdivision octree scheme.

To read a RAMPANT file, use the **File** → **Read** → **Mesh** menu item or the `file/read-mesh` text command.

A.3. Meshes from Third-Party CAD Packages

You can import grid files from third-party CAD packages by using the items in the **File** → **Import** menu. Alternatively, the `fe2ram` filter enables you to convert files created by several finite-element packages to the grid file format used by Fluent. You can convert surface or volume meshes from ANSYS, I-deas, NASTRAN, PATRAN, VRML files from VRML version 1.0, or other packages. ARIES files can be converted only if they are first saved as ANSYS Prep7 files, as described in ARIES Files (p. 858).

If you choose to convert the file manually before reading it, enter the following command:

```
utility fe2ram [dimension] read-format [merge] [zoning] [write-format] input-file output-file
```

The items in square brackets are optional.

- *`dimension`* indicates the dimension of the dataset.
  - For a 3D grid, do not specify *`dimension`* as 3D is the default.
  - For a 2D grid, replace *`dimension`* by `-d2`.
  - For a surface mesh, replace *`dimension`* by `-surface`.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- *read-format* indicates the format of the file you want to convert. Replace *read-format* as follows:
  - For an ANSYS file, use -tANSYS.
  - For an I-deas file, use -tIDEAS.
  - For a NASTRAN file, use -tNASTRAN.
  - For a PATRAN file, use -tPATRAN.

  For a list of conversion capabilities from other CAD packages, type utility fe2ram -cl -help.
- *merge* indicates the grid tolerance. The default is 10^{-6} (1.0e-06). To set another tolerance value, replace merge by -m*TOLERANCE*, where *TOLERANCE* is an appropriate real number value. To reset the tolerance to the default value, replace *merge* by -m.
- *zoning* indicates how zones were identified in the CAD package. Replace *zoning* as follows:
  - For a grid zoned by group, do not specify *zoning* as zoning by groups is the default.
  - For a grid that was zoned by property IDs, use -zID.
  - To ignore all zone groupings, use -zNONE.
- *write-format* indicates the output format for the file you want to convert. Replace *write-format* as follows:
  - To write the grid for use in Fluent, do not specify *write-format* as this is the default.
  - To write the grid in **FIDAP** format, use -oFIDAP7.
- *input-file* and *output-file* are the names of the original file and the file to which you want to write the converted grid information, respectively. Note that the *output_file* cannot be a CFF file (.msh.h5).

For example, to convert the 2D I-deas volume mesh file sample.unv to an output file called sample.msh, enter:

```
utility fe2ram -d2 -tIDEAS sample.unv sample.msh
```

After the output file has been written, you can read it using **File** → **Import** in the meshing mode of Fluent. For volume meshes, the resulting output file can also be read into the solution mode of Fluent.

Important:

All boundary types are considered to be wall zones. You can set the appropriate boundary types in meshing mode or in solution mode.

Note:

The fe2ram utility supports VRML files from VRML version 1.0.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A.3.1. I-deas Universal Files

For I-deas surface meshes, the filter reads triangular and quadrilateral elements that define the boundaries of the domain and have been grouped within I-deas to create zones. For volume meshes, the filter reads a 2D or 3D mesh that has its boundary nodes *or* 2D boundary elements appropriately grouped to create boundary zones. Do not include nodes and boundary elements in the same group. All boundary zones will be considered wall zones; you can set the appropriate boundary types in the meshing mode or in the solution mode. See Meshes from Third-Party CAD Packages (p. 853) for further details.

A.3.1.1. Recognized I-deas Datasets

The following Universal file datasets are recognized by the grid filter:

Node Coordinates

dataset number 15, 781, 2411

Elements

dataset number 780 or 2412

Permanent Groups

dataset number 752, 2417, 2429, 2430, 2432, 2435

Because Fluent uses linear elements, you should use linear elements to generate the grid inside the mesh areas. If parabolic elements exist in the dataset, the filter ignores the mid side nodes. This assumption is valid if the edges of the element are near linear. However, if this is not the case, an incorrect topology may result from this assumption. For example, in regions of high curvature the parabolic element may look much different than the linear element.

For volume meshes, note that mesh area/mesh volume datasets are *not* recognized. This implies that writing multiple mesh areas/mesh volumes to a single Universal file may confuse `fe2ram` or Fluent.

A.3.1.2. Grouping Elements to Create Zones for a Surface Mesh

The Group command in I-deas is used to create the boundary zones needed by Fluent. All faces grouped together are listed together in the output as a single zone. In Fluent, boundary conditions are set on a per-zone basis.

One technique is to generate groups automatically based on mesh areas—that is, every mesh area will be a different zone. Although this method may generate a large number of zones, the zones can be merged in the meshing mode or in the solution mode of Fluent. Another technique is to create a group of elements related to a given mesh area manually. This enables you to select multiple mesh areas for one group.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A.3.1.3. Grouping Nodes to Create Zones for a Volume Mesh

The Group command is used in I-deas to create the boundary zones needed by Fluent. All nodes grouped together are listed together in the output as a single zone. It is important not to group nodes of internal faces with nodes of boundary faces.

One technique is to generate groups automatically based on curves or mesh areas—that is, every curve or mesh area will be a different zone in Fluent. Another technique is to create the groups manually, generating groups consisting of all nodes related to a given mesh area (3D).

A.3.1.4. Periodic Boundaries

In general, it is difficult to generate a valid grid with periodic boundaries in I-deas. However, a special feature exists in the meshing mode in Fluent that enables you to generate a grid in a domain with periodic boundaries. See Creating Periodic Boundaries (p. 700) for further details.

A.3.1.5. Deleting Duplicate Nodes

I-deas often generates duplicate nodes in the process of creating triangular elements. These must be removed by using either the remove coincident node command in I-deas or the **Merge** button in the **Merge Boundary Nodes** dialog box (or the `/boundary/merge-duplicates` text command). This node merging process is usually faster in the meshing mode in Fluent but more visual in I-deas.

A.3.2. PATRAN Neutral Files

For PATRAN surface meshes, the filter reads triangular and quadrilateral linear elements that define the boundaries of the domain and have been grouped by named component or identified by property IDs within PATRAN to create zones. For volume meshes, the filter reads a 2D or 3D mesh that has its boundary nodes grouped by named component to create boundary zones. All boundary zones will be considered wall zones; you can set the appropriate boundary types in the meshing mode or in the solution mode of Fluent. See Meshes from Third-Party CAD Packages (p. 853) for details.

A.3.2.1. Recognized PATRAN Datasets

The following Neutral file datasets are recognized by the grid filter:

Node Data

Packet Type 01

Element Data

Packet Type 02

Distributed Load Data

Packet Type 06

Node Temperature Data

Packet Type 10

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name Components

Packet Type 21

File Header

Packet Type 25

A.3.2.2. Grouping Elements to Create Zones

In PATRAN, named components are applied to the nodes to create groups of faces called zones. In Fluent, boundary conditions are applied to each zone. For example, all nodes on a curve or patch can be put in a Name Component.

For 2D volume meshes, an additional constraint is placed on the elements: existence in the Z=0 plane.

A.3.2.3. Periodic Boundaries

In general, it is difficult to generate a valid grid with periodic boundaries in PATRAN. However, a special feature exists in the meshing mode in Fluent that enables you to generate a grid in a domain with periodic boundaries. See Creating Periodic Boundaries (p. 700) for further details.

A.3.3. ANSYS Files

For ANSYS surface meshes, the filter reads triangular and quadrilateral linear elements that define the boundaries of the domain and have been grouped within ANSYS using node and element selection. For volume meshes, the filter reads a 2D or 3D mesh that has its boundary nodes grouped within ANSYS using node and element selection. All boundary zones will be considered wall zones; you can set the appropriate boundary types in the meshing mode or in the solution mode in Fluent. See Meshes from Third-Party CAD Packages (p. 853) for details.

A.3.3.1. Recognized Datasets

The following datasets are recognized by the grid filter:

NBLOCK

node block data

EBLOCK

element block data

CMBLOCK

element/node grouping

The elements must be STIF63 linear shell elements. In addition, if element data without an explicit element ID is used, the filter assumes sequential numbering of the elements when creating the zones.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A.3.3.2. Periodic Boundaries

In general, it is difficult to generate a valid grid with periodic boundaries in ANSYS. However, a special feature exists in the meshing mode in Fluent that enables you to generate a grid in a domain with periodic boundaries. See Creating Periodic Boundaries (p. 700) for further details.

A.3.4. ARIES Files

ARIES provides a filter or you may write a Prep7 file from ARIES and use the `fe2ram` filter with arguments for an ANSYS file. For more information on importing ANSYS files, see ANSYS Files (p. 857).

In general, to write a Prep7 file within ARIES the following criteria must be met:

- Name the part in the Geom module.
- Create a material or read one from the mat_lib in the Material module. To create a material, you must supply density, Poisson's ratio, and elastic modulus.
- Generate face pressures for the surface in the Environment module. Later, when you write the Prep7 file, these will be transferred to the individual elements.
- Generate at least one restraint in the Environment module.
- Set the element type to be STIF63 (triangular shell elements) and specify some finite thickness.
- Write the Prep7 file, making sure you let it automatically assign the pressure to the elements.

You can filter the Prep7 file by using the ARIES or Fluent filter, whichever you find most convenient.

A.3.5. NASTRAN Files

For NASTRAN surface meshes, the filter reads triangular and quadrilateral linear elements that define the boundaries of the solution domain. For volume meshes, the filter reads a 2D or 3D mesh. All boundary zones are considered wall zones; you can set the appropriate boundary types in the meshing mode or in the solution mode in Fluent. For details, see Meshes from Third-Party CAD Packages (p. 853).

A.3.5.1. Recognized NASTRAN Bulk Data Entries

The following NASTRAN bulk entries are recognized by the grid filter:

GRID

single-precision node coordinates

GRID*

double-precision node coordinates

CBAR

line elements

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CTETRA, CTRIA3

tetrahedral and triangular elements

CHEXA, CQUAD4, CPENTA

hexahedral, quadrilateral, and wedge elements

Because Fluent uses linear elements, you should use linear elements in the mesh-generation process. If parabolic elements exist in the dataset, the filter ignores the mid-side nodes. This assumption is valid if the edges of the element are near linear. However, if this is not the case, an incorrect topology may result from this assumption. For example, in regions of high curvature the parabolic element may look much different than the linear element.

A.3.5.2. Periodic Boundaries

In general, it is difficult to generate a valid grid with periodic boundaries in NASTRAN. However, a special feature exists in the meshing mode in Fluent that enables you to generate a grid in a domain with periodic boundaries. For details, see Creating Periodic Boundaries (p. 700).

A.3.5.3. Deleting Duplicate Nodes

NASTRAN often generates duplicate nodes in the process of creating triangular elements. You must remove them by clicking **Merge** in the **Merge Boundary Nodes** dialog box (or by using the `/boundary/merge-duplicates` text command).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Appendix B. Mesh File Format

The content and format of Fluent mesh files is described in the following sections:

B.1. Guidelines

B.2. Formatting Conventions in Binary Files and Formatted Files

B.3. Grid Sections

B.4. Non-Grid Sections

B.5. Example Files

Note:

A *mesh* file is a subset of a case file; it contains only those sections of the case file that pertain to the mesh. The currently defined sections relevant for the mesh file are explained in the following sections. For information about sections in the case file, refer to Grid Sections (p. 5676) and Other (Non-Grid) Legacy Case Sections (p. 5688) in the Fluent User's Guide (p. 1).

B.1. Guidelines

The mesh files are broken into several sections according to the following guidelines:

- Each section is enclosed in parentheses and begins with a decimal integer indicating its type. This integer is different for formatted and binary files (see Formatting Conventions in Binary Files and Formatted Files (p. 861)).
- All groups of items are enclosed in parentheses. This makes skipping to ends of (sub)sections and parsing them very easy. It also allows for easy and compatible addition of new items in future releases.
- Header information for lists of items is enclosed in separate sets of parentheses, preceding the items, which are in their own parentheses.

B.2. Formatting Conventions in Binary Files and Formatted Files

For formatted files, examples of file sections are given in Grid Sections (p. 862) and Non-Grid Sections (p. 871). For binary files, the header indices described in subsequent sections (for example, `10` for the node section) are preceded by `20` for single-precision binary data, or by `30` for double-precision binary data (that is, `2010` or `3010` instead of 10). The end of the binary data is indicated by `End of Binary Section 2010` or `End of Binary Section 3010` before the closing parameters of the section.

An example with the binary data represented by periods is as follows:

```
(3010 (2 1 2aad 2 3)(
.
```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
.
.
)
End of Binary Section 3010)
```

B.3. Grid Sections

Grid sections are stored in the case file. A *mesh* file is a subset of a case file, containing only those sections pertaining to the mesh. The currently defined grid sections are explained in the following sections.

B.3.1. Comment
B.3.2. Header
B.3.3. Dimensions
B.3.4. Nodes
B.3.5. Periodic Shadow Faces
B.3.6. Cells
B.3.7. Faces
B.3.8. Edges
B.3.9. Face Tree
B.3.10. Cell Tree
B.3.11. Interface Face Parents

The section ID numbers are indicated in both symbolic and numeric forms. The symbolic representations are available as symbols in a Scheme source file (`xfile.scm`), which is available from Ansys, Inc., or as macros in a C header file (`xfile.h`), which is located in your installation area.

B.3.1. Comment

Index:	0
Scheme symbol:	`xf-comment`
C macro:	`XF_COMMENT`
Status:	optional

Comment sections can appear anywhere in the file (except within other sections) as:

```
(0 "comment text")
```

You should precede each long section or group of related sections, by a comment section explaining what is to follow.

For example,

```
(0 "Variables:")
 (60 (
 (max-skew-limit 1.)
 (max-cell-skew 0.85)
 (skewness-method 0)
 ))
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

B.3.2. Header

Index:	1
Scheme symbol:	`xf-header`
C macro:	XF_HEADER
Status:	optional

Header sections can appear anywhere in the file (except within other sections). The following is an example:

```
(1 "ANSYS(R) TGrid(TM) 3D, serial 15.0.0")
```

The purpose of this section is to identify the program that wrote the file. Although this section can appear anywhere, it is typically one of the first sections in the file. Additional header sections indicate other programs that may have been used in generating the file. This provides a history mechanism showing where the file came from and how it was processed.

B.3.3. Dimensions

Index:	2
Scheme symbol:	`xf-dimension`
C macro:	XF_DIMENSION
Status:	optional

The dimensions of the grid appear as:

```
(2 ND)
```

where *ND* is 2 or 3. This section is supported as a check that the grid has the appropriate dimensions.

B.3.4. Nodes

Index:	10
Scheme symbol:	`xf-node`
C macro:	XF_NODE
Status:	required

The nodes section appears as:

```
(10 (zone-id first-index last-index type ND)(
 x1 y1 z1
 x2 y2 z2
 .
 .
 .
 ))
```

- If `zone-id` is zero, this provides the total number of nodes in the mesh. In this case, `first-index` will be one, `last-index` will be the total number of nodes *in hexadecimal*, `type` is zero, *ND* is

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

omitted, and there are no coordinates following (the parentheses for the coordinates are omitted as well).

For example,

```
(10 (0 1 2d5 0))
```

- If `zone-id` is greater than zero, it indicates the zone to which the nodes belong. In this case, `first-index` and `last-index` are the indices of the nodes in the zone *in hexadecimal*. The values of `last-index` in each zone must be less than or equal to the value in the declaration section. `type` indicates the type of nodes in the zone. The following values are used to indicate the node type:

  – Zero for "virtual" nodes.

  – One for no (any) type.

  – Two for boundary nodes.

  Nodes of type zero are ignored but types one and two are read and written.

  ND is an optional argument that indicates the dimensionality of the node data, where *ND* is 2 or 3. If the number of dimensions in the grid is two, as specified in the Dimensions (p. 863) or in the node header, then only x and y coordinates are present on each line.

The following is an example of a two-dimensional grid:

```
(10 (1 1 2d5 1 2)(
 1.500000e-01 2.500000e-02
 1.625000e-01 1.250000e-02
  .
  .
  .
 1.750000e-01 0.000000e+00
 2.000000e-01 2.500000e-02
 1.875000e-01 1.250000e-02
 ))
```

As the grid connectivity is composed of integers representing pointers (see Cells (p. 865) and Faces (p. 866)), using hexadecimal conserves space in the file and provides for faster file input and output. The header indices are also in hexadecimal so that they match the indices in the bodies of the grid connectivity sections. The `zone-id` and `type` are also in hexadecimal for consistency.

B.3.5. Periodic Shadow Faces

Index:	18
Scheme symbol:	`xf-periodic-face`
C macro:	`XF_PERIODIC_FACE`
Status:	required only for grids with periodic boundaries

This section indicates the pairings of periodic faces on periodic boundaries. Grids without periodic boundaries do not have sections of this type.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The format of the section is as follows:

```
(18 (first-index last-index periodic-zone shadow-zone)(
 f00 f01
 f10 f11
 f20 f21
 .
 .
 .
 ))
```

where:

- `first-index` is the index of the first periodic face pair in the list.
- `last-index` is the index of the last periodic face pair in the list.
- `periodic-zone` is the zone ID of the periodic face zone.
- `shadow-zone` is the zone ID of the corresponding shadow face zone.

These are in hexadecimal format.

The indices in the section body (`f*`) refer to the faces on each of the periodic boundaries (in hexadecimal), the indices being offsets into the list of faces for the grid.

Note:

The `first-index` and `last-index` do *not* refer to face indices; they refer to indices in the list of periodic pairs.

An example of such a section is as follows:

```
(18 (1 2b a c) (
 12 1f
 13 21
 ad 1c2
 .
 .
 .
 ))
```

B.3.6. Cells

Index:	12
Scheme symbol:	`xf-cell`
C macro:	`XF_CELL`
Status:	required

The declaration section for cells is similar to that for nodes.

```
(12 (zone-id first-index last-index type element-type))
```

When `zone-id` is zero, it indicates that it is a declaration of the total number of cells. If `last-index` is zero, then there are no cells in the grid. This is useful when the file contains only a surface mesh as it serves to alert Fluent that it cannot be used in the solver.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In a declaration section, the `type` has a value of zero, while the `element-type` is not present.

For example,

```
(12 (0 1 3e3 0))
```

states that there are 3e3 (hexadecimal) = 995 cells in the grid. This declaration section is required and must precede the regular cell sections.

The `element-type` in a regular cell section header indicates the type of cells in the section, as follows:

element-type	description	nodes/cell	faces/cell
0	mixed		
1	triangular	3	3
2	tetrahedral	4	4
3	quadrilateral	4	4
4	hexahedral	8	6
5	pyramid	5	5
6	wedge	6	5
7	polyhedral	*NN*	*NF*

where *NN* and *NF* will vary, depending on the specific polyhedral cell.

Regular cell sections have no body, but they have a header of the same format where `first-index` and `last-index` indicate the range for the particular zone, `type` indicates whether the cell zone is fluid (`type` = 1) or solid (`type` = 17).

A `type` of zero indicates a dead zone and will be skipped when the file is read in solution mode in Fluent. If a zone is of mixed type (`element-type`=0), it will have a body that lists the `element-type` of each cell.

In the following example, there are 3d (hexadecimal) = 61 cells in cell zone 9, of which the first 3 are triangles, the next 2 are quadrilaterals, and so on.

```
(12 (9 1 3d 0 0)(
 1 1 1 3 3 1 1 3 1
 .
 .
 .
 ))
```

Note:

The cell section is not required in meshing mode when the file contains only a surface mesh.

B.3.7. Faces

Index:	13

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Scheme symbol:	xf-face
C macro:	XF_FACE
Status:	required

The face section has a header with the same format as that for cells, but with a section index of 13. The format is as follows:

```
(13 (zone-id first-index last-index bc-type face-type))
```

where:

- zone-id = zone ID of the face section
- first-index = index of the first face in the list
- last-index = index of the last face in the list
- bc-type = ID of the boundary condition represented by the face section
- face-type = ID of the type(s) of face(s) in the section

A zone-id of zero indicates a declaration section, which provides a count of the total number of faces in the mesh. Such a section omits the bc-type and is not followed by a body with further information.

A non-zero zone-id indicates a regular face section and will be followed by a body containing information about the grid connectivity. Each line describes one face and appears as follows:

```
n0 n1 n2 c0 c1
```

where n* are the defining nodes (vertices) of the face, and c* are the adjacent cells.

This is an example of the format for a 3D grid with a triangular face format. The actual number of nodes depends on the face-type. The order of the cell indices is important and is determined by the right-hand rule: if you curl the fingers of your right hand in the order of the nodes, your thumb will point toward c0.

If the face zone is of mixed type (face-type= 0) or of polygonal type (face-type= 5), each line of the section body will begin with a reference to the number of nodes that make up that particular face, and has the following format:

```
x n0 n1 ... nf c0 c1
```

where x = number of nodes (vertices) of the face and nf is the final node of the face.

All cells, faces, and nodes have positive indices. If a face has a cell only on one side, then either c0 or c1 is zero. For files containing only a boundary mesh, both these values are zero.

bc-type indicates the ID of the boundary condition represented by the face section. The current valid boundary condition types are defined in the following table:

bc-type	**Description**
2	interior
3	wall

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

bc-type	Description
4	pressure-inlet, inlet-vent, intake-fan
5	pressure-outlet, exhaust-fan, outlet-vent
7	symmetry
8	periodic-shadow
9	pressure-far-field
10	velocity-inlet
12	periodic
14	fan, porous-jump, radiator
20	mass-flow-inlet
24	interface
31	parent (hanging node)
36	outflow
37	axis

The faces resulting from the intersection of the non-conformal grids are placed in a separate face zone. A factor of 1000 is added to the `bc-type` of these sections; for example, 1003 is a wall zone.

`face-type` indicates the type of faces in the zone as defined in the following table:

face-type	description	nodes/face
0	mixed	
2	linear	2
3	triangular	3
4	quadrilateral	4
5	polygonal	*NN*

where *NN* will vary, depending on the specific polygonal face.

B.3.8. Edges

Index:	11
Scheme symbol:	`xf-edge`
C macro:	XF_EDGE
Status:	optional

The edge section has a header of the following format:

```
(11 (zone-id first-index last-index type element-type))
```

where:

- `zone-id` is the zone ID for the edge section
- `first-index` and `last-index` are the index of the first and last edge in the list, respectively

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- `type` indicates the edge type
- The `element-type` is ignored completely.

A `zone-id` of zero indicates a declaration section with no body, which provides a count of the total number of edges in the mesh. A non-zero `zone-id` indicates a regular edge section, and will be followed by a body containing information about the grid connectivity. Each line describes one edge and appears as follows:

```
v0 v1
```

where `v0`, `v1` are the vertices defining the edge.

`type` denotes the edge type as defined in the following table:

Description	type
boundary edge	5
interior edge	6

Note:

In case files written for Fluent, the Edges section will be omitted.

B.3.9. Face Tree

Index:	59
Scheme symbol:	`xf-face-tree`
C macro:	XF_FACE_TREE
Status:	only for grids with hanging-node adaption

This section indicates the face hierarchy of the grid containing hanging nodes. The format of the section is as follows:

```
(59 (face-id0 face-id1 parent-zone-id child-zone-id)
 (
  number-of-kids kid-id-0 kid-id-1 ... kid-id-n
  .
  .
  .
 ))
```

where

`face-id0` is the index of the first parent face in the section.

`face-id1` is the index of the last parent face in the section.

`parent-zone-id` is the ID of the zone containing the parent faces

`child-zone-id` is the ID of the zone containing the children faces.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

number-of-kids is the number of children of the parent face.

kid-id-n are the face IDs of the children.

These are in hexadecimal format. You can read files that contain this section in the meshing mode in Fluent.

B.3.10. Cell Tree

Index:	58
Scheme symbol:	xf-cell-tree
C macro:	XF_CELL_TREE
Status:	only for grids with hanging-node adaption

This section indicates the cell hierarchy of the grid containing hanging nodes.

The format of the section is as follows:

```
(58 (cell-id0 cell-id1 parent-zone-id child-zone-id)
 (
 number-of-kids kid-id-0 kid-id-1 ... kid-id-n
 .
 .
 .
 ))
```

where:

- cell-id0 is the index of the first parent cell in the section.
- cell-id1 is the index of the last parent cell in the section.
- parent-zone-id is the ID of the zone containing the parent cells.
- child-zone-id is the ID of the zone containing the children cells.
- number-of-kids is the number of children of the parent cell.
- kid-id-n are the cell IDs of the children.

These are in hexadecimal format. You cannot read files that contain this section in the meshing mode in Fluent.

B.3.11. Interface Face Parents

Index:	61
Scheme symbol:	xf-face-parents
C macro:	XF_FACE_PARENTS
Status:	only for grids with non-conformal interfaces

This section indicates the relationship between the intersection faces and original faces. The intersection faces (children) are produced from intersecting two non-conformal surfaces (parents) and are some fraction of the original face. Each child will refer to at least one parent.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The format of the section is as follows:

```
(61 (face-id0 face-id1)
 (
  parent-id-0 parent-id-1
  .
  .
  .
 ))
```

where:

- `face-id0` is the index of the first child face in the section.
- `face-id1` is the index of the last child face in the section.
- `parent-id-*` is the index of the parent faces.

These are in hexadecimal format.

If you set up and save a non-conformal mesh in the solution mode and then read it into the meshing mode of Fluent, this section will be skipped. Hence, all the information necessary to preserve the non-conformal interface will not be maintained. When you switch to the solution mode or read the mesh back into the solution mode, you will need to recreate the interface.

B.4. Non-Grid Sections

The non-grid sections contain the boundary conditions, material properties, and solver control settings.

B.4.1. Zone

B.4.1. Zone

Index:	39 or 45
Scheme symbol:	`xf-rp-tv`
C macro:	XF_RP_TV
Status:	required

Typically, there is one zone section for each zone referenced by the grid. Although some grid zones may not have corresponding sections, there cannot be more than one zone section for each zone.

The zone section has the following form:

```
(39 (zone-id zone-type zone-name domain-id)(
(condition1 . value1)
(condition2 . value2)
(condition3 . value3)
 .
 .
 .
  ))
```

Grid generators and preprocessors need only provide the section header and leave the list of conditions empty, as in:

```
(39 (zone-id zone-type zone-name domain-id)())
```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The empty parentheses at the end are required. The solver adds conditions as appropriate, depending on the zone type.

When only `zone-id`, `zone-type`, `zone-name`, and `domain-id` are specified, the index 45 may be used for a zone section. However, the index 39 must be used if boundary conditions are present, because any and all remaining information in a section of index 45 after `zone-id`, `zone-type`, `zone-name`, and `domain-id` will be ignored.

In meshing mode, the zone name and type can be extracted from the boundary condition section 39 (refer to the Fluent User's Guide for details) or 45, but only section 39 can be written.

The `zone-id` is in decimal format. This is in contrast to the use of hexadecimal in the grid sections.

The `zone-type` is one of the following:

```
degassing
exhaust-fan
fan
fluid
geometry
inlet-vent
intake-fan
interface
interior
internal
mass-flow-inlet
outflow
outlet-vent
parent-face
porous-jump
pressure-far-field
pressure-inlet
pressure-outlet
radiator
solid
symmetry
velocity-inlet
wall
wrapper
```

The `interior`, `fan`, `porous-jump`, and `radiator` types can be assigned only to zones of faces inside the domain. The `interior` type is used for the faces within a cell zone; the others are for interior faces that form infinitely thin surfaces within the domain. Fluent allows the `wall` type to be assigned to face zones both on the inside and on the boundaries of the domain.

Some zone types are valid only for certain types of grid components (for example, cell zones can be assigned only either `fluid` or `solid` type). All other types listed can be used for only boundary (face) zones.

The `zone-name` is a label for the zone. It must be a valid Scheme symbol and is written without quotes. The rules for a valid `zone-name` are as follows:

- The first character must be a lowercase letter or a special-initial.
- Each subsequent character must be a lowercase letter, a special-initial, a digit, or a special-subsequent.

  A special-initial character is one of the following:

  ```
  ! $ % & * / : < = > ? ~ _ ^
  ```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and a special subsequent is one of the following:

```
. + -
```

Some examples of zone sections produced in the meshing mode are as follows:

```
(39 (1 fluid fuel 1)())
(39 (8 pressure-inlet pressure-inlet-8 2)())
(39 (2 wall wing-skin 3)())
(39 (3 symmetry mid-plane 1)())
```

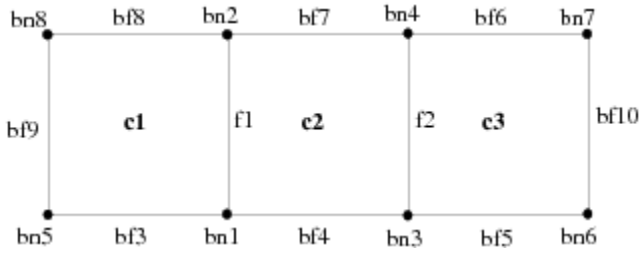
B.5. Example Files

The examples show 2D quadrilateral meshes for easier illustration. The same concepts are applied to 3D meshes.

Example 1

Figure 1: Quadrilateral Mesh (p. 873) illustrates a simple quadrilateral mesh with no periodic boundaries or hanging nodes.

Figure 1: Quadrilateral Mesh



The following describes this mesh:

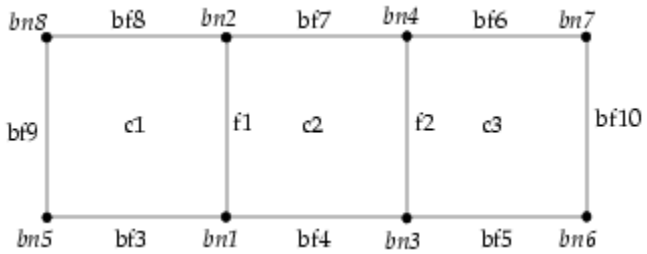
```
(0 "Grid:")

(0 "Dimensions:")
(2 2)

(12 (0 1 3 0))
(13 (0 1 a 0))
(10 (0 1 8 0 2))

(12 (7 1 3 1 3))

(13 (2 1 2 2 2)(
1 2 1 2
3 4 2 3))

(13 (3 3 5 3 2)(
5 1 1 0
1 3 2 0
3 6 3 0))

(13 (4 6 8 3 2)(
7 4 3 0
4 2 2 0
2 8 1 0))

(13 (5 9 9 a 2)(
8 5 1 0))
```

```
(13 (6 a a 24 2)(
6 7 3 0))

(10 (1 1 8 1 2)
(
1.00000000e+00 0.00000000e+00
1.00000000e+00 1.00000000e+00
2.00000000e+00 0.00000000e+00
2.00000000e+00 1.00000000e+00
0.00000000e+00 0.00000000e+00
3.00000000e+00 0.00000000e+00
3.00000000e+00 1.00000000e+00
0.00000000e+00 1.00000000e+00))
```

Example 2

Figure 2: Quadrilateral Mesh with Periodic Boundaries (p. 874) illustrates a simple quadrilateral mesh with periodic boundaries but no hanging nodes. In this example, `bf9` and `bf10` are faces on the periodic zones.

Figure 2: Quadrilateral Mesh with Periodic Boundaries



The following describes this mesh:

```
(0 "Dimensions:")
(2 2)

(0 "Grid:")

(12 (0 1 3 0))
(13 (0 1 a 0))
(10 (0 1 8 0 2))

(12 (7 1 3 1 3))

(13 (2 1 2 2 2)(
1 2 1 2
3 4 2 3))

(13 (3 3 5 3 2)(
5 1 1 0
1 3 2 0
3 6 3 0))

(13 (4 6 8 3 2)(
7 4 3 0
4 2 2 0
2 8 1 0))

(13 (5 9 9 c 2)(
8 5 1 0))

(13 (1 a a 8 2)(
6 7 3 0))
```

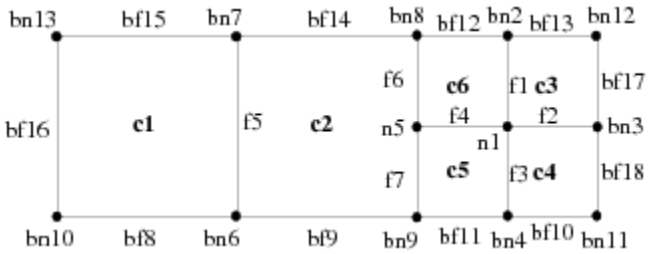

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
(18 (1 1 5 1)(
9 a))

(10 (1 1 8 1 2)(
1.00000000e+00 0.00000000e+00
1.00000000e+00 1.00000000e+00
2.00000000e+00 0.00000000e+00
2.00000000e+00 1.00000000e+00
0.00000000e+00 0.00000000e+00
3.00000000e+00 0.00000000e+00
3.00000000e+00 1.00000000e+00
0.00000000e+00 1.00000000e+00))
```

Example 3

Figure 3: Quadrilateral Mesh with Hanging Nodes (p. 875) illustrates a simple quadrilateral mesh with hanging nodes.

Figure 3: Quadrilateral Mesh with Hanging Nodes



The following describes this mesh:

```
(0 "Grid:")

(0 "Dimensions:")
(2 2)

(12 (0 1 7 0))
(13 (0 1 16 0))
(10 (0 1 d 0 2))

(12 (7 1 6 1 3))
(12 (1 7 7 20 3))

(58 (7 7 1 7)(
4 6 5 4 3))

(13 (2 1 7 2 2)(
1 2 6 3
1 3 3 4
1 4 4 5
1 5 5 6
6 7 1 2
5 8 2 6
9 5 2 5))

(13 (3 8 b 3 2)(
a 6 1 0
6 9 2 0
4 b 4 0
9 4 5 0))

(13 (4 c f 3 2)(
2 8 6 0
c 2 3 0
8 7 2 0
7 d 1 0))
```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
(13 (5 10 10 a 2)(
d a 1 0))

(13 (6 11 12 24 2)(
3 c 3 0
b 3 4 0))

(13 (b 13 13 1f 2)(
c 8 7 0))

(13 (a 14 14 1f 2)(
b c 7 0))

(13 (9 15 15 1f 2)(
9 b 7 0))

(13 (8 16 16 1f 2)(
9 8 2 7))

(59 (13 13 b 4)(
2 d c))

(59 (14 14 a 6)(
2 12 11))

(59 (15 15 9 3)(
2 b a))

(59 (16 16 8 2)(
2 7 6))

(10 (1 1 d 1 2)
(
2.50000000e+00 5.00000000e-01
2.50000000e+00 1.00000000e+00
3.00000000e+00 5.00000000e-01
2.50000000e+00 0.00000000e+00
2.00000000e+00 5.00000000e-01
1.00000000e+00 0.00000000e+00
1.00000000e+00 1.00000000e+00
2.00000000e+00 1.00000000e+00
2.00000000e+00 0.00000000e+00
0.00000000e+00 0.00000000e+00
3.00000000e+00 0.00000000e+00
3.00000000e+00 1.00000000e+00
0.00000000e+00 1.00000000e+00))
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Appendix C. Shortcut Keys

Creating a mesh and modifying or repairing it involves many operations. Various tools are available for making boundary repairs, enabling you to perform primitive operations on the boundary mesh, such as creating and deleting nodes/faces; moving nodes; swapping edges; merging/smoothing nodes; collapsing nodes, edges, and faces; and so on.

You can perform these operations using the appropriate dialog boxes. However, if you are handling a large and complicated mesh, you may find it difficult to perform these operations repetitively using the dialog boxes. To make your task easy, keyboard shortcuts are available.

C.1. Shortcut Key Actions

Important:

To make use of the keyboard shortcuts, display the geometry in the graphics window, click in the graphics window, then press the key combination. Many of the keyboard shortcuts are available as an onscreen tool button along the left side and bottom of the graphics window.

To repeat the same shortcut multiple times, after pressing the key combination, press **Ctrl** while using the right mouse button for multiple selections. This repeat shortcut is available only for some operations. The hot key help indicates the (previous) operation that will be performed in case of multiple selections.

C.1. Shortcut Key Actions

Feature	Action		Keys	Onscreen Tool
Hot key list	Print the available hot keys in the message and graphics window.		**Ctrl**+**H**	
CAD Entities **Note:** Some Keys and Onscreen Tools may have different behavior	Selection filters	CAD object filter	**Ctrl**+**B**	
		CAD zone filter	**Ctrl**+**Z**	
		Position filter	**Ctrl**+**X**	
	Display options	Highlight parent CAD object	**F6**	
		Highlight child CAD object	**F7**	
		Color by labels	**Ctrl**+**Shift**+**C**	
		Color by threads	**Ctrl**+**Shift**+**D**	

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Feature	Action		Keys	Onscreen Tool
for CAD Assemblies and Geometry/Mesh Objects.	Isolate tools	Isolate similar curvature	**Ctrl**+**C**	
		Isolate smaller area	**Ctrl**+**L**	
		Isolate similar area	**Ctrl**+**S**	
	Tree options	Delete inactive/suppressed CAD objects	**Ctrl**+**G**	
		Clear CAD tree selections	**Ctrl**+**Y**	
		Show CAD object path	**Ctrl**+**Shift**+**P**	
	Object management	Add to object	**Ctrl**+**P**	
		Replace object	**Ctrl**+**Q**	
		Create objects	**Ctrl**+**Shift**+**B**	
	Label management	Manage labels	**Ctrl**+**Shift**+**L**	
		CAD zone associated labels	**Ctrl**+**I**	
	Modify	Extract edge zones	**Ctrl**+**Shift**+**F**	
		Rename CAD objects	**Ctrl**+**Shift**+**N**	
	Update CAD objects		**Ctrl**+**Shift**+**R**	
	Suppress CAD objects		**Ctrl**+**Shift**+**S**	
	Restore deleted nodes		**Ctrl**+**J**	
Display bounds	Increase		**Up arrow**	
	Decrease		**Down arrow**	
Faces	Prints the coordinates of the centroid of the selected face to the console. This also works for edges and nodes.		**Ctrl**+**L**	
	Rezones the selected faces/cells to the target zone. Apply after setting a target zone and selecting source face(s) or cell(s).		**Ctrl**+**O**	

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Feature	Action		Keys	Onscreen Tool
	Mark faces based on selections and specified parameters. Marking faces allows for selective separation or remeshing. If help text display is active, a description of the marking options is displayed.		**Ctrl**+**Shift**+**J**	
	Marking options	Mark selected faces.	**Ctrl**+**S**	
		Mark faces by flood filling.	**Ctrl**+**D**	
		Mark faces in rings around selection.	**Ctrl**+**R**	
		Mark faces by quality.	**Ctrl**+**Q**	
		Mark faces by angle.	**Ctrl**+**G**	
		Mark island faces.	**Ctrl**+**L**	
		Unmark all/selected marked faces	**Ctrl**+**U**	
		Opens the **Mark Settings** dialog box, where you can set the parameters for marking faces.	**Ctrl**+**I**	
	Open the **Local Remesh** or **Zone Remesh** dialog box to set options and remesh faces (or zones) based on your selection in the graphics window. • Face selection will remesh the face and specified radial layers based on settings in the **Local Remesh** dialog box. • Face zone selection will remesh the zone based on settings in the **Zone Remesh** dialog box. • No selection will remesh marked faces.		**Ctrl**+**Shift**+**R**	
	Separates the face or edge zones based on the selected entity. • If a multi-region face zone is selected, then the face zones are separated by region. • If a single region face zone is selected, then the face zones are separated by angle. • If a face (or edge) is selected, then the face zone (edge zone) is separated by seed. • If edge zone with face seed selection, then the face zone is separated by edge loop. • If no selection, then the face zones are separated by mark.		**Ctrl**+**Shift**+**S**	
	Opens the Create Edge Zones dialog box to extract the edges of the surface(s) containing the selected face(s). See Extract Edge Zones (p. 707).			

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Feature	Action	Keys	Onscreen Tool
Edges	Show/hide edges on selected zones and objects independent of the mode of selection.If no object/zone is selected, then the edges on the entire geometry are shown/hidden.	**Ctrl**+**Shift**+**E**	
	Prints the coordinates of the center of the selected edge to the console. This also works for faces and nodes.	**Ctrl**+**L**	
Skewed faces	Display in decreasing order of skewness in the mesh/active group or the marked faces.	**Right arrow**	
	Clear	**Left arrow**	
Mouse Buttons dialog box	Toggle between the mouse-dolly and the option currently selected for the right-mouse button.	**F3**	
Mouse Probe	Toggle the selection of the mouse probe function between **polygon** and the option currently selected in the **Mouse Probe** dialog box. **Note:** When using **box** select or **polygon** select, by default, all entities (nodes, faces, zones, objects) will be selected. To select only visible entities instead, enable **Select Visible Entities** in the **Mouse Probe Functions** group in the ribbon. • If the mesh is not connected, all entities (nodes, faces, zones, objects) will be selected irrespective of whether they are visible or not. • This visual selection behavior works only on local displays and may generate warning messages when attempting selection on a remote system.	**F4**	or or
	Select **object** as the selection filter.	**Ctrl**+**B**	
	Select **cell** as the selection filter.	**Ctrl**+**C**	

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Feature	Action		Keys	Onscreen Tool
	This works only in the Linux version.			
	Select **edge** as the selection filter.		**Ctrl**+**E**	
	Select **face** as the selection filter.		**Ctrl**+**F**	
	Select **node** as the selection filter.		**Ctrl**+**N**	
	Select **position** as the selection filter.		**Ctrl**+**X**	
	Select **size** based on the computed size field as the selection filter.		**Ctrl**+**Y**	
	Select **zone** as the selection filter.		**Ctrl**+**Z**	
Entities	Create the selected entity.		**F5**	
	Smooth the selected entity.		**F6**	
	Perform the split operation on the selected entity (only face and edge).		**F7**	
	Perform the swap operation on the selected entity (only face and edge).		**F8**	
	Merge the selected pair.		**F9**	
	Perform the collapse operation on the selected nodes, edges, or faces.		**Ctrl**+**J** Alternatively, **Ctrl**+**^**	
	Delete the selected entities (nodes/faces/zones/objects) without confirming.		**Ctrl**+**W**	
	Toggle Edge Zone selection mode. This restricts selection to edge zones entities only.		**Ctrl**+**Shift**+**Z**	
	Viewing options only. These do not affect the mesh.	Hide the selected objects or zones in the display depending on the mode of selection set.	**Ctrl**+**Shift**+**H**	
		Isolate selected zones/objects.	**Ctrl**+**Shift**+**I**	
		Reverse the last show/hide operation.	**Ctrl**+**Shift**+**U**	
		Print detailed information about the selected entities in the message window. For more	**Ctrl**+**I**	

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Feature	Action		Keys	Onscreen Tool
		details, see Entity Information (p. 888). In addition, if a selected zone or object has been set as a **target**, this hotkey, or tool button, will toggle the identifying color.		
		Deselect the last selected entity.	**Ctrl**+**[** Alternatively: **Esc**	
		Clear all selection and visualization assists.	**F2**	
		Show all hidden zones/objects; show previously hidden zones/objects.	**Ctrl**+**Shift**+**A**	
		Calculate and display the distance between the two selected entities. The value is also printed in the console window. The shortcut prompts you when more selections are needed, otherwise, it uses the two most recent selections (nodes or coordinates) in the selections list. Distances can be measured between points, edges, faces, and zones. The distance between zones is computed using the global average centroid of each zone facet to the relative zone distance. So, for calculating the distance between two planar zones, the distance would be based on the starting and ending points being located directly on the zone surfaces; whereas for calculating the distance between two curved surfaces, the distance would be based on the starting and ending points being located offset from the zone surface(s).	**Ctrl**+**D**	
		Displays the neighboring face zones of the selected entity.	**Ctrl**+**G**	
		Inserts a clipped plane at the centroid of a selected entity/zone.	**Ctrl**+**/**	

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Feature	Action		Keys	Onscreen Tool
Nodes	Move nodes from their original location to a new position.		**Ctrl**+**V**	
	Prints the coordinates of the selected node to the console.		**Ctrl**+**L**	
Mesh/active group	Show only the worst skewed face and its ID; print the skewness value in the console.		**F11**	
Mesh operation	Undo latest operation (if possible).		**F12**	
Mesh display	Adjust to fit in the graphics window.		**Ctrl**+**A**	
Objects	Remove Boundary Gaps. Apply after setting a target zone and selecting a source zone.		**Ctrl**+**K**	
	Merge all the selected objects/zones into one object/zone depending on which mode of selection is set. • For object selection, only objects of the same type (geometry or mesh) can be merged. Enter a name for the merged object in the **Merge Objects** dialog box. • For zone selection, only zones of the same type or belonging to the same object can be merged. The resulting zone inherits the name of the zone selected first.		**Ctrl**+**Shift**+**M**	
	Open the **Change Object Properties** or **Change Zone Properties** dialog box, depending on your selection.		**Ctrl**+**Shift**+**N**	
	Randomly allocates colors to selected objects/zones, or, if no object/zone is selected, to the entire geometry.		**Ctrl**+**Shift**+**O**	
Graphics window	Selection Tools	Expands the on-screen selection tools menu.		
		Selects the object at the mouse pointer.		
		Click and drag to select objects within a rectangular box.		
		Click repeatedly to define an object selection polygon.		
		Selects all the visible objects/zones in the graphics window.	**Ctrl**+**Shift**+**V**	
		Display the previous view.	**Ctrl**+_	

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Feature	Action		Keys	Onscreen Tool
		Deselect the last selected entity.	**Ctrl**+[Alternatively: **Esc**	
		Toggle the display of the clipping plane's triad..	**F10**	
	Align objects.		**Ctrl**+**Shift**+**G**	
	Visualization Tools	Show all hidden zones/objects; show previously hidden zones/objects.	**Ctrl**+**Shift**+**A**	
		Hide the selected objects or zones in the display depending on the mode of selection set.	**Ctrl**+**Shift**+**H**	
		Isolate selected zones/objects.	**Ctrl**+**Shift**+**I**	
		Reverse the last show/hide operation.	**Ctrl**+**Shift**+**U**	
		Expands the on-screen Viewing tools menu		
		Toggle between a normal view and an exploded view of the objects in the geometry.	**Ctrl**+**Shift**+**X**	
		Toggle the transparency of the selected objects/zones depending on the mode of selection set. If no object/zone is selected then the entire geometry is made transparent so that internal objects/zones are visible.	**Ctrl**+**Shift**+**T**	
		Displays objects with similar curvature as the current selection.		
		Calculate and display the distance between the two selected entities. The value is also printed in the console window.	**Ctrl**+**D**	
		Preview size field at selected location.		
		Displays objects with similar area as the current selection.		
		Displays objects with smaller area than the current selection.		
Projection	Set/Unset a projection target. The target will be a line, plane, zone, or object depending on the selection.		**Ctrl**+**S**	

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Feature	Action	Keys	Onscreen Tool
	The target is used for easy object modification, for rezoning of faces/cells, for projecting nodes or zones, or for gap removal. If help text display is active, a description of the target options is displayed.		
	Project selected nodes/node loops/zones onto what you have set using **Ctrl**+**S**.	**Ctrl**+**P**	
	Displays information about the selected target and toggles the identifying target color.	**Ctrl**+**I**	
Zones	Auto patch zones by filling all holes based on the selected entity. Select the face zone to patch all holes associated with free faces. Select individual faces adjacent to the hole in the face zone to patch punched holes. See the User's Guide (p. 623) for details.	**Ctrl**+**R**	
	Open the **Change Object Properties** or **Change Zone Properties** dialog box, depending on the selection of objects or zones in the graphics window.	**Ctrl**+**Shift**+**N**	
	If zones are graphically selected, they are remeshed using the size field, if available.	**Ctrl**+**Shift**+**R**	
	Transfer the selected zone(s) to the target object, if applied after setting a target object and selecting source zone(s). If no target is set, the selected zone(s) will form a new geometry object as specified in the **Create Geometry Object** dialog box.	**Ctrl**+**Shift**+**Y**	
	Opens the Create Edge Zones dialog box to extract edges of the selected face zone. See Extract Edge Zones (p. 707).		
Construct geometry primitives	Enter the Construct Geometry mode to create a bounding box, cylinder, or frustum for selected or all zones displayed in the graphics window.	**Ctrl**+**Shift**+**K**	
	Create a bounding box for selected/all displayed zones.	**Ctrl**+**B**	
	Create a cylinder based on selections or bounding all displayed zones.	**Ctrl**+**C**	
	Create a frustum based on selections or bounding all displayed zones.	**Ctrl**+**F**	
	Open the **Create Object** dialog box to specify additional settings. Click **Create** to create the geometry object based on the selections and settings.	**F5**	

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Feature	Action			Keys	Onscreen Tool
Loop of nodes	Enter the Loop Selection Mode. Select a loop of nodes/positions for probe selection or for creating an edge loop or capping surface.			**Ctrl**+**Shift**+**L**	
	Selection options	Select position for loop creation.			
		Toggle between an open or closed loop.		**Ctrl**+**O**	
		Closed loop options	Toggle between using the direct or node loop closing path between the first and last nodes.	**Ctrl**+**D**	
			Toggle between using the direct or feature path between the first and last nodes.	**Ctrl**+**E**	
			Toggle between using the direct or zone boundary path between the first and last nodes.	**Ctrl**+**B**	
		Toggle between using the node path or the direct path between the last two nodes selected.		**Ctrl**+**P**	
		Toggle between using the feature path or the direct path between the last two nodes selected.		**Ctrl**+**F**	
		Toggle between using the zone boundary path or the direct path between the last two nodes selected.		**Ctrl**+**Z**	
	Create options	Select all nodes on the loop path.		**Ctrl**+**J**	
		Create a capping surface based on the nodes selected.		**Ctrl**+**K**	
		Create an edge loop based on the nodes selected.		**Ctrl**+**L**	
List Selection Options mode	Enter List Selection options mode. Contains options for controlling selection lists in dialog boxes based on zones or objects			**Ctrl**+**Shift**+**B**	

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Feature	Action		Keys	Onscreen Tool
	selected in the graphics window. You can also select face zone labels in the tree based on zones selected in the graphics window.			
	Zone selection	Add the selected zones to the current selections in the dialog box list.	**Ctrl**+**Q**	
		Remove the selected zones from the current selections in the dialog box list.	**Ctrl**+**S**	
		Set the selected zones as the current selections in the dialog box list.	**Ctrl**+**D**	
	Object selection	Add the selected objects to the current selections in the dialog box list.	**Ctrl**+**J**	
		Remove the selected objects from the current selections in the dialog box list.	**Ctrl**+**K**	
		Set the selected objects as the current selections in the dialog box list.	**Ctrl**+**L**	
	Face Zone Label selection	Add the face zone labels corresponding to zones selected to the current selections in the tree.	**Ctrl**+**F**	
		Remove the face zone labels corresponding to zones selected from the current selections in the tree.	**Ctrl**+**G**	
		Set the face zone labels corresponding to zones selected as the current selections in the tree.	**Ctrl**+**R**	
	Exit list selection options mode.		**F2**	
Color Options mode	Enter the Color Options mode.		**Ctrl**+**Shift**+**C**	
	Color the items by geometry recovery attribute.		**Ctrl**+**G**	
	Color the items by object.		**Ctrl**+**O**	
	Color the items by zone.		**Ctrl**+**Z**	
	Color the items based on normals.		**Ctrl**+**N**	
	Toggle the color palette.		**Ctrl**+**P**	
	Randomize the colors used in the display.		**Ctrl**+**R**	
	Exit color options mode.		**F2**	
Conditional Display Options mode	Enter the Conditional Display Options mode.		**Ctrl**+**Shift**+**D**	
	Isolate zones/objects with similar surface area to selected.		**Ctrl**+**S**	

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Feature	Action	Keys	Onscreen Tool
	Isolate zones/objects with surface area less than or equal to selected.	**Ctrl**+**I**	
	Isolate zones/objects of similar surface curvature.	**Ctrl**+**C**	
	Isolate zones/objects with neighborhood.	**Ctrl**+**N**	
	Show all zones/objects in the graphics window; unhide previously hidden zones/objects.	**Ctrl**+**Shift**+**A**	
	Exit conditional display options mode.	**F2**	
Miscellaneous Tools mode (only available when geometry and meshing objects are present)	Enter the Miscellaneous Tools mode.	**Ctrl**+**T**	
	Preview sizes on selected/all zones.	**Ctrl**+**P**	
	Draw size contours on selected/all zones.	**Ctrl**+**C**	
	Trace path between selected entities.	**Ctrl**+**T**	
	Join selected face zones using the options in the **Join** dialog box.	**Ctrl**+**J**	
	Intersect selected face zones using the options in the **Intersect** dialog box.	**Ctrl**+**I**	
Wrap face zones	Enables you to improve feature capture and/or rezone wrap face zones. • When wrap face zones are selected, they are rezoned. • When wrap face zones and edge zones are selected, feature capture on the wrap face zones is improved and the face zones are rezoned. • When wrap face zones and edge entities are selected, feature capture on the wrap face zones is improved.	**Ctrl**+**Shift**+**F**	

C.1.1. Entity Information

The information displayed for the selected entities using the hot key **Ctrl**+**I** is as follows:

Entity selected	Information displayed
Zone	Zone name Zone ID Location Zone type Object that the zone is included in Face zone labels that the zone is included in

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Entity selected	Information displayed
	Regions that the zone is included in Size functions
Face	Face name Zone ID Nodes Adjoining cell(s) Face skewness (if applicable) Periodic twins (if applicable) Child faces (if applicable)
Edge	Edge name Zone ID Adjoining faces
Cell	Cell name Zone ID Nodes Faces Cell center Cell skewness Cell simplex radius Cell size
Node	Node name Zone ID Location Adjoining faces (if applicable) Adjoining cells (if applicable)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Bibliography

Important:

Under U.S. and international copyright law, Ansys, Inc. is unable to distribute copies of the papers listed in the bibliography, other than those published internally by Ansys, Inc. Use your library or a document delivery service to obtain copies of copyrighted papers.

[1] T. J. Baker. Three Dimensional Mesh Generation By Triangulation of Arbitrary Point Sets. *Technical Report AIAA-87-1124*. American Institute of Aeronautics and Astronautics. 1987.

[2] T. J. Barth. Three-Dimensional Unstructured Grid Generation via Incremental Insertion and Local Optimization In Software Systems for Surface Modeling and Grid Generation. Langley Research Center, Hampton, Virginia, NASA Conference Publication. 3143. April 1992.

[3] K. R. Blake. A 3D Delaunay Unstructured Grid Generator Applied to Trains, Planes, and Automobiles. *Technical Report AIAA93-0673*. American Institute of Aeronautics and Astronautics. 1993.

[4] A. Bowyer. Computing Dirichlet Tessellations. *The Computer Journal*. 24(2). 162–166. May 1981.

[5] M. Garland. Surface Simplification Using Quadric Error Metrics. *24th Annual Conference on Computer Graphics and Interactive Techniques*. 209–216. 1997.

[6] D. G. Holmes. The Generation of Unstructured Triangular Meshes Using Delaunay Triangulation. J. Hauser. *Numerical Grid Generation in Computational Fluid Mechanics 88*. 643–652. Pineridge Press Limited. 1988.

[7] Y. Kallinderis. Hybrid Prismatic/Tetrahedral Grid Generation for Complex Geometries Technical Report AIAA-95-0211. *AIAA 33rd Aerospace Sciences Meeting and Exhibit Reno, Nevada*. 1995.

[8] Y. Kallinderis. Prismatic Grid Generation for Three-Dimensional Complex Geometries. *AIAA Journal*. 31(10). 1850–1856. October 1993.

[9] L. Kobbelt. A Shrink Wrapping Approach to Remeshing Polygonal Surfaces. *Computer Graphics Forum*. 18(3). 119–130. 1999.

[10] C. L. Lawson. Properties of n-dimensional Triangulations. *Computer-Aided Geometric Design*. 3. 231–246. December 1986.

[11] R. Löhner. Generation of Three-Dimensional Unstructured Grids by the Advancing Front Method Technical Report, AIAA-88-0515. *American Institute of Aeronautics and Astronautics*. 1988.

[12] D. J. Mavriplis. Adaptive Mesh Generation for Viscous Flows Using Delaunay Triangulation. *Journal of Computational Physics*. 90(12). 271–291. October 1990.

[13] J. D. Müller. A Frontal Approach for Node Generation in Delaunay Triangulations. *International Journal For Numerical Methods in Fluids*. 17(3). 241–255. August 1993.

[14] S. Pirzadeh. Unstructured Viscous Grid Generation by the Advancing-Layers Method. *AIAA Journal*. 32(8). 1735–1737. August 1994.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

[15] S. Pirzadeh. Three-Dimensional Unstructured Viscous Grids by the Advancing-Layers Method. *AIAA Journal*. 34(1). 43–49. January 1996.

[16] W. R. Quadros. Finite Element Mesh Sizing for Surfaces Using Skeleton. *In Proc 13th Int Meshing Roundtable*. 2004.

[17] J. Z. Wang. An Adaptive Cartesian Grid Generation Method for 'Dirty' Geometry. *International Journal For Numerical Methods in Fluids*. 39. 703–717. 2002.

[18] D. F. Watson. Computing the n-dimensional Delaunay tessellation with application to Voronoi polytopes. *The Computer Journal*. 24(2). 167–172. May 1981.

[19] N. P. Weatherill. A Method For Generating Irregular Computational Grids in Multiply Connected Planar Domains. *International Journal For Numerical Methods in Fluids*. 8(2). 181–197 . February 1988.

[20] M. A. Yerry. Automatic Three-Dimensional Mesh Generation by the Modified-Octree Technique. *International Journal for Numerical Methods in Engineering*. 20. 1965–1990. 1984.

[21] J. Zhu. Background Overlay Grid Size Functions. *In Proc 11th Int Meshing Roundtable*. 2002.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Part III: Solution Mode

The section describes how to use Ansys Fluent in solution mode.

- Using This Manual (p. dcccxcvii), provides typographical and mathematical conventions.
- Graphical User Interface (GUI) (p. 901), describes the mechanics of using the graphical user interface and the GUI on-line help.
- Text User Interface (TUI) (p. 945), describes the mechanics of using the text interface and the TUI on-line help. (See the Text Command List for information about specific text interface commands.)
- Reading and Writing Files (p. 947), describes the files that Ansys Fluent can read and write, including picture files.
- Unit Systems (p. 1033), describes how to use the standard and custom unit systems available in Ansys Fluent.
- Fluent Expressions Language (p. 1037), describes how to use the Ansys Fluent Expression Language.
- Reading and Manipulating Meshes (p. 1129), describes the various sources of computational meshes and explains how to obtain diagnostic information about the mesh and how to modify it by scaling, translating, and other methods. This chapter also contains information about the use of non-conformal meshes.
- Cell Zone and Boundary Conditions (p. 1299), describes the different types of boundary conditions available in Ansys Fluent, when to use them, how to define them, and how to define boundary profiles and volumetric sources and fix the value of a variable in a particular region. It also contains information about porous media and lumped parameter models.
- Physical Properties (p. 1597), describes the physical properties of materials and the equations that Ansys Fluent uses to compute the properties from the information that you input.
- Modeling Basic Fluid Flow (p. 1737), describes the governing equations and physical models used by Ansys Fluent to compute fluid flow (including periodic flow, swirling and rotating flows, compressible flows, and inviscid flows), as well as the inputs you need to provide to use these models.
- Modeling Flows with Moving Reference Frames (p. 1769), describes the use of single moving reference frames, multiple moving reference frames, and mixing planes in Ansys Fluent.
- Managing Motion Definitions (p. 1787), describes how to create and manage motion definitions in Ansys Fluent.
- Managing Auxiliary Geometry Definitions (p. 1791), describes how to create and manage auxiliary geometry definitions in Ansys Fluent.
- Modeling Flows Using Sliding and Dynamic Meshes (p. 1797), describes the use of sliding and deforming meshes in Ansys Fluent.

- Modeling Turbomachinery Flows (p. 1929), describes the turbomachinery-specific guided workflow and the various capabilities for modeling turbomachinary in Ansys Fluent.
- Modeling Turbulence (p. 2061), describes the use of the turbulent flow models in Ansys Fluent.
- Modeling Thermal Energy (p. 2163), describes the use of the physical models in Ansys Fluent to compute heat transfer(including convective and conductive heat transfer, natural convection, radiative heat transfer, periodic heat transfer, and heat exchanger models), as well as the inputs you need to provide to use these models
- Modeling Hypersonic Flow (p. ?), describes the use of the HIgh Speed Numerics
- Fluent's Virtual Blade Model (p. 2331), describes the use of Fluent's Virtual Blade Model (VBM)
- Modelling with Finite-Rate Chemistry (p. 2355), describes the use of the finite-rate chemistry models in Ansys Fluent. This chapter also provides information about modeling species transport in non-reacting flows.
- Modelling of Turbulent Combustion With Reduced Order (p. 2471), describes the use of the reduced order combustion models in Ansys Fluent.
- Modeling Pollutant Formation (p. 2585), describes the use of the models for the formation of NOx and soot in Ansys Fluent.
- Modeling Engine Ignition (p. 2571), describes the use of the engine ignition models in Ansys Fluent.
- Predicting Aerodynamically Generated Noise (p. 2625), describes the use of the acoustics model in Ansys Fluent.
- Modeling Discrete Phase (p. 2671), describes the use of the discrete phase models in Ansys Fluent.
- Modeling Macroscopic Particles (p. 2893), describes the use of the macroscopic particle models in Ansys Fluent.
- Modeling Multiphase Flows (p. 2915), describes the use of the general multiphase models in Ansys Fluent (VOF, mixture, and Eulerian).
- Modeling Solidification and Melting (p. 3219), describes the use of the solidification and melting model in Ansys Fluent.
- Modeling Fluid-Structure Interaction (FSI) Within Fluent (p. 3227), describes the use of a structural model for intrinsic fluid-structure interaction (FSI) simulations in Ansys Fluent.
- Modeling Eulerian Wall Films (p. 3239), describes the use of the Eulerian wall film model in Ansys Fluent.
- Modeling Electric Potential Field and Electrochemistry Models (p. 3265), describes the use of the electric potential model in Ansys Fluent.
- Modeling Batteries (p. 3299), describes the use of the battery models in Ansys Fluent.
- Modeling Fuel Cells (p. 3401), describes the use of the fuel cell models in Ansys Fluent.
- Modeling Magnetohydrodynamics (p. 3497), describes the use of the magnetohydrodynamics models in Ansys Fluent.

- Modeling Continuous Fibers (p. 3521), describes the use of the continuous fiber model in Ansys Fluent.
- Creating Reduced Order Models (ROMs) (p. 3561), describes the use of the reduced order model in Ansys Fluent.
- Using the Solver (p. 3571), describes the use of the Ansys Fluent solvers.
- Using the Fluent Native GPU Solver (p. 3755), describes the use of the GPU powered Ansys Fluent solver.
- Adapting the Mesh (p. 3773), describes the use of the solution-adaptive mesh refinement feature in Ansys Fluent.
- Creating Surfaces and Cell Registers for Displaying and Reporting Data (p. 3813), describes how to create surfaces in the domain on which you can examine Ansys Fluent solution data, and cell registers to visualize and perform operations on cell groupings.
- Displaying Graphics (p. 3869), describes the use of the graphics tools to examine your Ansys Fluent solution.
- Reporting Alphanumeric Data (p. 4097), describes how to obtain reports of fluxes, forces, surface integrals, and other solution data.
- Field Function Definitions (p. 4167), describes the flow variables that appear in the variable selection drop-down lists in Ansys Fluent dialog boxes, and tells you how to create your own custom field functions.
- Parallel Processing (p. 4293), describes the use of the parallel processing features in Ansys Fluent. This chapter also provides information about partitioning your mesh for parallel processing.
- Running Ansys Fluent on Arm Compute Nodes (p. 4355), describes the running of Ansys Fluent on the 64-bit extension of the Arm architecture (aarch64), along with the Armv8.2-A extensions.
- Using Simulation Reports (p. 4359), describes the creation of simulation reports in Ansys Fluent.
- Performing Parametric Studies (p. 4387), describes how to set up and analyze a parameric analysis directly within Ansys Fluent.
- Remotely Accessing Your Simulations Using Ansys Fluent's Web Interface (p. 4445), describes how to access your Ansys Fluent session using a browser-based web interface where you can interact with your settings and visualize your results remotely.
- Design Analysis and Optimization (p. 4491), describes the use of the adjoint solver and the mesh morpher/optimizer. These features allow you to modify the design and solve optimization problems in Ansys Fluent.
- Performing System Coupling Simulations Using Fluent (p. 4647), describes connecting Ansys Fluent to external solvers.
- Customizing Fluent (p. 4681), provides information about using Ansys ACT to create simulation wizards that can be run from either stand-alone Fluent or Fluent in Workbench.
- Task Page Reference Guide (p. 4683), describes the use of the task pages within the Ansys Fluent user interface.

- Ribbon Reference Guide (p. 5347), describes the **File** ribbon tab and other dialog boxes that are available from the ribbon at the top of the Ansys Fluent user interface.
- Appendix A: Ansys Fluent Model Compatibility (p. 5667), presents a series of tables outlining the compatibility of several Ansys Fluent model categories.
- Appendix B: Ansys Fluent File Formats (p. 5671), presents information about the contents and formats of Ansys Fluent case, data, and mesh morpher/optimizer files.
- Appendix C: Controlling CHEMKIN-CFD Solver Parameters Using Text Commands (p. 5697), describes the available text commands that can be used to control or improve the performance and convergence behavior of the CHEMKIN-CFD solver withinAnsys Fluent, as well as their default values and their recommended use.
- Appendix D: Nomenclature (p. 5709), presents a brief listing of the physical properties and fluid dynamic quantities used throughout the Ansys Fluent documentation.
- Bibliography (p. 5713) presents the bibliography for the previous chapters.

Using This Manual

This preface is divided into the following sections:

1. Typographical Conventions
2. Mathematical Conventions

1. Typographical Conventions

Several typographical conventions are used in this manual's text to help you find commands in the user interface.

- Different type styles are used to indicate graphical user interface items and text interface items. For example:

  Iso-Surface dialog box

  `surface/iso-surface` text command

- The text interface type style is also used when illustrating exactly what appears on the screen to distinguish it from the narrative text. In this context, user inputs are typically shown in boldface. For example,

```
solve/initialize/set-fmg-initialization

 Customize your FMG initialization:
   set the number of multigrid levels [5]

   set FMG parameters on levels ..

    residual reduction on level 1 is:  [0.001]
    number of cycles on level 1 is:  [10]  100

    residual reduction on level 2 is:  [0.001]
    number of cycles on level 2 is:  [50]  100
```

- Mini flow charts are used to guide you through the ribbon or the tree, leading you to a specific option, dialog box, or task page. The following tables list the meaning of each symbol in the mini flow charts.

Table 7: Mini Flow Chart Symbol Descriptions

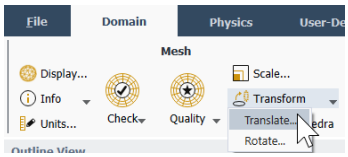
Symbol	Indicated Action
	Look at the ribbon
	Look at the tree
	Double-click to open task page
	Select from task page
	Right-click the preceding item

For example,

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Domain → Mesh → Transform → Translate...

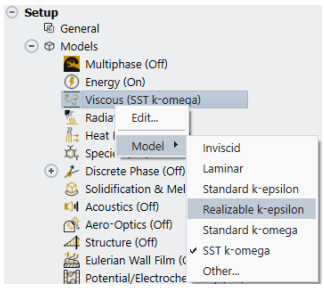
indicates selecting the **Domain** ribbon tab, clicking **Transform** (in the **Mesh** group box) and selecting **Translate...**, as indicated in the figure below:



And

Setup → Models → Viscous Model → Realizable k-epsilon

indicates expanding the **Setup** and **Models** branches, right-clicking **Viscous**, and selecting **Realizable k-epsilon** from the **Model** sub-menu, as shown in the following figure:

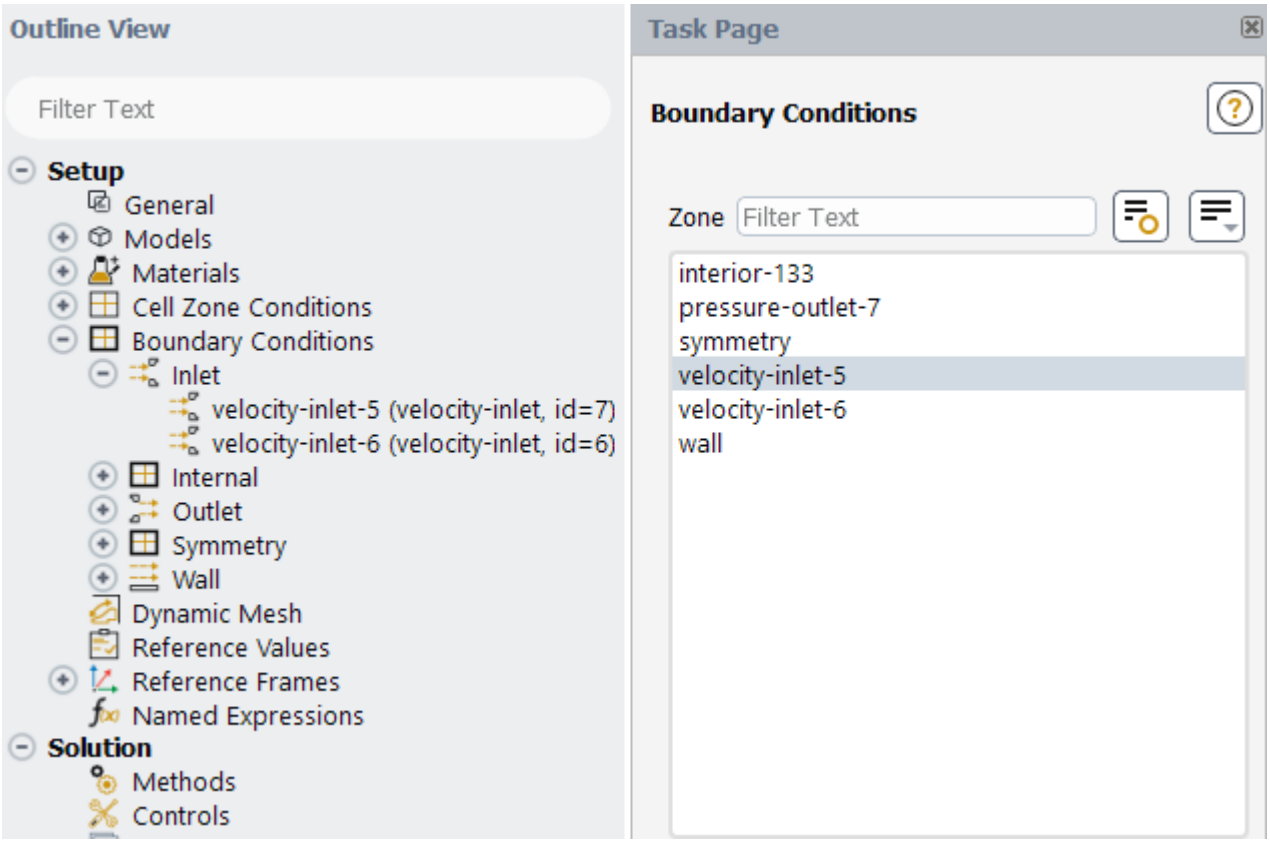


And

Setup → Boundary Conditions → velocity-inlet-5

indicates opening the task page as shown below:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



In this manual, mini flow charts usually accompany a description of a dialog box or command, or a screen illustration showing how to use the dialog box or command. They show you how to quickly access a command or dialog box without having to search the surrounding material.

- In-text references to **File** ribbon tab selections can be indicated using a "/". For example **File/Write/Case...** indicates clicking the **File** ribbon tab and selecting **Case...** from the **Write** submenu (which opens the **Select File** dialog box).

2. Mathematical Conventions

- Where possible, vector quantities are displayed with a raised arrow (for example, $\vec{a}$, $\vec{A}$). Boldfaced characters are reserved for vectors and matrices as they apply to linear algebra (for example, the identity matrix, $\mathbf{I}$).

- The operator ∇, referred to as grad, nabla, or del, represents the partial derivative of a quantity with respect to all directions in the chosen coordinate system. In Cartesian coordinates, ∇ is defined to be

$$\frac{\partial}{\partial x}\vec{\imath}+\frac{\partial}{\partial y}\vec{j}+\frac{\partial}{\partial z}\vec{k} \tag{10}$$

  ∇ appears in several ways:

  – The gradient of a scalar quantity is the vector whose components are the partial derivatives; for example,

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\nabla p = \frac{\partial p}{\partial x}\vec{i} + \frac{\partial p}{\partial y}\vec{j} + \frac{\partial p}{\partial z}\vec{k} \tag{11}$$

– The gradient of a vector quantity is a second-order tensor; for example, in Cartesian coordinates,

$$\nabla(\vec{v}) = \left(\frac{\partial}{\partial x}\vec{i} + \frac{\partial}{\partial y}\vec{j} + \frac{\partial}{\partial z}\vec{k}\right)\left(v_x\vec{i} + v_j\vec{j} + v_z\vec{k}\right) \tag{12}$$

This tensor is usually written as

$$\begin{pmatrix} \frac{\partial v_x}{\partial x} & \frac{\partial v_x}{\partial y} & \frac{\partial v_x}{\partial z} \\ \frac{\partial v_y}{\partial x} & \frac{\partial v_y}{\partial y} & \frac{\partial v_y}{\partial z} \\ \frac{\partial v_z}{\partial x} & \frac{\partial v_z}{\partial y} & \frac{\partial v_z}{\partial z} \end{pmatrix} \tag{13}$$

– The divergence of a vector quantity, which is the inner product between ∇ and a vector; for example,

$$\nabla \cdot \vec{v} = \frac{\partial v_x}{\partial x} + \frac{\partial v_y}{\partial y} + \frac{\partial v_z}{\partial z} \tag{14}$$

– The operator $\nabla \cdot \nabla$, which is usually written as ∇^2 and is known as the Laplacian; for example,

$$\nabla^2 T = \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} \tag{15}$$

$\nabla^2 T$ is different from the expression $(\nabla T)^2$, which is defined as

$$(\nabla T)^2 = \left(\frac{\partial T}{\partial x}\right)^2 + \left(\frac{\partial T}{\partial y}\right)^2 + \left(\frac{\partial T}{\partial z}\right)^2 \tag{16}$$

- An exception to the use of ∇ is found in the discussion of Reynolds stresses in Turbulence in the *Fluent Theory Guide*, where convention dictates the use of Cartesian tensor notation. In this chapter, you will also find that some velocity vector components are written as u, v, and w instead of the conventional v with directional subscripts.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 1: Graphical User Interface (GUI)

The Fluent graphical interface consists of a ribbon (tabs), a toolbar, a tree, a task page, a graphics toolbar, graphics windows, and a console, which is a textual command line interface (described in Text User Interface (TUI) (p. 945)). You will also have access to the dialog boxes via the ribbon or task pages.

1.1. GUI Components

1.2. Customizing the Graphical User Interface

1.3. Setting User Preferences/Options

1.4. Fluent Graphical User Interface Other Languages

1.5. Having the Session Close After Sitting Idle

1.6. Using the Help System

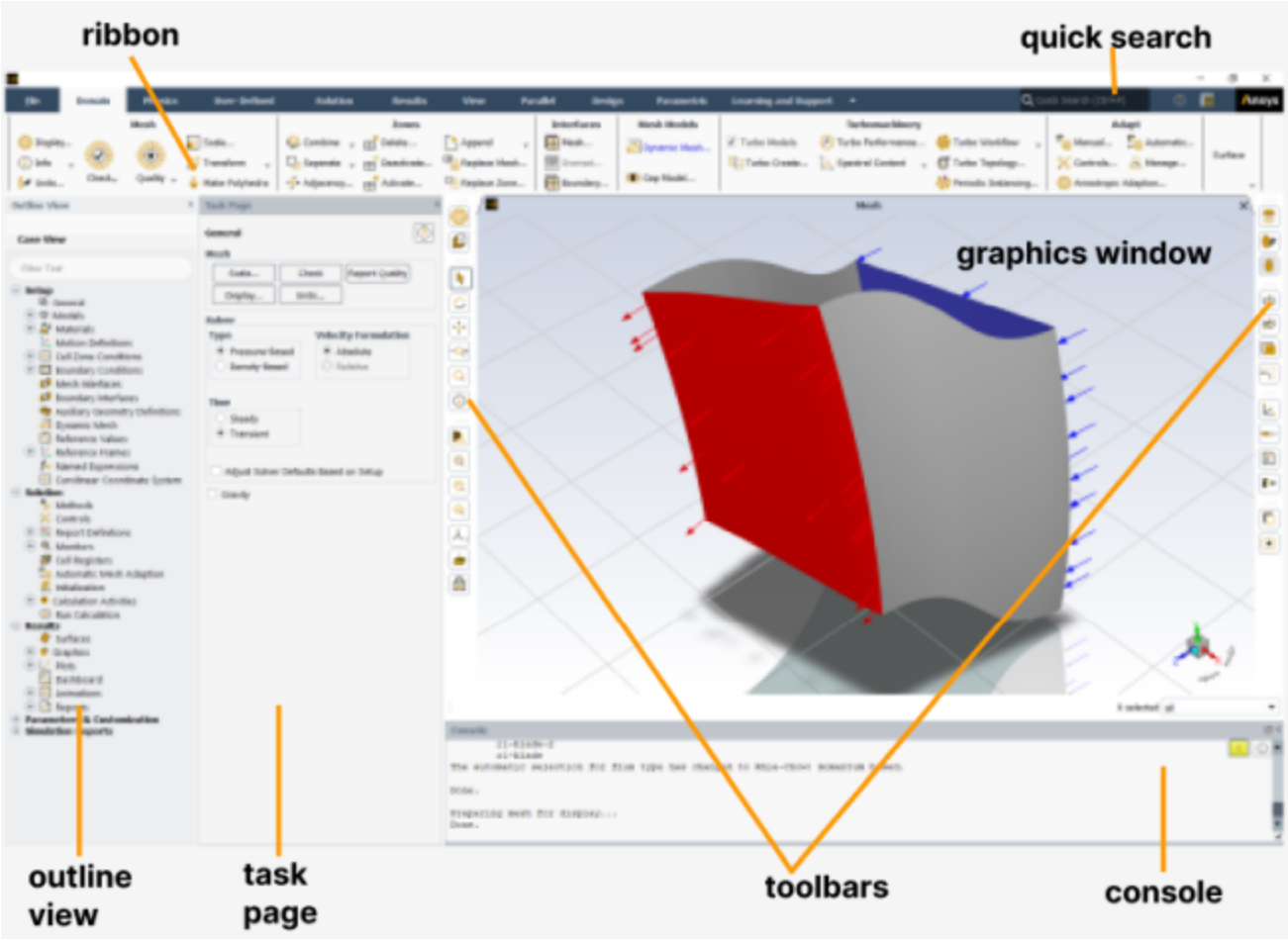
1.1. GUI Components

The graphical user interface (GUI) is made up of seven main components, which are described in detail in the subsequent sections: the ribbon, the tree, toolbars, task pages, a console, dialog boxes, and graphics windows. When you use the GUI, you will be interacting with one of these components at all times. The GUI will change depending on whether you are in meshing mode (as described in the Fluent Meshing section of the User's Guide) or solution mode (as described in this guide, and as shown in Figure 1.1: The GUI Components (p. 902)). For details on how to switch between these modes, see Switching Between Meshing and Solution Modes (p. 207) in the Getting Started (p. 1) part of this manual.

Many of the GUI elements can be moved or tabbed together to suit your preferences. You can also modify attributes of the GUI to better match your platform environment. These are described in Customizing the Graphical User Interface (p. 935) and Setting User Preferences/Options (p. 937).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.1: The GUI Components



For additional information, see the following sections:

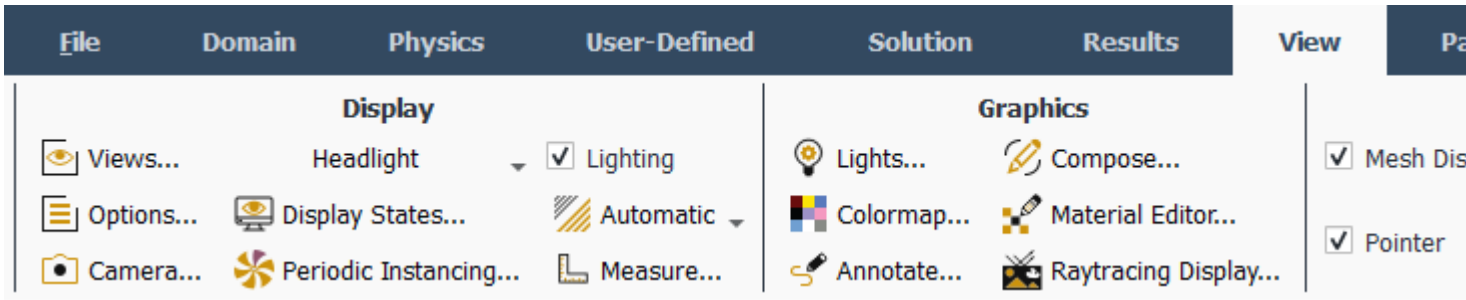
1.1.1. The Ribbon
1.1.2. The Outline View
1.1.3. Graphics Windows
1.1.4. Quick Search
1.1.5. Toolbars
1.1.6. Task Pages
1.1.7. The Console
1.1.8. Dialog Boxes
1.1.9. Quick Property Editor for Boundaries

1.1.1. The Ribbon

The ribbon, located at the top of the Fluent GUI, is the primary method for setting up and running your simulation. It facilitates access to the most commonly used items, with tabs nominally arranged in a left to right workflow for a typical simulation. Contents within each tab are grouped with related content and are organized logically to accommodate varied projects and priorities. You have the option to minimize the ribbon by clicking , located to the right of the ribbon tabs.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.2: The Fluent Ribbon



Often, making a selection in the ribbon opens a dialog box. In some instances, a ribbon selection opens a task page.

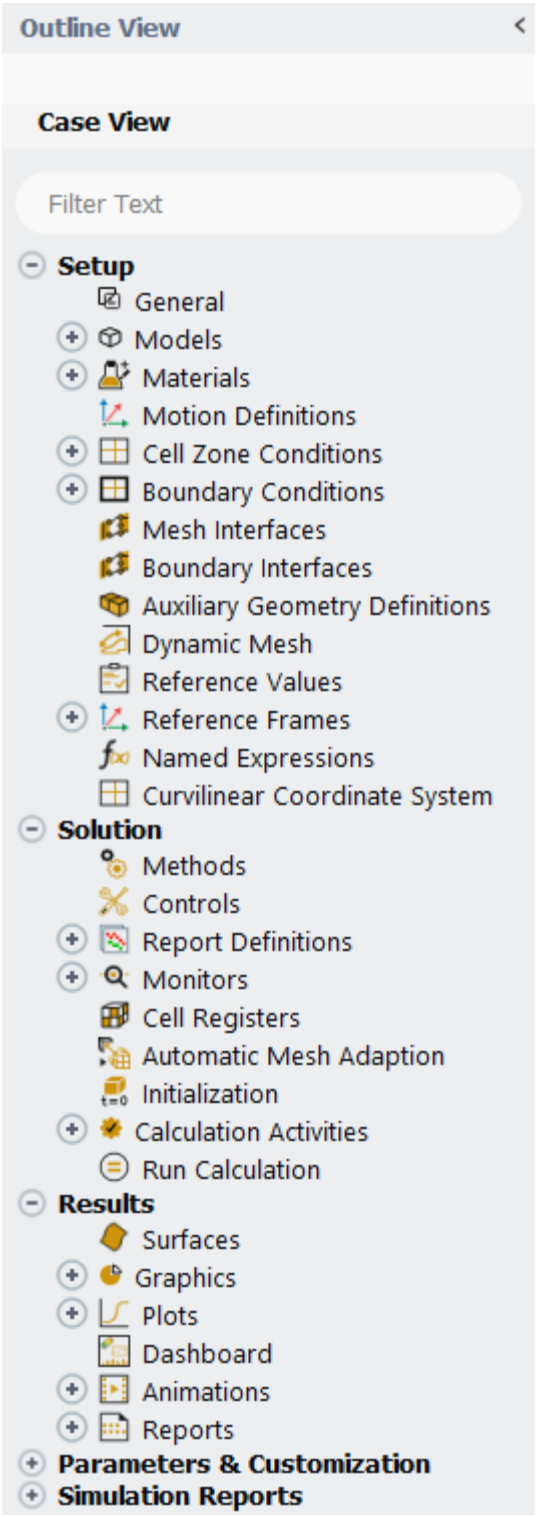
1.1.2. The Outline View

The outline view tree, located on the left side of the Fluent GUI, is an interactive representation of your project. Its items are listed in an order that complements the typical problem setup you follow in the ribbon. Branches can be expanded and collapsed to suit your preferences.

You can select multiple related items by holding **Shift** or **Ctrl** and left-clicking. Right-clicking selected items in the tree provides a list of common commands. Typographical Conventions (p. dcccxcvii) shows the convention used to describe this right-click action, when it is referred to in the documentation.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.3: The Fluent Outline View



You can perform wildcard and regular expression searches in the tree using the filter text entry box at the top of the tree; see Filter Text Entry Boxes (p. 926) for additional information.

Double-clicking many of the branches under **Setup**, **Solution**, or **Results** displays the corresponding **Task Page** to the right of the tree. For additional information, see (Task Pages (p. 920)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Copy and paste is available for copying items from the outline view to the clipboard so that you can paste them elsewhere in the same session or into another session without having to hold down the left-mouse-button like you do for a drag and drop operation. Copy and paste is available for:

1.1.2.1. Drag and Drop, Copy and Paste

You can use the drag and drop operation on the Outline View tree to varied effects, depending on what you are dragging and where you are dropping it.

Note that Fluent displays a plus (⊞) icon for valid drop operations and a invalid (🚫) icon for operations or locations that are not allowed. Tooltips indicate the type of operation that will be performed.

Within the Same Session:

- Open files by dropping them onto the open Fluent session.

  The supported file types include:

  – Case and data (`*.cas.h5`, `*.cas`, `*.cas.gz`, `*.dat.h5`, `*.dat`, `*.dat.gz`, `*.msh.h5`, `*.msh`, `*.msh.gz`)

  – Journals (`*.jou`, `*.log`)

  – Profiles (`*.csv`)

  – PDFs (`*.pdf`)

- **Cell Zones, Boundary Conditions, Mesh Interfaces**

  – Copies the conditions of the dragged boundary onto the boundary where the dragged boundary is dropped.

  – Drop cell zones and boundary conditions on the graphics window to display them (additive). *You can select a surface in the graphics window and hide it via the context menu option (accessed by right-clicking).*

- **Report Definitions**

  – Creates a copy of the dragged report definition when dropped on the **Report Definitions** branch.

  – When one report definition is dropped on another, the settings are copied from the first to the second.

- **Surfaces**

  – When one or more surfaces are dropped on the parent branch, new copies of those surfaces are created with default names.

  – When one surface is dropped on another, the settings are copied from the first surface to the second.

- **Graphics**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - When individual graphics objects are dropped onto the parent branch (for example, the **Mesh** branch when dragging a mesh object), a new copy of that object is created with a default name.

  - When an object is dropped on another child object of the same type (for example, dropping a mesh object onto another mesh object), the properties of the dragged object are copied to the dropped object.

  - Drop objects on the graphics window to display them. This functions like the **Add to** right-click option. *Note that once you have added multiple objects to the same window, you can right-click the graphics window and select **Save as Scene** to create a saved scene (Generating a Scene (p. 3935)) that persists with the case file (assuming you save it).*

- **Plots - XY and Cumulative**

  - When individual plot objects are dropped onto the parent branch a new copy of that object is created with a default name.

  - When an object is dropped on another child object of the same type (for example, dropping an XY plot object onto another XY plot object), the properties of the dragged object are copied to the dropped object.

  - Drop plot objects on the graphics window to display them. *This clears the previous display and only displays the most-recently dropped plot object.*

- **Scene**

  - When a scene object is dropped onto the parent branch a new copy of that scene is created with a default name.

  - When an scene object is dropped on another scene object, the properties of the dragged scene are copied to the dropped scene.

  - Drop scene objects on the graphics window to display them. This functions like the **Add to** right-click option.

Across Sessions:

Important:

The drag and drop of a single object will not create any more than one object in the new session, even if that object has dependencies that are not fulfilled in the new session. For example, if you drop **example-mesh-1** on a new session, you will get a **mesh-n** (where "n" is the next increment in mesh object names) object. But if **example-mesh-1** displays on **example-plane-surface** and that plane surface does not exist in the new session, the **mesh-n** object will be invalid, because the drop operation only created a new mesh object, not a new plane surface.

- **Cell Zones, Boundary Conditions, Mesh Interfaces**

  - Copies the conditions of the dragged boundary onto the boundary where the dragged boundary is dropped.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - Drop cell zones and boundary conditions on the graphics window to display them.

- **Report Definitions**

  - Creates a copy of the dragged report definition when dropped on the **Report Definitions** branch. If a report definition already exists with the same name, then a new default name is used.

  - When a single report definition is dropped on another, the settings are copied from the first session to the report definition in the second session.

- **Solution Methods, Solution Controls, Initialization, Run Calculation**

  - Copies the settings from the source session to the target session for any of these branches.

- **Animation Definitions**

  - Dragging the top-level **Animation Definitions** branch creates animation definition objects in the target session with objects of the same as in the source session. Any objects with the same names already exist in the target session are flagged. If you proceed with the operation, then the settings of the target objects of the same name are updated to match those of the source session.

  - Dragging individual animation definitions onto the top-level branch results in the same behavior as when you are dragging the whole **Animation Definitions** branch.

  - Dragging an individual animation definition onto a single animation definition in the target session will copy the settings of the source to the target.

- **Surfaces**

  - When one or more surfaces are dropped on the parent branch, new copies of those surfaces are created with default names, unless there are already surfaces with the same name in the target session. If you confirm to proceed with the drag and drop operation, the definitions of the conflicting surfaces will be updated to match that of the source surfaces.

  - When one surface is dropped on another, the settings are copied from the first surface to the second.

- **Graphics**

  - When individual graphics objects are dropped onto the parent branch (for example, the **Mesh** branch when dragging a mesh object), a new copy of that object is created with a default name.

  - When an object is dropped on another child object of the same type (for example, dropping a mesh object onto another mesh object), the properties of the dragged object are copied to the dropped object.

- **Plots - XY and Cumulative**

  - (XY Plots) When an individual plot object is dropped onto the parent branch a new copy of that object is created with a default name.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - (Cumulative Plots) When one or more cumulative plots are dropped on the parent branch, new copies of those plots are created with default names, unless there are already plots with the same names in the target session. If you confirm to proceed with the drag and drop operation, the definitions of the conflicting plots will be updated to match that of the source plots.

  - When an object is dropped on another child object of the same type (for example, dropping a plot object onto another plot object), the properties of the dragged object are copied to the dropped object.

- **Scene**

  - When scene objects are dropped onto the parent branch, new copies of those scene objects are created with default names.

  - When an scene object is dropped on another scene object, the properties of the dragged scene are copied to the dropped scene.

Copy and Paste

Copy and paste is available for copying items from the outline view to the clipboard so that you can paste them elsewhere in the same session or into another session without having to hold down the left-mouse-button like you do for a drag and drop operation. Copy and paste is available for:

- Boundary conditions
- Named expressions
- Report definitions, files, and plots
- Postprocessing objects (mesh, contours, and so on)

Note:

While drag and drop and copy and paste do work for multiphase boundary conditions, they are not available for phase level boundary conditions.

Export To File and Import From File

Export To File... and **Import From File...** are available as right-click options for certain branches in the outline view, including:

- Boundary conditions (except phase level boundary conditions)
- Named expressions
- Report definitions
- Postprocessing objects

These options allow you to save the settings of a single sub-branch (for example, a single boundary condition or named expression) or all of the items in the higher-level branch (for example, all of the boundaries or all of the named expressions) for later use. *These exports are saved in* `.tsv` *format.*

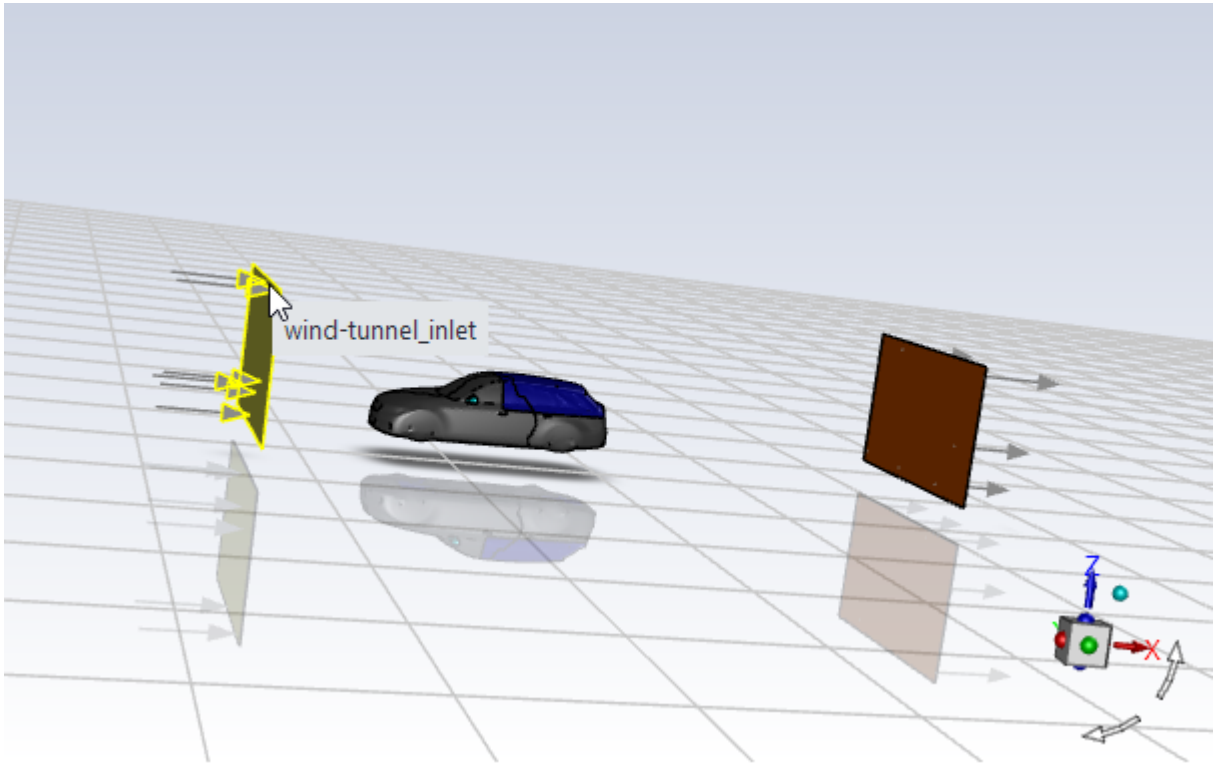
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.1.3. Graphics Windows

Graphics windows display the program's graphical output, and may be viewed within the Fluent application window or in separate windows. You can right-click the top of the graphics window to select whether you want new graphics windows to be tabbed or as additional sub-windows within the graphics window group space. By default, graphics windows viewed within the application window will be placed below the toolbar on the right, as shown in Figure 1.1: The GUI Components (p. 902).

If you have **Hover-Over Highlight** enabled under **Selections** in the **Appearance** branch of **Preferences** (accessed by selecting **Preferences...** in the **File** ribbon tab), then as you move the cursor over your model (with the mesh displayed) you will notice that the edges of the face you are hovering over are highlighted and the name of that surface is displayed in a tooltip, as shown in Figure 1.4: Hover-Over Highlight (p. 909). *You can control the thickness of the highlighted or selected edges using the **Edge Thickness** field, and the overall appearance of highlighted/selected surface using the **Front Edge Transparency** field (both listed under **Selections** in Preferences).*

Figure 1.4: Hover-Over Highlight



Using context menus in the graphics window you can access and modify settings directly on the model. Figure 1.5: Graphics Window Context Menu: Single-Selection (p. 910) and Figure 1.6: Graphics Window Context Menu: Multiple-Selection (p. 910) demonstrate some of the options available via right-click. *You can make multiple selections on the displayed graphics object by holding **Ctrl**.* Quickly Coloring Surfaces by Field Variable Value (p. 3897) contains more information on some of the right-click options.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.5: Graphics Window Context Menu: Single-Selection

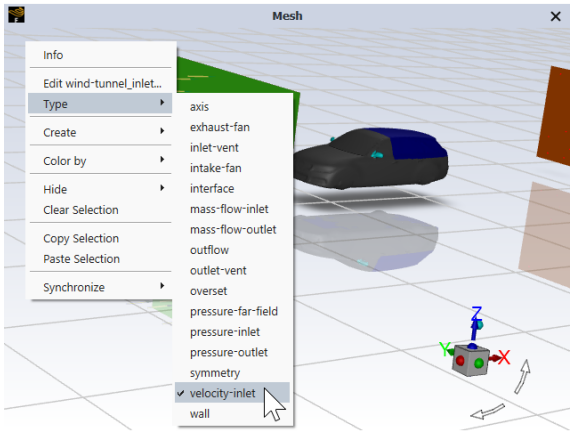
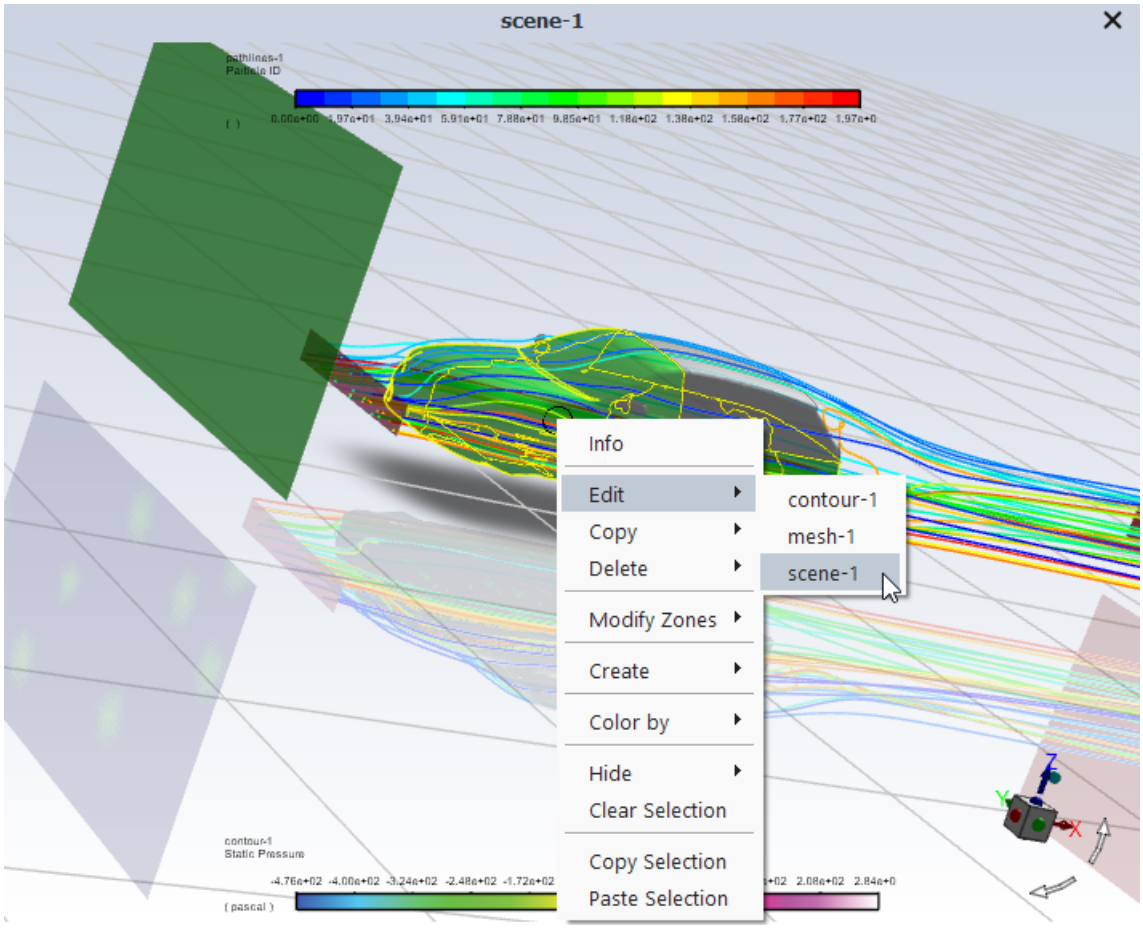


*Figure 1.5: Graphics Window Context Menu: Single-Selection (p. 910) shows right-clicking the inlet (selection highlighted in green) and specifying it as a **velocity-inlet**.*

Figure 1.6: Graphics Window Context Menu: Multiple-Selection



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.6: Graphics Window Context Menu: Multiple-Selection (p. 910) shows right-clicking the object when multiple surfaces are selected and a scene is displayed.

Note:

There are a few situations where a right-click in the graphics window will not open a context menu:

- Right-mouse-button click (**RMB Click**) is set as **Probe** and you right-click on the model when nothing is selected.
- Right-mouse-button click (**RMB Click**) is set as **Zoom** or **Fit to Window**.
- The graphics window is "reserved", that is, a window Fluent automatically generates while the solver is running (indicated by a gray Fluent icon). The reserved windows include residuals (Printing and Plotting Residuals (p. 3689)), report plots (Report Files and Report Plots (p. 4123)), and animation definitions (Creating an Animation Definition (p. 3714)).

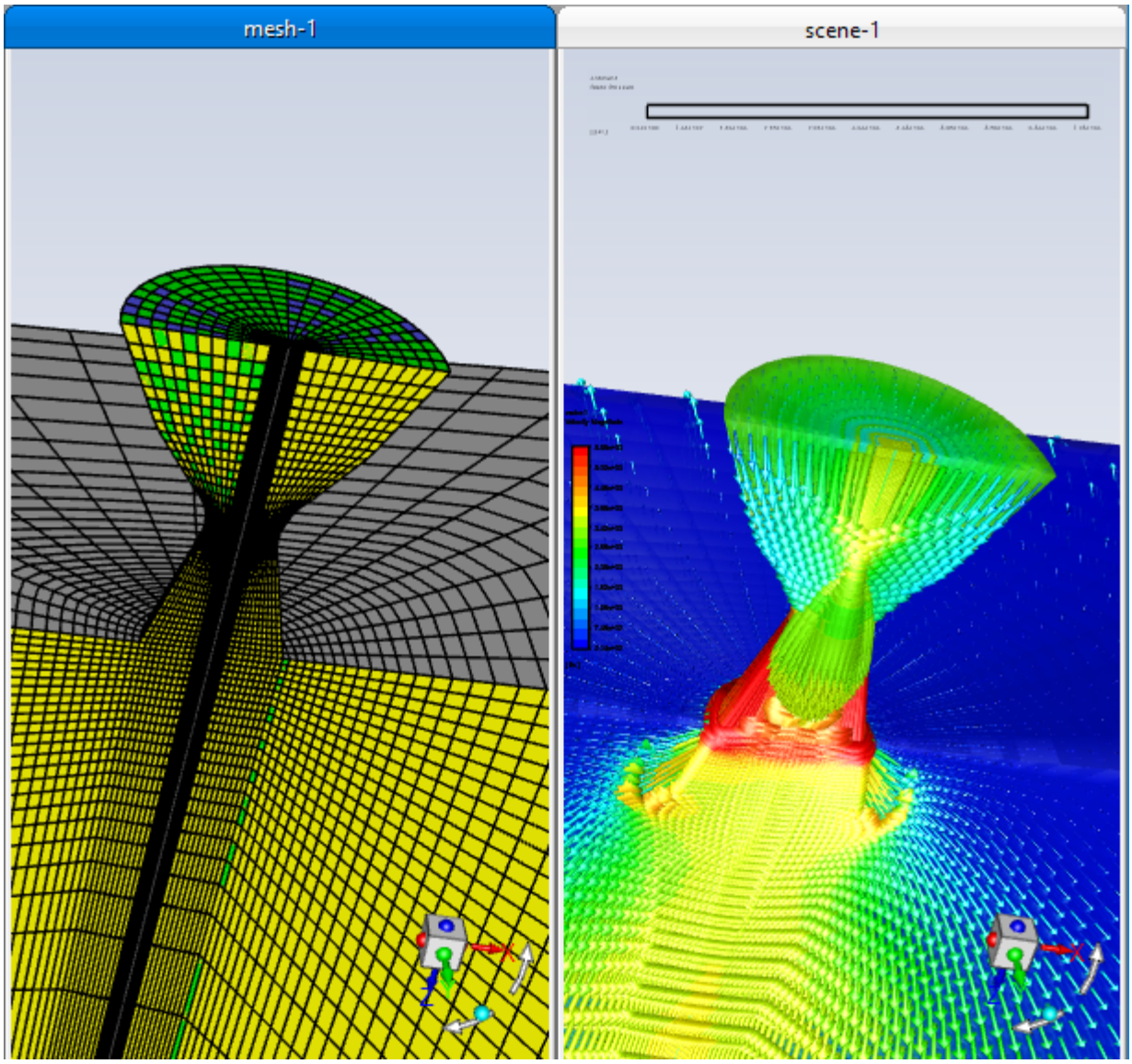
For additional information on mouse button settings, see Controlling the Mouse Button Functions (p. 4020).

In Figure 1.7: Displaying Two Graphics Windows (p. 912), two graphics windows are displayed by right-clicking the top of the graphics window and selecting **SubWindow View**, then creating two graphical displays and fitting them in the space accordingly.

In Figure 1.7: Displaying Two Graphics Windows (p. 912), two graphics windows are displayed by right-clicking the top of the graphics window and selecting **SubWindow View**, then creating two graphical displays and fitting them in the space accordingly.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.7: Displaying Two Graphics Windows



The **Display Options** dialog box (see Display Options Dialog Box (p. 5262)) can be used to change the attributes of the graphics window or to open another graphics window.

Important:

To cancel a display operation, press **Ctrl**+**c** while the data is being processed in preparation for graphical display. You cannot cancel the operation after the program begins to draw in the graphics window.

Important:

It is possible that the graphics window may become **[Out of Date]** (which would be indicated at the top of the graphics window), if you make changes to items that are already displayed. To resolve this out of date state, right-click in the graphics window and click **Refresh Display** in the context menu that appears.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

*If a context menu does not appear on a right-click of the graphics window, ensure that the right-mouse button is set to an action other than **Zoom**, **Probe**, or **Fit to Window**. Confirm and/or change mouse mappings in the **Navigation** branch of **Preferences** (**File** > **Preferences...**).*

*For the refresh of a mesh display, Fluent displays the surfaces that are currently selected in the global **Mesh Display** dialog box (accessed by clicking* *in the toolbar). In some instances this may result in the refreshed display not matching what was originally displayed.*

Copy to Clipboard

Right-clicking the graphics window tab allows you to select **Copy to Clipboard**. This places a copy of the current picture displayed in the selected graphics window tab onto your clipboard. The size of your graphics window affects the size of the text fonts used in the picture.

1.1.4. Quick Search

The search bar (upper right of the Fluent window or via **Ctrl** + **F**) allows you to quickly locate the commands or controls that you are looking for. Clicking the search results is equivalent to clicking the same control in the ribbon. Hovering over a search result highlights the location of the control in the ribbon. Clicking a text command search result automatically enters the text of the command, but it does not execute the command; you still have to press **Enter** in the console to execute the text command.

1.1.5. Toolbars

The Fluent GUI includes toolbars located within the application window. These toolbars provide shortcuts to performing common tasks in Fluent. The toolbars include a standard toolbar and an objects toolbar.

You can control which toolbars appear in the graphics window by enabling or disabling them in the **Graphics Toolbars** group box of the **View** ribbon tab (Figure 1.8: The View Ribbon Tab (p. 913)).

Figure 1.8: The View Ribbon Tab

1.1.5.1. The Standard Toolbar

The standard toolbar (Figure 1.9: The Standard Toolbar (p. 914)) contains options for getting help, arranging the graphical user interface, and visiting the Ansys website.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.9: The Standard Toolbar

The following is a brief description of each of the standard toolbar options:

- **Help** allows you to access the Fluent User's Guide (p. 1) for help topics, as well as additional learning resources. For more information, see Using the Help System (p. 942).
- **Arrange the workspace** provides you with several application window layout options. For example, you can choose to have the console docked in the **Messages** dialog box with the warning and error messages, allowing for a larger graphics window. These options also allow you to return to a standard layout if you accidentally rearrange (by dragging and dropping, or closing elements) the workspace in an undesirable manner.
- **Link to the Ansys Website** opens a link to the Ansys home page in your default browser.

1.1.5.2. The Graphics Toolbars

Figure 1.10: The Graphics Toolbars

The following tools are available in the graphics toolbar:

1.1.5.2.1. Mesh Display
1.1.5.2.2. Pointer Tools
1.1.5.2.3. View Tools
1.1.5.2.4. Visibility Tools
1.1.5.2.5. Copy Tools
1.1.5.2.6. Object Selection/Display Tools
1.1.5.2.7. Graphics Effects Tools
1.1.5.2.8. Additional Display Options

1.1.5.2.1. Mesh Display

The mesh display configuration options (Figure 1.11: Mesh Display (p. 914)) contains predefined settings for how a mesh and objects are displayed.

Figure 1.11: Mesh Display

The following is a brief description of the mesh display tools:

- **Mesh Display** opens the Mesh Display Dialog Box (p. 4688).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Front Faces Transparent** makes the front faces of the displayed object transparent, allowing you to see inside.

1.1.5.2.2. Pointer Tools

The pointer tools (Figure 1.12: The Pointer Tools (p. 915)) allow you to modify the way in which you view your model or select objects in the graphics window.

Figure 1.12: The Pointer Tools

The following is a description of each of the pointer tools:

- **Box Select** lets you select surfaces by box-selecting, that is clicking and dragging the mouse from left-to-right or right-to-left over your model. For more information, see Controlling the Mouse Button Functions (p. 4020).

- **Rotate View** lets you rotate your model about a central point in the graphics window. For more information, see Controlling the Mouse Button Functions (p. 4020).

- **Pan** allows you to pan horizontally or vertically across the view using the left mouse button. For more information, see Controlling the Mouse Button Functions (p. 4020).

- **Zoom In/Out** allows you to zoom in to and out of the model by holding the left mouse button down and moving the mouse down or up. For more information, see Controlling the Mouse Button Functions (p. 4020). You can also roll the view by holding the left mouse button down and moving the mouse left or right.

  Important:

  If you want to zoom in or out while the solver is iterating, click to assign **Zoom In/Out** to the left-mouse-button prior to beginning the calculation. *Roll zoom, using the middle scroll wheel, does not work while the solver is running.*

- **Zoom to Area** allows you to focus on any part of your model. After selecting this option, position the mouse pointer at a corner of the area to be magnified, hold down the left mouse button and drag open a box to the desired size, and then release the mouse button. The enclosed area will then fill the graphics window. Note that you must drag the mouse to the right in order to zoom in. To zoom out, you must drag the mouse to the left. For more information, see Controlling the Mouse Button Functions (p. 4020).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Probe - Print information about selected item** allows you to select items from the graphics windows and request information about displayed scenes. This behaves as a probe button. When you hover-over postprocessing graphics objects like contours, Fluent prints the name of the surface, the name of the graphics object, and the value of the displayed variable. For more information on clicking to probe, see Controlling the Mouse Button Functions (p. 4020).

  Note:

  (Parallel only) In some instances, the location selected using the mouse-probe in the graphics window may vary substantially from the location the code selects as the corresponding surface vertex. To make the mouse-probe and corresponding surface vertex match more closely, as they do when running with a single processor, disable the **Remove Partition Lines** option in the **Graphics** branch of **Preferences** (**File** > **Preferences...**).

  Important:

  For hover-over probe to work, **Hover-Over Probe Values** must be enabled in **Preferences** (located in the **Appearance** branch under **Selections**).

Hover Probe Limitations

– Not supported for global graphics objects (except for mesh) - these objects are hidden from the interface by default.

– Pathline and Vector graphics objects must be re-displayed after enabling the **Hover-Over Probe Values** option in **Preferences**, for the hover probe functionality to work.

– Vectors that are a part of a mirror plane or periodic repeat display are not highlighted when their value is shown.

– Pathlines:

→ The probe value will appear sometimes even when the probe marker is not directly on the pathline, but near to it.

→ When included in a scene, probe values are not available for pathlines.

→ Probe is not showing the surface name.

Note:

You can use **Ctrl**+**Shift**+**C** to copy a screenshot of the graphics window to the clipboard, which allows you to capture an image that includes the hover probe value.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.1.5.2.3. View Tools

The view tools (Figure 1.13: The View Tools (p. 917)) allow you to modify the way your model is displayed in the graphics window.

Figure 1.13: The View Tools

The following is a description of each of the view tools:

- **Perspective or Orthographic View** shows the 3D object on your 2D screen approximately how your eye would see it in real life. This button is only available when running Fluent in 3D.

  shows 3D objects to scale, ignoring perspective view. *When enabled, it makes it so the ruler is accurate*. This button is only available when running Fluent in 3D.

  Note:

  When **Orthographic** is selected for **Graphics view** in **Preferences** (**Appearance** branch), an additional **Ruler** option appears. Enabling **Ruler** makes it so that the ruler is displayed whenever the view is changed to orthographic. *You can still disable the ruler by toggling the ruler button* *).*

- **Fit to Window** adjusts the overall size of your model to take maximum advantage of the graphics window's width and height.

- **Last View** allows you to revert to the displayed objects previous location and orientation in the graphics window.

- **Next View** allows you to revert to the displayed objects previous location and orientation in the graphics window.

- **Set view** contains a drop-down of views, allowing you to display the model from the direction of the vector equidistant to all three axes, as well as in different axes orientations.

- **Synchronize Views** synchronizes the views of the displayed graphics objects in all the open graphics windows to that of the active graphics window. Graphics manipulations such

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

as panning and zooming performed in one window are simultaneously performed in the other linked graphics windows.

Note:

Any graphics windows opened after the **Synchronize Views** is clicked will not by synchronized by default. To synchronize newly opened windows with the other windows that are already linked, right-click in the window and select **Synchronize > Add this window** from the context menu that appears.

1.1.5.2.4. Visibility Tools

The visibility tools (Figure 1.14: The Visibility Tools (p. 918)) allow you to modify the appearance of objects in the graphics window.

Figure 1.14: The Visibility Tools

The following is a brief description of each of the visibility tools:

- **Axes Visibility** turns the axes display on and off.
- **Ruler Visibility** turns the ruler on and off. When enabled, it switches the view to *orthographic* (if it was previously set to *perspective*).
- **Titles Visibility** turns the titles display on and off.
- **Boundary Markers** enables/disables boundary markers at inlets and outlets when the mesh is displayed.

1.1.5.2.5. Copy Tools

The copy tools (Figure 1.15: The Copy Tools (p. 918)) allow you to copy an image of the graphics window to the clipboard and capture an image of the graphics window.

Figure 1.15: The Copy Tools

The following is a brief description of each of the copy tools:

- **Copy to Clipboard** copies the current picture displayed in the selected graphics window tab onto your clipboard. You can use **Ctrl**+**Shift**+**C** as a shortcut for this copy operation.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Save Picture** allows you to capture an image of the active graphics window. For more information, see Saving Picture Files (p. 1019).

1.1.5.2.6. Object Selection/Display Tools

The object selection/display tools (Figure 1.16: The Object Selection/Display Tools (p. 919)) allow you to modify the selection and display of objects in the graphics window.

Figure 1.16: The Object Selection/Display Tools

The following is a brief description of each of the object selection/display tools:

- **Show All** redisplays all hidden surfaces.
- **Hide Selected Surfaces** hides the selected surfaces from the display.
- **Make Selected Surfaces Transparent** makes any selected surfaces transparent. This operation is additive, so if you select surfaces and click this button, then select more surfaces and click it again, the second set of surfaces will also be made transparent. Clicking this button when no surfaces are selected removes all transparencies applied by this tool. Clicking this button when the selected surface is already transparent (from prior use of this button) removes the applied transparency from the selected surface.

  Note:

  Transparencies applied by this tool are temporary—they are not captured when you use the **Save as Scene** option available in the right-click context menu of the graphics window, nor are they saved in the case file.

- **Clear Selection** deselects the currently-selected surfaces in the graphics window. It will also remove any measurements that are displayed.

1.1.5.2.7. Graphics Effects Tools

The graphics effects tools (Figure 1.17: The Graphics Effects Tools (p. 919)) allow you to control which graphics effects are shown in the graphics window.

Figure 1.17: The Graphics Effects Tools

The following is a brief description of each of the graphics effects tools:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Reflections** redisplays all hidden surfaces.
- **Shadows** controls which shadowing effects are shown:
  - **Static Shadows** is the shadow of the model on the ground plane.
  - **Dynamic Shadows** is the shadow of the model on itself. These shadows are updated as the model is rotated. *This graphics effect may result in reduced graphics performance.*
- **Grid Plane** shows/hides the ground plane grid that is displayed beneath the model.

1.1.5.2.8. Additional Display Options

The additional display options (Figure 1.18: Additional Display Options (p. 920)) lets you control which surfaces are selected in the graphics window.

Figure 1.18: Additional Display Options

The following is a description of the additional display option:

- **Filter Selected Surfaces**

  1 selected wall lets you predefine which surfaces will be selected with a "box select" action or filter the surfaces that are already selected by a box select action. *The **mouse-multi-select**, **mouse-multi-select-add** and **mouse-multi-select-toggle** options for mouse controls allow you to perform box selection in the graphics window.*

1.1.6. Task Pages

Task pages appear on the right side of the tree when certain branches are double-clicked. They provide access to more advanced settings than are often available through a right-click of the same branch or sub-branches. The expected workflow is that you travel down the tree, using a right-click most branches and sub-branches to access the settings and controls required for to solve your problem.

Some of your setup will occur in dialog boxes, while others in task pages. For example, if you double-click **General** in the tree, the General Task Page (p. 4684) is displayed. Global settings are made in this task page, which are saved to the case definition.

Each task page has a **Help** button. Clicking this button opens the related help topic in the Task Page Reference Guide (p. 4683). See Using the Help System (p. 942) for more information.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.1.7. The Console

The console is typically located below the graphics window, as shown in Figure 1.1: The GUI Components (p. 902). However, you can move the console to have it in you preferred location, such as tabbed with the graphics window or docked as a tab within the **Messages** dialog box (Figure 1.19: Messages Dialog Box with Tabs for Console, Warnings, and Errors (p. 922)). *Dock the console by selecting the* ***Console*****>*****In Message Window*** *option from the "arrange the workspace" drop-down (* *) in the upper-right of the Fluent application window.*

Fluent communicates with you through the console. It is used to display various kinds of information (that is, messages relating to meshing or solution procedures, and so on). Console text is colored based on whether it is user-input, standard output, or a warning message. Fluent saves a certain amount of information that is written to the console into memory. You can review this information at any time by using the scroll bar on the right side of the console. You can enable/disable autoscrolling using the check box at the bottom right of the console to either have the cursor jump to the latest printed content or stay where it is. The size of the console can be adjusted by raising or lowering the bottom frame of the graphics window.

The console is similar in behavior to "xterm" or other Linux command shell tools, or to the MS-DOS Command Prompt window on Windows systems. It allows you to interact with the text user interface (TUI) menu. More information on the TUI can be found in Text User Interface (TUI) (p. 945).

The console accepts a "break" command (pressing **Ctrl**+**c** at the same time) to let you interrupt the program while it is working. It also lets you perform text copy and paste operations between the console and other X Window (or Windows) applications that support copy and paste. The following steps show you how to perform a copy and paste operation on a Windows system:

1. Move the pointer to the beginning of the text to be copied.
2. Press and hold down the left mouse button.
3. Move the pointer to the end of the text (text should be highlighted).
4. Release the left mouse button.
5. Press the **Ctrl** and `<Insert>` keys at the same time.
6. Move the pointer to the target window and click the left mouse button.
7. Press the **Ctrl**+**v** keys at the same time.

On a Linux system, you will follow the steps below to copy text to the clipboard:

1. Move the pointer to the beginning of the text to be copied.
2. Press and hold down the left mouse button.
3. Move the pointer to the end of the text (text should be highlighted).
4. Release the left mouse button.
5. Move the pointer to the target window.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Press the middle mouse button to “paste” the text.

Warning and Error Messages

In addition to being printed in the console, warning and error messages are also available for easier review in separate **Warning** and **Error** dialog boxes. These messaging dialog boxes can be opened by clicking the warning ( ) and error ( ) icons in the upper right of the console. When new messages are printed in the console, the icons are highlighted (,), indicating that there are new messages for review. The messages printed in the dialog box are hyperlinks to the location where they appear in the Console. *Note that clearing the messages from the **Warning** and **Error** dialog boxes does not remove them from the Console or transcript.*

When the console is docked with the warnings and errors messages in the **Messages** dialog box, the icon indicating a new warning or error is located at the bottom-right of the Fluent application window in the status bar. Messages such as when the system is ready after reading the mesh/case are printed to the status bar.

Figure 1.19: Messages Dialog Box with Tabs for Console, Warnings, and Errors

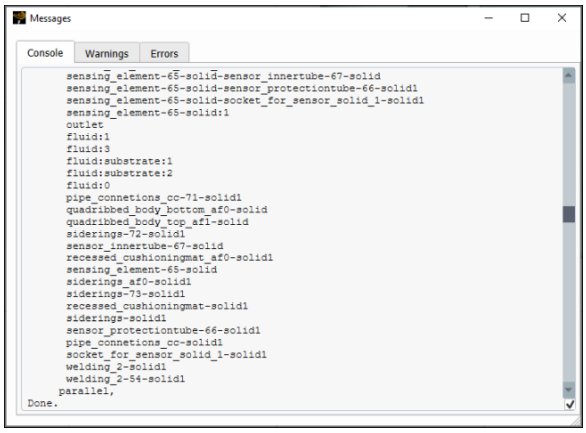
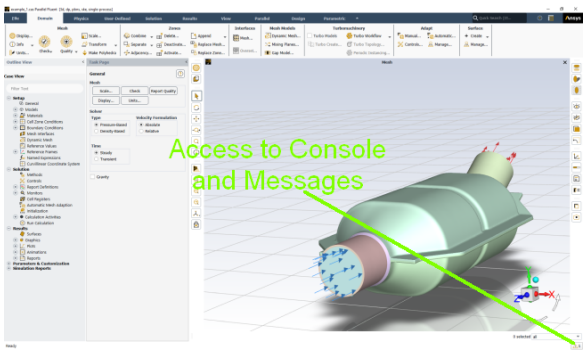


Figure 1.20: Location of Messages Dialog Box Access with Console in Message Window



Note:

The console has a 30,000 line retention limit. Once the number of lines passes 30,000, the auto-scroll checkbox (lower right of the Console) no longer has an effect.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.1.8. Dialog Boxes

There are two types of dialog boxes in Fluent. Some dialog boxes are used to perform simple input/output tasks, such as issuing warning and error messages, or asking a question requiring a yes or no answer. Other forms of dialog boxes allow you to perform more complicated input tasks.

A dialog box is a separate "temporary" window that appears when Fluent needs to communicate with you, or when various types of input controls are employed to set up your case. The types of controls you will see are described further in this section.

When you have finished entering data in a dialog box's controls, you must apply the changes you have made, or cancel the changes, if desired. For this task, each dialog box falls into one of two behavioral categories, depending on how it was designed.

The first category of dialog boxes is used in situations where it is desirable to apply the changes and immediately close the dialog box. This type of dialog box includes an **OK** and a **Cancel** button that function as described below:

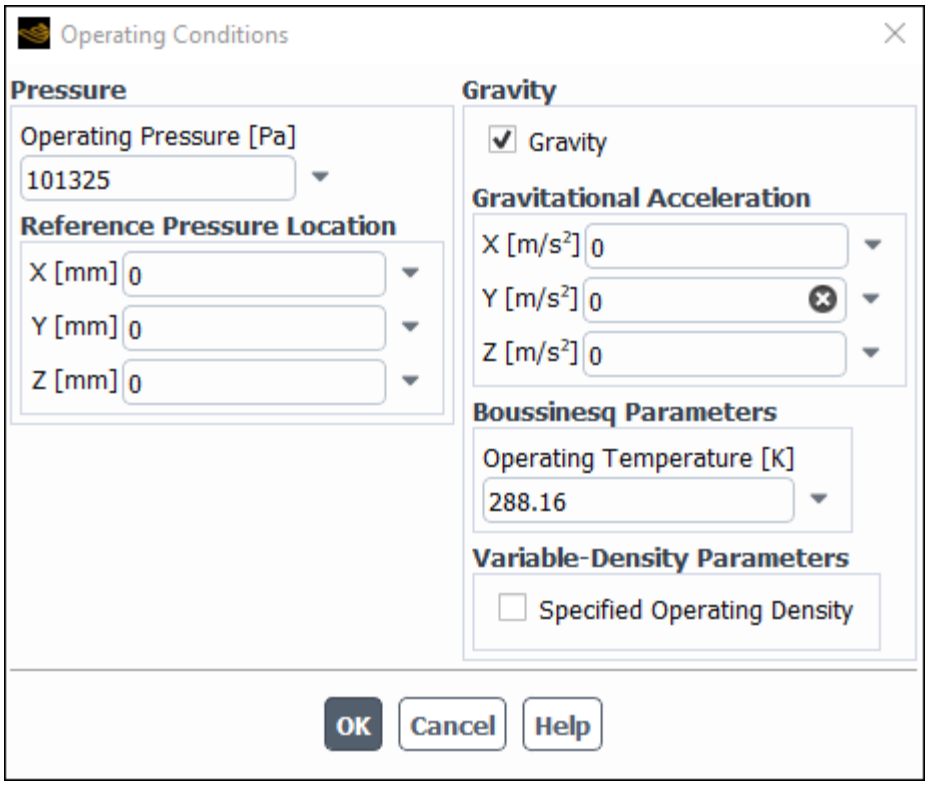
OK

applies any changes you have made to the dialog box, then closes the dialog box.

Cancel

closes the dialog box, ignoring any changes you have made.

An example of this type of dialog box is shown in the following figure:



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The other category of dialog boxes is used in situations where it is desirable to keep the dialog box displayed on the screen after changes have been applied. This makes it easy to quickly go back to that dialog box and make more changes. Dialog boxes used for postprocessing and mesh adaption often fall into this category. This type of dialog box typically includes an **Apply** button and a **Close** button as described below:

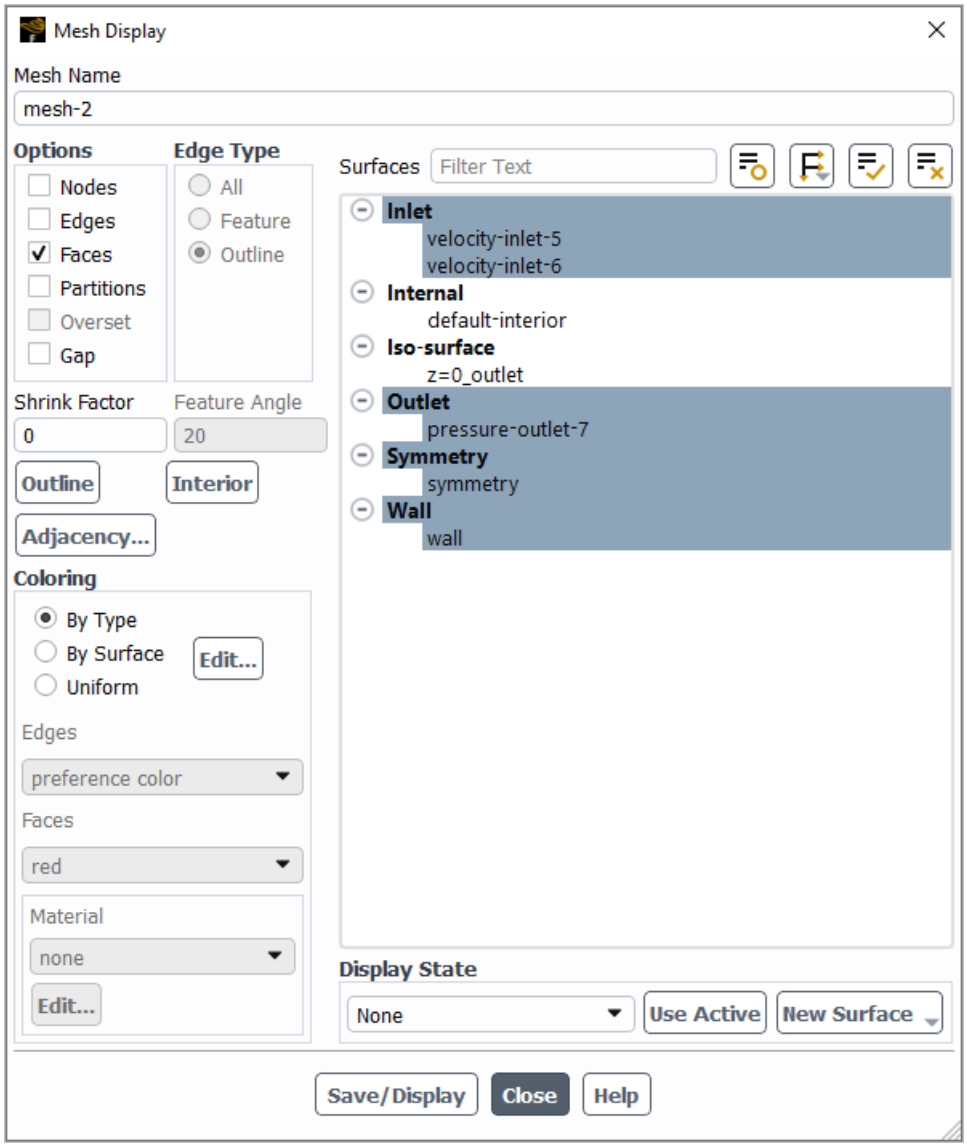
Apply

applies any changes you have made to the dialog box, but does not close the dialog box. The name of this button is often changed to something more descriptive. For example, many of the postprocessing dialog boxes use the name **Display** for this button, and the adaption dialog boxes use the name **Adapt**.

Close

closes the dialog box.

An example of this type of dialog box is shown in the following figure:



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

All dialog boxes include the following button used to access Ansys Help:

Help

displays information about the controls in the dialog box. The information appears on the Ansys help site.

Many dialog boxes also present additional buttons to accomplish specific tasks or open additional dialog boxes. You can find descriptions of the functions of these additional buttons in Ribbon Reference Guide (p. 5347) or Task Page Reference Guide (p. 4683).

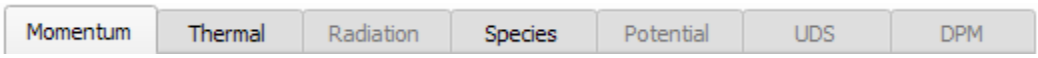
Note:

Dialog boxes containing lists are expandable.

1.1.8.1. Input Controls

Each type of input control utilized by the dialog boxes is described below. Note that the examples shown here are for a Windows system; if you are working on a Linux system, your dialog box controls may look slightly different, but they will work exactly as described here.

1.1.8.1.1. Tabs

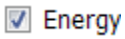


Much like the tabs on a notebook divider, tabs in dialog boxes are used to mark the different sections into which a dialog box is divided. A dialog box that contains many controls may be divided into different sections to reduce the amount of screen space it occupies. You can access each section of the dialog box by "clicking" the left mouse button on the corresponding tab. A click is one press and release of the mouse button.

1.1.8.1.2. Buttons

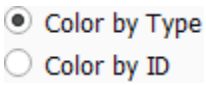
A button, also referred to as a push button, is used to perform a function indicated by the button label. To activate a button, place the pointer over the button and click the left mouse button.

1.1.8.1.3. Check Boxes



A check box, also referred to as a check button, is used to enable / disable an item or action indicated by the check box label. Click the left mouse button on the check box to toggle the state.

1.1.8.1.4. Radio Buttons

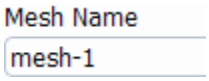


Radio buttons are a set of check boxes with the condition that only one can be set in the "on" position at a time. When you click the left mouse button on a radio button, it will be turned on,

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

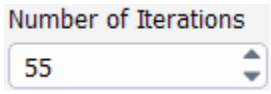
and all others will be turned off. Radio buttons appear either as diamonds (in Linux systems) or as circles (as shown above).

1.1.8.1.5. Text Entry Boxes



A text entry box lets you type text input. It will often have a label associated with it to indicate the purpose of the entry.

1.1.8.1.6. Integer Number Entry Boxes



An integer number entry box is similar to a text entry except it only allows integer numbers to be entered (for example, 10, -10, 50000 and 5E4). You may find it easier to enter large integer numbers using scientific notation. For example, you could enter `350000` or `3.5E5`.

The integer number entry also has arrow buttons that allow you to easily increase or decrease its value. For most integer number entry controls, the value will be increased (or decreased) by one when you click an arrow button. You can increase the size of the increment by holding down a keyboard key while clicking the arrow button. The keys used are shown below:

Key	Factor of Increase
Shift	`10`
Ctrl	`100`

1.1.8.1.7. Real Number Entry Boxes

A real number entry box is similar to a text entry, except it only allows real numbers to be entered (for example, 10, -10.538, 50000.45 and 5.72E-4). In most cases, the label will show the units associated with the real number entry.

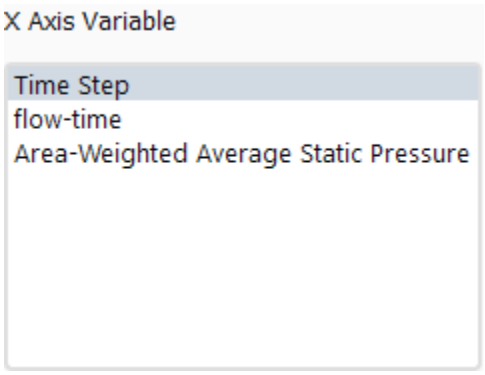
1.1.8.1.8. Filter Text Entry Boxes

The filter text entry box allows you to search and organize a list using a text string. The text string can include wildcards and regular expressions, which allow you to perform pattern matching. For example, searching `*let*` finds surfaces such as in*let*s and out*let*s, including those with longer names, such as "upper-inlet-5". If you have walls separated by a number, you can search `example-wall-?-23`, which would show `example-wall-1-23`, `example-wall-2-23`, and so on.

Filtering begins as soon as you enter text. Clicking **Enter** expands any sub-branches containing matches to the entered string.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

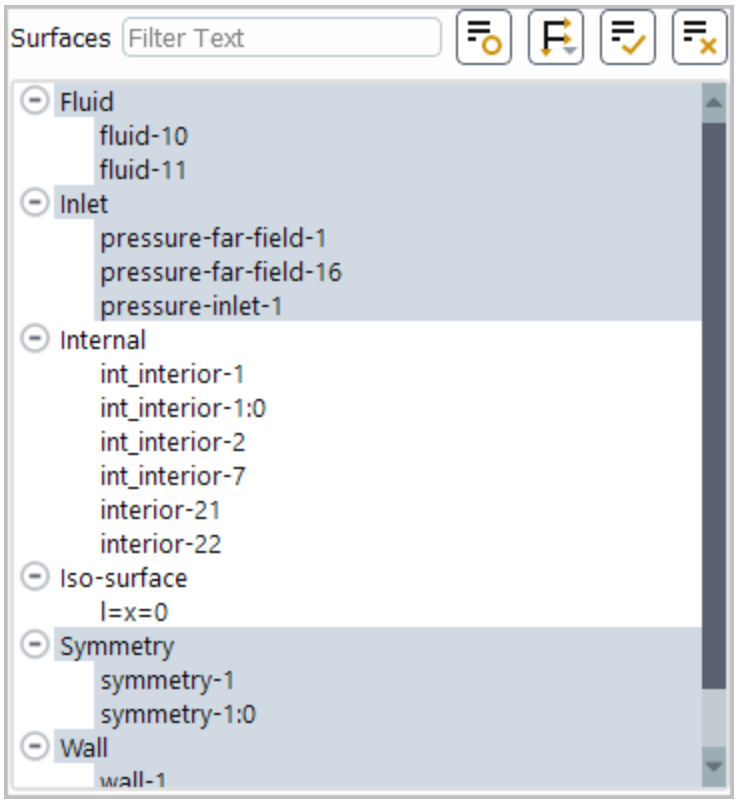
1.1.8.1.9. Single-Selection Lists



A single-selection list presents a list of items, with each item printed on a separate line. You can select an item by placing the pointer over the item line and clicking with the left mouse button. The selected item will become highlighted. Selecting another item will deselect the previously selected item in the list.

Many dialog boxes will also accept a double-click in order to invoke the dialog box action that is associated with the list selection (see information on the dialog box of interest for more details).

1.1.8.1.10. Multiple-Selection Lists



A multiple-selection list is similar to a single-selection list, except it allows for more than one selected item at a time. When you click the left mouse button on an item, its selection state will toggle. Clicking an unselected item will select it. Clicking a selected item will deselect it.

To select a range of items in a multiple-selection list, you can select the first desired item, and then select the last desired item while holding down the **Shift** key. The first and last items, and

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

all the items between them, will be selected. You can also click and drag the left mouse button to select multiple items.

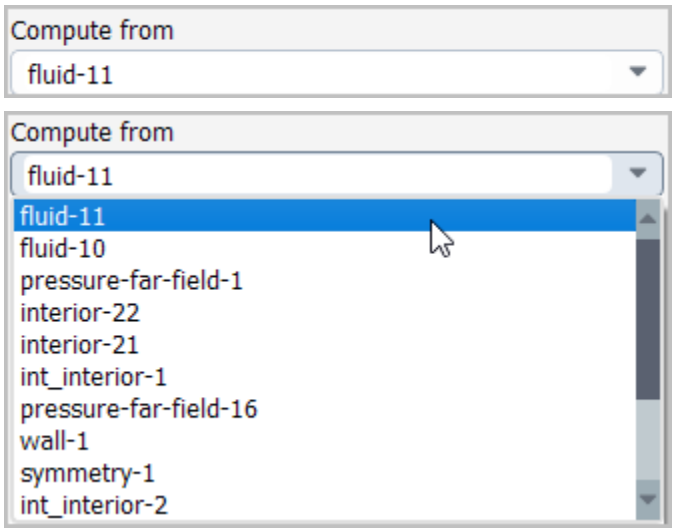
Note:

As you hover over surface names in a **Surfaces** list, the corresponding surface edges will be highlighted in the graphics window (as long as you have the mesh outline displayed and **Hover-Over Highlight** enabled in the **Appearance** branch of **Preferences**). Refer to Figure 1.4: Hover-Over Highlight (p. 909) for an example of how the surfaces appear when highlighted.

There are several small buttons in the upper right corner of the multiple selection list to accelerate selecting or deselecting items from the selection list.

- Click to display all of the items that are currently selected.
- Click or to toggle to a tree view of the selection list, and thereby group items by categories such as name or surface type. You can select or deselect all the items in a particular group by clicking the top-level branch. For example, if you have the surfaces grouped by **Surface Type**, clicking **Inlet** will select or deselect all inlets. Note that once you switch to a tree view, the icon changes to or , respectively.
- Click to select all the items displayed in the selection list.
- Click to deselect all the items displayed in the selection list.

1.1.8.1.11. Drop-Down Lists



A drop-down list is a hidden single-selection list that shows only the current selection to save space.

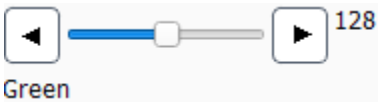
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When you want to change the selection, follow the steps below:

1. Click the arrow button to display the list.
2. Place the pointer over the new list item.
3. Click the left mouse button on the item to make the selection and close the list.

If you want to stop the selection operation while the list is displayed, you can move the pointer anywhere outside the list and click the left mouse button.

1.1.8.1.12. Scales



A scale is used to select a value from a predefined range by moving a slider. The number shows the current value. You can change the value by clicking the arrow buttons, or by following one of the procedures below:

1. Place the pointer over the slider.
2. Press and hold down the left mouse button.
3. Move the pointer along the slider bar to change the value.
4. Release the left mouse button.

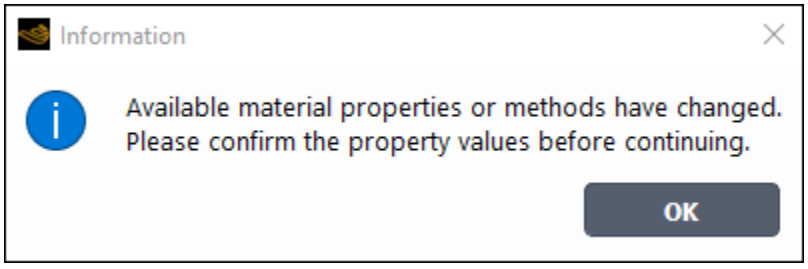
or

1. Place the pointer over the slider and click the left mouse button.
2. Using the arrow keys on the keyboard, move the slider bar left or right to change the value.

1.1.8.2. Types of Dialog Boxes

The following sections describe the various types of dialog boxes.

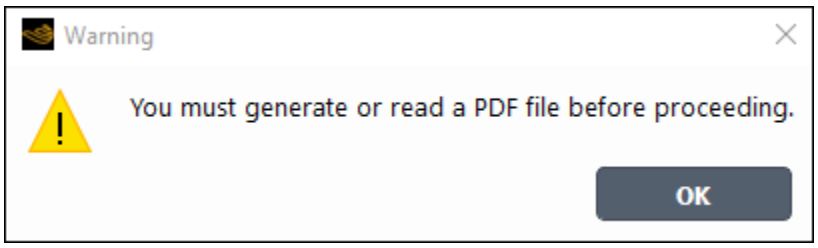
1.1.8.2.1. Information Dialog Boxes



The **Information** dialog box is used to report some information that Fluent thinks you should know. After you have read the information, you can click the **OK** button to close the dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.1.8.2.2. Warning Dialog Boxes

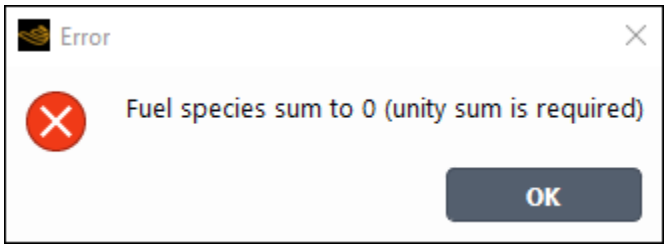


The **Warning** dialog box is used to warn you of a potential problem or deliver an important message. Your control of Fluent will be suspended until you acknowledge the warning by clicking the **OK** button.

Note:

All warnings are available for review in the **Warning** dialog box that you can access by clicking the warning icon ( ) in the upper-right of the console.

1.1.8.2.3. Error Dialog Boxes

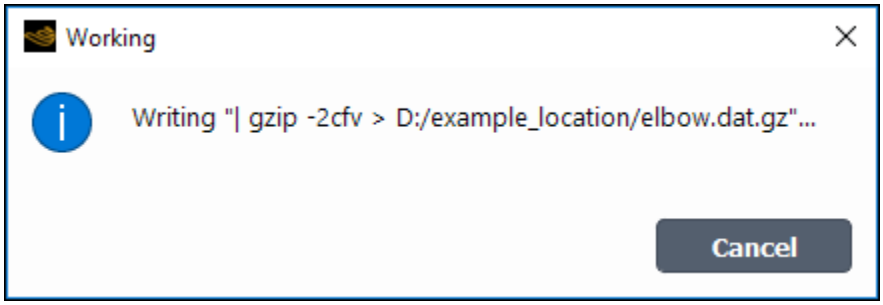


The **Error** dialog box is used to alert you of an error that has occurred. After you have read the error information, you can click the **OK** button to close the dialog box.

Note:

All errors are available for review in the **Error** dialog box that you can access by clicking the error icon ( ) in the upper-right of the console.

1.1.8.2.4. The Working Dialog Box

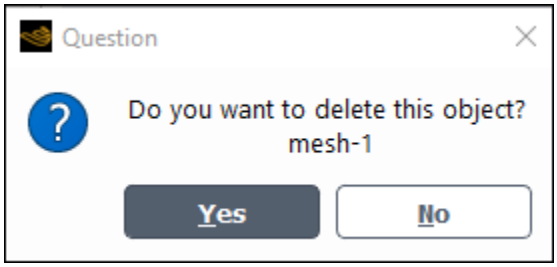


The **Working** dialog box is displayed when Fluent is busy performing a task. This is a special dialog box, because it requires no action by you. It is there to let you know that you must wait.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When the program is finished, it will close the dialog box automatically. You can, however, abort the task that is being performed by clicking the **Cancel** button.

1.1.8.2.5. Question Dialog Box

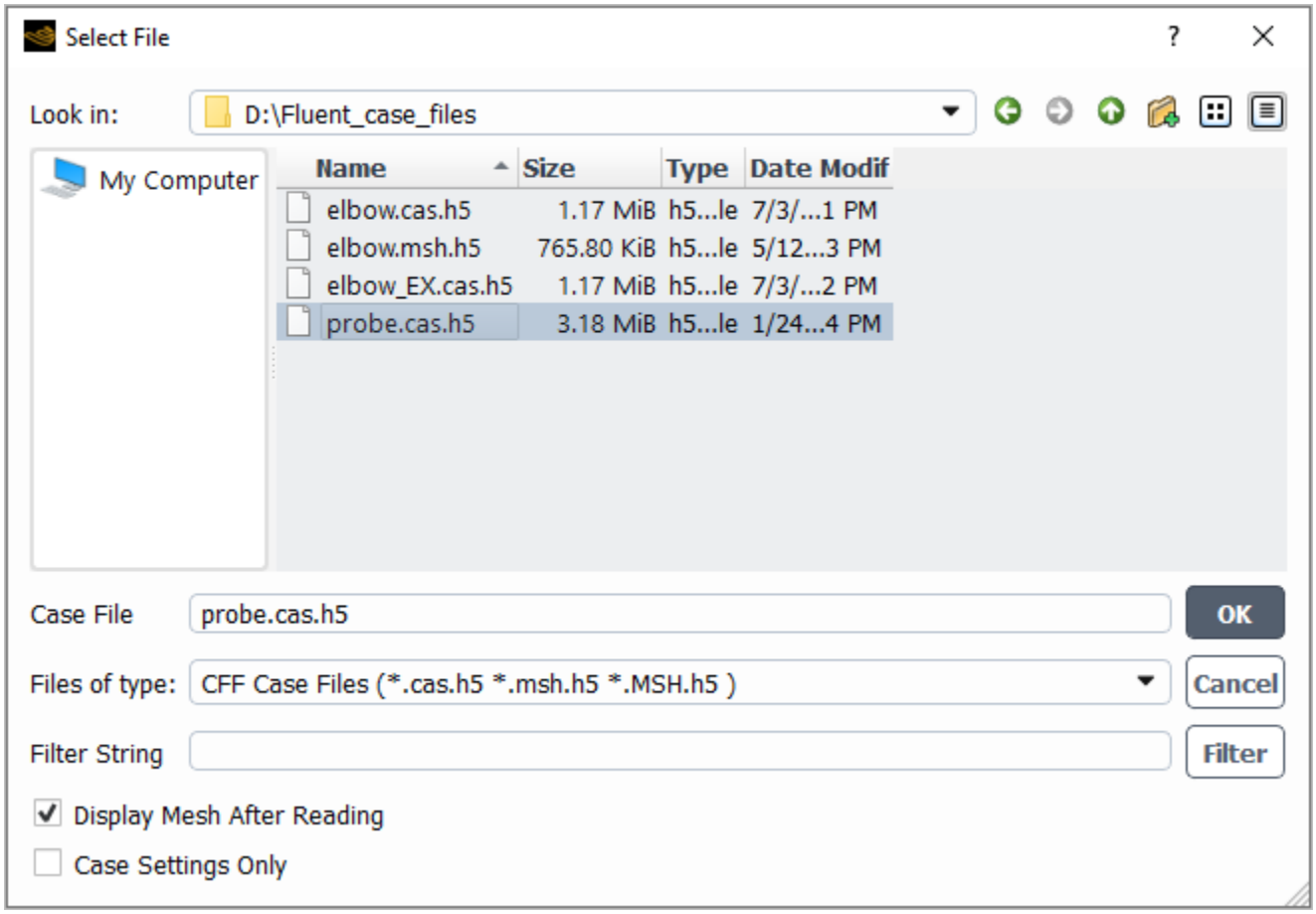


The **Question** dialog box is used to ask you a question. Sometimes the question will require a **Yes** or **No** answer, while other times it will require that you either allow an action to proceed (**OK**) or **Cancel** the action. You can click the appropriate button to answer the question.

1.1.8.2.6. The Select File Dialog Box

File selection is accomplished using the **Select File** dialog box (Figure 1.21: The Select File Dialog Box for Windows (p. 931) or Figure 1.22: The Select File Dialog Box for Linux Platforms (p. 932)).

Figure 1.21: The Select File Dialog Box for Windows

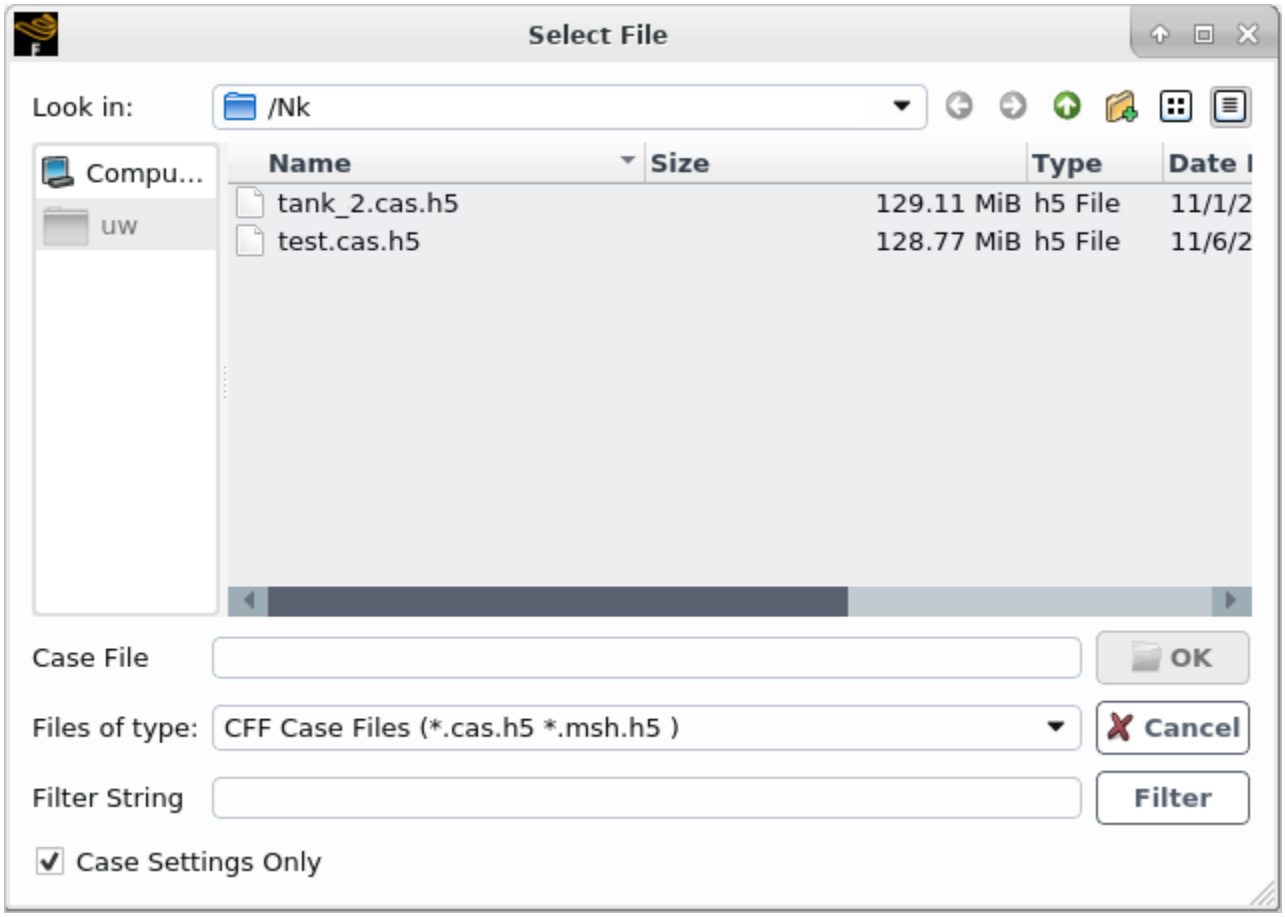


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For Linux systems, note that the appearance of the **Select File** dialog box will not always be the same.

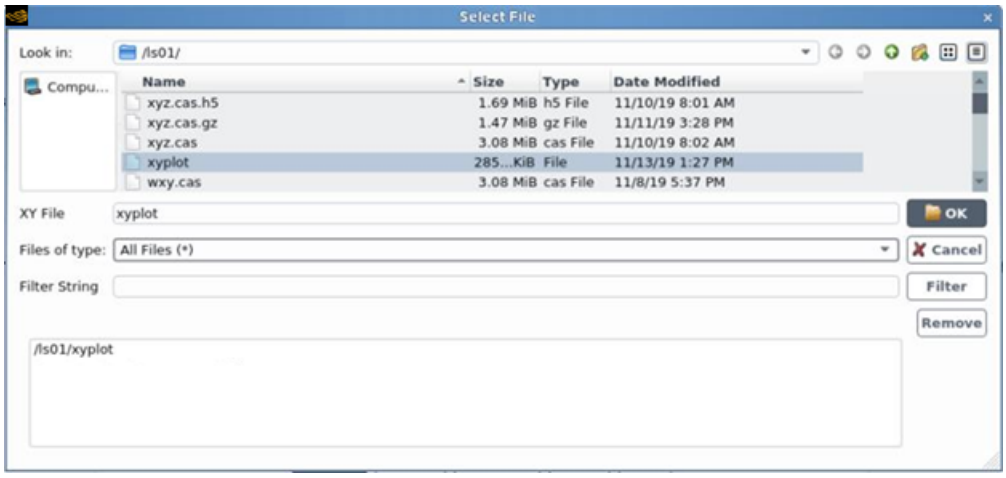
The version shown in Figure 1.22: The Select File Dialog Box for Linux Platforms (p. 932) will appear in almost all cases, but it will be different if you are loading external data files for use in an XY plot (see Including External Data in the Solution XY Plot (p. 4063) for more information). In such cases, the dialog box will look like Figure 1.23: Another Version of the Select File Dialog Box for Linux Platforms (p. 933).

Figure 1.22: The Select File Dialog Box for Linux Platforms



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.23: Another Version of the Select File Dialog Box for Linux Platforms



The steps for selecting a file are as follows:

1. Go to the appropriate directory. You can do this in two different ways:

   - Enter the path to the directory in the **Look in:** field.
   - Double-click a directory, and then a subdirectory, and so on, in the **Name** list until you reach the directory you want. You can also click once on a directory and then click the **Open** button, instead of double-clicking.

   Note:

   If you are accessing a file using a Universal Naming Convention (UNC) path, you must ensure that you have permission to access to all of the folders in the path or you will not be able to open the file.

   ---

2. Specify the file name by selecting it in the **Name** list or entering it in the **File** text entry box. The name of this text entry box changes depending on the type of file you are selecting (**Case File**, **Journal File**, and so on).

   Important:

   Note that if you are searching for an existing file with a nonstandard extension, you may need to modify the file filter by selecting a different type in the **Files of type:** drop-down list. For example, if you are reading a data file, only those files that have a `*.dat*` extension will appear in the **Name** list.

   If you want all files in the directory to be listed in the **Name** list, select **All Files (*)** from the **Files of type:** drop-down list.

   ---

3. If you are reading a mesh or case file, use the **Display Mesh After Reading** option to specify whether you want Fluent to automatically display the mesh after the file is read. All of the boundary zones will be displayed, except for the interior zones of 3D geometries. The default

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

status of this option (that is, enabled or disabled) is determined by your decision regarding the **Display Mesh After Reading** option in Fluent Launcher.

4. If you are reading a mesh or case file with a large cell count, enable the **Case Settings Only** option if you want to read the case settings only, without any of the data associated with the mesh. This allows you to review and/or modify the settings without having to wait for the reading of the mesh, on a machine that does not have the memory needed for the mesh (such as a personal computer or laptop). For further details, see Reading Settings Only for Mesh or Case Files with Large Cell Counts (p. 962).

5. If you are reading multiple data files (for example, XY plots), the selected file will be added to the list of files shown at the bottom of the dialog box. Each file you select in the **Name** list is automatically added to the list of selected files. (If you accidentally select the wrong file, you can select it in the list and click **Remove**.) Repeat until all of the desired files are in the list.

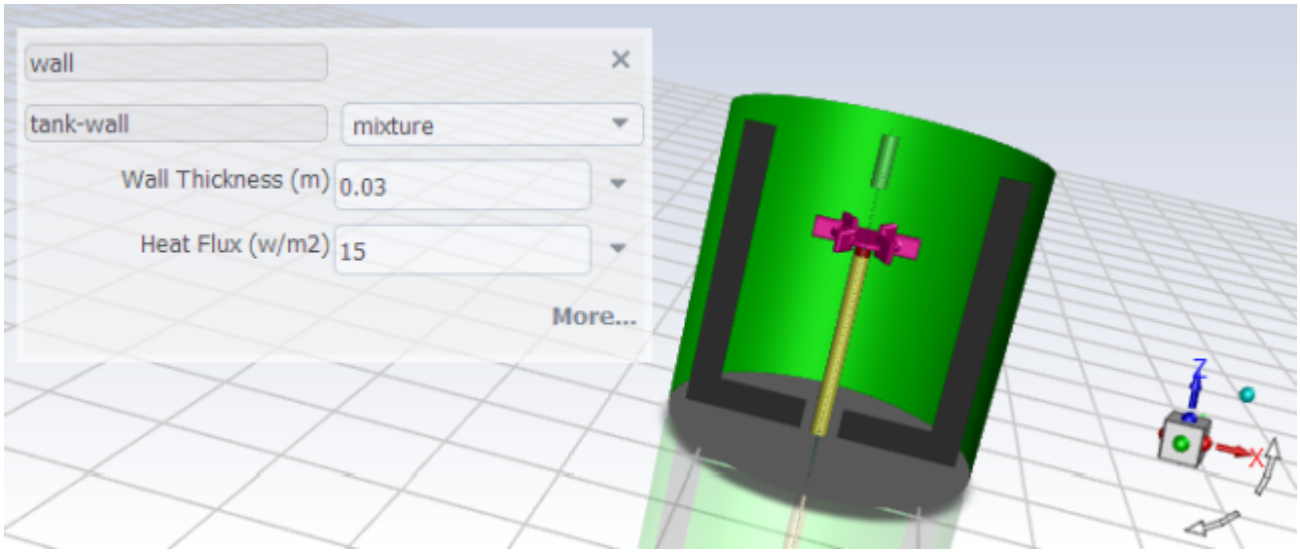
6. If you are writing a case, data, or radiation file, use the **Write Binary Files** check box to specify whether the file should be written as a text or binary file. You can read and edit a text file, but it will require more storage space than the same file in binary format. Binary files take up less space and can be read and written by Fluent more quickly. For further details, see Binary Files (p. 949).

7. Click the **OK** button to read or write the specified file.

1.1.9. Quick Property Editor for Boundaries

The "quick properties" editor allows you to specify certain key properties of selected boundaries directly in the graphics window without having to open the full version of relevant dialog box, as shown in Figure 1.24: Quick Editor for a Velocity Inlet (p. 934).

Figure 1.24: Quick Editor for a Velocity Inlet



To access this functionality, enable **Show quick property view** in the **Appearance** branch of Preferences.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

File → Preferences... → Appearance → Show quick property view

With the feature enabled, the quick property editor will appear as soon as you select a boundary. Inputs to the quick property editor are saved as soon as you leave the field. If you decide that you want to view the full version of the dialog box, you can click **More...** at the bottom right of the editor.

For multiphase models, You can edit the mixture as well as the other phases, by changing the selected phase in the drop-down list.

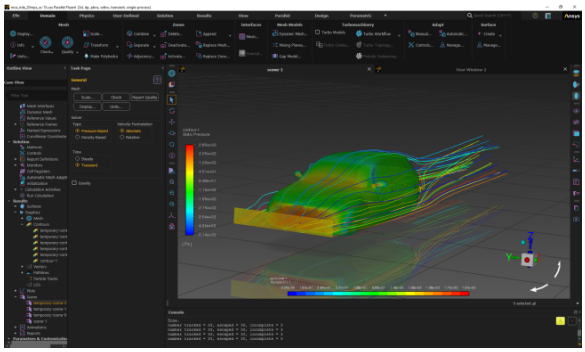
Note:

Settings changes made using the "quick-edit" property editor are printed in the console and saved with the transcript (if one is being written) including the previous value(s) and new value(s).

1.2. Customizing the Graphical User Interface

You may want to customize the graphical user interface by changing attributes such as text color, background color, text size (Modifying the Font Size (p. 936)), and text font. For example, you can run Ansys Fluent in **Dark** mode (Figure 1.25: Fluent in Dark Theme (p. 935)) by selecting **Dark** from the **Color Theme** drop-down list in the **Appearance** branch of **Preferences**. You can control many of these settings globally, using **Preferences** (see Setting User Preferences/Options (p. 937) for more information). The program will try to provide default text fonts that are satisfactory for your platform's display size, but in some cases customization may be necessary if the default text fonts make the GUI too small or too large on your display, or if the default colors are undesirable.

Figure 1.25: Fluent in Dark Theme



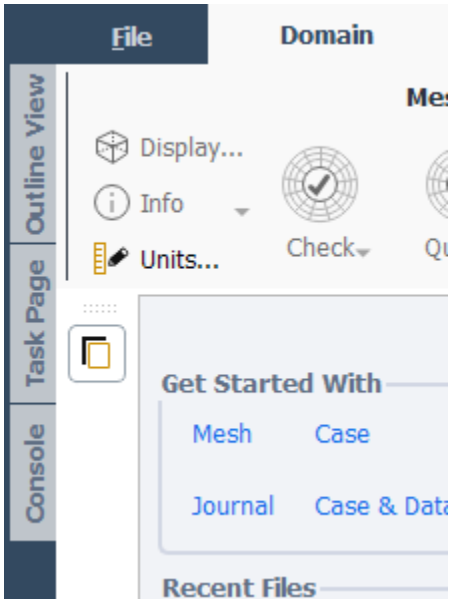
You may want to customize the graphical user interface by changing the way that the various elements are arranged and sized. This can be achieved by "dragging" elements and "dropping" them at a new location and by grabbing near the edge and expanding or shrinking the selected element. For example, certain items can be tabbed on top of other items, such as the graphics window on top of the console. The items that you can move include the console, the task page, and the toolbars. You can resize these elements while the solver is calculating to adjust the focus towards your area of interest, such as increasing the size of the graphics window. You can also hide or collapse the Outline View tree, Task Pages, and the Console, by clicking the collapse icon ( ) located in the upper-right of each component. The

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

console can also be docked as a tab within the **Messages** dialog box, using the **Console**>**In Message Window** option from the "arrange the workspace" drop-down ( ).

To restore items that you intentionally or unintentionally collapsed or hid, select them from the component dock, located at the left-hand side of the Fluent window (as shown in Figure 1.26: Component Dock (p. 936)). You can also click in the upper-right of the Fluent window, to select one of the predefined layouts and restore missing items. *You can also right-click the top portion of the* ***Outline View*** *or* ***Console*** *to restore missing items.*

Figure 1.26: Component Dock



Modifying the Font Size

You can control the font size of the Fluent application using the **Application Font Size** field in the **Appearance** branch of **Preferences**. This affects the size of labels and text in the ribbon, dialog boxes, console, outline view tree, and task pages. *Note that this setting does not affect the appearance of Simulation Reports (Using Simulation Reports (p. 4359)).*

The font size of plot legends and colormaps is controlled by a couple of fields under **Colormap Settings** in the **Graphics** branch of **Preferences**:

- **Text Behavior** drop-down list lets you specify whether the font is a fixed size or whether it resizes automatically based on the vertical size of the colormap.
- **Automatic Font Size | Fixed Font Size** fields (available when **Automatic** or **Fixed Size** are selected for **Text Behavior**, respectively) lets you specify either a ratio for the font size or the exact font size for legends and colormaps.

The GUI in Fluent is based on the **Qt Toolkit**. If you are unfamiliar with the **Qt Toolkit**, refer to any documentation you may have that describes how to use the **Qt Toolkit** or application. The graphical attributes can be modified in a Qt stylesheet file named `cxdisplay.qss` and placed in your home directory.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.3. Setting User Preferences/Options

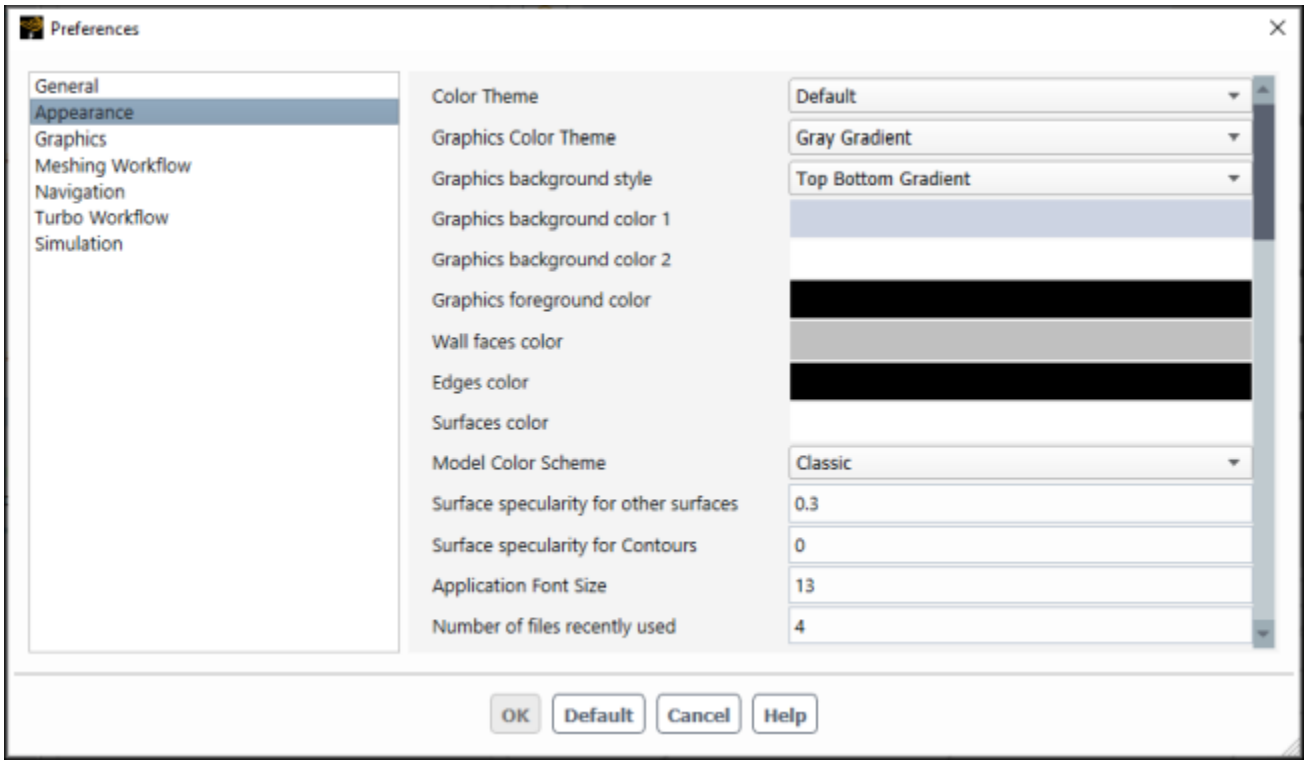
You can specify global settings that are applied whenever you are operating in Ansys Fluent. These settings are case-independent and are controlled using the **Preferences** dialog box.

To review and modify your preferences, open the **Preferences** dialog box by selecting **Preferences...** from the **File** menu.

Tooltips are available when you hover over setting labels that clarify the purpose/function of the setting.

File → Preferences...

Figure 1.27: Preferences Dialog Box



Note:

- Some settings, such as ruler and axis visibility can also be controlled locally within a Fluent session (that is, outside of the **Preferences** dialog box). Settings that deviate from the global settings specified in the **Preferences** dialog box will not be retained beyond the current session.
- If you change a setting locally that is also controlled by **Preferences**, the local change will be shown for this session only. If you open the **Preferences** dialog box and click **Apply** in an attempt to overwrite the local settings change, the global setting will only appear if it is a change from how the setting is already specified globally. For example, if you have **Ansys Logo** set as **Black** in **Preferences**, then you change the logo to **White** in the **Display Options** dialog box, the logo will change from black to white. If you then re-open the **Preferences** dialog box and click **Apply**, the logo will remain white, even though it is

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

specified as **Black** in the preferences dialog box. The reason being that globally, the logo is already specified as black, so clicking **Apply** does not change the current global color of the logo, which is being overruled by your local change to white.

- Not all preference settings apply in all workspaces. For example, the **Flow Model** option in the **General** branch only applies to the **Solution** workspace and **Meshing Workflow** settings only apply to the meshing workflows in the **Meshing** workspace.

The controlling preferences file location is:

- Windows: `%HOMEDRIVE%%HOMEPATH%\.fluentconf\24.2.0\preferences`

  Note:

  If you have `%HOME%` is defined as an environment variable, then the preference location is `%HOME%\.fluentconf\24.2.0\preferences`

- Linux: `%HOMEPATH%/.fluentconf/24.2.0/preferences`

If no preference file is found (for instance, after installation of a new version or deletion of the preference file), Fluent will attempt to load user preferences from a previous version. If no previous version's preferences are found, Fluent will create a new file using the default settings.

*At any time, you can revert your preferences to the default settings by clicking the **Default** button in the **Preferences** dialog box. Or you can delete the preferences file and Fluent will follow the above rule to create a new one.*

1.4. Fluent Graphical User Interface Other Languages

You can have the Fluent graphical user interface appear in Japanese, Korean, or Chinese, by defining the appropriate environment variable before launching Fluent (you can define it in the **Environment** tab of the Fluent Launcher).

These are the environment variables for each language:

- `lang=ja` for Japanese
- `lang=ko` for Korean
- `lang=zh` for Chinese

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.28: Fluent GUI in Japanese

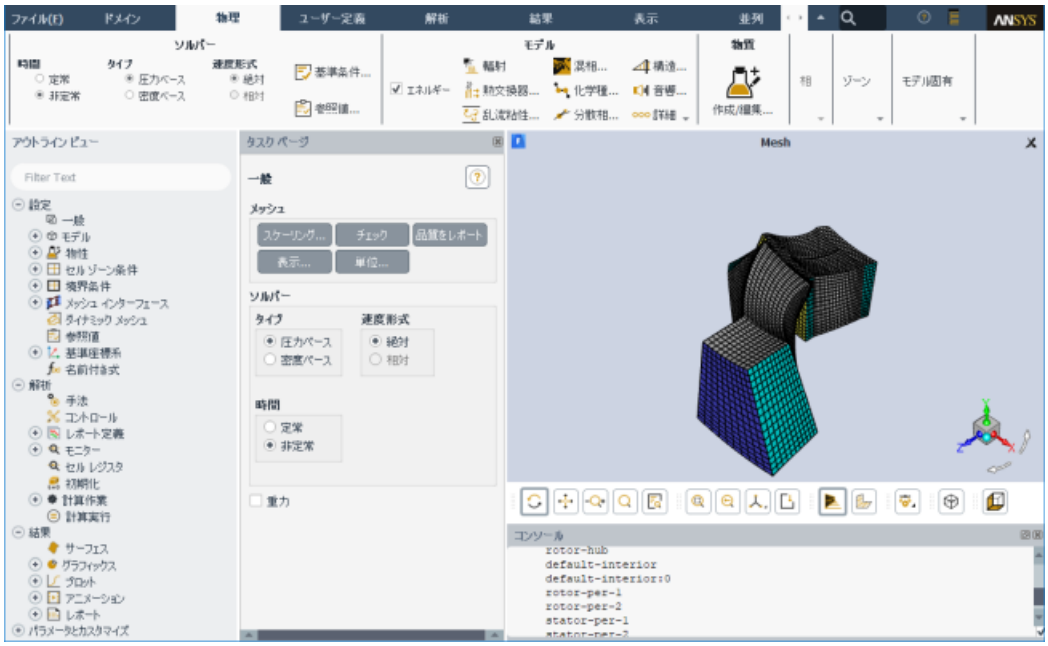


Figure 1.29: Fluent GUI in Korean

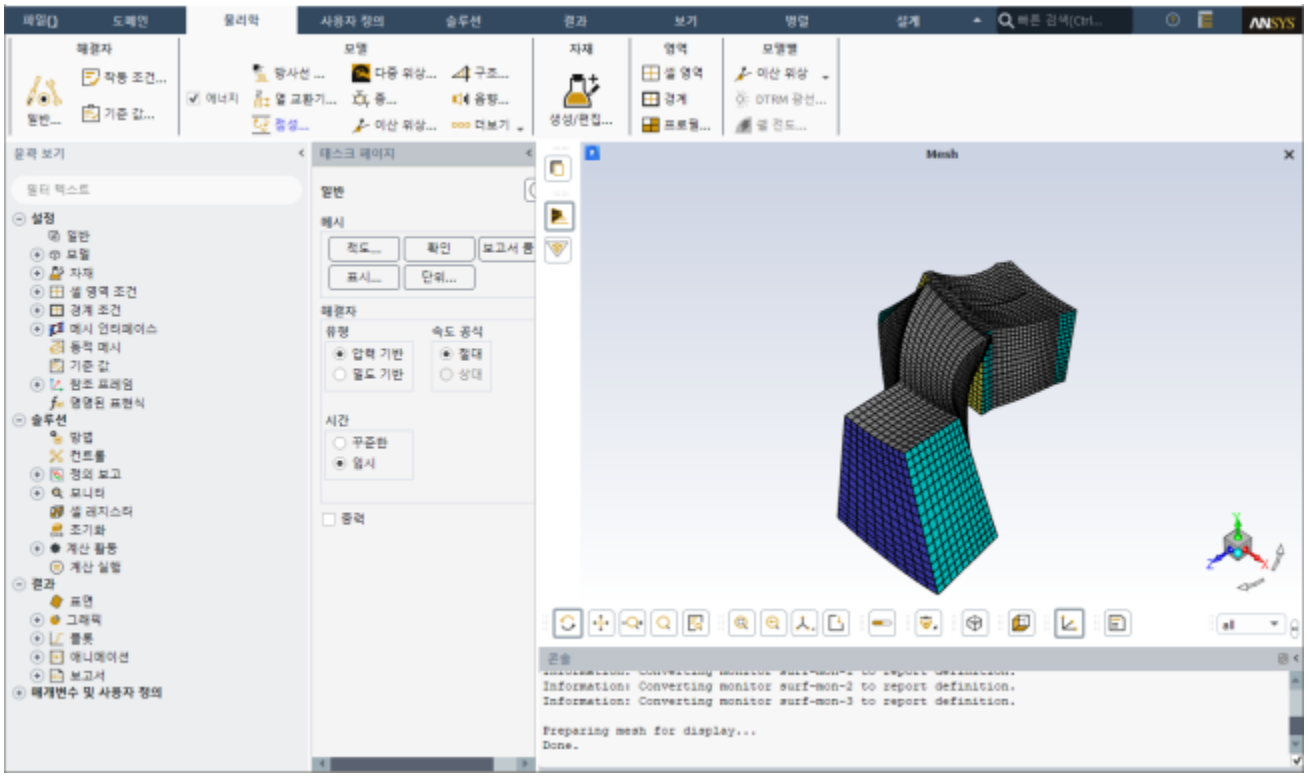
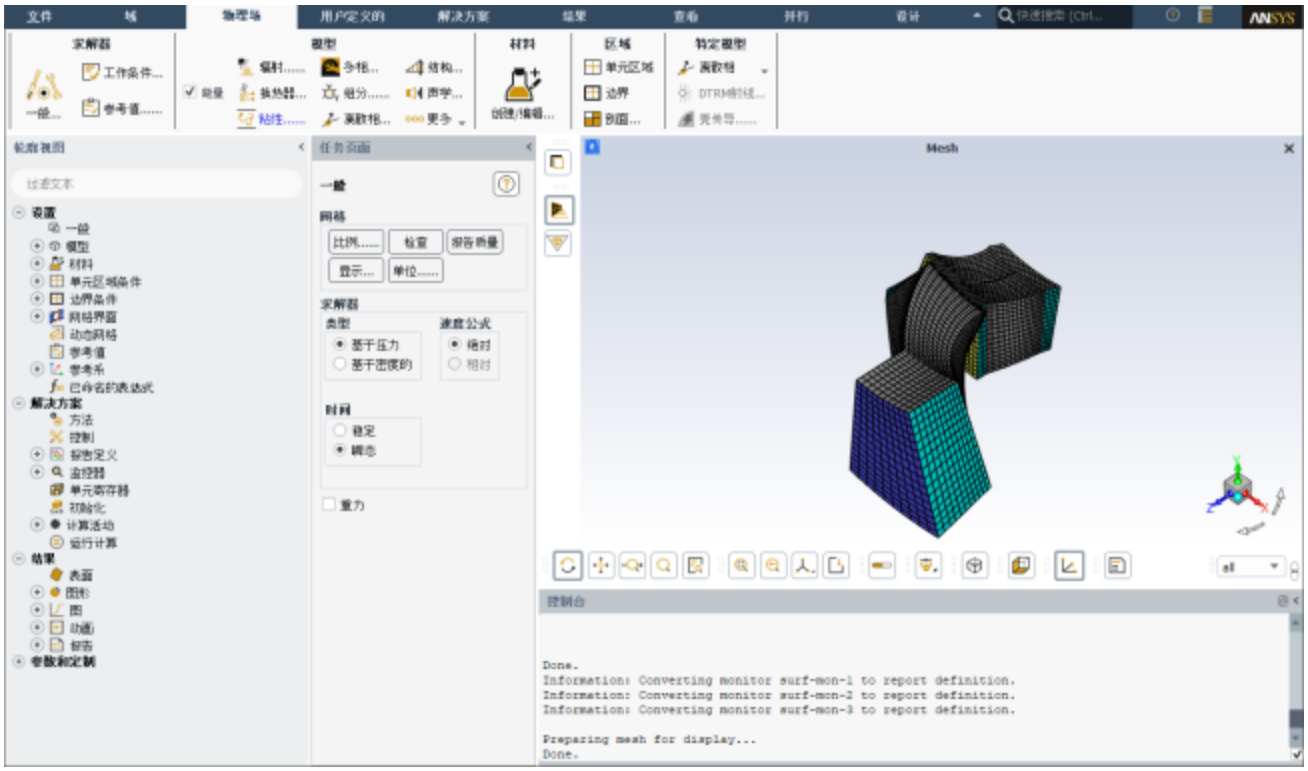


Figure 1.30: Fluent GUI in Chinese



Limitations for Localization

Some areas of the Fluent 2024 R2 GUI are not completely translated including:

- Drop-down menu/list items
- Quick search results
- Pop up messages
- Graphics window titles and labels
- Object names (boundary conditions, graphics objects, and so on)
- Interface (GUI labels) for items without meaningful Japanese/Chinese/Korean equivalents, such as X, Y, Z, Cp, OK, and so on
- The Console

Important:

Localization of the Fluent workspaces is a Beta-level exposure, meaning that it has not been fully tested or validated. *The workspaces include Aero, Icing, Material Processing, and Post-Analysis (which is itself Beta level).*

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1.5. Having the Session Close After Sitting Idle

You have the option of enabling an idle timeout so that Ansys Fluent closes after a set period of sitting unused. This can be beneficial if an idle Fluent session is consuming computing resources that could be used elsewhere.

There are two ways to set an idle timeout:

- Set the timeout using the Figure 1.31: Set Idle Timeout Dialog Box (p. 941), as discussed in Timeout Using the Set Idle Timeout Dialog Box (p. 941)
- Set the timeout using the `FLUENT_MAX_IDLE_TIMEOUT` environment variable, as discussed in Timeout Using FLUENT_MAX_IDLE_TIMEOUT (p. 942).

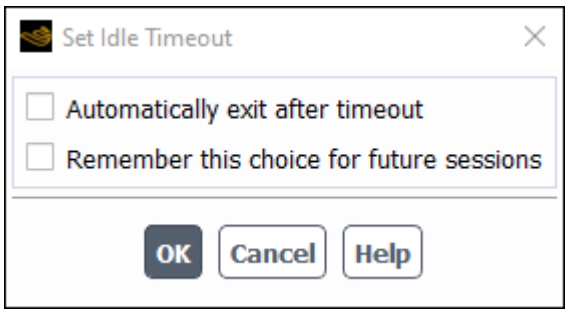
1.5.1. Timeout Using the Set Idle Timeout Dialog Box

Use the following steps to setup an idle timeout:

1. Open the **Set Idle Timeout** dialog box.

   File → Idle Timeout...

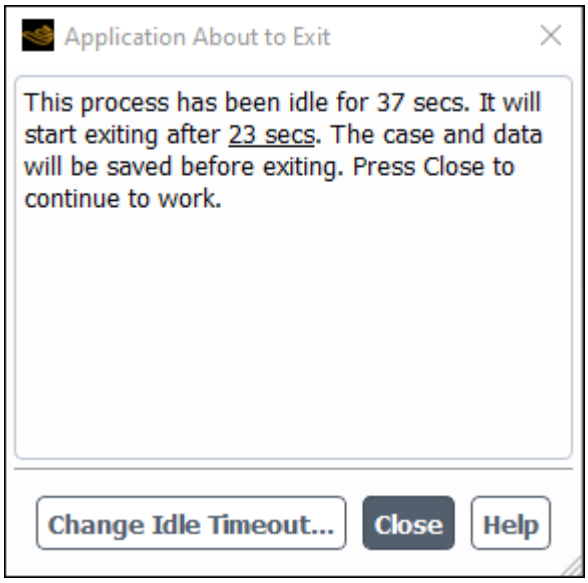
   Figure 1.31: Set Idle Timeout Dialog Box

2. Enable **Automatically exit after timeout**.
3. Enter how long you want Fluent to idle before closing in the **Idle timeout (min)** field. *Note that with five minutes remaining on the timer, the **Application About to Exit** dialog box opens, providing a count down to when Fluent will close. Before closing, Fluent saves case and data files in the working directory.*

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1.32: Application About to Exit Dialog Box



4. (Optional) Enable **Remember this choice for future sessions** if you want to have the idle timeout set for future instances of Fluent. *Note that next time you open the **Set Idle Timeout** dialog box, **Remember this choice for future sessions** will not be enabled. All this option does is ask if you want to save settings for a future session as they are currently shown in the dialog box.*

1.5.2. Timeout Using FLUENT_MAX_IDLE_TIMEOUT

You can use an environment variable to set the idle timeout. The advantage of using `FLUENT_MAX_IDLE_TIMEOUT` is that you can specify a maximum timeout at an administrator level, so that individual users cannot increase the timeout beyond the maximum you define. This is useful for managing cluster/HPC resources where Fluent is typically run in batch mode.

Set `FLUENT_MAX_IDLE_TIMEOUT` equal to the maximum time, in minutes, that you want a Fluent session to sit idle before it exits. This environment variable should be defined on the system where the Fluent batch jobs are run (such as, a Linux cluster managed through a job scheduler).

1.5.3. Idle Timeout Limitations

There are a few limitations associated with the idle timeout feature:

- It is not available for Fluent Meshing except when Fluent Meshing is running from within Workbench.
- (Windows only) it is not compatible with launching Fluent in `-g` or `-gu` mode.

1.6. Using the Help System

There are many ways to access Ansys help. You can get reference information from within a task page or dialog box. You can also go to the User's Guide (p. 1) contents page and use the links to find the information you are looking for. In addition to the User's Guide (p. 1), you can also access the other Fluent documentation (for example, the Theory Guide or Fluent Customization Manual).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that the last two chapters of the User's Guide (Task Page Reference Guide (p. 4683) and Ribbon Reference Guide (p. 5347)) (p. 1) are also referred to as the Reference Guide, and contain a description of each task page, and dialog box.

The sections that follow provide information on how to get help for a task page or dialog box, and brief descriptions of the help menu items in Fluent.

1.6.1. Learning and Support

1.6.2. Task Page and Dialog Box Help

1.6.3. Obtaining License Use Information

1.6.4. Version and Release Information

1.6.1. Learning and Support

The **Learning and Support** ribbon tab provides access to tutorials, training courses, videos, documentation, and various support resources.

Figure 1.33: Learning and Support Ribbon Tab

1.6.2. Task Page and Dialog Box Help

To get help about a task page or dialog box that you are currently using, click the **Help** button in the task page or dialog box. The help will open to the section of the User's Guide (p. 1) that explains the function of each item in the task page or dialog box. In this section, you will also find hypertext links to more specific section(s) of the User's Guide (p. 1) that discuss how to use the task page or dialog box and provide related information.

Note:

The latest versions of Google Chrome and Mozilla Firefox are recommended browsers for accessing Ansys Help. Microsoft Internet Explorer or Microsoft Edge are not supported.

1.6.3. Obtaining License Use Information

Your installation of Fluent is managed by the Ansys License Manager (ANSLIC_ADMIN). Information regarding the license being used is printed to the console. To learn more about Ansys licensing information, refer to the Licensing Documentation.

1.6.4. Version and Release Information

You can obtain information about the version and release of Fluent you are running by selecting the **Version...** option in help menu (accessed by clicking).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 2: Text User Interface (TUI)

The text user interface (TUI) in Fluent, also referred to as The Console (p. 921), is written in a dialect of Lisp called Scheme. Users familiar with Scheme will be able to use the interpretive capabilities of the interface to create customized commands. The text-based menu system provides a hierarchical interface to the underlying procedural interface of the program.

- You can easily manipulate its operation with standard text-based tools—input can be saved in files, modified using text editors, and read back in to be executed.
- The text menu system is tightly integrated with the Scheme extension language, so it can easily be programmed to provide sophisticated control and customized functionality.

A more complete description of the text-based interface, including a full list of commands is available in Fluent Text Command List.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 3: Reading and Writing Files

During an Ansys Fluent session you may need to import and export several kinds of files. Files that are read include mesh, case, data, profile, Scheme, and journal files. Files that are written include case, data, profile, journal, and transcript files. Ansys Fluent also has features that enable you to save pictures of graphics windows. You can also export data for use with various visualization and postprocessing tools. These operations are described in the following sections.

Important:

Ensure that you have read/write permission in the local user directory and that there are not any locked files in this directory, which could interfere with reading files into Ansys Fluent.

3.1. Shortcuts for Reading and Writing Files
3.2. Reading Mesh Files
3.3. Reading and Writing Case and Data Files
3.4. Reading Settings Only for Mesh or Case Files with Large Cell Counts
3.5. Reading Fluent/UNS and RAMPANT Case and Data Files
3.6. Reading and Writing Profile Files
3.7. Reading Files in Tabular Format
3.8. Reading and Writing Boundary Conditions
3.9. Writing a Boundary Mesh
3.10. Reading Scheme Source Files
3.11. Creating and Reading Journal Files
3.12. Creating Transcript Files
3.13. Importing Files
3.14. Exporting Solution Data
3.15. Exporting Solution Data after a Calculation
3.16. Exporting Steady-State Particle History Data
3.17. Exporting Data During a Transient Calculation
3.18. Exporting to Ansys CFD-Post
3.19. Parallel Exporting to Ansys EnSight
3.20. Managing Solution Files
3.21. Mesh-to-Mesh Solution Interpolation
3.22. Mapping Data for Fluid-Structure Interaction (FSI) Applications
3.23. Saving Picture Files
3.24. Setting Data File Quantities

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.25. The .fluent File
3.26. Coupled Simulations in Ansys Fluent with Functional Mock-up Unit (FMU) Files

3.1. Shortcuts for Reading and Writing Files

The following features in Ansys Fluent make reading and writing files convenient:

- Automatic appending or detection of default file name suffixes
- Binary file reading and writing
- Automatic detection of file format (text/binary)
- Recent file list
- Reading and writing of compressed files
- Tilde expansion
- Automatic numbering of files
- Ability to disable the overwrite confirmation prompt
- Standard toolbar buttons for reading and writing files

For additional information, see the following sections:

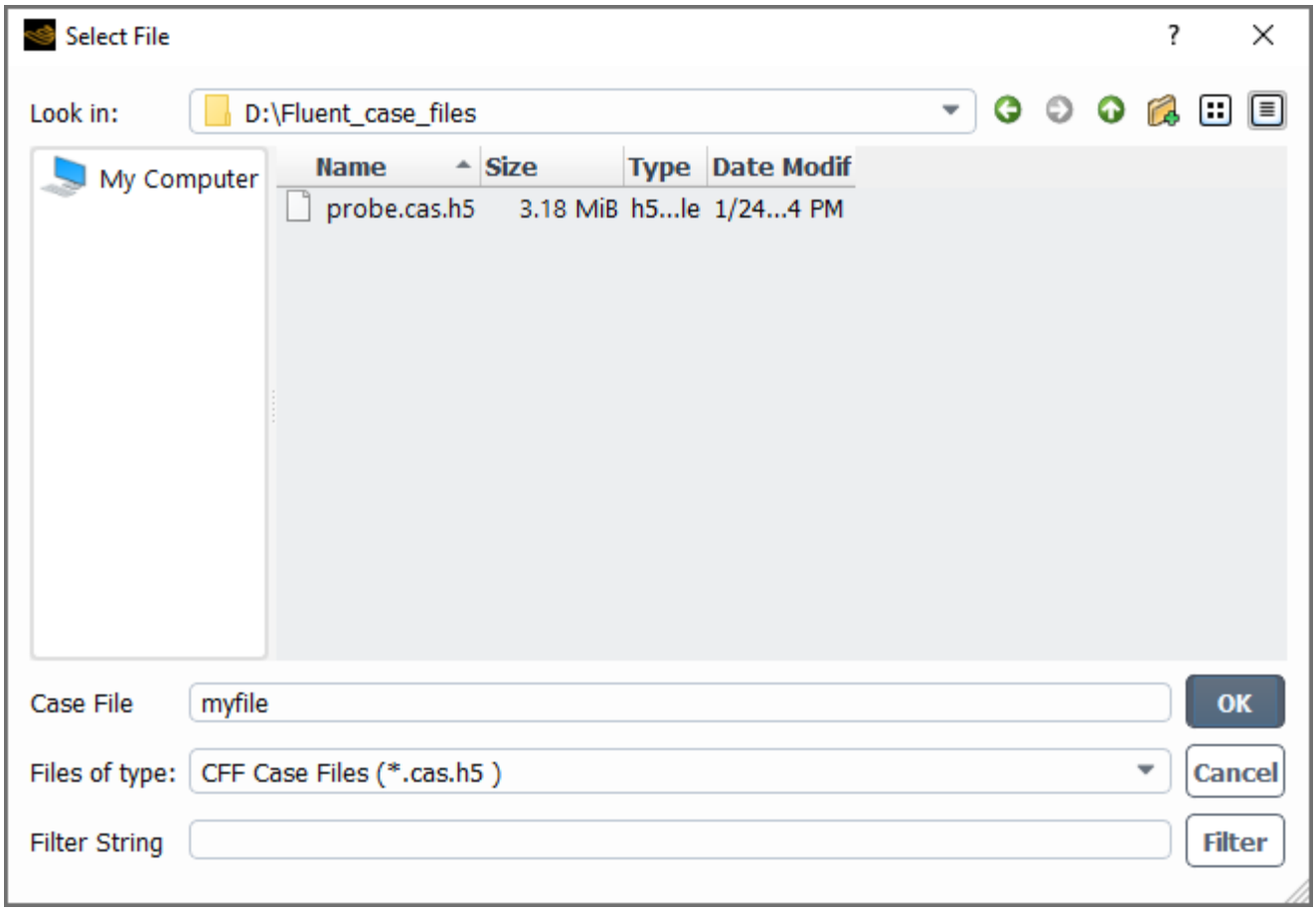
3.1.1. Default File Suffixes
3.1.2. Binary Files
3.1.3. Detecting File Format
3.1.4. Recent File List
3.1.5. Reading and Writing Compressed Files
3.1.6. Tilde Expansion (Linux Systems Only)
3.1.7. Automatic Numbering of Files
3.1.8. Disabling the Overwrite Confirmation Prompt

3.1.1. Default File Suffixes

Each type of file read or written in Ansys Fluent has a default file suffix associated with it. When you specify the first part of the file name (the prefix) for the commonly used files, Fluent automatically appends or detects an appropriate suffix. For example, to write a case file named `myfile.cas.h5` with the default settings, just specify the prefix `myfile` in the **Select File** dialog box (Figure 3.1: The Select File Dialog Box (p. 949)) and `.cas.h5` is automatically appended. Similarly, to read the case file named `myfile.cas.h5` into Fluent, you can just specify `myfile` and Ansys Fluent automatically searches for a file of that name with the suffix `.cas.h5`.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 3.1: The Select File Dialog Box



The default file suffix for case and data files, PDF (Probability Density Function) files, DTRM ray files, profiles, scheme files, journal files, and so on, are automatically detected and appended. The appropriate default file suffix appears in the **Select File** dialog box for each type of file. Note that for case, data, and mesh files, you can use the **Files of type** drop-down menu in the **Select File** dialog box to specify whether you want to read / write only the default CFF files or all formats (that is, both CFF and the legacy format); you can also change the order in which Fluent prioritizes these formats through the **Default Format for I/O** drop-down in the **Preferences** dialog box or through the `file/cff-files?` text command (for further details, see Reading and Writing Files in the Legacy Format (p. 958) and Reading Mesh Files (p. 953)).

3.1.2. Binary Files

When you write a case, data, or ray file, a binary file is saved by default. Binary files take up less memory than text files and can be read and written by Ansys Fluent more quickly.

Note:

You cannot read and edit a binary file, as you can do for a text file.

To save a text file, turn off the **Write Binary Files** option in the **Select File** dialog box when you are writing the file. Note that by default your case and data files will always be binary; if you require a

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

text file for a case or data file, you must write it in the legacy format (Reading and Writing Files in the Legacy Format (p. 958)).

3.1.3. Detecting File Format

When you read a case, data, mesh, PDF, or ray file, Fluent automatically determines whether it is a text (formatted) or binary file.

3.1.4. Recent File List

At the bottom of the **File/Read** ribbon tab submenu there is a list of four Ansys Fluent case files that you most recently read or wrote. To read one of these files into Ansys Fluent, select it in the list. This allows you to read a recently used file without selecting it in the **Select File** dialog box. If you want to change the number of recent files listed, you can change the **Number of files recently used** setting in the **Preferences** dialog box, which you can access by selecting **Preferences...** from the **File** ribbon tab.

Note that the files listed in this submenu may not be appropriate for your current session (for example, a 3D case file can be listed even if you are running a 2D version of Ansys Fluent). Also, if you read a case file using this shortcut, the corresponding data file is read only if it has the same base name as the case file (for example, `file1.cas.h5` and `file1.dat.h5`) *and* it was read/written with the case file the last time the case file was read/written.

Note:

Reading a case file from the **Recent Files** list in the Fluent Launcher will only load a case file and not the associated data file, even if the data file has the same base name and is stored in the same directory as the case file.

3.1.5. Reading and Writing Compressed Files

For more information, see the following sections:

3.1.5.1. Reading Compressed Files

3.1.5.2. Writing Compressed Files

3.1.5.1. Reading Compressed Files

You can use the **Select File** dialog box to read files (such as legacy mesh, case, and data files) that have been compressed using `compress` or `gzip`. If you select a compressed file with a `.Z` extension, Ansys Fluent automatically opens `zcat` to import the file. If you select a compressed file with a `.gz` extension, Fluent opens `gunzip` to import the file.

For example, if you select a file named `flow.msh.gz`, Fluent reports the following message indicating that the result of the `gunzip` is imported into Ansys Fluent via an operating system pipe.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
Reading "\"| gunzip -c \"Y:\flow.msh.gz\"\""...
```

Note:

In addition to `.gz` and `.Z` compression, Ansys Fluent can also handle `.bz2` compressed files.

Important:

- For Windows systems, only files that were compressed with `gzip` (that is, files with a `.gz` extension) can be read. Files that were compressed with `compress` cannot be read into Ansys Fluent on a Windows machine.
- Do not read a compressed ray file; Ansys Fluent cannot access the ray tracing information properly from a compressed ray file.

3.1.5.2. Writing Compressed Files

You can use the **Select File** dialog box to write a compressed file by appending a `.Z` or `.gz` extension onto the file name.

For example, if you enter `flow.gz` as the name for a legacy case file, Fluent reports the following message:

```
Writing "| gzip -cfv > Y:\flow.cas.gz"...
```

The status message indicates that the case file information is being piped into the `gzip` command, and that the output of the compression command is being redirected to the file with the specified name. In this particular example, the `.cas` extension is added automatically.

Note:

In addition to `.gz` and `.Z` compression, Ansys Fluent can also handle `.bz2` compressed files.

Important:

- For case and data files, such compression is not applied for the default Common Fluids Format (CFF), but instead the `.Z` or `.gz` extension is simply incorporated into the name (such as `.gz.cas.h5`). This compression is only used for the legacy format (such as when you have entered the following text command: `file/cff-files? no`).
- For Windows systems, such compression can be performed only with `gzip`. That is, you can write a compressed file by appending `.gz` to the name, but appending `.Z` does not compress the file.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Do not write a compressed ray file; Ansys Fluent cannot access the ray tracing information properly from a compressed ray file.

3.1.6. Tilde Expansion (Linux Systems Only)

On Linux systems, if you specify `~/` as the first two characters of a file name, the ~ is expanded as your home directory. Similarly, you can start a file name with `~username/`, and the `~username` is expanded to the home directory of "username". If you specify `~/file` as the case file to be written, Ansys Fluent saves the file `file.cas.h5` in your home directory. You can specify a subdirectory of your home directory as well: if you enter `~/cases/file.cas.h5`, Ansys Fluent saves the file `file.cas.h5` in the `cases` subdirectory.

3.1.7. Automatic Numbering of Files

There are several special characters that you can include in a file name. Using one of these character strings in your file name provides a shortcut for numbering the files based on various parameters (that is, iteration number, time step number, or total number of files saved so far), because you need not enter a new file name each time you save a file. (See also Automatic Saving of Case and Data Files (p. 958) for information about saving and numbering case and data files automatically.)

- For transient calculations, you can save files with names that reflect the time step number at which they are saved by including the character string `%t` in the file name. For example, you can specify `contours-%t.ps` for the file name, and Fluent saves a file with the appropriate name (for example, `contours-0001.ps` if the solution is at the first time step).

  This automatic saving of files with the time step number should not be used for steady-state cases, since the time step number will always remain zero.

- For transient calculations, you can save files with names that reflect the flow-time at which they are saved by including the character string `%f` in the file name. The usage is similar to `%t`. For example, when you specify `filename-%f.ps` for the file name, Fluent will save a file with the appropriate name (for example, `filename-005.000000.ps` for a solution at a flow-time of 5 seconds). By default, the flow-time that is included in the file name will have a field width of 10 and 6 decimal places. To modify this format, use the character string `%` *x* `.` *y* `f`, where *x* and *y* are the preferred field width and number of decimal places, respectively. Ansys Fluent will automatically add zeros to the beginning of the flow-time to achieve the prescribed field width. To eliminate these zeros and left align the flow-time, use the character string `%-` *x* `.` *y* `f` instead.

  This automatic saving of files with flow-time should not be used for steady-state cases, since the flow-time will always remain zero.

- To save a file with a name that reflects the iteration at which it is saved, use the character string `%i` in the file name. For example, you can specify `contours-%i.ps` for the file name, and Fluent saves a file with the appropriate name (for example, `contours-0010.ps` if the solution is at the 10th iteration).

- To save a picture file with a name that reflects the total number of picture files saved so far in the current Fluent session, use the character string %n in the file name. This option can be used only for picture files.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The default field width for `%i`, `%t`, and `%n` formats is 4. You can change the field width by using `%` *x* `i`, `%` *x* `t`, and `%` *x* `n` in the file name, where *x* is the preferred field width.

3.1.8. Disabling the Overwrite Confirmation Prompt

By default, if you ask Ansys Fluent to write a file with the same name as an existing file in that folder, it will ask you to confirm that it is "OK to overwrite" the existing file. If you do not want Fluent to ask you for confirmation before it overwrites existing files, you can use the **Batch Options** dialog box (for details, see Batch Execution Options (p. 206) in the Getting Started (p. 1) part of this manual). Alternatively, enter the `file/confirm-overwrite?` text command and answer `no` (see Text User Interface (TUI) (p. 945) for the text user interface commands).

3.2. Reading Mesh Files

Mesh files are created using the mesh generators (Ansys Meshing, the meshing mode of Fluent, Fluent Meshing, GAMBIT, GeoMesh, and PreBFC), or by several third-party CAD packages. From the point of view of Ansys Fluent, a mesh file is a subset of a case file (described in Reading and Writing Case Files (p. 956)). The mesh file contains the coordinates of all the nodes, connectivity information that tells how the nodes are connected to one another to form faces and cells, and the zone types and numbers of all the faces (for example, wall-1, pressure-inlet-5, symmetry-2).

The mesh file does not contain any information on boundary conditions, flow parameters. For information about meshes, see Reading and Manipulating Meshes (p. 1129).

To read a native-format mesh file (that is, a mesh file that is saved in the Ansys Fluent format, which includes FLUENT 5/6, Fluent/UNS, or RAMPANT meshes) into Fluent, use the **File/Read/Mesh...** ribbon tab item.

File → **Read** → **Mesh...**

Note that you can also use the **File/Read/Case...** ribbon tab item (described in Reading and Writing Case Files (p. 956)), because a mesh file is a subset of a case file. Ansys Meshing, the meshing mode of Fluent, Fluent Meshing, GAMBIT, and GeoMesh can all write a native-format mesh file. For information about these files and how they are created, see Ansys Meshing Mesh Files (p. 1151), Fluent Meshing Mode Mesh Files (p. 1151), Fluent Meshing Mesh Files (p. 1151), GAMBIT Mesh Files (p. 1151), and GeoMesh Mesh Files (p. 1152).

If after reading in a mesh file (or a case and data file), you would like to read in another mesh file, the Read Mesh Options Dialog Box (p. 5349) will open, where you can choose to

- Discard the case and read in a new mesh.
- Replace the existing mesh.

You also have the option to have the **Scale Mesh** dialog box appear automatically for you to check or scale your mesh, which in general is the recommended practice. For this to happen, enable **Show Scale Mesh Panel After Replacing Mesh**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

By default, when reading mesh files, Fluent uses the Common Fluids Format (CFF) which is built on the Hierarchical Data Format (HDF5). To read CFF files, use the ribbon tab item or text command and simply append `.msh.h5` to the file name.

Note:

In order to avoid changing preferences when reading mesh files, while using the **Select File** dialog box, as long as the file exists, Fluent will respect your choice of file regardless of the type that is selected for the **Files of type** field. Even if there is no explicit file type extension provided when specifying a mesh file, Fluent will initially look for a `.msh.h5` file type, followed by a `.msh.gz` file type, and finally a `.msh` file type.

CFF and HDF files are always binary and make use of built-in compression. Thus, they cannot be viewed in a text editor. However, third-party tools are available that enable you to open and explore the contents of files saved in HDF format.

You can read legacy mesh files (with the `.msh` extension) using one of the following options:

- When reading mesh files using the **File/Read/Mesh...** ribbon tab item, you can look for files written in the legacy mesh format by selecting **All Mesh Files (*.msh* *.MSH*)** from the **Files of type** drop-down list in the **Select File** dialog box.
- To read legacy mesh files in the current session, you can use the following text command: `file/cff-files? no`; to read legacy mesh files in the current and future sessions, open the **General** tab of the **Preferences** dialog box and select **Legacy** from the **Default Format for I/O** drop-down list. Note that both of these actions will also result in you reading and writing legacy case and data files.

For information on importing an unpartitioned mesh file into the parallel version of Fluent using the partition filter, see Using the Partition Filter (p. 4338).

For information about reading surface mesh files, see Reading Surface Mesh Files (p. 1159).

3.3. Reading and Writing Case and Data Files

Information related to the Ansys Fluent simulation is stored in both the case file and the data file. The commands for reading and writing these files are described in the following sections, along with commands for the automatic saving of case and data at specified intervals.

By default (that is, when the `file/cff-file?` text command is set to `yes`), the case and data files are written in the Common Fluids Format (CFF), which is built on the Hierarchical Data Format (HDF5). CFF files are always binary and make use of built-in compression. Thus, they cannot be viewed in a text editor. However, third-party tools are available that allow you to open and explore the contents of files saved in CFF / HDF.

You can specify the I/O mode and/or the CFF compression level through text commands:

```
>file/cff-options/compression-level
compression-level [0-9]? [1] 4

>file/cffio-options/io-mode
CFF I/O Modes:
 1. HOST
 2. NODE0
 3. PARALLEL INDEPENDENT
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
 4. PARALLEL COLLECTIVE
Enter Mode Number: [1] 2

NODE0 mode selected.
```

The compression level can be set from 0 (no compression; fastest) to 9 (maximum compression; slowest); the default level is 1. Note that increasing the compression level may not necessarily yield a corresponding decrease in file size.

The available I/O modes are:

1. `HOST`: I/O is done serially by the host process. This is the default mode when the file path is not accessible to compute-node 0.
2. `NODE0`: I/O is done serially by the node 0 process. This is the default mode when the file path is accessible to compute-node 0.
3. `PARALLEL INDEPENDENT`: I/O is done in a parallel file system using the independent mode of MPI I/O.
4. `PARALLEL COLLECTIVE`: I/O is done in a parallel file system using the collective mode of MPI I/O.

Note:

- The Parallel I/O modes do not support compression. If you have specified a non-zero level for compression and you switch to one of the parallel modes, the compression will be set to 0.
- For information on the supported parallel file systems (as well as specific configuration and usage issues), contact your Ansys Fluent support engineer.

When using the double-precision solver, you can specify that the data written in CFF data files is single precision (and thus sacrifice some precision in order to reduce the size of the files) by using the following text command prior to writing:

`file` → `cffio-options` → `single-precision-data?`

Important:

- If you adapt the mesh, you must save a new case file as well as a data file. Otherwise, the new data file will not correspond to the case file (for example, they will have different numbers of cells). If you have not saved the latest case or data file, Ansys Fluent will warn you when you try to exit the program.
- Note that if you define the time step size for a transient simulation and then subsequently read a data file that has a different value, the time step size will be overwritten with the value from the data file.

For additional information, see the following sections:

3.3.1. Reading and Writing Case Files

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.3.2. Reading and Writing Data Files
3.3.3. Reading and Writing Case and Data Files Together
3.3.4. Reading and Writing Files in the Legacy Format
3.3.5. Automatic Saving of Case and Data Files

3.3.1. Reading and Writing Case Files

Case files contain the mesh, boundary and cell zone conditions, and solution parameters for a problem. It also contains the information about the user interface and graphics environment. The commands used for reading case files can also be used to read native-format mesh files (as described in Reading Mesh Files (p. 953)) because the mesh information is a subset of the case information.

Note:

Prior to reading in a case file, you can review some of the case properties to help ensure you are loading the intended file. The available summary includes the Fluent version that last saved the case, case precision and dimension, and the total number of cells, faces, and nodes. To preview the case information:

- From the Fluent Launcher: Select the case file and click **Case File Info**.
- From a running Fluent session: Use the **File**>**Read**>**Case/Mesh Info...** menu option and select the desired case file.

Select the **File/Read/Case...** ribbon tab item to open the **Select File** dialog box.

File → **Read** → **Case...**

Read a case file using the **Select File** dialog box. Note that the **Display Mesh After Reading** option in the **Select File** dialog box allows you to have the mesh displayed automatically after it is read.

Select the **File/Write/Case...** ribbon tab item to open the **Select File** dialog box.

File → **Write** → **Case...**

Write a case file using the **Select File** dialog box.

Note:

The working directory is automatically updated to the directory where the case file is written, when the writing is initiated from within the graphical user interface (writing using the `file/write-case` text command does not update the working directory). *You can determine the present working directory by entering pwd in the Console.*

When Ansys Fluent reads a case file, it first looks for a file with the exact name you typed. If a file with that name is not found, it searches for the same file with different extensions, prioritizing CFF files by default. When Ansys Fluent writes a case file, `.cas.h5` is added by default to the name you type.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the single-precision version of Ansys Fluent, nodal coordinates are written by default in double precision, in order to ensure that there is no loss of precision if the case file is read back into the meshing mode. You can reduce the precision and case file size by using the following text command prior to writing:

`file → single-precision-coordinates?`

3.3.2. Reading and Writing Data Files

Data files contain the values of the specified flow field quantities in each mesh element and the convergence history (residuals) for that flow field.

1. After reading a mesh or case file, select the **File/Read/Data...** ribbon tab item to open the **Select File** dialog box.

   File → Read → Data...

2. Read a data file using the **Select File** dialog box.

3. After generating data for a case file, select the **File/Write/Data...** ribbon tab item to open the **Select File** dialog box.

   File → Write → Data...

4. Write a data file using the **Select File** dialog box.

When Ansys Fluent reads a data file, it first looks for a file with the exact name you typed. If a file with that name is not found, it searches for the same file with different extensions, prioritizing CFF files by default. When Ansys Fluent writes a data file, `.dat.h5` is added by default to the name you type.

3.3.3. Reading and Writing Case and Data Files Together

A case file and a data file together contain all the information required to restart a solution. Case files contain the mesh, boundary and cell zone conditions, and solution parameters. Data files contain the values of the flow field in each mesh element and the convergence history (residuals) for that flow field.

You can read a case file and a data file together by using the **Select File** dialog box opened by selecting the **File/Read/Case & Data...** ribbon tab item. To read both files, select the appropriate case file, and the corresponding data file (same name with `.dat.h5` suffix) is also read. Note that the **Display Mesh After Reading** option in the **Select File** dialog box allows you to have the mesh displayed automatically after it is read.

File → Read → Case & Data...

To write a case file and a data file, select the **File/Write/Case & Data...** ribbon tab item.

File → Write → Case & Data...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.3.4. Reading and Writing Files in the Legacy Format

When reading and writing case and data files, you can optionally use the legacy format (`.cas` and `.dat`, respectively), which was the default format prior to Release 2020 R1. To read or write legacy case/data files in the current session only, enter the following text command prior to reading or writing:

```
>file/cff-files? no
```

To read or write legacy files in both the current session and future sessions, open the **General** tab of the **Preferences** dialog box, select **Legacy** from the **Default Format for I/O** drop-down list, and click **OK**.

File → Preferences...

Ansys Fluent can read and write either text or binary case and data files in the legacy format. Binary files require less storage space and are faster to read and write. By default, Ansys Fluent writes files in binary format. To write a text file, disable the **Write Binary Files** check button in the **Select File** dialog box. In addition, you can read and write either text or binary files in compressed formats (For details, see Reading and Writing Compressed Files (p. 950)). Ansys Fluent automatically detects the file type when reading.

Important:

Computed values for walls, symmetry boundaries, and other surfaces where the results are interpolated from cell centers to the surface, are not retained when you save to the legacy format. This causes issues when legacy case and data files are read into postprocessing applications (such as Ansys EnSight) that do not have an integrated solver, because these applications cannot accurately interpolate the values back to these surfaces. This issue is resolved if you save case and data files in HDF format.

For legacy data files, you can output additional quantities for postprocessing in other applications using the Data File Quantities dialog box (see Setting Data File Quantities (p. 1026)).

For information about the format of legacy case and data files, see Legacy Case and Data File Formats (p. 5675).

3.3.5. Automatic Saving of Case and Data Files

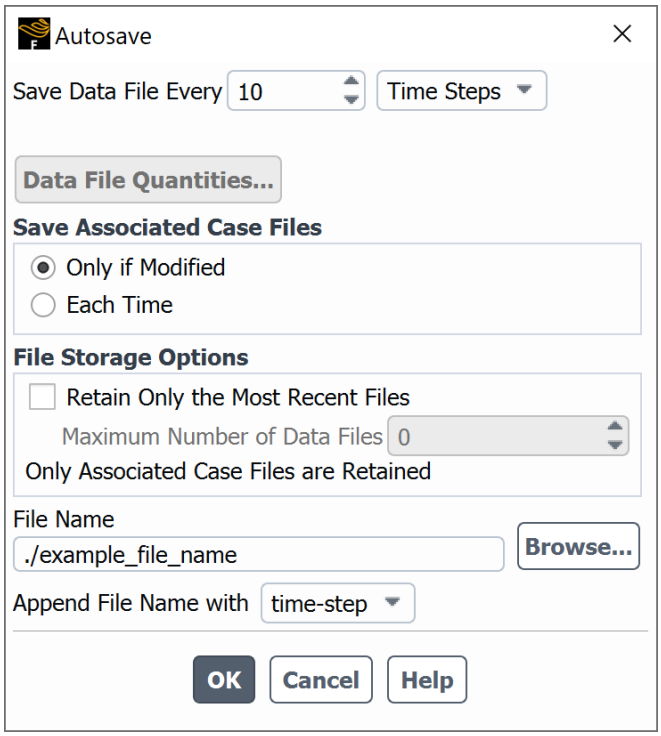
You can request Ansys Fluent to automatically save case and data files at specified intervals during a calculation. This is especially useful for time-dependent calculations, since it allows you to save the results at different time steps, flow times, or crank angles (for in-cylinder simulations) without stopping the calculation and performing the save manually. You can also use the autosave feature for steady-state problems, and thus examine the solution at different stages in the iteration history.

Automatic saving is specified using the Autosave Dialog Box (p. 5190) (Figure 3.2: The Autosave Dialog Box (p. 959)), which is opened by clicking the **Edit...** button next to the **Autosave Every** text box in the **Calculation Activities** task page.

Calculation Activities

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 3.2: The Autosave Dialog Box



Specify how often you would like to save your modified files by entering the frequency in the **Save Data File Every** number-entry field. **Save Data File Every** is set to zero by default, indicating that no automatic saving is performed.

For transient in-cylinder simulations, specify whether you want to save the data by **Time Steps** or **Crank Angles**.

If you choose to save the case file only if it is modified, then select **Only if Modified** under **Save Associated Case Files**. Note that the case file will be saved whether you make a manual change, or if Ansys Fluent makes a change internally during the calculation. If you choose to save the case file every time the data file is saved, then select **Each Time**.

Important:

To save only the data files, use the following TUI option:

`file` → `auto-save` → `case-frequency` → `if-mesh-is-modified`

This will result in the options in the **Save Associated Case Files** group box being disabled in the **Autosave** dialog box. In essence, this TUI command forces Ansys Fluent to the save case file only when the mesh is modified. It does not disable case file saving, but reduces it to an absolute minimum. This is necessary to do so since you cannot read a data file without a case file containing a matching mesh.

For steady-state solutions, specify the frequency in iterations. For transient solutions, specify it in time steps (unless you are using the explicit time stepping formulation, in which case specify the frequency in iterations). For transient in-cylinder solutions, you can choose to specify the frequency in crank

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

angles. If you define a frequency of 5, for example, a case file is saved every 5 iterations, time steps, or crank angles.

If you have limited disk space, restrict the number of files saved by Ansys Fluent by selecting the **Retain Only the Most Recent Files** option. When selected, enter the **Maximum Number of Data Files** you would like to retain. Note that the case and data files are treated separately with regard to the maximum number of files saved when overwriting. For example, if the value of **Maximum Number of Data Files** is set to five, Ansys Fluent saves a maximum of five case and five data files, irrespective of the frequency. After the maximum limit of files has been saved, Ansys Fluent begins overwriting the earliest existing file.

Note:

When the **Retain only the Most Recent Files** option is selected, the solution history currently in memory will be discarded and the solution history reset.

If you have requested the writing of files in the legacy format (see Reading and Writing Files in the Legacy Format (p. 958)), you may need to specify that additional data quantities are written if you want to postprocess in alternative applications. When you have generated data—either by initializing the solution or running the calculation—you can view the list of standard quantities that will be written to the legacy data file as a result of the autosave, and select additional quantities. Click the **Data File Quantities...** button to open the **Data File Quantities** dialog box, and make any necessary selections. For details, see Setting Data File Quantities (p. 1026).

Enter a root name for the autosave files in the **File Name** text box. When the files are saved, a number will be appended to this root name to indicate the point at which it was saved during the calculation: for steady-state solutions, this will be the iteration number, whereas for transient solutions it will be either the time step number, flow time, or crank angle (depending on your selection in the step that follows). An extension will also be automatically added to the root name (`.cas.h5` or `.dat.h5`, by default). If the specified **File Name** ends in `.gz` or `.Z` and you have requested the reading and writing of files in the legacy format, appropriate file compression is performed; for details about file compression, see Reading and Writing Compressed Files (p. 950).

For transient calculations, make a selection from the **Append File Name with** drop-down list to indicate whether you want the root file name to be appended with the **time-step**, **flow-time** or **crank-angle** (see Figure 3.2: The Autosave Dialog Box (p. 959)). If you select either of the latter two, you can set the **Decimal Places in File Name** to determine the ultimate width of the file name.

Consider a transient case for which you want to save your case and data files at known time steps. The procedure you would follow is to first set the frequency in the **Save Data File Every** text box. Select **Each Time** if you want both case and data files saved at the same interval. Then enter `my_file` for the **File Name**. Finally, select **time-step** from the **Append File Name with** drop-down list. An example of the resulting files saved would be

```
my_file-0005.cas.h5
```

```
my_file-0005.dat.h5
```

indicating that these files were saved at the fifth time step.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can revise the instructions for the previous example to instead save case and data files at known flow times, by selecting **flow-time** from the **Append File Name with** drop-down list. The default **Decimal Places in File Name** will be six. An example of the resulting files saved would be

```
my_file-0.500000.cas.h5
```

```
my_file-0.500000.dat.h5
```

indicating that these files were saved at a flow time of 0.5 seconds.

For steady-state and transient cases, you have the option of automatically numbering the files (as described in Automatic Numbering of Files (p. 952)), and thereby include further information about when the files were saved. This involves the addition of special characters to the **File Name**. For example, you may want the file names to convey the flow times with their corresponding time step numbers (transient cases only). Select **time-step** from the **Append File Name with**, and enter a **File Name** that ends with `-%f` to automatically number the files with the flow time. Thus, entering a **File Name** of `filename-%f` could result in a saved case file named `filename-000.500000-0010.cas.h5`. The conventions used in this example can be explained as follows:

- `filename-` is the file name you entered when autosaving your solution.
- `000.500000` is the result of the special character `%f` added to the file name, and is the flow time. This flow time has a field width of ten characters, which allows for six decimal places (as discussed in Automatic Numbering of Files (p. 952)).
- `-0010` is the appended **time-step**, as designated by the selection in the **Append File Name with** drop-down list.
- `.cas.h5` is the file extension automatically added when using the autosave option.

In addition to the case and data files, a single project file (`.flprj`) is also written during the calculation; it is recommended that you read this file whenever possible when postprocessing in CFD-Post and EnSight, as it contains metadata such as the time values for files exported from transient simulations. Note that if the root name of the autosave files is the same as that used for exporting Common Fluids Format - Post files during a transient calculation (using the **Automatic Export** dialog box) and Fluent is set up to save files in the default Common Fluids Format (CFF), then the metadata from both operations will be written to the same project file.

All of the autosave inputs are stored in memory when you click **OK** in the **Autosave** dialog box, and can then be saved with the case file.

Important:

The default path is `./<filename>` indicating that the autosaved files will be saved in the working directory of the current Fluent session. If you hard code a directory, then the files will always be saved at that location, regardless of the working directory of the current Fluent session.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.4. Reading Settings Only for Mesh or Case Files with Large Cell Counts

When setting up a simulation that has a large cell count, you can read the case settings only of a mesh or case file, without any of the data associated with the mesh. This allows you to review and/or modify the settings without having to wait for the reading of the mesh, on a machine that does not have the memory needed for the mesh (such as a personal computer or laptop). When you read the case settings, note that it is not possible to manipulate the mesh or run calculations, or set up any add-on modules besides the adjoint solver and the population balance model.

To read the case settings for a simulation, perform the following steps:

1. Open up a Fluent session.

   Note:

   - You can enable the **Pre/Post Only** option in Fluent Launcher / use the `-post` command line option.
   - Using a single processor (`-t1`) is recommended, as it will minimize the memory usage. Multiple processors are not necessary for reading the case settings only.

2. Read the case settings for a mesh or case file in the Common Fluids Format (with the `.msh.h5` or `.cas.h5` extension, respectively). This can be done using one of the following methods:

   - Select the **File/Read/Case Settings Only...** ribbon tab item, and use the **Select File** dialog box that opens to select a file.

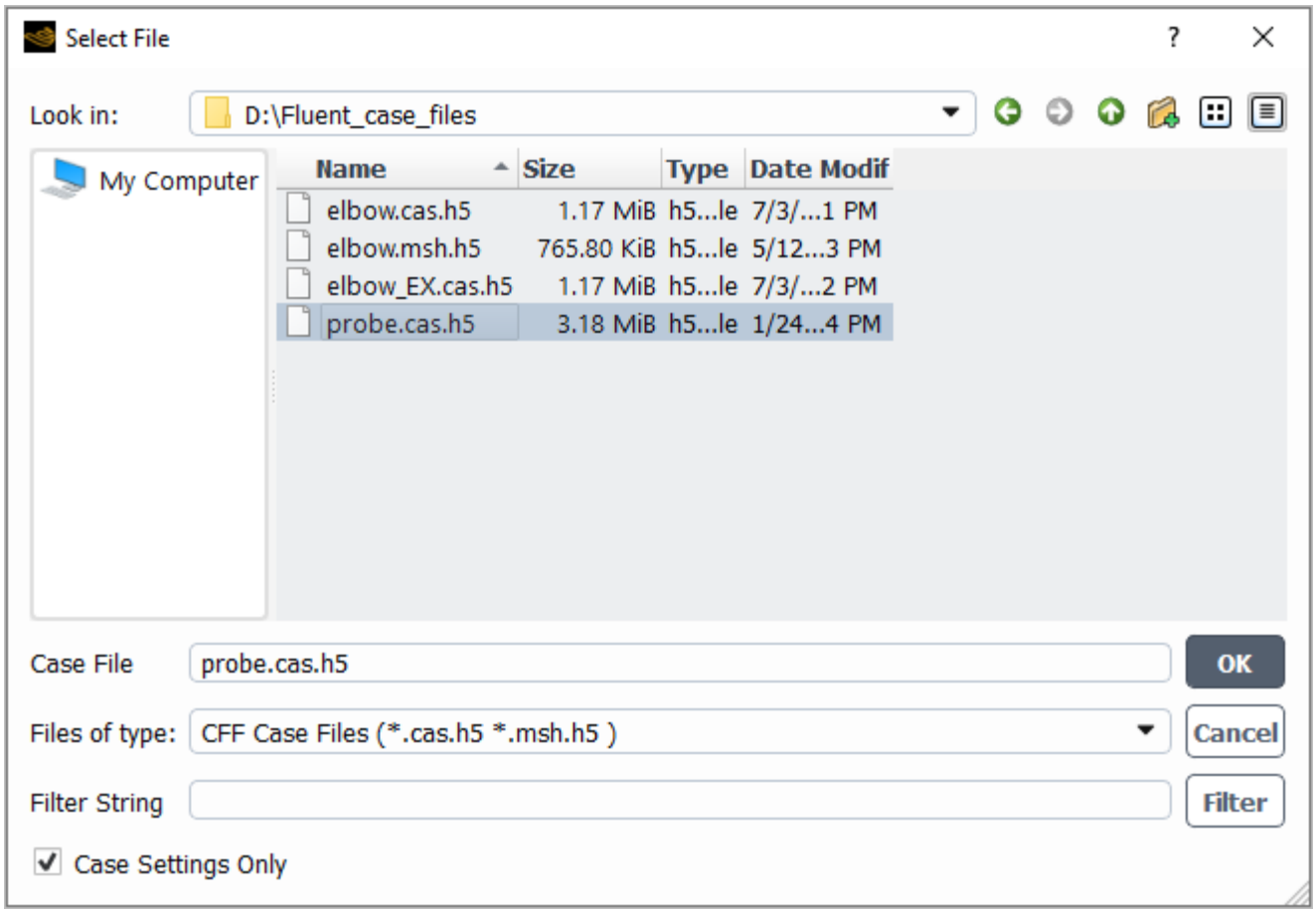
     File → Read → Case Settings Only...

   - Select the **File/Read/Case...** ribbon tab item to open the **Select File** dialog box.

     File → Read → Case...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 3.3: The Select File Dialog Box for Reading Case Settings Only



Enable the **Case Settings Only** option in the **Select File** dialog box, select a file, and click **OK**.

- Use the following text command:

  `file → read-case-settings`

3. Review and/or modify the settings as necessary. For any operation that is related to the manipulation of the mesh or the running of the calculation, the graphical user interface (GUI) is grayed out, and the text command is unavailable; additionally, the `define/models/addon-module` text command is unavailable. Note that you can initialize the case in order to access settings that are only available after initialization and/or warnings generated during initialization, but data will not be generated.

4. Write the revised settings to a case file. If the name of the case file to which you write is the same as the one you originally read, the writing will be fast and will not require a large amount of memory / file space; if you write using a new case file name, you will have to wait for the copying of the data associated with the mesh from the original case file, and you must have the memory / file space available to allow the copying.

3.5. Reading Fluent/UNS and RAMPANT Case and Data Files

Case files created by Fluent/UNS 3 or 4 or RAMPANT 2, 3, or 4 can be read into Ansys Fluent in the same way that current case files are read (see Reading and Writing Case and Data Files (p. 954)). If you read a case file created by Fluent/UNS, Ansys Fluent selects **Pressure-Based** in the **Solver** group box of the

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

General Task Page (p. 4684). If you read a case file created by RAMPANT, Ansys Fluent selects **Density-Based** in the **Solver** group box of the General Task Page (p. 4684), as well as **Explicit** from the **Formulation** drop-down menu in the Solution Methods Task Page (p. 5157).

Data files created by Fluent/UNS 4 or RAMPANT 4 can be read into Ansys Fluent in the same way that current data files are read (see Reading and Writing Case and Data Files (p. 954)).

3.6. Reading and Writing Profile Files

Boundary profiles are used to specify flow conditions on a boundary zone of the solution domain. For example, they can be used to prescribe a velocity field on an inlet plane. For information on boundary profiles, see Profiles (p. 1564). For information about transient profiles, see Transient Cell Zone and Boundary Conditions (p. 1523).

Profiles can also can also be used in expressions. Refer to Profiles (p. 1048) for additional information.

For additional information, see the following sections:

3.6.1. Reading Profile Files

3.6.2. Writing Profile Files

3.6.1. Reading Profile Files

To read the boundary profile files, open the **Select File** dialog box by selecting the **File/Read/Profile...** ribbon tab item.

File → **Read** → **Profile...**

This opens the **Select File** dialog box so that you can read a boundary profile with the standard extension `.prof` or a transient profile in tabular format with the standard extension `.ttab`. CSV profiles are also supported, see CSV Profiles (p. 1567) for additional information. If a profile in the file has the same name as an existing profile, the old profile will be overwritten.

Note:

The maximum number of profiles that can be read into a single Fluent session is 50.

3.6.2. Writing Profile Files

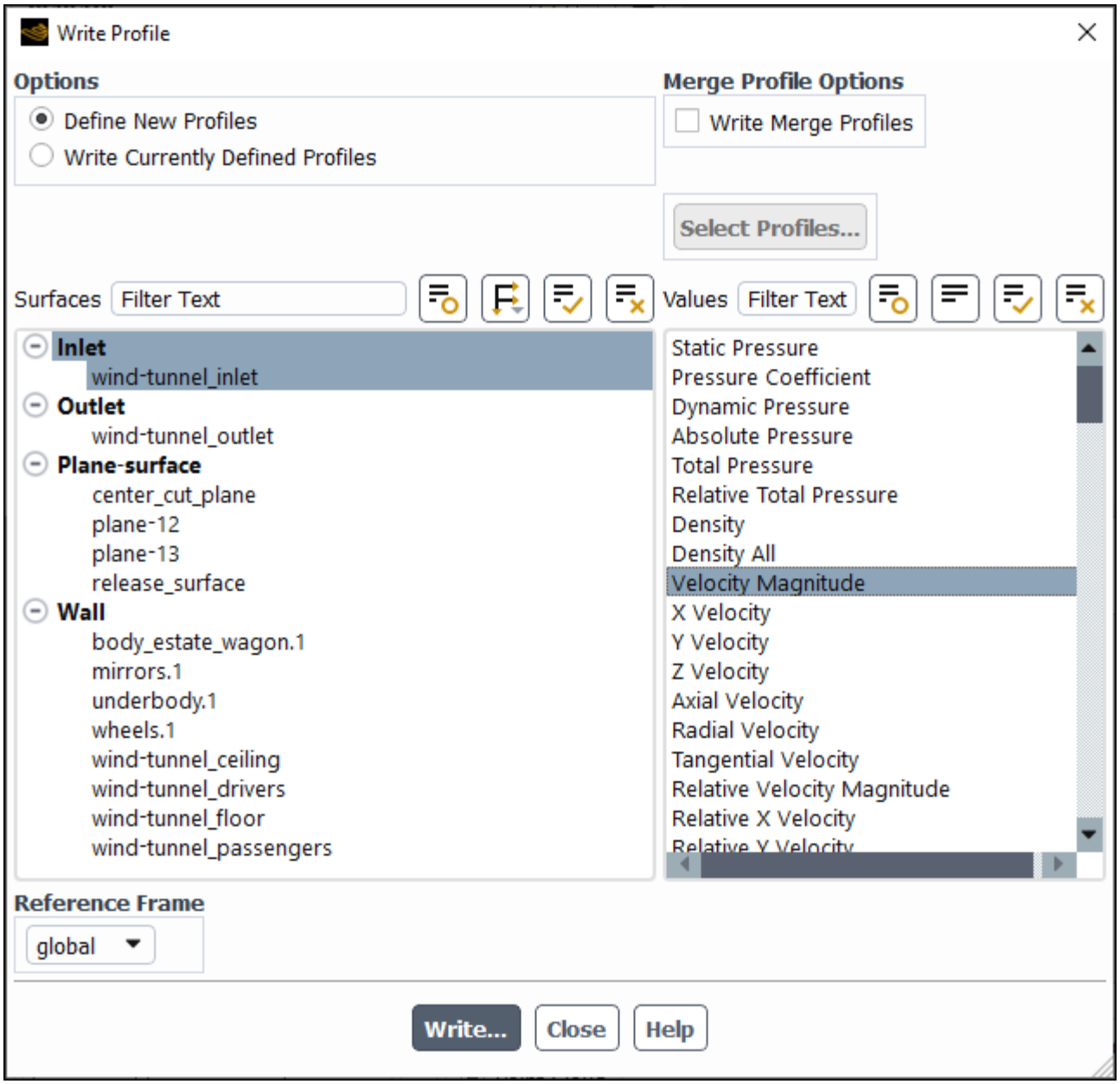
You can also create a profile file from the conditions on a specified boundary or surface. For example, you can create a profile file from the outlet conditions of one case. Then you can read that profile into another case and use the outlet profile data as the inlet conditions for the new case.

To write a profile file, use the Write Profile Dialog Box (p. 4988) (Figure 3.4: The Write Profile Dialog Box (p. 965)).

File → **Write** → **Profile...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 3.4: The Write Profile Dialog Box



1. Retain the default option of **Define New Profiles**.
2. Select the surface(s) where you want data extracted for the profile(s) in the **Surfaces** list.
3. Choose the variable(s) for which you want to create profiles in the **Values** list.
4. Optionally select **Write Merge Profiles**. This writes a `.csv` file with the selected surfaces consolidated into one set of data points. Note that this option is only available if you have selected 2 or more surfaces.
5. Select the desired **Reference Frame** from the drop-down list that will be used for creating this profile.
6. Click **Write...** and specify the profile file name in the resulting **Select File** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Fluent saves the mesh coordinates of the data points for the selected surface(s) and the values of the selected variables at those positions. When you read the profile file back into Fluent, you can select the profile values from the relevant drop-down lists in the boundary condition dialog boxes. The names of the profile values in the drop-down lists will consist of the surface name and the particular variable.

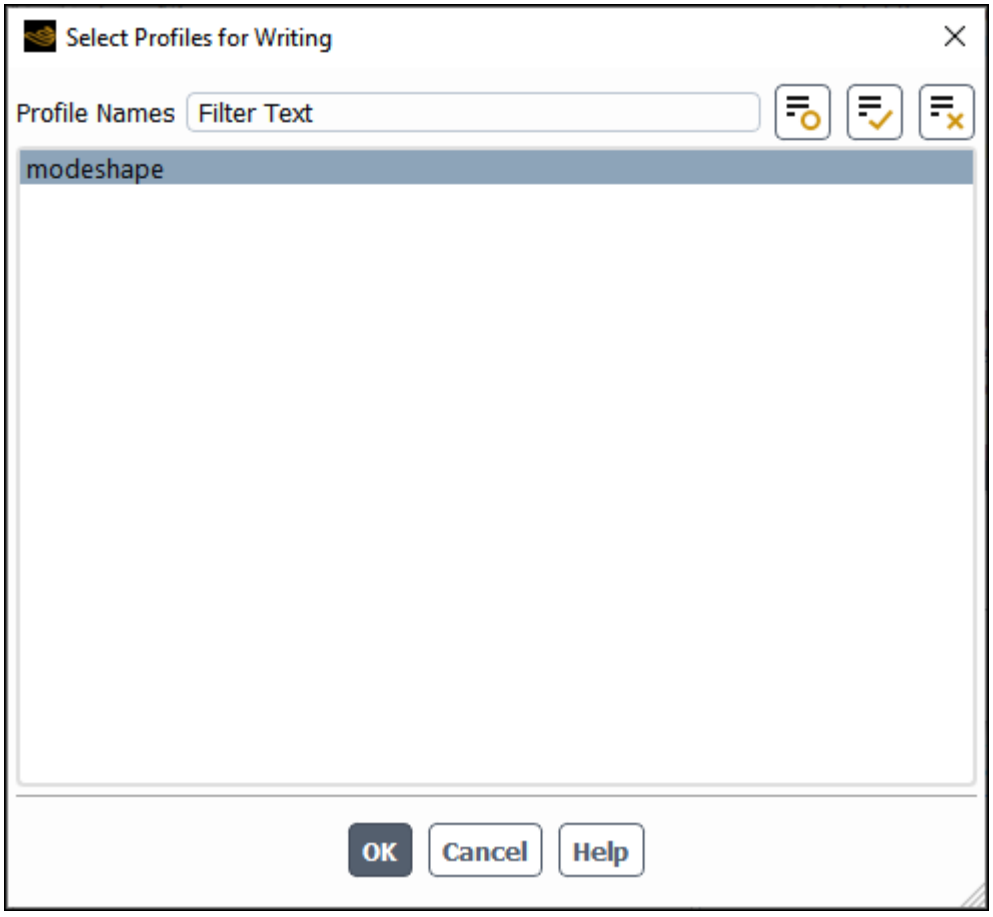
7. Select the **Write Currently Defined Profiles** option:

   - if you have made any modifications to the boundary profiles since you read them in (for example, if you reoriented an existing profile to create a new one).

   - if you want to store one or more of the profiles associated with the current case file in a separate file. *Note that you must click* ***Select Profiles...*** *to select the specific profiles that you want written.*

8. Click **Select Profiles...** to open the **Select Profiles for Writing** box. This allows you pick the individual profiles that you want written.

Figure 3.5: The Select Profiles for Writing Dialog Box



9. Click **OK** to confirm the profile selection.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10. Click **Write...** and specify the file name in the resulting **Select File** dialog box. This file can be read back into Fluent whenever you want to use the profile(s) again.

Note:

- Transformation of point profiles is only available for Cartesian reference frames. *Cylindrical and other reference frame systems are not supported.*
- Scalar point field profiles do not require transformation to a local reference frame.
- Transformation of vector point field profiles (for example, point velocities, acceleration) into local reference frames is not supported.

3.6.2.1. Writing Circumferential-Averaged Profiles

With **Turbo Models** enabled, you have the option to write circumferential-averaged profiles. Circumferential-averaged profiles are available for flow variables on various surfaces including inlet, outlet, interfaces, planes, iso-surfaces, and iso-clips. The determination of the profile type (radial or axial) is internally based on the specific type of machine.

To write a circumferential-average profile, perform the following steps:

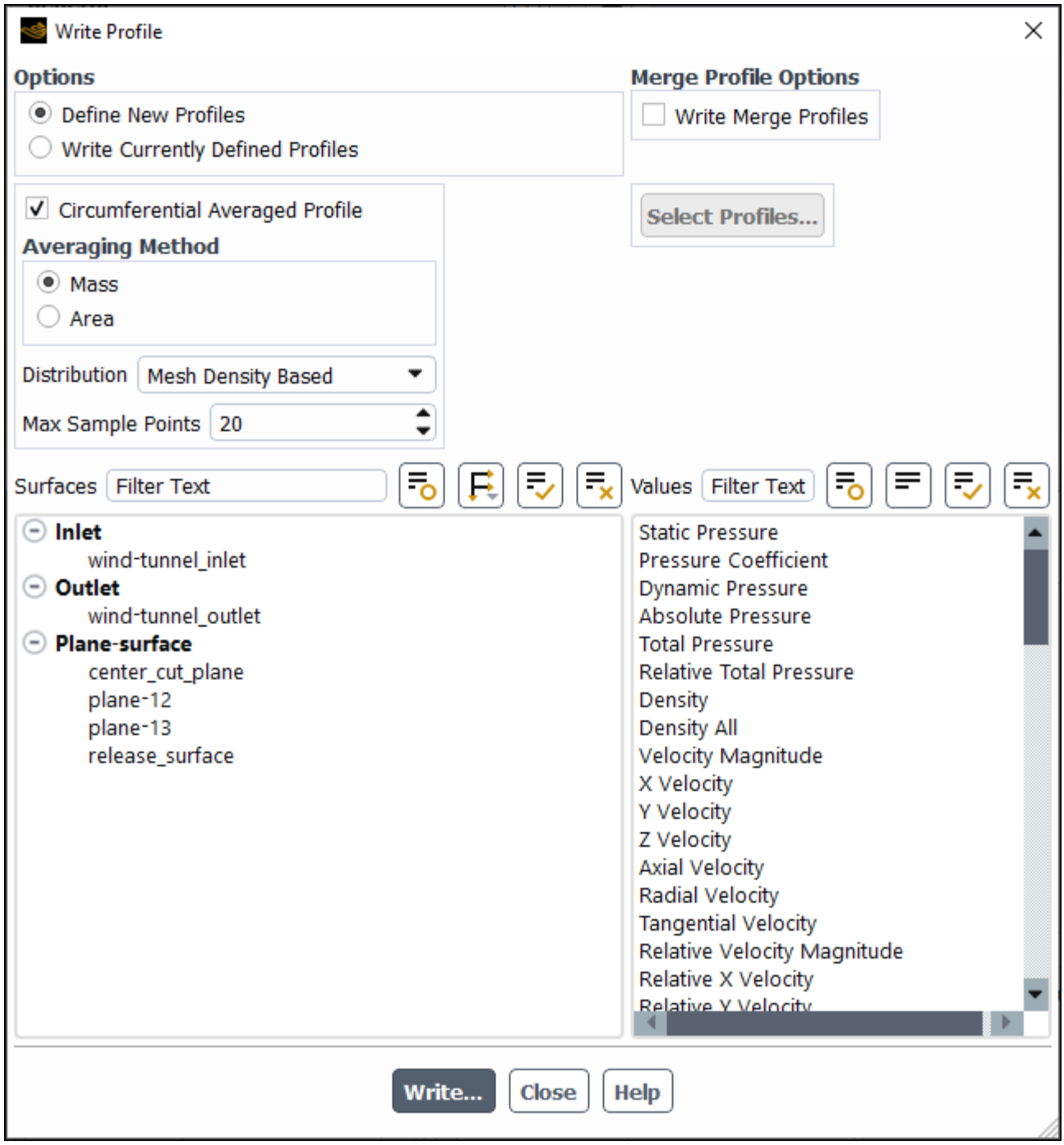
1. Enable **Turbo Models** from the ribbon.

   Domain → **Turbomachinery** → **Turbo Models**

2. On the **Write Profile** dialog box, select **Circumferential Averaged Profile** to reveal additional settings. In most cases, the default settings should be sufficient.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 3.6: The Write Profile Dialog Box



3. Under **Averaging Method**, select **Mass** or **Area**.

4. **Distribution** determines how the averaged values for the profile pitch-wise bands are created. The distribution of bands can either be **Mesh Density Based** or **Equal Distance Based**.

5. **Max Sample Points** specifies the maximum number of bands that are created for sampling.

6. Select the **Surfaces** and **Values** to be included in the profile. If you select `X Velocity`, `Y Velocity`, or `Z Velocity` then Fluent automatically converts them into cylindrical coordinates

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

so that extracted circumferential averaged velocity profile is always written in cylindrical components.

7. Click **Write...** to generate the profile in `.csv` or legacy `.prof` format. Note the main difference between the two formats is the `.csv` format contains additional information in the file header.

You can also access this feature by using the following console command and following the prompts:

```
file/write-circumferential-averaged-profile
```

3.7. Reading Files in Tabular Format

When the tabular input option is available for a solution variable, you can specify the variable as a tabular function of other solution quantities using one or more generic text files. The tabular method uses a bisection algorithm along with local linear interpolation to find the value of the tabulated quantity.

To read the table into Ansys Fluent:

1. Access the **Table File Manager** dialog box in one of the following ways:

   - In the **User-Defined** ribbon tab, click **Read Table** (**User-Defined** group).

     User-Defined → **User-Defined** → **Read Table**

   - In the **File** menu, click **Table File Manager...**.

     File → **Table File Manager...**

2. In the **Table File Manager** dialog box that opens, click **Read...**.

3. Use the **Select File** dialog box to select the file data in tabular format to be read.

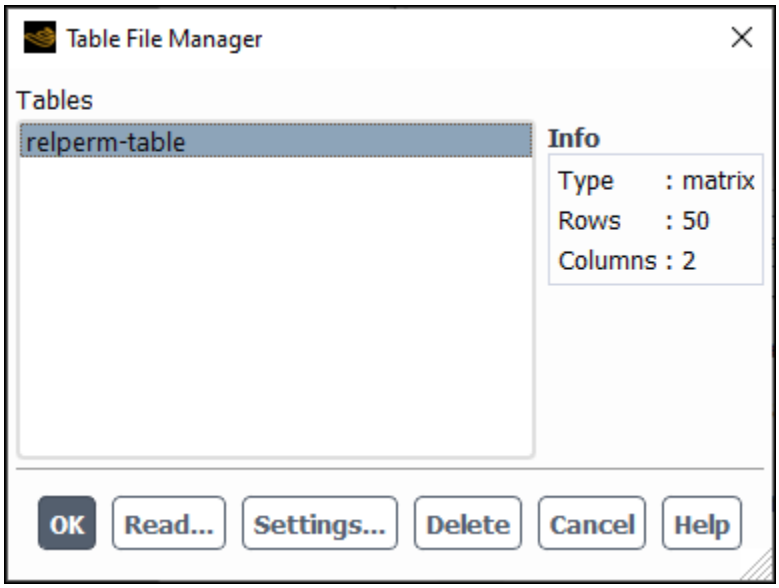
   Note:

   By default, the **Matrix Data Files** is selected as **Files of type**. With this option, all files in your directory will be shown in the **Select File** dialog box. If you want to show only RGP table files, you can select **Real Gas Property (RGP) Table Files (*.rgp)** for **Files of type**.

Once Ansys Fluent loads the file, the table name appears in the **Tables** selection list in the **Table File Manager** dialog box (see Figure 3.7: The Table File Manager Dialog Box (p. 970)). The table attributes, such as the table type and number of rows and columns, are displayed in the **Info** field.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 3.7: The Table File Manager Dialog Box



You can add or delete tables at any time by using the **Read...** or **Delete...** buttons.

3.7.1. Matrix Table Files

The text file can be any tab- (or space-) delimited file and must contain the entries described below. See Capillary Pressure Data in a Tabular Format (p. 1366) for example of a file in tabular format for tabular definition of capillary pressure.

Row 1 – The name of the table. The name should not exceed 128 characters and contain no spaces.

Row 2 – Column names for the table input data.

Row 3 and all consecutive rows – Data points. Note that although there are no limits on the number of rows and columns, the files are intended to be relatively small (a few hundred data points maximum), since they will be stored as part of the case file.

Each row in the table must be terminated by a newline or carriage return with no space or tab following the last entry in the row.

You must provide an adequate number of data points in order to obtain meaningful results for interpolated variables. The values of the interpolated quantity appearing in the first and last rows of the table are used as its lower and upper bounds, respectively.

Unlike locally stored files, these tables, once read in, are written out as part of the Fluent case file. You do not have to maintain local copies of the tables.

3.7.2. Real Gas Property (RGP) Table Files

Real Gas Property (RGP) tables provide thermodynamic and transport material properties as functions of temperature and pressure. Each table may contain data for supercritical state, vapor/liquid subcritical states, and saturation properties.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The detailed description of the contents and format of RGP files can be found in Real Gas Property (RGP) File Contents and Real Gas Property (RGP) File Format in the *CFX-Solver Modeling Guide*. See Using Real Gas Property (RGP) Table Files (p. 1734) for details on using RGP files.

3.8. Reading and Writing Boundary Conditions

To save all currently defined boundary conditions to a file, enter the `file/write-settings` text command and specify a name for the file.

`file` → `write-settings`

Ansys Fluent writes the boundary and cell zone conditions, the solver, and model settings to a file using the same format as the "zone" section of the case file.

To read boundary conditions from a file and to apply them to the corresponding zones in your model, enter the `file/read-settings` text command.

`file` → `read-settings`

Ansys Fluent sets the boundary and cell zone conditions in the current model by comparing the zone name associated with each set of conditions in the file with the zone names in the model. If the model does not contain a matching zone name for a set of boundary conditions, those conditions are ignored.

If you read boundary conditions into a model that contains a different mesh topology (for example, a cell zone has been removed), check the conditions at boundaries within and adjacent to the region of the topological change. This is important for wall zones.

Note:

If the boundary conditions are not checked and some remain uninitialized, the case will not run successfully.

When the `file/read-settings` text command is not used, all boundary conditions get the default settings when a mesh file is imported, allowing the case to run with the default values.

If you want Ansys Fluent to apply a set of conditions to multiple zones with similar names, or to a single zone with a name you are not sure of in advance, you can edit the boundary-condition file saved with the `file/write-settings` command to include *wildcards* (`*`) within the zone names. For example, if you want to apply a particular set of conditions to **wall-12**, **wall-15**, and **wall-17** in your current model, edit the boundary-condition file so that the zone name associated with the desired conditions is `wall-*`.

Note:

The settings file contains only your user-modified settings and does not include default Ansys Fluent parameters. The default parameters may be updated with each new release. Consequently, the usage of the same settings file in different releases of Ansys Fluent does not guarantee the same setup, which can cause solution differences.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.9. Writing a Boundary Mesh

You can write the boundary zones (surface mesh) to a file. This file can be read and used by the meshing mode of Fluent to produce a volume mesh. You may find this feature useful if you are unsatisfied with a mesh obtained from another mesh generation program.

A boundary mesh can be written using the **Select File** dialog box, opened by selecting the **File/Write/Boundary Mesh...** ribbon tab item.

File → Write → Boundary Mesh...

3.10. Reading Scheme Source Files

A Scheme source file can be loaded in three ways: through the menu system as a scheme file, through the menu system as a journal file, or through Scheme itself.

For large source files, use the **Select File** dialog box, opened by selecting the **File/Read/Scheme...** ribbon tab item

File → Read → Scheme...

or use the Scheme `load` function in the console, as shown in the following example:

```
> (load "file.scm")
```

Shorter files can also be loaded with the **File/Read/Journal...** ribbon tab item or the `file/ read-journal` command in the text interface (or its `.` or `source` alias, as shown in the example that follows).

```
>. file.scm

> source file.scm
```

In this case, each character of the file is echoed to the console as it is read, in the same way as if you were typing in the contents of the file.

3.11. Creating and Reading Journal Files

A journal file contains a sequence of Ansys Fluent commands, arranged as they would be typed interactively into the program or entered through the GUI or TUI. The GUI and TUI commands are recorded as Scheme code lines in journal files. You can also create journal files manually with a text editor. If you want to include comments in your file, be sure to put a semicolon (`;`) at the beginning of each comment line. For an example, see Background Execution on Linux Systems (p. 204) in the Getting Started (p. 1) part of this manual.

The purpose of a journal file is to automate a series of commands instead of entering them repeatedly on the command line. Another use is to produce a record of the input to a program session for later reference, although transcript files are often more useful for this purpose. (For details, see Creating Transcript Files (p. 976)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Command input is taken from the specified journal file until its end is reached, at which time control is returned to the standard input (usually the keyboard). Each line from the journal file is echoed to the standard output (usually the screen) as it is read and processed.

Important:

- A journal file is, by design, just a simple record and playback facility. It contains no information about the state in which it was recorded (other than the version of Ansys Fluent that it was created in) or the state in which it is being played back.
- The precision (single or double) that the journal is recorded in may affect playback when it is read into a session that is running in the other precision.

- Be careful not to change the folder while recording a journal file. Also, try to re-create the state in which the journal was written before you read it into the program. For example, if your journal file includes an instruction to save a new file with a specified name, you should check that if a file with that name exists in your folder before you read in your journal file. If a file with that name exists and you read in your journal file, when the program reaches the write instruction, it will prompt for a confirmation to overwrite the old file.

  Since the journal file does not contain any response to the confirmation request, Ansys Fluent cannot continue to follow the instructions of the journal file.

- Other conditions that may affect the program's ability to perform the instructions contained in a journal file can be created by modifications or manipulations that you make within the program.

  For example, if your journal file creates several surfaces and displays data on those surfaces, you must be sure to read in appropriate case and data files before reading the journal file.

  Important:

  At a point of time, only one journal file can be open for recording, but you can write a journal and a transcript file simultaneously. You can also read a journal file at any time.

- Whenever you start recording a journal file, the text command `/file/set-tui-version "XX.X"` is added at the top of the file (where `XX.X` corresponds to the version of Ansys Fluent that is recording the journal file). This text command can help journals created in an older version to work properly when used in a newer version, as it will hide the new text user interface (TUI) prompts and restore the deleted TUI prompts in that newer version.

  If you are writing a journal file in a text editor, it is recommended that you add `/file/set-tui-version "XX.X"` at the top of the file.

  Note:

  – The specified version must be within two full releases of the version that is running the journal.

  – To specify version 2024 R2, enter `"24.2"` for `"XX.X"`.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Whether you choose to type the text command in full or use partial strings (as described in Command Abbreviation), complete commands are recorded in the journal files. Consider the following examples:

- Typing in the TUI

  ```
  mesh/smooth-mesh , 5 ,
  ```

  will be recorded in the journal file as

  ```
  /mesh/smooth-mesh "quality based" 5 0.1
  ```

  where , or **Enter** signifies default values or entries, as described in Text Prompt System.

- Typing in the TUI

  ```
  so set ur mom 0.2 pres 0.4
  ```

  will be recorded in the journal file as two separate commands:

  ```
  /solve/set/under-relaxation/mom 0.2
  /solve/set/under-relaxation/pressure 0.4
  ```

Important:

- Only successfully completed commands are recorded. For example, if you stopped an execution of a command using **Ctrl**+**c**, it will not be recorded in the journal file.
- If a GUI event happens while a text command is in progress, the GUI event is recorded first.
- All default values are recorded (as in the first example above).

For additional information, see the following section:

3.11.1. Procedure

3.11.2. Multiple Journal Files

3.11.1. Procedure

To start the journaling process, select the **File/Write/Start Journal...** ribbon tab item.

File → Write → Start Journal...

After you enter a name for the file in the **Select File** dialog box, journal recording begins. The **Start Journal...** menu item becomes the **Stop Journal** menu item. You can end journal recording by selecting **Stop Journal**, or by exiting the program.

File → Write → Stop Journal

You can read a journal file into the program using the **Select File** dialog box opened by selecting the **File/Read/Journal...** ribbon tab item.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

File → Read → Journal...

Journal files are always loaded in the main (that is, top-level) text menu, regardless of where you are in the text menu hierarchy when you invoke the read command.

3.11.2. Multiple Journal Files

It is possible to read multiple journal files into Fluent at once.

To read multiple journal files through the **Select File** dialog box:

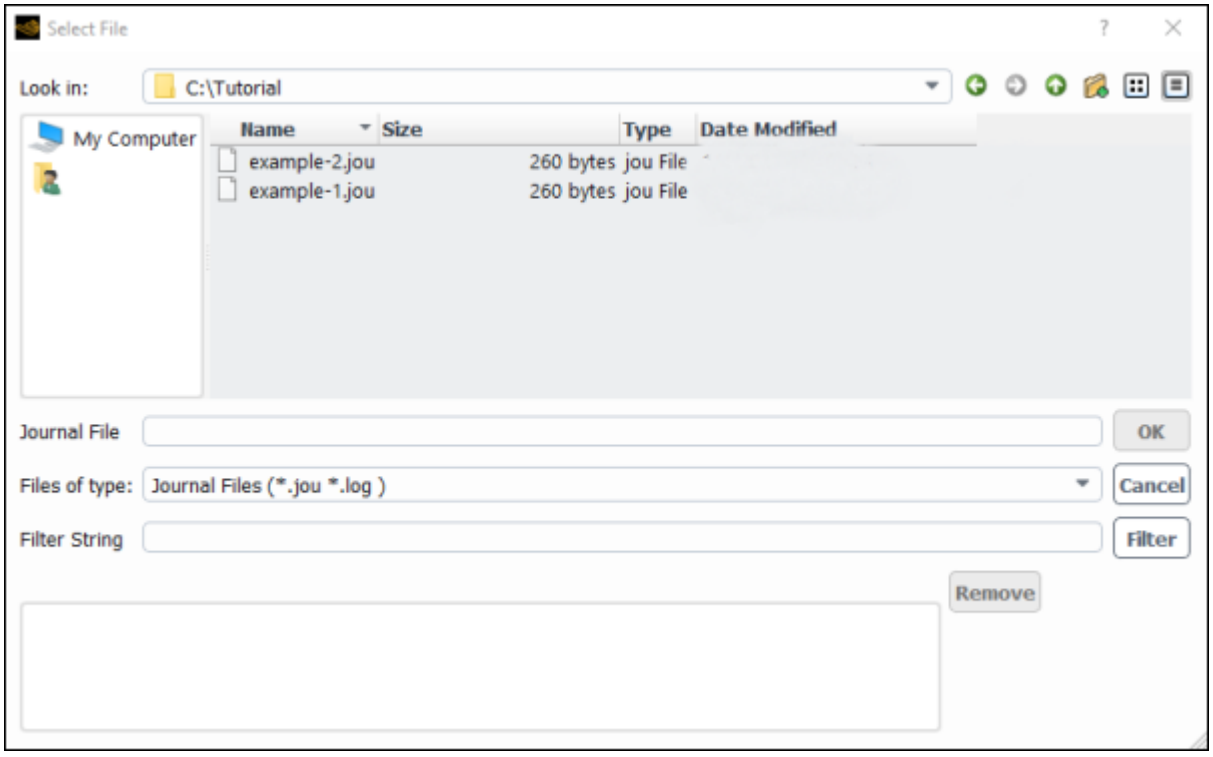
1. Open the **Select File** dialog box.

   File → Read → Journal...

2. Select your files in the order you want them read into Fluent.

   The order in which they will be read is indicated by the red box in Figure 3.8: Multiple Selection of Journal Files (p. 975).

   Figure 3.8: Multiple Selection of Journal Files



3. Click **OK** to load the selected files.

Nested Journal Files

It is possible to create a journal file that makes calls to other journal files.

The following is an example of a nested journal file:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
/file/read-journal "E:/Example_journals_example1.jou" ""
/file/read-journal "E:/Example_journals_example2.jou" ""
/file/read-journal "E:/Example_journals_example3.jou" ""
```

3.12. Creating Transcript Files

A transcript file contains a complete record of all standard input to and output from Ansys Fluent (usually all keyboard and GUI input and all screen output) including the start, end, and total time of the session, as well as the wall-clock time taken per iteration. GUI commands are recorded as Scheme code lines in transcript files. Ansys Fluent creates a transcript file by recording everything typed as input or entered through the GUI, and everything printed as output in the text window.

The purpose of a transcript file is to produce a record of the program session for later reference. Because they contain messages and other output, transcript files (unlike journal files), cannot be read back into the program.

You can set the **Automatic Transcript** option in the **General** branch of Preferences to automatically record a transcript file starting at the beginning of each new Ansys Fluent session. Automatically created transcripts are named as follows: `fluent-YYYYMMDD-HHMMSS-<processID>.trn`, where `YMD` stand for "year" "month" "day" and `HMS` stand for "hour" "minute" "second".

File → Preferences... → General → Automatic Transcript

The transcript is saved to the working directory if it is writable, otherwise it is saved in the temporary directory. *The absolute path of the temporary directory varies for Windows and Linux, but in all cases, the path where the transcript file is saved is printed in the Console.*

Important:

Only one transcript file can be open for recording at a time (except if you have **Automatic Transcript** enabled, then you can still create an additional transcript file), but you can write a transcript and a journal file simultaneously. You can also read a journal file while a transcript recording is in progress.

To start the transcription process, select the **File/Write/Start Transcript...** ribbon tab item.

File → Write → Start Transcript...

After you enter a name for the file in the **Select File** dialog box, transcript recording begins and the **Start Transcript...** menu item becomes the **Stop Transcript** menu item.

You can end transcript recording by selecting **Stop Transcript**, or by exiting the program.

File → Write → Stop Transcript

3.13. Importing Files

Ansys Fluent allows you to import the following file formats:

- ABAQUS `.inp`, `.fil`, and `.odb` files*

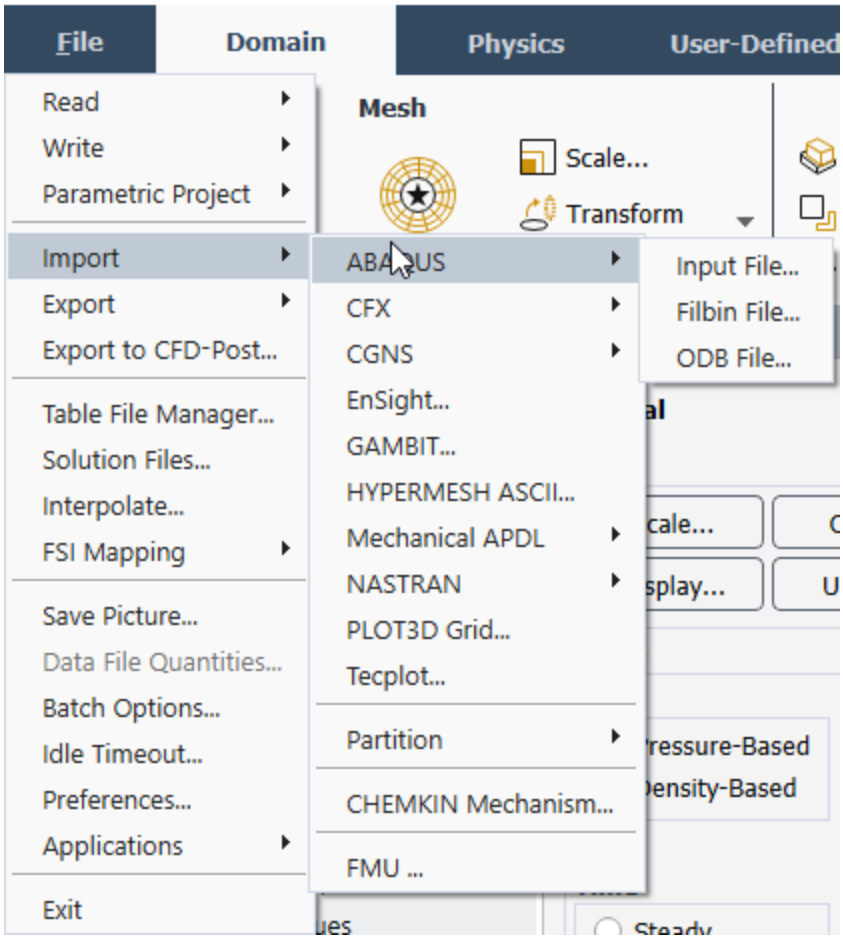
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- CFX `.def` and `.res` files
- CGNS files*
- EnSight files*
- GAMBIT files
- HYPERMESH ASCII files*
- Mechanical APDL `.inp`, `.cdb`, `.rst`, and `.rmg` files*
- NASTRAN Bulk Data files*
- PLOT3D mesh files*
- Tecplot files*

*Requires a VKI license

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 3.9: The Import Menu



Important:

Once you have imported any 3rd party file it is recommended that you confirm that the boundary zones and cell zones are set up as you expect.

For information on importing particle history data, see Importing Particle Data (p. 2844).

For additional information, see the following sections:

3.13.1. ABAQUS Files
3.13.2. CFX Files
3.13.3. Meshes and Data in CGNS Format
3.13.4. EnSight Files
3.13.5. GAMBIT and GeoMesh Mesh Files
3.13.6. HYPERMESH ASCII Files
3.13.7. NASTRAN Files
3.13.8. PLOT3D Files
3.13.9. Tecplot Files
3.13.10. Partition Files

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.13.11. CHEMKIN Mechanism

3.13.1. ABAQUS Files

To import an ABAQUS input file, use the **File/Import/ABAQUS/Input File...** ribbon tab item.

File → Import → ABAQUS → Input File...

Select this menu item to open the **Select File** dialog box. Specify the name of the ABAQUS **Input File** to be read. The ABAQUS input file (.inp) is a text file that contains the input description of a finite element model for the ABAQUS finite element program. The interface only produces datasets associated with the finite element model, no results of datasets are produced. Element types commonly associated with structural analysis are supported by this file format. There is a list of input keywords that are recognized in the ABAQUS **Input File** [4] (p. 5713).

To import an ABAQUS filbin file, use the **File/Import/ABAQUS/Filbin File...** ribbon tab item.

File → Import → ABAQUS → Filbin File...

Select this menu item to open the **Select File** dialog box. Specify the name of the ABAQUS **Filbin File** to be read. This output file has a `.fil` extension and consists of finite element model and results data.

To import an ABAQUS ODB file, use the **File/Import/ABAQUS/ODB File...** ribbon tab item.

File → Import → ABAQUS → ODB File...

Select this menu item to open the **Select File** dialog box. Specify the name of the ABAQUS **ODB File** to be read. This output database file has a `.odb` extension and consists of finite element model and results data in the OpenDocument format.

3.13.2. CFX Files

To import a CFX definition file, use the **File/Import/CFX/Definition File...** ribbon tab item.

File → Import → CFX → Definition File...

Select this menu item to open the **Select File** dialog box. Specify the name of the CFX **Definition File** to be read. Fluent reads mesh information from the CFX file with `.def` extensions. For information about importing CFX files, see CFX Files (p. 1153).

To import a CFX result file, use the **File/Import/CFX/Result File...** ribbon tab item.

File → Import → CFX → Result File...

In the **Select File** dialog box, specify the name of the CFX **Result File** to be read. Those imported files will have `.res` extensions.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that the **Create Zones from CCL Physics Data** option in the **Select File** dialog boxes allows you to create zones from the physics data objects or the primitive mesh region objects.

Important:

CFX file import is available for 3D cases only.

3.13.3. Meshes and Data in CGNS Format

To import meshes in CFD general notation system (CGNS) format (`.cgns`) into Ansys Fluent, use the **File/Import/CGNS/Mesh...** ribbon tab item.

File → Import → CGNS → Mesh...

To import a mesh and the corresponding CGNS data, use the **File/Import/CGNS/Mesh & Data...** ribbon tab item.

File → Import → CGNS → Mesh & Data...

To import only the CGNS data, use the **File/Import/CGNS/Data...** ribbon tab item.

File → Import → CGNS → Data...

Note:

While Fluent can import both structured and unstructured CGNS meshes, it cannot import data on CGNS Structured Zones.

Table 3.1: CGNS Variables Supported by Ansys Fluent

CGNS Variable Name	Ansys Fluent Name
Pressure	pressure
ReynoldsStressXX	uu-stress
ReynoldsStressXY	uv-stress
ReynoldsStressXZ	uw-stress
ReynoldsStressYY	vv-stress
ReynoldsStressYZ	vw-stress
ReynoldsStressZZ	ww-stress
Temperature	temperature
TurbulentDissipation	epsilon
TurbulentDissipationRate	omega
TurbulentEnergyKinetic	k
VelocityX	x-velocity
VelocityY	y-velocity

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CGNS Variable Name	Ansys Fluent Name
VelocityZ	z-velocity

Important:

- To import data correctly, first import the mesh using the mesh only option (**Mesh...**) and set up the boundary conditions. For example, if a boundary zone is of type **pressure-outlet** and is read as **outlet**, it should be changed to **pressure-outlet** before importing the data. Then select the appropriate model and read the data using the data only option (**Data...**)
- The new and original meshes should have the same zones, numbered in the same order. A warning is issued if they do not, because inconsistencies can create problems with the boundary conditions.
- Ansys Fluent defaults to the **Laminar Viscous Model** after reading a CGNS mesh. You must select the appropriate model before reading data since Ansys Fluent selectively imports only those values from the CGNS file required for a given model. For example, if **Epsilon** is available in the CGNS file, the model must be changed to **k-epsilon** to ensure that Ansys Fluent will import this quantity.
- (Linux only) Ensure that the `LC_NUMERIC` and `LC_ALL` are either not set or set to `en_US` prior to launching Ansys Fluent.

3.13.4. EnSight Files

You can import an EnSight file using the **File/Import/EnSight...** ribbon tab item.

File → Import → EnSight...

This file format is applied to both unstructured and structured data, where each part contains its own local coordinate array. The EnSight Gold software package, which uses this file format, allows you to analyze, visualize, and communicate engineering datasets. It allows you to take full advantage of parallel processing and rendering and supports a range of virtual reality devices. Furthermore, it enables real-time collaboration.

When selecting this option, the **Select File** dialog box will appear, where you will specify a file name. This file will have an `.encas` or `.case` extension.

Only the mesh file is read into Ansys Fluent, and any data present is discarded.

3.13.5. GAMBIT and GeoMesh Mesh Files

If you have saved a neutral file from GAMBIT, rather than an Ansys Fluent mesh file, you can import it into Ansys Fluent using the **File/Import/GAMBIT...** ribbon tab item.

File → Import → GAMBIT...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For information about importing files from GAMBIT and GeoMesh, see GAMBIT Mesh Files (p. 1151) and GeoMesh Mesh Files (p. 1152).

3.13.6. HYPERMESH ASCII Files

You can read a HYPERMESH ASCII file using the **File/Import/HYPERMESH ASCII...** ribbon tab item.

File → Import → HYPERMESH ASCII...

HYPERMESH is a high-performance finite element pre- and postprocessor for popular finite element solvers, allowing engineers to analyze product design performance in a highly interactive and visual environment.

When selecting this option, the **Select File** dialog box will appear, where you will specify a file name. This file should have an `.hm`, `.hma`, or `.hmascii` extension.

3.13.7. NASTRAN Files

You can read NASTRAN Bulkdata files into Ansys Fluent with the **File/Import/NASTRAN/Bulkdata File...** ribbon tab item.

File → Import → NASTRAN → Bulkdata File...

When you select the **Bulkdata File...** menu item, the **Select File** dialog box will appear and you will specify the name of the NASTRAN **File** to be read. This file will have `.nas*`, `.dat*`, `.bdf*` file extensions. Fluent reads mesh information from the NASTRAN file. For information about importing NASTRAN files, see NASTRAN Files (p. 1152).

To import NASTRAN Op2 files into Ansys Fluent, use the **File/Import/NASTRAN/Op2 File...** ribbon tab item.

File → Import → NASTRAN → Op2 File...

In the **Select File** dialog box, specify the name of the NASTRAN **Output2 File** to be read. This file is an output binary data file that contains data used in the NASTRAN finite element program. This file will have `.op2` file extension.

3.13.8. PLOT3D Files

To import a PLOT3D mesh file, use the **File/Import/PLOT3D Grid...** ribbon tab item.

File → Import → PLOT3D Grid...

The PLOT3D mesh files have `.p3d`, `.bin`, `.x`, `.xyz`, or `.grd` file extensions.

These file formats may be formatted, unformatted or binary.

3.13.9. Tecplot Files

To import a Tecplot file, use the **File/Import/Tecplot...** ribbon tab item.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

File → Import → Tecplot...

This will open the **Select File** dialog box. Specify the name of the neutral file to be read.

The Tecplot file is a binary file. Only the mesh is read into Ansys Fluent and any data present is discarded.

The form of the file must have the `.plt` extension. Ansys Fluent supports the importation of polyhedral cells and files created by Tecplot version 7.1–11.2, except for version 11.0 (which is unsupported).

3.13.10. Partition Files

To perform METIS partitioning on an unpartitioned mesh, use the **File/Import/Partition/Metis...** ribbon tab item.

File → Import → Partition → Metis...

You may also partition each cell zone individually, using the **File/Import/Partition/Metis Zone...** ribbon tab item.

File → Import → Partition → Metis Zone...

See Using the Partition Filter (p. 4338) for detailed information about partitioning.

3.13.11. CHEMKIN Mechanism

To import a CHEMKIN format, you can import the mechanism file into Ansys Fluent using the **File/Import/CHEMKIN Mechanism...** ribbon tab item (Figure 19.5: The Import CHEMKIN Format Mechanism Dialog Box for Volumetric Kinetics (p. 2370)).

File → Import → CHEMKIN Mechanism...

See Importing a Volumetric Kinetic Mechanism in CHEMKIN Format (p. 2368) for detailed information on importing a CHEMKIN Mechanism file.

3.14. Exporting Solution Data

The current release of Ansys Fluent allows you to export data to ABAQUS, Mechanical APDL Input, ASCII, **AVS**, CDAT for CFD-Post and EnSight, CGNS, Common Fluids Format - Post, EnSight Case Gold, EnSight DVS, **Data Explorer**, FAST, FIELDVIEW, **I-deas**, NASTRAN, **PATRAN**, TAITherm, and Tecplot formats. Exporting Solution Data after a Calculation (p. 985) explains how to export solution data in these formats after the calculation is complete, and ABAQUS Files (p. 986) to Tecplot Files (p. 999) provide specific information for each type of **File Type**. For information about exporting solution data during transient flow solutions, see Exporting Data During a Transient Calculation (p. 1001).

Fluent exports to third party software formats in the versions listed in Table 1.2: Supported Versions of Third-Party Software (p. 170).

For NASTRAN, ABAQUS, Mechanical APDL Input, I-deas Universal, and PATRAN file formats, the following quantities are exported [4] (p. 5713):

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Nodes, Elements
- Node Sets (Boundary Conditions)
- Temperature
- Pressure
- Heat Flux
- Heat Transfer Coefficient
- Force

  To generate the force data that is exported for nodes at boundaries, Ansys Fluent performs the following steps:

  1. Facial force for each wall face is calculated by summing the pressure force, viscous force and surface tension force of the face.
  2. Partial force for each wall face is calculated by dividing its facial force by its number of shared nodes.
  3. Total force for each wall node is calculated by summing the partial forces of all the wall faces sharing that node.

For additional information, see the following section:

3.14.1. Exporting Limitations

3.14.1. Exporting Limitations

Note the following limitations when exporting solution data:

- When using the parallel version of Ansys Fluent, you can only export to the following packages:
  - ABAQUS
  - CDAT for CFD-Post and EnSight
  - ASCII
  - CGNS
  - Common Fluids Format - Post
  - EnSight Case Gold
  - EnSight DVS
  - Fieldview Unstructured
  - Mechanical APDL Input
  - NASTRAN

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - TAITherm

    Note that parallel TAITherm export is only available for surface data.

  - Tecplot

  The exported file will be written by the host process (see Introduction to Parallel Processing (p. 4293)). Note that the memory required to write the file may exceed the memory available to the host process.

- When using the serial version of Ansys Fluent, you can only export surface data for TAITherm software.
- The solution mode of Fluent cannot import surfaces. Consequently, if you export a file from Ansys Fluent with surfaces selected, you may not be able to read these files back into the solution mode. However, the meshing mode of Fluent can import surface data (see the Fluent Meshing section of the User's Guide for details).
- The following file types require a VKI license: ABAQUS, CGNS, **Ideas-Universal**, NASTRAN, **PATRAN**, Tecplot.
- Ansys Fluent supports exporting polyhedral data only for ASCII, CDAT for CFD-Post and EnSight, CGNS, Common Fluids Format - Post, EnSight Case Gold, EnSight DVS, Fieldview Unstructured, NASTRAN, TAITherm, and Tecplot file formats. For further details, see ASCII Files (p. 988), CDAT for CFD-Post and EnSight (p. 988), CGNS Files (p. 989), Common Fluids Format - Post Files (p. 991), EnSight Case Gold Files (p. 992), FieldView Unstructured Files (p. 996), NASTRAN Files (p. 998), TAITherm Files (p. 998), and Tecplot Files (p. 999).
- If the files that are exported during multiple transient simulations are to be used as a set, you must make sure that all of the simulations are run on the same platform, using the same number of processors. This ensures that all of the files are compatible with each other.
- Data on user defined/created surfaces is not available for export to EnSight Case Gold file format.
- (Windows only) You cannot export files larger than 2GB to CGNS format. As a workaround, use a Linux machine to export files larger than 2GB to CGNS format.

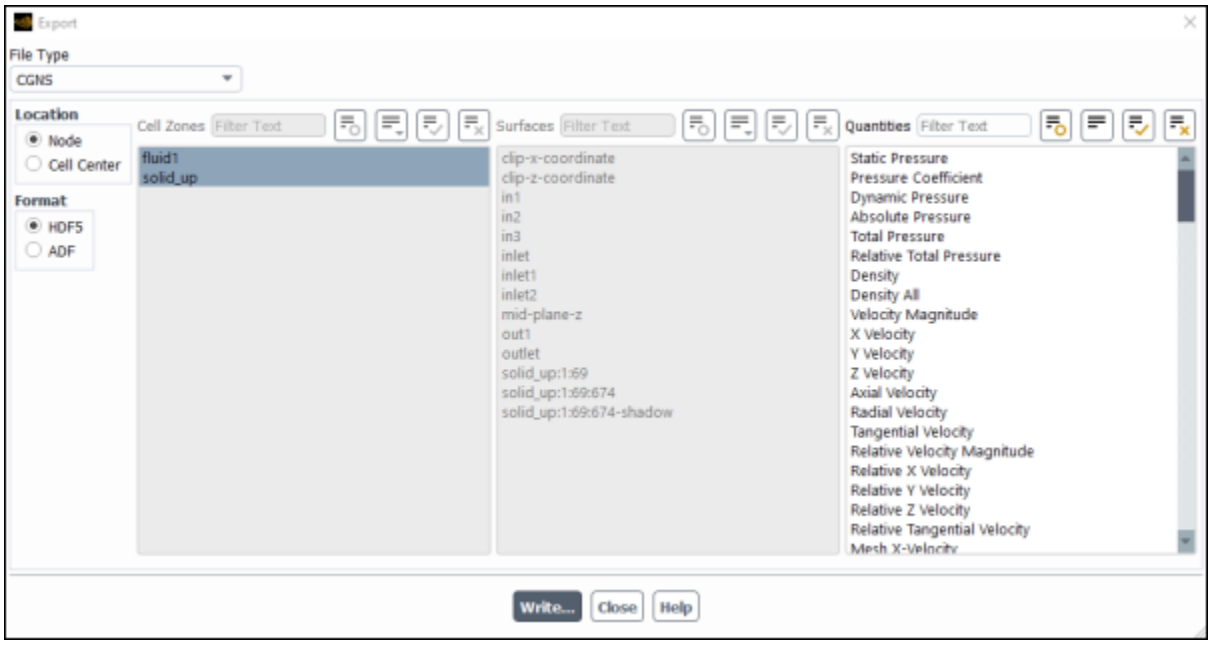
3.15. Exporting Solution Data after a Calculation

To export solution data to a different file format after a calculation is complete, use the Export Dialog Box (p. 5361) (Figure 3.10: The Export Dialog Box (p. 986)).

File → Export → Solution Data...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 3.10: The Export Dialog Box



Information concerning the necessary steps and available options for each **File Type** are listed in ABAQUS Files (p. 986) to Tecplot Files (p. 999).

For details about general limitations for exporting solution data and the manner in which it is exported, see Exporting Solution Data (p. 983).

For additional information, see the following sections:

3.15.1. ABAQUS Files
3.15.2. Mechanical APDL Input Files
3.15.3. ASCII Files
3.15.4. CDAT for CFD-Post and EnSight
3.15.5. CGNS Files
3.15.6. Common Fluids Format - Post Files
3.15.7. EnSight Case Gold Files
3.15.8. EnSight DVS
3.15.9. FAST Files
3.15.10. FAST Solution Files
3.15.11. FieldView Unstructured Files
3.15.12. NASTRAN Files
3.15.13. TAITherm Files
3.15.14. Tecplot Files

3.15.1. ABAQUS Files

Select ABAQUS from the **File Type** drop-down list and choose the surface(s) for which you want to write data in the **Surfaces** list. If no surfaces are selected, the entire domain is exported.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When the **Energy Equation** is enabled in the Energy Dialog Box (p. 4714), you can choose the loads to be written based on the kind of finite element analysis you intend to undertake. By selecting **Structural** in the **Analysis** list, you can select the following **Structural Loads**: **Force**, **Pressure**, and **Temperature**. By selecting **Thermal** in the **Analysis** list, you can select the following **Thermal Loads**: **Temperature**, **Heat Flux**, and **Heat Trans Coeff**. Note the following limitations with these loads:

- When the **Energy Equation** is disabled, only the **Structural Loads** options of **Force** and **Pressure** are available.
- Loads are written only on boundary walls when the entire domain is exported (that is, if no **Surfaces** are selected).

Click the **Write...** button to save the file, using the **Select File** dialog box. The exported file format of ABAQUS (`file.inp`) contains coordinates, connectivity, zone groups, and optional loads.

Export of data to ABAQUS is valid only for solid zones or for those surfaces that lie at the intersection of solid zones. Temperature data is exported for the whole domain.

You have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. See Exporting Data During a Transient Calculation (p. 1001) for the complete details.

3.15.2. Mechanical APDL Input Files

Select Mechanical APDL Input from the **File Type** drop-down list and choose the surface(s) for which you want to write data in the **Surfaces** list. If no surfaces are selected, the entire domain is exported.

When the **Energy Equation** is enabled in the Energy Dialog Box (p. 4714), you can choose the loads to be written based on the kind of finite element analysis you intend to undertake. By selecting **Structural** in the **Analysis** list, you can select the following **Structural Loads**: **Force**, **Pressure**, and **Temperature**. By selecting **Thermal** in the **Analysis** list, you can select the following **Thermal Loads**: **Temperature**, **Heat Flux**, and **Heat Trans Coeff**. Note the following limitations with these loads:

- When the **Energy Equation** is disabled, only the **Structural Loads** options of **Force** and **Pressure** are available.
- Loads are written only on boundary walls when the entire domain is exported (that is, if no **Surfaces** are selected).

Click the **Write...** button to save the file, using the **Select File** dialog box. Ansys Fluent exports an input file that contains Mechanical APDL finite element information including nodes, elements, and loads that can be used to do finite element analysis in Mechanical APDL with minimal effort. The file format is written in `.cdb` format. The export of Mechanical APDL Input files is in ASCII format and therefore is available on all platforms.

You have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. See Exporting Data During a Transient Calculation (p. 1001) for the complete details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.15.3. ASCII Files

Select **ASCII** from the **File Type** drop-down list and choose the surface(s) for which you want to write data in the **Surfaces** list. If no surfaces are selected, the entire domain is exported. Also select the variable(s) for which data is to be saved in the **Quantities** list.

When exporting ASCII files, you have the following options:

- Select the **Location** from which the values of scalar functions are to be taken. If you specify the data **Location** as **Node**, then the data values at the node points are exported. If you choose **Cell Center**, then the data values from the cell centers are exported. For boundary faces, it is the face center values that are exported when the **Cell Center** option is selected.
- Select the **Delimiter** separating the fields (**Comma** or **Space**).

Click the **Write...** button to save the file, using the **Select File** dialog box. Ansys Fluent will export a single ASCII file containing coordinates, optional loads, and specified scalar function data.

Important:

Ansys Fluent supports exporting polyhedral data to ASCII.

You have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. See Exporting Data During a Transient Calculation (p. 1001) for the complete details.

3.15.4. CDAT for CFD-Post and EnSight

Output from Ansys Fluent simulations can be read into CFD-Post and EnSight. The default output file for Ansys Fluent is a `.dat.h5` file (see Reading and Writing Data Files (p. 957)), which contains raw data directly from the solver; when viewed in CFD-Post and EnSight, such files may show results that are not entirely consistent with what you would see when postprocessing within Fluent (this is even more true for `.dat` files, which also have the limitation that they cannot be read in EnSight). You can instead use `.cdat` files so that the raw data is processed prior to export in order to reduce such inconsistencies.

Note:

- **CDAT for CFD-Post & EnSight** is not available for cases using the **Structural** model. For these cases, export to **Common Fluids Format - Post**.
- One known limitation of `.cdat` files is that they only contain nodal values, even for face zones.
- EnSight Case Gold files (EnSight Case Gold Files (p. 992)) are recommended for the best performance in EnSight, even over `.cdat` files.

To export `.cdat` files, select **CDAT for CFD-Post & EnSight** from the **File Type** drop-down list. Next, specify the **Cell Zones** that contain the data you want exported; note that by default, all of the cell zones will be exported. If you want to export solution data for particular surfaces, then select them

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

from the **Surfaces** list. Once you have chosen your cells zones and/or surfaces you must select the **Quantities** that you want to export. When selecting the variables, be sure to include any variable that was used to create an isosurface or clipped surface (as described in Iso-surfaces (p. 3838) and Clipping Surfaces (p. 3840), respectively); otherwise, the surface will not be created properly if you try to read the exported state file in CFD-Post.

Specify the format of the `.cdat` file by selecting either **Binary** or **ASCII** from the **Format** list. The advantage of the binary format is that it takes less time to load the exported data into Ansys CFD-Post and requires less storage space. Note that the format for the `.cst` will always be ASCII.

Fluent will write a legacy case file (that is, `.cas`) along with the **CDAT for CFD-Post & EnSight** file. This ensures that the data can be read into Ansys CFD-Post.

Click the **Write...** button to save the files, using the **Select File** dialog box. A `.cdat` file is written, containing the specified variable data for the specified cell zones and all of the boundary zones. A state file (that is, `.cst`) is also written, which contains the following surfaces that you created in Fluent for postprocessing: point surfaces, line surfaces, plane surfaces, isosurfaces, and clipped surfaces (see Creating Surfaces and Cell Registers for Displaying and Reporting Data (p. 3813)).

Note:

Imprinted surfaces defined in Ansys Fluent are not available for postprocessing in Ansys CFD-Post.

Important:

When you read the `.cdat` file into Ansys CFD-Post, the application will attempt to read in the case file that produced the data by looking in the folder for a `.cas` file with the same prefix. If such a case file does not exist in that folder, Ansys CFD-Post will prompt you to specify the appropriate case file.

Important:

Before loading the `.cst` file into Ansys CFD-Post, make sure that the `.cas` and `.cdat` files are already loaded for the session.

You have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. See Exporting Data During a Transient Calculation (p. 1001) for the complete details.

3.15.5. CGNS Files

Select **CGNS** from the **File Type** drop-down list, select the desired cell zone(s) or surface(s), and specify the scalars you want in the **Quantities** list.

Specify the **Format** of the exported file:

- **HDF5**—Hierarchical data format, which makes use of built-in compression. It cannot be viewed in a text editor. However, third-party tools are available that allow you to open and explore the contents of files saved in HDF format.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **ADF**—Legacy format for CGNS file export.

Specify the type of mesh that is exported:

- Mixed: The mesh is exported as CGNS mixed type elements (even if the original mesh is uniform).

  This is set by entering `file/export/settings/cgns-mesh-type 1` in the Console.

- Native: The mesh is exported with the same node ordering as the existing mesh.

  This is set by entering `file/export/settings/cgns-mesh-type 2` in the Console.

  Important:

  Files exported in the native mesh format are usually smaller than those exported in mixed or poly format.

- Polyhedral : The mesh is exported with the same node ordering as the existing mesh.

  This is set by entering `file/export/settings/cgns-mesh-type 3` in the Console.

Refer to https://cgns.github.io/CGNS_docs_current/sids/gridflow.html#element_example for additional information on CGNS mesh types.

Select the **Location** from which the values of scalar functions are to be taken. If you specify the data **Location** as **Node**, then the data values at the node points are exported. If you choose **Cell Center**, then the data values from the cell centers are exported. For boundary faces, it is the face center values that are exported when the **Cell Center** option is selected.

Note:

When you select a zone with more than 30 characters in its name, the name is cutoff and the zone ID is appended to the end of the shortened name. *This character limit is a CGNS standard.*

Click the **Write...** button to save the file, using the **Select File** dialog box. CGNS (CFD general notation system) is a single file (for example, `file.cgns`) containing coordinates, connectivity, zone information, velocity, and selected scalars.

Important:

- Ansys Fluent supports exporting polyhedral data to CGNS.
- (Windows only) You cannot export files larger than 2GB to CGNS in ADF format. There is no size limitation if you export to CGNS in HDF5 format. As a workaround if you required CGNS files in ADF format, use a Linux machine to export files larger than 2GB.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. See Exporting Data During a Transient Calculation (p. 1001) for the complete details.

3.15.6. Common Fluids Format - Post Files

Output from Ansys Fluent simulations can be read into EnSight. The default output file saved from Ansys Fluent is a `.dat.h5` file (see Reading and Writing Data Files (p. 957)), which contains raw data directly from the solver; when viewed in other applications (such as EnSight and CFD-Post) the results may not be entirely consistent with what you would see when postprocessing within Fluent. This is also true for legacy `.dat` files.

Within Fluent, it is possible to export Common Fluids Format - Post files to reduce such inconsistencies. The data exported to these files is made more consistent by processing the raw data within Fluent before writing the file.

The data written to the Common Fluids Format – Post file, does not contain the full mesh, topology, or physics information that is present within the default Fluent files. Instead, the export can be controlled to restrict the mesh and data to specific surfaces and quantities. This can be useful if you wish to monitor how quantities change across time on just a small subset of the problem domain because the reduced datasets can lead to improved read times.

Note:

- For the best performance in EnSight export to EnSight Case Gold (EnSight Case Gold Files (p. 992)).
- For CFD-Post, export CDAT files if you want to postprocess a full representation of the simulation mesh, topology, and physics used within Fluent with some corrected quantities.
- Common Fluids Format - Post files only contain nodal values, even for face zones, with the exception of cell-related field variables such as Active Cell Partition, Cell Volume Change, and so on, which are indicated with the *cve* code in Field Variables Listed by Category (p. 4171).
- When running on mixed Windows / Linux, you must specify a remote working directory at launch in order to be able to export Common Fluids Format - Post files, by using one of the following: the **Remote Working Directory** option and field in the **Remote** tab of Fluent Launcher; the `-nodehomedir=<directory>` command line option; or the `NODEHOMEDIR=<directory>` environment variable.

Select **Common Fluids Format - Post** from the **File Type** drop-down list and select the cell zone(s) and surface(s) where you want data exported in the **Cell Zones** and **Surfaces** lists, as well as the scalars you want in the **Quantities** list. After you select the cell zones, surfaces, and quantities that you want exported, click **>>** to add the quantity to the list. Click **<<** to remove selected zone/surface and quantities that you do not want exported from the **Quantity List**.

An advantage of exporting in the **Common Fluids Format - Post** is that you can export different quantities on different surfaces/cell zones. For example, you can export shear-stress on a wall surface

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and pressure on an outlet surface. This provides flexibility to limit the data you are exporting to the surfaces where you are interested in postprocessing those values.

Click the **Write...** button to save the file, using the **Select File** dialog box. The following files are written in the hierarchical data format (HDF) from the **Export** dialog box: `<filename>.cas.post` and `<filename>.dat.post`.

3.15.7. EnSight Case Gold Files

When exporting to EnSight, you can choose to export the data associated with a single data file, or you can export the data associated with multiple files that were generated by a transient solution.

- To export the solution data from a single data file, read in the case and data file and open the **Export** dialog box.

  File → Export → Solution Data...

  Then perform the following steps in the **Export** dialog box:

  1. Select **EnSight Case Gold** from the **File Type** drop-down list.
  2. Select the **Location** from which the values of scalar functions are to be taken. If you specify the data **Location** as **Node**, then the data values at the node points are exported. If you choose **Cell Center**, then the data values from the cell centers are exported. For boundary faces, it is the face center values that are exported when the **Cell Center** option is selected.
  3. Specify the format of the file by selecting either **Binary** or **ASCII** from the **Format** list. The advantage of the binary format is that it takes less time to load the exported files into EnSight.
  4. (optional) Select the **Cell Zones** from which you want data exported (you must select at least one zone). By default, all of the cell zones are selected.
  5. (optional) Select the **Interior Zone Surfaces** from which you want data exported. By default, the data being exported is taken from the entire Ansys Fluent domain. The **Interior Zone Surfaces** selection list allows you to also specify that the data be taken from selected zone surfaces whose **Type** is identified as **interior** in the **Boundary Conditions** task page.

     Note:

     Data on user defined/created surfaces is not available for export to EnSight Case Gold file format.

  6. Select the scalars you want to write from the **Quantities** selection list.
  7. Click the **Write...** button to open the **Select File** dialog box, which you can use to save a file that contains the specified variable data for the specified cell zones, the specified interior zone surfaces, and all of the boundary zones.

- To export solution data associated with multiple files that were generated by a transient solution, you must first make sure that the following criteria are met:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– All of the relevant case and data files must be in the working folder.

– The data files must be separated by a consistent number of time steps.

Next, enter the following text command in the console:

`file → transient-export → ensight-gold-from-existing-files`

Then, enter responses to the following prompts in the console:

1. `EnSight Case Gold file name`

   Enter the name you want assigned to the exported files.

2. `Case / Data file selection by base name?`

   Enter `yes` if the case and data files share a "base name" (that is, a common initial string of characters). The alphanumeric order of the full names must correspond to the order in which the files were created. You will then be prompted to enter the base name at the `Case / Data file base name` prompt that follows. For example, with a set of files named `elbow-0001`, `elbow-0002`, `elbow-0003`, and so on, enter `elbow-` for the base name.

   Enter `no` if you have created an ASCII file in the working folder that lists the names of the data files in order of when they were created. The file should list one data file name per line. You will then be prompted to enter the name of this file at the `Provide the file name which contains the data file names` prompt that follows. Note that you must include the file extension if any in your entry at the prompt.

3. `Specify Skip Value`

   Enter an integer value to specify the number of files you want to skip in between exporting files from the sequence. For example, enter `1` to export every other file, enter `2` to export every third file, and so on.

4. `Cell-Centered?`

   Enter `yes` if you want to export the data values from the cell centers (or face center values, for boundary faces).

   Enter `no` if you want to export the data values from the node points.

5. `Write separate file for each time step for each variable?`

   Enter `yes` if you want separate EnSight Case Gold files written for each time step. Otherwise, all of the data for the `.scl1` and `.vel` files will be combined into a single file for each.

6. `Write in binary format?`

   Enter `yes` to write the files in binary format. Otherwise, they will be written in ASCII format. The advantage of the binary format is that it takes less time to load the exported files into EnSight.

7. `Specify Data File Frequency`

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Enter the number of time steps between the data files being exported.

8. `Separate case file for each time step?`

   Enter `no` if all the data files were generated from the same case file (that is, the simulation involved a static mesh). Note that the name of the case file must be the same (not including the extension) as the name of the first data file in the sequence.

   Enter `yes` if the data files were generated from the different case files (that is, the simulation involved a sliding or dynamic mesh). Note that the names of the case files must be the same (not including the extension) as the names of the corresponding data files.

9. `Read the case file?`

   Enter `no` if the first (or only, for a static mesh) case file is already in memory.

   Enter `yes` if the first (or only, for a static mesh) case file is *not* already in memory.

10. `cell zone id/name(1)`

    Enter the name or ID of any cell zone from which you want data exported. By default, the data being exported is taken from the entire Ansys Fluent domain. After you specify the first cell zone, you will be prompted to specify the second one, and so on, until you press **Enter** without typing any characters.

11. `Interior Zone Surfaces(1)`

    Enter the name of any interior zone surface from which you want data exported. By default, the data being exported is taken from the entire Ansys Fluent domain. This prompt allows you to also specify that the data be taken from selected zone surfaces whose **Type** is identified as **interior** in the **Boundary Conditions** task page. After you specify the first interior zone surface, you will be prompted to specify the second one, and so on, until you press **Enter** without typing any characters.

12. `EnSight Case Gold scalar(1) else q to continue`

    Enter in the first scalar quantity you want exported. You can press the **Enter** key to print a list of available scalar quantities in the console. After you enter the first quantity, you will be prompted to enter the second quantity, and so on, until you enter `q`. The EnSight Case Gold files will then be written.

When exporting to EnSight Case Gold, files will be created with the following four formats:

- A geometry file (for example, `file.geo`) containing the coordinates and connectivity information.
- A velocity file (for example, `file.vel`) containing the velocity.
- A scalar file (for example, `file.scl1`) for each selected variable or function.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- An EnSight case file (for example, `file.encas`) that contains details about the other exported files.

Important:

- For non-stationary reference zones, all the velocities are exported to EnSight as velocities relative to the selected reference zone.
- Ansys Fluent supports exporting polyhedral data to EnSight.

Note:

If you are planning to import files exported to EnSight into a non-Ansys third-party post-processing software, such as Paraview, then you must ensure that no surface has an ID of zero. If there is, then create a zone surface for that surface and export the newly created zone surface instead of the original.

You also have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. For details, see Exporting Data During a Transient Calculation (p. 1001).

Refer to Transition Assistance from Fluent to EnSight for Post Processing in the *Fluent to EnSight Considerations* for additional guidance on choosing the best format for your case.

3.15.8. EnSight DVS

You can export Fluent solution data in EnSight DVS format. Refer to Transition Assistance from Fluent to EnSight for Post Processing in the *Fluent to EnSight Considerations* for additional guidance on choosing the best format to use in EnSight for your case.

To export in EnSight DVS format:

1. Open the **Export** dialog box.

   File → **Export** → **Solution Data...**

2. Select **EnSight DVS** from the **File Type** drop-down list.
3. Select the desired **Cell Zones** or **Surfaces** (not both), and **Quantities** from their respective lists.

   Note:

   Duplicate surfaces cannot be exported to EnSight.

4. Click **Write...** to open the **Select File** dialog box, so that you can save the **EnSight DVS** files in the preferred location.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.15.9. FAST Files

This file type is valid only for triangular and tetrahedral meshes. Select **FAST** from the **File Type** drop-down list and select the scalars you want to write in the **Quantities** list.

Click the **Write...** button to save the file for the specified function(s), using the **Select File** dialog box. The following files are written:

- A mesh file in extended Plot3D format containing coordinates and connectivity.
- A velocity file containing the velocity.
- A scalar file for each selected variable or function.

You have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. See Exporting Data During a Transient Calculation (p. 1001) for the complete details.

3.15.10. FAST Solution Files

This file type is valid only for triangular and tetrahedral meshes. Select **FAST Solution** from the **File Type** drop-down list and click the **Write...** button. A single file is written containing density, velocity, and total energy data.

You have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. See Exporting Data During a Transient Calculation (p. 1001) for the complete details.

3.15.11. FieldView Unstructured Files

Select **Fieldview Unstructured** from the **File Type** drop-down list. Next, specify the cell zones and surfaces from which you want data exported by making selections in the **Cell Zones** or **Surfaces** lists (you must select at least one zone or surface); note that by default, all of the cell zones will be exported. Then select the scalars you want to write in the **Quantities** list. Finally, click the **Write...** button to open the **Select File** dialog box, which you can use to save a file that contains the specified function(s) for the specified cell zones and associated boundary zones or surfaces.

The following files are written:

- A binary file (for example, `file.fvuns`) containing coordinate and connectivity information and specified scalar function data. In parallel, each partition writes to this file with a grid for each cell zone or surface that is exported. This allows parallel postprocessing in FieldView.
- A regions file (for example, `file.fvuns.fvreg`) containing information about the cell zones or surfaces and the frame of reference. It also includes information about which grids make up the various cell zones or surfaces.

The cell zone and surface information includes the names of the cell zones/surfaces along with the mesh numbers. For the moving frame of reference, the regions file contains information about the origin, the axis of rotation and the rotation speed. Volume data is written using the absolute frame of reference.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you are running multiple steady-state solutions on the same mesh, you can export only the data files and avoid the repeated writing of the mesh file by using the following TUI command for surfaces and cell zones:

`file/export/fieldview-unstruct-data` *file_name* `(` *list_of_surfaces* `) (` *list_of_zones* `)` *list_of_scalars* `q`

where

- *file_name* specifies the name (without the extension) of the file that you want to write.
- *list_of_surfaces* specifies the list of surfaces (separated by spaces, that is, no commas) from which you want data exported. If you want to specify all of the surfaces, enter `*` within the parentheses.
- *list_of_zones* specifies the list of cell zones (separated by spaces, that is, no commas) from which you want data exported. If you want to specify all of the cell zones, enter `*` within the parentheses.
- *list_of_scalars* specifies the list of cell functions (separated by spaces, that is, no commas) that you want to write to the exported file. The `q` input terminates the list. For example, the input `x-velocity cell-zone q` will select x velocity and the cell volume.

To export only the data files for surfaces, use the following TUI command:

`file/export/fieldview-unstruct-surfaces 2` *file_name* `(` *list_of_surfaces* `)` *list_of_scalars* `q`

where

- `2` specifies that you want to write a Fieldview unstructured results only file.
- *file_name* specifies the name (without the extension) of the file that you want to write.
- *list_of_surfaces* specifies the list of surfaces (separated by spaces, that is, no commas) from which you want data exported. If you want to specify all of the surfaces, enter `*` within the parentheses.
- *list_of_scalars* specifies the list of cell functions (separated by spaces, that is, no commas) that you want to write to the exported file. The `q` input terminates the list. For example, the input `x-velocity q` will select only the x velocity.

You have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. See Exporting Data During a Transient Calculation (p. 1001) for the complete details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.15.12. NASTRAN Files

Select **NASTRAN** from the **File Type** drop-down list. Select the surface(s) containing the data you want to export in the **Surfaces** list. If you do not select any surfaces, the entire domain is exported. You can specify which scalars you want in the **Quantities** list.

Note:

If you are exporting data on a polyhedral mesh, only zone surfaces will be available for selection (no volumetric data or user-defined surfaces).

You have the option of selecting loads to be included in the exported file. When the **Energy Equation** is enabled in the Energy Dialog Box (p. 4714), you can choose the loads based on the kind of finite element analysis you intend to undertake. By selecting **Structural** in the **Analysis** list, you can select the following **Structural Loads**: **Force**, **Pressure**, and **Temperature**. By selecting **Thermal** in the **Analysis** list, you can select the following **Thermal Loads**: **Temperature**, **Heat Flux**, and **Heat Trans Coeff**.

These loads have the following limitations :

- When the **Energy Equation** is disabled, only the **Structural Loads** options of **Force** and **Pressure** are available.
- Loads are written only on boundary walls when the entire domain is exported (that is, if no **Surfaces** are selected).

Click the **Write...** button to save the file, using the **Select File** dialog box. A single file (for example, `file.bdf`) is written containing coordinates, connectivity, optional loads, zone groups, and velocity. Pressure is written as PLOAD4, and heat flux is written as QHBDYE data. If you select wall zones in the **Surfaces** list, nodal forces are written for the walls. When data is written for the heat transfer coefficient, it is based on the wall faces rather than the nodes.

You have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. For details, see Exporting Data During a Transient Calculation (p. 1001).

3.15.13. TAITherm Files

The option to export a TAITherm file type is available only when the **Energy Equation** is enabled in the Energy Dialog Box (p. 4714). Select **TAITherm** from the **File Type** drop-down list and select the surface(s) for which you want to write data in the **Surfaces** list. If no surfaces are selected, the entire domain is exported. If the mesh contains polyhedral cells, Ansys Fluent will export only boundary data. No volume data will be exported.

Select the method of writing the heat transfer coefficient (**Heat Transfer Coefficient**), which can be **Flux Based** or, if a turbulence model is enabled, **Wall Function** based. The flux-based heat transfer coefficient is calculated using Equation 7.123 (p. 1481). If no turbulence model is enabled, the wall-function based heat transfer coefficient is calculated as k/h, where k is the thermal conductivity, and h is the cell height of the wall-adjacent cell.

Click the **Write...** button to save the file, using the **Select File** dialog box. A PATRAN neutral file (for example, `file.neu`) is written containing element velocity components (that is, the element that

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

is just touching the wall), heat transfer coefficients, and temperatures of the wall for any selected wall surface. If the wall is one-sided, the data is written for one side of the wall. If the wall is two-sided (a wall-wall shadow pair), the values are written only for the original wall face, not for the shadow face (which is a duplicate).

You have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. See Exporting Data During a Transient Calculation (p. 1001) for the complete details.

Note:

If no turbulence model is enabled, the **Wall Function** based Heat Transfer Coefficient is equal to k/h, where k is the thermal conductivity, and h is the cell height of the wall-adjacent cell.

3.15.14. Tecplot Files

Select **Tecplot** from the **File Type** drop-down list and choose the surface(s) where you want data written in the **Surfaces** list. *If no surfaces are selected, the data is exported at cell centers for the entire domain. If surfaces are selected, the data is exported on their face centers*. Select the variable(s) you want exported in the **Quantities** list.

Click the **Write...** button to save the file, using the **Select File** dialog box. A single file is written containing the coordinates and scalar functions in the appropriate tabular format.

Important:

- The **utility fe2ram** can import Tecplot files only in **FEPOINT** format.
- If you intend to postprocess Ansys Fluent data with Tecplot, you can either export data from Ansys Fluent and import it into Tecplot, or use the Tecplot Ansys Fluent Data Loader included with your Tecplot distribution. The data loader reads native Ansys Fluent case and data files directly. If you are interested in this option, contact Tecplot, Inc. for assistance or visit www.tecplot.com.

You have the option of exporting data at specified intervals during a transient calculation through the **Automatic Export** dialog box. See Exporting Data During a Transient Calculation (p. 1001) for the complete details.

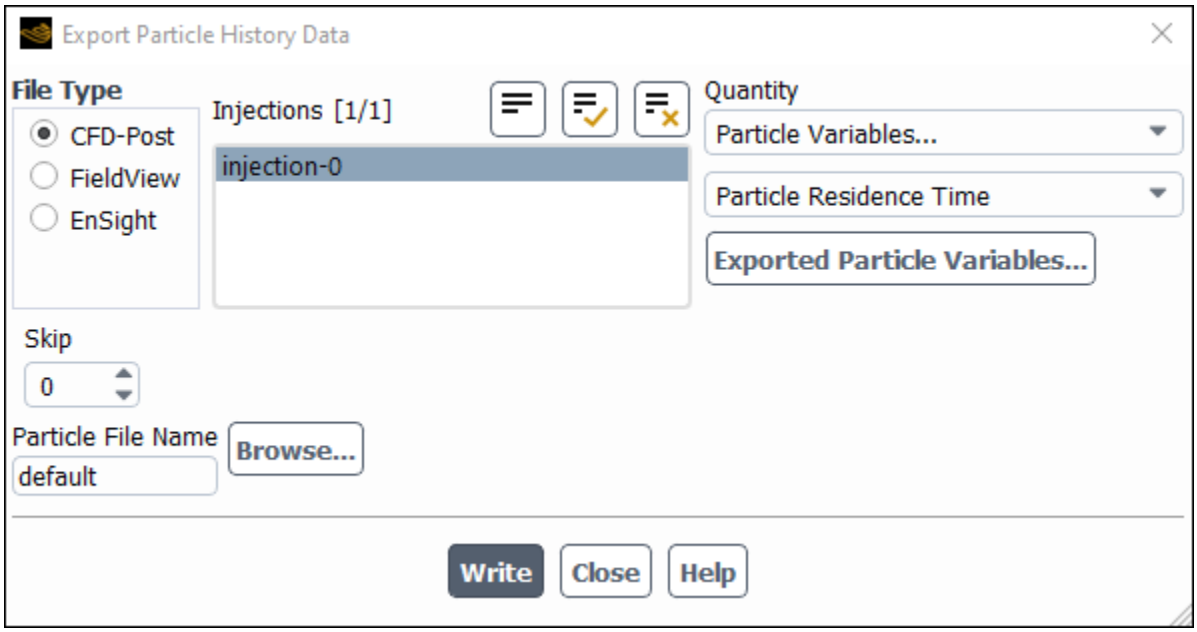
3.16. Exporting Steady-State Particle History Data

Particle history data can be exported for steady-state solutions or for single transient particle steps by selecting the **Particle History Data...** option under the **File/Export** ribbon tab and performing the steps described in this section. For details about exporting particle history data automatically during transient simulations, see Creating Automatic Export Definitions for Transient Particle History Data (p. 1004).

File → Export → Particle History Data...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 3.11: The Export Particle History Data Dialog Box



1. Specify the **File Type** you want to export by selecting one of the following:

   - **CFD-Post** for the CFD-Post compatible format
   - **FieldView** for the FIELDVIEW format
   - **EnSight** for the EnSight format
   - **Geometry** for the `.ibl` format (not available when **Unsteady Particle Tracking** is enabled in the **Discrete Phase Model** dialog box)

   Important:

   If you plan to export particle data to EnSight, you should first verify that you have already written the files associated with the EnSight Case Gold file type by using the **File/Export/Solution Data...** ribbon tab option (see EnSight Case Gold Files (p. 992)).

   Select the predefined injections that are the source of the particles from the **Injections** selection list. See Creating and Modifying Injections (p. 2753) for details about creating injections.

2. Select the particle variables contained in the export file by clicking the **Exported Particle Variables...** button and selecting the variables appearing in the **Reporting Variables** dialog box (Figure 24.68: The Reporting Variables Dialog Box (p. 2857)), as described in Reporting of Current Positions for Unsteady Tracking (p. 2858). Note that although the variable **Particle Residence Time** does not appear in the **Available Particle Variables** selection list, it is always exported.

3. If you have added the **Color by** variable in the **Reporting Variables** dialog box, select an appropriate category and variable under **Quantity** for the particle data to be exported.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. If your exported particle history file is too large to postprocess because there are too many tracks or particles written to the file, you can reduce the number of particle tracks by increasing the **Skip** value.

5. To control the exported file size, the number of points of the particle trajectories can be reduced using the **Coarsen** value. This is only valid for steady-state particle trajectories.

   When exporting to an EnSight file, you can use this value to reduce the temporary file size written during tracking with a small effect on accuracy of the trajectory. The number of steps per trajectory finally exported to the EnSight file is the same as the **Number of Particle Time Steps**.

6. Enter the name (and folder path, if you do not want it to be written in the current folder) for the exported particle data file in the **Particle File Name** text box. Alternatively, you can specify it through the **Select File** dialog box, which is opened by clicking the **Browse...** button.

7. If you selected **EnSight** under **File Type**, you should specify the **EnSight Encas File Name**. Use the **Browse...** button to select the `.encas` file that was created when you exported the file with the **File/Export/Solution Data...** ribbon tab option. The selected file will be modified and renamed as a new file that contains information about all of the related particle files that are generated during the export process (including geometry, velocity, scalars, particle and particle scalar files).

   The name of the new file will be the root of the original file with `.new` appended to it (for example, if `test.encas` is selected, a file named `test.new.encas` will be written). It is this new file that should be read into EnSight. If you do not specify a **EnSight Encas File Name**, then you must create an appropriate `.encas` file manually.

8. If you selected **EnSight** under **File Type**, and you are exporting steady-state particle tracks, enter the **Number of Particle Time Steps**.

9. Click **Write** to export the particle history data. If you selected **EnSight** under **File Type**, data files will be written in both `.mpg` and `.mscl` formats.

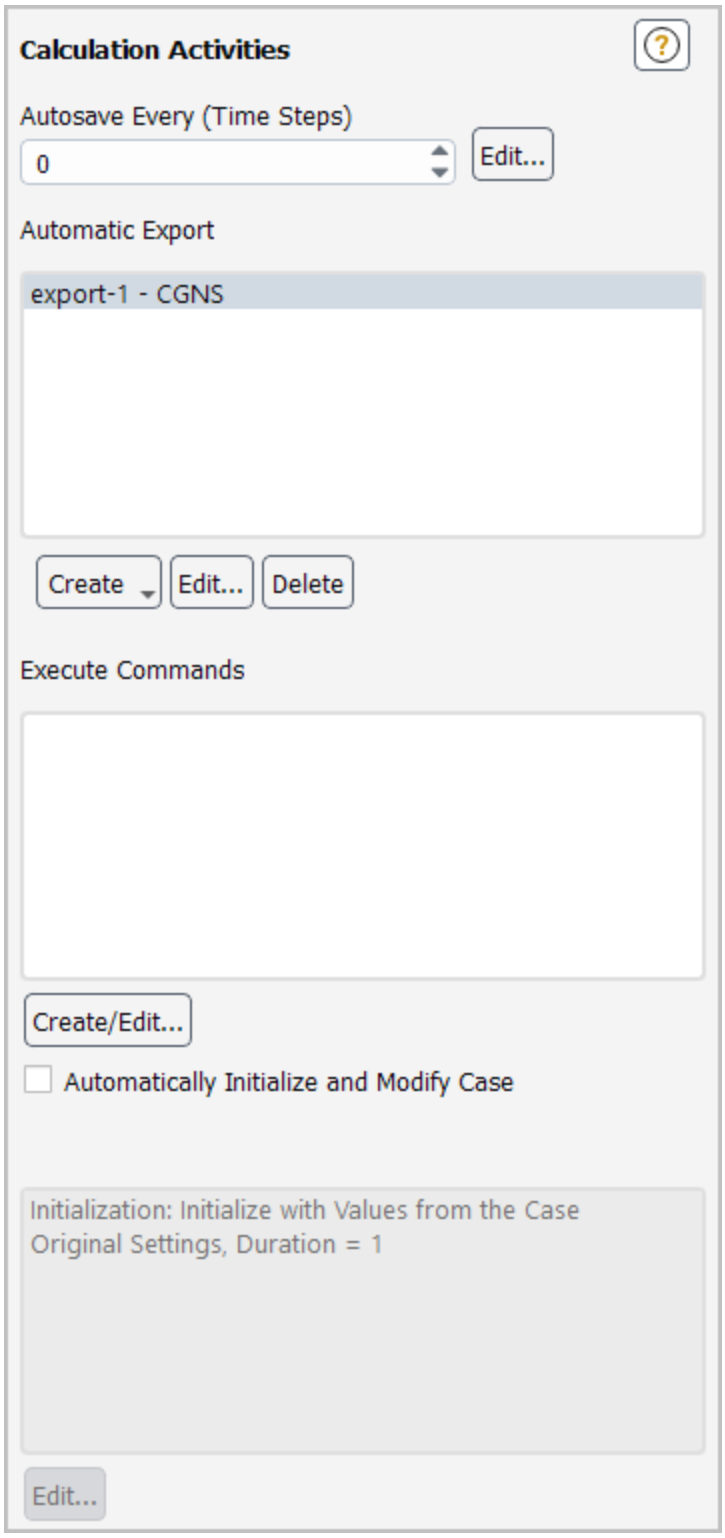
10. Click **Close** to close the dialog box.

3.17. Exporting Data During a Transient Calculation

Before you run a transient flow solution, you can set up the case file so that solution data and particle history data is exported as the calculation progresses. This is accomplished by creating automatic export definitions using the Calculation Activities Task Page (p. 5188) (Figure 3.12: The Calculation Activities Task Page (p. 1002)), as described in the following sections.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 3.12: The Calculation Activities Task Page



The names of the automatic export definitions you create are displayed in the **Automatic Export** selection list, along with the format in which it will be exported. You can edit or delete the definition by selecting a definition in the list and clicking the **Edit...** or **Delete** button,.

For additional information, see the following sections:

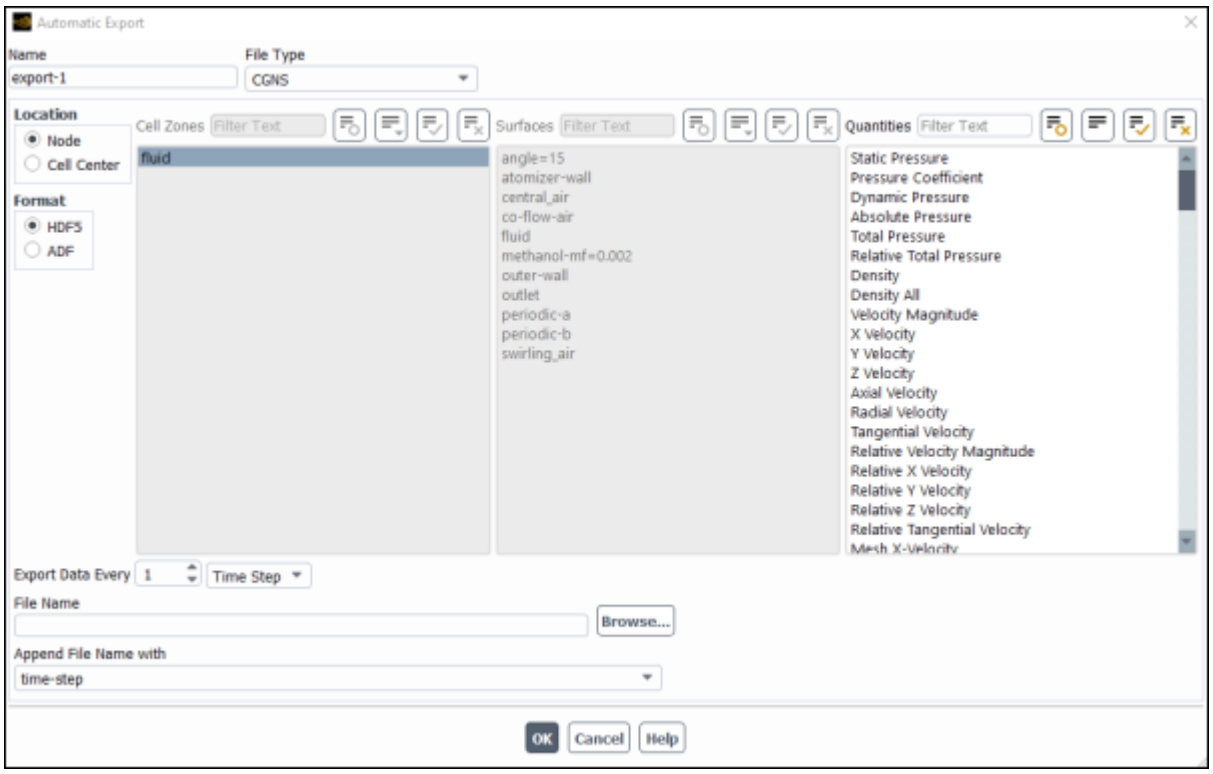
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3.17.1. Creating Automatic Export Definitions for Solution Data
3.17.2. Creating Automatic Export Definitions for Transient Particle History Data

3.17.1. Creating Automatic Export Definitions for Solution Data

To create an automatic export definition for solution data, begin by making sure that **Transient** is selected for **Time** in the General Task Page (p. 4684). Next, click the **Create** button under the **Automatic Export** selection list in the **Calculation Activities** task page (a drop-down list will appear). Select **Solution Data Export...** from the drop-down list to open the **Automatic Export** dialog box (Figure 3.13: The Automatic Export Dialog Box (p. 1003)).

Figure 3.13: The Automatic Export Dialog Box



Then perform the following steps:

1. Enter a name for the automatic export definition in the **Name** text box. This is the name that will be displayed in the **Automatic Export** selection list in the **Calculation Activities** task page.

2. Define the data to be exported by making selections in the relevant group boxes and selection lists: **File Type**, **Cell Zones**, **Surfaces**, **Interior Zone Surfaces**, **Quantities**, **Analysis**, **Structural Loads**, **Thermal Loads**, **Location**, **Delimiter**, **Format**, and **Heat Transfer Coefficient**. See ABAQUS Files (p. 986) – Tecplot Files (p. 999) for details about the specific options available for the various file types.

3. Set the frequency at which the solution data will be exported during the calculation using the **Export Data Every** field and specify whether you want the save rate based on **Time Steps** or **Flow Time**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. If you selected **EnSight Case Gold** from the **File Type** drop-down list, the **Separate Files for Each Time Step** option allows you to specify that separate files are written at the prescribed time steps. This option is enabled by default and is the recommended practice, as it ensures that all of the data is not lost if there is a disruption to the calculation (for example, from a network failure) before it is complete. If you choose to disable this option, all of the data for the `.scl1` and `.vel` files will be combined into a single file for each.

5. If you selected **CDAT for CFD-Post & EnSight** from the **File Type** drop-down list, by default Ansys Fluent will save a legacy case file (`.cas`) with every `.cdat` file (that is, at the specified **Frequency**), but you have the option of picking **Only if Case/Mesh Changes**, which can reduce the total space required to save all files associated with this export. A single `.cst` file will also be written when exporting during a transient calculation.

   For more information about the **CDAT for CFD-Post & EnSight** file type, refer to CDAT for CFD-Post and EnSight (p. 988).

6. Specify how the exported files will be named. Every file saved will begin with the characters entered in the **File Name** text box (note that a file extension is not necessary). You can specify a folder path if you do not want it written in the current folder. The **File Name** can also be specified through the **Select File** dialog box, which is opened by clicking the **Browse...** button.

   Next, make a selection in the **Append File Name with** drop-down list, to specify that the **File Name** be followed by either the time step number or flow time at which it was saved. Note that this selection is not available when exporting to EnSight. When **EnSight Case Gold** is selected from the **File Type** drop-down list, the time step number is always appended if the **Separate Files for Each Time Step** option is enabled; otherwise, no digits are appended.

   When appending the file name with the flow time, you can specify the number of decimal places that will be used by making an entry in the **Decimal Places in File Name** text box. By default, six decimal places will be used.

7. Click **OK** to save the settings for the automatic export definition.

For details about general limitations for exporting solution data and the manner in which it is exported, see Exporting Solution Data (p. 983).

Important:

If the files that are exported during multiple transient simulations are to be used as a set, you should run all of the simulations on the same platform, using the same number of processors. This ensures that all of the files are compatible with each other.

Note that if you selected **EnSight Case Gold** from the **File Type** drop-down list, Ansys Fluent does not support exporting data files to EnSight during a transient calculation in which a new cell zone or surface is created after the calculation has begun (as can be the case for an in-cylinder simulation, for example).

3.17.2. Creating Automatic Export Definitions for Transient Particle History Data

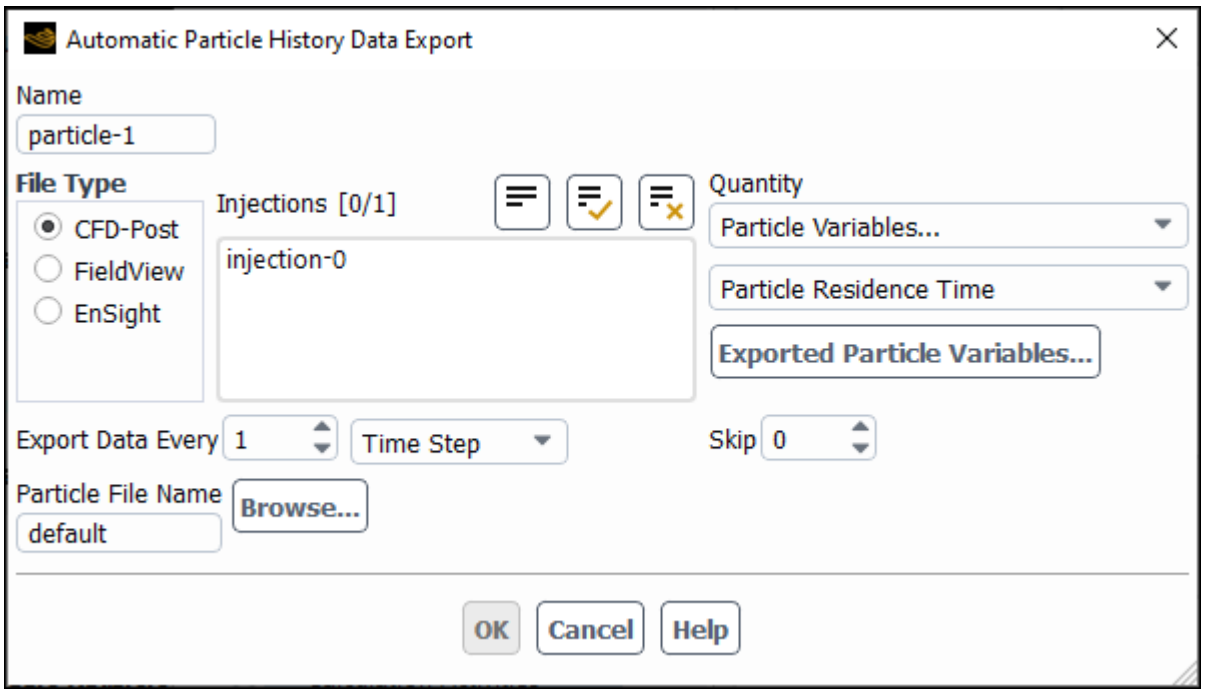
To create an automatic export definition for particle history data:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Make sure that **Unsteady Particle Tracking** is selected in the **Discrete Phase Model** dialog box.

2. In the **Calculation Activities** task page, under the **Automatic Export** selection list, click the **Create** drop-down list and select **Particle History Data Export...** to open the **Automatic Particle History Data Export** dialog box (Figure 3.14: The Automatic Particle History Data Export Dialog Box (p. 1005)).

   Figure 3.14: The Automatic Particle History Data Export Dialog Box



3. Enter a name for the automatic export definition in the **Name** text box. This is the name that will be displayed in the **Automatic Export** selection list in the **Calculation Activities** task page. Make sure that the chosen name is not already used for another **Automatic Export**.

4. Choose the **File Type** you want to export by selecting one of the following:

   - **CFD-Post** for the CFD-Post compatible format
   - **FieldView** for the FIELDVIEW format
   - **EnSight** for the EnSight format

   Important:

   If you plan to export particle data to EnSight, you should first set up an automatic export definition so that solution data is also exported to EnSight during this calculation (see Creating Automatic Export Definitions for Solution Data (p. 1003)). As described in the steps that follow, some of the settings must correspond between the two automatic export definitions.

5. Select the predefined injections that are the source of the particles from the **Injections** selection list. See Creating and Modifying Injections (p. 2753) for details about creating injections.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Select the particle variables contained in the export file by clicking the **Exported Particle Variables...** button and selecting the variables appearing in the **Reporting Variables** dialog box (Figure 24.68: The Reporting Variables Dialog Box (p. 2857)), as described in Reporting of Current Positions for Unsteady Tracking (p. 2858). Note that although the variable **Particle Residence Time** does not appear in the **Available Particle Variables** selection list, it is always exported.

7. If you have added the **Color by** variable in the **Reporting Variables** dialog box, select an appropriate category and variable under **Quantity** for the particle data to be exported.

8. In the **Export Data Every** integer number entry box, specify the frequency for appending the data during the solution process. You can export particle history data frequency based on either **Time Step** or **Flow Time** for unsteady flow cases, or based on **DPM Iteration** for steady flow cases.

9. If your exported particle history file is too large to postprocess because there are too many tracks or particles written to the file, you can reduce the number of particle tracks by increasing the **Skip** value.

10. If you selected **EnSight Case Gold** for the **File Type**, the **Separate Files for Each Time Step** option allows you to specify that separate files are written at the prescribed time steps. This option is enabled by default and is the recommended practice, as it ensures that all of the data is not lost if there is a disruption to the calculation (for example, from a network failure) before it is complete. If you choose to disable this option, all of the data for the `.mscl` and `.mpg` files will be combined into a single file for each.

    Important:

    The setting for the **Separate Files for Each Time Step** option should be the same (that is, enabled or disabled) as that of the automatic export definition you set up to export solution data to EnSight during this calculation. For details, see Creating Automatic Export Definitions for Solution Data (p. 1003).

11. Enter the name (and folder path, if you do not want it to be written in the current folder) for the exported particle data file in the **Particle File Name** text box. Alternatively, you can specify it through the **Select File** dialog box, which is opened by clicking the **Browse...** button. Make sure the file name is different from other existing **Automatic Export** definitions to avoid overwriting data.

    If you selected **FieldView** under **File Type**, you can write particle history data for each time step to separate files by including the character string `%t` in the particle file name. At the time of export, Fluent will substitute `%t` with the current time step number. The saved files can be processed by FieldView 14 or newer.

12. If you selected **EnSight** under **File Type**, you should specify the **EnSight Encas File Name**. Enter the same name (and folder path, if necessary) that you entered in the **File Name** text box when you set up the automatic export definition for exporting solution data to EnSight during this calculation. (For details, see Creating Automatic Export Definitions for Solution Data (p. 1003)). The `.encas` file created during the solution data export will be modified and renamed as a new file that contains information about all of the related particle files that are generated after every time step during the export process (including geometry, velocity, scalars, particle and particle scalar files). The name of the new file will be the root of the original `.encas` file with `.new` appended

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

to it (for example, if the solution data export creates `test.encas`, a file named `test.new.encas` will be written for the particle data export). It is this new file that should be read into EnSight.

Important:

If you do not specify an **EnSight Encas File Name**, you will have to manually create an appropriate `.encas` file.

13. Click **OK** to save the settings for the automatic export definition.

The particle data will be exported as it is generated during the transient calculation. If you selected **EnSight** under **File Type**, data files will be written in both `.mpg` and `.mscl` formats.

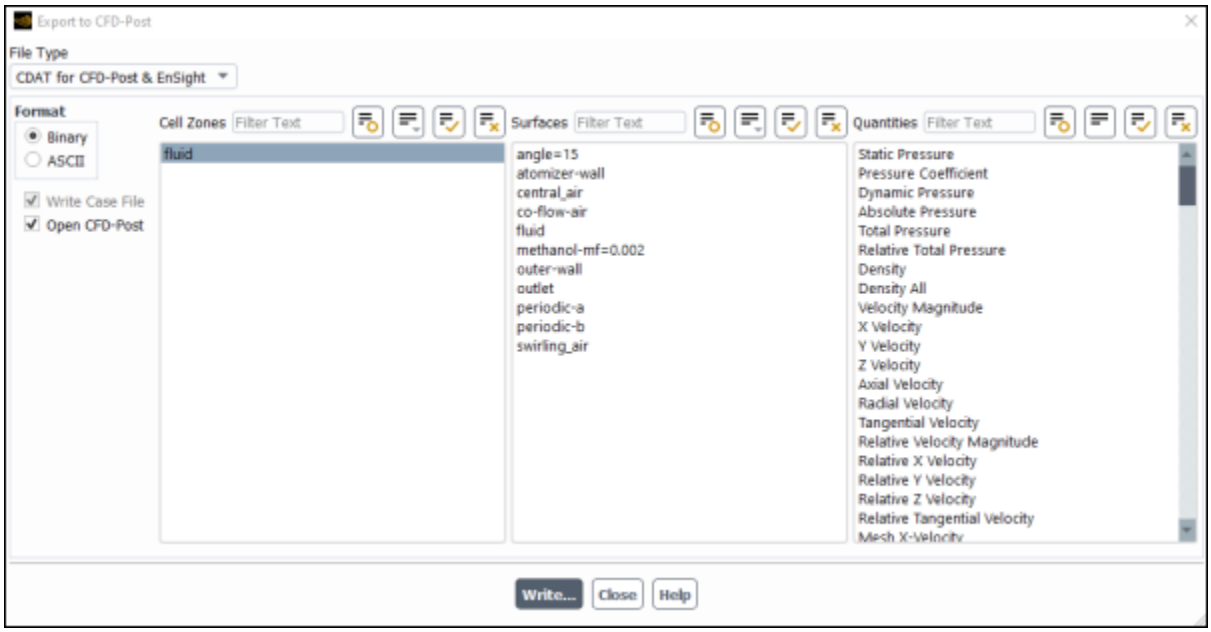
3.18. Exporting to Ansys CFD-Post

You can use the **Export to CFD-Post** dialog box (Figure 3.15: The Export to CFD-Post Dialog Box (p. 1007)) to export the results from cell zones and/or surfaces to files that are compatible with Ansys CFD-Post (and EnSight).

For instructions on exporting variables in Ansys CFD-Post compatible file format (`.cdat`) using this dialog box, see CDAT for CFD-Post and EnSight (p. 988).

File → Export to CFD-Post...

Figure 3.15: The Export to CFD-Post Dialog Box



By default, the **Open CFD-Post** option is enabled so that the following actions will occur after the files are exported:

1. A CFD-Post session opens automatically.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. The case and `.cdat` files are loaded in CFD-Post.

3. CFD-Post displays the results.

If you disable the **Open CFD-Post** option, you can open CFD-Post manually and use the **Load Results** item in the **File** drop-down menu to load the results files. For more information about this feature in CFD-Post, see the separate Ansys CFD-Post manual.

Note:

- When exporting to CFD-Post, Fluent only saves a new case file if the mesh or case has changed since the last time the case file was saved, which conserves storage space.
- CFD-Post may have difficulty reading transient cases that have changing mesh topology and/or zones added/removed during the simulation.

3.19. Parallel Exporting to Ansys EnSight

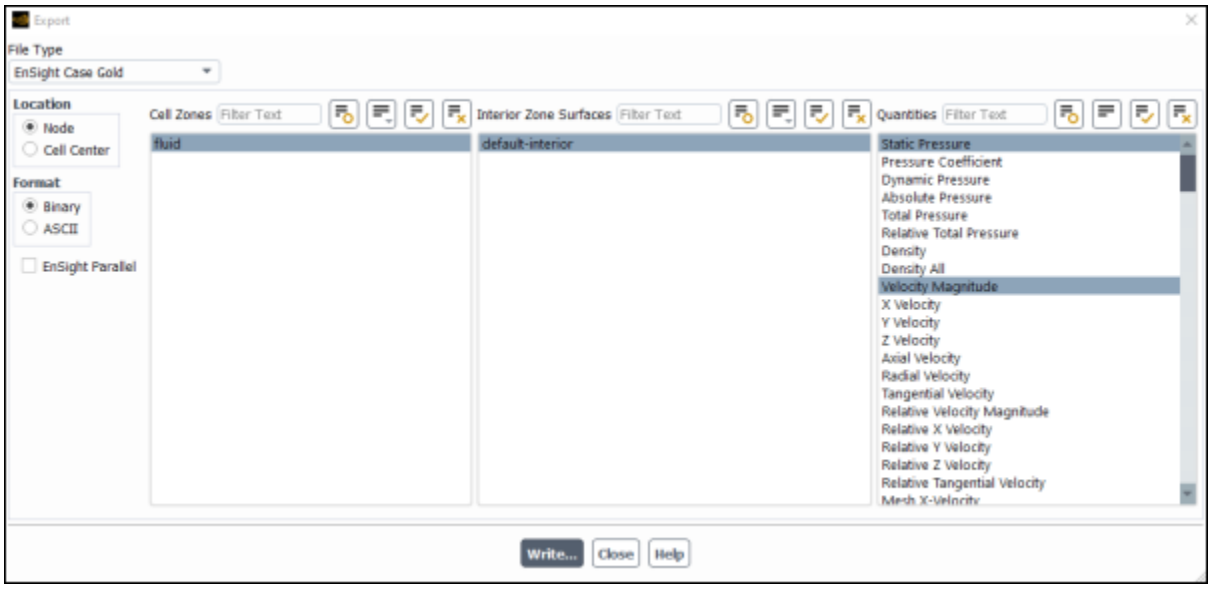
Using the **EnSight Parallel** option in the **Export** dialog box you can save Fluent solution case and data files for postprocessing in Ansys EnSight Enterprise (EnSight in parallel).

To setup parallel EnSight export in Fluent:

1. Ensure that you are running Fluent in parallel. *This functionality is only available when Fluent is running in parallel.*

2. Open the **Export** dialog box.

   File → **Export** → **Solution Data...**



3. Select **EnSight Case Gold** from the **File Type** drop-down.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Select either **Node** or **Cell Center** for the **Location** where the data will be written.
5. Select the **Format** for how the data will be saved.
6. Enable **EnSight Parallel**.
7. Select either the **Cell Zones** or **Surfaces** that you want to export.
8. Select the **Quantities** that you want to export.
9. Click **Write...** to specify the location where the data will be saved.

Note that you can also setup for automatic exporting of solution data using the same steps as described above but using the Automatic Export Dialog Box (p. 5193).

Important:

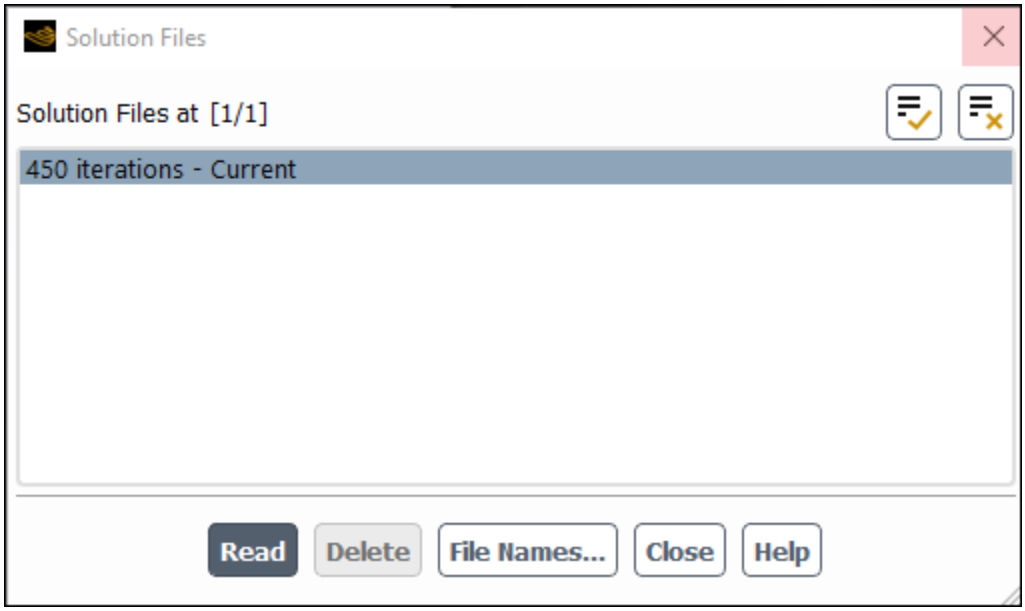
You can either export solution data from **Cell Zones** or **Surfaces**, but you cannot do both at once.

3.20. Managing Solution Files

You can manage your solution files effectively and efficiently using the **Solution Files** dialog box. Here, you can select previously saved files created using the **Autosave** dialog box and read or delete them.

File → Solution Files...

Figure 3.16: The Solution Files Dialog Box



The **Solution Files** dialog box (Figure 3.16: The Solution Files Dialog Box (p. 1009)) lists all of the solution files that have been automatically saved. They are listed by iteration number or time step/flow time. For the file that is currently read in, the status of `current` will appear in the **Solution Files at** list. You can make any of the files in the list current by clicking the **Read** button. Note that if more than one file

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

is selected, the **Read** button is disabled. When an earlier solution is made current, the solution files that were generated for a later iteration/time step will be removed from this list when the calculation continues.

You can delete solution files by selecting an entry in the list and clicking **Delete**. Note that a currently loaded solution file cannot be deleted, however multiple (non-current) files can be selected and deleted. If multiple files are selected and one of those files is a currently loaded solution file, clicking **Delete** will result in the current solution file being skipped.

You can click the **File Names...** button to obtain information about the solution files and the path of the associated files.

When running Fluent under Workbench, you can recover the latest case and data files from the system folder (`<project name>\dp0\FFF\Fluent\` for analysis system and `<project name>\dp0\FLU \Fluent\` for component system) by clicking **Recover Missing Solution...** The recovered files will appear in the **Solution Files at** list. You can use this option when the latest matching/compatible case and data files are missing from the **Solution Files at** list.

The **Solution Files** dialog box is particularly useful for reading in files that were saved during the autosave session, since case and data files may not necessarily have the same file name.

3.21. Mesh-to-Mesh Solution Interpolation

Ansys Fluent can interpolate solution data for a given geometry from one mesh to another, allowing you to compute a solution using one mesh (for example, hexahedral) and then change to another mesh (for example, hybrid) and continue the calculation using the first solution as a starting point.

Important:

Ansys Fluent does zeroth-order interpolation for interpolating the solution data from one mesh to another.

For additional information, see the following sections:

3.21.1. Performing Mesh-to-Mesh Solution Interpolation

3.21.2. Format of the Interpolation File

3.21.1. Performing Mesh-to-Mesh Solution Interpolation

The procedure for mesh-to-mesh solution interpolation is as follows:

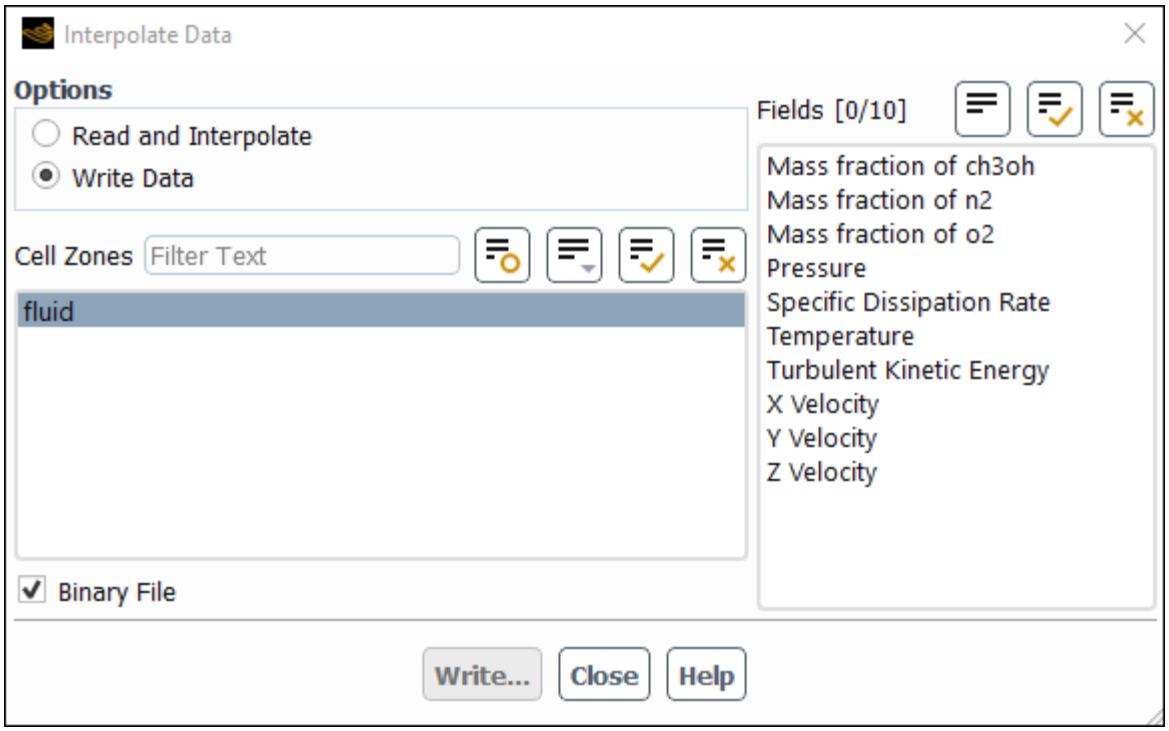
1. Set up the model and calculate a solution on the initial mesh.

2. Write an interpolation file for the solution data to be interpolated onto the new mesh, using the **Interpolate Data** dialog box (Figure 3.17: The Interpolate Data Dialog Box (p. 1011)).

   File → **Interpolate...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 3.17: The Interpolate Data Dialog Box



a. Under **Options**, select **Write Data**.

b. In the **Cell Zones** selection list, select the cell zones for which you want to save data to be interpolated.

> **Note:**
>
> If your case includes both fluid and solid zones, write the data for the fluid zones and the data for the solid zones to separate files.

c. Select the variable(s) for which you want to interpolate data in the **Fields** selection list. All Ansys Fluent solution variables are available for interpolation.

d. Select the **Binary File** check box if you want a binary interpolation file to be generated.

> **Note:**
>
> Writing a binary interpolation file is significantly faster and requires less memory than writing a text file.

e. Click **Write...** and specify the interpolation file name in the resulting **Select File** dialog box. The file format is described in Format of the Interpolation File (p. 1012).

3. Set up a new case.

a. Read in the new mesh, using the appropriate ribbon tab item (**File/Read/** or **File/Import/**).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. Define the appropriate models.

Important:

Enable all of the models that were enabled in the original case. For example, if the energy equation was enabled in the original case and you forget to enable it in the new case, the temperature data in the interpolation file will not be interpolated.

c. Define the boundary conditions, material properties, and so on.

Important:

An alternative way to set up the new case is to save the boundary conditions from the original model using the `write-settings` text command, and then read in those boundary conditions with the new mesh using the `read-settings` text command. See Reading and Writing Boundary Conditions (p. 971) for further details.

4. Read in the data to be interpolated.

File → Interpolate...

a. Under **Options**, select **Read and Interpolate**.

b. In the **Cell Zones** list, select the cell zones for which you want to read and interpolate data.

If the solution has not been initialized, computed, or read, all zones in the **Cell Zones** list are selected by default, to ensure that no zone remains without data after the interpolation. If all zones already have data (from initialization or a previously computed or read solution), select a subset of the **Cell Zones** to read and interpolate data onto a specific zone (or zones).

c. Click the **Read...** button and specify the interpolation file name in the resulting **Select File** dialog box.

Important:

If your case includes both fluid and solid zones, the two sets of data are saved to separate files. Hence perform these steps twice, once to interpolate the data for the fluid zones and once to interpolate the data for the solid zones.

5. Reduce the under-relaxation factors and calculate on the new mesh for a few iterations to avoid sudden changes due to any imbalance of fluxes after interpolation. Then increase the under-relaxation factors and compute a solution on the new mesh.

3.21.2. Format of the Interpolation File

An example of an interpolation file is shown below:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
3
2
34800
3
x-velocity
pressure
y-velocity
(-0.068062
-0.0680413
...
```

The format of the interpolation file is as follows:

- The first line is the interpolation file version. It is 2 for files generated using Ansys Fluent 12.0 through 14.0, 3 for text files generated using Ansys Fluent 14.5, 4 for binary files generated using single precision Ansys Fluent 14.5, and 5 for files generated using double precision Ansys Fluent 14.5.
- The second line is the dimension (2 or 3).
- The third line is the total number of points.
- The fourth line is the total number of fields (temperature, pressure, and so on) included.
- Starting at the fifth line is a list of field names. To see a complete list of the field names used by Ansys Fluent, enter the `display/contour` text command and view the available choices by pressing **Enter** at the `contours of>` prompt. The list depends on the models turned on.
- After the field names is a section for each list of x, y, and (in 3D) z coordinates for all the data points.
- At the end is a section for each list of the field values at all the points in the same order as their names. The number of coordinate and field points should match the number given in line 3.
- For version 3 interpolation files, the sections are bounded by "(" and ")".
- For version 4 and 5 interpolation files, the sections are bounded by "(" and "\nEnd of Binary Section 0)".
- The delimiters help skip a section if the associated model is not enabled. With version 2 interpolation files (where the delimiters do not exist), sections may not be skipped properly if the size of the field cannot be determined without enabling the associated model. This may result in incorrect interpolation of the subsequent field variables.

Important:

An interpolation file written with Ansys Fluent 14.5 and above is not readable in prior versions of Ansys Fluent.

3.22. Mapping Data for Fluid-Structure Interaction (FSI) Applications

Ansys Fluent allows you to map variables (for example, temperature, pressure) from the cell or face zones of an Ansys Fluent simulation onto locations associated with a finite element analysis (FEA) mesh.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The results are written to a file for inclusion into an FEA simulation. During this process, both the original and the new mesh can be viewed simultaneously. Ansys Fluent maps the data using zeroth-order interpolation, and can write the output file in a variety of formats.

This capability is useful when solving fluid-structure interaction (FSI) problems, and allows you to perform further analysis on the solid portion of your model using FEA software. Mapping the data may be preferable to simply exporting the Ansys Fluent data file (as described in Exporting Solution Data (p. 983)), since the meshes used in CFD analysis are typically finer than those used in finite element analysis.

For additional information, see the following sections:

3.22.1. FEA File Formats

3.22.2. Using the FSI Mapping Dialog Boxes

3.22.1. FEA File Formats

The FEA software types that are compatible with Ansys Fluent's FSI mapping capability include ABAQUS, I-deas, ANSYS, NASTRAN, and PATRAN. For details about the kinds of files that can be read or written during this process, see Table 3.2: FEA File Extensions for FSI Mapping (p. 1014).

Table 3.2: FEA File Extensions for FSI Mapping

Type	Input File	Output File
ABAQUS	`.inp`	`.inp`
I-deas	`.unv`	`.unv`
ANSYS	`.cdb`, `.neu`	`.cdb`
NASTRAN	`.bdf`	`.bdf`
PATRAN	`.neu`, `.out`, `.pat`	`.out`

3.22.2. Using the FSI Mapping Dialog Boxes

To begin the process of mapping Ansys Fluent data, you must first create a mesh file that can be used as the input file in the steps that follow. The resolution of the mesh should be appropriate for your eventual finite element analysis. You are free to use the method and preprocessor of your choice in the creation of this file, but the end result must correspond to one of the entries in the Input File column of Table 3.2: FEA File Extensions for FSI Mapping (p. 1014).

When creating the input file, note the following:

- While the input file may be scaled when it is read into Ansys Fluent, the volumes or surfaces on which the data is to be mapped must otherwise be spatially coincident with their counterparts in the Ansys Fluent simulation.
- Ansys Fluent can map volume and surface data only for 3D cases; data mapping is not supported for 2D cases since data mapping for edges is not supported.
- The input file can be only a portion of the overall FEA model (that is, you can exclude the parts of the model on which you are not mapping Ansys Fluent data). When this is the case, note that the numbering of the nodes and elements in the input file must match the numbering of the nodes and elements in the complete file you will use for your finite element analysis.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Next, read a case file in Ansys Fluent and make sure data is available for mapping, either by running the calculation or by reading a data file.

Finally, perform the following steps to generate an output file in which the Ansys Fluent data has been mapped to the mesh of the input file:

1. Open the Ansys Fluent dialog box that is appropriate for the zones from which the data is to be taken. If the data you are mapping is from a volume (for example, the cell zone of a solid region), open the **Volume FSI Mapping** dialog box using the **File/FSI Mapping/Volume...** ribbon tab item (Figure 3.18: The Volume FSI Mapping Dialog Box for Cell Zone Data (p. 1015)). If instead the data is from a surface (for example, a face boundary zone), open the **Surface FSI Mapping** dialog box using the **File/FSI Mapping/Surface...** ribbon tab item (Figure 3.19: The Surface FSI Mapping Dialog Box for Face Zone Data (p. 1016)).

   File → FSI Mapping → Volume...

   or

   File → FSI Mapping → Surface...

Figure 3.18: The Volume FSI Mapping Dialog Box for Cell Zone Data

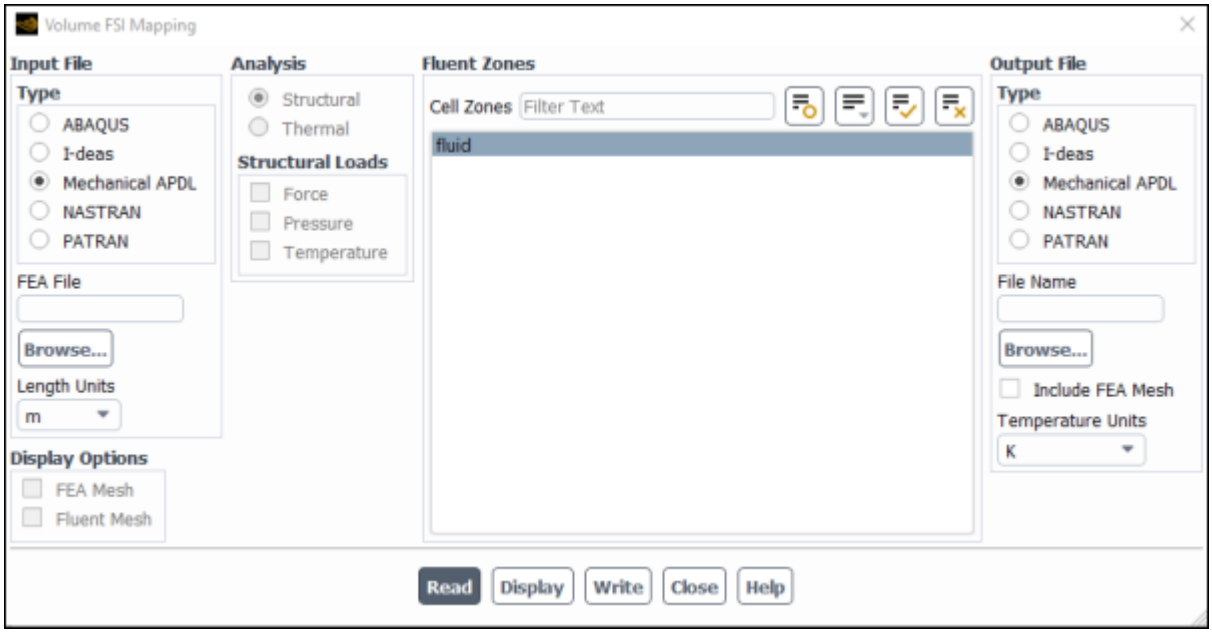
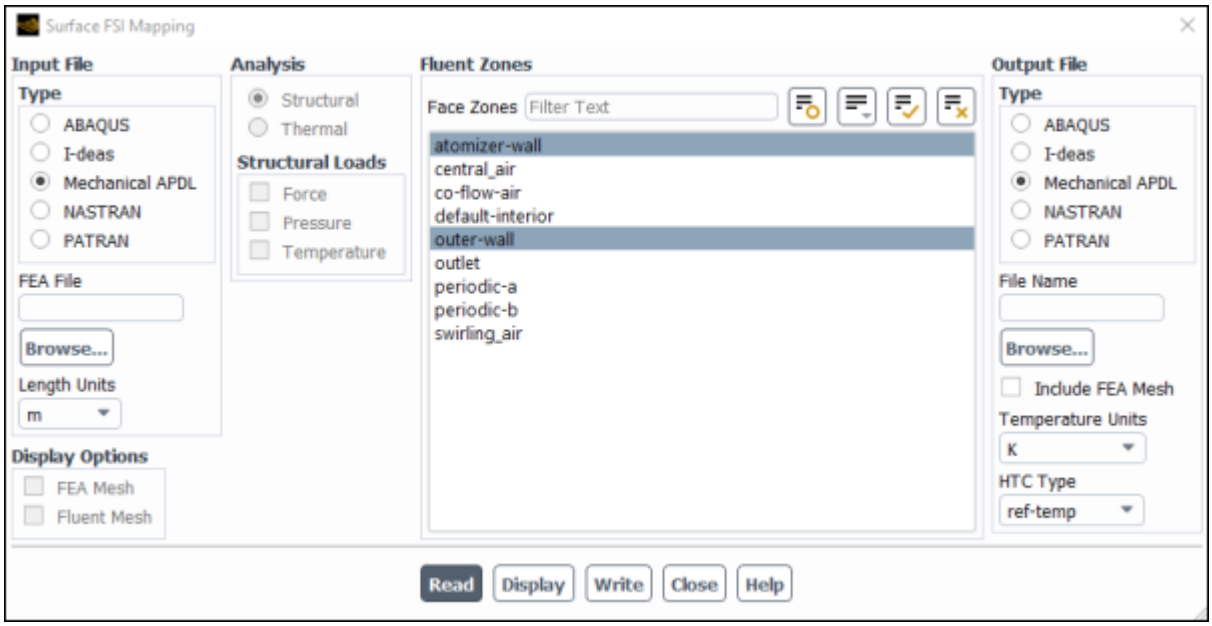


Figure 3.19: The Surface FSI Mapping Dialog Box for Face Zone Data



2. Specify the parameters of the input file and read it into Ansys Fluent.

   a. Select the format of the input file from the **Type** list in the **Input File** group box, based on the FEA software with which it is associated. The choices include:

      - **ABAQUS**
      - **I-deas**
      - **Mechanical APDL**
      - **NASTRAN**
      - **PATRAN**

      For a list of the file extensions associated with these types, see the Input File column of Table 3.2: FEA File Extensions for FSI Mapping (p. 1014).

   b. Enter the name and extension (along with the folder path, if it is not in the current folder) of the input file in the **FEA File** text-entry box. Alternatively, you can specify it through the **Select File** dialog box, which is opened by clicking the **Browse...** button.

   c. Specify the length units that were used in the creation of the input file by making a selection from the **Length Units** drop-down menu. This ensures that the input file is scaled appropriately relative to the Ansys Fluent file.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

d. Click the **Read** button to read the input file into memory.

Important:

Note that the input file will only be held in memory until the output file is written, or until the FSI mapping dialog box is closed.

3. Display the meshes so that you can visually verify that the input file is properly scaled and aligned with the Ansys Fluent mesh file.

   a. Make sure that the **FEA Mesh** and **Fluent - Mesh** options are selected in the **Display Options** group box. Note that you can disable either of these options if you want to examine one of the meshes independently.

   b. Click the **Display** button to display the meshes in the graphics window.

   Important:

   For the Ansys Fluent mesh, only the zones selected in the **Fluent Zones** group box will be displayed—in this case, the default selections. If the default zones are not appropriate, you should redisplay the meshes after you make your zone selections in a later step.

4. Specify the type of data variables to be mapped.

   a. Select either **Structural** or **Thermal** in the **Analysis** group box. Your selection should reflect the kind of further analysis you intend to pursue, and will determine what variables are available for mapping.

   b. Enable the variables you want to map in the **Structural Loads** or **Thermal Loads** group box. When mapping volume data, you can enable only **Temperature**. When mapping surface data, you can enable **Force**, **Pressure**, and **Temperature** for structural analysis, or **Temperature**, **Heat Flux**, and **Heat Trans Coeff** for thermal analysis.

   Important:

   Note that the **Energy Equation** must be enabled in the **Energy** dialog box if you want to map temperature for a structural analysis or any variable for a thermal analysis.

5. Select the zones that contain the data to be mapped in the **Fluent Zones** group box. You can select individual zones in the **Cell Zones** or **Face Zones** selection lists, or select all zones of a particular type in the **Zone Type** selection list. If you modify the default selections, you should display the meshes again, as described previously.

   Important:

   - Note that all wall zones in the **Face Zones** selection list are selected by default in the **Surface FSI Mapping** dialog box, and this includes the shadow walls created

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

for two-sided walls. If your Ansys Fluent file contains a wall/shadow pair (for example, separating a solid zone from a fluid zone), you should make sure that only the correct wall or shadow of the pair is selected.

- Inlet zones do not have heat transfer coefficient data, and so any attempts to map this combination will be ignored.

6. Specify the parameters of the output file and write it.

   a. Select the format of the output file from the **Type** list in the **Output File** group box, based on the software with which you plan to perform your finite element analysis. The choices in this list are the same as those for the input file type. Note that you can select an output file type that is different from the input file type.

      For details about the file extensions associated with the various types of output files, see the Output File column of Table 3.2: FEA File Extensions for FSI Mapping (p. 1014).

   b. Enter the name (with the folder path, if appropriate) of the output file in the **File Name** text-entry box. Alternatively, you can specify it through the **Select File** dialog box, which is opened by clicking the **Browse...** button.

   c. To include additional FEA information like node/element information in the exported output file, enable **Include FEA Mesh**. By default, this option is disabled and therefore, only the selected boundary condition values are exported.

   d. When mapping temperature for a structural analysis or any variable for a thermal analysis, make a selection in the **Temperature Units** drop-down menu. Table 3.3: Units Associated with the Temperature Units Drop-Down List Selections (p. 1018) shows the units for the mapped variables, depending on the **Temperature Units** selection.

      Table 3.3: Units Associated with the Temperature Units Drop-Down List Selections

Temperature Units Selection	Temperature	Heat Flux	Heat Transfer Coefficient
K	K	W/m^2	W/m^2-K
C	$^\circ C$	W/m^2	W/m^2- $^\circ C$
F	$^\circ F$	BTU/ft^2-hr	BTU/ft^2-hr- $^\circ F$

   e. When mapping the heat transfer coefficient for a thermal analysis, make a selection in the **HTC Type** drop-down menu to determine how the heat transfer coefficient h_{eff} is calculated.

      ref-temp

      calculates h_{eff} using Equation 42.50 (p. 4269), where T_{ref} is the reference temperature defined in the Reference Values Task Page (p. 5154). Note that this option has the same definition as the field variable **Surface Heat Transfer Coef.**, as described in Alphabetical Listing of Field Variables and Their Definitions (p. 4205).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

cell-temp

calculates h_{eff} using the general form of Equation 42.50 (p. 4269), but defines T_{ref} as the temperature of the cell adjacent to the face.

wall-func-htc

calculates h_{eff} using Equation 42.69 (p. 4281). Note that this option has the same definition as the field variable **Wall Func. Heat Tran. Coef.**, as described in Alphabetical Listing of Field Variables and Their Definitions (p. 4205).

f. Click **Write** to write an output file in which the Ansys Fluent data has been mapped to the mesh of the input file.

The input file will be released from memory when the output file is written.

3.23. Saving Picture Files

Graphic window displays can be saved in various 2D and 3D formats (see Choosing the Picture File Format (p. 1021) for a full listing of the supported file types). There can be slight differences between pictures and the displayed graphics windows depending on your settings and hardware, as the pictures may be generated using the internal software renderer while the graphics windows may use specialized graphics hardware for optimum performance. To eliminate such differences and save these files at the fastest rate possible, you must follow all of the following best practices:

- Run Cortex on a suitable machine with an appropriate graphics card and the latest drivers (for details, see the Ansys website). Note that you can assign Cortex to a particular machine using the `-gui_machine=<hostname>` command line option, or by selecting **Specify Machine** from the **Graphics Display Machine** list in the **Scheduler** tab of Fluent Launcher.
- Ensure that Cortex / the host process is run on a separate machine than that used for compute node 0. For example, do not include the machine assigned using the `-gui_machine` option as the first machine in the hosts file / machine list (specified using the `-cnf=x` command line option).
- Do not set the graphics driver to `null`, `x11` (for Linux), or `msw` (for Windows).
- When saving picture files, enable the **Fast hardcopy** option in the **Preferences** dialog box (under **Graphics**).

Many systems provide a utility to "dump" the contents of a graphics window into a raster file. This is generally the fastest method of generating a picture (since the scene is already rendered in the graphics window), and guarantees that the picture is identical to the window.

Note:

You can control the line thickness in saved images by specifying the **Line width** setting in **Preferences**, under **Save Picture Settings**.

For additional information, see the following sections:

3.23.1. Using the Save Picture Dialog Box

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

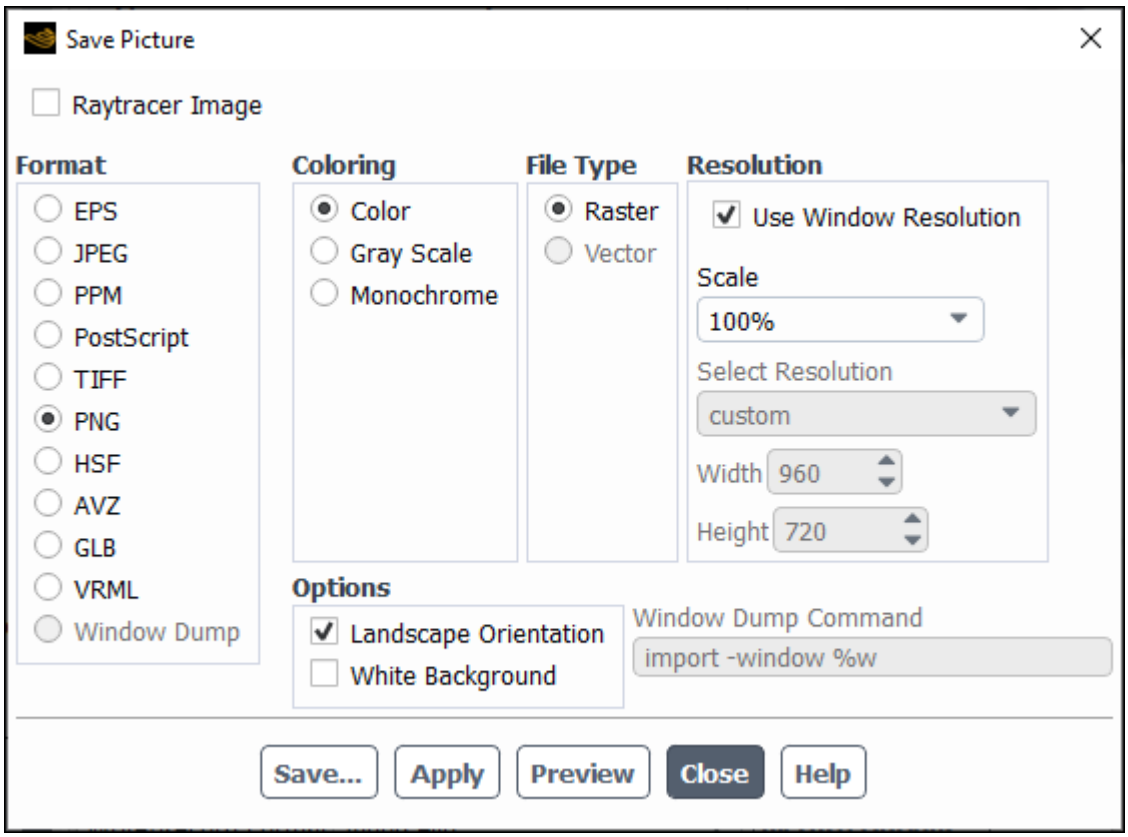
3.23.2. Picture Options for PostScript Files

3.23.1. Using the Save Picture Dialog Box

To set picture parameters and save picture files, use the Save Picture Dialog Box (p. 5254) (Figure 3.20: The Save Picture Dialog Box (p. 1020)).

File → Save Picture...

Figure 3.20: The Save Picture Dialog Box



For your convenience, this dialog box may also be opened using the **Save Picture** button ( ) in the standard toolbar.

The procedure for saving a picture file is as follows:

1. (Optional) Enable **Raytracer Image** if you want a raytracer to render the image prior to saving. Refer to Realistic Rendering Using Raytracing (p. 3995) for additional information.
2. Choose the picture file **Format**.
3. Set the **Coloring**.
4. Specify the **File Type**, if applicable.
5. Define the **Resolution**, if applicable.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Set the appropriate **Options**.
7. If you are generating a window dump, specify the **Window Dump Command**.
8. (optional) Preview the result by clicking **Preview**.

   Note:

   The preview for a **Monochrome** picture may not match the end result. If you see gray, it will be black when the picture is saved.

9. Click the **Save...** button and enter the file name in the resulting **Select File** dialog box. See Automatic Numbering of Files (p. 952) for information on special features related to file name specification.

If you are not ready to save a picture but want to save the current picture settings, click the **Apply** button instead of the **Save...** button. The applied settings become the defaults for subsequent pictures.

3.23.1.1. Choosing the Picture File Format

To choose the picture file format, select one of the following items in the **Format** list:

EPS

(Encapsulated PostScript) output is the same as PostScript output, with the addition of Adobe Document Structuring Conventions (v2) statements. Currently, no preview bitmap is included in EPS output. Often, programs that import EPS files use the preview bitmap to display on-screen, although the actual vector PostScript information is used for printing (on a PostScript device). You can save EPS files in raster or vector format.

JPEG

is a common raster file format.

PPM

output is a common raster file format.

PostScript

is a common vector file format. You can also choose to save a PostScript file in raster format.

TIFF

is a common raster file format.

PNG

is a common raster file format.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

HSF

is HOOPS Visualize Stream Format, a highly-compressible and streamable 2D/3D file format.

AVZ

is the free Ansys Viewer Format, which is a 3D file format allowing you to visualize, share, collaborate, and interactively manipulate the displayed object. For additional information on the Ansys Viewer, refer to Ansys Viewer User's Guide.

Note:

Graphics effects such as reflections and the ground plane grid are not exported to AVZ format. (211327)

Limitations for AVZ Export:

- Only mesh faces and edges (not nodes) can be displayed in the Ansys Viewer.
- Contour values may be displayed incorrectly in the Ansys Viewer, if the **Node Values** option was disabled when the `.avz` file was saved.
- Contour lines are not exported to AVZ format.
- Lighting effects and the brightness of the display may vary in AVZ when compared to the Fluent graphics windows.
- When the entire colormap represents a single value, the color on the surface(s) may display differently in the Ansys Viewer than it does when it is displayed in Ansys Fluent.
- Scenes containing multiple colormaps may show the colormaps overlapping when displayed in the Ansys Viewer. You can rearrange the colormaps by dragging and dropping them to different locations in the graphics window (top, bottom, right) in Ansys Fluent before saving in AVZ format to avoid this overlap issue.
- Colormap properties such as font type and size and number format are not retained when imported into the Ansys Viewer.
- The Ansys Viewer has a fixed size for pathline and particle track displays that may not match the exact settings of the pathline or particle track object when it was saved in `.avz` format in Ansys Fluent.
- 2D plots cannot be saved in `.avz` format, including report plots, residuals, and XY plots.
- Mesh included as part of a scene may appear black, if the transparency has been modified from **0**.
- Hovering over individual vectors will not display their values in the Ansys Viewer. This also applies for pathline displays with the **Style** set to either **line**, **point**, or **sphere**.
- The colormap type and precision displayed in the Ansys Viewer may not match how the colormap appeared when the `.avz` file was saved in Ansys Fluent.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Cylindrical/Spherical silhouettes (artificial edges rendered for surfaces characterized by roundness that do not have a 'sharp edge') are not properly preserved and displayed in the Ansys Viewer.
- Mesh displays that are colored by a material are not rendered properly in the Ansys Viewer.

GLB

is a 3D file format frequently used in virtual reality (VR) and augmented reality (AR) applications. The file size is relatively small as it is a binary format of the GL Transmission Format (glTF).

Note:

- You cannot save animation files in GLB format.
- You can only export and successfully view faces or faces + edges. Exporting only edges is not supported.

VRML

is a graphics interchange format that allows export of 3D geometrical entities that you can display in the Ansys Fluent graphics window. This format can commonly be used by VR systems and the 3D geometry can be viewed and manipulated in a web-browser graphics window.

Important:

Non-geometric entities such as text, titles, color bars, and orientation axis are not exported. In addition, most display or visibility characteristics set in Ansys Fluent, such as lighting, shading method, transparency, face and edge visibility, and the making of front faces transparent, are not explicitly exported but are controlled by the software used to view the VRML file.

Window Dump

(Linux systems only) selects a window dump operation for generating the picture. With this format, you must specify the appropriate **Window Dump Command**.

Important:

The advantage to saving the animation sequence using the **HSF File** option is that these files are highly compressible and can be viewed and interacted with using a “HOOPS Viewer” application on iOS and Android equipped devices. The “HOOPS Viewer” application can be downloaded to iOS and Android equipped devices from the “App Store” and “Google Play” respectively.

An advantage to saving the animation sequence using the **PPM Image** option is that you can use the separate pixmap image files for the creation of a single GIF file. GIF file creation can be done quickly with graphics tools provided by other third-party graphics packages such as ImageMagick, that is, `animate` or `convert`. For example, if you save the PPM files starting with the string `sequence-2`, and you are using the ImageMa-

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

gick software, you can use the `convert` command with the `-adjoin` option to create a single GIF file out of the sequence using the following command.

```
convert -adjoin sequence-2_00*.ppm sequence2.gif
```

3.23.1.2. Specifying the Color Mode

For all formats except VRML, HSF, and the window dump, specify the type of **Coloring** you want to use for the picture file.

- Select **Color** for a color-scale copy.
- Select **Gray Scale** for a gray-scale copy.
- Select **Monochrome** for a black-and-white copy.

Note:

The preview for a **Monochrome** picture may not match the end result. If you see gray, it will be black when the picture is saved.

Most monochrome PostScript devices render **Color** images in shades of gray, but to ensure that the color ramp is rendered as a linearly-increasing gray ramp, you should select **Gray Scale**.

3.23.1.3. Choosing the File Type

When you save an EPS (Encapsulated PostScript) or PostScript file, choose one of the following under **File Type**:

- A **Raster** file defines the color of each individual pixel in the image. Raster files have a fixed resolution. The supported raster formats are EPS, JPEG, PPM, PostScript, TIFF, and PNG.
- A **Vector** file defines the graphics image as a combination of geometric primitives like lines, polygons, and text. Vector files are usually scalable to any resolution. The supported vector formats include EPS, PostScript, and VRML.

Important:

For the quickest print time, you can save vector files for simple 2D displays and raster files for complicated scenes.

3.23.1.4. Defining the Resolution

For raster picture files (that is, JPEG, PPM, TIFF, and PNG), you can control the resolution of the picture image by specifying the size (in pixels). Set the desired **Width** and **Height** under **Resolution**. If the **Width** and **Height** are both zero, the picture is generated at the same resolution as the active graphics window. To check the size of the active window in pixels, click **Info** in the **Display Options** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For EPS and PostScript files, specify the resolution in dots per inch (**DPI**) instead of setting the width and height.

3.23.1.5. Picture Options

For all picture formats except VRML, HSF, and the window dump, you can control two additional settings under **Options**:

- Specify the orientation of the picture using the **Landscape Orientation** button. If this option is turned on, the picture is made in landscape mode; otherwise, it is made in portrait mode.
- Control the foreground/background color using the **White Background** option. If this option is enabled, the picture is saved with a white background and, for 2D plots, the axes and labels are saved in black. This option allows for the production of consistent images, regardless of variations in user preferences, such as when running in Dark theme or with a custom background color.

3.23.2. Picture Options for PostScript Files

Ansys Fluent provides options that allow you to save PostScript files that can be printed more quickly. The following options are found in the `display/set/picture/driver/ post-format` text menu:

`fast-raster`

enables a raster file that may be larger than the standard raster file, but will print much more quickly.

`raster`

enables the standard raster file.

`rle-raster`

enables a run-length encoded raster file that is about the same size as the standard raster file, but will print slightly more quickly. This is the default file type.

`vector`

enables the standard vector file.

3.23.2.1. Window Dumps (Linux Systems Only)

If you select the **Window Dump** format, the program uses the specified **Window Dump Command** to save the picture file. For example, if you want to use `xwd` to capture a window, set the **Window Dump Command** to

```
xwd -id %w >
```

When the dump occurs, Ansys Fluent automatically interprets `%w` to be the ID number of the active window.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When you click the **Save...** button, the **Select File** dialog box appears. Enter the file name for the output from the window dump (for example, `myfile.xwd`).

If you are planning to make an animation, save the window dumps into numbered files, using the %n variable. To do this, use the **Window Dump Command** (`xwd -id %w`), but for the file name in the **Select File** dialog box enter `myfile%n.xwd`. Each time a new window dump is created, the value of %n increases by one. So there is no need to tack numbers onto the picture file names manually.

To use the **ImageMagick** animate program, saving the files in MIFF format (the native **ImageMagick** format) is more efficient. In such cases, use the **ImageMagick** tool `import`. Set the default **Window Dump Command** enter

```
import -window %w
```

Click **Save...** to open the **Select File** dialog box. Specify the output format to be MIFF by using the `.miff` suffix at the end of file name.

The window dump feature is both, system and graphics-driver specific. Thus the commands available for dumping windows depends on the particular configuration.

The window dump captures the window exactly as it is displayed, including resolution, colors, transparency, and so on. For this reason, all of the inputs that control these characteristics are disabled in the **Save Picture** dialog box when you enable the **Window Dump** format. If you are using an 8-bit graphics display, use one of the built-in raster drivers (for example, TIFF) to generate higher-quality 24-bit color output rather than dumping the 8-bit window.

3.23.2.2. Previewing the Picture Image

Before saving a picture file, you have the option of previewing what the saved image will look like. Click **Preview** to open a new window that will display the graphics using the current settings. This allows you to investigate the effects of different options interactively before saving the final, approved picture.

3.24. Setting Data File Quantities

If you have requested the writing of files in the legacy format (see Reading and Writing Files in the Legacy Format (p. 958)), the information saved in a data file by default includes a standard set of quantities that were computed during the calculation. These quantities are specifically suitable for postprocessing and restarting solutions in Ansys Fluent. If, however, you plan to postprocess the data file in an application other than Ansys Fluent (such as Ansys CFD-Post) you may want to include additional quantities that are derived from the standard quantities.

Note:

- If you plan to postprocess using EnSight, it is recommended that you export EnSight Case Gold Files (p. 992) or Common Fluids Format - Post Files (p. 991) instead of saving legacy data files with additional quantities.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

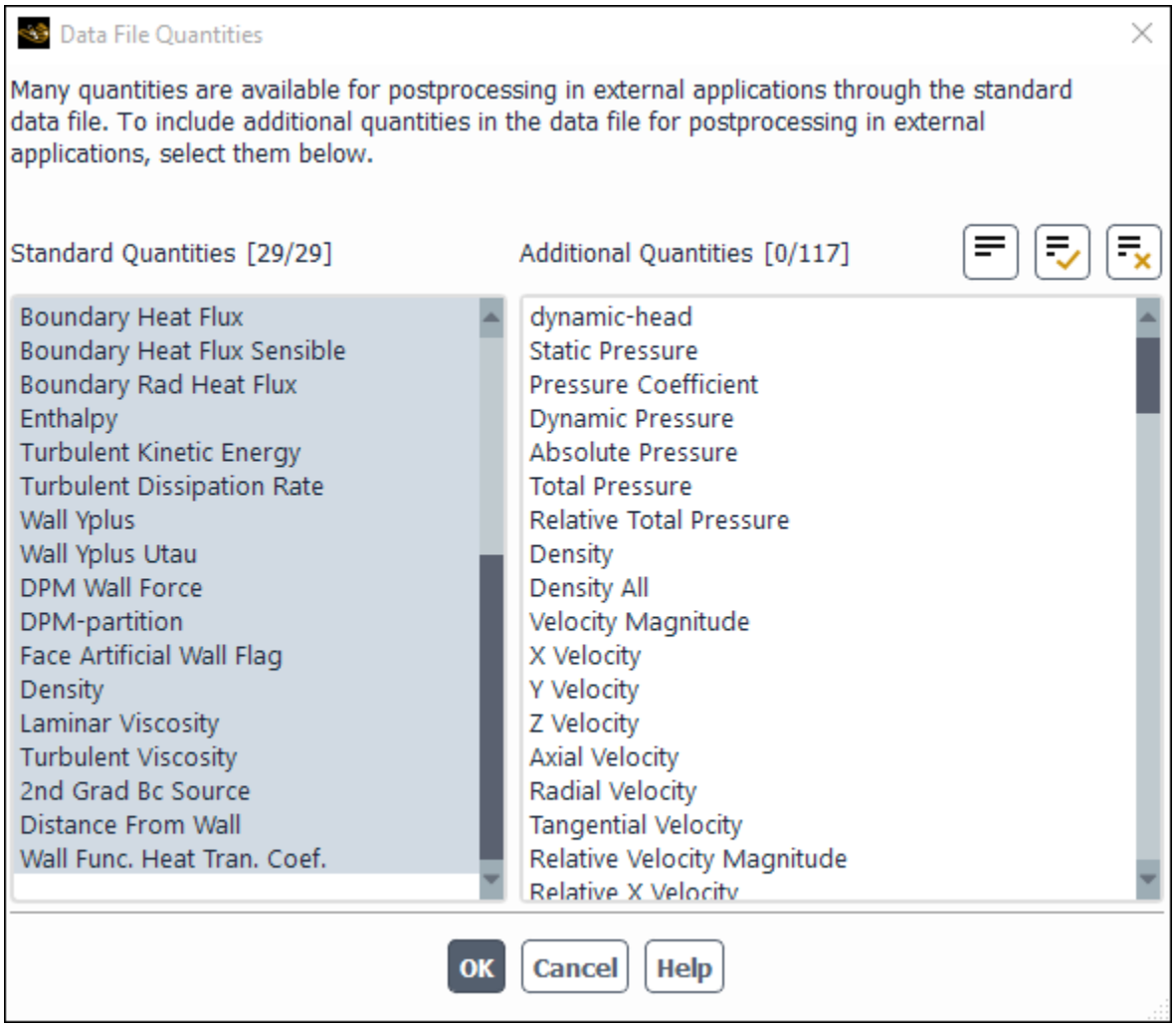
- If you plan to postprocess using Ansys CFD-Post, note that some standard quantities are also listed as additional quantities; where a standard quantity has a corresponding entry in the additional quantity list, the latter should be selected. This is because Ansys CFD-Post requires that some standard quantities are derived in a specific form.
- Not all standard quantities are available in CFD-Post for postprocessing. An example of such a quantity is Mach number.

The procedure for generating a legacy data file with additional quantities is as follows:

1. Read the case file for your simulation.
2. Initialize the solution using the Solution Initialization Task Page (p. 5179).
3. Specify the quantities to be written in the legacy data file, using the **Data File Quantities** dialog box (Figure 3.21: The Data File Quantities Dialog Box (p. 1027)).

   File → Data File Quantities...

Figure 3.21: The Data File Quantities Dialog Box



a. View the **Standard Quantities** list to see what will be saved in the data file by default. Note that you cannot deselect any of the **Standard Quantities**.

b. Select the additional quantities you want saved from the **Additional Quantities** selection list.

c. Click **OK**.

4. Save the legacy case file. Note that the legacy data file quantities specified in the previous step will be saved as part of the case file.

5. Run the calculation and save the legacy data file. This can be done as separate steps, or as one step if you have selected the automatic saving of data files via the Calculation Activities Task Page (p. 5188).

The **Data File Quantities** dialog box can also be opened by clicking the **Data File Quantities...** button in the **Autosave Case/Data** dialog box.

3.25. The .fluent File

When starting up, Ansys Fluent looks in your home folder for an optional file called `.fluent`. If it finds the file, it loads it with the Scheme `load` function. This file can contain Scheme functions that customize the code's operation.

The `.fluent` file can also contain TUI commands that are executed via the Scheme function `ti-menu-load-string`. For example, if the `.fluent` file contains

```
(ti-menu-load-string "file read-case test.cas.h5")
```

then Ansys Fluent will read in the case file `test.cas.h5`. For more details about the function `ti-menu-load-string`, see Text Menu Input from Character Strings.

Important:

Another optional file, `.tgrid`, if present, is also loaded at start up. This file may contain Scheme functions that customize the operation of the code in meshing mode. When both the `.fluent` and `.tgrid` files are present, the `.tgrid` file will be loaded first, followed by the `.fluent` file, when the solution mode is launched. Hence, the functions in the `.fluent` file will take precedence over those in the `.tgrid` file for the solution mode.

The `.fluent` file is not loaded automatically when switching to solution mode from meshing mode. You will need to load the file separately using the Scheme load function, if needed.

3.26. Coupled Simulations in Ansys Fluent with Functional Mock-up Unit (FMU) Files

Ansys Fluent can run co-simulations with Functional Mock-up Unit (FMU) files using the Functional Mock-up Interface technology (FMI). For more information about the FMI standard, see http://fmi-standard.org/. Co-simulation type FMU contains the model and the solver. When FMU is coupled to

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Fluent, the two simulation participants run independently and only exchange data. Ansys Fluent supports both the import and the use of FMU files.

Note:

There are two types of FMU in general: co-simulation and model exchange. Ansys Fluent only supports co-simulation FMU.

The following limitations exist when co-simulation type FMU in Ansys Fluent:

- Only co-simulation type FMU using FMI 2.0 is supported.
- Co-simulations with FMU are available only with the pressure-based solver.
- Only one FMU file can be imported into Ansys Fluent.
- A simulation with FMU can be restarted only if the FMU file supports restarting. (For information on how to determine whether restarts are supported, see this document).
- For co-simulations with FMU files that support restarting, only case files that have been saved in the default format (***.h5**) can be restarted.

To import an FMU file and set up a coupled simulation in Ansys Fluent:

1. Read a case or mesh file into Ansys Fluent.
2. Select the **File/Import/FMU...** menu to import an FMU file.
3. In the **Import FMU File** dialog box, click **Import...**.
4. In the **Select File** dialog box that opens, select the FMU file (`*.fmu`) to be imported. Once you click **OK**, the FMU file is imported into Ansys Fluent.

   During the import process, Ansys Fluent automatically recognizes the FMU output variables, creates corresponding input parameters, and links them to the FMU output parameters. If the imported FMU does not support restarts, a warning message will be displayed.

   Once the import operation is complete, the **Import FMU File** dialog box displays the following Fluent parameters and FMU variables (see Figure 3.22:**Import FMU File** Dialog Box (p. 1030)):

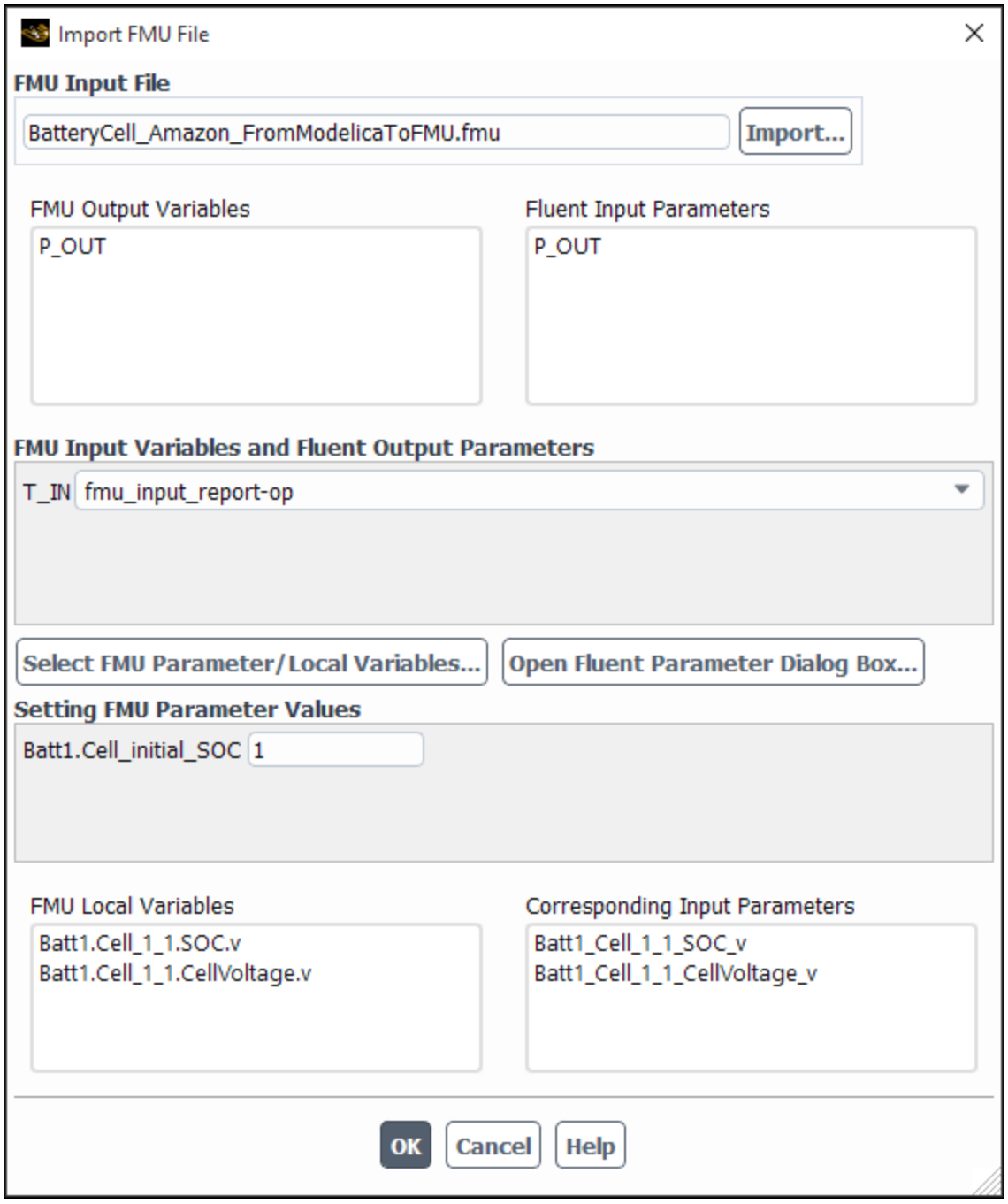
   - FMU output variables and the corresponding Fluent input parameters.

     Fluent input parameters are generated automatically and always have the same names as FMU output variables. The **FMU Output Variables** and **Fluent Input Parameters** lists are for information only. You do not need to do anything else.

   - FMU input variables and Fluent output parameters that have been defined in your simulation. All output parameters that have been defined in Ansys Fluent appear in the drop-down lists next to the FMU input variables.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 3.22: Import FMU File Dialog Box



5. For each FMU input variable, select an appropriate Fluent output parameter from the corresponding drop-down list.

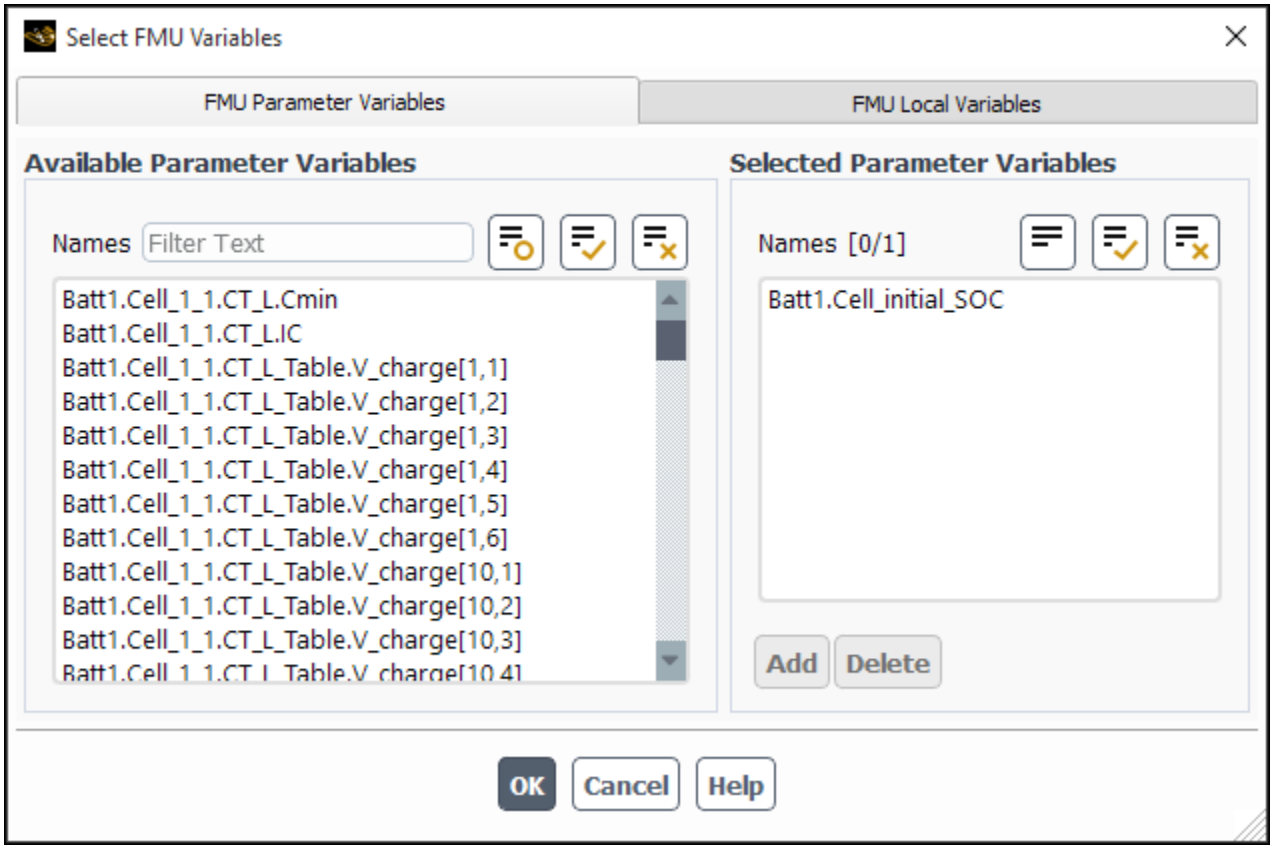
   Right after the FMU file has been imported, FMU input variables are not yet defined. You must hook them to the Fluent output parameters. If Fluent output parameters are not yet defined in your case, you can define them as described in Creating Output Parameters (p. 4133), and then revisit the **Import FMU File** dialog box and complete this step later on.

6. (optional) If you want to modify values of one or more FMU parameter variables, select them in the **Select FMU Variables** dialog box as follows:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. Click the **Select FMU Parameters and Local Variables** button.

Figure 3.23: Import FMU File Dialog Box



b. In the **FMU Parameter Variables** tab, use the **Add** and **Delete** buttons to modify a list of the FMU parameter variables:

- To add a parameter variable to the **Selected Parameter Variables** list, select it in the **Available Parameter Variables** multiple-selection list variables (use **Ctrl** or **Shift** to select multiple variables) and click **Add**.
- To delete a parameter variable from the **Selected Parameter Variables** list and move it back to the **Available Parameter Variables** multiple-selection list, select it and click **Delete**.

c. Click **OK** to close the **Select FMU Variables** dialog box.

The selected FMU parameter variables and the corresponding data entry boxes will appear under **Setting FMU Parameters Values** in the **Import FMU File** dialog box.

d. Adjust the values of the FMU parameters as desired.

Note:

- If you want to conduct parametric studies with different combinations of FMU parameters to evaluate their effects on the model behavior, you need to revisit the **Import FMU File** dialog box and edit the FMU parameter settings as appropriate for each run.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To review the input and output parameters defined in your case, click the **Open Fluent Parameter Dialog Box** button.

7. (optional) Select the FMU local variables you want to monitor using the **Select FMU Variables** dialog box in a manner similar to that for selecting FMU parameter variables:

   a. Click the **Select FMU Parameters and Local Variables** button.

   b. In the **FMU Local Variables** tab, use the **Add** and **Delete** buttons to modify a list of the FMU local variables.

   c. Click **OK** to close the **Select FMU Variables** dialog box.

      The selected parameters are listed in the **FMU Local Variables** list. Ansys Fluent automatically creates corresponding input parameters with matching names and displays them in the **Corresponding Input Parameters** list. You can then use the Ansys Fluent reporting functionality to create report definitions for these parameters (see Expression Report Definition Dialog Box (p. 5469) for details).

8. Click **OK** to close the **Import FMU File** dialog box.

   Ansys Fluent links the FMU input variables and the selected Fluent output parameters and establishes the data exchange connection between FMU and Fluent. The newly created input parameters will appear in the tree under **Parameters & Customization/Parameters/Input Parameters** branch and in the **Parameters** dialog box.

9. Use the new Fluent input parameters that are linked to the FMU output variables in the usual way.

   For example, if the FMU output variable is a heat source term, you can hook this source term to proper cell zones in the energy equation. (Note that the units of the input parameter and the quantity for which you want to use it must be consistent. In case of inconsistent units, you can easily create additional input parameters to either scale the units or to use the input parameter in an expression. See Fluent Expressions Language (p. 1037) for more details about using expressions.)

10. Initialize your solution. During the solution initialization, Ansys Fluent creates an instance of FMU and initializes it.

11. Run the calculation.

    During the calculation, FMU is called at the first iteration in an unsteady simulation or every iteration in a steady simulation.

To run your simulation from an existing case file that has already imported FMU.

1. Load the case file. Make sure that the FMU file is in the same folder as the case file or saved in the absolute path you specified when setting up the case.

2. If FMU does not support restarts, re-initialize the solution and run the calculation from beginning.

3. If FMU supports restarts, load the data file and continue the simulation.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 4: Unit Systems

This chapter describes the units used in Ansys Fluent and how you can control them. Information is organized into the following sections:

4.1. Restrictions on Units

4.2. Units in Mesh Files

4.3. Built-In Unit Systems in Ansys Fluent

4.4. Customizing Units

Ansys Fluent enables you to work in any unit system, including inconsistent units. Thus, for example, you may work in British units with heat input in Watts or you may work in SI units with length defined in inches. This is accomplished by providing Ansys Fluent with a correct set of conversion factors between the units you want to use and the standard SI unit system that is used internally by the solver. Ansys Fluent uses these conversion factors for input and output, internally storing all parameters and performing all calculations in SI units. Both solvers always prompt you for the units required for all dimensional inputs.

Units can be altered part-way through a problem setup and/or after you have completed your calculation. If you have entered some parameters in SI units and then you switch to British, all of your previous inputs (and the default prompts) are converted to the new unit system. If you have completed a simulation in SI units but you would like to report the results in any other units, you can alter the unit system and Ansys Fluent will convert all of the problem data to the new unit system when results are displayed. As noted above, all problem inputs and results are stored in SI units internally. This means that the parameters stored in the case and data files are in SI units. Ansys Fluent simply converts these values to your unit system at the interface level.

4.1. Restrictions on Units

It is important to note that the units for some inputs in Ansys Fluent are different from the units used for the rest of the problem setup.

- You must always define the following in SI units, regardless of the unit system you are using:
  - Boundary profiles (see Profiles (p. 1564))
  - Source terms (see Defining Mass, Momentum, Energy, and Other Sources (p. 1382))
  - Custom field functions (see Custom Field Functions (p. 4285))
  - Data in externally-created XY plot files (see Creating an XY Plot From Multiple Data Sources (Including Files) (p. 4064))
  - User-defined functions (See the Fluent Customization Manual for details about user-defined functions.)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you define a material property by specifying a temperature-dependent polynomial or piecewise-polynomial function, remember that temperature in the function is always in units of Kelvin or Rankine. If you are using Celsius or Kelvin as your temperature unit, then polynomial coefficient values must be entered in terms of Kelvin; if you are using Fahrenheit or Rankine as the temperature unit, values must be entered in terms of Rankine. See Defining Properties Using Temperature-Dependent Functions (p. 1616) for information about temperature-dependent material properties.

4.2. Units in Mesh Files

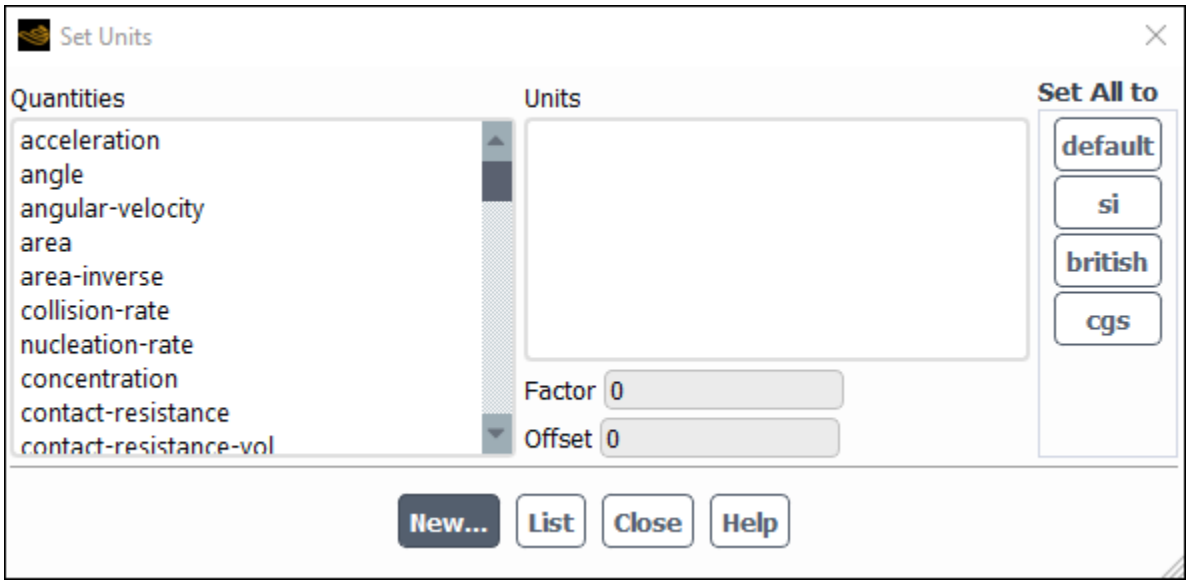
Some mesh generators allow you to define a set of units for the mesh dimensions. However, when you read the mesh into Ansys Fluent, it is always assumed that the unit of length is meters. If this is not true, you must scale the mesh, as described in Scaling the Mesh (p. 1285).

4.3. Built-In Unit Systems in Ansys Fluent

Ansys Fluent provides four built-in unit systems: British, SI, CGS, and "default", all of which can be selected in the **Set Units** dialog box (Figure 4.1: The Set Units Dialog Box (p. 1034)), using the buttons under the **Set All to** heading. To display the **Set Units** dialog box, right-click **General** and click **Units...**, under **Setup** in the tree.

Setup → General → Units

Figure 4.1: The Set Units Dialog Box



To choose the English Engineering standard for all units, click the **british** button; to select the International System of units (SI) standard for all units, click the **si** button; to choose the CGS (centimeter-gram-second) standard for all units, click the **cgs** button; and to return to the "default" system, click the **default** button. The default system of units is similar to the SI system, but uses degrees instead of radians for angles. Clicking on one of the buttons under **Set All to** will immediately change the unit system. You can then close the dialog box if you are not interested in customizing any units.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Changing the unit system in the **Set Units** dialog box causes all future inputs that have units to be based on the newly selected unit system.

4.4. Customizing Units

If you would like a mixed unit system, or any unit system different from the four supplied by Ansys Fluent (and described in Built-In Unit Systems in Ansys Fluent (p. 1034)), you can use the **Set Units** dialog box (Figure 4.1: The Set Units Dialog Box (p. 1034)) to select an available unit or specify your own unit name and conversion factor for each quantity.

For additional information, see the following sections:

4.4.1. Listing Current Units

4.4.2. Changing the Units for a Quantity

4.4.3. Defining a New Unit

4.4.1. Listing Current Units

Before customizing units for one or more quantities, you may want to list the current units. You can do this by clicking the **List** button at the bottom of the **Set Units** dialog box. Ansys Fluent will print out a list (in the text window) containing all quantities and their current units, conversion factors, and offsets.

4.4.2. Changing the Units for a Quantity

Ansys Fluent will enable you to modify the units for individual quantities. This is useful for problems in which you want to use one of the built-in unit systems, but you want to change the units for one quantity (or for a few). For example, you may want to use SI units for your problem, but the dimensions of the geometry are given in inches. You can select the SI unit system, and then change the unit of length from meters to inches.

To change the units for a particular quantity, you will follow these two steps:

1. Select the quantity in the **Quantities** list (they are arranged in alphabetical order).

2. Choose a new unit from those that are available in the **Units** list.

For the example cited above, you would choose **length** in the **Quantities** list, and then select **in** in the **Units** list. The **Factor** will automatically be updated to show 0.0254 meters/inch. (See Figure 4.1: The Set Units Dialog Box (p. 1034).) If there was a nonzero offset for the new unit, the **Offset** field would also be updated. For example, if you were using SI units but wanted to define temperature in Celsius instead of Kelvin, you would select **temperature** in the **Quantities** list and **c** in the **Units** list. The **Factor** would change to 1, and the **Offset** would change to 273.15. Once you have selected the quantity and the new unit, no further action is needed, unless you want to change the units for another quantity by following the same procedure.

4.4.3. Defining a New Unit

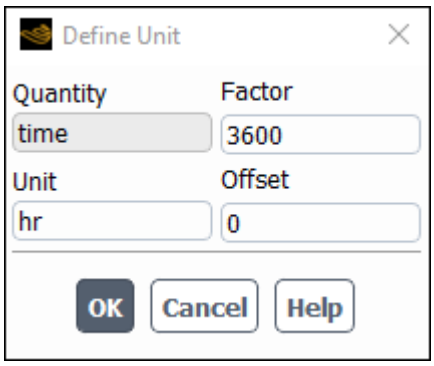
To create a new unit to be used for a particular quantity, you will follow the procedure below:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the **Set Units** dialog box, select the quantity in the **Quantities** list.

2. Click the **New...** button and the **Define Unit** dialog box (Figure 4.2: The Define Unit Dialog Box (p. 1036)) will open. In this dialog box, the selected quantity will be shown in the **Quantity** field.

   Figure 4.2: The Define Unit Dialog Box



3. Enter the name of your new unit in the **Unit** field, the conversion factor in the **Factor** field, and the offset in the **Offset** field.

4. Click **OK** in the **Define Unit** dialog box, and the new unit will appear in the **Set Units** dialog box.

For example, if you want to use hours as the unit of time, select **time** in the **Quantities** list in the **Set Units** dialog box and click the **New...** button. In the resulting **Define Unit** dialog box, enter `hr` for the **Unit** and `3600` for the **Factor**, as in Figure 4.2: The Define Unit Dialog Box (p. 1036). Then click **OK**. The new unit **hr** will appear in the **Units** list in the **Set Units** dialog box, and it will be selected.

4.4.3.1. Determining the Conversion Factor

The conversion factor you specify (**Factor** in the **Define Unit** dialog box) tells Ansys Fluent the number to multiply by to obtain the SI unit value from your customized unit value. Thus the conversion factor should have the form SI units/custom units. For example, if you want the unit of length to be inches, you should enter a conversion factor of 0.0254 meters/inch. If you want the unit of velocity to be feet/min, you can determine the conversion factor by using the following equation:

$$x\frac{\text{ft}}{\text{min}} \times \frac{0.3048\text{m}}{\text{ft}} \times \frac{\text{min}}{60\text{s}} = y\frac{\text{m}}{\text{s}} \tag{4.1}$$

You should enter a conversion factor of 0.0051, which is equal to 0.3048/60.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 5: Fluent Expressions Language

5.1. Introduction to Expressions

The Fluent Expression Language is an interpreted, declarative language based on Python, that enables you to:

- Specify complex boundary conditions and source terms with respect to time, iteration number, position, and solution variables.
- Specify various model and solver settings in terms of time or iteration.

Important:

Expression values are updated at the same frequency as the **Profile Update Interval** (specified in the **Run Calculation** task page).

5.1.1. Expression Syntax

An expression is a string representing a combination of values, variables, operators, and function calls that returns a value when evaluated with appropriate values for the variables.

For example: `Vmax*(5.0*exp(-t-0.3 [s]/2.8 [s]))`

The following sections cover the basic elements of proper expression syntax:

5.1.1.1. Expression Data Types

5.1.1.2. Expression Values

5.1.1.3. Expression Operations and Functions

5.1.1.1. Expression Data Types

The evaluated result of an expression can be a real number, a boolean, a real field or a boolean field. For example, `2*StaticPressure` evaluates to a real field when computed on a zone. Whereas, `average(2*StaticPressure, ["inlet"])` evaluates to a single real values.

Note:

Cell registers evaluate as boolean.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.1.1.2. Expression Values

Values can be real numbers (for example, `1.0e-3`), integers (for example, -10, 5, 37), booleans (true or false), or quantities. Quantities are real numbers with units associated. The syntax supported for quantities is `<number> [ <unit> ]`, for example, `2324.0 [Pa kg^-3 s]`. The unit specification is based on CFX (Units Syntax in the *CFX-Pre User's Guide*).

5.1.1.3. Expression Operations and Functions

All of the following mathematical functions take inputs in the form of an expression that evaluates to a real number or a real field and return a real number or real field:

Table 5.1: Operations and Functions

Description	Function
Operators	+, -, *, /, ** (power), ^ (power), >, >=, <, <=, ==, !=

Conditional	`AND(<expr>, <expr>, …)`
	`IF(<cond>, <true_value>, <false_value>)`
	`NOT(<expr>)`
	`OR(<expr>, <expr>, …)`
	`XOR(<expr>)`

Flux	`FilmHeatTransferRate(['location', …],phase = <phase>)`
	`FilmMassFlowRate(['location', …],phase = <phase>)`
	`HeatTransferRate(['location', …],phase = <phase>)`
	`MassFlow(['location', …],phase = <phase>)`
	`RadiationHeatTransferRate(['location', …],phase = <phase>)`
	`SensibleHeatTransferRate(['location', …],phase = <phase>)`

Hyperbolic	`acosh(<expr>)`
	`asinh(<expr>)`
	`atanh(<expr>)`
	`cosh(<expr>)`
	`sinh(<expr>)`
	`tanh(<expr>)`

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Function

Mathematical	`abs(<expr>)`
	`*besselJ(n**, <expr>)`
	`*besselY(n**, <expr>)`
	`ceil(<expr>)`
	`exp(<expr>)`
	`floor(<expr>)`
	`gradient(<expr>)`
	`log(<expr>)`
	`log10(<expr>)`
	`max(<expr>, <expr>, …)`
	`min(<expr>,<expr>, …)`
	`mod(<expr>, <expr>)`
	`normalize(<expr>,['location',…])`
	`round(<expr>)`
	`sqrt(<expr>)`
	`step(<expr>)`
	`trunc(<expr>)`
	*`besselJ` is a Bessel function of the first kind and `besselY` is a Bessel function of the second kind.
	**n must be a constant of type 'double'.

Reduction	`Area(['location', 'location', …])`
	`AreaAve(Field, ['location'])`
	`AreaInt(Field, ['location'])`
	`*Average(<expr>, ['location', 'location', …], Weight= <None| 'Area'| 'Volume'| 'Mass'| 'MassFlowRate'| 'AbsMassFlowRate'>)`
	`Centroid(['location'])`
	`Count(['location'])`
	`CountIf(Boolean_Expr, ['location'])`

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Function
	Force (['location'], …) MassAve(Field,['location']) MassFlowAve(Field, ['location']) MassFlowAveAbs(Field, ['location']) MassFlowInt(Field, ['inelt1'], ['inlet2']) MassInt(Field, ['location']) Maximum(<expr>, ['location', 'location', …]) Minimum(<expr>, ['location', 'location', …]) Moment(<point>, ['location']) PressureForce(['location'], …) *Sum(<expr>, ['location', 'location', …], Weight= <None| 'Area'| 'Volume'| 'Mass'| 'MassFlowRate'| 'AbsMassFlowRate'>) SumIf(Field, Boolean_Expr, ['location'], Weight=[Weight]) ViscousForce([<location>, <location>, …]) Volume([<location>, <location>, …]) VolumeAve(Field, ['location']) VolumeInt(Field, ['location'])
Trigonometric	acos(<expr>) asin(<expr>) atan(<expr>) atan2(<expr>, <expr>) cos(<expr>) sin(<expr>) tan(<expr>)
Vector	cross(<expr>, <expr>) dot <vector1>, <vector2>)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Function

	`unitVector(<x>, <y>, <z>)`
	
	`<vector>.dir`
	
	`vector(<x>, <y>, <z>, ["unit"])`

*For mass-weighted reduction operations (`Sum` and `Average`), weight. `MassFlowRate` is equivalent to a mass-weighted surface integral (`Sum`) or surface average (`Average`), `AbsMassFlowRate` is equivalent to the absolute value mass-weighted surface integral (`Sum`) or surface average (`Average`) , and `Mass` is equal to a mass-weighted volume integral.

Operations

Components of vectors can be accessed with the `.x`, `.y`, and the `.z` suffixes; magnitude can be accessed with the `.mag` suffix.

Important:

Do not use multiple comparison operators within a single expression, as the operation will not work as intended. For example, `300[K]< StaticTemperature < 400[K]` will not work. To accomplish this expression, use `AND(StaticTemperature>300[K], StaticTemperature<400[K])`.

IF Statements

The `IF(<cond>, <true_value>, <false_value>)` function returns the `<true_value>` if `<cond>` is true, else it returns the `<false_value>`. Any of the arguments can be fields. The type and unit dimension of `<true_value>` and `<false_value>` must be the same.

OR Statements

`OR(<cond1>, <cond2>)` returns true if any of the conditions (any of which can be a boolean field) returns true.

AND Statements

`AND(<cond1>, <cond2>)` returns true if all of the conditions (any of which can be a boolean field) returns true.

Flux Operations

The following flux operations are explained in greater detail:

- **FilmHeatTransferRate([<location>, ...],phase = <phase>)**—compute the heat transfer rate of the film at the specified location for the specified phase. *Note that you do not need to provide a phase for single-phase flow.*
- **FilmMassFlowRate([<location>, ...],phase = <phase>)**—compute the mass flow rate of the film at the specified location for the specified phase. *Note that you do not need to provide a phase for single-phase flow.*

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **HeatTransferRate([<location>, ...],phase = <phase>)**—compute the heat transfer rate at the specified location for the specified phase. *Note that you do not need to provide a phase for single-phase flow.*
- **MassFlow([<location>, ...],phase = <phase>)**—compute the mass flow rate at the specified location for the specified phase. *Note that you do not need to provide a phase for single-phase flow.*
- **RadiationHeatTransferRate([<location>, ...],phase = <phase>)**—compute the radiation heat transfer rate at the specified location for the specified phase. *Note that you do not need to provide a phase for single-phase flow.*
- **SensibleHeatTransferRate([<location>, ...],phase = <phase>)**—compute the sensible heat transfer rate at the specified location for the specified phase. *Note that you do not need to provide a phase for single-phase flow.*

Important:

Fluxes are evaluated on boundaries and face zones, but not on user-defined post-processing surfaces (such as planes and iso-surfaces). *This input requirement is similar to flux reports.*

Mathematical Operations

The following mathematical operations are explained in greater detail:

- **gradient(<expr>)**—compute the spatial gradient of an expression. For example: `Average(gradient(StaticPressure).mag,['wall-top','wall-bottom'])`.

  The **gradient()** function computes the gradient of the expression using the currently selected gradient method (see Choosing the Spatial Discretization Scheme (p. 3575) for additional information). In some cases, the values obtained may not match the solver gradients for the same variable, as special techniques are employed when computing solver gradients to improve solver robustness. These techniques are not applied when computing the gradients of expressions.

- **normalize(<expr>, [<location>,...])**—normalize the value of an expression by scaling according to the maximum value of the expression at the specified location(s), which is equivalent to: `(<expr> - Minimum(<expr>, [<location>]))/(Maximum(<expr>, [<locations>]) - Minimum(<expr>, [<location>]))`.

- **Step (<expr>)**

  Step functions work as follows (`x` must be a non-dimensional expression of type real):

  If `step(x)` < `0.0` return `0.0`

  If `step(x)` == `0.0` return `0.5`

  If `step(x)` > `0.0` return `1.0`

Reduction Operations

The following reduction operations provide simpler access to more complex operations:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **AreaAve(Field, ['location'])**—calculates the area averaged value for the specified field variable and location. This reduction operation is equivalent to `Average(Field, ['location'], Weight='Area')`.
- **AreaInt(Field, ['location'])**—calculates the area integrated averaged value for the specified field variable and location. This reduction operation is equivalent to `Average(Field, ['location', Weight], Weight='Area')`.
- **Centroid(['location'])**—calculates the geometric centroid of the specified location(s) as a vector.
- **Count(['location'])**—calculates the number of *solve* cells included in the specified location. This reduction operation is equivalent to `Sum(1, ['location'])`.
- **CountIf(Boolean_Expr, ['location'])**—calculates the number of *solve* cells in the specified location according to the provided boolean expression. This reduction operation is equivalent to Sum(IF(Boolean_Expr, 1, 0),['location']).
- **Force(['location'])**—calculates the force on the location(s) specified (should be walls) as a vector. *The expression may include porous walls*.
- **MassAve(Field,['location'])**—calculates the mass-weighted average value for the specified field variable and location. This reduction operation is equivalent to `Average(Field, ['location'], Weight='Mass')`.
- **MassFlowAve(Field, ['location'])**—calculates the mass-flow-weighted average value for the specified field variable and location. This reduction operation is equivalent to `Average(Field, ['location'], Weight='MassFlowRate')`.
- **MassFlowAveAbs(Field, ['location'])**—calculates the absolute value of the mass-flow-weighted average for the specified field variable and location. This reduction operation is equivalent to `Average(Field, ['location'], Weight='AbsMassFlowRate')`.
- **MassFlowInt(Field, ['inelt1'], ['inlet2'])**—calculates the total mass flow across the specified inlets. This reduction operation is equivalent to `Sum(Field, ['inlet1', 'inlet2'], Weight='MassFlowRate')`.
- **MassInt(Field, ['location'])**—computes the total mass-weighted value for the specified field and location. This reduction operation is equivalent to `Sum(Field, ['location'], Weight='Mass')`.
- **Maximum(Field, ['location'])**—computes maximum value for the specified field and location.
- **Minimum(Field, ['location'])**—computes minimum value for the specified field and location.
- **Moment([[<PointExpression>], 'location'])**—calculates the moment vector about the specified point (which can be single-valued expression) for the given location(s).
- **PressureForce(['location'])**—calculates the pressure force on the location(s) specified (should be walls) as a vector.
- **Sum(Field, ['location'], Weight=[Weight])**—computes the sum of the specified field variable at the specified location.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **SumIf(Field, Boolean_Expr, ['location'], Weight=[Weight])**—computes the sum of the specified field variable at the specified location according to the provided boolean operation. This reduction operation is equivalent to `Sum(IF(Boolean_Expr, Field, 0), ['location'], Weight=[Weight])`.
- **ViscousForce(['location'])**—calculates the viscous force on the location(s) specified (should be walls) as a vector.
- **VolumeAve(Field, ['location'])**—calculates the volume-weighted average value for the specified field variable and location. This reduction operation is equivalent to `Average(Field, ['location'], Weight='Volume')`.
- **VolumeInt(Field, ['location'])**—calculates the volume-weighted total for the specified value and location. This reduction operation is equivalent to `Sum(Field, ['location'], Weight='Volume')`.

Vector Operations

- **cross(<expr>, location)**—compute the cross product of an expression and a location. For example: `cross(Force(['wall-top', 'wall-bottom']),Centroid(['wall-top', 'wall-bottom'])).mag`.
- **dot(<v1>, <v2>)**—calculates the dot product of vectors.
- **<v>.dir**—calculates a unit vector along the direction of <v>, which is equivalent to <v>/<v>.mag.
- **vector (<Xexpression>, <Yexpression>, <Zexpression>)**—calculates a vector from scalar component expressions. The components should evaluate to the same units, but they can be a mix of field and single-valued vector expressions. For example, `vector(-y, x, 0[m])`.
- **vector (<X>, <Y>, <Z>, <unit>)**—calculates a vector in the specified units from constant values. For example, `vector(0.4, 0.5, 0.2,'cm')`.
- **unitVector (<X>, <Y>, <Z>)**—calculates a unit vector from constant values.

5.1.2. Units Validation

Expressions are validated to ensure they have consistent unit types. For example, Fluent flags `1 [cm] + TotalPressure` because centimeters are not units of pressure. Note that units can still be provided in different unit systems, for example, `1 [atm] + 200 [Pa]`.

Important:

- Even though radians is a dimensionless unit for measuring angles, it is still considered in expression units validation. For example, $1[\mathrm{m\ rad}]+1[\mathrm{m}]$ is not allowed. You must either divide the first term by $1[\mathrm{rad}]$ or multiply the second term by $1[\mathrm{rad}]$ to make the units consistent.
- Consider using absolute units when specifying temperature (either Kelvin or Rankine). *If an expression contains multiple instances of a temperature expressed in Celsius or*

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fahrenheit, Fluent will convert each value separately into absolute units, which could lead to unexpected results.

5.2. Expression Sources

Expression sources are the variables that can be used in expressions.

5.2.1. Field Variables
5.2.2. Solution Variables
5.2.3. Scientific Constants
5.2.4. Aliases
5.2.5. Profiles

5.2.1. Field Variables

A subset of the Fluent postprocessing field variables are available for use in expressions. See Appendix: Supported Field Variables (p. 1078) for a listing of all the available field variables.

Note:

Some variables require additional context, which is specified in the following format: `Variable(<context>="<context value>")`. For example, to get the species mass fraction of carbon dioxide: `MassFraction(species="co2")`.

In multiphase simulations, the phase context must be specified. For example: `MassFraction(species="co2", phase="air")`.

The context for variables requiring it is provided in the **Variables (type/context)** column of Appendix: Supported Field Variables (p. 1078).

5.2.2. Solution Variables

The following solution variables are available:

Table 5.2: Solution Variables

Variable	Description
`Time`	Current time* *In steady state, time evaluates as 0, unless the case was run in transient, then switched to steady state, in which case it will evaluate as the

Variable	Description
	latest time from the transient run.
Timestep	Current time step number* *In steady state, timestep evaluates as 0, unless the case was run in transient, then switched to steady state, in which case it will evaluate as the latest timestep from the transient run.
DeltaTime	Current time step size* *In steady state, delta time evaluates as 0, unless the case was run in transient, then switched to steady state, in which case it will evaluate as the latest delta time from the transient run.
Iteration	Global iteration count

5.2.3. Scientific Constants

Table 5.3: Scientific Constants

Variable	Description	Value
PI	Pi	3.14159265358979323846
e	e (base of the natural logarithm)	2.71828182845904523536
R	Gas constant	8.314472 [J K^-1 mol^-1]
avogadro	Avogadro's number	6.02214199e23 [mol^-1]
boltzmann	Boltzmann constant	1.3806503 [J K^-1]
clight	Light velocity	2.99792458e8 [m s^-1]
echarge	Electron charge	1.60217653e-19 [A s]
g	Acceleration due to gravity	9.8066502 [m s^-2]
planck	Planck's constant	6.62606876e-34 [J s]
stephan	Stephan-Boltzmann constant	5.670400e-08 [W m^-2 K^-2]
mupermo	Magnetic permeability	4.0*PI*1.0e-7 [N A^-2]
epspermo	Electric constant	1./(clight*clight*mupermo)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.2.4. Aliases

Aliases provide simplified syntax to access frequently used variables.

Table 5.4: Variable Aliases

Alias	Variable
x	Position.x
y	Position.y
z	Position.z
u	Velocity.x
v	Velocity.y
w	Velocity.z
t	Time* *In steady state, time evaluates as 0, unless the case was run in transient, then switched to steady state, in which case it will evaluate as the latest time from the transient run.
dt	DeltaTime* *In steady state, delta time evaluates as 0, unless the case was run in transient, then switched to steady state, in which case it will evaluate as the latest delta time from the transient run.
iter	Global iteration count
T	StaticTemperature
P	StaticPressure
mf	MassFraction
Amag	FaceAreaMagnitude
vol	CellVolume
mass	CellVolume*Density

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.2.5. Profiles

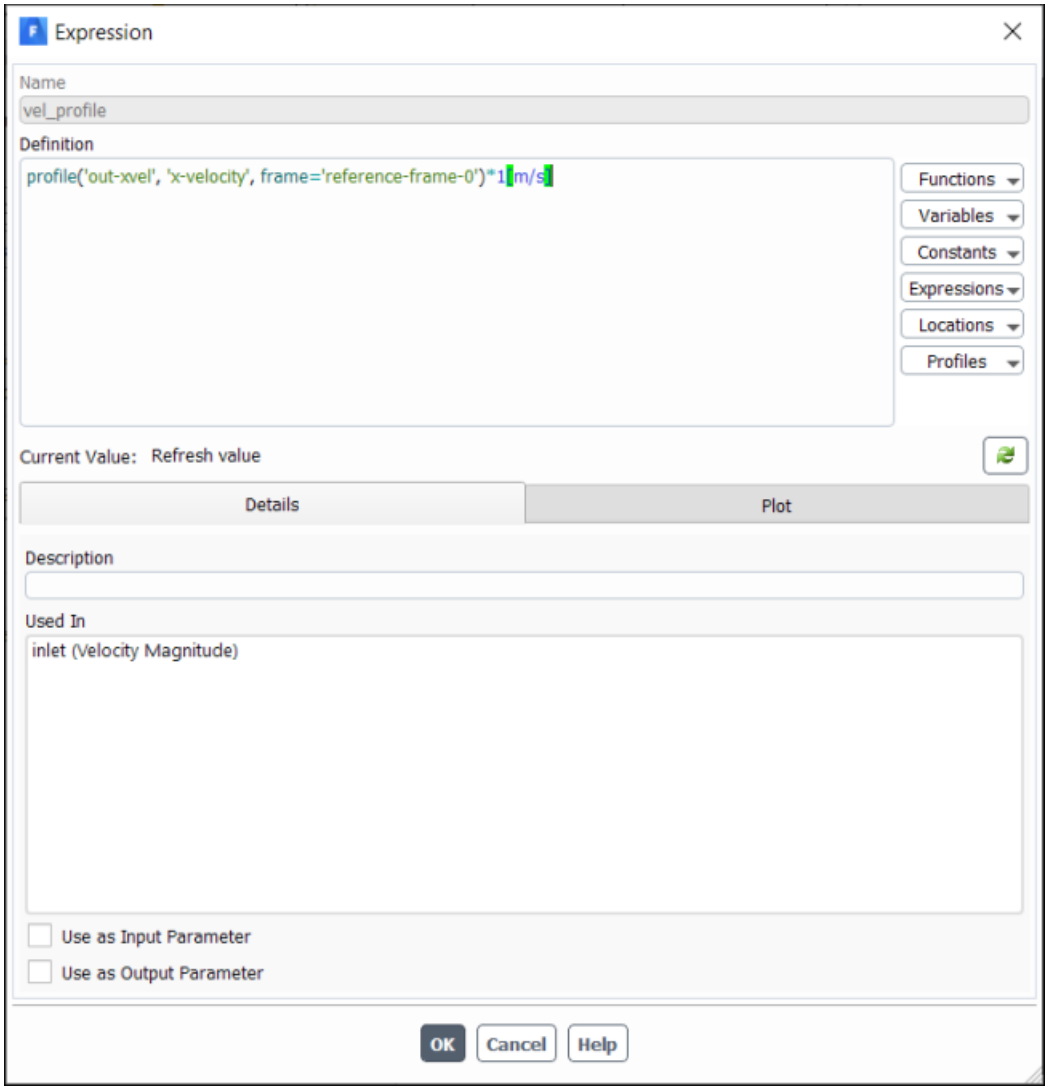
You can use profiles in expressions to specify boundary and cell zone conditions, for postprocessing, and in reduction operations. Profiles are tabular data that can be imported into Fluent, which typically contain X, Y, and Z columns (for spatial profiles) or `Time` (for transient profiles), in addition to one or more dependent variable columns.

Prior to using profiles in an expression you must load one or more profiles. Refer to Reading and Writing Profile Files (p. 964) for additional information.

The profile expression requires the following inputs: **profile('<profile-name>', '<field-name>', frame='<reference-frame-name>')** (the reference frame argument is only for spatial profiles). Note that if specified, profile coordinates are assumed to be of the specified frame and are therefore transformed to global coordinates prior to interpolation (profile coordinates are not transformed if you specify the reference frame as **frame='global'**).

Once the profile expression is defined, you can select it for the desired boundary condition or cell zone condition field. Refer to Expressions for Cell Zones and Boundary Conditions (p. 1050)

Figure 5.1: Example Profile Expression



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.3. Creating and Using Expressions

There are two ways that you can create expressions in Ansys Fluent. The first way is by creating an expression directly in the field where it will be applied. The second is by creating a named expression, which can be reused at multiple locations. These two formats are discussed in the following sections:

5.3.1. Directly Applied Expressions
5.3.2. Named Expressions
5.3.3. Context Specification
5.3.4. Plotting Expressions
5.3.5. Postprocessing Expressions
5.3.6. Expression Manager

Important:

- Be careful to avoid situations where you are creating a circular dependency for a field or property. For example, setting the **Velocity Magnitude** field of a velocity inlet to `Velocity.mag`, without specifying a value for the velocity magnitude elsewhere, results in a 0-value for the velocity magnitude.
- 2D axisymmetric cases—expressions report quantities such as **Area**, on the actual cells/faces or on a per radian basis. Therefore, these quantities must be multiplied by 2π to get the value for the whole domain.
- 2D cases—expressions report quantities such as **Area**, on the actual cells/faces or on a per unit basis. Therefore, these quantities must be multiplied by the reference depth to get the value for the whole domain (otherwise the depth is assumed to be 1).

Note:

For postprocessing, expressions are evaluated using an algorithm that is suitable for a wide range of computations. First cell values are evaluated, then node values are interpolated from the cell values. Custom field functions (Custom Field Functions (p. 4285)) interpolate cell values to nodes prior to performing the computations of the provided definition. The observable difference in these two approaches appears in cases where there are steep gradients in cell values.

Depending on the specific expression, you may notice small differences in postprocessing for node-based variables (such as mesh or structures-based variables) if comparing expression results with directly evaluated postprocessing of the same variables. For example, if you compare a contour plot of **Total Mesh Displacement** with a contour plot of an expression evaluating total mesh displacement, you may see minute differences due to small approximation errors.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.3.1. Directly Applied Expressions

You can use expressions at boundary and cell zone conditions, at certain material properties, and at fields and properties where profiles and parameters can also be defined. Expressions can also be used at most other non-cell zone or boundary condition locations where parameters can be defined.

The following sections deal with defining expressions directly:

5.3.1.1. Expressions for Cell Zones and Boundary Conditions
5.3.1.2. Expressions for Material Properties
5.3.1.3. Expressions in the Console

5.3.1.1. Expressions for Cell Zones and Boundary Conditions

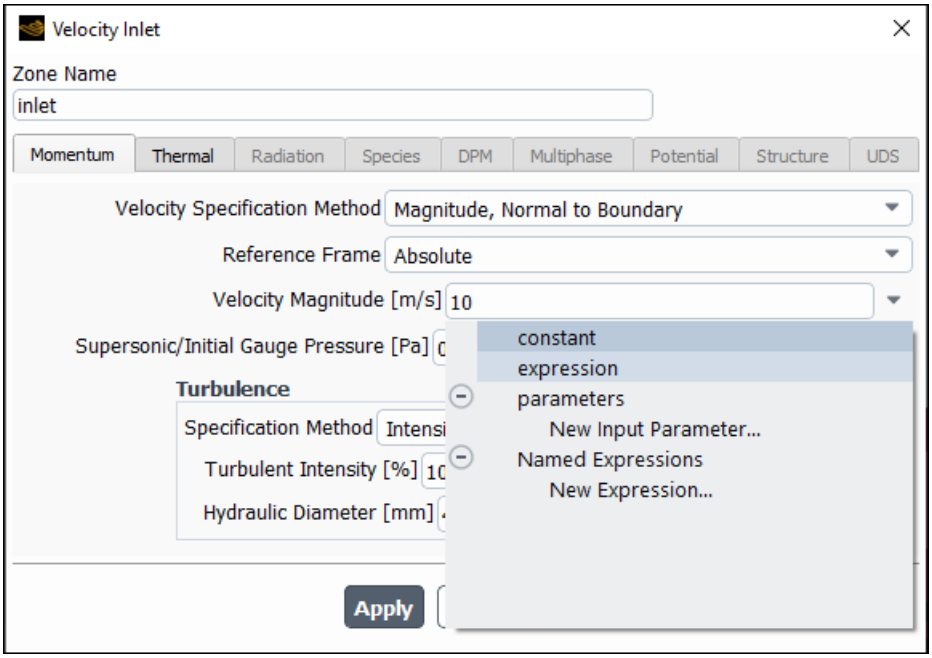
To define an expression for a boundary or cell zone condition:

1. Open the dialog box for the boundary/cell zone where you want to create an expression. *For example, the* **Velocity Inlet** *dialog box.*
2. Click the drop-down arrow to the right of the field that you would like to define using an expression and select **expression**.

Note:

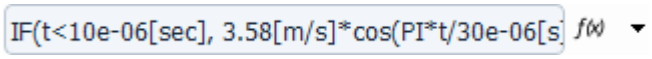
Alternatively, you can:

- Create a new named expression (Named Expressions (p. 1054)).
- Select a valid expression from the list.



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

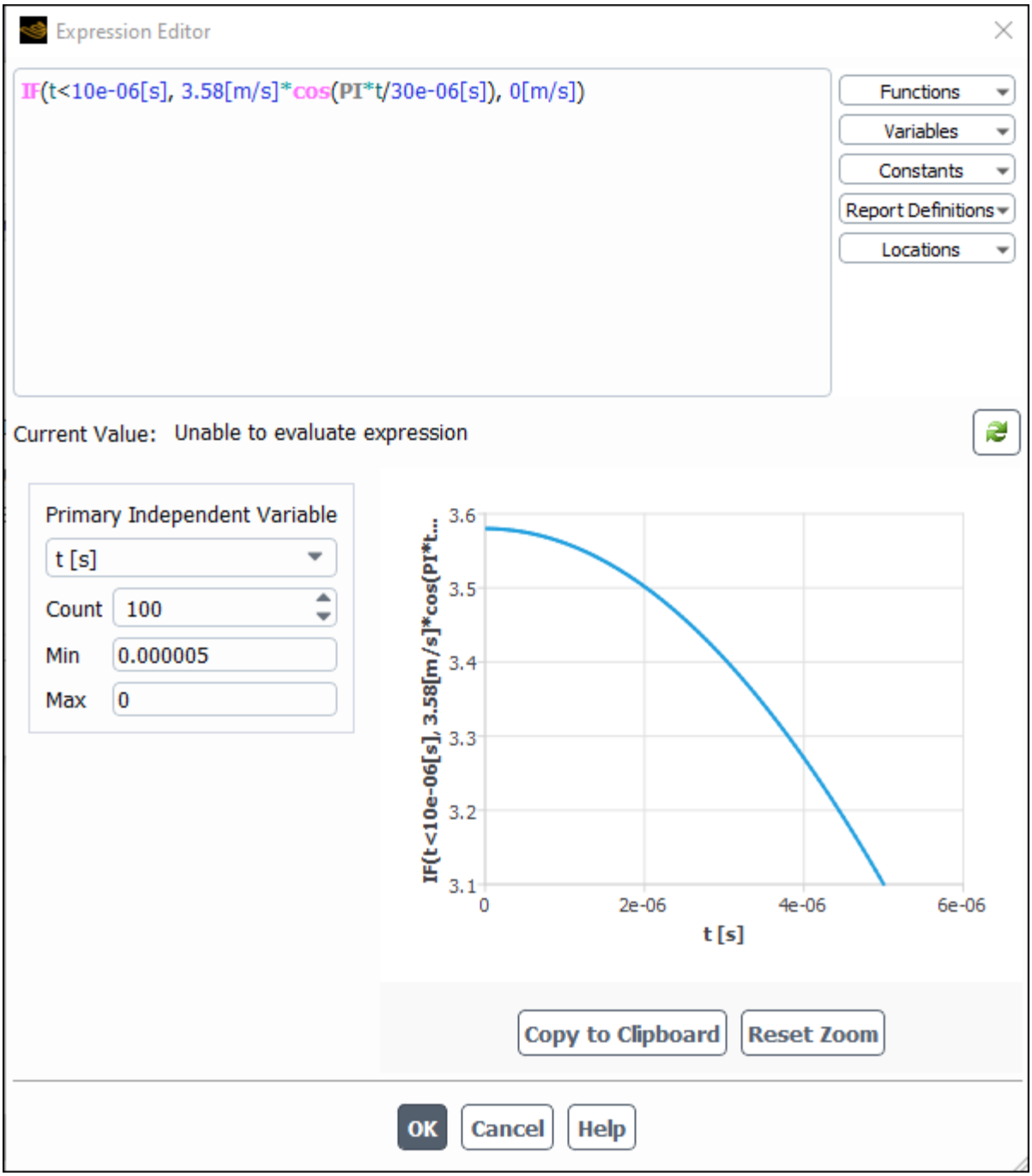
3. You can enter your expression directly in the text field. *Ansys Fluent provides messaging regarding the validity of the expression as soon as you begin entering it.*



Alternatively, you can click $f_{(x)}$ to open a bigger editor (Figure 5.2: The Expression Editor Dialog Box (p. 1052)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.2: The Expression Editor Dialog Box



Important:

Expressions allow for implicit locations. For example, you could specify the mass flux at a mass-flow-inlet boundary as `1 [kg/s]/Area()`. However, if you explicitly provide a location, that will be used (for example, `1 [kg/s]/Area(['outlet'])`.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5.3.1.2. Expressions for Material Properties

You can use expressions to define the following material properties:

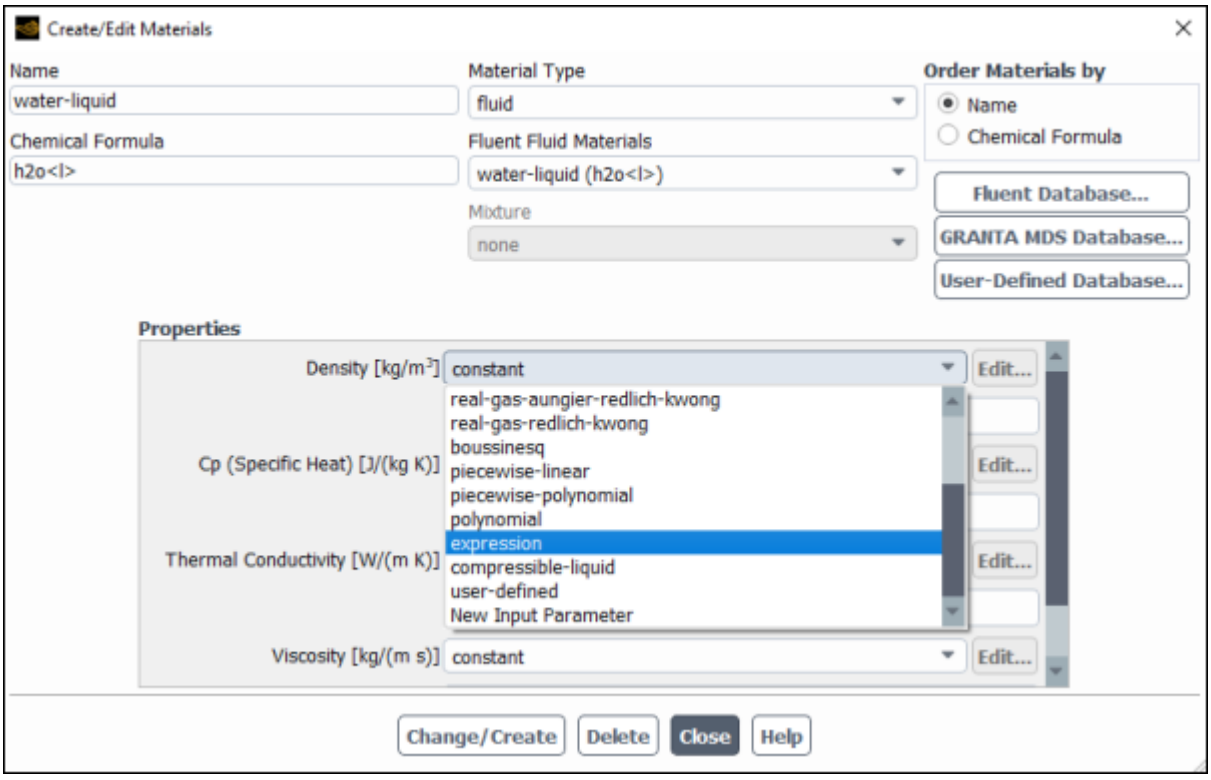
- Density
- Viscosity
- Thermal conductivity
- Absorption coefficient
- Scattering coefficient
- Lithium diffusivity
- Partition coefficient
- UDS diffusivities

To define an expression for a material property:

1. Open the **Create/Edit Materials** dialog box.

   Setup → **Materials** → **Fluid | Solid** → **<material name>** **Edit...**

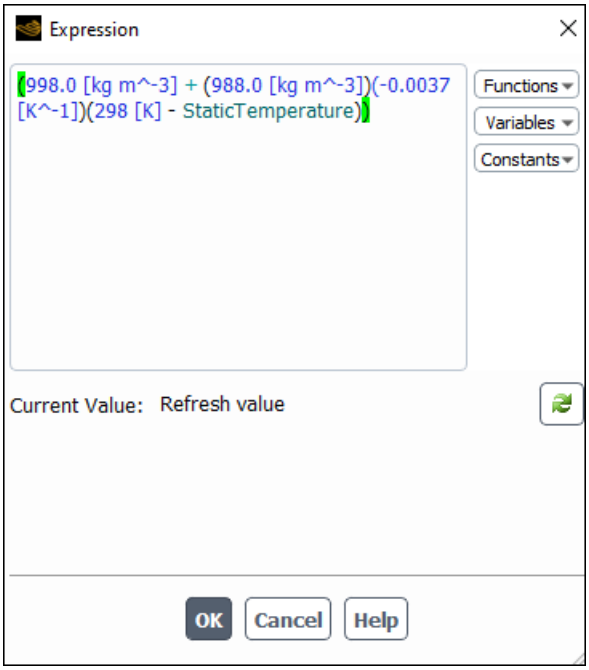
2. Click the drop-down arrow to the right of the property field that you would like to define using an expression and select **expression**.



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Enter the expression in the **Expression** editor dialog box that opens.

Figure 5.3: Example Expression for Water Density



You can use the **Functions**, **Variables**, **Constants**, and **Expressions** drop-down lists when creating your expression. These lists contain the valid expression inputs.

Note:

Density can be a function of static temperature or absolute pressure, but not both. Density specification as a function of absolute pressure is primarily for modeling mild compressibility, such as compressible liquids. If density is specified as a function of absolute pressure, Fluent will automatically compute the speed of sound.

---

4. Click **OK** to save the expression.

5.3.1.3. Expressions in the Console

You can specify expressions for settings through the text user interface (Console). To do so, enter a string instead of a real value to specify a setting as an expression. *For example:*

```
/define/operating-conditions> operating-pressure
operating pressure (atm) [0] "1[atm]*sin(t/0.1[s])"
```

5.3.2. Named Expressions

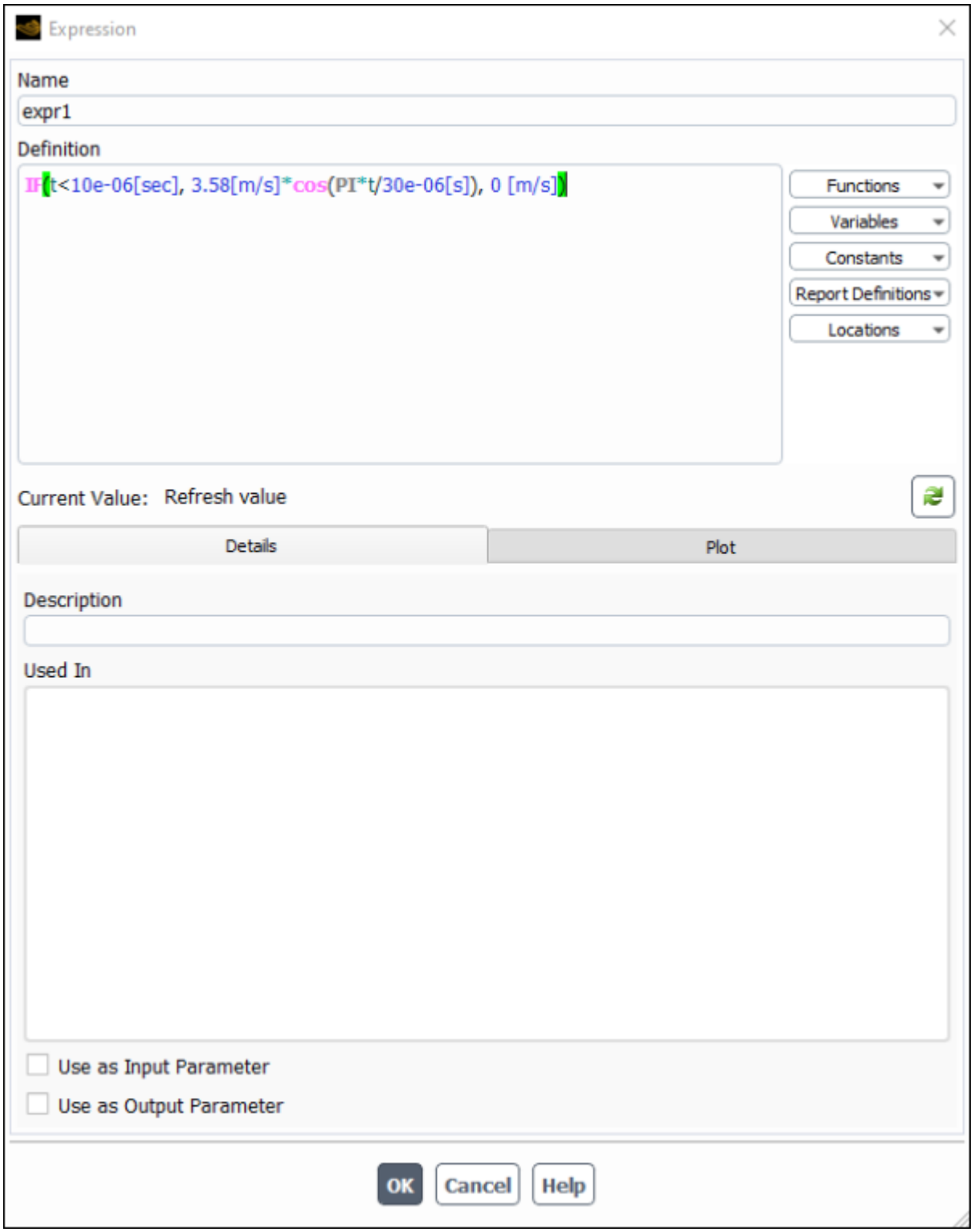
You can create, edit, and delete named expressions through the graphical user interface (GUI) via the right-click context menu in the Outline View tree. *Right-click options are also available for importing/exporting expressions from/to a file, computing expressions, marking them as input/output parameters, and printing expression details to the console.*

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also managed named expression using the **Expression Manager** (Expression Manager (p. 1063)).

Setup → Named Expressions New...

Figure 5.4: The Expression Dialog Box



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Provide a **Name** for the expression. *The name must start with a letter and it may contain numbers and underscores.*

Naming Guidance:

- Expression names must be unique. That is, they cannot use the same name as other expressions, report definitions, or expression constants or variables such as "PI", "avogadro", "T", "vol", "mass", and so on. *Note that expression names cannot contain dashes "-", so if, for example, you have a report definition named* `surface-temperature-1`, *you cannot name your expression* `surface_temperature_1`, *because the expression language automatically converts dashes to underscores, resulting in a name conflict.*
- It may cause confusion if you create named expressions with names that match expression functions such as "exp" and "abs".

2. Enter the **Definition** of the expression. *You can use the drop-downs to the right of the* **Definition** *text box to add functions, variables, cell registers, constants, existing expressions, report definitions, locations, and profiles to this expression definition.*

   All of the variables supported by the Fluent Expression Language are listed in the drop-down list.

   Note:

   - When you use curly brackets "{}" to specify the name of a defined object, such as a report definition or cell register, automatic suggestions are disabled and you must provide the display name of that object.
   - The instantaneous value of a report definition is always used when a report definition is included in an expression.

3. (Optional) Add a **Description** of the expression.

4. (Optional) Enable **Use as Input Parameter**, if you want to use the named expression as an input parameter. Refer to Defining and Viewing Parameters (p. 1309) for additional information on parameters.

   Note:

   Only constant-value expressions can be marked as input parameters.

5. (Optional) Enable **Use as Output Parameter**, if you want to use the named expression as an output parameter. Refer to Creating Output Parameters (p. 4133) and Working With Input and Output Parameters in Workbench in the *Fluent in Workbench User's Guide* for additional information on parameters.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. (Optional) Click to refresh the **Current Value** field to display the current value of the expression.

   Note:

   This feature is only available for expressions that evaluate to a single value, for example `Average(StaticPressure, ["inlet_1"])`, and not for expressions that evaluate as a field, such as `2*AbsolutePressure`.

7. Click **OK** to create the expression.

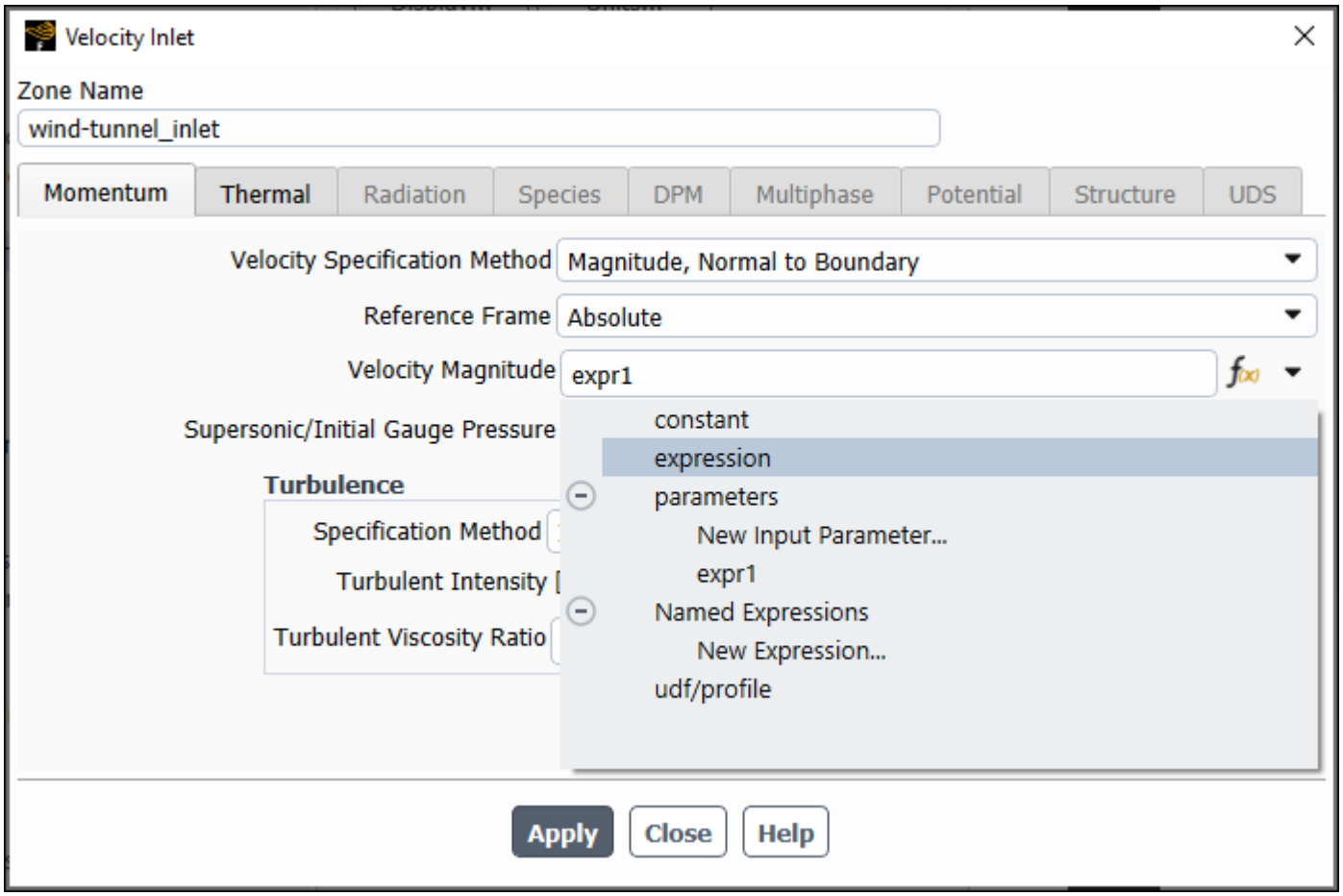
The **Used In** *field lists the locations where the expression is used, including other expressions.*

Important:

When you select a named expression for a field, either as just an expression or as an input parameter, the expression only shows as being selected under **Named Expressions** or **parameters** until you save the changes and close the dialog box. When the dialog box is reopened, you will see that the expression name is now displayed in the field and **expression** is selected from the drop-down. This is only a visual change in how this selection is shown; the field is still defined using that expression as it was when it was selected earlier.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

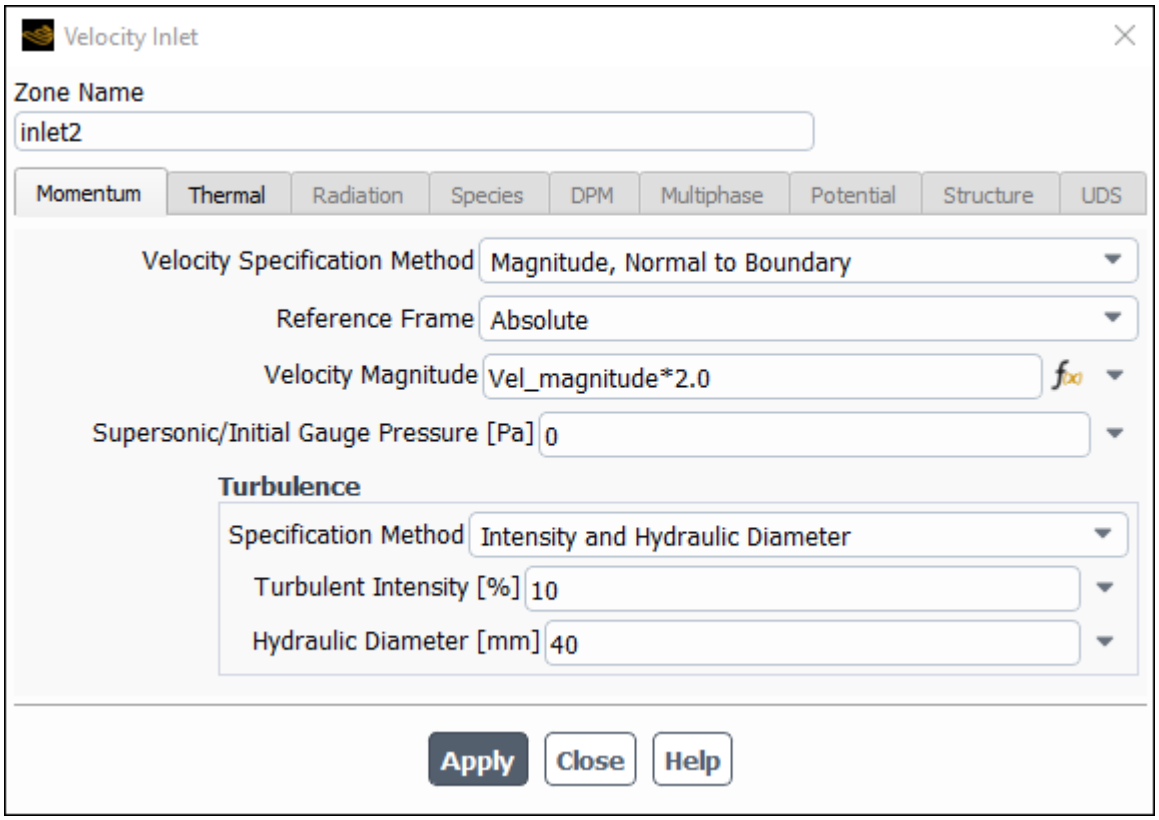
Figure 5.5: Velocity Magnitude Defined as "expr1", An Expression Input Parameter



In Figure 5.5: Velocity Magnitude Defined as "expr1", An Expression Input Parameter (p. 1058), the **Velocity Magnitude** field was previously set as an input parameter called `expr1`, but this is how it appears after the change is applied and the dialog box closed, and then re-opened.

You can use a named expression to define other expressions—both inline expressions and named expressions. For example, a named expression with the name `Vel_magnitude` can be used to setup the velocity magnitude setting of an inlet as follows:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

- Expressions that are in use cannot be deleted.
- You can create and modify named expressions through the `define/named-expressions/` text command in the console.

Important:

- Expressions must be dimensionally consistent. *For example, you cannot add two quantities that have different unit dimensions.*
- Report definitions that are included in expressions and have the **Per Surface | Per Zone** option enabled, must use an **Average Over** value of `1`.

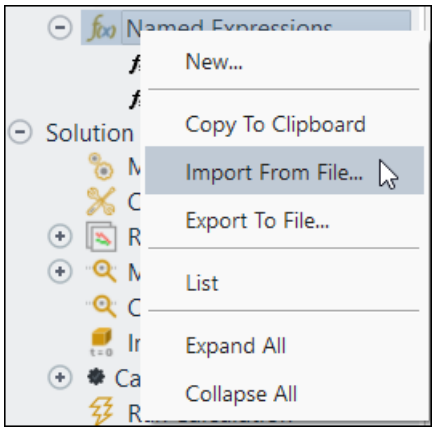
Saving and Importing Named Expressions

You have the option of saving some or all of the named expressions you create to a file for future use.

Click **Export To File...** to choose a directory and save your expression(s) to a file.

Click **Import From File...** to select and load an expression file.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



5.3.3. Context Specification

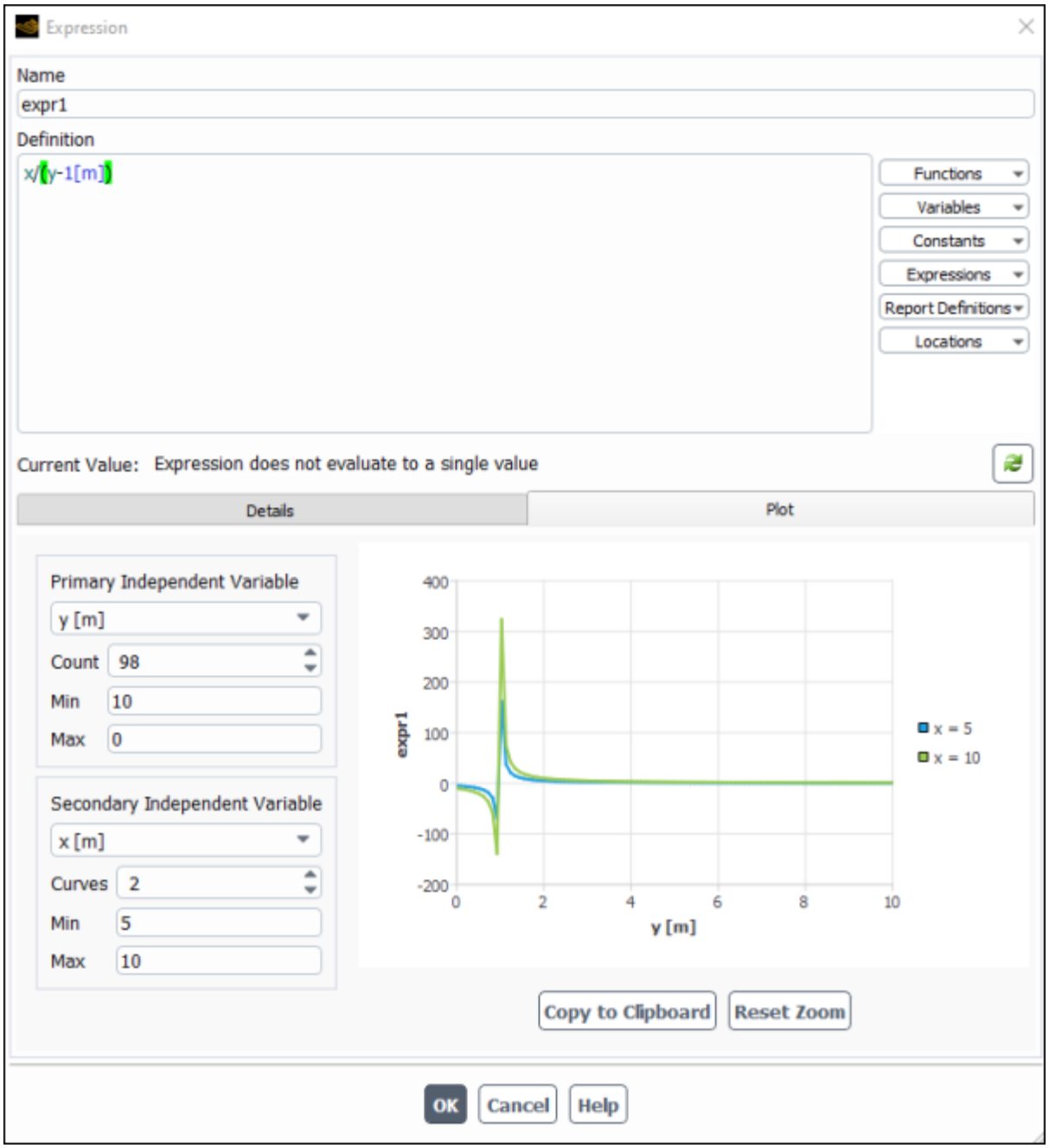
Some variables require additional context before they can be evaluated. For example, `MassFraction` requires the species context. Velocity may require phase context when it is evaluated in a multiphase simulation. The context is specified as a keyword argument to a function call on the variable. For example, `MassFraction(species="co2", phase="smoke")`.

5.3.4. Plotting Expressions

You can plot expressions to evaluate and visualize their behavior, as shown in Figure 5.6: Plotting an Expression (p. 1061).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.6: Plotting an Expression



To plot an expression:

1. Open the **Plot** tab.
2. Specify the **Primary Independent Variable** from the drop-down list.
3. Specify the number of plot points in the **Count** field.
4. Set the **Min** and **Max** range for the plot.
5. If you have more than one variable in your expression, you can define an additional variable as the **Secondary Independent Variable**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For the secondary variable, you can specify:

a. The number of **Curves**.

b. The **Min** and **Max** values for this variable.

6. (Optional) Click **Copy to Clipboard** to copy the plot image onto your clipboard.

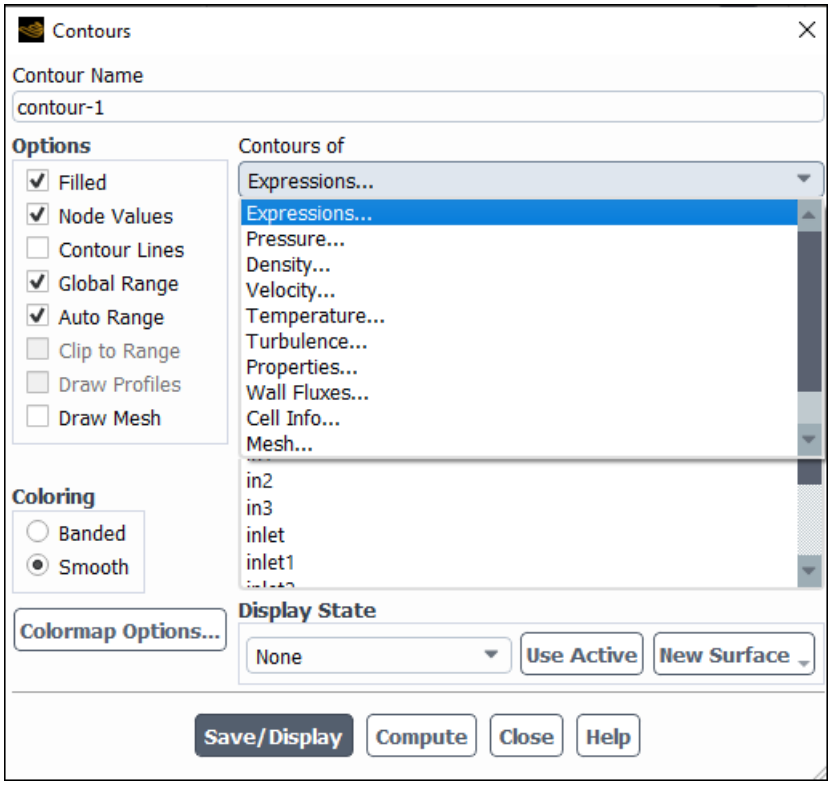
7. You can use the left and right-mouse buttons to zoom in and out on the plot. Click **Reset Zoom** to return to the original view for the plot.

5.3.5. Postprocessing Expressions

You can visualize expression results using the **Expressions...** field (Figure 5.7: Expressions... Postprocessing Field (p. 1062)) in postprocessing objects (contours, vectors, pathlines, particle tracks, XY plots). You also have the option of computing expression derivatives, which may also be visualized.

In addition to postprocessing expressions as field variables, you can also create expression volumes, which allow you to define a volume surface where you can display other field variables. Refer to Expression Volumes (p. 3845) for additional information.

Figure 5.7: Expressions... Postprocessing Field



Expressions available for postprocessing must:

- be of type **Double** or **Double Vector**. *If it is of type **Double Vector**, the components and magnitude are available for postprocessing.*

- evaluate to a field (and not a single-value)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- be fully specified. *For example, if the expression computes mass fraction, the specie must also be included in the definition (*`MassFraction(species='co2')`*. Similarly, for a multiphase case, the phase must be specified, where required (*`VolumeFraction(phase='water')`*.*

  Important:

  – For multiphase cases, the **Phase** drop-down is not relevant, because the available expressions are already fully specified. *Phase specification is not required for "mixture only" variables, such as static pressure.*

  – Expressions do not work reliably when exported to other postprocessors. They are not exported to CFD-Post.

  Note:

  While the plot displayed in the graphics window will show the units in the colormap, the **Min** and **Max** fields in the dialog box for the same plot, will not.

Computing Statistics on Expressions

You can select expressions that evaluate to a field in the **Zone-Specific Sampling Options** dialog box. Refer to Inputs for Time-Dependent Problems (p. 3645) and Data Sampling for Steady Statistics (p. 3641) for additional information on statistics.

Important:

RMSE values computed on expressions may differ from RMSE values computed on equivalent custom field functions. The difference is due to the number of significant digits used in expression computations (9 decimal places).

5.3.6. Expression Manager

As expressions are created, they are added to the Outline View tree under the **Named Expressions** branch. You can perform various operations on defined expressions (such as copying and deleting) using context menus available via right-mouse-button click of the expression(s) in the Outline View tree. The **Expression Manager** (Figure 5.8: Expression Manager Dialog Box (p. 1064)) allows you to perform all of these operations within a single dialog box.

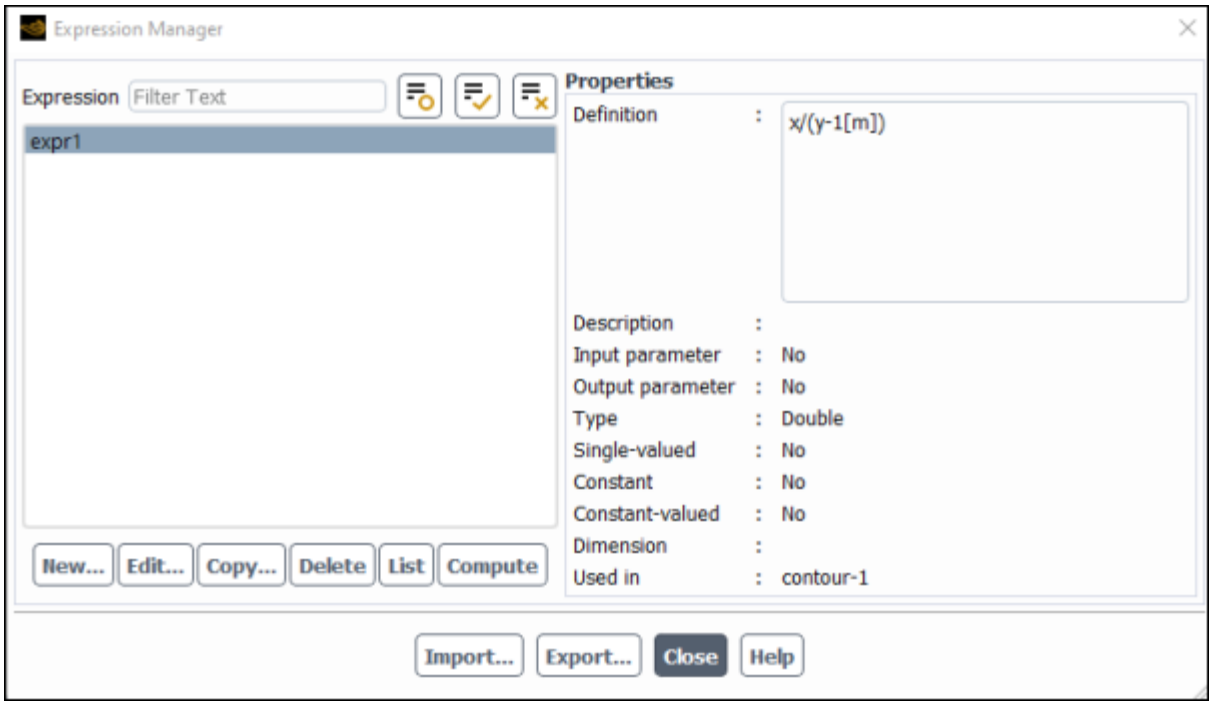
To use the **Expression Manager**:

1. Open the **Expression Manager** dialog box.

   Setup → Named Expressions → Manage...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.8: Expression Manager Dialog Box



2. Select the desired expression(s) and click the relevant button to perform the desired operation. *Not all operations are available when multiple expressions are selected, for example,* ***Edit...*** *is grayed-out.*

3. Click **Close** when you are done modifying the expressions.

Refer to Expression Manager Dialog Box (p. 5468) for detailed explanations of each button function.

5.4. Expression Examples

The following are a few examples that demonstrate why or where you may want to use an expression and how you would do so for these situations.

5.4.1. Parabolic Inflow Profile

5.4.2. Time-Varied Parabolic Inflow

5.4.3. Controlled Outlet Temperature

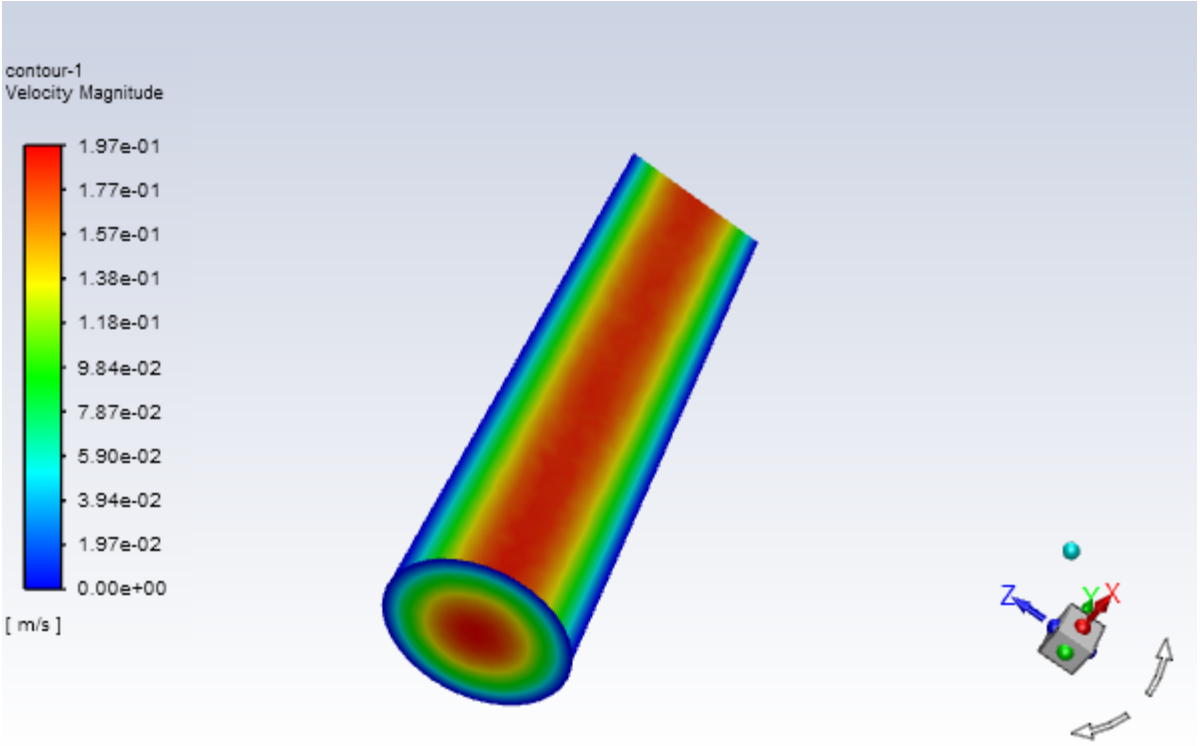
5.4.4. Computing Forces with Parameterized Angle of Attack

5.4.1. Parabolic Inflow Profile

The following example shows you how to define a parabolic inlet profile for laminar pipe flow, as shown in Figure 5.9: Contours of Velocity - Parabolic Inflow (p. 1065). In this example, the pipe is centered in the X and Z directions and the pipe axis is aligned to the Y-direction.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.9: Contours of Velocity - Parabolic Inflow



The expression for defining the parabolic inflow shown in Figure 5.9: Contours of Velocity - Parabolic Inflow (p. 1065) is Equation 5.1 (p. 1065),

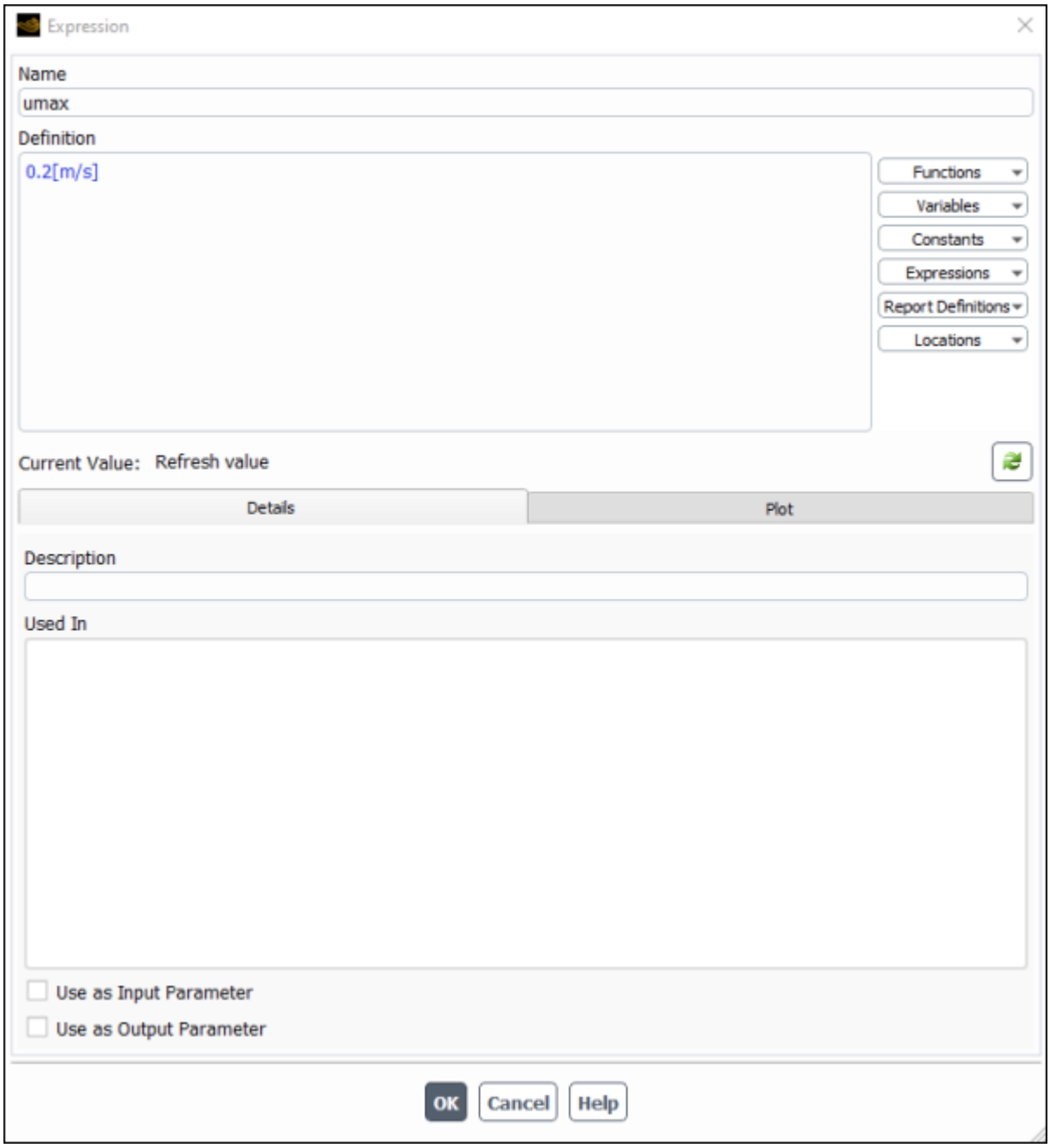
$$u_{\max}*\left(1-\left(radius/Radius\right)^{2}\right) \tag{5.1}$$

Where $u_{\max}$ is the velocity at the axis, $Radius$ is the radius of the pipe, and $radius$ is the local radial coordinate.

To fully define this example:

1. Create a named expression for the maximum velocity called **umax**.

   Setup → **Named Expressions** **New...**



   a. Enter **umax** for the **Name**.

   b. Enter **0.2 [m/s]** for the **Definition** and click **OK**.

2. Create an expression for the radius of the pipe.

   a. Open the **Expressions** dialog box by right-clicking **Named Expressions** in the outline view tree and selecting **New...**.

   b. Enter **Radius** for the **Name**.

   c. Enter **sqrt(Area(["in"])/PI)** for the **Definition**. *"in" is the name of the inlet boundary. PI is the expression constant for Pi.*

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

d. Click **OK** to create the named expression.

3. Create an expression for the local radial profile.

   a. Open the **Expressions** dialog box and enter **radius** for the **Name**.

   b. Enter **sqrt(x**2+z**2)** for the **Definition**. *This expression uses the square root mathematical expression operator.*

   c. Click **OK** to create the named expression.

4. Create an expression for the inlet velocity profile. *This expression combines the other expressions you created.*

   a. Open the **Expressions** dialog box and enter **uprofile** for the **Name**.

   b. Enter **umax*(1- (radius/Radius)**2)** for the **Definition**. *You can use the **Expressions** drop-down to the right of the **Definition** box to add named expressions to your expression definition as an alternative to typing the names manually.*

   c. Click **OK** to create the named expression.

5. Assign **uprofile** to the velocity inlet.

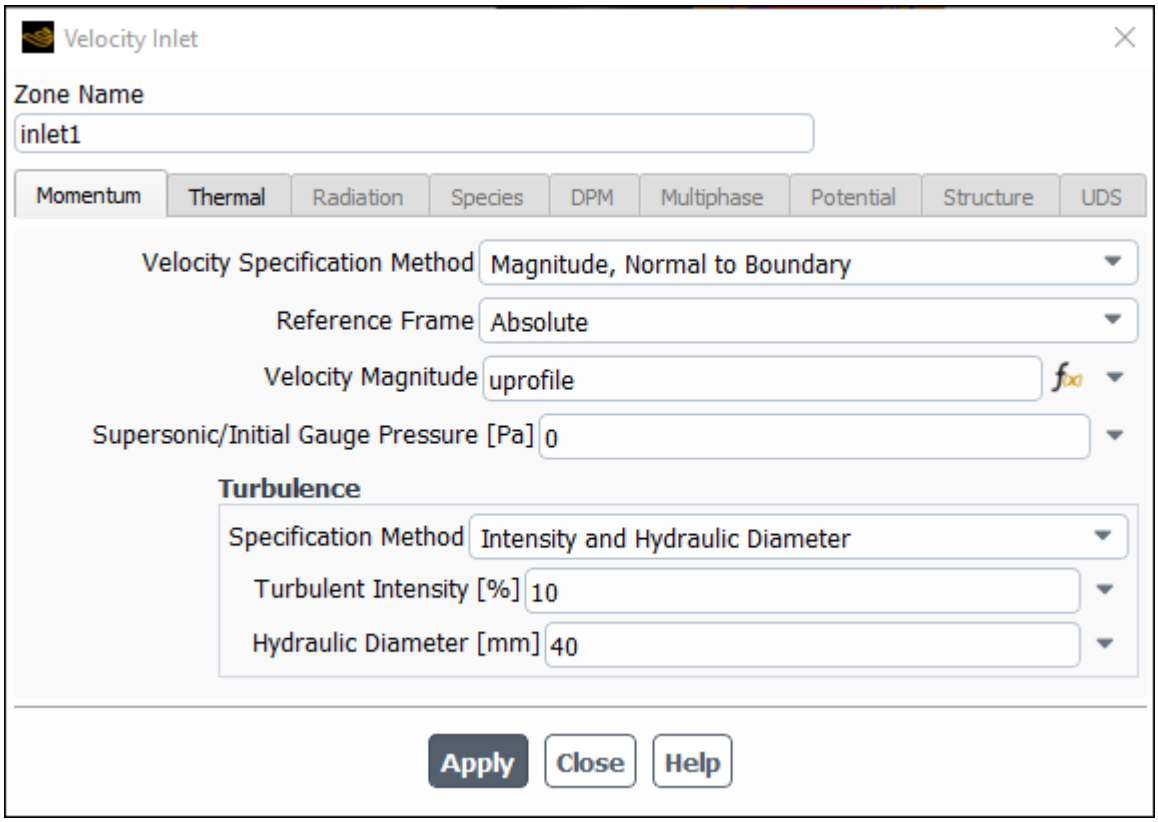
   Setup → Boundary Conditions → Inlet → in Edit...

   *You can group the boundary conditions by type to organize the boundaries and reduce the size of the list. This can be accomplished by right-clicking **Boundary Conditions** in the tree and selecting **Group By**> **Zone Type**.*

   a. Select **expression** from the drop-down to the right of **Velocity Magnitude**.

   b. Enter **uprofile** in the **Velocity Magnitude** field and click **OK**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

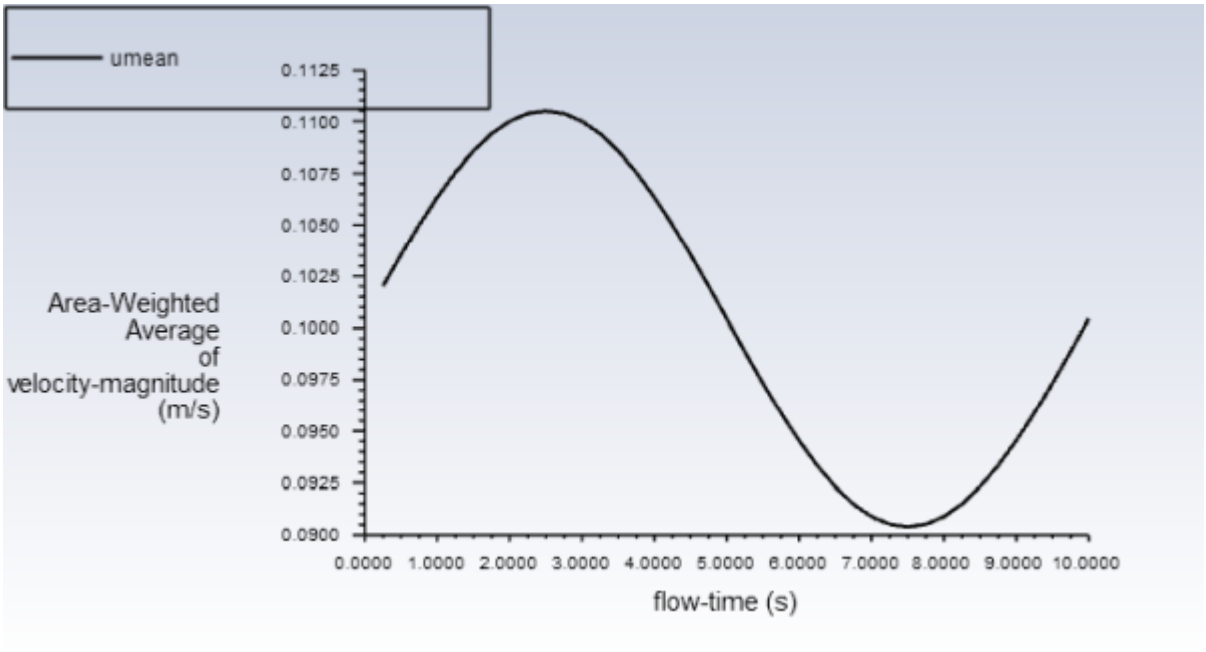


5.4.2. Time-Varied Parabolic Inflow

The following example shows you how to define a parabolic inlet profile for laminar pipe flow that varies over time (a sine function with a period of 10 seconds). The velocity profile in this example is the same as for Parabolic Inflow Profile (p. 1064) except it is for a transient simulation because the velocity varies over time. Figure 5.10: Parabolic Inflow Velocity Over Time (p. 1069) plots the area weighted average of the velocity magnitude over 10 seconds.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.10: Parabolic Inflow Velocity Over Time

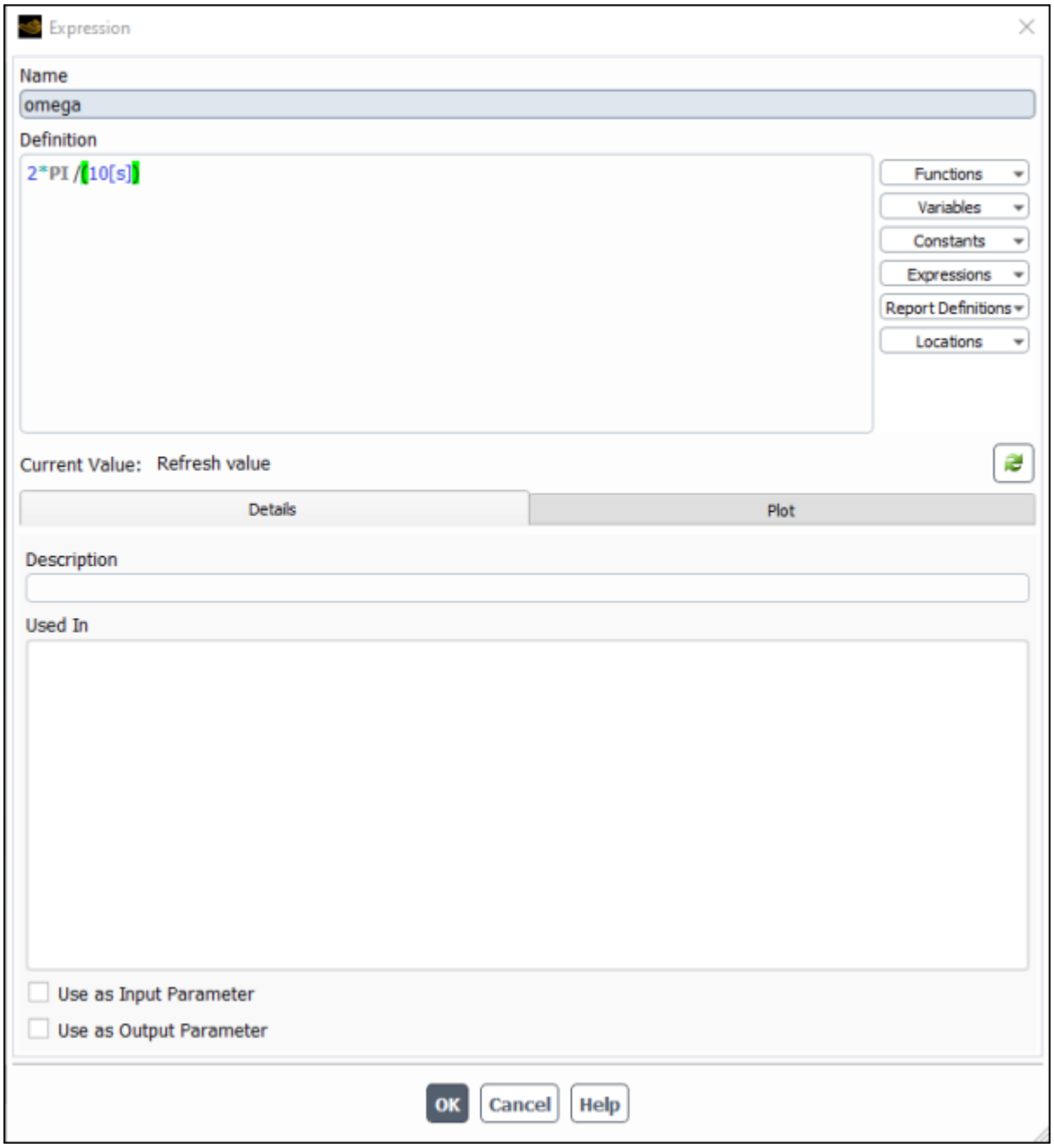


To create an expression for this time-varied velocity profile:

1. Define all of the named expressions as described in Parabolic Inflow Profile (p. 1064).
2. Create an expression for omega.

   Setup → Named Expressions New...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



a. Enter **omega** for the **Name**.

b. Enter **2*PI / (10[s])** for the **Definition** and click **OK**.

3. Create an expression for the velocity profile that varies over time.

   a. Open the **Expressions** dialog box and enter **uprofile_transient** for the **Name**.

   b. Enter **uprofile*(sin(omega*Time)*0.1 + 1)** for the **Definition** and click **OK**.

4. Assign **uprofile_transient** to the velocity inlet.

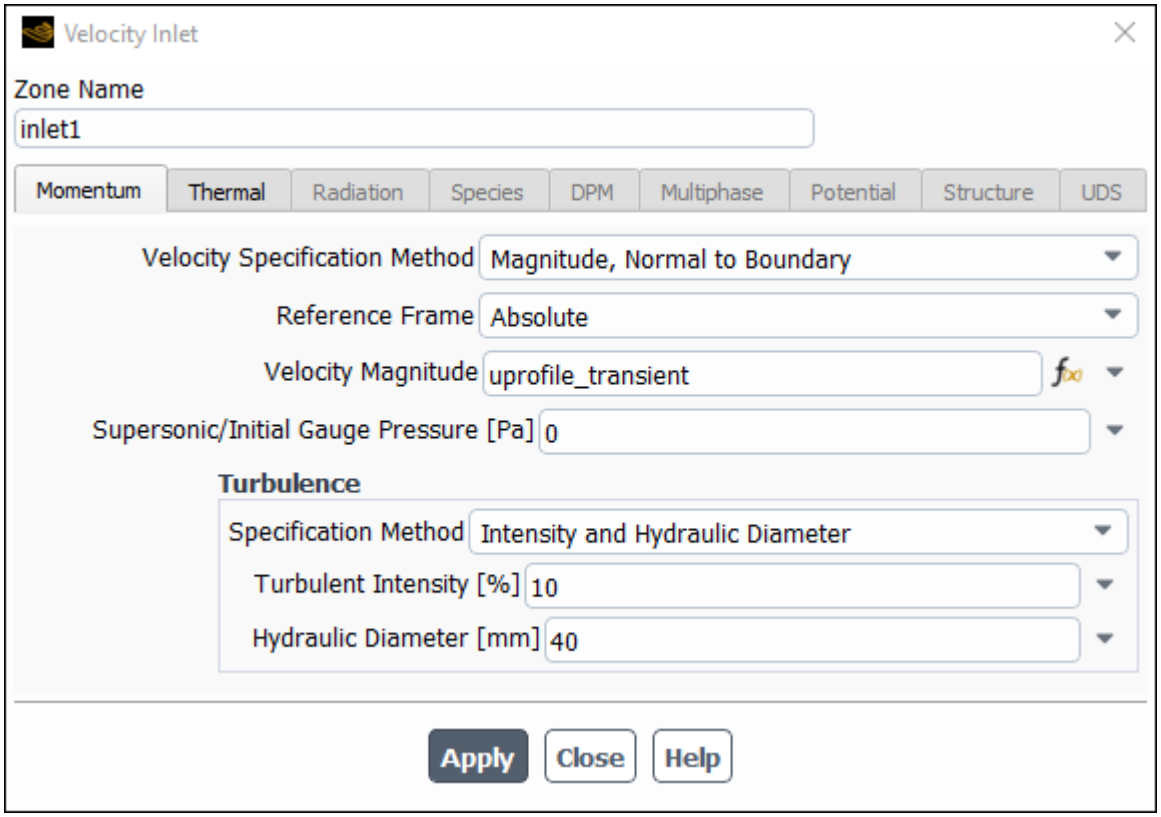
   Setup → Boundary Conditions → Inlet → in → Edit...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can group the boundary conditions by type to organize the boundaries and reduce the size of the list. This can be accomplished by right-clicking ***Boundary Conditions*** *in the tree and selecting* ***Group By*** *>* ***Zone Type****.*

a. Select **expression** from the drop-down to the right of **Velocity Magnitude**.

b. Enter **uprofile_transient** in the **Velocity Magnitude** field and click **OK**.



5.4.3. Controlled Outlet Temperature

The following example shows you how to drive the maximum temperature at the outlet to a target temperature by varying the inlet temperature. For this scenario, a pipe with laminar flow has a part of the wall that is heated to 350 Kelvin and a target outlet temperature of 320 Kelvin (Figure 5.11: Pipe Geometry Colored by ID (Heated Wall is Green) (p. 1072) shows the portion of the wall that is heated). A controller turns on every 10th iteration (steady-state case) and adjusts the inlet temperature according to Equation 5.2 (p. 1071). In the other iterations, the controller leaves the temperature unchanged. This is achieved by using the conditional statement given in Equation 5.3 (p. 1071). Figure 5.12: Contours of Temperature (outlet is closest) (p. 1072) shows the developed temperature profile for this case.

$$T_{Adjust} = T_{InCurrent} + (T_{\mathrm{Target}} - T_{OutMax}) * relax \tag{5.2}$$

$$IF(iter<5, TStart, IF(\mathrm{mod}(iter+1,10)==0, Tadjust, TInCurrent)) \tag{5.3}$$

T_{Adjust} is the formula used to adjust the inlet temperature, $T_{InCurrent}$ is the current inlet temperature, T_{Target} is the target temperature for the outlet, which is set to 320 Kelvin, T_{OutMax} is the maximum temperature at the outlet, and $relax$ is a relaxation factor.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5.11: Pipe Geometry Colored by ID (Heated Wall is Green)

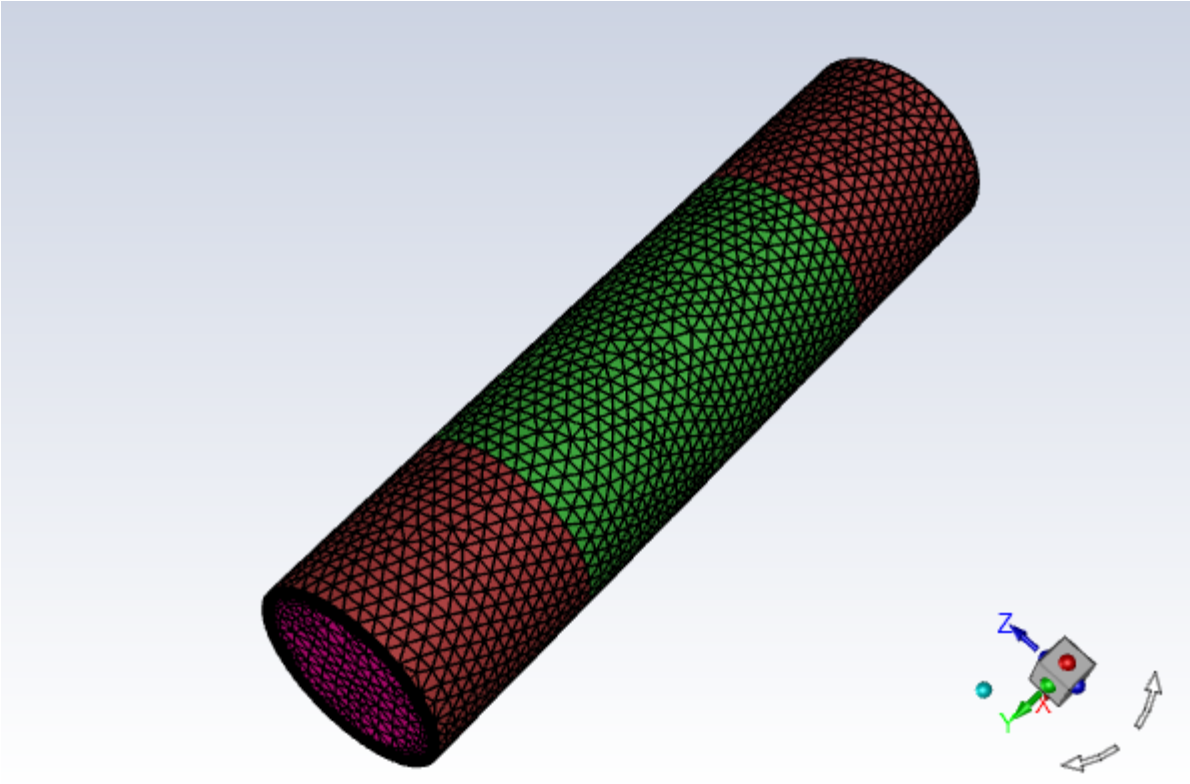
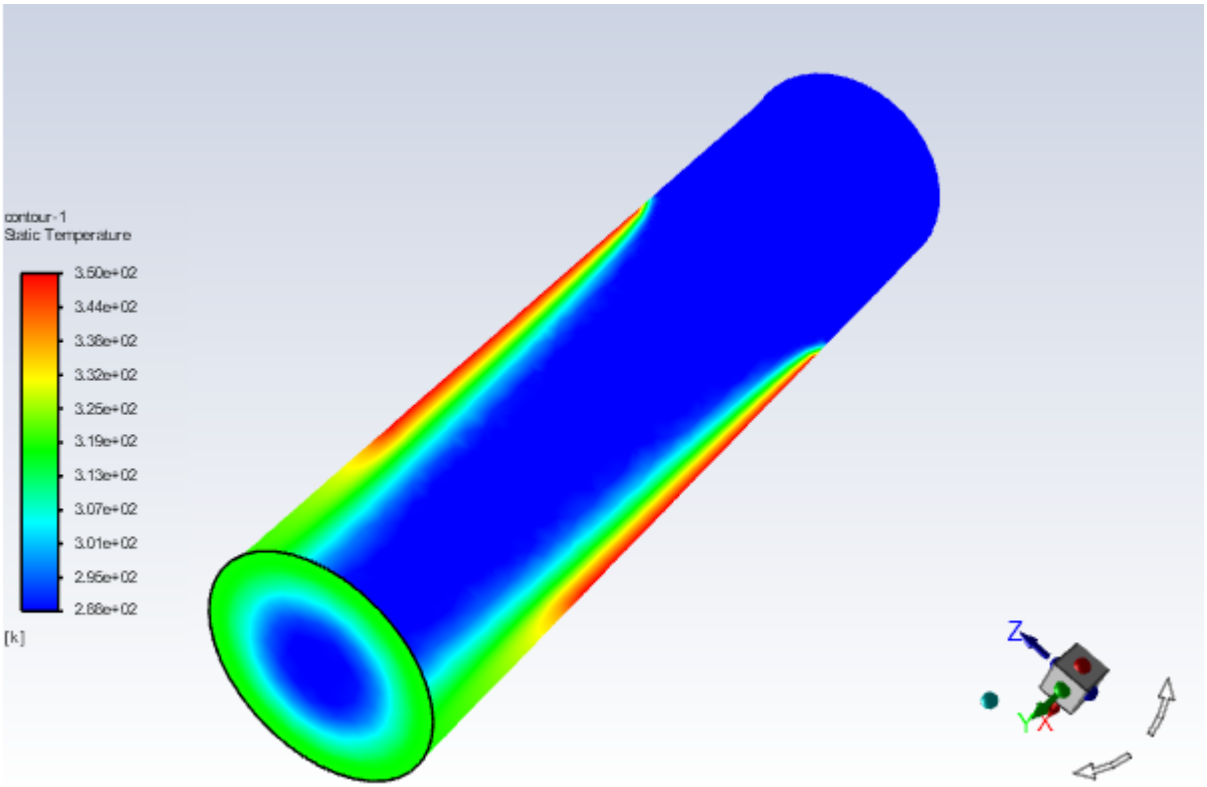
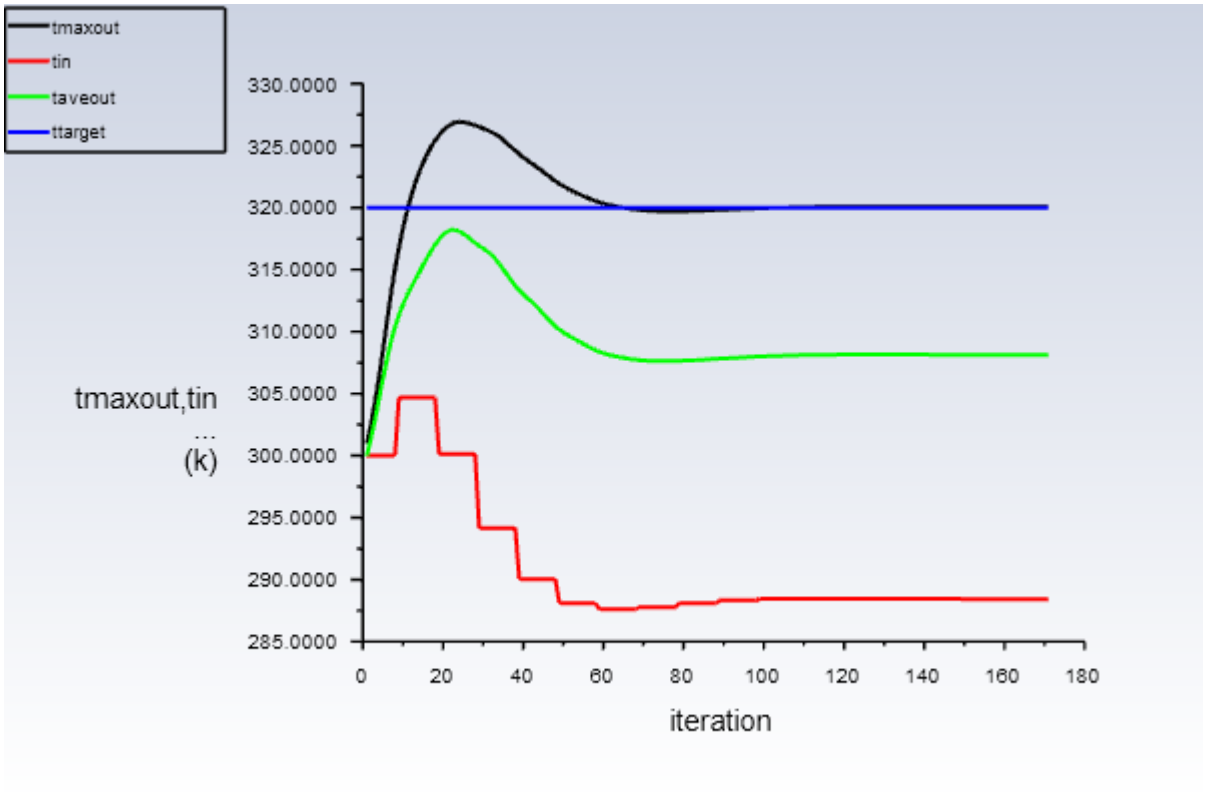


Figure 5.12: Contours of Temperature (outlet is closest)



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

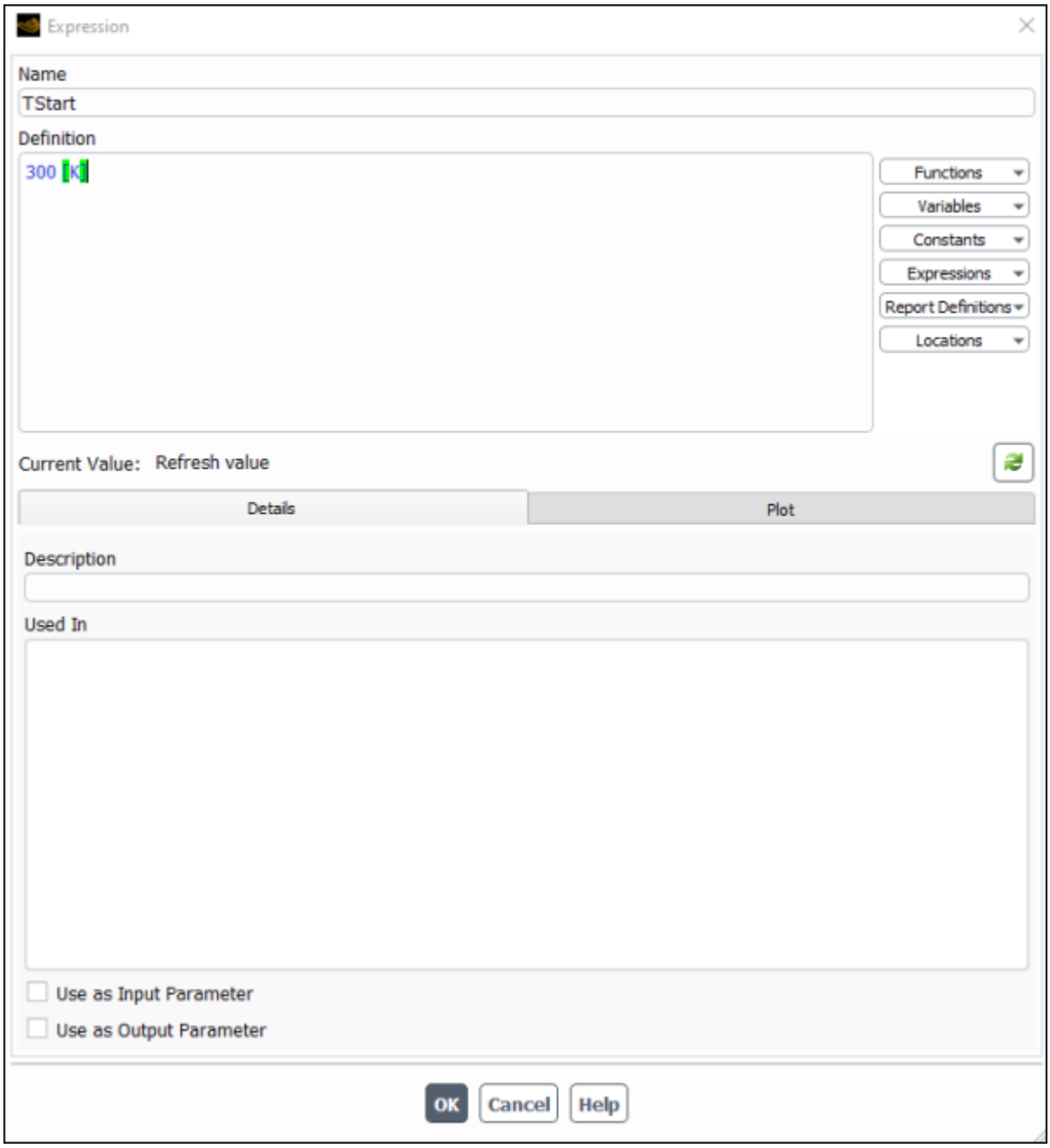
Figure 5.13: Plots of Inlet Temperature, Average Outlet Temperature, and Maximum Outlet Temperature



The following steps show how to create the expressions used in this example:

1. Create a named expression for the initial temperature, called **TStart**.

   Setup → Named Expressions New...



a. Enter **TStart** for the **Name**.

b. Enter **300 [K]** for the **Definition** and click **OK**.

2. Create an expression for the target temperature for the outlet, called **TTarget**, and set it equal to **320 [K]**. *Use the same process that you used to create **TStart**.*

3. Create an expression for the maximum temperature that occurs at the outlet, called **TMaxOut**. *In this expression you will be using a function to determine the maximum temperature.*

   a. Click the **Functions** drop-down and select **Reduction** → **Maximum**.

   b. Enter **StaticTemperature** for the first field of the function and **["out"]** for the location. The total expression should appear as **Maximum(StaticTemperature, ["out"])**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Create an expression for the current inlet temperature, called **TInCurrent**, and set it equal to **Average(StaticTemperature, ["in"], Weight=None)**. *Use a similar process to what you used to create **TMaxOut**, but with selecting the **Average** function instead.*

5. Create an expression for the relaxation factor, called **relax**, and set it equal to **0.9**.

6. Create an expression for adjusting the inlet temperature, called **TAdjust**, and set it equal to **TInCurrent + (TTarget - TMaxOut)*relax**.

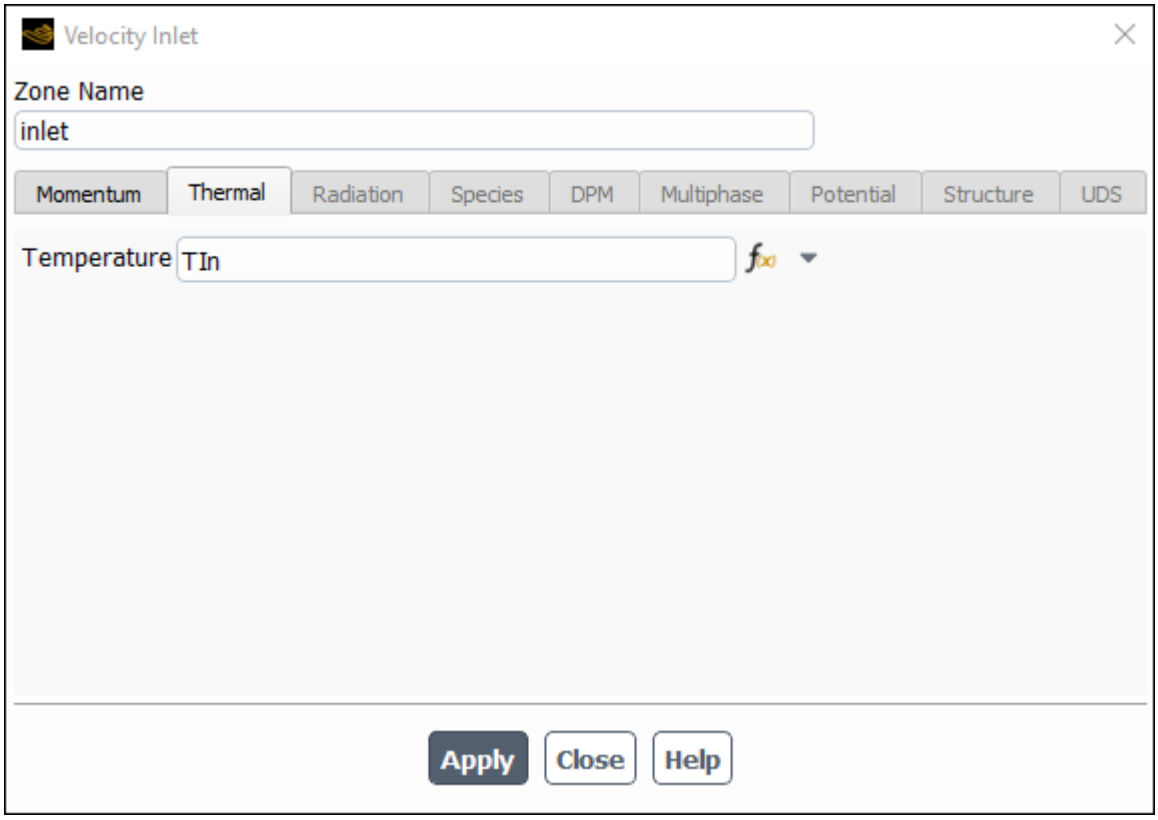
7. Create an expression for the inlet temperature, which will be called **TIn**, and set it equal to **IF(iter<5, TStart, IF(mod(iter+1, 10)==0, TAdjust, TInCurrent))**. *Use a similar process to what you used to create **TMaxOut**, but with selecting the **IF** conditional function instead.*

8. Assign **TIn** to the inlet.

   Setup → Boundary Conditions → Inlet → in Edit...

   a. Select **expression** from the drop-down to the right of **Temperature**.

   b. Enter **TIn** in the **Temperature** field and click **OK**.



5.4.4. Computing Forces with Parameterized Angle of Attack

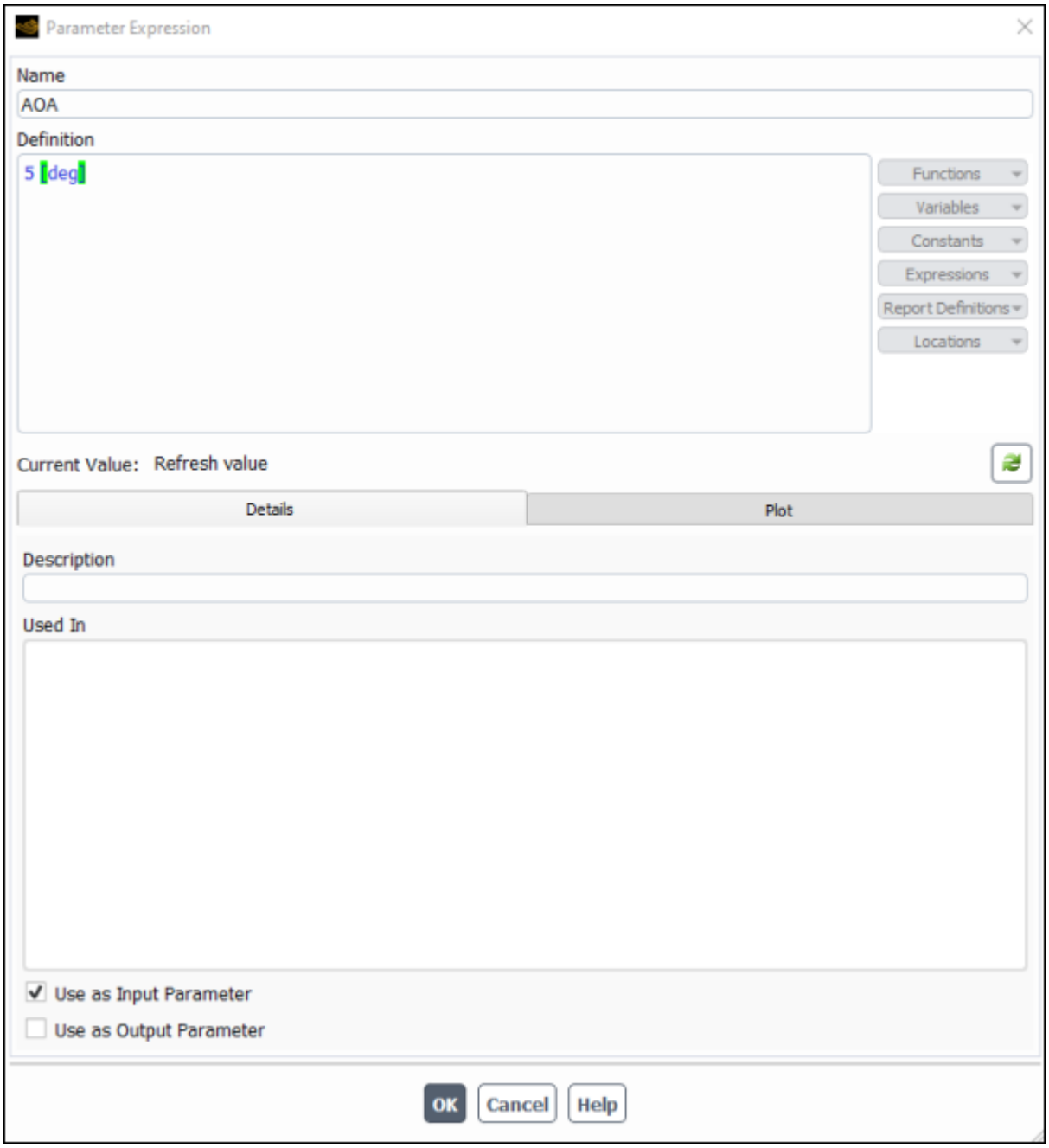
The following example shows you how to setup to use `dot()`, `vector()`, `moment()`, and `force()` expression operations in Ansys Fluent. Six expressions are defined: AOA, dragForce, flowDir, liftForce, moment, and totalForce.

To begin, define the angle of attack expression as an input parameter.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Open the **Expression** dialog box.

   Setup → Named Expressions → New...



   *Note that the dialog box appears as **Parameter Expression** because the setup is already completed and **Use as Input Parameter** is enabled.*

2. Enter A0A as the **Name**.

3. Enter `5[deg]` in the **Definition** field.

4. Enable **Use as Input Parameter**.

5. Click **OK** to create the angle of attack input parameter expression.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Define the flow direction.

1. Open the **Expression** dialog box.

   Setup → **Named Expressions** → **New...**

2. Enter `flowDir` as the **Name**.
3. Enter `vector(cos(AOA), sin(AOA), 0.0)` in the **Definition** field.

   Note:

   You select expressions that you have already defined, such as **AOA** from the **Expressions** drop-down list on the right-hand side of the dialog box.

4. Click **OK** to create the expression for the flow direction.

Define the total force.

1. Open the **Expression** dialog box.

   Setup → **Named Expressions** → **New...**

2. Enter `totalForce` as the **Name**.
3. Enter `Force(['wall-top', 'wall-bottom'])` in the **Definition** field.
4. Click **OK** to create the expression for the total force.

Define the moment about the centroid of the airfoil.

1. Open the **Expression** dialog box.

   Setup → **Named Expressions** → **New...**

2. Enter `moment` as the **Name**.
3. Enter `Moment(Centroid(['fluid-16'], ['wall-top', 'wall-bottom']).mag` in the **Definition** field.
4. Enable **Use as Output Parameter** to use the moment as an output parameter.
5. Click **OK** to create the expression for the total force.

Define the lift force normal to the flow.

1. Open the **Expression** dialog box.

   Setup → **Named Expressions** → **New...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Enter `liftForce` as the **Name**.
3. Enter `dot(totalForce, vector(-flowDir.y, flowdir.x, 0))` in the **Definition** field.

   Note:

   You select expressions that you have already defined, such as **totalForce** from the **Expressions** drop-down list on the right-hand side of the dialog box.

4. Enable **Use as Output Parameter** to use the lift force as an output parameter.
5. Click **OK** to create the expression for the lift force.

Define the drag force.

1. Open the **Expression** dialog box.

   Setup → **Named Expressions** → **New...**

2. Enter `dragForce` as the **Name**.
3. Enter `dot(totalForce, flowDir)` in the **Definition** field.

   Note:

   You select expressions that you have already defined, such as **totalForce** from the **Expressions** drop-down list on the right-hand side of the dialog box.

4. Enable **Use as Output Parameter** to use the drag force as an output parameter.
5. Click **OK** to create the expression for the total force.

Now that these expressions are defined, you can include them in expression report definitions for monitoring and judging convergence.

5.5. Appendix: Supported Field Variables

Note:

Postprocessing variables related to structures and the Structural Model are only available on solid cell zones. Therefore, any computations made on variables available under **Structure...** should only include solid zones to ensure they are evaluated correctly.

Description	Variable	Section	Type
Static Pressure	StaticPressure	Pressure...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
Pressure Correction Ap	PressureCorrectionAp	Pressure...	Scalar
Pressure Correction Source	PressureCorrectionSource	Pressure...	Scalar
Reconstruction dp/d	Reconstructiondpd	Pressure...	Vector
Pressure Coefficient	PressureCoefficient	Pressure...	Scalar
Dynamic Pressure	DynamicPressure	Pressure...	Scalar
Absolute Pressure	AbsolutePressure	Pressure...	Scalar
Total Pressure	TotalPressure	Pressure...	Scalar
Capillary-Pressure	CapillaryPressure	Pressure...	Scalar
User Mass Source	UserMassSource	Pressure...	Scalar
Pressure Discontinuity Sensor	PressureDiscontinuitySensor	Pressure...	Scalar
Density	Density	Density...	Scalar
Density All	DensityAll	Density...	Scalar
Reconstruction drho/d	Reconstructiondrhod	Density...	Vector
Velocity Magnitude	VelocityMagnitude	Velocity...	Scalar
Velocity	Velocity	Velocity...	Vector
Reconstruction dX-Velocity/d	ReconstructiondXVelocityd	Velocity...	Vector
Reconstruction dY-Velocity/d	ReconstructiondYVelocityd	Velocity...	Vector
Reconstruction dZ-Velocity/d	ReconstructiondZVelocityd	Velocity...	Vector
Velocity Ap	VelocityAp	Velocity...	Vector
Velocity Source	VelocitySource	Velocity...	Vector
Axial Velocity	AxialVelocity	Velocity...	Scalar
Radial Velocity	RadialVelocity	Velocity...	Scalar
Tangential Velocity	TangentialVelocity	Velocity...	Scalar
Mach Number	MachNumber	Velocity...	Scalar
Normal Mach Number	NormalMachNumber	Velocity...	Scalar
Mesh -Velocity	MeshVelocity	Velocity...	Vector
Moving Mesh Courant Number	MovingMeshCourantNumber	Velocity...	Scalar
Velocity Angle	VelocityAngle	Velocity...	Scalar
Vorticity Magnitude	VorticityMagnitude	Velocity...	Scalar
Helicity	Helicity	Velocity...	Scalar
Vorticity	Vorticity	Velocity...	Vector
Cell Reynolds Number	ElementReynoldsNumber	Velocity...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
Compressible Wall Reynolds Number	CompressibleWallReynoldsNumber	Velocity...	Scalar
Cell Convective Courant Number	ElementConvectionCourantNumber	Velocity...	Scalar
Multiphase Minimum Time Scale	MultiphaseMinimumTimeScale	Velocity...	Scalar
Cell Acoustic Courant Number	ElementAcousticCourantNumber	Velocity...	Scalar
Preconditioning Reference Velocity	PreconditioningReferenceVelocity	Velocity...	Scalar
X Velocity Mesh Smoothing	XVelocityMeshSmoothing	Velocity...	Scalar
Y Velocity Mesh Smoothing	YVelocityMeshSmoothing	Velocity...	Scalar
Z Velocity Mesh Smoothing	ZVelocityMeshSmoothing	Velocity...	Scalar
X Velocity Mesh Smoothing	XVelocityMeshSmoothing	Velocity...	Scalar
Y Velocity Mesh Smoothing	YVelocityMeshSmoothing	Velocity...	Scalar
Z Velocity Mesh Smoothing	ZVelocityMeshSmoothing	Velocity...	Scalar
BCD Normalized Variable U	BCDNormalizedVariableU	Velocity...	Scalar
BCD Normalized Variable V	BCDNormalizedVariableV	Velocity...	Scalar
BCD Normalized Variable W	BCDNormalizedVariableW	Velocity...	Scalar
Q Criterion Normalized	QCriterionNormalized	Velocity...	Scalar
Q Criterion Raw	QCriterionRaw	Velocity...	Scalar
Lambda 2 Criterion	Lambda2Criterion	Velocity...	Scalar
Tagged Face	TaggedFace	Perforated...	Scalar
Injection Hole Mass Flow Rate	InjectionHoleMassFlowRate	Perforated...	Scalar
Injection Pressure	InjectionPressure	Perforated...	Scalar
Static Temperature	StaticTemperature	Temperature...	Scalar
RMS Temperature	RMSTemperature	Temperature...	Scalar
Reconstruction dT/d	ReconstructiondTd	Temperature...	Vector
Temperature Ap	TemperatureAp	Temperature...	Scalar
Temperature Source	TemperatureSource	Temperature...	Scalar
User Energy Source	UserEnergySource	Temperature...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
User Volumetric Energy Source	UserVolumetricEnergySource	Temperature...	Scalar
Temperature EM Volume Source	TemperatureEMVolumeSource	Temperature...	Scalar
Temperature EM Surface Source	TemperatureEMSurfaceSource	Temperature...	Scalar
Mean Static Temperature	PDFMeanTemperature	Temperature...	Scalar
RMS Static Temperature	RMSStaticTemperature	Temperature...	Scalar
Total Temperature	TotalTemperature	Temperature...	Scalar
Sensible Enthalpy	SensibleEnthalpy	Temperature...	Scalar
Enthalpy	SpecificEnthalpy	Temperature...	Scalar
Reconstruction dH/d	ReconstructiondHd	Temperature...	Vector
Enthalpy Ap	EnthalpyAp	Temperature...	Scalar
Enthalpy Source	EnthalpySource	Temperature...	Scalar
Wall Temperature	WallTemperature	Temperature...	Scalar
Wall Temperature (Shell Cell/Thin)	WallTemperatureShellCellThin	Temperature...	Scalar
Wall Adjacent Temperature	WallAdjacentTemperature	Temperature...	Scalar
Yplus Based Heat Tran. Ref. Temperature	YplusBasedHeatTranRefTemperature	Temperature...	Scalar
Wall Temperature (Thin)	WallTemperatureThin	Temperature...	Scalar
External Temperature (Shell)	ExternalTemperatureShell	Temperature...	Scalar
Total Enthalpy	SpecificTotalEnthalpy	Temperature...	Scalar
Total Enthalpy Deviation	TotalEnthalpyDeviation	Temperature...	Scalar
Entropy	SpecificEntropy	Temperature...	Scalar
Total Energy	SpecificTotalEnergy	Temperature...	Scalar
Internal Energy	SpecificInternalEnergy	Temperature...	Scalar
Fine Scale Temperature	FineScaleTemperature	Temperature...	Scalar
Equilibrium Temperature	EquilibriumTemperature	Temperature...	Scalar
Non-Equilibrium Thermal Model Source	NonEquilibriumThermalModelSource	Temperature...	Scalar
Heat Exchanger Source	HeatExchangerSource	Temperature...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
Wall Work Density	WallWorkDensity	Temperature...	Scalar
Wall Power Density	WallPowerDensity	Temperature...	Scalar
Incident Radiation	IncidentRadiation	Radiation...	Scalar
Incident Radiation (Inner)	IncidentRadiationInner	Radiation...	Scalar
Absorption Coefficient	AbsorptionCoefficient	Radiation...	Scalar
Scattering Coefficient	ScatteringCoefficient	Radiation...	Scalar
Refractive Index	RefractiveIndex	Radiation...	Scalar
Radiation Temperature	RadiationTemperature	Radiation...	Scalar
Volumetric Absorbed Radiation	VolumetricAbsorbedRadiation	Radiation...	Scalar
Volumetric Emitted Radiation	VolumetricEmittedRadiation	Radiation...	Scalar
Volume Glob Source	VolumeGlobSource	Radiation...	Scalar
Volume Glob Factor	VolumeGlobFactor	Radiation...	Scalar
Surface Cluster ID	SurfaceClusterID	Radiation...	Scalar
Face Radiation	FaceRadiation	Radiation...	Scalar
Radiation Intensity.Normalized Std Deviation	RadiationIntensityNormalizedStdDeviation	Radiation...	Scalar
Wall Irradiation Flux.Normalized Std Deviation	WallIrradiationFluxNormalizedStdDeviation	Radiation...	Scalar
Liquid Fraction	LiquidFraction	Solidification / Melting...	Scalar
Contact Resistivity	ContactResistivity	Solidification / Melting...	Scalar
Pull Velocity	PullVelocity	Solidification / Melting...	Vector
Liquidus Temperature	LiquidusTemperature	Solidification / Melting...	Scalar
Solidus Temperature	SolidusTemperature	Solidification / Melting...	Scalar
Air - Density	AirDensity	Icing...	Scalar
Air - Pressure	AirPressure	Icing...	Scalar
Air - Velocity magnitude	AirVelocitymagnitude	Icing...	Scalar
Air - Velocity	AirVelocity	Icing...	Vector
Air - Temperature	AirTemperature	Icing...	Scalar
Air - Wall Distance	AirWallDistance	Icing...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
Air - UPlus	AirUPlus	Icing...	Scalar
Air - YPlus	AirYPlus	Icing...	Scalar
Air - Turbulent viscosity	AirTurbulentviscosity	Icing...	Scalar
Air - Laminar viscosity	AirLaminarviscosity	Icing...	Scalar
Air - Shear stress X	AirShearstressX	Icing...	Scalar
Air - Shear stress Y	AirShearstressY	Icing...	Scalar
Air - Shear stress Z	AirShearstressZ	Icing...	Scalar
Air - Classical Heat Flux	AirClassicalHeatFlux	Icing...	Scalar
Air - Gresho Heat Flux	AirGreshoHeatFlux	Icing...	Scalar
Air - Wall roughness height	AirWallroughnessheight	Icing...	Scalar
Air - Wall roughness constant	AirWallroughnessconstant	Icing...	Scalar
Droplet - LWC	DropletLWC	Icing...	Scalar
Droplet - Velocity-X	DropletVelocityX	Icing...	Scalar
Droplet - Velocity-Y	DropletVelocityY	Icing...	Scalar
Droplet - Velocity-Z	DropletVelocityZ	Icing...	Scalar
Droplet - Diameter	DropletDiameter	Icing...	Scalar
Droplet - Collection efficiency	DropletCollectionefficiency	Icing...	Scalar
Crystal - LWC	CrystalLWC	Icing...	Scalar
Crystal - Velocity-X	CrystalVelocityX	Icing...	Scalar
Crystal - Velocity-Y	CrystalVelocityY	Icing...	Scalar
Crystal - Velocity-Z	CrystalVelocityZ	Icing...	Scalar
Crystal - Diameter	CrystalDiameter	Icing...	Scalar
Crystal - Collection efficiency	CrystalCollectionefficiency	Icing...	Scalar
Vapor - Concentration	VaporConcentration	Icing...	Scalar
Vapor - Condensation	VaporCondensation	Icing...	Scalar
Vapor - Saturation Vapor Pressure	VaporSaturationVaporPressure	Icing...	Scalar
Vapor - Vapor pressure	VaporVaporpressure	Icing...	Scalar
Vapor - Relative Humidity	VaporRelativeHumidity	Icing...	Scalar
Vapor - Vapor diffusivity	VaporVapordiffusivity	Icing...	Scalar
Ice - Mass caught	IceMasscaught	Icing...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
Ice - Film thickness	IceFilmthickness	Icing...	Scalar
Ice - Instant. Ice Growth	IceInstantIceGrowth	Icing...	Scalar
Ice - Growth	IceGrowth	Icing...	Scalar
Ice - Instant. mass evaporation	IceInstantmassevaporation	Icing...	Scalar
Ice - Wall temperature	IceWalltemperature	Icing...	Scalar
Ice - Density	IceDensity	Icing...	Scalar
Ice - Cfx	IceCfx	Icing...	Scalar
Ice - Cfy	IceCfy	Icing...	Scalar
Ice - Cfz	IceCfz	Icing...	Scalar
Ice - Flow solver convective flux	IceFlowsolverconvectiveflux	Icing...	Scalar
Ice - Ch	IceCh	Icing...	Scalar
Ice - Current convective heat flux	IceCurrentconvectiveheatflux	Icing...	Scalar
Ice - Evaporative heat flux	IceEvaporativeheatflux	Icing...	Scalar
Ice - Anti-icing heat fluxx	IceAntiicingheatfluxx	Icing...	Scalar
Ice - Ice thickness	IceIcethickness	Icing...	Scalar
Ice - Anti-icing wall temp	IceAntiicingwalltemp	Icing...	Scalar
Ice - Bead height	IceBeadheight	Icing...	Scalar
Mean Static Pressure	MeanStaticPressure	Unsteady Statistics...	Scalar
Mean Velocity Magnitude	MeanVelocityMagnitude	Unsteady Statistics...	Scalar
Mean Velocity	MeanVelocity	Unsteady Statistics...	Vector
Mean Volume fraction	MeanVolumefraction	Unsteady Statistics...	Scalar
Mean Static Temperature	MeanStaticTemperature	Unsteady Statistics...	Scalar
Mean Mass Fraction of	MeanMassFractionof	Unsteady Statistics...	Scalar
Mean of Mixture Fraction	MeanofMixtureFraction	Unsteady Statistics...	Scalar
Mean Reaction Progress	MeanReactionProgress	Unsteady Statistics...	Scalar
Mean Phase Diameter	MeanPhaseDiameter	Unsteady Statistics...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
Mean Granular Temperature	MeanGranularTemperature	Unsteady Statistics...	Scalar
RMSE Static Pressure	RMSEStaticPressure	Unsteady Statistics...	Scalar
RMSE Velocity Magnitude	RMSEVelocityMagnitude	Unsteady Statistics...	Scalar
RMSE Velocity	RMSEVelocity	Unsteady Statistics...	Vector
LES Resolution Quality	LESResolutionQuality	Unsteady Statistics...	Scalar
RMSE Volume fraction	RMSEVolumefraction	Unsteady Statistics...	Scalar
RMSE Static Temperature	RMSEStaticTemperature	Unsteady Statistics...	Scalar
RMSE Mass Fraction of	RMSEMassFractionof	Unsteady Statistics...	Scalar
RMSE of Mixture Fraction	RMSEofMixtureFraction	Unsteady Statistics...	Scalar
RMSE Reaction Progress	RMSEReactionProgress	Unsteady Statistics...	Scalar
RMSE Phase Diameter	RMSEPhaseDiameter	Unsteady Statistics...	Scalar
RMSE Granular Temperature	RMSEGranularTemperature	Unsteady Statistics...	Scalar
Mean Sound WaveEq Model Source	MeanSoundWaveEqModelSource	Unsteady Statistics...	Scalar
Mean Sound Pressure	MeanSoundPressure	Unsteady Statistics...	Scalar
RMSE Sound WaveEq Model Source	RMSESoundWaveEqModelSource	Unsteady Statistics...	Scalar
RMSE Sound Pressure	RMSESoundPressure	Unsteady Statistics...	Scalar
Granular Pressure	GranularPressure	Granular Pressure...	Scalar
Granular Temperature	GranularTemperature	Granular Temperature...	Scalar
Turbulent Kinetic Energy (k)	TurbulentKineticEnergyk	Turbulence...	Scalar
Laminar Kinetic Energy	LaminarKineticEnergy	Turbulence...	Scalar
Total Fluctuation Energy	TotalFluctuationEnergy	Turbulence...	Scalar
Turbulent Intensity	TurbulenceIntensity	Turbulence...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
Turb Kin. Energy Ap	TurbKinEnergyAp	Turbulence...	Scalar
Turb Kin. Energy Source	TurbKinEnergySource	Turbulence...	Scalar
dk-d	dkd	Turbulence...	Vector
Reconstruction dk/d	Reconstructiondkd	Turbulence...	Vector
Solved Turbulent Kinetic Energy (k)	SolvedTurbulentKineticEnergyk	Turbulence...	Scalar
Subgrid Kinetic Energy	SubgridKineticEnergy	Turbulence...	Scalar
Turbulent Dissipation Rate (Epsilon)	TurbulenceDissipationRate	Turbulence...	Scalar
Intermittency	TurbulenceIntermittency	Turbulence...	Scalar
Intermittency Ap	IntermittencyAp	Turbulence...	Scalar
Intermittency Source	IntermittencySource	Turbulence...	Scalar
Intermittency Effective	IntermittencyEffective	Turbulence...	Scalar
Intermittency Effective Smooth	IntermittencyEffectiveSmooth	Turbulence...	Scalar
Intermittency	Intermittency	Turbulence...	Scalar
Intermittency Smooth	IntermittencySmooth	Turbulence...	Scalar
Critical Reynolds Number Correlation	CriticalReynoldsNumberCorrelation	Turbulence...	Scalar
Momentum Thickness Re	MomentumThicknessRe	Turbulence...	Scalar
Mom. Thickn. Re Ap	MomThicknReAp	Turbulence...	Scalar
Mom. Thickn. Re Source	MomThicknReSource	Turbulence...	Scalar
Geometric Roughness Height	GeometricRoughnessHeight	Turbulence...	Scalar
Turb Diss Ap	TurbDissAp	Turbulence...	Scalar
Turb Diss Source	TurbDissSource	Turbulence...	Scalar
de-d	ded	Turbulence...	Vector
Reconstruction de/d	Reconstructionded	Turbulence...	Vector
Velocity Variance Scale (v2)	VelocityVarianceScalev2	Turbulence...	Scalar
Elliptic Relaxation Function	EllipticRelaxationFunction	Turbulence...	Scalar
Specific Dissipation Rate (Omega)	SpecificDissipationRateOmega	Turbulence...	Scalar
Spec Diss Ap	SpecDissAp	Turbulence...	Scalar
Spec Diss Source	SpecDissSource	Turbulence...	Scalar
Turbulent Length Scale	TurbulentLengthScale	Turbulence...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
Turbulent Time Scale	TurbulentTimeScale	Turbulence...	Scalar
Turbulent Velocity Scale	TurbulentVelocityScale	Turbulence...	Scalar
Production of k	Productionofk	Turbulence...	Scalar
Production of laminar k	Productionoflaminark	Turbulence...	Scalar
Production of Entropy	ProductionofEntropy	Turbulence...	Scalar
Modified Turbulent Viscosity	ModifiedTurbulenceViscosity	Turbulence...	Scalar
Modified Turb Viscosity Ap	ModifiedTurbViscosityAp	Turbulence...	Scalar
Modified Turb Viscosity Source	ModifiedTurbViscositySource	Turbulence...	Scalar
dnut-d	dnutd	Turbulence...	Vector
Reconstruction dnut/d	Reconstructiondnutd	Turbulence...	Vector
Turbulent Viscosity	TurbulenceViscosity	Turbulence...	Scalar
LES Subgrid Turbulent Viscosity	LESSubgridTurbulentViscosity	Turbulence...	Scalar
Turbulent Viscosity (large-scale)	TurbulentViscositylargescale	Turbulence...	Scalar
Turbulent Viscosity (small-scale)	TurbulentViscositysmallscale	Turbulence...	Scalar
Effective Viscosity	EffectiveViscosity	Turbulence...	Scalar
Turbulent Viscosity Ratio	TurbulenceViscosityRatio	Turbulence...	Scalar
Subgrid Turbulent Viscosity	SubgridTurbulentViscosity	Turbulence...	Scalar
Subgrid Effective Viscosity	SubgridEffectiveViscosity	Turbulence...	Scalar
Subgrid Turbulent Viscosity Ratio	SubgridTurbulentViscosityRatio	Turbulence...	Scalar
Subgrid Filter Length	SubgridFilterLength	Turbulence...	Scalar
Subgrid Test-Filter Length	SubgridTestFilterLength	Turbulence...	Scalar
Subgrid Dissipation Rate	SubgridDissipationRate	Turbulence...	Scalar
Subgrid Dynamic Viscosity Const	SubgridDynamicViscosityConst	Turbulence...	Scalar
Subgrid Dynamic Prandtl Number	SubgridDynamicPrandtlNumber	Turbulence...	Scalar
Subgrid Dynamic Sc of Fmean	SubgridDynamicScofFmean	Turbulence...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
Subgrid Dynamic Sc of Progress Variable	SubgridDynamicScofProgressVariable	Turbulence...	Scalar
Subgrid Dynamic Sc of	SubgridDynamicScof	Turbulence...	Scalar
Subtest Kinetic Energy	SubtestKineticEnergy	Turbulence...	Scalar
Effective Thermal Conductivity	EffectiveThermalConductivity	Turbulence...	Scalar
Effective Prandtl Number	EffectivePrandtlNumber	Turbulence...	Scalar
Wall Ystar	WallYstar	Turbulence...	Scalar
Wall Yplus	WallYplus	Turbulence...	Scalar
Turbulent Reynolds Number (Re_y)	TurbulentReynoldsNumberRe_y	Turbulence...	Scalar
Two Layer Blending Function	TwoLayerBlendingFunction	Turbulence...	Scalar
SST F1 Blending Function	SSTF1BlendingFunction	Turbulence...	Scalar
SST F2 Blending Function	SSTF2BlendingFunction	Turbulence...	Scalar
F-mu Damping Function	FmuDampingFunction	Turbulence...	Scalar
F-1 Damping Function	F1DampingFunction	Turbulence...	Scalar
F-2 Damping Function	F2DampingFunction	Turbulence...	Scalar
Turbulent Reynolds Number (Re_t)	TurbulentReynoldsNumberRe_t	Turbulence...	Scalar
Mean Deformation	MeanDeformation	Turbulence...	Scalar
Relative Strain	RelativeStrain	Turbulence...	Scalar
Spatial Spectral Bandwidth	SpatialSpectralBandwidth	Turbulence...	Scalar
Temporal Spectral Bandwidth	TemporalSpectralBandwidth	Turbulence...	Scalar
Rng Strainrate Term	RngStrainrateTerm	Turbulence...	Scalar
DES Length Scale	DESLengthScale	Turbulence...	Scalar
DES TKE Dissipation Multiplier	DESTKEDissipationMultiplier	Turbulence...	Scalar
Blending Function for GEKO	BlendingFunctionforGEKO	Turbulence...	Scalar
Built-in Correlation for GEKO Blending Function	BuiltinCorrelationforGEKOBlendingFunction	Turbulence...	Scalar
Shielding Function for SBES or SDES	ShieldingFunctionforSBESorSDES	Turbulence...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
Curvature Correction Function fr	CurvatureCorrectionFunctionfr	Turbulence...	Scalar
Machine Learning Tke Coefficient	MachineLearningTkeCoefficient	Turbulence...	Scalar
Machine Learning Omega Coefficient	MachineLearningOmegaCoefficient	Turbulence...	Scalar
Machine Learning Algebraic Transition Coefficient	MachineLearningAlgebraicTransitionCoefficient	Turbulence...	Scalar
SAS Term Strain Rate Magnitude	SASTermStrainRateMagnitude	Turbulence...	Scalar
LES Resolution Estimate	LESResolutionEstimate	Turbulence...	Scalar
Sound Pressure	SoundPressure	Acoustics...	Scalar
Sound Potential	SoundPotential	Acoustics...	Scalar
Sound dP/dt	SounddPdt	Acoustics...	Scalar
Sound d2P/dt2	Soundd2Pdt2	Acoustics...	Scalar
Sound WaveEq Model Source	SoundWaveEqModelSource	Acoustics...	Scalar
Sound WaveEq Model Source Smoothed	SoundWaveEqModelSourceSmoothed	Acoustics...	Scalar
Sound WaveEq Model Source Final	SoundWaveEqModelSourceFinal	Acoustics...	Scalar
Sound WaveEq Model Source Mask	SoundWaveEqModelSourceMask	Acoustics...	Scalar
Sound Sponge Layer Marker	SoundSpongeLayerMarker	Acoustics...	Scalar
Sound WaveEq dVar/d	SoundWaveEqdVard	Acoustics...	Vector
Sound WaveEq Reconstruction dVar/d	SoundWaveEqReconstructiondVard	Acoustics...	Vector
Sound WaveEq Ap	SoundWaveEqAp	Acoustics...	Scalar
Sound WaveEq Source	SoundWaveEqSource	Acoustics...	Scalar
Surface dpdt RMS	SurfacedpdtRMS	Acoustics...	Scalar
Acoustic Power Level (dB)	AcousticPowerLeveldB	Acoustics...	Scalar
Acoustic Power	AcousticPower	Acoustics...	Scalar
Jet Acoustic Power Level (dB)	JetAcousticPowerLeveldB	Acoustics...	Scalar
Jet Acoustic Power	JetAcousticPower	Acoustics...	Scalar
Surface Acoustic Power Level (dB)	SurfaceAcousticPowerLeveldB	Acoustics...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
Surface Acoustic Power	SurfaceAcousticPower	Acoustics...	Scalar
Lilley's Self-Noise Source	LilleysSelfNoiseSource	Acoustics...	Scalar
Lilley's Shear-Noise Source	LilleysShearNoiseSource	Acoustics...	Scalar
Lilley's Total Noise Source	LilleysTotalNoiseSource	Acoustics...	Scalar
LEE Self-Noise X-Source	LEESelfNoiseXSource	Acoustics...	Scalar
LEE Shear-Noise X-Source	LEEShearNoiseXSource	Acoustics...	Scalar
LEE Total Noise X-Source	LEETotalNoiseXSource	Acoustics...	Scalar
LEE Self-Noise Y-Source	LEESelfNoiseYSource	Acoustics...	Scalar
LEE Shear-Noise Y-Source	LEEShearNoiseYSource	Acoustics...	Scalar
LEE Total Noise Y-Source	LEETotalNoiseYSource	Acoustics...	Scalar
LEE Self-Noise Z-Source	LEESelfNoiseZSource	Acoustics...	Scalar
LEE Shear-Noise Z-Source	LEEShearNoiseZSource	Acoustics...	Scalar
LEE Total Noise Z-Source	LEETotalNoiseZSource	Acoustics...	Scalar
SPL for Octave Band at 1Hz (dB)	SPLforOctaveBandat1HzdB	Acoustics...	Scalar
SPL for Octave Band at 2Hz (dB)	SPLforOctaveBandat2HzdB	Acoustics...	Scalar
SPL for Octave Band at 4Hz (dB)	SPLforOctaveBandat4HzdB	Acoustics...	Scalar
SPL for Octave Band at 8Hz (dB)	SPLforOctaveBandat8HzdB	Acoustics...	Scalar
SPL for Octave Band at 16Hz (dB)	SPLforOctaveBandat16HzdB	Acoustics...	Scalar
SPL for Octave Band at 31.5Hz (dB)	SPLforOctaveBandat315HzdB	Acoustics...	Scalar
SPL for Octave Band at 63Hz (dB)	SPLforOctaveBandat63HzdB	Acoustics...	Scalar
SPL for Octave Band at 125Hz (dB)	SPLforOctaveBandat125HzdB	Acoustics...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
SPL for Octave Band at 250Hz (dB)	SPLforOctaveBandat250HzdB	Acoustics...	Scalar
SPL for Octave Band at 500Hz (dB)	SPLforOctaveBandat500HzdB	Acoustics...	Scalar
SPL for Octave Band at 1kHz (dB)	SPLforOctaveBandat1kHzdB	Acoustics...	Scalar
SPL for Octave Band at 2kHz (dB)	SPLforOctaveBandat2kHzdB	Acoustics...	Scalar
SPL for Octave Band at 4kHz (dB)	SPLforOctaveBandat4kHzdB	Acoustics...	Scalar
SPL for Octave Band at 8kHz (dB)	SPLforOctaveBandat8kHzdB	Acoustics...	Scalar
SPL for Octave Band at 16kHz (dB)	SPLforOctaveBandat16kHzdB	Acoustics...	Scalar
SPL for Octave Band at 31.5kHz (dB)	SPLforOctaveBandat315kHzdB	Acoustics...	Scalar
SPL for Octave Band at 63kHz (dB)	SPLforOctaveBandat63kHzdB	Acoustics...	Scalar
PSD of dp/dt for Octave Band at 1Hz	PSDofdpdtforOctaveBandat1Hz	Acoustics...	Scalar
PSD of dp/dt for Octave Band at 2Hz	PSDofdpdtforOctaveBandat2Hz	Acoustics...	Scalar
PSD of dp/dt for Octave Band at 4Hz	PSDofdpdtforOctaveBandat4Hz	Acoustics...	Scalar
PSD of dp/dt for Octave Band at 8Hz	PSDofdpdtforOctaveBandat8Hz	Acoustics...	Scalar
PSD of dp/dt for Octave Band at 16Hz	PSDofdpdtforOctaveBandat16Hz	Acoustics...	Scalar
PSD of dp/dt for Octave Band at 31.5Hz	PSDofdpdtforOctaveBandat315Hz	Acoustics...	Scalar
PSD of dp/dt for Octave Band at 63Hz	PSDofdpdtforOctaveBandat63Hz	Acoustics...	Scalar
PSD of dp/dt for Octave Band at 125Hz	PSDofdpdtforOctaveBandat125Hz	Acoustics...	Scalar
PSD of dp/dt for Octave Band at 250Hz	PSDofdpdtforOctaveBandat250Hz	Acoustics...	Scalar
PSD of dp/dt for Octave Band at 500Hz	PSDofdpdtforOctaveBandat500Hz	Acoustics...	Scalar
PSD of dp/dt for Octave Band at 1kHz	PSDofdpdtforOctaveBandat1kHz	Acoustics...	Scalar
PSD of dp/dt for Octave Band at 2kHz	PSDofdpdtforOctaveBandat2kHz	Acoustics...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
PSD of dp/dt for Octave Band at 4kHz	PSDofdpdtforOctaveBandat4kHz	Acoustics...	Scalar
PSD of dp/dt for Octave Band at 8kHz	PSDofdpdtforOctaveBandat8kHz	Acoustics...	Scalar
PSD of dp/dt for Octave Band at 16kHz	PSDofdpdtforOctaveBandat16kHz	Acoustics...	Scalar
PSD of dp/dt for Octave Band at 31.5kHz	PSDofdpdtforOctaveBandat315kHz	Acoustics...	Scalar
PSD of dp/dt for Octave Band at 63kHz	PSDofdpdtforOctaveBandat63kHz	Acoustics...	Scalar
SPL for 1/3-Octave Band at 0.5Hz (dB)	SPLfor13OctaveBandat05HzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 0.63Hz (dB)	SPLfor13OctaveBandat063HzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 0.8Hz (dB)	SPLfor13OctaveBandat08HzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 1Hz (dB)	SPLfor13OctaveBandat1HzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 1.25Hz (dB)	SPLfor13OctaveBandat125HzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 1.6Hz (dB)	SPLfor13OctaveBandat16HzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 2Hz (dB)	SPLfor13OctaveBandat2HzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 2.5Hz (dB)	SPLfor13OctaveBandat25HzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 3.15Hz (dB)	SPLfor13OctaveBandat315HzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 4Hz (dB)	SPLfor13OctaveBandat4HzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 5Hz (dB)	SPLfor13OctaveBandat5HzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 6.3Hz (dB)	SPLfor13OctaveBandat63HzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 8Hz (dB)	SPLfor13OctaveBandat8HzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 10Hz (dB)	SPLfor13OctaveBandat10HzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 12.5Hz (dB)	SPLforOneThirdOctaveBandat12p5HzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 16Hz (dB)	SPLforOneThirdOctaveBandat16HzdB	Acoustics...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
SPL for 1/3-Octave Band at 20Hz (dB)	SPLfor13OctaveBandat20HzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 25Hz (dB)	SPLforOneThirdOctaveBandat25HzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 31.5Hz (dB)	SPLforOneThirdOctaveBandat31p5HzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 40Hz (dB)	SPLfor13OctaveBandat40HzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 50Hz (dB)	SPLfor13OctaveBandat50HzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 63Hz (dB)	SPLforOneThirdOctaveBandat63HzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 80Hz (dB)	SPLfor13OctaveBandat80HzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 100Hz (dB)	SPLfor13OctaveBandat100HzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 125Hz (dB)	SPLforOneThirdOctaveBandat125HzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 160Hz (dB)	SPLfor13OctaveBandat160HzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 200Hz (dB)	SPLfor13OctaveBandat200HzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 250Hz (dB)	SPLfor13OctaveBandat250HzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 315Hz (dB)	SPLforOneThirdOctaveBandat315HzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 400Hz (dB)	SPLfor13OctaveBandat400HzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 500Hz (dB)	SPLfor13OctaveBandat500HzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 630Hz (dB)	SPLfor13OctaveBandat630HzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 800Hz (dB)	SPLfor13OctaveBandat800HzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 1kHz (dB)	SPLfor13OctaveBandat1kHzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 1.25kHz (dB)	SPLfor13OctaveBandat125kHzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 1.6kHz (dB)	SPLfor13OctaveBandat16kHzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 2kHz (dB)	SPLfor13OctaveBandat2kHzdB	Acoustics...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
SPL for 1/3-Octave Band at 2.5kHz (dB)	SPLfor13OctaveBandat25kHzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 3.15kHz (dB)	SPLfor13OctaveBandat315kHzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 4kHz (dB)	SPLfor13OctaveBandat4kHzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 5kHz (dB)	SPLfor13OctaveBandat5kHzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 6.3kHz (dB)	SPLfor13OctaveBandat63kHzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 8kHz (dB)	SPLfor13OctaveBandat8kHzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 10kHz (dB)	SPLfor13OctaveBandat10kHzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 12.5kHz (dB)	SPLforOneThirdOctaveBandat125kHzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 16kHz (dB)	SPLforOneThirdOctaveBandat16kHzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 20kHz (dB)	SPLfor13OctaveBandat20kHzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 25kHz (dB)	SPLforOneThirdOctaveBandat25kHzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 31.5kHz (dB)	SPLforOneThirdOctaveBandat315kHzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 40kHz (dB)	SPLfor13OctaveBandat40kHzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 50kHz (dB)	SPLfor13OctaveBandat50kHzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 63kHz (dB)	SPLforOneThirdOctaveBandat63kHzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 80kHz (dB)	SPLfor13OctaveBandat80kHzdB	Acoustics...	Scalar
SPL for 1/3-Octave Band at 100kHz (dB)	SPLfor13OctaveBandat100kHzdB	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 0.5Hz	PSDofdpdtfor13OctaveBandat05Hz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 0.63Hz	PSDofdpdtfor13OctaveBandat063Hz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 0.8Hz	PSDofdpdtfor13OctaveBandat08Hz	Acoustics...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
PSD of dp/dt for 1/3-Octave Band at 1Hz	PSDofdpdtfor13OctaveBandat1Hz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 1.25Hz	PSDofdpdtfor13OctaveBandat125Hz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 1.6Hz	PSDofdpdtfor13OctaveBandat16Hz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 2Hz	PSDofdpdtfor13OctaveBandat2Hz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 2.5Hz	PSDofdpdtfor13OctaveBandat25Hz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 3.15Hz	PSDofdpdtfor13OctaveBandat315Hz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 4Hz	PSDofdpdtfor13OctaveBandat4Hz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 5Hz	PSDofdpdtfor13OctaveBandat5Hz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 6.3Hz	PSDofdpdtfor13OctaveBandat63Hz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 8Hz	PSDofdpdtfor13OctaveBandat8Hz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 10Hz	PSDofdpdtfor13OctaveBandat10Hz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 12.5Hz	PSDofdpdtfor13OctaveBandat125Hz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 16Hz	PSDofdpdtfor13OctaveBandat16Hz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 20Hz	PSDofdpdtfor13OctaveBandat20Hz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 25Hz	PSDofdpdtfor13OctaveBandat25Hz	Acoustics...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
PSD of dp/dt for 1/3-Octave Band at 31.5Hz	PSDofdpdtfor13OctaveBandat315Hz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 40Hz	PSDofdpdtfor13OctaveBandat40Hz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 50Hz	PSDofdpdtfor13OctaveBandat50Hz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 63Hz	PSDofdpdtfor13OctaveBandat63Hz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 80Hz	PSDofdpdtfor13OctaveBandat80Hz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 100Hz	PSDofdpdtfor13OctaveBandat100Hz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 125Hz	PSDofdpdtfor13OctaveBandat125Hz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 160Hz	PSDofdpdtfor13OctaveBandat160Hz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 200Hz	PSDofdpdtfor13OctaveBandat200Hz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 250Hz	PSDofdpdtfor13OctaveBandat250Hz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 315Hz	PSDofdpdtfor13OctaveBandat315Hz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 400Hz	PSDofdpdtfor13OctaveBandat400Hz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 500Hz	PSDofdpdtfor13OctaveBandat500Hz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 630Hz	PSDofdpdtfor13OctaveBandat630Hz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 800Hz	PSDofdpdtfor13OctaveBandat800Hz	Acoustics...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
PSD of dp/dt for 1/3-Octave Band at 1kHz	PSDofdpdtfor13OctaveBandat1kHz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 1.25kHz	PSDofdpdtfor13OctaveBandat125kHz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 1.6kHz	PSDofdpdtfor13OctaveBandat16kHz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 2kHz	PSDofdpdtfor13OctaveBandat2kHz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 2.5kHz	PSDofdpdtfor13OctaveBandat25kHz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 3.15kHz	PSDofdpdtfor13OctaveBandat315kHz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 4kHz	PSDofdpdtfor13OctaveBandat4kHz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 5kHz	PSDofdpdtfor13OctaveBandat5kHz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 6.3kHz	PSDofdpdtfor13OctaveBandat63kHz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 8kHz	PSDofdpdtfor13OctaveBandat8kHz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 10kHz	PSDofdpdtfor13OctaveBandat10kHz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 12.5kHz	PSDofdpdtfor13OctaveBandat125kHz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 16kHz	PSDofdpdtfor13OctaveBandat16kHz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 20kHz	PSDofdpdtfor13OctaveBandat20kHz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 25kHz	PSDofdpdtfor13OctaveBandat25kHz	Acoustics...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
PSD of dp/dt for 1/3-Octave Band at 31.5kHz	PSDofdpdtfor13OctaveBandat315kHz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 40kHz	PSDofdpdtfor13OctaveBandat40kHz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 50kHz	PSDofdpdtfor13OctaveBandat50kHz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 63kHz	PSDofdpdtfor13OctaveBandat63kHz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 80kHz	PSDofdpdtfor13OctaveBandat80kHz	Acoustics...	Scalar
PSD of dp/dt for 1/3-Octave Band at 100kHz	PSDofdpdtfor13OctaveBandat100kHz	Acoustics...	Scalar
Pressure Spectrum Re 0	PressureSpectrumRe0	Acoustics...	Scalar
Pressure Spectrum Im 0	PressureSpectrumIm0	Acoustics...	Scalar
Pressure Spectrum Re 1	PressureSpectrumRe1	Acoustics...	Scalar
Pressure Spectrum Im 1	PressureSpectrumIm1	Acoustics...	Scalar
Pressure Spectrum Re 2	PressureSpectrumRe2	Acoustics...	Scalar
Pressure Spectrum Im 2	PressureSpectrumIm2	Acoustics...	Scalar
Pressure Spectrum Re 3	PressureSpectrumRe3	Acoustics...	Scalar
Pressure Spectrum Im 3	PressureSpectrumIm3	Acoustics...	Scalar
Pressure Spectrum Re 4	PressureSpectrumRe4	Acoustics...	Scalar
Pressure Spectrum Im 4	PressureSpectrumIm4	Acoustics...	Scalar
Pressure Spectrum Re 5	PressureSpectrumRe5	Acoustics...	Scalar
Pressure Spectrum Im 5	PressureSpectrumIm5	Acoustics...	Scalar
Pressure Spectrum Re 6	PressureSpectrumRe6	Acoustics...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
Pressure Spectrum Im 6	PressureSpectrumIm6	Acoustics...	Scalar
Pressure Spectrum Re 7	PressureSpectrumRe7	Acoustics...	Scalar
Pressure Spectrum Im 7	PressureSpectrumIm7	Acoustics...	Scalar
Pressure Spectrum Re 8	PressureSpectrumRe8	Acoustics...	Scalar
Pressure Spectrum Im 8	PressureSpectrumIm8	Acoustics...	Scalar
Pressure Spectrum Re 9	PressureSpectrumRe9	Acoustics...	Scalar
Pressure Spectrum Im 9	PressureSpectrumIm9	Acoustics...	Scalar
Pressure Spectrum Re 10	PressureSpectrumRe10	Acoustics...	Scalar
Pressure Spectrum Im 10	PressureSpectrumIm10	Acoustics...	Scalar
Pressure Spectrum Re 11	PressureSpectrumRe11	Acoustics...	Scalar
Pressure Spectrum Im 11	PressureSpectrumIm11	Acoustics...	Scalar
Pressure Spectrum Re 12	PressureSpectrumRe12	Acoustics...	Scalar
Pressure Spectrum Im 12	PressureSpectrumIm12	Acoustics...	Scalar
Pressure Spectrum Re 13	PressureSpectrumRe13	Acoustics...	Scalar
Pressure Spectrum Im 13	PressureSpectrumIm13	Acoustics...	Scalar
Pressure Spectrum Re 14	PressureSpectrumRe14	Acoustics...	Scalar
Pressure Spectrum Im 14	PressureSpectrumIm14	Acoustics...	Scalar
Pressure Spectrum Re 15	PressureSpectrumRe15	Acoustics...	Scalar
Pressure Spectrum Im 15	PressureSpectrumIm15	Acoustics...	Scalar
Pressure Spectrum Re 16	PressureSpectrumRe16	Acoustics...	Scalar
Pressure Spectrum Im 16	PressureSpectrumIm16	Acoustics...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
Pressure Spectrum Re 17	PressureSpectrumRe17	Acoustics...	Scalar
Pressure Spectrum Im 17	PressureSpectrumIm17	Acoustics...	Scalar
Pressure Spectrum Re 18	PressureSpectrumRe18	Acoustics...	Scalar
Pressure Spectrum Im 18	PressureSpectrumIm18	Acoustics...	Scalar
Pressure Spectrum Re 19	PressureSpectrumRe19	Acoustics...	Scalar
Pressure Spectrum Im 19	PressureSpectrumIm19	Acoustics...	Scalar
SPL for Const Width Band 0 (dB)	SPLforConstWidthBand0dB	Acoustics...	Scalar
SPL for Const Width Band 1 (dB)	SPLforConstWidthBand1dB	Acoustics...	Scalar
SPL for Const Width Band 2 (dB)	SPLforConstWidthBand2dB	Acoustics...	Scalar
SPL for Const Width Band 3 (dB)	SPLforConstWidthBand3dB	Acoustics...	Scalar
SPL for Const Width Band 4 (dB)	SPLforConstWidthBand4dB	Acoustics...	Scalar
SPL for Const Width Band 5 (dB)	SPLforConstWidthBand5dB	Acoustics...	Scalar
SPL for Const Width Band 6 (dB)	SPLforConstWidthBand6dB	Acoustics...	Scalar
SPL for Const Width Band 7 (dB)	SPLforConstWidthBand7dB	Acoustics...	Scalar
SPL for Const Width Band 8 (dB)	SPLforConstWidthBand8dB	Acoustics...	Scalar
SPL for Const Width Band 9 (dB)	SPLforConstWidthBand9dB	Acoustics...	Scalar
SPL for Const Width Band 10 (dB)	SPLforConstWidthBand10dB	Acoustics...	Scalar
SPL for Const Width Band 11 (dB)	SPLforConstWidthBand11dB	Acoustics...	Scalar
SPL for Const Width Band 12 (dB)	SPLforConstWidthBand12dB	Acoustics...	Scalar
SPL for Const Width Band 13 (dB)	SPLforConstWidthBand13dB	Acoustics...	Scalar
SPL for Const Width Band 14 (dB)	SPLforConstWidthBand14dB	Acoustics...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
SPL for Const Width Band 15 (dB)	SPLforConstWidthBand15dB	Acoustics...	Scalar
SPL for Const Width Band 16 (dB)	SPLforConstWidthBand16dB	Acoustics...	Scalar
SPL for Const Width Band 17 (dB)	SPLforConstWidthBand17dB	Acoustics...	Scalar
SPL for Const Width Band 18 (dB)	SPLforConstWidthBand18dB	Acoustics...	Scalar
SPL for Const Width Band 19 (dB)	SPLforConstWidthBand19dB	Acoustics...	Scalar
PSD of dp/dt for Const Width Band 0	PSDofdpdtforConstWidthBand0	Acoustics...	Scalar
PSD of dp/dt for Const Width Band 1	PSDofdpdtforConstWidthBand1	Acoustics...	Scalar
PSD of dp/dt for Const Width Band 2	PSDofdpdtforConstWidthBand2	Acoustics...	Scalar
PSD of dp/dt for Const Width Band 3	PSDofdpdtforConstWidthBand3	Acoustics...	Scalar
PSD of dp/dt for Const Width Band 4	PSDofdpdtforConstWidthBand4	Acoustics...	Scalar
PSD of dp/dt for Const Width Band 5	PSDofdpdtforConstWidthBand5	Acoustics...	Scalar
PSD of dp/dt for Const Width Band 6	PSDofdpdtforConstWidthBand6	Acoustics...	Scalar
PSD of dp/dt for Const Width Band 7	PSDofdpdtforConstWidthBand7	Acoustics...	Scalar
PSD of dp/dt for Const Width Band 8	PSDofdpdtforConstWidthBand8	Acoustics...	Scalar
PSD of dp/dt for Const Width Band 9	PSDofdpdtforConstWidthBand9	Acoustics...	Scalar
PSD of dp/dt for Const Width Band 10	PSDofdpdtforConstWidthBand10	Acoustics...	Scalar
PSD of dp/dt for Const Width Band 11	PSDofdpdtforConstWidthBand11	Acoustics...	Scalar
PSD of dp/dt for Const Width Band 12	PSDofdpdtforConstWidthBand12	Acoustics...	Scalar
PSD of dp/dt for Const Width Band 13	PSDofdpdtforConstWidthBand13	Acoustics...	Scalar
PSD of dp/dt for Const Width Band 14	PSDofdpdtforConstWidthBand14	Acoustics...	Scalar
PSD of dp/dt for Const Width Band 15	PSDofdpdtforConstWidthBand15	Acoustics...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
PSD of dp/dt for Const Width Band 16	PSDofdpdtforConstWidthBand16	Acoustics...	Scalar
PSD of dp/dt for Const Width Band 17	PSDofdpdtforConstWidthBand17	Acoustics...	Scalar
PSD of dp/dt for Const Width Band 18	PSDofdpdtforConstWidthBand18	Acoustics...	Scalar
PSD of dp/dt for Const Width Band 19	PSDofdpdtforConstWidthBand19	Acoustics...	Scalar
Mass fraction of	MassFraction	Species...	Scalar
Conserved Mass fraction of	ConservedMassFraction	Species...	Scalar
Liquid species mass fraction of	LiquidSpeciesMassFraction	Species...	Scalar
Mole fraction of	MoleFraction	Species...	Scalar
Mass Concentration of (Phase Level)	MassConcentrationofPhaseLevel	Species...	Scalar
Mass Concentration of (Mixture Level)	MassConcentrationofMixtureLevel	Species...	Scalar
Molar Concentration of	MolarConcentration	Species...	Scalar
Lam Diff Coef of	LamDiffCoefof	Species...	Scalar
Eff Diff Coef of	EffDiffCoefof	Species...	Scalar
Thermal Diff Coef of	ThermalDiffCoefof	Species...	Scalar
Specific Heat of	SpecificHeatof	Species...	Scalar
Enthalpy of	Enthalpyof	Species...	Scalar
Entropy of	Entropyof	Species...	Scalar
Source Term	SourceTerm	Species...	Scalar
Surface Deposition Rate of	SurfaceDepositionRate	Species...	Scalar
Porous Deposition Rate of	PorousDepositionRate	Species...	Scalar
Surface Coverage of	SurfaceCoverage	Species...	Scalar
Bulk Activity of	BulkActivity	Species...	Scalar
Surface Corrosion Rate of	SurfaceCorrosionRate	Species...	Scalar
Relative Humidity	RelativeHumidity	Species...	Scalar
Time Step Scale	TimeStepScale	Species...	Scalar
Fine Scale Mass fraction of	FineScaleMassfractionof	Species...	Scalar
Equilibrium Mass fraction of	EquilibriumMassfractionof	Species...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
Cell Time Scale	CellTimeScale	Species...	Scalar
EDC Cell Volume Fraction	EDCCellVolumeFraction	Species...	Scalar
Chemical Time Scale	ChemicalTimeScale	Species...	Scalar
Turbulent Time Scale	SpeciesTurbulentTimeScale	Species...	Scalar
Rich Flammability Factor	RichFlammabilityFactor	Species...	Scalar
Mean Mass fraction	MeanMassfraction	Species...	Scalar
RMS Mass fraction	RMSMassfraction	Species...	Scalar
Schmidt Number	SchmidtNumber	Species...	Scalar
TFM Thickening Factor	TFMThickeningFactor	Species...	Scalar
TFM Omega	TFMOmega	Species...	Scalar
TFM Efficiency Factor	TFMEfficiencyFactor	Species...	Scalar
Laminar Flame Speed	LaminarFlameSpeed	Species...	Scalar
Laminar Flame Thickness	LaminarFlameThickness	Species...	Scalar
DRG Reduced Number of Species	DRGReducedNumberofSpecies	Species...	Scalar
Net Reaction Rate of	NetReactionRateof	Reactions...	Scalar
Kinetic Rate of	KineticRate	Reactions...	Scalar
Heat of Reaction	HeatofReaction	Reactions...	Scalar
DRG Reduced Number of Reactions	DRGReducedNumberofReactions	Reactions...	Scalar
Reactor Net Zone ID	ReactorNetZoneID	Reactor Network...	Scalar
Reactor Net Temperature	ReactorNetTemperature	Reactor Network...	Scalar
Mean Mixture Fraction	MeanMixtureFraction	Pdf...	Scalar
Fmean Ap	FmeanAp	Pdf...	Scalar
Fmean Source	FmeanSource	Pdf...	Scalar
Secondary Mean Mixture Fraction	SecondaryMeanMixtureFraction	Pdf...	Scalar
Fmean2 Ap	Fmean2Ap	Pdf...	Scalar
Fmean2 Source	Fmean2Source	Pdf...	Scalar
Mixture Fraction Variance	MixtureFractionVariance	Pdf...	Scalar
Fvar Ap	FvarAp	Pdf...	Scalar
Fvar Source	FvarSource	Pdf...	Scalar
Secondary Mixture Fraction Variance	SecondaryMixtureFractionVariance	Pdf...	Scalar
Fvar2 Ap	Fvar2Ap	Pdf...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
Fvar2 Source	Fvar2Source	Pdf...	Scalar
Fvar Prod	FvarProd	Pdf...	Scalar
Fvar2 Prod	Fvar2Prod	Pdf...	Scalar
Scalar Dissipation	ScalarDissipation	Pdf...	Scalar
Heat Release Rate	HeatReleaseRate	Pdf...	Scalar
PDF Table Adiabatic Enthalpy	PDFTableAdiabaticEnthalpy	Pdf...	Scalar
PDF Table Heat Loss/Gain	PDFTableHeatLossGain	Pdf...	Scalar
PDF Departure Enthalpy	PDFDepartureEnthalpy	Pdf...	Scalar
Inertial-Convective Variance	InertialConvectiveVariance	Pdf...	Scalar
Viscous-Convective Variance	ViscousConvectiveVariance	Pdf...	Scalar
Viscous-Diffusive Variance	ViscousDiffusiveVariance	Pdf...	Scalar
Progress Variable	ProgressVariable	Premixed Combustion...	Scalar
Progress Variable Variance	ProgressVariableVariance	Premixed Combustion...	Scalar
Progress Variable Ap	ProgressVariableAp	Premixed Combustion...	Scalar
Progress Variable Source	ProgressVariableSource	Premixed Combustion...	Scalar
G-equation Ap	GequationAp	Premixed Combustion...	Scalar
G-equation Source	GequationSource	Premixed Combustion...	Scalar
G-variance Ap	GvarianceAp	Premixed Combustion...	Scalar
G-variance Source	GvarianceSource	Premixed Combustion...	Scalar
Damkohler Number	DamkohlerNumber	Premixed Combustion...	Scalar
Stretch Factor	StretchFactor	Premixed Combustion...	Scalar
Turbulent Flame Speed	TurbulentFlameSpeed	Premixed Combustion...	Scalar
Unburnt Thermal Diffusivity	UnburntThermalDiffusivity	Premixed Combustion...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
Unburnt Density	UnburntDensity	Premixed Combustion...	Scalar
Static Temperature	PDFStaticTemperature	Premixed Combustion...	Scalar
Product Formation Rate	ProductFormationRate	Premixed Combustion...	Scalar
Laminar Flame Speed	PDFLaminarFlameSpeed	Premixed Combustion...	Scalar
Strained Laminar Flame Speed	StrainedLaminarFlameSpeed	Premixed Combustion...	Scalar
Total Strain Rate	TotalStrainRate	Premixed Combustion...	Scalar
Critical Strain Rate	CriticalStrainRate	Premixed Combustion...	Scalar
Adiabatic Flame Temperature	AdiabaticFlameTemperature	Premixed Combustion...	Scalar
Unburnt Fuel Mass Fraction	UnburntFuelMassFraction	Premixed Combustion...	Scalar
Progress Variable Gradient Magnitude	ProgressVariableGradientMagnitude	Premixed Combustion...	Scalar
Flame Area Density	FlameAreaDensity	Premixed Combustion...	Scalar
Net Flame Area Production	NetFlameAreaProduction	Premixed Combustion...	Scalar
Flame Area Production P1	FlameAreaProductionP1	Premixed Combustion...	Scalar
Flame Area Production P2	FlameAreaProductionP2	Premixed Combustion...	Scalar
Flame Area Production P3	FlameAreaProductionP3	Premixed Combustion...	Scalar
Flame Area Production P4	FlameAreaProductionP4	Premixed Combustion...	Scalar
Intermittent Turb Net Flame Stretch	IntermittentTurbNetFlameStretch	Premixed Combustion...	Scalar
Flame Area Destruction	FlameAreaDestruction	Premixed Combustion...	Scalar
Mean Distance from Flame	MeanDistancefromFlame	Premixed Combustion...	Scalar
Variance of Distance from Flame	VarianceofDistancefromFlame	Premixed Combustion...	Scalar
Un-Normalized Progress Variable	UnNormalizedProgressVariable	Premixed Combustion...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
Un-Normalized Progress Variable Variance	UnNormalizedProgressVariableVariance	Premixed Combustion...	Scalar
Finite-Rate Source	FiniteRateSource	Premixed Combustion...	Scalar
Turbulent Flame Speed Source	TurbulentFlameSpeedSource	Premixed Combustion...	Scalar
Dynamic Algebraic Variance Constant	DynamicAlgebraicVarianceConstant	Premixed Combustion...	Scalar
Un-Normalized Progress Variable Burned	UnNormalizedProgressVariableBurned	Premixed Combustion...	Scalar
Un-Normalized Progress Variable Unburned	UnNormalizedProgressVariableUnburned	Premixed Combustion...	Scalar
Flame Diffusion Enhancement	FlameDiffusionEnhancement	Premixed Combustion...	Scalar
Flame Marker	FlameMarker	Premixed Combustion...	Scalar
Finite Rate Source Enhancement	FiniteRateSourceEnhancement	Premixed Combustion...	Scalar
Q	Q	PDF Transport...	Scalar
Q Time-Average	QTimeAverage	PDF Transport...	Scalar
PHI	PHI	PDF Transport...	Scalar
U Correction Velocity	UCorrectionVelocity	PDF Transport...	Scalar
V Correction Velocity	VCorrectionVelocity	PDF Transport...	Scalar
W Correction Velocity	WCorrectionVelocity	PDF Transport...	Scalar
Convection Time Step	ConvectionTimeStep	PDF Transport...	Scalar
Diffusion Time Step	DiffusionTimeStep	PDF Transport...	Scalar
Turbulent Time Step	TurbulentTimeStep	PDF Transport...	Scalar
Particle Time Step	ParticleTimeStep	PDF Transport...	Scalar
Mean Temperature	FlameletMeanTemperature	Unsteady Flamelet...	Scalar
Electric Potential	ElectricPotential	Potential...	Scalar
Electric Current Magnitude	ElectricCurrentMagnitude	Potential...	Scalar
Faradaic Voltage Jump	FaradaicVoltageJump	Potential...	Scalar
Faradaic Voltage Resistance	FaradaicVoltageResistance	Potential...	Scalar
Faradaic Current Density	FaradaicCurrentDensity	Potential...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
Electrode Surface Potential	ElectrodeSurfacePotential	Potential...	Scalar
Total Surface Corrosion Rate	TotalSurfaceCorrosionRate	Potential...	Scalar
Total Accumulated Corrosion Rate	TotalAccumulatedCorrosionRate	Potential...	Scalar
Joule Heat Source	JouleHeatSource	Potential...	Scalar
Faradaic Heat Source	FaradaicHeatSource	Potential...	Scalar
Total EChem Heat Source	TotalEChemHeatSource	Potential...	Scalar
Migration Diff Coef of	MigrationDiffCoefof	Potential...	Scalar
Lithium Concentration	LithiumConcentration	Lithium...	Scalar
Lithium Migration Source	LithiumMigrationSource	Lithium...	Scalar
Cathode Potential	CathodePotential	Battery Variables...	Scalar
Anode Potential	AnodePotential	Battery Variables...	Scalar
Passive Zone Potential	PassiveZonePotential	Battery Variables...	Scalar
Cell Voltage	CellVoltage	Battery Variables...	Scalar
Potential Field	PotentialField	Battery Variables...	Scalar
Current Magnitude	CurrentMagnitude	Battery Variables...	Scalar
Cathode Electrical Conductivity	CathodeElectricalConductivity	Battery Variables...	Scalar
Anode Electrical Conductivity	AnodeElectricalConductivity	Battery Variables...	Scalar
Passive Zone Electrical Conductivity	PassiveZoneElectricalConductivity	Battery Variables...	Scalar
Electrical Conductivity	BatteryElectricalConductivity	Battery Variables...	Scalar
State of Charge (SOC)	StateofChargeSOC	Battery Variables...	Scalar
Network Current	NetworkCurrent	Battery Variables...	Scalar
Network Voltage	NetworkVoltage	Battery Variables...	Scalar
Network State of Charge (SOC)	NetworkStateofChargeSOC	Battery Variables...	Scalar
Joule Heat Source	BatteryJouleHeatSource	Battery Variables...	Scalar
Echem Heat Source	EchemHeatSource	Battery Variables...	Scalar
Short-Circuit Heat Source	ShortCircuitHeatSource	Battery Variables...	Scalar
Thermal-Abuse Heat Source: SEI Decomposition	ThermalAbuseHeatSourceSEIDecomposition	Battery Variables...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
Thermal-Abuse Heat Source: Negative-Solvent	ThermalAbuseHeatSourceNegativeSolvent	Battery Variables...	Scalar
Thermal-Abuse Heat Source: Positive-Solvent	ThermalAbuseHeatSourcePositiveSolvent	Battery Variables...	Scalar
Thermal-Abuse Heat Source: Electrolyte Decomposition	ThermalAbuseHeatSourceElectrolyteDecomposition	Battery Variables...	Scalar
Thermal-Abuse Heat Source: Internal Short	ThermalAbuseHeatSourceInternalShort	Battery Variables...	Scalar
Thermal-Abuse Heat Source	ThermalAbuseHeatSource	Battery Variables...	Scalar
Total Heat Source	TotalHeatSource	Battery Variables...	Scalar
Short Current Source	ShortCurrentSource	Battery Variables...	Scalar
Echem Current Source	EchemCurrentSource	Battery Variables...	Scalar
Total Current Source	TotalCurrentSource	Battery Variables...	Scalar
Cell Cluster ID	CellClusterID	Battery Variables...	Scalar
Short-Circuit Resistance	ShortCircuitResistance	Battery Variables...	Scalar
Abuse: alpha	Abusealpha	Battery Variables...	Scalar
Abuse: Csei	AbuseCsei	Battery Variables...	Scalar
Abuse: Cpe	AbuseCpe	Battery Variables...	Scalar
Abuse: Cne	AbuseCne	Battery Variables...	Scalar
Abuse: Ce	AbuseCe	Battery Variables...	Scalar
Abuse: internal short	Abuseinternalshort	Battery Variables...	Scalar
X Current Density for phi+	XCurrentDensityforphiPlus	Battery Variables...	Scalar
Y Current Density for phi+	YCurrentDensityforphiPlus	Battery Variables...	Scalar
Z Current Density for phi+	ZCurrentDensityforphiPlus	Battery Variables...	Scalar
X Current Density for phi-	XCurrentDensityforphiMinus	Battery Variables...	Scalar
Y Current Density for phi-	YCurrentDensityforphiMinus	Battery Variables...	Scalar
Z Current Density for phi-	ZCurrentDensityforphiMinus	Battery Variables...	Scalar
X Current Density in Battery	XCurrentDensityinBattery	Battery Variables...	Scalar
Y Current Density in Battery	YCurrentDensityinBattery	Battery Variables...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
Z Current Density in Battery	ZCurrentDensityinBattery	Battery Variables...	Scalar
Newman P2D: Average ce in Cathode	NewmanP2DAverageceinCathode	Battery Variables...	Scalar
Newman P2D: Average ce in Separator	NewmanP2DAverageceinSeparator	Battery Variables...	Scalar
Newman P2D: Average ce in Anode	NewmanP2DAverageceinAnode	Battery Variables...	Scalar
Newman P2D: Average cs in Cathode	NewmanP2DAveragecsinCathode	Battery Variables...	Scalar
Newman P2D: Average cs in Anode	NewmanP2DAveragecsinAnode	Battery Variables...	Scalar
Newman P2D: Average phis in Cathode	NewmanP2DAveragephisinCathode	Battery Variables...	Scalar
Newman P2D: Average phis in Anode	NewmanP2DAveragephisinAnode	Battery Variables...	Scalar
Newman P2D: Average phi in electrolyte	NewmanP2DAveragephiinelectrolyte	Battery Variables...	Scalar
Newman P2D: phi_s-phi_e at anode-separator interface	NewmanP2Dphi_sphi_eatanodeseparatorinterface	Battery Variables...	Scalar
Translational-Rotational Temperature	TranslationalRotationalTemperature	Two-Temperature Model...	Scalar
Vibrational-Electronic Temperature	VibrationalElectronicTemperature	Two-Temperature Model...	Scalar
Internal Energy	InternalEnergy	Two-Temperature Model...	Scalar
Translational-Rotational Energy	TranslationalRotationalEnergy	Two-Temperature Model...	Scalar
Vibrational-Electronic Energy	VibrationalElectronicEnergy	Two-Temperature Model...	Scalar
Total Displacement	TotalDisplacement	Structure...	Scalar
X Displacement	XDisplacement	Structure...	Scalar
Y Displacement	YDisplacement	Structure...	Scalar
Z Displacement	ZDisplacement	Structure...	Scalar
Sigma XX	SigmaXX	Structure...	Scalar
Sigma YY	SigmaYY	Structure...	Scalar
Sigma XY	SigmaXY	Structure...	Scalar
Sigma ZZ	SigmaZZ	Structure...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
Sigma YZ	SigmaYZ	Structure...	Scalar
Sigma XZ	SigmaXZ	Structure...	Scalar
von Mises stress	vonMisesstress	Structure...	Scalar
Pollutant Correction Ap	PollutantCorrectionAp	NOx...	Scalar
Pollutant Correction Source	PollutantCorrectionSource	NOx...	Scalar
Concentration of	SiteConcentration	Soot...	Scalar
Site Fraction of	SiteFraction	Soot...	Scalar
Volume fraction	Volumefraction	Phases...	Scalar
Normalized volume fraction	Normalizedvolumefraction	Phases...	Scalar
Phase ID	PhaseID	Phases...	Scalar
Smoothed VOF	SmoothedVOF	Phases...	Scalar
Smoothed Curvature	SmoothedCurvature	Phases...	Scalar
Smoothed VOF Gradient-d	SmoothedVOFGradientd	Phases...	Vector
Smoothed VOF Gradient Magnitude	SmoothedVOFGradientMagnitude	Phases...	Scalar
Body-Force-d	BodyForced	Phases...	Vector
Body Force Magnitude	BodyForceMagnitude	Phases...	Scalar
Surface-Tension-Force-d	SurfaceTensionForced	Phases...	Vector
Surface Tension Force Magnitude	SurfaceTensionForceMagnitude	Phases...	Scalar
MP P Source	MPPSource	Phases...	Scalar
VOF Ap	VOFAp	Phases...	Scalar
VOF S	VOFS	Phases...	Scalar
VOF Gradient-d	VOFGradientd	Phases...	Vector
Artificial Interfacial Viscosity	ArtificialInterfacialViscosity	Phases...	Scalar
Max. Packing Limit	MaxPackingLimit	Phases...	Scalar
Heat of Heterogeneous Reaction	HeatofHeterogeneousReaction	Phase Interaction...	Scalar
DPM Accretion Rate	DPMAccretionRate	Discrete Phase Variables...	Scalar
DPM Accretion	DPMAccretion	Discrete Phase Variables...	Scalar
DPM Velocity Magnitude	DPMVelocityMagnitude	Discrete Phase Variables...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
DPM Absorption Coefficient	DPMAbsorptionCoefficient	Discrete Phase Variables...	Scalar
DPM Number Density	DPMNumberDensity	Discrete Phase Variables...	Scalar
DPM Volume Fraction	DPMVolumeFraction	Discrete Phase Variables...	Scalar
DPM Volume	DPMVolume	Discrete Phase Variables...	Scalar
DPM Number of Particles	DPMNumberofParticles	Discrete Phase Variables...	Scalar
DPM Number of Parcels	DPMNumberofParcels	Discrete Phase Variables...	Scalar
DPM Collision Rate	DPMCollisionRate	Discrete Phase Variables...	Scalar
DPM Emission	DPMEmission	Discrete Phase Variables...	Scalar
DPM Scattering Coefficient	DPMScatteringCoefficient	Discrete Phase Variables...	Scalar
DPM Concentration	DPMConcentration	Discrete Phase Variables...	Scalar
DPM Concentration	DPMSpeciesConcentration	Discrete Phase Variables...	Scalar
DPM Steps per Cell	DPMStepsperCell	Discrete Phase Variables...	Scalar
DPM Approaching Packing Limit	DPMApproachingPackingLimit	Discrete Phase Variables...	Scalar
DPM Wall Force	DPMWallForce	Discrete Phase Variables...	Vector
DPM Wall Normal Pressure	DPMWallNormalPressure	Discrete Phase Variables...	Scalar
DPM Liquid Volume Fraction	DPMLiquidVolumeFraction	Discrete Phase Variables...	Scalar
DPM Pressure	DPMPressure	Discrete Phase Variables...	Scalar
DPM Press Grad	DPMPressGrad	Discrete Phase Variables...	Vector
DPM Mass Source	DPMMassSource	Discrete Phase Sources...	Scalar
DPM Momentum Source	DPMMomentumSource	Discrete Phase Sources...	Vector
DPM Turbulent Kinetic Energy Source	DPMTurbulentKineticEnergySource	Discrete Phase Sources...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
DPM Turbulent Dissipation Source	DPMTurbulentDissipationSource	Discrete Phase Sources...	Scalar
DPM Specific Turbulent Dissipation Source	DPMSpecificTurbulentDissipationSource	Discrete Phase Sources...	Scalar
DPM UU Reynolds Stress Source	DPMUUReynoldsStressSource	Discrete Phase Sources...	Scalar
DPM VV Reynolds Stress Source	DPMVVReynoldsStressSource	Discrete Phase Sources...	Scalar
DPM WW Reynolds Stress Source	DPMWWReynoldsStressSource	Discrete Phase Sources...	Scalar
DPM UV Reynolds Stress Source	DPMUVReynoldsStressSource	Discrete Phase Sources...	Scalar
DPM VW Reynolds Stress Source	DPMVWReynoldsStressSource	Discrete Phase Sources...	Scalar
DPM UW Reynolds Stress Source	DPMUWReynoldsStressSource	Discrete Phase Sources...	Scalar
DPM Sensible Enthalpy Source	DPMSensibleEnthalpySource	Discrete Phase Sources...	Scalar
DPM Enthalpy Source	DPMEnthalpySource	Discrete Phase Sources...	Scalar
DPM Burnout	DPMBurnout	Discrete Phase Sources...	Scalar
DPM Evaporation / Devolatilization	DPMEvaporationDevolatilization	Discrete Phase Sources...	Scalar
DPM Source	DPMSource	Discrete Phase Sources...	Scalar
DPM Mixture Fraction Source	DPMMixtureFractionSource	Discrete Phase Sources...	Scalar
DPM Mixture Fraction Secondary Source	DPMMixtureFractionSecondarySource	Discrete Phase Sources...	Scalar
DPM Inert Source	DPMInertSource	Discrete Phase Sources...	Scalar
Wall Film Height	WallFilmHeight	Wall Film...	Scalar
Wall Film Impingement Mass Flux	WallFilmImpingementMassFlux	Wall Film...	Scalar
Wall Film Mass	WallFilmMass	Wall Film...	Scalar
Wall Film Mass	WallFilmMassM0	Wall Film...	Scalar
Wall Film Temperature	WallFilmTemperature	Wall Film...	Scalar
Wall Film Heat Flux	WallFilmHeatFlux	Wall Film...	Scalar
Wall Coverage	WallCoverage	Wall Film...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
Wall Film -Velocity	WallFilmVelocity	Wall Film...	Vector
Wall Film Velocity Magnitude	WallFilmVelocityMagnitude	Wall Film...	Scalar
Wall Leidenfrost Temperature	WallLeidenfrostTemperature	Wall Film...	Scalar
Ignition Variable	IgnitionVariable	Ignition...	Scalar
Ignition Variable Production	IgnitionVariableProduction	Ignition...	Scalar
Local Ignition Time	LocalIgnitionTime	Ignition...	Scalar
Scalar-	UDS	User Defined Scalars...	Scalar
Diffusion Coef. of Scalar-	UDSDiff	User Defined Scalars...	Scalar
Cells with realizable moments	Cellswithrealizablemoments	Population Balance Variables...	Scalar
DQMOM-m4	DQMOMm4	Population Balance Variables...	Scalar
Sources of DQMOM-m4	SourcesofDQMOMm4	Population Balance Variables...	Scalar
Sources of DQMOM-vof	SourcesofDQMOMvof	Population Balance Variables...	Scalar
Interfacial Area Concentration	InterfacialAreaConcentration	Interfacial Area Concentration...	Scalar
Level-set Function	LevelsetFunction	Level-set Function...	Scalar
UDM-%d	UDM	User Defined Memory...	Scalar
Molecular Viscosity	DynamicViscosity	Properties...	Scalar
Diameter	Diameter	Properties...	Scalar
Granular Conductivity	GranularConductivity	Properties...	Scalar
Granular Bulk Viscosity	GranularBulkViscosity	Properties...	Scalar
Frictional Viscosity	FrictionalViscosity	Properties...	Scalar
Relative Permeability	RelativePermeability	Properties...	Scalar
Thermal Conductivity	ThermalConductivity	Properties...	Scalar
Specific Heat (Cp)	SpecificHeatCapacity	Properties...	Scalar
Specific Heat Ratio (gamma)	SpecificHeatRatiogamma	Properties...	Scalar
Gas Constant (R)	GasConstantR	Properties...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
Molecular Prandtl Number	PrandtlNumber	Properties...	Scalar
Mean Molecular Weight	MolecularMass	Properties...	Scalar
Sound Speed	SpeedOfSound	Properties...	Scalar
Electrical Conductivity	ElectricalConductivity	Properties...	Scalar
Lithium Diffusivity	LithiumDiffusivity	Properties...	Scalar
Compressibility Factor	CompressibilityFactor	Properties...	Scalar
Reduced Temperature	ReducedTemperature	Properties...	Scalar
Reduced Pressure	ReducedPressure	Properties...	Scalar
Critical Temperature	CriticalTemperature	Properties...	Scalar
Critical Pressure	CriticalPressure	Properties...	Scalar
Acentric Factor	AcentricFactor	Properties...	Scalar
Critical Specific Volume	CriticalSpecificVolume	Properties...	Scalar
Spinodal Temperature	SpinodalTemperature	Properties...	Scalar
Subcritical Condition	SubcriticalCondition	Properties...	Scalar
Diffusivity Mesh Smoothing	DiffusivityMeshSmoothing	Properties...	Scalar
Liquid Mass Fraction	LiquidMassFraction	Wet Steam...	Scalar
Liquid Mass Generation Rate	LiquidMassGenerationRate	Wet Steam...	Scalar
Log10(Droplets Per Unit Volume)	Log10DropletsPerUnitVolume	Wet Steam...	Scalar
Log10(Droplets Nucleation Rate)	Log10DropletsNucleationRate	Wet Steam...	Scalar
Steam Density (Gas-Phase)	SteamDensityGasPhase	Wet Steam...	Scalar
Liquid Density (Liquid-Phase)	LiquidDensityLiquidPhase	Wet Steam...	Scalar
Mixture Density	MixtureDensity	Wet Steam...	Scalar
Saturation Ratio	SaturationRatio	Wet Steam...	Scalar
Saturation Pressure	SaturationPressure	Wet Steam...	Scalar
Saturation Temperature	SaturationTemperature	Wet Steam...	Scalar
Subcooled Vapor Temperature	SubcooledVaporTemperature	Wet Steam...	Scalar
Droplet Surface Tension	DropletSurfaceTension	Wet Steam...	Scalar
Droplet Critical Radius [um]	DropletCriticalRadiusum	Wet Steam...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
Droplet Average Radius [um]	DropletAverageRadiusum	Wet Steam...	Scalar
Droplet Growth Rate [um/s]	DropletGrowthRateums	Wet Steam...	Scalar
Inert Mass Fraction	InertMassFraction	Inert...	Scalar
Inert Specific Heat	InertSpecificHeat	Inert...	Scalar
Inert Density	InertDensity	Inert...	Scalar
Inert Enthalpy	InertEnthalpy	Inert...	Scalar
Pdf Enthalpy	PdfEnthalpy	Inert...	Scalar
Pdf Fmean	PdfFmean	Inert...	Scalar
Pdf Fvar	PdfFvar	Inert...	Scalar
Film Thickness	FilmThickness	Eulerian Wall Film...	Scalar
Film Mass	FilmMass	Eulerian Wall Film...	Scalar
Film Temperature	FilmTemperature	Eulerian Wall Film...	Scalar
Film -Velocity	FilmVelocity	Eulerian Wall Film...	Vector
Film Velocity Magnitude	FilmVelocityMagnitude	Eulerian Wall Film...	Scalar
Film Surface -Velocity	FilmSurfaceVelocity	Eulerian Wall Film...	Vector
Film Surface Velocity Magnitude	FilmSurfaceVelocityMagnitude	Eulerian Wall Film...	Scalar
Film Surface Temperature	FilmSurfaceTemperature	Eulerian Wall Film...	Scalar
Film Passive Scalar	FilmPassiveScalar	Eulerian Wall Film...	Scalar
Film Density	FilmDensity	Eulerian Wall Film...	Scalar
Film Effective Pressure	FilmEffectivePressure	Eulerian Wall Film...	Scalar
Film Coverage	FilmCoverage	Eulerian Wall Film...	Scalar
Film Courant Number	FilmCourantNumber	Eulerian Wall Film...	Scalar
Film Weber Number	FilmWeberNumber	Eulerian Wall Film...	Scalar
Film DPM Mass Source	FilmDPMMassSource	Eulerian Wall Film...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
Film DPM Energy Source	FilmDPMEnergySource	Eulerian Wall Film...	Scalar
Film DPM X-Momentum Source	FilmDPMXMomentumSource	Eulerian Wall Film...	Scalar
Film DPM Y-Momentum Source	FilmDPMYMomentumSource	Eulerian Wall Film...	Scalar
Film DPM Z-Momentum Source	FilmDPMZMomentumSource	Eulerian Wall Film...	Scalar
Film Stripping Weber Number	FilmStrippingWeberNumber	Eulerian Wall Film...	Scalar
Film Stripped Mass	FilmStrippedMass	Eulerian Wall Film...	Scalar
Film Stripped Diam	FilmStrippedDiam	Eulerian Wall Film...	Scalar
Film Stripping Rate	FilmStrippingRate	Eulerian Wall Film...	Scalar
Film Separated Mass	FilmSeparatedMass	Eulerian Wall Film...	Scalar
Film Separated Diam	FilmSeparatedDiam	Eulerian Wall Film...	Scalar
Film Separation Rate	FilmSeparationRate	Eulerian Wall Film...	Scalar
Film Phase Change Rate	FilmPhaseChangeRate	Eulerian Wall Film...	Scalar
Film Outflow Mass	FilmOutflowMass	Eulerian Wall Film...	Scalar
Film Secondary Phase Mass	FilmSecondaryPhaseMass	Eulerian Wall Film...	Scalar
Film Secondary Phase Collection Coef	FilmSecondaryPhaseCollectionCoef	Eulerian Wall Film...	Scalar
Film VOF Equivalent Thickness	FilmVOFEquivalentThickness	Eulerian Wall Film...	Scalar
Film Total Equivalent Thickness	FilmTotalEquivalentThickness	Eulerian Wall Film...	Scalar
Film Volume Fraction	FilmVolumeFraction	Eulerian Wall Film...	Scalar
Film Total Equivalent Volume Fraction	FilmTotalEquivalentVolumeFraction	Eulerian Wall Film...	Scalar
Film Height Fraction	FilmHeightFraction	Eulerian Wall Film...	Scalar
Film Total Equivalent Height Fraction	FilmTotalEquivalentHeightFraction	Eulerian Wall Film...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
Film VOF Mass Exchange	FilmVOFMassExchange	Eulerian Wall Film...	Scalar
Film VOF Exchanged Mass	FilmVOFExchangedMass	Eulerian Wall Film...	Scalar
Adjoint GEKO CSEP	AdjointGEKOCSEP	Sensitivities...	Scalar
Adjoint GEKO CNW	AdjointGEKOCNW	Sensitivities...	Scalar
Adjoint GEKO CMIX	AdjointGEKOCMIX	Sensitivities...	Scalar
Adjoint GEKO BF	AdjointGEKOBF	Sensitivities...	Scalar
Adjoint CCURV	AdjointCCURV	Sensitivities...	Scalar
TKE Source Coefficient	TKESourceCoefficient	Sensitivities...	Scalar
Omega Source Coefficient	OmegaSourceCoefficient	Sensitivities...	Scalar
Adjoint GEKO CSEP (Trained)	AdjointGEKOCSEPTrained	Sensitivities...	Scalar
Adjoint GEKO CNW (Trained)	AdjointGEKOCNWTrained	Sensitivities...	Scalar
Adjoint GEKO CMIX (Trained)	AdjointGEKOCMIXTrained	Sensitivities...	Scalar
Adjoint GEKO BF (Trained)	AdjointGEKOBFTrained	Sensitivities...	Scalar
Adjoint CCURV (Trained)	AdjointCCURVTrained	Sensitivities...	Scalar
TKE Source Coefficient (Trained)	TKESourceCoefficientTrained	Sensitivities...	Scalar
Omega Source Coefficient (Trained)	OmegaSourceCoefficientTrained	Sensitivities...	Scalar
Non-Equilibrium Parameter	NonEquilibriumParameter	Sensitivities...	Scalar
Second Invariant	SecondInvariant	Sensitivities...	Scalar
Third Invariant	ThirdInvariant	Sensitivities...	Scalar
Fourth Invariant	FourthInvariant	Sensitivities...	Scalar
Fifth Invariant	FifthInvariant	Sensitivities...	Scalar
Length Ratio	LengthRatio	Sensitivities...	Scalar
Turbulent Reynolds Number (Scaled)	TurbulentReynoldsNumberScaled	Sensitivities...	Scalar
Turbulent Viscosity Ratio (Scaled)	TurbulentViscosityRatioScaled	Sensitivities...	Scalar
Magnitude of Sensitivity to Body Forces (Cell Values)	MagnitudeofSensitivitytoBodyForcesCellValues	Sensitivities...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
Sensitivity to Body Force X-Component (Cell Values)	SensitivitytoBodyForceXComponentCellValues	Sensitivities...	Scalar
Sensitivity to Body Force Y-Component (Cell Values)	SensitivitytoBodyForceYComponentCellValues	Sensitivities...	Scalar
Sensitivity to Body Force Z-Component (Cell Values)	SensitivitytoBodyForceZComponentCellValues	Sensitivities...	Scalar
Sensitivity to Mass Sources (Cell Values)	SensitivitytoMassSourcesCellValues	Sensitivities...	Scalar
Sensitivity to Energy Sources (Cell Values)	SensitivitytoEnergySourcesCellValues	Sensitivities...	Scalar
Sensitivity to Turbulent Kinetic Energy Sources (Cell Values)	SensitivitytoTurbulentKineticEnergySourcesCellValues	Sensitivities...	Scalar
Sensitivity to Specific Dissipation Rate Sources (Cell Values)	SensitivitytoSpecificDissipationRateSourcesCellValues	Sensitivities...	Scalar
Magnitude of Sensitivity to Body Forces	MagnitudeofSensitivitytoBodyForces	Sensitivities...	Scalar
Sensitivity to Body Force X-Component	SensitivitytoBodyForceXComponent	Sensitivities...	Scalar
Sensitivity to Body Force Y-Component	SensitivitytoBodyForceYComponent	Sensitivities...	Scalar
Sensitivity to Body Force Z-Component	SensitivitytoBodyForceZComponent	Sensitivities...	Scalar
Sensitivity to Mass Sources	SensitivitytoMassSources	Sensitivities...	Scalar
Sensitivity to Energy Sources	SensitivitytoEnergySources	Sensitivities...	Scalar
Adjoint Ap	AdjointAp	Sensitivities...	Scalar
Artificial Dissipation	ArtificialDissipation	Sensitivities...	Scalar
Mesh Variation Effect	MeshVariationEffect	Sensitivities...	Scalar
Sensitivity To Viscosity	SensitivityToViscosity	Sensitivities...	Scalar
Shape Sensitivity Magnitude	ShapeSensitivityMagnitude	Sensitivities...	Scalar
Normal Shape Sensitivity	NormalShapeSensitivity	Sensitivities...	Scalar
Normal Surface Shape Sensitivity	NormalSurfaceShapeSensitivity	Sensitivities...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
Normal Optimal Displacement	NormalOptimalDisplacement	Sensitivities...	Scalar
Optimal Displacement X-Component	OptimalDisplacementXComponent	Sensitivities...	Scalar
Optimal Displacement Y-Component	OptimalDisplacementYComponent	Sensitivities...	Scalar
Optimal Displacement Z-Component	OptimalDisplacementZComponent	Sensitivities...	Scalar
Normal Displacement from Reference	NormalDisplacementfromReference	Sensitivities...	Scalar
Displacement from Reference X-Component	DisplacementfromReferenceXComponent	Sensitivities...	Scalar
Displacement from Reference Y-Component	DisplacementfromReferenceYComponent	Sensitivities...	Scalar
Displacement from Reference Z-Component	DisplacementfromReferenceZComponent	Sensitivities...	Scalar
log10(Shape Sensitivity Magnitude)	log10ShapeSensitivityMagnitude	Sensitivities...	Scalar
Shape Sensitivity Component	ShapeSensitivityComponent	Sensitivities...	Vector
Sensitivity to Boundary -Velocity	SensitivitytoBoundaryVelocity	Sensitivities...	Vector
Sensitivity to Boundary Pressure	SensitivitytoBoundaryPressure	Sensitivities...	Scalar
Sensitivity to Boundary Temperature	SensitivitytoBoundaryTemperature	Sensitivities...	Scalar
Sensitivity to Boundary Heat Flux	SensitivitytoBoundaryHeatFlux	Sensitivities...	Scalar
Sensitivity to Flow Blockage	SensitivitytoFlowBlockage	Sensitivities...	Scalar
Topology Density	TopologyDensity	Sensitivities...	Scalar
Topology Density (Reconstructed)	TopologyDensityReconstructed	Sensitivities...	Scalar
Optimal Topology Density Change	OptimalTopologyDensityChange	Sensitivities...	Scalar
Sensitivity to Topology Density	SensitivitytoTopologyDensity	Sensitivities...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
Sensitivity to Topology Density (Reconstructed)	SensitivitytoTopologyDensityReconstructed	Sensitivities...	Scalar
Sensitivity to GEKO CSEP	SensitivitytoGEKOCSEP	Sensitivities...	Scalar
Sensitivity to GEKO CNW	SensitivitytoGEKOCNW	Sensitivities...	Scalar
Sensitivity to GEKO CMIX	SensitivitytoGEKOCMIX	Sensitivities...	Scalar
Sensitivity to GEKO Blending Function	SensitivitytoGEKOBlendingFunction	Sensitivities...	Scalar
Sensitivity to CCURV	SensitivitytoCCURV	Sensitivities...	Scalar
Sensitivity to Algebraic Intermittency	SensitivitytoAlgebraicIntermittency	Sensitivities...	Scalar
Sensitivity to Critical Reynolds Number	SensitivitytoCriticalReynoldsNumber	Sensitivities...	Scalar
Sensitivity to TKE Source Coefficient	SensitivitytoTKESourceCoefficient	Sensitivities...	Scalar
Sensitivity to Omega Source Coefficient	SensitivitytoOmegaSourceCoefficient	Sensitivities...	Scalar
Sensitivity to Algebraic Transition Coefficient	SensitivitytoAlgebraicTransitionCoefficient	Sensitivities...	Scalar
Adjoint Local Solution Marker	AdjointLocalSolutionMarker	Sensitivities...	Scalar
Adjoint X-Velocity Residual	AdjointXVelocityResidual	Sensitivities...	Scalar
Adjoint Y-Velocity Residual	AdjointYVelocityResidual	Sensitivities...	Scalar
Adjoint Z-Velocity Residual	AdjointZVelocityResidual	Sensitivities...	Scalar
Adjoint Continuity Residual	AdjointContinuityResidual	Sensitivities...	Scalar
Adjoint Local Flow Rate Residual	AdjointLocalFlowRateResidual	Sensitivities...	Scalar
Adjoint Energy Residual	AdjointEnergyResidual	Sensitivities...	Scalar
Adjoint Turbulent Kinetic Energy Residual	AdjointTurbulentKineticEnergyResidual	Sensitivities...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
Adjoint Specific Dissipation Rate Residual	AdjointSpecificDissipationRateResidual	Sensitivities...	Scalar
Spark Heat Source	SparkHeatSource	Spark ...	Scalar
Spark Diffusion Multiplier	SparkDiffusionMultiplier	Spark ...	Scalar
Lump ID	LumpID	Lump Detection...	Scalar
Wall Shear Stress	WallShearStress	Wall Fluxes...	Scalar
Wall Shear Stress Vector	WallShearStressVector	Wall Fluxes...	Vector
Skin Friction Coefficient	SkinFrictionCoefficient	Wall Fluxes...	Scalar
Total Surface Heat Flux	SurfaceHeatFlux	Wall Fluxes...	Scalar
Evaporation Heat Flux	EvaporationHeatFlux	Wall Fluxes...	Scalar
Thin Film Heat Flux	ThinFilmHeatFlux	Wall Fluxes...	Scalar
Quenching Heat Flux	QuenchingHeatFlux	Wall Fluxes...	Scalar
Liquid Phase Heat Flux	LiquidPhaseHeatFlux	Wall Fluxes...	Scalar
Vapor Phase Heat Flux	VaporPhaseHeatFlux	Wall Fluxes...	Scalar
Radiation Heat Flux	RadiativeHeatFlux	Wall Fluxes...	Scalar
Surface Incident Radiation	SurfaceIncidentRadiation	Wall Fluxes...	Scalar
Absorbed Radiation Flux	AbsorbedRadiationFlux	Wall Fluxes...	Scalar
Reflected Radiation Flux	ReflectedRadiationFlux	Wall Fluxes...	Scalar
Transmitted Radiation Flux	TransmittedRadiationFlux	Wall Fluxes...	Scalar
Beam Irradiation Flux	BeamIrradiationFlux	Wall Fluxes...	Scalar
Solar Heat Flux	SolarHeatFlux	Wall Fluxes...	Scalar
Absorbed Visible Solar Flux	AbsorbedVisibleSolarFlux	Wall Fluxes...	Scalar
Absorbed IR Solar Flux	AbsorbedIRSolarFlux	Wall Fluxes...	Scalar
Reflected Visible Solar Flux	ReflectedVisibleSolarFlux	Wall Fluxes...	Scalar
Reflected IR Solar Flux	ReflectedIRSolarFlux	Wall Fluxes...	Scalar
Transmitted Visible Solar Flux	TransmittedVisibleSolarFlux	Wall Fluxes...	Scalar
Transmitted IR Solar Flux	TransmittedIRSolarFlux	Wall Fluxes...	Scalar
Solar BIN ID	SolarBINID	Wall Fluxes...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
Surface Heat Transfer Coef.	HeatTransferCoefficient	Wall Fluxes...	Scalar
Wall Adjacent Heat Transfer Coef.	WallAdjacentHeatTransferCoef	Wall Fluxes...	Scalar
Yplus Based Heat Tran. Coef.	YplusBasedHeatTranCoef	Wall Fluxes...	Scalar
Wall Func. Heat Tran. Coef.	SurfaceHeatTransferCoefficient	Wall Fluxes...	Scalar
Surface Nusselt Number	SurfaceNusseltNumber	Wall Fluxes...	Scalar
Surface Stanton Number	SurfaceStantonNumber	Wall Fluxes...	Scalar
Wall Surface Tension Force	WallSurfaceTensionForce	Wall Fluxes...	Scalar
Helicity	WallSurfaceTensionForce	Wall Fluxes...	Vector
Mean Pressure Coefficient	MeanPressureCoefficient	Unsteady Wall Statistics...	Scalar
Mean Wall Shear Stress	MeanWallShearStress	Unsteady Wall Statistics...	Scalar
Mean -Wall Shear Stress	MeanWallShearStressVector	Unsteady Wall Statistics...	Vector
Mean Skin Friction Coefficient	MeanSkinFrictionCoefficient	Unsteady Wall Statistics...	Scalar
Mean Surface Heat Flux	MeanSurfaceHeatFlux	Unsteady Wall Statistics...	Scalar
Mean Surface Heat Transfer Coef.	MeanSurfaceHeatTransferCoef	Unsteady Wall Statistics...	Scalar
SC Mean Thermal Data Heat Transfer Coef.	SCMeanThermalDataHeatTransferCoef	Unsteady Wall Statistics...	Scalar
SC Mean Thermal Data Ref Temp.	SCMeanThermalDataRefTemp	Unsteady Wall Statistics...	Scalar
Mean Surface Nusselt Number	MeanSurfaceNusseltNumber	Unsteady Wall Statistics...	Scalar
Mean Surface Stanton Number	MeanSurfaceStantonNumber	Unsteady Wall Statistics...	Scalar
Active Cell Partition	ActiveElementPartition	Cell Info...	Scalar
Stored Cell Partition	StoredElementPartition	Cell Info...	Scalar
Cell Id	CellId	Cell Info...	Scalar
Cell Index	CellIndex	Cell Info...	Scalar
Cell Machine	CellMachine	Cell Info...	Scalar
Cell Map	CellMap	Cell Info...	Scalar
Cell Element Type	CellElementType	Cell Info...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
Cell Zone Type	CellZoneType	Cell Info...	Scalar
Cell Zone Index	CellZoneIndex	Cell Info...	Scalar
Cell Zone Counter	CellZoneCounter	Cell Info...	Scalar
Connected Region	ConnectedRegion	Cell Info...	Scalar
Partition Neighbors	PartitionNeighbors	Cell Info...	Scalar
Cell Weight	CellWeight	Cell Info...	Scalar
DPM Domain Partition	DPMDomainPartition	Cell Info...	Scalar
Overset Cell Type	OversetCellType	Cell Info...	Scalar
Overset Donor Count	OversetDonorCount	Cell Info...	Scalar
Overset Receptor Count	OversetReceptorCount	Cell Info...	Scalar
Overset Bounding Cell Count	OversetBoundingCellCount	Cell Info...	Scalar
Overset Flood Fill Level	OversetFloodFillLevel	Cell Info...	Scalar
Gap ID	GapID	Cell Info...	Scalar
Gap Type	GapType	Cell Info...	Scalar
Flow-Blocking Gap Interface	FlowBlockingGapInterface	Cell Info...	Scalar
Contact Cell Mark	ContactCellMark	Cell Info...	Scalar
HTC Cell Mark	HTCCellMark	Cell Info...	Scalar
HTC LB Volume Fraction	HTCLBVolumeFraction	Cell Info...	Scalar
HTC LB Boundary Zones	HTCLBBoundaryZones	Cell Info...	Scalar
HTC LB Region	HTCLBRegion	Cell Info...	Scalar
HTC LB Region Count	HTCLBRegionCount	Cell Info...	Scalar
HTC LB Region 0	HTCLBRegion0	Cell Info...	Scalar
HTC LB Region 1	HTCLBRegion1	Cell Info...	Scalar
HTC LB Region 2	HTCLBRegion2	Cell Info...	Scalar
HTC LB Region Type 0	HTCLBRegionType0	Cell Info...	Scalar
HTC LB Region Type 1	HTCLBRegionType1	Cell Info...	Scalar
HTC LB Region Type 2	HTCLBRegionType2	Cell Info...	Scalar
Position	Position	Mesh...	Vector
Axial Coordinate	AxialCoordinate	Mesh...	Scalar
Angular Coordinate	AngularCoordinate	Mesh...	Scalar
Abs. Angular Coordinate	AbsAngularCoordinate	Mesh...	Scalar
Radial Coordinate	RadialCoordinate	Mesh...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
Meridional Coordinate	MeridionalCoordinate	Mesh...	Scalar
Abs Meridional Coordinate	AbsMeridionalCoordinate	Mesh...	Scalar
Spanwise Coordinate	SpanwiseCoordinate	Mesh...	Scalar
Abs (H-C) Spanwise Coordinate	AbsHCSpanwiseCoordinate	Mesh...	Scalar
Abs (C-H) Spanwise Coordinate	AbsCHSpanwiseCoordinate	Mesh...	Scalar
Pitchwise Coordinate	PitchwiseCoordinate	Mesh...	Scalar
Abs Pitchwise Coordinate	AbsPitchwiseCoordinate	Mesh...	Scalar
Face Area Magnitude	FaceAreaMagnitude	Mesh...	Scalar
X Face Area	XFaceArea	Mesh...	Scalar
Y Face Area	YFaceArea	Mesh...	Scalar
Z Face Area	ZFaceArea	Mesh...	Scalar
Cell Volume	CellVolume	Mesh...	Scalar
Orthogonal Quality	OrthogonalQuality	Mesh...	Scalar
Cell Equiangle Skew	CellEquiangleSkew	Mesh...	Scalar
Cell Equivolume Skew	CellEquivolumeSkew	Mesh...	Scalar
Face Skew	FaceSkew	Mesh...	Scalar
Face Squish Index	FaceSquishIndex	Mesh...	Scalar
Cell Squish Index	CellSquishIndex	Mesh...	Scalar
Face Handedness	FaceHandedness	Mesh...	Scalar
Accumulated Deformation	AccumulatedDeformation	Mesh...	Scalar
Gradient Quality Measure	GradientQualityMeasure	Mesh...	Scalar
Solution and Cell Quality Measure	SolutionandCellQualityMeasure	Mesh...	Scalar
Combined Cell Quality Residual Measure	CombinedCellQualityResidualMeasure	Mesh...	Scalar
Mark Poor Elements	MarkPoorElements	Mesh...	Scalar
Interface Overlap Fraction	InterfaceOverlapFraction	Mesh...	Scalar
Face Warpage	FaceWarpage	Mesh...	Scalar
Cell Wall Distance	ElementWallDistance	Mesh...	Scalar
Cell Boundary Distance Mesh Smoothing	CellBoundaryDistanceMeshSmoothing	Mesh...	Scalar
Cell Boundary Distance Overset	CellBoundaryDistanceOverset	Mesh...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
Node Pair Check	NodePairCheck	Mesh...	Scalar
Interface Wall IDs	InterfaceWallIDs	Mesh...	Scalar
Aspect Ratio	ElementAspectRatio	Mesh...	Scalar
Edge Length Ratio	EdgeLengthRatio	Mesh...	Scalar
Expansion Ratio	ExpansionRatio	Mesh...	Scalar
Cell Normalized Sum Face Area	CellNormalizedSumFaceArea	Mesh...	Scalar
Cell Aspect Ratio Tag	CellAspectRatioTag	Mesh...	Scalar
Cell AR	CellAR	Mesh...	Scalar
Cell AR Directional	CellARDirectional	Mesh...	Scalar
Cell AR Ratio	CellARRatio	Mesh...	Scalar
Cell Min d	CellMind	Mesh...	Scalar
Cell Min d Directional	CellMindDirectional	Mesh...	Scalar
Cell Min d Ratio	CellMindRatio	Mesh...	Scalar
Cell CFL Multiplier	CellCFLMultiplier	Mesh...	Scalar
Cell Volume Derivative	CellVolumeDerivative	Mesh...	Scalar
Cell Volume Error	CellVolumeError	Mesh...	Scalar
Dynamic Cell Volume Change	DynamicCellVolumeChange	Mesh...	Scalar
Dynamic Prism Face Cav. Marks	DynamicPrismFaceCavMarks	Mesh...	Scalar
Background Sizing Function	BackgroundSizingFunction	Mesh...	Scalar
Skew/Size Cav. Marking	SkewSizeCavMarking	Mesh...	Scalar
Skew/Size Cav. Mark Type	SkewSizeCavMarkType	Mesh...	Scalar
Skew/Size Cav. Seeds	SkewSizeCavSeeds	Mesh...	Scalar
Cell Surface Area	CellSurfaceArea	Mesh...	Scalar
Cell Warpage	CellWarpage	Mesh...	Scalar
Cell Refine Level	CellRefineLevel	Mesh...	Scalar
Cell Parent Index	CellParentIndex	Mesh...	Scalar
Additional Refinement Layers	AdditionalRefinementLayers	Mesh...	Scalar
Excluded Adaption Cells	ExcludedAdaptionCells	Mesh...	Scalar
Boundary Layer Cells	BoundaryLayerCells	Mesh...	Scalar
Boundary Cell Distance	BoundaryCellDistance	Mesh...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
Boundary Normal Distance	BoundaryNormalDistance	Mesh...	Scalar
Boundary Volume Distance	BoundaryVolumeDistance	Mesh...	Scalar
Cell Volume Change	CellVolumeChange	Mesh...	Scalar
Sponge Layer Distance	SpongeLayerDistance	Mesh...	Scalar
Sponge Blending Function	SpongeBlendingFunction	Mesh...	Scalar
Overset Donor Size Ratio	OversetDonorSizeRatio	Mesh...	Scalar
X Real Displacement	XRealDisplacement	Mesh...	Scalar
Y Real Displacement	YRealDisplacement	Mesh...	Scalar
Z Real Displacement	ZRealDisplacement	Mesh...	Scalar
Passage Number	PassageNumber	Mesh...	Scalar
X Imaginary Displacement	XImaginaryDisplacement	Mesh...	Scalar
Y Imaginary Displacement	YImaginaryDisplacement	Mesh...	Scalar
Z Imaginary Displacement	ZImaginaryDisplacement	Mesh...	Scalar
X Mesh Velocity	XMeshVelocity	Mesh...	Scalar
Y Mesh Velocity	YMeshVelocity	Mesh...	Scalar
Z Mesh Velocity	ZMeshVelocity	Mesh...	Scalar
X Relative Mesh Displacement	XRelativeMeshDisplacement	Mesh...	Scalar
Y Relative Mesh Displacement	YRelativeMeshDisplacement	Mesh...	Scalar
Z Relative Mesh Displacement	ZRelativeMeshDisplacement	Mesh...	Scalar
X Total Mesh Displacement	XTotalMeshDisplacement	Mesh...	Scalar
Y Total Mesh Displacement	YTotalMeshDisplacement	Mesh...	Scalar
Z Total Mesh Displacement	ZTotalMeshDisplacement	Mesh...	Scalar
Mass Imbalance	MassImbalance	Residuals...	Scalar
Time Step	Timestep	Residuals...	Scalar
Velocity Divergence	VelocityDivergence	Derivatives...	Scalar
Strain Rate	StrainRate	Derivatives...	Scalar
d-Velocity/dx	dVelocitydx	Derivatives...	Vector
d-Velocity/dy	dVelocitydy	Derivatives...	Vector

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Variable	Section	Type
d-Velocity/dz	dVelocitydz	Derivatives...	Vector
dT-d	dTd	Derivatives...	Vector
dH-d	dHd	Derivatives...	Vector
dp-d	dpd	Derivatives...	Vector
dp-dt	dpdt	Derivatives...	Scalar
drho-d	drhod	Derivatives...	Vector
dTKE/d	dTKEd	Derivatives...	Vector
dTED/d	dTEDd	Derivatives...	Vector
dSDR/d	dSDRd	Derivatives...	Vector
dvof-d	dvofd	Derivatives...	Vector
Granular Pressure G cmpn	GranularPressureGcmpn	Derivatives...	Vector
dMass Fraction -dx	dMassFractiondx	Derivatives...	Scalar
dMass Fraction -dy	dMassFractiondy	Derivatives...	Scalar
dMass Fraction -dz	dMassFractiondz	Derivatives...	Scalar
Hvisc Norm	HviscNorm	Derivatives...	Scalar
Time Gradient Limiter	TimeGradientLimiter	Derivatives...	Scalar
Beta PH	BetaPH	Derivatives...	Scalar
Shock Wave Identification Parameter	ShockWaveIdentificationParameter	Derivatives...	Scalar
Shock Wave Identification Parameter (Limited)	ShockWaveIdentificationParameterLimited	Derivatives...	Scalar
Shock Wave Identification Parameter (Non-Limited)	ShockWaveIdentificationParameterNonLimited	Derivatives...	Scalar
BCD Scheme Boundedness	BCDSchemeBoundedness	Derivatives...	Scalar
Pressure Hessian Indicator	PressureHessianIndicator	Derivatives...	Scalar
Reconstruction dvof/d	Reconstructiondvofd	Reconstruction...	Vector
Hrecon Norm	HreconNorm	Reconstruction...	Scalar
<quantity> tonal analysis magnitude	<quantity>-tonal-analysis-<#>-magnitude	Runtime DFT...	Scalar
<quantity> tonal analysis phase	<quantity>-tonal-analysis-<#>-phase	Runtime DFT...	Scalar

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 6: Reading and Manipulating Meshes

Ansys Fluent can import different types of meshes from various sources. You can modify the mesh by translating or scaling node coordinates, partitioning the domain for parallel processing, and merging or separating zones. You can convert all 3D meshes to polyhedral cells, except for pure hex meshes. Hexahedral cells are preserved during conversion. You can also obtain diagnostic information on the mesh, including memory usage and simplex, topological, and domain information. You can find out the number of nodes, faces, and cells in the mesh, determine the minimum and maximum cell volumes in the domain, and check for the proper numbers of nodes and faces per cell. These and other capabilities are described in the following sections.

6.1. Mesh Topologies

6.2. Mesh Requirements and Considerations

6.3. Mesh Sources

6.4. Reference Frames

6.5. Curvilinear Coordinate Systems

6.6. Non-Conformal Meshes

6.7. Overset Meshes

6.8. Controlling Flow in Narrow Gaps for Valves and Pumps

6.9. Checking the Mesh

6.10. Reporting Mesh Statistics

6.11. Converting the Mesh to a Polyhedral Mesh

6.12. Modifying the Mesh

See Adapting the Mesh (p. 3773) for information about adapting the mesh based on solution data and related functions, and Mesh Partitioning and Load Balancing (p. 4315) for details on partitioning the mesh for parallel processing.

6.1. Mesh Topologies

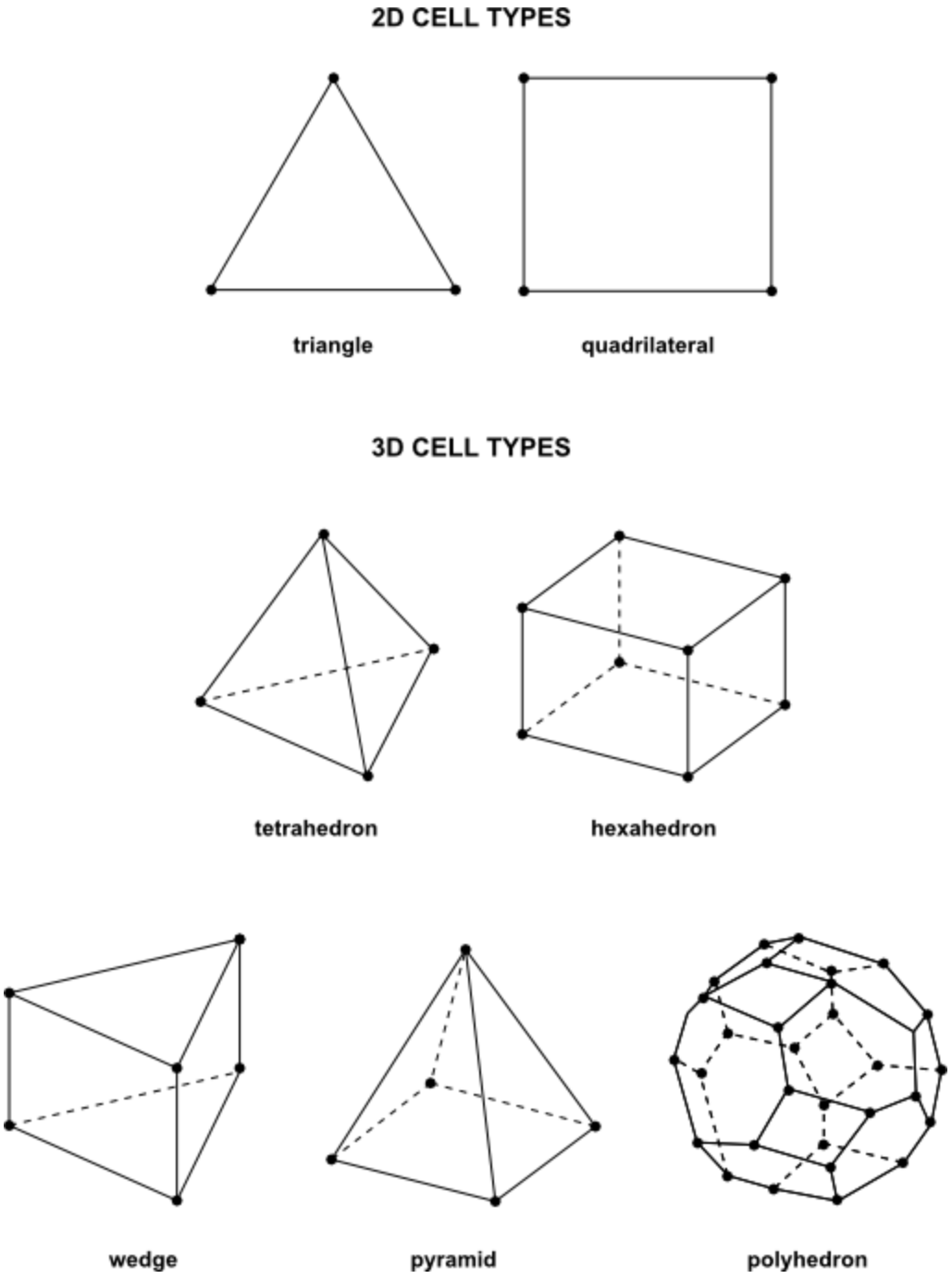
As an unstructured solver, Ansys Fluent uses internal data structures to assign an order to the cells, faces, and grid points in a mesh and to maintain contact between adjacent cells. Therefore, it does not require i,j,k indexing to locate neighboring cells. This gives you the flexibility to use the best mesh topology for your problem, as the solver does not force an overall structure or topology on the mesh.

For 2D meshes, quadrilateral and triangular cells are accepted, and for 3D meshes, hexahedral, tetrahedral, pyramid, wedge, and polyhedral cells can be used. Figure 6.1: Cell Types (p. 1130) depicts each of these cell types. Both single-block and multi-block structured meshes, as well as hybrid meshes containing quadrilateral and triangular cells or hexahedral, tetrahedral, pyramid, and wedge cells are acceptable. Ansys Fluent also can accept meshes with hanging nodes (that is, nodes on edges and faces that are not vertices of all the cells sharing those edges or faces) and hanging edges (that is, edges on faces that do not act as edges for both of the cells sharing those faces). While such hanging nodes / edges

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

can be accepted, in 3D cases by default the adjacent cells are automatically converted to polyhedra when read to remove them; if you opt to disable this conversion, you must ensure that the hanging nodes / edges are removed from interior walls. See Hanging Node Adaption in the Theory Guide for information about acceptable hanging nodes, and Removing Hanging Nodes / Edges (p. 1155) and Converting Cells with Hanging Nodes / Edges to Polyhedra (p. 1259) for details on removing hanging nodes / edges. Meshes with non-conformal boundaries (that is, meshes with multiple subdomains in which the mesh node locations at the internal subdomain boundaries are not identical) are also acceptable. For details, see Non-Conformal Meshes (p. 1170).

Figure 6.1: Cell Types



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Some examples of meshes that are valid for Ansys Fluent are presented in Examples of Acceptable Mesh Topologies (p. 1131). Different cell shapes and their face-node connectivity are explained in Face-Node Connectivity in Ansys Fluent (p. 1136). Choosing the Appropriate Mesh Type (p. 1143) explains how to choose the mesh type that is best suited for your problem.

6.1.1. Examples of Acceptable Mesh Topologies

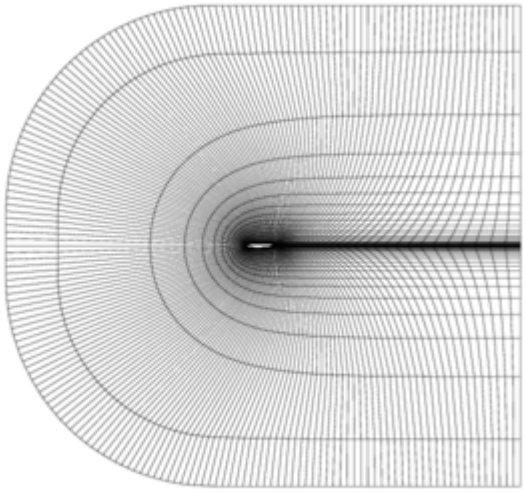
Ansys Fluent can solve problems on a wide variety of meshes. Figure 6.2: Structured Quadrilateral Mesh for an Airfoil (p. 1131)-Figure 6.13: Polyhedral Mesh (p. 1136) show examples of meshes that are valid for Ansys Fluent.

O-type meshes, meshes with zero-thickness walls, C-type meshes, conformal block-structured meshes, multiblock structured meshes, non-conformal meshes, and unstructured triangular, tetrahedral, quadrilateral, hexahedral, and polyhedral meshes are all acceptable.

Note:

Though Ansys Fluent does not require a cyclic branch cut in an O-type mesh, it will accept a mesh that contains one.

Figure 6.2: Structured Quadrilateral Mesh for an Airfoil



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.3: Unstructured Quadrilateral Mesh

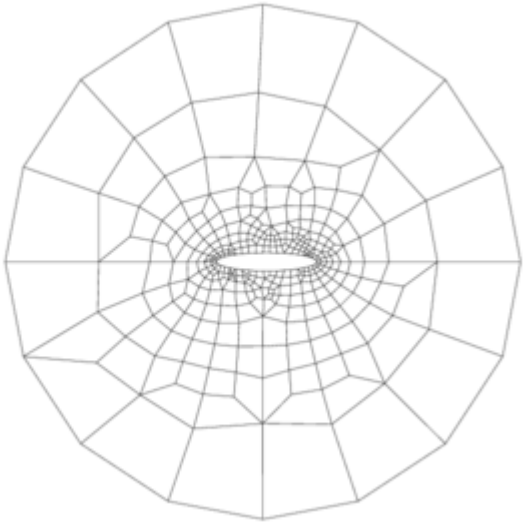


Figure 6.4: Multiblock Structured Quadrilateral Mesh

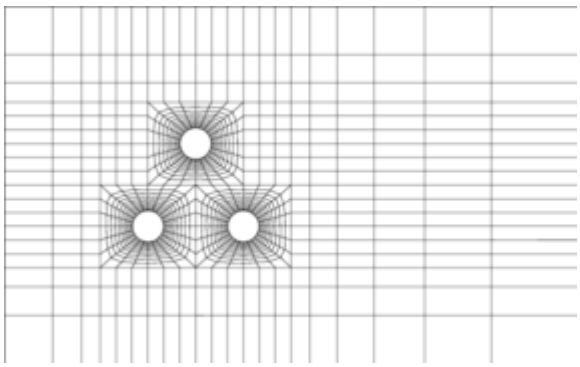
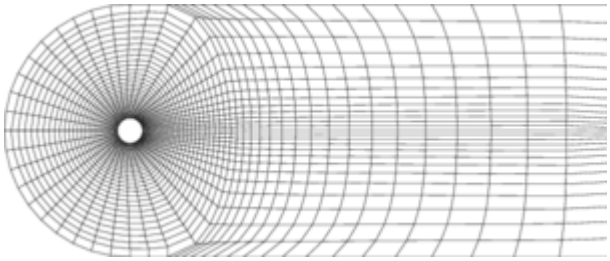


Figure 6.5: O-Type Structured Quadrilateral Mesh



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.6: Parachute Modeled With Zero-Thickness Wall

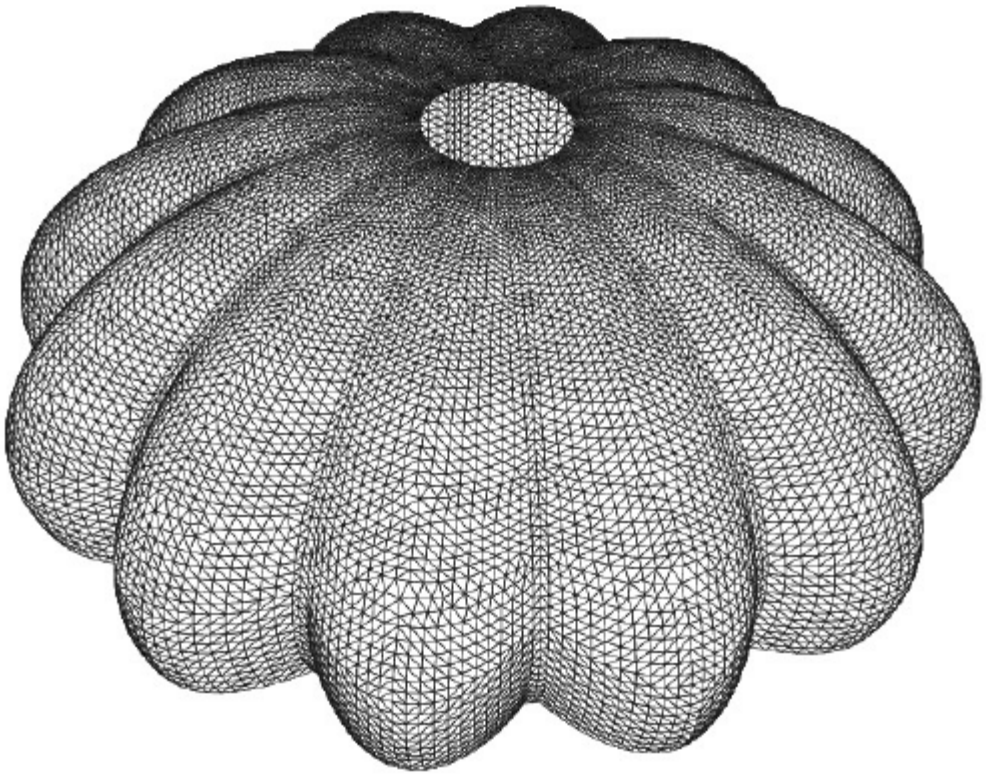
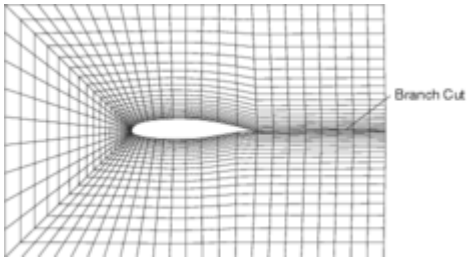


Figure 6.7: C-Type Structured Quadrilateral Mesh



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.8: 3D Multiblock Structured Mesh

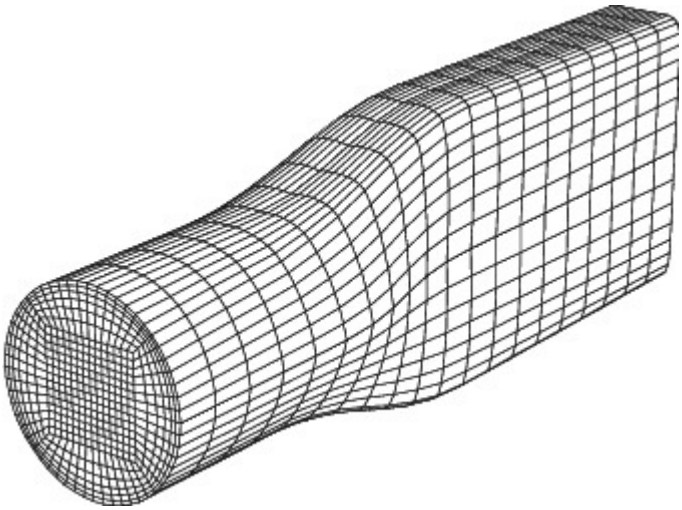
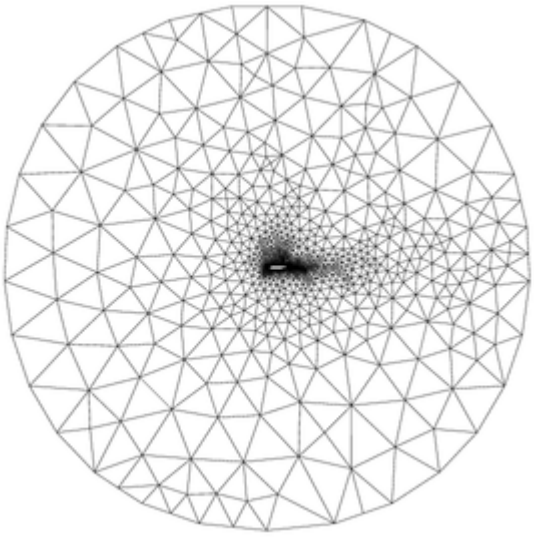


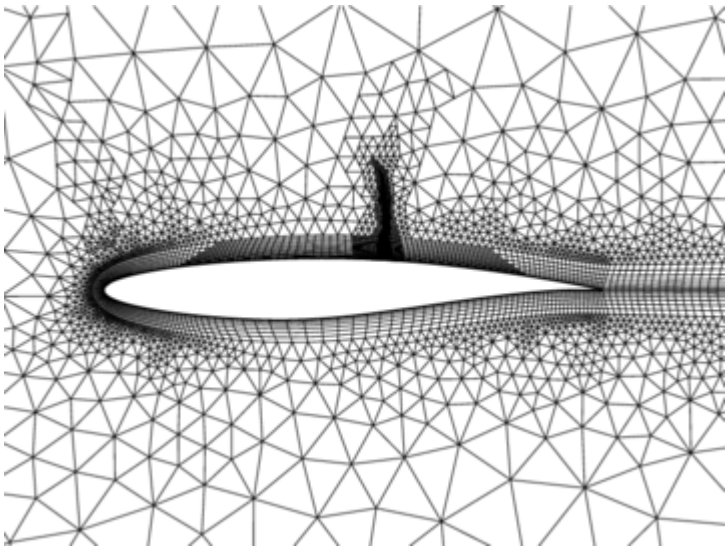
Figure 6.9: Unstructured Triangular Mesh for an Airfoil



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.10: Unstructured Tetrahedral Mesh

Figure 6.11: Hybrid Triangular/Quadrilateral Mesh with Hanging Nodes



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.12: Non-Conformal Hybrid Mesh for a Rotor-Stator Geometry

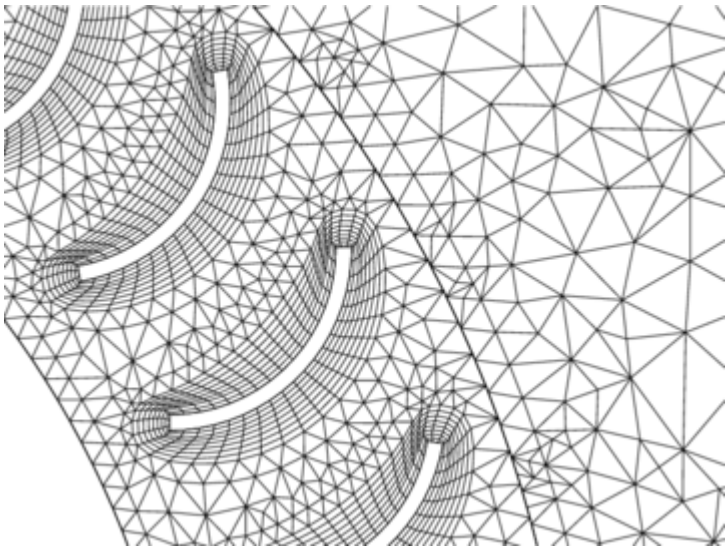


Figure 6.13: Polyhedral Mesh

6.1.2. Face-Node Connectivity in Ansys Fluent

This section contains information about the connectivity of faces and their related nodes in terms of node number and face number.

Face-node connectivity for the following cell shapes is explained here:

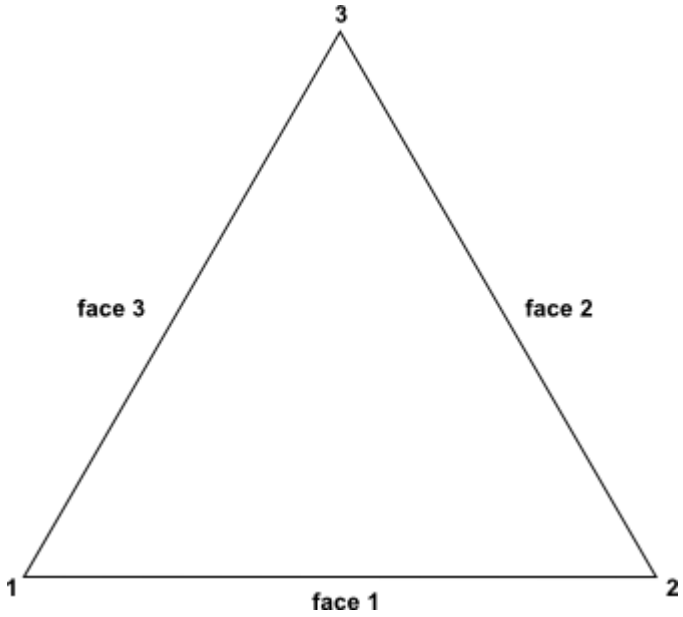
- triangular (Figure 6.14: Face and Node Numbering for Triangular Cells (p. 1137))
- quadrilateral (Figure 6.15: Face and Node Numbering for Quadrilateral Cells (p. 1138))
- tetrahedral (Figure 6.16: Face and Node Numbering for Tetrahedral Cells (p. 1139))
- wedge (Figure 6.17: Face and Node Numbering for Wedge Cells (p. 1140))
- pyramidal (Figure 6.18: Face and Node Numbering for Pyramidal Cells (p. 1141))
- hex (Figure 6.19: Face and Node Numbering for Hex Cells (p. 1142))
- polyhedral (Figure 6.20: An Example of a Polyhedral Cell (p. 1143))

This information is useful in interfacing with Ansys Fluent.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.1.2.1. Face-Node Connectivity for Triangular Cells

Figure 6.14: Face and Node Numbering for Triangular Cells

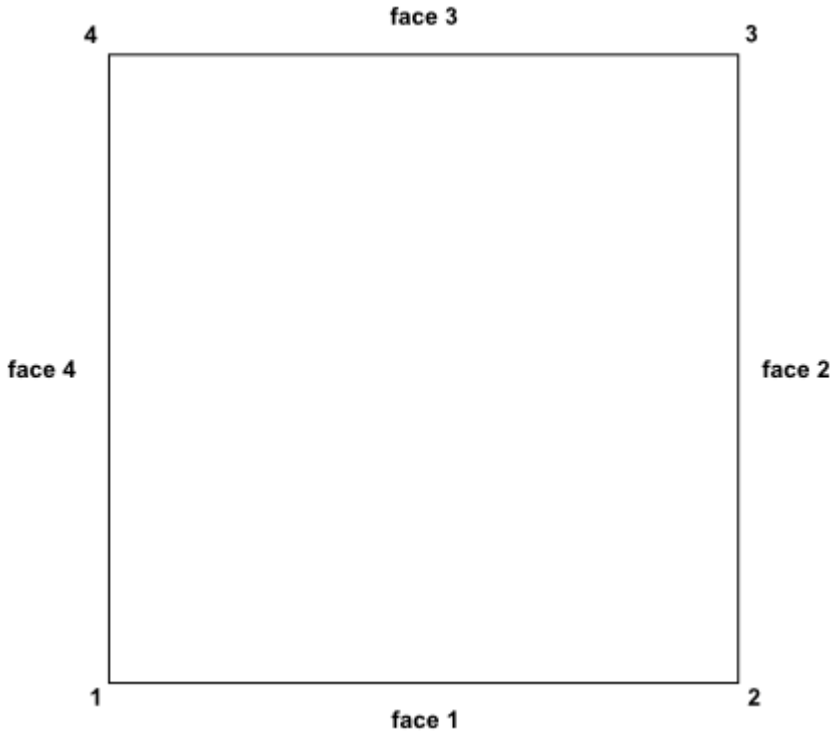


Face	Associated Nodes
Face 1	1-2
Face 2	2-3
Face 3	3-1

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.1.2.2. Face-Node Connectivity for Quadrilateral Cells

Figure 6.15: Face and Node Numbering for Quadrilateral Cells

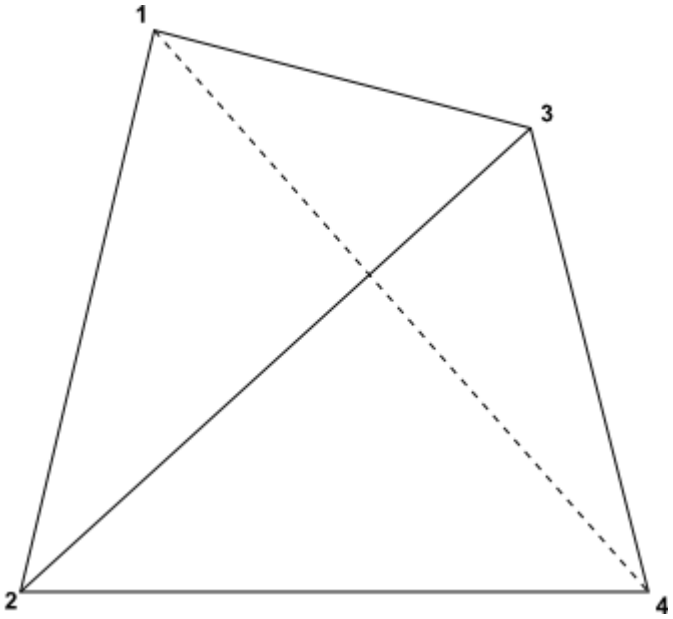


Face	Associated Nodes
Face 1	1-2
Face 2	2-3
Face 3	3-4
Face 4	4-1

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.1.2.3. Face-Node Connectivity for Tetrahedral Cells

Figure 6.16: Face and Node Numbering for Tetrahedral Cells

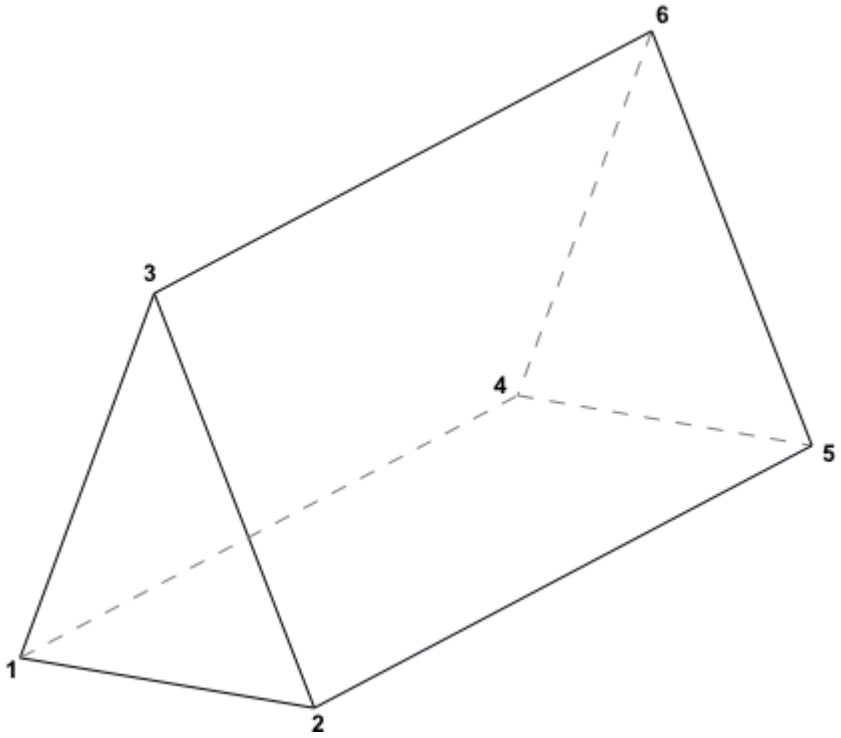


Face	Associated Nodes
Face 1	4-3-2
Face 2	3-4-1
Face 3	2-1-4
Face 4	3-1-2

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.1.2.4. Face-Node Connectivity for Wedge Cells

Figure 6.17: Face and Node Numbering for Wedge Cells

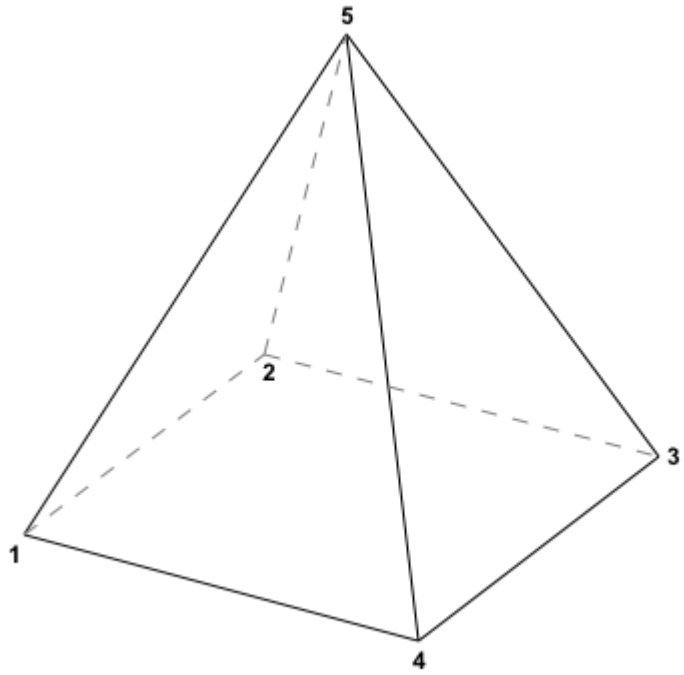


Face	Associated Nodes
Face 1	1-2-3
Face 2	1-4-5-2
Face 3	2-5-6-3
Face 4	3-6-4-1
Face 5	4-6-5

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.1.2.5. Face-Node Connectivity for Pyramidal Cells

Figure 6.18: Face and Node Numbering for Pyramidal Cells

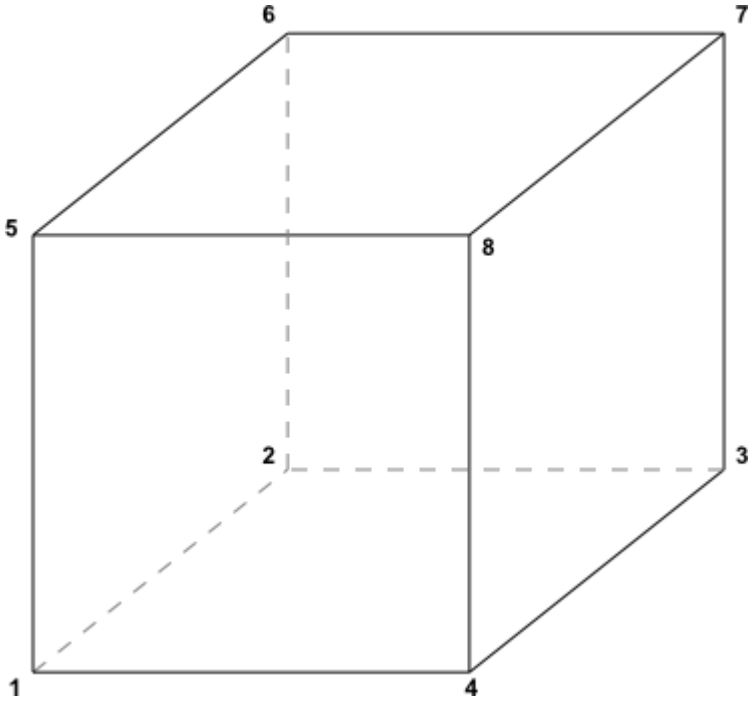


Face	Associated Nodes
Face 1	1-2-3-4
Face 2	1-5-2
Face 3	2-5-3
Face 4	3-5-4
Face 5	4-5-1

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.1.2.6. Face-Node Connectivity for Hex Cells

Figure 6.19: Face and Node Numbering for Hex Cells

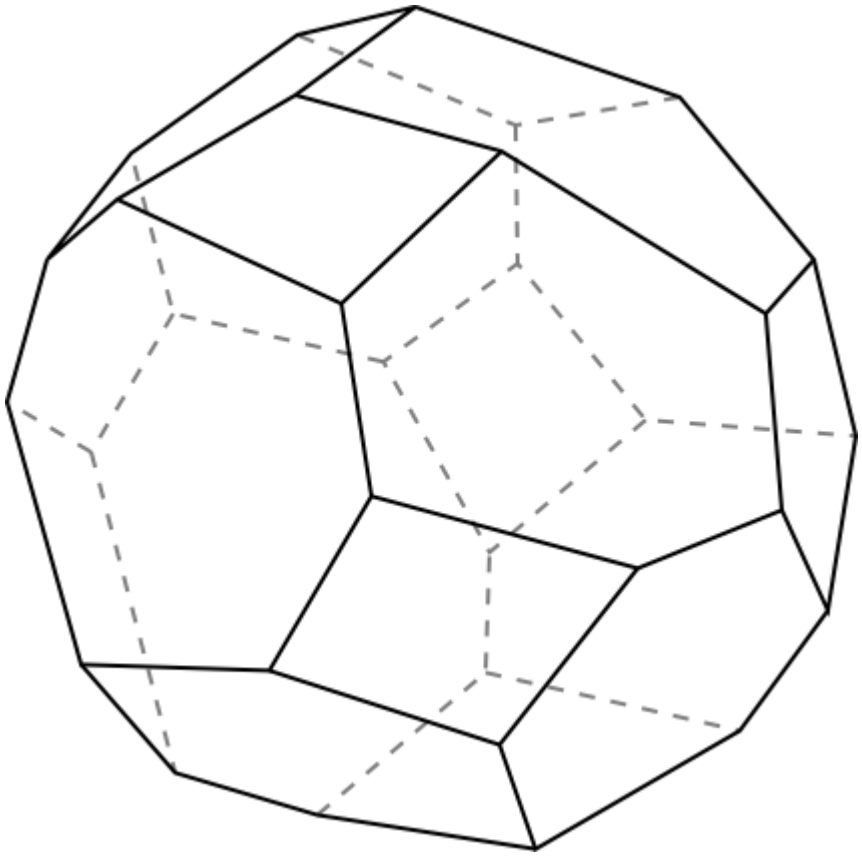


Face	Associated Nodes
Face 1	1-2-3-4
Face 2	1-5-6-2
Face 3	2-6-7-3
Face 4	4-3-7-8
Face 5	1-4-8-5
Face 6	5-8-7-6

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.1.2.7. Face-Node Connectivity for Polyhedral Cells

Figure 6.20: An Example of a Polyhedral Cell



For polyhedral cells, there is no explicit face and node numbering as with the other cell types.

6.1.3. Choosing the Appropriate Mesh Type

Ansys Fluent can use meshes composed of triangular or quadrilateral cells (or a combination of the two) in 2D, and tetrahedral, hexahedral, polyhedral, pyramid, or wedge cells (or a combination of these) in 3D. The choice of which mesh type to use will depend on your application. When choosing mesh type, consider the following issues:

- setup time
- computational expense
- numerical diffusion

6.1.3.1. Setup Time

Many flow problems solved in engineering practice involve complex geometries. The creation of structured or block-structured meshes (consisting of quadrilateral or hexahedral elements) for such problems can be extremely time-consuming if not impossible. Therefore, setup time for complex geometries is the major motivation for using unstructured meshes employing triangular or tetra-

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

hedral cells. However, if your geometry is relatively simple, there may be no saving in setup time with either approach.

Other risks of using structured or block-structured meshes with complicated geometries include the oversimplification of the geometry, mesh quality issues, and a less efficient mesh distribution (for example, fine resolution in areas of less importance) that results in a high cell count.

If you already have a mesh created for a structured code, it will save you time to use this mesh in Ansys Fluent rather than regenerate it. This can be a motivation for using quadrilateral or hexahedral cells in your Ansys Fluent simulation.

Note:

Ansys Fluent has a range of filters that allow you to import structured meshes from other codes, including FLUENT 4. For details, see Mesh Sources (p. 1150).

6.1.3.2. Computational Expense

When geometries are complex or the range of length scales of the flow is large, a triangular/tetrahedral mesh can be created with far fewer cells than the equivalent mesh consisting of quadrilateral/hexahedral elements. This is because a triangular/tetrahedral mesh allows clustering of cells in selected regions of the flow domain. Structured quadrilateral/hexahedral meshes will generally force cells to be placed in regions where they are not needed. Unstructured quadrilateral/hexahedral meshes offer many of the advantages of triangular/tetrahedral meshes for moderately-complex geometries.

A characteristic of quadrilateral/hexahedral elements that might make them more economical in some situations is that they permit a much larger aspect ratio than triangular/tetrahedral cells. A large aspect ratio in a triangular/tetrahedral cell will invariably affect the skewness of the cell, which is undesirable as it may impede accuracy and convergence. Therefore, if you have a relatively simple geometry in which the flow conforms well to the shape of the geometry, such as a long thin duct, use a mesh of high-aspect-ratio quadrilateral/hexahedral cells. The mesh is likely to have far fewer cells than if you use triangular/tetrahedral cells.

Converting the entire domain of your (tetrahedral) mesh to a polyhedral mesh will result in a lower cell count than your original mesh. Although the result is a coarser mesh, convergence will generally be faster, possibly saving you some computational expense.

In summary, the following practices are generally recommended:

- For simple geometries, use quadrilateral/hexahedral meshes.
- For moderately complex geometries, use unstructured quadrilateral/hexahedral meshes.
- For relatively complex geometries, use triangular/tetrahedral meshes with wedge elements in the boundary layers.
- For extremely complex geometries, use pure triangular/tetrahedral meshes.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.1.3.3. Numerical Diffusion

A dominant source of error in multidimensional situations is numerical diffusion (false diffusion). The term *false diffusion* is used because the diffusion is not a real phenomenon, yet its effect on a flow calculation is analogous to that of increasing the real diffusion coefficient.

The following comments can be made about numerical diffusion:

- Numerical diffusion is most noticeable when the real diffusion is small, that is, when the situation is convection-dominated.
- All practical numerical schemes for solving fluid flow contain a finite amount of numerical diffusion. This is because numerical diffusion arises from truncation errors that are a consequence of representing the fluid flow equations in discrete form.
- The second-order and the MUSCL discretization scheme used in Ansys Fluent can help reduce the effects of numerical diffusion on the solution.
- The amount of numerical diffusion is inversely related to the resolution of the mesh. Therefore, one way of dealing with numerical diffusion is to refine the mesh.
- Numerical diffusion is minimized when the flow is aligned with the mesh.

  This is the most relevant to the choice of the mesh. If you use a triangular/tetrahedral mesh, the flow can *never* be aligned with the mesh. If you use a quadrilateral/hexahedral mesh, this situation might occur, but *not for complex flows*. It is only in a *simple* flow, such as the flow through a long duct, in which you can rely on a quadrilateral/hexahedral mesh to minimize numerical diffusion. In such situations, it is advantageous to use a quadrilateral/hexahedral mesh, since you will be able to get a better solution with fewer cells than if you were using a triangular/tetrahedral mesh.
- If you would like higher resolution for a gradient that is perpendicular to a wall, you can create prism layers with higher aspect ratios near the wall.

6.2. Mesh Requirements and Considerations

This section contains information about special geometry/mesh requirements and general comments on mesh quality.

6.2.1. Geometry/Mesh Requirements

6.2.2. Mesh Quality

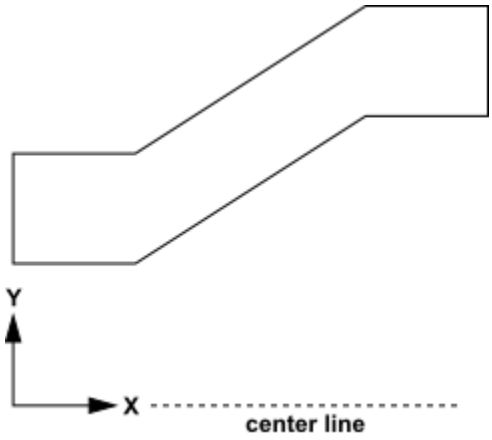
6.2.1. Geometry/Mesh Requirements

You should be aware of the following geometry setup and mesh construction requirements at the beginning of your problem setup:

- Axisymmetric geometries must be defined such that the axis of rotation is the x axis of the Cartesian coordinates used to define the geometry (Figure 6.21: Setup of Axisymmetric Geometries with the x Axis as the Centerline (p. 1146)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.21: Setup of Axisymmetric Geometries with the x Axis as the Centerline



- Ansys Fluent allows you to set up periodic boundaries using either conformal or non-conformal periodic zones. For conformal periodic boundaries, the periodic zones must have identical meshes.

  The conformal periodic boundary zones can be created in the meshing mode of Fluent or GAMBIT when you are generating the volume mesh. See the Fluent Meshing section of the User's Guide or the GAMBIT Modeling Guide for more information.

  You can also use the solution mode of Fluent to create a periodic boundary zone pair from two conformal zones, as well as a periodic interface from non-conformal zones; the latter is necessary if you created your mesh using a CAD package that is not able to produce true periodic boundaries. For details on creating such interfaces, see Creating Periodic Zones and Interfaces (p. 1269).

6.2.2. Mesh Quality

The quality of the mesh plays a significant role in the accuracy and stability of the numerical computation. Regardless of the type of mesh used in your domain, checking the quality of your mesh is essential. One important indicator of mesh quality that Ansys Fluent allows you to check is a quantity referred to as the orthogonality. In order to determine the orthogonality of a given cell, the following quantities are calculated for each face i:

- the normalized dot product of the area vector of a face ($\vec{A}_i$) and a vector from the centroid of the cell to the centroid of that face ($\vec{f}_i$):

$$\frac{\vec{A}_i \cdot \vec{f}_i}{|\vec{A}_i||\vec{f}_i|} \tag{6.1}$$

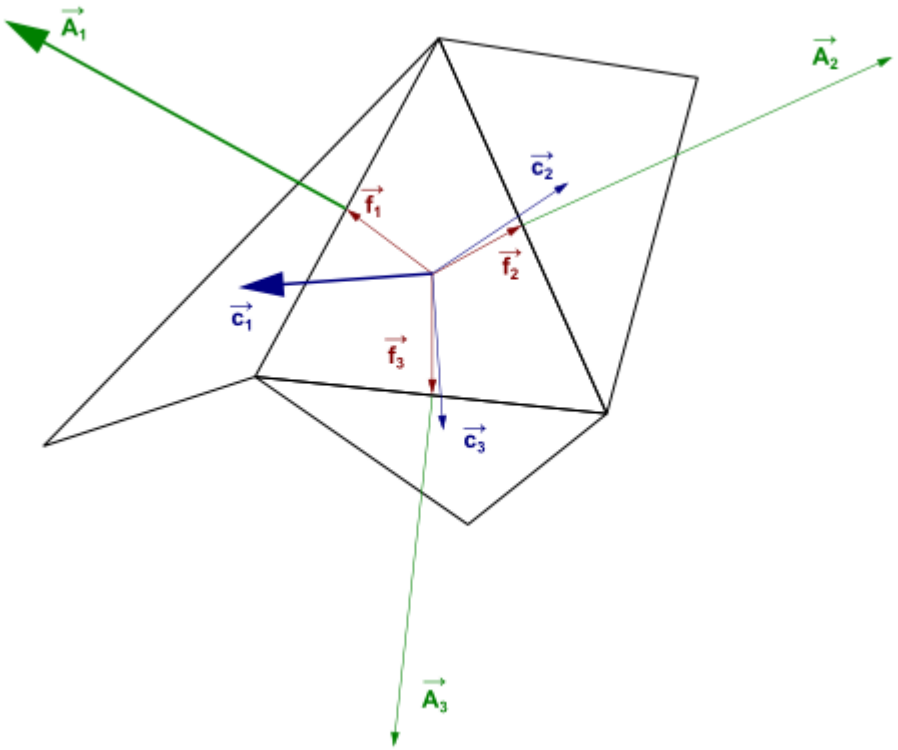
- the normalized dot product of the area vector of a face ($\vec{A}_i$) and a vector from the centroid of the cell to the centroid of the adjacent cell that shares that face ($\vec{c}_i$):

$$\frac{\vec{A}_i \cdot \vec{c}_i}{|\vec{A}_i||\vec{c}_i|} \tag{6.2}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The minimum value that results from calculating Equation 6.1 (p. 1146) and Equation 6.2 (p. 1146) for all of the faces is then defined as the orthogonality for the cell. Therefore, the worst cells will have an orthogonality closer to 0 and the best cells will have an orthogonality closer to 1. Figure 6.22: The Vectors Used to Compute Orthogonality (p. 1147) illustrates the relevant vectors.

Figure 6.22: The Vectors Used to Compute Orthogonality



This calculation of the orthogonality is used in (but is not always equal to) a metric referred to as the "orthogonal quality". By default, the orthogonal quality depends on the cell type:

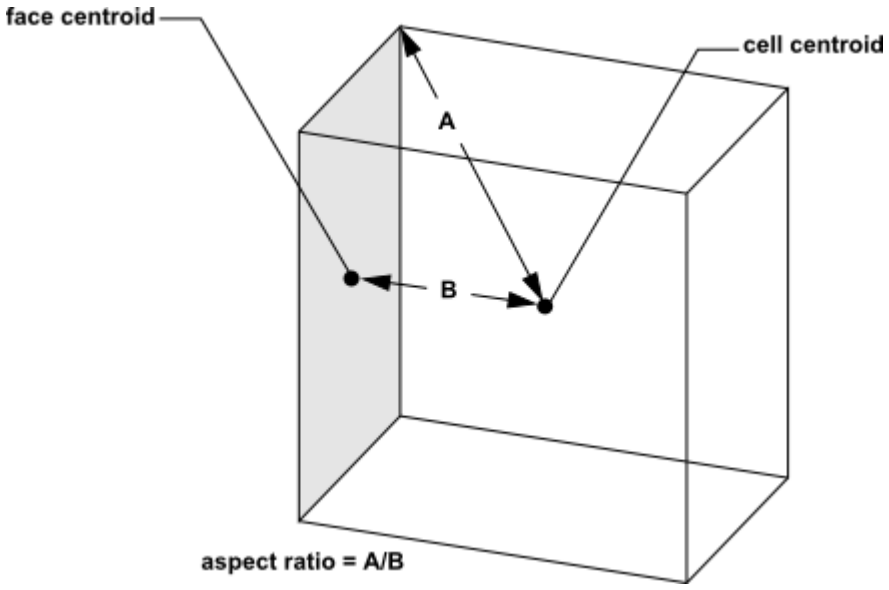
- For tetrahedral, prism, and pyramid cells, the orthogonal quality is the minimum of the orthogonality and (1- cell skewness).
- For hexahedral and polyhedral cells, the orthogonal quality is the same as the orthogonality.

Note that you can use the following text command to specify that the orthogonal quality uses an enhanced definition: `mesh/enhanced-orthogonal-quality?`. When enabled, the orthogonal quality is defined using a variety quality measures, including: the orthogonality of a face relative to vectors from the cell centroid to the face centroid and to the neighboring cell centroid; a metric that detects poor cell shape at a local edge (such as twisting and/or concavity); and the variation of normals between the faces that can be constructed from the cell face. This enhanced definition is optimal for evaluating thin prism cells.

Another important indicator of the mesh quality is the aspect ratio. The aspect ratio is a measure of the stretching of a cell. It is computed as the ratio of the maximum value to the minimum value of any of the following distances: the normal distances between the cell centroid and face centroids (computed as a dot product of the distance vector and the face normal), and the distances between the cell centroid and nodes. For a unit cube (see Figure 6.23: Calculating the Aspect Ratio for a Unit Cube (p. 1148)), the maximum distance is 0.866, and the minimum distance is 0.5, so the aspect ratio is 1.732. This type of definition can be applied on any type of mesh, including polyhedral.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.23: Calculating the Aspect Ratio for a Unit Cube



To check the quality of your mesh, you can use the **Report Quality** button in the **General** task page:

Setup → **General** → **Report Quality**

A message will be displayed in the console, such as the example that follows:

```
Mesh Quality:
 Orthogonal Quality ranges from 0 to 1, where values close to 0 correspond to low quality.
Minimum Orthogonal Quality = 6.07960e-01
Maximum Aspect Ratio = 5.42664e+00
```

If you would like more information about the quality displayed in the console (including additional quality metrics and the zones that have the cells with the lowest quality), set the `mesh/check-verbosity` text command to 2 (valid values are `0`, `1`, 2) prior to using the **Report Quality** button.

You can also get information about the number of cells and cell quality on a per cell zone basis. This can allow you to diagnose which cell zone contains poor quality cells.

Setup → **Cell Zone Conditions** → **<Zone Type>** **Info**

*You can also perform the same **Info** check on individual cell zones.*

For information about how to improve poor quality cells, see Repairing Meshes (p. 1247). For information about avoiding divergence when cells with poor face orthogonality are near coupled two-sided walls, see Orthogonality-Based Secondary Gradient Limiting at Coupled Two-Sided Walls (p. 1473).

When evaluating whether the quality of your mesh is sufficient for the problem you are modeling, it is important to consider attributes such as mesh element distribution, cell shape, smoothness, and flow-field dependency. These attributes are described in the sections that follow.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.2.2.1. Mesh Element Distribution

Since you are discretely defining a continuous domain, the degree to which the salient features of the flow (such as shear layers, separated regions, shock waves, boundary layers, and mixing zones) are resolved depends on the density and distribution of mesh elements. In many cases, poor resolution in critical regions can dramatically affect results. For example, the prediction of separation due to an adverse pressure gradient depends heavily on the resolution of the boundary layer upstream of the point of separation.

Resolution of the boundary layer (that is, mesh spacing near walls) also plays a significant role in the accuracy of the computed wall shear stress and heat transfer coefficient. This is particularly true in laminar flows where the mesh adjacent to the wall should obey

$$y_p\sqrt{\frac{u_\infty}{\nu x}} \leq 1 \tag{6.3}$$

where

y_p = distance to the wall from the adjacent cell centroid

u_∞ = free-stream velocity

ν = kinematic viscosity of the fluid

x = distance along the wall from the starting point of the boundary layer

Equation 6.3 (p. 1149) is based upon the Blasius solution for laminar flow over a flat plate at zero incidence [137] (p. 5721).

Proper resolution of the mesh for turbulent flows is also very important. Due to the strong interaction of the mean flow and turbulence, the numerical results for turbulent flows tend to be more susceptible to mesh element distribution than those for laminar flows. In the near-wall region, different mesh resolutions are required depending on the near-wall model being used. See Model Hierarchy (p. 2080) for guidelines.

In general, no flow passage should be represented by fewer than 5 cells. Most cases will require many more cells to adequately resolve the passage. In regions of large gradients, as in shear layers or mixing zones, the mesh should be fine enough to minimize the change in the flow variables from cell to cell. Unfortunately, it is very difficult to determine the locations of important flow features in advance. Moreover, the mesh resolution in most complicated 3D flow fields will be constrained by CPU time and computer resource limitations (that is, memory and disk space). Although accuracy increases with larger meshes, the CPU and memory requirements to compute the solution and postprocess the results also increase. Solution-adaptive mesh refinement can be used to increase and/or decrease mesh density based on the evolving flow field, and therefore provides the potential for more economical use of grid points (and hence reduced time and resource requirements). See Adapting the Mesh (p. 3773) for information on solution adaption.

6.2.2.2. Cell Quality

The quality of the cell (including its orthogonal quality, aspect ratio, and skewness) also has a significant impact on the accuracy of the numerical solution.

- *Orthogonal quality* is computed as described in Mesh Quality (p. 1146). The worst cells will have an orthogonal quality closer to 0, with the best cells closer to 1. The minimum orthogonal quality for all types of cells should be more than 0.01, with an average value that is significantly higher.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- *Aspect ratio* is a measure of the stretching of the cell. As discussed in Computational Expense (p. 1144), for highly anisotropic flows, extreme aspect ratios may yield accurate results with fewer cells. Generally, it is best to avoid sudden and large changes in cell aspect ratios in areas where the flow field exhibit large changes or strong gradients.

- *Skewness* is defined as the difference between the shape of the cell and the shape of an equilateral cell of equivalent volume. Highly skewed cells can decrease accuracy and destabilize the solution. For example, optimal quadrilateral meshes will have vertex angles close to 90 degrees, while triangular meshes should preferably have angles of close to 60 degrees and have all angles less than 90 degrees. A general rule is that the maximum skewness for a triangular/tetrahedral mesh in most flows should be kept below 0.95, with an average value that is significantly lower. A maximum value above 0.95 may lead to convergence difficulties and may require changing the solver controls, such as reducing under-relaxation factors and/or switching to the pressure-based coupled solver.

Cell size change and face warp are additional quality measure that could affect stability and accuracy. See the Fluent Meshing section of the User's Guide for more details.

6.2.2.3. Smoothness

Truncation error is the difference between the partial derivatives in the governing equations and their discrete approximations. Rapid changes in cell volume between adjacent cells translate into larger truncation errors. Ansys Fluent provides the capability to improve the smoothness by refining the mesh based on the change in cell volume or the gradient of cell volume. For information on marking cells by cell volume and then refining the mesh, see Volume (p. 3861) and Refining and Coarsening (p. 3777).

6.2.2.4. Flow-Field Dependency

The effect of resolution, smoothness, and cell shape on the accuracy and stability of the solution process is dependent on the flow field being simulated. For example, very skewed cells can be tolerated in benign flow regions, but can be very damaging in regions with strong flow gradients.

Since the locations of strong flow gradients generally cannot be determined a priori, you should strive to achieve a high-quality mesh over the entire flow domain.

6.3. Mesh Sources

Since Ansys Fluent can handle a number of different mesh topologies, there are many sources from which you can obtain a mesh to be used in your simulation. You can generate a mesh using Ansys Meshing, the meshing mode of Fluent, Fluent Meshing, GAMBIT, GeoMesh, ICEM CFD, NASTRAN, ARIES, CFX, or other preprocessors. You can also use the mesh contained in a Fluent/UNS or RAMPANT. You can also prepare multiple mesh files and combine them to create a single mesh.

6.3.1. Ansys Meshing Mesh Files

6.3.2. Fluent Meshing Mode Mesh Files

6.3.3. Fluent Meshing Mesh Files

6.3.4. GAMBIT Mesh Files

6.3.5. GeoMesh Mesh Files

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.6. NASTRAN Files
6.3.7. CFX Files
6.3.8. Using the fe2ram Filter to Convert Files
6.3.9. Removing Hanging Nodes / Edges
6.3.10. Fluent/UNS and RAMPANT Case Files
6.3.11. Ansys FIDAP Neutral Files
6.3.12. Reading Multiple Mesh/Case/Data Files
6.3.13. Reading Surface Mesh Files

6.3.1. Ansys Meshing Mesh Files

You can use Ansys Meshing to create your mesh. Follow the meshing procedure and export to the Ansys Fluent mesh format (see Fluent Mesh Export in the Ansys Meshing User's Guide for details). To import the mesh into Fluent, use the **File/Read/Mesh...** ribbon tab item (as described in Reading Mesh Files (p. 953)).

6.3.2. Fluent Meshing Mode Mesh Files

You can use the meshing mode of Fluent (which was previously a stand-alone program named Fluent Meshing) to create 3D unstructured triangular/tetrahedral meshes from boundary or surface meshes. Switch to meshing mode and then follow the meshing procedure described in the Fluent Meshing section of the User's Guide. You can save your mesh using the **File/Write/Mesh...** menu item.

Then switch to the solution mode to set up the case file for the transferred mesh, or import the saved mesh into a new Fluent session using the **File/Read/Mesh...** ribbon tab item (as described in Reading Mesh Files (p. 953)).

For information about switching between meshing and solution modes, see Switching Between Meshing and Solution Modes (p. 207) in the Getting Started (p. 1) part of this manual.

6.3.3. Fluent Meshing Mesh Files

Prior to version 14.5, Fluent Meshing was a stand-alone product that was used to create 3D unstructured triangular/tetrahedral meshes from boundary or surface meshes. Fluent Meshing has been integrated as the Fluent meshing mode. To import meshes created by Fluent Meshing into Fluent, use the **File/Read/Mesh...** ribbon tab item (as described in Reading Mesh Files (p. 953)).

6.3.4. GAMBIT Mesh Files

You can use GAMBIT to create 2D and 3D structured/unstructured/hybrid meshes. To create any of these meshes for Ansys Fluent, follow the procedure described in the GAMBIT Modeling Guide, and export your mesh in Fluent 5/6 format. All such meshes can be imported directly into Ansys Fluent using the **File/Read/Mesh...** ribbon tab item, as described in Reading Mesh Files (p. 953).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.5. GeoMesh Mesh Files

You can use GeoMesh to create complete 2D quadrilateral or triangular meshes, 3D hexahedral meshes, and triangular surface meshes for 3D tetrahedral meshes. To create any of these meshes for Ansys Fluent, follow the procedure described in the GeoMesh User's Guide.

To complete the generation of a 3D tetrahedral mesh, read the surface mesh into the meshing mode of Fluent and generate the volume mesh there (see the Fluent Meshing section of the User's Guide for details). All other meshes can be read directly into the solution mode of Fluent. Use the **File/Read/Mesh...** ribbon tab item to read the mesh files, as described in Reading Mesh Files (p. 953).

6.3.6. NASTRAN Files

There are three different ways in which you can import a NASTRAN file into Ansys Fluent:

- You can generate a NASTRAN surface or volume mesh containing triangular, quadrilateral, tetrahedral, wedge, and/or hexahedral elements, and import it into the meshing mode of Fluent using the commands described in the Fluent Meshing section of the User's Guide. Adhere to the restrictions described in Appendix B: Mesh File Format (p. 861) in the Fluent Meshing section of the User's Guide. In meshing mode, complete the mesh generation (if necessary); then you can switch to the solution mode of Fluent or read it into a new Fluent session as described in Fluent Meshing Mode Mesh Files (p. 1151).

- You can generate a NASTRAN volume mesh with linear triangular, quadrilateral, tetrahedral, wedge, or hexahedral elements, and import it directly using the **File/Import/NASTRAN** ribbon tab item, as described in NASTRAN Files (p. 982).

- You can generate a NASTRAN volume mesh with linear triangular, quadrilateral, tetrahedral, wedge, or hexahedral elements. Then use the `fe2ram` filter to convert the NASTRAN file to the format used by Ansys Fluent. To convert an input file in NASTRAN format to an output file in Ansys Fluent format, follow the instructions below in Using the fe2ram Filter to Convert Files (p. 1154). After the output file has been written, you can read it into Ansys Fluent using the **File/Read/Mesh...** ribbon tab item, as described in Reading Mesh Files (p. 953).

After reading a triangular or tetrahedral NASTRAN volume mesh using the latter methods, improve its quality through smoothing (as described in Quality-Based Smoothing (p. 1292)).

6.3.6.1. Recognized NASTRAN Bulk Data Entries

The following NASTRAN file datasets are recognized by the Ansys Fluent mesh import utility:

- **GRID** single-precision node coordinates
- **GRID*** double-precision node coordinates
- **CBAR** line elements
- **CTETRA, CTRIA3** tetrahedral and triangular elements
- **CHEXA, CQUAD4, CPENTA** hexahedral, quadrilateral, and wedge elements

For 2D volume meshes, the elements must exist in a constant z plane.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.6.2. Deleting Duplicate Nodes

NASTRAN may generate duplicate or coincident nodes in the process of creating elements. These nodes must be removed in NASTRAN before writing the file for import into Ansys Fluent.

6.3.7. CFX Files

You can import the meshes from 3D CFX files, such as definition (`.def`) and result (`.res`) files into Ansys Fluent, using the **File/Import/CFX** ribbon tab item, as described in CFX Files (p. 979). The `fe2ram` utility is used as the import filter, which can be used as a stand-alone program to obtain an Ansys Fluent mesh file. See Using the fe2ram Filter to Convert Files (p. 1154) for information about `fe2ram`.

Important:

Note that you have the ability to import only the mesh from a CFX file, and not any results or data.

When importing a mesh from a CFX definition or results file, select whether you want the Ansys Fluent zones to be created from the physics data objects or the primitive mesh region objects. The former option is the default and is called "Zoning by CCL" and the latter option is called "Zoning by Group". This will allow you to choose the type of mesh topology you would like to preserve when importing the file:

- If you want zones to be created from the physics data objects, enable **Create Zones from CCL Physics Data** in the **Select File** dialog box.
- If you want zones to be created from the primitive mesh region objects, disable **Create Zones from CCL Physics Data** in the **Select File** dialog box. This will result in the group zoning.

Important:

The primitive mesh topology may contain additional regions that do not appear in the physics definition.

The default import method is the **Create Zones from CCL Physics Data** method. This method will not import CFX subdomain regions, however, the zoning by group method can import subdomain regions.

The 3D element set corresponding to zones/domains present in these files are imported as cell zones in Ansys Fluent. They may contain tetrahedral, pyramidal, wedge, and hexahedral elements. The boundary zones in these files are a group of faces with a boundary condition name/type and are imported as face zones with the boundary condition name/type retained in Ansys Fluent. The following boundary condition types are retained:

- **inlet**
- **outlet**
- **symmetry**
- **interface**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **wall**

The boundaries of type `Interface` may be conformal or non-conformal. If they are non-conformal, they are retained. However, conformal interfaces contain coincident nodes that are merged and changed to type `Interior`. For some cases, for the merge to work correctly, the merge tolerance may need to be adjusted. Alternatively, the Fuse Face Zones Dialog Box (p. 5483) in Ansys Fluent can be used to merge the conformal interfaces. For details, see Fusing Face Zones (p. 1267).

6.3.8. Using the fe2ram Filter to Convert Files

The `fe2ram` filter can be used to manually convert files of certain formats into Ansys Fluent mesh files, which can then be read into Ansys Fluent. To use the `fe2ram` filter, enter the following at a command prompt in a terminal or command window:

`utility fe2ram` [*dimension*] *format* [*zoning*] *input_file output_file*

Note:

The items enclosed in square brackets are optional. Do not type the square brackets.

- *dimension* indicates the dimension of the dataset. Replace *dimension* with `-d2` to indicate that the mesh is two dimensional. For a 3D mesh, do not enter any value for *dimension*, because 3D is the default.
- *format* indicates the format of the file you want to convert. For example, replace *format* with `-tANSYS` for a Mechanical APDL file, `-tIDEAS` for an I-deas file, `-tNASTRAN` for a NASTRAN file, and so on. To print a list of the formats which `fe2ram` can convert, type

  `utility fe2ram -cl -help`.

  Note:

  fe2ram does not support VRML 2.0 or later.

- *zoning* indicates how zones were identified in the original format. Replace *zoning* by `-zID` for a mesh that was zoned by property IDs, or `-zNONE` to ignore all zone groupings. For a mesh zoned by group, do not enter anything for *zoning*, because zoning by groups is the default.
- *input_file* is the name of the original file. *output_file* is the name of the file to which you want to write the converted mesh information. Note that the *output_file* cannot be a CFF file (`.msh.h5`).

  For example, if you wanted to convert the 2D I-deas volume mesh file `sample.unv` to an output file called `sample.msh`, you will enter the following command:

  ```
  utility fe2ram -d2 -tIDEAS sample.unv sample.msh
  ```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.9. Removing Hanging Nodes / Edges

As noted in Mesh Topologies (p. 1129), while Ansys Fluent can accept meshes that contain hanging nodes or hanging edges, in 3D cases by default the adjacent cells are automatically converted to polyhedra when read to remove such hanging nodes / edges. Each cell that is converted will retain the same overall dimensions, but the number of faces associated with the cell may increase. You can opt to disable this conversion by using the following text command prior to reading: `file/convert-hanging-nodes-during-read? no`. Note that when the conversion is disabled, the creation of interior walls can yield an error if hanging nodes / edges are located on the zone that is turned into an interior wall. This problem can occur in the following cases:

- If you read a mesh file that has an interior (or two-sided) wall boundary condition setup, but the shadow wall has not been created yet.
- If you turn an interior surface into a wall.
- If you slit an interior surface (for example, turn it into a non-conformal interface).

Such error-producing hanging nodes / edges may be present in hexcore or CutCell meshes, for example. If you find you need to remove the hanging nodes / edges after reading, you can convert the associated cells to polyhedra by using the `mesh/polyhedra/convert-hanging-nodes` text command.

6.3.9.1. Limitations

Meshes with polyhedra have the following limitations:

- The following mesh manipulation tools are not available on polyhedral meshes:
  - `extrude-face-zone` under the `modify-zone` option
  - skewness smoothing
  - swapping (will not affect polyhedral cells)
- The polyhedral cells that result from the conversion are not eligible for adaption with the hanging node method, though they can be refined with the default polyhedral unstructured mesh adaption (PUMA) method. For more information about adaption, see Adapting the Mesh (p. 3773).
- Note the following with regard to the polyhedral cells that result from this conversion and the update methods available for dynamic mesh problems:
  - None of the remeshing methods will modify polyhedral cells.
  - Smoothing is allowed for polyhedral cells, and radial basis function smoothing is the recommended method (see Radial Basis Function Smoothing (p. 1830) for details). The linearly elastic solid smoothing method is not compatible with polyhedral cells.

6.3.10. Fluent/UNS and RAMPANT Case Files

If you have a Fluent/UNS 3 or 4 case file or a RAMPANT 2, 3, or 4 case file and you want to run an Ansys Fluent simulation using the same mesh, you can read it into Ansys Fluent using the

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

File/Read/Case... ribbon tab item, as described in Reading Fluent/UNS and RAMPANT Case and Data Files (p. 963).

6.3.11. Ansys FIDAP Neutral Files

If you have an Ansys FIDAP Neutral file and you want to run an Ansys Fluent simulation using the same mesh, import it using the ribbon item. Ansys Fluent will read mesh information and zone types from the Ansys FIDAP file.

You can manually convert an input file in Ansys FIDAP format to an output file in Ansys Fluent (as described in detail in Using the fe2ram Filter to Convert Files (p. 1154)):

`utility fe2ram [` *dimension* `] -tFIDAP7` *input_file output_file*

The item in square brackets is optional. Do not type the square brackets. For a 2D file, replace *dimension* with `-d2`. For a 3D file, do not enter anything for *dimension*, because 3D is the default.

After the output file has been written, read it into Ansys Fluent using the **File/Read/Case...** ribbon tab item, as described in Reading Mesh Files (p. 953).

6.3.12. Reading Multiple Mesh/Case/Data Files

There may be some cases in which you will need to read multiple mesh files (subdomains) to form your computational domain.

- To solve on a multiblock mesh, generate each block of the mesh in the mesh generator and save it to a separate mesh file.
- For very complicated geometries, it may be more efficient to save the mesh for each part as a separate mesh file.

The mesh node locations need not be identical at the boundaries where two separate meshes meet. Ansys Fluent can handle non-conformal mesh interfaces. See Non-Conformal Meshes (p. 1170) for details about non-conformal mesh boundaries.

There are three ways of reading multiple mesh files in Ansys Fluent:

- Read multiple mesh files into the solution mode of Fluent.
- Read multiple mesh files into the meshing mode of Fluent.
- Use `tmerge` to manipulate the individual files and merge them into a single file that can then be read into Fluent. Note that CFF files (`.msh.h5`) are not supported with `tmerge`.

Note:

If your multiple meshes are not properly scaled, located, and/or oriented relative to each other, you have the ability to manipulate the cell zones in the solution mode of Ansys Fluent, as described in Scaling Individual Cell Zones (p. 1288), Translating Individual Cell Zones (p. 1289), and Rotating Individual Cell Zones (p. 1290).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.3.12.1. Reading Multiple Mesh Files via the Solution Mode of Fluent

The solution mode of Fluent allows you to handle more than one mesh at a time within the same solver settings. This capability of handling multiple meshes saves time, since you can directly read in the different mesh files and start setting up the solution, without having to switch over from meshing mode or employ another tool such as `tmerge`.

The steps to take when reading more than one mesh file are:

1. Read in your first mesh file.

   File → **Read** → **Mesh...**

   In the **Select File** dialog box, select the mesh file and click **OK**.

2. Read in your second mesh file and append it to the first mesh selected in the first step.

   Domain → **Zones** → **Append** → **Append Case File...**

   In The Select File Dialog Box (p. 931), select the second mesh file and click **OK**.

3. (optional). Display your meshes using the **Mesh Display** dialog box.

   Setup → **General** → **Display...**

   You will find that the second mesh is appended to the first.

Ansys Fluent also allows you to append the data on the mesh. To do that, follow the procedure above. For the second step, use the following ribbon tab item:

Domain → **Zones** → **Append** → **Append Case & Data Files...**

Select the case file in the **Select File** dialog box, and click **OK**. Both the case and data files will be appended.

Important:

Review your case setup after appending multiple mesh and/or data files before proceeding with the calculation.

6.3.12.2. Reading Multiple Mesh Files via the Meshing Mode of Fluent

1. Generate the mesh for the whole domain in the mesh generator, and save each cell zone (or block or part) to a separate mesh file for Fluent.

2. In the meshing mode of Fluent, combine the meshes into one mesh file.

   a. Read all of the mesh files. As the mesh files are read, they will be automatically merged into a single mesh.

   b. Save the merged mesh file.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See the Fluent Meshing section of the User's Guide for information about reading and writing files in meshing mode.

3. Switch over to the solution mode of Fluent or read the combined mesh file into a new session using the **File/Read/Mesh...** ribbon tab item.

For a conformal mesh, if you do not want a boundary between the adjacent cell zones, use the **Fuse Face Zones** dialog box to fuse the overlapping boundaries. For details, see Fusing Face Zones (p. 1267). The matching faces will be moved to a new zone with a boundary type of **interior** and the original zone(s) will be discarded.

Important:

If you are planning to use sliding meshes, or if you have non-conformal boundaries between adjacent cell zones, do not combine the overlapping zones. Instead, change the boundary zone type as needed and continue the setup described in Non-Conformal Meshes (p. 1170).

6.3.12.3. Reading Multiple Mesh Files via tmerge

1. Generate the mesh for the whole domain in the mesh generator, and save each cell zone (or block or part) to a separate legacy mesh file (`.msh`) for Ansys Fluent.

2. Before launching Fluent, use the `tmerge` filter to combine the meshes into one mesh file. The `tmerge` method allows you to rotate, scale, and/or translate the meshes before they are merged. Note that the `tmerge` filter allows you to merge large meshes with very low memory requirement.

   a. For 3D problems, type `utility tmerge -3d`. For 2D problems, type `utility tmerge -2d`.

   b. When prompted, specify the names of the input files (the separate mesh files) and the name of the output file in which to save the complete mesh. Be sure to include the `.msh` extension.

   c. For each input file, specify scaling factors, translation distances, and rotation information.

   For information about the various options available when using `tmerge`, type `utility tmerge -h`.

3. Read the combined mesh file into the solution mode of Fluent in the usual manner (using the **File/Read/Mesh...** ribbon tab item).

For a conformal mesh, if you do not want a boundary between the adjacent cell zones, use the **Fuse Face Zones** dialog box to fuse the overlapping boundaries. For details, see Fusing Face Zones (p. 1267). The matching faces will be moved to a new zone with a boundary type of **interior** and the original zone(s) will be discarded.

Important:

If you are planning to use sliding meshes, or if you have non-conformal boundaries between adjacent cell zones, do not combine the overlapping zones. Instead, change

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the boundary zone type as needed and continue the setup described in Non-Conformal Meshes (p. 1170).

In this example, scaling, translation, or rotation is not requested. Hence you can simplify the inputs to the following:

```
 user@mymachine:> utility tmerge -2d

Starting /ansys_inc/v242/fluent/fluent24.2.0/utility/tmerge/lnamd64/tmerge_2d

 Append 2D grid files.
 tmerge2D ANSYS Inc, stream

 Enter name of grid file (ENTER to continue) :  my1.msh

 x,y scaling factor, eg. 1 1          : 1 1

 x,y translation, eg. 0 1             : 0 0

 rotation angle (deg), eg. 45         : 0

 Enter name of grid file (ENTER to continue) :  my2.msh

 x,y scaling factor, eg. 1 1          : 1 1

 x,y translation, eg. 0 1             : 0 0

rotation angle (deg), eg. 45          : 0

 Enter name of grid file (ENTER to continue) :  {Enter}

 Enter name of output file            : final.msh

 Reading...
  node zone: id 1, ib 1, ie 1677, typ 1
  node zone: id 2, ib 1678, ie 2169, typ 2
    .
    .
    .
 done.
 Writing...
  492 nodes, id 1, ib 1678, ie 2169, type 2.
1677 nodes, id 2, ib 1, ie 1677, type 1.
    .
    .
    .
 done.
 Appending done.
```

6.3.13. Reading Surface Mesh Files

Surface meshes are used as background meshes for geometry-based adaption. Perform the following steps to read the surface mesh file into Ansys Fluent:

1. Open the **Surface Meshes** dialog box (Figure 6.24: The Surface Meshes Dialog Box (p. 1160)). How you open this dialog box depends on the adaption method you plan to use:

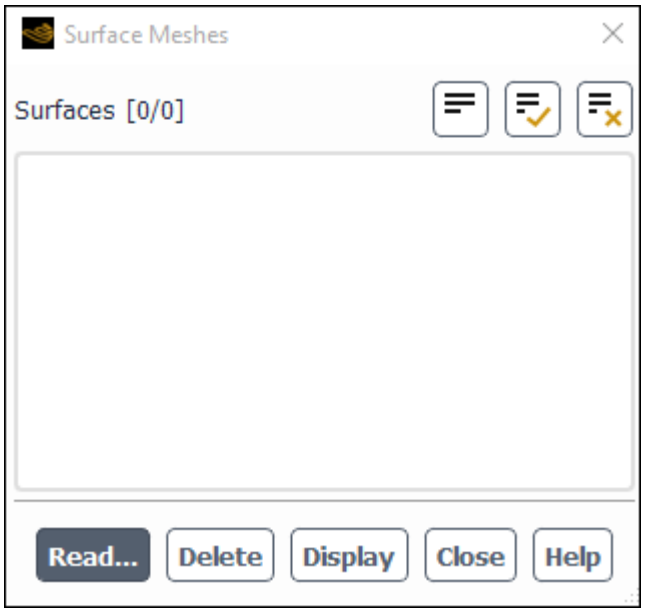
   - If you are using the PUMA adaption method (the default for 3D), open the **Auxiliary Geometry Definition** dialog box (as described in Managing Auxiliary Geometry Definitions (p. 1791)), select **Surface Mesh** from the **Type** drop-down list, and then click the **Surfaces Meshes...** button.

- If you are using the hanging node adaption method, open the **Geometry Based Adaption** dialog box (as described in Performing Geometry-Based Adaption with the Hanging Node Method (p. 3811)), enable the **Reconstruct Geometry** option, and then click the **Surfaces Meshes...** button.

2. In the Surface Meshes Dialog Box (p. 5624), click **Read...** and select the surface mesh file using The Select File Dialog Box (p. 931).

   Specify the units the mesh was created in (default is meters).

   Figure 6.24: The Surface Meshes Dialog Box



   Note that you can also display and delete the surfaces using Figure 6.24: The Surface Meshes Dialog Box (p. 1160).

3. Continue performing the adaption:

   - If you are using the PUMA adaption method (the default for 3D), enable the **Reconstruct Geometry** option in the **General Adaption Controls** dialog box, set up the geometry-based adaption using the **Manage Geometry-Based Adaption** dialog box, and perform adaption as described in Refining and Coarsening (p. 3777).

   - If you are using the hanging node adaption method, make a selection from the **Wall Zones** list in the **Geometry Based Adaption** dialog box, click **Set...**, set up the adaption in the Geometry Based Adaption Controls Dialog Box (p. 5490) that opens (as described in Performing Geometry-Based Adaption with the Hanging Node Method (p. 3811)), and perform adaption as described in Refining and Coarsening (p. 3777).

6.4. Reference Frames

You can create multiple local reference frames for use in setting up Ansys Fluent cases.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The reference frames feature allows you to create local coordinate systems with a given position and orientation, either with or without motion. Without local reference frames, you would have to specify things such as surfaces based on the global reference frame/coordinate system.

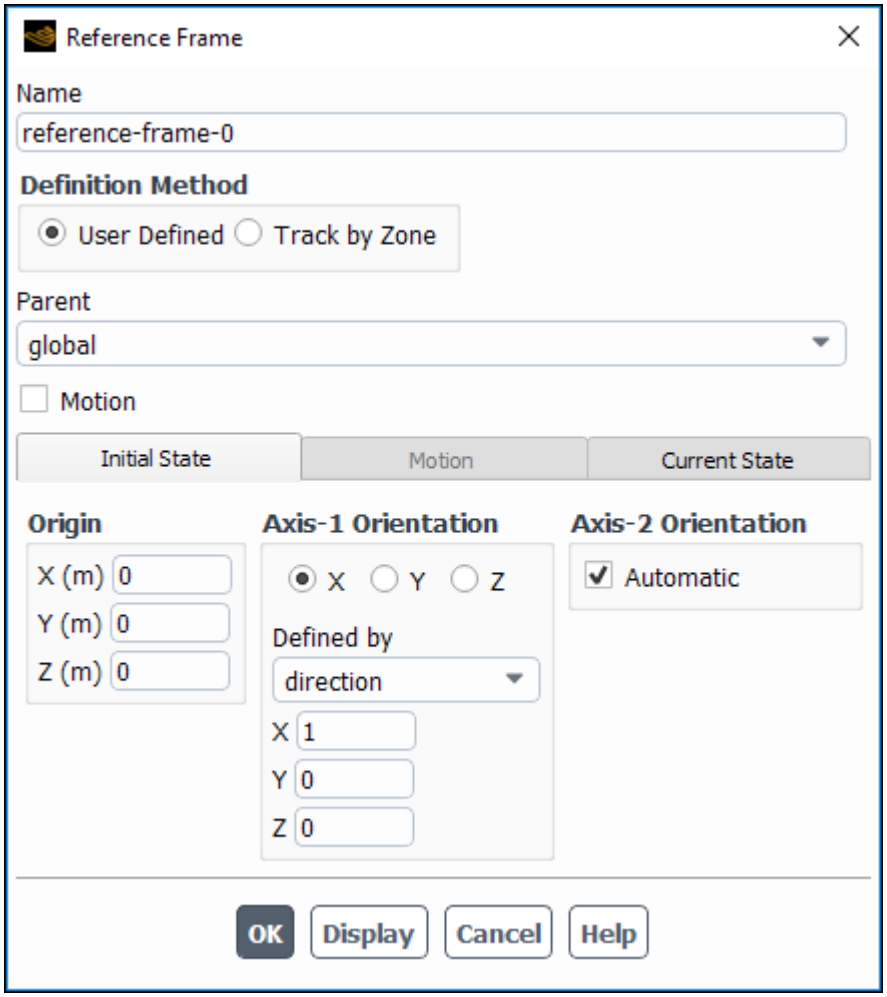
6.4.1. Creating and Using Reference Frames

To create a reference frame:

1. Open the **Reference Frame** dialog box by right-clicking **Reference Frames** in the outline view and selecting **New...**.

   Setup → Reference Frames → New...

   Figure 6.25: The Reference Frame Dialog Box



2. Enter a **Name** for the reference frame.

3. Specify how you want to define the reference frame.

   - **User Defined** allows you to completely specify the details of where the reference frame is located and oriented and how it moves.

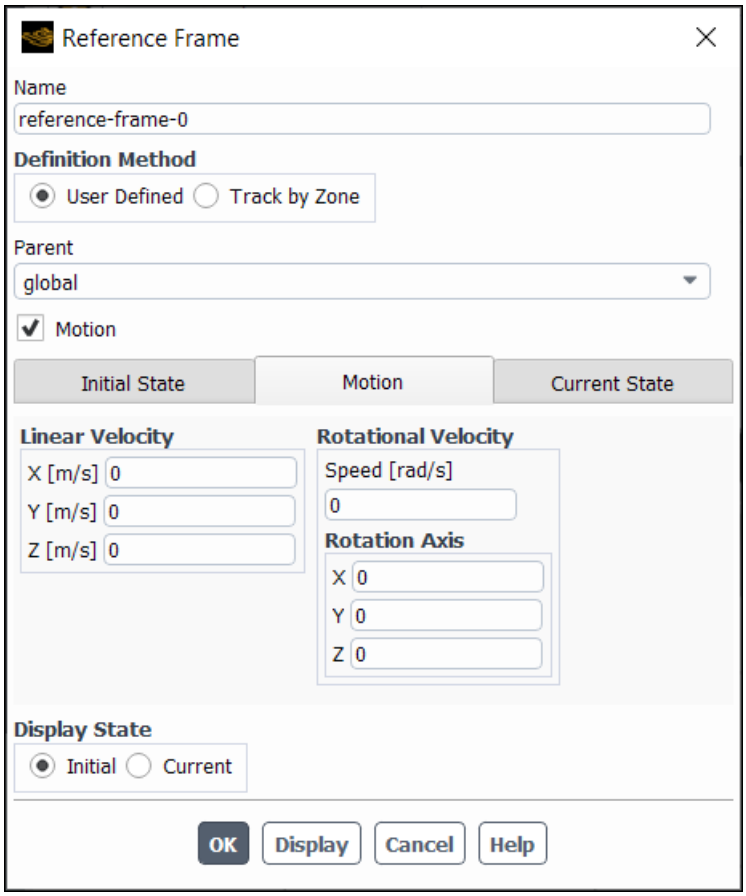
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Track by Zone** allows you to "attach" a reference frame to a cell zone. You can still specify where the reference frame is located and how it is oriented, but you cannot specify any motion. If the zone it is tracking moves, so does the reference frame.

4. Specify either the **Parent** reference frame (when you select **User Defined**) or the **Zone** that the reference frame will track (when you select **Track by Zone**).

5. (Optional, **User Defined** only) enable **Motion** and specify the additional settings that appear in the **Motion** tab.

Figure 6.26: The Motion Tab of the Reference Frame Dialog Box



- **Linear Velocity** is the linear velocity with respect to the parent reference frame orientation.

- **Rotational Velocity** is the rotational velocity with respect to the parent reference frame orientation.

6. Specify the **Origin** of your reference frame.

7. Specify the orientation of the first axis of your reference frame in the **Axis-1 Orientation** group box. Select which axis you are specifying using by selecting either **X**, **Y**, or **Z**. *You can either specify the orientation by providing a point or a direction.*

8. (Optional) Specify the orientation of the second axis of your reference frame. By default, Fluent automatically provides an orientation for the second axis, but you can specify it manually by disabling **Automatic** and providing your own inputs.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

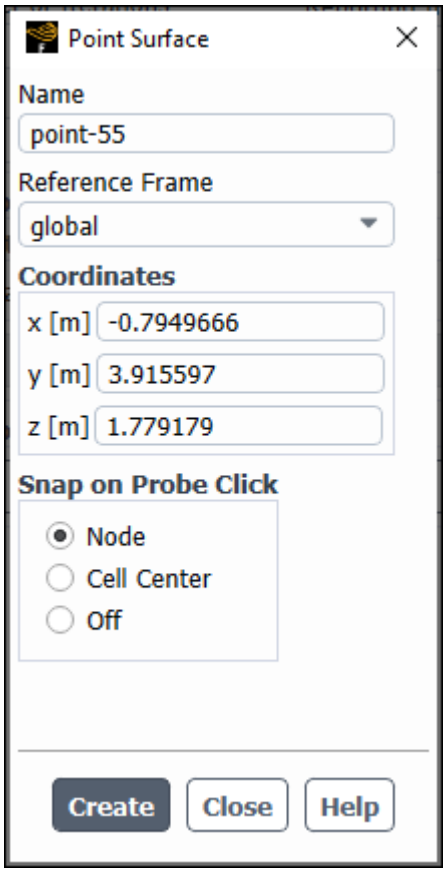
9. Click **OK** to create the reference frame.

The **Current State** field shows you the current origin and axes directions of the reference frame with respect to the global reference frame.

Using Reference Frames

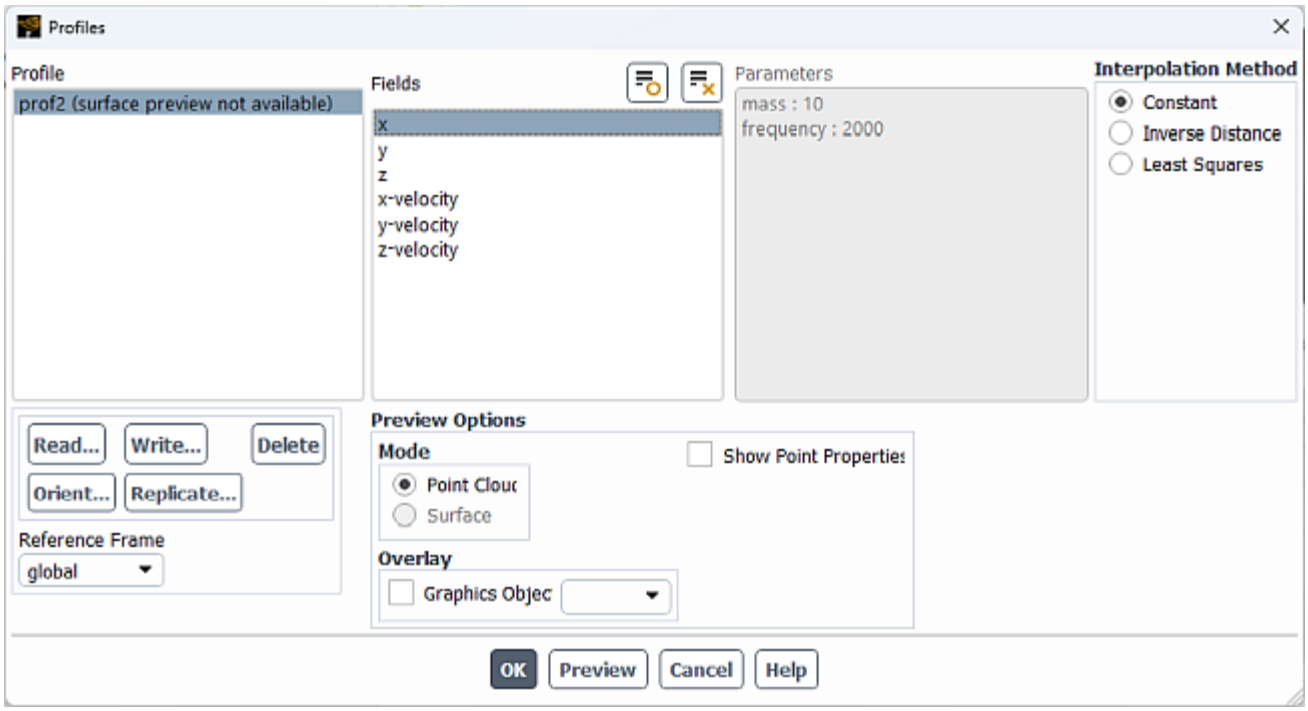
You can use any local reference frame when defining point surfaces and profiles. Just select the desired reference frame from the **Reference Frame** drop-down in the **Point Surface** or **Profile** dialog boxes.

Figure 6.27: Point Surface Creation on Local Reference Frame



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.28: Profile Definition on Local Reference Frame



6.5. Curvilinear Coordinate Systems

You can create multiple Curvilinear Coordinate Systems (CCS) for use in setting up Ansys Fluent cases.

The CCS feature allows you to create local coordinate systems that follow the geometry of the model. For example, you can create a CCS where one of the directions follow the curved shaped of a motor winding. This is especially useful for defining anisotropic thermal conductivity, as described in Anisotropic Thermal Conductivity with Curvilinear Coordinate System (CCS) (p. 2190).

6.5.1. Defining a Curvilinear Coordinate System

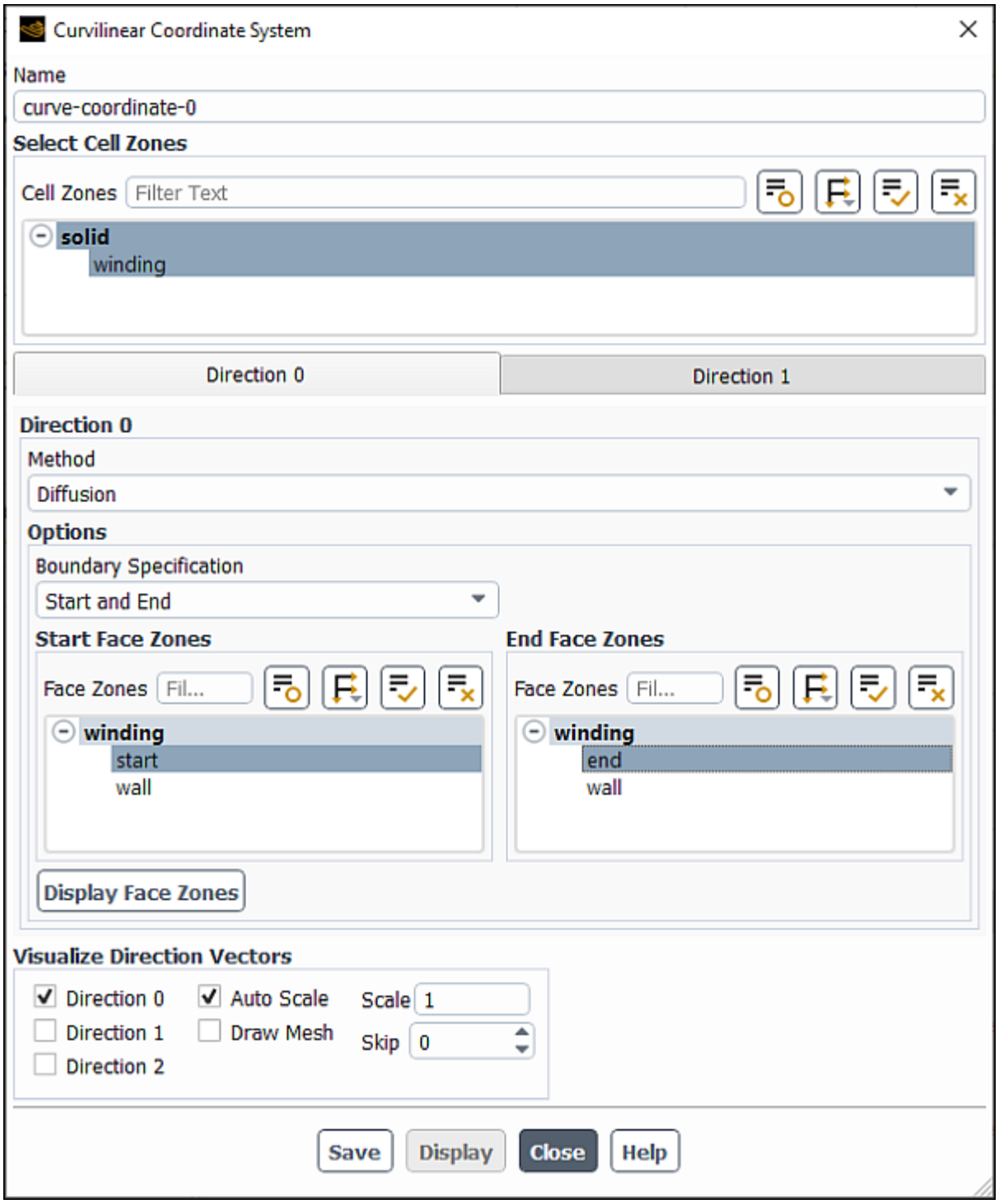
Define a CCS using the following steps:

1. Open the **Curvilinear Coordinate System** dialog box.

   Setup → **Curvilinear Coordinate System** **New...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.29: The Curvilinear Coordinate System Dialog Box



2. Under **Name**, enter a name for the CCS.
3. Under **Select Cell Zones**, select the cell zones for which you want to define the CCS.
4. Define directions for the CCS. **Direction 0** and **Direction 1** must be specified. Direction 2 is computed automatically using the cross product of Direction 0 and 1.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. For **Direction 0** and **Direction 1**, you must specify the direction using the geometry in the chosen cell zone.

   Under **Method**, choose one of the following options:

   - `Diffusion` - The direction of the vector field is specified with **Start and End** face zones. If the geometry is a closed loop, you can use the **Jump** boundary by specifying a coupled wall as a cross section though the loop, which is then split into start and end faces.
   - `Base Vector` - You have the option of specifying user defined vectors if you know one of the directions is aligned with the global coordinate system over the entire geometry.
   - `Vector Projection` (available only for **Direction 1**) - Requires the specification of the **X**, **Y**, and **Z** components of an **Alignment Vector**. The direction of the vector field is defined by the projection of the alignment vector onto the principal vector field (**Direction 0**).

   Important:

   The accuracy of the Vector Projection method is dependent on the specified Alignment Vector and principal vector (Direction 0), therefore it is highly recommended that you visualize and verify the computed vectors when using the Vector Projection method. For example on the importance of visualizing the vector fields, see Curvilinear Coordinate System Example (p. 1167).

6. Under **Visualize Direction Vectors**, there are various options to affect how the CCS is displayed in the graphics window.

   - **Direction 0** - Selects Direction 0 to display in the graphics viewer.
   - **Direction 1** - Selects Direction 1 to display in the graphics viewer.
   - **Direction 2** - Selects Direction 2 to display in the graphics viewer.
   - **Auto Scale** - Scales the vectors to an appropriate size.
   - **Draw Mesh** - Opens the Mesh Display dialog box.
   - **Scale** - Specifies the size of the vectors for display.
   - **Skip** - Thins the number of vectors displayed to improve visibility.

7. Click **Display** to visualize the direction vectors.

   Note:

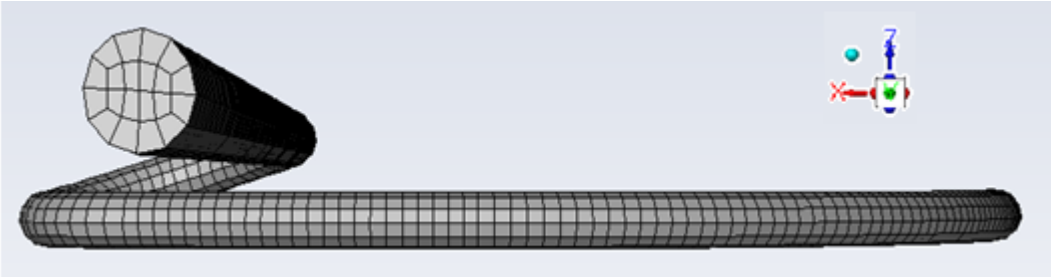
   Direction vectors are not displayed on all interior boundaries.

8. Click **Save** to save the coordinate system.

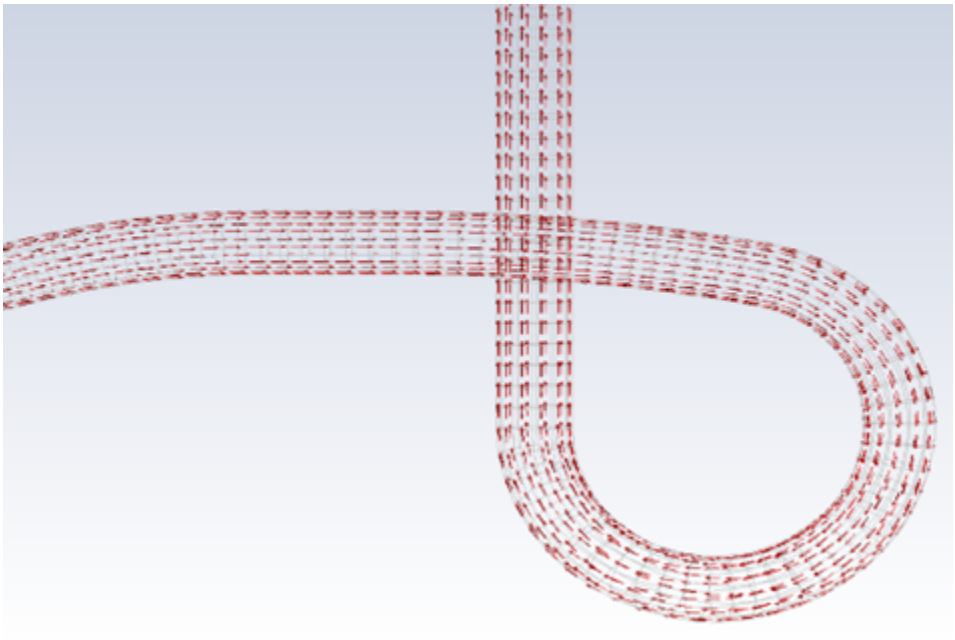
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.5.2. Curvilinear Coordinate System Example

Consider creating a curvilinear coordinate system that follows this coil geometry:

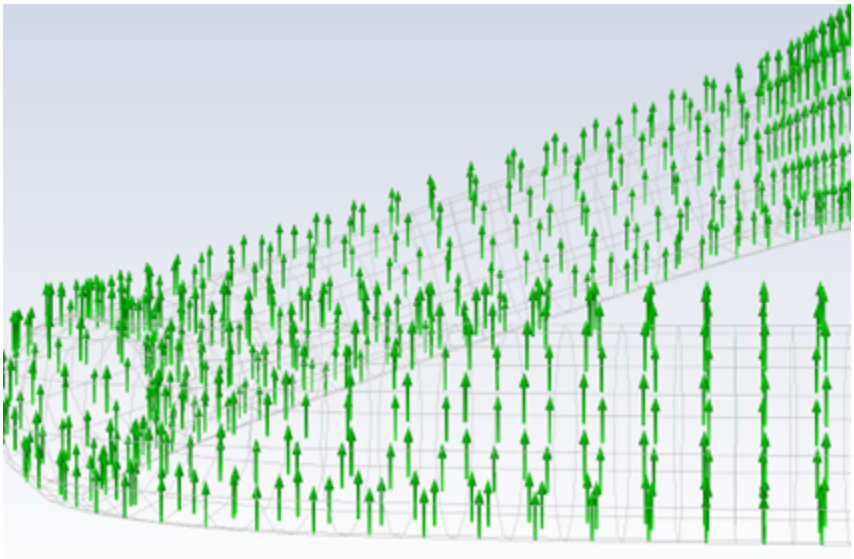


- For **Direction 0**, the `Diffusion` method can be used with the circular end faces being used as the Start Face Zone and End Face Zone. The result for Direction 0 is a vector field that follows the lengthwise curvature of the coil.

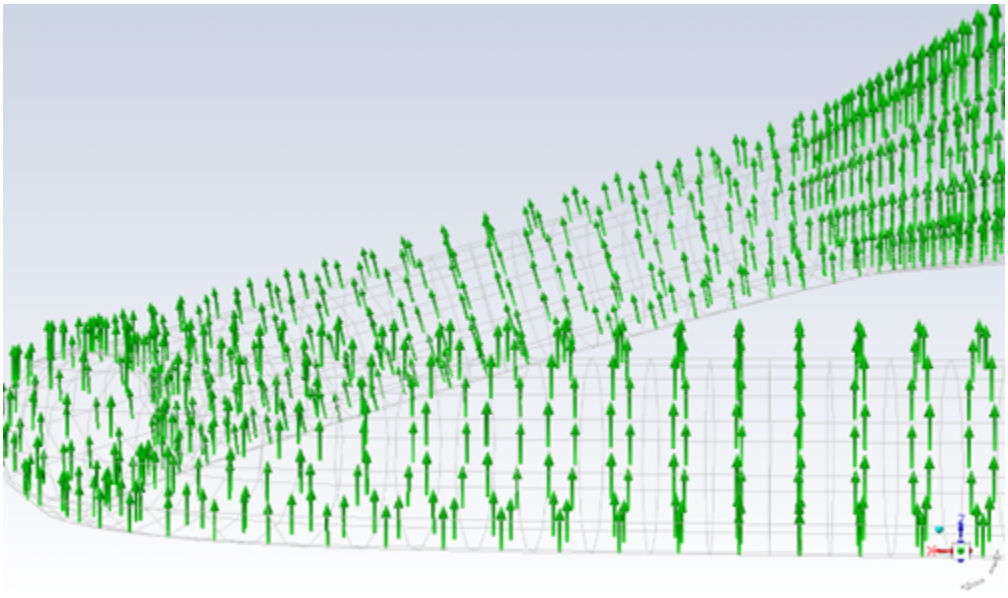


- For **Direction 1**, the `Diffusion` method cannot be used because there is only a cylindrical face (no start or end face exists). If the `Base Vector` method was used with the direction vector in +Z direction (0,0,1), Direction 1 would still not be aligned with curvature of coil.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- For **Direction 1**, the `Vector Projection` method should be used to align the secondary direction with the curvature of the coil. Specifying the **Alignment Vector** in the +Z direction (0,0,1) ensures that the vector field for Direction 1 is normal to Direction 0.



6.5.3. Limitations of Curvilinear Coordinate Systems

The following limitations exist for Curvilinear Coordinate Systems:

- Periodic Boundary face zones are not available in the Face Zone lists for selection of Start, End Jump condition faces.

6.5.4. Limitations with Jump Boundary

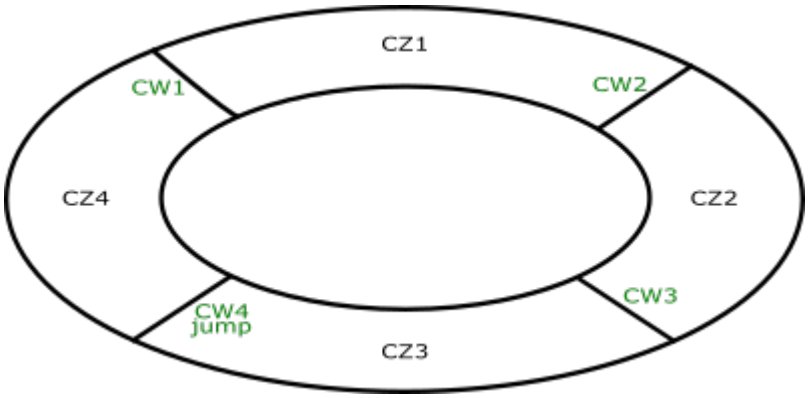
1. Jump boundaries are valid for closed-loop geometries only.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

An open ended geometry with jump boundary will lead to incorrect results. See below for an example of an open-ended geometry. Left and right sides are open, and jump is provided on a coupled wall in between two cell zones C1 and C2.

2. Closed loops cannot have more than one jump boundary.

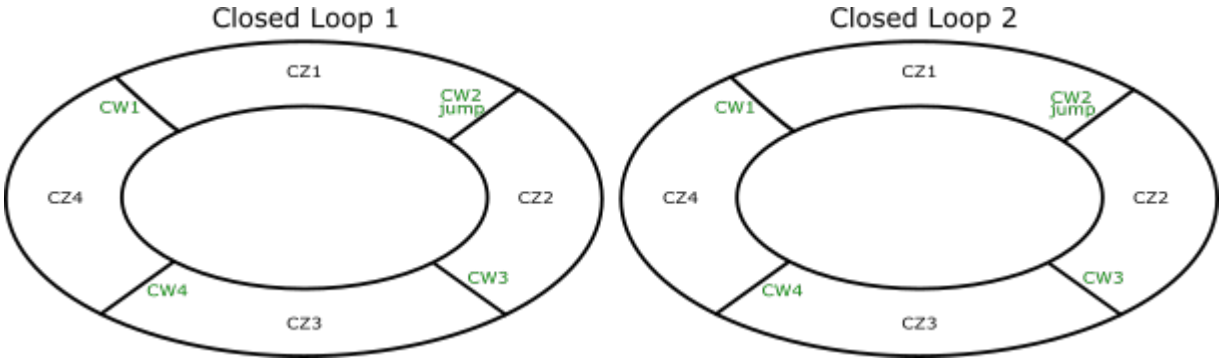
   In the example below, there are four cell zones CZ1, CZ2, CZ3 and CZ4, and 4 coupled walls CW1, CW2, CW3 and CW4. You can only specify one coupled wall inside the closed loop geometry as the jump boundary (CW4 in this case).



   Using the same example, you cannot have all coupled walls with same thread ID. In such a scenario, specifying a jump boundary on one of the coupled walls would result in all four coupled walls having a jump boundary, invalidating the condition of only one jump per closed loop.

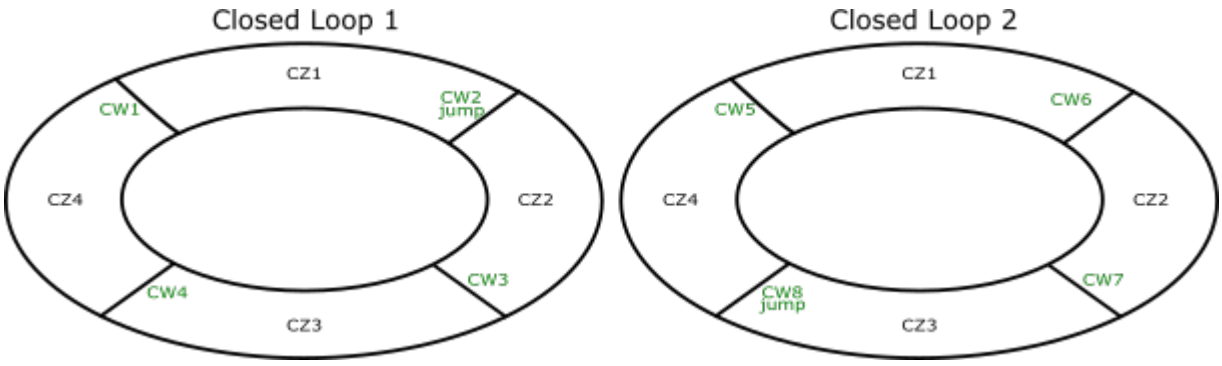
3. Considerations for multiple closed loops

   Multiple closed loops with the same thread ID for coupled walls can work only if one coupled wall is selected as the jump boundary (CW2 in example below). Only one of CW1, CW2, CW3 or CW4 can be selected in this case. Otherwise, there would be more than one jump boundary in each closed loop.



   If different closed loop geometries have different coupled wall thread IDs, then you can select one of the coupled walls in each loop as the jump boundary (CW2 and CW8 in example below).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



6.6. Non-Conformal Meshes

In Ansys Fluent it is possible to use a mesh that has non-conformal interfaces, that is, boundaries between cell zones that connect to each other in which the mesh node locations are not identical. Such non-conformal interfaces permit the cell zones to pass fluxes from one mesh to another.

6.6.1. Non-Conformal Mesh Calculations

6.6.2. Non-Conformal Interface Algorithm

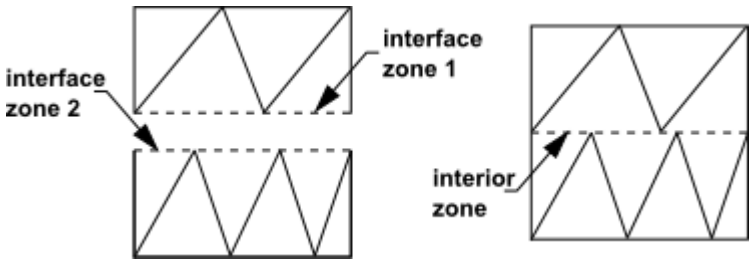
6.6.3. Requirements and Limitations of Non-Conformal Meshes

6.6.4. Using a Non-Conformal Mesh in Ansys Fluent

6.6.1. Non-Conformal Mesh Calculations

To compute the flux across the non-conformal boundary, Ansys Fluent must first compute the intersection between the interface zones that make up the boundary. In the case of a solid-to-solid zone interface of the same material or a fluid-to-fluid zone interface, the resulting intersection produces an interior zone where the two interface zones overlap (see Figure 6.30: Completely Overlapping Mesh Interface Intersection (p. 1170)). In the case of a solid-to-solid zone interface of different materials or a fluid-to-solid zone interface, the boundary is treated as a coupled wall (see The Coupled Wall Option (p. 1176)).

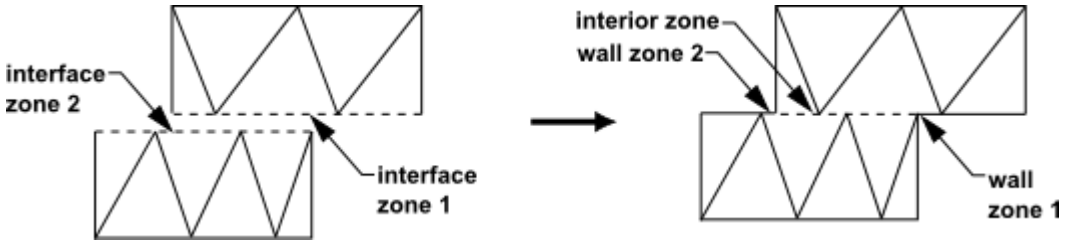
Figure 6.30: Completely Overlapping Mesh Interface Intersection



If one of the interface zones extends beyond the other (Figure 6.31: Partially Overlapping Mesh Interface Intersection (p. 1171)), by default Ansys Fluent will create additional boundary zones for the portion(s) of the boundary where the two interface zones do not overlap; these are referred to as "non-overlapping zones", and you can change their settings and/or zone type at your discretion.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

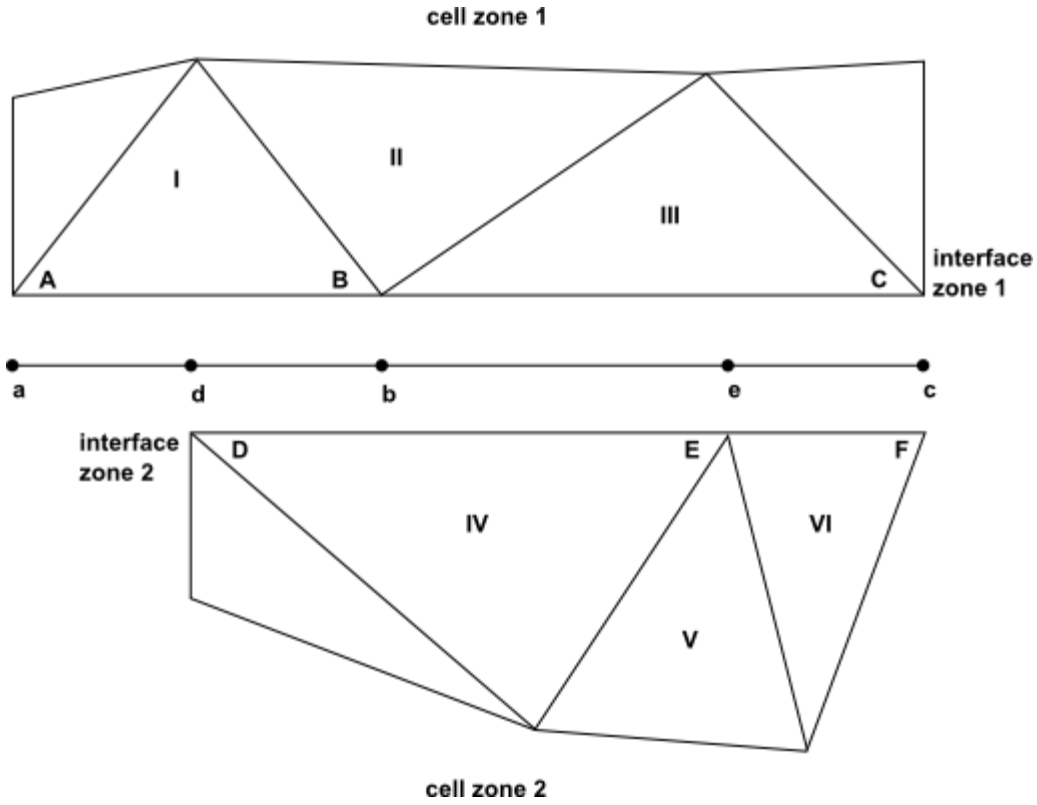
Figure 6.31: Partially Overlapping Mesh Interface Intersection



Fluxes across the mesh interface are computed using the faces resulting from the intersection of the two interface zones, not from the interface zone faces.

In the example shown in Figure 6.32: Two-Dimensional Non-Conformal Mesh Interface (p. 1171), the interface zones are composed of faces A-B and B-C, and faces D-E and E-F.

Figure 6.32: Two-Dimensional Non-Conformal Mesh Interface



The intersection of these zones produces the faces a-d, d-b, b-e, and e-c. Faces produced in the region where the two cell zones overlap (d-b, b-e, and e-c) are grouped to form an interior or coupled wall zone (depending on the types of the adjacent cell zones), while the remaining face (a-d) forms a wall zone.

To compute the flux across the interface into cell IV, face D-E is ignored and instead faces d-b and b-e are used to bring information into cell IV from cells I and III.

While the previous discussion described the default treatment of a non-conformal interface, there are several options you can enable at the interface in order to revise the treatment of the fluxes and/or reduce the memory usage and processing time:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- periodic boundary condition
- periodic repeats
- coupled wall
- matching
- mapped
- static

Note:

Only the coupled wall option is available for mesh interfaces created using the default one-to-one interface creation method.

These non-conformal interface options are described in the following sections.

6.6.1.1. The Periodic Boundary Condition Option
6.6.1.2. The Periodic Repeats Option
6.6.1.3. The Coupled Wall Option
6.6.1.4. Matching Option
6.6.1.5. The Mapped Option
6.6.1.6. The Static Option
6.6.1.7. Interface Zones Automatic Naming Conventions

6.6.1.1. The Periodic Boundary Condition Option

Non-conformal interfaces can be used to implement a periodic boundary condition like that described for conformal periodic boundaries (see Periodic Boundary Conditions (p. 1501)). The advantage of using a mesh interface is that, unlike the standard periodic boundary condition, the nodes of the two zones do not have to match one-for-one.

The interface zones that utilize the periodic boundary condition option (Figure 6.33: Non-Conformal Periodic Boundary Condition (Translational) (p. 1173) and Figure 6.34: Non-Conformal Periodic Boundary Condition (Rotational) (p. 1174)) are coupled in the manner described in the previous section, except that the zones do not overlap (that is, the zones are not spatially coincident at any point). In order to generate the new faces that will be used to compute the fluxes across the interface, the nodes of the first zone are either translated or rotated (about a given axis) onto the other zone. The distance/angle that the nodes are translated/rotated is called the "periodic offset". The new faces will be defined between all of the combined nodes, and then applied to each of the original zones.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.33: Non-Conformal Periodic Boundary Condition (Translational)

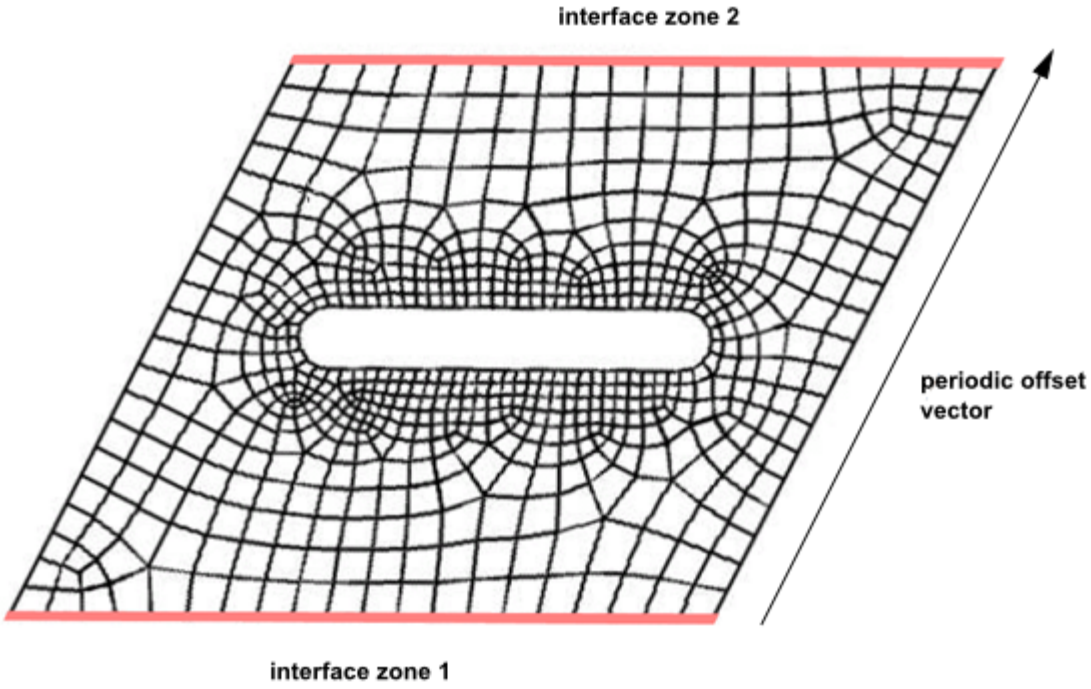
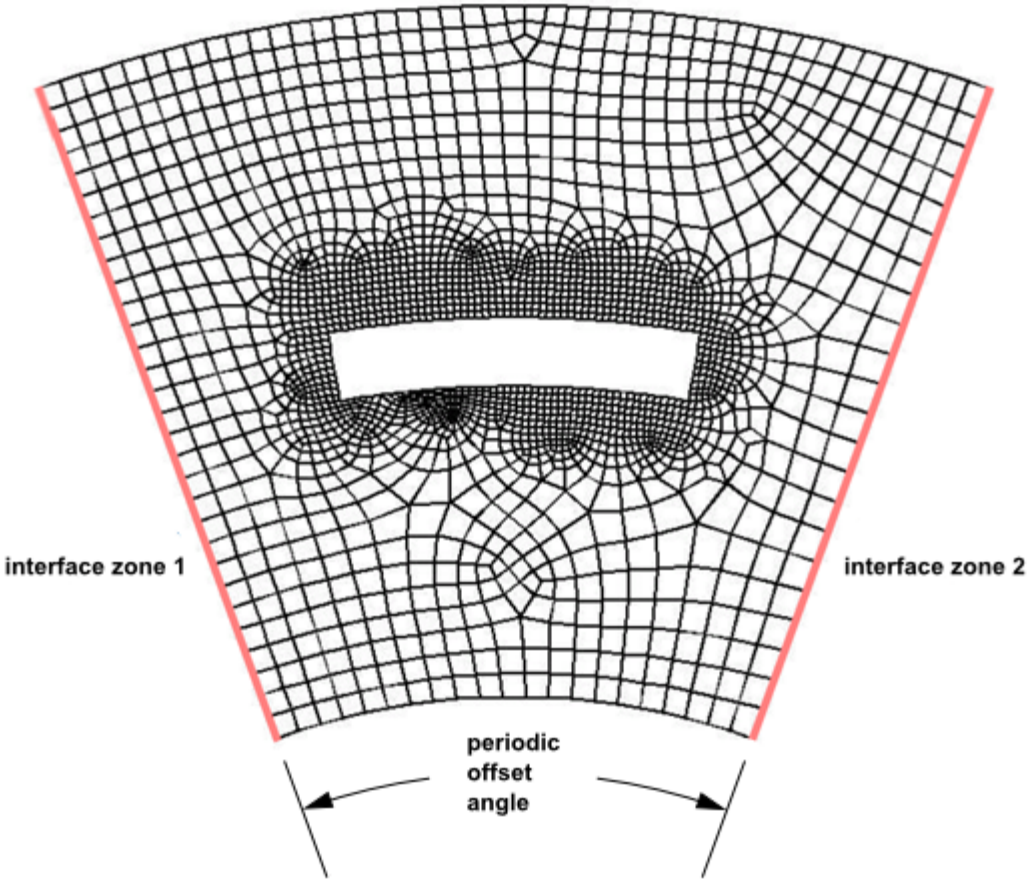


Figure 6.34: Non-Conformal Periodic Boundary Condition (Rotational)



Note:

The **Matching** option is enabled by default with the **Periodic Boundary Condition** option.

6.6.1.2. The Periodic Repeats Option

The periodic repeats option is appropriate when each of the interface zones is adjacent to a non-conformal periodic interface or a pair of conformal periodic zones (see Figure 6.35: Translational Non-Conformal Interface with the Periodic Repeats Option (p. 1175) and Figure 6.36: Rotational Non-Conformal Interface with the Periodic Repeats Option (p. 1176)). The periodic repeats option takes into account the repeating nature of the flow solutions in the two cell zones in the following manner. Wherever the interface zones overlap (that is, wherever interface zone 1 and 2 are spatially coincident), the fluxes on either side of the interface are coupled in the usual way. The portion of interface zone 1 that does not overlap is coupled to the non-overlapping portion of interface zone 2, by translating or rotating the fluxes by the periodic offset. This is similar to the treatment of non-

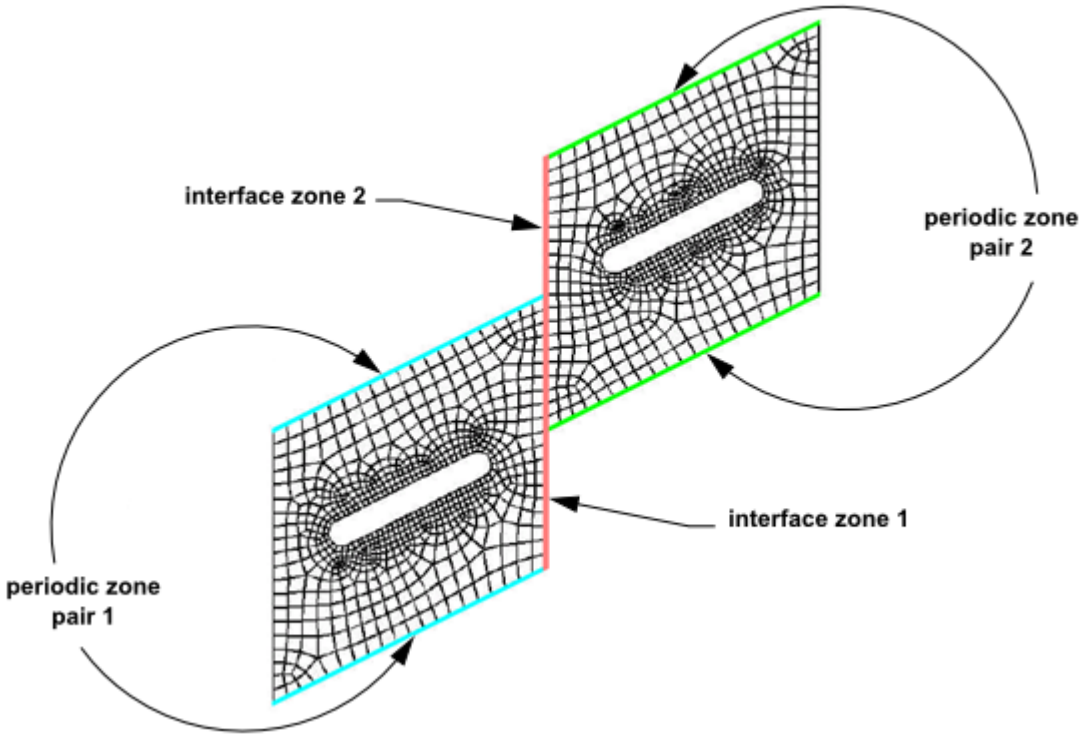
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

conformal periodic boundary conditions. The periodic repeats option is typically used in conjunction with the sliding mesh model when simulating the interface between a rotor and stator.

Note:

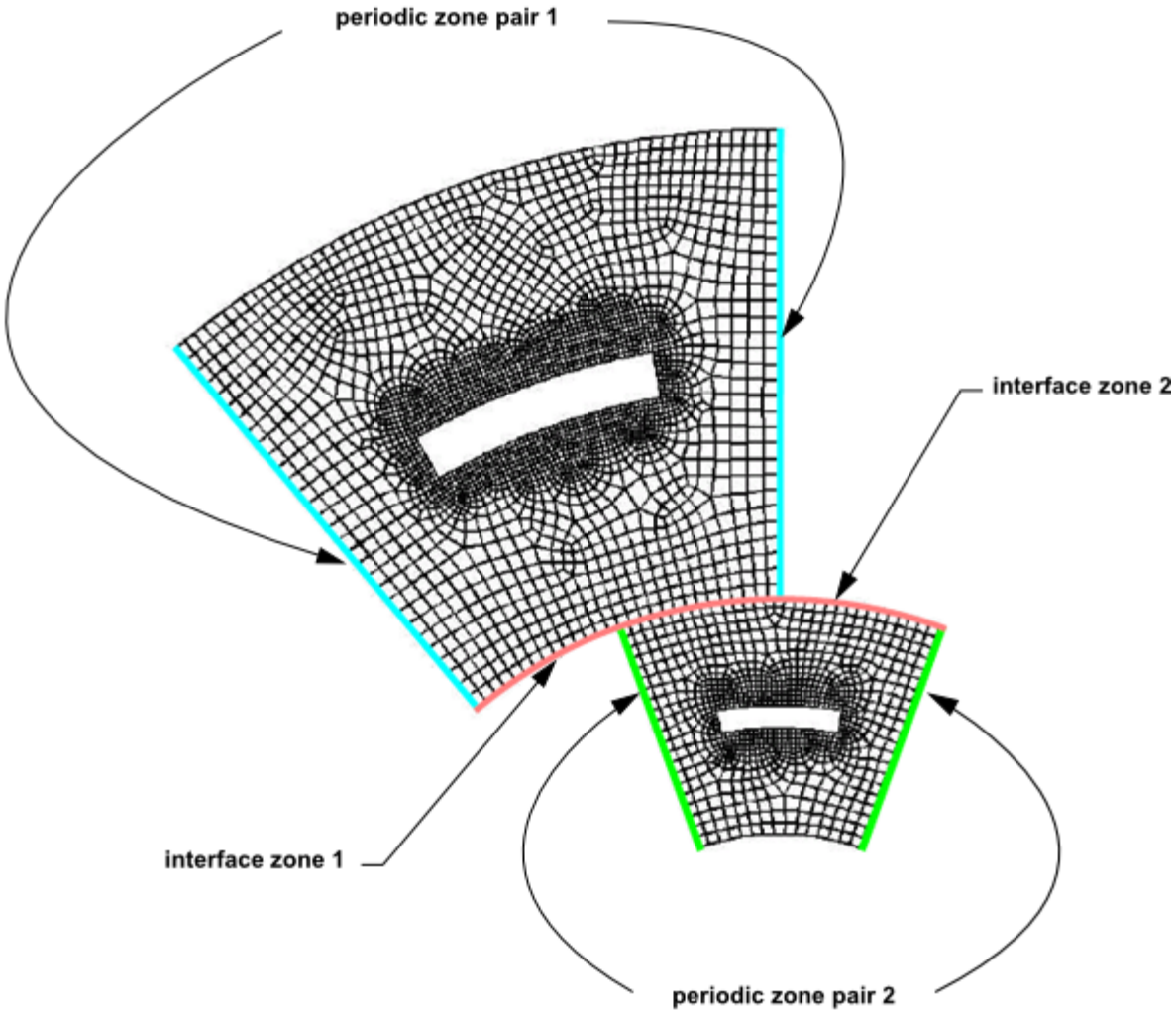
The periodic repeats option is not supported if the interface zone is adjacent to a solid zone.

Figure 6.35: Translational Non-Conformal Interface with the Periodic Repeats Option



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.36: Rotational Non-Conformal Interface with the Periodic Repeats Option



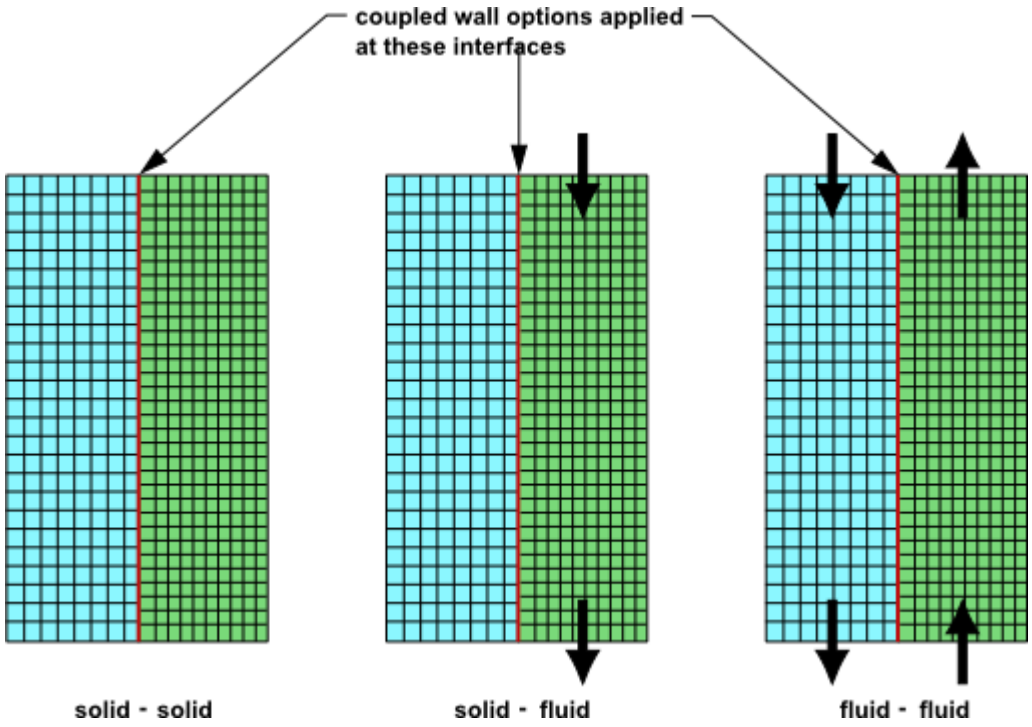
6.6.1.3. The Coupled Wall Option

As described previously, the typical function of non-conformal interfaces is to couple fluid zones, so as to permit fluid flow to pass from one mesh interface to the other. Another available option is to create a coupled wall boundary at the interface. In such a case, fluid flow would not pass across the interface, as the interface is acting as a wall zone. Coupled wall heat transfer, on the other hand, would be permitted. Such treatment is applied by default in the case of a solid-to-solid zone interface of different materials or a fluid-to-solid zone interface. For fluid-to-fluid zone interfaces, you must enable the **Coupled Wall** option to have such treatment; for example, you can model a thin wall or baffle separating the two fluid zones. Figure 6.37: Non-Conformal Coupled Wall Interfaces (p. 1177) illustrates coupled walls with both solid and fluid zones.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that coupled walls can also make use of the periodic repeats option. That is, both options can be invoked simultaneously. For details see Using a Non-Conformal Mesh in Ansys Fluent (p. 1185).

Figure 6.37: Non-Conformal Coupled Wall Interfaces



6.6.1.4. Matching Option

When two interface zones do not match well (which could result from different mesh topologies, geometric misalignment, or the existence of large gaps), wall zones will be created on the interface. If, in such cases, you want to have only internal zones generated on the interface (that is, interior or wall / shadow pairs), it is best to use a matching interface. If the interface zones are completely overlapping (see Figure 6.30: Completely Overlapping Mesh Interface Intersection (p. 1170)) as opposed to partially overlapping (see Figure 6.31: Partially Overlapping Mesh Interface Intersection (p. 1171)) and so no wall zones are expected to be created on the mesh interface, it is recommended that you select the **Matching** option.

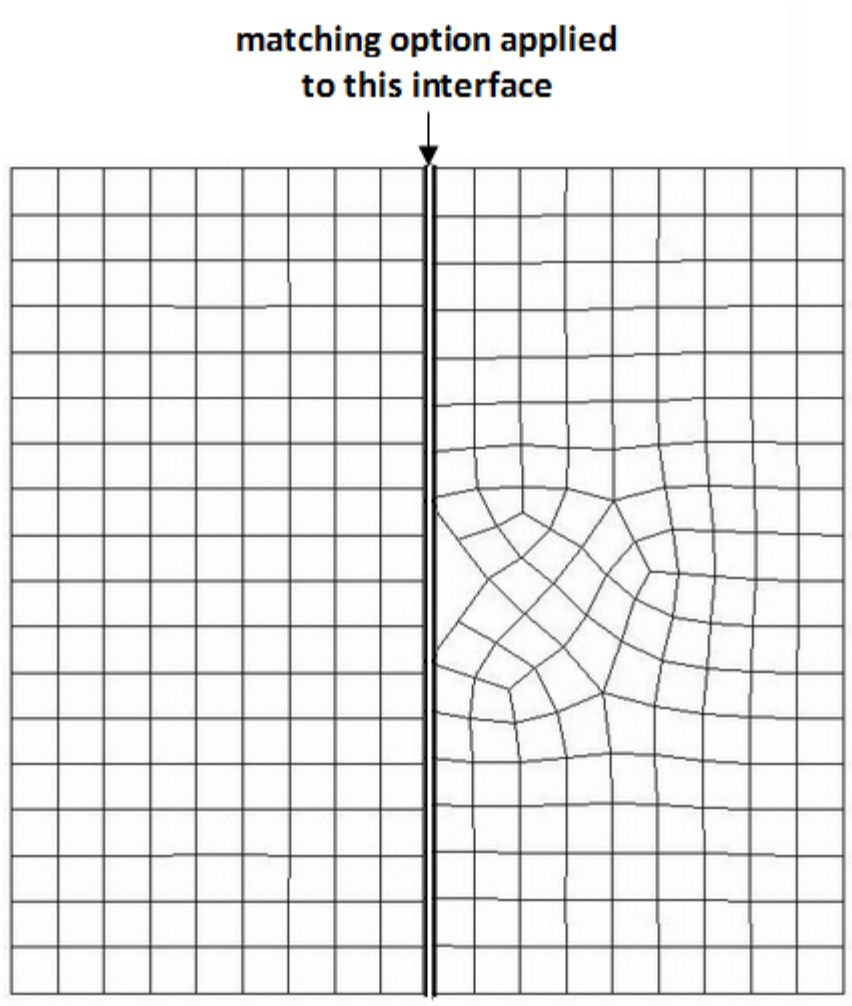
Note:

The **Matching** option is enabled by default with the **Periodic Boundary Condition** option.

Figure 6.38: Matching Non-Conformal Wall Interfaces (p. 1178) below shows two interface zones that do not match up well in the center—the interface zone on the right has an inverted spike. Even with such a large mismatch, the **Matching** option still enables you to create an interface with only the interior zone.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.38: Matching Non-Conformal Wall Interfaces



When the **Matching** option is selected, Ansys Fluent checks if one interface zone sits on top of the other and displays messages if there is a penetration of cell zones that are connected through a mesh interface (therefore making the interface invalid for use). Since the severity of zone penetration is different for matching interfaces and non-matching interfaces, Ansys Fluent displays different messages accordingly.

For example, for non-matching interfaces:

```
Info: Interface zones penetrate each other for mesh interface (interface-name). This could adversely
affect your solution.
```

Likewise, for matching interfaces:

```
Warning: Interface zones penetrate each other for mesh interface (interface-name).
         This could generate left-handed faces on the interface and make it invalid for use.
```

6.6.1.5. The Mapped Option

The **Mapped** option is an alternative approach for modeling coupled walls between zones. This approach provides better handling for interface zones that penetrate each other or have gaps

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

between them. The **Mapped** option is useful in situations where using the standard non-conformal interface formulations causes the simulation to fail (as may occur when there is interface penetration or gaps between interfaces, causing the creation of left-handed faces at the interface).

Figure 6.39: Non-Conformal Mapped Interface with a Gap and Penetration

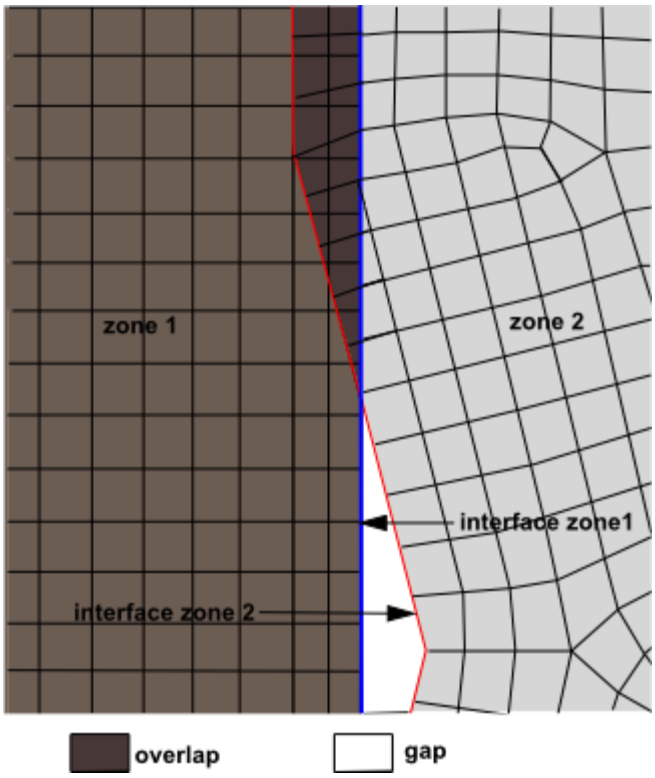


Figure 6.39: Non-Conformal Mapped Interface with a Gap and Penetration (p. 1179) shows significant penetration between the two interface zones and large gap. The mapped option allows you to create a coupled wall even with the large amount of penetration and gap shown in this image.

The **Mapped** option only works in conjunction with the **Coupled Wall** option. Enabling the **Mapped** automatically enables **Coupled Wall**.

You can use the Mapped Interface Options Dialog Box (p. 5520) to modify the global settings for the mapped interfaces, as well as to convert existing mesh interfaces (either all of them or just those that have penetrating zones) to use the **Mapped** option.

You can confirm that you created good quality mapped interfaces by performing a mesh check. If you have mapped interfaces that did not pass the quality check, Fluent will suggest that you use the text command `define/mesh-interface/improve-quality`, which allows you to increase the interface tolerance to improve the quality of the mapped interfaces.

In problems set up in version 19.2 or earlier, where you have a solid zone adjacent to a fluid zone (conjugate heat transfer) with a mapped mesh interface between them, it is recommended you switch from the explicit coupling method to the implicit method using the text command `/define/models/cht/implicit-coupling?`. The implicit method produces better convergence behavior compared to the explicit method for these fluid-solid pairs. Note that the implicit method is the now the default.

Note the following limitations when using the **Mapped** option:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The **Mapped** option can only be applied if at least one side of the interface consists of only solid zones.
- The **Mapped** option is not compatible with shell conduction.
- Thermal coupling calculations will not be performed at mapped interfaces for the Eulerian multiphase model.
- No other interface options besides **Coupled Wall** are allowed with the **Mapped** option.
- For the S2S model, the **Face to Face** basis for the view factors is not compatible with mapped interfaces for polyhedral meshes.
- Mapped interfaces are not supported with the density-based solver.
- Mapped interfaces are not supported with the Lagrangian discrete phase model.
- Mapped interfaces are not supported with moving / deforming meshes.

6.6.1.6. The Static Option

For a default mesh interface or when the **Coupled** and/or **Matching** options are enabled, Ansys Fluent will automatically create zones that will be needed if the interface should change due to motion (from moving reference frames or sliding meshes) or deformation. If you know in advance that the interface zones will not be moving or deforming relative to each other at the interface, then (and only then) it is recommended that you enable the **Static** option. The **Static** option prevents the creation of such zones, and thus reduces the memory usage and processing time (for both the creation of the interface and the running of the calculation). This reduction can be significant, especially when there are many zones on both sides of the interface.

> **Warning:**
>
> If the **Static** option is applied at an interface that does end up undergoing relative motion or deformation, the results will be incorrect.

Note that the **Static** option is not limited to steady-state cases.

6.6.1.7. Interface Zones Automatic Naming Conventions

The following sub-sections describe the automatic naming conventions for each type of non-conformal interface. Note that in these descriptions:

- <*mesh interface name*> is user specified
- <*connection number side 1*> is a value between 1 and *m*, where *m* is the number of zones in interface zone side 1
- <*connection number side 2*> is a value between 1 and *n*, where *n* is the number of zones in interface zone side 2
- <*side number*> is 1 or 2

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.6.1.7.1. Default (No Options Enabled)
6.6.1.7.2. Periodic Boundary Condition
6.6.1.7.3. Periodic Repeats
6.6.1.7.4. Coupled Wall
6.6.1.7.5. Matching
6.6.1.7.6. Mapped
6.6.1.7.7. Static

6.6.1.7.1. Default (No Options Enabled)

The automatic naming convention for the non-overlapping zones is as follows: <*interface zone name*> - *non-overlapping*.

The automatic naming convention for interface wall zone side 1 is as follows: <*mesh interface name*> - *wall1* - <*connection number side 1*> - <*connection number side 2*>.

The automatic naming convention for interface wall zone side 2 is as follows: <*mesh interface name*> - *wall1* - <*connection number side 1*> - <*connection number side 2*> - *shadow*.

The automatic naming convention for interface interior zones from a one-to-one pairing (the default) is as follows: <*mesh interface name*>.

The automatic naming convention for interface interior zones that are from a many-to-many pairing is as follows: <*mesh interface name*> - *interior* - <*connection number side 1*> - <*connection number side 2*>.

6.6.1.7.2. Periodic Boundary Condition

The automatic naming convention for the non-overlapping zones is as follows: <*interface zone name*> - *non-overlapping*.

The automatic naming convention for interface interior zones is as follows: <*mesh interface name*>.

6.6.1.7.3. Periodic Repeats

The automatic naming convention for interface interior zones is as follows: <*mesh interface name*> - *periodic*. Note that unnamed zones may be created for use in the solver and cannot be changed or modified by the user.

6.6.1.7.4. Coupled Wall

The automatic naming convention for the non-overlapping zones is as follows: <*interface zone name*> - *non-overlapping*.

The automatic naming convention for interface wall zones from a one-to-one pairing (the default) is as follows: <*mesh interface name*> and <*mesh interface name*> - *shadow*.

The automatic naming convention for side 1 of interface wall zones that are from a many-to-many pairing is as follows: <*mesh interface name*> - *wall1* - <*connection number side 1*> - <*connection number side 2*>; and then interface wall zone side 2 is named: <*mesh interface name*> - *wall1* - <*connection number side 1*> - <*connection number side 2*> - *shadow*. For example:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
> define/mesh-interfaces/list

List of Mesh Interfaces


Interface Name:     cw:01
Interface Options:  Coupled


       Description                    Name                 ID         Area        Area Percentage
---------------------------  -------------------------  -------  --------------  ---------------
      Interface-Zone-Side-1                        a2       17    0.1707190424
      Interface-Zone-Side-1                        a1       13    0.1707191467
      Interface-Zone-Side-2                        b1        4   36.7499923706
Non-Overlapping-Zone-Side-2          b1-non-overlapping       53   36.4085540771        99.070915%
      Interface-Wall-Zone-1             cw:01-wall1-1-1       50    0.1706306934        99.948250%
      Interface-Wall-Zone-1             cw:01-wall1-2-1       49    0.1706308126        99.948257%
      Interface-Wall-Zone-2      cw:01-wall1-1-1-shadow       54    0.1706306934         0.464301%
      Interface-Wall-Zone-2      cw:01-wall1-2-1-shadow       55    0.1706308126         0.464302%
```

6.6.1.7.5. Matching

The automatic naming convention for interface interior zones is as follows: <*mesh interface name*> - *interior* - <*connection number side 1*> - <*connection number side 2*>.

6.6.1.7.6. Mapped

The automatic naming convention for the non-overlapping zones is as follows: <*interface zone name*> - *non-overlapping*.

The automatic naming convention for interface wall zones is as follows: <*mesh interface name*> - *wall* <*side number*> - <*connection number side 1*> - <*connection number side 2*>.

For example,

```
> define/mesh-interfaces/list

List of Mesh Interfaces


Interface Name:     map:01
Interface Options:  Mapped


   Description                     Name                 ID          Area         Area Percentage
---------------------------  -------------------------  -------  --------------  ---------------
      Interface-Zone-Side-1                       a1       18   1.01046020e-03
      Interface-Zone-Side-2                       b1        6   1.01029174e-03
Non-Overlapping-Zone-Side-1         a1-non-overlapping       30   1.10513203e-04         10.9369%
Non-Overlapping-Zone-Side-2         b1-non-overlapping       31   1.10303146e-04         10.9179%
 Interface-Wall-Zone-Side-1           map:01-wall1-1-1       14   8.99947016e-04         89.0631%
 Interface-Wall-Zone-Side-2           map:01-wall2-1-1       29   8.99988518e-04         89.0820%
```

6.6.1.7.7. Static

The **Static** option does not affect the naming conventions of the zones; that is, the zone names will be based entirely on the other settings in the **Interface Options** group box of the **Edit Mesh Interfaces** or **Create/Edit Mesh Interfaces** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.6.2. Non-Conformal Interface Algorithm

In the current version of Ansys Fluent, non-conformal interface calculations are handled using one of three approaches depending on the type of interface. **Coupled Wall** interfaces are handled using interface boundary metrics and **Mapped** interfaces use a projection approach. The interface boundary metrics and projection approach are both robust algorithms that avoid creating left-handed faces and are particularly useful when you have complex geometries.

The remaining types of non-conformal interfaces are handled using a virtual polygon approach. The virtual polygon approach stores the area vector and centroid of the polygon faces. This approach does not involve node movement and cells are not necessarily water-tight cells. Hence gradients are corrected to take into account the missing face area. Note that in transient cases, the stationary non-conformal interfaces will be automatically preserved by Ansys Fluent during mesh update.

Previous versions (16.0 and earlier) of Fluent used the virtual polygon approach for all non-conformal interfaces. The interface boundary metrics is a more robust approach for creating coupled wall interfaces than the virtual polygon approach because it avoids creating left-handed faces. Older versions of Fluent (6.1 or earlier) used a triangular face approach, which triangulated the polygon intersection faces and stored triangular faces. This approach involved node movement and water-tight cells, and was not as stable as the virtual polygon approach. Note that case files created in releases prior to 16.0 can be read and run normally in the current version of Fluent unless you recreate any of the interfaces. Recreated coupled wall interfaces will use interface boundary metrics and all other recreated interfaces will use the virtual polygon approach.

For a sliding mesh case that has interface zones that do not match well at the mesh interface, the resulting mesh may have left-handed faces. Such faces can cause the mesh to be invalid, and so Ansys Fluent automatically attempts to make the faces "right handed". If any left-handed faces remain, you can remove them using the `define/mesh-interfaces/remove-left-handed-interface-faces?` text command.

Left-handed cells can also be created for the geometries that contain sharp edges and corners, which may affect the final solution. For such geometries, you should first separate the zones and then create the interfaces separately to get the better solution.

The additional input of the angle/translation vector at the `angle/translation-vector` prompt in the console may be required to recreate face-periodic interfaces. Also, with the current mesh interface algorithm in parallel, there is no need for encapsulation.

6.6.3. Requirements and Limitations of Non-Conformal Meshes

This section describes the requirements and limitations of non-conformal meshes:

- The mesh interface can be of any shape (including a non-planar surface, in 3D), provided that the two interface boundaries are based on the same geometry. If there are sharp features (for example, 90-degree angles) or curvature in the mesh, it is especially important that both sides of the interface closely follow that feature.

  For example, consider the case of two concentric circles that define two fluid zones with a circular, non-conformal interface between them, as shown in Figure 6.40: A Circular Non-Conformal Interface (p. 1184). Because the node spacing on the interface edge of the outer fluid zone is coarse

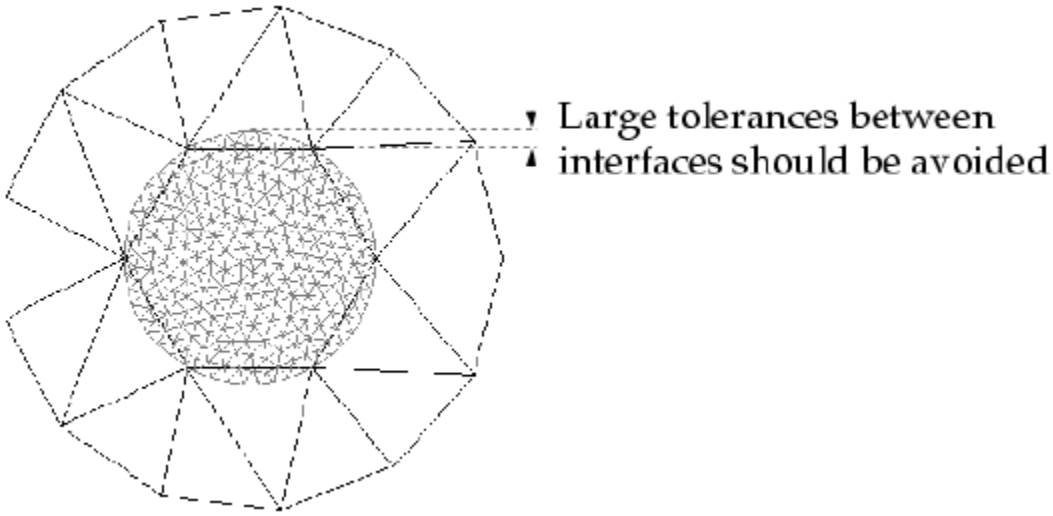
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

compared to the radius of curvature, the interface does not closely follow the feature (in this case, the circular edge.)

Important:

The maximum tolerance between two interfaces should not be larger than their adjacent cell size at that location; that is, no cell should be completely enclosed between two interfaces.

Figure 6.40: A Circular Non-Conformal Interface



- Non-conformal interface surfaces can be visualized, but they do not participate in postprocessing operations. That is, they will show zero values while the corresponding parent walls are non-zero.
- If you create a single mesh with multiple cell zones separated by a non-conformal boundary, you must be sure that each cell zone has a distinct face zone on the non-conformal boundary.

  The face zones for two adjacent cell zones will have the same position and shape, but one will correspond to one cell zone and one to the other. It is also possible to create a separate mesh file for each of the cell zones, and then merge them as described in Reading Multiple Mesh/Case/Data Files (p. 1156).
- All periodic zones must be correctly oriented (either rotational or translational) before you create the non-conformal interface.
- In order for the periodic boundary condition option or periodic repeats option to be valid, the edges of the second interface zone must be offset from the corresponding edges of the first interface zone by a uniform amount (either a uniform translational displacement or a uniform rotation angle). This is not true for non-conformal interfaces in general.

  The periodic boundary condition option has the additional requirement that the angle associated with a rotationally periodic must be able to divide 360 without remainder.
- The periodic repeats option requires that some portion of the two interface zones must overlap (that is, be spatially coincident).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The periodic repeats option requires that the non-overlapping portions of the interface zones must have identical shape and dimensions. If the interface is part of a sliding mesh, you must define the mesh motion such that this criterion is met at all times.
- Note that for 3D cases, you cannot have more than one pair of conformal periodic zones adjacent to each of the interface zones.
- You must not have a single non-conformal interface where part of the interface is made up of a coupled two-sided wall, while another part is not coupled (that is, the normal interface treatment). In such cases, you must break the interface up into two interfaces: one that is a coupled interface, and the other that is a standard fluid-fluid interface. See Using a Non-Conformal Mesh in Ansys Fluent (p. 1185) for information about creating coupled interfaces.
- For simulations that involve the Fluent, Mechanical, and Meshing applications, meshing problems can arise in instances where there are multiple regions and contacts between them. In Fluent, a zone can only exist in a single contact region. The Mechanical and Meshing applications both use a different approach concerning contact regions when compared to Fluent.
- Fluid zones designated as 3D fan zones cannot have non-conformal interfaces.
- When you read a non-conformal interface case file into meshing mode and later switched to the solution mode, note the following limitations:
  - Zone IDs may match, however, the corresponding zone names may be inconsistent.
  - Boundary conditions on intersected threads are not preserved.
  - Unassociated (dangling) non-conformal interface (NCI) surfaces remain present.

6.6.4. Using a Non-Conformal Mesh in Ansys Fluent

If your multiple-zone mesh includes non-conformal boundaries, check if the mesh meets all the requirements (listed in Requirements and Limitations of Non-Conformal Meshes (p. 1183)). This ensures that Ansys Fluent can obtain a solution on the mesh. Then do the following:

1. Read the mesh into Ansys Fluent. If you have multiple mesh files that have not yet been merged, first follow the instructions in Reading Multiple Mesh/Case/Data Files (p. 1156) to merge them into a single mesh.
2. Change the type for all of the zones that make up the non-conformal boundaries as needed. When using the default one-to-one interface creation method, change them to **interface** or **wall** if you plan to use the **Mesh Interfaces** or **Create Mesh Interfaces** dialog box; if you instead plan to use the right-click menu in the outline view or the text command interface, they can be any external boundary zone type. When using the many-to-many interface creation method, you must change them all to type **interface**. For details on changing the zone type, see Changing Cell and Boundary Zone Types (p. 1301).

   Setup → Boundary Conditions

3. By default, the one-to-one interface method is enabled for creating mesh interfaces, so that a single zone is assigned to each side of all of the resulting mesh interfaces. This method has the

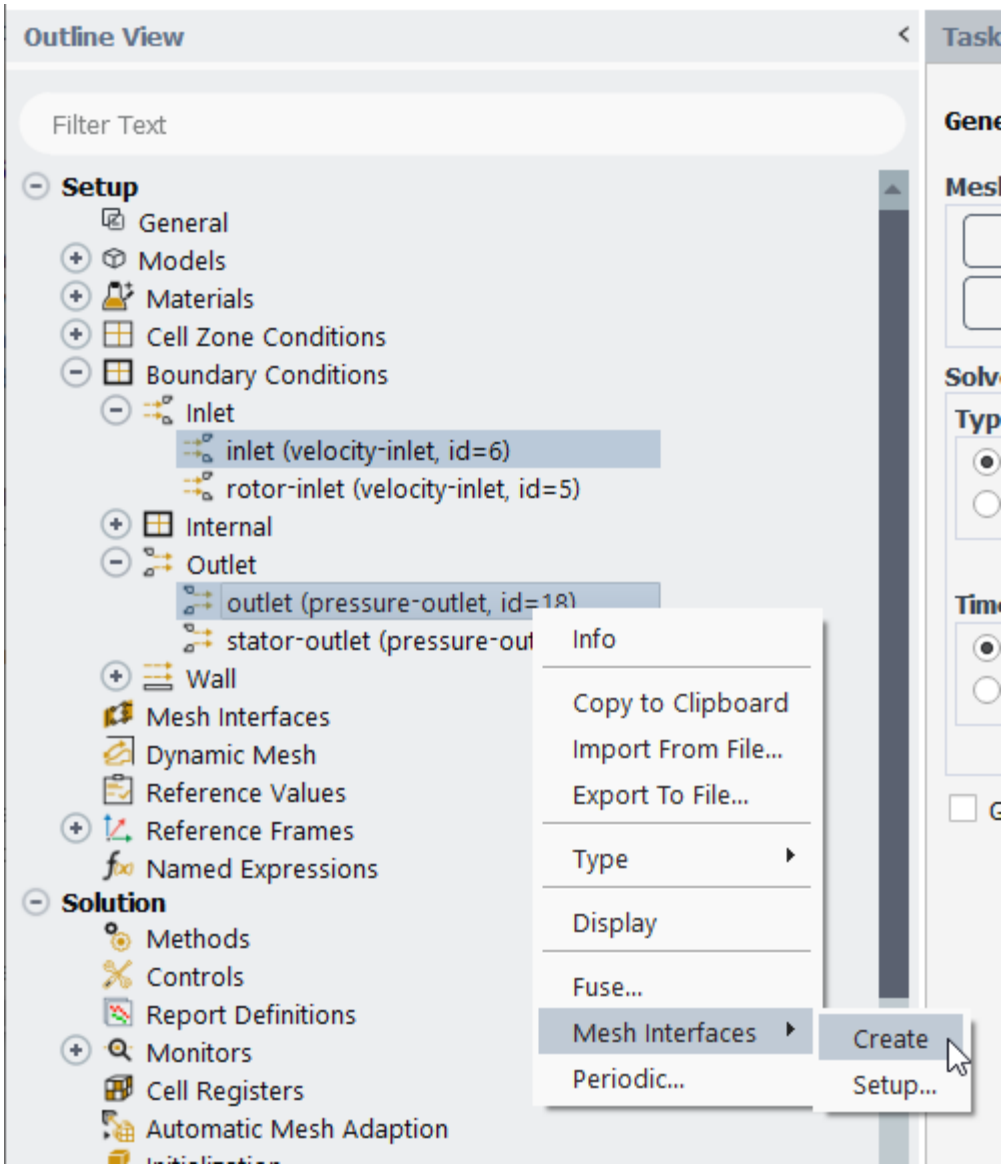
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

following advantages over the many-to-many method (which can produce mesh interfaces that have multiple zones on one or both sides):

- It allows you to have single boundary zones be a part of multiple mesh interfaces, so that it is not necessary to break up large boundary zones in the meshing mode of Fluent prior to creating mesh interfaces (which may compromise the quality of the interface, and is not available for polyhedral meshes). This can improve the accuracy / robustness of the simulation.
- It allows additional boundary zone types besides interface zones to form the two sides a mesh interface:
  - When using the **Mesh Interfaces** or **Create Mesh Interfaces** dialog box (as described in later steps), you can select wall zones. This can simplify your setup when your case has a large number of wall zones, if only some of them will be part of a non-conformal interface.
  - When using the text command interface or selecting them in the outline view and using the right-click menu (**Mesh Interfaces** / **Create**), you can select any type of external boundary zone (that is, any zone that is connected with the cells on only side).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.41: Pairing Boundary Zones Using the Right-Click Menu



- It allows you to create a mesh interface between two zones that are not overlapping at all. This can be useful when setting up a sliding mesh simulation in which the zones begin without overlap, and then slide into a position that includes overlap. Note that to do this, you must select only two zones at a time in the **Mesh Interfaces** dialog box, or you can select two groups of zones in the **Create Mesh Interfaces** dialog box.

You must weigh the advantages listed above against the following limitations:

- No interface options can be enabled at the time of creation, except for the coupled wall option that is automatically enabled for solid-to-fluid/solid mesh interfaces.
- The only editing you can do to such a mesh interface after it has been created is to change the name and/or apply a coupled wall boundary condition (as described in a later step).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You may want to disable the one-to-one interface creation method in order to use certain options, such as the **Periodic Boundary Condition** option or the **Mapped** option. You can disable the one-to-one method by using the following text command:

`define → mesh-interfaces → one-to-one-pairing?`

Note that all of the mesh interfaces in a case file should be created using the same method. If mesh interfaces already exist and you use the previous text command to change the method, you will be prompted to authorize that the existing mesh interfaces are deleted (when switching to the many-to-many method) or converted (when switching to the one-to-one method). When converting, all incompatible settings will be disabled. Conversion can be useful if you want to update old case files that were created in an earlier version of Fluent when the one-to-one method was not available or was not enabled by default.

4. (optional) When using the default one-to-one interface creation method, there are optional settings that may be useful:

   - You can define settings related to the names of the resulting mesh interfaces.

     One option pertains to the suffix added to the end of the name: by default the suffix includes the names of the associated boundary zones, and you can use the following text command to instead specify that the suffix is omitted, replaced with the IDs of the boundary zones, or replaced with the names of the adjacent cell zones.

     `define → mesh-interfaces → auto-options → naming-option`

     You can also change the default interface name prefix by using the following text command:

     `define → mesh-interfaces → auto-options → set-default-name-prefix`

   - You can enable the following text command so that the boundary conditions of the original zones are copied to the resulting coupled wall pair when a mesh interface is created at a solid-to-solid zone interface of different materials or a fluid-to-solid zone interface. This can save you from having to edit such zones later on. Note that the coupled wall pair is thermally coupled.

     `define → mesh-interfaces → coupled-interfaces-inherit-bcs?`

   - If you want to ensure that mesh interfaces are only created between different cell zones, you can enable the following text command:

     `define → mesh-interfaces → auto-options → pairing-between-different-cell-zones-only?`

   - When more than two zones are selected in the **Boundary Zones** group box in the **Mesh Interface** dialog box, default tolerances are used to determine which zones can be paired up to make the mesh interfaces. These default tolerances may not be suitable for your application if there are gaps between interfaces, multiple mesh interfaces in proximity, or under-resolved interfaces with a large curvature. If you find that the default tolerances are not producing the expected number of mesh interfaces, or the correct mesh interface areas, you can use the following text command to enable adjustable tolerances:

     `define → mesh-interfaces → auto-options → set-one-to-one-pairing-tolerance`

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

With adjustable tolerances enabled, the tolerances are evaluated using the product of a specified length factor (with a default value of 1) and the largest mesh length scale of the interface boundaries. You can reduce the length factor if the resulting interface areas are too large or if boundary zones that you do not want paired are paired. An increase in the length factor is recommended if any expected interfaces are not created, or if the intersection areas are smaller than expected.

- You can use the following text command to set the minimum area percentage of overlap for mesh interface creation. If zones are paired using the automatic method (as described in a later step), after the mesh interfaces are created, a check will be done: if the area that the two zones overlap divided by the area of the zone is less than this value for both zones in any mesh interface, then that mesh interface will be deleted. This can help you avoid unnecessary or unintentional mesh interfaces. By default the minimum area percentage is set to 0, so no pairings are deleted. When you are setting up a sliding mesh simulation in which the zones begin with little or no overlap and then slide into a position that includes more overlap, you should make sure the minimum area percentage is set such that it does not delete your intended mesh interface.

  `define` → `mesh-interfaces` → `auto-options` → `set-minimum-area-percentage`

5. Open the Mesh Interfaces Dialog Box (p. 5525).

   Setup → **Mesh Interfaces** **New...**

Figure 6.42: The Mesh Interfaces Dialog Box for the One-to-One Method

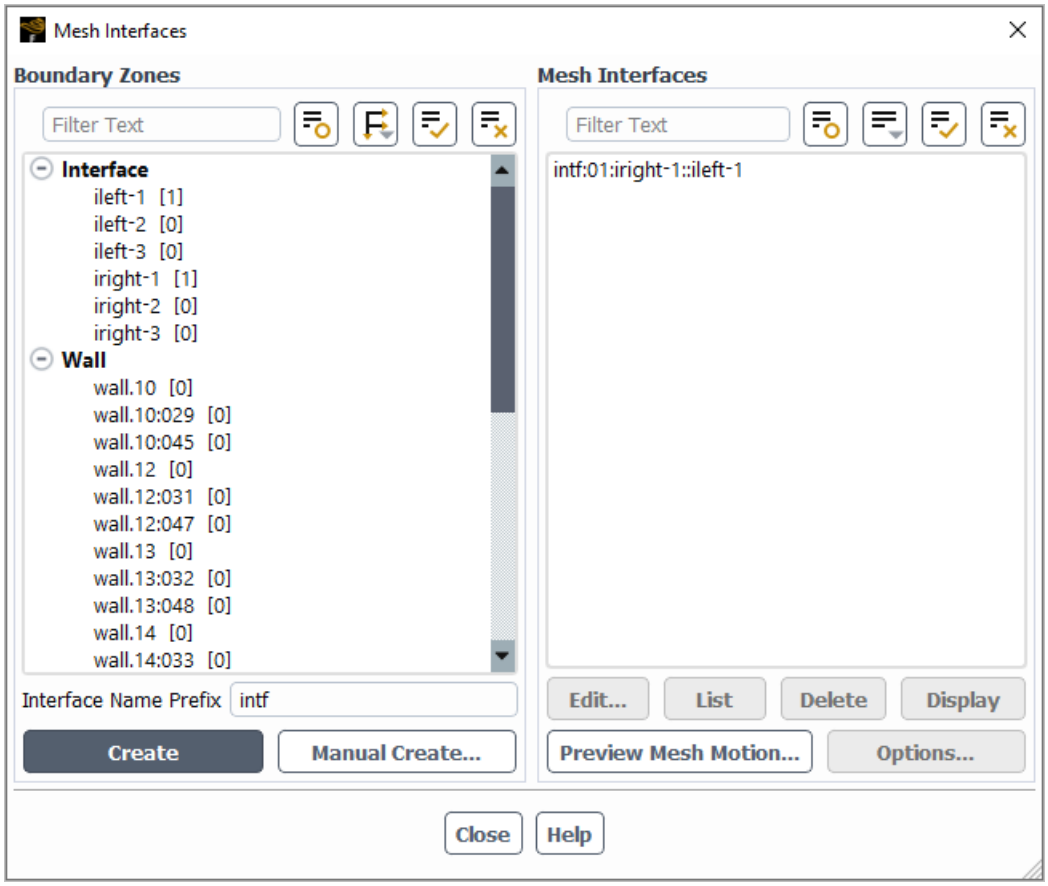
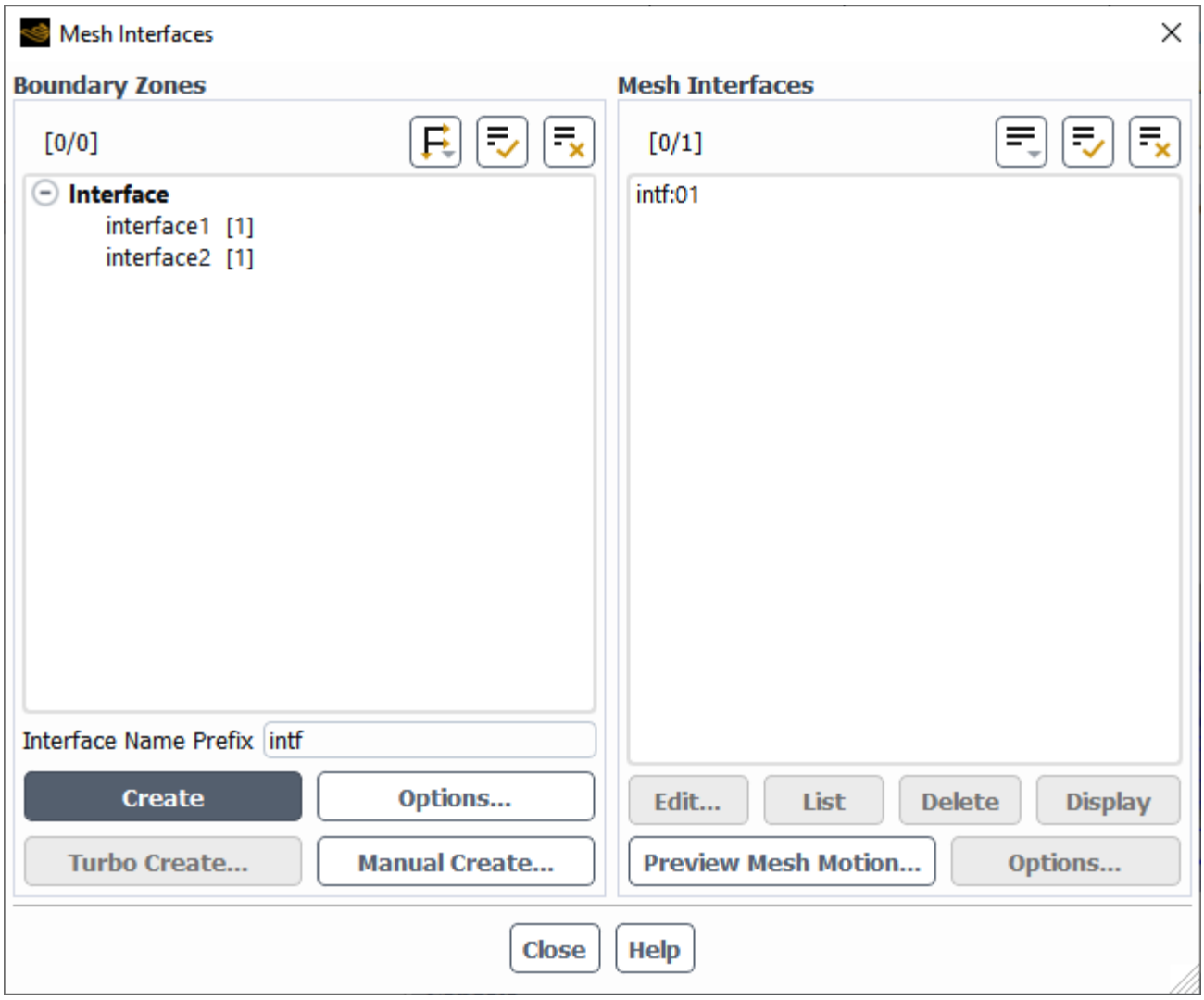


Figure 6.43: The Mesh Interfaces Dialog Box for the Many-to-Many Method



6. Create all of the mesh interfaces that use the periodic or periodic repeats option. To do so, you must disable the one-to-one interface creation method and use the manual method by clicking the **Manual Create...** button to open the Create/Edit Mesh Interfaces Dialog Box (p. 5419). For details, see Manually Creating Many-to-Many Mesh Interfaces (p. 1201).

7. If you want to easily create many mesh interfaces between boundary zones that do not currently overlap (for example, for a sliding mesh simulation), you must ensure that the default one-to-one interface creation method is selected and click the **Manual Create...** button to open the Create Mesh Interfaces (p. 5418). For details, see Manually Creating Many Non-Overlapping Mesh Interfaces (p. 1204).

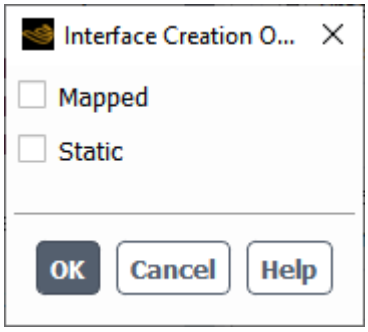
8. Create all of the mesh interfaces that *do not* use the periodic or periodic repeats option, using the automatic method. As part of this method, Fluent will automatically determine which of the selected interface boundary zones can be grouped together to form the two sides of one or more mesh interfaces.

   To use the automatic method, perform the following steps in the **Mesh Interfaces** dialog box:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. (optional) If you have disabled the one-to-one interface creation method, you can click the **Options...** button in the **Boundary Zones** group box to open the Interface Creation Options Dialog Box (p. 5498), and enable the settings that you want applied to all of the resulting interfaces.

Figure 6.44: The Interface Creation Options Dialog Box



- You can specify that the mapped option is applied to all of the resulting interfaces for which at least one side of the interface consists of only solid zones, by enabling the **Mapped** option. Note that the mapped option will be applied regardless of the mesh quality, and the tolerance used for mapping will be that specified in the Mapped Interface Options Dialog Box (p. 5520). For more information about the mapped option, see The Mapped Option (p. 1178).

- You can specify that all of the resulting interfaces use the **Static** option, if and only if you know that the interface zones will not be moving or deforming relative to each other at the interface. While you will have the ability to enable the static option after the interfaces are created, doing so prior to creation avoids the creation of unnecessary zones (and thus saves memory usage and processing time, which can be significant when you have selected many **Boundary Zones** in the **Mesh Interfaces** dialog box).

b. Select the zones in the selection list in the **Boundary Zones** group box that you want paired. You may want to select only some of the zones, such as when you want the resulting mesh interfaces to have different settings in the **Interface Creation Options** dialog box (as described in the previous step), or different name prefixes (as described in the steps that follow); for example, you could select only the interface zones from the front half of a model, and then have all of the resulting mesh interfaces named with the prefix "front". For a case in which the zones begin with no overlap, and will later move into a position that includes overlap (such as a sliding mesh simulation), you must select only two zones at a time if you want to use the **Mesh Interfaces** dialog box; if you want to select more than two zones at a time, you would need to use the **Create Mesh Interfaces** dialog box instead, as described in Manually Creating Many Non-Overlapping Mesh Interfaces (p. 1204).

   Note that each zone in this list has a number in brackets on the right side (for example, **[0]**). This number indicates how many mesh interfaces it is already a part of, as a result of any previous acts of mesh interface creation. Note that a boundary zone can only be used in multiple mesh interfaces when those mesh interfaces are created when the default one-to-one interface creation option is enabled.

c. Enter the **Interface Name Prefix** you want applied to all of the resulting interfaces.

d. Click the **Create** button in the **Mesh Interfaces** dialog box to create the interfaces. The resulting interfaces will be added to the **Mesh Interfaces** list; note that if you select an item in the list,

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

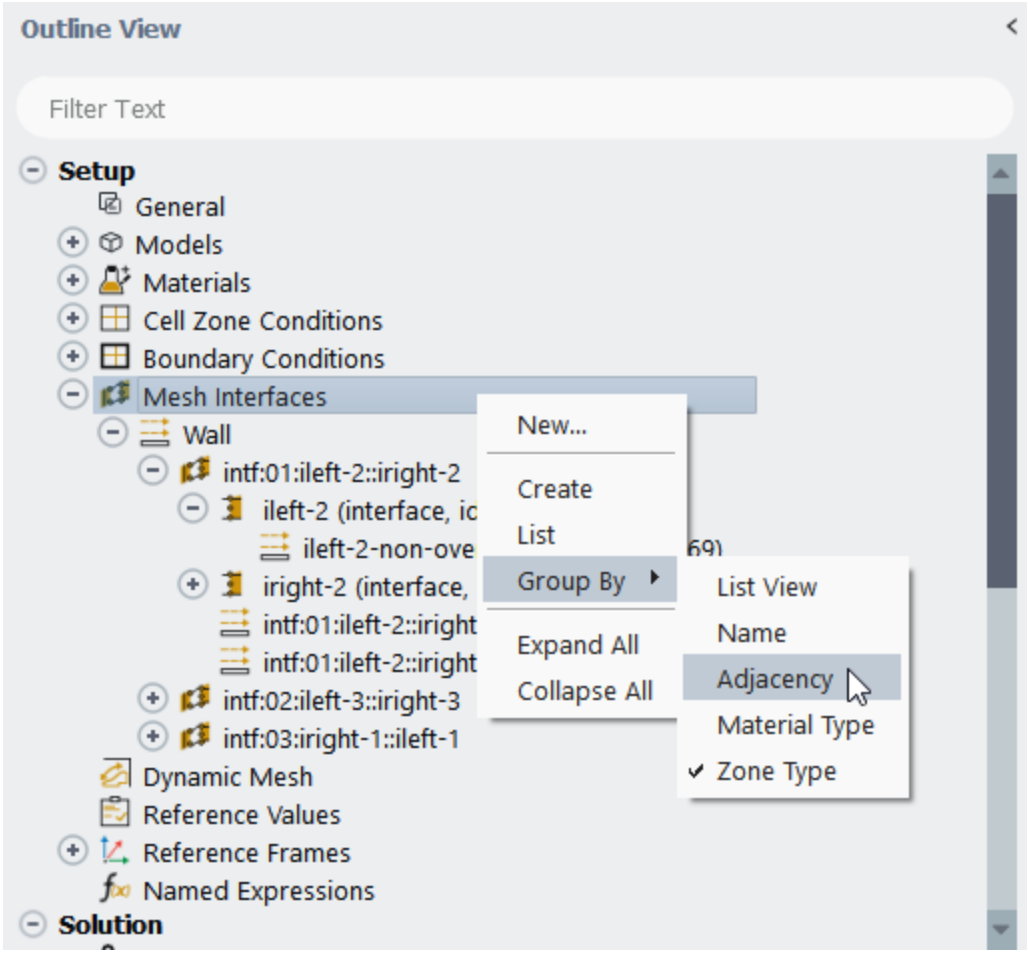
the associated zones will be highlighted in the **Boundary Zones** group box (and vice versa). Details about the creation process will be printed in the console, including (for one-to-one pairings) a list of the resulting mesh interfaces with the area percentage that each boundary zone overlaps the other.

9. After creating mesh interfaces, you can manage them from the outline view tree (under **Setup** / **Mesh Interfaces**) or from the **Mesh Interfaces** dialog box. Note that for cases with only one-to-one mesh interfaces:

   - The individual mesh interfaces listed in the outline view tree have nodes that can be expanded to list all of the associated boundary zones (which have icons to indicate their type). By default, this also includes the associated non-overlapping zones and interior zones, and this is the only place they are listed in the outline view; you can choose to instead have them listed under **Setup** / **Boundary Conditions** under their zone types, by enabling the **Show interface children** option in **Appearance** branch of the **Preferences** dialog box (accessed through **File**/**Preferences...**).

     Using the right-click menu, you can choose to display the mesh in a list view or grouped by the names, the adjacent cell zones, the materials on either side (for example, **fluid-fluid**), and the zone types (that is, whether it includes an internal boundary or a wall because the adjacent cell zones are similar or dissimilar material types, respectively).

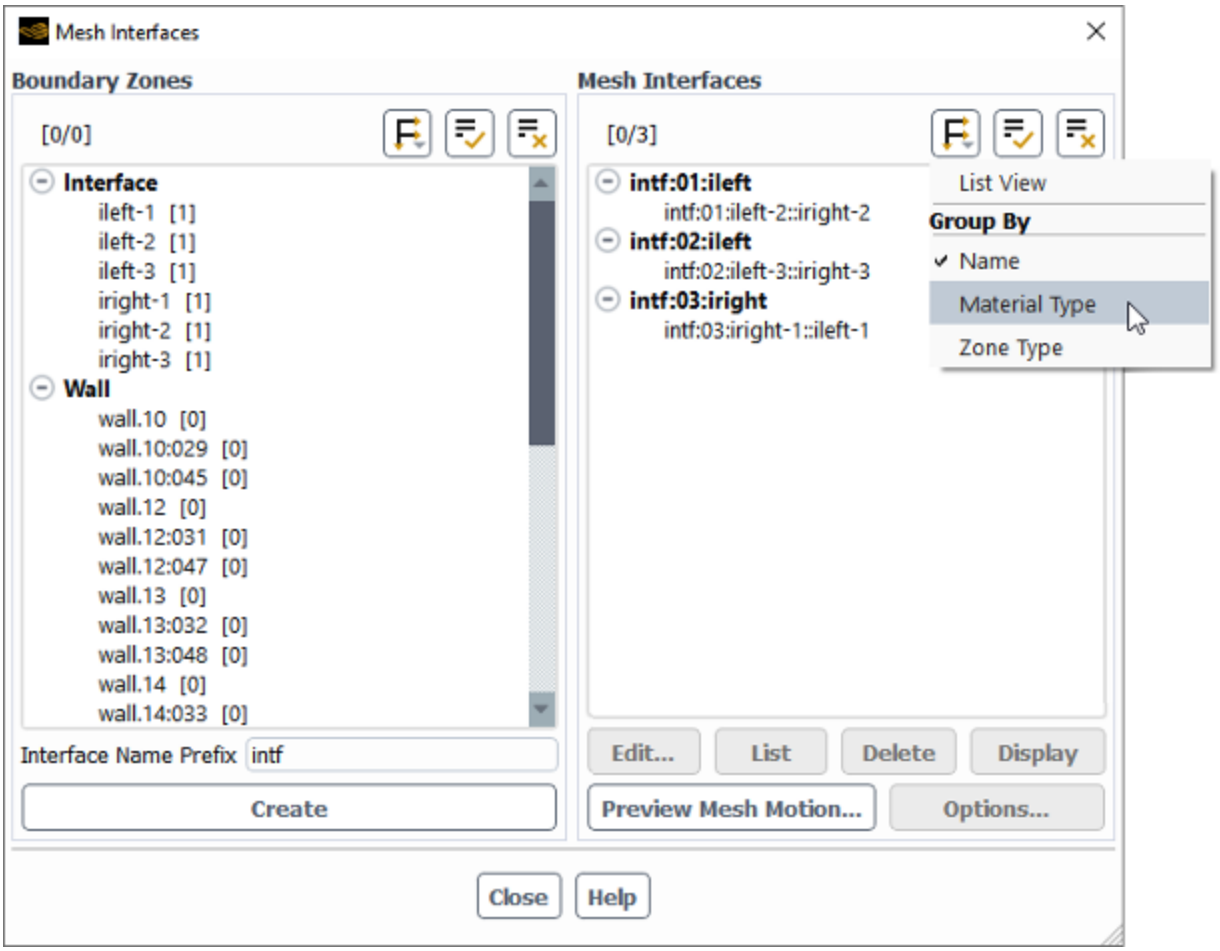
     Figure 6.45: Managing One-to-One Interfaces from the Outline View Tree



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- All of the outline view tree grouping options (except for the adjacent cell zones option) are also available in the **Mesh Interfaces** group box of the **Mesh Interfaces** dialog box, through the **Toggle Tree View** button.

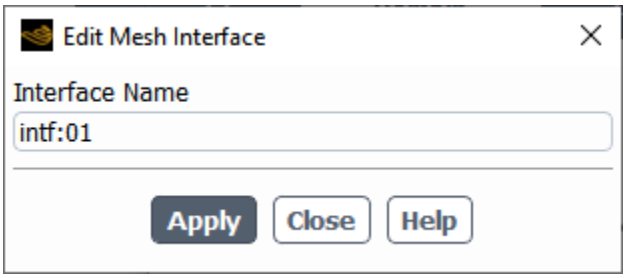
Figure 6.46: Managing One-to-One Interfaces from the Mesh Interfaces Dialog Box



10. For mesh interfaces that were created using the default one-to-one interface creation method, you can edit them in the following ways:

- To edit the name, make a selection from the **Mesh Interfaces** list, click **Edit...**, and then revise the **Interface Name** in the **Edit Mesh Interface** dialog box that opens.

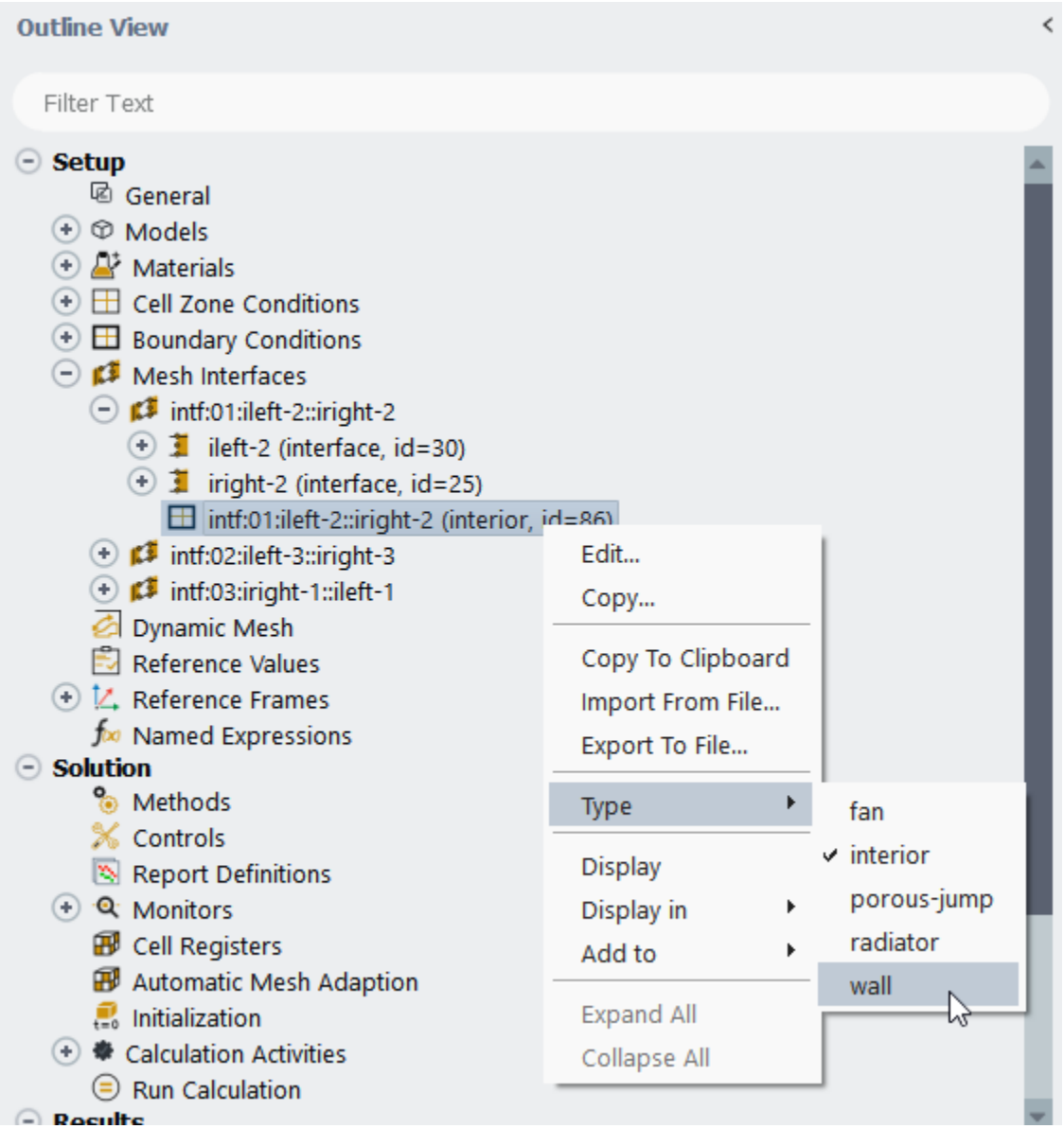
Figure 6.47: The Edit Mesh Interface Dialog Box



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To model a thermally coupled wall at a mesh interface between two fluid zones, right-click the associated interface interior zone in the outline view; by default it is under the associated mesh interface under **Setup** / **Mesh Interfaces**, and has the same name as the mesh interface. Then hover over **Type** and select **wall** from the menu that opens.

Figure 6.48: Creating a Coupled Wall at a One-to-One Mesh Interface



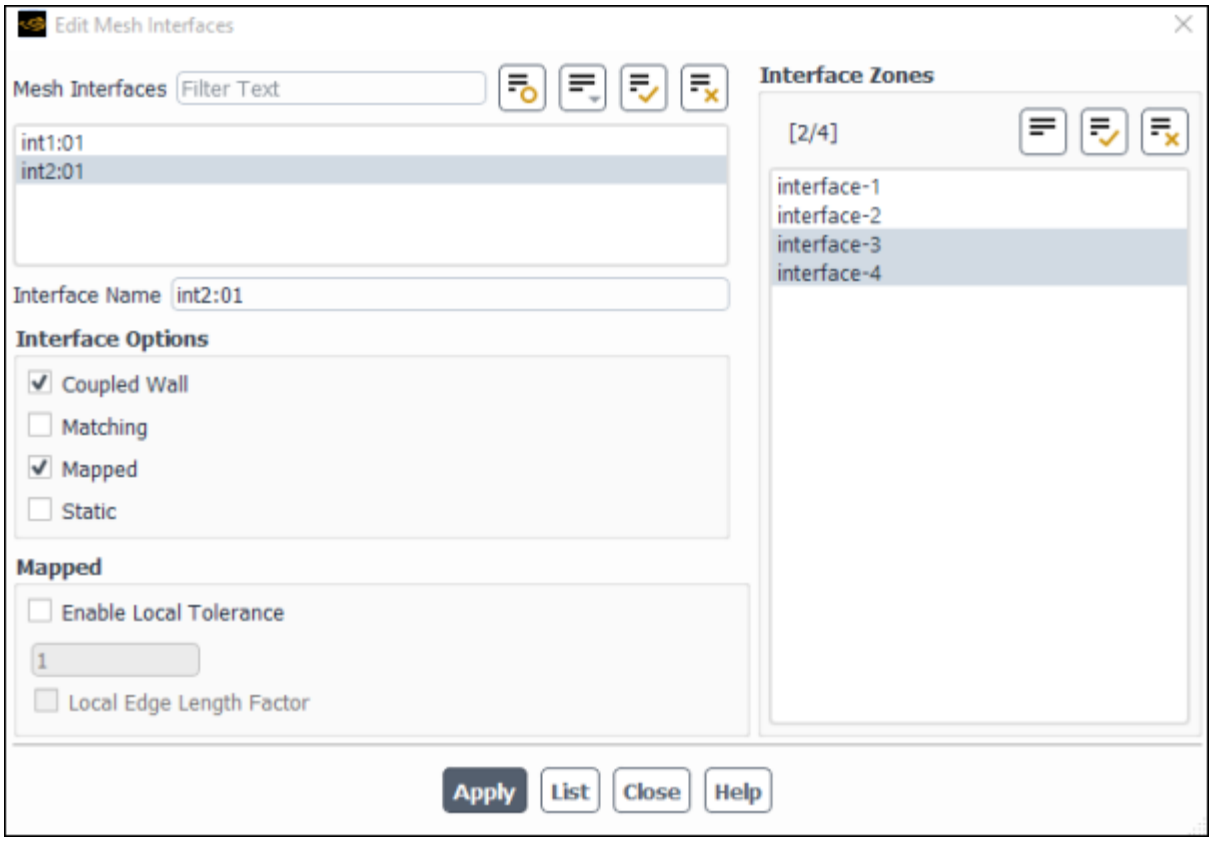
Note:

In the case of a solid-to-solid zone interface of different materials or a fluid-to-solid zone interface, by default it is treated as a coupled wall. Therefore, no action is required in the outline view tree to set up such an interface.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

11. For mesh interfaces that were created with the one-to-one interface creation method disabled, you can revise them by making a selection from the **Mesh Interfaces** list, clicking **Edit...**, and then completing the setup in the Edit Mesh Interfaces Dialog Box (p. 5454) that opens.

Figure 6.49: The Edit Mesh Interfaces Dialog Box



In the **Edit Mesh Interfaces** dialog box, select the **Mesh Interfaces** you want to change, revise the settings, and click **Apply**. You can edit the **Interface Name** and/or the **Interface Zones** that make up the interface (provided that the zones you select are unassigned and located appropriately), though such changes are only allowed if you select a single interface. You can select one or more interfaces and enable the following **Interface Options**; note that you can click the button to group the interfaces in the list by their currently enabled interface option(s), which allows you to select / deselect all those of a particular type by simply clicking the top-level branch.

- Enable **Coupled Wall** if you would like to model a thermally coupled wall between two fluid zones that share a non-conformal interface.

  Note:

  In the case of a solid-to-solid zone interface of different materials or a fluid-to-solid zone interface, by default it is treated as a coupled wall. Therefore, no action is required in the **Edit Mesh Interfaces** dialog box to set up such an interface.

- Enable the **Matching** option if only internal zones should be created (that is, interior or wall / shadow pairs), since the interface zones on both sides are aligned. With the **Matching** option,

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

even interface zones that are not perfectly aligned are treated as if they are; however, if the discrepancy between the interface zones on both sides exceeds default thresholds, then warning messages will be displayed. See Matching Option (p. 1177) for more information about the recommended uses of this option.

- Enable the **Mapped** option when there is penetration or large gaps between interface zones. Note that at least one side of the interface must consist of only solid zones. For additional information, see The Mapped Option (p. 1178).

  Enable **Enable Local Tolerance** in the **Mapped** group box for the selected interface, if you want to override the global tolerance defined for **Mapped Tolerance** in the Mapped Interface Options Dialog Box (p. 5520).

- Enable **Static** if and only if you know that the interface zones will not be moving or deforming relative to each other at the interface. This option will reduce memory usage and processing time during the running of the calculation, especially when there are many zones on both sides of the interface. The only options it is compatible with are the **Coupled Wall** and/or **Matching** options. See The Static Option (p. 1180) for details.

When all your changes are applied, close the **Edit Mesh Interfaces** dialog box.

Note the following:

- The **Periodic Boundary Condition** and **Periodic Repeats** options are not available in the **Edit Mesh Interfaces** dialog box, and so cannot be changed from enabled to disabled (or vice-versa) for an existing interface; to change them, you must delete the interface and create it again with those options revised.

- For interfaces that have the **Periodic Boundary Condition** option enabled, you can edit the **Type** and **Offset** from the **Edit Mesh Interfaces** dialog box. Note that the **Auto Compute Offset** option must be disabled to edit the **Offset**.

12. The buttons at the bottom of the **Mesh Interfaces** group box allow you to take further actions, such as deleting or displaying a selected interface. Note that the **Options...** button opens the Mapped Interface Options Dialog Box (p. 5520), which allows you to update which mesh interfaces use the mapped option and the related settings.

13. Additional wall and interior zones will be created (and deleted) along with the mesh interfaces. These zones will be named as described in Interface Zones Automatic Naming Conventions (p. 1180), and are visible in the outline view tree. There are times when you may need to take action on these additional zones, including the following:

    - If the two interface zones did not overlap entirely, check the boundary zones created for the non-overlapping portions to ensure that they are the proper boundary type and/or defined appropriately. The non-overlapping zones are listed in the outline view tree (as children of the associated interface boundary zones), as well as in the relevant **Interface** dialog boxes. If the zone type is not correct, you can use the Boundary Conditions Task Page (p. 4991) to change it; if the zone is not defined appropriately, double-click it in the outline view tree or click the **Edit...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

button in the **Interface** dialog box, and make the necessary changes in the dialog box that opens.

Note:

If you delete a mesh interface that was created using the default one-to-one interface method, the type and settings of the non-overlapping zones (including any changes that you made) will be applied to the associated parent boundary zones after the deletion.

- If you have any coupled wall interfaces, define boundary conditions (if relevant) by updating the interface wall zones using the **Boundary Conditions** task page.

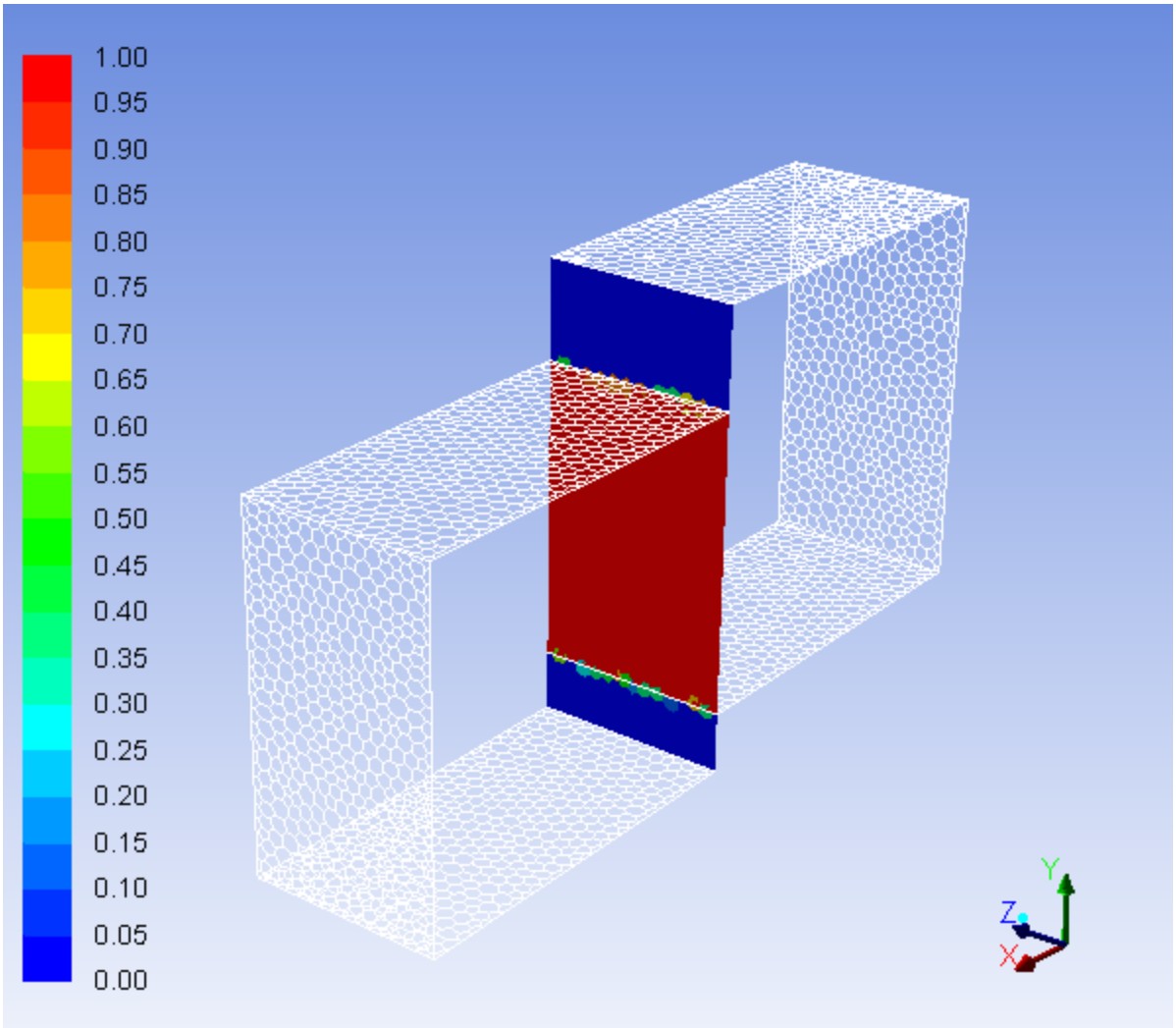
  Setup → **Boundary Conditions**

14. It is always a good practice to initialize the solution and review your interfaces:

    - You can visualize the mesh interface(s) you have created by displaying contours of **Mesh** and **Interface Overlap Fraction** in the Contours Dialog Box (p. 5412). See Figure 6.50: Contours of Interface Overlap Fraction (p. 1198) for an example.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

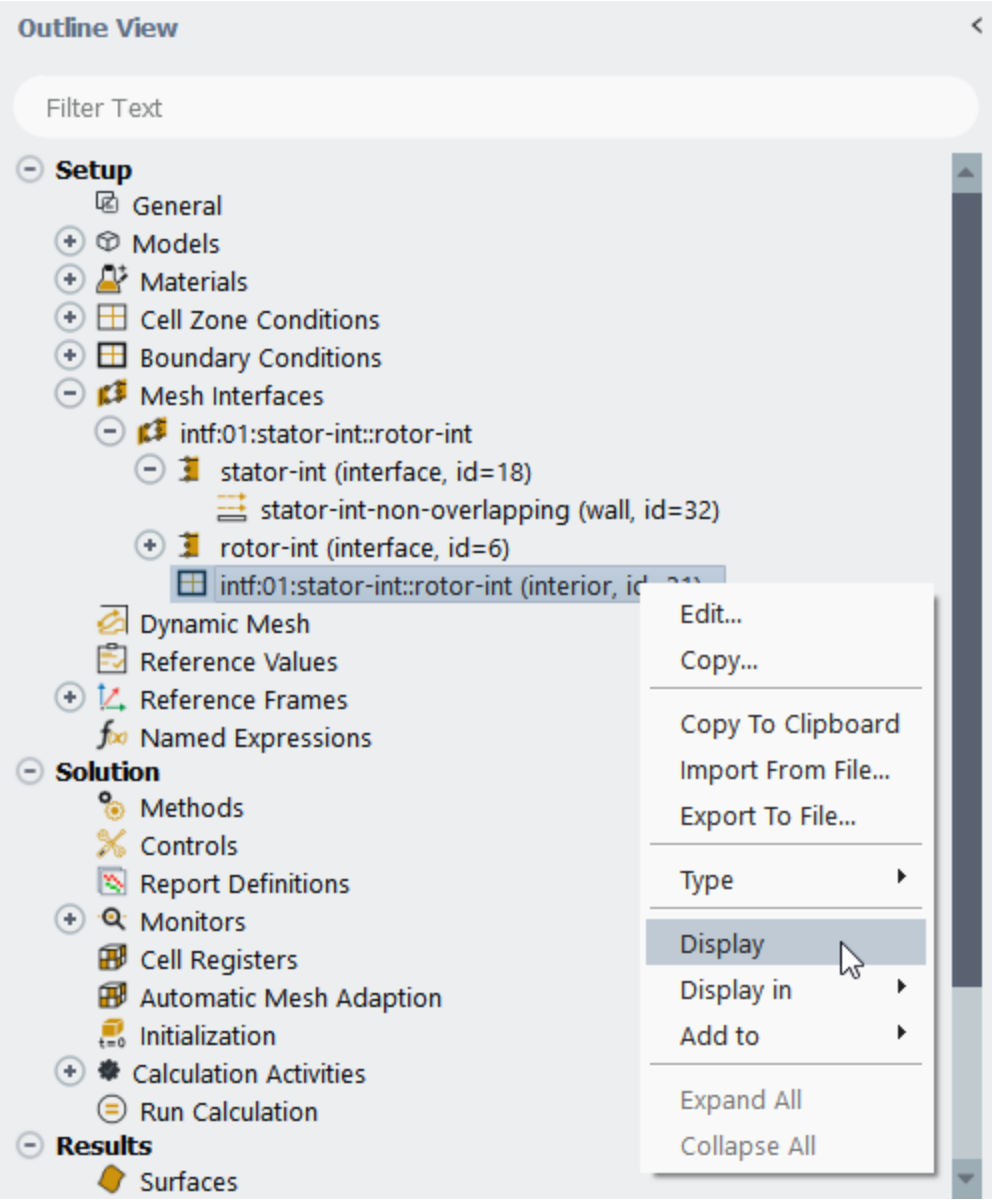
Figure 6.50: Contours of Interface Overlap Fraction



For a one-to-one mesh interface, you also have the option of displaying the intersected zone in the graphics window by using the right-click menu in the outline view tree (under the mesh interface node under **Setup** / **Mesh Interfaces**, by default). The intersected zone is of type interior or is a coupled wall (wall or wall shadow) that is generated during the mesh interface creation, and it represents the overlapping area between the two interface zones. In the same way you can also display the non-overlapping zones; note that for non-overlapping zones, the display is approximate at the boundaries of intersected zones, as whole faces will be shown or not shown depending on the location of their centroid.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.51: Displaying the Intersected Zone



- You can print information in the console about the areas of the zones associated with interfaces either by using the **List** button in the **Mesh Interfaces** dialog box or by entering the text command `define/mesh-interface/list`. The following is the information for a one-to-one mesh interface:

```
>define/mesh-interfaces/list

List of Mesh Interface Boundary Zones


  Interface Boundary:  s-1
  Area:   1.01046027e-03

                     Name                       ID         Area         Area Percentage
  ----------------------------------------  ----  -----------------  ---------------
                           intf:01:s-1::r-1    14     8.99861833e-04         89.0546%
                      s-1-non-overlapping    29     1.10513145e-04         10.9369%
```

```
  Interface Boundary:  r-1
  Area:   1.01029151e-03

                     Name                       ID         Area         Area Percentage
   ----------------------------------------  ----  -----------------  ---------------
                           intf:01:s-1::r-1    14     8.99861833e-04         89.0695%
                        r-1-non-overlapping    30     1.10303113e-04         10.9179%



List of Mesh Interface Area Percentage

                    Name                       ID          Area                Area Percentage
   ------------------------------------  --------  -----------------  ------------------------
                          intf:01:s-1::r-1        14    8.99861833e-04          89.05% / 89.07%
```

The `Area Percentage` is the `Area` of a mesh interface / non-overlapping zone divided by the area of an interface boundary. The final line lists the `Area Percentage` for both of the interface boundaries, that is, `s-1` and `r-1`, respectively.

The following is the information for a many-to-many mesh interface:

```
>define/mesh-interfaces/list

List of Mesh Interfaces


  Interface Name:     intf:01
  Interface Options:  none


       Description                    Name               ID         Area         Area Percentage
---------------------------  -----------------------  -------  -------------  ----------------
      Interface-Zone-Side-1                      s-1       18  1.01046027e-03
      Interface-Zone-Side-2                      r-1        6  1.01029151e-03
Non-Overlapping-Zone-Side-1      s-1-non-overlapping       32  1.10513145e-04          10.9369%
Non-Overlapping-Zone-Side-2      r-1-non-overlapping       33  1.10303113e-04          10.9179%
    Interface-Interior-Zone     intf:01-interior-1-1       31  8.99861833e-04          89.0546%
```

The `Area Percentage` for `intf:01-interior-1-1` is calculated as the area of `intf:01-interior-1-1` divided by the area of `s-1`.

- If you created one-to-one mesh interfaces, you can use the following text command to delete mesh interfaces that have a small area of overlap. You define the area percentage that is used for deletion: if the area that the two zones overlap divided by the area of the zone is less than this value for both zones in any mesh interface, then that mesh interface will be deleted. This can help you easily delete unnecessary or unintentional mesh interfaces.

  `define` → `mesh-interfaces` → `delete-interfaces-with-small-overlap`

15. For dynamic meshes, by default any motion is automatically transferred across a mesh interface when only one side is moving as a result of user-defined or system coupling motion, to ensure that the coupling is maintained on both sides. To change the method by which the displacement of the passive nodes is calculated or to disable this transfer altogether, use the `define/mesh-interfaces/transfer-motion-across-interfaces?` text command.

    With regard to the method by which the motion is transferred, you have two options: `transfer-displacements` (the default) interpolates nodal displacement from the active side of the interface to the passive side, and is recommended when there are gaps and/or penetrations in the mesh

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

interface that must be maintained; `project-nodes` projects the passive nodes onto the faces of active side, and is recommended when the active side undergoes significant tangential motion (as only the normal displacement is effectively transferred to the passive side in this method). For further details, see Transferring Motion Across a Mesh Interface (p. 1205).

Note:

For case files in which the interface was created in Ansys Fluent version 17.2 or earlier, you may reduce quality issues (such as left-handed faces) and/or convergence problems if you recreate the interface in version 18.0 or later by entering the following Scheme command in the console: (`recreate-sliding-interfaces`).

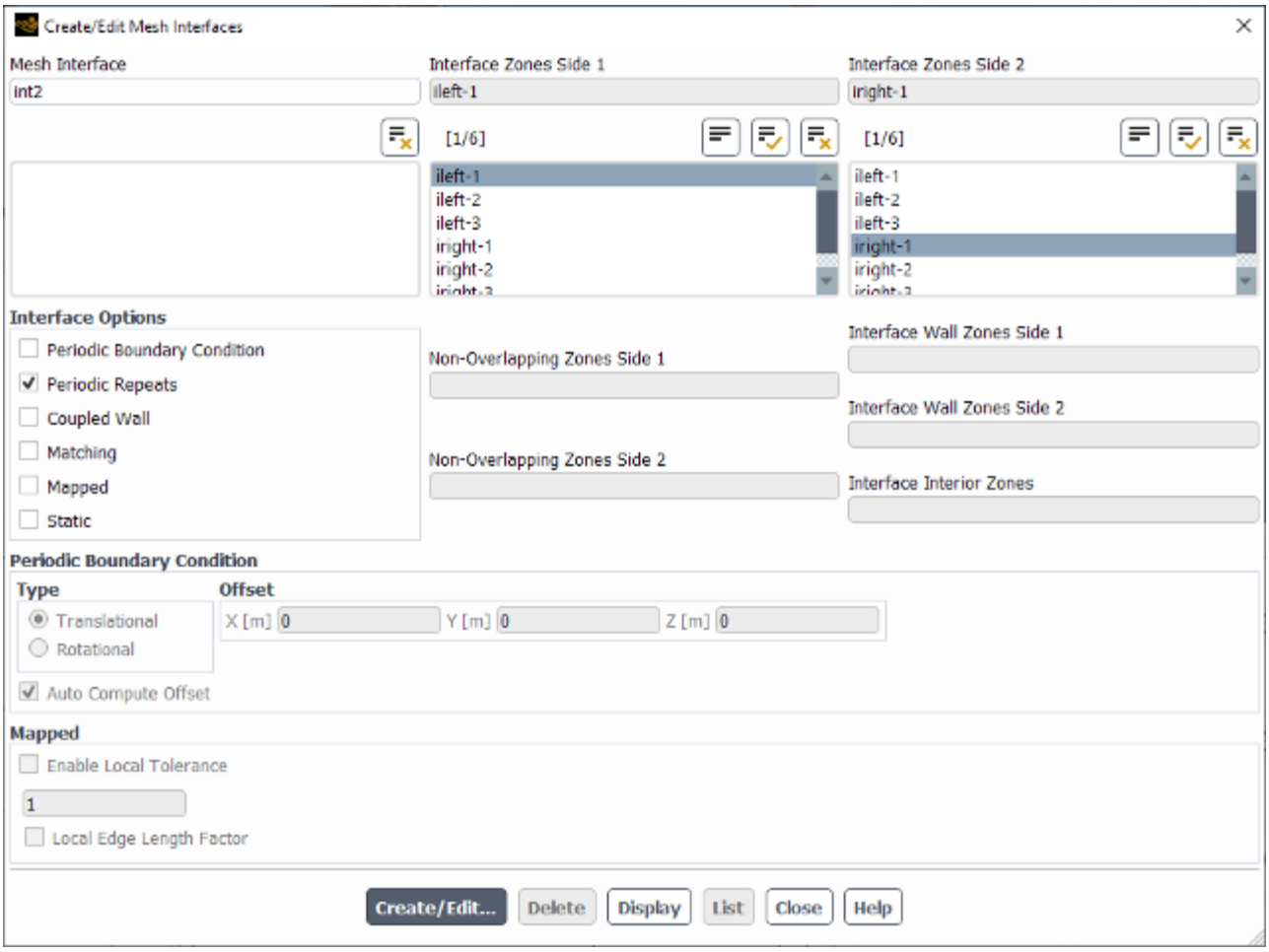
6.6.4.1. Manually Creating Many-to-Many Mesh Interfaces

While it is possible to create every type of many-to-many mesh interface using the manual method, it is only necessary when you want your interface to use the periodic repeats option. If you want your interface to use the periodic option, you can create it manually or by using the **Create Periodic** dialog box (as described in Creating Periodic Zones and Interfaces (p. 1269)). For all other types it is more convenient to use the automatic method (as described in the preceding section), especially when you have many interface zones and/or are unfamiliar with their names / locations.

To manually create a many-to-many mesh interface, you must ensure that the one-to-one interface creation method is disabled and then use the Create/Edit Mesh Interfaces Dialog Box (p. 5419), which is opened by clicking the **Manual Create...** button in the Mesh Interfaces Dialog Box (p. 5525).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.52: The Create/Edit Mesh Interfaces Dialog Box



1. Enter a name for the interface in the **Mesh Interface** text-entry box.

2. Specify the interface zones that make up the mesh interface by selecting one or more zones in the **Interface Zones Side 1** list and one or more zones in the **Interface Zones Side 2** list. Note that you must select only one zone per side if you plan to use the periodic or periodic repeats option.

3. Enable the desired **Interface Options**:

   - Enable **Periodic Boundary Condition** to create a non-conformal periodic boundary condition interface.

     – Select either **Translational** or **Rotational** as the periodic boundary condition **Type** to define the type of periodicity.

     – Retain the enabled default setting of **Auto Compute Offset** if you want Ansys Fluent to automatically compute the offset. After creating the interface, the offsets will be displayed in these fields. The fields will be uneditable when the **Auto Compute Offset** is enabled.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– Disable **Auto Compute Offset** if you decide that you do not want Ansys Fluent to find the offset. In this case, you will have to provide the offset coordinates or angle in the required fields, depending on whether **Translational** or **Rotational** periodicity is selected.

Important:

→ Auto computation means that the rotational angle or the translational offset will be automatically calculated and used while creating a non-conformal periodic boundary condition interface. However, it still relies on the **Rotational Axis Origin** and the **Rotational Axis Direction** that was entered for the cell zone in the cell zone condition dialog box (for example, the **Fluid** dialog box). Therefore, before proceeding with the creation of the non-conformal periodic boundary condition interface, you have to correctly enter the rotational axis for the corresponding cell zone.

Note that auto computation of the non-conformal periodic boundary condition offset does not mean that the **Rotational Axis Origin** and **Rotational Axis Direction** are also detected automatically and updated. It is still your responsibility to set up these values correctly.

→ When using auto computation for 3D cases, you should ensure that the mesh resolution on the periodic interface zone pairs are similar if both of the following conditions are met: the periodic interface zone pairs are not planar; and they are not connected by at least two face zones that meet at an edge that points from one periodic zone to the other. In such situations, having a similar mesh resolution will produce more precise offset calculations (and subsequently, more accurate solutions). It is recommended that the average mesh size on one boundary should not be more than twice that of the other.

- Enable **Periodic Repeats** when each of the two cell zones has a single pair of conformal or non-conformal periodics adjacent to the interface (see Figure 6.35: Translational Non-Conformal Interface with the Periodic Repeats Option (p. 1175)). This option is typically used in conjunction with the sliding mesh model, when simulating the interface between a rotor and stator.

- Other options are available, as described in the preceding section. Note that with **Periodic Boundary Condition** enabled, the **Matching** option is also enabled by default and no other options are compatible; with **Periodic Repeats** enabled, only the **Coupled Wall** option is compatible.

4. Click **Create/Edit...** to create a new mesh interface. Ansys Fluent will automatically create additional wall and/or interior zones for the interface, which will appear under **Non-Overlapping Zones Side 1**, **Non-Overlapping Zones Side 2**, **Interface Wall Zones Side 1**, **Interface Wall Zones Side 2**, and **Interface Interior Zones**.

5. The buttons at the bottom of the dialog box allow you to take further actions, such as the following:

   - If you create an incorrect mesh interface, you can select it in the selection list and click the **Delete** button to delete it. Any additional wall and/or interior zones that were created when the interface was created will also be deleted.

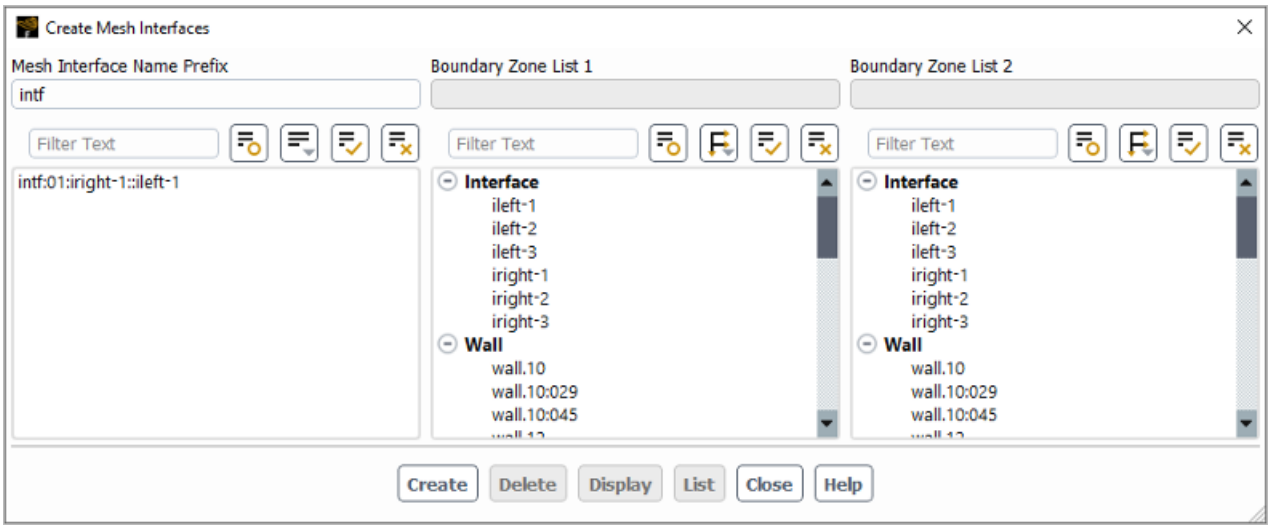
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To edit an interface, select it in the selection list and click **Create/Edit...**, and then revise the settings using the Edit Mesh Interfaces Dialog Box (p. 5454) that opens.
- You can click the **Display** button to display interface zones or mesh interfaces in the graphics window. Note that you can only select and display interface zones from **Interface Zones Side 1** or **Interface Zones Side 2** if no interfaces are selected in the selection list and if the zone in question is not yet assigned to an interface. After an interface is defined, you can select the appropriate mesh interface and click the **Display** button to display the zones under **Interface Zones Side 1** and **Interface Zones Side 2** together. Displaying is particularly useful if you want to check the location of the interface zones prior to setting up a mesh interface.

6.6.4.2. Manually Creating Many Non-Overlapping Mesh Interfaces

If you want to easily create many mesh interfaces between boundary zones that do not currently overlap (for example, for a sliding mesh simulation), you can use the **Create Mesh Interfaces** dialog box. To open this dialog box, you must ensure that the default one-to-one interface creation method is enabled and then click the **Manual Create...** button in the Mesh Interfaces Dialog Box (p. 5525).

Figure 6.53: The Create Mesh Interfaces Dialog Box



1. Enter the **Mesh Interface Name Prefix** you want applied to all of the resulting interfaces.
2. Select the zones that you want to pair up to make mesh interfaces by selecting one or more zones in the list under **Boundary Zone List 1** and one or more zones in the list under **Boundary Zone List 2**. Note that mesh interfaces will be created for every possible combination between these two lists, regardless of whether the zones in question currently overlap.
3. Click the **Create** button in the **Create Mesh Interfaces** dialog box to create the mesh interfaces. The resulting mesh interfaces will be displayed in the list under the **Mesh Interface Name Prefix** field; note that if you select an item in the list, the associated zones will be highlighted in the lists under **Boundary Zone List 1** and **Boundary Zone List 2**. Details about the creation process will be printed in the console, including a list of the resulting mesh interfaces with the area percentage that each boundary zone overlaps the other.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. The buttons at the bottom of the **Create Mesh Interfaces** dialog box allow you to take further actions, such as the following:

   - If you create an incorrect mesh interface, you can select it in the list under the **Mesh Interface Name Prefix** field and click the **Delete** button to delete it. Any additional wall and/or interior zones that were created when the interface was created will also be deleted.

   - You can click the **Display** button to display any of the selected boundary zones and/or mesh interfaces in the graphics window. Note that after you have selected a mesh interface, you can modify what zones are displayed by selecting / deselecting zones under **Boundary Zone List 1** and/or **Boundary Zone List 2**.

   - You can click the **List** button to print information in the console about the areas of the zones associated with selected mesh interfaces.

6.6.4.3. Transferring Motion Across a Mesh Interface

Ansys Fluent allows motion to be specified on either side of a mesh interface, and in some instances, can transfer the motion from one side of a mesh interface to the other. When motion is specified on one side of the interface (but not on the other) using either User-Defined Motion (p. 1913) or System Coupling Motion (p. 1916), by default Ansys Fluent will automatically interpolate and transfer this motion so that the coupling is maintained on both sides. To disable this transfer or change the method by which the displacement of the nodes on the passive side is calculated, use the following text command:

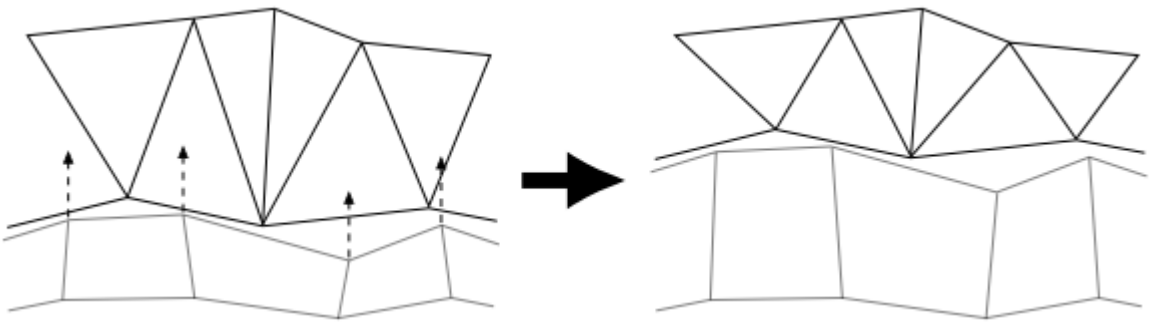
`define` → `mesh-interfaces` → `transfer-motion-across-interfaces?`

When such transfer is disabled, the sides of the mesh interface may separate and/or penetrate over the course of the simulation, which may lead to convergence difficulties and/or an invalid mesh. When it is enabled, you have the following options:

- the `transfer-displacements` approach (default)

  This approach interpolates nodal displacement from the active side of the interface to the passive side; it uses an inverse-distance weighted interpolation, so that the closer active nodes have more influence. This may be beneficial in situations where gaps and/or penetrations are present at the interface, and they need to be maintained as the simulation progresses. Note that any motion specified tangential to the interface will also be interpolated to the passive side, which can lead to unintended mesh dragging.

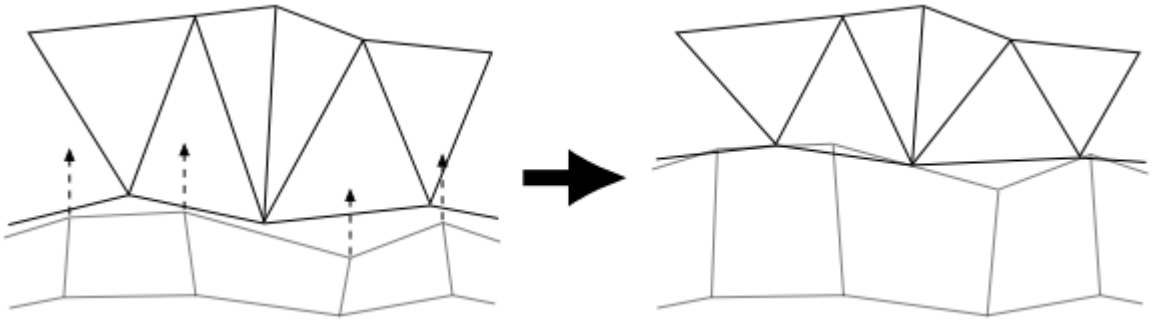
  Figure 6.54: Transferring Displacements



- the `project-nodes` approach

  This approach projects the nodes of the passive side of the interface onto the faces of the active side, in order to maintain conformity. It is beneficial in situations that involve significant tangential motions on the active side of the interface, as only the normal displacement is effectively transferred to the passive side. Note that for mesh interfaces that involve significant gaps or penetration, the `project-nodes` approach may cause the mesh to become invalid.

  Figure 6.55: Projecting Nodes



Note:

If you expect the motion to produce large displacements, you should consider applying a mesh update method on the cell zone of the passive side (such as smoothing or remeshing, as described in Dynamic Mesh Update Methods (p. 1816)).

6.7. Overset Meshes

Fluent allows you to build up the computational domain from overlapping meshes—also known as Chimera or overset methodology. This gives you another way to build the complete mesh, in addition to using conformally connected cell zones and non-conformal interfaces.

6.7.1. Introduction

6.7.2. Overset Topologies

6.7.3. Overset Domain Connectivity

6.7.4. Diagnosing Overset Interface Issues

6.7.5. Overset Mesh Adaption

6.7.6. Overset Meshing Best Practices

6.7.7. Overset Meshing Limitations and Compatibilities

6.7.8. Setting up an Overset Interface

6.7.9. Postprocessing Overset Meshes

6.7.10. Writing and Reading Overset Files

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.7.1. Introduction

Whereas non-conformal interfaces connect cell zones along matching face zones, overset interfaces connect cell zones by interpolating cell data in the overlapping regions. For overset meshing to be successful, the cell zones must overlap sufficiently. An advantage of overset meshing is that the individual parts of an overset mesh can be generated independently and with fewer constraints than if the parts had to fit together conformally or along non-conformal interfaces. This can make it easier to replace parts of a mesh without having to remesh large parts or even the complete mesh.

Figure 6.56: Overset Component and Background Mesh

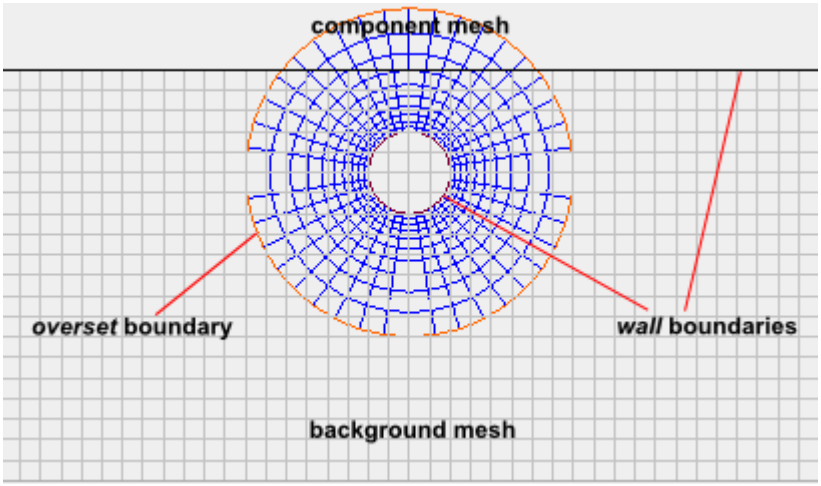
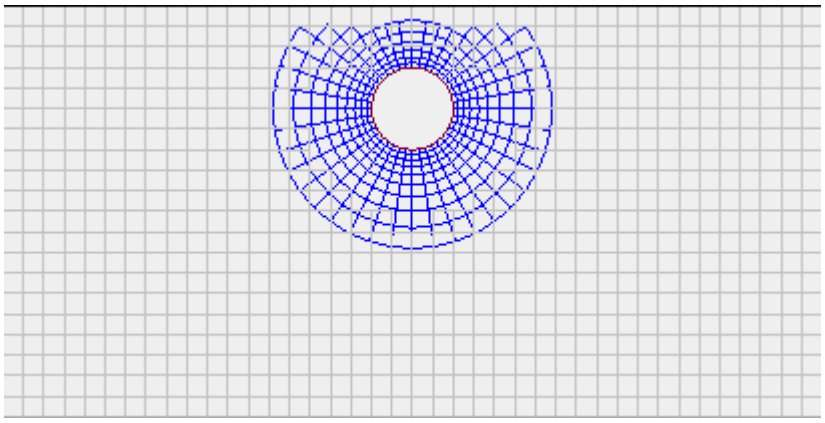


Figure 6.56: Overset Component and Background Mesh (p. 1207) shows a simplified mesh for a simulation of flow over a cylinder in a duct. The mesh consists of two parts—a *background* mesh representing the duct and a separate *component* mesh around the cylinder. The case is set up in Fluent as an overset case by specifying the outer boundary of the cylinder mesh as **overset** (boundary type), and by creating an overset interface containing the two cell zones. With these two steps complete, Fluent automatically establishes the necessary connectivity between the meshes when the flow is initialized. In this process, cells that fall outside the computational domain are classified as *dead* cells. The cells where the flow equations are solved are referred to as *solve* cells. *Receptor* cells receive data interpolated from another mesh. The *donor* cells—the cells where the receptors get their data—are a subset of the solve cells. For the above case, Figure 6.57: Solve Cells After Initialization (p. 1208) shows the solve cells of the mesh after the flow is initialized. Note that in parallel, you can print partition statistics to get information about some of the cell classifications, as described in Interpreting Partition Statistics (p. 4339).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.57: Solve Cells After Initialization



There is no limit on the number of cell zones that can be paired in an overset interface. Fluent classifies the cell zones in an overset interface as either *background* or *component*:

- Background zones are the cell zones that make up the background mesh of the computational domain. Background zones cannot have **overset** boundaries.
- Component zones overlay background zones and have **overset** boundaries near where they connect to the background and other component zones. If multiple component zones are connected, at least one of these zones needs to have an overset boundary.

All cell types generally supported in Fluent are supported with overset meshing, including polyhedral cells in 3D. The different zones paired in an overset interface can have different element types or can have mixed element types.

Overset meshing in Fluent is compatible with mesh adaption. The mesh zones of an overset interface can be locally adapted using all available adaption tools.

Overset meshing can be used for steady-state or transient solutions. Note that for steady-state cases in parallel that use the default **Metis** partitioning, Ansys Fluent will automatically repartition overset meshes when the solution is initialized or data is read; a model-weighted partitioning that is designed for optimal performance is used.

6.7.2. Overset Topologies

There are a multitude of configurations that can be constructed by superposing background and component meshes in an overset interface. Fluent analyzes the topology of an interface with minimal user input, which requires that the superposed meshes adhere to some topological constraints.

A case can have multiple overset interfaces, but a cell zone can only belong to a single interface. That is, interfaces cannot share component or background meshes. It is always possible to have a single overset interface for an entire model. However, it is more efficient to declare multiple interfaces if the zonal topology allows it. Only including cell zones that are absolutely required in an interface also increases efficiency.

In general, most overset interfaces will have at least one background and one component mesh. However, Ansys Fluent also allows you to set up overset interfaces that have no background mesh and consist only of two or more component meshes.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

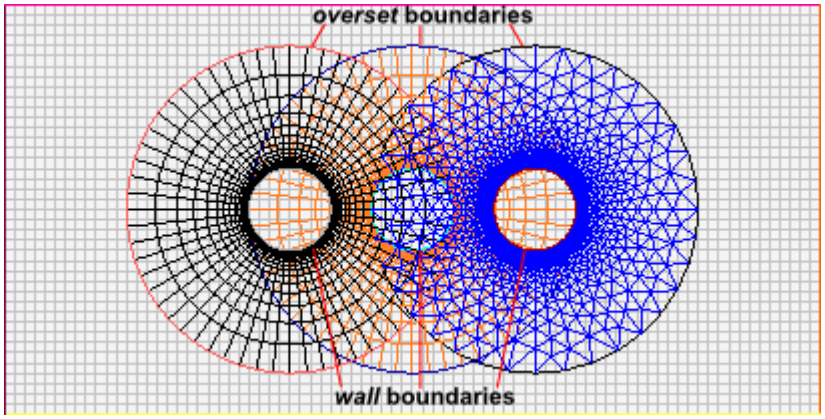
An overset interface can contain multiple component and background meshes, provided that the background meshes do not intersect each other.

Currently, non-conformal interfaces are not fully compatible with overset meshing. Non-conformal mesh interfaces are permissible inside overset cell zones if the mesh interface boundaries do not spatially overlap with where the active overset interface lies. The proximity between the mesh interfaces and overset interfaces is checked during initialization and when the overset interfaces are updated. A warning is issued and the mesh check will fail if not enough separation is detected between the interfaces.

The component meshes of an overset interface are allowed to overlap arbitrarily and multiple component meshes can be combined to create an object of interest. However, an important topological constraint is that physical boundary zones (wall, inlet, outlet, symmetry, and so on) are not allowed to *intersect* each other. That is, a wall boundary cannot cross another wall boundary. For example, in the model shown in Figure 6.56: Overset Component and Background Mesh (p. 1207), the component mesh cannot be placed such that its wall boundary zone intersects with the wall of the background mesh.

Overset boundaries can intersect with other overset boundaries and with physical boundaries. In Figure 6.56: Overset Component and Background Mesh (p. 1207) the overset boundary of the component mesh extends outside of the intended computational domain, crossing the wall boundary of the background mesh—this is allowed. Figure 6.58: Valid Overset Meshes with Components in Close Proximity (p. 1209) shows a valid configuration of three cylinders in proximity (none of the physical boundaries intersect with other physical boundaries).

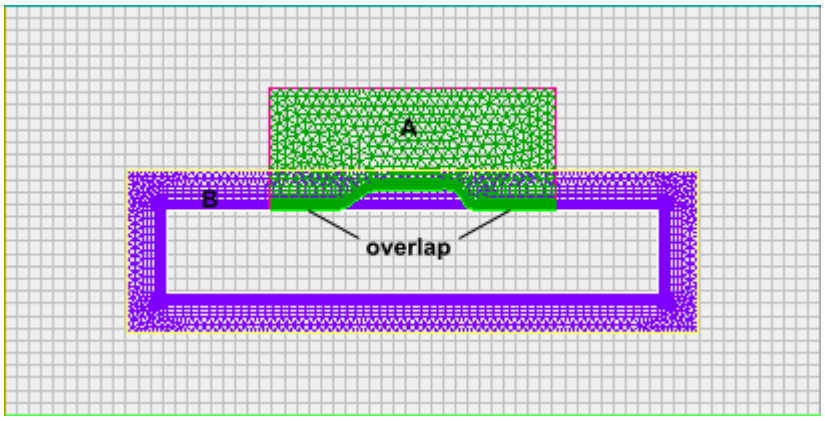
Figure 6.58: Valid Overset Meshes with Components in Close Proximity



Physical boundary zones are allowed to *overlap*, that is, a wall boundary (or parts of a wall boundary) can be coincident with another wall boundary, as long as they do not cross. This powerful functionality lets you build a body from overlapping parts or modify a body by adding parts. Figure 6.59: Second Component Modifying Existing Body (p. 1210) shows an overset configuration where a second component mesh is added to an existing body to modify its shape. Fluent analyzes and automatically detects the overlapping boundaries (walls in this case) of the different meshes in an overset interface, when deciding which cells are solve cells and which cells are dead cells.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

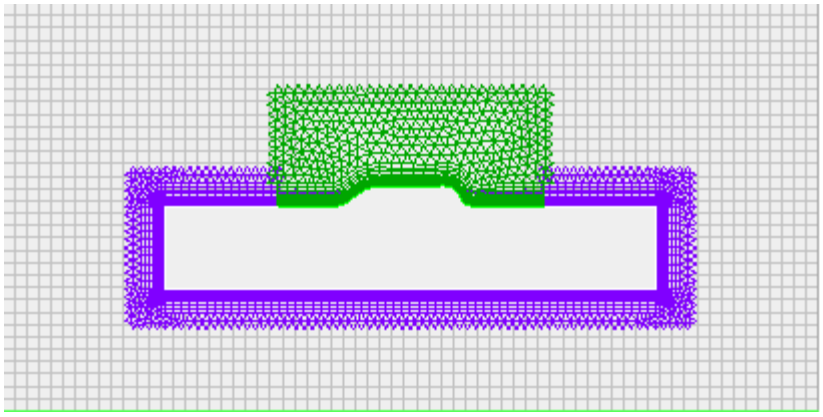
Figure 6.59: Second Component Modifying Existing Body



*Note that part of the wall boundary of mesh **A** lays on top of part of the wall boundary of mesh **B**, overlapping it, but the walls do not cross or intersect.*

Figure 6.60: Existing Body Modification After Initialization (p. 1210) shows the solve cells of the mesh after the flow field is initialized.

Figure 6.60: Existing Body Modification After Initialization



The previous example shows a case where physical boundaries overlap between two component meshes. Overlapping boundaries can also exist between component and background meshes.

If a body can only be built from components that have intersecting boundaries, then you must use a *collar* mesh to properly connect them. A collar mesh is an additional component mesh that bridges the intersecting physical boundaries, essentially transforming the intersection problem (not allowed) into a problem of overlapping boundaries (allowed). Figure 6.61: Multiple Components Bridged by Collars Meshes (p. 1211) shows an overset interface consisting of two component meshes that have intersecting wall boundaries. Two collar meshes (labeled **C** and **D**) are manually created to correctly define the connectivity where the wall boundaries of the original component meshes (labeled **A** and **B**) intersect. The wall boundaries of meshes **C** and **D** overlap with the wall boundaries of meshes **A** and **B**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.61: Multiple Components Bridged by Collars Meshes

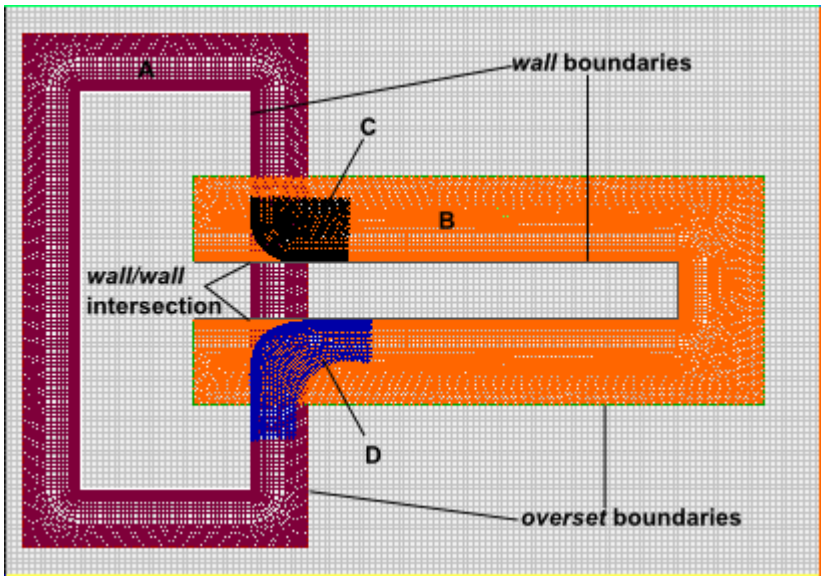
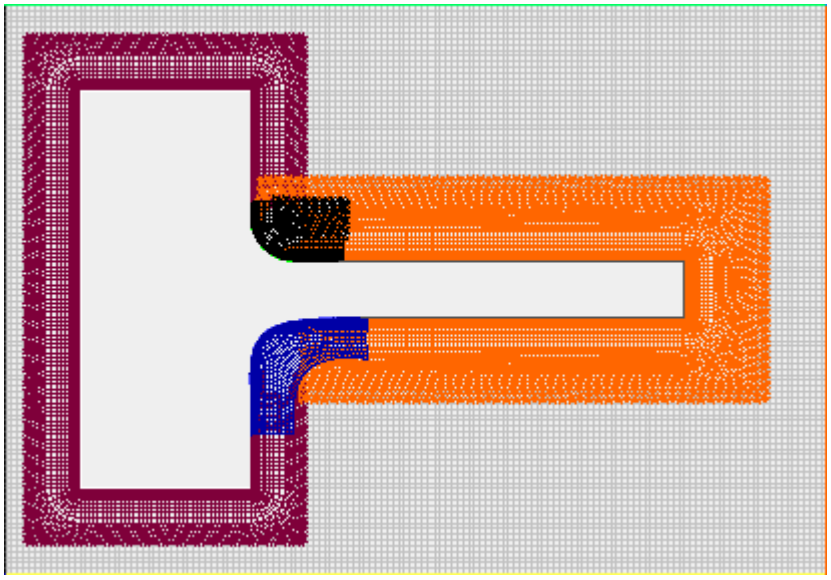


Figure 6.62: Multiple Components with Collar Meshes Initialized (p. 1211) shows the resulting solve cells after the flow is initialized. The collar meshes act as fillet meshes that place the actual boundary intersections into the interior of the body and outside of the fluid domain.

Figure 6.62: Multiple Components with Collar Meshes Initialized



Special considerations are required when you intend to add a fluid region that expands the bounds of an existing region, as shown in Figure 6.63: Adding Fluid to a Region Using Cut Control (p. 1212). In this example, a duct (**A**) is modified by adding a cavity (**B**). The boundaries of the two meshes intersect, and by default the lower boundary of the duct would cut off the added cavity. In order to avoid this, you need to use cut control to specify that the duct wall is not cutting the cavity cell zone, and, similarly, that the cavity boundary is not cutting into the duct cell zone. See Hole Cutting Control (p. 1213) for details on the available cut controls. The resulting mesh for the duct / cavity example after initialization is shown in Figure 6.64: Cut Controlled Region After Initialization (p. 1212).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.63: Adding Fluid to a Region Using Cut Control

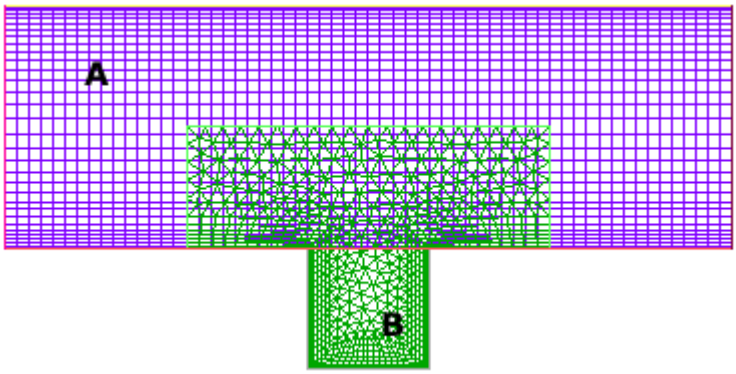
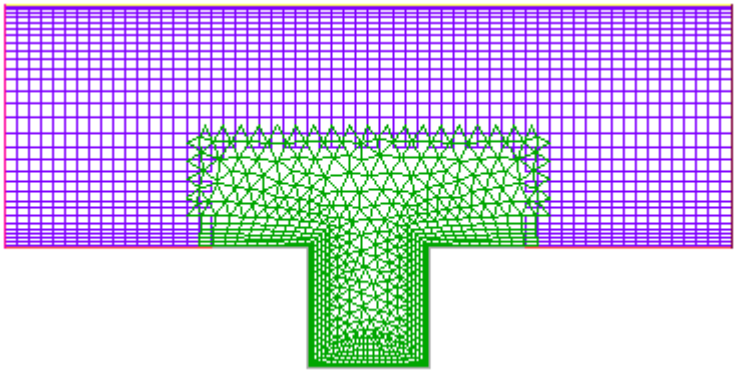


Figure 6.64: Cut Controlled Region After Initialization



Baffles (coupled walls) are permissible in overset meshing—both in background and component meshes.

6.7.3. Overset Domain Connectivity

When an overset interface is initialized, there are three main steps that Fluent completes to establish connectivity between the participating zones:

6.7.3.1. Hole Cutting

6.7.3.2. Overlap Minimization

6.7.3.3. Donor Search

6.7.3.1. Hole Cutting

Hole cutting is the process by which cells lying outside of the flow region (that is, inside bodies and outside of the computational domain) are marked as dead cells. This is achieved by marking all the cells that are cut by physical boundary zones (wall, inlet, outlet, symmetry, and so on), and marking seed cells determined to lie outside the flow region for subsequent flood filling of dead cells. The result of this flood filling is a valid overset mesh with maximum mesh overlap.

Figure 6.65: Overset Component and Background Meshes Before Hole Cutting (p. 1213) and Figure 6.66: Overset Component and Background Meshes After Hole Cutting (p. 1213) show an overset mesh before and after hole cutting. All of the background cells that are bounded or cut by the cylinder wall are classified as dead during hole cutting. The maximum overlap mesh shown in Figure 6.66: Overset Component and Background Meshes After Hole Cutting (p. 1213) is a valid overset

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

mesh, however, it may not be ideal. An overly large overlap between component and background meshes is computationally inefficient (the equations are solved in more cells than are necessary). Additionally, the cell sizes of the overlapping meshes may vary greatly—this affects the data interpolation and is detrimental to solution quality. Ideally meshes should transition in regions of similar resolution.

Figure 6.65: Overset Component and Background Meshes Before Hole Cutting

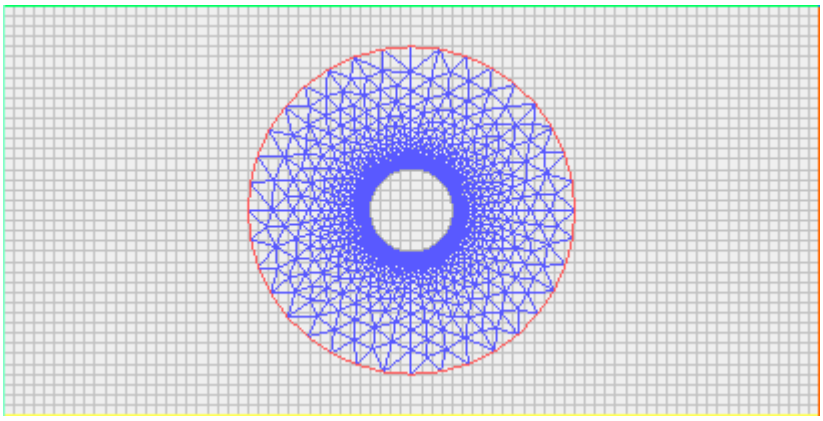
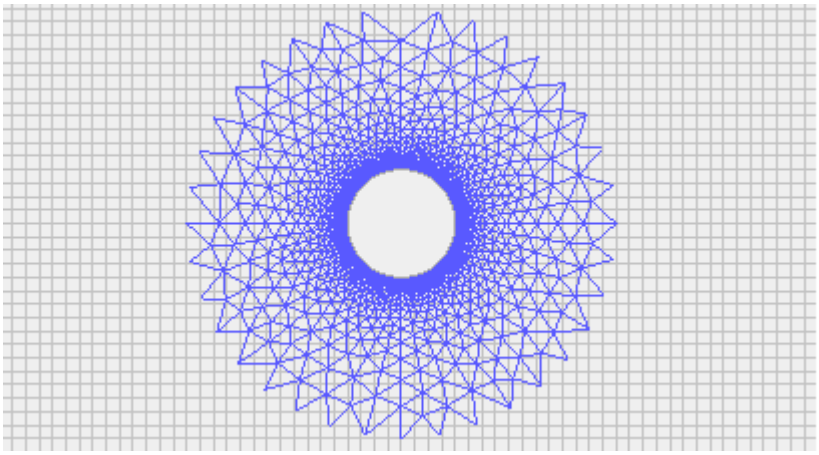


Figure 6.66: Overset Component and Background Meshes After Hole Cutting



6.7.3.1.1. Hole Cutting Control

By default all boundary zones (other than overset boundaries) cut all other cell zones and mark dead cells. If this is not the desired behavior (see Overset Topologies (p. 1208)), you can manually specify that a boundary zone does not cut certain cell zones. This optional cut control is available using the following text command:

`define` → `overset-interfaces` → `cut-control` → `add`

A component mesh with only overset boundaries (for example, a refinement mesh zone overlaying other zones) will cut into other mesh zones at its center and mark a single dead cell (cut seed) in the overlaying mesh zones. This acts as a pilot hole that is then enlarged during the overlap minimization. In cases where all cut boundaries of a component mesh are overlapping with other boundaries, you can force cut seeds in all component zones using the following text command:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

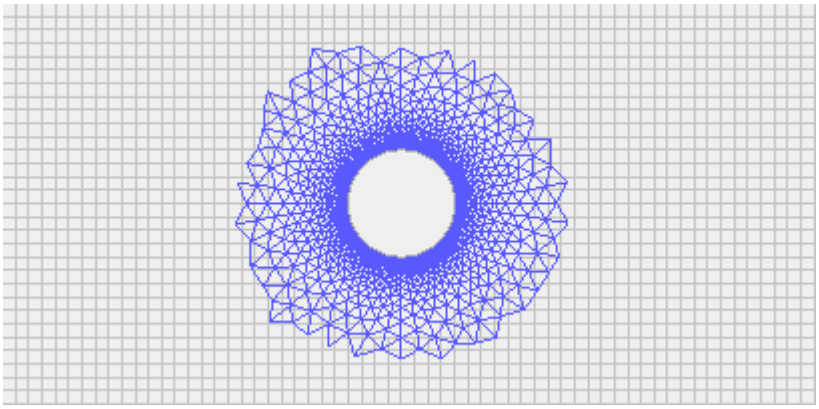
`define → overset-interfaces → cut-control → cut-seeds → cut-seeds-for-all-component-zones?`

Cut seeds are automatically enabled for all component zones when a cut control is added.

6.7.3.2. Overlap Minimization

Overlap minimization is used to minimize mesh overlap among different component and background meshes by converting additional solve cells into receptor cells and turning unnecessary receptors into dead cells. During this process, a solve cell is turned into a receptor cell if the cell can find a suitable donor cell with higher donor priority. By default, smaller cells have a higher donor priority. Thus, in mesh overlap areas, without additional user input the solver attempts to obtain the solution on the finest local mesh. The resulting mesh interface moves to an area where the meshes are more comparable in cell size, leading to better solution quality. Figure 6.67: Overset Component and Background Meshes After Overlap Minimization (p. 1214) shows the cylinder case from the previous figure after the overlap minimization step.

Figure 6.67: Overset Component and Background Meshes After Overlap Minimization



Note that overlap minimization is not literally minimizing the mesh overlap. The overlap minimization is based on a flood filling of receptor cells, wherever suitable solve cells with higher donor priority can be found. The resulting actual mesh overlap depends on the local donor priority distribution.

Also note that with overlap minimization, data interpolation between cell zones does not necessarily occur at the overset boundaries. The purpose of specifying an overset boundary is primarily to specify that overset mesh coupling should happen, and not where it should occur.

You have the option to specify different donor priority methods in order to control the location of the overset interface. The text user interface command `define/overset-interfaces/options/donor-priority-method` allows you to specify that the donor priority is based either on the cell size (proportional to the inverse of the cell volume) or the boundary distance (proportional to the inverse of the distance to the closest boundary). Overlap minimization with donor priority based on cell size (default) works best if the component mesh resolution is fine near walls and increases away from walls, to spacings similar to—or larger than—the background mesh (see Figure 6.65: Overset Component and Background Meshes Before Hole Cutting (p. 1213) for an example of this type of mesh). If the overlapping meshes have uniform and nearly identical resolutions, then overlap minimization based on cell size will not reduce the overlap and a donor priority based on the boundary distance should be used instead. The following two figures illustrate an overset mesh before and after initialization where a donor priority based on boundary distance is used. Note that

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

for the evaluation of donor priorities based on boundary distance, all boundaries (including inlets, outlets, and so on) are used for the distance evaluation.

Figure 6.68: Overset Mesh Before Hole Cutting

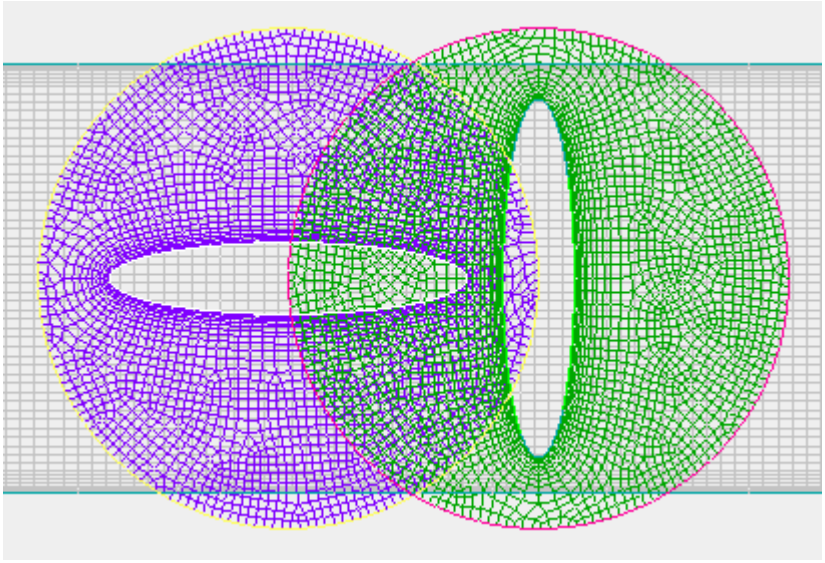
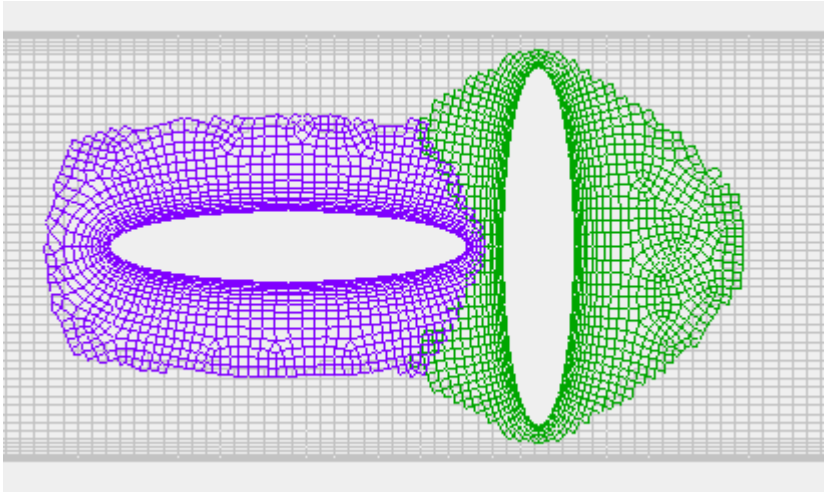


Figure 6.69: Overset Mesh After Minimization Based on Boundary Distance



You have the option for additional control over the overlap minimization process by assigning grid priorities to component and background meshes, using the text user interface command `define/overset-interfaces/grid-priorities`. Meshes (that is, cell zones) with higher grid priorities are favored during the minimization process, irrespective of the local cell donor priority. If two cell zones have the same grid priority, then the cell donor priorities are used for the overlap minimization. For example, by assigning a higher grid priority to a coarser mesh, you can force Fluent to minimize the finer mesh and obtain the solution on the coarser mesh, even if the donor priority is based on cell size. Using grid priorities, similar to using donor priority based on boundary distance, is helpful if local mesh distributions are irregular, and produce overlap regions that are irregular as well. Note that the grid priorities take precedence over the donor priorities.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

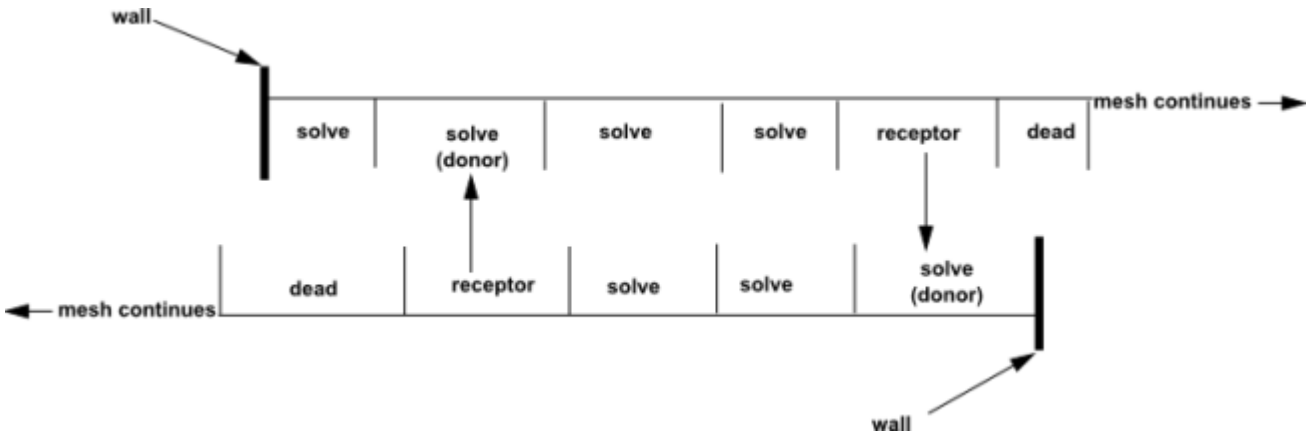
You have the option of disabling overlap minimization by using the text command `define/overset-interfaces/options/minimize-overlap?`. This command is only available globally and applies to all overset interfaces in the domain.

6.7.3.3. Donor Search

The donor search is the final step in establishing the domain connectivity. Fluent searches other meshes for valid solve cells for each receptor. The solve cell containing the cell centroid of the receptor cell, along with its connected solve cells, are used as donor candidates for a given receptor. Each receptor must have at least one valid donor cell.

There must be four or more cells in the overlap of both meshes to ensure a successful donor search. The receptor cells, which form the fringe layer of a mesh zone, must overlap sufficiently with the opposite mesh, such that they find valid solve cells as donors. For an example of valid mesh overlap, see Figure 6.70: Valid Overlap (p. 1216).

Figure 6.70: Valid Overlap



6.7.4. Diagnosing Overset Interface Issues

Topology detection of overset interfaces in Fluent is automatic and does not require user input. However, for it to work well, it requires that the provided meshes are of sufficient quality, particularly when dealing with meshes that have overlapping boundaries.

Overset meshes are built from parts that can be meshed independently, which can lead people to think of this as a much simpler process for mesh generation. This is not always the case, as considerable thought is often required to generate good meshes for the individual components to avoid problems during overset interface initialization.

If you are faced with issues during the initialization of an overset interface, there is most often no recovery without making changes to the mesh. You must analyze the cause of the issue, using the available postprocessing and diagnostic tools, and improve the mesh before the simulation can continue.

When diagnosing overset interface failures, it can be convenient to use the text commands `define/overset-interfaces/intersect` and `define/overset-interfaces/clear` (available after enabling the expert tools available under `define/overset-interfaces/options/expert?`) to execute and clear the hole cutting without having to initialize the solution.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The issues most often encountered are:

6.7.4.1. Flood Filling Fails During Hole Cutting

6.7.4.2. Donor Search Fails Due to Orphan Cells

6.7.4.1. Flood Filling Fails During Hole Cutting

These are the issues that can cause problems with flood filling:

6.7.4.1.1. Incorrect Seed Cells

6.7.4.1.2. Leakage Between Overlapping Boundaries

6.7.4.1.1. Incorrect Seed Cells

Flood filling of dead cells will fail if Fluent inadvertently identifies seed cells inside of the fluid zone instead of outside. This can occur as a result of complexities in the topology or geometry of the boundary zones, or because of mesh resolution issues. It manifests itself as an entire fluid region, or a complete cell zone, being marked as dead.

This issue can be diagnosed by using the list command with the overset verbosity set to `1` (see Overset Interface Listing (p. 1233)).

You can use the TUI command `define/overset-interfaces/debug-hole-cut` (available after enabling the expert tools available under `define/overset-interfaces/options/expert?`) to identify the problematic seed cells. This command allows you to execute the hole cutting process on a specific overset interface and mark dead and seed cells.

When the flood fill option of the debug command is disabled, it will create two registers with dead cells and seed cells which you can then display using the `define/overset-interface/display-cells` text command. The dead marks show the cells cut by the physical boundary zones. The seed cells (mark0 register) show the cell candidates that will be used for flood filling.

If the flood fill option of the debug command is enabled, Fluent will then also create a register of the seed cells from where flood filling actually started (mark1 register). If flood filling fails, displaying these mark1 cells is the most efficient way to detect the problematic locations in the model.

It is critical that all the seed cells are located inside bodies or outside of the computational domain. If there is a seed cell in the fluid region, it will cause a flood fill failure. It is not possible to recover from flood filling failure without making changes to the model.

6.7.4.1.2. Leakage Between Overlapping Boundaries

If no bad seed cells are found, as described in Incorrect Seed Cells (p. 1217), and there are overlapping boundaries, it is likely that the overlapping boundaries do not match sufficiently and the flood filling of dead cells leaks from the inside of the body to the outside. As a result, a complete cell zone can disappear, due to having only dead cells. The permissible tolerance between overlapping boundary zones is at most one cell-size in the wall normal direction. This can be restrictive when dealing with prismatic boundary layer meshes with high cell aspect ratios at the wall. If your model suffers from this leakage, you generally need to improve the quality of the mesh, by making the overlapping boundary zones match better.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

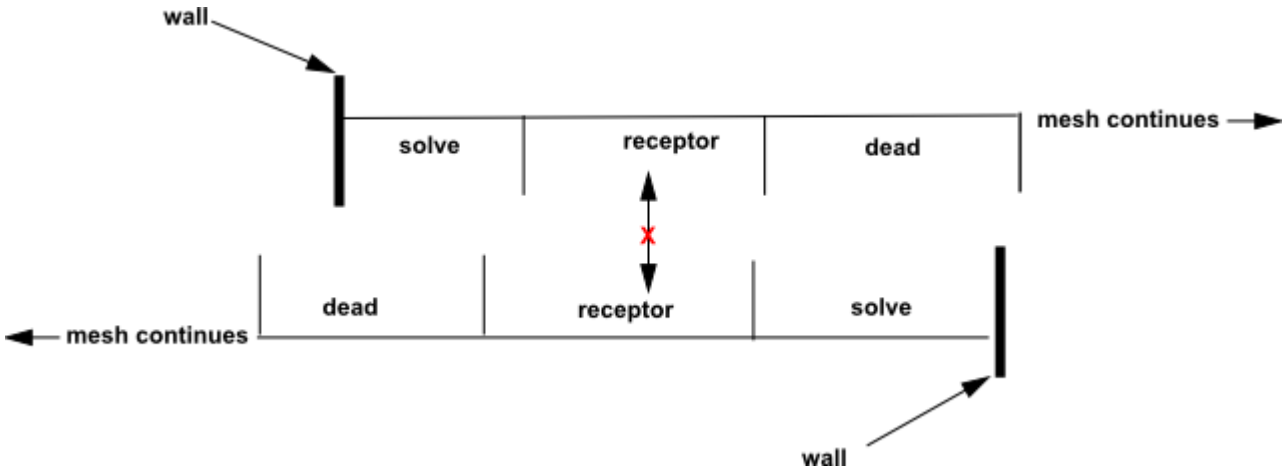
6.7.4.2. Donor Search Fails Due to Orphan Cells

If a receptor cell cannot find a valid donor cell, then it becomes an *orphan* cell. The presence of orphan cells generally indicates that there is insufficient overlap between meshes or that the mesh resolutions do not match well. As a result, the receptors are either finding no candidate donor cells, or invalid candidates (dead or receptor cells).

Orphan cells can be an issue when walls of different meshes are in proximity. You must have at least four cells in a gap, such as the ones between the walls in Figure 6.58: Valid Overset Meshes with Components in Close Proximity (p. 1209), in order to have a good connection between the meshes—otherwise orphan cells are created. When a receptor overlays another receptor, it becomes an orphan.

See Figure 6.71: Invalid Overlap Creating Orphans (p. 1218) for an example of insufficient overlap causing the creation of orphan cells. For information on visualizing orphan cells, see Postprocessing Overset Meshes (p. 1229). Figure 6.70: Valid Overlap (p. 1216) shows meshes overlapping sufficiently to avoid the creation of orphan cells.

Figure 6.71: Invalid Overlap Creating Orphans



You can mark the orphan cells, as described in Overset Cell Marks (p. 1233), which will likely indicate where the mesh overlap must be increased. Sometimes, increasing the mesh resolution can lead to a sufficient increase in mesh overlap. If mesh resolution is the issue, you can also use local mesh adaption, along with mesh modification, to refine the mesh. Most often the mesh requiring adaption is the mesh where the orphan should find its donors, and not the mesh zone containing the orphan. For details on overset-specific adaption tools in Fluent, see Overset Mesh Adaption (p. 1219).

If the presence of orphan cells in the simulation is unavoidable and you proceed with a calculation, by default the solver applies a numerical treatment that attempts to assign reasonable values to the orphan cells. This treatment has been found to prevent divergence in some instances, allowing the solution to proceed; however, the orphan cell treatment does not guarantee the solution quality and robustness a priori. The success of this treatment is case dependent, and difficult to predict based on orphan cell counts. It is therefore highly recommended that you make every effort to eliminate orphan cells by modifying the mesh, as described previously.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.7.5. Overset Mesh Adaption

Fluent has functionality that allows you to attempt to remove orphan cells, reduce size mismatches between donor and receptor cells, and/or increase the mesh resolution in gaps as needed (in order to prevent the creation of orphan cells) by locally refining the mesh. These are overset-specific adaption tools that analyze the initialized overset interface and can perform marking and adaption of cells in a single step.

It is important to note that there are different reasons for the existence of orphan cells, and that not all orphans can be removed by locally refining the mesh. Overset adaption will proceed iteratively, up to a maximum of five adaption cycles: cells are marked at the start of each cycle, and by default two sweeps of adaption are applied during each cycle. Multiple adaption sweeps are performed for a given set of adaption markings in order to reduce the time taken for the adaption process. The overset interface needs to be reestablished after each adaption cycle, which can be computationally costly. In cases where mesh resolution is the reason for orphans, this adaption process should effectively improve the mesh and reduce the orphan count.

Overset mesh adaption can be used to improve the mesh either manually on demand, or automatically for moving and dynamic mesh simulations. If automatic overset adaption is enabled, then the mesh is marked and adapted automatically at every time step after the mesh update. Since it is computationally costly to establish the overset connectivity at the end of each adaption cycle, Fluent performs only a single cycle during automatic overset adaption. Thus, a single automatic overset adaption step may not be as thorough in removing orphan cells as an instance of manual adaption; however, automatic overset adaption will be executed at every time step if orphan cells persist.

6.7.5.1. Marking for Orphan Adaption

For each orphan, all of the cells in the overlapping meshes that bound the orphan's cell centroid are marked for adaption; by default, an additional buffer layer of cells surrounding those bounding cells are also marked. The mesh zone that contains the orphan cell is not marked for adaption.

6.7.5.2. Marking for Size Adaption

Excessively large donor cells are marked for adaption if the ratio of the donor cell volume (V_d) divided by its smallest receptor cell volume (V_r) exceeds a limit. This limit is controlled by a maximum length scale ratio R_l. Donor cells with a V_d / V_r value greater than R_l^3 in 3D and greater than R_l^2 in 2D are marked for adaption. The default value for R_l is 3.

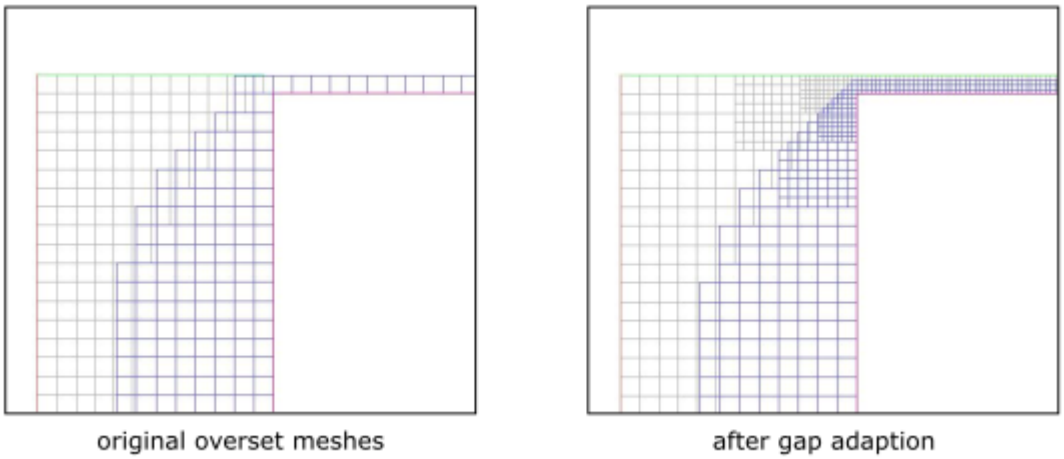
After initialization, the value for V_d / V_r can be postprocessed by selecting the **Overset Donor Size Ratio** field variable (under the **Mesh...** category).

6.7.5.3. Marking for Gap Adaption

For overset meshes, it is recommended that the gaps between walls are resolved by a minimum of 4 cells, in order to create a valid overset interface and prevent the creation of orphan cells. You can specify that the cells in gaps with a resolution that is less than this recommendation (or another gap resolution value of your choice) are marked for adaption, so that any subsequent overset adaption (manual or automatic) increases the resolution to meet or exceed the gap resolution value. Note that marking cells for gap adaption is not enabled by default, but can be enabled as described in Overset Adaption Controls (p. 1221).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.72: Overset Gap Adaption



6.7.5.4. Using Manual Overset Adaption

Overset adaption can be applied manually on demand in order to improve the mesh. Typically it is used in steady simulations or simulations without mesh motion.

The overset mesh is marked and adapted in a single step using the following text command:

`define` → `overset-interfaces` → `adapt` → `adapt-mesh`

You can perform just the marking step for overset adaption by entering the following text command: `define/overset-interfaces/adapt/mark-adaption`. Note that it is not necessary to perform this marking step before using the command to adapt; the purpose of such marking is primarily to verify the adaption parameters and evaluate whether they need to be changed. The text command to mark for adaption will mark cells (mark0 for orphan adaption, size adaption, and gap adaption, and mark1 for coarsening), so that you can then create cell registers and thereby postprocess them as described in Overset Cell Marks (p. 1233). Note that mark0 and mark1 are only available after enabling the expert tools through the following text command: `define/overset-interfaces/options/expert?`.

You can use the following text command to set a verbosity level of 2 if you want to print the number of cells marked for refinement in the console when the overset mesh is adapted: `define/overset-interfaces/options/verbosity`.

6.7.5.5. Using Automatic Overset Adaption

With moving and dynamic mesh simulations, it can be difficult to create an initial mesh that meets all of the needs throughout the simulation while also being computationally efficient. Rather than refining the initial mesh in regions that only require such refinement temporarily, it can be preferable to use automatic overset adaption so that Fluent analyzes the mesh at every time step and then refines or coarsens the cells as required; the goal is to remove orphans and/or reduce size mismatches between donor and receptor cells.

To enable automatic overset adaption, you can use the Automatic Mesh Adaption Dialog Box (p. 5403) as described in Overset Adaption (p. 3799), or you can use the following text command:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

`define` → `overset-interfaces` → `adapt` → `set` → `automatic?`

Automatic overset adaption is fully compatible with conventional automatic mesh adaption, such as when it is used to more accurately track Volume of Fluid (VOF) interfaces (as described in VOF Adaption (p. 3795)). The two adaption methods can be enabled at the same time, and will not interfere with each other.

6.7.5.6. Overset Adaption Controls

If the default settings for overset adaption are not suitable for your case, you can revise them using the text commands in the following menu:

`define` → `overset-interfaces` → `adapt` → `set`

The text commands in this menu include:

`automatic?`

Enables/disables the option to automatically adapt overset meshes during the calculation, to remove orphans, reduce size mismatches between donor and receptor cells, and/or increase the mesh resolution in gaps as needed (in order to prevent the creation of orphan cells). This option is disabled by default.

`maximum-refinement-level`

Sets the maximum level of refinement during overset adaption, in conjunction with the value set using the `mesh/adapt/set/maximum-refinement-level` text command (the larger of the two values is used). This is set to 4 by default.

`mark-size?`

Enables/disables the option to adapt based on donor-receptor cell size differences.

`length-ratio-max`

Sets the length scale ratio threshold used to determine which cells are marked for adaption based on donor-receptor cell size differences. This is set to 3 by default and is only available when `define/overset-interfaces/adapt/set/mark-size?` is enabled.

`mark-orphans?`

Enables/disables the option to adapt for orphan reduction.

`buffer-layers`

Sets the number of cell layers marked in addition to the cells marked for orphan adaption. This is set to 1 by default.

`mark-fixed-orphans?`

Enables/disables the option to also adapt based on cells that are not actual orphans because they were fixed by accepting neighbor donors. This option is only available when `define/overset-interfaces/adapt/set/mark-orphans?` is enabled.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

mark-gaps?

Enables/disables the option to adapt under-resolved gaps in order to prevent the creation of orphan cells. This option is disabled by default.

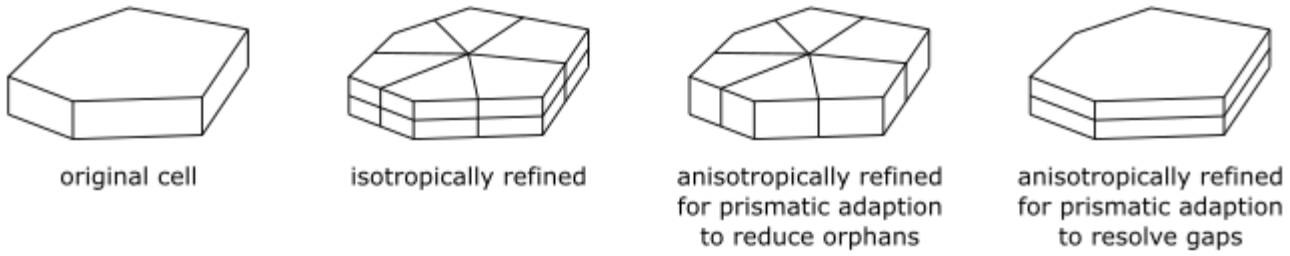
gap-resolution

Sets the target (minimum) gap resolution used when marking cells for gap adaption. This is set to 4 by default and is only available when `define/overset-interfaces/adapt/set/mark-gaps?` is enabled.

prismatic?

Enables/disables the option to use prismatic adaption for the cells that are refined in order to reduce orphans and/or better resolve gaps. Instead of being isotropically refined, these cells will be anisotropically refined (that is, they will not be split on particular edges), in order to reduce the resulting cell count. This adaption option is referred to as "prismatic", because it is only available for 3D prismatic cells that are deemed suitable. For orphan adaption, the goal is to produce cells that have a lower aspect ratio; for gap adaption, the goal is to produce a mesh resolution that is suitable for boundary layers near walls.

Figure 6.73: Prismatic Adaption Examples



mark-coarsening?

Enables/disables the option to coarsen the mesh if mesh refinement is no longer needed. This option is enabled by default.

adaption-sweeps

Sets the number of sweeps of adaption applied during each adaption cycle. This is set to 2 by default.

6.7.6. Overset Meshing Best Practices

Receptor and donor cell sizes should be comparable to ensure that interpolation errors are minimized. You can adapt the mesh to reduce the mismatch between donors and receptors (see Overset Mesh Adaption (p. 1219)).

You must have a minimum of four cells in a gap to create a valid overset interface, but a higher number is recommended to ensure robust coupling between meshes and to prevent the creation of orphan cells (see Figure 6.71: Invalid Overlap Creating Orphans (p. 1218) and Figure 6.70: Valid Overlap (p. 1216)). You can adapt the mesh to refine under-resolved gaps (see Overset Mesh Adaption (p. 1219)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When using the pressure-based solver, the default pressure-velocity coupling scheme for overset meshes is **Coupled**, and this is generally recommended. However, for transient simulations using time step sizes comparable to CFL limits, the greater computational efficiency of a segregated scheme (**SIMPLE**, **SIMPLEC**, or **PISO**) can help in reducing the total simulation time. Note that as you employ larger time step sizes with respect to CFL limits, the tighter pressure-velocity coupling offered by the default **Coupled** scheme makes it the preferred scheme for such cases. For details on these schemes, see Choosing the Pressure-Velocity Coupling Method (p. 3585).

The use of standard initialization with defaults computed from all zones has been found to be useful for turbulent simulations starting from quiescent flow, especially for the segregated pressure-velocity coupling schemes.

It is recommended that you start transient simulations from a converged steady-state solution.

If you are experiencing startup issues with a steady-state case, it is recommended that you ramp-up to the final boundary conditions. For example, you can begin with a lower inlet velocity and a higher viscosity to establish flow, before increasing to the target velocity and the desired final material properties.

When using automatic time step calculation for the pseudo time method (with the global time step method selected), it is recommended that you use the user-specified length scale method.

If you replace a zone that is part of an overset interface, you should either reinitialize the solution or patch the solution on the replaced zone(s) before continuing with the calculation.

It is recommended that you use the double-precision solver for overset mesh simulations.

Closed domain cases are defined as setups that do not have a flow inlet or outlet boundary; this type of setup is not supported with overset meshing if the working fluid is incompressible. If such a setup is unavoidable, creating a small pressure-outlet boundary far away from the region of interest is recommended before attempting to run the case using overset meshes.

For dynamic and sliding mesh cases, note the following:

- The ideal time step size should be chosen such that the relative mesh motion does not exceed the length of the smallest cell at the overset interface for a given time step for first order in time, and half that length for second order in time. The mesh length used for calculating the time step size should be restricted to cells in the overset overlap region. This ensures a more gradual evolution of the overset interface with the motion and is recommended for better accuracy and stability.

- With Volume of Fluid (VOF) simulations using second order in time, the solution strategies for multiphase modeling could be beneficial in improving the stability of your simulations (see VOF Solution Stability Controls (p. 3195)). When using a large CFL value (that is, greater than 1), it is recommended that you run with first order in time to ensure the simulations remain stable.

- In general, the motion of the overset interface may change the status for individual cells from dead to receptor or solve, and vice versa. The exact instances depend on the relative mesh sizes of the moving cell zones participating in the overset interface, the magnitude of the relative mesh velocity, and the time step size. There is currently no mechanism to eliminate this type of cell status change for general cases.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- While the solver is capable of handling the cell status change, it is recommended that you actively monitor these instances by setting the overset mesh verbosity to 1 using the `define/overset-interface/options/verbosity` text command. With such a verbosity, the solver reports the instances of dead to solve cells in the console. Keeping these to a minimum is highly recommended for better accuracy and stability.

- Do not have large variations in mesh resolution in the motion path that could trigger large changes in the overset interface as the motion proceeds. The use of grid priorities can minimize large variations in the overset interface.

- When using an overset mesh to simulate flow through narrow gaps that open or close over time (such as in a valve or a rotary pump), you can use the gap model to block or decelerate the flow when the gaps are closed (as described in Controlling Flow in Narrow Gaps for Valves and Pumps (p. 1234)).

You can specify that the overset interfaces use either the least squares interpolation method (the default) or the inverse distance interpolation method by using the following text command.

`solve` → `set` → `overset` → `interpolation-method`

When the porous zone option is enabled with an overset mesh, it is recommended that you use the default least squares overset interpolation method, especially if you have a porous zone that overlaps with another porous zone. If you need to use the inverse distance interpolation method, you may also need to use pressure interpolation schemes other than second order, as well as enable the use of high-order pressure extrapolation at the overset interface (available through the following text command: `solve/set/overset/high-order-pressure? yes`). This improves the accuracy of the pressure gradients at the overset interface, which is vital for predicting the pressure drop and flow in the porous zone.

6.7.7. Overset Meshing Limitations and Compatibilities

Overset meshing has the following limitations and compatibilities:

6.7.7.1. Limitations

Overset meshing currently has the following limitations:

- Overset interfaces cannot contain **solid** cell zones.

- Non-conformal mesh interfaces inside overset cell zones are only permitted if the mesh interfaces do not spatially overlap with the cells where the overset interfaces lie.

- Component zones cannot have **periodic** boundary conditions.

- Background zones cannot have **overset** boundaries.

- Component mesh boundaries cannot overlap with coupled walls.

- FMG initialization is not available.

- The FAS multigrid scheme is not used.

- Implicit residual smoothing cannot be enabled.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Overset meshing is not supported for incompressible flows in a closed domain.
- The use of node weights for node-based gradients in postprocessing is not available for cases with overset meshes.
- Overset meshing is not compatible with the **Coupled Level Set + VOF** hybrid model.
- The turbulence models that are supported with overset meshing (listed in the section that follows) cannot be used in combination with Scale-Resolving Simulation (SRS) models / options.
- When using an overset mesh with the DPM model:
  - The **High-Res Tracking** option must be enabled
  - Wall film particles cannot move between different overset meshes

6.7.7.2. Compatibilities

Overset meshing is compatible with the transient and steady-state solvers. Both laminar and turbulent flow regimes with heat transfer are allowed as well as single phase flows, and Volume of Fluid (VOF) and mixture multiphase flows. These applications are valid with overset for the listed solver options and configurations. Unless otherwise specified, consider all other solver features and models to be unsupported.

Supported Options and Models:

- Pressure-based solver using:
  - 2D (planar and axisymmetric) and 3D flows in the absolute velocity formulation
  - Compressible flows
  - Cell-based gradient methods: **Green-Gauss Cell Based** and **Least Squares Cell Based**
  - All pressure interpolation schemes
  - First-order and second-order spatial schemes
  - **SIMPLE**, **SIMPLEC**, **PISO**, **Fractional Step**, and **Coupled** schemes for pressure-velocity coupling
  - All mass flux types (**Rhie-Chow: distance based** or **Rhie-Chow: momentum based**)
  - A pseudo time method
  - First-order, second-order, and (if available) bounded second-order transient formulations
  - The Non-Iterative Time Advancement (NITA) option
  - The hybrid NITA option (for cases that involve the Volume of Fluid model)
- Density-based solver using:
  - 2D (planar and axisymmetric) and 3D flows in the absolute velocity formulation
  - Compressible flows

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - Cell-based gradient methods: **Green-Gauss Cell Based** and **Least Squares Cell Based**
  - All convective flux types (**Roe-FDS**, **AUSM**, and **Low Diffusion Roe-FDS**)
  - First-order and second-order spatial schemes
  - A pseudo time method
  - First-order, second-order, and (if available) bounded second-order transient formulations
- k-ε turbulence models: Standard, RNG, and Realizable
- k-ω turbulence models: Standard, BSL, and SST
- Spalart-Allmaras turbulence model
- Laminar-turbulent transition models: Transition SST, Intermittency Transition, and Transition k-kl-ω
- Discrete Ordinates (DO) radiation model
- Species transport model:
  - single-phase flow only
  - reactions must be disabled
- VOF models:
  - Open channel flow and surface gravity waves
  - Cavitation and evaporation/condensation mass transfer
- Mixture multiphase model:
  - Non-granular flows
  - Cavitation and evaporation/condensation mass transfer
- Cavitation and evaporation/condensation mass transfer in VOF/Mixture multiphase applications
- Non-granular flows in multiphase applications
- Dynamic and sliding meshes with the first-order transient formulation
- Gap model
- User-defined scalars
- User-defined functions (for information about overset-specific macros that can be useful when writing user-defined functions, see Overset Mesh Looping Macros in the Fluent Customization Manual)
- All AMG options except for the **Laplace Coarsening** option (for the scalar or coupled parameters)
- All boundary conditions are supported except for:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - the following non-interior face zone boundary conditions: exhaust fan, inlet vent, intake fan, outflow, and outlet vent
  - the following interior face zone boundary conditions: fan, porous-jump, and radiator
- Cell zone conditions
  - Source terms may only be defined on cell zones not participating in an overset interface
  - Fixed values may only be defined on cell zones not participating in an overset interface
  - Multiple Reference Frame (MRF) model
  - The porous zone option is supported with the superficial velocity formulation
- Hybrid and standard initialization
- An overset meshing solution may be obtained on any mesh type, including adapted meshes

6.7.8. Setting up an Overset Interface

Once you have read all of the related meshes into Fluent, use the following steps to setup an overset interface.

1. Change the boundary zones of component meshes to **overset** (boundary type), where needed.

   Important:

   You must correctly specify the **overset** boundaries for all zones in an overset interface before creating the overset interface. (Once overset interfaces are created you cannot change any boundaries from **overset** to another type or from another type to **overset**.)

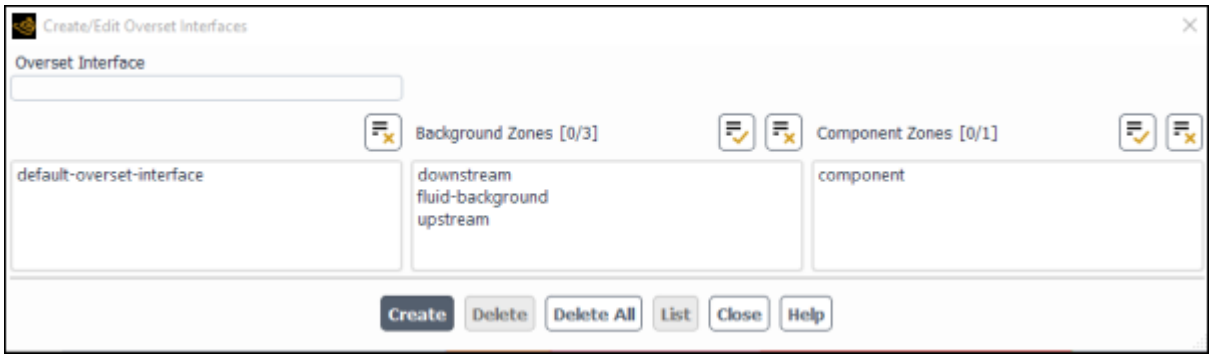
   Setup → **Boundary Conditions** **<boundary>** → **Type** → **overset**

   The Overset Dialog Box (p. 5041) will open, which you can use to revise the **Zone Name**.

2. Create an overset interface between the cell zones that must be coupled. A case can have multiple interfaces. Each interface must have at least one background and one component mesh.

   Domain → **Interfaces** → **Overset**

Figure 6.74: Create/Edit Overset Interfaces Dialog Box



a. Enter a name under **Overset Interface**.

b. Select the zones of the interface in the **Background Zones** and **Component Zones** selection lists. Ansys Fluent allows you to create interfaces that contain only component zones; in such cases, you do not select a background zone, but instead select two or more component zones.

c. Click **Create**.

3. (optional) If your overset case doesn't have overlapping boundaries, and you don't need Fluent to analyze the case for boundary overlaps, you can improve the overset performance by disabling overlap boundaries using the following text command:

   `define → overset-interfaces → options → overlap-boundaries?`

4. (optional) Use the text user interface for additional control over the hole cutting step of the interface initialization.

   - If certain boundary zones should not cut particular cell zones, use the `define/overset-interfaces/cut-control/add` text command to set up cut control.

   - If component meshes have no boundaries that will cut a cell zone, you can use the `define/overset-interfaces/cut-control/cut-seeds/cut-seeds-for-all-component-zones?` text command to ensure that all component zones get cut seeds (see Hole Cutting Control (p. 1213)).

5. (optional) Use the text user interface for additional control over the overlap minimization of the overset interface. It can be helpful to revise the default settings if your overlapping meshes have uniform and nearly identical resolutions or if local mesh distributions are irregular, as this may improve the computational efficiency and/or the solution quality. See Overlap Minimization (p. 1214) for details.

   - To specify whether the cell donor priorities used during minimization are based on the cell size or distance to the nearest boundary, use the following text command:

     `define → overset-interfaces → options → donor-priority-method`

   - To revise the grid priorities during minimization, use the following text command. The grid priority for each zone is an integer, and only the relative values between the zones matter.

     `define → overset-interfaces → grid-priorities`

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To disable overlap minimization entirely, use the following text command. This command is only available globally and applies to all overset interfaces in the domain.

  `define` → `overset-interfaces` → `options` → `minimize-overlap?`

Note that these steps only define an overset interface. Hole cutting occurs during solution initialization.

If a case contains **overset** face zones, but no defined overset interfaces, then Fluent creates a default interface with the name `default-overset-interface` upon initialization. This default overset interface contains all cell zones in the domain. If you know that some cell zones do not need to be included in an overset interface, then you can increase the solution performance by manually creating the interface and only including the relevant background and component meshes.

6.7.9. Postprocessing Overset Meshes

Fluent has several overset-specific postprocessing tools to help you analyze/understand the domain connectivity. These tools are only available after the flow field is initialized and the domain connectivity is established.

6.7.9.1. Overset Mesh Display

6.7.9.2. Overset Field Functions

6.7.9.3. Overset Cell Marks

6.7.9.4. Overset Interface Listing

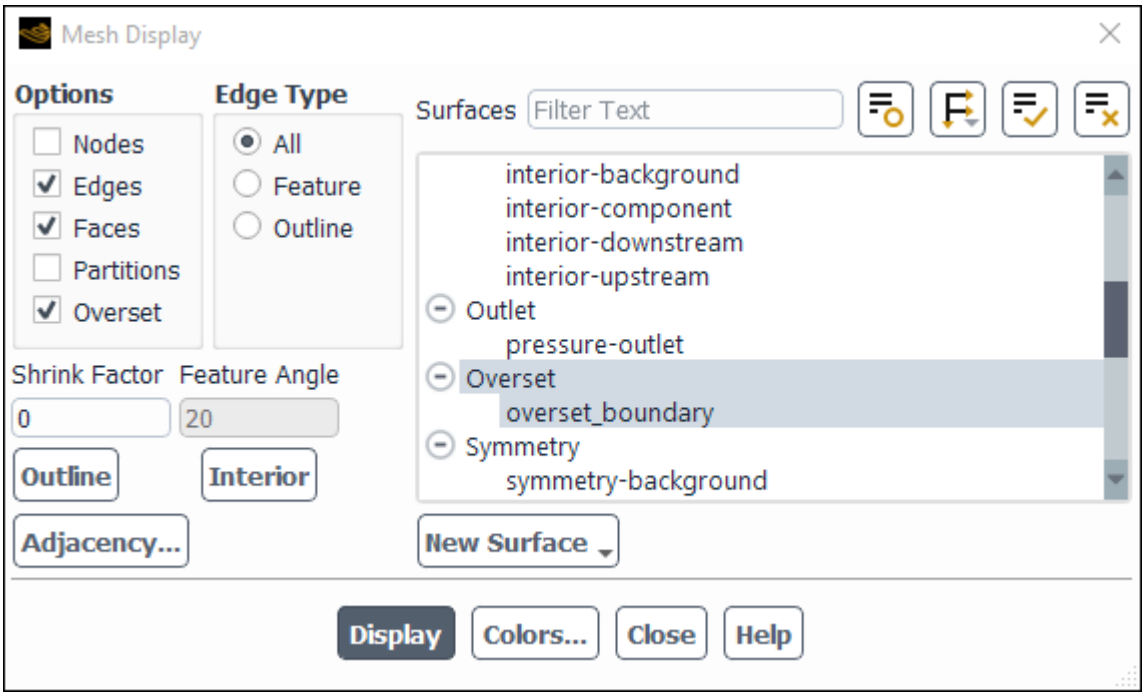
6.7.9.5. Overset Postprocessing Limitations

6.7.9.1. Overset Mesh Display

You can either visualize the complete mesh or just the active cells. If the **Overset** option is enabled in the **Mesh Display** dialog box, by default only solve cells are shown. Disabling the **Overset** option displays the complete surfaces.

Domain → **Mesh** → **Display**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You have the option to have receptor cells displayed along with the solve cells when **Overset** is selected. You can enable this option via the TUI command `define/overset-interfaces/options/render-receptor-cells?`.

Note:

- When displaying contours of postprocessing quantities, dead cell values are never shown, regardless of whether or not **Overset** is enabled in the **Mesh Display** dialog box.
- You may notice a z-fighting effect when displaying overset meshes, particularly when coloring the mesh by ID. This occurs due to the nature of overset meshes, where two meshes are coincident and the rendering tool is conflicted in determining which color to display.

6.7.9.2. Overset Field Functions

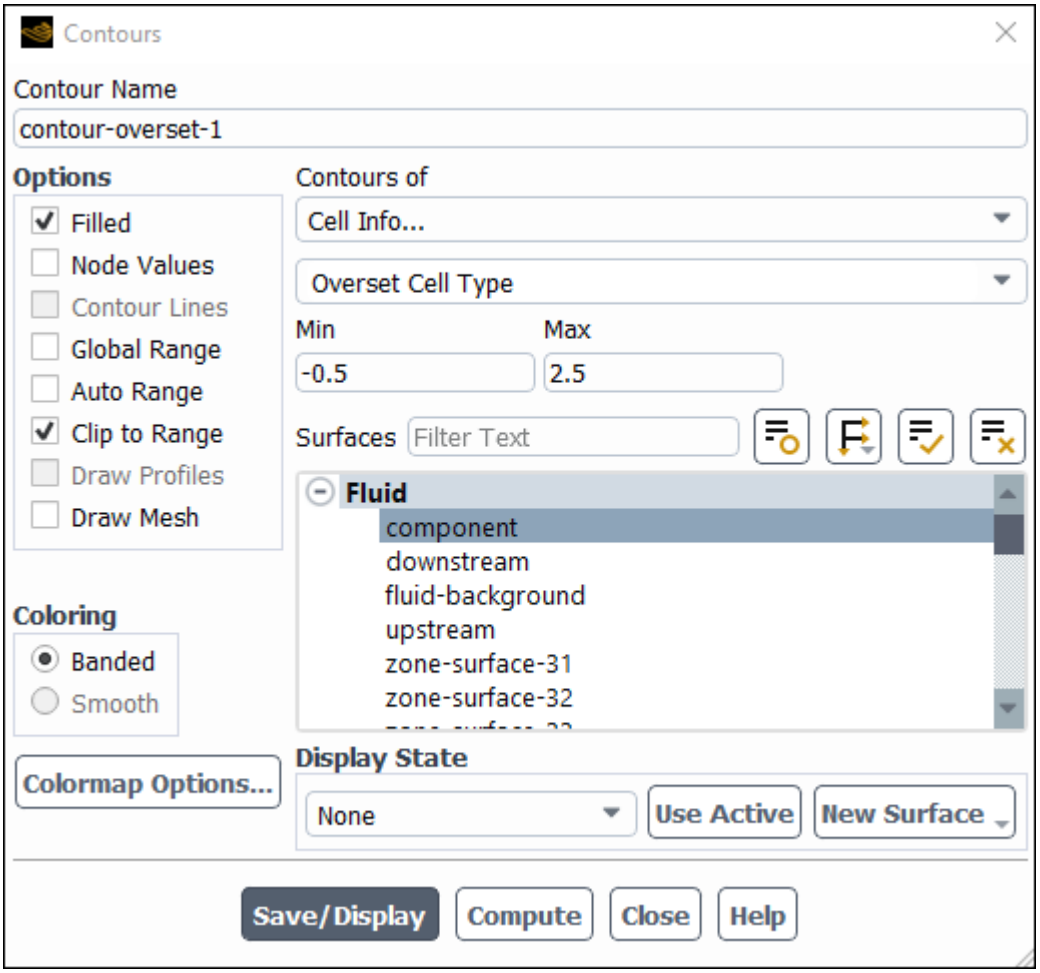
There is an **Overset Cell Type** function available in the **Cell Info...** category in the Contours Dialog Box (p. 5412). The integer function value depends on the overset cell type:

Cell Type	Integer Function Value
donor	2
solve	1
receptor	0
orphan	–1
dead	–2

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This field function is also available in the **Contours** dialog box.

Results → **Graphics** → **Contours** → **New...**



1. Select **Cell Info...** and **Overset Cell Type** from the **Contours of** drop-down lists.
2. Choose the surfaces containing the overset cells you want to visualize and click **Display**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.75: Contours of Overset Cell Type: Background Mesh

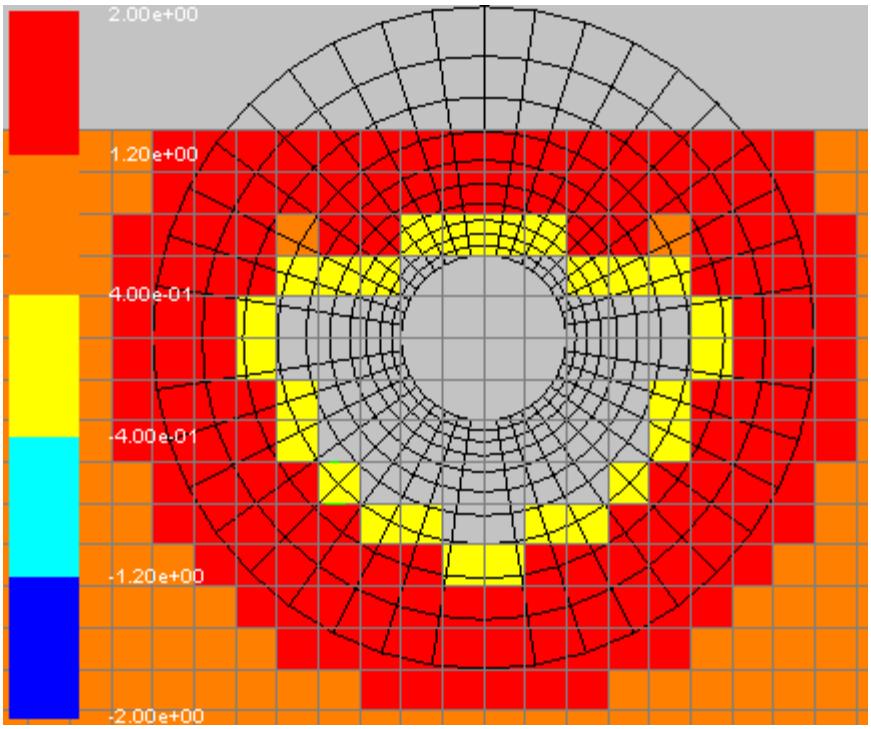


Figure 6.76: Contours of Overset Cell Type: Component Mesh

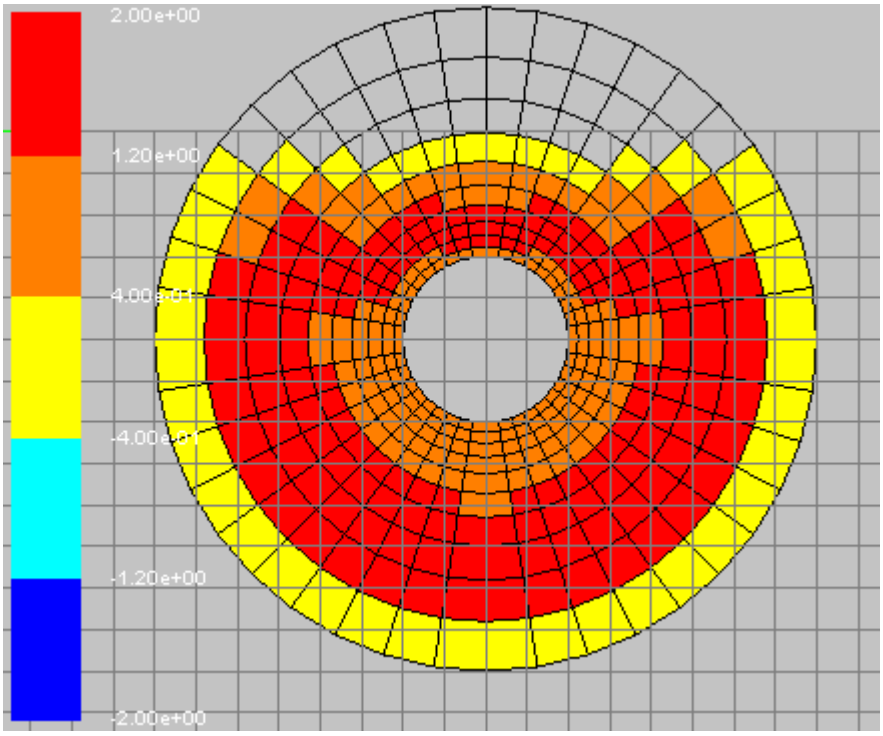


Figure 6.75: Contours of Overset Cell Type: Background Mesh (p. 1232) and Figure 6.76: Contours of Overset Cell Type: Component Mesh (p. 1232) display a background mesh and component mesh

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(sequentially) of **Overset Cell Type** for the cylinder problem shown in Figure 6.56: Overset Component and Background Mesh (p. 1207).

Note:

If you want to display orphan cells, then you must enable the TUI command `define/overset-interfaces/options/render-receptor-cells?`.

If the expert options are enabled under `define/overset-interfaces/options/expert?`, then you can also choose to display **Overset Donor Count** and **Overset Receptor Count** under **Cell Info…**. The **Overset Donor Count** is filled for each receptor and shows the number of donors. Similarly, the **Overset Receptor Count** is filled for donor cells and shows the number of receptors that are referencing the donors.

Note that the donor counts reported are the maximum number of donors available for the data interpolation. The actual number of donors used by the solver may be less.

6.7.9.3. Overset Cell Marks

It can be difficult to interpret contours of overset cells in 3D. Often, it is more illustrative to mark a certain type of overset cells and display these marked cells.

Using the text command `define/overset-interfaces/mark-cells` you can mark cells of specified overset cell type (solve, receptor, donor, orphan, or dead), either globally or on a single cell zone. This command automatically fills registers that can be displayed using the `define/overset-interfaces/display-cells` text command.

6.7.9.4. Overset Interface Listing

Use the text command `define/overset-interfaces/list` to print overset interface related information to the console. If the overset verbosity, `define/overset-interfaces/options/verbosity`, is set to `1`, then counts of each overset cell type are printed per cell zone as well.

6.7.9.5. Overset Postprocessing Limitations

Volume integrals are taken over all solve cells in the domain. This results in double counting where solve cells overlap, leading to errors in volume integrals on overset meshes.

Similarly, surface integrals, including force and moment reports, will include errors when taken on surfaces created for postprocessing (such as plane surfaces or rake surfaces) in regions where solve cells or boundary zones overlap. If surface integrals are taken on boundary zones rather than created surfaces and/or in a region without overlap, then the surface integrals are exact.

Important:

Errors in integral evaluations scale with the size of the overlap.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.7.10. Writing and Reading Overset Files

When writing overset case files, it is recommended that you use the default Common Fluids Format (CFF), as in this format the overset domain connectivity is saved in the `.cas.h5` file. This makes it unnecessary to reestablish the domain connectivity when reading an overset case into a new session, as is the case with legacy case files (`.cas`). It also guarantees an identical connectivity when a case is read on a different number of compute nodes.

When you write an overset case file in CFF and the case has not yet been initialized, Ansys Fluent will by default establish the domain connectivity prior to writing the file. This can be disabled using the `define/overset-interfaces/options/update-before-case-write?` text command, which is available after you enable the expert tools using the `define/overset-interfaces/options/expert?` command.

6.8. Controlling Flow in Narrow Gaps for Valves and Pumps

When using dynamic meshes, overset meshes, and/or sliding meshes to simulate flow through narrow gaps that open or close over time (such as in a valve or a rotary pump), you can use the gap model to block or decelerate the flow when the gaps are closed. The following sections provide details:

6.8.1. The Gap Model Approach

6.8.2. Limitations of the Gap Model

6.8.3. Recommendations for the Setup of a Simulation with Gaps

6.8.4. Using the Gap Model

6.8.1. The Gap Model Approach

For transient flows with moving face zones, narrow gaps may open or close during every time step. You can use the gap model to simulate the blockage of flow when a gap is closed. A gap region is made up of a pair of face zones, so that when one moves within a specified proximity threshold distance of the other, the fluid cells within the threshold of the zones are marked, and then the flow is blocked at the boundary faces of the marked region so that the marked cells no longer participate in the solution (for what is called a flow-blocking type region) or else decelerated in the marked region (for what is called a flow-modeling type region). The gap model has the following advantages:

- With the flow-blocking type, you can ensure that there is no flow leakage. This is a realistic flow condition for valves in the closed position.
- When the flow-blocking type does not produce a stable solution or is otherwise not preferred, you can use the flow-modeling type to decelerate the flow inside the gap regions and minimize leakage.
- It can reduce computational cost in narrow gaps by reducing the need for a high cell count to resolve high solution gradients in the flow between walls in close proximity (for example, a valve opening and closing).
- The flow-blocking type can avoid the difficulties of meshing and/or remeshing in narrow passages, especially with moving deforming meshes.
- You can define multiple gap regions, each with a unique proximity threshold.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.8.2. Limitations of the Gap Model

The following are not supported with the gap model:

- density-based solver
- axisymmetric swirl
- Eulerian multiphase model
- ablation model
- Eulerian wall film model
- Lagrangian wall film model
- solidification/melting model
- electric potential model
- Lithium-ion battery model
- acoustics models
- structural model
- discrete phase model (DPM)
- radiation models
- species models (with the exception of species transport for mixing but not reacting)
- pollutant formation models
- reactor network model
- decoupled detailed chemistry pollutant model
- multiphase flow with species transport
- the following turbulence models / options: Reynolds stress models (RSM), Scale-Adaptive Simulation (SAS) model, Detached eddy simulation (DES) model, Large Eddy Simulation (LES) model, Scale-Resolving Simulation Options (SAS, DES, or SBES/SDES)
- **Green-Gauss Node Based** gradient method (for flow-blocking type only)
- **QUICK** or **MUSCL** discretization schemes (for flow-blocking type only)
- Non-Iterative Time Advancement (NITA) scheme
- coupled walls are not allowed to be selected for a gap region
- the gap model cannot be used in a case that also has the **Flow Control** option enabled in the **Contact Detection** tab of the Options Dialog Box (p. 5121)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- the porous zone option cannot be enabled in cell zones where gaps are located

6.8.3. Recommendations for the Setup of a Simulation with Gaps

Note the following recommendations when setting up a case that uses the gap model:

Boundary Conditions

If flow blocking is used to control the flow in the gap regions of a single passage domain and the flow stream is detached from the upstream region to the downstream region, it is recommended that you do not use boundary conditions of the following types: **velocity-inlet**, **outflow**, and/or **mass-flow-inlet**; instead, the **pressure-inlet** and **pressure-outlet** types are preferred.

Important:

You must avoid boundary conditions of type **fan**, **porous-jump**, and/or **periodic** in the vicinity of either type of gap region.

Materials

In domains with blocking gap regions, incompressible liquids and constricting and expanding volumes can create unphysical pressure spikes. Therefore, it is recommended that you select the compressible-liquid option for liquids, or use compressible gas options such as the ideal gas law for gases. In case of incompressible liquids, you can also use a user-defined function (UDF) and add compressibility to provide a realistic setup.

Stability and Convergence

The time step size requires special attention when the gap model is used. A larger time step size can be used when there is no blocked gap region in the domain compared with the condition where one or multiple blocking regions exist in the domain. It is recommended that you use adaptive time stepping to dynamically adjust the time step size depending the flow condition, and set the Courant (CFL) number to 1.

Whenever poor solution convergence behavior is observed, some or all of the following recommendations may help: ensure that the initialization settings are appropriate (especially the turbulence settings); start the simulations with lower-order numerics and then move to second-order discretization; reduce the time step size; and/or reduce the momentum under-relaxation factor to 0.2. You may also want to enable the **High Order Term Relaxation** option in case the solution field variation is very considerable per each time step.

If these suggestions do not help, you can gradually increase the **Stabilization Level** from **None** in the **Advanced Options** dialog box (as described in the section that follows) and increase the maximum iterations per time step. You can also refine the mesh in the vicinity of the gap regions.

6.8.4. Using the Gap Model

To use the gap model, perform the following steps:

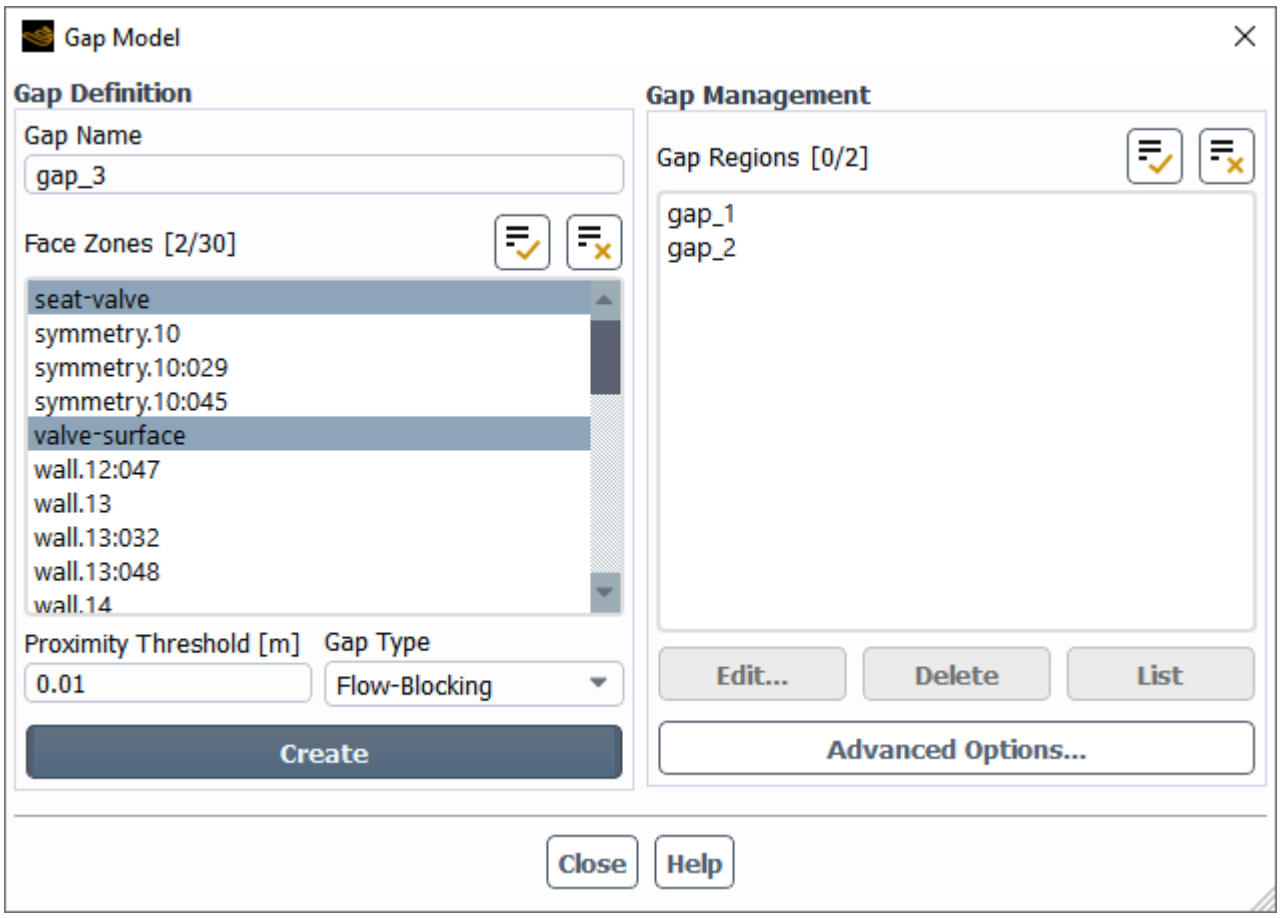
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Set up a dynamic mesh, overset mesh, and/or sliding mesh with a narrow gap that opens or closes over time (such as in a valve or a rotary pump). You must ensure that the setup respects the limitations and recommendations noted in the preceding sections.

2. Open the **Gap Model** dialog box by clicking the **Gap Model...** button in the **Domain** ribbon tab (**Mesh Models** group):

   Domain → Mesh Models → Gap Model...

   Figure 6.77: The Gap Model Dialog Box



   a. Create one or more gap regions using the input controls in the **Gap Definition** group box. For each one:

      i. Provide a **Gap Name**. The name cannot consist solely of numbers.

      ii. Select two or more face zones (of type wall or symmetry) that could come in close proximity to each other from the **Face Zones** list; note that a given face zone may be used in multiple gap regions, but two or more face zones used in one gap region may not all be used in one other gap region. It is recommended that you use the minimum number of face zones as possible per gap region, as additional unnecessary face zones will adversely affect the calculation speed.

      iii. Specify a **Proximity Threshold**, that is, the maximum distance between the face zones when flow blockage / deceleration occurs.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

iv. Select one of the following from the **Gap Type** drop-down list:

- **Flow-Blocking**

  This type completely blocks the flow in the gap region, as a numerical zero-mass-flux boundary condition is applied at the boundary faces of the marked region so that the marked cells no longer participate in the solution. This can be appropriate for a valve that completely closes. It is recommended that you use this for meshes that only move translationally. Meshes that undergo rotational motion (such as gears or pumps) may experience divergence; in such cases, it may be possible to proceed using a larger proximity threshold so that the region of cells marked for blockage is larger (and you thus avoid creating "islands" of unmarked cells surrounded by marked cells).

- **Flow-Modeling**

  This type decelerates the flow in the cells of the marked region by manipulating the viscosity or source terms. Since it allows some leakage (though this can be negligible, based on your setup), it can be more robust if appropriately set up. It is recommended for meshes that undergo rotational motion, such as gears or pumps.

  If you plan on controlling the flow based on the viscosity (the default method), you should specify an appropriate **Gap Reynolds Number** in order to obtain the desired level of sealing. A lower **Gap Reynolds Number** results in better sealing. You can determine a suitable value through trial and error during the simulation, or estimate it using the expected properties in the gap region according to the following equation:

$$\mathrm{Re}_{\mathrm{gap}}=\frac{\rho_{\mathrm{avg}}u_{\mathrm{avg}}L}{\mu_{\mathrm{avg}}} \tag{6.4}$$

  where

  $\mathrm{Re}_{\mathrm{gap}}$ = gap Reynolds number

  ρ_{avg} = averaged density

  u_{avg} = averaged velocity

  L = proximity threshold

  μ_{avg} = averaged laminar viscosity

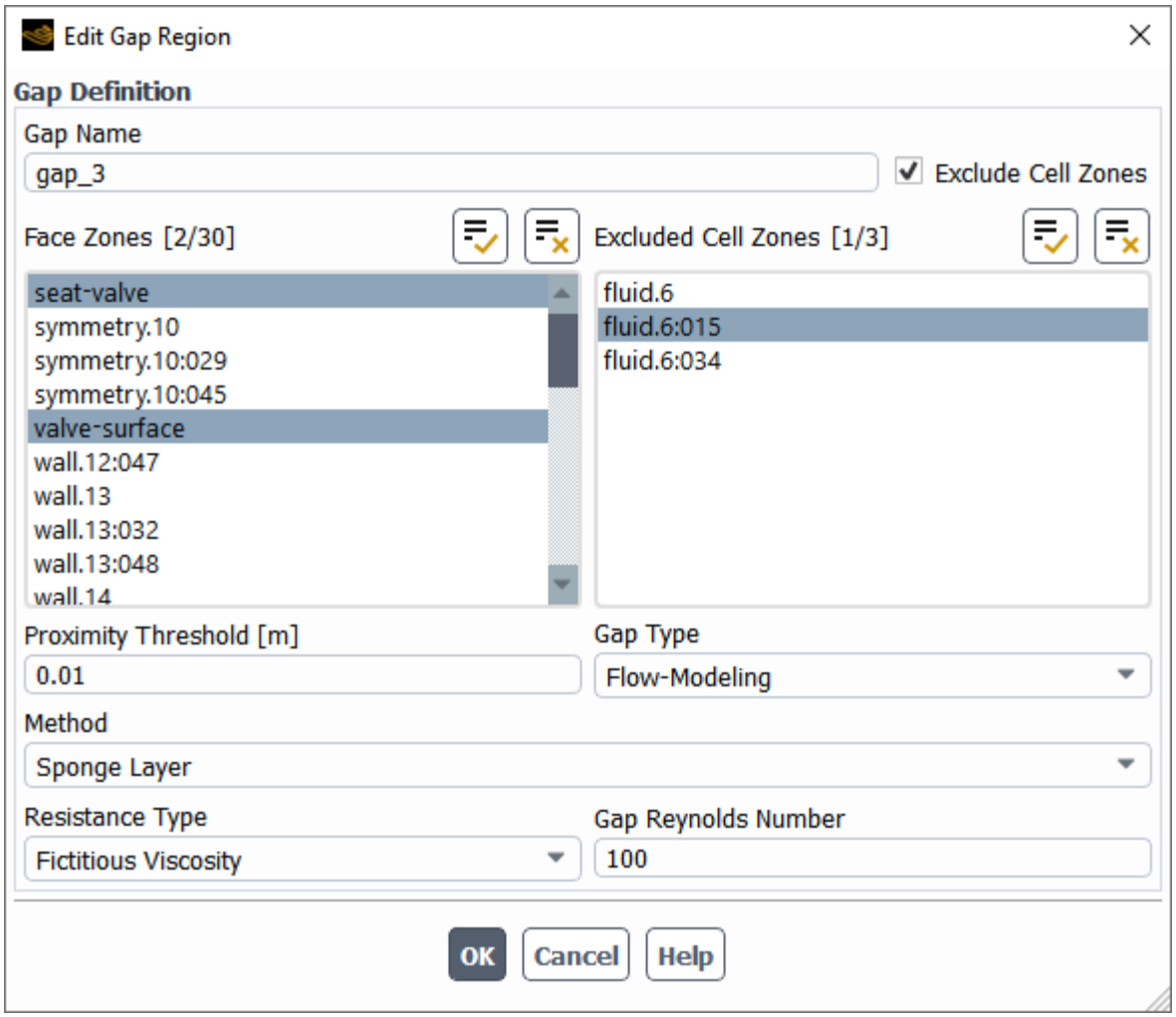
v. Click **Create**.

b. After you have created gap regions, you can revise the setup using the input controls in the **Gap Management** group box.

i. Revise a gap region by making a single selection from the **Gap Regions** list, clicking **Edit...**, and then completing the setup in the **Edit Gap Region** dialog box that opens.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.78: The Edit Gap Region Dialog Box

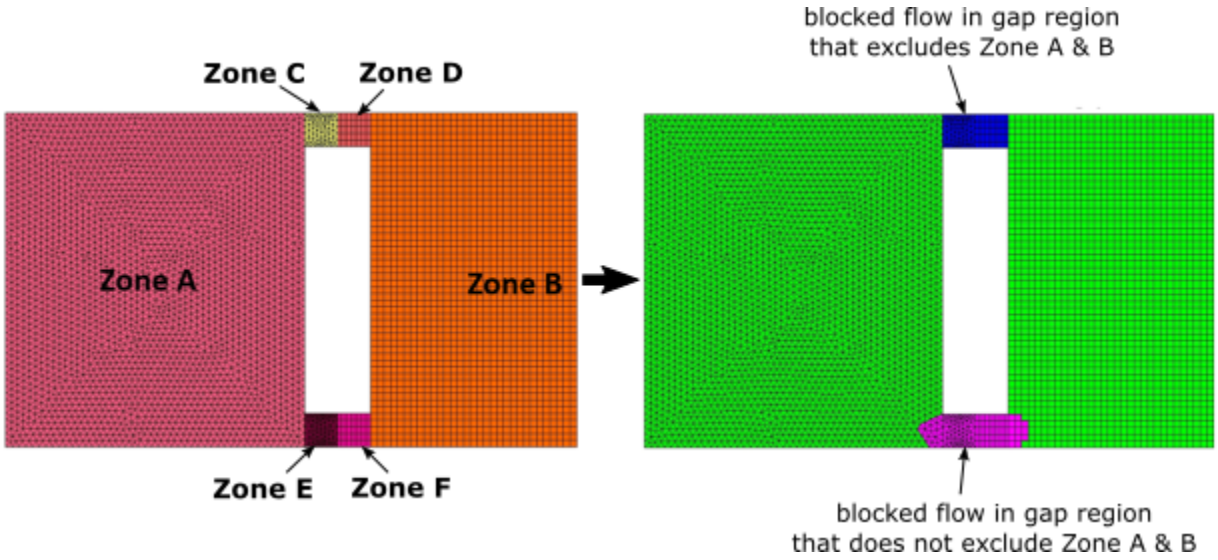


In addition to the input controls you used when initially creating the gap region, the following settings can be defined:

- You can enable the **Exclude Cell Zones** option and then make selections from the **Excluded Cell Zones** list, in order to specify that cells in particular cell zones are not marked for flow blockage / deceleration as part of the gap region. In this way you can have greater control over the shape of the gap region, as shown in the figure that follows.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.79: Controlling Gap Shape by Excluding Zones



Note:

The **Exclude Cell Zones** option is automatically disabled for simulations that use overset meshes, as this combination is not supported.

- If you selected **Flow-Modeling** for the **Gap Type**, you can select a **Method**:

  – **Sponge Layer**

  This specifies that the viscosity inside the gap regions is scaled based on the specified **Gap Reynolds Number**. As the viscosity increases, the velocity in the gap region decreases. As part of this method, you can specify the **Resistance Type**, which affects the nature of the viscosity scaling: by default, the **Fictitious Viscosity** approach is used, as it is more robust and has less impact on thermal simulations. It scales the laminar viscosity inside the gap region during discretization of the viscous forces for the momentum equation; thus, its impact is explicitly seen in the velocity and pressure computation, and it has an implicit effect on the other solution fields through the changes in the velocity and pressure field. You can instead select the **Real Viscosity** approach, which changes the actual laminar viscosity inside the gap region based on your specified **Gap Reynolds Number**; therefore, the effect of modeling will be seen not only in flow equation but also in other equations. The flow inside the gap region modeled by this approach is indeed using a modified viscosity value over the course of the entire simulation.

  – **User-Defined Source**

  This allows you to use a user-defined function (UDF) to specify the source terms for the momentum and energy equations in the gap region using the `DEFINE_GAP_MODEL_SOURCE` macro (for details, see `DEFINE_GAP_MODEL_SOURCE` in the *Fluent Customization Manual*). After you have compiled the UDF, you must then select it in the **Gap Source Terms** field. Note that interpreted UDFs are not supported.

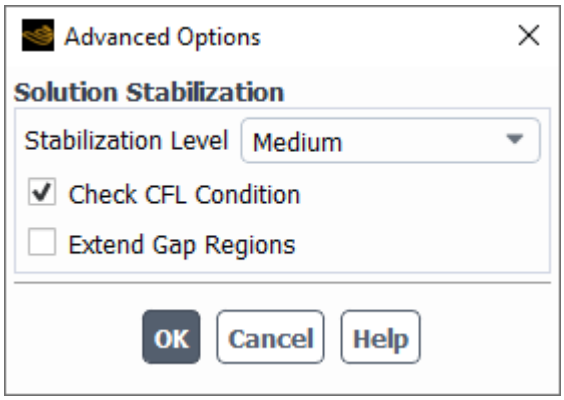
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ii. To delete gap regions, select one or more in the **Gap Regions** list and click the **Delete** button.

iii. To print a list in the console of the properties of gap regions, select one or more in the **Gap Regions** list and click the **List** button. Note that the number of cells marked for blockage / deceleration will not be updated until after you have generated data by initializing or calculating.

iv. A gap region can introduce large gradients in the flow in the vicinity of the gap region boundary. You have the option of applying solution stabilization to improve the stability of the simulation by clicking the **Advanced Options...** button and selecting a **Stabilization Level** in the dialog box that opens. For each level, the solver adjusts the numerical settings, discretization schemes, under-relaxations, AMG settings, and so on, and controls the flow behavior in the vicinity of the gap regions; note that these changes are applied globally to all gap regions, and a level other than **None** will override the local stabilization applied by default to flow-modeling regions that use the **Sponge Layer** method. You should use the lowest level as possible, as the cost of computation increases with higher levels. Information about any changes made as part of this stabilization is printed in the console when you click **OK**.

Figure 6.80: The Advanced Options Dialog Box



You can enable the **Check CFL Condition** option if you want Fluent to determine a desirable acoustic Courant (CFL) number based on your selected **Stabilization Level**, and then provide a warning message in the console if this CFL number would require you to specify a smaller time step for the calculation in order to improve the solver stability.

You can also enable the **Extend Gap Regions** if the shape of the marked cells is negatively affecting solution stability or convergence behavior. The gap regions will then be enlarged during marking by including additional cells in the vicinity of the gap interfaces.

c. When all of the gap regions are properly defined, click the **OK** button to close the **Gap Model** dialog box.

3. When you have a **Flow-Modeling** type gap region that uses the **Sponge Layer** method, by default a local stabilization is enabled to mitigate bad convergence behavior (though it can be disabled for individual gap regions using the `define/gap-model/edit` text command, or overridden generally through stabilization level set in the **Advanced Options** dialog box). You have the ability to revise the settings for all of the gap regions that use this local stabilization through the following text command:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

`define` → `gap-model` → `advanced-options` → `sponge-layer`

- You can set the viscosity variation factor, which determines how much the viscosity linearly varies within the gap region as you approach the center. The default value of `0` produces maximum variability, which is appropriate when the gap size is larger and/or the viscosity within the gap region is very high compared to the flow domain. You can enter values closer to `1` (so that the viscosity is more fixed / constant throughout) when the gap is smaller and/or the viscosity is lower. Minimizing the variability restricts the flow better, but can lead to solution instability.
- You can set minimum limit for the velocity in the sponge layers. Viscosity scaling is disabled if the average velocity inside the gap region is less than this minimum.
- You can set the maximum limit for the viscosity change ratio in the sponge layers. The local stabilization method measures the variation of maximum viscosity to minimum viscosity inside the gap region, and constrains this ratio to be bounded between 0 and the value you enter. This can help avoid numerical instability when the variation of the viscosity inside the gap region is quite substantial compared to the flow viscosity.

4. During the solution, information about the gap model is printed in the console. You can set the level of detail that is provided by using the following text command:

   `define` → `gap-model` → `advanced-options` → `verbosity`

5. Though it is not typically necessary, you can enable the following text command to make additional expert-level text commands available.

   `define` → `gap-model` → `advanced-options` → `expert?`

   The expert-level text commands include the following:

   `define/gap-model/advanced-options/enhanced-data-interpolation?`

   This can enable the use of enhanced data interpolation for updating information in gap regions. It may produce more accurate results, but with a higher computational cost.

   `define/gap-model/advanced-options/fill-data-in-gap-regions?`

   This can enable the interpolation of solution data throughout the gap regions.

   `define/gap-model/advanced-options/precise-gap-marking?`

   This can enable the use of a more accurate search algorithm for marking cells in gap regions. Note that it can be costly, particularly for 3D cases or those with a large number of cells inside the gap regions.

   `define/gap-model/advanced-options/reduce-gap-regions?`

   This can enable a more restrictive algorithm for marking cells in gap regions.

   `define/gap-model/advanced-options/render-flow-modeling-gaps?`

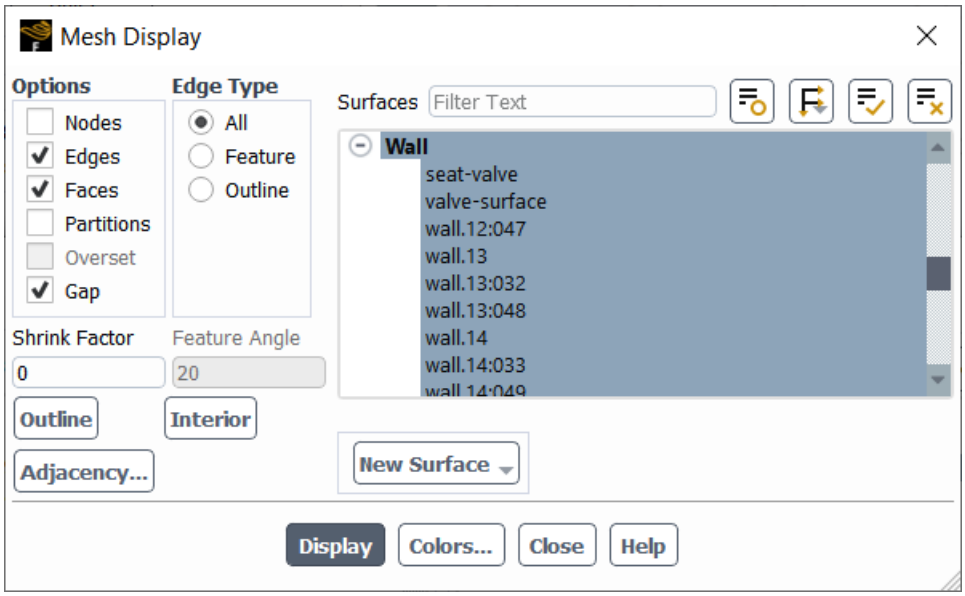
   This can disable the rendering of the solution in the cells of flow-modeling gap regions during postprocessing.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. You can use the **Mesh Display** dialog box to view the mesh. By default, the complete mesh is displayed. If you enable the **Gap** option, only the cells in which flow is allowed are displayed (that is, those that are not marked for blockage in the gap region).

   Domain → **Mesh** → **Display**

Figure 6.81: The Mesh Display Dialog Box for a Gap Model Simulation



Note:

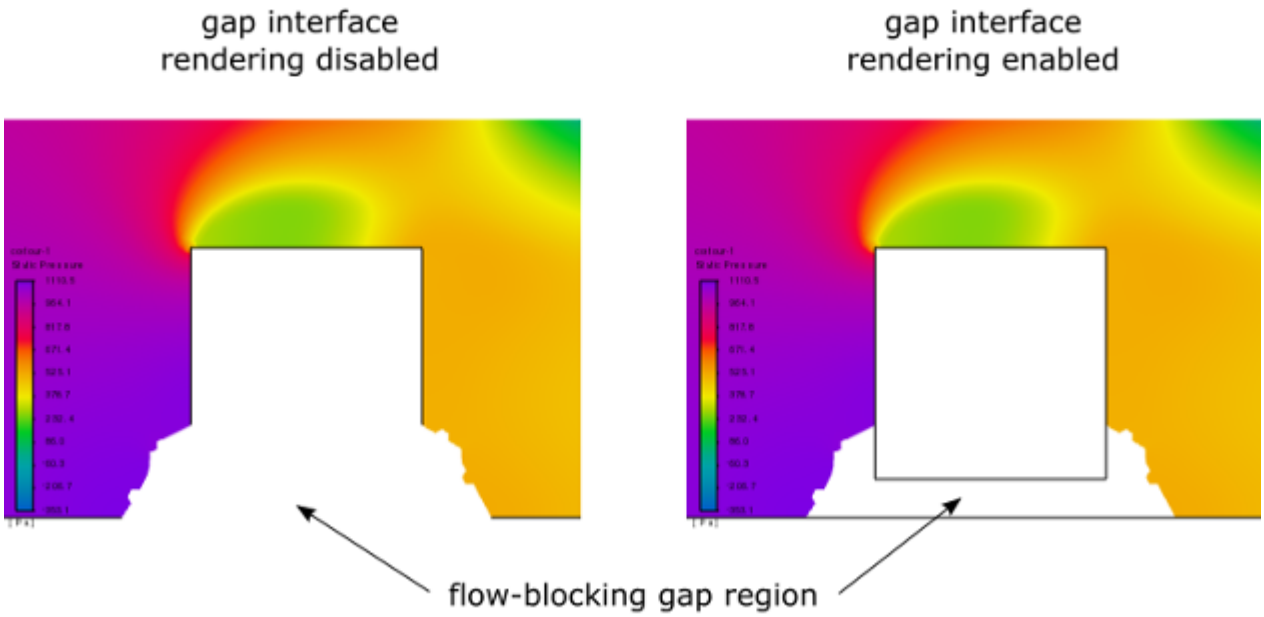
When displaying contours of postprocessing quantities, cells marked for blockage in a gap region are not shown (except for the **Gap ID** or **Gap Type** field variables), regardless of whether or not the **Gap** option is enabled in the **Mesh Display** dialog box.

When displaying the mesh with contours, you can use the following text command to allow the rendering of the mesh surfaces inside the gap regions. Note that the solution is still not rendered inside the flow-blocking gap regions.

`define` → `gap-model` → `advanced-options` → `render-gap-interface?`

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.82: Contours and Mesh with a Flow-Blocking Gap



7. When postprocessing a gap model simulation, note that the **Node Values** option is not supported. The following field variables are available (under the **Cell Info...** category), and may visualize the gap regions:

   - **Gap ID**

     This is the gap region ID, as identified using the **List** button in the Gap Model Dialog Box (p. 5484). All cells that are not marked for a gap region have a value of 0.

   - **Gap Type**

     This is the type of gap region. Cells that are marked as part of a flow-blocking gap region have a value of 1, cells that are marked as part of a flow-modeling gap region have a value of 2, and cells that are not marked for a gap region have a value of 0.

   - **Flow-Blocking Gap Interface**

     This identifies the interfaces of flow-blocking gap regions. All cells that have at least one face that applies the zero-mass-flux boundary for a gap region have a value of 1, whereas all other cells have a value of 0.

Note:

- For steady-state simulations, if you have initialized the solution and then revise a gap region definition or delete a gap region such that there is a change in the cells that are marked for blockage / deceleration, then you must initialize again before proceeding.
- It is recommended that you do not have two or more gap regions overlap (that is, mark the same cells), especially if they are not of the same **Gap Type**. If you enable the verbosity text command (`define/gap-model/advanced-options/verbosity`

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1), you will receive a warning message in the console when gaps overlap. For flow-blocking gaps, you can replace the multiple overlapping gap regions with a single gap region that includes all of the related face zones and has a proximity threshold that is suitable for all.

6.9. Checking the Mesh

The mesh checking capability in Ansys Fluent examines various aspects of the mesh, including the mesh topology, periodic boundaries, simplex counters, and (for axisymmetric cases) node position with respect to the x axis, and provides a mesh check report with details about domain extents, statistics related to cell volume and face area, and information about any problems associated with the mesh. You can check the mesh by clicking the **Check** button in the **General** task page.

Setup → **General** → **Check**

Important:

It is generally a good idea to check your mesh right after reading it into Fluent, in order to detect any mesh trouble before you get started with the problem setup.

The mesh check examines the topological information, beginning with the number of faces and nodes per cell. A triangular cell (2D) should have 3 faces and 3 nodes, a tetrahedral cell (3D) should have 4 faces and 4 nodes, a quadrilateral cell (2D) should have 4 faces and 4 nodes, and a hexahedral cell (3D) should have 6 faces and 8 nodes. Polyhedral cells (3D) will have an arbitrary number of faces and nodes.

Next, the face handedness and face node order for each zone is checked. The zones should contain all right-handed faces, and all faces should have the correct node order.

The last topological verification is checking the element-type consistency. If a mesh does not contain mixed elements (quadrilaterals and triangles or hexahedra and tetrahedra), Ansys Fluent will determine that it does not need to keep track of the element types. By doing so, it can eliminate some unnecessary work.

For axisymmetric cases, the number of nodes below the x axis is listed. Nodes below the x axis are forbidden for axisymmetric cases, since the axisymmetric cell volumes are created by rotating the 2D cell volume about the x axis; therefore nodes below the x axis would create negative volumes.

For solution domains with rotationally periodic boundaries, the minimum, maximum, average, and prescribed periodic angles are computed. A common mistake is to specify the angle incorrectly. For domains with translationally periodic boundaries, the boundary information is checked to ensure that the boundaries are truly periodic.

Finally, the simplex counters are verified. The actual numbers of nodes, faces, and cells that Fluent has constructed are compared to the values specified in the corresponding header declarations in the mesh file. Any discrepancies are reported.

6.9.1. Mesh Check Report

6.9.2. Repairing Meshes

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.9.1. Mesh Check Report

When you click the **Check** button in the **General** task page, a mesh check report will be displayed in the console. The following is a sample of a successful output :

```
 Mesh Check

 Domain Extents:
   x-coordinate: min (m) = -4.000000e-002, max (m) = 2.550000e-001
   y-coordinate: min (m) = 0.000000e+000, max (m) = 2.500000e-002
 Volume statistics:
   minimum volume (m3): 2.463287e-009
   maximum volume (m3): 4.508038e-007
    total volume (m3): 4.190433e-004
   minimum 2d volume (m3): 3.000589e-007
   maximum 2d volume (m3): 3.019523e-006
 Face area statistics:
   minimum face area (m2): 4.199967e-004
   maximum face area (m2): 2.434403e-003
 Checking mesh.....................
Done.
```

The mesh check report begins by listing the domain extents. The domain extents include the minimum and maximum x, y, and z coordinates in meters.

Then the volume statistics are provided, including the minimum, maximum, and total cell volume in m^3. A negative value for the minimum volume indicates that one or more cells have improper connectivity. Cells with a negative volume can often be identified and viewed by creating an iso-value field variable cell register (Field Variable (p. 3855)). You can try adapting the mesh to resolve the negative volume cells (Adapting the Mesh (p. 3773)). You *must* eliminate these negative volumes before continuing the flow solution process.

Next, the mesh report lists the face area statistics, including the minimum and maximum areas in m^2. A value of 0 for the minimum face area indicates that one or more cells have degenerated. As with negative volume cells, you *must* eliminate such faces. It is also recommended to correct cells that have nonzero face areas, if the values are very small.

Besides the information about the domain extents and statistics, the mesh check report will also display warnings based on the results of the checks previously described. You can specify that the mesh check report displays more detailed information about the various checks and the mesh failures, by using the following text command prior to performing the mesh check:

`mesh` → `check-verbosity`

You will then be prompted to enter the level of verbosity for the mesh check report. The possible levels include:

- `0`

  This is the default level, and only notifies you that checks are being performed (for example, **Checking mesh...**). The report will look like the previous example; while warnings will be displayed below the domain extents and statistics, the names of the individual checks will not be listed as they are conducted.

- `1`

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This level lists the individual checks as they are performed (for example, **Checking right-handed cells**). Any warnings that result will be displayed immediately below the check that produced it.

- 2

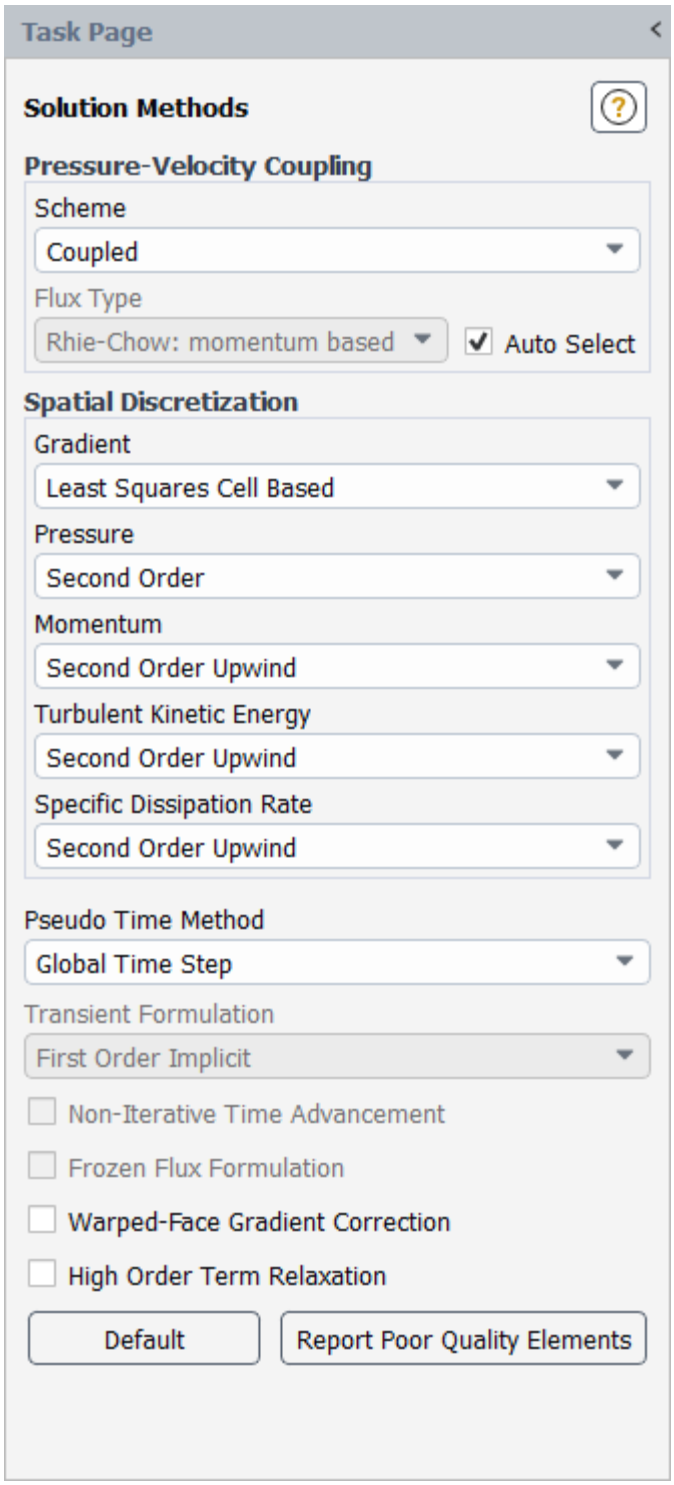
  This level provides the maximum information about the mesh check. The report will list the individual checks as they are performed (for example, **Checking right-handed cells**); any warnings that result will be displayed immediately below the check that produced it. Additional details about the check failure will also be displayed, such as the location of the problem or the affected cells.

6.9.2. Repairing Meshes

If the mesh check report indicates a mesh problem or if you receive warnings, you can investigate the extent of the problem by printing the poor element statistics in the console. This can be accomplished using the **Report Poor Quality Elements** button, located at the bottom of the **Solution Methods** task page (Figure 6.83: The Solution Methods Task Page (p. 1248)). Note that this button is only available when poor quality elements have been detected by the mesh check or have been marked for the application of poor mesh numerics (for details, see Robustness with Meshes of Poor Quality (p. 3741)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.83: The Solution Methods Task Page



Alternatively, you can print the poor element statistics via the following text command:

`mesh` → `repair-improve` → `report-poor-elements`

You can also visualize the invalid and poor elements by marking and displaying them by using the **Mesh...** and **Mark Poor Elements** selections from the **Field Value of** drop-down lists of the **Field Variable Register** dialog box (see Field Variable (p. 3855) for further details). Similarly, you can display

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

them using the **Contours** dialog box, by selecting **Mesh...** and **Mark Poor Elements** from the **Contours of** drop-down lists.

The solver variable can take a value of 1 to 6 and can be visualized:

- 1.0 user_defined_poor_elements
- 2.0 poor_quality_cells
- 3.0 modified_centroid_cells
- 4.0 cell adjacent to small_face_area
- 5.0 invalid_cells
- 6.0 solver_identified_poor_cells

By default 3.0, 4.0 and 5.0 are active. 2.0 is recommended.

The mesh check report will indicate if the mesh has problems that must be repaired, such as left-handed faces and/or faces that have the wrong node order. If repairable failures are detected by the mesh check, the **Repair** button in the **Domain** ribbon tab (**Mesh** group box) becomes available:

Domain → **Mesh** → **Repair**

Using the **Repair** button is the simplest way to attempt to correct your mesh problems. It will attempt to correct a number of problems identified by the mesh check, including cells that have:

- the wrong node order
- the wrong face handedness or that are not convex
- faces that are small or nonexistent
- very poor quality (see Mesh Quality (p. 1146) for additional details)

Note that by default, the **Repair** button will only adjust the positions of interior nodes. If you want to also modify the nodes on the boundaries of the mesh, use the following text command *before* you repair the mesh:

```
mesh → repair-improve → allow-repair-at-boundaries
```

The **Repair** button may convert degenerate cells into polyhedra, based on skewness criteria (for more information on how cells are converted, see Converting Skewed Cells to Polyhedra (p. 1258)). If you want to ensure that there are no polyhedra in the repaired mesh (for example, if you if plan on performing hanging node adaption), you must disable such conversions using the following text command *before* you repair the mesh:

```
mesh → repair-improve → include-local-polyhedra-conversion-in-repair
```

If you would like to only attempt to improve the poor quality cells, you can use the following text command:

```
mesh → repair-improve → improve-quality
```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can use the `improve-quality` text command multiple times, until the mesh is improved to your satisfaction. For greater control over the degree to which the mesh is improved, you can perform quality-based smoothing (as described in Smoothing (p. 1291)).

It should be noted that both the **Repair** button and the `improve-quality` text command can be CPU intensive, if there are a large number of poor quality cells in the mesh. If you are not as concerned about cell quality, you can attempt to fix only the left-handed faces and faces with the wrong node order. Begin by repairing the node order with the following text command:

`mesh` → `repair-improve` → `repair-face-node-order`

Because the left-handed faces may be a result of improper face node order, the previous text command may resolve both issues at the same time. Be sure to perform another mesh check after entering the `repair-face-node-order` command, to see if the mesh has been fully repaired.

If at any point the mesh check reveals that the mesh contains left-handed faces without any node order issues, you can attempt to repair the face handedness by modifying the cell centroids with the following text command:

`mesh` → `repair-improve` → `repair-face-handedness`

Once again, perform a mesh check to see if the text command was successful. The `repair-face-handedness` text command is most effective for cells with high aspect ratios.

If the mesh check report includes a warning message such as

```
WARNING: node on face thread 2 has multiple shadows.
```

it indicates the existence of duplicate shadow nodes. This error occurs only in meshes with periodic-type walls. You can repair such a mesh using the following text command:

`mesh` → `repair-improve` → `repair-periodic`

If the interface is rotationally periodic, you will be prompted for the rotation angle.

6.10. Reporting Mesh Statistics

There are several methods for reporting information about the mesh after it has been read into Ansys Fluent. You can report the amount of memory used by the current problem, the mesh size, and statistics about the mesh partitions. Zone-by-zone counts of cells and faces can also be reported.

Information about mesh statistics is provided in the following sections:

6.10.1. Mesh Size
6.10.2. Memory Usage
6.10.3. Mesh Zone Information
6.10.4. Partition Statistics

6.10.1. Mesh Size

You can print out the numbers of nodes, faces, cells, and partitions in the mesh by clicking **Info** and selecting **Size**, in the **Domain** ribbon tab (**Mesh** group box).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Domain → Mesh → Info → Size

A partition is a piece of a mesh that has been segregated for parallel processing (see Parallel Processing (p. 4293)).

A sample of the resulting output follows:

```
Mesh Size

Level     Cells       Faces       Nodes        Partitions
    0      7917       12247       4468                1

2 cell zones, 11 face zones.
```

If you are interested in how the cells and faces are divided among the different zones, click **Info** and select **Zones**, in the **Domain** ribbon tab (**Mesh** group box), as described in Mesh Zone Information (p. 1252).

You can determine the number of faces at each boundary or at each boundary type by clicking **Info** in the context menu of the selected boundary or boundary type in the Outline View tree.

Setup → Boundary Conditions → <Boundary Type> → Info

If you are using the density-based coupled explicit solver, the mesh information will be printed for each grid level. The grid levels result from creating coarse grid levels for the FAS multigrid convergence acceleration (see Full-Approximation Storage (FAS) Multigrid in the Theory Guide). A sample of the resulting output is shown below:

```
Mesh Size

Level   Cells    Faces    Nodes     Partitions
 0       7917    12247     4468              1
 1       1347     3658        0              1
 2        392     1217        0              1
 3        133      475        0              1
 4         50      197        0              1
 5         17       78        0              1
2 cell zones, 11 face zones.
```

6.10.2. Memory Usage

During an Ansys Fluent session you may want to check the amount of memory used and allocated in the present analysis. Ansys Fluent has a feature that will report the following information: the numbers of nodes, faces, cells, edges, and object pointers (generic pointers for various mesh and graphics utilities) that are used and allocated; the amount of array memory (scratch memory used for surfaces) used and allocated; and the amount of memory used by the solver process.

You can obtain this information by clicking **Info** and selecting **Memory Usage**, in the **Domain** ribbon tab (**Mesh** group box).

Domain → Mesh → Info → Memory Usage

The memory information will be different for Linux and Windows systems.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.10.2.1. Linux Systems

On Linux systems, note the following definitions related to process memory information:

- Process static memory is essentially the size of the code itself.
- Process dynamic memory is the allocated heap memory used to store the mesh and solution variables.
- Process total memory is the sum of static and dynamic memory.

6.10.2.2. Windows Systems

On Windows systems, note the following definitions related to process memory information:

- Process physical memory is the allocated heap memory currently resident in RAM.
- Process virtual memory is the allocated heap memory currently swapped to the Windows system page file.
- Process total memory is the sum of physical and virtual memory.

Note the following:

- The memory information does not include the static (code) memory.
- Cortex runs in its own process, so the heap memory value includes storage for the mesh and solution variables only.

On Windows systems, you can also get more information on the Ansys Fluent process (or processes) by using the Task Manager (see your Windows documentation for details). Examples of process image names are as follows: `cx2420.exe` (Cortex), `fl2420.exe` (solver host), and `fl_mpi2420.exe` (solver node).

6.10.3. Mesh Zone Information

You can print information in the console about the nodes, faces, and cells in each zone the by clicking **Info** and selecting **Zones**, in the **Domain** ribbon tab (**Mesh** group box).

Domain → **Mesh** → **Info** → **Zones**

The mesh zone information includes the total number of nodes and, for each face and cell zone, the number of faces or cells, the cell (and, in 3D, face) type (triangular, quadrilateral, and so on), the boundary condition type, and the zone ID. Sample output is shown below:

```
Zone sizes on domain 1:
21280 hexahedral cells, zone 4.
  532 quadrilateral velocity-inlet faces, zone 1.
  532 quadrilateral pressure-outlet faces, zone 2.
  1040 quadrilateral symmetry faces, zone 3.
  1040 quadrilateral symmetry faces, zone 7.
 61708 quadrilateral interior faces, zone 5.
   1120 quadrilateral wall faces, zone 6.
23493 nodes.
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.10.4. Partition Statistics

You can print mesh partition statistics in the console by clicking **Info** and selecting **Partitions**, in the **Domain** ribbon tab (**Mesh** group box).

Domain → Mesh → Info → Partitions

The statistics include the numbers of cells, faces, interfaces, and neighbors of each partition. See Interpreting Partition Statistics (p. 4339) for further details, including sample output.

6.11. Converting the Mesh to a Polyhedral Mesh

Since the Ansys Fluent solver is face based, it supports polyhedral cells. The advantages that polyhedral meshes have shown over some of the tetrahedral or hybrid meshes is the lower overall cell count, almost 3-5 times lower for unstructured meshes than the original cell count. Currently, there are three options in Ansys Fluent that allow you to convert your non-polyhedral cells to a polyhedra:

- Converting the entire domain into polyhedral cells (applicable only for meshes that contain tetrahedral and/or wedge cells).
- Converting skewed tetrahedral cells to polyhedral cells.
- Converting cells with hanging nodes / edges to polyhedral cells.

Information about polyhedral mesh conversion is provided in the following sections:

6.11.1. Converting the Domain to a Polyhedra

6.11.2. Converting Skewed Cells to Polyhedra

6.11.3. Converting Cells with Hanging Nodes / Edges to Polyhedra

6.11.1. Converting the Domain to a Polyhedra

Conversion of a mesh to polyhedra only applies to 3D meshes that contain tetrahedral and/or wedge cells.

To begin the conversion process, Ansys Fluent automatically decomposes each non-hexahedral cell into multiple sub-volumes called “duals” (the shaded regions seen in the 2D example in Figure 6.84: Connection of Edge Centroids with Face Centroids (p. 1254)). Each dual is associated with one of the original nodes of the cell. These duals are then agglomerated into polyhedral cells around the original nodes. Therefore, the collection of duals from all cells sharing a particular node makes up each polyhedral cell (see Figure 6.85: A Polyhedral Cell (p. 1255)). The node that is now within the polyhedral cell is no longer needed and is removed.

To better understand how duals are formed, you can consider the straightforward case of a tetrahedral mesh. Each of the cells are decomposed in the following manner: first, new edges are created on each face between the face centroid and the centroids of the edges of that face. Then, new faces are created within the cell by connecting the cell centroid to the new edges on each face. These interior faces establish the boundaries between the duals of a cell, and divide the cell into 4 sub-volumes.

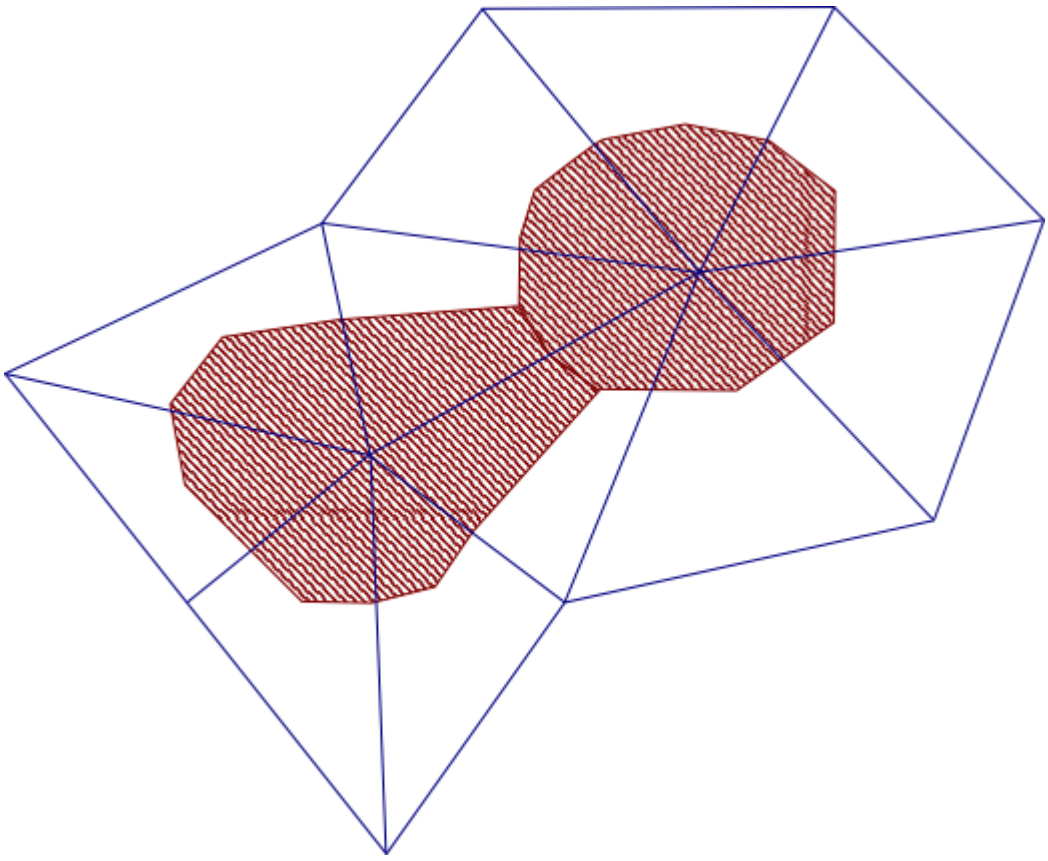
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These dividing faces may be adjusted and merged with neighboring faces during the agglomeration process, in order to minimize the number of faces on the resultant polyhedral cell.

Note:

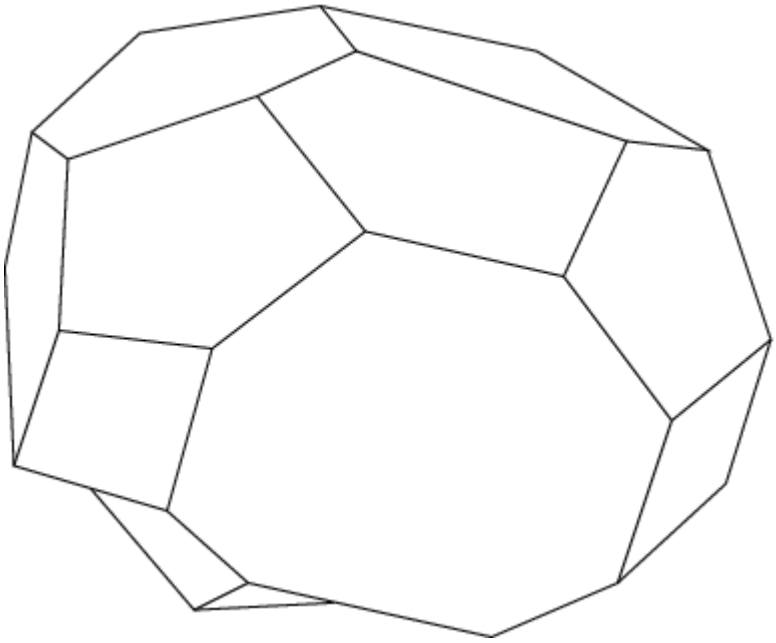
By default, Ansys Fluent checks the aspect ratio of boundary layer cells and if the aspect ratios are high, Ansys Fluent asks whether you want to preserve cells at the boundary layer (which could be useful for keeping cell counts low when retaining boundary features). You can override this default with the text command: `mesh/polyhedra/options/preserve-boundary-layer?` (for additional information about these options, see `mesh/` in the *Fluent Text Command List*).

Figure 6.84: Connection of Edge Centroids with Face Centroids



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

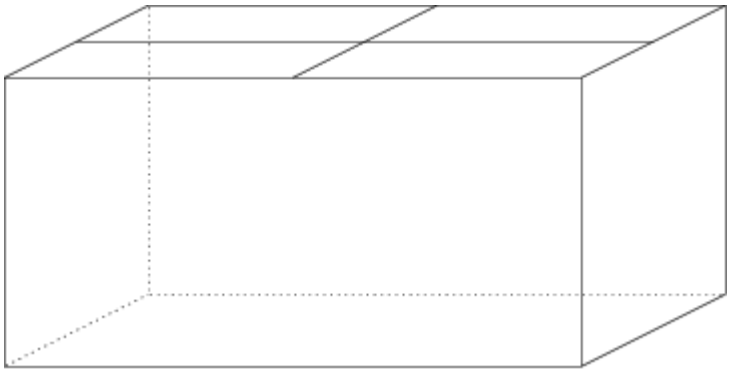
Figure 6.85: A Polyhedral Cell



Important:

Hexahedral cells are not converted to polyhedra when the domain is converted, except when they border non-hexahedral cells. When the neighboring cell is reconfigured as polyhedra, the shared face of the hexahedral cell is decomposed into multiple faces as well, resulting in a polyhedral cell. In such a case the shape of the original hexahedral cell is preserved (that is the overall dimensions of the cell stay the same), but the converted cell has more than the original 6 faces (see Figure 6.86: A Converted Polyhedral Cell with Preserved Hexahedral Cell Shape (p. 1255)).

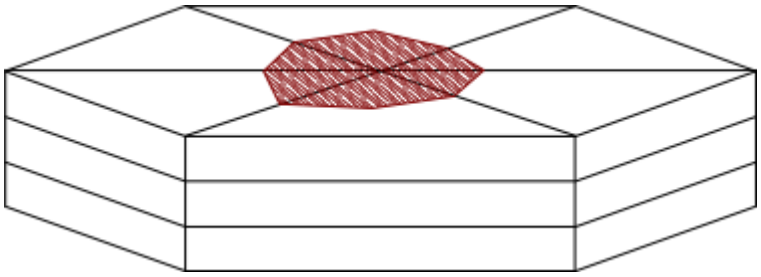
Figure 6.86: A Converted Polyhedral Cell with Preserved Hexahedral Cell Shape



Conversion proceeds in a slightly different manner in boundary layers that are modeled using thin wedge cells. These cells are decomposed in the plane of the boundary surface, but not in the direction normal to the surface. The resulting polyhedra will therefore preserve the thickness of the original wedge cells (Figure 6.87: Treatment of Wedge Boundary Layers (p. 1256)). In most cases, the cell count in the new polyhedral boundary layer will be lower than the original boundary layer.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.87: Treatment of Wedge Boundary Layers



To convert the entire domain of your mesh, click **Make Polyhedra** in the **Domain** ribbon tab (**Mesh** group box).

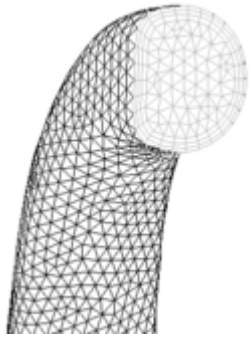
Domain → Mesh → Make Polyhedra

The following is an example of the resulting message printed in the console:

```
 Setup conversion to polyhedra.
 Converting domain to polyhedra...

      Creating polyhedra zones.
      Processing face zones..........
      Processing cell zones...
      Building polyhedra mesh..................
      Optimizing polyhedra mesh.....
>> Reordering domain using Reverse Cuthill-McKee method:
    zones, cells, faces, done.
 Bandwidth reduction = 1796/247 = 7.27
 Done.
```

Figure 6.88: The Original Tetrahedral Mesh (p. 1256), the original tetrahedral mesh of a section of a manifold, is compared to Figure 6.89: The Converted Polyhedral Mesh (p. 1257), which is the resulting mesh after the entire domain is converted to a polyhedra.

Figure 6.88: The Original Tetrahedral Mesh



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.89: The Converted Polyhedral Mesh

Note that by default, the surfaces (that is, manifold zones of type **interior**) will be lost during the conversion to polyhedra. If you would like to preserve any of these zones (in order to utilize them for postprocessing, for example), use the following text command prior to the conversion:

`mesh → polyhedra → options → preserve-interior-zones`

You will be prompted to enter a string of characters, and only those interior surfaces with a name that includes the string you specify will be preserved.

In parallel, a new partitioning will be stored after the conversion, but by default these new partitions will not be made active (that is, migrated to the compute-nodes) as this requires significant additional memory. The new partitions will be saved with the case file and used automatically when it is opened in a new Fluent session; it is recommended that you then manually reorder the domain prior to calculating (using the `mesh/reorder/reorder-domain` text command). If you want to use the new partitions in the current Fluent session, you should first ensure that no more than half of the available memory of this system is currently used, and then you can manually migrate the partitions (using the `parallel/partition/use-stored-partitions` text command) and reorder. If you know prior to conversion that you will want to calculate a solution in the same session (and you have sufficient memory), you can enable the following text command, so that the migration and reordering is performed automatically as part of the conversion:

`mesh → polyhedra → options → migrate-and-reorder?`

6.11.1.1. Limitations

Some limitations you will find with polyhedral meshes that you generally do not experience with other cell types include:

- Meshes that already contain polyhedral cells cannot be converted.
- Meshes with hanging nodes / edges will not be converted. This includes meshes that have undergone hanging node adaption (see Hanging Node Adaption in the Theory Guide), as well as CutCell meshes and hexcore meshes (generated when using the **Hex Core** meshing scheme in GAMBIT or the **Hexcore** menu option in the meshing mode of Fluent) which you have read after disabling the automatic conversion of hanging nodes / edges (see Converting Cells with Hanging Nodes / Edges to Polyhedra (p. 1259) for details).
- The following mesh manipulation tools are not available for polyhedral meshes:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - the `mesh/modify-zones/extrude-face-zone-delta` text command
  - the `mesh/modify-zones/extrude-face-zone-para` text command
  - skewness smoothing
  - swapping

- Meshes in which the domain has been converted to polyhedral cells are not eligible for adaption with the hanging node method, though they can be refined with the default polyhedral unstructured mesh adaption (PUMA) method. For more information about adaption, see Adapting the Mesh (p. 3773).

- A mesh that is composed entirely of polyhedral cells has limitations when used in a dynamic mesh problem: it cannot undergo remeshing. Smoothing is allowed for polyhedral cells, and radial basis function smoothing is the recommended method (see Radial Basis Function Smoothing (p. 1830) for details). The linearly elastic solid smoothing method is not compatible with polyhedral cells.

- Since polyhedral meshes can have a much higher ratio of nodes to cells then tet or hex meshes, the use of the node-based gradient method may result in a significant increase in memory consumption compared with other gradient methods.

6.11.2. Converting Skewed Cells to Polyhedra

Another method of cell agglomeration is the skewness-based cluster approach. This type of conversion is designed to convert only part of the domain. The objective is to convert only skewed tetrahedral cells above a specified cell equivolume skewness threshold into polyhedra. By converting the highly skewed tetrahedral cells, the quality of the mesh can be improved significantly.

A different algorithm is used for local conversion. This algorithm evaluates each highly skewed tetrahedral cell and all of the surrounding cells, to select an edge on the highly skewed cell that best matches criteria for cell agglomeration. Then all of the cells that share this edge are combined into a polyhedral cell. During the process, the data is interpolated from the original cells to the resultant polyhedra.

6.11.2.1. Limitations

There are certain limitations with this type of conversion:

- The following mesh manipulation tools are not available on polyhedral meshes:
  - the `mesh/modify-zones/extrude-face-zone-delta` text command
  - the `mesh/modify-zones/extrude-face-zone-para` text command
  - skewness smoothing
  - swapping will not affect polyhedral cells

- The polyhedral cells that result from the conversion are not eligible for adaption with the hanging node method, though they can be refined with the default polyhedral unstructured mesh adaption (PUMA) method. For more information about adaption, see Adapting the Mesh (p. 3773).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Only tetrahedral cells are converted, as all other cells are skipped.
- Meshes with hanging nodes / edges will not be converted. This includes meshes that have undergone hanging node adaption (see Hanging Node Adaption in the Theory Guide), as well as CutCell meshes and hexcore meshes (generated when using the **Hex Core** meshing scheme in GAMBIT or the **Hexcore** menu option in the meshing mode of Fluent) which you have read after disabling the automatic conversion of hanging nodes / edges (see Converting Cells with Hanging Nodes / Edges to Polyhedra (p. 1259) for details). Note that if the mesh is a CutCell / hexcore mesh in which the transitional cells have been converted to polyhedra, then it does not have hanging nodes / edges and can therefore be converted.
- Note the following with regard to the polyhedral cells that result from this conversion and the update methods available for dynamic mesh problems:
  - None of the remeshing methods will modify polyhedral cells.
  - Smoothing is allowed for polyhedral cells, and radial basis function smoothing is the recommended method (see Radial Basis Function Smoothing (p. 1830) for details). The linearly elastic solid smoothing method is not compatible with polyhedral cells.

6.11.3. Converting Cells with Hanging Nodes / Edges to Polyhedra

3D meshes that contain hanging nodes or hanging edges are by default converted when the file is read, so that the cells adjacent to the hanging nodes / edges become polyhedra. Each cell that is converted will retain the same overall dimensions, but the number of faces associated with the cell may increase (see Figure 6.86: A Converted Polyhedral Cell with Preserved Hexahedral Cell Shape (p. 1255) for an example). Such a conversion may be helpful for CutCell meshes, as well as hexcore meshes (generated when using the **Hex Core** meshing scheme in GAMBIT or the **Hexcore** menu option in the meshing mode of Fluent), and may prevent errors associated with interior walls (see Removing Hanging Nodes / Edges (p. 1155)). You can opt to disable this conversion by using the following text command prior to reading: `file/convert-hanging-nodes-during-read? no`.

Ansys Fluent also provides a text command that allows you to convert 3D cells that have hanging nodes / edges into polyhedra after reading. For example, you may want to convert the cells with hanging nodes in a mesh that has undergone hanging node adaption (see Adapting the Mesh (p. 3773)); note that you want to be sure that no further refinement / coarsening of the mesh will be necessary. To convert the cells with hanging nodes / edges after reading, use the following text command:

`mesh → polyhedra → convert-hanging-nodes`

6.11.3.1. Limitations

There are certain limitations with this type of conversion:

- The following mesh manipulation tools are not available on polyhedral meshes:
  - the `mesh/modify-zones/extrude-face-zone-delta` text command
  - the `mesh/modify-zones/extrude-face-zone-para` text command
  - skewness smoothing
  - swapping will not affect polyhedral cells

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The polyhedral cells that result from the conversion are not eligible for adaption with the hanging node method, though they can be refined with the default polyhedral unstructured mesh adaption (PUMA) method. For more information about adaption, see Adapting the Mesh (p. 3773).
- Note the following with regard to the polyhedral cells that result from this conversion and the update methods available for dynamic mesh problems:
  - None of the remeshing methods will modify polyhedral cells.
  - Smoothing is allowed for polyhedral cells, and radial basis function smoothing is the recommended method (see Radial Basis Function Smoothing (p. 1830) for details). The linearly elastic solid smoothing method is not compatible with polyhedral cells.

6.12. Modifying the Mesh

There are several ways in which you can modify or manipulate the mesh after it has been read into Ansys Fluent. You can scale, translate, or rotate the mesh / zones, copy, merge, or separate zones, create or decouple periodic zones, fuse boundaries, and smooth and swap faces. Methods for partitioning meshes to be used in a parallel solver are discussed in Mesh Partitioning and Load Balancing (p. 4315).

Important:

Whenever you modify the mesh, you should be sure to save a new case file (and a data file, if data exists). If you have old data files that you would like to be able to read in again, be sure to retain the original case file as well, as the data in the old data files may not correspond to the new case file.

Information about mesh manipulation is provided in the following sections:

6.12.1. Merging Zones
6.12.2. Separating Zones
6.12.3. Fusing Face Zones
6.12.4. Creating Periodic Zones and Interfaces
6.12.5. Decoupling Periodic Zones
6.12.6. Slitting Face Zones
6.12.7. Orienting Face Zones
6.12.8. Extruding Face Zones
6.12.9. Replacing, Deleting, Deactivating, and Activating Zones
6.12.10. Copying Cell Zones
6.12.11. Replacing the Mesh
6.12.12. Managing Adjacent Zones
6.12.13. Reordering the Domain
6.12.14. Scaling the Mesh
6.12.15. Translating the Mesh
6.12.16. Rotating the Mesh
6.12.17. Improving the Mesh by Smoothing and Swapping

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.12.18. Boundary Layer Redistribution
6.12.19. Deleting Cells

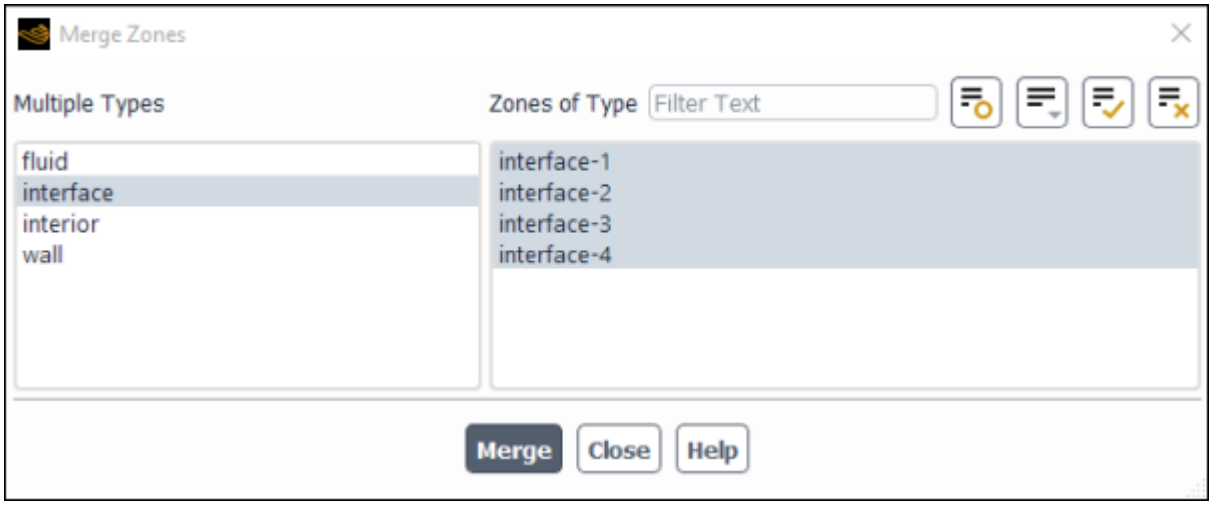
6.12.1. Merging Zones

To simplify the solution process, you may want to merge zones. Merging zones involves combining multiple zones of similar type into a single zone. Setting boundary conditions and postprocessing may be easier after you have merged similar zones.

Zone merging is performed in the Merge Zones Dialog Box (p. 5525) (Figure 6.90: The Merge Zones Dialog Box (p. 1261)).

Domain → Zones → Combine → Merge...

Figure 6.90: The Merge Zones Dialog Box



6.12.1.1. When to Merge Zones

Ansys Fluent allows you to merge zones of similar type into a single zone. This is not necessary unless the number of zones becomes prohibitive to efficient setup or postprocessing of the numerical analysis. For example, setting the same boundary condition parameters for a large number of zones can be time-consuming and may introduce inconsistencies. In addition, the postprocessing of the data often involves surfaces generated using the zones. A large number of zones often translates into a large number of surfaces that must be selected for the various display options, such as color contouring. Fortunately, surfaces can also be merged (see Grouping, Prioritizing, Editing, Renaming, and Deleting Surfaces (p. 3847)), minimizing the negative impact of a large number of zones on postprocessing efficiency.

Although merging zones can be helpful, there may be cases where you will want to retain a larger number of zones. Since the merging process is not fully reversible, a larger number of zones provides more flexibility in imposing boundary conditions. Although a large number of zones can make selection of surfaces for display tedious, it can also provide more choices for rendering the mesh and the flow-field solution. For instance, it can be difficult to render an internal flow-field solution. If the outer domain is composed of several zones, the meshes of subsets of these zones can be plotted along with the solution to provide the relationship between the geometry and solution field. Merging zones may also adversely affect dynamic zones and mesh interfaces.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.12.1.2. Using the Merge Zones Dialog Box

The procedure for merging multiple zones of the same type into a single zone is as follows:

1. Select the zone type in the **Multiple Types** list. This list contains all the zone types for which there are multiple zones. When you choose a type from this list, the corresponding zones will appear in the **Zones of Type** list.
2. Select two or more zones in the **Zones of Type** list.
3. Click the **Merge** button to merge the selected zones. Note that if your case file has dynamic zones or mesh interfaces, the **Warning** dialog box will open before the merge is initiated, allowing you to specify whether you want to delete such zones or interfaces first (see Warning Dialog Box (p. 5662) for details).

Important:

Remember to save a new case file (and a data file, if data exists).

6.12.2. Separating Zones

Upon reading a mesh file, Ansys Fluent automatically performs zone separations in two conditions. If a face zone is attached to multiple cells zones in the preprocessor, the face zone will be separated so that each one is attached to only one cell zone. Furthermore, if you have defined an internal face as a wall type, an additional shadow wall zone will be generated (for example, for a wall named **baffle**, a shadow wall zone named **baffle-shadow** will be generated).

There are several methods available in Ansys Fluent that allow you to manually separate a single face or cell zone into multiple zones of the same type. If your mesh contains a zone that you want to break up into smaller portions, you can make use of these features. For example, if you created a single wall zone when generating the mesh for a duct, but you want to specify different temperatures on specific portions of the wall, you will need to break that wall zone into two or more wall zones. If you plan to solve a problem using the sliding mesh model or multiple reference frames, but you forgot to create different fluid zones for the regions moving at different speeds, you will need to separate the fluid zone into two or more fluid zones.

Important:

- After performing any of these separations, you should save a new case file. If data exists, it is automatically assigned to the proper zones when separation occurs, so you should also write a new data file. The old data cannot be read on top of the case file in which the zones have changed.
- The maximum number of zones into which you can separate any one face zone or cell zone is 32.

There are four ways to separate face zones and two ways to separate cell zones. The face separation methods will be described first, followed by the cell separation tools. Decoupling of periodic zones is discussed in Decoupling Periodic Zones (p. 1272).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that all of the separation methods allow you to report the result of the separation before you commit to performing it.

6.12.2.1. Separating Face Zones

For more information, see the following sections:

6.12.2.1.1. Methods for Separating Face Zones

6.12.2.1.2. Inputs for Separating Face Zones

6.12.2.1.1. Methods for Separating Face Zones

For geometries with sharp corners, it is often easy to separate face zones based on significant angle. Faces with normal vectors that differ by an angle greater than or equal to the specified significant angle will be placed in different zones. For example, if your mesh consists of a cube, and all 6 sides of the cube are in a single wall zone, you would specify a significant angle of 89°. Since the normal vector for each cube side differs by 90° from the normals of its adjacent sides, each of the 6 sides will be placed in a different wall zone.

If you have a small face zone and would like to put each face in the zone into its own zone, you can do so by separating the faces based on face. Each individual face (triangle, quad, or polygon) will be separated into different zones.

You can also separate face zones based on cell registers. For example, you can mark cells based on their location in the domain (region register), their boundary closeness (boundary register), field values of some variable, or any of the cell marking methods discussed in Using Cell Registers (p. 3851). When you specify which register is to be used for the separation of the face zone, all faces of cells that are marked will be placed into a new face zone.

Finally, you can separate face zones based on contiguous regions. For example, when you use coupled wall boundary conditions you need the faces on the zone to have a consistent orientation. Consistent orientation can only be guaranteed on contiguous regions, so you may need to separate face zones to allow proper boundary condition specification.

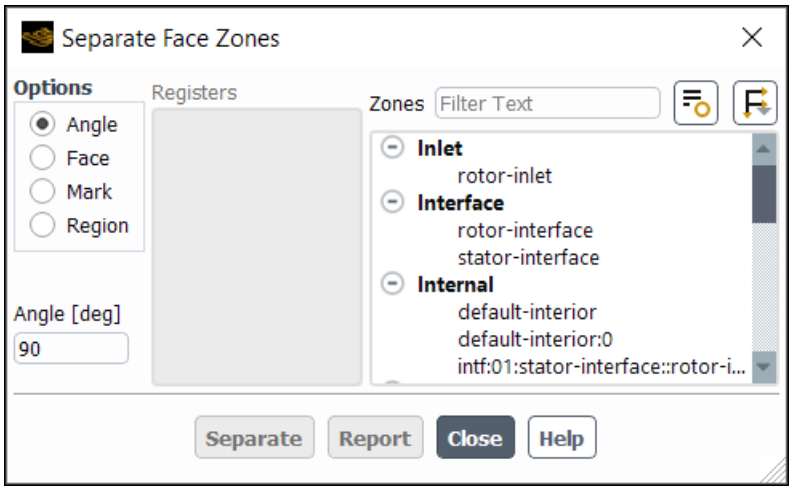
6.12.2.1.2. Inputs for Separating Face Zones

To break up a face zone based on angle, face, cell register mark, or region, use the Separate Face Zones Dialog Box (p. 5606) (Figure 6.91: The Separate Face Zones Dialog Box (p. 1264)).

Domain → **Zones** → **Separate** → **Faces...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.91: The Separate Face Zones Dialog Box



Important:

If you are planning to separate face zones, you should do so before performing any adaptions using the hanging node adaption method. Face zones that contain hanging nodes cannot be separated.

The steps for separating faces are as follows:

1. Select the separation method (**Angle**, **Face**, **Mark**, or **Region**) under **Options**.
2. Specify the face zone to be separated in the **Zones** list.
3. If you are separating by face or region, skip to the next step. Otherwise, do one of the following:
   - If you are separating faces by angle, specify the significant angle in the **Angle** field.
   - If you are separating faces by mark, select the cell register to be used in the **Registers** list.
4. (optional) To check what the result of the separation will be before you actually separate the face zone, click the **Report** button. The report will look like the following example:

```
45 faces in contiguous region 0
30 faces in contiguous region 1
11 faces in contiguous region 2
14 faces in contiguous region 3
Separates zone 4 into 4 zone(s).
```

5. To separate the face zone, click the **Separate** button. A report will be printed in the console like the following example:

```
45 faces in contiguous region 0
30 faces in contiguous region 1
11 faces in contiguous region 2
14 faces in contiguous region 3
Separates zone 4 into 4 zone(s).
Updating new zone information ...
   created new zone wall-4:001 from wall-4
   created new zone wall-4:002 from wall-4
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
   created new zone wall-4:010 from wall-4
done.
```

Important:

When you separate the face zone by cell register mark, you may sometimes find that a face of a corner cell will be placed in the wrong face zone. You can usually correct this problem by performing an additional separation, based on angle, to move the offending face to a new zone. You can then merge this new zone with the zone in which you want the face to be placed, as described in Merging Zones (p. 1261).

6.12.2.2. Separating Cell Zones

For more information, see the following sections:

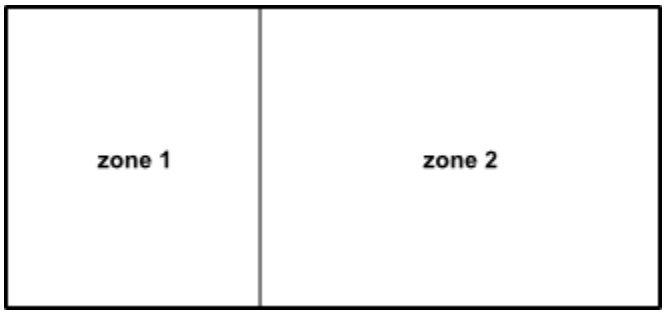
6.12.2.2.1. Methods for Separating Cell Zones

6.12.2.2.2. Inputs for Separating Cell Zones

6.12.2.2.1. Methods for Separating Cell Zones

If you have two or more enclosed cell regions sharing internal boundaries (as shown in Figure 6.92: Cell Zone Separation Based on Region (p. 1265)), but all of the cells are contained in a single cell zone, you can separate the cells into distinct zones using the separation-by-region method. Note that if the shared internal boundary is of type **interior**, you must change it to another double-sided face zone type (**fan**, **radiator**, and so on) prior to performing the separation.

Figure 6.92: Cell Zone Separation Based on Region



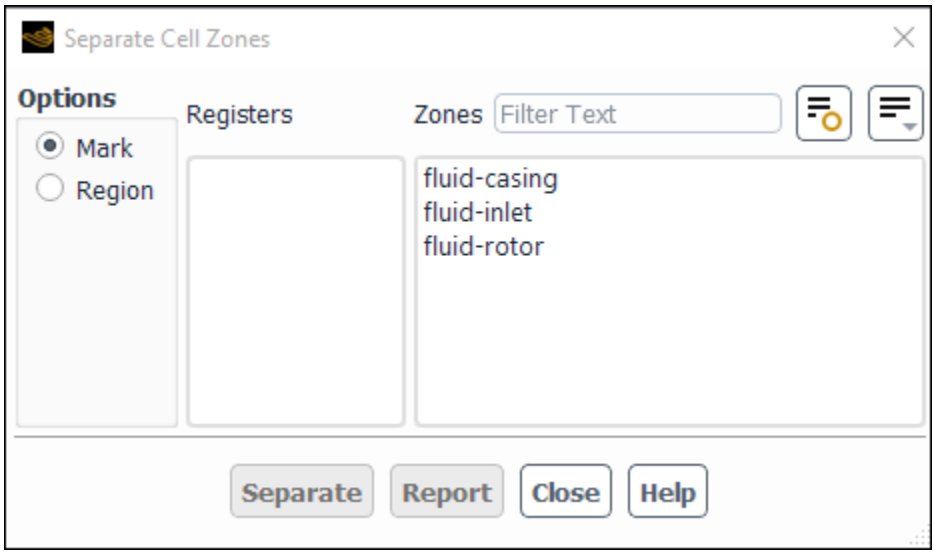
You can also separate cell zones based on the marks stored in cell registers. You can mark cells using any of the methods discussed in Using Cell Registers (p. 3851) (for example, you can mark cells with a certain field value range or cells inside or outside a specified region). When you specify which register is to be used for the separation of the cell zone, cells that are marked will be placed into a new cell zone.

6.12.2.2.2. Inputs for Separating Cell Zones

To break up a cell zone based on region or cell register mark, use the Separate Cell Zones Dialog Box (p. 5605) (Figure 6.93: The Separate Cell Zones Dialog Box (p. 1266)).

Domain → Zones → Separate → Cells...

Figure 6.93: The Separate Cell Zones Dialog Box



Important:

If you are planning to separate cell zones, you should do so before performing any adaptions using the hanging node adaption method. Cell zones that contain hanging nodes cannot be separated.

The steps for separating cells are as follows:

1. Select the separation method (**Mark** or **Region**) under **Options**.
2. Specify the cell zone to be separated in the **Zones** list.
3. If you are separating cells by mark, select the cell register to be used in the **Registers** list.
4. (optional) To check what the result of the separation will be before you actually separate the cell zone, click the **Report** button. The report will look like this:

```
Separates zone 14 into two zones, with 1275 and 32 cells.
```

5. To separate the cell zone, click the **Separate** button. Ansys Fluent will print the following information:

```
Separates zone 14 into two zones, with 1275 and 32 cells.
No faces marked on thread, 2
No faces marked on thread, 3
No faces marked on thread, 1
No faces marked on thread, 5
No faces marked on thread, 7
No faces marked on thread, 8
No faces marked on thread, 9
No faces marked on thread, 61
Separates zone 62 into two zones, with 1763 and 58 faces.
All faces marked on thread, 4
No faces marked on thread, 66
Moved 32 cells from cell zone 14 to zone 10
Updating new zone information ...
   created new zone interior-4:010 from interior-4
   created new zone interior-6:009 from interior-6
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
   created new zone fluid-14:008 from fluid-14
done.
```

As shown in the example above, separation of a cell zone will often result in the separation of face zones as well. When you separate by mark, faces of cells that are moved to a new zone will be placed in a new face zone. When you separate by region, faces of cells that are moved to a new zone will not necessarily be placed in a new face zone.

If you find that any faces are placed incorrectly, see the comment above, at the end of the inputs for face zone separation.

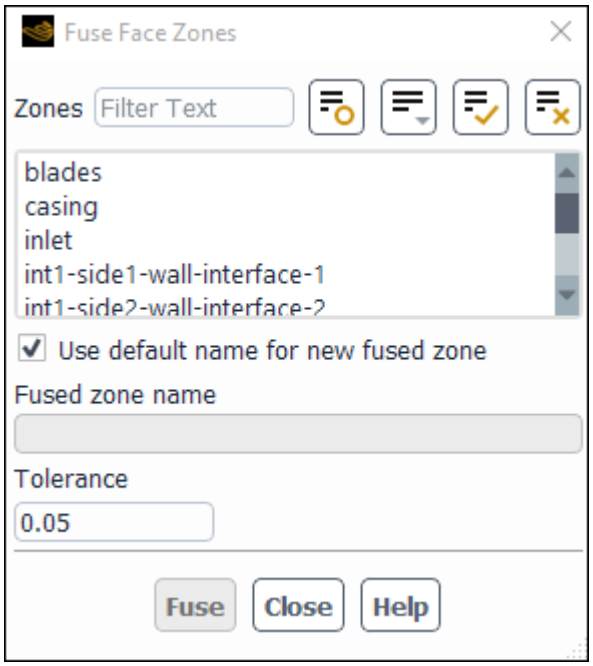
6.12.3. Fusing Face Zones

The face-fusing utility is a convenient feature that can be used to fuse boundaries (and merge duplicate nodes and faces) created by assembling multiple mesh regions. When the domain is divided into subdomains and the mesh is generated separately for each subdomain, you will combine the subdomains into a single file before reading the mesh into Fluent. For details, see Reading Multiple Mesh/Case/Data Files (p. 1156). This situation could arise if you generate each block of a multiblock mesh separately and save it to a separate mesh file. Another possible scenario is that you decided, during mesh generation, to save the mesh for each part of a complicated geometry as a separate part file. (Note that the mesh node locations need not be identical at the boundaries where two subdomains meet; see Non-Conformal Meshes (p. 1170) for details.)

The **Fuse Face Zones** dialog box (Figure 6.94: The Fuse Face Zones Dialog Box (p. 1267)) allows you to merge the duplicate nodes and delete these artificial internal boundaries.

Domain → Zones → Combine → Fuse...

Figure 6.94: The Fuse Face Zones Dialog Box



The boundaries on which the duplicate nodes lie are assigned zone ID numbers (just like any other boundary) when the mesh files are combined, as described in Reading Multiple Mesh/Case/Data

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Files (p. 1156). You need to keep track of the zone ID numbers when `tmerge` or the meshing mode of Fluent reports its progress or, after the complete mesh is read in, display all boundary mesh zones and use the mouse-probe button to determine the zone names (see Controlling the Mouse Button Functions (p. 4020) for information about the mouse button functions). Alternatively, you can specify a name for the fused zone.

Note:

For adapted meshes, fusing of face zones is only possible in serial with the PUMA adaption method (which is the default method for 3D).

6.12.3.1. Inputs for Fusing Face Zones

The steps for fusing face zones are as follows:

1. Select the zones to be fused in the **Zones** list.

   Important:

   When using the Fuse Face Zone utility, each of the selected face zones must be fused in its entirety with another face zone having the same connectivity. Partial fusing of face zones is not supported.

2. (Optional) If you would like to specify a name for the fused zone rather than use an automatically generated name (for instance, to preserve the original name of one of the face zones), disable **Use default name for new fused zone** and enter the desired name.

3. Click the **Fuse** button to fuse the selected zones.

If all of the appropriate faces do not get fused using the default **Tolerance**, you should increase it and attempt to fuse the zones again. (This tolerance is the same as the matching tolerance discussed in Creating Periodic Zones and Interfaces (p. 1269).) The **Tolerance** should not exceed 0.5, or you may fuse the wrong nodes.

You can also fuse zones using the `mesh/modify-zones/fuse-face-zones` text command, as shown in the following example.

```
/mesh/modify-zones > fuse-face-zones

()
Zone to fuse(1) [()]  w1.top

Zone to fuse(2) [()]  w3

Zone to fuse(3) [()]  [Enter]

Choose from list (automatic w3 w1.top) or pick your own
Enter the fused zone name: [automatic]  w3

  Name of zone 14 is changed into w3.
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
Fused list of zones successfully.
```

Important:

Remember to save a new case file (and a data file, if data exists) after fusing faces.

6.12.3.1.1. Fusing Zones on Branch Cuts

Meshes imported from structured mesh generators or solvers (such as FLUENT 4) can often be O-type or C-type meshes with a reentrant branch cut where coincident duplicate nodes lie on a periodic boundary. Since Ansys Fluent uses an unstructured mesh representation, there is no reason to retain this artificial internal boundary. (You can preserve these periodic boundaries and the solution algorithm will solve the problem with periodic boundary conditions.)

To fuse this periodic zone with itself, you must first decouple the periodic zone, as described in Decoupling Periodic Zones (p. 1272). This will create two symmetry zones that you can fuse using the procedure above.

Note that if you need to fuse portions of a non-periodic zone with itself, you must use the `mesh/modify-zones/fuse-face-zones` text command.

`mesh` → `modify-zones` → `fuse-face-zones`

This command will prompt you for the name or ID of each zone to be fused. (You will enter the same zone twice.) To change the node tolerance, use the `mesh/modify-zones/matching-tolerance` text command.

6.12.4. Creating Periodic Zones and Interfaces

Ansys Fluent allows you to assign periodicity to your mesh by coupling a pair of face zones. If the two zones are conformal (that is, have identical node and face distributions), then you can change them to be a pair of coupled zones that are both of type **periodic**. If the two zones are not identical at the boundaries within a specified tolerance, then you can change them both to be of type **interface** and include them in a non-conformal mesh interface that is periodic.

Important:

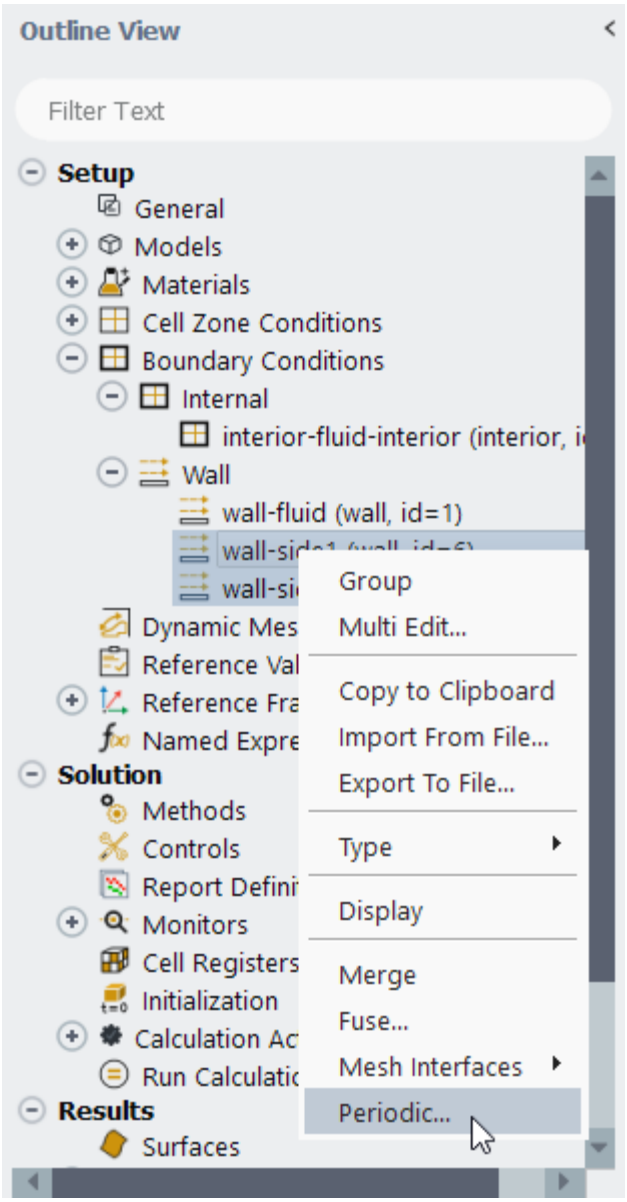
Remember to save a new case file (and a data file, if data exists) after creating or decoupling a periodic boundary pair.

To create a periodic boundary zone pair or interface, perform the following steps:

1. Select the two boundary zones that make up the periodic pair in the Outline View (under **Setup**/**Boundary Conditions**). Note that you can select multiple zones by holding the **Ctrl** key while selecting. The zones cannot be of type **interior** or **periodic**. The order of selection will not affect the calculation results, though it may affect the naming: for conformal pairings, the first zone you select is considered the primary zone of the pair and will be renamed according to your specifications (while the second zone you select is considered the shadow zone).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

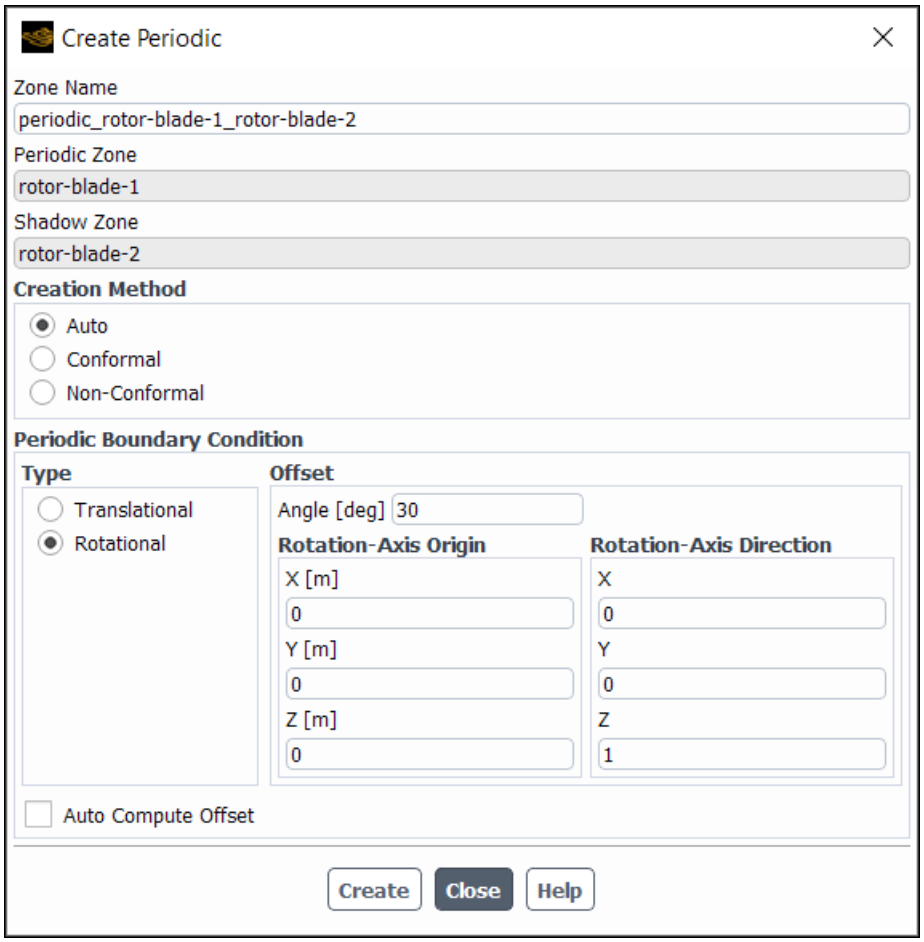
2. Right-click one of the selected zones and select **Periodic...** from the menu that opens, in order to open the **Create Periodic** dialog box.



3. Perform the following steps in the **Create Periodic** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.95: The Create Periodic Dialog Box



a. Enter a **Zone Name**. For conformal pairings, the first boundary zone you selected in step 1. is considered the primary zone and will be renamed with this **Zone Name** during the creation process; for non-conformal pairings, this **Zone Name** will be used for both the resulting mesh interface and a single periodic boundary zone.

b. Make a selection from the **Creation Method** list to indicate whether the face zones that make up the periodic pair are **Conformal** or **Non-Conformal**. If you are unsure, you can select **Auto** and Fluent will automatically handle this decision (in which case a conformal pairing will be used if possible).

Note:

For pairings that are non-conformal, the resulting periodic mesh interface will use the **Matching** option by default. For details about the **Matching** option, see Matching Option (p. 1177).

---

c. Define the settings in the **Periodic Boundary Condition** group box:

- Select either **Translational** or **Rotational** from the **Type** list to define the type of periodicity for the pairing.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Retain the enabled default setting of **Auto Compute Offset** if you want Ansys Fluent to automatically compute the translational offset or the angle of the rotational offset. Otherwise, you can disable the **Auto Compute Offset** option and manually enter values to define the offset.

  Important:

  When using auto computation for 3D cases, you should ensure that the mesh resolution on the periodic interface zone pairs are similar if both of the following conditions are met: the periodic interface zone pairs are not planar; and they are not connected by at least two face zones that meet at an edge that points from one periodic zone to the other. In such situations, having a similar mesh resolution will produce more precise offset calculations (and subsequently, more accurate solutions). It is recommended that the average mesh size on one boundary should not be more than twice that of the other.

- For rotationally periodic pairings, enter values for the **Rotation-Axis Origin** and the **Rotation-Axis Direction**. Note that auto computation of the offset does not mean that these fields are also detected automatically and updated.

d. Click **Create** to create the periodic pairing. If it is a conformal pairing, the face zones are both changed to be of type **periodic**; the primary zone is renamed according to your specified **Zone Name**, and the shadow zone will retain its original name or (if it uses a default name) will be renamed automatically to reflect the new type (as described in Changing Cell and Boundary Zone Types (p. 1301)). If it is a non-conformal pairing, both face zones are changed to be of type **interface**, and a face zone of type **periodic** and a mesh interface is created.

4. When attempting to create a conformal periodic boundary zone pair, Fluent will check to see if the faces on the selected zones "match" (that is, whether or not the nodes on corresponding faces are coincident). The matching tolerance for a face is a fraction of the minimum edge length of the face. If the creation of a conformal periodic boundary zone pair fails, you can change the matching tolerance using the following text command. The tolerance should not exceed 0.5, or you may match up the periodic zones incorrectly and corrupt the mesh.

   `mesh` → `modify-zones` → `matching-tolerance`

5. When using the density-based solver, you can specify a pressure jump for a conformal periodic zone pair; for details, see Periodic Flows (p. 1747).

Note that the functionality of the **Create Periodic** dialog box can also be accessed through the `define/boundary-conditions/modify-zones/create-periodic-interface` text command.

For more information about non-conformal periodic boundaries and an alternate way to create them, see Non-Conformal Meshes (p. 1170) and Using a Non-Conformal Mesh in Ansys Fluent (p. 1185), respectively.

6.12.5. Decoupling Periodic Zones

If you want to decouple the zones in a periodic pair (also referred to as "deleting" or "slitting" a periodic pair), right-click the periodic zone in the Outline View (under **Setup**/**Boundary Conditions**/**Peri-**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

odic), and select **Delete** in the menu that opens. For conformal pairings, both of the zones in the periodic pair are changed to be of type **symmetry**; for non-conformal pairings, the original face zones remain as type **interface** and the associated mesh interface and periodic zone are deleted.

6.12.6. Slitting Face Zones

The face-zone slitting feature has two uses:

- You can slit a single boundary zone of any double-sided type (that is, any boundary zone that has cells on both sides of it) into two distinct zones.
- You can slit a coupled wall zone into two distinct, uncoupled wall zones.

When you slit a face zone, Fluent will duplicate all faces and nodes, except those nodes that are located at the ends (2D) or edges (3D) of the zone. One set of nodes and faces will belong to one of the resulting boundary zones and the other set will belong to the other zone. The only ill effect of the shared nodes at each end is that you may see some inaccuracies at those points when you graphically display solution data with a slit boundary. (Note that if you adapt the boundary after slitting, you will not be able to fuse the boundaries back together again.)

Before you can slit, you first need to select the zone in the **Boundary Condition** task page and change the **Type** to **wall**. Upon changing the **Type** to **wall**, another shadow zone will be created (that is, if the original zone is called **rad-outlet**, another zone called **rad-outlet-shadow** will be created). Then you can apply the `mesh/modify-zones/slit-face-zone` text command on either of the walls (that is, **rad-outlet** or **rad-outlet-shadow**) to separate them into two distinct walls.

Important:

When a wall and wall-shadow pair is created by slitting a face zone, the wall will inherit the zone id number and face orientation of the original zone. A new zone id number will be created and assigned to the wall-shadow.

For example, the outlet of the radiator in an underhood application is typically of **interior** type (that is, has cells on both sides). If you know the mass flow rate through this zone (either from other CFD models or from test data) and want to apply it as a boundary condition at the radiator outlet, you first need to slit the radiator outlet. To be able to slit it, select **wall** from the **Type** drop-down list in the **Boundary Conditions** task page for this outlet. It will create a wall and a shadow. Then you can use the TUI command (`mesh/modify-zones/slit-face-zone`) to slit them. After it is slit, two additional walls will be created, one facing one side of the outlet and another facing the other. Then you can select the appropriate wall and change the **Type** to **mass-flow-inlet** and specify the mass flow rate using the Mass-Flow Inlet Dialog Box (p. 5020). There is no option of **mass-flow-inlet** without first slitting it.

Important:

You should not confuse “slitting” a face zone with “separating” a face zone. When you slit a face zone, additional faces and nodes are created and placed in a new zone. When you separate a face zone, a new zone will be created, but no new faces or nodes are created; the existing faces and nodes are simply redistributed among the zones.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.12.6.1. Inputs for Slitting Face Zones

If you want to slit a face zone, you can use the `mesh/modify-zones/slit-face-zone` text command.

`mesh` → `modify-zones` → `slit-face-zone`

You will specify the face zone's name or ID, and Fluent will replace the zone with two zones:

```
/mesh/modify-zones > slit-face-zone

 Face zone id/name [] wall-4

   zone 4 deleted
   face zone 4 created
   face zone 10 created
```

Important:

Remember to save a new case file (and a data file, if data exists) after slitting faces.

6.12.7. Orienting Face Zones

The faces of a boundary face zone (which have cells only on one side) are oriented such that the normals are all pointing in one direction. However, for internal face zones (which have cells on both sides), the normals are allowed to point in either direction. To orient them so that they all point in one direction, you can use the following TUI command:

`mesh` → `modify-zones` → `orient-face-zone`

Having all of the normals oriented in one direction is needed for some boundary condition types. For example, the **fan** boundary condition type determines the flow direction based on its normals. If some of the normals are pointing in one direction and some in the other, the correct flow orientation cannot be determined, which leads to incorrect results.

6.12.8. Extruding Face Zones

The ability to extrude a boundary face zone allows you to extend the solution domain without having to exit Fluent. A typical application of the extrusion capability is to extend the solution domain when recirculating flow is impinging on a flow outlet. The current extrusion capability creates prismatic or hexahedral layers based on the shape of the face and normal vectors computed by averaging the face normals to the face zone's nodes. You can define the extrusion process by specifying a list of displacements (in SI units) or by specifying a total distance (in SI units) and parametric coordinates.

Important:

- Note that extrusion is not possible from boundary face zones that have hanging nodes.
- Extruding face zones is not allowed on polygonal face zones.
- Extruding face zones is only allowed in the 3D version of Ansys Fluent.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.12.8.1. Specifying Extrusion by Displacement Distances

You can specify the extrusion by entering a list of displacement distances (in SI units) using the `mesh/modify-zones/extrude-face-zone-delta` text command.

`mesh` → `modify-zones` → `extrude-face-zone-delta`

Note:

This text command is not available in the parallel version of Ansys Fluent.

You will be prompted for the boundary face zone ID or name and a list of displacement distances.

6.12.8.2. Specifying Extrusion by Parametric Coordinates

You can specify the extrusion by specifying a distance (in SI units) and parametric coordinates using the `mesh/modify-zones/extrude-face-zone-para` text command.

`mesh` → `modify-zones` → `extrude-face-zone-para`

Note:

This text command is not available in the parallel version of Ansys Fluent.

You will be prompted for the boundary face zone ID or name, a total distance, and a list of parametric coordinates. The list of parametric coordinates should begin with 0.0 and end with 1.0. For example, the following list of parametric coordinates would create two equally spaced extrusion layers: 0.0, 0.5, 1.0.

6.12.9. Replacing, Deleting, Deactivating, and Activating Zones

Ansys Fluent allows you to append or replace an existing cell zone in the mesh. You can also permanently delete a cell zone and all associated face zones, or temporarily deactivate and activate zones from your Ansys Fluent case.

6.12.9.1. Replacing Zones

This feature allows you to replace a small region of a computational domain with a new region of different mesh quality. This functionality will be required where you may want to make changes to the geometry or mesh quality for any part of the domain. This ability of Ansys Fluent will save you time, since you can modify only the required part of the domain without remeshing the whole domain every time.

The replacement mesh must be prepared in advance, with the following considerations:

- The replacement mesh must contain a single cell zone.
- The cell zone in the replacement mesh must be of the same zone type as the zone that is being replaced.

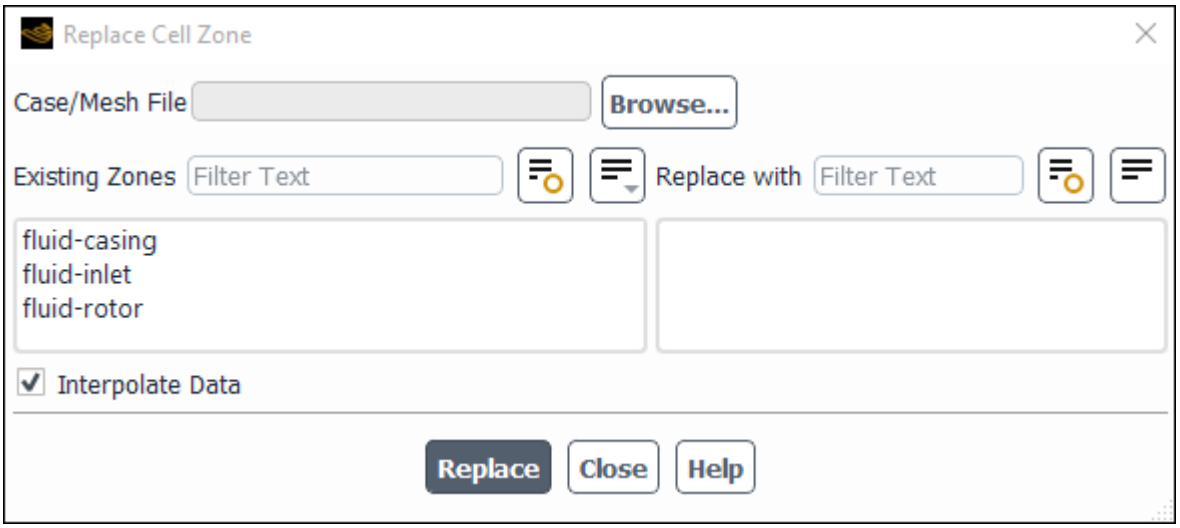
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If the boundaries of the replacement mesh will form a non-conformal interface with existing zones (that is, zones that are not replaced in the existing mesh), these boundaries should be based on the same geometry as the existing zones. If there are sharp features (for example, 90-degree angles) or curvature in the mesh, it is especially important that both sides of the interface closely follow that feature.
- The replacement cell zone does not need to have the same name as the existing zone being replaced.
- When naming the boundary zones of the replacement cell zone, note that boundary conditions will be automatically transferred from the existing cell zone based on name. All boundaries of the replacement cell zone that have unique names will be assigned the default boundary condition for that zone type.
- When selecting the file with the replacement cell zone, note that legacy mesh files (`.msh`) and legacy case files (`.cas`) may be selected, but default CFF mesh files (`.msh.h5`) and CFF case files (`.cas.h5`) may not.

Replacing a zone is performed using the Replace Cell Zone Dialog Box (p. 5593) (Figure 6.96: The Replace Cell Zone Dialog Box (p. 1276)).

Domain → Zones → Replace Zone...

Figure 6.96: The Replace Cell Zone Dialog Box



To replace a zone, do the following:

1. Click **Browse...** and select the new or modified mesh containing the cell zone that will replace one of the cell zones in the current mesh.

   *The name of the cell zone in the replacement mesh will be displayed in the **Replace with** list.*

2. Under **Existing Zones**, select the zone you want to replace.
3. Under **Replace with**, select the zone from the replacement mesh.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Enable/Disable **Interpolate Data**, if data already exists. If the replacement cell zone is geometrically different, then **Interpolate Data** can be turned off to prevent data interpolation over the non-matching geometries.

   Important:

   Disable **Interpolate Data** if the zone being replaced is part of an overset interface.

5. Click **Replace** to replace the selected zone.

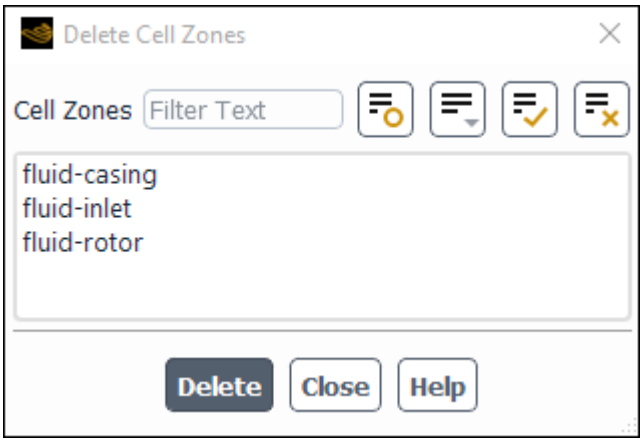
6. Set up the boundary conditions for any of the boundaries of the replacement mesh that do not share a name with one of the existing boundaries.

6.12.9.2. Deleting Zones

To permanently delete zones, select them in the Delete Cell Zones Dialog Box (p. 5434) (Figure 6.97: The Delete Cell Zones Dialog Box (p. 1277)), and click **Delete**.

Domain → Zones → Delete...

Figure 6.97: The Delete Cell Zones Dialog Box



All of the cells, faces, and nodes associated with the cell zone will be deleted. If one of the faces is of type **interior** and borders another cell zone, the face will automatically be changed to a wall and will stay attached to the remaining cell zone.

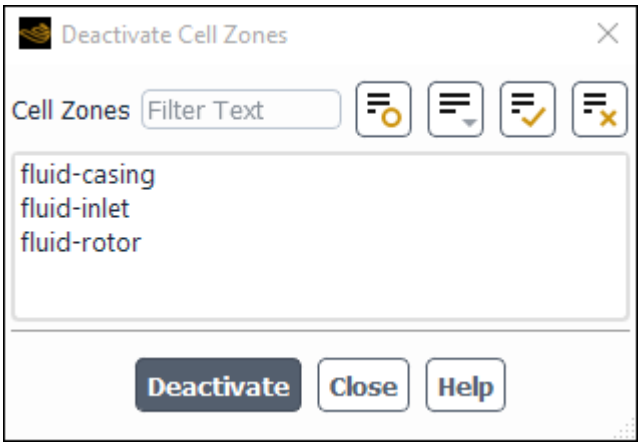
6.12.9.3. Deactivating Zones

To deactivate zones, select them in the Deactivate Cell Zones Dialog Box (p. 5432) (Figure 6.98: The Deactivate Cell Zones Dialog Box (p. 1278)), and click **Deactivate**.

Domain → Zones → Deactivate...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.98: The Deactivate Cell Zones Dialog Box



Deactivation will separate all relevant interior face zones (that is, fan, interior, porous-jump, or radiator) into wall and wall-shadow pairs.

Note:

When you deactivate a zone using the **Deactivate Cell Zones** dialog box, Ansys Fluent will remove the zone from the mesh and from all relevant solver loops.

Important:

If a cell zone contains mesh interfaces it cannot be deactivated until the solution has been initialized.

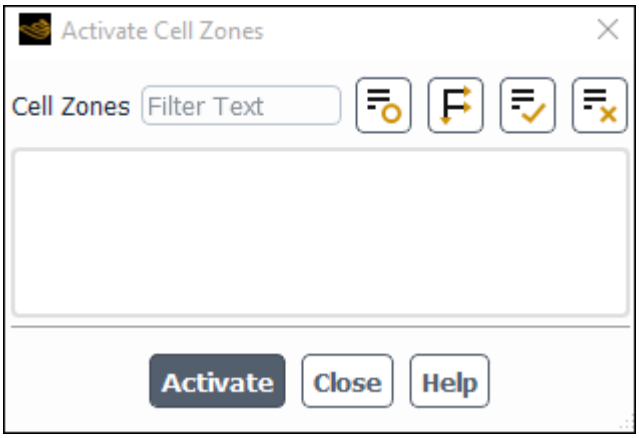
6.12.9.4. Activating Zones

You can reactivate the zones and recover the last data available for them using the Activate Cell Zones Dialog Box (p. 5386) (Figure 6.99: The Activate Cell Zones Dialog Box (p. 1279)).

Domain → **Zones** → **Activate...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.99: The Activate Cell Zones Dialog Box



Note:

The **Activate Cell Zones** dialog box will only be populated with zones that were previously deactivated.

After reactivation, you need to make sure that the boundary conditions for the wall and wall-shadow pairs are restored correctly to what you assigned before deactivating the zones. If you plan to reactivate them at a later time, make sure that the face zones that are separated during deactivation are not modified. Adaption, however, is supported.

6.12.10. Copying Cell Zones

You can create a copy of a cell zone that is offset from the original either by a translational distance or a rotational angle. Note that in the copied zone, the bounding face zones are all converted to walls, any existing cell data is initialized to a constant value, and non-conformal interfaces and dynamic zones are not copied; otherwise, the model settings are the same as in the original zone.

To copy a zone, use the following text command:

`mesh` → `modify-zones` → `copy-move-cell-zone`

You will then be prompted to enter the ID of the zone you want to copy, and either the distance for translation in each of the axes or the rotational angle, origin, and axis.

Note that if you want the copied zone to be connected to existing zones, you must either fuse the boundaries (as described in Fusing Face Zones (p. 1267)) or set up a non-conformal interface (as described in Using a Non-Conformal Mesh in Ansys Fluent (p. 1185)).

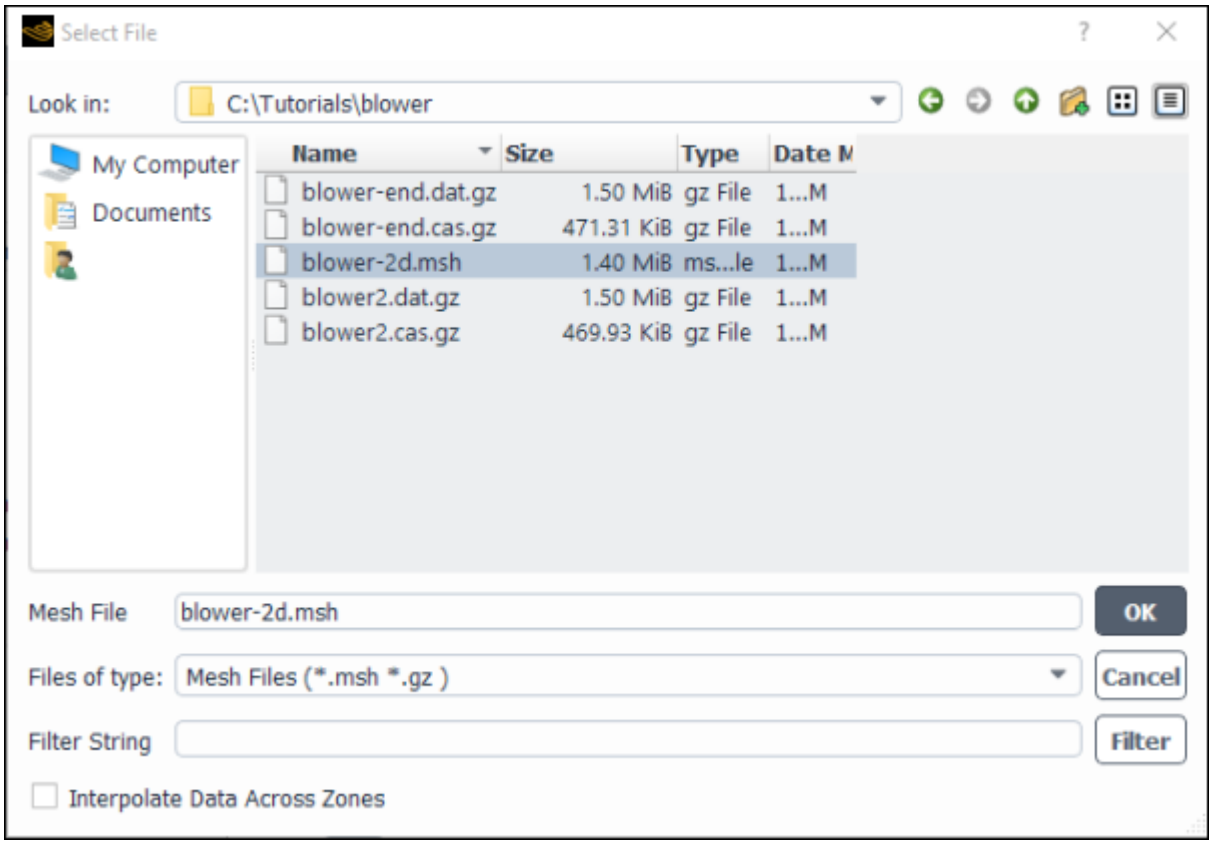
6.12.11. Replacing the Mesh

Ansys Fluent allows you to replace your mesh, so that you can continue to run your simulation on a new mesh without having to manually copy the case file settings or interpolate the existing data. Replacing the mesh globally may be helpful, for example, if you would like to perform a mesh refinement study with a prepared series of meshes, or if your results indicate that your current mesh is not of sufficient quality. Note that the data interpolation is done automatically during the replacement if data exists (that is, you have initialized the flow field or run a calculation).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Global mesh replacement is performed by clicking **Replace Mesh...** in the **Domain** ribbon tab **Zones** group box), which opens the **Select File** dialog box (Figure 6.100: The Select File Dialog Box (p. 1280)).

Figure 6.100: The Select File Dialog Box



6.12.11.1. Inputs for Replacing the Mesh

The steps for replacing the mesh using the **Select File** dialog box (Figure 6.100: The Select File Dialog Box (p. 1280)) are as follows:

1. Select the replacement mesh from the appropriate folder.

2. By default, data is interpolated between matching zone pairs (that is, between the zones with the same names in both the current mesh and the replacement mesh). You can enable the **Interpolate Data Across Zones** option if you want to interpolate data across cell zones when replacing the mesh. This option is appropriate when the matching zone pairs do not have the same interior zone boundaries. Note that global conservation of data is not enforced when the **Interpolate Data Across Zones** option is enabled, so it should only be used when absolutely necessary. For best data interpolation, the zone boundaries of the replacement mesh should be coincident with those of the current mesh.

   When running in parallel the replacement mesh gets partitioned such that matching cells reside on the same compute node. In some cases, differences in mesh resolution can cause the mesh to not be able to find its interpolation cell on the same compute node during interpolation. Fluent then automatically enables **Interpolate Data Across Zones** to ensure successful data interpolation across zones, even if you have not enabled the option. To avoid this behavior use the Scheme command: (`rpsetvar 'dynamesh/replace-mesh/partition-per-zone?`

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

`#t`). Note that using this command comes at the expense of requiring additional repartitioning steps.

3. Click the **OK** button to replace the mesh.

6.12.11.2. Limitations

The following limitations apply when replacing meshes:

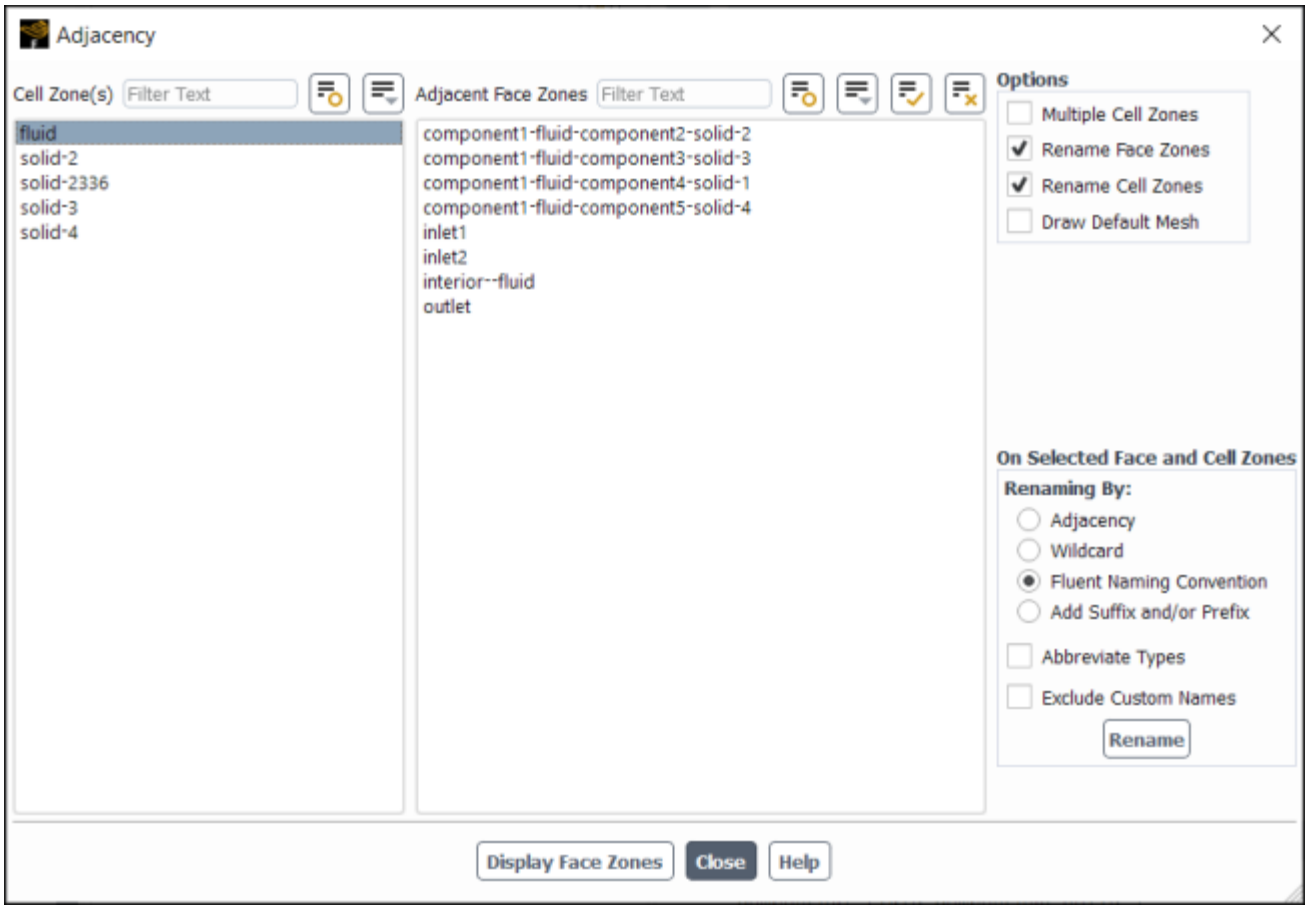
- The boundary conditions of the zones in the current mesh are mapped onto the zones with the same names in the replacement mesh, as described in Reading and Writing Boundary Conditions (p. 971). If your replacement mesh contains zones for which no match is found, these zones will have the default boundary conditions for that zone type.
- The replacement mesh is expected to be an Ansys Fluent mesh file in the legacy format (`.msh`). You may select an Ansys Fluent legacy case file instead, but be aware that only the mesh information will be used and all of the setup information associated with that case file will be ignored. The default CFF files (`.msh.h5` or `.cas.h5`) may not be used as a replacement mesh.
- If you intend to select an Ansys Fluent legacy case file (`.cas`) as the replacement mesh, you must first delete any defined non-conformal mesh interfaces in the case file (as described in Using a Non-Conformal Mesh in Ansys Fluent (p. 1185)).
- You should ensure that the replacement mesh has the same mesh scaling as your current mesh. The data interpolation will not work properly if the meshes are scaled differently.

6.12.12. Managing Adjacent Zones

In some cases you may want to identify and display the face zones which are adjacent to a cell zone. You can do this using the **Adjacency** dialog box.

Domain → **Zones** → **Adjacency...**

Figure 6.101: The Adjacency Dialog Box



Using the **Adjacency** dialog box (Figure 6.101: The Adjacency Dialog Box (p. 1282)) you can also optionally rename zones based on their adjacency or with default names based on zone type. The steps to use the **Adjacency** dialog box are:

1. Open the **Adjacency** dialog box by clicking **Adjacency...** in the **Domain** ribbon tab (**Zones** group box).

2. Select a zone under **Cell Zone(s)** to populate the **Adjacent Face Zones** list. You can use the 'Toggle Tree View' icon ( ) to group the cell zones by type. If the **Multiple Cell Zones** option is disabled, only one cell zone can be selected at a time.

3. Select the face zones you want to display in **Adjacent Face Zones** and click the **Display Face Zones** button.

4. Enable **Draw Default Mesh** to open the **Mesh Display** dialog box where you may choose to display mesh zones. These will be displayed permanently while others will be displayed as currently selected in the **Adjacent Face Zones** list. This is useful for finding your way through a new and complex mesh.

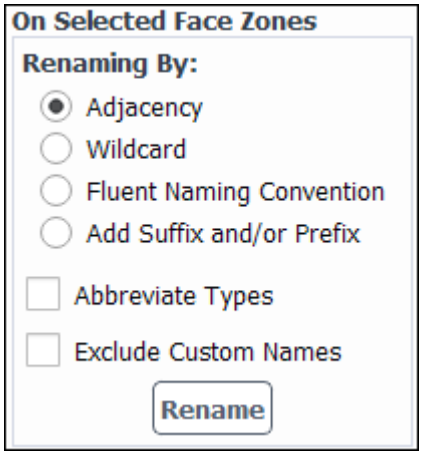
5. Enable **Rename Face Zones** if you want to rename or clean up some of the names that may cause confusion. See Renaming Zones Using the Adjacency Dialog Box (p. 1283) for more information.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.12.12.1. Renaming Zones Using the Adjacency Dialog Box

You can rename selected cell zones and/or face zones from the Adjacency dialog box after enabling **Rename Face Zones** and/or **Rename Cell Zones**. The following renaming methods are available:

Rename by Adjacency

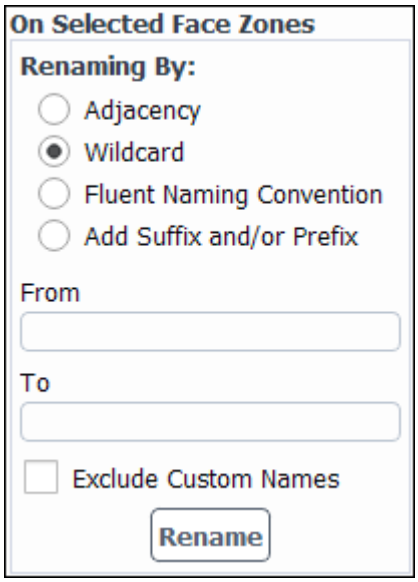


appends the name of the adjacent cell zone to the selected face zone(s). For example, if **fluid** is selected in the **Cell Zone(s)** and **interior** is selected in the **Adjacent Face Zones** list, then renaming by adjacency produces **interior-fluid**. If the name is already in use, the original zone ID is appended in addition in order to create a unique name. Note that it is best if you avoid using long cell zone names.

Note:

Renaming by adjacency is only available for face zones and is not available for renaming cell zones.

Rename by Wildcard



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

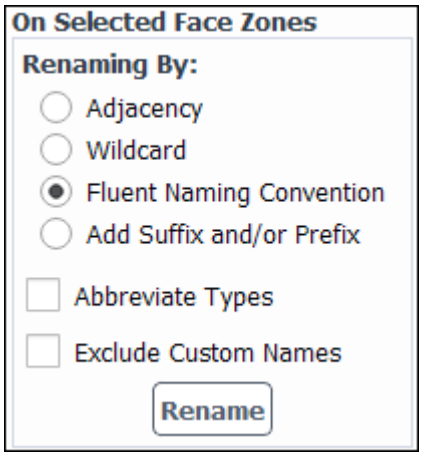
performs pattern matching of the selected zone names against the pattern in the **From** text box (with optional wildcards) and replaces a matching substring with the literal string in the **To** text box. Wildcard characters * and ? are interpreted as follows:

- A * at the beginning and/or end of the **From** pattern matches zero or more characters at the beginning and/or end of the zone name that are retained during renaming with the rest of the name replaced by the **To** string.

  A * used anywhere except at the beginning or end of the **From** pattern is not supported and may have unpredictable results.

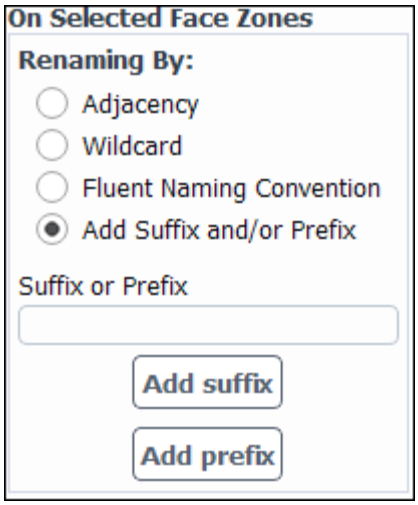
- A ? matches exactly one character and is considered part of the string to be replaced. Multiple ? characters can be used to match substrings of more than one character.

Fluent Naming Convention



renames the selected surface to be a combination of the surface type and the surface ID. In the case of cell zones, the zone is renamed to be a combination of the zone type and zone ID.

Add Suffix and/or Prefix



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

adds the provided text in the **Suffix or Prefix** field to either the beginning or end of the selected surface(s) name(s) or cell zone names. Click **Add Suffix** to add the provided text at the end of the name and click **Add Prefix** to add the provided text at the beginning of the name.

There are two additional options available for some renaming techniques:

- **Exclude Custom Names**—excludes customized names (those that do not conform to a recognized pattern or match any default names) from renaming. This is a protective measure so as not to accidentally destroy your desired naming. Disabling this option will unconditionally rename all selected zones according to the specified renaming method and customized names will be permanently lost.
- **Abbreviate Types**—abbreviates the zone type (for example, `vi` for velocity inlet, `int` for interior, `ifc` for interface, and so on).

6.12.13. Reordering the Domain

In a Fluent session that uses multiple processes, the mesh domain is automatically reordered when it is read, in order to increase memory access efficiency. The reordering uses the Reverse Cuthill-McKee algorithm [35] (p. 5715), and is applied to cells and faces.

A typical output produced during the domain reordering at read is shown below:

```
Building...
     mesh
 auto partitioning mesh by Metis (fast),
 distributing mesh
  parts...,
  faces...,
  nodes...,
  cells...,
        bandwidth reduction using Reverse Cuthill-McKee: 3875/150 = 25.8333
```

The bandwidth is the maximum difference between neighboring cells in the zone—that is, if you numbered each cell in the zone list sequentially and compared the differences between these indices.

6.12.14. Scaling the Mesh

Internally, Ansys Fluent stores the computational mesh in meters, the SI unit of length. When mesh information is read, it is assumed that the mesh was generated in units of meters. If your mesh was created using a different unit of length (inches, feet, centimeters, and so on), you must scale the mesh to meters. To do this, you can select from a list of common units to convert the mesh or you can supply your own custom scale factors. Each node coordinate will be multiplied by the corresponding scale factor.

Scaling can also be used to change the physical size of the mesh. For instance, you could stretch the mesh in the x direction by assigning a scale factor of 2 in the x direction and 1 in the y and z direc-

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

tions. This would double the extent of the mesh in the x direction. However, you should use anisotropic scaling with caution, since it will change the aspect ratios of the cells in your mesh.

Important:

- If you plan to scale the mesh in any way, you should do so before you compute the view factors (as part of an S2S radiation problem), initialize the flow, or begin calculations. Any data that exists when you scale the mesh will be invalid.
- It is a good practice to scale the mesh before setting up the case, especially when you plan to create mesh interfaces or shell conduction zones.
- Scaling the mesh makes the graphics window **[Out of Date]**. Right-click in the graphics window and select **Refresh Display** to update the window.

  You may have to click *to see the object in the graphics window after refreshing the display.*

  *For the refresh of a mesh display, Fluent displays the surfaces that are currently selected in the global **Mesh Display** dialog box (accessed by clicking* *in the toolbar). In some instances this may result in the refreshed display not matching what was originally displayed.*

You have the option of scaling the entire mesh, or individual cell zones. For details, see the following:

6.12.14.1. Scaling the Entire Mesh

6.12.14.2. Scaling Individual Cell Zones

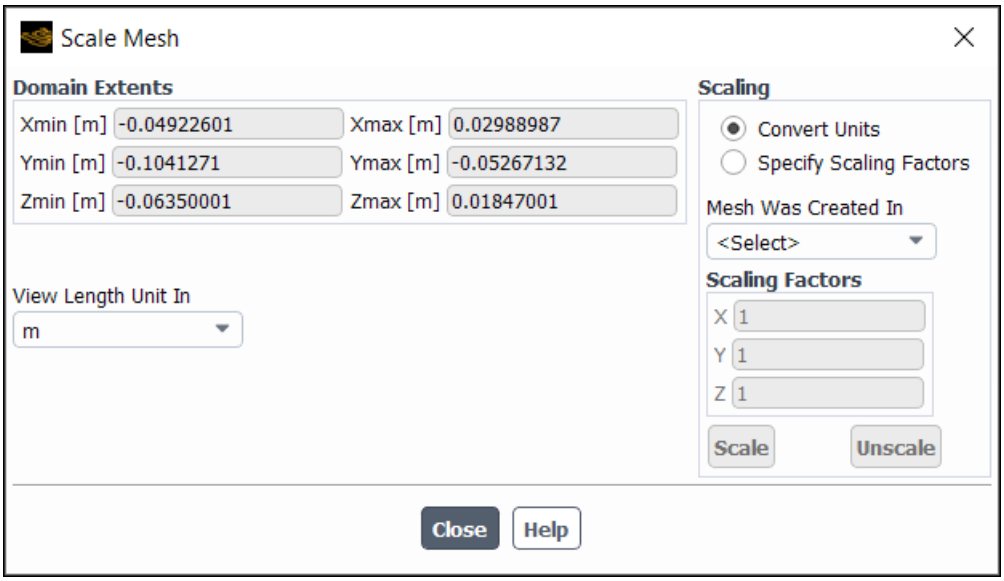
6.12.14.1. Scaling the Entire Mesh

You will use the Scale Mesh Dialog Box (p. 4686) (Figure 6.102: The Scale Mesh Dialog Box (p. 1287)) to scale the entire mesh, either to change the unit of measurement or to apply custom scaling factors.

Setup → **General** → **Mesh** → **Scale...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.102: The Scale Mesh Dialog Box



The procedure for scaling the mesh is as follows:

1. Use the conversion factors provided by Ansys Fluent by selecting **Convert Units** in the **Scaling** group box. Then indicate the units used when creating the mesh by selecting the appropriate abbreviation for meters, centimeters, millimeters, inches, or feet from the **Mesh Was Created In** drop-down list. The **Scaling Factors** will automatically be set to the correct values (for example, 0.0254 meters/inch).

   If you created your mesh using units other than those in the **Mesh Was Created In** drop-down list, you can select **Specify Scaling Factors** and enter values for **X**, **Y**, and **Z** manually in the **Scaling Factors** group box (for example, the number of meters per yard).

2. Click the **Scale** button. The **Domain Extents** will be updated to show the correct range in meters. If you prefer to use your original unit of length during the Ansys Fluent session, you can follow the procedure described below to change the unit.

6.12.14.1.1. Changing the Unit of Length

As mentioned in Step 2. of the previous section, when you scale the mesh you do not change the units; you just convert the original dimensions of your mesh points from your original units to meters by multiplying each node coordinate by the specified **Scaling Factors**. If you want to work in your original units, instead of in meters, you can make a selection from the **View Length Unit In** drop-down list. This updates the **Domain Extents** to show the range in your original units and automatically changes the length unit in the Set Units Dialog Box (p. 4693) (see Customizing Units (p. 1035)). Note that this unit will be used for all future inputs of length quantities.

6.12.14.1.2. Unscaling the Mesh

If you use the wrong scale factor, accidentally click the **Scale** button twice, or want to undo the scaling for any other reason, you can click the **Unscale** button. "Unscaling" simply divides each of the node coordinates by the specified **Scale Factors**. (Selecting **m** in the **Mesh Was Created In** list and clicking on **Scale** will *not* unscale the mesh.)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6.12.14.1.3. Changing the Physical Size of the Mesh

You can also use the Scale Mesh Dialog Box (p. 4686) to change the physical size of the mesh. For example, if your 2D mesh is 5 feet by 8 feet, and you want to model the same geometry with dimensions twice as big (10 feet by 16 feet), you can enter 2 for **X** and **Y** in the **Scaling Factors** group box and click **Scale**. The **Domain Extents** will be updated to show the new range.

6.12.14.2. Scaling Individual Cell Zones

You have the ability to scale one or more individual cell zones. This could be useful, for example, if you append a case file to a mesh that is of a different scale. To scale zones, use the following text command:

`mesh` → `modify-zones` → `scale-zone`

You will be prompted to specify one or more cell zones, as well as the scaling factors in the coordinate directions.

Note:

You cannot scale a zone that is conformally connected to a zone that is not being scaled.

6.12.15. Translating the Mesh

You can "move" the mesh by applying prescribed offsets to the Cartesian coordinates of all the nodes in the mesh. This would be necessary for a rotating problem if the mesh were set up with the axis of rotation not passing through the origin, or for an axisymmetric problem if the mesh were set up with the axis of rotation not coinciding with the x axis. It is also useful if, for example, you want to move the origin to a particular point on an object (such as the leading edge of a flat plate) to make an XY plot have the desired distances on the x axis.

You have the option of translating the entire mesh, or individual cell zones. For details, see the following:

6.12.15.1. Translating the Entire Mesh

6.12.15.2. Translating Individual Cell Zones

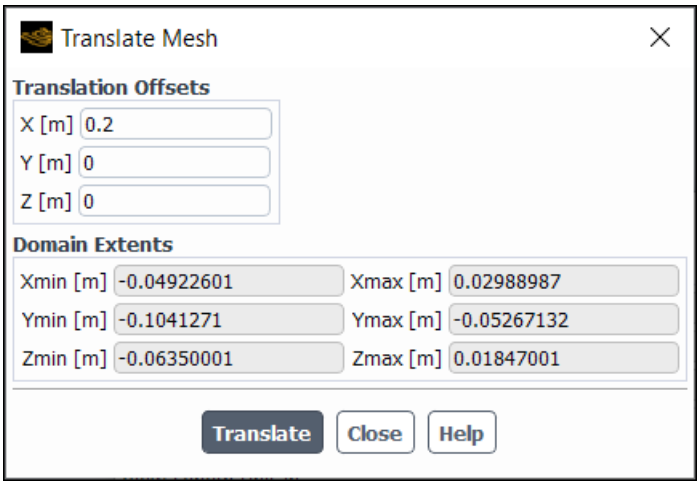
6.12.15.1. Translating the Entire Mesh

You can translate the entire mesh using the Translate Mesh Dialog Box (p. 5634) (Figure 6.103: The Translate Mesh Dialog Box (p. 1289)).

Domain → **Mesh** → **Transform** → **Translate...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.103: The Translate Mesh Dialog Box



The procedure for translating the mesh is as follows:

1. Enter the desired translations in the x, y, and (for 3D) z directions (that is, the desired delta in the axes) in the **X**, **Y**, and **Z** text-entry boxes in the **Translation Offsets** group box. You can specify positive or negative real numbers in the current unit of length.

2. Click the **Translate** button and redisplay the mesh. The **Domain Extents** will be updated to display the new extents of the translated mesh. (Note that the **Domain Extents** are purely informational; you cannot edit them manually.)

6.12.15.2. Translating Individual Cell Zones

You have the ability to translate one or more individual cell zones. This could be useful, for example, if you append a case file to a mesh and they are not properly located relative to each other. To translate zones, use the following text command:

`mesh` → `modify-zones` → `translate-zone`

You will be prompted to specify one or more cell zones, as well as the offsets in the coordinate directions.

Note:

You cannot translate a zone that is conformally connected to a zone that is not being translated.

6.12.16. Rotating the Mesh

The ability to rotate the mesh is analogous to the ability to translate the mesh in Ansys Fluent. You can rotate the mesh about the x, y, or (for 3D) z axis and also specify the rotation origin. This option is useful in the cases where the structural mesh and the CFD mesh are offset by a small angle.

You have the option of rotating the entire mesh, or individual cell zones. For details, see the following:

6.12.16.1. Rotating the Entire Mesh

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

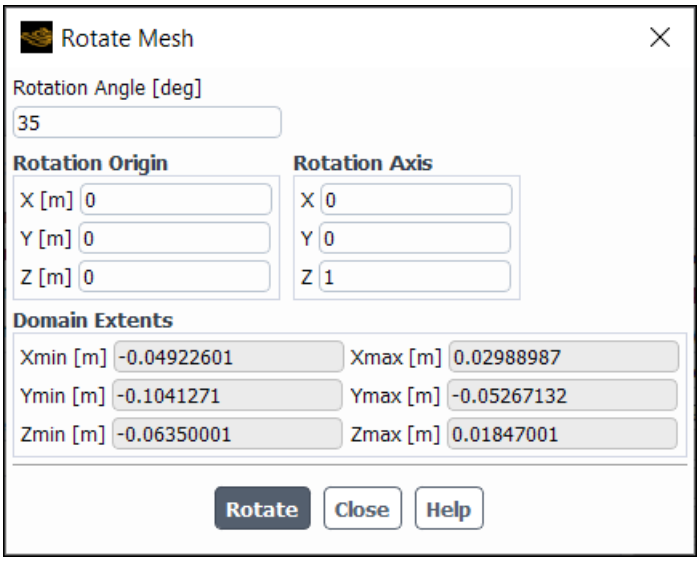
6.12.16.2. Rotating Individual Cell Zones

6.12.16.1. Rotating the Entire Mesh

You can rotate the entire mesh using the Rotate Mesh Dialog Box (p. 5602) (Figure 6.104: The Rotate Mesh Dialog Box (p. 1290)).

Domain → Mesh → Transform → Rotate...

Figure 6.104: The Rotate Mesh Dialog Box



The procedure for rotating the mesh is as follows:

1. Specify the required **Rotation Angle** for the mesh. You can specify any positive or negative real number in the correct unit of angle.

2. In the **Rotation Origin** group box, enter **X**, **Y**, and (for 3D) **Z** coordinates to specify a new origin for the axis of rotation.

3. In the **Rotation Axis** group box, enter values for the **X**, **Y**, and (for 3D) **Z** axes to specify the vector for the axis of rotation.

4. Click the **Rotate** button and redisplay the mesh.

   The Domain Extents will be updated to display the new extents of the rotated mesh. (Note that the Domain Extents are purely informational; you cannot edit them manually.)

6.12.16.2. Rotating Individual Cell Zones

You have the ability to rotate one or more individual cell zones. This could be useful, for example, if you append a case file to a mesh and they are not properly oriented relative to each other. To rotate zones, use the following text command:

`mesh → modify-zones → rotate-zone`

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You will be prompted to specify one or more cell zones, as well as the rotation angle, origin, and axis components.

Note:

You cannot rotate a zone that is conformally connected to a zone that is not being rotated.

6.12.17. Improving the Mesh by Smoothing and Swapping

Smoothing and face swapping are tools that complement mesh adaption by increasing the quality of the final numerical mesh. Smoothing repositions the nodes, and face swapping modifies the cell connectivity to achieve these improvements in quality.

Important:

Only smoothing that is based on quality is recommended; face swapping and smoothing based on skewness or a Laplacian operator have more limitations, and support for them may be removed in a future release.

Important:

Face swapping is applicable only to meshes with triangular or tetrahedral cells.

Important:

Face swapping and smoothing based on skewness or a Laplacian operator are available only for serial cases; only quality-based smoothing can be used for parallel cases.

For additional information, see the following sections:

6.12.17.1. Smoothing

6.12.17.2. Face Swapping

6.12.17.3. Combining Skewness-Based Smoothing and Face Swapping

6.12.17.1. Smoothing

The three smoothing methods that are available in Ansys Fluent are:

- quality-based smoothing

  This method is the only recommended method, and is available for all types of meshes in serial and parallel.

- Laplacian smoothing

  This method is not recommended, though it is available for all types of meshes in serial (and is best suited for quadrilateral and hexahedral meshes).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- skewness-based smoothing

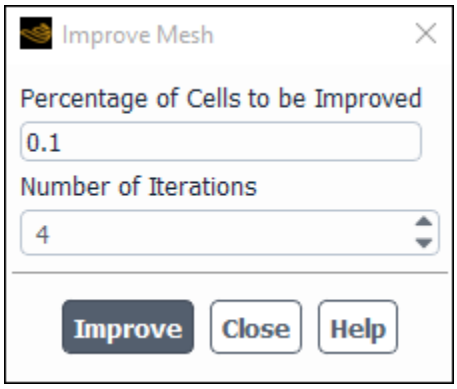
  This method is not recommended, though it is available for triangular and tetrahedral meshes in serial; it can be used alternatively with face swapping (see Combining Skewness-Based Smoothing and Face Swapping (p. 1297)).

6.12.17.1.1. Quality-Based Smoothing

Smoothing that is based on quality is performed using the Improve Mesh Dialog Box (p. 5495) (Figure 6.105: The Improve Mesh Dialog Box (p. 1292)).

Domain → Mesh → Quality → Improve Mesh Quality...

Figure 6.105: The Improve Mesh Dialog Box



Ansys Fluent divides the mesh into a number of "bins", each of which contain a certain number of cells. Improvements are attempted on the cells in those bins that exhibit the lowest orthogonal quality (as defined in Mesh Quality (p. 1146)). As part of this method, you specify the percentage of the total number of cells, in order to determine how many bins are modified.

Note that this smoothing is similar to when you use the `mesh/repair-improve/improve` text command. The advantage of using the **Improve Mesh** dialog box rather than the text command is that you can control the percentage of the cells that Ansys Fluent attempts to improve.

To perform quality-based smoothing, do the following steps:

1. In the Improve Mesh Dialog Box (p. 5495) (Figure 6.105: The Improve Mesh Dialog Box (p. 1292)), enter the **Percentage of Cells to be Improved**.

   Important:

   Quality-based smoothing will be CPU intensive if you specify a large value for the **Percentage of Cells to be Improved**. It is recommended that you enter a small value initially, and then perform the smoothing process multiple times, if necessary. Note that the maximum percentage allowed is 10%.

2. Specify the number of successive smoothing sweeps to be performed on the mesh in the **Number of Iterations** number-entry box. The default value is 4.

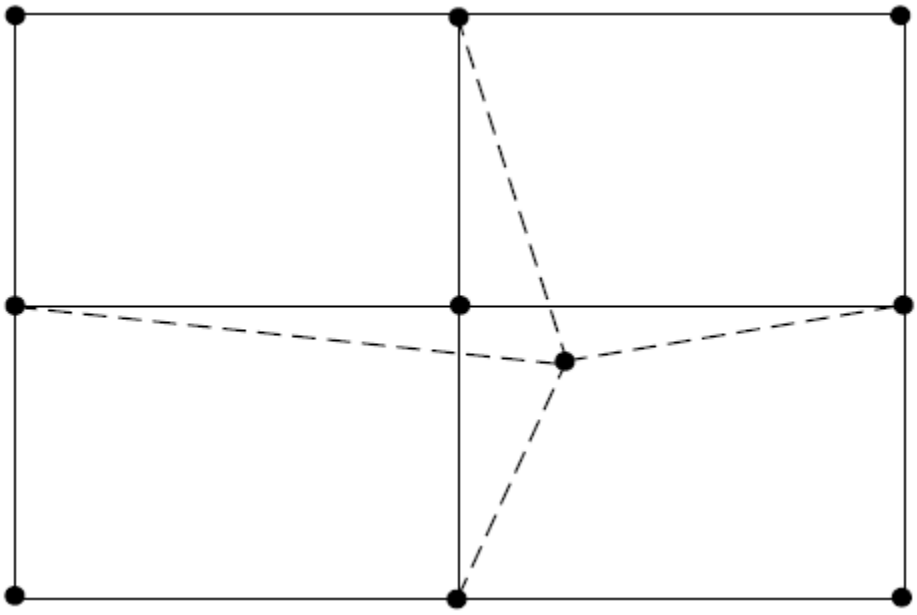
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Click the **Improve** button.

6.12.17.1.2. Laplacian Smoothing

When you use this method, a Laplacian smoothing operator is applied to the unstructured mesh to reposition nodes. The new node position is the average of the positions of its node neighbors. The computed node position increment is multiplied by the relaxation factor (which is set to a value between 0.0 and 1.0). A value of zero for the relaxation factor results in no movement of the node, and a value of unity results in movement equivalent to the entire computed increment. Figure 6.106: Result of Smoothing Operator on Node Position (p. 1293) illustrates the new node position for a typical configuration of quadrilateral cells. The dashed line is the original mesh and the solid line is the final mesh.

Figure 6.106: Result of Smoothing Operator on Node Position



This repositioning strategy improves the skewness of the mesh, but relaxes the clustering of node points. In extreme circumstances, the present operator may create mesh lines that cross over the boundary, creating negative cell volumes. This is most likely to occur near sharp or coarsely resolved convex corners, especially if you perform multiple smoothing operations with a large relaxation factor. Figure 6.107: Initial Mesh Before Smoothing Operation (p. 1294) illustrates an initial tetrahedral mesh before one unrelaxed smoothing iteration creates mesh lines that cross over each other (Figure 6.108: Mesh Smoothing Causing Mesh-Line Crossing (p. 1294)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6.107: Initial Mesh Before Smoothing Operation

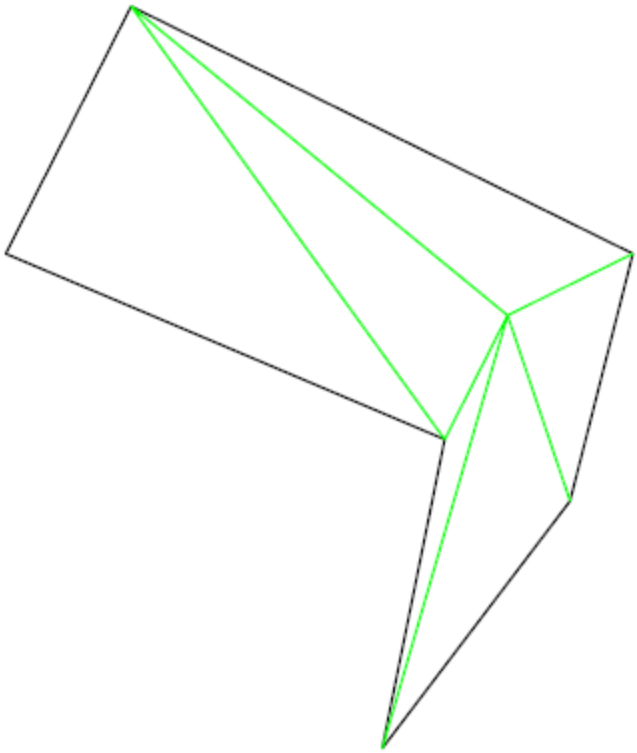
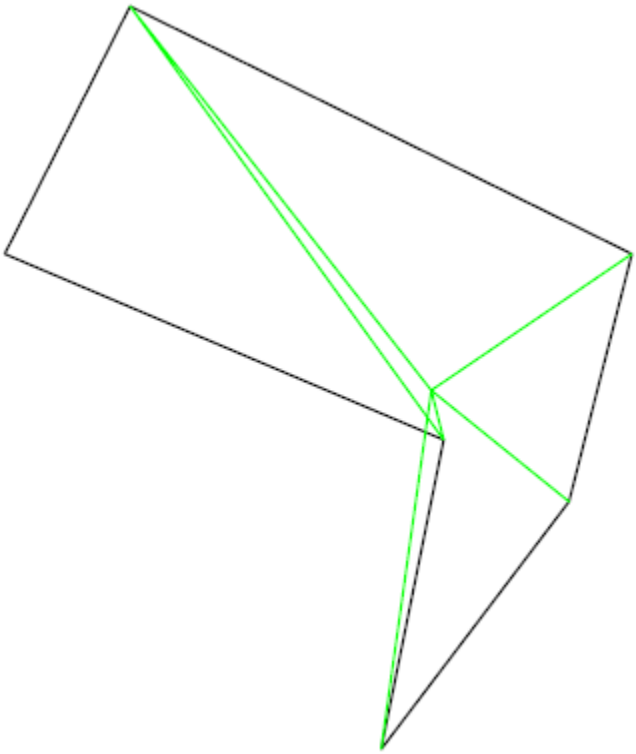


Figure 6.108: Mesh Smoothing Causing Mesh-Line Crossing



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The default smoothing parameters are designed to improve mesh quality with minimal adverse effects, but it is recommended that you save a case file before smoothing the mesh. If you apply a conservative relaxation factor and start with a good quality initial mesh, the frequency of failure due to smoothing is extremely low in two dimensions. However, corruption of the mesh topology occurs much more frequently in three dimensions, particularly with tetrahedral meshes.

The smoothing operator can also be applied repeatedly, but as the number of smoothing sweeps increase, the node points have a tendency to pull away from boundaries and the mesh tends to lose any clustering characteristics.

To perform Laplacian smoothing, enter the `mesh/smooth-mesh` text command in the console and respond to the prompts as follows:

1. Enter `"laplace"` for the `type of smoothing`.
2. Specify the number of successive smoothing sweeps to be performed on the mesh in response to the `number of iterations` prompt. The default value is 4.
3. Specify the factor by which to multiply the computed position increment for the node in response to the `relaxation factor` prompt. The lower the factor, the more reduction in node movement.

6.12.17.1.3. Skewness-Based Smoothing

When you use skewness-based smoothing, Ansys Fluent applies a smoothing operator to the mesh, repositioning interior nodes to lower the maximum skewness of the mesh. Ansys Fluent will try to move interior nodes to improve the skewness of cells with skewness greater than the specified "skewness threshold". This process can be very time consuming, so perform smoothing only on cells with high skewness.

Improved results can be obtained by smoothing the nodes several times. There are internal checks that will prevent a node from being moved if moving it causes the maximum skewness to increase, but it is common for the skewness of some cells to increase when a cell with a higher skewness is being improved. Thus, you may see the average skewness increase while the maximum skewness is decreasing.

Important:

Carefully consider whether the improvements to the mesh due to a decrease in the maximum skewness are worth the potential increase in the average skewness. Performing smoothing only on cells with very high skewness (for example, 0.8 or 0.9) may reduce the adverse effects on the average skewness.

To perform skewness-based smoothing, enter the `mesh/smooth-mesh` text command in the console and respond to the prompts as follows:

1. Enter `"skewness"` for the `type of smoothing`.
2. Specify the number of successive smoothing sweeps to be performed on the mesh in response to the `number of iterations` prompt. The default value is 4.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Specify the minimum cell skewness value for which node smoothing will be attempted in response to the `skewness threshold` prompt. Ansys Fluent will try to move interior nodes to improve the skewness of cells with skewness greater than this value. By default, skewness threshold is set to 0.4 for 2D cases and 0.8 for 3D cases.

6.12.17.2. Face Swapping

While it is not recommended over smoothing (and support for it may be removed in future releases), face swapping can be used to improve the quality of a triangular or tetrahedral mesh in the serial version of Ansys Fluent.

To perform face swapping, enter the `mesh/swap-mesh-faces` text command repeatedly until the console report declares that 0 faces were swapped out. The console report will display the total number of faces that were visited and tested for possible face swapping.

6.12.17.2.1. Triangular Meshes

The approach for triangular meshes is to use the Delaunay circle test to decide if a face shared by two triangular cells should be swapped. A pair of cells sharing a face satisfies the circle test if the circumcircle of one cell does not contain the unshared node of the second cell. Figure 6.109: Examples of Cell Configurations in the Circle Test (p. 1296) illustrates cell neighbors in the circle test. In cases where the circle test is not satisfied, the diagonal or face is swapped, as illustrated in Figure 6.110: Swapped Faces to Satisfy the Delaunay Circle Test (p. 1296).

Figure 6.109: Examples of Cell Configurations in the Circle Test

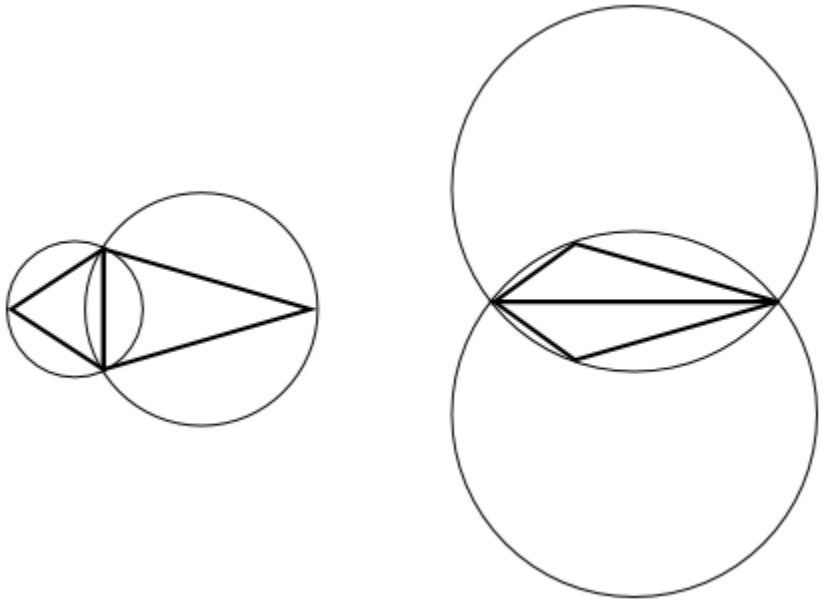


Figure 6.110: Swapped Faces to Satisfy the Delaunay Circle Test

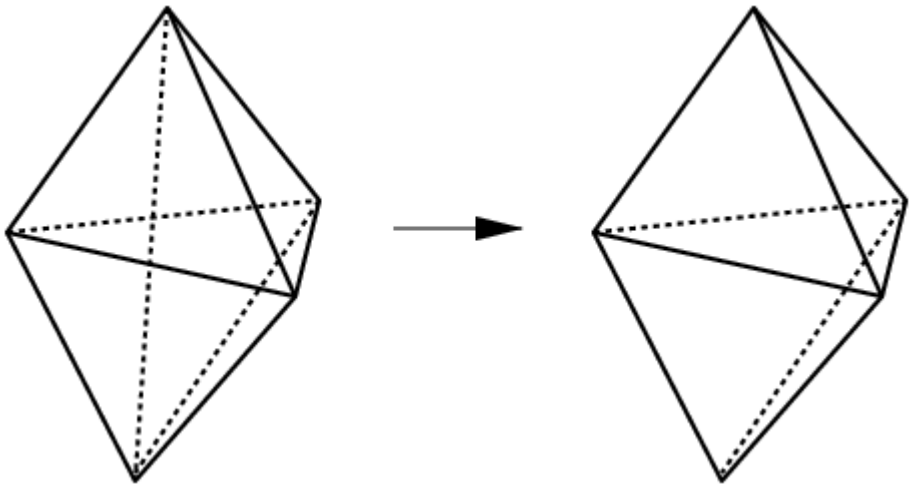
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Repeated application of the face-swapping technique will produce a constrained Delaunay mesh. If you have a Delaunay mesh, it is a unique triangulation that maximizes the minimum angles in the mesh. Thus, the triangulation tends toward equilateral cells, providing the most equilateral mesh for the given node distribution. For more information on Delaunay mesh generation, see Generating Tetrahedral Meshes (p. 780) in the Fluent Meshing section of the User's Guide.

6.12.17.2.2. Tetrahedral Meshes

For tetrahedral meshes, face swapping consists of searching for configurations of three cells sharing an edge and converting them into two cells sharing a face to decrease skewness and the cell count (see Figure 6.111: 3D Face Swapping (p. 1297)).

Figure 6.111: 3D Face Swapping



6.12.17.3. Combining Skewness-Based Smoothing and Face Swapping

As mentioned in Skewness-Based Smoothing (p. 1295), skewness-based smoothing should usually be alternated with face swapping. Guidelines for this procedure are presented here.

- Perform four smoothing iterations using a `skewness threshold` of 0.8 for 3D cases, or 0.4 for 2D cases.
- Swap until the number of faces swapped decreases to 0.
- For 3D meshes, decrease the `skewness threshold` to 0.6 and repeat the smoothing/swapping procedure.

6.12.18. Boundary Layer Redistribution

After performing mesh operations, the boundary layer zone may not satisfy the original growth rate. When the boundary layer is composed of hex and/or wedge cells, you can restore the desired growth rate by using the `mesh/redistribute-boundary-layer` text command. You will be prompted to enter a face zone and the desired growth rate. The face zone you specify must be adjacent to the boundary layer zone that you want to redistribute.

```
>  mesh redistribute-boundary-layer
face zone id/name [] wall-fluid
growth rate [1.1] 1.2
```

Fluent will redistribute the nodes in a boundary region that extends into the mesh from the face zone you specify until one of the following criteria is encountered:

- the edge of the domain
- a change in zone
- a change in element type

6.12.19. Deleting Cells

You can delete cells that are marked by a specified cell register. After the cells are deleted, one boundary zone of type symmetry is created for each cell zone at the faces that the deleted cells shared with their neighboring cells.

To delete cells marked by a cell register, perform the following steps:

1. Create a cell register to mark the cells you want to delete. For details, see Using Cell Registers (p. 3851). Note that to create some kinds of cell registers (such as one based on a field variable), you must first initialize the solution.
2. Delete the cells by using the following text command:

   `mesh` → `modify-zones` → `delete-cell-by-mark`

   The names of any symmetry boundary zones created as part of this deletion are printed in the console.

Note:

- The deletion of cells in a cell register is not supported with the following:
  - overset meshes
  - shell conduction walls
- There is no way to undo the deletion, so it is recommended that you save a copy of the case file before deleting.
- overset meshes
- shell conduction walls

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 7: Cell Zone and Boundary Conditions

This chapter describes the cell zone and boundary condition options available in Ansys Fluent. Details regarding the cell zone and boundary condition inputs and the internal treatment at boundaries are provided.

The information in this chapter is divided into the following sections:

7.1. Overview
7.2. Cell Zone Conditions
7.3. Operating Conditions
7.4. Boundary Conditions
7.5. Editing Multiple Boundary Conditions at Once
7.6. Transient Cell Zone and Boundary Conditions
7.7. Boundary Acoustic Wave Models
7.8. User-Defined Fan Model
7.9. Profiles
7.10. Coupling Boundary Conditions with GT-POWER
7.11. Coupling Boundary Conditions with WAVE

7.1. Overview

Cell zone and boundary conditions specify the flow and thermal variables on the boundaries of your physical model. They are, therefore, a critical component of your Ansys Fluent simulations and it is important that they are specified appropriately.

In this chapter, most of the cell zones and boundary conditions will be described in detail, and an explanation of how to set them and where they are most appropriately used will be provided.

7.1.1. Available Cell Zone and Boundary Types
7.1.2. The Cell Zone and Boundary Conditions Task Pages
7.1.3. Changing Cell and Boundary Zone Types
7.1.4. Setting Cell Zone and Boundary Conditions
7.1.5. Copying Cell Zone and Boundary Conditions
7.1.6. Exporting Boundary Zones in CSV Format
7.1.7. Changing Cell or Boundary Zone Names
7.1.8. Defining Non-Uniform Cell Zone and Boundary Conditions
7.1.9. Defining and Viewing Parameters
7.1.10. Selecting Cell or Boundary Zones in the Graphics Display
7.1.11. Operating and Periodic Conditions

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.1.12. Saving and Reusing Cell Zone and Boundary Conditions

7.1.1. Available Cell Zone and Boundary Types

The cell zone and boundary zone types available in Ansys Fluent are classified as follows:

Table 7.1: Zone Types by Category

Category	Zone Types
External	axis, degassing, exhaust fan, inlet vent, intake fan, interface, mass-flow inlet, mass-flow outlet, outflow, outlet vent, overset, pressure far-field, pressure inlet, pressure outlet, symmetry, velocity inlet, and wall (one-sided only)
Internal	fan, interior, porous jump, radiator, RANS/LES interface, and wall (two-sided only)
Periodic	periodic
Cells	fluid, solid (porous media and 3D fans are a type of fluid cell)

Note that internal boundary zones are connected with the cells on both sides, whereas external boundary zones are connected with the cells on only one side.

7.1.2. The Cell Zone and Boundary Conditions Task Pages

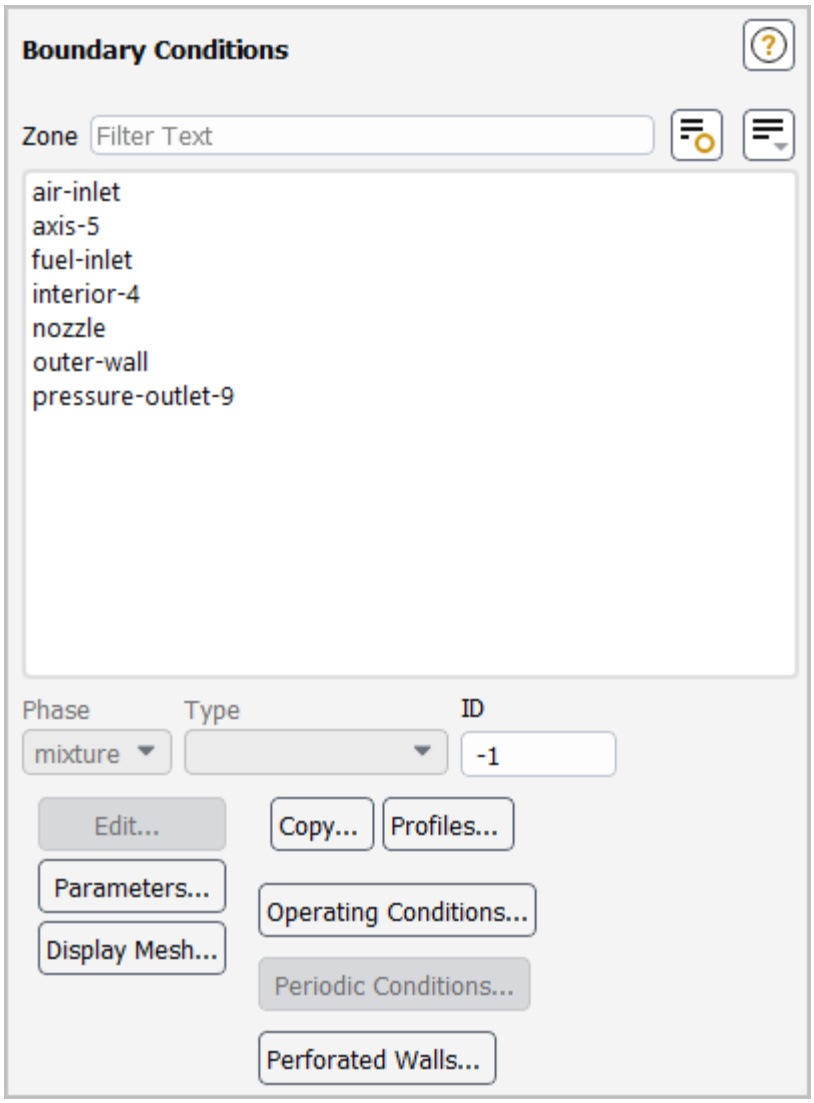
The **Cell Zone Conditions** and **Boundary Conditions** task pages (Figure 7.1: The Boundary Conditions Task Page (p. 1301)) display lists of the zones (which can be grouped by name or zone type), and allow you to revise their settings and perform other related actions. They provide an alternate setup method to working directly in the Outline View tree.

Setup → **Cell Zone Conditions**

Setup → **Boundary Conditions**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.1: The Boundary Conditions Task Page



The sections that follow explain how to perform operations, mostly using the Outline View tree, however these same operations can also be completed using the **Cell Zone Conditions** or **Boundary Conditions** task page. The sections also explain how to use the mouse and the graphics display in conjunction with the dialog box.

Note — You can modify individual boundary conditions and cell zones by expanding the **Cell Zones** and **Boundary Conditions** branches in the tree, and right-clicking the boundary condition or cell zone you want to modify.

7.1.3. Changing Cell and Boundary Zone Types

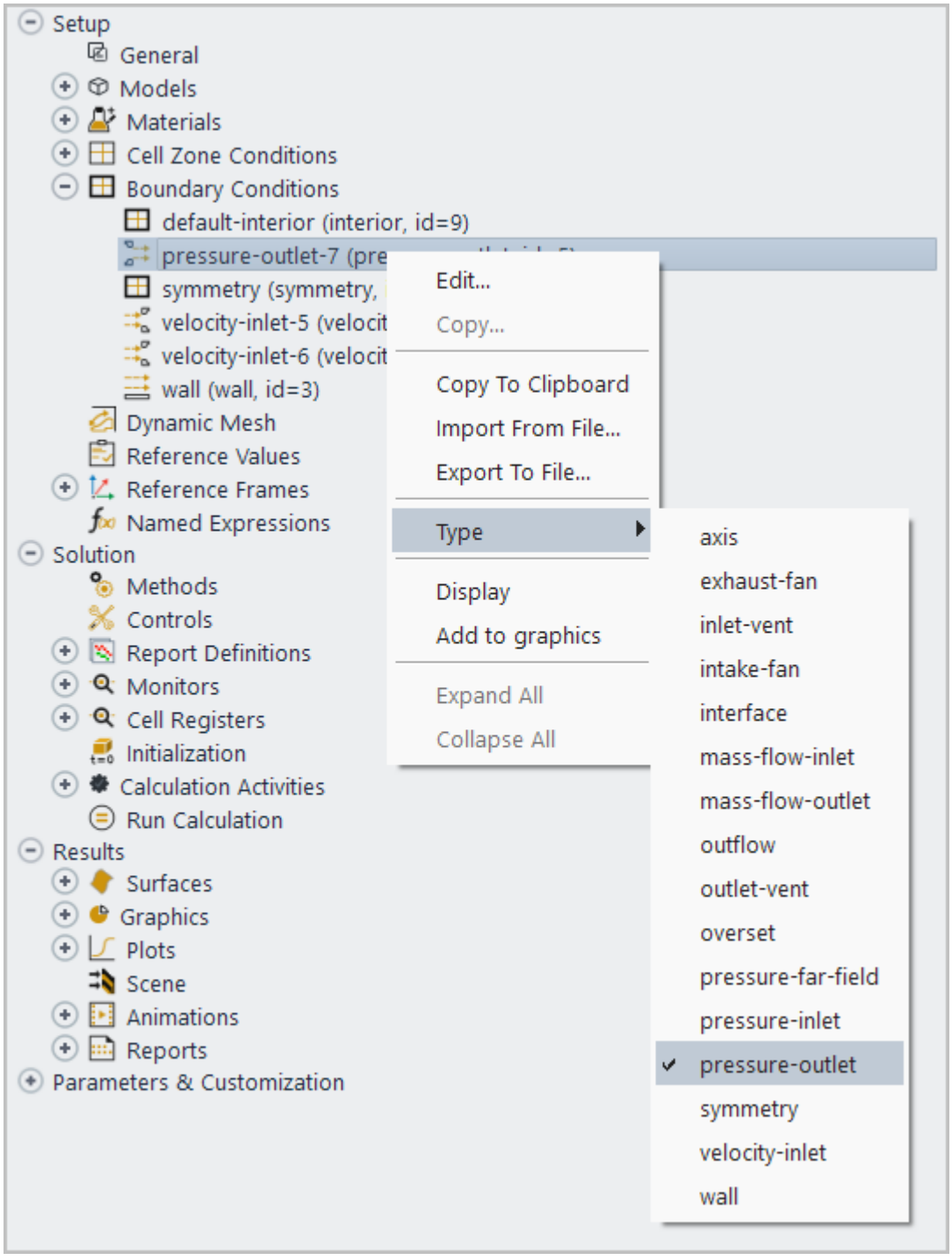
Before you set any cell zone or boundary conditions, you should check the zone types of all boundary zones and change any if necessary. For example, if your mesh includes a pressure inlet, but you want to use a velocity inlet instead, you will need to change the pressure-inlet zone to a velocity-inlet zone. Note that external and internal boundary zones (as listed in Table 7.1: Zone Types by Category (p. 1300)) can only be changed to a zone of the same category.

The steps for changing a zone type using the graphical user interface are as follows:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the boundary or cell zone in the tree (under **Setup**/**Boundary Conditions** or **Setup**/**Cell Zone Conditions**). Note that you can select multiple zones by holding the **Shift** or **Ctrl** keys when selecting, though you must ensure that all of the selections are of the same category.

2. Right-click a selected boundary or cell zone, and select the correct zone type from the **Type** sub-menu.



You can also change the zone type using either the `define/boundary-conditions/zone-type` or the `define/boundary-conditions/modify-zones/zone-type` text command. To

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

change the type for multiple zones of the same category through a single action, you can enter a list of names or IDs contained within a pair of parentheses at the prompt, as shown in the following example (which changes zone 4 and 9 to be pressure inlets):

```
>
define/boundary-conditions/zone-type
zone id/name/list []
(4 9)
external-type>
pressure-inlet
```

When changing the cell or boundary zone type, note the following:

- For zones that have a default name (that is, <*type*>-<*ID*>, <*type*>_<*ID*>, or <*type*>.<*ID*>), the <*type*> in the name will be changed automatically to match the new type.
- When you have a large number of internal boundary zones that you want to change to or from a wall / shadow pair, it is recommended that you change their type with a single action (either by multi-selecting in the tree or by using parentheses at the text command prompt); this will greatly reduce the processing time associated with the change.
- When changing a wall / shadow pair to one of the other internal types (fan, interior, porous jump, radiator, or RANS/LES interface), you can select either the wall or the wall-shadow to change. In either case, the resulting zone will retain the ID and orientation of the wall zone and the wall-shadow zone will be deleted.
- Note that you cannot use this method to change zone types to or from the periodic type, since additional restrictions exist for this boundary type. For details on creating or decoupling periodic zones, see Creating Periodic Zones and Interfaces (p. 1269) or Decoupling Periodic Zones (p. 1272), respectively.
- If you are using one of the general multiphase models (VOF, mixture, or Eulerian), the procedure for changing types is slightly different. See Steps for Setting Boundary Conditions (p. 2957) for details.

7.1.4. Setting Cell Zone and Boundary Conditions

In Ansys Fluent, boundary conditions are associated with zones, not with individual faces or cells. If you want to combine two or more zones that will have the same boundary conditions, see Merging Zones (p. 1261) for information about merging zones.

To set cell zone and boundary conditions for a particular zone, perform one of the following sequences:

1. Right-click the name of zone in the tree (under **Setup**/**Boundary Conditions** or **Setup**/**Cell Zone Conditions**).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Select **Edit...**.

Note:

You can group **Boundary Conditions** based on adjacency, allowing you to see which boundaries belong to the same cell zone. To do this, right-click the **Boundary Conditions** branch in the **Outline View** tree and select **Group By**/**Adjacency**.

Setup → **Boundary Conditions** **Group By** → **Adjacency**

---

or

1. Select the zone from the **Zone** list in the **Cell Zone Conditions** or **Boundary Conditions** task page.

2. Click the **Edit...** button.

The dialog box for the selected cell or boundary zone will open, and you can specify the appropriate conditions.

Important:

If you are using one of the general multiphase models (VOF, mixture, or Eulerian), the procedure for setting conditions is slightly different from that described above. See Steps for Setting Boundary Conditions (p. 2957) for details.

You can use the text command: `define/boundary-conditions/set/` to define one or more settings at a single or multiple boundaries/cell zones of a given type. This facilitates quick and efficient setup of cell/boundary conditions for cases with large numbers of zones.

To edit multiple boundaries you can either provide a list of boundary names / IDs enclosed in parenthesis:

```
                /define/boundary-conditions/set>
                wall
                (rotor-hub rotor-shroud rotor-blade-1 rotor-blade-2 stator-hub stator-shroud stator-blade-1 stato
zone(s) to set boundary conditions(1) [rotor-hub]
                (rotor-blade-1 rotor-blade-2)
```

or you can use a wildcard pattern for the boundary names:

```
                /define/boundary-conditions/set>
                wall
                (rotor-hub rotor-shroud rotor-blade-1 rotor-blade-2 stator-hub stator-shroud stator-blade-1 stato
zone(s) to set boundary conditions(1) [rotor-hub]
                (rotor-blade-*)
```

For additional information on formatting lists in the console, see Lists.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

After entering the zone(s) you want to change you will be prompted for what setting to change. Pressing **Enter** will list the available settings.

When you have changed all of the desired settings, press **q** to exit the `define/boundary-conditions/set/<type>` command. Note that some settings will not take effect until after you have exited the command, by pressing **q**.

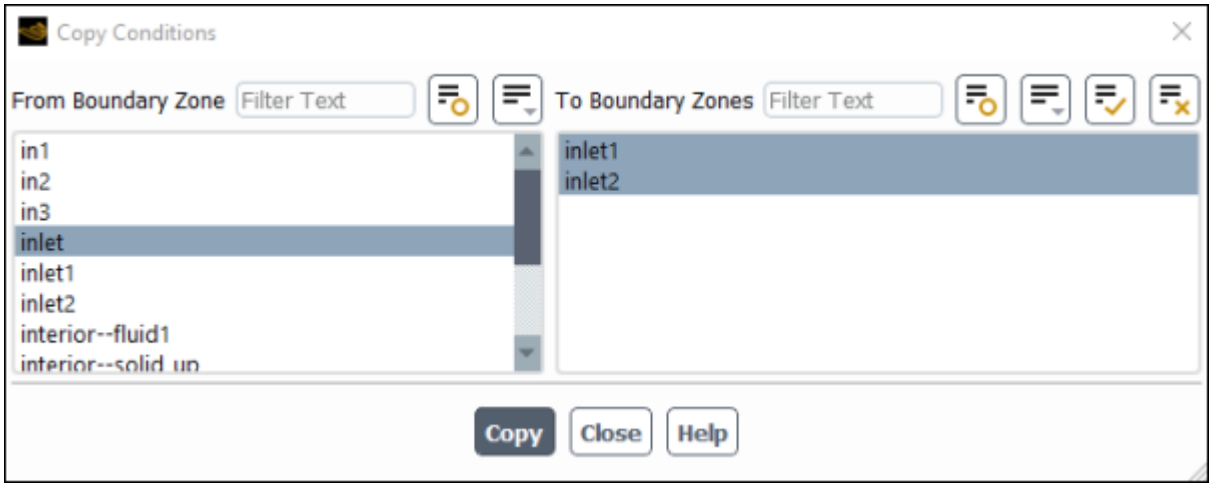
7.1.5. Copying Cell Zone and Boundary Conditions

You can copy cell zones and boundary conditions from one zone to other zones of the same type. If, for example, you have several wall zones in your model and they all have the same boundary conditions, you can set the conditions for one wall, and then simply copy them to the others.

The procedure for copying cell zone or boundary conditions is as follows:

1. Right-click the name of zone in the tree that you want to copy (under **Setup**/**Boundary Conditions** or **Setup** / **Cell Zone Conditions**) and select **Copy...**. This will open the **Copy Conditions** dialog box (Figure 7.2: The Copy Conditions Dialog Box (p. 1305)).

   Figure 7.2: The Copy Conditions Dialog Box



2. In the **From Cell Zone** or **From Boundary Zone** list, confirm that the zone that has the conditions you want to copy is selected.

3. In the **To Cell Zones** or **To Boundary Zones** list, select the zone or zones to which you want to copy the conditions.

4. Click **Copy**. Ansys Fluent will set *all* of the cell zones or boundary conditions for the zones selected in the **To Cell Zones** or **To Boundary Zones** list to be the same as the conditions for the zone selected in the **From Cell Zone** or **From Boundary Zone** list. (You cannot copy a subset of the conditions, such as only the thermal conditions.)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

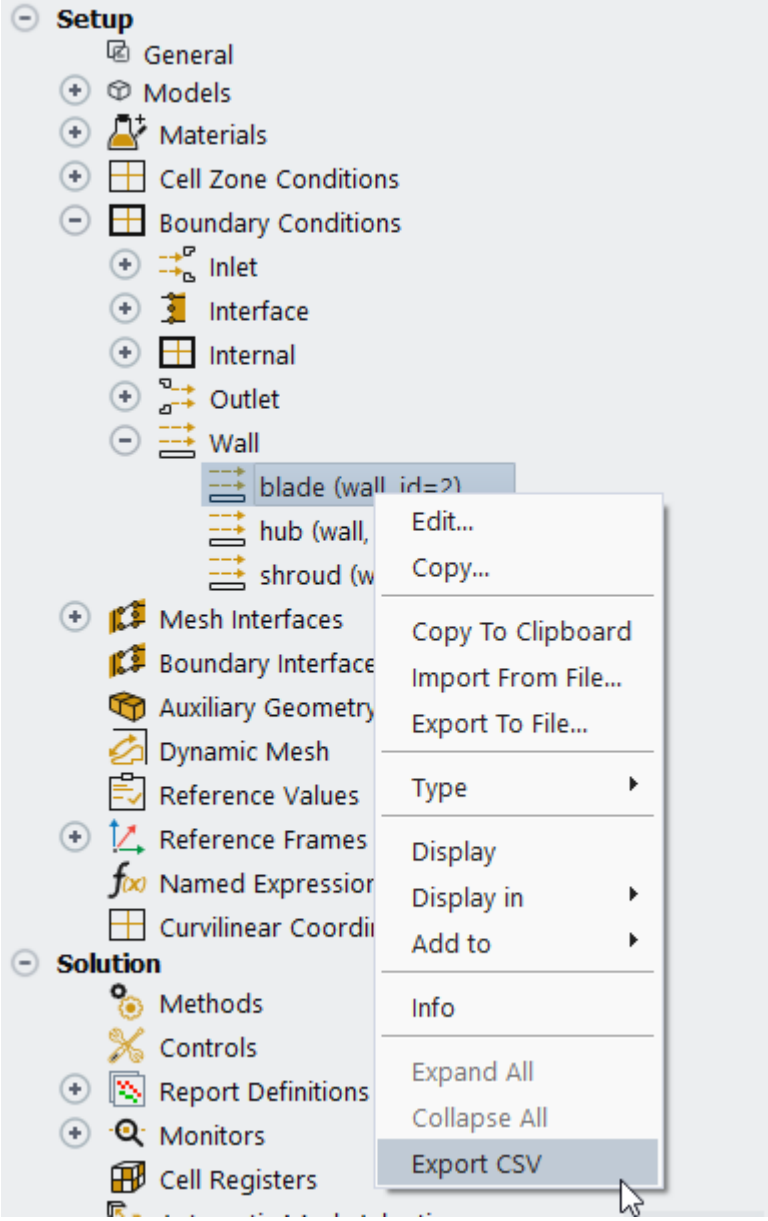
Note that you cannot copy conditions from external walls to internal (that is, two-sided) walls, or vice versa, if the energy equation is being solved, since the thermal conditions for external and internal walls are different.

Important:

If you are using one of the general multiphase models (VOF, mixture, or Eulerian), the procedure for copying boundary conditions is slightly different. See Steps for Copying Cell Zone and Boundary Conditions (p. 2974) for details.

7.1.6. Exporting Boundary Zones in CSV Format

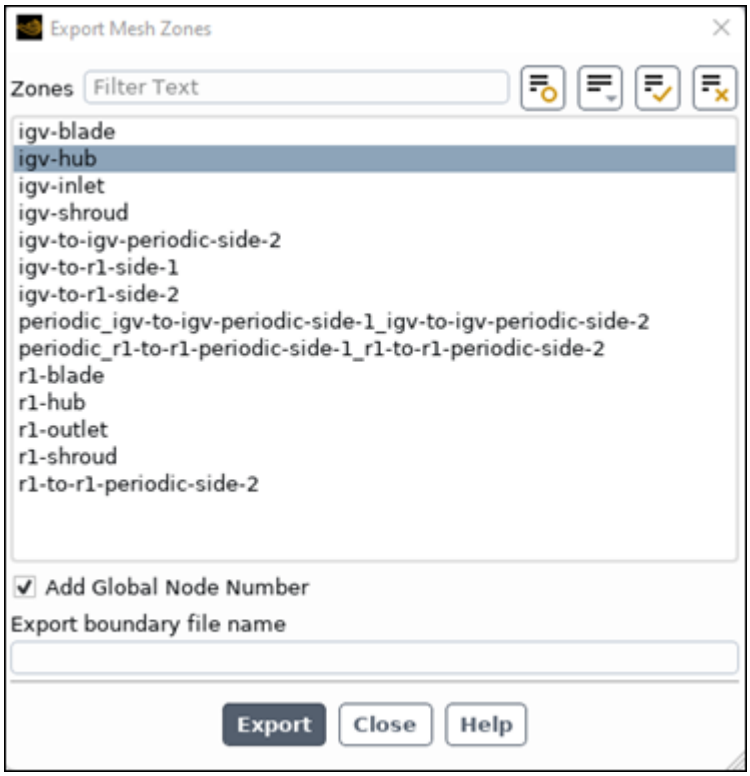
You can export a `.csv` file containing data from one or more boundaries by right-clicking one or more boundaries and selecting **Export CSV**.



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Clicking **Export CSV** for one or more boundaries opens the **Export Mesh Zones** dialog box as shown in Figure 7.3: Export Mesh Zones Dialog Box (p. 1307).

Figure 7.3: Export Mesh Zones Dialog Box



1. Under **Zones**. select the boundary zone(s) from which you would like to export mesh data.

2. Optionally select **Add Global Node Number**.

   Adds the "`Global Node Number`" column to the exported data. Doing so can help with the identification of shared nodes, for example, nodes shared among multiple exported surfaces. In particular, if you want the System Coupling Service to be able to identify which nodes are shared between multiple surfaces in the exported CSV data, you should select **Add Global Node Number**. For additional information, see System Coupling's Mapping Capabilities in the *System Coupling User's Guide*.

3. Under **Export boundary file name**, provide a file name for the exported mesh data. Note that `.csv` is appended to the name you provide.

4. Click **Export** to write the `.csv` to your working directory.

The same functionality is provided by entering the following command in the console and following the prompts:

```
/define/turbo-model/export-boundary-mesh
```

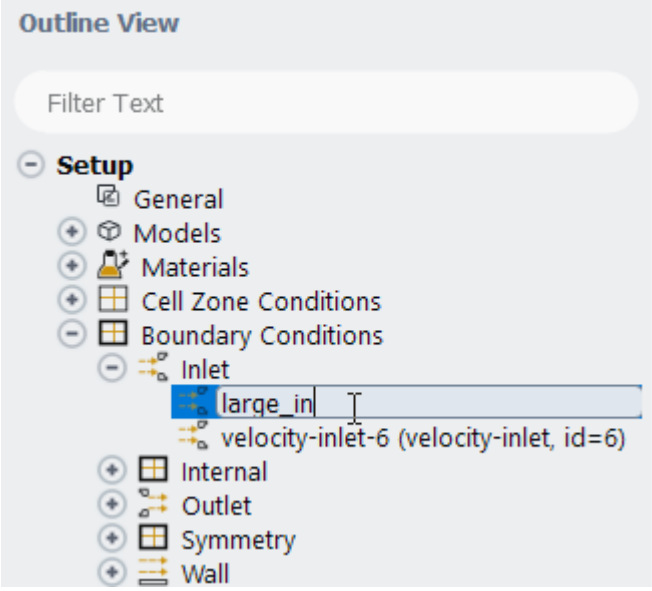
When exporting CSV data, the resulting `.csv` file contains surface meshes in a form that can be imported into CFX-Pre or CFD-Post as a user surface, or used in System Coupling mapping (see Aerodynamic Damping Analysis in the *System Coupling User's Guide*).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.1.7. Changing Cell or Boundary Zone Names

The default name for a zone is *<type>* **-** *<ID>* (for example, **pressure-inlet-7**). In some cases, you may want to assign more descriptive names to the boundary zones. If you have two pressure-inlet zones, for example, you might want to rename them **small-inlet** and **large-inlet**. (Changing the name of a zone will *not* change its type. Instructions for changing a zone's type are provided in Changing Cell and Boundary Zone Types (p. 1301).)

You can rename boundaries and cell zones directly within the tree, by selecting the zone that you want to rename, then clicking the name (slow double-click).



Alternatively, you can rename a zone by following these steps:

1. Right-click the name of zone in the tree (under **Setup**/**Boundary Conditions** or **Setup**/**Cell Zone Conditions**).
2. Select **Edit...** to open the dialog box for the selected zone.
3. Enter a new name under **Zone Name**.
4. Click the **OK** button.

Note:

If you specify a non-default name for a zone and then change its type, the name you specified will be retained; the automatic name change that accompanies a change in type occurs only for those with the default name.

7.1.8. Defining Non-Uniform Cell Zone and Boundary Conditions

Most conditions at each type of boundary zone can be defined as profile functions instead of constant values. You can use a profile contained in an externally generated profile file, or a function that you

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

create using a user-defined function (UDF). Profiles are described in Profiles (p. 1564), and user-defined functions are described in the separate Fluent Customization Manual.

7.1.9. Defining and Viewing Parameters

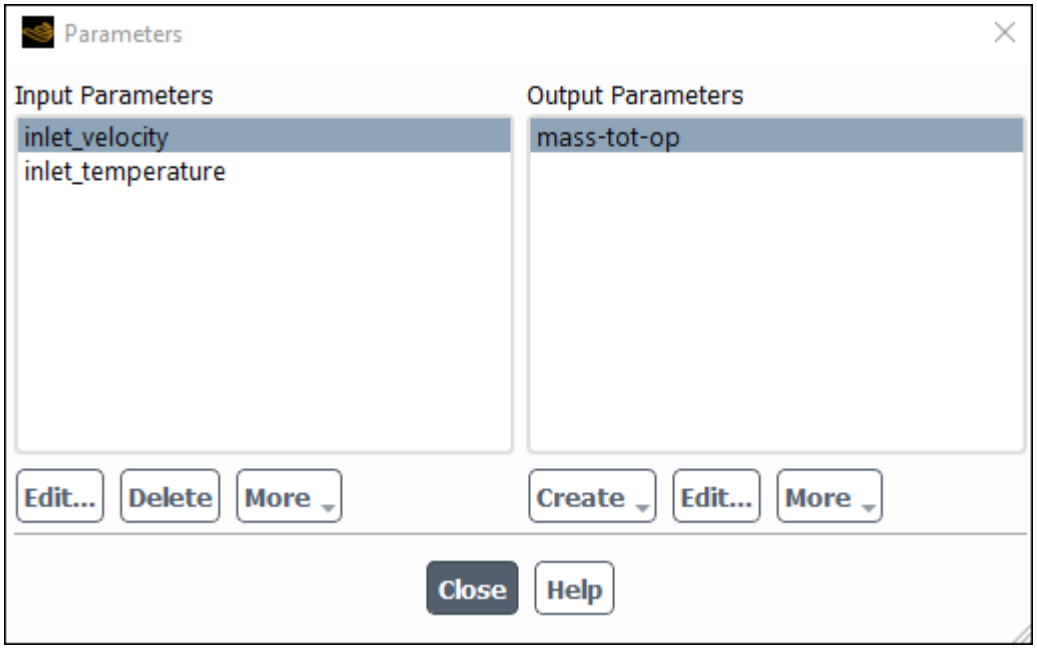
You can define a series of cases based on a set of parametric values. These parameters may be defined for numeric cell zone and boundary condition settings. This is especially useful if you are using Workbench and performing parametric studies (optimization), comparing cases with different boundary settings; information about such usage can be found in see Working with Parameters and Design Points in the Workbench User's Guide. If you are not running Ansys Fluent through Workbench, then you can use the parameter settings to define the same boundary condition value to different boundaries having the same units, or to use the **Gradient-Based Optimizer** to optimize a geometry for multiple objectives at multiple operating conditions (see Using the Gradient-Based Optimizer (p. 4583)).

Note:

For more information about using parameters with Ansys Fluent in Ansys Workbench, see Working With Input and Output Parameters in Workbench in the separate Fluent in Workbench User's Guide.

The parameters that you have defined in the various boundary condition dialog boxes are accessed by clicking the **Parameters...** button in the **Cell Zone Conditions** or **Boundary Conditions** task page. The **Parameters** dialog box will open, as shown in Figure 7.4: The Parameters Dialog Box (p. 1309), listing all of the input parameters that you have created in the various boundary condition dialog boxes.

Figure 7.4: The Parameters Dialog Box



In the **Parameters** dialog box, under the **Input Parameters** list, you can

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Edit the input properties using the **Parameter Expression** dialog box, which is the same dialog box used to create parameters. This dialog box can also be accessed by selecting **New Input Parameter...** from the drop-down lists in the boundary conditions dialog boxes, as described later in this section.

  Important:

  If you are using Ansys Fluent in Ansys Workbench, you cannot edit the input parameters, you can only view them. For more information, see the separate Ansys Fluent in Ansys Workbench User's Guide.

- Delete input parameters that are not assigned to a setting.
- More options exist under the **More** drop-down list (see Working With Advanced Parameter Options (p. 1313)):

  Use in Scheme Procedure

  displays the Use Input Parameter in Scheme Procedure Dialog Box (p. 5341) where you can apply an input parameter using a Scheme procedure.

  Use in UDF

  displays the Use Input Parameter for UDF Dialog Box (p. 5342) where you can make an input parameter available in a user-defined function.

In the **Parameters** dialog box, under the **Output Parameters** list, you can also

- Create output parameters. These are single values generated by existing reports or monitors. You can generate the output parameters from report definitions (surface, volume, lift, drag, and so on). These output parameters are discussed in greater detail in Creating Output Parameters (p. 4133).
- Edit existing output parameters.
- More options exist under the **More** drop-down list:

  Delete

  removes the selected output parameter from the list of **Output Parameters**.

  Rename

  allows you to edit the name of the output parameter through the **Rename** dialog box.

  Print to Console

  will output values to the console window. If you select multiple output parameters, then the output includes values from multiple output parameters.

  Print All to Console

  outputs the values from all output parameters to the console window.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Write...

allows you to store the output to a file. A dialog box is displayed allowing you to provide a file name.

Write All...

prompts you for a file name and then writes the values for all of the output parameters to a file.

Important:

Changing the units for a quantity changes the value for any input parameter using that quantity.

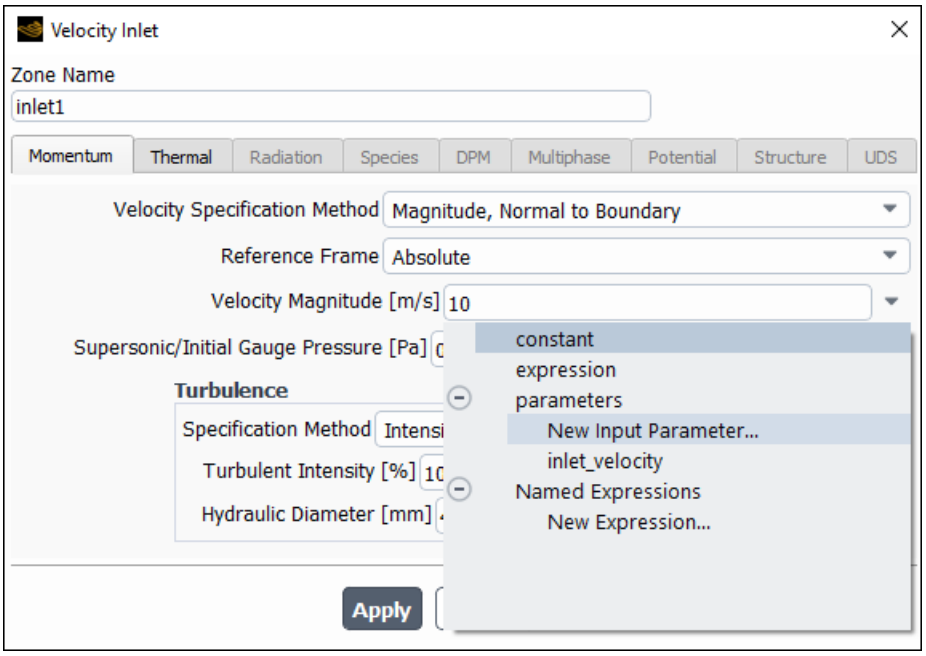
Note:

Various Ansys Fluent setup-related input quantities (of type real and profile) can be assigned to an input parameter (indicated by the **New Input Parameter...** option in the corresponding drop-down list or menu next to the field). Clicking this option opens the Figure 7.6: The Parameter Expression Dialog Box (p. 1312) where you can create the input parameter.

7.1.9.1. Creating a New Parameter

You can create a new cell zone or boundary condition parameter, as shown in Figure 7.5: The New Input Parameter... Selection (p. 1311).

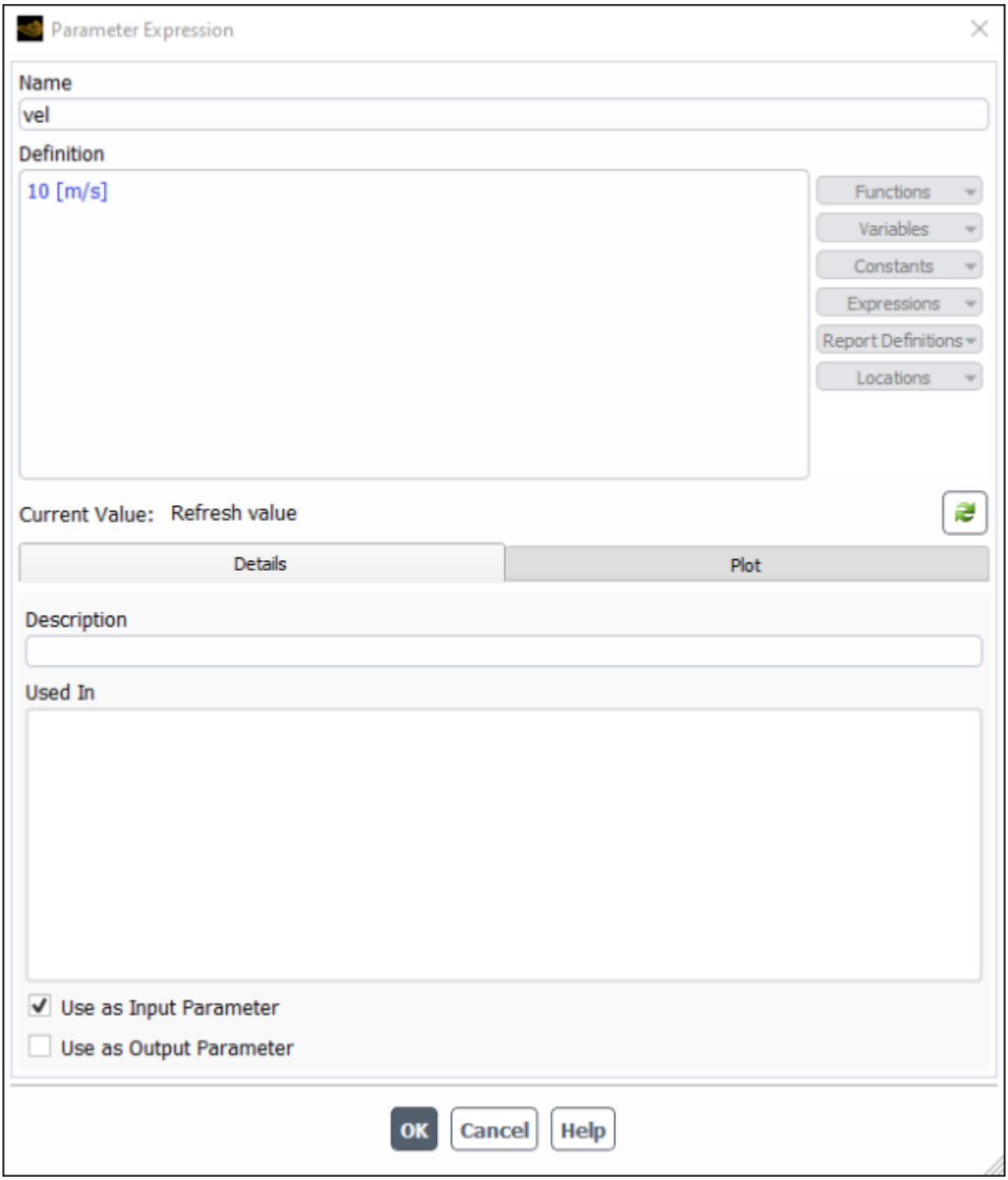
Figure 7.5: The New Input Parameter... Selection



When you select **New Input Parameter...** from the drop-down list, the **Parameter Expression** dialog box (Figure 7.6: The Parameter Expression Dialog Box (p. 1312)) will open where you will

- Enter the **Name** of the parameter expression.
- Specify the **Definition** as a constant.
- The **Used In** field displays information about where the parameter is utilized. Note that this field will be populated after you create an input parameter.

Figure 7.6: The Parameter Expression Dialog Box



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Once the parameter is defined in the **Parameter Expression** dialog box, the name of the parameter will appear in the drop-down list of the property you are defining, as seen in Figure 7.5: The New Input Parameter... Selection (p. 1311).

Important:

Input parameter expressions must be directly defined as a constant value including appropriate units; it may not reference another expression.

Note:

Ansys Fluent automatically creates generic default names for new input and output parameters (for example, `parameter_1`, `parameter_2`, `parameter_3`, and so on) If a parameter is deleted, the default name is not reused. For example, if you have `parameter_1`, `parameter_2`, and `parameter_3`, then delete `parameter_2` and create a new parameter, the default name for the new parameter will be `parameter_4`.

7.1.9.2. Working With Advanced Parameter Options

Various Ansys Fluent setup related input quantities can be assigned to an input parameter. You can define a series of simulations based on a set of parametric values that are managed both in Ansys Fluent and in Workbench. These parameters may be defined for numeric cell zone and boundary condition settings using the **New Input Parameter…** option in the corresponding drop-down list or menu adjacent to a specific input setting. However, various Ansys Fluent settings are not supported by these methods.

You can mitigate this limitation using input parameters through Scheme procedures and user-defined functions (UDFs), and define input parameters for various Ansys Fluent simulation related settings.

7.1.9.2.1. Defining Scheme Procedures With Input Parameters

7.1.9.2.2. Defining UDFs With Input Parameters

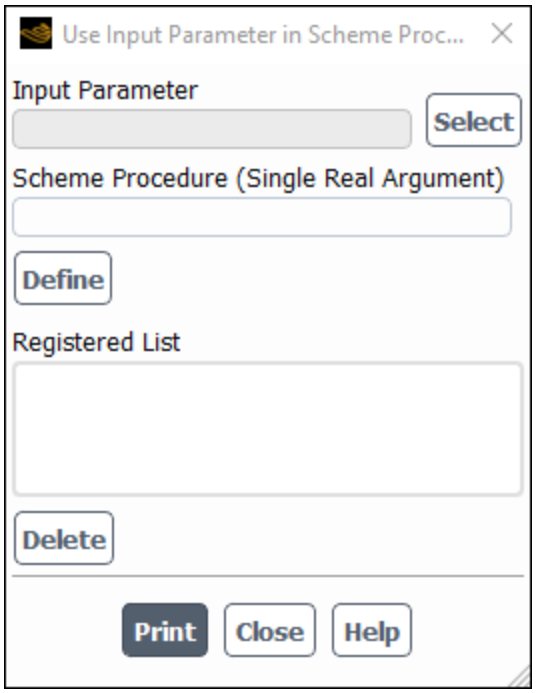
7.1.9.2.3. Using the Text User Interface to Define UDFs and Scheme Procedures With Input Parameters

7.1.9.2.1. Defining Scheme Procedures With Input Parameters

To use a Scheme procedure with input parameters, in the **Parameters** dialog box under **Input Parameters**, select **More**, then choose **Use in Scheme Procedure** to display the Use Input Parameter in Scheme Procedure Dialog Box (p. 5341) where you can register the input parameters that can be used in Scheme procedures during calculations. A text user interface command is also available, described in Using the Text User Interface to Define UDFs and Scheme Procedures With Input Parameters (p. 1316).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.7: Use Input Parameter in Scheme Procedure Dialog Box



Use the **Select** button to open the Select Input Parameter Dialog Box (p. 4980) where an input parameter can be chosen. Enter a **Scheme Procedure** name or body starting with `lambda` that should receive a real argument (the value of the selected input parameter). For example, if the following Scheme procedure called `my-funct` is defined as a Scheme file that gets loaded into the current session,

```
(define my-funct
 (lambda (value )
  (ti-menu-load-string (format #f "/solve/set/under-relaxation/pressure ~g" value))))
```

then enter `my-funct` for the **Scheme Procedure**. Otherwise, you can directly enter the following code for the **Scheme Procedure**:

```
(lambda (value)(ti-menu-load-string (args->string '/solve/set/under-relaxation/pressure value)))
```

Click **Define** to populate the **Registered List**. Use **Delete** to delete the use of the selected input variable, but not the associated input parameter (the input parameter has to be deleted separately in the **Parameters** dialog box). Use **Print** to print out details of the registered input parameters.

Important:

While writing Scheme procedures, you should use the "args->string" help procedure instead of using double quotes. For example:

```
(lambda ( value1)
 (display (args->string 'Changing-Theta-Divisions-value=   value1))
 (ti-menu-load-string
    (args->string '/define/models/radiation/discrete-ordinates? 'yes value1  '_  '_  '_)) )
```

Alternatively, you could also write:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
(lambda ( value1)
 (display (args->string 'Changing-Theta-Divisions-value=  value1))
 (ti-menu-load-string
   (apply args->string `(/define/models/radiation/discrete-ordinates? yes ,value1  _  _  _) )) )
```

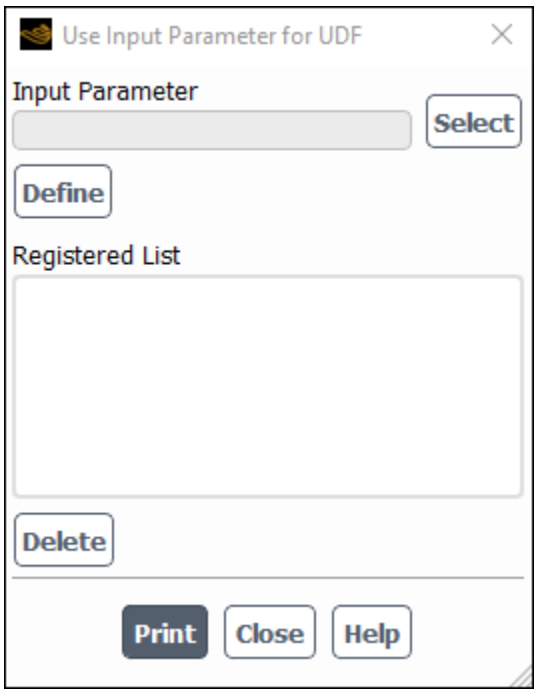
where the ` character after `args->string` is the grave accent character (not a single quote) and the _ character stores the default value of the current text user interface prompt. Note that the first occurrence of `value1` is the Scheme variable, while the second occurrence of `value1` will be replaced by the value stored in the variable. This method eliminates the need to specify a single quote for all arguments.

Also note that if you enter a scheme procedure body in the **Use Input Parameter in Scheme Procedure** dialog box, you should combine the scheme procedure lines into one single line.

7.1.9.2.2. Defining UDFs With Input Parameters

To use a UDF with input parameters, in the **Parameters** dialog box under **Input Parameters**, select **More**, then choose **Use in UDF** to display the Use Input Parameter for UDF Dialog Box (p. 5342) where you can register the input parameters that can be used in UDF functions during calculations. A text user interface command is also available, described in Using the Text User Interface to Define UDFs and Scheme Procedures With Input Parameters (p. 1316).

Figure 7.8: Use Input Parameter for UDF Dialog Box



Use the **Select** button to open the Select Input Parameter Dialog Box (p. 4980) where an input parameter can be chosen. Click **Define** to populate the **Registered List**. Use **Delete** to delete the use of the selected input variable, but not the associated input parameter (the input parameter has to be deleted separately in the **Parameters** dialog box). Click **Print** to print out the parameter ID of the registered input parameter. This ID should be used to access the value of the parameter in UDF.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

By registering the input parameter, Ansys Fluent knows that this input parameter is being used by a UDF. In the UDF C function, you can access the registered input parameter value using the `RP_Get_Input_Parameter` macro. For example,

```
real value = RP_Get_Input_Parameter("real-4"),
```

where `real-4` is an input parameter ID obtained using the **Print** feature.

7.1.9.2.3. Using the Text User Interface to Define UDFs and Scheme Procedures With Input Parameters

The `/define/parameters/input-parameters/advance/use-in` text interface command can also be used to define input parameters and Scheme procedures and user-defined functions.

The Scheme procedures can use other text user interface (TUI) commands in Ansys Fluent to change the desired simulation settings in a parametric manner. Each numerical component of the TUI command string can be marked and treated as a parameter. Setting up advanced input parameters requires using Scheme functions that use text interface commands to apply new values. Once defined, these input parameters are displayed in the **Parameters** dialog box alongside other input parameters. The input parameter passes a constant numeric value to the registered/provided Scheme function. Therefore, the associated Scheme function (and corresponding Ansys Fluent text command) uses the constant parameter values using the units that were already defined for the designated text command quantity.

`/define/parameters/input-parameters/advance`

Enter the advanced input parameters menu.

`use-in/`

Allows you to use an input parameter in a Scheme procedure or in a UDF. The following examples demonstrates how to use an input parameter using a Scheme procedure. The Scheme procedure body can be written at the prompt itself:

```
                                                /define/parameters/input-parameters/advance>
                                                use-in
                                                Select type:>
                                                scheme-proc
                                                Name of parameter ["parameter-1"]
parameter-1 value [0]
                                                0.3
                                                Enter the name/body of apply-function [()]
                                                (lambda (value )
        (ti-menu-load-string (format #f "/solve/set/under-relaxation/pressure ~g" value)))
```

or it can be written in a Scheme file (for example, `my-funct`)

```
                                                /define/parameters/custom-input-parameters>
                                                use-in
                                                Select type:>
                                                scheme-proc
                                                Name of parameter ["parameter-1"]
parameter-1 value [0]
                                                0.3
                                                Enter the name/body of apply-function [()]
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
                                                  my-funct
```

where the `my-funct` Scheme file contents are:

```
(define my-funct
    (lambda (value )
        (ti-menu-load-string (format #f "/solve/set/under-relaxation/pressure ~g" value))))
```

The following example demonstrates how to use an input parameter using a UDF:

```
                                                  /define/parameters/input-parameters/advance>
                                                  use-in
                                                  Select type:>
                                                  udf-side
                                                  Name of parameter ["parameter-1"]
parameter-1 value [0]
                                                  0.3
```

delete

Deletes the use of the selected input variable, but not the associated input parameter (the input parameter has to be deleted separately). Using the wildcard '*' allows you to delete all custom input variable at once.

list

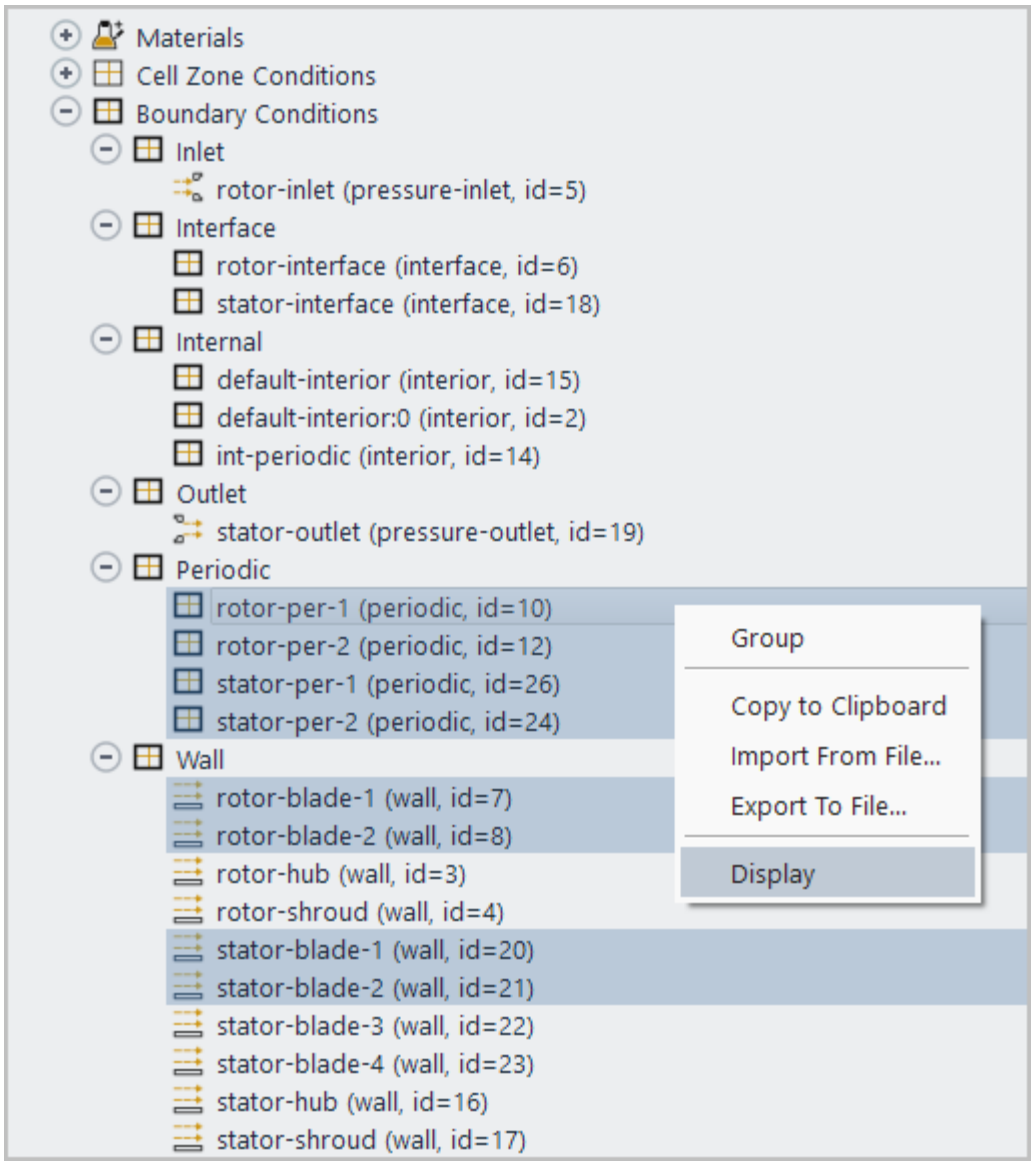
Shows usage of input parameters for Scheme procedures and for UDFs

7.1.10. Selecting Cell or Boundary Zones in the Graphics Display

During setup you will want to display boundary conditions, which can be accomplished using the Mesh Display Dialog Box (p. 4688). You also have the option to highlight boundaries in the tree, right-click them, and select **Display**, **Display in** or **Add to**. This feature is particularly useful if you are setting up a problem for the first time or if you want to visually confirm the location of a particular boundary condition.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

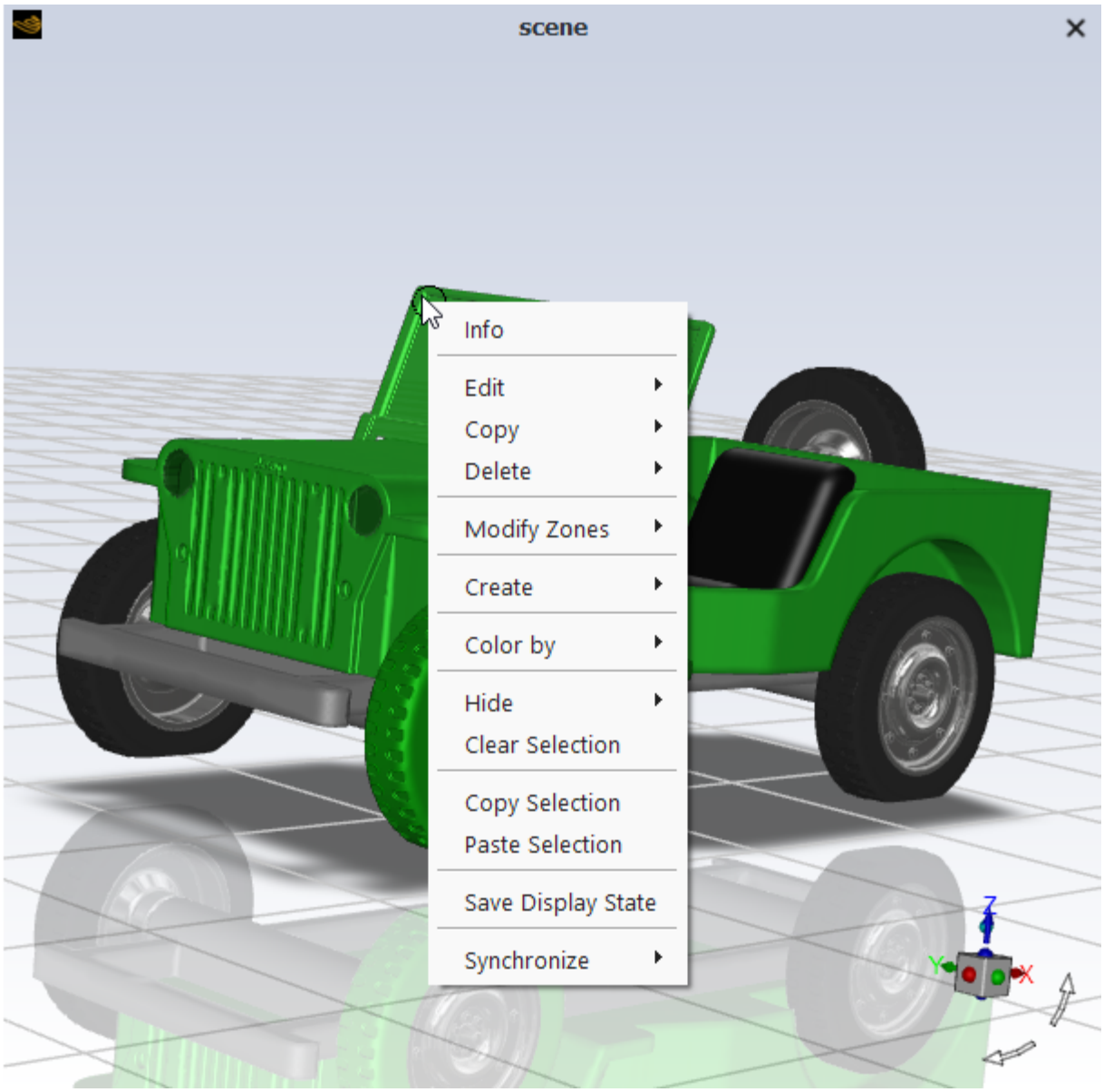
Figure 7.9: Selecting Multiple Boundaries for Display in the Graphics Window



The boundary description in the tree includes the zone IDs, which can help you differentiate the zones. You can also right-click selected boundaries in the graphics window to perform a number of options including printing information about the boundaries (surface IDs, zone IDs, surface groups, and so on), fusing zones, and creating graphics objects. For example, refer to Figure 7.10: Example Operations for Multiple Selected Surfaces in the Graphics Window (p. 1319), which shows some operations that are available for the selected surfaces (highlighted in light green).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.10: Example Operations for Multiple Selected Surfaces in the Graphics Window



7.1.11. Operating and Periodic Conditions

The **Cell Zone Conditions** and **Boundary Conditions** task pages allow you to access the **Operating Conditions** dialog box, where you can set the operating pressure, reference pressure location, include the effects of gravity, and specify other operating variables, as discussed in Modeling Basic Fluid Flow (p. 1737).

Setup → **Cell Zone Conditions** → **Operating Conditions...**

Setup → **Boundary Conditions** → **Operating Conditions...**

The **Periodic Conditions** dialog box can be accessed from the **Boundary Conditions** task page. For a detailed description of this dialog box's inputs, refer to Periodic Flows (p. 1747).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setup → **Boundary Conditions** → **Periodic Conditions...**

7.1.12. Saving and Reusing Cell Zone and Boundary Conditions

You can save cell zone and boundary conditions to a file so that you can use them to specify the same conditions for a different case, as described in Reading and Writing Boundary Conditions (p. 971).

7.2. Cell Zone Conditions

Cell zones consist of fluids and solids. Porous zones and 3D fans in Ansys Fluent are treated as fluid zones. A detailed description of the various cell zones is given in the sections that follow.

7.2.1. Fluid Conditions
7.2.2. Solid Conditions
7.2.3. Porous Media Conditions
7.2.4. 3D Fan Zones
7.2.5. Fixing the Values of Variables
7.2.6. Locking the Temperature for Solid and Shell Zones
7.2.7. Defining Mass, Momentum, Energy, and Other Sources

7.2.1. Fluid Conditions

A fluid zone is a group of cells for which all active equations are solved. The only required input for a fluid zone is the type of fluid material. You must indicate which material the fluid zone contains so that the appropriate material properties will be used.

Important:

If you are modeling multiphase flow, you will not specify the materials here; you will choose the phase material when you define the phases, as described in Defining the Phases for the VOF Model (p. 3009).

Important:

If you are modeling species transport and/or combustion, you can specify the material as either a mixture or a fluid. The mixture material has to be the same as that specified in the **Species Model** dialog box when you enable the model. The fluid zones, being of different material types, must not be contiguous.

Optional inputs allow you to set sources or fixed values of mass, momentum, heat (temperature), turbulence, species, and other scalar quantities. You can also define motion for the fluid zone. If there are rotationally periodic boundaries adjacent to the fluid zone, you must specify the rotation axis. If you are modeling turbulence using one of the k - ε models, the k - ω model, or the Spalart-Allmaras

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

model, you can choose to define the fluid zone as a laminar flow region. If you are modeling radiation using the DO model, you can specify whether or not the fluid participates in radiation.

Warning:

In general, disabling **Participates in Radiation** for fluid zones is not recommended, as it can produce erroneous results. There are rare cases when it is acceptable: for example, if the domain contains multiple fluid zones, disabling this option for zones where radiation is negligible may save computational time without affecting the results.

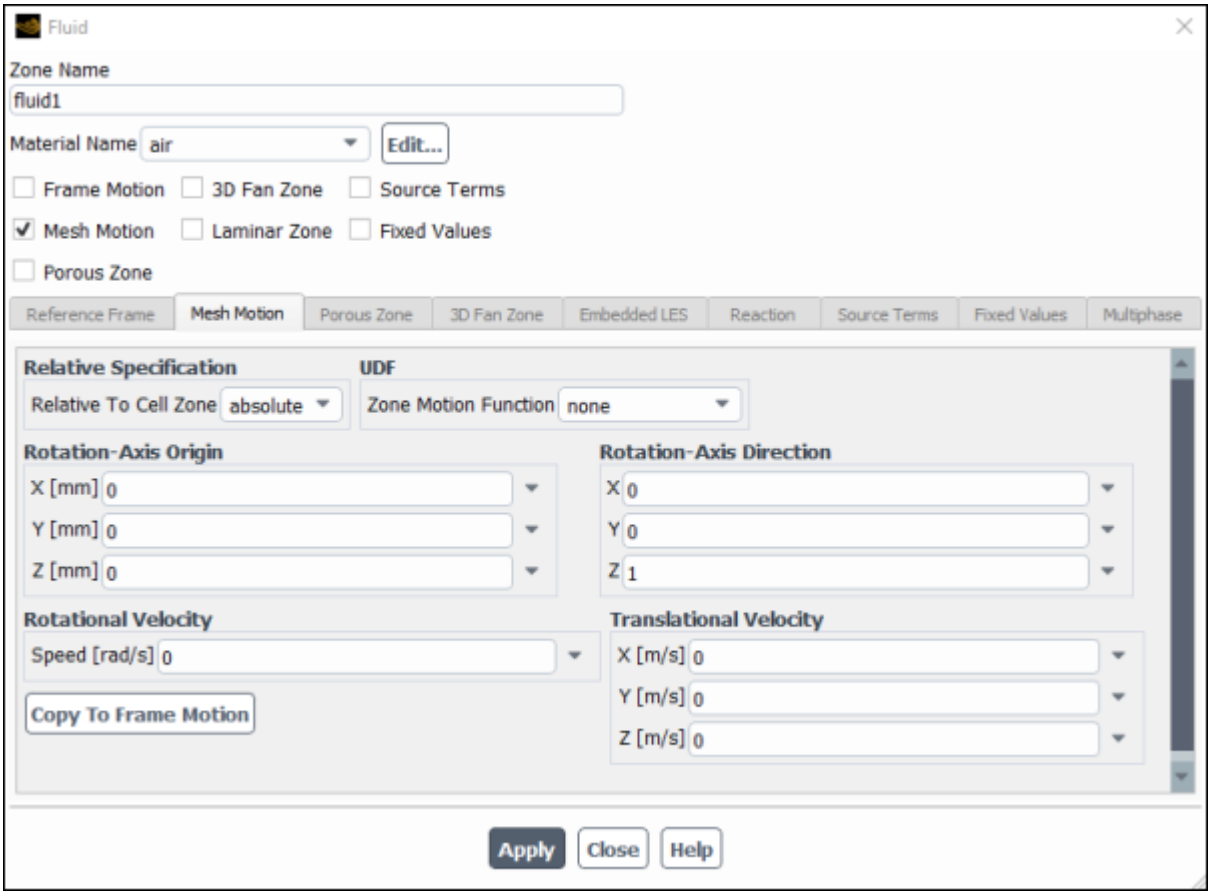
Important:

For information about porous zones and 3D fan zones, see Porous Media Conditions (p. 1333) and 3D Fan Zones (p. 1372), respectively.

7.2.1.1. Inputs for Fluid Zones

You will set all fluid conditions in the Fluid Dialog Box (p. 4961) (Figure 7.11: The Fluid Dialog Box (p. 1321)), which is accessed from the **Cell Zone Conditions** task page (as described in Setting Cell Zone and Boundary Conditions (p. 1303)).

Figure 7.11: The Fluid Dialog Box



7.2.1.1.1. Defining the Fluid Material

To define the material contained in the fluid zone, select the appropriate item in the **Material Name** drop-down list. This list will contain all fluid materials database) in the Create/Edit Materials Dialog Box (p. 4885). If you want to check or modify the properties of the selected material, you can click **Edit...** to open the **Edit Material** dialog box; this dialog box contains just the properties of the selected material, not the full contents of the standard **Create/Edit Materials** dialog box.

Important:

If you are modeling species transport or multiphase flow, the **Material Name** list will not appear in the **Fluid** dialog box. For species calculations, the mixture material for all fluid zones will be the material you specified in the Species Model Dialog Box (p. 4768). For multiphase flows, the materials are specified when you define the phases, as described in Defining the Phases for the VOF Model (p. 3009).

7.2.1.1.2. Defining Sources

If you want to define a source of heat, mass, momentum, turbulence, species, or other scalar quantity within the fluid zone, you can do so by enabling the **Source Terms** option. See Defining Mass, Momentum, Energy, and Other Sources (p. 1382) for details.

7.2.1.1.3. Defining Fixed Values

If you want to fix the value of one or more variables in the fluid zone, rather than computing them during the calculation, you can do so by enabling the **Fixed Values** option. See Fixing the Values of Variables (p. 1377) for details.

7.2.1.1.4. Specifying a Laminar Zone

When you are calculating a turbulent flow, it is possible to "turn off" turbulence modeling in a specific fluid zone. To disable turbulence modeling, turn on the **Laminar Zone** option in the **Fluid** dialog box. This is useful if you know that the flow in a certain region is laminar. For example, if you know the location of the transition point on an airfoil, you can create a laminar/turbulent transition boundary where the laminar cell zone borders the turbulent cell zone. This feature allows you to model turbulent transition on the airfoil.

By default, the **Laminar Zone** option will set the turbulent viscosity, μ_t, to zero and disable turbulence production in the fluid zone. Turbulent quantities will still be transported through the zone, but effects on fluid mixing and momentum will be ignored. If you want to keep the turbulent viscosity, you can do so using the text command `define/ boundary-conditions/fluid`. You will be asked if you want to `Set Turbulent Viscosity to zero within laminar zone?`. If your response is `no`, Ansys Fluent will set the production term in the turbulence transport equation to zero, but will retain a nonzero μ_t.

Disabling turbulence modeling in a fluid zone can be applied to all the turbulence models except the Large Eddy Simulation (LES) model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.2.1.1.5. Specifying a Reaction Mechanism

If you are modeling species transport with reactions, you can enable a reaction mechanism in a fluid zone by turning on the **Reaction** option and selecting an available mechanism from the **Reaction Mechanism** drop-down list. See Defining Zone-Based Reaction Mechanisms (p. 2387) for more information about defining reaction mechanisms.

7.2.1.1.6. Specifying the Rotation Axis

If there are rotationally periodic boundaries adjacent to the fluid zone or if the zone is rotating, either the mesh or its reference frame, you must specify the rotation axis. To define the axis for a moving reference frame problem, set the **Rotation-Axis Direction** and **Rotation-Axis Origin** under the **Reference Frame** tab. To define the axis for a moving mesh problem, set the **Rotation-Axis Direction** and **Rotation-Axis Origin** under the **Mesh Motion** tab.

Note:

If a frame motion and a mesh motion are specified at the same zone and this zone has rotationally periodic boundaries adjacent to it, then both axes have to be coaxial. Otherwise, the periodicity assumption is not valid and you will receive a warning message. In addition, the mesh check will fail.

The cell zone axis is independent of the axis of rotation used by any adjacent wall zones or any other cell zones. For 3D problems, the axis of rotation is the vector from the **Rotation-Axis Origin** in the direction of the vector given by your **Rotation-Axis Direction** inputs for the frame of reference and the mesh motion. For 2D non-axisymmetric problems, you will specify only the **Rotation-Axis Origin**; the axis of rotation is the z -direction vector passing through the specified point. (The z direction is normal to the plane of your geometry so that rotation occurs in the plane.) For 2D axisymmetric problems, you will not define the axis: the rotation will always be about the x axis, with the origin at (0,0).

7.2.1.1.7. Defining Zone Motion

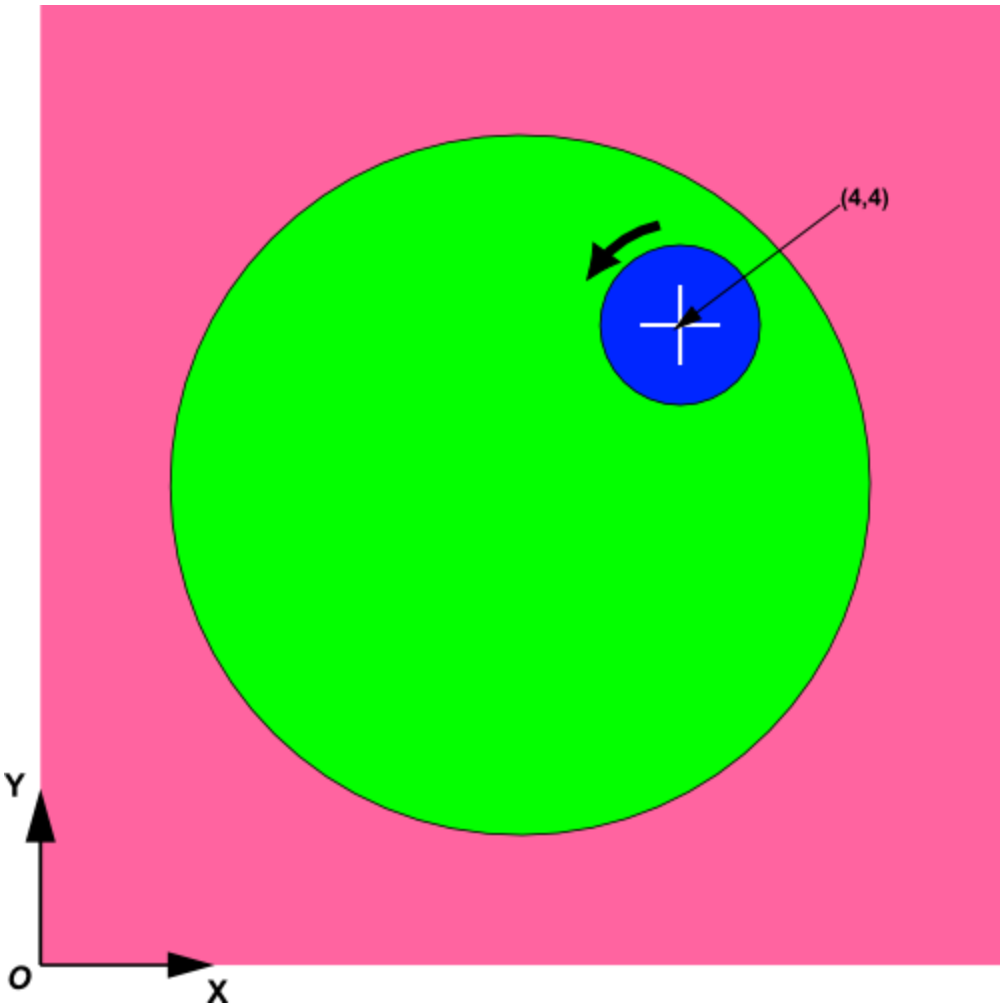
To define zone motion for a moving reference frame (MRF), enable the **Frame Motion** option in the **Fluid** dialog box. Set the appropriate parameters in the expanded portion of the dialog box, under the **Reference Frame** tab. For further details on such simulations, see Modeling Flows with Moving Reference Frames (p. 1769).

To define zone motion for a moving (sliding) mesh, enable the **Mesh Motion** option in the **Fluid** dialog box. Set the appropriate parameters in the expanded portion of the dialog box, under the **Mesh Motion** tab. See Setting Up the Sliding Mesh Problem (p. 1805) for details.

For cases that do not contain zones with motion specified in a relative frame to another zone, select **absolute** from the **Relative To Cell Zone** drop-down list. Here, the velocity and rotation components are specified in an **absolute** reference frame, which is the default setting, as shown in Figure 7.12: Rotation Specified in the Absolute Reference Frame (p. 1324). If no moving zones are present in the simulation, then **absolute** will be the only available selection. See The Multiple Reference Frame Model for more information.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.12: Rotation Specified in the Absolute Reference Frame

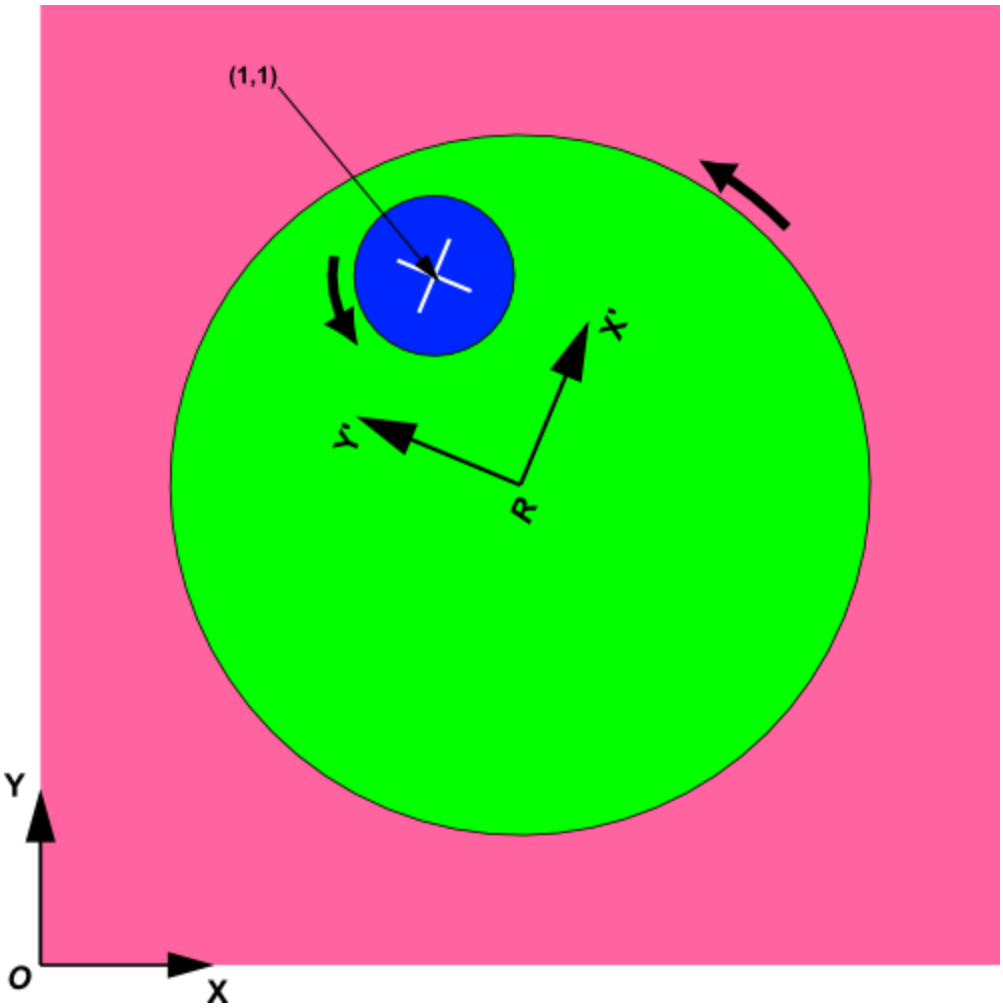


For cases where you have a moving zone specified relative to another moving zone, select the cell zone carrying the primary motion from the **Relative To Cell Zone** drop-down list under the **Reference Frame** tab or the **Mesh Motion** tab. Note that for such cases, **Rotation-Axis Origin (Relative)** will appear in the interface, signifying coordinates relative to the zone selected from the **Relative To Cell Zone** drop-down list.

Figure 7.13: Rotation Specified Relative to a Moving Zone (p. 1325) illustrates that the rotational axis origin of the small rotating zone is specified relative to the cell zone carrying the primary motion (having local coordinate system R).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.13: Rotation Specified Relative to a Moving Zone



Note:

The **Relative To Cell Zone** list will consist of all moving cell zones with an absolute motion specification (that is zones that are moving, but their motion is not relative to some other zone), excluding the current cell zone.

For problems that include linear, translational motion of the fluid zone, specify the **Translational Velocity** by setting the **X**, **Y**, and **Z** components under the **Mesh Motion** tab. For problems that include rotational motion, specify the rotational **Speed** under **Rotational Velocity**. The rotation axis is defined as described above. Note that the speed can be specified as a constant value or a transient profile. The transient profile may be in a file format, as described in Transient Cell Zone and Boundary Conditions (p. 1523), or a UDF macro (DEFINE_TRANSIENT_PROFILE). Specifying the individual velocities as either a profile or a UDF allows you to specify a single component of the frame motion individually. However, you can also specify the frame motion using a user-defined function. This may prove to be quite convenient if you are modeling a more complicated

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

motion of the moving reference frame, where the hooking of many different user-defined functions or profiles can be cumbersome.

Important:

If you need to switch between the MRF and moving mesh models, simply click the **Copy To Mesh Motion** for zones with a moving frame of reference and **Copy to Frame Motion** for zones with moving meshes to transfer motion variables, such as the axes, frame origin, and velocity components between the two models. The variables used for the origin, axis, and velocity components, as well as for the UDF `DEFINE_ZONE_MOTION` will be copied. This is particularly useful if you are doing a steady-state MRF simulation to obtain an initial solution for a transient Moving Mesh simulation in a turbomachine.

Note that when a fluid zone undergoes normal zone motion relative to an adjacent solid zone, energy transfer at that boundary (that is, the total heat transfer rate) includes pressure work when the absolute velocity formulation is used. This pressure work is on the fluid side only, not on the solid side. For details on reporting the total heat transfer rate and/or pressure work rate, see Generating a Flux Report (p. 4136).

See Modeling Flows with Moving Reference Frames (p. 1769) for details about modeling flows in moving reference frames. Details about the frame motion UDF can be found in `DEFINE_ZONE_MOTION` in the Fluent Customization Manual.

7.2.1.1.8. Defining Radiation Parameters

If you are using the DO radiation model, you can specify whether or not the fluid zone participates in radiation using the **Participates in Radiation** option. See Defining Boundary Conditions for Radiation (p. 2217) for details.

7.2.2. Solid Conditions

A "solid" zone is a group of cells for which no flow equations are solved: in a fluid simulation, only a heat conduction problem is solved; in an intrinsic fluid-structure interaction (FSI) simulation, displacement and stress tensor components are calculated. The material being treated as a solid may actually be a fluid, but it is assumed that no convection is taking place. The only required input for a solid zone is the type of solid material. You must indicate which material the solid zone contains so that the appropriate material properties will be used. Optional inputs allow you to set a volumetric heat generation rate (heat source) or a fixed value of temperature. You can also define motion for the solid zone. If there are rotationally periodic boundaries adjacent to the solid zone, you must specify the rotation axis. If you are modeling radiation using the DO model, you can specify whether or not the solid material participates in radiation.t.

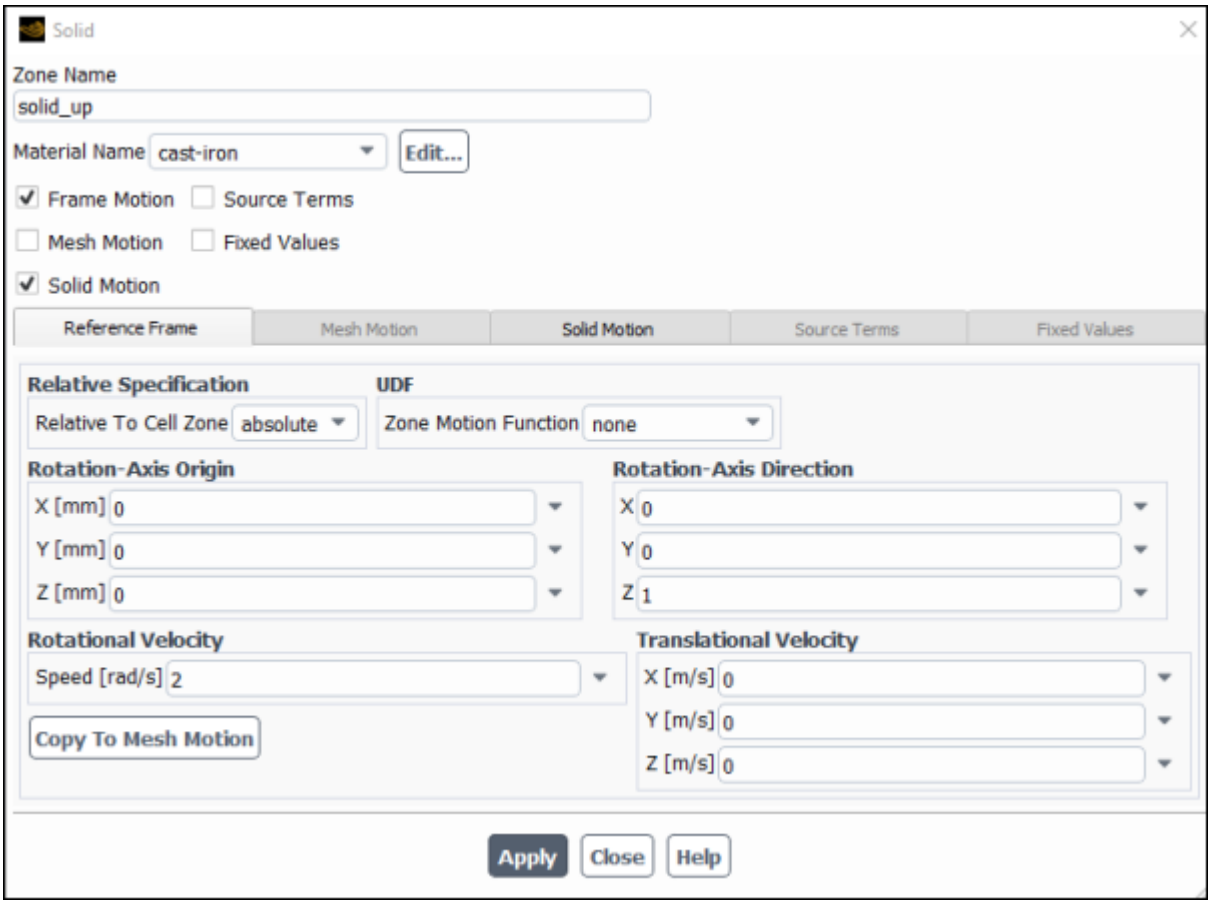
Note that the solver will always visit the active equations, including flow and possibly turbulence equations, during solver iterations. This will happen even if all the cell zones existing in the domain are solid and there is no fluid zone in the domain. In order to improve the solver performance, you may want to deactivate the flow, and turbulence equations manually when there is only solid cell zones in the domain and fluid properties are not important.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.2.2.1. Inputs for Solid Zones

You will set all solid conditions in the Solid Dialog Box (p. 4973) (Figure 7.14: The Solid Dialog Box (p. 1327)), which is opened from the **Cell Zone Conditions** task page (as described in Setting Cell Zone and Boundary Conditions (p. 1303)).

Figure 7.14: The Solid Dialog Box



7.2.2.1.1. Defining the Solid Material

To define the material contained in the solid zone, select the appropriate item in the **Material Name** drop-down list. This list will contain all solid materials database) in the Create/Edit Materials Dialog Box (p. 4885). If you want to check or modify the properties of the selected material, you can click **Edit...** to open the **Edit Material** dialog box; this dialog box contains just the properties of the selected material, not the full contents of the standard **Create/Edit Materials** dialog box.

7.2.2.1.2. Defining a Heat Source

If you want to define a source of heat within the solid zone, you can do so by enabling the **Source Terms** option. See Defining Mass, Momentum, Energy, and Other Sources (p. 1382) for details.

7.2.2.1.3. Defining a Fixed Temperature

If you want to fix the value of temperature in the solid zone, rather than computing it during the calculation, you can do so by enabling the **Fixed Values** option. See Fixing the Values of Variables (p. 1377) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.2.2.1.4. Specifying the Rotation Axis for Boundary Zones

If there are rotationally periodic boundaries adjacent to the solid zone and you are not defining zone motion (as described in Defining Zone Motion (p. 1328)), you must specify the cell zone axis by defining the **Rotation-Axis Origin** and **Rotation-Axis Direction** in the **Reference Frame** tab.

The cell zone axis is independent of the axis of rotation used by any adjacent wall zones or any other cell zones. For 3D problems, the axis of rotation is the vector from the **Rotation-Axis Origin** in the direction of the vector given by your **Rotation-Axis Direction**. For 2D non-axisymmetric problems, you will specify only the **Rotation-Axis Origin**; the axis of rotation is the z -direction vector passing through the specified point. (The z direction is normal to the plane of your geometry so that rotation occurs in the plane.) For 2D axisymmetric problems, you will not define the axis: the rotation will always be about the x axis, with the origin at (0,0).

7.2.2.1.5. Defining Zone Motion

When defining motion for a solid zone, you have the following options:

- You can enable the **Frame Motion** option for a moving reference frame (MRF) simulation only if the solid zone is not moving relative to an adjacent solid zone. Then define the settings in the **Reference Frame** tab. For further details on such simulations, see Modeling Flows with Moving Reference Frames (p. 1769).
- You can enable the **Solid Motion** option if the solid zone is moving relative to an adjacent solid zone. Then define the settings in the **Solid Motion** tab.

  To complete the setup for such a simulation, you must also define the velocity boundary conditions at coupled walls between the solid zone and any adjacent fluid zones; note that this is not necessary for the walls between the solid zone and any adjacent solid zones. For the fluid zone wall, you can choose to define the wall velocity based on the solid zone motion either as an absolute velocity or relative to the adjacent fluid zone. Details about these inputs are presented in Velocity Conditions for Moving Walls (p. 1462). When the energy equation is enabled, after you define the translational or rotational velocity for the solid in the **Solid Mesh** tab (as described in the steps that follow), you should enable the **Boundary Advection** option for the boundary walls through which heat is advected into or out of the domain to a significant degree; for details, see Boundary Advection for Solid Motion (p. 1474).
- You can enable the **Mesh Motion** option for a moving (sliding) mesh, whether or not the solid zone is moving relative to an adjacent solid zone. Then define the settings in the **Mesh Motion** tab. For further details on such simulations, see Setting Up the Sliding Mesh Problem (p. 1805).

When defining the settings in the **Reference Frame** tab, **Moving Mesh** tab, and/or **Solid Mesh** tab, perform the following steps:

1. Make a selection from the **Relative To Cell Zone** drop-down list, in a manner similar to that described for fluid zones in Defining Zone Motion (p. 1323).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Define the **Rotation-Axis Origin** and **Rotation-Axis Direction** for the motion.

   Note:

   If mesh motion is specified in the same zone as frame motion and/or solid motion and this zone has rotationally periodic boundaries adjacent to it, then both axes have to be coaxial. Otherwise, the periodicity assumption is not valid and you will receive a warning message. In addition, the mesh check will fail.

   The only time it is possible to have separate axes defined is when combining solid motion with frame motion. For example, you could simulate an automotive disc brake assembly where a spinning rotor that is in contact with a brake pad is modeled using solid motion, while the rotation of the assembly about the vertical axis (due to input from the steering wheel) is modeled using frame motion (see Figure 7.19: Two solids in contact where one is stationary and the other is rotating. The rotational motion of the moving solid should be described in the Solid Motion tab. Both solids may also have rotation about the y-axis described in the Frame Motion tab. (p. 1332)).

   The cell zone axis is independent of the axis used by any adjacent wall zones or any other cell zones. For 3D problems, the axis is the vector from the **Rotation-Axis Origin** in the direction of the vector given by your **Rotation-Axis Direction**. For 2D non-axisymmetric problems, you will specify only the **Rotation-Axis Origin**; the axis of rotation is the z -direction vector passing through the specified point. (The z direction is normal to the plane of your geometry so that rotation occurs in the plane.) For 2D axisymmetric problems, you will not define the axis: the rotation will always be about the x axis, with the origin at (0,0).

3. For problems that include linear, translational motion of the solid zone, specify the **Translational Velocity** by setting the **X**, **Y**, and **Z** components. For problems that include rotational motion, specify the rotational **Speed** under **Rotational Velocity**. The rotation axis is defined as described above. Note that the speed can be specified as a constant value or a transient profile. The transient profile may be in a file format, as described in Transient Cell Zone and Boundary Conditions (p. 1523), or a UDF macro (`DEFINE_TRANSIENT_PROFILE`). Specifying the individual velocities as either a profile or a UDF allows you to specify a single component of the motion individually. However, you can also specify the motion using a user-defined function that uses the UDF macro `DEFINE_ZONE_MOTION`. This may prove to be quite convenient if you are modeling a more complicated motion, where the hooking of many different user-defined functions or profiles can be cumbersome.

   Important:

   Note that simultaneous translation and rotation can be modeled only if the rotation axis and the translation direction are the same (that is, the origin is fixed).

4. Note that if you decide to switch between the motion options for a solid zone, the user interface makes it easy to transfer motion variables, such as the axes, frame origin, and velocity components between the models:

   - If you need to switch from frame motion or solid motion to mesh motion, click the **Copy To Mesh Motion** button. This is particularly useful if you are doing a steady-state MRF

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

simulation to obtain an initial solution for a transient moving mesh simulation in a turbomachine.

- If you need to switch from mesh motion to frame motion, simply click the **Copy to Frame Motion** button.
- If you need to switch from frame motion to solid motion, enter the following text command:

  `mesh → modify-zones → convert-all-solid-mrf-to-solid-motion`

  Note that this text command applies to all solid zones with frame motion enabled. It can be useful if you are updating a frame motion simulation that was set up in a previous release before solid motion was available.

For frame motion and/or solid motion, the velocity must be divergence-free, which requires that the specified motion be tangential to the boundaries. If the motion has a normal component, the solver cannot achieve energy conservation. The only exception, for which normal motion is permitted, is a coupled wall or mesh interface between two solid zones having the same material and motion. The solver will print a warning to the solver transcript the first time it encounters a solid velocity having a significant normal component at any other boundary type. The following examples illustrate valid solid zone motion definitions:

Figure 7.15: Single Rotating Solid Zone

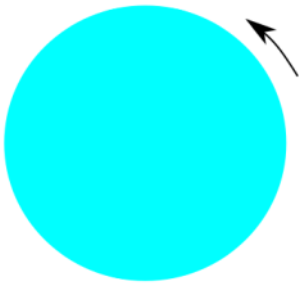
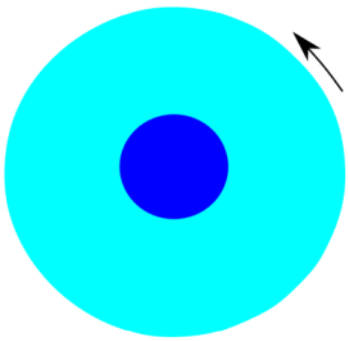


Figure 7.16: Rotating solid zone separated from another fluid or solid zone separated by a surface of revolution.



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.17: Multiple rotating solid zones having the same material and motion specifications, separated by mesh interfaces or coupled walls.

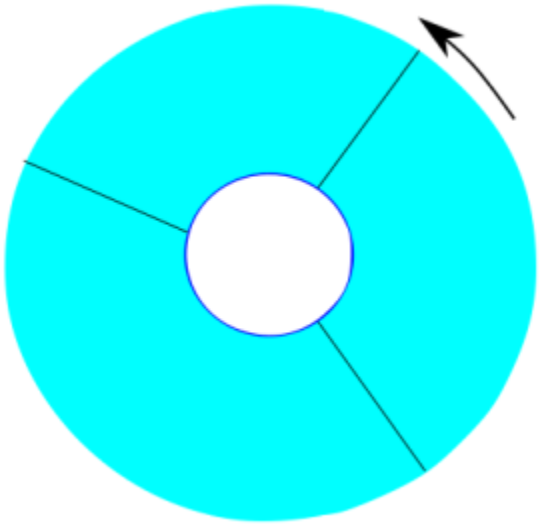
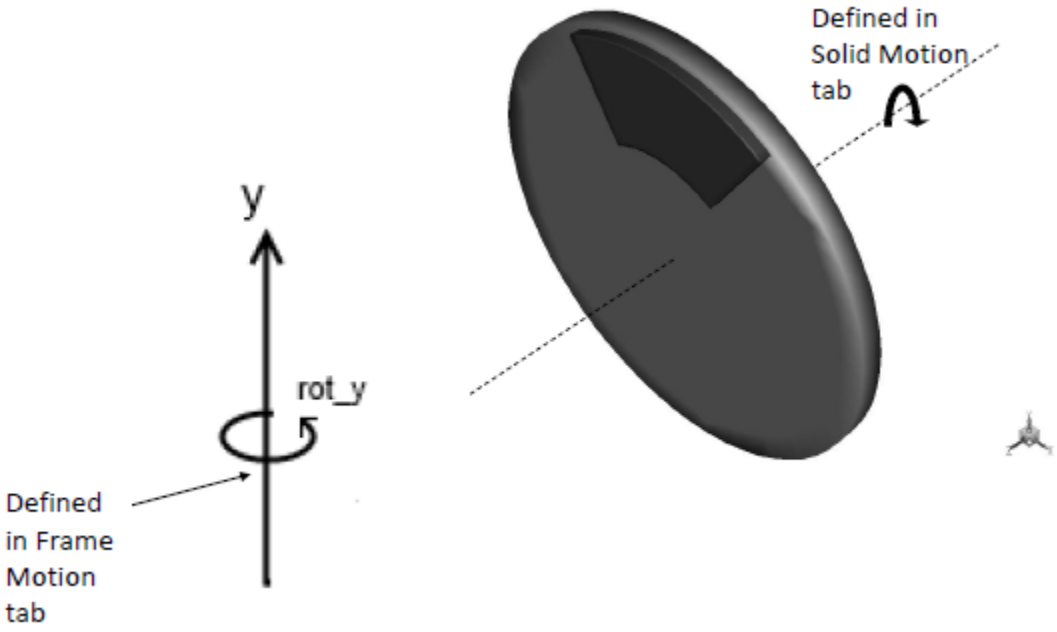


Figure 7.18: Two solids in contact where one is stationary and the other is moving with translational motion. The translational motion of the moving solid should be described in the Solid Motion tab.

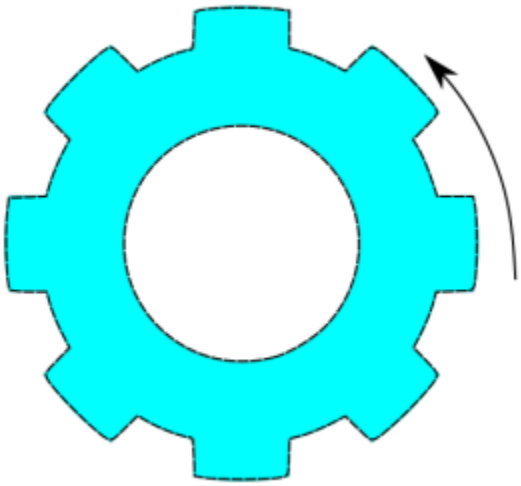
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.19: Two solids in contact where one is stationary and the other is rotating. The rotational motion of the moving solid should be described in the Solid Motion tab. Both solids may also have rotation about the y-axis described in the Frame Motion tab.



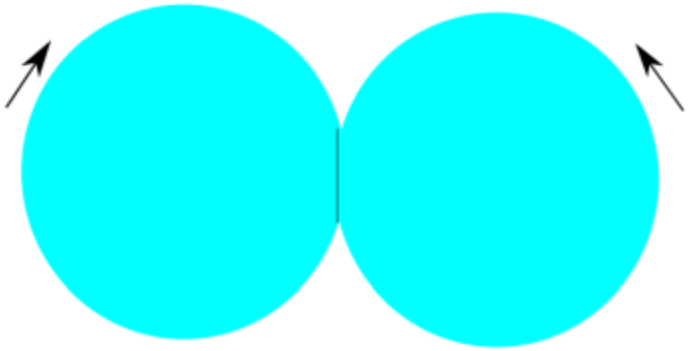
The following examples illustrate invalid frame motion or solid motion definitions for solid zones:

Figure 7.20: Rotating solid with boundaries which are not tangential to the motion.



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.21: Two solids in contact with some squish. At the contact, the rotational motion has some normal component, so the solver will not achieve global energy conservation. However, the temperature field might still be acceptable for engineering purposes.



Limitations:

- The effects of motion on the solid zone are applied only to the energy equation and the structural model calculations. User-defined scalars do not include the effects of motion.
- Motion through a boundary zone adjacent to the solid zone is not permitted, though you can define a wall boundary with specified temperature condition; that is, there is no support for a 'solid inlet' or 'solid outlet'.
- The **Solid Motion** option must not be enabled in a cell zone that has dynamic smoothing and/or layering enabled.

7.2.2.1.6. Defining Radiation Parameters

If you are using the DO radiation model, you can specify whether or not the solid material participates in radiation using the **Participates in Radiation** option. See Defining Boundary Conditions for Radiation (p. 2217) for details.

7.2.3. Porous Media Conditions

The porous media model can be used for a wide variety of single phase and multiphase problems, including flow through packed beds, filter papers, perforated plates, flow distributors, and tube banks. When you use this model, you define a cell zone in which the porous media model is applied and the pressure loss in the flow is determined via your inputs as described in Momentum Equations for Porous Media (p. 1335). Heat transfer through the medium can also be represented, with or without the assumption of thermal equilibrium between the medium and the fluid flow (as described in Treatment of the Energy Equation in Porous Media (p. 1337)).

A 1D simplification of the porous media model, termed the "porous jump," can be used to model a thin membrane with known velocity/pressure-drop characteristics. The porous jump model is applied to a face zone, not to a cell zone, and should be used (instead of the full porous media model) whenever possible because it is more robust and yields better convergence. See Porous Jump Boundary Conditions (p. 1519) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.2.3.1. Limitations and Assumptions of the Porous Media Model

The porous media model incorporates an empirically determined flow resistance in a region of your model defined as "porous". In essence, the porous media model adds a momentum sink in the governing momentum equations. Consequently, the following modeling assumptions and limitations should be readily recognized:

- Since the volume blockage that is physically present is not represented in the model, by default Ansys Fluent uses and reports a superficial velocity inside the porous medium, based on the volumetric flow rate, to ensure continuity of the velocity vectors across the porous medium interface. This superficial velocity formulation does not take porosity into account when calculating the convection and diffusion terms of the transport equations. You can choose to use a more accurate alternative, in which the true (physical) velocity is calculated inside the porous medium and porosity is included in the differential terms of the transport equations. See Modeling Porous Media Based on Physical Velocity (p. 1340) for details. In a multiphase flow system, all phases share the same porosity.

- The effect of the porous medium on the turbulence field is only approximated. See Treatment of Turbulence in Porous Media (p. 1339) for details.

- In general, the Ansys Fluent porous medium model, for both single phase and multiphase, assumes the porosity is isotropic, and it can vary with space and time.

- The Superficial Velocity Formulation and the Physical Velocity Formulation are available for multiphase porous media. See User Inputs for Porous Media (p. 1343) for details.

- The porous media momentum resistance and heat source terms are calculated separately on each phase. See Momentum Equations for Porous Media (p. 1335) for details.

- A unique pressure interpolation scheme is always used inside porous media zones regardless of the pressure scheme selected in the **Solution Methods** task page.

- The interactions between a porous medium and shock waves are not considered.

- When applying the porous media model in a moving reference frame, Ansys Fluent will either apply the relative reference frame or the absolute reference frame when you enable the **Relative Velocity Resistance Formulation**. This allows for the correct prediction of the source terms.

- **Standard Initialization** is the recommended initialization method for porous media simulations. The default **Hybrid Initialization** method does not account for the porous media properties, and depending on boundary conditions, may produce an unrealistic initial velocity field. For porous media simulations, the **Hybrid Initialization** method can only be used if the **Maintain Constant Velocity Magnitude** option is selected in the **Hybrid Initialization** dialog box.

- The physical velocity porous formulation may produce non-physical flow fields and poor convergence when porous resistance (Inertial or Viscous) values are less than or comparable to the change in the dynamic pressure across the porous interface (interior face zone separating the porous and non-porous cell zone). Switching the porous interface zone to a porous jump boundary is an effective way to overcome this issue.

- Solid motion is not supported in a solid zone that is coincident with a porous fluid zone that uses the non-equilibrium thermal model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The Non-Premixed Combustion model is not compatible for use in porous fluid zone that uses the non-equilibrium thermal model.

7.2.3.2. Momentum Equations for Porous Media

The porous media models for single phase flows and multiphase flows use the **Superficial Velocity Porous Formulation** as the default. Ansys Fluent calculates the superficial phase or mixture velocities based on the volumetric flow rate in a porous region. The porous media model is described in the following sections for single phase flow, however, it is important to note the following for multiphase flow:

- In the Eulerian multiphase model (Eulerian Model Theory in the Theory Guide), the general porous media modeling approach, physical laws, and equations described below are applied to the corresponding phase for mass continuity, momentum, energy, and all the other scalar equations.
- The **Superficial Velocity Porous Formulation** generally gives good representations of the bulk pressure loss through a porous region. However, since the superficial velocity values within a porous region remain the same as those outside the porous region, it cannot predict the velocity increase in porous zones and therefore limits the accuracy of the model.

Porous media are modeled by the addition of a momentum source term to the standard fluid flow equations. The source term is composed of two parts: a viscous loss term (Darcy, the first term on the right-hand side of Equation 7.1 (p. 1335), and an inertial loss term (the second term on the right-hand side of Equation 7.1 (p. 1335))

$$S_i = -\left(\sum_{j=1}^{3} D_{ij} \mu v_j + \sum_{j=1}^{3} C_{ij} \frac{1}{2} \rho |v| v_j \right) \tag{7.1}$$

where S_i is the source term for the i th (x, y, or z) momentum equation, $|v|$ is the magnitude of the velocity and D and C are prescribed matrices. This momentum sink contributes to the pressure gradient in the porous cell, creating a pressure drop that is proportional to the fluid velocity (or velocity squared) in the cell.

Computation of forces on porous zones is based on viscous and inertial loss terms. The definition of these terms is in Equation 7.1 (p. 1335). Forces in simple homogenous porous media are defined in Equation 7.2 (p. 1335). Setting viscous and inertial resistance coefficients will determine the properties of the momentum sink, and subsequently the forces.

To recover the case of simple homogeneous porous media

$$S_i = -\left(\frac{\mu}{\alpha} v_i + C_2 \frac{1}{2} \rho |v| v_i \right) \tag{7.2}$$

where α is the permeability and C_2 is the inertial resistance factor, simply specify D and C as diagonal matrices with $1/\alpha$ and C_2, respectively, on the diagonals (and zero for the other elements).

Ansys Fluent also allows the source term to be modeled as a power law of the velocity magnitude:

$$S_i = -C_0 |v|^{C_1} = -C_0 |v|^{(C_1 - 1)} v_i \tag{7.3}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where C_0 and C_1 are user-defined empirical coefficients.

Important:

In the power-law model, the pressure drop is isotropic and the units for C_0 are SI.

7.2.3.2.1. Darcy's Law in Porous Media

In laminar flows through porous media, the pressure drop is typically proportional to velocity and the constant C_2 can be considered to be zero. Ignoring convective acceleration and diffusion, the porous media model then reduces to Darcy's Law:

$$\nabla p = -\frac{\mu}{\alpha}\vec{v} \tag{7.4}$$

The pressure drop that Ansys Fluent computes in each of the three (x, y, z) coordinate directions within the porous region is then

$$\begin{aligned} \Delta p_x &= \sum_{j=1}^{3} \frac{\mu}{\alpha_{xj}} v_j \Delta n_x \\ \Delta p_y &= \sum_{j=1}^{3} \frac{\mu}{\alpha_{yj}} v_j \Delta n_y \\ \Delta p_z &= \sum_{j=1}^{3} \frac{\mu}{\alpha_{zj}} v_j \Delta n_z \end{aligned} \tag{7.5}$$

where $1/\alpha_{ij}$ are the entries in the matrix D in Equation 7.1 (p. 1335), v_j are the velocity components in the x, y, and z directions, and Δn_x, Δn_y, and Δn_z are the thicknesses of the medium in the x, y, and z directions.

Here, the thickness of the medium (Δn_x, Δn_y, or Δn_z) is the *actual* thickness of the porous region in your model. Therefore if the thicknesses used in your model differ from the actual thicknesses, you must make the adjustments in your inputs for $1/\alpha_{ij}$.

7.2.3.2.2. Inertial Losses in Porous Media

At high flow velocities, the constant C_2 in Equation 7.1 (p. 1335) provides a correction for inertial losses in the porous medium. This constant can be viewed as a loss coefficient per unit length along the flow direction, thereby allowing the pressure drop to be specified as a function of dynamic head.

If you are modeling a perforated plate or tube bank, you can sometimes eliminate the permeability term and use the inertial loss term alone, yielding the following simplified form of the porous media equation:

$$\nabla p = -\sum_{j=1}^{3} C_{2_{ij}} \left(\frac{1}{2} \rho v_j |v| \right) \tag{7.6}$$

or when written in terms of the pressure drop in the x, y, z directions:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\begin{aligned}\Delta p_x &\approx \sum_{j=1}^{3} C_{2_{xj}} \Delta n_x \tfrac{1}{2} \rho v_j |v| \\ \Delta p_y &\approx \sum_{j=1}^{3} C_{2_{yj}} \Delta n_y \tfrac{1}{2} \rho v_j |v| \\ \Delta p_z &\approx \sum_{j=1}^{3} C_{2_{zj}} \Delta n_z \tfrac{1}{2} \rho v_j |v|\end{aligned} \tag{7.7}$$

Again, the thickness of the medium (Δn_x, Δn_y, or Δn_z) is the thickness you have defined in your model.

7.2.3.3. Relative Viscosity in Porous Media

For viscous flows, an effective viscosity, μ_e, is introduced to account for the effect of the porous medium on the diffusion term in the momentum equations:

$$\mu_e = \mu_r \mu \tag{7.8}$$

where μ is the fluid viscosity and μ_r is the relative viscosity. In Fluent the following sub-models are implemented to calculate the relative viscosity. The effective viscosity is then calculated from Equation 7.8 (p. 1337).

Brinkman Correction [26] (p. 5714)

$$\mu_r = (1-\gamma)^{-2.5} \tag{7.9}$$

Einstein Formula [25] (p. 5714)

$$\mu_r = 1 + 2.5(1-\gamma) \tag{7.10}$$

Breugem Correlation [24] (p. 5714)

$$\mu_r = \begin{cases} \frac{1}{2}\left(\gamma - \frac{3}{7}\right) & \gamma \geq \frac{3}{7} \\ 0 & \gamma < \frac{3}{7} \end{cases} \tag{7.11}$$

7.2.3.4. Treatment of the Energy Equation in Porous Media

Ansys Fluent solves the standard energy transport equation (Equation 5.1 in the Theory Guide) in porous media regions with modifications to the conduction flux and the transient terms only.

7.2.3.4.1. Equilibrium Thermal Model Equations

For simulations in which the porous medium and fluid flow are assumed to be in thermal equilibrium, the conduction flux in the porous medium uses an effective conductivity and the transient term includes the thermal inertia of the solid region on the medium:

$$\begin{aligned}\frac{\partial}{\partial t}(\gamma \rho_{\mathrm{f}} E_{\mathrm{f}} + (1-\gamma)\rho_{\mathrm{s}} E_{\mathrm{s}}) &+ \nabla \cdot (\vec{v}(\rho_{\mathrm{f}} E_{\mathrm{f}} + p)) \\ &= S_{\mathrm{f}}^{h} + \nabla \cdot \left[k_{\mathrm{eff}} \nabla T - \left(\sum_i h_i J_i \right) + (\bar{\bar{\tau}} \cdot \vec{v}) \right]\end{aligned} \tag{7.12}$$

where

E_f = total fluid energy

E_s = total solid medium energy

ρ_f = fluid density

ρ_s = solid medium density

γ = porosity of the medium

k_{eff} = effective thermal conductivity of the medium

S_f^h = fluid enthalpy source term

The effective thermal conductivity in the porous medium, k_{eff}, is computed by Ansys Fluent as the volume average of the fluid conductivity and the solid conductivity:

$$k_{eff} = \gamma k_f + (1-\gamma) k_s \tag{7.13}$$

where

k_f = fluid phase thermal conductivity (including the turbulent contribution, k_t)

k_s = solid medium thermal conductivity

The fluid thermal conductivity k_f and the solid thermal conductivity k_s can be computed via user-defined functions.

The anisotropic effective thermal conductivity can also be specified via user-defined functions. In this case, the isotropic contributions from the fluid, γk_f, are added to the diagonal elements of the solid anisotropic thermal conductivity matrix.

7.2.3.4.2. Non-Equilibrium Thermal Model Equations

For simulations in which the porous medium and fluid flow are not assumed to be in thermal equilibrium, a dual cell approach is used. In such an approach, a solid zone that is spatially coincident with the porous fluid zone is defined, and this solid zone only interacts with the fluid with regard to heat transfer. The conservation equations for energy are solved separately for the fluid and solid zones. The conservation equation solved for the fluid zone is

$$\frac{\partial}{\partial t}\left(\gamma \rho_f E_f\right) + \nabla \cdot \left(\vec{v}\left(\rho_f E_f + p\right)\right) = \nabla \cdot \left(\gamma k_f \nabla T_f - \left(\sum_i h_i J_i\right) + \left(\bar{\bar{\tau}} \cdot \vec{v}\right)\right) + S_f^h + h_{fs} A_{fs}\left(T_s - T_f\right) \tag{7.14}$$

and the conservation equation solved for the solid zone is

$$\frac{\partial}{\partial t}\left((1-\gamma)\rho_s E_s\right) = \nabla \cdot \left((1-\gamma) k_s \nabla T_s\right) + S_s^h + h_{fs} A_{fs}\left(T_f - T_s\right) \tag{7.15}$$

where

E_f = total fluid energy

E_s = total solid medium energy

ρ_f = fluid density

ρ_s = solid medium density

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

γ = porosity of the medium

k_{f} = fluid phase thermal conductivity (including the turbulent contribution, k_t)

k_{s} = solid medium thermal conductivity

h_{fs} = heat transfer coefficient for the fluid / solid interface

A_{fs} = interfacial area density, that is, the ratio of the area of the fluid / solid interface and the volume of the porous zone

T_{f} = temperature of the fluid

T_{s} = temperature of the solid medium

S_{f}^{h} = fluid enthalpy source term

S_{s}^{h} = solid enthalpy source term

The fluid thermal conductivity k_{f} and the solid thermal conductivity k_{s} can be computed via user-defined functions.

The source term due to the non-equilibrium thermal model is represented in Equation 7.14 (p. 1338) and Equation 7.15 (p. 1338) by $h_{\mathrm{fs}}A_{\mathrm{fs}}(T_{\mathrm{f}}-T_{\mathrm{s}})$ and $h_{\mathrm{fs}}A_{\mathrm{fs}}(T_{\mathrm{s}}-T_{\mathrm{f}})$, respectively.

7.2.3.5. Treatment of Turbulence in Porous Media

Ansys Fluent will, by default, solve the standard conservation equations for turbulence quantities in the porous medium. In this default approach, turbulence in the medium is treated as though the solid medium has no effect on the turbulence generation or dissipation rates. This assumption may be reasonable if the medium's permeability is quite large and the geometric scale of the medium does not interact with the scale of the turbulent eddies. In other instances, however, you may want to suppress the effect of turbulence in the medium.

If you are using one of the turbulence models (with the exception of the Large Eddy Simulation (LES) model), you can suppress the effect of turbulence in a porous region by enabling the **Laminar Zone** option in the Fluid Dialog Box (p. 4961). Refer to Specifying a Laminar Zone (p. 1322) for details about using the **Laminar Zone** option.

7.2.3.6. Effect of Porosity on Transient Scalar Equations

For transient porous media calculations, the effect of porosity on the time-derivative terms is accounted for in all scalar transport equations and the continuity equation. When the effect of porosity is taken into account, the time-derivative term becomes $\frac{\partial}{\partial t}\left(\gamma\rho\phi\right)$, where ϕ is the scalar quantity (k, ε, and so on) and γ is the porosity.

The effect of porosity is enabled automatically for transient calculations, and the porosity is set to 1 by default.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.2.3.7. Modeling Porous Media Based on Physical Velocity

As stated in Limitations and Assumptions of the Porous Media Model (p. 1334), by default Ansys Fluent calculates the superficial velocity based on volumetric flow rate. The superficial velocity in the governing equations can be represented as

$$\vec{v}_{superficial} = \gamma \vec{v}_{physical} \tag{7.16}$$

where γ is the porosity of the media defined as the ratio of the volume occupied by the fluid to the total volume.

The superficial velocity values within the porous region remain the same as those outside of the porous region, and porosity is not taken into account in the differential terms of the transport equations. This limits the accuracy of the porous model in cases where there should be an increase in velocity throughout the porous region, and does not yield accurate results when velocity values and gradients are important. For more accurate simulations of porous media flows, it becomes necessary to solve for the true, or physical, velocity throughout the flowfield rather than the superficial velocity, as well as to include porosity in all terms of the transport equations.

Ansys Fluent allows the calculation of the physical velocity using the **Porous Formulation**, available in the **Cell Zone Conditions** task page. By default, the **Superficial Velocity** option is turned on.

7.2.3.7.1. Single Phase Porous Media

Using the physical velocity formulation, and assuming a general scalar ϕ, the governing equation in an isotropic porous media has the following form:

$$\frac{\partial(\gamma\rho\phi)}{\partial t} + \nabla\cdot(\gamma\rho\vec{v}\phi) = \nabla\cdot(\gamma\Gamma\nabla\phi) + S_\phi \tag{7.17}$$

Assuming isotropic porosity and single phase flow, the volume-averaged mass and momentum conservation equations are as follows:

$$\frac{\partial(\gamma\rho)}{\partial t} + \nabla\cdot(\gamma\rho\vec{v}) = 0 \tag{7.18}$$

$$\frac{\partial(\gamma\rho\vec{v})}{\partial t} + \nabla\cdot(\gamma\rho\vec{v}\vec{v}) = -\gamma\nabla p + \nabla\cdot(\gamma\vec{\tau}) + \gamma\vec{B}_f - \left(\frac{\gamma^2\mu}{K}\vec{v} + \frac{\gamma^3 C_2}{2}\rho|\vec{v}|\vec{v}\right) \tag{7.19}$$

The last term in Equation 7.19 (p. 1340) represents the viscous and inertial drag forces imposed by the pore walls on the fluid.

Important:

Note that even when you solve for the physical velocity in Equation 7.19 (p. 1340), the two resistance coefficients can still be derived using the superficial velocity as given in Defining the Viscous and Inertial Resistance Coefficients (p. 1346). Ansys Fluent assumes that the inputs for these resistance coefficients are based upon well-established empirical correlations that are usually based on superficial velocity. Therefore, Ansys Fluent

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

automatically converts the inputs for the resistance coefficients into those that are compatible with the physical velocity formulation.

Important:

Note that the inlet mass flow is also calculated from the superficial velocity. Therefore, for the same mass flow rate at the inlet and the same resistance coefficients, for either the physical or superficial velocity formulation you should obtain the same pressure drop across the porous media zone.

Important:

The effect of porosity is not applied in Ansys Fluent's treatment of user defined sources. If porosity effects need to be included, they should be incorporated in the definition of the source terms. This applies to all user defined sources in Ansys Fluent.

7.2.3.7.2. Multiphase Porous Media

You can simulate porous media multiphase flows using the **Physical Velocity Porous Formulation** to solve the true or physical velocity field throughout the entire flow field, including both porous and non-porous regions. In this approach, assuming a general scalar in the q^{th} phase, ϕ_q, the governing equation in an isotropic porous medium takes on the following form:

$$\frac{\partial\left(\gamma\alpha_q\rho_q\phi_q\right)}{\partial t}+\nabla\cdot\left(\gamma\alpha_q\rho_q\vec{v}_q\phi_q\right)=\nabla\cdot\left(\gamma\Gamma_q\nabla\phi_q\right)+S_{\phi,q} \tag{7.20}$$

Here γ is the porosity, which may vary with time and space; ρ_q is the phase density; α_q is the volume fraction; $\vec{v}_q$ is the phase velocity vector; $S_{\phi,q}$ is the source term; and Γ_q is the diffusion coefficient.

Important:

The effect of porosity is not applied in Ansys Fluent's treatment of user defined sources. If porosity effects need to be included, they should be incorporated in the definition of the source terms. This applies to all user defined sources in Ansys Fluent.

The general scalar equation Equation 7.20 (p. 1341) applies to all other transport equations in the Eulerian multiphase model, such as the granular phase momentum and energy equations, turbulence modeling equations, and the species transport equations.

Assuming isotropic porosity and multiphase flows, the governing equations in the q^{th} phase, Equation 14.193, Equation 14.194, and Equation 14.197 in the Theory Guide take the general forms described below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.2.3.7.2.1. The Continuity Equation

$$\frac{\partial}{\partial t}\left(\gamma\alpha_q\rho_q\right)+\nabla\cdot\left(\gamma\alpha_q\rho_q\vec{v}_q\right)=\gamma\sum_{\substack{p=1\\p\neq q}}^{n}\left(\dot{m}_{pq}-\dot{m}_{qp}\right)+S_q \tag{7.21}$$

7.2.3.7.2.2. The Momentum Equation

$$\begin{aligned}\frac{\partial}{\partial t}\left(\gamma\alpha_q\rho_q\vec{v}_q\right)+\nabla\cdot\left(\gamma\alpha_q\rho_q\vec{v}_q\vec{v}_q\right)= \quad & -\gamma\alpha_q\nabla\left(p-p_c\right)+\nabla\cdot\left(\gamma\bar{\bar{\tau}}_q\right)+\gamma\alpha_q\rho_q\vec{B}_f\\ & -\left(\alpha_q^2\gamma^2\frac{\mu_q\vec{v}_q}{K K_{r,q}}+\alpha_q^3\gamma^3\frac{C_2\rho_q|\vec{v}_q|\vec{v}_q}{2}\right)\\ & +\gamma\sum_{p=1}^{n}\left(\vec{F}_{pq}^{D}+\vec{F}_{pq}^{TD}+\dot{m}_{pq}\vec{v}_{pq}-\dot{m}_{qp}\vec{v}_{qp}\right)\\ & +\gamma\left(\vec{F}_q+\vec{F}_q^{L}+\vec{F}_q^{vm}\right)\end{aligned} \tag{7.22}$$

where p_c is capillary pressure for the wetting phase, $\bar{\bar{\tau}}_q$ is phase shear stress, $\vec{B}_f$ is the body force, $\dot{m}_{pq}$ and $\dot{m}_{qp}$ are the mass transfers from phase p to phase q and vice versa, K is the absolute permeability, and $K_{r,q}$ is the relative permeability. The term in the brackets is the porous sink, having zero value in non-porous flows/regions. The last two terms includes the general interphase momentum exchange forces: $\vec{F}_{pq}^{D}$ is the drag force for non-porous flows/regions, $\vec{F}_{pq}^{TD}$ is the turbulent dispersed force, $\vec{v}_{pq}$ and $\vec{v}_{qp}$ are the relative velocity vectors, and $\vec{F}_q$, $\vec{F}_q^{L}$, and $\vec{F}_q^{vm}$ are the external body, lift and virtual mass exchange forces. If the capillary pressure model is not enabled, $K_{r,q}$ =1.

Details about the user inputs related to the momentum resistance sources can be found in User Inputs for Porous Media (p. 1343).

7.2.3.7.2.3. The Energy Equation

$$\begin{aligned}\frac{\partial}{\partial t}\left(\alpha_q\left(\gamma\rho_q h_q+\left(1-\gamma\right)\rho_s h_s\right)\right)+\nabla\cdot\left(\gamma\alpha_q\rho_q\vec{v}_q h_q\right)= \quad & -\gamma\alpha_q\frac{\partial p_q}{\partial t}+\gamma\bar{\bar{\tau}}_q:\nabla\vec{v}_q-\nabla\cdot\left(\alpha_q\left(\gamma k_q+\left(1-\gamma\right)k_s\right)\nabla T_q\right)\\ & +S_q+\gamma\sum_{n}^{p=1}\left(Q_{pq}+\dot{m}_{pq}h_{pq}-\dot{m}_{qp}h_{qp}\right)\end{aligned} \tag{7.23}$$

where:

Subscripts q, p, and s denote phase q, phase p, and the solid material, respectively

h is the total enthalpy

k is the thermal conductivity

Q_{pq} is the heat transfer between phase p and phase q

ρ is the density

S is the heat source

α is the phase volume fraction

$\vec{v}$ is the phase velocity

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

h_{pq} and h_{qp} are the total enthalpy differences between phases p and q and vice versa

7.2.3.8. User Inputs for Porous Media

When you are modeling a porous region, the additional inputs for the problem setup are as follows. Optional inputs are indicated as such.

1. Define the porous zone (as described in Defining the Porous Zone (p. 1344)).
2. (optional) Define the porous velocity formulation in the **Cell Zone Conditions** task page (as described in Defining the Porous Velocity Formulation (p. 1345)).
3. Identify the fluid material flowing through the porous medium (as described in Defining the Fluid Passing Through the Porous Medium (p. 1345)).
4. Enable reactions for the porous zone, if appropriate, and select the reaction mechanism (as described in Enabling Reactions in a Porous Zone (p. 1345)).
5. Enable the **Relative Velocity Resistance Formulation**. By default, this option is already enabled and takes the moving porous media into consideration (as described in Including the Relative Velocity Resistance Formulation (p. 1345)).
6. Set the viscous resistance coefficients (D_{ij} in Equation 7.1 (p. 1335), or $1/\alpha$ in Equation 7.2 (p. 1335)) and the inertial resistance coefficients (C_{ij} in Equation 7.1 (p. 1335), or C_2 in Equation 7.2 (p. 1335)), and define the direction vectors for which they apply. Alternatively, specify the coefficients for the power-law model. (See Defining the Viscous and Inertial Resistance Coefficients (p. 1346) for more details.)
7. Specify the porosity of the porous medium (as described in Defining Porosity (p. 1354)).
8. (optional) Specify the settings for heat transfer (as described in Specifying the Heat Transfer Settings (p. 1354)).
9. (optional) Specify a model for relative viscosity (as described in Specifying the Relative Viscosity (p. 1358)).
10. (optional, Eulerian multiphase models only) Specify relative permeability (as described in Specifying the Relative Permeability (p. 1358).
11. (optional) Set the volumetric heat generation rates in the porous medium (or any other sources, such as mass or momentum). (See Defining Sources (p. 1370) for more details.)
12. (optional) Set any fixed values for solution variables in the fluid region (as described in Defining Fixed Values (p. 1370).
13. Suppress the turbulent viscosity in the porous region, if appropriate. (See Suppressing the Turbulent Viscosity in the Porous Region (p. 1370) for more details.)
14. Specify the rotation axis and/or zone motion, if relevant. (See Specifying the Rotation Axis and Defining Zone Motion (p. 1370) for more details.)

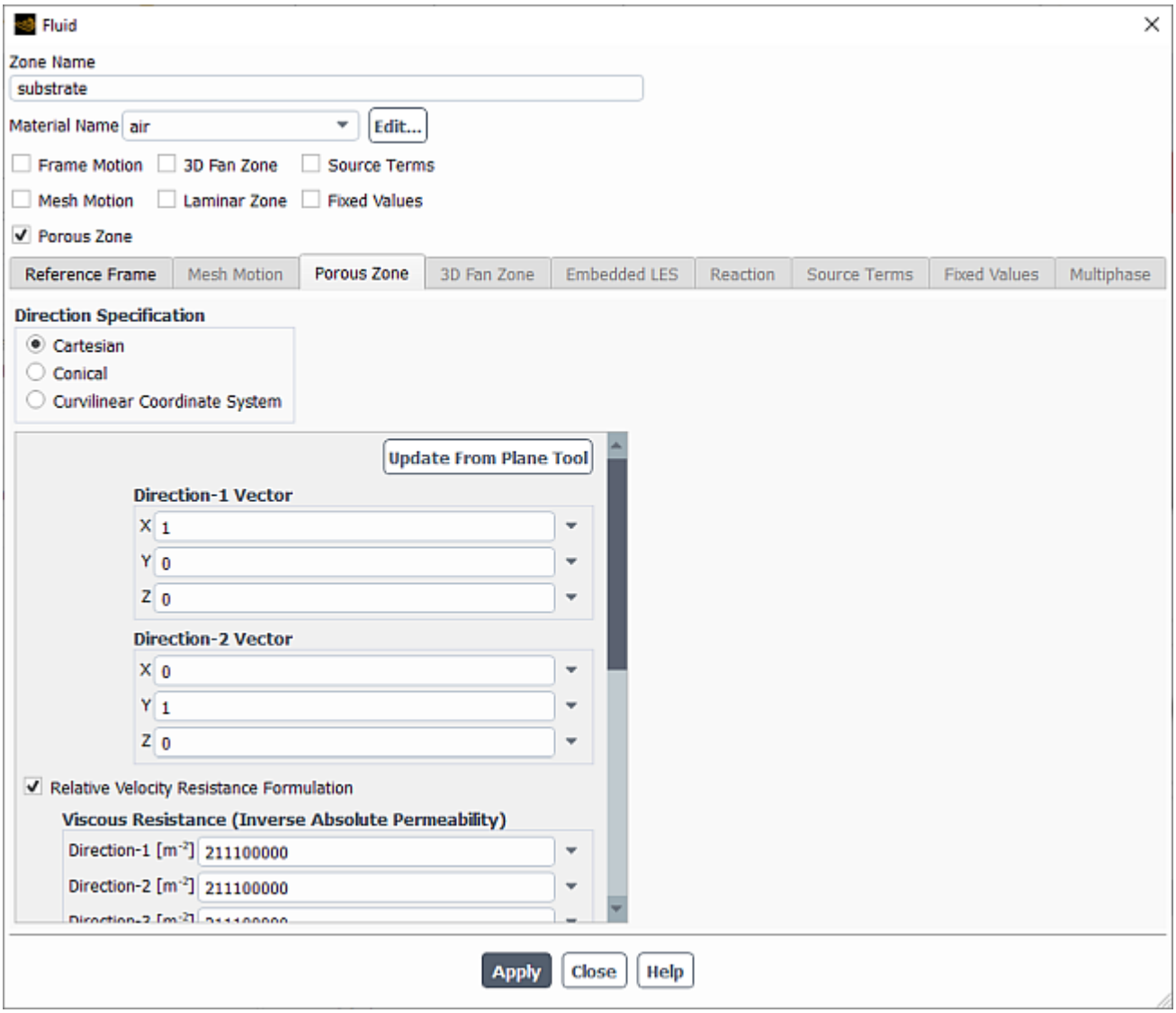
Methods for determining the resistance coefficients and/or permeability are presented below. If you choose to use the power-law approximation of the porous-media momentum source term, you

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

will enter the coefficients C_0 and C_1 in Equation 7.3 (p. 1335) instead of the resistance coefficients and flow direction.

You will set all parameters for the porous medium in the Fluid Dialog Box (p. 4961) (Figure 7.22: The Fluid Dialog Box for a Porous Zone (p. 1344)), which is opened from the **Cell Zone Conditions** task page (as described in Setting Cell Zone and Boundary Conditions (p. 1303)).

Figure 7.22: The Fluid Dialog Box for a Porous Zone



7.2.3.8.1. Defining the Porous Zone

As mentioned in Overview (p. 1299), a porous zone is modeled as a special type of fluid zone. To indicate that the fluid zone is a porous region, enable the **Porous Zone** option in the **Fluid** dialog box. The dialog box will expand to show the porous media inputs (as shown in Figure 7.22: The Fluid Dialog Box for a Porous Zone (p. 1344)).

In a coupled analysis involving Fluent and System Coupling, a porous jump boundary of zero-thickness next to the porous zone can be used to transfer data between the porous zone and

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the System Coupling system. For more about System Coupling and the variable transferred from a porous jump boundary, see Performing System Coupling Simulations Using Fluent (p. 4647).

7.2.3.8.2. Defining the Porous Velocity Formulation

The **Cell Zone Conditions** task page contains a **Porous Formulation** region where you can instruct Ansys Fluent to use either a superficial or physical velocity in the porous medium simulation. By default, the velocity is set to **Superficial Velocity**. For details about using the **Physical Velocity** formulation, see Modeling Porous Media Based on Physical Velocity (p. 1340).

7.2.3.8.3. Defining the Fluid Passing Through the Porous Medium

To define the fluid that passes through the porous medium, select the appropriate fluid in the **Material Name** drop-down list in the Fluid Dialog Box (p. 4961). If you want to check or modify the properties of the selected material, you can click **Edit...** to open the **Edit Material** dialog box; this dialog box contains just the properties of the selected material, not the full contents of the standard **Create/Edit Materials** dialog box.

Important:

If you are modeling species transport or multiphase flow, the **Material Name** list will not appear in the **Fluid** dialog box. For species calculations, the mixture material for all fluid/porous zones will be the material you specified in the Species Model Dialog Box (p. 4768). For multiphase flows, the materials are specified when you define the phases, as described in Defining the Phases for the VOF Model (p. 3009).

7.2.3.8.4. Enabling Reactions in a Porous Zone

If you are modeling species transport with reactions, you can enable reactions in a porous zone by turning on the **Reaction** option in the **Fluid** dialog box and selecting a mechanism in the **Reaction Mechanism** drop-down list.

If your mechanism contains wall surface reactions, you also must specify a value for the **Surface-to-Volume Ratio**. This value is the surface area of the pore walls per unit volume ($\frac{A}{V}$), and can be thought of as a measure of catalyst loading. With this value, Ansys Fluent can calculate the total surface area on which the reaction takes place in each cell by multiplying $\frac{A}{V}$ by the volume of the cell and the porosity. See Defining Zone-Based Reaction Mechanisms (p. 2387) for details about defining reaction mechanisms. See Wall Surface Reactions and Chemical Vapor Deposition (p. 2400) for details about wall surface reactions.

Note:

The wall surface reactions in porous medium are allowed only for the pressure-based solver.

7.2.3.8.5. Including the Relative Velocity Resistance Formulation

For cases involving moving meshes, dynamic meshes or moving reference frames (MRF), the **Relative Velocity Resistance Formulation** option allows you to better predict porous media sources. The porous media source terms are calculated using relative velocities in the porous zone. For

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

more information, see Momentum Equations for Porous Media (p. 1335). The **Relative Velocity Resistance Formulation** option works well for cases with moving and stationary porous media. It is turned on by default.

Note the following limitation when using the **Relative Velocity Resistance Formulation** option:

- The relative velocity resistance formulation is not supported with axisymmetric-swirl when there are non-zero swirl resistances.

Note:

In Ansys Fluent 6.3, this option was only supported for moving mesh and MRF (with the exception of power law for modeling porous sources). Currently, this option also supports dynamic mesh and takes care of proper treatment of porous sources using power law.

7.2.3.8.6. Defining the Viscous and Inertial Resistance Coefficients

The viscous and inertial resistance coefficients are both defined in the same manner. The basic approach for defining the coefficients using a Cartesian coordinate system is to define one direction vector (**Direction-1 Vector**) in 2D or two direction vectors (**Direction-1 Vector** and **Direction-2 Vector**) in 3D, and then specify the viscous and/or inertial resistance coefficients in each direction. The **Direction-1 Vector** usually represents the primary flow direction (parallel to the porous zone axis), and the other two directions represent the transverse directions.

Ansys Fluent automatically determines the second direction vector in 2D or the third direction vector in 3D, which is not explicitly defined:

- In 2D, the second direction is normal to the plane defined by the **Direction-1 Vector** and the z-direction vector.
- In 3D, the third direction is normal to the plane defined by the **Direction-1 Vector** and **Direction-2 Vector**. Note that the **Direction-2 Vector** must be normal to the **Direction-1 Vector**. If you fail to specify two normal directions, the Ansys Fluent solver will orthogonalize the direction specification by removing the projection of the **Direction-2 Vector** on the **Direction-1 Vector** before computing the third mutually orthogonal direction. You should therefore be certain that the first direction is correctly specified.

If the porous zone faces are not aligned with the global Cartesian axis, you can use the plane tool in 3D or the line tool in 2D for the direction vector definitions as further described.

You can also define the viscous and/or inertial resistance coefficients in each direction using a user-defined function (UDF). The user-defined options become available in the corresponding drop-down list when the UDF has been created and loaded into Ansys Fluent. Note that the coefficients defined in the UDF must utilize the `DEFINE_PROFILE` macro. For more information on creating and using user-defined function, see the Fluent Customization Manual.

If you are modeling axisymmetric swirling flows, you can specify an additional direction component for the viscous and/or inertial resistance coefficients. This direction component is always tangential to the other two specified directions. This option is available for both density-based and pressure-based solvers.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In 3D, it is also possible to define the coefficients using a conical (or cylindrical) coordinate system, as described below.

Important:

Note that the viscous and inertial resistance coefficients are generally based on the superficial velocity of the fluid in the porous media.

The procedure for defining resistance coefficients is as follows:

1. Define the direction vectors.

   - To use a Cartesian coordinate system, simply specify the **Direction-1 Vector** and, for 3D, the **Direction-2 Vector**. The unspecified direction will be determined as described above. These direction vectors correspond to the principle axes of the porous media.

     Note:

     The units for the inertial resistance coefficients (**Direction-1 Vector** and **Direction-2 Vector**) is the inverse of length. Should you want to define different units, you can do so by opening the **Units** dialog box and selecting **resistance** from the **Quantities** list.

     For some problems in which the principal axes of the porous medium are not aligned with the coordinate axes of the domain, you may not know a priori the direction vectors of the porous medium. In such cases, the plane tool (available via the **Plane Surface** dialog box) in 3D or the line tool (available via the **Line/Rake Surface** dialog box) in 2D can help you to determine these direction vectors.

     a. Open the **Plane Surface** dialog box in 3D or the **Line/Rake Surface** dialog box in 2D to enable the plane or line tool.

     b. Rotate the axes of the tool appropriately until they are aligned with the porous medium. You may want to move the plane tool (or the line tool) onto the boundary of the porous region. (Follow the instructions in Using the Plane Tool (p. 3833) or Using the Line Tool (p. 3828) for initializing the tool to a position on an existing surface.)

     c. Once the axes are aligned, click the **Update From Plane Tool** or **Update From Line Tool** button in the **Fluid** dialog box. Ansys Fluent will automatically set the **Direction-1 Vector** to the direction of the red arrow of the tool, and (in 3D) the **Direction-2 Vector** to the direction of the green arrow.

     Important:

     The Plane Surface Dialog Box (p. 5580) or the Line/Rake Surface Dialog Box (p. 5514) must be open for the plane or line tool to be active.

   - To use a conical coordinate system (for example, for an annular, conical filter element), follow the steps below. This option is available only in 3D cases.

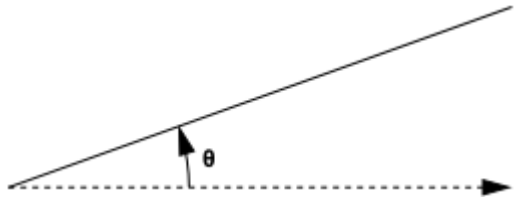
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. Turn on the **Conical** option.

b. Set the **Cone Half Angle** (the angle between the cone's axis and its surface, shown in Figure 7.23: Cone Half Angle (p. 1348)). To use a cylindrical coordinate system, set the **Cone Half Angle** to 0.

c. Specify the **Cone Axis Vector** and **Point on Cone Axis**. The cone axis is specified as being in the direction of the **Cone Axis Vector** (unit vector), and passing through the **Point on Cone Axis**. The cone axis may or may not pass through the origin of the coordinate system.

Figure 7.23: Cone Half Angle



For some problems in which the axis of the conical filter element is not aligned with the coordinate axes of the domain, you may not know a priori the direction vector of the cone axis and coordinates of a point on the cone axis. In such cases, the plane tool can help you to determine the cone axis vector and point coordinates. One method is as follows:

a. Select a boundary zone of the conical filter element that is normal to the cone axis vector in the drop-down list next to the **Snap to Zone** button.

b. Click the **Snap to Zone** button. Ansys Fluent will automatically "snap" the plane tool onto the boundary. It will also set the **Cone Axis Vector** and the **Point on Cone Axis**. (Note that you will still have to set the **Cone Half Angle** yourself.)

An alternate method is as follows:

a. Move the plane tool onto the boundary of the porous region. (Follow the instructions in Using the Plane Tool (p. 3833) for moving the tool to a position on an existing surface.) *Open the **Plane Surface** dialog box to enable the plane tool.*

b. Rotate and translate the axes of the tool appropriately until the red arrow of the tool is pointing in the direction of the cone axis vector and the origin of the tool is on the cone axis.

c. Once the axes and origin of the tool are aligned, click the **Update From Plane Tool** button in the **Fluid** dialog box. Ansys Fluent will automatically set the **Cone Axis Vector** and the **Point on Cone Axis**. (Note that you will still have to set the **Cone Half Angle** yourself.)

Open the Plane Surface Dialog Box (p. 5580) or the Line/Rake Surface Dialog Box (p. 5514) to make the plane or line tool appear.

- To use a curvilinear coordinate system (such as for a winding or coil), follow the steps below. This option is available only in 3D cases.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. Under **Direction Specification**, select **Curvilinear Coordinate System**.

b. For **Coordinate System**, select an already created curvilinear coordinate system or you can create a new one by clicking **New...**. For details on creating a curvilinear coordinate system, see Curvilinear Coordinate Systems (p. 1164).

c. For **Viscous Resistance** and **Inertial Resistance**, the direction components will correspond to the directions defined for the chosen curvilinear coordinate system.

2. Under **Viscous Resistance**, specify the viscous resistance coefficient $1/\alpha$ in each direction.

   Under **Inertial Resistance**, specify the inertial resistance coefficient C_2 in each direction. (You can scroll down with the scroll bar to view these inputs.)

   For porous media cases containing highly anisotropic inertial resistances, enable **Alternative Formulation** under **Inertial Resistance**. The **Alternative Formulation** option provides better stability to the calculation when your porous medium is anisotropic. The pressure loss through the medium depends on the magnitude of the velocity vector of the i th component in the medium. Using the formulation of Equation 7.6 (p. 1336) yields the expression below:

$$S_i = \frac{1}{2}\rho C_i |v_i| v_i \tag{7.24}$$

   Whether or not you use the **Alternative Formulation** option depends on how well you can fit your experimentally determined pressure drop data to the Ansys Fluent model. For example, if the flow through the medium is aligned with the mesh in your Ansys Fluent model, then it will not make a difference whether or not you use the formulation. In general, however, the default and the **Alternative Formulation** are not equivalent, and solution differences can occur when switching between the two options.

   Note:

   After you change the status of the **Alternative Formulation** option, you need to rerun the simulation to obtain consistent force reports and cumulative plots (Cumulative Force, Moment, and Coefficients Plots (p. 4094)) for lift, drag, moments, and corresponding coefficients computed on porous zones. This is because the porous force contribution is always obtained on demand by recomputing the porous sources used in the momentum equations, which can change with enabling or disabling the **Alternative Formulation** option.

   For more information about simulations involving highly anisotropic porous media, see Solution Strategies for Porous Media (p. 1371).

   Important:

   Note that the alternative formulation is compatible only with the pressure-based solver.

   If you are using the **Conical** specification method, **Direction-1** is the tangential direction of the cone, **Direction-2** is the normal to the cone surface (radial (r) direction for a cylinder), and **Direction-3** is the circumferential (θ) direction.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In 3D there are three possible categories of coefficients, and in 2D there are two:

- In the isotropic case, the resistance coefficients in all directions are the same (for example, a sponge). For an isotropic case, you must explicitly set the resistance coefficients in each direction to the same value.
- When (in 3D) the coefficients in two directions are the same and those in the third direction are different or (in 2D) the coefficients in the two directions are different, you must be careful to specify the coefficients properly for each direction. For example, if you had a porous region consisting of cylindrical straws with small holes in them positioned parallel to the flow direction, the flow would pass easily through the straws, but the flow in the other two directions (through the small holes) would be very little. If you had a plane of flat plates perpendicular to the flow direction, the flow would not pass through them at all; it would instead move in the other two directions.
- In 3D the third possible case is one in which all three coefficients are different. For example, if the porous region consisted of a plane of irregularly-spaced objects (for example, pins), the movement of flow between the blockages would be different in each direction. You would therefore need to specify different coefficients in each direction.

Methods for deriving viscous and inertial loss coefficients are described in the sections that follow.

7.2.3.8.7. Deriving Porous Media Inputs Based on Superficial Velocity, Using a Known Pressure Loss

When you use the porous media model, you must remember that the porous cells in Ansys Fluent are *100% open*, and that the values that you specify for $1/\ \alpha_{ij}$ and/or $C_{2_{ij}}$ must be based on this assumption. Suppose, however, that you know how the pressure drop varies with the velocity through the actual device, which is only partially open to flow. The following exercise is designed to show you how to compute a value for C_2 which is appropriate for the Ansys Fluent model.

Consider a perforated plate that has 25% area open to flow. The pressure drop through the plate is known to be 0.5 times the dynamic head in the plate. The loss factor, K_L, defined as

$$\Delta p = K_L\left(\frac{1}{2}\rho v^2_{25\%open}\right) \tag{7.25}$$

is therefore 0.5, based on the actual fluid velocity in the plate, that is, the velocity through the 25% open area. To compute an appropriate value for C_2, note that in the Ansys Fluent model:

1. The velocity through the perforated plate assumes that the plate is 100% open.
2. The loss coefficient must be converted into dynamic head loss per unit length of the porous region.

Noting item 1, the first step is to compute an adjusted loss factor, $K_L^{'}$, which would be based on the velocity of a 100% open area:

$$\Delta p = K_L^{'}\left(\frac{1}{2}\rho v^2_{100\%open}\right) \tag{7.26}$$

or, noting that for the same flow rate, $v_{25\%open} = 4 \times v_{100\%open}$,

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$K_L' = K_L \times \frac{v_{25\%open}^2}{v_{100\%open}^2} = 0.5 \times \left(\frac{4}{1}\right)^2 = 8 \tag{7.27}$$

The adjusted loss factor has a value of 8. Noting item 2, you must now convert this into a loss coefficient per unit thickness of the perforated plate. Assume that the plate has a thickness of 1.0 mm (10^{-3} m). The inertial loss factor would then be

$$C_2 = \frac{K_{K_L'}}{thickness} = \frac{8}{10^{-3}} = 8000 m^{-1} \tag{7.28}$$

Note that, for anisotropic media, this information must be computed for each of the 2 (or 3) coordinate directions.

7.2.3.8.8. Using the Ergun Equation to Derive Porous Media Inputs for a Packed Bed

As a second example, consider the modeling of a packed bed. In turbulent flows, packed beds are modeled using both a permeability and an inertial loss coefficient. One technique for deriving the appropriate constants involves the use of the Ergun equation [42] (p. 5715), a semi-empirical correlation applicable over a wide range of Reynolds numbers and for many types of packing:

$$\frac{|\Delta p|}{L} = \frac{150\mu}{D_p^2}\frac{(1-\varepsilon)^2}{\varepsilon^3} v_\infty + \frac{1.75\rho}{D_p}\frac{(1-\varepsilon)}{\varepsilon^3} v_\infty^2 \tag{7.29}$$

When modeling laminar flow through a packed bed, the second term in the above equation may be dropped, resulting in the Blake-Kozeny equation [42] (p. 5715):

$$\frac{|\Delta p|}{L} = \frac{150\mu}{D_p^2}\frac{(1-\varepsilon)^2}{\varepsilon^3} v_\infty \tag{7.30}$$

In these equations, μ is the viscosity, D_p is the mean particle diameter, L is the bed depth, and ε is the void fraction, defined as the volume of voids divided by the volume of the packed bed region. Comparing Equation 7.4 (p. 1336) and Equation 7.6 (p. 1336) with Equation 7.29 (p. 1351), the permeability and inertial loss coefficient in each component direction may be identified as

$$\alpha = \frac{D_p^2}{150}\frac{\varepsilon^3}{(1-\varepsilon)^2} \tag{7.31}$$

and

$$C_2 = \frac{3.5}{D_p}\frac{(1-\varepsilon)}{\varepsilon^3} \tag{7.32}$$

7.2.3.8.9. Using an Empirical Equation to Derive Porous Media Inputs for Turbulent Flow Through a Perforated Plate

As a third example we will take the equation of Van Winkle et al. [118] (p. 5719) [150] (p. 5721) and show how porous media inputs can be calculated for pressure loss through a perforated plate with square-edged holes.

The expression, which is claimed by the authors to apply for turbulent flow through square-edged holes on an equilateral triangular spacing, is

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\dot{m}=CA_f\sqrt{\left(2\rho\Delta p\right)/\left(1-\left(A_f/A_p\right)^2\right)} \tag{7.33}$$

where

$\dot{m}$ = mass flow rate through the plate

A_f = the free area or total area of the holes

A_p = the area of the plate (solid and holes)

C = a coefficient that has been tabulated for various Reynolds-number ranges and for various D/t

D/t = the ratio of hole diameter to plate thickness

for $t/D>1.6$ and for $Re>4000$ the coefficient C takes a value of approximately 0.98, where the Reynolds number is based on hole diameter and velocity in the holes.

Rearranging Equation 7.33 (p. 1352), making use of the relationship

$$\dot{m}=\rho v A_p \tag{7.34}$$

and dividing by the plate thickness, $\Delta x=t$, we obtain

$$\frac{\Delta p}{\Delta x}=\left(\frac{1}{2}\rho v^2\right)\frac{1}{C^2}\frac{\left(A_p/A_f\right)^2-1}{t} \tag{7.35}$$

where v is the superficial velocity (not the velocity in the holes). Comparing with Equation 7.6 (p. 1336) is seen that, for the direction normal to the plate, the constant C_2 can be calculated from

$$C_2=\frac{1}{C^2}\frac{\left(A_p/A_f\right)^2-1}{t} \tag{7.36}$$

7.2.3.8.10. Using Tabulated Data to Derive Porous Media Inputs for Laminar Flow Through a Fibrous Mat

Consider the problem of laminar flow through a mat or filter pad which is made up of randomly-oriented fibers of glass wool. As an alternative to the Blake-Kozeny equation (Equation 7.30 (p. 1351)) we might choose to employ tabulated experimental data. Such data is available for many types of fiber [71] (p. 5717).

Volume Fraction of Solid Material	Dimensionless Permeability B of Glass Wool
0.262	0.25
0.258	0.26
0.221	0.40
0.218	0.41
0.172	0.80

where $B=\alpha/a^2$ and a is the fiber diameter. α, for use in Equation 7.4 (p. 1336), is easily computed for a given fiber diameter and volume fraction.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.2.3.8.11. Deriving the Porous Coefficients Based on Experimental Pressure and Velocity Data

Experimental data that is available in the form of pressure drop against velocity through the porous component, can be extrapolated to determine the coefficients for the porous media. To effect a pressure drop across a porous medium of thickness, Δn, the coefficients of the porous media are determined in the manner described below.

If the experimental data is:

Velocity (m/s)	Pressure Drop (Pa)
20.0	197.8
50.0	948.1
80.0	2102.5
110.0	3832.9

then an xy curve can be plotted to create a trendline through these points yielding the following equation

$$\Delta p = 0.27394v^2 + 4.68816v \tag{7.37}$$

where Δp is the pressure drop and v is the velocity.

Important:

Although the best fit curve may yield negative coefficients, it should be avoided when using the porous media model in Ansys Fluent.

Note that a simplified version of the momentum equation, relating the pressure drop to the source term, can be expressed as

$$\nabla p = S_i \tag{7.38}$$

or

$$\Delta p = -S_i \Delta n \tag{7.39}$$

Hence, comparing Equation 7.37 (p. 1353) to Equation 7.2 (p. 1335), yields the following curve coefficients:

$$0.27394 = C_2 \frac{1}{2} \rho \Delta n \tag{7.40}$$

with $\rho = 1.225$ kg/ m^3, and a porous media thickness, Δn, assumed to be 1 m in this example, the inertial resistance factor, $C_2 = 0.447$.

Likewise,

$$4.68816 = \frac{\mu}{\alpha} \Delta n \tag{7.41}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

with $\mu = 1.7894 \times 10^{-5}$, the viscous inertial resistance factor, $\frac{1}{\alpha} = 261996$.

Important:

Note that this same technique can be applied to the porous jump boundary condition. Similar to the case of the porous media, you have to take into account the thickness of the medium Δn. Your experimental data can be plotted in an xy curve, yielding an equation that is equivalent to Equation 7.152 (p. 1519). From there, you can determine the permeability α and the pressure jump coefficient C_2.

7.2.3.8.12. Using the Power-Law Model

If you choose to use the power-law approximation of the porous-media momentum source term (Equation 7.3 (p. 1335)), the only inputs required are the coefficients C_0 and C_1. Under **Power Law Model** in the **Fluid** dialog box, enter the values for **C0** and **C1**. Note that the power-law model can be used in conjunction with the Darcy and inertia models.

C0 must be in SI units, consistent with the value of **C1**.

7.2.3.8.13. Defining Porosity

To define the porosity, scroll down below the resistance inputs in the **Fluid** dialog box, and set the **Porosity** under **Fluid Porosity**. Note that the value of the **Porosity** must be in the range of `0 – 1` ; furthermore, the lower and upper limits (that is, `0` and `1`, respectively) are not allowed for the non-equilibrium thermal model.

You can also define the porosity using a user-defined function (UDF). The user-defined option becomes available in the corresponding drop-down list when the UDF has been created and loaded into Ansys Fluent. Note that the porosity defined in the UDF must utilize the `DEFINE_PROFILE` macro. For more information on creating and using user-defined functions, see the separate Fluent Customization Manual.

The porosity, γ, is the volume fraction of fluid within the porous region (that is, the open volume fraction of the medium). The porosity is used in the prediction of heat transfer in the medium, as described in Treatment of the Energy Equation in Porous Media (p. 1337), and in the time-derivative term in the scalar transport equations for unsteady flow, as described in Effect of Porosity on Transient Scalar Equations (p. 1339). It also impacts the calculation of reaction source terms and body forces in the medium. These sources will be proportional to the fluid volume in the medium. If you want to represent the medium as completely open (no effect of the solid medium), you should set the porosity equal to 1.0 (the default). When the porosity is equal to 1.0, the solid portion of the medium will have no impact on heat transfer or thermal/reaction source terms in the medium.

7.2.3.8.14. Specifying the Heat Transfer Settings

You can model heat transfer in the porous material, with or without the assumption of thermal equilibrium between the medium and the fluid flow. Note that heat transfer is not available for inviscid flow.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.2.3.8.14.1. Equilibrium Thermal Model

To specify that the porous medium and the fluid flow are in thermal equilibrium, scroll down below the resistance inputs in the **Fluid** dialog box to the **Heat Transfer Settings** group box and select **Equilibrium** from the **Thermal Model** list (this is the default selection). Then you must specify the material contained in the porous medium by selecting the appropriate solid in the **Solid Material Name** drop-down list.

If you want to check or modify the properties of the selected material, you can click **Edit...** to open the **Edit Material** dialog box; this dialog box contains just the properties of the selected material, not the full contents of the standard **Create/Edit Materials** dialog box. You can define the **Thermal Conductivity** of the porous material using a user-defined function (UDF), in order to define a non-isotropic thermal conductivity. The user-defined option becomes available in the corresponding drop-down list when the UDF has been created and loaded into Ansys Fluent. Note that the non-isotropic thermal conductivity defined in the UDF must utilize the `DEFINE_PROPERTY` macro. For more information on creating and using user-defined function, see the separate Fluent Customization Manual.

7.2.3.8.14.2. Non-Equilibrium Thermal Model

Note:

The non-equilibrium thermal model is incompatible with the density-based solver.

The assumption of thermal equilibrium between the solid medium and the fluid flow is not appropriate for all simulations, as the presence of different geometric length scales (for example, pore sizes) and physical properties of solid and liquid phases may result in local temperature differences between the phases. Examples of when a non-equilibrium thermal model may be suitable include the simulation of light-off for exhaust after-treatment, fuel cells, catalytic converters, and so on.

The non-equilibrium thermal model available for porous media in Ansys Fluent is based on a dual cell approach. This approach is referred to as "dual cell" because it involves a second solid cell zone that overlaps (that is, is spatially coincident with) the porous fluid zone; the two zones are solved simultaneously and are coupled only through heat transfer. Ansys Fluent can automatically create a solid zone for you that is a duplicate of the porous fluid zone; otherwise, you can create a duplicate cell zone manually using the `mesh/modify-zones/copy-move-cell-zone` text command before you enable the non-equilibrium thermal model, or make sure that the mesh you read contains two cell zones in the region where the porous medium will be defined. You should ensure that the duplicate zone is defined as a solid zone, and that the two zones have similar levels of mesh refinement (as one-to-one mapping will be employed between the cell centroids of the fluid zone and the solid zone).

Important:

When setting up both the porous fluid zone and the overlapping solid zone, note that the non-equilibrium thermal model is not supported for zones that undergo (or are adjacent to zones that undergo) changes to the geometry or mesh, such as dynamic mesh zones or zones that undergo hanging node adaption, utilize the mesh morpher/optimizer, or are involved in fluid-structure interaction (FSI) applications.

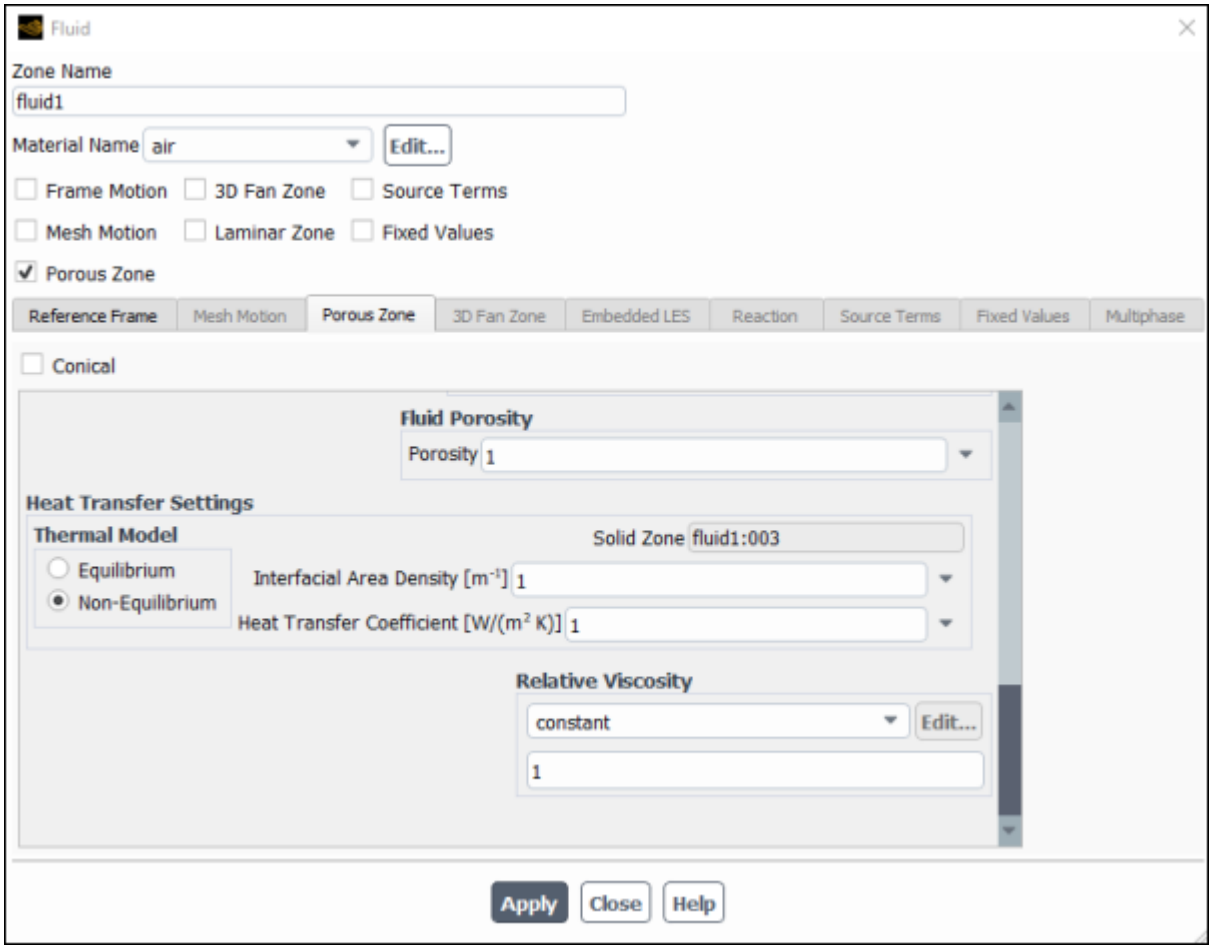
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The instructions that follow assume you have already performed steps 1–7 in User Inputs for Porous Media (p. 1343). Before you enable the non-equilibrium thermal model, click **OK** in the **Fluid** dialog box, in order to save your settings in the **Porous Zone** tab; note that if you do not save your settings, they will be reset to the default values when you enable the non-equilibrium thermal model. Then reopen the **Fluid** dialog box and scroll down below the resistance inputs in the **Porous Zone** tab. Select **Non-Equilibrium** from the **Thermal Model** list in the **Heat Transfer Settings** group box (see Figure 7.24: The Heat Transfer Settings Group Box of the Fluid Dialog Box (p. 1356)); this will enable the dual cell approach and display the associated GUI input controls. If a solid zone that is spatially coincident with the porous fluid zone does not already exist, use the **Question** dialog box that opens to automatically create one. The name of the newly created solid cell zone will then be displayed in the **Solid Zone** text box of the **Fluid** dialog box.

Important:

Note that the **Non-Equilibrium** option button is not available if a radiation and/or multiphase model is enabled, as such a combination is not supported.

Figure 7.24: The Heat Transfer Settings Group Box of the Fluid Dialog Box



Next, define A_{fs} and h_{fs} (as described in Non-Equilibrium Thermal Model Equations (p. 1338)) for **Interfacial Area Density** and **Heat Transfer Coefficient**, respectively; you can define these settings as a **Constant**, a **New Input Parameter...**, or as a user-defined function. The **user-**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

defined option becomes available in the corresponding drop-down list when the UDF has been created and loaded into Ansys Fluent. Note that the UDF must utilize the `DEFINE_PROFILE` macro. For more information on creating and using user-defined functions, see the separate Fluent Customization Manual.

When you are finished setting up the porous fluid zone, you should verify that the solid zone created in the previous step has the appropriate settings. Note the name of the zone displayed in the **Solid Zone** text box of the **Heat Transfer Settings** group box (see Figure 7.24: The Heat Transfer Settings Group Box of the Fluid Dialog Box (p. 1356)), and then double-click that zone in the **Zone** list of the **Cell Zone Conditions** task page to open the **Solid** dialog box. Then, verify that an appropriate selection is made for **Material Name**. If you want to check or modify the properties of the selected material, you can click **Edit...** to open the **Edit Material** dialog box; this dialog box contains just the properties of the selected material, not the full contents of the standard **Create/Edit Materials** dialog box. You can define the **Thermal Conductivity** of the solid material using a user-defined function (UDF), in order to define a non-isotropic thermal conductivity. The user-defined option becomes available in the corresponding drop-down list when the UDF has been created and loaded into Ansys Fluent. Note that the non-isotropic thermal conductivity defined in the UDF must utilize the `DEFINE_PROPERTY` macro. For more information on creating and using user-defined functions, see the separate Fluent Customization Manual.

Important:

For transient simulations using the Non-Equilibrium Thermal Model with low porosity (typically < 0.2), it is recommended to increase the Solid Time Step Size as described in Specifying the Solid Time Step Size (p. 2169). A value ~1s or the Automatic option can help with solution stability and convergence.

Note that when postprocessing simulations that utilize the non-equilibrium thermal model, you can display the energy source due to the temperature difference between the fluid and solid zones using the **Non-Equilibrium Thermal Model Source** variable in the **Temperature...** category. Furthermore, attention must be paid to the fact that there are overlapping cell zones (that is, a fluid and a solid zone) in the porous region. For example:

- When creating an isosurface and making selections from the **From Zones** selection list of the **Iso-Surface** dialog box, you should never select both the porous fluid zone and the overlapping solid zone at the same time. Note that if no selections are made in this list, then all the cell zones are selected.
- When making selections from the **Surfaces** list of the **Contours** dialog box, you should not select a surface that is associated with the porous fluid zone and a surface associated with the overlapping solid zone at the same time, if those surfaces are spatially coincident.
- When postprocessing the thermal conductivity property in the porous region, the value that is reported for the fluid zone is the thermal conductivity of the fluid material scaled by the porosity (γk_f) and for the overlapping solid zone the value reported is the thermal conductivity of the solid zone scaled by the complement of the porosity ($(1-\gamma)k_s$); where γ is the porosity, k_f is the fluid phase thermal conductivity (including the turbulent contribution, k_t), and k_s is the solid medium thermal conductivity.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.2.3.8.15. Specifying the Relative Viscosity

The relative viscosity can be used to account for the effect of the porous media on the diffusion term in the momentum equation. By default, the relative viscosity is set to a constant value of 1 (that is, the effective viscosity is equal to the molecular viscosity). You can choose to model the relative viscosity using one of the models described in Relative Viscosity in Porous Media (p. 1337), or you can specify a user-defined function using a `DEFINE_PROPERTY` macro.

When you define multiple porous zones, the last value you enter for the relative viscosity will be applied to compute the effective viscosity for all porous zones in your simulation. If you want to simulate different effective viscosities in different porous zones, specify the **Viscosity** property of the porous material directly via user-defined function using the `DEFINE_PROPERTY` macro (in the **Create/Edit Materials** dialog box).

7.2.3.8.16. Specifying the Relative Permeability

The relative permeability of a phase denotes its ability to transmit the phase material when its saturation (volume fraction) is less than 100%.

Reservoir engineers often resort to empirical models or measurement curves to determine the relative and, therefore, the effective permeabilities. In Fluent, the Corey power law model [27] (p. 5714) is available to account for the relative permeability for the two-phase flow through porous media. The Corey correlations of the relative permeability can be represented as follows:

$$K_{r,q}=K_{r,q}^{0}\tilde{S}_{q}^{nq} \tag{7.42}$$

where:

$K_{r,q}^{0}$ is the reference phase relative permeability

$\tilde{S}_{q}$ is the normalized saturation in the q th phase

nq is the Corey exponent for the q th phase

In Fluent the phase saturation is the phase volume fraction. Thus $\tilde{S}_{q}$ can be expressed as:

$$\tilde{S}_{q}=\tilde{\alpha}_{q}=\frac{\alpha_{q}-\alpha_{qr}}{1-\sum_{q=0}^{n-1}\alpha_{qr}} \tag{7.43}$$

where α_{q} and α_{qr} are the volume fraction and the volume fraction residual in the q th phase, respectively.

For the two-phase oil-water flow in a water-wet-system, the relative permeabilities can be obtained from:

$$\begin{aligned} K_{r,w}&=K_{r,w}^{0}\tilde{\alpha}_{w}^{nw} \\ K_{r,o}&=K_{r,o}^{0}\left(1-\tilde{\alpha}_{w}\right)^{no} \end{aligned} \tag{7.44}$$

where:

Subscripts w and o refer to the water and oil phases, respectively

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$K^0_{r,w}$ and $K^0_{r,o}$ are the reference phase relative permeabilities for the wetting and non-wetting phases, respectively

nw and no are the Corey exponents for the two phases

The Corey correlation is often used for approximation of the relative permeabilities, particularly for water flooding type scenarios. The relative permeability curves of the oil (or the non-wetting phase) is usually an S-shaped curve and of the water (or the wetting phase) is a concave upward curve.

Alternatively, if relative permeability data for one or two phases are available as tabular functions of other solution variables, you can use generic text files to input such data to define phase relative permeability as described in Reading Files in Tabular Format (p. 969).

To model the relative permeability in Ansys Fluent:

1. Set parameters for the **mixture** phase.

   Note:

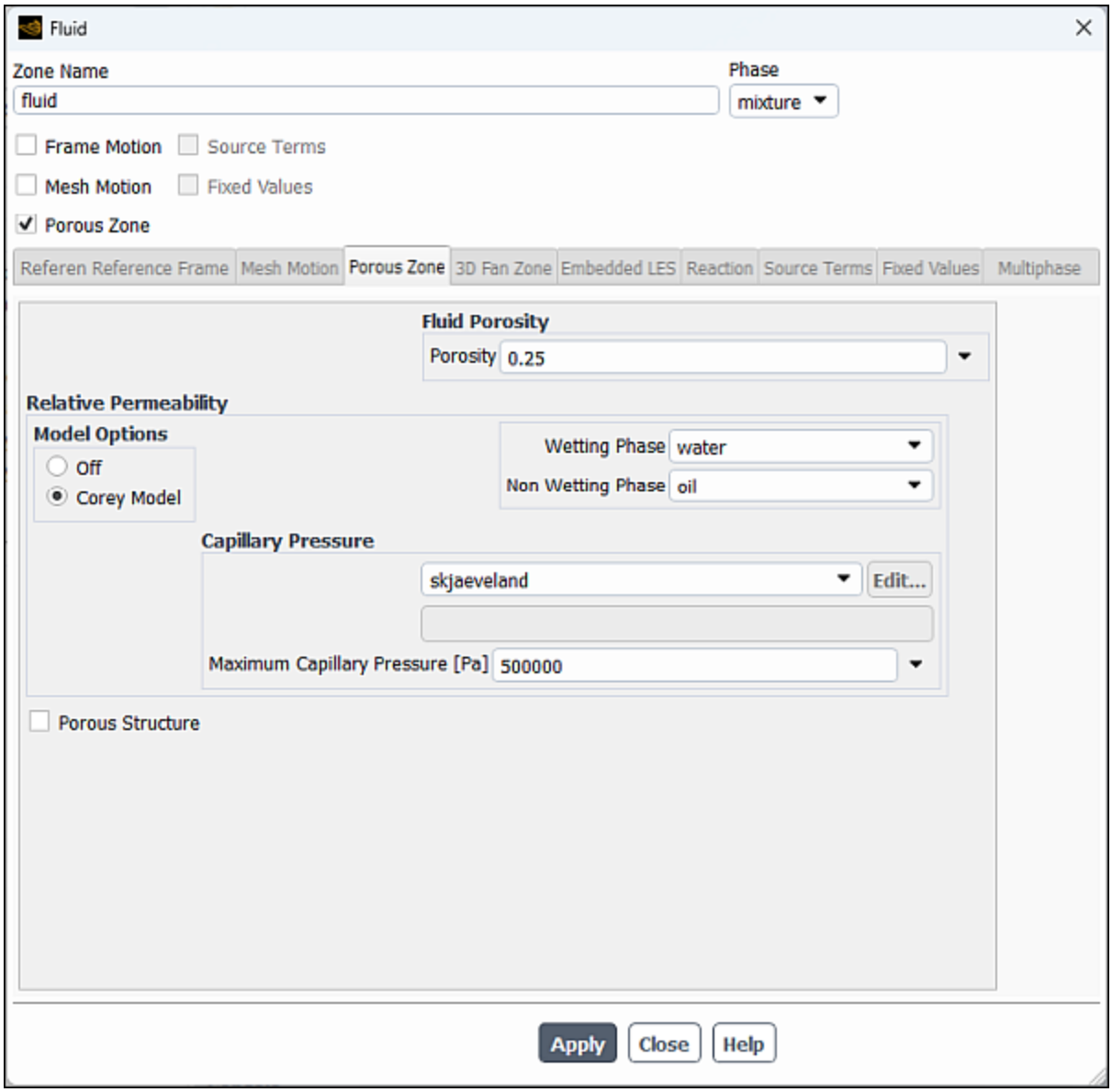
   The phase can be selected from the **Phase** drop-down list either in the **Cell Zone Conditions** task page or in the **Fluid** dialog box.

   a. In the **Fluid** dialog box, make sure that **mixture** is selected from the **Phase** drop-down list.

   b. In the **Porous Zone** tab, under **Relative Permeability**, select **Corey Model**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.25: The Fluid Dialog Box: Relative Permeability



c. From the drop-down lists, select **Wetting Phase** and **Non Wetting Phase**.

> **Important:**
>
> Although Equation 7.42 (p. 1358) through Equation 7.44 (p. 1358) are valid for any given phase, care must be exercised in defining the wetting and non-wetting phases.

d. Click **Apply**.

2. Set conditions for the wetting phase.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. In the **Fluid** dialog box, make sure that the wetting phase is selected from the **Phase** drop-down list.

b. In the **Porous Zone** tab, specify the following parameters in the **Wetting Phase Relative Permeability** group box:

Minimum Relative Permeability

sets a lower limit for the relative permeability for the wetting phase. A low value increases impact of the resistance due to relative permeability effects on the flow. This helps to avoid saturation overshoots and undershoots (that is, phase saturation dropping below residual values) in simulations carried out for a long period of time. The default value is 1.e-8. The default should work for most cases, but may be lowered in case undershoot problems persist.

Reference Relative Permeability

is $K^0_{r,w}$ in Equation 7.44 (p. 1358). This parameter is available only when the **tabular** option is disabled.

Corey Exponent

is nw in Equation 7.44 (p. 1358).

Residual Saturation

is the residual volume fraction of the wetting phase. This parameter is available only when the **tabular** option is disabled.

If, instead, you wish to specify the relative permeability parameters for the wetting phase in the tabular format, follow these steps:

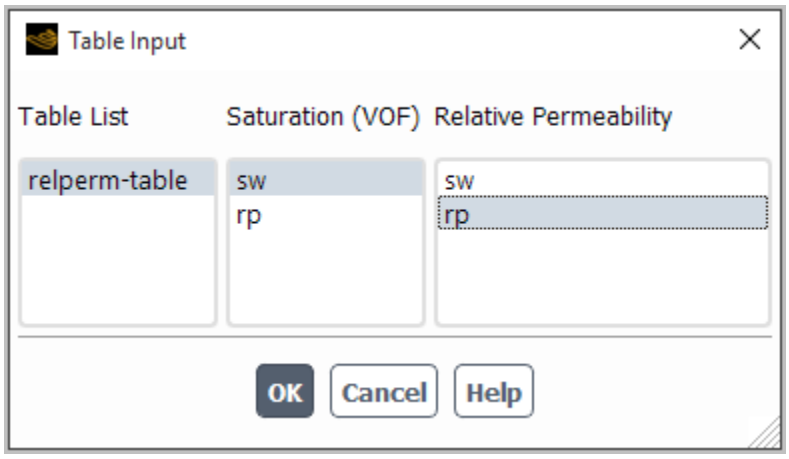
i. Select **tabular** and click **Edit...**.

> **Note:**
>
> Prior to using the **tabular** option, you must read a text file containing phase relative permeability data as described in Capillary Pressure Data in a Tabular Format (p. 1366).

ii. In the **Table Input** dialog box that opens, select the table appropriate for your fluid zone from the **Table List** and the names of the columns from the **Saturation (VOF)** and **Relative Permeability** selection lists.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.26: The Table Input Dialog Box for Relative Permeability



See Capillary Pressure Data in a Tabular Format (p. 1366) for details about the text file format.

c. Click **Apply**.

3. Similarly, set conditions for the non-wetting phase.

a. In the **Fluid** dialog box, make sure that the non-wetting phase is selected from the **Phase** drop-down list.

b. In the **Porous Zone** tab, specify the following parameters in the **Non Wetting Phase Relative Permeability** group:

Minimum Relative Permeability

sets a lower limit for the relative permeability for the non-wetting phase. This item is similar to **Minimum Relative Permeability** for the wetting phase.

Reference Relative Permeability

is $K^0_{r,o}$ in Equation 7.44 (p. 1358). This parameter is available only when the **tabular** option is disabled.

Corey Exponent

is no in Equation 7.44 (p. 1358).

Residual Saturation

is the residual volume fraction of the non-wetting phase. This parameter is available only when the **tabular** option is disabled.

If, instead, you wish to specify the relative permeability parameters for the non-wetting phase in the tabular format, select **tabular**. The inputs are similar to the tabular specification for the wetting phase.

c. Click **Apply**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

On rare occasions, the saturation may still fall below residual saturation. You can fix the affected phase to its residual saturation and allow the other phase to pass through normally by issuing the following text commands:

```
solve/set/multiphase-numerics/porous-media/relative-permeability

enable fix option below residual saturation? [no] yes
```

Note:

The previous text command is not supported for cases that use the tabular specification method for phase relative permeability.

7.2.3.8.17. Specifying the Capillary Pressure

When using the Eulerian multiphase model, you can optionally specify a sub-model for capillary pressure.

When two or more phases are present in pores, one of the phase (usually water) has a preference for the pore surfaces. This phase is referred to as the wetting phase, while the other phase is called the non-wetting phase. The boundary between the phases present in a porous medium is also curved, due to interfacial tension between the fluid, and this gives rise to a difference in the pressure across the interface:

$$P_c = P_{nw} - P_w \tag{7.45}$$

where P_{nw} and P_w are the non-wetting and wetting phase pressures, respectively. The difference, P_c is the capillary pressure. It presents a significant driving force under the conditions of low flow pressure and high volume fraction.

At the pore scale, capillary pressure depends primarily on the pore geometric characteristics (sizes, distributions, etc.), interfacial tensions and wettability (contact angles). At the Macro scale the capillary pressure is generally assumed to be a function of the saturation of the wetting fluid:

$$P_c = P_c\left(S_{wetting}\right) = P_c\left(\alpha_{wetting}\right) \tag{7.46}$$

Its value is often obtained either by measurements or correlations.

In Fluent, the capillary pressure effect is modeled as an additional source term in the wetting phase. Additional information is provided in the following sections:

7.2.3.8.17.1. Brooks-Corey Model

7.2.3.8.17.2. Van-Genuchten Model

7.2.3.8.17.3. Leverett J-Function

7.2.3.8.17.4. Skjaeveland Model

7.2.3.8.17.5. Capillary Pressure Data in a Tabular Format

7.2.3.8.17.6. Capillary Pressure Usage

7.2.3.8.17.7. Modeling Capillary Pressure as Diffusion

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.2.3.8.17.1. Brooks-Corey Model

The Brooks-Corey model [27] (p. 5714) proposed the following relationship for capillary pressure for primary drainage, in which the porous media initially saturated with water is invaded with oil phase:

$$P_{c,q}=P_e\left(S_{eq}\right)^{-1/nb} \tag{7.47}$$

where S_{eq} is the resulting normalized (effective) saturation:

$$S_{eq}=\alpha_{eq}=\frac{\alpha-\alpha_q^{\min}}{\alpha_q^{\max}-\alpha_q^{\min}} \tag{7.48}$$

where $\alpha_q^{\min}$ and $\alpha_q^{\max}$ are the minimum and maximum volume fraction (saturation) values, respectively. By default $\alpha_q^{\min}=\alpha_{\mathrm{qr}}$ and $\alpha_q^{\max}=1$ where α_{qr} is the residual saturation of phase q The value of α_{eq} is limited to values greater than 0.001 in order to avoid an infinite value for capillary pressure.

In order to solve Equation 7.47 (p. 1364), the entry pressure, P_e, and the parameter nb related to the pore size are required. The default values in Fluent are $P_e=440Pa$ and $nb=1.124$ which are appropriate for air-water flow in sand [29] (p. 5714).

7.2.3.8.17.2. Van-Genuchten Model

Another well established parametric model was proposed by van Genuchten [48] (p. 5715):

$$P_{c,q}=P_g\left[\left(S_{eq}\right)^{-1/m_g}-1\right]^{1/n_g} \tag{7.49}$$

where P_g is the entry pressure. n_g is the pore size distribution and $m_g=1-{}^{1}/_{n_g}$. The default values inFluent are $P_g=667Pa$ and $n_g=2.68$ which are appropriate for air-water flow in sand [29] (p. 5714).

7.2.3.8.17.3. Leverett J-Function

The J-function was developed by Leverett in an attempt to extend the pore-scale capillary pressure formulation by Washburn [29] (p. 5714) to a Macro-scale capillary pressure model. Leverett proposed that the averaged pore radius can be expressed in terms of porosity and permeability:

$$r_p=\sqrt{\frac{8K_{e,q}}{\gamma}} \tag{7.50}$$

Furthermore, he acknowledged that the capillary pressure should depend on porosity and permeability. By analogy to the Washburn equation, he developed a dimensionless J-function [85] (p. 5718):

$$J\left(S_q\right)=J\left(\alpha_q\right)=\frac{P_{c,q}}{\sigma\cos\theta_q}\sqrt{\frac{K_{e,q}}{\gamma}} \tag{7.51}$$

$$J(S_q)=a\left(S_q\right)^b \tag{7.52}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where σ is the surface tension, θ_q is the wetting or contact angle, and a and b are fitting parameters. Various empirical J-functions can be developed to estimate capillary pressure. In the Fluent, the following formulation is used:

$$P_{c,q}=\begin{cases}\sigma\cos\theta_q\sqrt{\frac{\gamma}{K_{e,q}}}\left[1.417\left(1-\alpha_q\right)-2.12\left(1-\alpha_q\right)^2+1.263\left(1-\alpha_q\right)^3\right] & \theta_q\leq 90^\circ \\ \sigma\cos\theta_q\sqrt{\frac{\gamma}{K_{e,q}}}\left[1.417\alpha_q-2.12\alpha_q^2+1.263\alpha_q^3\right] & \theta_q>90^\circ\end{cases} \tag{7.53}$$

This is the same formulation as is used in the Fluent Fuel Cell Module (see Liquid Water Formation, Transport, and its Effects (Low-Temperature PEMFC Only) in the *Fluent Theory Guide*).

7.2.3.8.17.4. Skjaeveland Model

All of the above-mentioned capillary pressure models (Brooks-Corey, Van-Genuchten, Leverett J-Function) are limited to primary drainage processes. Water saturation decreases in a primary drainage process. To model a displacement process where flooding is important, such as water-drive, models that describe imbibition are required. However, most reservoirs are mixed wettabilty and for these systems a more comprehensive model that describes both drainage and imbibition processes is required. The Skjaeveland model for capillary pressure encompasses these features.

The Skjaeveland model [145] (p. 5721) for capillary pressure treats the two fluids symmetrically for wettability. The Skjaeveland capillary pressure curve is defined by Equation 7.54 (p. 1365).

$$P_{c,q}=\frac{c_w}{\left(\left(\alpha_q-\alpha_{qr}\right)/\left(1-\alpha_{qr}\right)\right)^{a_w}}-\frac{c_{nw}}{\left(\left(1-\alpha_q-\alpha_{pr}\right)/\left(1-\alpha_{pr}\right)\right)^{a_{nw}}} \tag{7.54}$$

Where $P_{c,q}$ is the capillary pressure of the q^{th} phase, a_w and a_{nw} are fitting parameters, c_w and c_{nw} are constants, and α_q and α_p are the volume fractions of the two phases. w and nw subscripts refer to wetting and non-wetting phases, respectively.

The positive part of the capillary pressure curve is defined by c_w and a_w. c_{nw} and a_{nw} define the negative portion of the capillary pressure curve. The effects of varying c_w, a_w, c_{nw}, and a_{nw} on the capillary pressure curve are shown in Figure 7.27: Skjaeveland Correlation Behavior (p. 1366) [145] (p. 5721).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.27: Skjaeveland Correlation Behavior

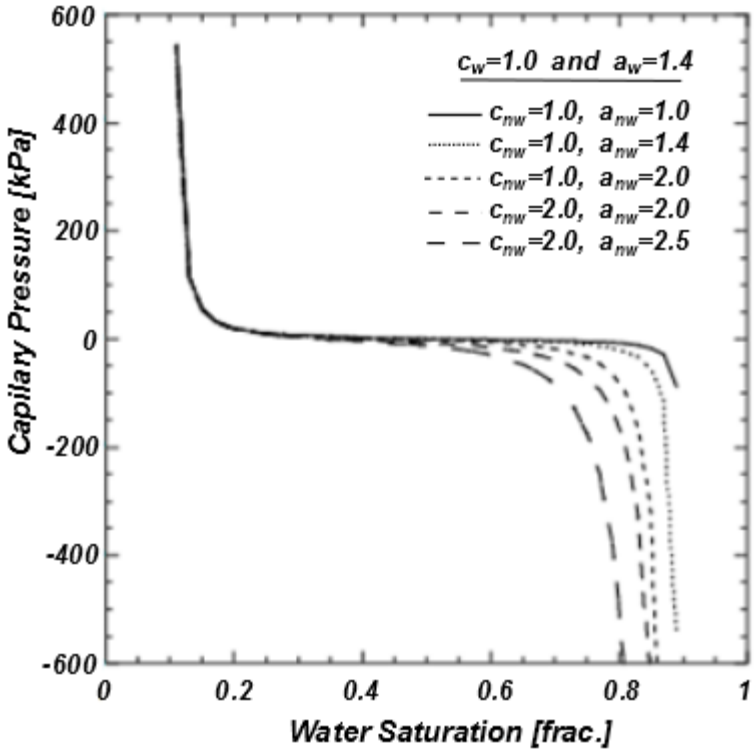


Figure 7.27: Skjaeveland Correlation Behavior (p. 1366) shows that there is a lot of flexibility in the capillary pressure curve. You can add further flexibility to the capillary pressure curve by estimating the residual oil saturation. Note that by varying c_w and a_w you can modify the cross-over point (while keeping c_{nw} and a_{nw} unchanged).

Parameter	Default Value
c_w	9.e5
c_{nw}	9.e5
a_w	0.05
a_{nw}	0.05

7.2.3.8.17.5. Capillary Pressure Data in a Tabular Format

Often, field data for capillary pressure are available as tabular functions of other solution variables. In, Ansys Fluent, you can use a tabular method to specify capillary pressure. Multiple files can be used to specify zone-specific properties, for example, different models for capillary pressure for different zones.

The text file format and the table reading procedure are described in Reading Files in Tabular Format (p. 969).

An example of the data file for the capillary pressure variable is shown below.

```
Default-Skjaevland-Model
Sw          Pc
0.2122      195399.0883
0.2245      156423.6016
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
0.2367      134012.8962
0.249       117936.4991
0.2612      105469.0929
0.2735      95079.83957
0.2857      86245.04955
0.298       78405.02466
```

Row 1 – The name of the table. The name should not exceed 128 characters and contain no spaces. For example, `Default-Skjaevland-Model` and `Default_Skjaevland_Model` are acceptable names.

Row 2 – Column names for the table input data (`Sw` and `Pc` in the above example).

Row 3 and all consecutive rows – Data points of the saturation and the corresponding capillary pressure values as a function of the saturation.

See Capillary Pressure Usage (p. 1367) for more information on tabular input for capillary pressure.

7.2.3.8.17.6. Capillary Pressure Usage

To model the capillary pressure:

1. In the **Fluid** dialog box for the phase mixture, in the **Porous Zone** tab, select **Corey Model** (**Model Options** group box).

2. Select a model from the drop-down list under **Capillary Pressure**. You can choose from the following models:

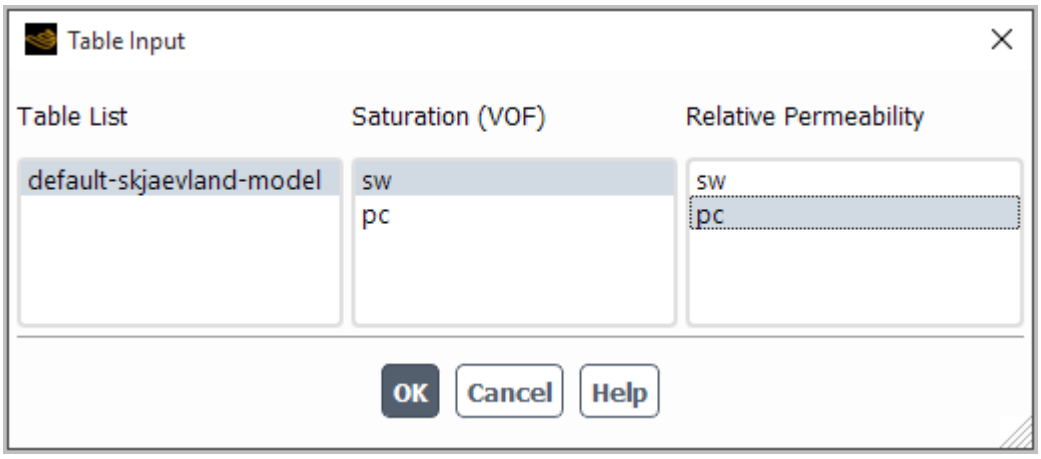
   - **brooks-corey**
   - **van-genuchten**
   - **leverett-j-function**
   - **skjaeveland**
   - **tabular**

     Note that prior to using the **tabular** option, you must load a text file containing capillary pressure data as described in Capillary Pressure Data in a Tabular Format (p. 1366).

3. (tabular only) In the **Table Input** dialog box that opens when you select **tabular**, select the table appropriate for your fluid zone from the **Table List** and the names of the columns from the **Saturation (VOF)** and **Capillary Pressure** selection lists.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.28: The Table Input Dialog Box for Capillary Pressure



For details about capillary pressure table files, see Capillary Pressure Data in a Tabular Format (p. 1366).

Note:

You can also define capillary pressure as a tabular function by using the text commands available under `define/boundary-conditions/set/fluid`. Once you enter `mixture` for `domain id/name`, you can select the `capillary-pressure` TUI option, set the method to `tabular` and specify its parameters.

4. Specify **Maximum Capillary Pressure**.

   The capillary pressure graph exhibits asymptotic behavior at the residual saturation of wetting and non-wetting phases where the capillary pressure approaches plus and minus infinity (see Figure 6.99: The Activate Cell Zones Dialog Box (p. 1279)). For this reason, the capillary pressure must be limited to avoid numerical problems.

5. (Brooks-Corey model only) In the secondary phase **Fluid** dialog box, in the **Porous Zone** tab, specify the model-specific parameters (**Brooks-Corey Model Inputs** group box):

   Entry pressure

   is P_e in Equation 7.47 (p. 1364).

   Pore size exponent

   is the parameter nb in Equation 7.47 (p. 1364).

6. (Van-Genuchten model only) In the secondary phase **Fluid** dialog box, in the **Porous Zone** tab, specify the model-specific parameters (**Van-Genuchten Model Inputs** group box):

   Entry pressure

   is P_g in Equation 7.49 (p. 1364).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Pore size exponent

is n_g in Equation 7.49 (p. 1364).

7. (Skjaeveland model only) In the **Fluid** dialog boxes for the primary and secondary phases, in the **Porous Zone** tab, specify the model-specific parameters (**Skjaeveland Model Inputs** group box):

   Entry pressure

   is a constant c_w for the wetting phase and c_{nw} for the non-wetting phase in Equation 7.54 (p. 1365).

   Pore size exponent

   is a_w for the wetting phase and a_{nw} for the non-wetting phase in Equation 7.54 (p. 1365).

8. (Leverett J-Function model only) Specify the following model-specific parameters:

   1. In the **Fluid** dialog boxes for the secondary phase, in the **Porous Zone** tab, in the **Leverett J-Function Model Inputs** group box, specify **Contact Angle** (θ_q in Equation 7.51 (p. 1364)).

   2. In the **Multiphase Model** dialog box, go to the **Phase Interaction** > **Forces** tab, and select **Surface Tension Force Modeling** (**Global Options** group box). Then in the **Force Setup** group box, specify **Surface Tension Coefficient** (σ in Equation 7.51 (p. 1364)).

Alternatively, you can specify a user-defined function that calculates the capillary pressure using a `DEFINE_EXCHANGE_PROPERTY` macro.

Note:

When modeling the capillary pressure, you should ensure that the wetting phases are chosen as the secondary phases.

Postprocessing

The following additional field variable is available for secondary phases:

- **Capillary-Pressure**

  Note that, although this item is available for both non-wetting and wetting phases, meaningful values are reported only for the wetting phase.

7.2.3.8.17.7. Modeling Capillary Pressure as Diffusion

By default, the capillary pressure term is included as a source in the wetting phase momentum equation as shown in Equation 7.22 (p. 1342). For two-phase cases, you can use an alternative approach where capillary pressure is modeled as a diffusion term in the wetting phase continuity equation. This approach provides numerical robustness and stability for certain cases.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The gradient of capillary pressure is expressed as a function of the wetting phase saturation α_w:

$$\nabla p_c = \frac{\partial p_c}{\partial \alpha_w} \nabla \alpha_w \tag{7.55}$$

The calculation of the gradient of capillary pressure with respect to the wetting phase volume fraction can be done analytically.

To model capillary pressure as diffusion, issue the following text command:

```
solve/set/multiphase-numerics/porous-media/capillary-pressure-as-diffusion?
model capillary pressure as diffusion? [no] yes
```

Note:

- When modeling the capillary pressure as diffusion, the secondary phase must be the wetting phase.
- Regardless of whether the flow is laminar or turbulent, Ansys Fluent uses only the viscous resistance in the calculation of capillary pressure gradient.

7.2.3.8.18. Defining Sources

If you want to include effects of the heat generated by the porous medium in the energy equation, enable the **Source Terms** option and set a nonzero **Energy** source. For porous media, energy source terms on the solid and gas sides are applied exactly as entered with no adjustment for porosity. A source term of 100 W/m ^{3} specified for a solid zone of volume 1.0 m ^{3} will, therefore, deliver 100 W even if the solids make up only 60% of that volume. You may also define sources of mass, momentum, turbulence, species, or other scalar quantities, as described in Defining Mass, Momentum, Energy, and Other Sources (p. 1382).

7.2.3.8.19. Defining Fixed Values

If you want to fix the value of one or more variables in the fluid region of the zone, rather than computing them during the calculation, you can do so by enabling the **Fixed Values** option. See Fixing the Values of Variables (p. 1377) for details.

7.2.3.8.20. Suppressing the Turbulent Viscosity in the Porous Region

As discussed in Treatment of Turbulence in Porous Media (p. 1339), turbulence will be computed in the porous region just as in the bulk fluid flow. If you are using one of the turbulence models (other than the Large Eddy Simulation model) and you want the turbulence generation to be zero in the porous zone, turn on the **Laminar Zone** option in the **Fluid** dialog box. Refer to Specifying a Laminar Zone (p. 1322) for more information about suppressing turbulence generation.

7.2.3.8.21. Specifying the Rotation Axis and Defining Zone Motion

Inputs for the rotation axis and zone motion are the same as for a standard fluid zone. See Inputs for Fluid Zones (p. 1321) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.2.3.9. Solution Strategies for Porous Media

In general, you can use the standard solution procedures and solution parameter settings when your Ansys Fluent model includes porous media. You may find, however, that the rate of convergence slows when you define a porous region through which the pressure drop is relatively large in the flow direction (for example, the permeability, α, is low or the inertial factor, C_2, is large). This slow convergence can occur because the porous media pressure drop appears as a momentum source term—yielding a loss of diagonal dominance—in the matrix of equations solved. The best remedy for poor convergence of a problem involving a porous medium is to supply a good initial guess for the pressure drop across the medium.

Similarly, **Standard Initialization** is the recommended initialization method for porous media simulations. The default **Hybrid Initialization** method does not account for the porous media properties, and depending on boundary conditions, may produce an unrealistic initial velocity field. For porous media simulations, the **Hybrid Initialization** method should only be used with the **Maintain Constant Velocity Magnitude** option.

Another possible way to deal with poor convergence is to temporarily disable the porous media model (by turning off the **Porous Zone** option in the Fluid Dialog Box (p. 4961)) and obtain an initial flow field without the effect of the porous region. With the porous media model turned off, Ansys Fluent will treat the porous zone as a fluid zone and calculate the flow field accordingly. Once an initial solution is obtained, or the calculation is proceeding steadily to convergence, you can enable the porous media model and continue the calculation with the porous region included. (This method is not recommended for porous media with high resistance.)

Simulations involving highly anisotropic porous media may, at times, pose convergence troubles. You can address these issues by limiting the anisotropy of the porous media coefficients ($1/\ \alpha_{ij}$ and $C_{2_{i,j}}$) to two or three orders of magnitude. Even if the medium's resistance in one direction is infinite, you do not need to set the resistance in that direction to be greater than 1000 times the resistance in the primary flow direction.

For cases that involve the non-equilibrium thermal model, using a default meshing partitioning method may cause convergence issues and /or solution dependence on the number of processors. In such a case, you should chose a meshing partition method that does not place the partition interface in the regions of significant variations in thermal conductivity and/or mesh size. For more information, see Mesh Partitioning Methods (p. 4331).

7.2.3.10. Postprocessing for Porous Media

The impact of a porous region on the flow field can be determined by examining either velocity components or pressure values. Graphical plots (including report plots of force|lift|drag|moment report definitions— Force and Moment Report Definitions (p. 4106), XY plots— XY Plots of Solution Data (p. 4060) , cumulative force/moment plots— Cumulative Force, Moment, and Coefficients Plots (p. 4094), and contour— Displaying Contours and Profiles (p. 3896) or vector— Displaying Vectors (p. 3912) plots) or alphanumeric reports of the following variables/functions may be of interest:

- **X**, **Y**, **Z Velocity** (in the **Velocity...** category)
- **Static Pressure** (in the **Pressure...** category)

These variables are contained in the specified categories of the variable selection drop-down list that appears in postprocessing dialog boxes.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that thermal reporting in the porous region is defined as follows:

$$k_{eff} = \gamma k_f + \left(1 - \gamma\right) k_s \tag{7.56}$$

where

γ = porosity of the medium

k_f = fluid phase thermal conductivity (including the turbulent contribution, k_t)

k_s = solid medium thermal conductivity

Important:

- Volume and mass integral reports do not account for porosity effects in the porous media.
- For porous media involving surface reactions, you can display/report the surface reaction rates using the **Arrhenius Rate of Reaction-n** in the **Reactions...** category of the variable selection drop-down list.

Important:

Special care must be taken when postprocessing a porous media simulation that utilizes the non-equilibrium thermal model. See Non-Equilibrium Thermal Model (p. 1355) for details.

7.2.4. 3D Fan Zones

3D fan zones are fluid cell zones that simulate the effect of an axial fan by applying a distributed momentum source in a toroid-shaped fluid volume (that is, a blade-swept volume). This is in contrast to fan boundary conditions (for example, a boundary zone of type **fan** or **intake-fan**), which are based on a lumped parameter model in which the fan is considered to be infinitely thin and a discontinuous pressure rise is specified as a function of velocity. 3D fan zones have the following advantages:

- 3D fan zones can calculate swirl and radial velocities, as opposed to fan boundary conditions, which are mainly used to simulate the axial flow and require you to input swirl and radial coefficients.
- The results for 3D fan zones can be comparable to moving reference frame (MRF) simulations, and do not require modeling 3D rotating fan blade geometries.
- 3D fan zones can simulate a fan even if the available fan curve data was measured at different operating conditions than that of the simulation.

Note:

Fluid zones designated as 3D fan zones cannot have non-conformal interfaces.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.2.4.1. Momentum Equations for 3D Fan Zones

The following equations are used in the 3D fan zone for the momentum sources in the axial, tangential, and radial directions, which mimic the effect of the fan on the fluid. The tangential momentum source is based on a turbomachinery relation, whereas the radial momentum source is based on a centrifugal force balance.

$$\text{Axial: } S_a = \Delta P(Q)/h \tag{7.57}$$

$$\text{Tangential: } S_t = \begin{cases} \frac{2W_{fan}r}{c_1|\Omega_{operating}|} & \text{for } R_h < r \le R_{ip} \\ \frac{2W_{fan}R_{ip}^2}{c_1 r|\Omega_{operating}|} & \text{for } R_{ip} < r < R_t \end{cases} \tag{7.58}$$

$$\text{Radial: } S_r = \frac{\rho V_\phi^2}{r} \tag{7.59}$$

where

$\Delta P(Q)$ = pressure rise across the fan for a given axial flow rate Q

h = thickness of the toroid region swept by the blades in the axial direction

W_{fan} = fan power

r = local radial distance from the fan axis

$\Omega_{operating}$ = fan operating angular velocity

R_h = radius of the fan hub

R_{ip} = radius of a point on the fan blade based on the inflection point ratio

R_t = radius of the fan blade tip

ρ = fluid density

V_ϕ = local tangential velocity

In order to calculate the tangential source, three other expressions are necessary:

$$W_{fan} = (\mathbf{v}_{exit} \cdot \mathbf{a}_{exit}) \Delta P(Q) \tag{7.60}$$

$$R_{ip} = R_h + (R_t - R_h) b_{ip} \tag{7.61}$$

$$c_1 = \pi h \left(R_{ip}^2 \left(2R_t^2 - R_{ip}^2 \right) - R_h^4 \right) \tag{7.62}$$

where

$\mathbf{v}_{exit}$ = velocity vector at the fan exit plane

$\mathbf{a}_{exit}$ = area vector at the fan exit plane

b_{ip} = the inflection point ratio, that is, the fraction of the fan blade length (starting at the hub) over which the tangential velocity of the fan discharge is increasing with increasing radius and peaks, before tapering off (see Figure 7.30: The Inflection Point Ratio of a Pitched Blade Turbine (p. 1376))

The pressure rise across the 3D fan zone is specified as a function of the volumetric flow rate through the fan. The relationship may be a constant or a fan curve, that is, a polynomial or piecewise-linear function. In the case of a polynomial, the relationship is of the form

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\Delta P(Q) = \sum_{n=1}^{N} f_n Q^{n-1} \tag{7.63}$$

where f_n are the pressure head polynomial coefficients (calculated by Fluent), and Q is the flow rate normal to the fan.

Important:

The flow rate Q can be either positive or negative. You must be careful to model the fan so that a pressure rise occurs for forward flow through the fan.

The fan curve data is typically collected at a prescribed set of test conditions. However, the operating conditions of your CFD simulation may be inconsistent with this data (for example, the rotational speed or fluid temperature may be different). To account for this, the 3D fan model can adjust the fan curve data according to the following scaling relationships:

$$P_{\text{operating}}(Q) = P_{\text{test}}(Q)\sqrt{\frac{\Omega_{\text{operating}}}{\Omega_{\text{test}}}} \tag{7.64}$$

$$Q_{\text{operating}} = Q_{\text{test}}\sqrt{\frac{\Omega_{\text{operating}}}{\Omega_{\text{test}}}} \tag{7.65}$$

$$W_{\text{fan,operating}} = W_{\text{fan,test}}\frac{T_{\text{test}}}{T_{\text{avg}}} \tag{7.66}$$

where T_{test} and T_{avg} are the test temperature and the volume-averaged temperature of the fan cell zone (which is calculated internally), respectively.

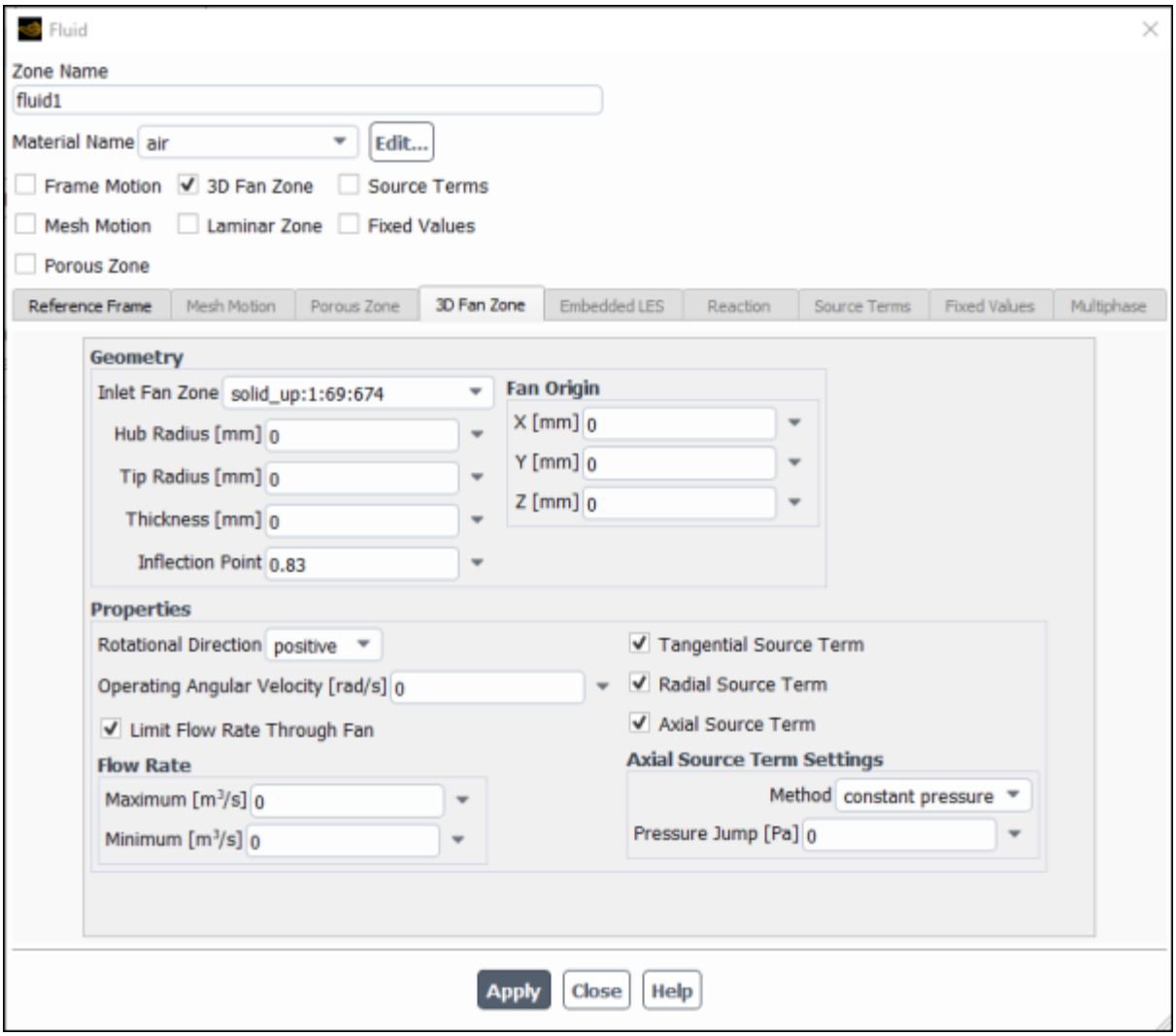
7.2.4.2. User Inputs for 3D Fan Zones

A 3D fan zone is modeled as a special type of fluid zone. The cell zone should have a toroid shape that is sized to match the blade-swept volume of the fan you are simulating (that is, has an inner and outer radius that matches the radius of the fan's hub and blade tips, respectively, as well as a length that matches the thickness of the toroid region swept by the blades in the axial direction). Furthermore, the fluid cell zone should be "interior", that is, have at least two boundary zones of type **interior** that border another fluid cell zone and act as the fan inlet and outlet (the other boundaries can be of type **interior** as well).

To designate a cell zone as a 3D fan zone, enable the **3D Fan Zone** option in the **Fluid** dialog box and then define the geometry and properties of the fan in the **3D Fan Zone** tab.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.29: The Fluid Dialog Box for a 3D Fan Zone



7.2.4.2.1. Defining the Geometry of a 3D Fan Zone

After enabling the **3D Fan Zone** option in the **Fluid** dialog box, you can begin setting it up by defining the settings in the **Geometry** group box in the **3D Fan Zone** tab of the fluid cell zone:

1. Select a boundary zone from the **Inlet Fan Zone** drop-down list, which will act as the inlet zone for the 3D fan zone and the surface on which the fan will be located. This boundary must meet certain requirements: it must be of type **interior**; it must be planar; and it must act as a boundary between the 3D fan zone and another fluid cell zone. Note that the direction the fan blows will be *into* the 3D fan zone, perpendicular to this boundary zone.

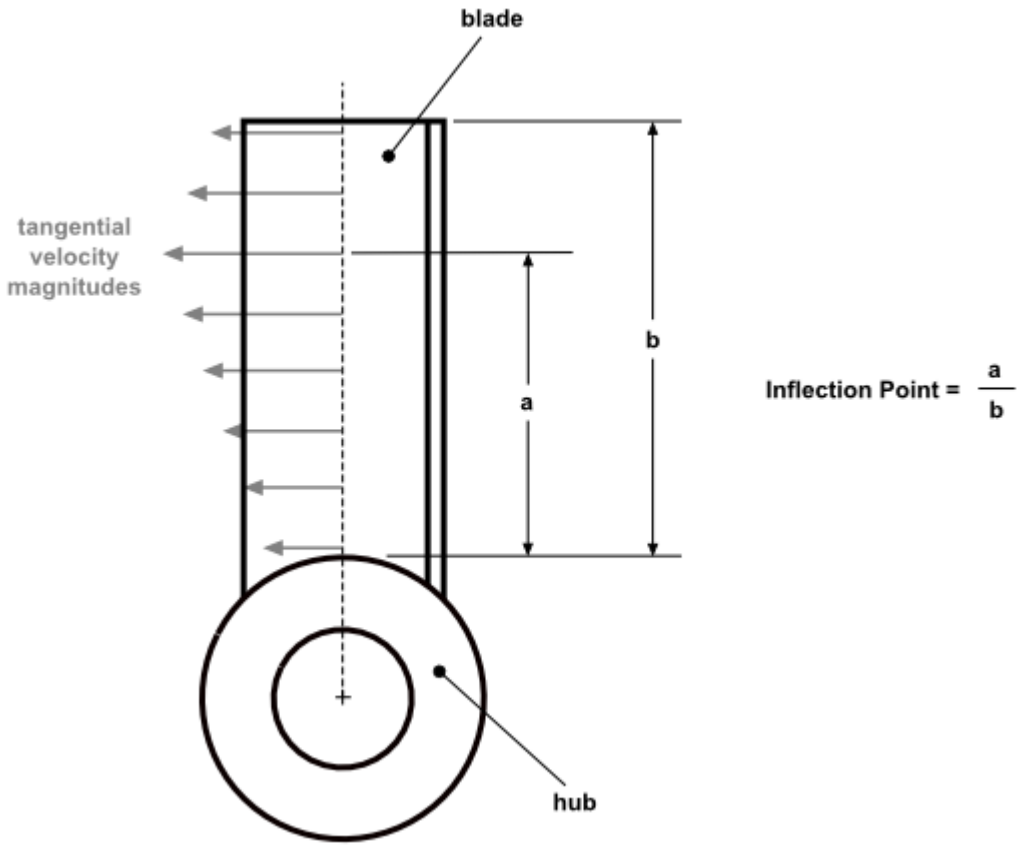
2. Define the overall dimensions of the fan by entering values for the **Hub Radius** and the **Tip Radius**, as well as the **Thickness** of the toroid region swept by the blades in the axial direction. Note that the values must fit within the confines of the 3D fan zone, and that the **Thickness** will be applied from the **Inlet Fan Zone** into the 3D fan zone.

3. Specify the **Inflection Point** ratio. The inflection point ratio represents the fraction of the fan blade length (starting at the hub) over which the tangential velocity of the fan discharge is increasing with increasing radius and peaks, before tapering off. See Figure 7.30: The Inflec-

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

tion Point Ratio of a Pitched Blade Turbine (p. 1376) for an illustration. You can derive this ratio empirically from experimental data; otherwise, you can first use Ansys Fluent to model the actual fan geometry (for example, using a moving reference frame simulation) and then use those results to determine the input parameters for simpler simulations using the **3D Fan Zone** option.

Figure 7.30: The Inflection Point Ratio of a Pitched Blade Turbine



4. Enter the coordinates for the **Fan Origin**. This must define a point that is on the plane of the **Inlet Fan Zone** and located such that the diameters of the fan fall within boundaries of the **Inlet Fan Zone**.

7.2.4.2.2. Defining the Properties of a 3D Fan Zone

To complete the setup of a 3D fan zone, you must define the settings in the **Properties** group box in the **3D Fan Zone** tab of the fluid cell zone:

1. Define the **Rotational Direction** of the fan. The rotation is deemed **positive** if it follows the right-hand rule with respect to the fan direction vector (that is, a vector from the **Fan Origin** that points into the 3D fan zone perpendicular to the **Inlet Fan Zone**).

2. Specify the **Operating Angular Velocity** of the fan. This must be a positive nonzero number.

3. You can enable the **Limit Flow Rate Through Fan** option, and thereby specify the **Maximum** and **Minimum** flow rates that are allowed.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. You can enable the **Tangential Source Term**, **Radial Source Term**, and/or **Axial Source Term** options, in order to enable momentum sources in the specified directions. Note that the tangential and radial sources are defined according to impeller theory, and so no further inputs are necessary, whereas the axial source term requires additional input (as described in the step that follows).

5. If you have enabled the **Axial Source Term** option, you must define this source in the **Axial Source Term Settings** group box. For the **Method**, you can select one of the following choices:

   - **constant pressure**

     With this method, you must then specify a constant **Pressure Jump** that is applied to all of the cells in the 3D fan zone.

   - **fan curve**

     This method allows you to specify the pressure jump through a fan curve (pressure rise vs. volumetric flow rate). Using the **Read Fan Curve...** button, you can read in a tab-delimited ASCII file that defines the relationship between pressure and flow rate for the 3D fan cell zone (see 3D Fan Curve File Format (p. 5694) for details). You must then specify the **Curve Fitting Method** Fluent should use on the data points in your file, selecting either **polynomial** or **piecewise-linear** (for the former, you must then define the **Order of Polynomial**). Finally, you must provide an estimate of the **Initial Flow Rate** through your fan zone, as well as the **Test Angular Velocity** and **Test Temperature** of the experiment that produced the data points.

     Note that a single file can define multiple cell zones, but the file must be read in each of the applicable **Fluid** dialog boxes.

7.2.4.3. 3D Fan Zone Limitations

Note the following limitations when setting up a 3D fan zone:

- 3D fan zones are not supported with the premixed combustion model.
- 3D fan zones are not supported with the Eulerian multiphase model.
- The **Energy Equation** must be enabled in the Energy Dialog Box (p. 4714) for 3D fan zones when **fan curve** is selected for the axial source term method.

7.2.5. Fixing the Values of Variables

The option to fix values of variables in Ansys Fluent allows you to set the value of one or more variables in a fluid or solid zone, essentially setting a boundary condition for the variables within the cells of the zone. When a variable is fixed in a given cell, the transport equation for that variable is not solved in the cell (and the cell is not included when the residual sum is computed for that variable). The

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

fixed value is used for the calculation of face fluxes between the cell and its neighbors. The result is a smooth transition between the fixed value of a variable and the values at the neighboring cells.

Important:

You can fix values for temperature, species mass fractions, and/or velocity components only if you are using the pressure-based solver. (Refer to Pressure-Based Solver in the Theory Guide for information about the pressure-based solver.)

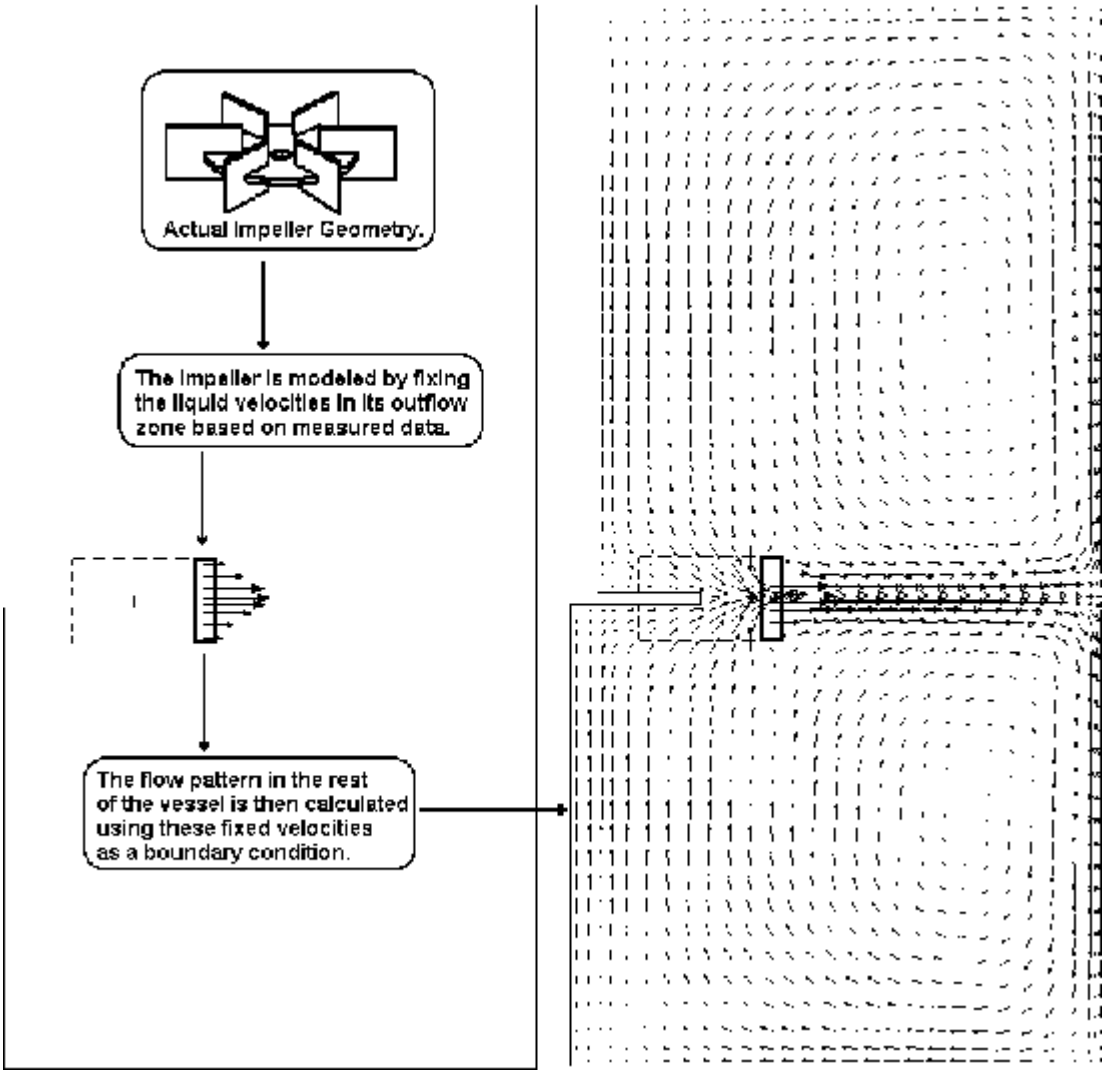
7.2.5.1. Overview of Fixing the Value of a Variable

The ability to fix the value of a variable has a wide range of applications. The velocity fixing method is often used to model the flow in stirred tanks. This approach provides an alternative to the use of a moving reference frame (solution in the reference frame of the blade) and can be used to model baffled tanks. In both 2D and 3D geometries, a fluid cell zone may be used in the impeller regions, and velocity components can be fixed based on measured data.

Although the actual impeller geometry can be modeled and the flow pattern calculated using the sliding mesh model, experimental data for the velocity profile in the outflow region are available for many impeller types. If you do not need to know the details of the flow around the blades for your problem, you can model the impeller by fixing the experimentally-obtained liquid velocities in its outflow zone. The velocities in the rest of the vessel can then be calculated using this fixed velocity profile as a boundary condition. Figure 7.31: Fixing Values for the Flow in a Stirred Tank (p. 1379) shows an example of how this method is used to model the flow pattern created by a disk-turbine in an axisymmetric stirred vessel.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.31: Fixing Values for the Flow in a Stirred Tank



7.2.5.1.1. Variables That Can Be Fixed

The variables that can be fixed include velocity components (pressure-based solver only), turbulence quantities, temperature (pressure-based solver only), enthalpy, species mass fractions (pressure-based solver only), and user-defined scalars. For turbulence quantities, different values can be set depending on your choice of turbulence model. You can fix the value of the temperature in a fluid or solid zone if you are solving the energy equation. If you are using the non-premixed combustion model, you can fix the enthalpy in a fluid zone. If you have more than one species in your model, you can specify fixed values for the species mass fractions for each individual species except the last one you defined. See the Fluent Customization Manual for details about defining user-defined scalars.

If you are using the Eulerian multiphase model, you can fix the values of velocity components and (depending on which multiphase turbulence model you are using) turbulence quantities on a per-phase basis. See Eulerian Model (p. 2968) for details about setting boundary conditions for Eulerian multiphase calculations.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.2.5.2. Procedure for Fixing Values of Variables in a Zone

To fix the values of one or more variables in a cell zone, follow these steps (remembering to use only SI units):

1. In the Fluid Dialog Box (p. 4961) or Solid Dialog Box (p. 4973), enable the **Fixed Values** option.

2. Fix the values for the appropriate variables in the **Fixed Values** tab, noting the comments below.

   - To specify a constant value for a variable, choose **constant** in the drop-down list next to the relevant field and then enter the constant value in the field.

   - To specify a non-constant value for a variable, you can use a profile (see Profiles (p. 1564)) or a user-defined function for a profile (see the Fluent Customization Manual). Select the appropriate profile or UDF in the drop-down list next to the relevant field.

     If you specify a radial-type profile (see Profile Specification Types (p. 1564)) for temperature, enthalpy, species mass/mole fractions, or turbulence quantities for the k - ε, Spalart-Allmaras, or k - ω model, the local coordinate system upon which the radial profile is based is defined by the **Rotation-Axis Origin** and **Rotation-Axis Direction** for the fluid zone. See Specifying the Rotation Axis (p. 1323) for information about setting these parameters. (Note that it is acceptable to specify the rotation axis and direction for a non-rotating zone. This will not cause the zone to rotate; it will not rotate unless it has been explicitly defined as a moving zone.)

   - If you do not want to fix the value for a variable, choose (or keep) **none** in the drop-down list next to the relevant field. This is the default for all variables.

7.2.5.2.1. Fixing Velocity Components

To fix the velocity components, you can specify **X**, **Y**, and (in 3D) **Z Velocity** values, or, for axisymmetric cases, **Axial**, **Radial**, and (for axisymmetric swirl) **Swirl Velocity** values. The units for a fixed velocity are m/s.

For 3D cases, you can choose to specify cylindrical velocity components instead of Cartesian components. Turn on the **Local Coordinate System For Fixed Velocities** option, and then specify the **Axial**, **Radial**, and/or **Tangential Velocity** values. The local coordinate system is defined by the **Rotation-Axis Origin** and **Rotation-Axis Direction** for the fluid zone. See Specifying the Rotation Axis (p. 1323) for information about setting these parameters. (Note that it is acceptable to specify the rotation axis and direction for a non-rotating zone. This will not cause the zone to rotate; it will not rotate unless it has been explicitly defined as a moving zone.)

Important:

Note the following:

- You can fix values for velocity components only if you are using the pressure-based solver. (Refer to Pressure-Based Solver in the Theory Guide for information about the pressure-based solver.)

- If all components of velocity are set to fixed values, the solver will also set the gauge pressure to zero on all cells in the cell zone.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.2.5.2.2. Fixing Temperature and Enthalpy

If you are solving the energy equation, you can fix the temperature in a zone by specifying the value of the **Temperature**. The units for a fixed temperature are K.

If you are using the non-premixed combustion model, you can fix the enthalpy in a zone by specifying the value of the **Enthalpy**. The units for a fixed enthalpy are J/kg.

If you specify a radial-type profile (see Profile Specification Types (p. 1564)) for temperature or enthalpy, the local coordinate system upon which the radial profile is based is defined by the **Rotation-Axis Origin** and **Rotation-Axis Direction** for the fluid zone. See above for details.

Important:

You can fix the value of temperature only if you are using the pressure-based solver.

7.2.5.2.3. Fixing Species Mass Fractions

If you are using the species transport model, you can fix the values of the species mass fractions for individual species. Ansys Fluent allows you to fix the species mass fraction for each species (for example, **h2**, **o2**) except the last one you defined.

If you specify a radial-type profile (see Profile Specification Types (p. 1564)) for a species mass fraction, the local coordinate system upon which the radial profile is based is defined by the **Rotation-Axis Origin** and **Rotation-Axis Direction** for the fluid zone. See above for details.

Important:

You can fix values for species mass fractions only if you are using the pressure-based solver.

7.2.5.2.4. Fixing Turbulence Quantities

To fix the values of k and ε in the k - ε equations, specify the **Turbulence Kinetic Energy** and **Turbulence Dissipation Rate** values. The units for k are m^2/s^2 and those for ε are m^2/s^3.

To fix the value of the modified turbulent viscosity ($\tilde{\nu}$) for the Spalart-Allmaras model, specify the **Modified Turbulent Viscosity** value. The units for the modified turbulent viscosity are m^2 /s.

To fix the values of k and ω in the k - ω equations, specify the **Turbulence Kinetic Energy** and **Specific Dissipation Rate** values. The units for k are m^2/s^2 and those for ω are 1/s.

To fix the value of k, ε, or the Reynolds stresses in the RSM transport equations, specify the **Turbulence Kinetic Energy**, **Turbulence Dissipation Rate**, **Specific Dissipation Rate**, **UU Reynolds Stress**, **VV Reynolds Stress**, **WW Reynolds Stress**, **UV Reynolds Stress**, **VW Reynolds Stress**, and/or **UW Reynolds Stress**. The units for k and the Reynolds stresses are m^2/s^2, and those for ε are m^2/s^3.

If you specify a radial-type profile (see Profile Specification Types (p. 1564)) for k, ε, ω, or $\tilde{\nu}$, the local coordinate system upon which the radial profile is based is defined by the **Rotation-Axis**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Origin and **Rotation-Axis Direction** for the fluid zone. See above for details. Note that you cannot specify radial-type profiles for the Reynolds stresses.

7.2.5.2.5. Fixing User-Defined Scalars

To fix the value of a user-defined scalar, specify the **User-defined scalar-n** value. (There will be one for each user-defined scalar you have defined.) The units for a user-defined scalar will be the appropriate SI units for the scalar quantity. See the Fluent Customization Manual for information on user-defined scalars.

7.2.6. Locking the Temperature for Solid and Shell Zones

You can lock (or "freeze") the temperature values for all the cells in solid zones (including cells to which you have applied an energy source using a UDF) and in walls that have shell conduction enabled, so that the values do not change during further solver iterations. When the temperature is locked for a given cell, the transport equation will still be solved in the cell, and the cell will be included when the residual sum is computed.

You can lock / unlock the temperature for solid and shell zones by using the following text command:

`solve → set → lock-solid-temperature?`

Note the following about the option for locking the temperature of solid and shell zones:

- It is only available when energy is enabled and the pressure-based solver is selected.
- When this option is used, the **BCGSTAB** option from the **Stabilization Method** drop-down list of the **Advanced Solution Controls** dialog box is not supported.
- Alternative means of monitoring the convergence of energy may be used.

7.2.7. Defining Mass, Momentum, Energy, and Other Sources

You can define volumetric sources of mass (for single or multiple species), momentum, energy, turbulence, and other scalar quantities in a fluid zone, or a source of energy for a solid zone. This feature is useful when you want to input a known value for these sources. (For more complicated sources with functional dependency, you can create a user-defined function as described in the separate Fluent Customization Manual.) To add source terms to a cell or group of cells, you must place the cell(s) in a separate zone. The sources are then applied to that cell zone. Typical uses for this feature are listed below:

- A flow source that cannot be represented by an inlet, for example, due to an issue of scale. If you need to model an inlet that is smaller than a cell, you can place the cell where the tiny "inlet" is located in its own fluid zone and then define the mass, momentum, and energy sources in that cell zone. For the example shown in Figure 7.32: Defining a Source for a Tiny Inlet (p. 1383), you should set a mass source of $\frac{\dot{m}}{V} = \frac{\rho_j A_j v_j}{V}$ and a momentum source of $\frac{\dot{m}v}{V} = \frac{\dot{m}v_j}{V}$, where V is the cell volume.
- Heat release due to a source (for example, fire) that is not explicitly defined in your model. For this case, you can place the cell(s) into which the heat is originally released in its own fluid zone and then define the energy source in that cell zone.

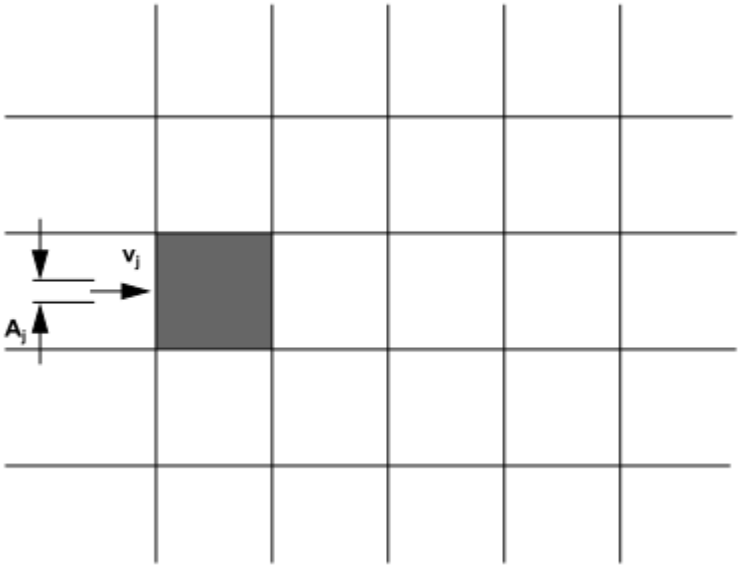
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- An energy source in a solid zone, for conjugate heat transfer applications. For this case, you can place the cell(s) into which the heat is originally released in its own solid zone and then define the energy source in that cell zone.
- A species source due to a reaction that is not explicitly included in the model. In the above example of simulating a fire, you might need to define a source for a species representing smoke generation.

Important:

Note that if you define a mass source for a cell zone, you should also define a momentum source and, if appropriate for your model, energy and turbulence sources. If you define only a mass source, that mass will assume the velocity, temperature/enthalpy of the corresponding cell. The mass will therefore have to be accelerated and heated by the flow and, consequently, there may be a drop in velocity or temperature. This drop may or may not be perceptible, depending on the size of the source. (Note that defining only a momentum, energy, or turbulence source is acceptable.)

Figure 7.32: Defining a Source for a Tiny Inlet



7.2.7.1. Sign Conventions and Units

All positive source terms indicate sources, and all negative source terms indicate sinks. All sources must be specified in SI units.

7.2.7.2. Procedure for Defining Sources

To define one or more source terms for a zone, follow these steps (remembering to use only SI units):

1. In the Fluid Dialog Box (p. 4961) or Solid Dialog Box (p. 4973), turn on the **Source Terms** option.
2. Set the appropriate source terms under the **Source Terms** tab, noting the comments below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To specify a source, click the **Edit...** button next to the mass, momentum, energy, or other source. The sources dialog box will open where you will define the number of sources. For each source, choose **none**, **constant**, or **New Input Parameter...** in the drop-down list.
- If you do not want to specify a source term for a variable, choose (or keep) **none** in the drop-down list next to the relevant field. This is the default for all variables.
- To specify a constant source, choose **constant** in the drop-down list and then enter the constant value in the field.
- To specify a new source parameter, choose **New Input Parameter...** in the drop-down list and enter the **Name** or the parameter and the **Current Value** as a constant.
- It is not recommended to only define a mass source without defining the other sources, as described in Defining Mass, Momentum, Energy, and Other Sources (p. 1382) above.
- Since the sources you specify are defined per unit volume, to determine the appropriate value of your source term you will often need to first determine the volume of the cell(s) in the zone for which you are defining the source. To do this, you can use the Volume Integrals Dialog Box (p. 5329).
- In this guide, source is used as a generic term for both source and sink; where source means addition, and sink means subtraction to the physics model. Addition of a source usually leads to the opposite outcome of a sink. For example, a momentum source will accelerate the flow, and in contrast, a momentum sink will decelerate the flow. In general, a source term can be replaced with a sink term with opposite expected outcome.

More details are discussed in the sections that follow.

7.2.7.2.1. Mass Sources

7.2.7.2.2. Momentum Sources

7.2.7.2.3. Energy Sources

7.2.7.2.4. Turbulence Sources

7.2.7.2.5. Mean Mixture Fraction and Variance Sources

7.2.7.2.6. P-1 Radiation Sources

7.2.7.2.7. Progress Variable Sources

7.2.7.2.8. NO, HCN, and NH3 Sources for the NOx Model

7.2.7.2.9. Discrete Bin Fraction Sources for the Population Balance Model

7.2.7.2.10. User-Defined Scalar (UDS) Sources

7.2.7.2.1. Mass Sources

If you have only one species in your problem, you can simply define a **Mass** source for that species. The units for the mass source are $kg/m^3–s$. In the continuity equation (Equation 1.1 in the Theory Guide), the defined mass source will appear in the S_m term.

If you have more than one species, you can specify mass sources for each individual species. There will be a total **Mass** source term as well as a source term listed explicitly for each species (for example, **h2**, **o2**) except the last one you defined. If the total of all species mass sources (including the last one) is 0, then you should specify a value of 0 for the **Mass** source, and also

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

specify the values of the nonzero individual species mass sources. Since you cannot specify the mass source for the last species explicitly, Ansys Fluent will compute it by subtracting the sum of all other species mass sources from the specified total **Mass** source.

For example, if the mass source for hydrogen in a hydrogen-air mixture is 0.01, the mass source for oxygen is 0.02, and the mass source for nitrogen (the last species) is 0.015, you will specify a value of 0.01 in the **h2** field, a value of 0.02 in the **o2** field, and a value of 0.045 in the **Mass** field. This concept also applies within each cell if you use user-defined functions for species mass sources.

The units for the species mass sources are kg/m^3–s. In the conservation equation for a chemical species (Equation 7.1 in the Theory Guide), the defined mass source will appear in the S_i term.

Note:

When a mass source is added in the continuity equation, a corresponding energy source is added in the energy equation at a reference temperature of 0 K for inviscid flows and 298.15 K for viscous flows.

7.2.7.2.2. Momentum Sources

To define a source of momentum, specify the **X Momentum**, **Y Momentum**, and/or **Z Momentum** term. The units for the momentum source are N/ m^3. In the momentum equation (Equation 1.3 in the Theory Guide), the defined momentum source will appear in the $\vec{F}$ term.

7.2.7.2.3. Energy Sources

To define a source of energy, specify an **Energy** term. The units for the energy source are W/ m^3. In the energy equation (Equation 5.1 in the Theory Guide), the defined energy source will appear in the S_h term.

7.2.7.2.4. Turbulence Sources

7.2.7.2.4.1. Turbulence Sources for the k-ε Model

To define a source of k or ε in the k - ε equations, specify the **Turbulent Kinetic Energy** or **Turbulent Dissipation Rate** term. The units for the k source are kg/m- s^3 and those for ε are kg/m- s^4.

The defined k source will appear in the S_k term on the right-hand side of the turbulent kinetic energy equation (for example, Equation 4.39 in the Theory Guide).

The defined ε source will appear in the S_ε term on the right-hand side of the turbulent dissipation rate equation (for example, Equation 4.40 in the Theory Guide).

7.2.7.2.4.2. Turbulence Sources for the Spalart-Allmaras Model

To define a source of modified turbulent viscosity, specify the **Modified Turbulent Viscosity** term. The units for the modified turbulent viscosity source are kg/m- s^2. In the transport equation for the Spalart-Allmaras model (Equation 4.15 in the Theory Guide), the defined modified turbulent viscosity source will appear in the $S_{\tilde{\nu}}$ term.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.2.7.2.4.3. Turbulence Sources for the k- ω Model

To define a source of k or ω in the k - ω equations, specify the **Turbulent Kinetic Energy** or **Specific Dissipation Rate** term. The units for the k source are kg/m- s^3 and those for ω are kg/ m^3 - s^2.

The defined k source will appear in the S_k term on the right-hand side of the turbulent kinetic energy equation (Equation 4.71 in the Theory Guide).

The defined ω source will appear in the S_ω term on the right-hand side of the specific turbulent dissipation rate equation (Equation 4.72 in the Theory Guide).

7.2.7.2.4.4. Turbulence Sources for the Reynolds Stress Model

To define a source of k, ε, or the Reynolds stresses in the RSM transport equations, specify the **Turbulence Kinetic Energy**, **Turbulence Dissipation Rate**, **Specific Dissipation Rate**, **UU Reynolds Stress**, **VV Reynolds Stress**, **WW Reynolds Stress**, **UV Reynolds Stress**, **VW Reynolds Stress**, and/or **UW Reynolds Stress** terms. The units for the k source and the sources of Reynolds stress are kg/m- s^3, and those for ε are kg/m- s^4.

The defined Reynolds stress sources will appear in the S_{user} term on the right-hand side of the Reynolds stress transport equation (Equation 4.226 in the Theory Guide).

The defined k source will appear in the S_k term on the right-hand side of Equation 4.254 in the Theory Guide.

The defined ε will appear in the S_ε term on the right-hand side of Equation 4.257 in the Theory Guide.

7.2.7.2.5. Mean Mixture Fraction and Variance Sources

To define a source of the mean mixture fraction or its variance for the non-premixed combustion model, specify the **Mean Mixture Fraction** or **Mixture Fraction Variance** term. The units for the mean mixture fraction source are kg/m^3–s, and those for the mixture fraction variance source are kg/m^3–s.

The defined mean mixture fraction source will appear in the S_{user} term in the transport equation for the mixture fraction (Equation 8.4 in the Theory Guide).

The defined mixture fraction variance source will appear in the S_{user} term in the transport equation for the mixture fraction variance (Equation 8.5 in the Theory Guide).

If you are using the two-mixture-fraction approach, you can also specify sources of the **Secondary Mean Mixture Fraction** and **Secondary Mixture Fraction Variance**.

7.2.7.2.6. P-1 Radiation Sources

To define a source for the P-1 radiation model, specify the **P1** term. The units for the radiation source are W/ m^3, and the defined source will appear in the S_G term in Equation 5.57 in the Theory Guide.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that, if the source term you are defining represents a transfer from internal energy to radiative energy (for example, absorption or emission), you must specify an **Energy** source of the same magnitude as the **P1** source, but with the opposite sign, in order to ensure overall energy conservation.

7.2.7.2.7. Progress Variable Sources

To define a source of the progress variable for the premixed combustion model, specify the **Progress Variable** term. The units for the progress variable source are kg/ m^3 -s, and the defined source will appear in the ρS_c term in Equation 8.70 in the Theory Guide.

7.2.7.2.8. NO, HCN, and NH3 Sources for the NOx Model

To define a source of NO, HCN, or NH_3 for the NOx model, specify the **no**, **hcn**, or **nh3** term. The units for these sources are kg/ m^3 -s, and the defined sources will appear in the S_{NO}, S_{HCN}, and S_{NH3} terms of Equation 9.1, Equation 9.2, and Equation 9.3 in the Theory Guide.

7.2.7.2.9. Discrete Bin Fraction Sources for the Population Balance Model

For the **Discrete** and **Inhomogeneous Discrete** population balance models, you can specify **Bin-*i*-fraction** source terms (where i is the bin label). Note that bins are labeled from 0 to $N-1$, where N is the number of bins that you specified in the **Population Balance Model** tab of the **Multiphase Model** dialog box. The units for the fraction source are kg/m^3 s.

7.2.7.2.10. User-Defined Scalar (UDS) Sources

You can specify source term(s) for each UDS transport equation you have defined in your model. See Setting Up UDS Equations in Ansys Fluent (p. 1739) for details.

7.3. Operating Conditions

7.3.1. Buoyancy-Driven Flows and Natural Convection

A buoyancy-driven flow is induced due to the force of gravity acting on the density variations. When the density variation is caused by temperature, such buoyancy-driven flows are termed as natural convection flows.

For more information about the theory behind natural convection and buoyancy-driven flows, see Natural Convection and Buoyancy-Driven Flows Theory in the Theory Guide.

7.3.1.1. Modeling Natural Convection in a Closed Domain

When you model natural convection inside a closed domain, the solution will depend on the mass inside the domain. Since this mass will not be known unless the density is known, you must model the flow in one of the following ways:

- Perform a transient calculation. In this approach, the initial density will be computed from the initial pressure and temperature, so the initial mass is known. As the solution progresses over time, this mass will be properly conserved. If the temperature differences in your domain are large, you must follow this approach.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Perform a steady-state calculation using the Boussinesq model (described in The Boussinesq Model (p. 1388)). In this approach, you will specify a constant density, so the mass is properly specified. This approach is valid only if the temperature differences in the domain are small. If not, you use the transient approach.

  Important:

  For a closed domain, you can use the *incompressible* ideal gas law only with a *fixed* operating pressure. It *cannot* be used with a floating operating pressure. You can use the *compressible* ideal or real gas law with either *floating* or *fixed* operating pressure.

See Floating Operating Pressure (p. 1763) for more information about the floating operating pressure option.

7.3.1.2. The Boussinesq Model

For many natural-convection flows, you can get faster convergence with the Boussinesq model than you can get by setting up the problem with fluid density as a function of temperature. This model treats density as a constant value in all solved equations, except for the buoyancy term in the momentum equation:

$$\left(\rho-\rho_0\right)g \approx -\rho_0\beta\left(T-T_0\right)g \tag{7.67}$$

where ρ_0 is the (constant) density of the flow, T_0 is the operating temperature, and β is the thermal expansion coefficient. Equation 7.67 (p. 1388) is obtained by using the Boussinesq approximation $\rho=\rho_0\left(1-\beta\Delta T\right)$ to eliminate ρ from the buoyancy term. This approximation is accurate as long as changes in actual density are small; specifically, the Boussinesq approximation is valid when $\beta\left(T-T_0\right)\ll 1$.

7.3.1.3. Limitations of the Boussinesq Model

The Boussinesq model should not be used if the temperature differences in the domain are large. In addition, it cannot be used with species calculations, combustion, or reacting flows.

7.3.1.4. Steps in Solving Buoyancy-Driven Flow Problems

The procedure for including buoyancy forces in the simulation of mixed or natural convection flows is described below.

1. To activate the calculation of heat transfer, enable the energy equation by right-clicking **Energy** in the tree (under **Setup/Models**) and clicking **On** in the menu that opens (Figure 16.1: Enabling the Energy Equation (p. 2164)).

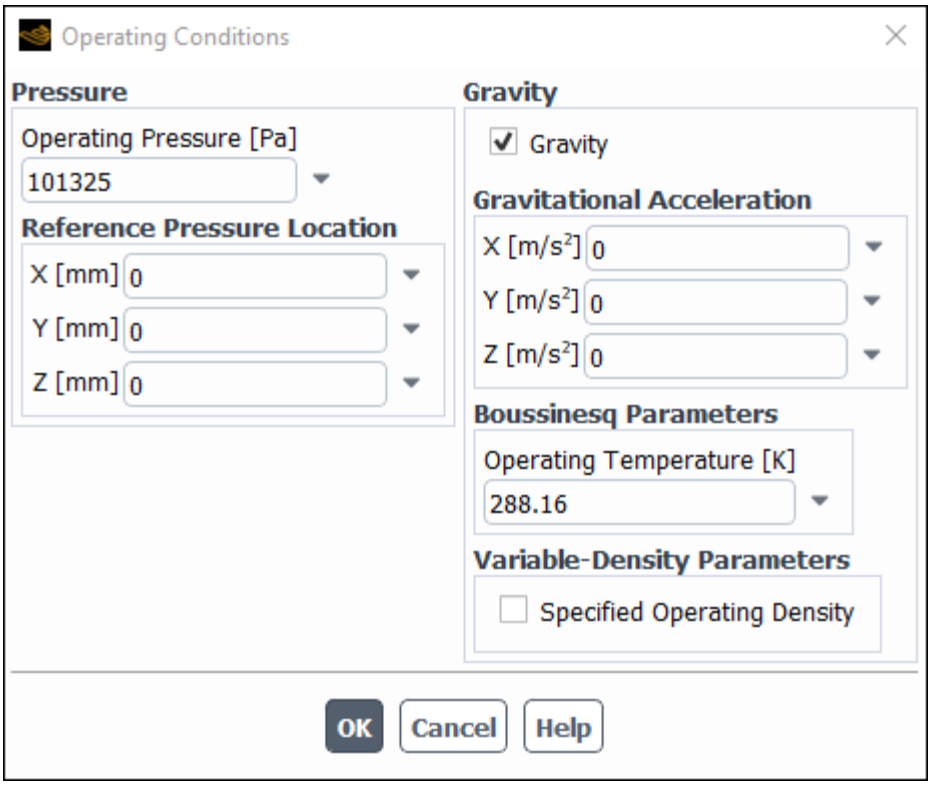
   Setup → **Models** → **Energy** → **On**

2. Define the operating conditions in the **Operating Conditions** dialog box (Figure 7.33: The Operating Conditions Dialog Box (p. 1389)).

   Setup → **Cell Zone Conditions** → **Operating Conditions...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.33: The Operating Conditions Dialog Box



a. Enable the **Gravity** option under **Gravity**.

b. Enter the appropriate values in the **X**, **Y**, and (for 3D) **Z** fields for **Gravitational Acceleration** for each Cartesian coordinate direction. (Note that the default gravitational acceleration in Ansys Fluent is zero.)

c. If you are using the incompressible ideal gas law, check that the **Operating Pressure** is set to an appropriate (nonzero) value.

d. Depending on whether or not you use the Boussinesq approximation, specify the appropriate parameters described below:

   - If you are not using the Boussinesq model, the inputs are as follows:

     i. If necessary, define the operating density:

        – For a single-phase flow, enable the **Specified Operating Density** option in the **Operating Conditions** dialog box, and enter a value for the **Operating Density**. See below for details.

> **Note:**
>
> For compressible (ideal and real) gas models with buoyancy, you should specify a value of zero for **Operating Density**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– For a multiphase flow, Ansys Fluent uses the **minimum-phase-averaged** operating density method by default. If you want to provide your own input, select the **user-input** method from the **Operating Density** drop-down list in the **Operating Conditions** dialog box (**Variable-Density Parameters** group box) and enter the value of the operating density. See Setting the Operating Density for a Buoyancy-Driven Multiphase Flow (p. 2930) for details.

ii. Define the fluid density as a function of temperature as described in Defining Properties Using Temperature-Dependent Functions (p. 1616) and Density (p. 1624).

Setup → **Materials**

- If you are using the Boussinesq model (described in The Boussinesq Model (p. 1388)) the inputs are as follows:

i. Enter the **Operating Temperature** (T_0 in Equation 7.67 (p. 1388)) in the **Operating Conditions** dialog box.

ii. Select **boussinesq** in the drop-down list for **Density** in the **Create/Edit Materials** dialog box as described in Defining Properties Using Temperature-Dependent Functions (p. 1616) and Density (p. 1624), and enter a constant value.

iii. Also in the **Create/Edit Materials** dialog box, enter an appropriate value for the **Thermal Expansion Coefficient** (β in Equation 7.67 (p. 1388)) for the fluid material.

Note that if your model involves multiple fluid materials you can choose whether or not to use the Boussinesq model for each material. As a result, you may have some materials using the Boussinesq model and others not. In such cases, you will need to set all the parameters described above in this step.

3. Define the boundary conditions. The boundary condition dialog boxes can be opened by right-clicking the boundary name in the tree (under **Setup/Boundary Conditions**) and clicking **Edit...** in the menu that opens; alternatively, you can open them from the **Boundary Conditions** task page:

Setup → **Boundary Conditions**

The boundary pressures that you specify at pressure inlet and outlet boundaries are the redefined pressures as given by Equation 7.68 (p. 1391). In general you should enter equal pressures, p', at the inlet and exit boundaries of your Ansys Fluent model if there are no externally imposed pressure gradients.

4. Set the parameters that control the solution, using the **Solution Methods** task page.

Solution → **Methods**

a. If you are using the pressure-based solver, selecting **PRESTO!** for **Pressure** under **Spatial Discretization** is recommended.

b. Add cells near the walls to resolve boundary layers, if necessary.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See also Solving Heat Transfer Problems (p. 2164) for information on setting up heat transfer calculations.

7.3.1.5. If you are using the incompressible ideal gas law, check the Operating Density

When the Boussinesq approximation is not used, the operating density ρ_0 appears in the body-force term in the momentum equations as $\left(\rho-\rho_0\right)g$. This form of the body-force term follows from the redefinition of pressure in Ansys Fluent as

$$p_s'=p_s-\rho_0 g x \tag{7.68}$$

The hydrostatic pressure in a fluid at rest is then

$$p_s'=0 \tag{7.69}$$

7.3.1.5.1. Setting the Operating Density for a Single Phase Flow

By default, Ansys Fluent will compute the operating density by averaging over all cells. In some cases, you may obtain better results if you explicitly specify the operating density instead of having the solver compute it for you. For example, if you are solving a natural-convection problem with a pressure boundary, it is important to understand that the pressure you are specifying is p_s' in Equation 7.68 (p. 1391). Although you will know the actual pressure p_s, you will need to know the operating density ρ_0 in order to determine p_s' from p_s. Therefore, you should explicitly specify the operating density rather than use the computed average. The specified value should, however, be representative of the average value.

In some cases the specification of an operating density will improve convergence behavior, rather than the actual results. For such cases use the approximate bulk density value as the operating density and be sure that the value you choose is appropriate for the characteristic temperature in the domain.

Note that if you are using the Boussinesq approximation for all fluid materials, the operating density ρ_0 does not appear in the body-force term of the momentum equation. Consequently, you need not specify it.

7.3.1.5.2. Setting the Operating Density for a Multiphase Flow

Details on setting the operation density for multiphase flows can be found in Modeling Buoyancy-Driven Multiphase Flow (p. 2930).

7.3.1.6. Solution Strategies for Buoyancy-Driven Flows

For high-Rayleigh-number flows you may want to consider the solution guidelines below. In addition, the guidelines presented in Solution Strategies for Heat Transfer Modeling (p. 2167) for solving other heat transfer problems can also be applied to buoyancy-driven flows. No steady-state solution exists for some laminar, high-Rayleigh-number flows.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.3.1.6.1. Guidelines for Solving High-Rayleigh-Number Flows

When you are solving a high-Rayleigh-number flow ($Ra > 10^8$), you should follow one of the procedures outlined below for best results.

The first procedure uses a steady-state approach:

1. Start the solution with a lower value of Rayleigh number (for example, 10^7) and run it to convergence using the first-order scheme.
2. To change the effective Rayleigh number, change the value of gravitational acceleration (for example, from 9.8 to 0.098 to reduce the Rayleigh number by two orders of magnitude).
3. Use the resulting data file as an initial guess for the higher Rayleigh number and start the higher-Rayleigh-number solution using the first-order scheme.
4. After you obtain a solution with the first-order scheme you may continue the calculation with a higher-order scheme.

The second procedure uses a time-dependent approach to obtain a steady-state solution [62] (p. 5716):

1. Start the solution from a steady-state solution obtained for the same or a lower Rayleigh number.
2. Estimate the time constant as [17] (p. 5714)

$$\tau = \frac{L}{U} \sim \frac{L^2}{\alpha}\left(PrRa\right)^{-1/2} = \frac{L}{\sqrt{g\beta\Delta TL}} \tag{7.70}$$

   where L and U are the length and velocity scales, respectively. Use a time step size Δt such that

$$\Delta t \approx \frac{\tau}{4} \tag{7.71}$$

   Using a larger time step size may lead to divergence.

3. After oscillations with a typical frequency of $f\tau =$ 0.05–0.09 have decayed, the solution reaches steady-state. Note that τ is the time constant estimated in Equation 7.70 (p. 1392) and f is the oscillation frequency in Hz. In general this solution process may take as many as 5000 time steps to reach steady-state.

7.4. Boundary Conditions

Boundary conditions consist of external, internal, and periodic boundaries. Most of the boundary conditions are discussed in the sections that follow.

7.4.1. Flow Inlet and Exit Boundary Conditions
7.4.2. Using Flow Boundary Conditions
7.4.3. Pressure Inlet Boundary Conditions
7.4.4. Velocity Inlet Boundary Conditions

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.5. Mass-Flow Inlet Boundary Conditions
7.4.6. Mass-Flow Outlet Boundary Conditions
7.4.7. Inlet Vent Boundary Conditions
7.4.8. Intake Fan Boundary Conditions
7.4.9. Pressure Outlet Boundary Conditions
7.4.10. Pressure Far-Field Boundary Conditions
7.4.11. Outflow Boundary Conditions
7.4.12. Outlet Vent Boundary Conditions
7.4.13. Exhaust Fan Boundary Conditions
7.4.14. Degassing Boundary Conditions
7.4.15. Wall Boundary Conditions
7.4.16. Perforated Wall Boundary Conditions
7.4.17. Symmetry Boundary Conditions
7.4.18. Periodic Boundary Conditions
7.4.19. Axis Boundary Conditions
7.4.20. Fan Boundary Conditions
7.4.21. Radiator Boundary Conditions
7.4.22. Porous Jump Boundary Conditions

You can find details about the following types here:

- interface zones: Using Sliding Meshes (p. 1804)
- interior zones: Interior Dialog Box (p. 5020)
- overset zones: Overset Meshes (p. 1206)
- RANS/LES interface zones: Setting Up the Embedded Large Eddy Simulation (ELES) Model (p. 2124)

Note:

By default, boundaries are grouped by type, but you also have the option of grouping them by:

- Alphabetical (List View)
- Name
- Cell Zone (Adjacency)

To change the grouping, right-click the **Boundary Conditions** branch in the Outline View tree and select **Group By**/**List View** or **Name** or **Adjacency**.

Setup → Boundary Conditions **Group By → List View | Name | Adjacency**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can set the default boundary organization method using the **Group boundary conditions by** drop-down in Preferences (**Appearance** branch).

File → **Preferences**

7.4.1. Flow Inlet and Exit Boundary Conditions

Ansys Fluent has a wide range of boundary conditions that permit flow to enter and exit the solution domain. To help you select the most appropriate boundary condition for your application, this section includes descriptions of how each type of condition is used, and what information is needed for each one. Recommendations for determining inlet values of the turbulence parameters are also provided.

7.4.2. Using Flow Boundary Conditions

This section provides an overview of flow boundaries in Ansys Fluent and how to use them.

Ansys Fluent provides 12 types of boundary zone types for the specification of flow inlets and exits: velocity inlet, pressure inlet, mass-flow inlet, mass-flow outlet, pressure outlet, pressure far-field, outflow, inlet vent, intake fan, outlet vent, exhaust fan, and degassing.

The inlet and exit boundary condition options in Ansys Fluent are as follows:

- Velocity inlet boundary conditions are used to define the velocity and scalar properties of the flow at inlet boundaries.
- Pressure inlet boundary conditions are used to define the total pressure and other scalar quantities at flow inlets.
- Mass-flow inlet boundary conditions are used in compressible flows to prescribe a mass flow rate at an inlet. It is not necessary to use mass-flow inlets in incompressible flows because when density is constant, velocity inlet boundary conditions will fix the mass flow. Like pressure and velocity inlets, other inlet scalars are also prescribed.
- Pressure outlet boundary conditions are used to define the static pressure at flow outlets (and also other scalar variables, in case of backflow). The use of a pressure outlet boundary condition instead of an outflow condition often results in a better rate of convergence when backflow occurs during iteration.
- Pressure far-field boundary conditions are used to model a free-stream compressible flow at infinity, with free-stream Mach number and static conditions specified. This boundary type is available only for compressible flows.
- Outflow boundary conditions are used to model flow exits where the details of the flow velocity and pressure are not known prior to solution of the flow problem. They are appropriate where the exit flow is close to a fully developed condition, as the outflow boundary condition assumes a zero streamwise gradient for all flow variables except pressure. They are not appropriate for compressible flow calculations.
- Inlet vent boundary conditions are used to model an inlet vent with a specified loss coefficient, flow direction, and ambient (inlet) total pressure and temperature.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Intake fan boundary conditions are used to model an external intake fan with a specified pressure jump, flow direction, and ambient (intake) total pressure and temperature.
- Outlet vent boundary conditions are used to model an outlet vent with a specified loss coefficient and ambient (discharge) static pressure and temperature.
- Exhaust fan boundary conditions are used to model an external exhaust fan with a specified pressure jump and ambient (discharge) static pressure.
- Degassing boundary conditions are used to model a free surface through which dispersed gas bubbles are allowed to escape, but the continuous liquid phase is not. A typical application is a bubble column in which you want to reduce computational cost by not including the freeboard region in the simulation. The degassing boundary condition is only available for two-phase liquid-gas flows using the Eulerian multiphase model.

7.4.2.1. Determining Turbulence Parameters

When the flow enters the domain at an inlet, outlet, or far-field boundary, Ansys Fluent requires specification of transported turbulence quantities. This section describes which quantities are needed for specific turbulence models and how they must be specified. It also provides guidelines for the most appropriate way of determining the inflow boundary values.

7.4.2.1.1. Specification of Turbulence Quantities Using Profiles

If it is important to accurately represent a boundary layer or fully-developed turbulent flow at the inlet, you should ideally set the turbulence quantities by creating a profile file (see Profiles (p. 1564)) from experimental data or empirical formulas. If you have an analytical description of the profile, rather than data points, you can either use this analytical description to create a profile file, or create a user-defined function to provide the inlet boundary information. (See the Fluent Customization Manual for information on user-defined functions.)

Once you have created the profile function, you can use it as described below:

- Spalart-Allmaras model: Choose **Modified Turbulent Viscosity** or **Turbulent Viscosity Ratio** in the **Turbulence Specification Method** drop-down list and select the appropriate profile name in the drop-down list next to **Turbulent Viscosity** or **Turbulent Viscosity Ratio**. Ansys Fluent computes the boundary value for the modified turbulent viscosity, $\tilde{\nu}$, by combining $\frac{\mu_t}{\mu}$ with the appropriate values of density and molecular viscosity.
- k - ε models: Choose **K and Epsilon** in the **Turbulence Specification Method** drop-down list and select the appropriate profile names in the drop-down lists next to **Turbulent Kinetic Energy** and **Turbulent Dissipation Rate**.
- k - ω models: Choose **K and Omega** in the **Turbulence Specification Method** drop-down list and select the appropriate profile names in the drop-down lists next to **Turbulent Kinetic Energy** and **Specific Dissipation Rate**.
- ε -based Reynolds stress models: Choose **K and Epsilon** in the **Turbulence Specification Method** drop-down list and select the appropriate profile names in the drop-down lists next to **Turbulent Kinetic Energy** and **Turbulent Dissipation Rate**. Choose **Reynolds-Stress Components** in the **Reynolds-Stress Specification Method** drop-down list and select the

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

appropriate profile name in the drop-down list next to each of the individual Reynolds-stress components.

- ω -based Reynolds stress models: Choose **K and Omega** in the **Turbulence Specification Method** drop-down list and select the appropriate profile names in the drop-down lists next to **Specific Dissipation Rate**. Choose **Reynolds-Stress Components** in the **Reynolds-Stress Specification Method** drop-down list and select the appropriate profile name in the drop-down list next to each of the individual Reynolds-stress components.

7.4.2.1.2. Uniform Specification of Turbulence Quantities

In some situations, it is appropriate to specify a uniform value of the turbulence quantity at the boundary where inflow occurs. Examples are fluid entering a duct, far-field boundaries, or even fully-developed duct flows where accurate profiles of turbulence quantities are unknown.

In most turbulent flows, higher levels of turbulence are generated within shear layers than enter the domain at flow boundaries, making the result of the calculation relatively insensitive to the inflow boundary values. Nevertheless, caution must be used to ensure that boundary values are not so unphysical as to contaminate your solution or impede convergence. This is particularly true of external flows where unphysically large values of effective viscosity in the free stream can "swamp" the boundary layers.

You can use the turbulence specification methods described above to enter uniform constant values instead of profiles. Alternatively, you can specify the turbulence quantities in terms of more convenient quantities such as turbulence intensity, turbulent viscosity ratio, hydraulic diameter, and turbulence length scale. The default Turbulence Specification Method is set to **Turbulent Viscosity Ratio** (for the Spalart-Allmaras model) or **Intensity and Viscosity Ratio** (for the k - ε models, the k - ω models, or the RSM). These quantities are discussed further in the following sections.

7.4.2.1.3. Turbulence Intensity

The turbulence intensity, I, is defined as the ratio of the root-mean-square of the velocity fluctuations, u', to the mean flow velocity, u_{avg}.

A turbulence intensity of 1% or less is generally considered low and turbulence intensities greater than 10% are considered high. Ideally, you will have a good estimate of the turbulence intensity at the inlet boundary from external, measured data. For example, if you are simulating a wind-tunnel experiment, the turbulence intensity in the free stream is usually available from the tunnel characteristics. In modern low-turbulence wind tunnels, the free-stream turbulence intensity may be as low as 0.05%.

For internal flows, the turbulence intensity at the inlets is totally dependent on the upstream history of the flow. If the flow upstream is under-developed and undisturbed, you can use a low turbulence intensity. If the flow is fully developed, the turbulence intensity may be as high as a few percent. The turbulence intensity at the core of a fully-developed duct flow can be derived from an empirical correlation by Blasius for pipe flows (see Chapter 20 Equations (20-4) and (20-5) in [137] (p. 5721)), using the additional estimation of the RMS velocity fluctuation at the pipe core $u'=0.8*u_\tau$:

$$I \equiv \frac{u'}{u_{avg}} = 0.16\left(Re_{D_H}\right)^{-1/8} \tag{7.72}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

At a Reynolds number of 50,000, for example, the turbulence intensity will be 4%, according to this formula.

The default value for turbulence intensity is 5% (medium intensity).

7.4.2.1.4. Turbulence Length Scale and Hydraulic Diameter

The turbulence length scale, ℓ, is a physical quantity related to the size of the large eddies that contain the energy in turbulent flows.

In fully-developed duct flows, ℓ is restricted by the size of the duct, since the turbulent eddies cannot be larger than the duct. An approximate relationship between ℓ and the physical size of the duct is

$$\ell=\frac{0.07L}{C_\mu^{3/4}} \tag{7.73}$$

where L is the relevant dimension of the duct and $1/C_\mu^{3/4}$ ensures consistency with the definition of the turbulent length scales for one- and two-equation turbulence models. The factor of 0.07 is based on the maximum value of the mixing length in fully-developed turbulent pipe flow, where L is the diameter of the pipe. In a channel of non-circular cross-section, you can base L on the hydraulic diameter.

If the turbulence derives its characteristic length from an obstacle in the flow, such as a perforated plate, it is more appropriate to base the turbulence length scale on the characteristic length of the obstacle rather than on the duct size.

It should be noted that the relationship of Equation 7.73 (p. 1397), which relates a physical dimension (L) to the turbulence length scale (ℓ), is not necessarily applicable to all situations. For most cases, however, it is a suitable approximation.

Guidelines for choosing the characteristic length L or the turbulence length scale ℓ for selected flow types are listed below:

- For fully-developed internal flows, choose the **Intensity and Hydraulic Diameter** specification method and specify the hydraulic diameter $L=D_H$ in the **Hydraulic Diameter** field.
- For flows downstream of turning vanes, perforated plates, and so on, choose the **Intensity and Length Scale** method and specify the characteristic length of the flow opening for L in the **Turbulent Length Scale** field.
- For wall-bounded flows in which the inlets involve a turbulent boundary layer, choose the **Intensity and Length Scale** method and use the boundary-layer thickness, δ_{99}, to compute the turbulence length scale, ℓ, from $\ell=0.4\delta_{99}$. Enter this value for ℓ in the **Turbulence Length Scale** field.

7.4.2.1.5. Turbulent Viscosity Ratio

The turbulent viscosity ratio, $\frac{\mu_t}{\mu}$, is directly proportional to the turbulent Reynolds number ($Re_t \equiv k^2/(\varepsilon\nu)$). Re_t is large (on the order of 100 to 1000) in high-Reynolds-number boundary layers, shear layers, and fully-developed duct flows. However, at the free-stream boundaries of

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

most external flows, $\frac{\mu_t}{\mu}$ is fairly small. Typically, the turbulence parameters are set so that $1<\frac{\mu_t}{\mu}<10$. For internal flows values up to 100 are sensible for the turbulent viscosity ratio, $\frac{\mu_t}{\mu}$. The default value for the turbulent viscosity ratio is set to 10.

To specify quantities in terms of the turbulent viscosity ratio, you can choose **Turbulent Viscosity Ratio** (for the Spalart-Allmaras model) or **Intensity and Viscosity Ratio** (for the k - ε models, the k - ω models, or the RSM). The default value for the turbulence intensity is set to 5% (medium intensity) and the turbulent viscosity ratio, $\frac{\mu_t}{\mu}$, has a default value of 10.

7.4.2.1.6. Relationships for Deriving Turbulence Quantities

To obtain the values of transported turbulence quantities from more convenient quantities such as I, L, or $\frac{\mu_t}{\mu}$, you must typically resort to an empirical relations in addition to the formal definitions of these quantities. Several formal definitions and useful empirical relations, most of which are used within Ansys Fluent, are presented below.

7.4.2.1.7. Estimating Modified Turbulent Viscosity from Turbulence Intensity and Length Scale

To obtain the modified turbulent viscosity, $\tilde{\nu}$, for the Spalart-Allmaras model from the turbulence intensity, I, and length scale, ℓ, the following equation can be used. Note that this equation assumes that there is no viscous damping, that is, $\tilde{\nu}=\nu_t$ (see Modeling the Turbulent Viscosity in the Theory Guide).

$$\tilde{\nu}=C_\mu\sqrt{\frac{3}{2}}\,u_{avg}I\ell \tag{7.74}$$

This formula is also used in Ansys Fluent if you select the **Intensity and Hydraulic Diameter** specification method with the Spalart-Allmaras model. In this case, ℓ is obtained from Equation 7.73 (p. 1397). C_μ ensures consistency with the definition of the eddy viscosity for k - ε and k - ω turbulence models. The value of C_μ is 0.09.

When using a moving reference frame, be aware of the order of the conversion operations applied to the inlet velocity and to the turbulence quantities, which is explained in the next section. It concerns the values of u_{avg} in Equation 7.74 (p. 1398).

7.4.2.1.8. Estimating Turbulent Kinetic Energy from Turbulence Intensity

The relationship between the turbulent kinetic energy, k, and turbulence intensity, I, is

$$k=\frac{3}{2}\left(u_{avg}I\right)^2 \tag{7.75}$$

where u_{avg} is the magnitude of the mean flow velocity.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This relationship is used in Ansys Fluent whenever the **Intensity and Hydraulic Diameter**, **Intensity and Length Scale**, or **Intensity and Viscosity Ratio** method is used instead of specifying explicit values for k and ε or ω.

Important:

The reference frame for the mean flow velocity in Equation 7.75 (p. 1398) corresponds to the Velocity Formulation selected in the General Task Page (p. 4684), see also Choosing the Relative or Absolute Velocity Formulation (p. 1775). When selecting the Reference Frame in any of the inlet boundary condition dialog boxes (see, for example, Figure 7.37: The Velocity Inlet Dialog Box (p. 1412) for the Velocity Inlet Boundary Conditions (p. 1411)), be aware that the specified inlet velocity is first converted to the selected Velocity Formulation and then applied to compute the turbulent kinetic energy from turbulence intensity using Equation 7.75 (p. 1398). If an inlet boundary type does not require the inlet velocity specification (for example, Pressure Inlet Boundary Conditions (p. 1401)), then u_{avg} in Equation 7.75 (p. 1398) is the current local solution velocity magnitude, which corresponds to the selected Velocity Formulation.

7.4.2.1.9. Estimating Turbulent Dissipation Rate from a Length Scale

If you know the turbulence length scale, ℓ, you can determine ε from the relationship

$$\varepsilon = \frac{k^{3/2}}{\ell} \tag{7.76}$$

The determination of ℓ was discussed previously.

This relationship is used in Ansys Fluent whenever the **Intensity and Hydraulic Diameter** or **Intensity and Length Scale** method is used instead of specifying explicit values for k and ε.

7.4.2.1.10. Estimating Turbulent Dissipation Rate from Turbulent Viscosity Ratio

The value of ε can be obtained from the turbulent viscosity ratio $\frac{\mu_t}{\mu}$ and k using the following relationship:

$$\varepsilon = \rho C_\mu \frac{k^2}{\mu} \left(\frac{\mu_t}{\mu} \right)^{-1} \tag{7.77}$$

where C_μ is an empirical constant specified in the turbulence model.

This relationship is used in Ansys Fluent whenever the **Intensity and Viscosity Ratio** method is used instead of specifying explicit values for k and ε.

7.4.2.1.11. Estimating Turbulent Dissipation Rate for Decaying Turbulence

If you are simulating a wind-tunnel situation in which the model is mounted in the test section downstream of a mesh and/or wire mesh screens, you can choose a value of ε such that

$$\varepsilon \approx \frac{\Delta k U_\infty}{L_\infty} \tag{7.78}$$

where Δk is the approximate decay of k you want to have across the flow domain (say, 10% of the inlet value of k), U_∞ is the free-stream velocity, and L_∞ is the streamwise length of the flow

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

domain. Equation 7.78 (p. 1399) is a linear approximation to the power-law decay observed in high-Reynolds-number isotropic turbulence. Its basis is the exact equation for k in decaying turbulence, $U\partial k/\partial x=-\varepsilon$.

If you use this method to estimate ε, you should also check the resulting turbulent viscosity ratio $\frac{\mu_t}{\mu}$ to make sure that it is not too large, using Equation 7.77 (p. 1399).

Although this method is not used internally by Ansys Fluent, you can use it to derive a constant free-stream value of ε that you can then specify directly by choosing **K and Epsilon** in the **Turbulence Specification Method** drop-down list. In this situation, you will typically determine k from I using Equation 7.75 (p. 1398).

7.4.2.1.12. Estimating Specific Dissipation Rate from a Length Scale

If you know the turbulence length scale, ℓ, you can determine ω from the relationship

$$\omega=\frac{k^{1/2}}{C_\mu \ell} \tag{7.79}$$

where C_μ is an empirical constant specified in the turbulence model. The determination of ℓ was discussed previously.

This relationship is used in Ansys Fluent whenever the **Intensity and Hydraulic Diameter** or **Intensity and Length Scale** method is used instead of specifying explicit values for k and ω.

7.4.2.1.13. Estimating Specific Dissipation Rate from Turbulent Viscosity Ratio

The value of ω can be obtained from the turbulent viscosity ratio $\frac{\mu_t}{\mu}$ and k using the following relationship:

$$\omega=\rho\frac{k}{\mu}\left(\frac{\mu_t}{\mu}\right)^{-1} \tag{7.80}$$

This relationship is used in Ansys Fluent whenever the **Intensity and Viscosity Ratio** method is used instead of specifying explicit values for k and ω.

7.4.2.1.14. Estimating Reynolds Stress Components from Turbulent Kinetic Energy

When the RSM is used, if you do not specify the values of the Reynolds stresses explicitly at the inlet using the **Reynolds-Stress Components** option in the **Reynolds-Stress Specification Method** drop-down list, they are approximately determined from the specified values of k. The turbulence is assumed to be isotropic such that

$$\overline{u_i' u_j'}=0 \tag{7.81}$$

and

$$\overline{u_\alpha' u_\alpha'}=\frac{2}{3}k \tag{7.82}$$

(no summation over the index α).

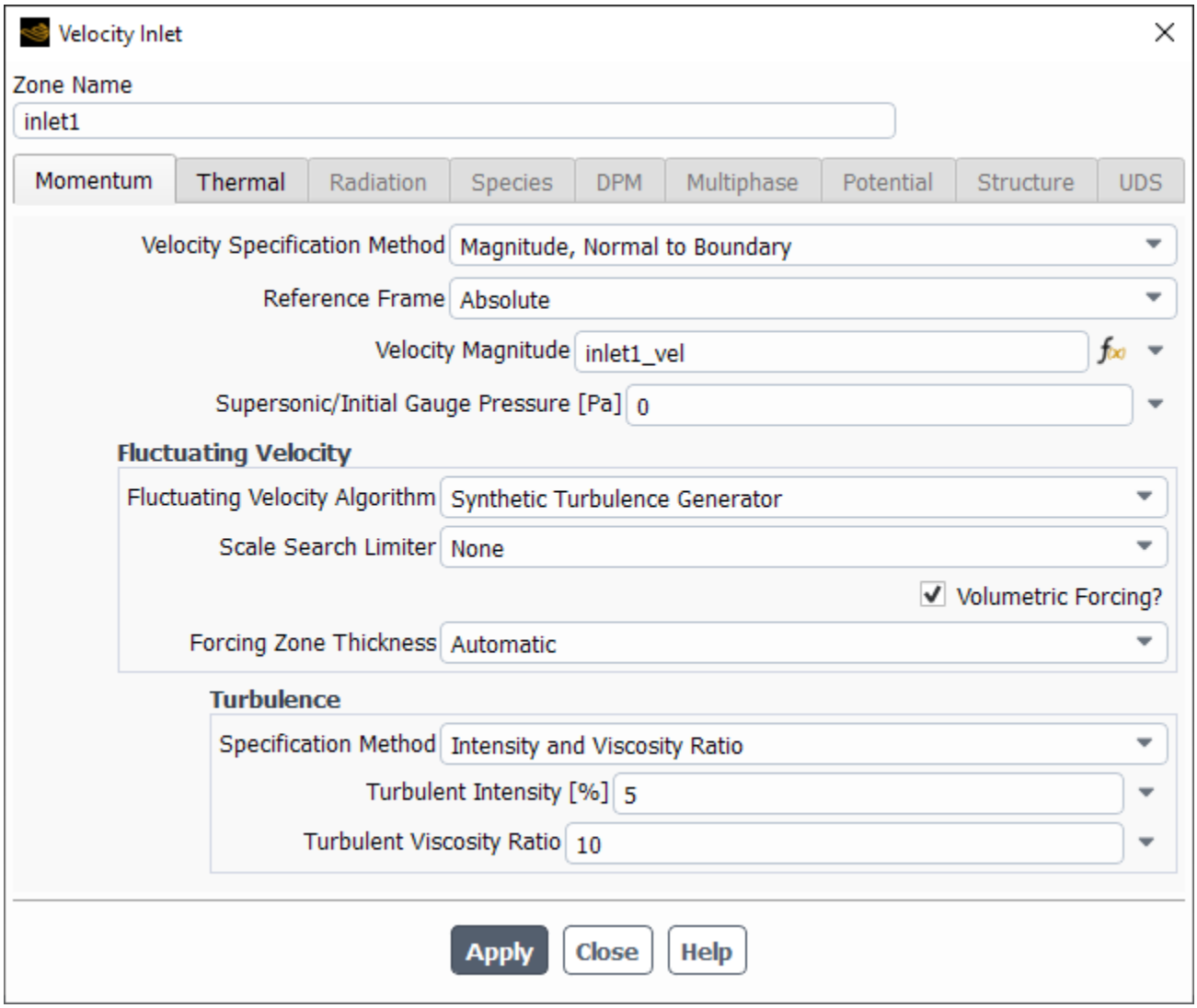
Ansys Fluent will use this method if you select **K or Turbulence Intensity** in the **Reynolds-Stress Specification Method** drop-down list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.2.1.15. Specifying Inlet Turbulence for Scale Resolving Simulations

When using Scale Resolving Simulations (SAS, DES, SDES, SBES, and LES models), it is possible to specify unsteady **Fluctuating Velocity** at velocity and pressure inlets to generate realistic turbulent content. Therefore, additional input is required for velocity and pressure inlets such as the choice of **Fluctuating Velocity Algorithm** and corresponding parameters (as shown in Figure 7.34: Specifying Inlet Turbulence for Scale Resolving Simulation (p. 1401)). The available methods along with their options are described in Inlet Boundary Conditions for Scale Resolving Simulations in the *Fluent Theory Guide*. Note that even though some LES models do not require specification of the **Turbulence** parameters at the boundary, it is required when a **Fluctuating Velocity Algorithm** is selected. Additional parameters available for the **Fluctuating Velocity Algorithm** are listed in Pressure Inlet Dialog Box (p. 5051) and Velocity Inlet Dialog Box (p. 5072).

Figure 7.34: Specifying Inlet Turbulence for Scale Resolving Simulation



7.4.3. Pressure Inlet Boundary Conditions

Pressure inlet boundary conditions are used to define the fluid pressure at flow inlets, along with all other scalar properties of the flow. They are suitable for both incompressible and compressible flow

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

calculations. Pressure inlet boundary conditions can be used when the inlet pressure is known but the flow rate and/or velocity is not known. This situation may arise in many practical situations, including buoyancy-driven flows. Pressure inlet boundary conditions can also be used to define a "free" boundary in an external or unconfined flow.

For an overview of flow boundaries, see Flow Inlet and Exit Boundary Conditions (p. 1394).

7.4.3.1. Inputs at Pressure Inlet Boundaries

7.4.3.1.1. Summary

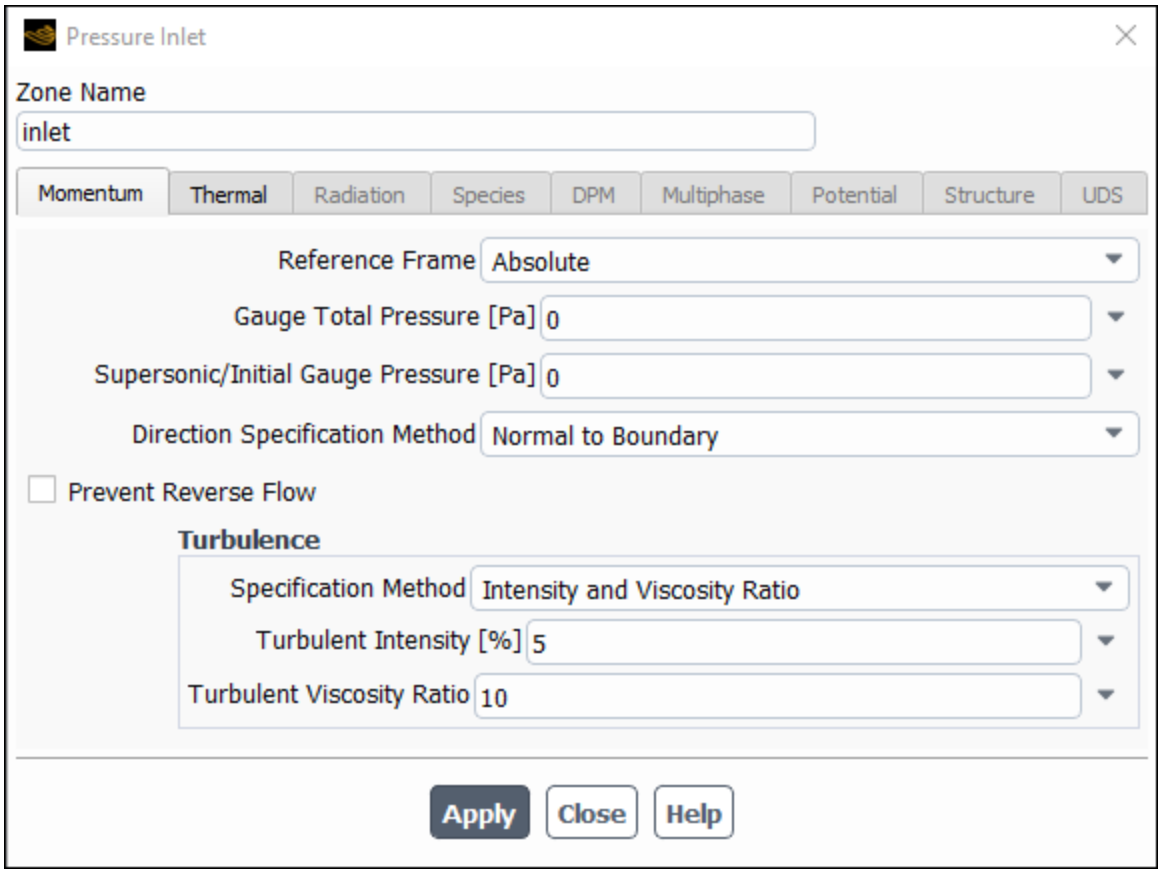
You will enter the following information for a pressure inlet boundary:

- type of reference frame
- total (stagnation) pressure
- total (stagnation) temperature
- flow direction
- static pressure
- turbulence parameters (for turbulent calculations)
- radiation parameters (for calculations using the P-1, DTRM, DO, surface-to-surface, or MC models)
- chemical species mass or mole fractions (for species calculations)
- mixture fraction and variance (for non-premixed or partially premixed combustion calculations)
- progress variable (for premixed or partially premixed combustion calculations)
- discrete phase boundary conditions (for discrete phase calculations)
- multiphase boundary conditions (for general multiphase calculations)
- open channel flow parameters (for open channel flow calculations using the VOF multiphase model)
- acoustic wave model settings

All values are entered in the Pressure Inlet Dialog Box (p. 5051) (Figure 7.35: The Pressure Inlet Dialog Box (p. 1403)), which is opened from the **Boundary Conditions** task page (as described in Setting Cell Zone and Boundary Conditions (p. 1303)). Note that open channel boundary condition inputs are described in Modeling Open Channel Flows (p. 2979), and acoustic wave model settings are described in Boundary Acoustic Wave Models (p. 1527).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.35: The Pressure Inlet Dialog Box



7.4.3.1.1.1. Pressure Inputs and Hydrostatic Head

When gravitational acceleration is activated in the **Operating Conditions** dialog box (accessed from the **Boundary Conditions** task page), the pressure field (including all pressure inputs) will include the hydrostatic head. This is accomplished by redefining the pressure in terms of a modified pressure that includes the hydrostatic head (denoted p') as follows:

$$p' = p - \rho_0 \vec{g} \cdot \vec{r} \tag{7.83}$$

where ρ_0 is a constant operating density, $\vec{g}$ is the gravity vector (also a constant), and

$$\vec{r} = x\hat{i} + y\hat{j} + z\hat{k} \tag{7.84}$$

is the position vector. Noting that

$$\nabla\left(\rho_0 \vec{g} \cdot \vec{r}\right) = \rho_0 \vec{g} \tag{7.85}$$

it follows that

$$\nabla p' = \nabla\left(p - \rho_0 \vec{g} \cdot \vec{r}\right) = \nabla p - \rho_0 \vec{g} \tag{7.86}$$

The substitution of this relation in the momentum equation gives pressure gradient and gravitational body force terms of the form

$$-\nabla p' + \left(\rho - \rho_0\right)\vec{g} \tag{7.87}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where ρ is the fluid density. Therefore, if the fluid density is constant, we can set the operating density ρ_0 equal to the fluid density, thereby eliminating the body force term. If the fluid density is not constant (for example, density is given by the ideal gas law), then the operating density should be chosen to be representative of the average or mean density in the fluid domain, so that the body force term is small.

An important consequence of this treatment of the gravitational body force is that your inputs of pressure (now defined as p') should not include hydrostatic pressure differences. Moreover, reports of static and total pressure will not show any influence of the hydrostatic pressure. See Buoyancy-Driven Flows and Natural Convection (p. 1387) for additional information.

7.4.3.1.1.2. Defining Total Pressure and Temperature

Enter the value for total pressure in the **Gauge Total Pressure** field in the **Pressure Inlet** dialog box. Total temperature is set in the **Thermal** tab, in the **Total Temperature** field.

Remember that the total pressure value is the gauge pressure with respect to the operating pressure defined in the Operating Conditions Dialog Box (p. 4978). Total pressure for an incompressible fluid is defined as

$$p_0 = p_s + \frac{1}{2}\rho|\vec{v}|^2 \tag{7.88}$$

and for a compressible fluid of constant c_p as

$$p_0 = p_s\left(1 + \frac{\gamma-1}{2}M^2\right)^{\gamma/(\gamma-1)} \tag{7.89}$$

where
p_0 = total pressure
p_s = static pressure
M = Mach number
γ = ratio of specific heats $\left(c_p / c_v\right)$

If you are modeling axisymmetric swirl, $\vec{v}$ in Equation 7.88 (p. 1404) will include the swirl component.

The **Total Temperature**, **Gauge Total Pressure**, and flow directions are in absolute or relative to the adjacent cell zone reference frames, based on the **Reference Frame** setting in the **Pressure Inlet** dialog box.

If the cell zone adjacent to a pressure inlet is defined as a moving reference frame zone, and you are using the pressure-based solver, the velocity in Equation 7.88 (p. 1404) (or the Mach number in Equation 7.89 (p. 1404)) will be absolute or relative to the mesh velocity, depending on whether or not the **Absolute** velocity formulation is enabled in the **General** task page. For the density-based solver, the **Absolute** velocity formulation is always used; hence, the velocity in Equation 7.88 (p. 1404) (or the Mach number in Equation 7.89 (p. 1404)) is always the **Absolute** velocity.

For the Eulerian multiphase model, the total temperature, and velocity components must be specified for the individual phases. The **Reference Frame** (**Relative to Adjacent Cell Zone** or

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Absolute) for each of the phases is the same as the reference frame selected for the mixture phase. Note that the total pressure values must be specified in the mixture phase.

Important:

- If the flow is incompressible, then the temperature assigned in the **Pressure Inlet** dialog box will be considered the static temperature.
- For the mixture multiphase model, if a boundary allows a combination of compressible and incompressible phases to enter the domain, then the temperature assigned in the **Pressure Inlet** dialog box will be considered the static temperature at that boundary. If a boundary allows *only a compressible* phase to enter the domain, then the temperature assigned in the **Pressure Inlet** dialog box will be taken as the total temperature (relative/absolute) at that boundary. The total temperature will depend on the **Reference Frame** option selected in the **Pressure Inlet** dialog box.
- For the VOF multiphase model, if a boundary allows a *compressible phase* to enter the domain, then the temperature assigned in the **Pressure Inlet** dialog box will be considered the total temperature at that boundary. The total temperature (relative/absolute) will depend on the **Reference Frame** option chosen in the dialog box. Otherwise, the temperature assigned to the boundary will be considered the static temperature at the boundary.
- For the Eulerian multiphase model, if a boundary allows a mixture of compressible and incompressible phases in the domain, then the temperature of each of the phases will be the total or static temperature, depending on whether the phase is compressible or incompressible.
- Total temperature (relative/absolute) will depend on the **Reference Frame** option chosen in the **Pressure Inlet** dialog box.

7.4.3.1.1.3. Defining the Flow Direction

The flow direction is defined as a unit vector ($\vec{d}$) which is aligned with the local velocity vector, $\vec{v}$. This can be expressed simply as

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\vec{d}=\frac{\vec{v}}{|\vec{v}|} \tag{7.90}$$

Important:

For the inputs in Ansys Fluent, the flow direction $\vec{d}$ need not be a unit vector, as it will be automatically normalized before it is applied.

Important:

For a moving reference frame, the relative flow direction $\vec{d}_r$ is defined in terms of the relative velocity, $\vec{v}_r$. Thus,

$$\vec{d}_r=\frac{\vec{v}_r}{|\vec{v}_r|} \tag{7.91}$$

You can define the flow direction at a pressure inlet explicitly, or you can define the flow to be normal to the boundary. If you choose to specify the direction vector, you can set either the (Cartesian) x, y, and z components, or the (cylindrical) radial, tangential, and axial components.

For moving zone problems calculated using the pressure-based solver, the flow direction will be absolute or relative to the mesh velocity, depending on whether or not the **Absolute** velocity formulation is selected in the **General** task page. For the density-based solver, the flow direction will always be in the absolute frame.

The procedure for defining the flow direction is as follows (refer to Figure 7.35: The Pressure Inlet Dialog Box (p. 1403)):

1. Specify the flow direction by selecting **Direction Vector** or **Normal to Boundary** in the **Direction Specification Method** drop-down list.

2. If you selected **Normal to Boundary** in step 1 and you are modeling axisymmetric swirl, enter the appropriate value for the **Tangential-Component of Flow Direction**. If you chose **Normal to Boundary** and your geometry is 3D or 2D without axisymmetric swirl, there are no additional inputs for flow direction.

3. If you selected **Direction Vector** in step 1, and your geometry is 3D, choose **Cartesian (X, Y, Z)**, **Cylindrical(Radial, Tangential, Axial)**, **Local Cylindrical (Radial, Tangential, Axial)**, or **Local Cylindrical Swirl** from the **Coordinate System** drop-down list. Some notes on these selections are provided below:

   - The **Cartesian** coordinate option is based on the Cartesian coordinate system used by the geometry. Enter appropriate values for the **X**, **Y**, and **Z-Component of Flow Direction**.

   - The **Cylindrical** coordinate system uses the axial, radial, and tangential components based on the following coordinate systems:

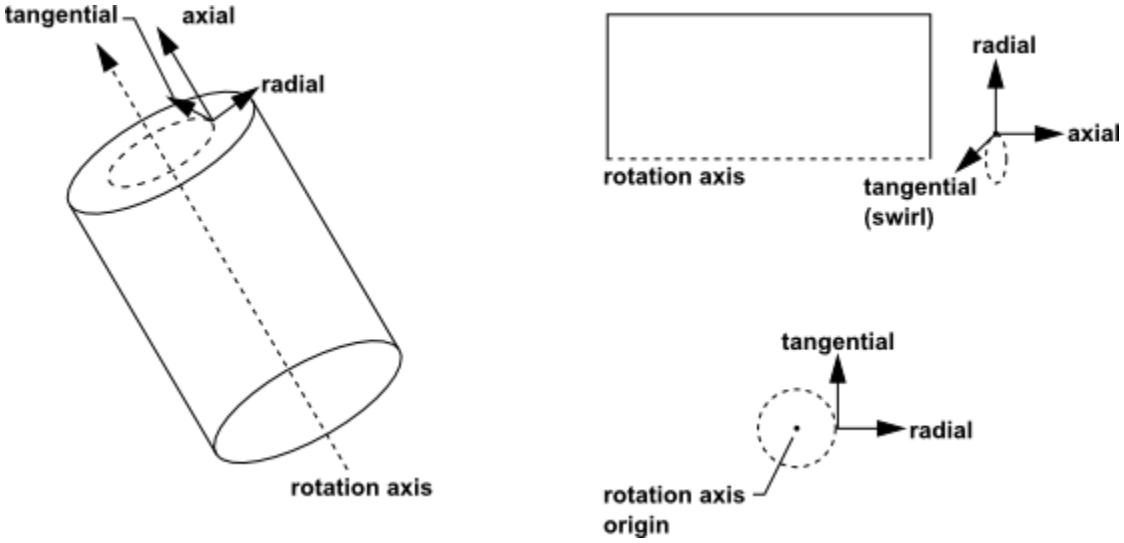
     – For problems involving a single cell zone, the coordinate system is defined by the rotation axis and origin specified in the Fluid Dialog Box (p. 4961).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– For problems involving multiple zones (for example, multiple reference frames or sliding meshes), the coordinate system is defined by the rotation axis specified in the **Fluid** (or **Solid**) dialog box for the fluid (or solid) zone that is adjacent to the inlet.

For all of the above definitions of the cylindrical coordinate system, positive radial velocities point radially outward from the rotation axis, positive axial velocities are in the direction of the rotation axis vector, and positive tangential velocities are based on the right-hand rule using the positive rotation axis (see Figure 7.36: Cylindrical Velocity Components in 3D, 2D, and Axisymmetric Domains (p. 1407)).

Figure 7.36: Cylindrical Velocity Components in 3D, 2D, and Axisymmetric Domains



- The **Local Cylindrical** coordinate system allows you to define a coordinate system specifically for the inlet. When you use the local cylindrical option, you will define the coordinate system right here in the **Pressure Inlet** dialog box. The local cylindrical coordinate system is useful if you have several inlets with different rotation axes. Enter appropriate values for the **Axial**, **Radial**, and **Tangential-Component of Flow Direction**, and then specify the **X**, **Y**, and **Z** components of the Axis Origin and Axis Direction.

- The **Local Cylindrical Swirl** coordinate system option allows you to define a coordinate system specifically for the inlet where the total pressure, swirl velocity, and the components of the velocity in the axial and radial planes are specified. Enter appropriate values for the **Axial** and **Radial-Component of Flow Direction**, and the **Tangential-Velocity**. Specify the **X**, **Y**, and **Z** components of the Axis Origin and Axis Direction. It is recommended that you start your simulation with a smaller swirl velocity and then progressively increase the velocity to obtain a stable solution.

Important:

Local Cylindrical Swirl should not be used for open channel boundary conditions and on the mixing plane boundaries while using the mixing plane model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. If you selected **Direction Vector** in step 1, and your geometry is 2D, define the vector components as follows:

   - For a 2D planar geometry, enter appropriate values for the **X**, **Y**, and **Z-Component of Flow Direction**.
   - For a 2D axisymmetric geometry, enter appropriate values for the **Axial**, **Radial-Component of Flow Direction**.
   - For a 2D axisymmetric swirl geometry, enter appropriate values for the **Axial**, **Radial**, and **Tangential-Component of Flow Direction**.

   Figure 7.36: Cylindrical Velocity Components in 3D, 2D, and Axisymmetric Domains (p. 1407) shows the vector components for these different coordinate systems.

7.4.3.1.1.4. Defining Static Pressure

The static pressure (termed the **Supersonic/Initial Gauge Pressure**) must be specified if the inlet flow is supersonic *or* if you plan to initialize the solution based on the pressure inlet boundary conditions. Solution initialization is discussed in Initializing the Solution (p. 3627).

Remember that the static pressure value you enter is relative to the operating pressure set in the Operating Conditions Dialog Box (p. 4978). Note the comments in Pressure Inputs and Hydrostatic Head (p. 1403) regarding hydrostatic pressure.

The **Supersonic/Initial Gauge Pressure** is ignored by Ansys Fluent whenever the flow is sub-sonic, in which case it is calculated from the specified stagnation quantities. If you choose to initialize the solution based on the pressure-inlet conditions, the **Supersonic/Initial Gauge Pressure** will be used in conjunction with the specified stagnation pressure to compute initial values according to the isentropic relations (for compressible flow) or Bernoulli's equation (for incompressible flow). Therefore, for a sub-sonic inlet it should generally be set based on a reasonable estimate of the inlet Mach number (for compressible flow) or inlet velocity (for incompressible flow).

7.4.3.1.1.5. Prevent Reverse Flow

When this option is selected, Fluent will erect artificial walls on the boundary mesh faces to prevent flow out of the domain. The artificial walls are removed when the flow is no longer leaving the domain and when a favorable pressure gradient is recovered at the boundary mesh faces.

When artificial walls are created, Fluent will print a message in the console window indicating the number of faces on which artificial walls have been erected and the equivalent area percentage of the boundary it represents.

Note:

This feature is not available with multiphase flow.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.3.1.1.6. Defining Turbulence Parameters

For turbulent calculations, there are several ways in which you can define the turbulence parameters. Instructions for deciding which method to use and determining appropriate values for these inputs are provided in Determining Turbulence Parameters (p. 1395). Turbulence modeling in general is described in Modeling Turbulence (p. 2061).

7.4.3.1.1.7. Defining Radiation Parameters

The options available depend on which radiation model is active. For details, see Defining Boundary Conditions for Radiation (p. 2217).

7.4.3.1.1.8. Defining Species Mass or Mole Fractions

If you are modeling species transport, you will set the species mass or mole fractions under **Species Mole Fractions** or **Species Mass Fractions**. For details, see Defining Cell Zone and Boundary Conditions for Species (p. 2394).

7.4.3.1.1.9. Defining Non-Premixed Combustion Parameters

If you are using the non-premixed or partially premixed combustion model, you will set the **Mean Mixture Fraction** and **Mixture Fraction Variance** (and the **Secondary Mean Mixture Fraction** and **Secondary Mixture Fraction Variance**, if you are using two mixture fractions), as described in Defining Non-Premixed Boundary Conditions (p. 2526).

7.4.3.1.1.10. Defining Premixed Combustion Boundary Conditions

If you are using the premixed or partially premixed combustion model, you will set the **Progress Variable**, as described in Setting Boundary Conditions for the Progress Variable (p. 2541).

7.4.3.1.1.11. Defining Discrete Phase Boundary Conditions

If you are modeling a discrete phase of particles, you can set the fate of particle trajectories at the pressure inlet. See Setting Boundary Conditions for the Discrete Phase (p. 2779) for details.

7.4.3.1.1.12. Defining Multiphase Boundary Conditions

If you are using the VOF, mixture, or Eulerian model for multiphase flow, you will need to specify volume fractions for secondary phases and (for some models) additional parameters. See Defining Multiphase Cell Zone and Boundary Conditions (p. 2957) for details.

7.4.3.1.1.13. Defining Open Channel Boundary Conditions

If you are using the VOF model for multiphase flow and modeling open channel flows, you will need to specify the **Free Surface Level**, **Bottom Level**, and additional parameters. See Modeling Open Channel Flows (p. 2979) for details.

7.4.3.2. Default Settings at Pressure Inlet Boundaries

Default settings (in SI) for pressure inlet boundary conditions are as follows:

Gauge Total Pressure	0

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Supersonic/Initial Gauge Pressure	0
Total Temperature	300
Direction Specification Method	Normal to Boundary
Turbulent Intensity	5%
Turbulent Viscosity Ratio	10

7.4.3.3. Calculation Procedure at Pressure Inlet Boundaries

The treatment of pressure inlet boundary conditions by Ansys Fluent can be described as a loss-free transition from stagnation conditions to the inlet conditions. For incompressible flows, this is accomplished by application of the Bernoulli equation at the inlet boundary. In compressible flows, the equivalent isentropic flow relations for an ideal gas are used.

7.4.3.3.1. Incompressible Flow Calculations at Pressure Inlet Boundaries

When flow enters through a pressure inlet boundary, Ansys Fluent uses the boundary condition pressure you input as the total pressure of the fluid at the inlet plane, p_0. In incompressible flow, the inlet total pressure and the static pressure, p_s, are related to the inlet velocity via Bernoulli's equation:

$$p_0 = p_s + \frac{1}{2}\rho v^2 \tag{7.92}$$

With the resulting velocity magnitude and the flow direction vector you assigned at the inlet, the velocity components can be computed. The inlet mass flow rate and fluxes of momentum, energy, and species can then be computed as outlined in Calculation Procedure at Velocity Inlet Boundaries (p. 1416).

For incompressible flows, density at the inlet plane is either constant or calculated as a function of temperature and/or species mass/mole fractions, where the mass or mole fractions are the values you entered as an inlet condition.

If flow exits through a pressure inlet, the total pressure specified is used as the static pressure. For incompressible flows, total temperature is equal to static temperature.

7.4.3.3.2. Compressible Flow Calculations at Pressure Inlet Boundaries

In compressible flows, isentropic relations for an ideal gas are applied to relate total pressure, static pressure, and velocity at a pressure inlet boundary. Your input of total pressure, p_0', at the inlet and the static pressure, p_s', in the adjacent fluid cell are therefore related as

$$\frac{p_0' + p_{op}}{p_s' + p_{op}} = \left(1 + \frac{\gamma - 1}{2} M^2\right)^{\gamma/(\gamma-1)} \tag{7.93}$$

where

$$M \equiv \frac{v}{c} = \frac{v}{\sqrt{\gamma R T_s}} \tag{7.94}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c = the speed of sound, and $\gamma = c_p / c_v$. Note that the operating pressure, p_{op}, appears in Equation 7.93 (p. 1410) because your boundary condition inputs are in terms of pressure relative to the operating pressure. Given p_0' and p_s', Equation 7.93 (p. 1410) and Equation 7.94 (p. 1410) are used to compute the velocity magnitude of the fluid at the inlet plane. Individual velocity components at the inlet are then derived using the direction vector components.

For compressible flow, the density at the inlet plane is defined by the ideal gas law in the form

$$\rho = \frac{p_s' + p_{op}}{RT_s} \tag{7.95}$$

For multi-species gas mixtures, the specific gas constant, R, is computed from the species mass or mole fractions, Yi that you defined as boundary conditions at the pressure inlet boundary. The static temperature at the inlet, T_s, is computed from your input of total temperature, T_0, as

$$\frac{T_0}{T_s} = 1 + \frac{\gamma - 1}{2} M^2 \tag{7.96}$$

7.4.4. Velocity Inlet Boundary Conditions

Velocity inlet boundary conditions are used to define the flow velocity, along with all relevant scalar properties of the flow, at flow inlets. In this case, the total (or stagnation) pressure is not fixed but will rise (in response to the computed static pressure) to whatever value is necessary to provide the prescribed velocity distribution. This boundary condition is applicable to incompressible and compressible flows.

Important:

Use caution when specifying the velocity inlet boundary condition for a compressible fluid in internal flow modeling. If the flow path is completely choked with no secondary flow path for relief, the velocity inlet boundary condition is not recommended, as it can cause numerical instabilities. In this case, you should switch to the pressure inlet boundary condition or refrain from reaching operating conditions that choke the internal flow. The choking condition can happen in high speed supersonic and transonic flow.

In special instances, a velocity inlet may be used in Ansys Fluent to define the flow velocity at flow exits. (The scalar inputs are not used in such cases.) In such cases you must ensure that overall continuity is maintained in the domain.

For an overview of flow boundaries, see Flow Inlet and Exit Boundary Conditions (p. 1394).

7.4.4.1. Inputs at Velocity Inlet Boundaries

7.4.4.1.1. Summary

You will enter the following information for a velocity inlet boundary:

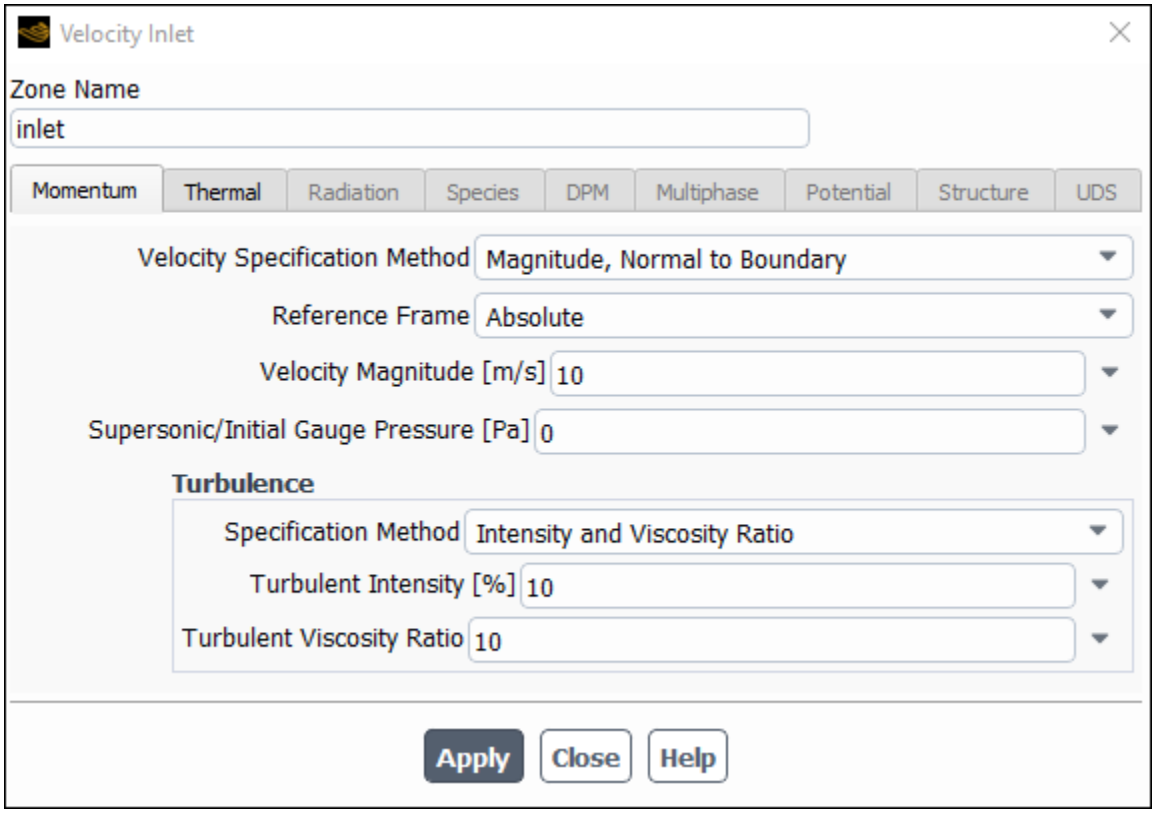
- type of reference frame
- velocity magnitude and direction or velocity components
- swirl velocity (for 2D axisymmetric problems with swirl)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- static pressure
- temperature (for energy calculations)
- outflow gauge pressure (for calculations with the density-based solver)
- turbulence parameters (for turbulent calculations)
- radiation parameters (for calculations using the P-1, DTRM, DO, surface-to-surface, or MC models)
- chemical species mass or mole fractions (for species calculations)
- mixture fraction and variance (for non-premixed or partially premixed combustion calculations)
- progress variable (for premixed or partially premixed combustion calculations)
- discrete phase boundary conditions (for discrete phase calculations)
- multiphase boundary conditions (for general multiphase calculations)
- acoustic wave model settings (see Boundary Acoustic Wave Models (p. 1527))

All values are entered in the Velocity Inlet Dialog Box (p. 5072) (Figure 7.37: The Velocity Inlet Dialog Box (p. 1412)), which is opened from the **Boundary Conditions** task page (as described in Setting Cell Zone and Boundary Conditions (p. 1303)). Note that acoustic wave model settings are described in Boundary Acoustic Wave Models (p. 1527).

Figure 7.37: The Velocity Inlet Dialog Box



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.4.1.2. Defining the Velocity

You can define the inflow velocity by specifying the velocity magnitude and direction, the velocity components, or the velocity magnitude normal to the boundary. If the cell zone adjacent to the velocity inlet is moving (that is, if you are using a moving reference frame, multiple reference frames, or sliding meshes), you can specify either relative or absolute velocities. For axisymmetric problems with swirl in Ansys Fluent, you will also specify the swirl velocity.

The procedure for defining the inflow velocity is as follows:

1. Specify the flow direction by selecting **Magnitude and Direction**, **Components**, or **Magnitude, Normal to Boundary** in the **Velocity Specification Method** drop-down list.

2. If the cell zone adjacent to the velocity inlet is moving, you can choose to specify relative or absolute velocities by selecting **Relative to Adjacent Cell Zone** or **Absolute** in the **Reference Frame** drop-down list. If the adjacent cell zone is not moving, **Absolute** and **Relative to Adjacent Cell Zone** will be equivalent, so you need not visit the list.

3. If you are going to set the velocity magnitude and direction or the velocity components, and your geometry is 3D, choose **Cartesian (X, Y, Z)**, **Cylindrical (Radial, Tangential, Axial)**, or **Local Cylindrical (Radial, Tangential, Axial)** from the **Coordinate System** drop-down list. See Defining the Flow Direction (p. 1405) for information about Cartesian, cylindrical, and local cylindrical coordinate systems.

4. Set the appropriate velocity parameters, as described below for each specification method.

7.4.4.1.3. Setting the Velocity Magnitude and Direction

If you selected **Magnitude and Direction** as the **Velocity Specification Method** in step 1 above, you will enter the magnitude of the velocity vector at the inflow boundary (the **Velocity Magnitude**) and the direction of the vector:

- If your geometry is 2D non-axisymmetric, or you chose in step 3 to use the **Cartesian** coordinate system, you will define the **X**, **Y**, and (in 3D) **Z-Component of Flow Direction**.

- If your geometry is 2D axisymmetric, or you chose in step 3 to use a **Cylindrical** coordinate system, enter the appropriate values of **Radial**, **Axial**, and (if you are modeling axisymmetric swirl or using cylindrical coordinates) **Tangential-Component of Flow Direction**.

- If you chose in step 3 to use a **Local Cylindrical** coordinate system, enter appropriate values for the **Axial**, **Radial**, and **Tangential-Component of Flow Direction**, and then specify the **X**, **Y**, and **Z** components of the **Axis Origin** and the **Axis Direction**.

Figure 7.36: Cylindrical Velocity Components in 3D, 2D, and Axisymmetric Domains (p. 1407) shows the vector components for these different coordinate systems.

7.4.4.1.4. Setting the Velocity Magnitude Normal to the Boundary

If you selected **Magnitude, Normal to Boundary** as the **Velocity Specification Method** in step 1 above, you will enter the magnitude of the velocity vector at the inflow boundary (the **Velocity Magnitude**).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.4.1.5. Setting the Velocity Components

If you selected **Components** as the **Velocity Specification Method** in step 1 above, you will enter the components of the velocity vector at the inflow boundary as follows:

- If your geometry is 2D non-axisymmetric, or you chose in step 3 to use the Cartesian coordinate system, you will define the **X**, **Y**, and (in 3D) **Z-Velocity**.
- If your geometry is 2D axisymmetric without swirl, you will set the **Radial** and **Axial-Velocity**.
- If your model is 2D axisymmetric with swirl, you will set the **Axial**, **Radial**, and **Swirl-Velocity**, and (optionally) the **Angular Velocity**, as described below.
- If you chose in step 3 to use a **Cylindrical** coordinate system, you will set the **Radial**, **Tangential**, and **Axial-Velocity**, and (optionally) the **Angular Velocity**, as described below.
- If you chose in step 3 to use a **Local Cylindrical** coordinate system, you will set the **Radial**, **Tangential**, and **Axial-Velocity**, and (optionally) the **Angular Velocity**, as described below, and then specify the **X**, **Y**, and **Z** component of the **Axis Origin** and the **Axis Direction**.

Important:

Remember that positive values for x, y, and z velocities indicate flow in the positive x, y, and z directions. If flow enters the domain in the negative x direction, for example, you will need to specify a negative value for the x velocity. The same holds true for the radial, tangential, and axial velocities. Positive radial velocities point radially out from the axis, positive axial velocities are in the direction of the axis vector, and positive tangential velocities are based on the right-hand rule using the positive axis.

7.4.4.1.6. Setting the Angular Velocity

If you chose **Components** as the **Velocity Specification Method** in step 1 above, and you are modeling axisymmetric swirl, you can specify the inlet **Angular Velocity** Ω in addition to the **Swirl-Velocity**. Similarly, if you chose **Components** as the **Velocity Specification Method** and you chose in step 3 to use a **Cylindrical** or **Local Cylindrical** coordinate system, you can specify the inlet **Angular Velocity** Ω in addition to the **Tangential-Velocity**.

If you specify Ω, v_θ is computed for each face as Ωr, where r is the radial coordinate in the coordinate system defined by the rotation axis and origin. If you specify both the **Swirl-Velocity** and the **Angular Velocity**, or the **Tangential-Velocity** and the **Angular Velocity**, Ansys Fluent will add v_θ and Ωr to get the swirl or tangential velocity at each face.

7.4.4.1.7. Defining Static Pressure

The static pressure (termed the **Supersonic/Initial Gauge Pressure**) must be specified if the inlet flow is supersonic *or* if you plan to initialize the solution based on the velocity inlet boundary conditions. Solution initialization is discussed in Initializing the Solution (p. 3627).

The **Supersonic/Initial Gauge Pressure** is ignored by Ansys Fluent whenever the flow is subsonic. If you choose to initialize the flow based on the velocity inlet conditions, the **Supersonic/Initial Gauge Pressure** will be used in conjunction with the specified stagnation quantities to compute initial values according to isentropic relations.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Remember that the static pressure value you enter is relative to the operating pressure set in the Operating Conditions Dialog Box (p. 4978). Note the comments in Pressure Inputs and Hydrostatic Head (p. 1403)

7.4.4.1.8. Defining the Temperature

For calculations in which the energy equation is being solved, you will set the static temperature of the flow at the velocity inlet boundary in the **Thermal** tab in the **Temperature** field.

7.4.4.1.9. Defining Outflow Gauge Pressure

If you are using the density-based solver, you can specify an **Outflow Gauge Pressure** for a velocity inlet boundary. If the flow exits the domain at any face on the boundary, that face will be treated as a pressure outlet with the pressure prescribed in the **Outflow Gauge Pressure** field.

7.4.4.1.10. Defining Turbulence Parameters

For turbulent calculations, there are several ways in which you can define the turbulence parameters. Instructions for deciding which method to use and determining appropriate values for these inputs are provided in Determining Turbulence Parameters (p. 1395). Turbulence modeling in general is described in Modeling Turbulence (p. 2061).

7.4.4.1.11. Defining Radiation Parameters

The options available depend on which radiation model is active. For details, see Defining Boundary Conditions for Radiation (p. 2217).

7.4.4.1.12. Defining Species Mass or Mole Fractions

If you are modeling species transport, you will set the species mass or mole fractions under **Species Mole Fractions** or **Species Mass Fractions**. For details, see Defining Cell Zone and Boundary Conditions for Species (p. 2394).

7.4.4.1.13. Defining Non-Premixed Combustion Parameters

If you are using the non-premixed or partially premixed combustion model, you will set the **Mean Mixture Fraction** and **Mixture Fraction Variance** (and the **Secondary Mean Mixture Fraction** and **Secondary Mixture Fraction Variance**, if you are using two mixture fractions), as described in Defining Non-Premixed Boundary Conditions (p. 2526).

7.4.4.1.14. Defining Premixed Combustion Boundary Conditions

If you are using the premixed or partially premixed combustion model, you will set the **Progress Variable**, as described in Setting Boundary Conditions for the Progress Variable (p. 2541).

7.4.4.1.15. Defining Discrete Phase Boundary Conditions

If you are modeling a discrete phase of particles, you can set the fate of particle trajectories at the velocity inlet. See Setting Boundary Conditions for the Discrete Phase (p. 2779) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.4.1.16. Defining Multiphase Boundary Conditions

If you are using the VOF, mixture, or Eulerian model for multiphase flow, you will need to specify volume fractions for secondary phases and (for some models) additional parameters. See Defining Multiphase Cell Zone and Boundary Conditions (p. 2957) for details.

7.4.4.2. Default Settings at Velocity Inlet Boundaries

Default settings (in SI) for velocity inlet boundary conditions are as follows:

Temperature	300
Velocity Magnitude	0
X-Component of Flow Direction	1
Y-Component of Flow Direction	0
Z-Component of Flow Direction	0
X-Velocity	0
Y-Velocity	0
Z-Velocity	0
Turbulent Intensity	5%
Turbulent Viscosity Ratio	10
Outflow Gauge Pressure	0

7.4.4.3. Calculation Procedure at Velocity Inlet Boundaries

Ansys Fluent uses your boundary condition inputs at velocity inlets to compute the mass flow into the domain through the inlet and to compute the fluxes of momentum, energy, and species through the inlet. This section describes these calculations for the case of flow entering the domain through the velocity inlet boundary and for the less common case of flow exiting the domain through the velocity inlet boundary.

7.4.4.3.1. Treatment of Velocity Inlet Conditions at Flow Inlets

When your velocity inlet boundary condition defines flow entering the physical domain of the model, Ansys Fluent uses both the velocity components and the scalar quantities that you defined as boundary conditions to compute the inlet mass flow rate, momentum fluxes, and fluxes of energy and chemical species.

The mass flow rate entering a fluid cell adjacent to a velocity inlet boundary is computed as

$$\dot{m} = \int \rho \vec{v} \cdot d\vec{A} \tag{7.97}$$

Note that only the velocity component normal to the control volume face contributes to the inlet mass flow rate.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.4.3.2. Treatment of Velocity Inlet Conditions at Flow Exits

Sometimes a velocity inlet boundary is used where flow exits the physical domain. This approach might be used, for example, when the flow rate through one exit of the domain is known or is to be imposed on the model.

Important:

In such cases you must ensure that overall continuity is maintained in the domain.

In the pressure-based solver, when flow exits the domain through a velocity inlet boundary condition, Ansys Fluent uses the boundary velocity to compute the flow flux through the exit flow area and the viscous term in the momentum equation. The convection term in all equations including momentum uses upwind discretization and takes the values of the upstream cells.

In the density-based solver, if the flow exits the domain at any face on the boundary, that face will be treated as a pressure outlet with the pressure prescribed in the **Outflow Gauge Pressure** field.

7.4.4.3.3. Density Calculation

Density at the inlet plane is either constant or calculated as a function of temperature, pressure, and/or species mass/mole fractions, where the mass or mole fractions are the values you entered as an inlet condition.

7.4.5. Mass-Flow Inlet Boundary Conditions

Mass flow boundary conditions can be used in Ansys Fluent to provide a prescribed mass flow rate or mass flux distribution at an inlet. As with a velocity inlet, specifying the mass flux permits the total pressure to vary in response to the interior solution. This is in contrast to the pressure inlet boundary condition (see Pressure Inlet Boundary Conditions (p. 1401)), where the total pressure is fixed while the mass flux varies. The mass-flow inlet is applicable to incompressible and compressible flows.

Important:

Use caution when specifying the mass-flow inlet boundary condition for a compressible fluid in internal flow modeling. If the flow path is completely choked with no secondary flow path for relief, the mass-flow inlet boundary condition is not recommended, as it can cause numerical instabilities. In this case, you should switch to the pressure inlet boundary condition or refrain from reaching operating conditions that choke the internal flow. The choking condition can happen in high speed supersonic and transonic flow.

A mass-flow inlet is often used when it is more important to match a prescribed mass flow rate than to match the total pressure of the inflow stream. An example is the case of a small cooling jet that is bled into the main flow at a fixed mass flow rate, while the velocity of the main flow is governed primarily by a (different) pressure inlet/outlet boundary condition pair. A mass-flow inlet boundary condition can also be used as an outflow by specifying the flow direction away from the solution domain.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.5.1. Limitations and Special Considerations

- The adjustment of inlet total pressure might result in a slower convergence, so if both the pressure inlet boundary condition and the mass-flow inlet boundary condition are acceptable choices, you should choose the former.
- It is not necessary to use mass-flow inlets in incompressible flows because when density is constant, velocity inlet boundary conditions will fix the mass flow.

For an overview of flow boundaries, see Flow Inlet and Exit Boundary Conditions (p. 1394).

7.4.5.2. Inputs at Mass-Flow Inlet Boundaries

7.4.5.2.1. Summary

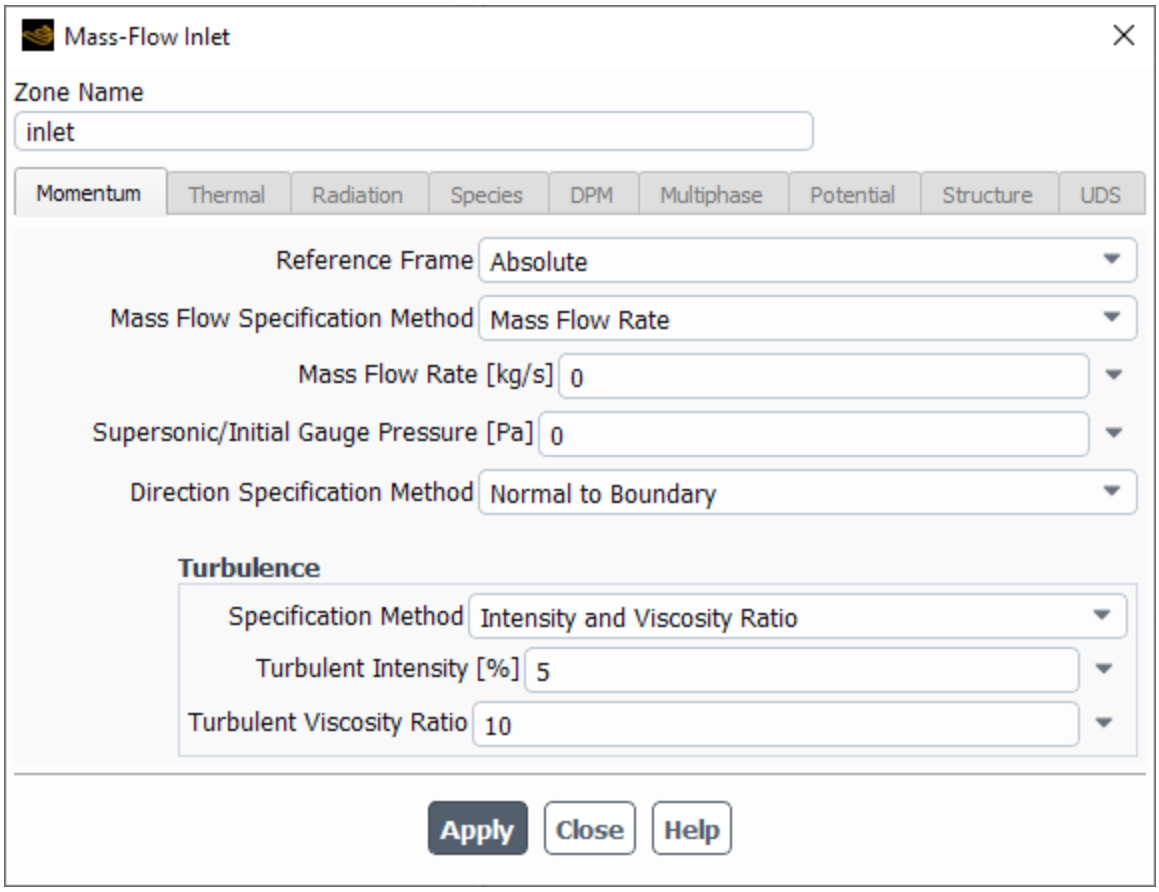
You will enter the following information for a mass-flow inlet boundary:

- type of reference frame
- mass flow rate, mass flux, or (primarily for the mixing plane model) mass flux with average mass flux
- total (stagnation) temperature
- static pressure
- flow direction
- turbulence parameters (for turbulent calculations)
- radiation parameters (for calculations using the P-1, DTRM, DO, surface-to-surface, or MC models)
- chemical species mass or mole fractions (for species calculations)
- mixture fraction and variance (for non-premixed or partially premixed combustion calculations)
- progress variable (for premixed or partially premixed combustion calculations)
- discrete phase boundary conditions (for discrete phase calculations)
- open channel flow parameters (for open channel flow calculations using the VOF multiphase model)
- acoustic wave model settings

All values are entered in the Mass-Flow Inlet Dialog Box (p. 5020) (Figure 7.38: The Mass-Flow Inlet Dialog Box (p. 1419)), which is opened from the **Boundary Conditions** task page (as described in Setting Cell Zone and Boundary Conditions (p. 1303)). Note that open channel boundary condition inputs are described in Modeling Open Channel Flows (p. 2979), and acoustic wave model settings are described in Boundary Acoustic Wave Models (p. 1527).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.38: The Mass-Flow Inlet Dialog Box



7.4.5.2.2. Selecting the Reference Frame

You will have the option to specify the mass flow boundary conditions either in the absolute or relative reference frame, when the cell zone adjacent to the mass-flow inlet is moving. For such a case, choose **Absolute** (the default) or **Relative to Adjacent Cell Zone** in the **Reference Frame** drop-down list. If the cell zone adjacent to the mass-flow inlet is not moving, both formulations are equivalent.

7.4.5.2.3. Defining the Mass Flow Rate or Mass Flux

You can specify the mass flow rate through the inlet zone and have Ansys Fluent convert this value to mass flux, or specify the mass flux directly. For cases where the mass flux varies across the boundary, you can also specify an average mass flux; see below for more information about this specification method.

You can define the mass flux or mass flow rate using a profile or a user-defined function.

The inputs for mass flow rate or flux are as follows:

1. Make a selection in the **Mass Flow Specification Method** drop-down list to specify whether you will define a **Mass Flow Rate**, **Mass Flux**, or **Mass Flux with Average Mass Flux**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. If you selected **Mass Flow Rate** (the default), enter the prescribed **Mass Flow Rate** when **constant** is selected from the drop-down list. Otherwise, select your hooked user-defined function (UDF) or transient profile.

   Important:

   The hooked UDF or transient profile can only be used to provide time-varying specification of mass flow rate. Therefore, the transient solver must be used to run the simulation. Note that the variation of profile with position in space is not applicable with this hookup.

   See `DEFINE_PROFILE` in the Fluent Customization Manual for an example of a mass-flow inlet UDF.

   Important:

   Note that for axisymmetric problems, this mass flow rate is the flow rate through the entire (2π -radian) domain, not through a 1-radian slice.

3. If you selected **Mass Flux**, enter the prescribed **Mass Flux**, or select your hooked UDF or profile.

4. If you selected **Mass Flux with Average Mass Flux**, enter the prescribed **Mass Flux** and **Average Mass Flux**.

   Important:

   Note that for axisymmetric problems, the **Mass Flux** and **Average Mass Flux** is the flux through a 1-radian slice of the domain.

7.4.5.2.4. More About Mass Flux and Average Mass Flux

As noted previously, you can specify an average mass flux with the mass flux. If, for example, you specify a mass flux profile such that the average mass flux integrated over the zone area is 4.7, but you actually want to have a total mass flux of 5, you can keep the profile unchanged, and specify an average mass flux of 5. Ansys Fluent will maintain the profile shape but adjust the values so that the resulting mass flux across the boundary is 5.

The mass flux with average mass flux specification method is also used by the mixing plane model described in Legacy Mixing Plane Model (p. 1966). If the mass-flow inlet boundary is going to represent one of the mixing planes, then you do *not* need to specify the mass flux or flow rate; you can keep the default **Mass Flow Rate** of 1. When you create the mixing plane later on in the problem setup, Ansys Fluent will automatically select the **Mass Flux with Average Mass Flux** method in the **Mass-Flow Inlet** dialog box and set the **Average Mass Flux** to the value obtained by integrating the mass flux profile for the upstream zone. This will ensure that mass is conserved between the upstream zone and the downstream (mass-flow inlet) zone.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.5.2.5. Defining the Total Temperature

Enter the value for the total (stagnation) temperature of the inflow stream in the **Total Temperature** field in the **Thermal** tab.

The total temperature is specified either in the absolute reference frame or relative to the adjacent cell zone, depending on your setting for the **Reference Frame**.

For the Eulerian multiphase model, the total temperature, and mass flux components need to be specified for the individual phases. The **Reference Frame** (**Relative to Adjacent Cell Zone** or **Absolute**) for each of the phases is the same as the reference frame selected for the mixture phase.

Important:

Note that you can only set the reference frame for the mixture, however, the total temperature can only be set for the individual phases.

Important:

- If the flow is incompressible, then the temperature assigned in the **Mass-Flow Inlet** dialog box is considered to be the static temperature.
- For the mixture multiphase model, if a boundary allows a combination of compressible and incompressible phases to enter the domain, then the temperature assigned in the **Mass-Flow Inlet** dialog box is considered to be the static temperature at that boundary. If a boundary allows *only a compressible phase* to enter the domain, then the temperature assigned in the **Mass-Flow Inlet** dialog box is the total temperature (relative/absolute) at that boundary. The total temperature depends on the **Reference Frame** option selected in the **Mass-Flow Inlet** dialog box.
- For the VOF multiphase model, if a boundary allows a *compressible phase* to enter the domain, then the temperature assigned in the **Mass-Flow Inlet** dialog box is considered to be the total temperature at that boundary. The total temperature (relative/absolute) depends on the **Reference Frame** option chosen in the dialog box. Otherwise, the temperature assigned to the boundary is considered to be the static temperature at the boundary.
- For the Eulerian multiphase model, if a boundary allows a mixture of compressible and incompressible phases in the domain, then the temperature of each of the phases is the total or static temperature, depending on whether the phase is compressible or incompressible. Total temperature (relative/absolute) depends on the **Reference Frame** option chosen in the **Mass-Flow Inlet** dialog box.

7.4.5.2.6. Defining Static Pressure

The static pressure (termed the **Supersonic/Initial Gauge Pressure**) must be specified if the inlet flow is supersonic *or* if you plan to initialize the solution based on the pressure inlet boundary conditions. Solution initialization is discussed in Initializing the Solution (p. 3627).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Supersonic/Initial Gauge Pressure** is ignored by Ansys Fluent whenever the flow is subsonic. If you choose to initialize the flow based on the mass-flow inlet conditions, the **Supersonic/Initial Gauge Pressure** will be used in conjunction with the specified stagnation quantities to compute initial values according to isentropic relations.

Remember that the static pressure value you enter is relative to the operating pressure set in the Operating Conditions Dialog Box (p. 4978). Note the comments in Pressure Inputs and Hydrostatic Head (p. 1403) regarding hydrostatic pressure.

7.4.5.2.7. Defining the Flow Direction

You can define the flow direction at a mass-flow inlet explicitly, or you can define the flow to be normal to the boundary.

The procedure for defining the flow direction is as follows, referring to Figure 7.38: The Mass-Flow Inlet Dialog Box (p. 1419):

1. Specify the flow direction by selecting **Direction Vector** or **Normal to Boundary** in the **Direction Specification Method** drop-down list. The default value for this option is **Normal to Boundary**.

2. If you selected **Direction Vector** and your geometry is 2D, go to the next step. If your geometry is 3D, choose **Cartesian (X, Y, Z)**, **Cylindrical (Radial, Tangential, Axial)**, **Local Cylindrical (Radial, Tangential, Axial)**, or **Local Cylindrical Swirl** in the **Coordinate System** drop-down list. See Defining the Flow Direction (p. 1405) for information about Cartesian, cylindrical, local cylindrical, and local cylindrical swirl coordinate systems.

3. If you selected **Direction Vector**, set the vector components as follows:

   - If your geometry is 2D non-axisymmetric, or you chose to use a 3D **Cartesian** coordinate system, enter appropriate values for the **X-**, **Y-**, and (in 3D) **Z-Component of Flow Direction**.

   - If your geometry is 2D axisymmetric, or you chose to use a 3D **Cylindrical** coordinate system, enter appropriate values for the **Axial-**, **Radial-**, and (if you are modeling swirl or using cylindrical coordinates) **Tangential-Component of Flow Direction**.

   - If you chose to use a 3D **Local Cylindrical** coordinate system, enter appropriate values for the **Axial-**, **Radial-**, and **Tangential-Component of Flow Direction**, and then specify the **X**, **Y**, and **Z** components of **Axis Origin** and the **Axis Direction**.

   - If you chose to use a 3D **Local Cylindrical Swirl** coordinate system, enter appropriate values for the **Axial-** and **Radial-Component of Flow Direction** in the axial and radial planes, and the **Tangential-Velocity**. Specify the **X**, **Y**, and **Z** components of the **Axis Origin** and the **Axis Direction**.

     Important:

     Local Cylindrical Swirl should not be used for open channel boundary conditions and on the mixing plane boundaries, while using the mixing plane model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. If you selected **Normal to Boundary**, there are no additional inputs for flow direction.

   Important:

   Note that if you are modeling axisymmetric swirl, the flow direction will be normal to the boundary; that is, there will be no swirl component at the boundary for axisymmetric swirl.

7.4.5.2.8. Defining Turbulence Parameters

For turbulent calculations, there are several ways in which you can define the turbulence parameters. Instructions for deciding which method to use and determining appropriate values for these inputs are provided in Determining Turbulence Parameters (p. 1395). Turbulence modeling is described in Modeling Turbulence (p. 2061).

7.4.5.2.9. Defining Radiation Parameters

The options available depend on which radiation model is active. For details, see Defining Boundary Conditions for Radiation (p. 2217).

7.4.5.2.10. Defining Species Mass or Mole Fractions

If you are modeling species transport, you will set the species mass or mole fractions under **Species Mole Fractions** or **Species Mass Fractions**. For details, see Defining Cell Zone and Boundary Conditions for Species (p. 2394).

7.4.5.2.11. Defining Non-Premixed Combustion Parameters

If you are using the non-premixed or partially premixed combustion model, you will set the **Mean Mixture Fraction** and **Mixture Fraction Variance** (and the **Secondary Mean Mixture Fraction** and **Secondary Mixture Fraction Variance**, if you are using two mixture fractions), as described in Defining Non-Premixed Boundary Conditions (p. 2526).

7.4.5.2.12. Defining Premixed Combustion Boundary Conditions

If you are using the premixed or partially premixed combustion model, you will set the **Progress Variable**, as described in Setting Boundary Conditions for the Progress Variable (p. 2541).

7.4.5.2.13. Defining Discrete Phase Boundary Conditions

If you are modeling a discrete phase of particles, you can set the fate of particle trajectories at the mass-flow inlet. See Setting Boundary Conditions for the Discrete Phase (p. 2779) for details.

7.4.5.2.14. Defining Open Channel Boundary Conditions

If you are using the VOF model for multiphase flow and modeling open channel flows, you will need to specify the **Free Surface Level**, **Bottom Level**, and additional parameters. See Modeling Open Channel Flows (p. 2979) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.5.3. Default Settings at Mass-Flow Inlet Boundaries

Default settings (in SI) for mass-flow inlet boundary conditions are as follows:

Mass Flow Rate	1
Total Temperature	300
Supersonic/Initial Gauge Pressure	0
X-Component of Flow Direction	1
Y-Component of Flow Direction	0
Z-Component of Flow Direction	0
Turbulent Intensity	5%
Turbulent Viscosity Ratio	10

7.4.5.4. Calculation Procedure at Mass-Flow Inlet Boundaries

When mass flow boundary conditions are used for an inlet zone, a velocity is computed for each face in that zone, and this velocity is used to compute the fluxes of all relevant solution variables into the domain. With each iteration, the computed velocity is adjusted so that the correct mass flow value is maintained.

To compute this velocity, your inputs for mass flow rate, flow direction, static pressure, and total temperature are used.

There are two ways to specify the mass flow rate. The first is to specify the total mass flow rate, $\dot{m}$, for the inlet. The second is to specify the mass flux, ρv_n (mass flow rate per unit area). If a total mass flow rate is specified, Ansys Fluent converts it internally to a uniform mass flux by dividing the mass flow rate by the total inlet area:

$$\rho v_n = \frac{\dot{m}}{A} \tag{7.98}$$

If the direct mass flux specification option is used, the mass flux can be varied over the boundary by using profile files or user-defined functions. If the average mass flux is also specified (either explicitly by you or automatically by Ansys Fluent), it is used to correct the specified mass flux profile, as described earlier in this section.

Once the value of ρv_n at a given face has been determined, the density, ρ, at the face must be determined in order to find the normal velocity, v_n. The manner in which the density is obtained depends upon whether the fluid is modeled as an ideal gas or not. Each of these cases is examined below.

7.4.5.4.1. Flow Calculations at Mass Flow Boundaries for Ideal Gases

If the fluid is an ideal gas, the static temperature and static pressure are required to compute the density:

$$p = \rho R T \tag{7.99}$$

If the inlet is supersonic, the static pressure used is the value that has been set as a boundary condition. If the inlet is subsonic, the static pressure is extrapolated from the cells inside the inlet face.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The static temperature at the inlet is computed from the total enthalpy, which is determined from the total temperature that has been set as a boundary condition. The total enthalpy is given by

$$h_0(T_0) = h(T) + \frac{1}{2}v^2 \tag{7.100}$$

where the velocity magnitude is related to the mass flow rate given by Equation 7.98 (p. 1424) and the known user-specified flow direction vector. Using Equation 7.99 (p. 1424) to relate density to the (known) static pressure and (unknown) temperature, Equation 7.100 (p. 1425) can be solved to obtain the static temperature.

7.4.5.4.2. Flow Calculations at Mass Flow Boundaries for Incompressible Flows

When you are modeling incompressible flows, the static temperature is equal to the total temperature. The density at the inlet is either constant or readily computed as a function of the temperature and (optionally) the species mass or mole fractions. The velocity is then computed using Equation 7.98 (p. 1424).

7.4.5.4.3. Flux Calculations at Mass Flow Boundaries

To compute the fluxes of all variables at the inlet, the flux velocity, v_n, is used along with the inlet value of the variable in question. For example, the flux of mass is ρv_n, and the flux of turbulence kinetic energy is $\rho k v_n$. These fluxes are used as boundary conditions for the corresponding conservation equations during the course of the solution.

7.4.6. Mass-Flow Outlet Boundary Conditions

Mass-flow outlet boundaries can be used in Ansys Fluent to pump flow out of the domain at a prescribed mass flow rate or mass flux distribution. A mass-flow outlet is often used when it is more important to match a prescribed mass flow rate than to match the static pressure of the outflow stream. By default, the mass-flow outlet boundary condition is set to 0.

7.4.6.1. Limitations

The mass-flow outlet boundary condition is designed to be strictly used only when flow is leaving the domain. It should not be used for mixed flow conditions (where flow is flowing in and out) or when flow is changing directions.

The mass-flow outlet boundary condition has the following limitations:

- Mass-flow outlet boundaries are not supported with the Wet Steam model in the density-based solver.
- For Eulerian multiphase cases, the mass flow outlet boundary condition should only be used for cases where only one of the phases exits the domain at the boundary. Otherwise, the solution results may be inaccurate.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.6.2. Inputs at Mass-Flow Outlet Boundaries

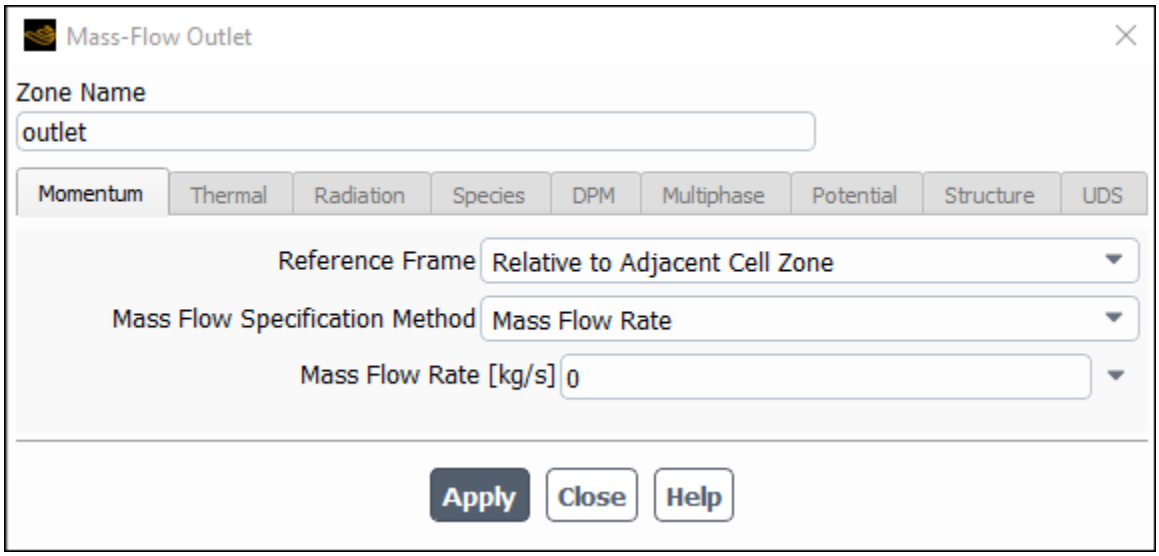
7.4.6.2.1. Summary

You will enter the following information for a mass-flow outlet boundary:

- mass flow rate, mass flux, mass flux with average mass flux, or exit corrected mass flow rate with reference temperature and gauge pressure
- radiation parameters (for calculations using the P-1, DTRM, DO, surface-to-surface, or MC models)
- discrete phase boundary conditions (for discrete phase calculations)

All values are entered in the Mass-Flow Outlet Dialog Box (p. 5028) (Figure 7.39: The Mass-Flow Outlet Dialog Box (p. 1426)), which is opened from the **Boundary Conditions** task page (as described in Setting Cell Zone and Boundary Conditions (p. 1303)).

Figure 7.39: The Mass-Flow Outlet Dialog Box



7.4.6.2.2. Selecting the Reference Frame

Mass-flow outlets only support the **Relative to Adjacent Cell Zone** specification for the **Reference Frame**.

7.4.6.2.3. Defining the Mass Flow Rate or Mass Flux

You can specify the mass flow rate through the outlet zone and have Ansys Fluent convert this value to mass flux, or specify the mass flux directly. For cases where the mass flux varies across the boundary, you can specify an average mass flux. For compressible problems, you can specify that the mass flow rate is adjusted to total conditions at the outlet, thus maintaining a constant exit corrected mass flow rate.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can define the mass flow or mass flux rate using a profile or a user-defined function (UDF). For an example of a mass-flow outlet UDF, see `DEFINE_PROFILE` in the Fluent Customization Manual.

Important:

The hooked UDF or transient profile can only be used to provide time-varying specification of mass flow or mass flux rate. Therefore, the transient solver must be used to run the simulation. Note that the variation of profile with position in space is not applicable with this hookup.

The inputs for mass flow rate or flux are as follows:

1. Make a selection in the **Mass Flow Specification Method** drop-down list to specify whether you will define a **Mass Flow Rate**, **Mass Flux**, **Mass Flux with Average Mass Flux**, or (if the density of the material is defined either as an ideal gas or using a real gas model) **Exit Corrected Mass Flow Rate**.

2. If you selected **Mass Flow Rate** (the default), enter the prescribed **Mass Flow Rate** when **constant** is selected from the drop-down list. Otherwise, select your hooked user-defined function (UDF) or transient profile.

3. If you selected **Mass Flux**, enter the prescribed **Mass Flux** (or select your hooked UDF or profile).

4. If you selected **Mass Flux with Average Mass Flux**, enter the prescribed **Mass Flux** (or select your hooked UDF or profile) and **Average Mass Flux**.

   Note that if you specify a mass flux profile such that the average mass flux integrated over the zone area is 4.7, but you actually want to have a total mass flux of 5, you can keep the profile unchanged, and specify an average mass flux of 5. Ansys Fluent will maintain the profile shape but adjust the values so that the resulting mass flux across the boundary is 5.

5. If you selected **Exit Corrected Mass Flow Rate**, enter the prescribed **Exit Corrected Mass Flow Rate** (or select your hooked UDF or profile); this corresponds to $\dot{m}_{\text{cor}}$ in Equation 7.102 (p. 1429). Then enter the **ECMF Reference Temperature** (T_{ref}) and the **ECMF Reference Gauge Pressure** (P_{ref}). For complete details about this method, see Exit Corrected Mass Flow Rate (p. 1428).

Important:

Note that for axisymmetric problems:

- If you provide a mass flow rate, you must specify the flow rate through the entire (2π -radian) domain, not through a 1-radian slice.
- If you provide a mass flux rate, you must specify the flux through a 1-radian slice of the domain.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The mass flow boundary will be pumping flow out of the domain normal to the boundary at the prescribed rate or flux. If the mass flow rate is specified, then by default, the fluxes on the boundary will be allowed to vary to preserve the flow profile out of the domain. At convergence, the total mass flow rate should match the specified value.

Important:

Note that if constant mass flux is needed rather than the default variable fluxes to preserve the profiles, then you can do so via the text command `define/boundary-conditions/bc-settings/mass-flow`. Answer `no` when asked to `preserve profile while flow leaves`.

7.4.6.2.4. Defining Radiation Parameters

The options available depend on which radiation model is active. For details, see Defining Boundary Conditions for Radiation (p. 2217).

7.4.6.2.5. Defining Discrete Phase Boundary Conditions

If you are modeling a discrete phase of particles, you can set the fate of particle trajectories at the mass-flow outlet. See Setting Boundary Conditions for the Discrete Phase (p. 2779) for details.

7.4.6.3. Calculation Procedure at Mass-Flow Outlet Boundaries

For the **Mass Flow Rate** specification method, you specify the total mass flow rate, $\dot{m}$, for the outlet. For the **Mass Flux** or **Mass Flux with Averaged Mass Flux** specification methods, you specify the mass flux, ρv_n (mass flow rate per unit area). If a total mass flow rate is specified, Ansys Fluent converts it internally to a uniform mass flux by dividing the mass flow rate by the total outlet area:

$$\rho v_n = \frac{\dot{m}}{A} \tag{7.101}$$

If the direct mass flux specification option is used, the mass flux can be varied over the boundary by using profile files or user-defined functions. If the average mass flux is also specified, it is used to correct the specified mass flux profile, as described previously in this section.

Once the value of ρv_n at a given face has been determined, the density, ρ, at the face must be determined in order to find the normal velocity, v_n.

Boundary velocities are scaled to achieve the specified value of mass flow rate or mass flux. When the mass-flow outlet is used with the profile preserving feature, a scaling factor of the specified mass flow rate over the computed mass flow rate at the boundary is used to scale the normal face velocities at the boundary. The other velocity components will be extrapolated from the interior. Flow variables such as pressure, temperature, species, or other scalar quantities will be also extrapolated from adjacent cell centers.

7.4.6.3.1. Exit Corrected Mass Flow Rate

The **Exit Corrected Mass Flow Rate** specification method adjusts the mass flow rate to the total conditions at the outlet, maintaining a constant exit corrected mass flow rate. This boundary option is mainly intended for turbomachinery applications. This method allows you to sweep through the complete machine operational range, including machine operating points from

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

choked flow to stall conditions. When using the **Exit Corrected Mass Flow Rate** specification method, the boundary operates like a mass-flow outlet on the left side of the speedline curve away from the best efficiency point, and more like a pressure outlet boundary to the right of the best efficiency point all the way beyond the choke point. This method allows you to specify the equivalent mass flow, based on similar criteria, corrected to a specified reference temperature and pressure.

For compressible flows, the exit corrected mass flow rate ($\dot{m}_{\text{cor}}$) is calculated as:

$$\dot{m}_{\text{cor}} = \dot{m} \frac{\sqrt{\tilde{T}_0 / T_{\text{ref}}}}{\tilde{P}_0 / P_{\text{ref}}} \tag{7.102}$$

where $\tilde{P}_0$ and $\tilde{T}_0$ are mass-averaged values of total pressure and temperature in the absolute frame at the outlet. P_{ref} and T_{ref} are the reference conditions, which are constants in the equation. It should be noted that the numerical behavior of the boundary condition is not affected by the choice of reference conditions; rather, the reference conditions simply provide a dimensional meaning to the otherwise non-dimensional mass flow rate. It is typical to specify these reference conditions to be the same as the inflow total pressure and temperature.

In order to maintain a constant exit corrected mass flow, you can observe that the resulting mass flow rate must be proportional to the exit total pressure and inversely proportional to the square root of the exit total temperature (or equivalently, inversely proportional to the stagnation speed of sound). This allows the boundary condition to adapt dynamically to varying operating conditions, while remaining stable as the flow develops from the initial guess.

Note that the **Exit Corrected Mass Flow Rate** specification method is only available for ideal gas and real gas materials.

You can obtain an initial exit corrected mass flow rate value ($\dot{m}_{\text{cor}}$) from a previously converged case that uses standard inlet and outlet boundaries (for example, a pressure inlet and a pressure outlet). Use the converged solution to obtain the mass-flow-averaged value of $\tilde{T}_0$ and $\tilde{P}_0$ at the outlet boundary, as well as the mass flow rate ($\dot{m}$). These values can then be applied to Equation 7.102 (p. 1429) along with P_{ref} and T_{ref} (which are usually taken as the inflow total temperature and pressure). After you have calculated the exit corrected mass flow rate, you can adjust this value up or down in order to traverse the speed line from choke to stall.

7.4.7. Inlet Vent Boundary Conditions

Inlet vent boundary conditions are used to model an inlet vent with a specified loss coefficient, flow direction, and ambient (inlet) pressure and temperature.

7.4.7.1. Inputs at Inlet Vent Boundaries

You will enter the following information for an inlet vent boundary:

- type of reference frame
- total (stagnation) pressure
- total (stagnation) temperature
- flow direction

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

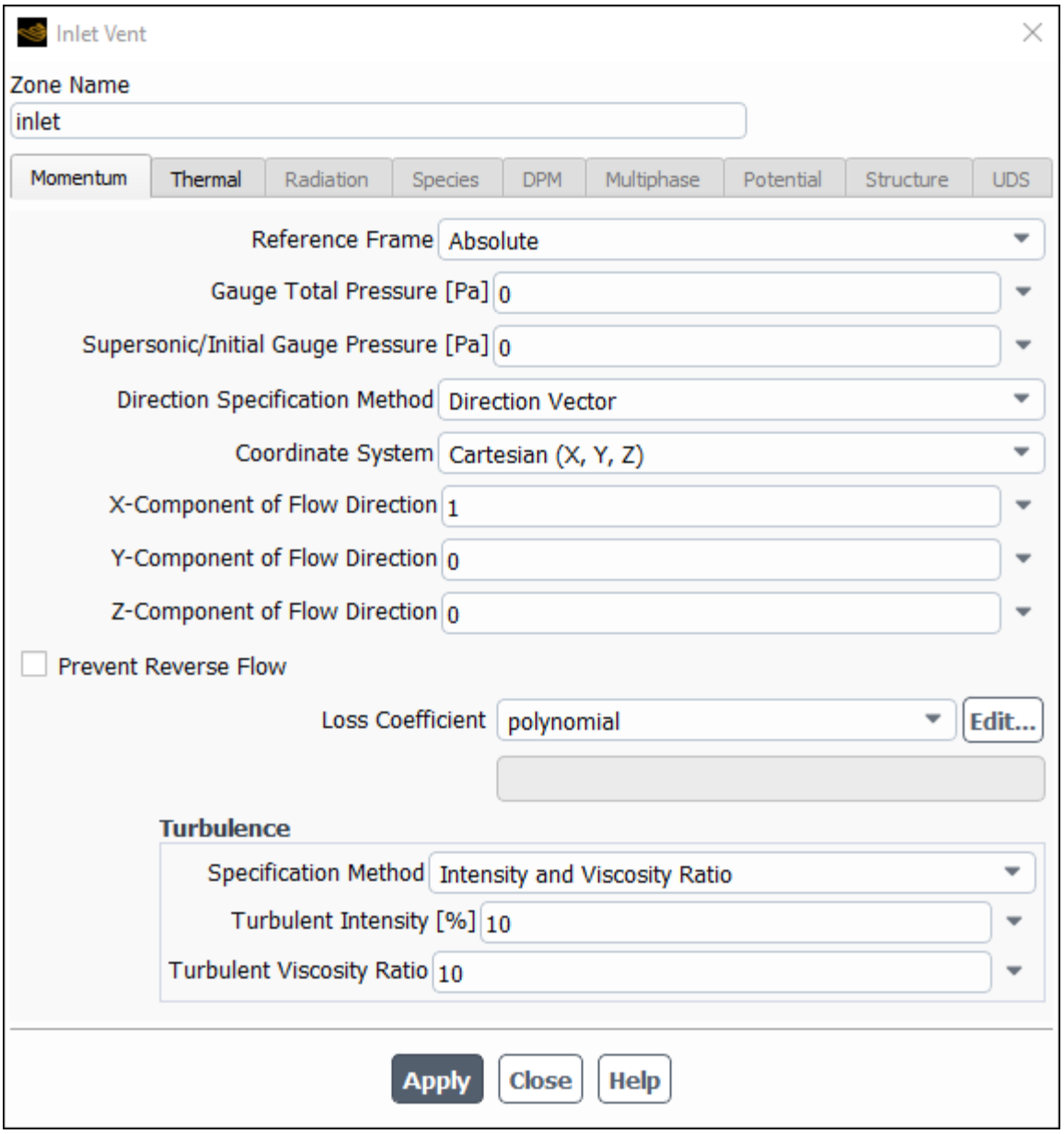
- static pressure
- turbulence parameters (for turbulent calculations)
- radiation parameters (for calculations using the P-1, DTRM, DO, surface-to-surface, or MC models)
- chemical species mass or mole fractions (for species calculations)
- mixture fraction and variance (for non-premixed or partially premixed combustion calculations)
- progress variable (for premixed or partially premixed combustion calculations)
- discrete phase boundary conditions (for discrete phase calculations)
- multiphase boundary conditions (for general multiphase calculations)
- loss coefficient
- open channel flow parameters (for open channel flow calculations using the VOF multiphase model)

All values are entered in the Inlet Vent Dialog Box (p. 5005) (Figure 7.40: The Inlet Vent Dialog Box (p. 1431)), which is opened from the **Boundary Conditions** task page (as described in Setting Cell Zone and Boundary Conditions (p. 1303)).

The first 12 items listed above are specified in the same way that they are specified at pressure inlet boundaries. See Inputs at Pressure Inlet Boundaries (p. 1402) for details. Specification of the loss coefficient is described here. Open channel boundary condition inputs are described in Modeling Open Channel Flows (p. 2979).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.40: The Inlet Vent Dialog Box



7.4.7.1.1. Specifying the Loss Coefficient

An inlet vent is considered to be infinitely thin, and the pressure drop through the vent is assumed to be proportional to the dynamic head of the fluid, with an empirically determined loss coefficient that you supply. That is, the pressure drop, Δp, varies with the normal component of velocity through the vent, v, as follows:

$$\Delta p = k_L \frac{1}{2} \rho v^2 \tag{7.103}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where ρ is the fluid density, and k_L is the non-dimensional loss coefficient.

Important:

Δp is the pressure drop in the direction of the flow; therefore the vent will appear as a resistance even in the case of backflow.

You can define the **Loss-Coefficient** across the vent as a **constant**, **polynomial**, **piecewise-linear**, or **piecewise-polynomial** function of the normal velocity. The dialog boxes for defining these functions are the same as those used for defining temperature-dependent properties. See Defining Properties Using Temperature-Dependent Functions (p. 1616) for details.

7.4.8. Intake Fan Boundary Conditions

Intake fan boundary conditions are used to model an external intake fan with a specified pressure jump, flow direction, and ambient (intake) pressure and temperature.

Note:

The intake fan boundary condition considers only momentum. Diffusion processes are not considered.

7.4.8.1. Inputs at Intake Fan Boundaries

You will enter the following information for an intake fan boundary:

- type of reference frame
- total (stagnation) pressure
- total (stagnation) temperature
- flow direction
- static pressure
- turbulence parameters (for turbulent calculations)
- radiation parameters (for calculations using the P-1, DTRM, DO, surface-to-surface, or MC models)
- chemical species mass or mole fractions (for species calculations)
- mixture fraction and variance (for non-premixed or partially premixed combustion calculations)
- progress variable (for premixed or partially premixed combustion calculations)
- discrete phase boundary conditions (for discrete phase calculations)
- multiphase boundary conditions (for general multiphase calculations)
- pressure jump

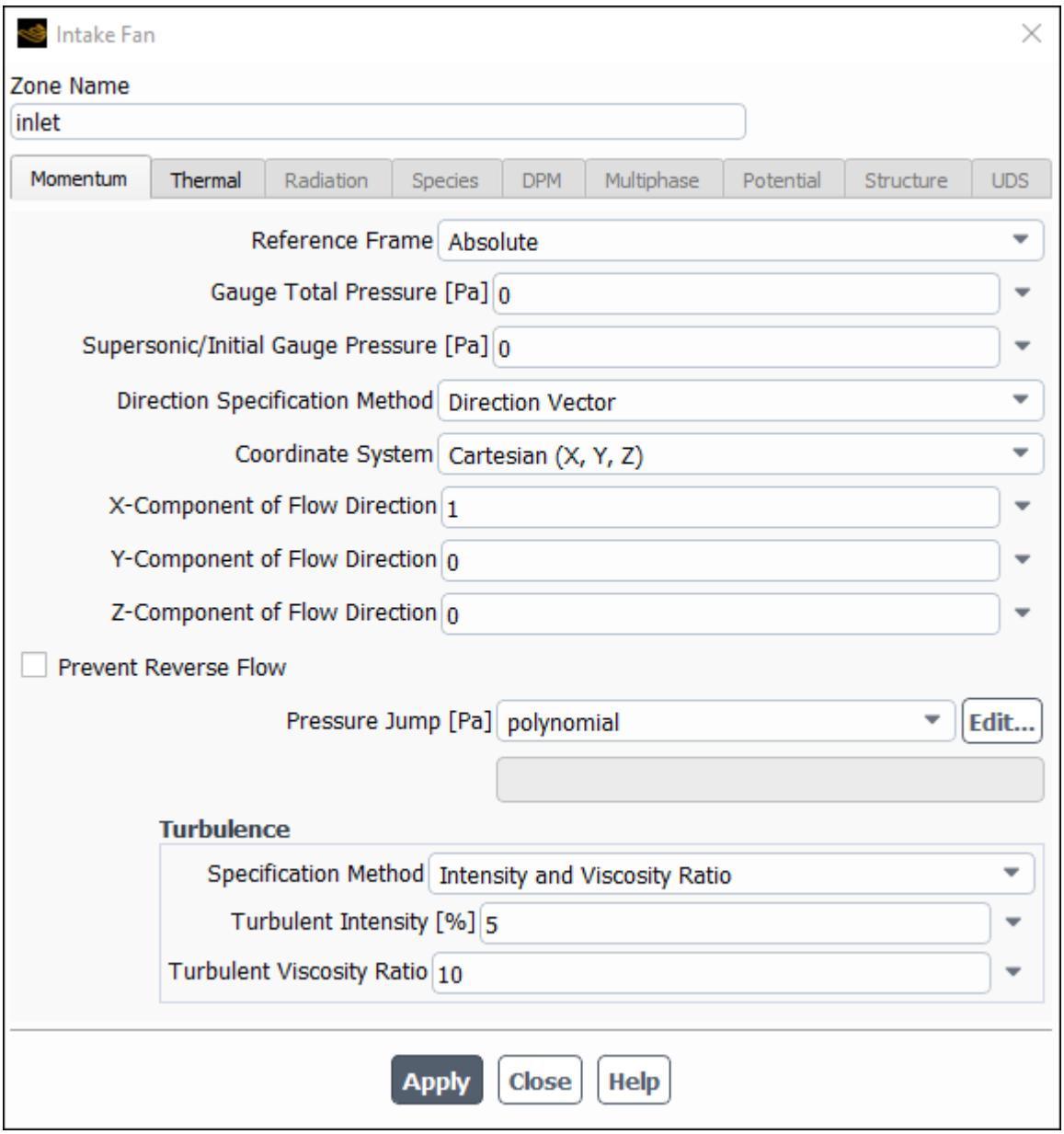
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- open channel flow parameters (for open channel flow calculations using the VOF multiphase model)

All values are entered in the Intake Fan Dialog Box (p. 5013) (shown in Figure 7.41: The Intake Fan Dialog Box (p. 1433)), which is opened from the **Boundary Conditions** task page (as described in Setting Cell Zone and Boundary Conditions (p. 1303)).

The first 12 items listed above are specified in the same way that they are specified at pressure inlet boundaries. See Inputs at Pressure Inlet Boundaries (p. 1402) for details. Specification of the pressure jump is described here. Open channel boundary condition inputs are described in Modeling Open Channel Flows (p. 2979).

Figure 7.41: The Intake Fan Dialog Box



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.8.1.1. Specifying the Pressure Jump

An intake fan is considered to be infinitely thin, and the discontinuous pressure rise across it is specified as a function of the velocity through the fan. In the case of reversed flow, the fan is treated like an outlet vent with a loss coefficient of unity.

You can define the **Pressure-Jump** across the fan as a **constant**, **polynomial**, **piecewise-linear**, or **piecewise-polynomial** function of the normal velocity. The dialog boxes for defining these functions are the same as those used for defining temperature-dependent properties. See Defining Properties Using Temperature-Dependent Functions (p. 1616) for details.

7.4.9. Pressure Outlet Boundary Conditions

Pressure outlet boundary conditions require the specification of a static (gauge) pressure at the outlet boundary. The value of the specified static pressure is used only while the flow is subsonic. Should the flow become locally supersonic, the specified pressure will no longer be used; pressure will be extrapolated from the flow in the interior. All other flow quantities are extrapolated from the interior.

A set of "backflow" conditions is also specified should the flow reverse direction at the pressure outlet boundary during the solution process. Convergence difficulties will be minimized if you specify realistic values for the backflow quantities.

Several options in Ansys Fluent exist, where a radial equilibrium outlet boundary condition can be used (see Defining Static Pressure (p. 1436) and a target mass flow rate for pressure outlets (see Target Mass Flow Rate Option (p. 1442) for details) can be specified.

For an overview of flow boundaries, see Flow Inlet and Exit Boundary Conditions (p. 1394).

7.4.9.1. Inputs at Pressure Outlet Boundaries

7.4.9.1.1. Summary

You will enter or select the following information for a pressure outlet boundary:

- static pressure
- pressure profile multiplier
- backflow conditions
  - total (stagnation) temperature (for energy calculations)
  - backflow pressure specification
  - backflow direction specification method
  - turbulence parameters (for turbulent calculations)
  - chemical species mass or mole fractions (for species calculations)
  - mixture fraction and variance (for non-premixed or partially premixed combustion calculations)
  - progress variable (for premixed or partially premixed combustion calculations)

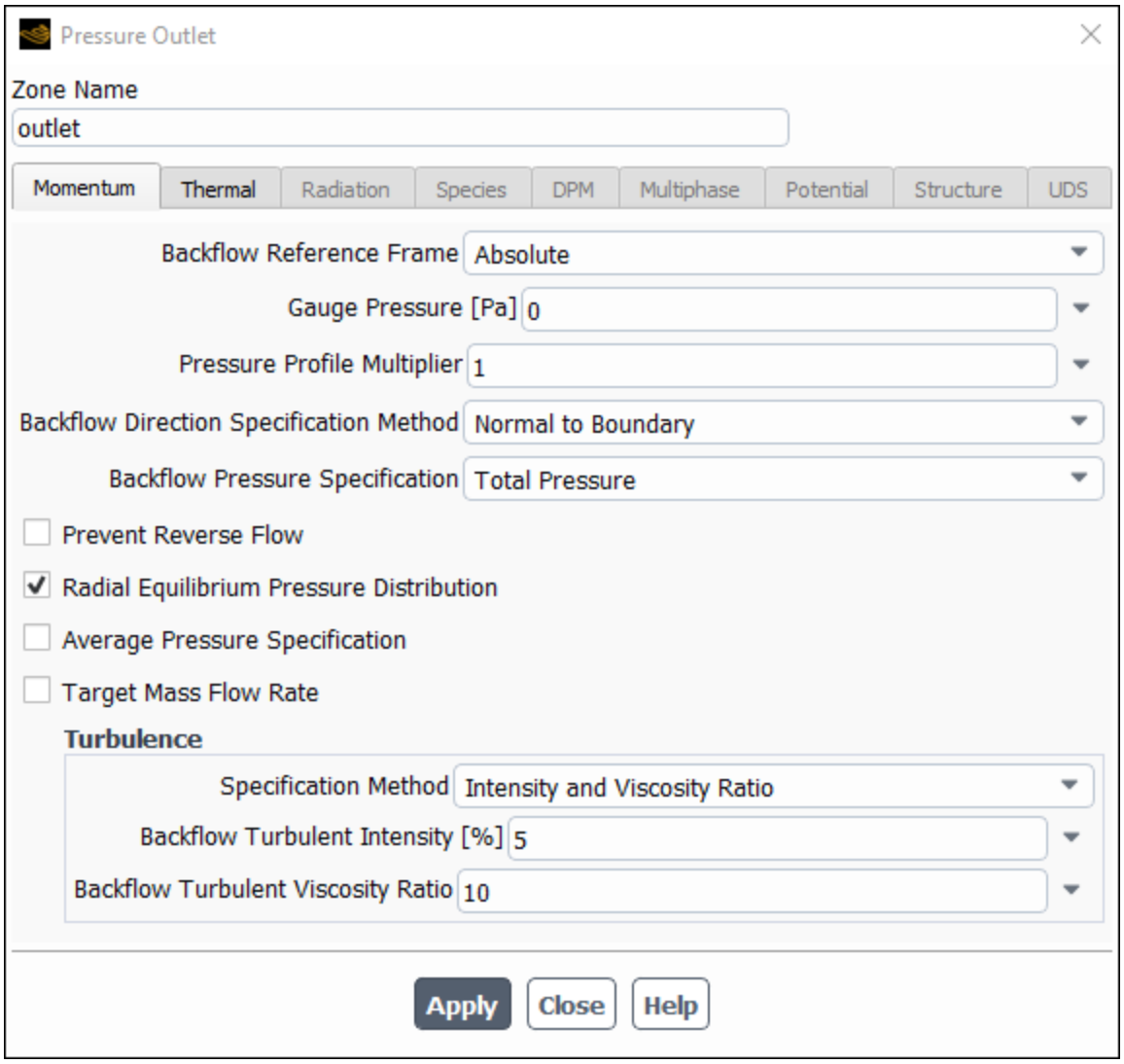
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - multiphase boundary conditions (for general multiphase calculations)
  - type of backflow reference frame (not available for multiphase flows or with the acoustic wave model)
- radiation parameters (for calculations using the P-1, DTRM, DO, surface-to-surface, or MC models)
- discrete phase boundary conditions (for discrete phase calculations)
- open channel flow parameters (for open channel flow calculations using the VOF multiphase model)
- radial equilibrium pressure distribution
- average pressure specification (not available for multiphase flows)
- target mass flow rate (not available for multiphase flows)
- acoustic wave model settings

All values are entered in the Pressure Outlet Dialog Box (p. 5060) (Figure 7.42: The Pressure Outlet Dialog Box (p. 1436)), which is opened from the **Boundary Conditions** task page (as described in Setting Cell Zone and Boundary Conditions (p. 1303)). Note that open channel boundary condition inputs are described in Modeling Open Channel Flows (p. 2979), and acoustic wave model settings are described in Boundary Acoustic Wave Models (p. 1527).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.42: The Pressure Outlet Dialog Box



7.4.9.1.2. Defining Static Pressure

To set the static pressure at the pressure outlet boundary, enter the appropriate value for **Gauge Pressure** in the **Pressure Outlet** dialog box. This value will be used when the flow is locally subsonic. Should the flow become locally supersonic, the pressure will be extrapolated from the upstream conditions. By default the subsonic and supersonic local flow condition at the boundary is determined based on the local normal velocity component.

Remember that the static pressure value you enter is relative to the operating pressure set in the Operating Conditions Dialog Box (p. 4978). Refer to Pressure Inputs and Hydrostatic Head (p. 1403) regarding hydrostatic pressure.

Ansys Fluent also provides an option to use a radial equilibrium outlet boundary condition. This option is used to model the exit flow in turbomachinery flow problems. To turn on this option, enable **Radial Equilibrium Pressure Distribution**. When this feature is active, the specified gauge pressure applies only to the position of minimum radius (relative to the axis of rotation) at the boundary. The static pressure on the rest of the zone is calculated from the assumption that radial velocity is negligible, so that the pressure gradient is given by

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\frac{\partial p}{\partial r} = \frac{\rho v_\theta^2}{r} \tag{7.104}$$

where r is the distance from the axis of rotation and v_θ is the tangential velocity. Note that this boundary condition can be used even if the rotational velocity is zero. For example, it could be applied to the calculation of the flow through an annulus containing guide vanes.

Important:

Note that the radial equilibrium outlet condition is available only for 3D and axisymmetric swirl calculations.

Ansys Fluent also provides an option to use an **Average Pressure Specification** method at the pressure outlet boundary. This option allows the pressure along the outlet boundary to vary, but maintain an average equivalent to the specified value in the **Gauge Pressure** input field. The pressure variation allowed in this boundary implementation slightly diminishes the reflectivity of the boundary as compared with the default uniform pressure specification. In the density-based solver two averaging methods are available, **Strong Averaging** and **Weak Averaging**. The average pressure specification in the pressure-based solver is the same as the strong pressure averaging in the density-based solver. For more details, see Calculation Procedure at Pressure Outlet Boundaries (p. 1439).

The **Pressure Profile Multiplier** allows you to specify a constant factor by which the **Gauge Pressure** is multiplied. This is provided mainly for cases where a non-uniform distribution of the static pressure at the pressure outlet boundary is specified by means of a profile file or a user-defined function. The scaling of an outlet profile with a multiplier can be convenient, for example, when computing a performance map for a turbomachine at different mass flow rates. The option will also work for a constant **Gauge Pressure** simply by scaling its value by a factor. This scaled pressure value will be used consistently when additional options are enabled in the **Pressure Outlet** dialog box; for example, when the **Radial Equilibrium Pressure Distribution** option is enabled, the scaled gauge pressure will be applied to the position of minimum radius (relative to the axis of rotation) at the boundary.

7.4.9.1.3. Defining Backflow Conditions

Backflow properties consistent with the models you are using will appear in the **Pressure Outlet** dialog box. The specified values will be used only if flow is pulled in through the outlet.

- The **Backflow Total Temperature** (in the **Thermal** tab) should be set for problems involving energy calculation.
- When the direction of the backflow re-entering the computational domain is known, and deemed to be relevant to the flow field solution, you can specify it choosing one of the options available in the **Backflow Direction Specification Method** drop-down list. The default value for this field is **Normal to Boundary**, and requires no further input. If you choose **Direction Vector**, the dialog box will expand to show the inputs for the components of the direction vector for the backflow, and if you are running the 3D version of Ansys Fluent, the dialog box will display a **Coordinate System** drop-down list. If you choose **From Neighboring Cell**, Ansys Fluent will determine the direction of the backflow using the direction of the flow in the cell layer adjacent to the pressure outlet.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For turbulent calculations, there are several ways in which you can define the turbulence parameters. Instructions for deciding which method to use in determining appropriate values for these inputs are provided in Determining Turbulence Parameters (p. 1395). Turbulence modeling in general is described in Modeling Turbulence (p. 2061).

- If you are modeling species transport, you will set the backflow species mass or mole fractions under **Species Mass Fractions** or **Species Mole Fractions**. For details, see Defining Cell Zone and Boundary Conditions for Species (p. 2394).

- If you are modeling combustion using the non-premixed or partially premixed combustion model, you will set the backflow mixture fraction and variance values. See Defining Non-Premixed Boundary Conditions (p. 2526) for details.

- If you are modeling combustion using the premixed or partially premixed combustion model, you will set the backflow **Progress Variable** value. See Setting Boundary Conditions for the Progress Variable (p. 2541) for details.

- If you are using the VOF, mixture, or Eulerian model for multiphase flow, you will need to specify volume fractions for secondary phases and (for some models) additional parameters. See Defining Multiphase Cell Zone and Boundary Conditions (p. 2957) for details.

- If backflow occurs,

  1. The pressure you specified as the **Gauge Pressure** will be used to determine the total pressure or the static pressure (based on your selection from the **Backflow Pressure Specification** drop-down list), so you need not specify a backflow pressure value explicitly. When **Total Pressure** is selected for **Backflow Pressure Specification**, boundary face pressure values are calculated by combining the **Gauge Pressure** with a dynamic contribution that is based on the velocity in the adjacent cell zone; when **Static Pressure** is selected for **Backflow Pressure Specification**, the specified **Gauge Pressure** value is directly imposed as the boundary face pressure.

  2. The flow direction will be based on your selection from the **Backflow Direction Specification Method** drop-down list.

  3. The total temperature, total pressure, and flow directions are in *absolute* or *relative to the adjacent cell zone* reference frames, based on the **Backflow Reference Frame** setting specified in the **Pressure Outlet** dialog box.

  If the cell zone adjacent to the pressure outlet is moving (that is, if you are using a moving reference frame, multiple reference frames, mixing planes, or sliding meshes) and you are using the pressure-based solver, the velocity in the dynamic contribution to total pressure (see Equation 7.88 (p. 1404)) will be absolute or relative to the motion of the cell zone, depending on whether or not the **Absolute** velocity formulation is selected in the **General** task page. For the density-based solver, the velocity in Equation 7.88 (p. 1404) (or the Mach number in Equation 7.89 (p. 1404)) is always in the absolute frame.

Important:

Even if no backflow is expected in the converged solution, you should always set realistic values to minimize convergence difficulties in the event that backflow does occur during the calculation.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Backflow Reference Frame** option is not available for all multiphase models or non-reflecting boundary condition options.

7.4.9.1.3.1. Prevent Reverse Flow

When this option is selected, Fluent will erect artificial walls on the boundary mesh faces to prevent flow back into the domain. The artificial walls are removed when the flow is no longer entering the domain and when a favorable pressure gradient is recovered at the boundary mesh faces.

When artificial walls are created, Fluent will print a message in the console window indicating the number of faces on which artificial walls have been erected and the equivalent area percentage of the boundary it represents.

Note:

This feature is not available with multiphase flow.

7.4.9.1.4. Defining Radiation Parameters

The options available depend on which radiation model is active. For details, see Defining Boundary Conditions for Radiation (p. 2217).

7.4.9.1.5. Defining Discrete Phase Boundary Conditions

If you are modeling a discrete phase of particles, you can set the fate of particle trajectories at the pressure outlet. See Setting Boundary Conditions for the Discrete Phase (p. 2779) for details.

7.4.9.1.6. Defining Open Channel Boundary Conditions

If you are using the VOF model for multiphase flow and modeling open channel flows, you will need to specify the **Free Surface Level**, **Bottom Level**, and additional parameters. See Modeling Open Channel Flows (p. 2979) for details.

7.4.9.2. Default Settings at Pressure Outlet Boundaries

Default settings (in SI) for pressure outlet boundary conditions are as follows:

Gauge Pressure	0
Pressure Profile Multiplier	1
Backflow Total Temperature	300
Backflow Turbulent Intensity	5%
Backflow Turbulent Viscosity Ratio	10

7.4.9.3. Calculation Procedure at Pressure Outlet Boundaries

At pressure outlets, if the flow exiting the boundary is subsonic, then Ansys Fluent uses the pressure specified in the **Pressure Outlet** dialog box as the static pressure of the fluid at the outlet plane,

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

P_s, and extrapolates all other conditions from the interior of the domain. However, if the flow exiting the boundary becomes supersonic, then the face pressure values are extrapolated from the interior cell pressure.

7.4.9.3.1. Average Pressure Specification

When this option is selected, the exit pressure is not kept constant over the pressure outlet boundary, but instead is allowed to vary across the boundary while maintaining an average boundary pressure close to the specified static exit pressure. This option can be used for pressure specified as a constant or as a profile (for example, in turbomachinery cases with radial, axial, or radial equilibrium pressure distributions).

There are two implementations of the **Average Pressure Specification**. These are **Strong Averaging** and **Weak Averaging**. The strong averaging approach is available in both the density-based and the pressure-based solver. The weak averaging approach is available only in the density-based solver.

Note:

- The **Average Pressure Specification** option is not available with multiphase flows.
- When using the density-based solver, **Strong Averaging** is the recommended method and is used by default. The weak averaging method is maintained primarily for legacy simulations.
- When one of the **Acoustic Wave Models** is active, the Average Pressure Specification option is not offered because the pressure will be obtained based on the **Acoustic Wave Model** chosen.
- Using the Average Pressure Specification option in unsteady simulations can result in nonphysical pressure feedback and is not recommended.

7.4.9.3.1.1. Strong Averaging

In the **Strong Averaging** approach, the face pressure value P_f for subsonic exit flow is computed using the following expression:

$$P_f = P_e + \left(1 - F\right)\left(P_c - P_{c_avg}\right) \tag{7.105}$$

where

P_c = interior cell pressure at neighboring exit face, f

P_{c_avg} = averaged interior cell pressure at a boundary

P_e = specified exit pressure

F = pressure blending factor; $F=0$ recovers the fully averaged pressure, $F=1$ recovers the specified pressure.

When the strong averaging option is used with a profile specification method (radial, axial, or radial equilibrium pressure distribution) then the outlet boundary will be subdivided into radial or axial containers called "bins", and the above averaging procedure is applied to each bin.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

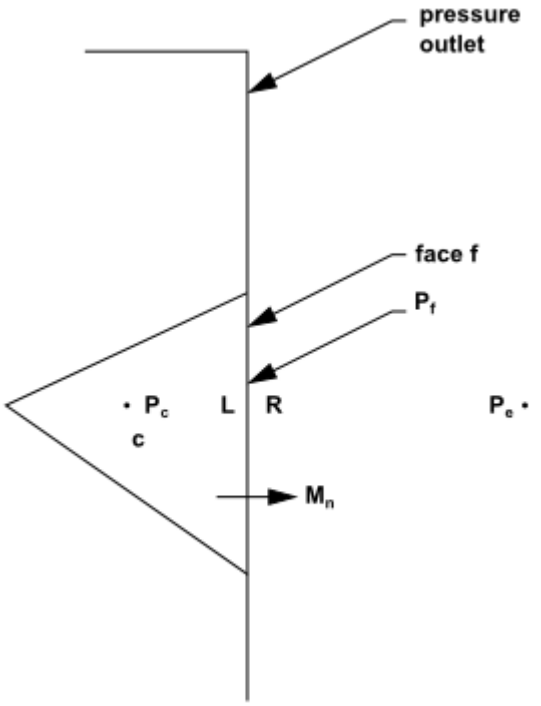
You can use the following TUI command to change the blending factor and the number of bins:

```
/define/boundary-conditions/bc-settings/pressure-outlet
```

7.4.9.3.1.2. Weak Averaging

When the weak averaging method is used and the flow is subsonic, the pressure at the faces of the outlet boundary is computed using a weighted average of the left and right state of the face boundary. This weighting is a blend of fifth-order polynomials based on the exit face normal Mach number [91] (p. 5718). Therefore, the face pressure P_f is a function of (P_c, P_e, M_n), where P_c is the interior cell pressure neighboring the exit face f, P_e is the specified exit pressure, and M_n is the face normal Mach number.

Figure 7.43: Pressures at the Face of a Pressure Outlet Boundary



For incompressible flows, the face pressure is computed as an average between the specified pressure and the interior pressure.

$$P_f = 0.5\left(P_c + P_e\right) \tag{7.106}$$

In this boundary implementation, the exit pressure is not constant along the pressure outlet boundary. However, upon flow convergence, the average boundary pressure will be close to the specified static exit pressure.

7.4.9.4. Other Optional Inputs at Pressure Outlet Boundaries

7.4.9.4.1. Non-Reflecting Boundary Conditions Option

One of the options that may be used at pressure outlets is non-reflecting boundary conditions (NRBC). This option is used when waves are made to pass through the boundaries while avoiding

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

false reflections. Details of non-reflecting boundary conditions can be found in General Non-Reflecting Boundary Conditions (p. 1538).

7.4.9.4.2. Target Mass Flow Rate Option

The simple Bernoulli's equation is used to adjust the pressure at every iteration on a pressure outlet zone in order to meet the desired mass flow rate. The change in pressure, based on Bernoulli's equation is given by the following equation:

$$dP = 0.5\rho_{ave}\left(\dot{m}^2 - \dot{m}_{req}^2\right) / \left(\rho_{ave}A\right)^2 \tag{7.107}$$

where dP is the change in pressure, $\dot{m}$ is the current computed mass flow rate at the pressure-outlet boundary, $\dot{m}_{req}$ is the required mass flow rate, ρ_{ave} is the computed average density at the pressure-outlet boundary, and A is the area of the pressure-outlet boundary.

Limitations

- The target mass flow rate option is not available with multiphase flows or when any of the non-reflecting boundary conditions models are used.
- If the pressure-outlet zone is used in the mixing-plane model, the target mass flow rate option will not be available for that particular zone.
- The pressure outlet will not achieve the target mass flow rate if the flow becomes choked (that is, the Mach number of the fluid in the pressure-outlet zone becomes equal to 1).
- The pressure outlet may not achieve the target mass flow rate if porous zones are present in the flow domain causing a large pressure drop.

Target Mass Flow Rate Settings

To use the target mass flow rate option

1. Enable **Target Mass Flow Rate** in the **Pressure Outlet** dialog box.
2. Specify the **Target Mass Flow** as either a constant value or hook a UDF to set the target mass flow rate.

   The settings for the target mass flow rate option can be accessed from the `target-mass-flow-rate-settings` text command:

   `define` → `boundary-conditions` → `target-mass-flow-rate-settings`

   You will be prompted to

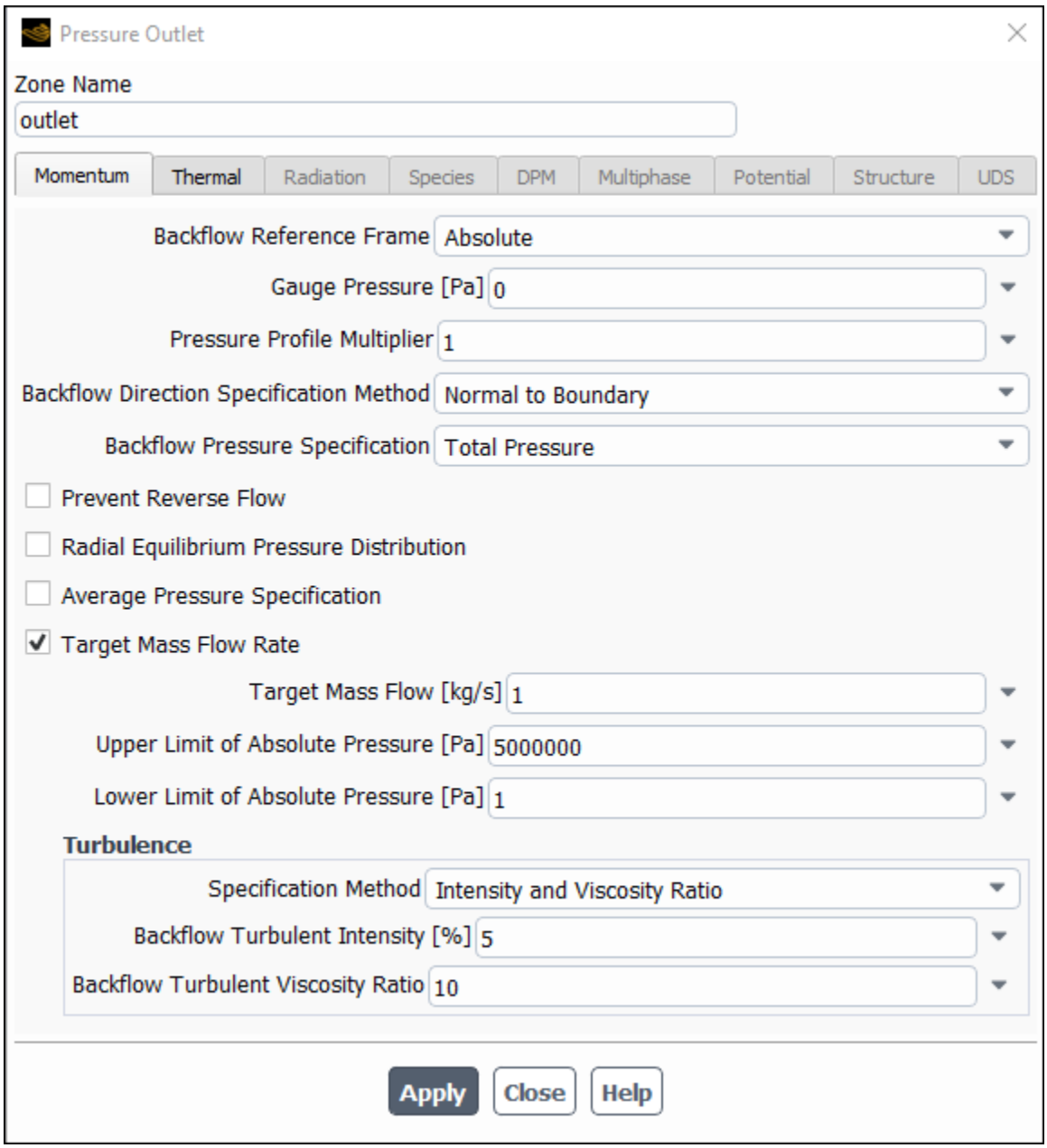
   a. Set the under-relaxation factor (the default setting is 0.05).

   b. Enable the targeted mass flow rate verbosity (the default is `no`). If enabled, it prints to the console window the required mass flow rate, computed mass flow rate, mean pressure, the new pressure imposed on the outlet, and the change in pressure in SI units.
3. In the **Pressure Outlet** dialog box, specify the **Upper Limit of Absolute Pressure** and **Lower Limit of Absolute Pressure**. Specifying the range of the pressure limits improves convergence in cases with a large number of outlet boundaries, which have different pressure variations

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

on different boundaries. You can also use the `define/boundary-conditions/pressure-outlet` text command to specify these limits.

Figure 7.44: The Pressure Outlet Dialog Box with the Target Mass Flow Rate Option Enabled



Solution Strategies When Using the Target Mass Flow Rate Option

If convergence difficulties are encountered or if the solution is not converging at the desired mass flow rate, then try to lower the under-relaxation factor from the default value. Otherwise, you can use the alternate method to converge at the required mass flow rate.

In some cases, you may want to switch off the target mass flow rate option initially, then guess an exit pressure that will bring the solution closer to the target mass flow rate. After the solution stabilizes, you can turn on the target mass flow rate option and iterate to convergence. For many complex flow problems, this strategy is usually very successful.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The use of Full Multigrid Initialization is also very helpful in obtaining a good starting solution and in general will reduce the time required to get a converged solution on a target mass flow rate. For further information on Full Multigrid Initialization, see Full Multigrid (FMG) Initialization (p. 3633).

Setting Target Mass Flow Rates Using UDFs

For some unsteady problems it is desirable that the target mass flow rate be a function of the physical flow time. This enforcement of boundary condition can be done by attaching a UDF with `DEFINE_PROFILE` functions to the target mass flow rate field.

Important:

Note that the mass flow rate profile is a function of time and only one constant value should be applied to all zone faces at a given time.

An example of a simple UDF using a `DEFINE_PROFILE` that will adjust the mass flow rate can be found in `DEFINE_PROFILE` in the Fluent Customization Manual.

7.4.10. Pressure Far-Field Boundary Conditions

Pressure far-field conditions are used in Ansys Fluent to model a free-stream condition at infinity, with free-stream Mach number and static conditions being specified. The pressure far-field boundary condition is often called a characteristic boundary condition, since it uses characteristic information (Riemann invariants) to determine the flow variables at the boundaries.

7.4.10.1. Limitations

Note the following limitations and restrictions when using pressure far-field boundary conditions:

- This boundary condition is applicable only when the density is calculated using the ideal-gas law (see Density (p. 1624)). Using it for other flows is not permitted. To effectively approximate true infinite-extent conditions, you must place the far-field boundary far enough from the object of interest. For example, in lifting airfoil calculations, it is not uncommon for the far-field boundary to be a circle with a radius of 20 chord lengths.
- It is incompatible with the multiphase models (VOF, mixture, and Eulerian) that are available with the pressure-based solver.
- It cannot be applied to flows that employ constant density, the real gas model, and the wet steam model, which are available in the density-based solver.

For an overview of flow boundaries, see Flow Inlet and Exit Boundary Conditions (p. 1394).

7.4.10.2. Inputs at Pressure Far-Field Boundaries

7.4.10.2.1. Summary

You will enter the following information for a pressure far-field boundary:

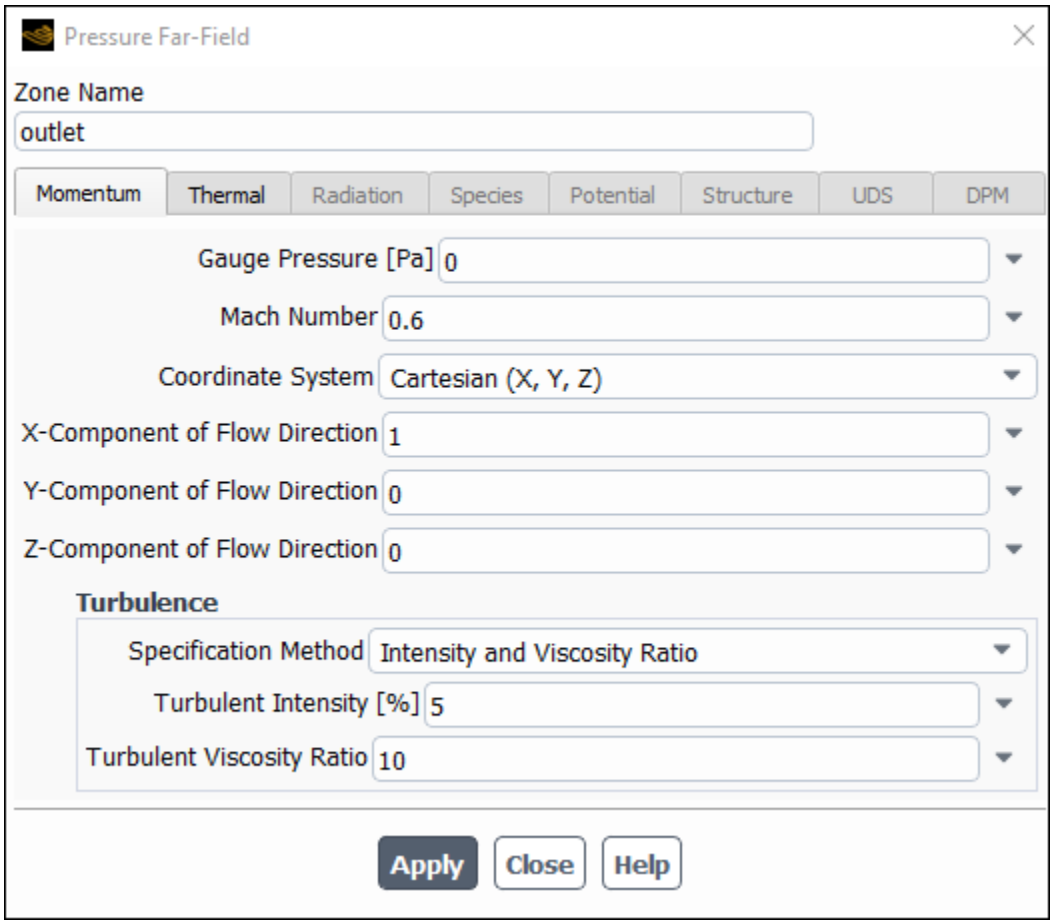
- static pressure

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Mach number
- temperature
- flow direction
- turbulence parameters (for turbulent calculations)
- radiation parameters (for calculations using the P-1, DTRM, DO, surface-to-surface, or MC models)
- chemical species mass or mole fractions (for species calculations)
- discrete phase boundary conditions (for discrete phase calculations)

All values are entered in the Pressure Far-Field Dialog Box (p. 5045) (Figure 7.45: The Pressure Far-Field Dialog Box (p. 1445)), which is opened from the **Boundary Conditions** task page (as described in Setting Cell Zone and Boundary Conditions (p. 1303)).

Figure 7.45: The Pressure Far-Field Dialog Box



7.4.10.2.2. Defining Static Pressure, Mach Number, and Static Temperature

To set the static pressure and temperature at the far-field boundary in the **Pressure Far-Field** dialog box, enter the appropriate values for **Gauge Pressure** and **Mach Number** in the **Momentum**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

tab. The Mach number can be subsonic, sonic, or supersonic. Set the **Temperature** in the **Thermal** tab.

7.4.10.2.3. Defining the Flow Direction

You can define the flow direction at a pressure far-field boundary by setting the components of the direction vector. If your geometry is 2D non-axisymmetric enter appropriate values for **X** and **Y** in the **Pressure Far-Field** dialog box (Figure 7.45: The Pressure Far-Field Dialog Box (p. 1445)). If your geometry is 2D axisymmetric, enter the appropriate values for **Axial**, **Radial**, and (if you are modeling axisymmetric swirl) **Tangential-Component of Flow Direction**.

If your geometry is 3D, you can choose a **Coordinate System** that is **Cartesian**, **Cylindrical**, or **Local Cylindrical**. In the Cartesian coordinate system, enter the appropriate values for **X**, **Y**, and **Z-Component of Flow Direction**. If the direction cosine data on the boundary is available, then use the cylindrical or local cylindrical coordinate system and specify the **Axial**, **Radial**, **Tangential-Component of Flow Direction**. For **Cylindrical**, axis parameters need to be specified on the adjacent cell zone of the boundary face. For **Local Cylindrical Swirl**, specify the **Axis Origin** and **Axis Direction**.

7.4.10.2.4. Defining Turbulence Parameters

For turbulent calculations, there are several ways in which you can define the turbulence parameters. Instructions for deciding which method to use and determining appropriate values for these inputs are provided in Determining Turbulence Parameters (p. 1395). Turbulence modeling is described in Modeling Turbulence (p. 2061).

7.4.10.2.5. Defining Radiation Parameters

The options available depend on which radiation model is active. For details, see Defining Boundary Conditions for Radiation (p. 2217).

7.4.10.2.6. Defining Species Transport Parameters

If you are modeling species transport, you will set the species mass or mole fractions under **Species Mass Fractions** or **Species Mole Fractions**. See Defining Cell Zone and Boundary Conditions for Species (p. 2394) for details.

7.4.10.3. Defining Discrete Phase Boundary Conditions

If you are modeling a discrete phase of particles, you can set the fate of particle trajectories at the pressure far-field boundary. See Setting Boundary Conditions for the Discrete Phase (p. 2779) for details.

7.4.10.4. Default Settings at Pressure Far-Field Boundaries

Default settings (in SI) for pressure far-field boundary conditions are as follows:

Gauge Pressure	0
Mach Number	0.6
Temperature	300
X-Component of Flow Direction	1

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Y-Component of Flow Direction	0
Z-Component of Flow Direction	0
Turbulent Intensity	5%
Turbulent Viscosity Ratio	10

7.4.10.5. Calculation Procedure at Pressure Far-Field Boundaries

The pressure far-field boundary condition is a non-reflecting boundary condition based on the introduction of Riemann invariants (that is, characteristic variables) for a one-dimensional flow normal to the boundary. For flow that is subsonic there are two Riemann invariants, corresponding to incoming and outgoing waves:

$$R_{\infty}=v_{n_{\infty}}-\frac{2c_{\infty}}{\gamma-1} \tag{7.108}$$

$$R_i=v_{n_i}+\frac{2c_i}{\gamma-1} \tag{7.109}$$

where v_n is the velocity magnitude normal to the boundary, c is the local speed of sound and γ is the ratio of specific heats (ideal gas). The subscript ∞ refers to conditions being applied at infinity (the boundary conditions), and the subscript i refers to conditions in the interior of the domain (that is, in the cell adjacent to the boundary face). These two invariants can be added and subtracted to give the following two equations:

$$v_n=\frac{1}{2}\left(R_i+R_{\infty}\right) \tag{7.110}$$

$$c=\frac{\gamma-1}{4}\left(R_i-R_{\infty}\right) \tag{7.111}$$

where v_n and c become the values of normal velocity and sound speed applied on the boundary. At a face through which flow exits, the tangential velocity components and entropy are extrapolated from the interior; at an inflow face, these are specified as having free-stream values. Using the values for v_n, c, tangential velocity components, and entropy the values of density, velocity, temperature, and pressure at the boundary face can be calculated.

7.4.10.6. Flux-based Pressure Far-Field

Fluent's density-based solver offers an alternative method for the pressure far-field boundary condition. This method employs a Riemann solver to evaluate the numerical flux at the far-field boundary, thus providing the same level of accuracy and robustness as the interior faces. This treatment is a global application of the local tangency-correction used to regain robustness on boundary faces tangent to the imposed flow direction (as described in Tangency Correction (p. 1448)). It is also advantageous for calculations where boundary layers or wakes extend beyond the boundary of the computational domain. This flux-based pressure far-field can be selected with the following text command:

```
/define/boundary-condition/bc-settings/pressure-far-field/type?
```

Following the prompts, enter **1** for `Flux-based`.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.10.7. Tangency Correction

Depending on the choice of computational domain, certain boundary-faces might be tangent to the imposed free-stream flow direction. Flow tangency along a pressure far-field boundary hinders the solver robustness. To regain robustness in such situations, a local tangency-correction has been implemented in Fluent's density-based solver. This correction locally replaces the standard Riemann-invariants boundary treatment by a numerical-flux evaluation, analogous to using a Riemann solver. The correction is active by default, and can be deactivated with the following text command:

```
/define/boundary-condition/bc-settings/pressure-far-field/riemann-in-
variants-tangency-correction?
```

7.4.11. Outflow Boundary Conditions

Outflow boundary conditions in Ansys Fluent are used to model flow exits where the details of the flow velocity and pressure are not known prior to solving the flow problem. You do not define any conditions at outflow boundaries (unless you are modeling radiative heat transfer, a discrete phase of particles, or split mass flow): Ansys Fluent extrapolates the required information from the interior. It is important, however, to understand the limitations of this boundary type.

Important:

Note that outflow boundaries cannot be used in the following cases:

- If a problem includes pressure inlet boundaries; use pressure outlet boundary conditions (see Pressure Outlet Boundary Conditions (p. 1434)) instead.
- If you are modeling compressible flow.
- If you are modeling unsteady flows with varying density (even if the fluid is incompressible), it is preferable to use a pressure outlet.

In general, an outflow condition may be used in incompressible cases using the Eulerian or Mixture multiphase models. However, if the flow may produce a recirculation at the outlet or if the flow field is not stable and fully developed at the outlet, then a pressure outlet boundary condition is preferred.

For an overview of flow boundaries, see Flow Inlet and Exit Boundary Conditions (p. 1394).

7.4.11.1. Ansys Fluent's Treatment at Outflow Boundaries

The boundary conditions used by Ansys Fluent at outflow boundaries are as follows:

- A zero diffusion flux for all flow variables.
- An overall mass balance correction.

The zero diffusion flux condition applied at outflow cells means that the conditions of the outflow plane are extrapolated from within the domain and have no impact on the upstream flow. The extrapolation procedure used by Ansys Fluent updates the outflow velocity and pressure in a

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

manner that is consistent with a fully-developed flow assumption, as noted below, when there is no area change at the outflow boundary.

The zero diffusion flux condition applied by Ansys Fluent at outflow boundaries is approached physically in fully-developed flows. Fully-developed flows are flows in which the flow velocity profile (and/or profiles of other properties such as temperature) is unchanging in the flow direction.

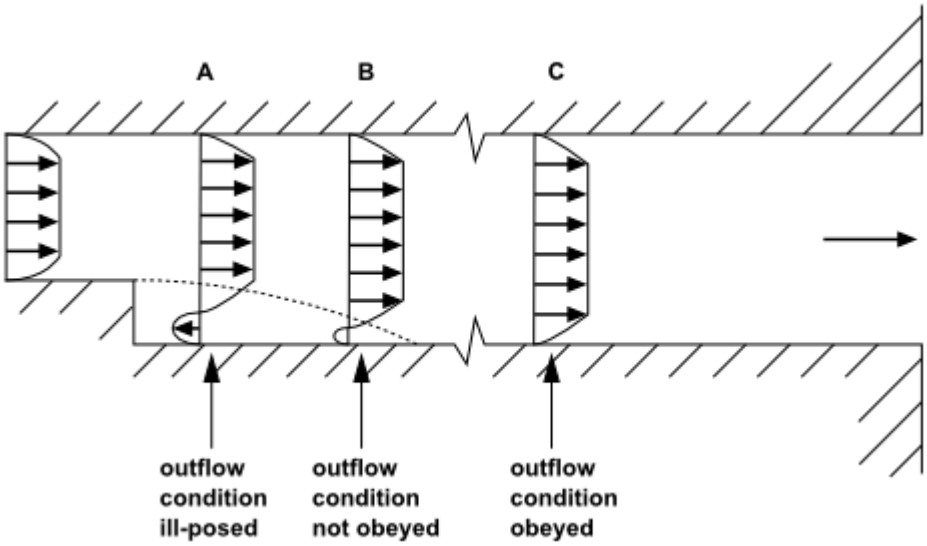
It is important to note that gradients in the cross-stream direction may exist at an outflow boundary. Only the diffusion fluxes in the direction normal to the exit plane are assumed to be zero.

7.4.11.2. Using Outflow Boundaries

As noted in Ansys Fluent's Treatment at Outflow Boundaries (p. 1448), the outflow boundary condition is obeyed in fully-developed flows where the diffusion flux for all flow variables in the exit direction are zero. However, you may also define outflow boundaries at physical boundaries where the flow is not fully developed—and you can do so with confidence if the assumption of a zero diffusion flux at the exit is expected to have a small impact on your flow solution. The appropriate placement of an outflow boundary is described by example below.

- Outflow boundaries where normal gradients are negligible: Figure 7.46: Choice of the Outflow Boundary Condition Location (p. 1449) shows a simple two-dimensional flow problem and several possible outflow boundary location choices. Location C shows the outflow boundary located upstream of the plenum exit but in a region of the duct where the flow is fully-developed. At this location, the outflow boundary condition is exactly obeyed.

  Figure 7.46: Choice of the Outflow Boundary Condition Location



- Ill-posed outflow boundaries: Location B in Figure 7.46: Choice of the Outflow Boundary Condition Location (p. 1449) shows the outflow boundary near the reattachment point of the recirculation in the wake of the backward-facing step. This choice of outflow boundary condition is ill-posed as the gradients normal to the exit plane are quite large at this point and can be expected to have a significant impact on the flow field upstream. Because the outflow boundary condition ignores these axial gradients in the flow, location B is a poor choice for an outflow boundary. The exit location should be moved downstream from the reattachment point.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.46: Choice of the Outflow Boundary Condition Location (p. 1449) shows a second ill-posed outflow boundary at location A. Here, the outflow is located where flow is pulled into the Ansys Fluent domain through the outflow boundary. In situations like this the Ansys Fluent calculation typically does not converge and the results of the calculation have no validity. This is because when flow is pulled into the domain through an outflow, the mass flow rate through the domain is "floating" or undefined. In addition, when flow enters the domain through an outflow boundary, the scalar properties of the flow are not defined. For example, the temperature of the flow pulled in through the outflow is not defined. (Ansys Fluent chooses the temperature using the temperature of the fluid adjacent to the outflow, inside the domain.) Therefore you should view all calculations that involve flow entering the domain through an outflow boundary with skepticism. For such calculations, pressure outlet boundary conditions (see Pressure Outlet Boundary Conditions (p. 1434)) are recommended.

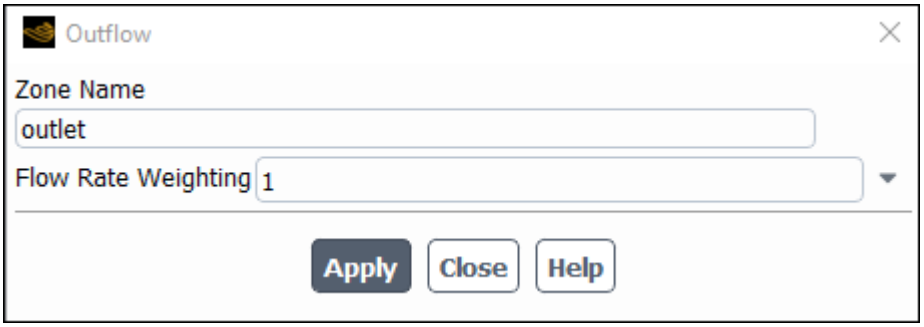
Important:

Note that convergence may be affected if there is recirculation through the outflow boundary at any point during the calculation, even if the final solution is not expected to have any flow reentering the domain. This is particularly true of turbulent flow simulations.

7.4.11.3. Mass Flow Split Boundary Conditions

In Ansys Fluent, it is possible to use multiple outflow boundaries and specify the fractional flow rate through each boundary. In the Outflow Dialog Box (p. 5033), set the **Flow Rate Weighting** to indicate what portion of the outflow is through the boundary.

Figure 7.47: The Outflow Dialog Box



The **Flow Rate Weighting** is a weighting factor:

$$\text{percentage flow through boundary} = \frac{\text{Flow Rate Weighting specified on boundary}}{\text{sum of all Flow Rate Weightings}} \tag{7.112}$$

By default, the **Flow Rate Weighting** for all outflow boundaries is set to 1. If the flow is divided equally among all of your outflow boundaries (or if you have just one outflow boundary), you need not change the settings from the default; Ansys Fluent will scale the flow rate fractions to obtain equal fractions through all outflow boundaries. Therefore, if you have two outflow boundaries and you want half of the flow to exit through each one, no inputs are required from you. If, however,

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

you want 75% of the flow to exit through one, and 25% through the other, you will need to explicitly specify *both* **Flow Rate Weighting** values, that is, 0.75 for one boundary and 0.25 for the other.

Important:

If you specify a **Flow Rate Weighting** of 0.75 at the first exit and leave the default **Flow Rate Weighting** (1.0) at the second exit, then the flow through each boundary will be

$$\text{Boundary 1} = \frac{0.75}{0.75+1.0} = 0.429 \text{ or } 42.9\%$$

$$\text{Boundary 2} = \frac{1.0}{0.75+1.0} = 0.571 \text{ or } 57.1\%$$

7.4.11.4. Other Inputs at Outflow Boundaries

7.4.11.4.1. Radiation Inputs at Outflow Boundaries

In general, there are no boundary conditions for you to set at an outflow boundary. If, however, you are using the P-1, DTRM, DO, surface-to-surface, or MC models, you will set the **Internal Emissivity** and (optionally) **External Black Body Temperature Method** in the **Outflow** dialog box. These parameters are described in Defining Boundary Conditions for Radiation (p. 2217). The default value for **Internal Emissivity** is 1 and the default value for **Black Body Temperature** is 300.

7.4.11.4.2. Defining Discrete Phase Boundary Conditions

If you are modeling a discrete phase of particles, you can set the fate of particle trajectories at the outflow boundary. See Setting Boundary Conditions for the Discrete Phase (p. 2779) for details.

7.4.12. Outlet Vent Boundary Conditions

Outlet vent boundary conditions are used to model an outlet vent with a specified loss coefficient and ambient (discharge) pressure and temperature.

7.4.12.1. Inputs at Outlet Vent Boundaries

You will enter the following information for an outlet vent boundary:

- static pressure
- backflow conditions
  - total (stagnation) temperature (for energy calculations)
  - backflow pressure specification
  - backflow direction specification method
  - turbulence parameters (for turbulent calculations)
  - chemical species mass or mole fractions (for species calculations)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

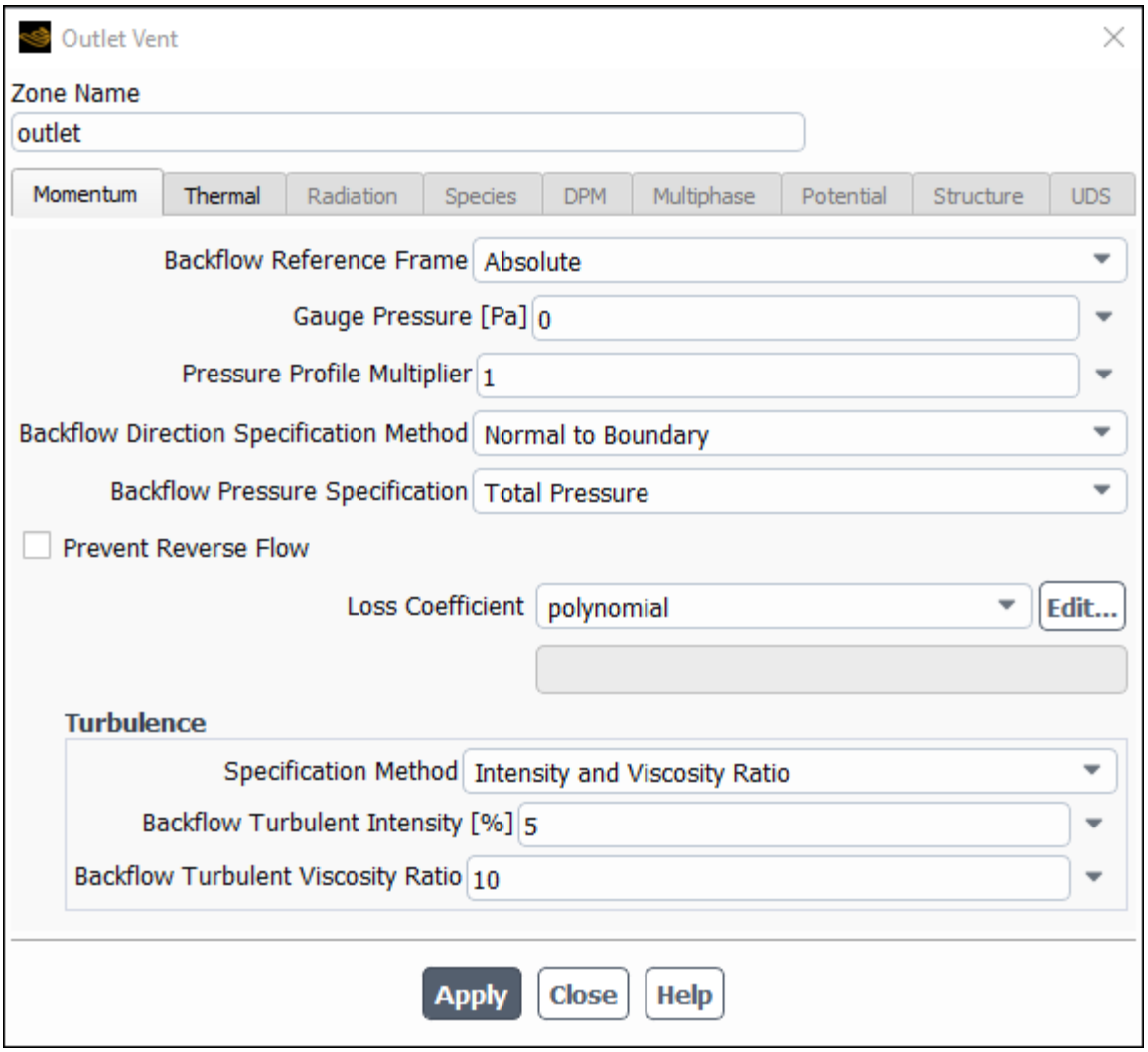
  - mixture fraction and variance (for non-premixed or partially premixed combustion calculations)
  - progress variable (for premixed or partially premixed combustion calculations)
  - multiphase boundary conditions (for general multiphase calculations)
  - type of backflow reference frame (not available for multiphase flows)
- radiation parameters (for calculations using the P-1, DTRM, DO, surface-to-surface, or MC models)
- discrete phase boundary conditions (for discrete phase calculations)
- loss coefficient
- open channel flow parameters (for open channel flow calculations using the VOF multiphase model)

All values are entered in the Outlet Vent Dialog Box (p. 5035) (Figure 7.48: The Outlet Vent Dialog Box (p. 1453)), which is opened from the **Boundary Conditions** task page (as described in Setting Cell Zone and Boundary Conditions (p. 1303)).

The first 4 items listed above are specified in the same way that they are specified at pressure outlet boundaries. See Inputs at Pressure Outlet Boundaries (p. 1434) for details. Specification of the loss coefficient is described here. Open channel boundary condition inputs are described in Modeling Open Channel Flows (p. 2979).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.48: The Outlet Vent Dialog Box



7.4.12.1.1. Specifying the Loss Coefficient

An outlet vent is considered to be infinitely thin, and the pressure drop through the vent is assumed to be proportional to the dynamic head of the fluid, with an empirically determined loss coefficient that you supply. That is, the pressure drop, Δp, varies with the normal component of velocity through the vent, v, as follows:

$$\Delta p = k_L \frac{1}{2} \rho v^2 \tag{7.113}$$

where ρ is the fluid density, and k_L is the nondimensional loss coefficient.

Important:

Δp is the pressure drop in the direction of the flow; therefore the vent will appear as a resistance even in the case of backflow.

You can define a **constant**, **polynomial**, **piecewise-linear**, or **piecewise-polynomial** function for the **Loss Coefficient** across the vent. The dialog boxes for defining these functions are the

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

same as those used for defining temperature-dependent properties. See Defining Properties Using Temperature-Dependent Functions (p. 1616) for details.

7.4.13. Exhaust Fan Boundary Conditions

Exhaust fan boundary conditions are used to model an external exhaust fan with a specified pressure jump and ambient (discharge) pressure.

Note:

The exhaust fan boundary condition considers only momentum. Diffusion processes are not considered.

7.4.13.1. Inputs at Exhaust Fan Boundaries

You will enter the following information for an exhaust fan boundary:

- static pressure
- backflow conditions
  - total (stagnation) temperature (for energy calculations)
  - backflow pressure specification
  - backflow direction specification method
  - turbulence parameters (for turbulent calculations)
  - chemical species mass or mole fractions (for species calculations)
  - mixture fraction and variance (for non-premixed or partially premixed combustion calculations)
  - progress variable (for premixed or partially premixed combustion calculations)
  - multiphase boundary conditions (for general multiphase calculations)
  - type of backflow reference frame (not available for multiphase flows)
  - user-defined scalar boundary conditions (for user-defined scalar calculations)
- radiation parameters (for calculations using the P-1, DTRM, DO, surface-to-surface, or MC models)
- discrete phase boundary conditions (for discrete phase calculations)
- pressure jump
- open channel flow parameters (for open channel flow calculations using the VOF multiphase model)

All values are entered in the Exhaust Fan Dialog Box (p. 4995) (Figure 7.49: The Exhaust Fan Dialog Box (p. 1456)), which is opened from the **Boundary Conditions** task page (as described in Setting Cell Zone and Boundary Conditions (p. 1303)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

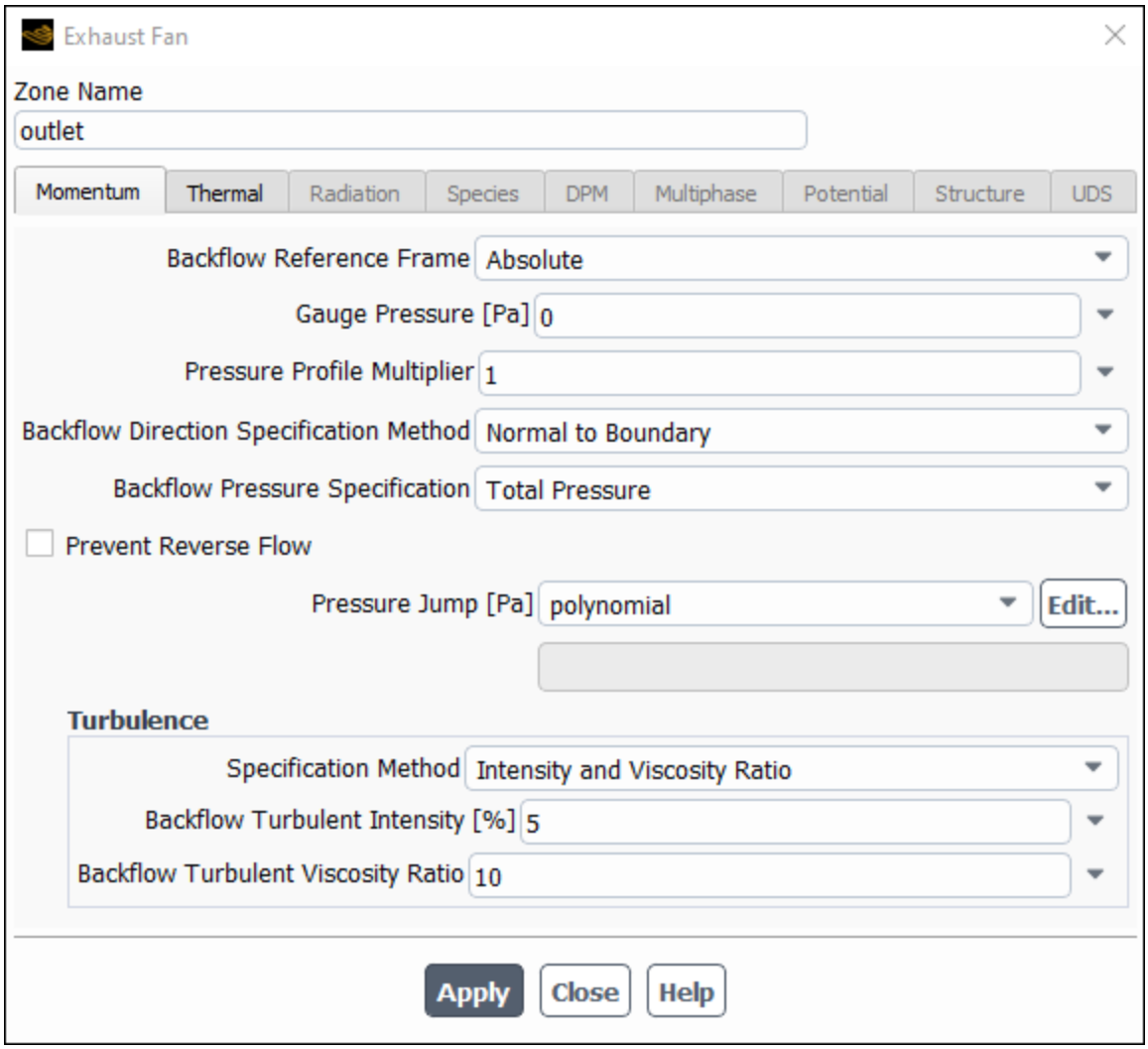
The first 4 items listed above are specified in the same way that they are specified at pressure outlet boundaries. See Inputs at Pressure Outlet Boundaries (p. 1434) for details. Specification of the pressure jump is described here. Open channel boundary condition inputs are described in Modeling Open Channel Flows (p. 2979).

7.4.13.1.1. Specifying the Pressure Jump

An exhaust fan is considered to be infinitely thin, and the discontinuous pressure rise across it is specified as a function of the local fluid velocity normal to the fan. You can define a **constant**, **polynomial**, **piecewise-linear**, or **piecewise-polynomial** function for the **Pressure Jump** across the fan. The dialog boxes for defining these functions are the same as those used for defining temperature-dependent properties. See Defining Properties Using Temperature-Dependent Functions (p. 1616) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.49: The Exhaust Fan Dialog Box



Important:

You must be careful to model the exhaust fan so that a pressure rise occurs for forward flow through the fan. In the case of reversed flow, the fan is treated like an inlet vent with a loss coefficient of unity.

7.4.14. Degassing Boundary Conditions

Degassing boundary conditions are used to model a free surface through which dispersed gas bubbles are allowed to escape, but the continuous phase is not. A typical application is a bubble column in which you want to reduce computational cost by not including the freeboard region in the simulation.

When the degassing boundary condition is specified for an outlet, the continuous liquid phase sees the boundary as a free-slip wall and does not leave the domain. The dispersed gas phase sees the boundary as an outlet. The outlet pressure is not specified. Instead, Ansys Fluent automatically specifies

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a mass sink for the dispersed gas phase in the cells adjacent to the degassing outlet. The mass sink is calculated using the flux normal to the boundary at the cell center.

Note:

The degassing boundary condition is only available for two-phase liquid-gas flows using the Eulerian multiphase model.

7.4.14.1. Limitations

The following limitations apply to the degassing boundary condition in Ansys Fluent:

- The degassing boundary condition is only available for liquid-gas two-phase flow using the Eulerian model. The primary phase must be liquid.
- In order for the gas to escape from the degassing boundary, gravity must be switched on in the model.
- The degassing boundary condition is only recommended for modeling situations like bubble columns without the freeboard region.
- When postprocessing on a degassing boundary, there is no normal velocity for either phase since the gas escape is modeled by a mass sink in the neighboring cells.

7.4.14.2. Inputs at Degassing Boundaries

No inputs are necessary for the degassing boundary condition. However, the initial condition for volume fraction must be set appropriately. It is recommended that the gas phase volume fraction be initialized with a nonzero value smaller than the steady-state gas holdup value.

7.4.15. Wall Boundary Conditions

Wall boundary conditions are used to bound fluid and solid regions. In viscous flows, the no-slip boundary condition is enforced at walls by default, but you can specify a tangential velocity component in terms of the translational or rotational motion of the wall boundary, or model a "slip" wall by specifying shear. (You can also model a slip wall with zero shear using the symmetry boundary type, but using a symmetry boundary will apply symmetry conditions for *all* equations. See Symmetry Boundary Conditions (p. 1499) for details.)

The shear stress and heat transfer between the fluid and wall are computed based on the flow details in the local flow field.

The following topics are discussed in this section:

7.4.15.1. Inputs at Wall Boundaries
7.4.15.2. Stationary Wall
7.4.15.3. Wall Roughness Effects in Turbulent Wall-Bounded Flows
7.4.15.4. Wall Motion
7.4.15.5. Thermal Boundary Conditions at Walls
7.4.15.6. Species Boundary Conditions for Walls

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.15.7. Radiation Boundary Conditions for Walls
7.4.15.8. Discrete Phase Model (DPM) Boundary Conditions for Walls
7.4.15.9. User-Defined Scalar (UDS) Boundary Conditions for Walls
7.4.15.10. Wall Film Conditions for Walls
7.4.15.11. Structural Model Conditions for Walls
7.4.15.12. Default Settings at Wall Boundaries
7.4.15.13. Shear-Stress Calculation Procedure at Wall Boundaries
7.4.15.14. Heat Transfer Calculations at Wall Boundaries

7.4.15.1. Inputs at Wall Boundaries

7.4.15.1.1. Summary

You will enter the following information for a wall boundary:

- wall roughness (for turbulent flows, optional)
- wall motion conditions (for moving or rotating walls)
- shear conditions (for slip walls, optional)
- thermal boundary conditions (for heat transfer calculations)
- species boundary conditions (for species calculations)
- chemical reaction boundary conditions (for surface reactions)
- radiation boundary conditions (for calculations using the P-1, DTRM, DO, surface-to-surface, or MC models)
- discrete phase boundary conditions (for discrete phase calculations)
- wall adhesion contact angle (for VOF calculations, optional)
- displacement boundary conditions for intrinsic fluid-structure interaction (FSI) simulations

7.4.15.2. Stationary Wall

For a stationary wall, choose the **Stationary Wall** option under **Wall Motion**.

7.4.15.3. Wall Roughness Effects in Turbulent Wall-Bounded Flows

Fluids flows over rough surfaces are encountered in diverse situations. Examples are, among many others, flows over the surfaces of airplanes (especially due to ice accretion), ships, turbomachinery, heat exchangers, piping systems, and atmospheric boundary layers over terrain of varying roughness. Wall roughness affects drag (resistance) as well as heat and mass transfer on the walls.

If you are modeling turbulent wall-bounded flow in which the wall roughness effects are considered to be significant, you can include the wall roughness effects through the Standard Law-of-the-Wall Modified for Roughness (for details, see Wall Roughness Effects in Turbulent Wall-Bounded Flows

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

in the *Fluent Theory Guide*) or you can use one of the Additional Roughness Models for Icing Simulations (p. 1460) (for Spalart-Allmaras or the SST k-ω models only).

Note:

Rough walls cannot be used together with the following model combinations:

- An -equation model with enhanced wall treatment or the Menter-Lechner near-wall treatment.

  Note that the following are the relevant -equations:

  – all of the k- models (that is, standard, RNG, and realizable).

  – the Reynolds stress model with Linear Pressure-Strain model selected.

  – the detached eddy simulation (DES) with the Realizable k-epsilon option selected.

- the Reynolds stress model with the Stress-Omega or Stress-BSL model selected.
- the transition k-kl-ω model.
- the Large Eddy Simulation (LES) model with Kader Blending Wall Functions and Werner Wengle Wall Treatment.

7.4.15.3.1. Setting the Roughness Parameters

The roughness parameters are in the **Momentum** tab of the Wall Dialog Box (p. 5085) (see Figure 7.54: The Wall Dialog Box for Marangoni Stress (p. 1467)), which is opened from the Boundary Conditions Task Page (p. 4991) (as described in Setting Cell Zone and Boundary Conditions (p. 1303)).

To model the wall roughness effects, you must specify two roughness parameters: the **Roughness Height**, K_s, and the **Roughness Constant**, C_s. The default roughness height (K_s) is zero, which corresponds to smooth walls. For the roughness to take effect, you must specify a nonzero value for K_s. For a uniform sand-grain roughness, the height of the sand-grain can simply be taken for K_s. For a non-uniform sand-grain, however, the mean diameter (D_{50}) would be a more meaningful roughness height. For other types of roughness, an "equivalent" sand-grain roughness height could be used for K_s. The above approaches are only relevant if the height is considered constant per surface. However, if the roughness constant or roughness height is not constant, then you can specify a profile (see Profiles (p. 1564)). Similarly, user-defined functions may be used to define a wall roughness height that is not constant. For details on the format of user-defined functions, refer to the Fluent Customization Manual.

Choosing a proper roughness constant (C_s) is dictated mainly by the type of the given roughness. The default roughness constant (C_s=0.5) was determined so that, when used with k - ε turbulence models, it reproduces Nikuradse's resistance data for pipes roughened with tightly-packed, uniform sand-grain roughness. You may need to adjust the roughness constant when the roughness you want to model departs much from uniform sand-grain. For instance, there is some experimental evidence that, for non-uniform sand-grains, ribs, and wire-mesh roughness, a higher value (C_s=0.5~1.0) is more appropriate. Unfortunately, a clear guideline for choosing C_s for arbitrary types of roughness is not available.

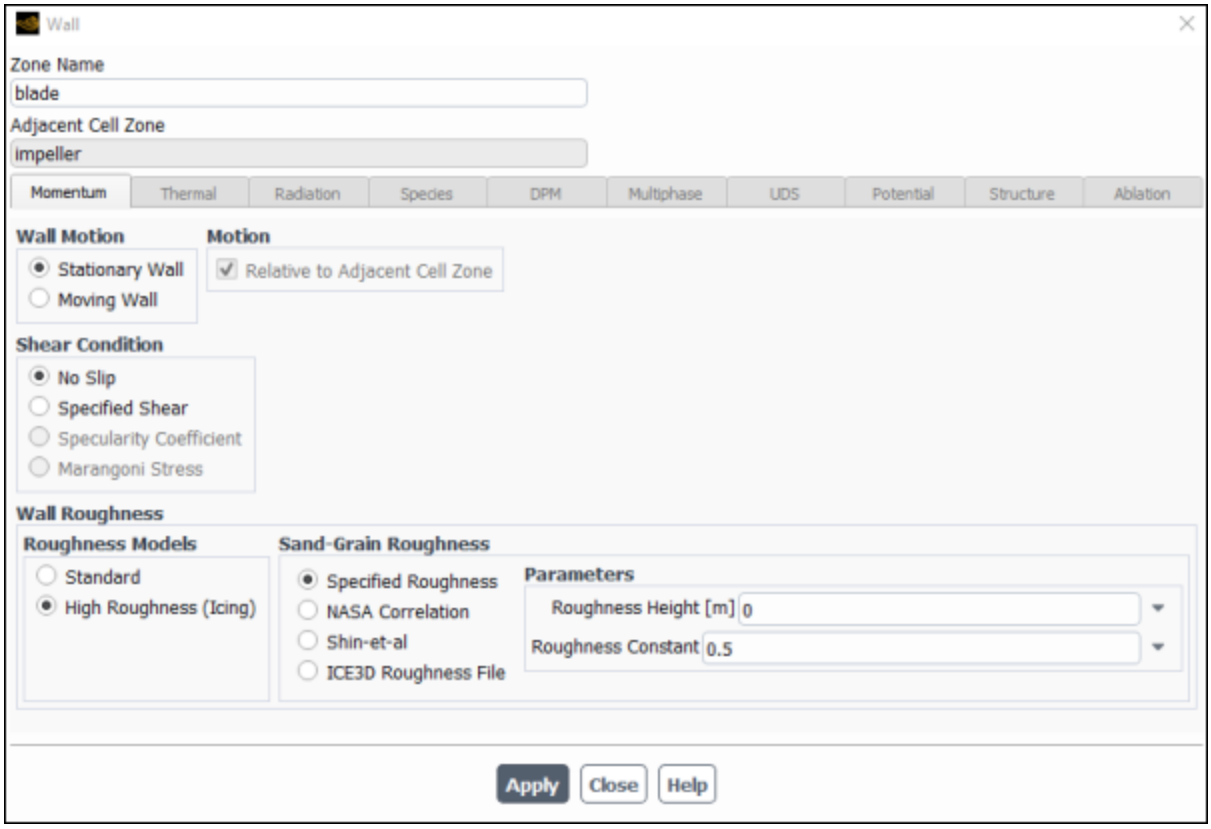
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.15.3.2. Additional Roughness Models for Icing Simulations

Additional roughness models are available if you have selected the **Spalart-Allmaras** or the **SST** k-ω model in the **Viscous Model** dialog box. See the Steps in Using a Turbulence Model (p. 2081) for more information. With one of these viscous models active, the additional roughness models will be available on the **Wall** boundary condition panel on the **Momentum** tab under **Wall Roughness** → **Roughness Models** → **High Roughness (Icing)**. A discussion of the treatment of the turbulence models with the **High Roughness (Icing)** enabled is given in Treatment of the Spalart-Allmaras Model for Icing Simulations and Treatment of the SST Model for Icing Simulations. These models are primarily designed and tested for simulations of icing applications, however they may also be useful for other applications in which the boundary layer is fully resolved and there is surface roughness that is large relative to the near-wall mesh.

Note that the **High Roughness (Icing)** models are only valid for low-Re number turbulence, or need fine near-wall meshes (mesh should fully resolve the boundary layer).

Figure 7.50: The Wall Dialog Box for High Roughness (Icing) Models



You have the following options:

- Specified Roughness
- NASA Correlation
- Shin-et-al
- ICE3D Roughness File

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.15.3.2.1. Specified Roughness

A **Specified Roughness** can be applied in the same manner as described in the section on the **Standard Roughness** model. See Setting the Roughness Parameters (p. 1459) for more details.

7.4.15.3.2.2. NASA Correlation

If the **NASA Correlation** option is selected, the surface sand-grain roughness height is computed with an empirical NASA correlation for icing in air flows [141] (p. 5721). The sand-grain roughness height is computed from the product of the following coefficients:

$$\left[\frac{h_s/c}{\left(h_s/c\right)_{base}}\right]_{V_\infty} = 0.4286 + 0.0044139 V_\infty \tag{7.114}$$

$$\left[\frac{h_s/c}{\left(h_s/c\right)_{base}}\right]_{LC} = 0.5714 + 0.2457\,(LC) + 1.2571\,(LC)^2 \tag{7.115}$$

$$\left[\frac{h_s/c}{\left(h_s/c\right)_{base}}\right]_{T_s} = 0.04\,T_s - 11.27 \tag{7.116}$$

Where V_∞ is the **Free Stream Velocity**, T_s is the **Free Stream Temperature**, LC is the **Liquid Content** (in most cases water for ice accretion), c is the **Characteristic Length**, and $\left(h_s/c\right)_{base} = 0.001177$.

The sand-grain roughness height is then obtained from the formulation:

$$h_s = \left[\frac{h_s/c}{\left(h_s/c\right)_{base}}\right]_{T_s}\left[\frac{h_s/c}{\left(h_s/c\right)_{base}}\right]_{LC}\left[\frac{h_s/c}{\left(h_s/c\right)_{base}}\right]_{V_\infty} \cdot \left(h_s/c\right)_{base} \cdot c \tag{7.117}$$

7.4.15.3.2.3. Shin-et-al

If the **Shin-et-al** option is selected the empirical correlation for the surface sand-grain roughness is computed with the Shin and Bond formulation [141] (p. 5721), which modifies the NASA correlation (see the NASA Correlation (p. 1461) for a description of variables) with the following factor:

$$\left[\frac{h_s/c}{\left(h_s/c\right)_{base}}\right]_{MVD} = \begin{cases} 1 & MVD \le 20 \\ 1.667 - 0.0333(MVD) & MVD > 20 \end{cases} \tag{7.118}$$

where MVD is the droplet mean diameter (**Droplet Diameter**). The corresponding value of the sand-grain roughness height is obtained from:

$$h_s = 0.6839\left[\frac{h_s/c}{\left(h_s/c\right)_{base}}\right]_{LC}\left[\frac{h_s/c}{\left(h_s/c\right)_{base}}\right]_{T_s}\left[\frac{h_s/c}{\left(h_s/c\right)_{base}}\right]_{MVD} \cdot \left(h_s/c\right)_{base} \cdot c \tag{7.119}$$

7.4.15.3.2.4. ICE3D Roughness File

If the **ICE3D Roughness File** is selected, the sand-grain roughness height is read from a node-based input file obtained from FENSAP-ICE. The file must have the format $\left\{x_i, y_i, z_i, h_{s,i}\right\}$, and

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

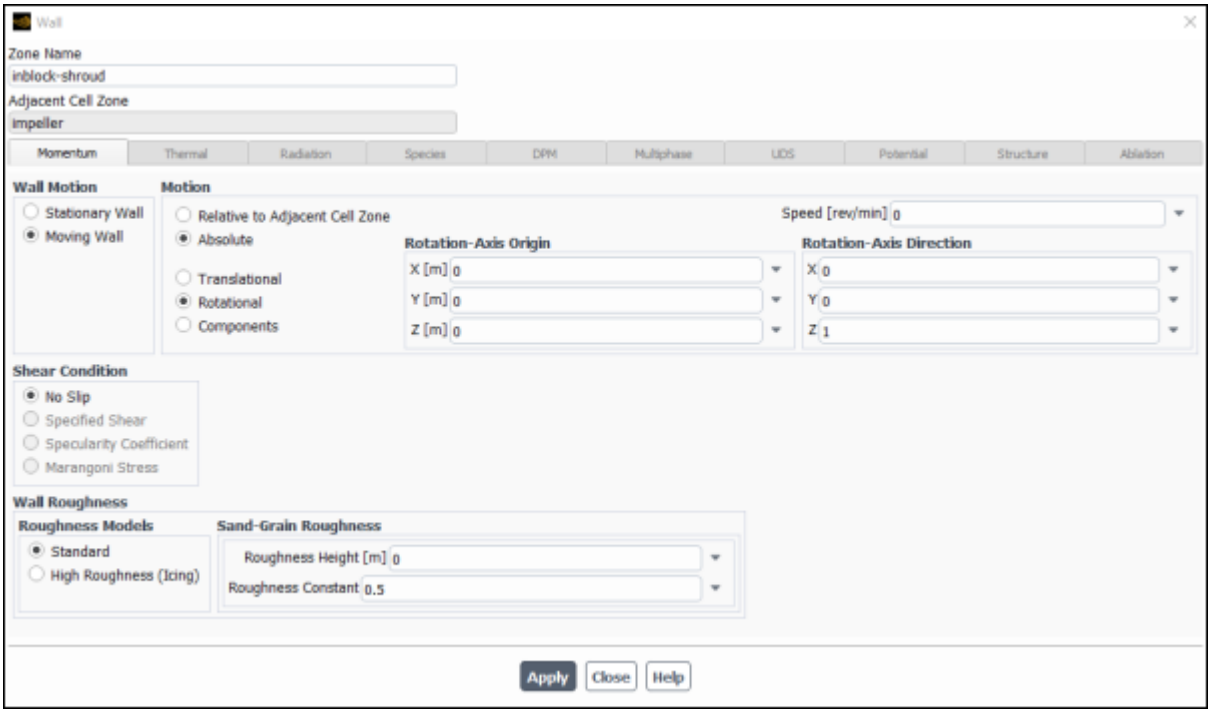
be read in Fluent via **File** → **Read** → **Profile**. After the FENSAP-ICE roughness file is read into Fluent, the file name will appear in the dropdown list.

7.4.15.4. Wall Motion

Wall boundaries can be either stationary or moving. The stationary boundary condition specifies a fixed wall, whereas the moving boundary condition can be used to specify the translational or rotational velocity of the wall, or the velocity components.

Wall motion conditions are entered in the **Momentum** tab of the Wall Dialog Box (p. 5085) (Figure 7.51: The Wall Dialog Box for a Moving Wall (p. 1462)), which is opened from the Boundary Conditions Task Page (p. 4991) (as described in Setting Cell Zone and Boundary Conditions (p. 1303)). To view the wall motion conditions, click the **Momentum** tab.

Figure 7.51: The Wall Dialog Box for a Moving Wall



7.4.15.4.1. Velocity Conditions for Moving Walls

If you want to include tangential motion of the wall in your calculation, you need to define the translational or rotational velocity, or the velocity components. Select the **Moving Wall** option under **Wall Motion**. The **Wall** dialog box will expand, as shown in Figure 7.51: The Wall Dialog Box for a Moving Wall (p. 1462), to show the wall velocity conditions.

Note that you cannot use the moving wall condition to model problems where the wall motion with respect to the adjacent cell zone has a component that is normal to the wall itself. For such problems, consider using a Sliding or Dynamic Mesh approach as discussed in Modeling Flows Using Sliding and Dynamic Meshes (p. 1797). Ansys Fluent will neglect any normal component of wall motion that you specify using the methods below.

- **Specifying Relative or Absolute Velocity**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the cell zone adjacent to the wall is moving (for example, if you are using a moving reference frame or a sliding mesh), you can choose to specify velocities relative to the zone motion by enabling the **Relative to Adjacent Cell Zone** option. If you choose to specify relative velocities, a velocity of zero means that the wall is stationary in the relative frame, and therefore moving at the speed of the adjacent cell zone in the absolute frame. If you choose to specify absolute velocities (by enabling the **Absolute** option), a velocity of zero means that the wall is stationary in the absolute frame, and therefore moving at the speed of the adjacent cell zone—but in the opposite direction—in the relative reference frame.

Important:

If you are using one or more moving reference frames, sliding meshes, or mixing planes, and you want the wall to be fixed in the moving frame, it is recommended that you specify relative velocities (the default) rather than absolute velocities. Then, if you modify the speed of the adjacent cell zone, you will not need to make any changes to the wall velocities, as you would if you specified absolute velocities.

Note that if the adjacent cell zone is not moving, the absolute and relative options are equivalent.

- **Translational Wall Motion**

  For problems that include linear translational motion of the wall boundary (for example, a rectangular duct with a moving belt as one wall) you can enable the **Translational** option and specify the wall's **Speed** and **Direction** (**X,Y,Z** vector). By default, wall motion is "disabled" by the specification of **Translational** velocity with a **Speed** of zero. You can define time-varying translational speed using a transient profile (Standard Transient Profiles (p. 1524)) or user-defined function for the speed of the wall.

  If you need to define nonlinear translational motion, you will need to use the **Components** option, described below.

- **Rotational Wall Motion**

  For problems that include rotational wall motion you can enable the **Rotational** option and define the rotational **Speed** about a specified axis. To define the axis, set the **Rotation-Axis Direction** and **Rotation-Axis Origin**. This axis is independent of the axis of rotation used by the adjacent cell zone, and independent of any other wall rotation axis. For 3D problems, the axis of rotation is the vector passing through the specified **Rotation-Axis Origin** and parallel to the vector from (0,0,0) to the (**X,Y,Z**) point specified under **Rotation-Axis Direction**. For 2D problems, you will specify only the **Rotation-Axis Origin**; the axis of rotation is the z -direction vector passing through the specified point. For 2D axisymmetric problems, you will not define the axis: the rotation will always be about the x axis, with the origin at (0,0). You can define time-varying rotational speed using a transient profile (Standard Transient Profiles (p. 1524)) profile or a user-defined function for the speed of the wall.

  Note that the modeling of tangential rotational motion will be correct only if the wall bounds a surface of revolution about the prescribed axis of rotation (for example, a circle or cylinder). Note also that rotational motion can be specified for a wall in a stationary reference frame.

- **Wall Motion Based on Velocity Components**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For problems that include linear or nonlinear translational motion of the wall boundary you can enable the **Components** option and specify the **X-Velocity**, **Y-Velocity**, and **Z-Velocity** of the wall. You can define nonlinear translational motion using a profile or a user-defined function for the **X-Velocity**, **Y-Velocity**, and/or **Z-Velocity** of the wall.

- **Wall Motion for Two-Sided Walls**

  As discussed earlier in this section, when you read a mesh with a two-sided wall zone (which forms the interface between fluid/solid regions) into Ansys Fluent, a "shadow" zone will automatically be created so that each side of the wall is a distinct wall zone. For two-sided walls, it is possible to specify different motions for the wall and shadow zones, whether or not they are coupled. Note, however, that you cannot specify motion for a wall (or shadow) that is adjacent to a solid zone.

7.4.15.4.2. Shear Conditions at Walls

Four types of shear conditions are available:

- no-slip
- specified shear
- specularity coefficient
- Marangoni stress

The no-slip condition is the default, and it indicates that the fluid sticks to the wall and moves with the same velocity as the wall, if it is moving. The specified shear and Marangoni stress boundary conditions are useful in modeling situations in which the shear stress (rather than the motion of the fluid) is known. Examples of such situations are applied shear stress, slip wall (zero shear stress), and free surface conditions (zero shear stress or shear stress dependent on surface tension gradient). The specified shear boundary condition allows you to specify the x, y, and z components of the shear stress as constant values or profiles. The Marangoni stress boundary condition allows you to specify the gradient of the surface tension with respect to the temperature at this surface. The shear stress is calculated based on the surface gradient of the temperature and the specified surface tension gradient. The Marangoni stress option is available only for calculations in which the energy equation is being solved.

The specularity coefficient shear condition is specifically used in multiphase with granular flows. The specularity coefficient is a measure of the fraction of collisions that transfer momentum to the wall and its value ranges between zero and unity. This implementation is based on the Johnson and Jackson [74] (p. 5717) boundary conditions for granular flows.

Shear conditions are entered in the **Momentum** tab of the Wall Dialog Box (p. 5085), which is opened from the Boundary Conditions Task Page (p. 4991) (as described in Setting Cell Zone and Boundary Conditions (p. 1303)).

7.4.15.4.3. No-Slip Walls

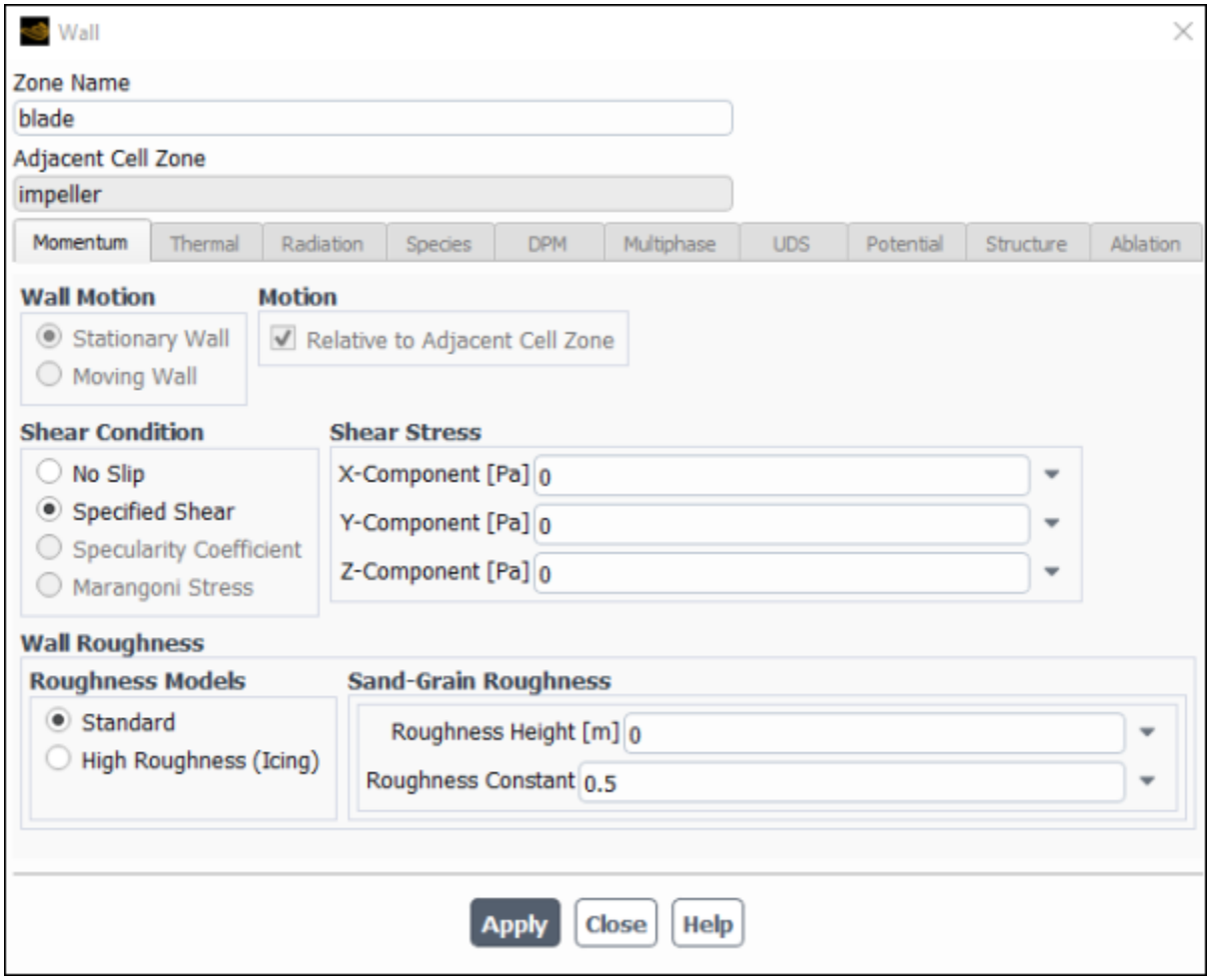
You can model a no-slip wall by selecting the **No Slip** option under **Shear Condition**. This is the default for all walls in viscous flows.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.15.4.4. Specified Shear

In addition to the no-slip wall that is the default for viscous flows, you can model a slip wall by specifying zero or nonzero shear. For nonzero shear, the shear to be specified is the shear at the wall by the fluid. To specify the shear, select the **Specified Shear** option under **Shear Condition** (see Figure 7.52: The Wall Dialog Box for Specified Shear (p. 1465)). You can then enter x, y, and z components of shear under **Shear Stress**.

Figure 7.52: The Wall Dialog Box for Specified Shear



7.4.15.4.5. Specularity Coefficient

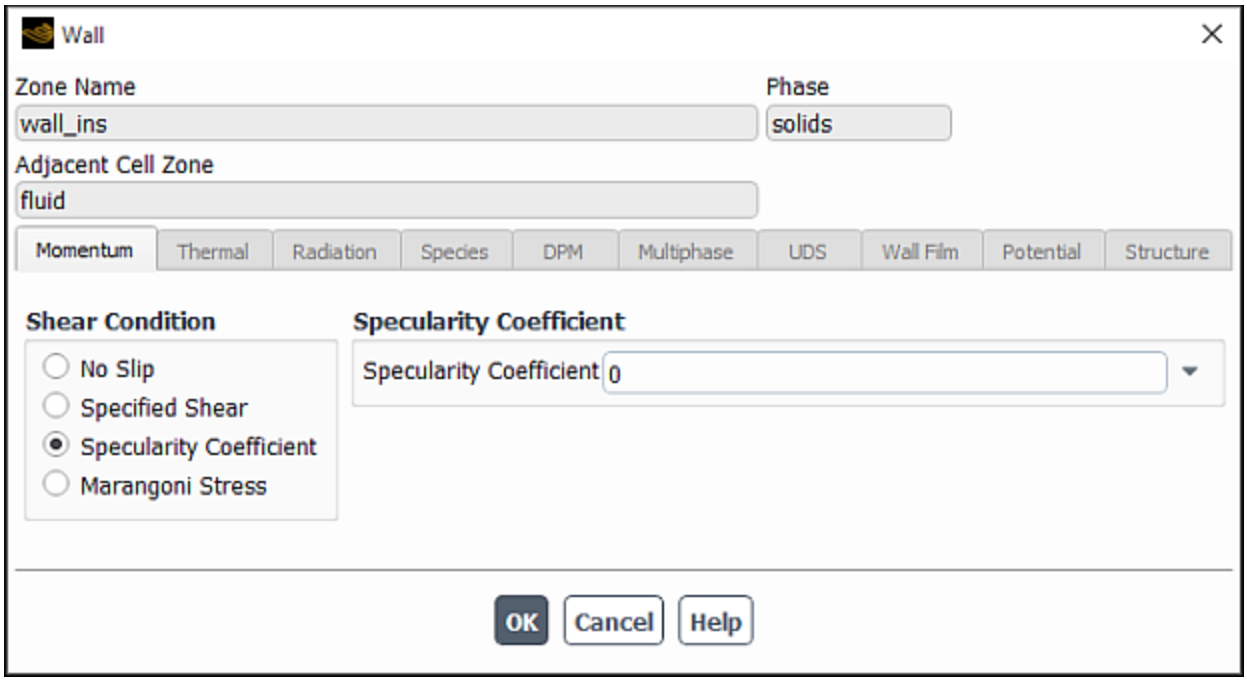
For multiphase granular flow, you can specify the specularity coefficient such that when the value is zero, this condition is equivalent to zero shear at the wall, but when the value is near unity, there is a significant amount of lateral momentum transfer. To specify the specularity coefficient, select the **Specularity Coefficient** option under **Shear Condition** (see Figure 7.53: The Wall Dialog Box for the Specularity Coefficient (p. 1466)) and enter the desired value in the text-entry box under **Specularity Coefficient**.

Note:

The specularity coefficient is not available for moving walls.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.53: The Wall Dialog Box for the Specularity Coefficient



7.4.15.4.6. Marangoni Stress

Ansys Fluent can also model shear stresses caused by the variation of surface tension due to temperature. The shear stress applied at the wall is given by

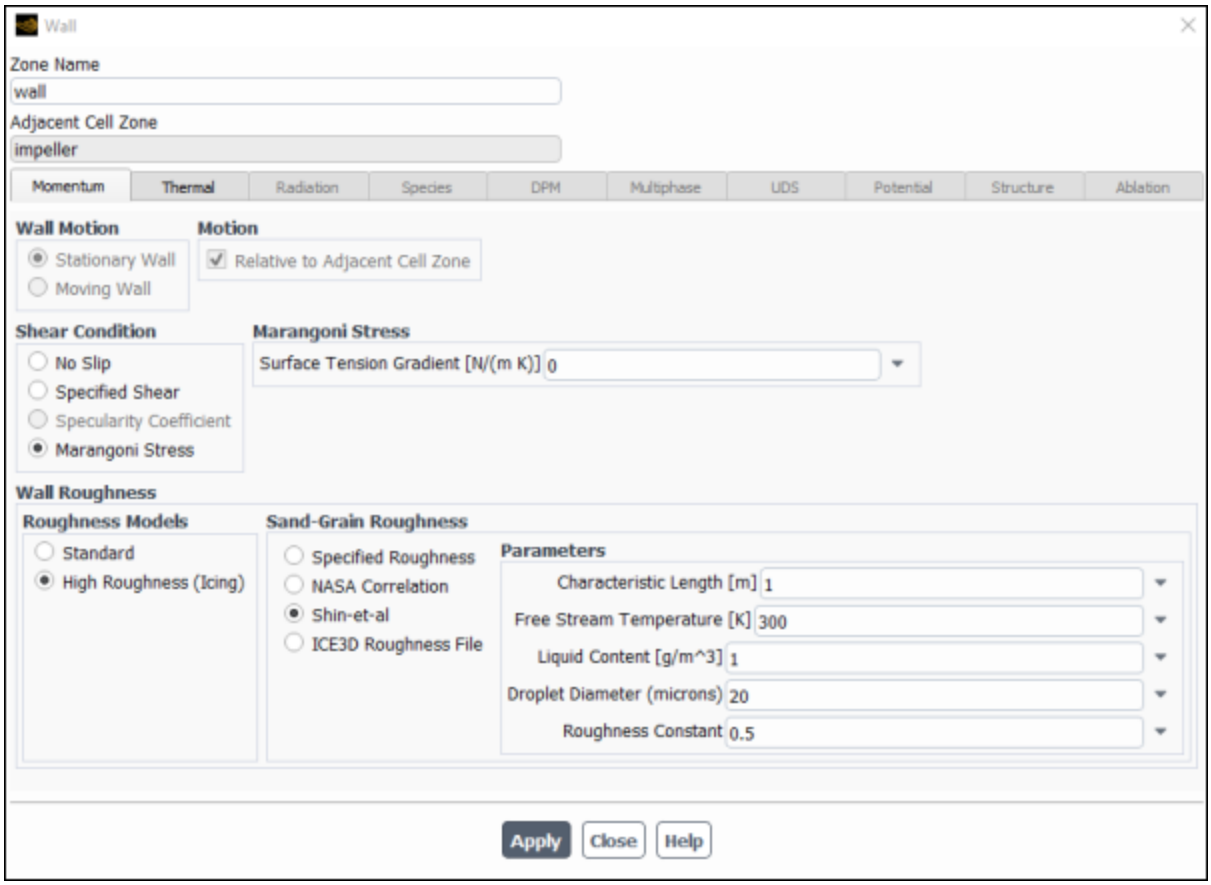
$$\tau = \frac{d\sigma}{dT} \nabla_s T \tag{7.120}$$

where $d\sigma / dT$ is the surface tension gradient with respect to temperature, and $\nabla_s T$ is the surface gradient. This shear stress is then applied to the momentum equation.

To model Marangoni stress for the wall, select the **Marangoni Stress** option under **Shear Condition** (see Figure 7.54: The Wall Dialog Box for Marangoni Stress (p. 1467)). This option is available only for calculations in which the energy equation is being solved. You can then enter the surface tension gradient ($d\sigma / dT$ in Equation 7.120 (p. 1466)) in the **Surface Tension Gradient** field. Wall functions for turbulence are not used with the **Marangoni Stress** option.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.54: The Wall Dialog Box for Marangoni Stress



7.4.15.4.7. Partial Slip for Rarefied Gases

Ansys Fluent offers a partial-slip boundary condition for rarefied gases. In the rarefied regime, the finite amount of friction is no longer sufficient to allow the full adhesion of gas particles onto solid walls, see Partial Slip for Rarefied Gases (p. ?).

7.4.15.5. Thermal Boundary Conditions at Walls

When you are solving the energy equation, you need to define thermal boundary conditions at wall boundaries. Seven types of thermal conditions are available:

- fixed heat flux
- fixed temperature
- convective heat transfer
- external radiation heat transfer
- combined external radiation and convection heat transfer
- thermal data transferred between another system in Workbench using System Coupling
- thermal coupling across a mapped interface

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the wall zone is a "two-sided wall" (a wall that forms the interface between two regions, such as the fluid/solid interface for a conjugate heat transfer problem) a subset of these thermal conditions will be available, but you will also be able to choose whether or not the two sides of the wall are "coupled". See below for details.

The inputs for each type of thermal condition are described below. If the wall has a nonzero-thickness, you should also set parameters for calculating thin-wall thermal resistance and heat generation in the wall, as described below.

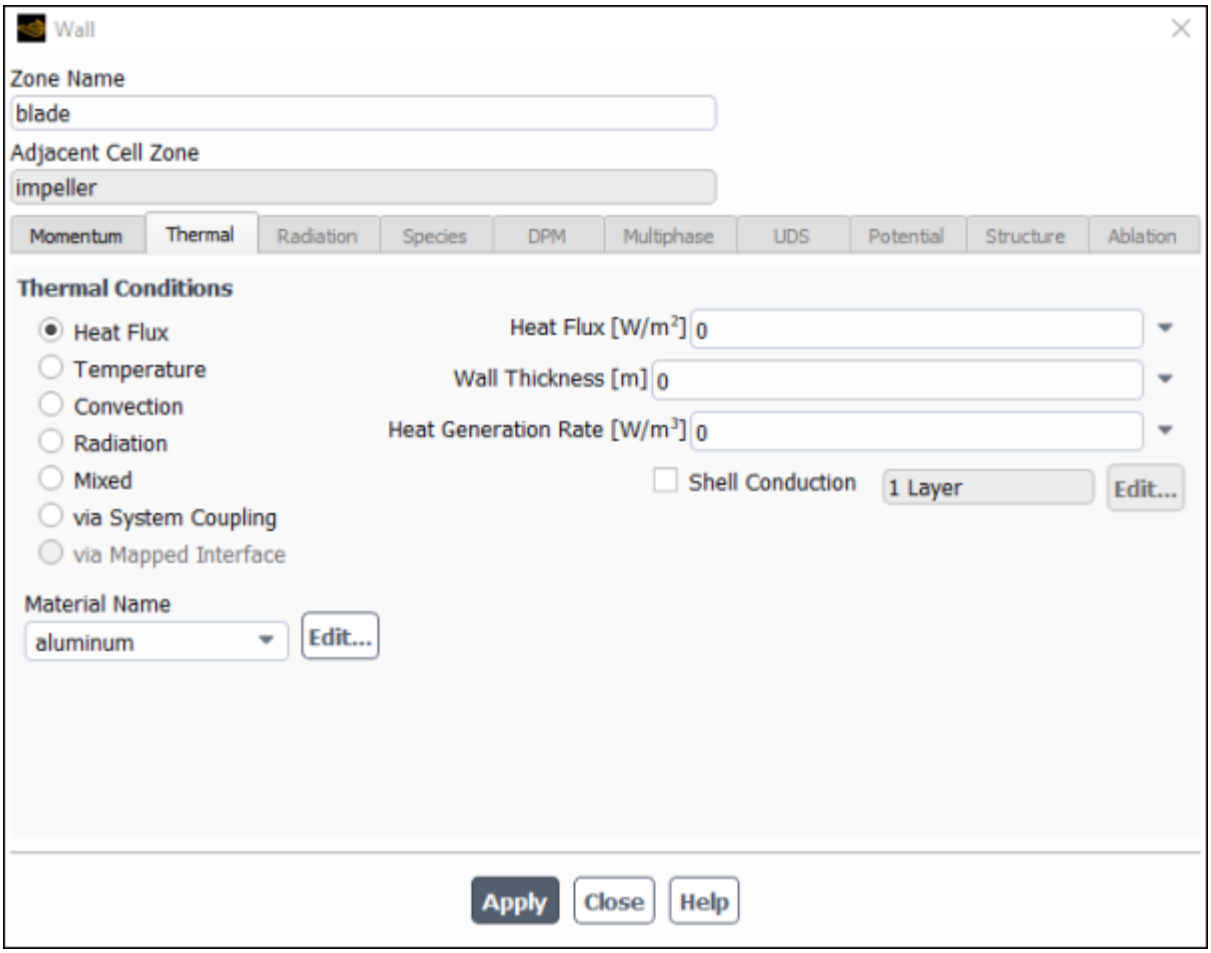
For a boundary wall that is adjacent to a solid cell zone that undergoes solid motion, you can model the advection of heat into or out of the domain as a result of the motion that is normal to the boundary. Details are provided in a section that follows.

You can model conduction within boundary walls and internal (that is, two-sided) walls of your model. This type of conduction, called shell conduction, allows you to more conveniently model heat conduction on walls where the wall thickness is small with respect to the overall geometry (for example, finned heat exchangers or sheet metal in automobile underhoods). Meshing these walls with solid cells would lead to high-aspect-ratio meshes and a significant increase in the total number of cells. See below for details about shell conduction.

Thermal conditions are entered in the **Thermal** tab of the Wall Dialog Box (p. 5085) (Figure 7.55: The Wall Dialog Box (Thermal Tab) (p. 1469)), which is opened from the **Boundary Conditions** task page (as described in Setting Cell Zone and Boundary Conditions (p. 1303)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.55: The Wall Dialog Box (Thermal Tab)



7.4.15.5.1. Heat Flux Boundary Conditions

For a fixed heat flux condition, choose the **Heat Flux** option under **Thermal Conditions**. You will then need to set the appropriate value for the heat flux at the wall surface in the **Heat Flux** field. You can define an adiabatic wall by setting a zero heat flux condition. This is the default condition for all walls.

7.4.15.5.2. Temperature Boundary Conditions

To select the fixed temperature condition, choose the **Temperature** option under **Thermal Conditions** in the **Wall** dialog box. You will need to specify the temperature at the wall surface (**Temperature**). The heat transfer to the wall is computed using Equation 7.123 (p. 1481) or Equation 7.124 (p. 1481).

7.4.15.5.3. Convective Heat Transfer Boundary Conditions

For a convective heat transfer wall boundary, select **Convection** under **Thermal Conditions**. Your inputs of **Heat Transfer Coefficient** and **Free Stream Temperature** will allow Ansys Fluent to compute the heat transfer to the wall using Equation 7.127 (p. 1482).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.15.5.4. External Radiation Boundary Conditions

If radiation heat transfer from the exterior of your model is of interest, you can enable the **Radiation** option in the **Wall** dialog box and set the **External Emissivity** and **External Radiation Temperature**.

7.4.15.5.5. Combined Convection and External Radiation Boundary Conditions

You can choose a thermal condition that combines the convection and radiation boundary conditions by selecting the **Mixed** option. With this thermal condition, you will need to set the **Heat Transfer Coefficient**, **Free Stream Temperature**, **External Emissivity**, and **External Radiation Temperature**.

7.4.15.5.6. Augmented Heat Transfer

When modeling the heat transfer of applications that have perturbed flow and/or disturbed boundary layers, it can be necessary to augment the calculation of the diffusive heat flux with a convective augmentation factor. Such applications include the modeling of underhood and underbody heat loads, as wells as transient heat transfer in fully warmed-up exhaust systems.

The convective augmentation factor represents the ratio of the measured Nusselt number to the Nusselt number of an ideal flow. You can define it by using the following text command:

`define` → `boundary-conditions` → `wall`

You will be prompted to define the `Convective Augmentation Factor` as either a profile or a single value. Note that a value of 1 (the default value) represents no augmentation of the diffusive heat flux, whereas values greater than 1 initiate augmentation. For further details, see the equation for `qid` in `DEFINE_HEAT_FLUX` of the Fluent Customization Manual.

7.4.15.5.7. Thin-Wall Thermal Resistance Parameters

By default, a wall will have a thickness of zero. You can, however, in conjunction with any of the thermal conditions, model a thin layer of material on the wall. For example, you can model the effect of a piece of sheet metal between two fluid zones, a coating on a solid zone, or contact resistance between two solid regions. Ansys Fluent will solve a 1D steady heat conduction equation to compute the thermal resistance offered by the wall and the heat generation in the wall.

To include these effects in the heat transfer calculation you will need to specify the type of material, the thickness of the wall, and the heat generation rate in the wall:

1. Select the material type in the **Material Name** drop-down list, and

   If you want to check or modify the properties of the selected material, you can click **Edit...** to open the **Edit Material** dialog box; this dialog box contains just the properties of the selected material, not the full contents of the standard **Create/Edit Materials** dialog box.

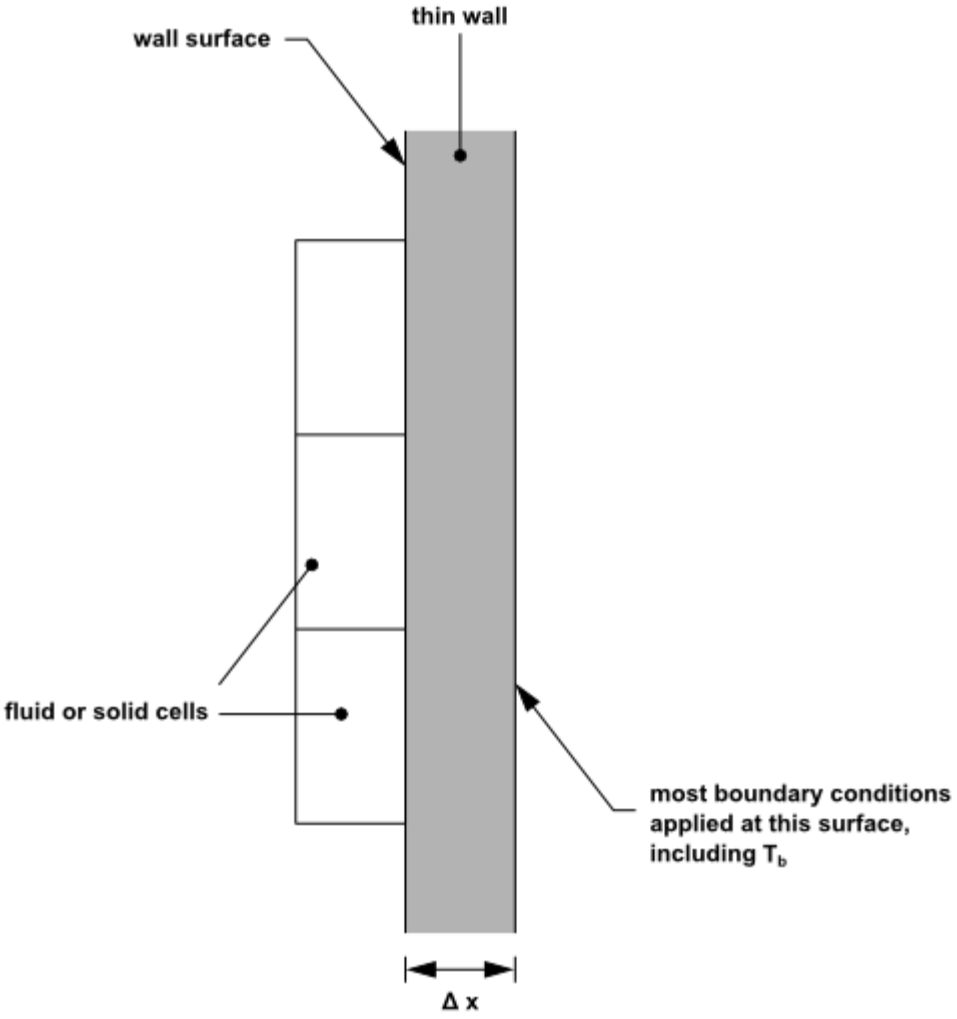
2. Specify the thickness in the **Wall Thickness** field.

   You can specify a constant thickness or you can specify a variable thickness using an expression, UDF, or profile.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When you specify a thickness, the wall is then treated as a coupled wall, where the surface that is adjacent to the fluid / solid cells is referred to as the "wall surface". See Figure 7.56: A Thin Wall (p. 1471).

Figure 7.56: A Thin Wall



The thermal resistance of the wall is $\Delta x / k$, where k is the conductivity of the wall material and Δx is the wall thickness. The thermal wall boundary condition you set will be specified on the surface that is separated from the fluid / solid cells by the wall thickness. The temperature specified at this side of the wall is T_b.

Important:

Note that for thin walls, you can only specify a constant thermal conductivity. If you want to use a non-constant thermal conductivity for a wall with nonzero-thickness, you should use the shell conduction model (see Shell Conduction (p. 1475) for details).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Specify the heat generation rate inside the wall in the **Heat Generation Rate** field. This option is useful if, for example, you are modeling printed circuit boards where you know the electrical power dissipated in the circuits.

When postprocessing a wall that has a thickness but does not have shell conduction enabled, the **Temperature...** category provides three options: the temperature of the adjacent fluid / solid cells are stored as **Static Temperature**; the temperature of the wall surface itself is stored as **Wall Temperature**; and the temperature of the surface that is separated from the fluid / solid cells by the wall thickness is stored as **Wall Temperature (Thin)**. If a more detailed analysis of the solid zone and surfaces is required, then you should consider creating layers of solid cells in your meshing application.

7.4.15.5.8. Thermal Conditions for Two-Sided Walls

If the wall zone has a fluid or solid region on each side, it is called a "two-sided wall". When you read a mesh with this type of wall zone into Ansys Fluent, a "shadow" zone will automatically be created so that each side of the wall is a distinct wall zone. In the **Wall** dialog box, the shadow zone's name will be shown in the **Shadow Face Zone** field. You can choose to specify different thermal conditions on each zone, or to couple the two zones:

- To couple the two sides of the wall (creating a "coupled wall"), select the **Coupled** option under **Thermal Conditions**. (This option will appear in the **Wall** dialog box only when the wall is a two-sided wall.) No additional thermal boundary conditions are required, because the solver will calculate heat transfer directly from the solution in the adjacent cells. You can, however, specify the material type, wall thickness, and heat generation rate for thin-wall thermal resistance calculations, as described in Thin-Wall Thermal Resistance Parameters (p. 1470). Note that the resistance parameters you set for one side of the wall will automatically be assigned to its shadow wall zone. Specifying the heat generation rate inside the wall is useful if, for example, you are modeling printed circuit boards where you know the electrical power dissipated in the circuits but not the heat flux or wall temperature.

  Note:

  – For inviscid flows with two-sided walls, the **Thermal** tab is greyed out and the reported heat flux on the CHT surface will be 0.

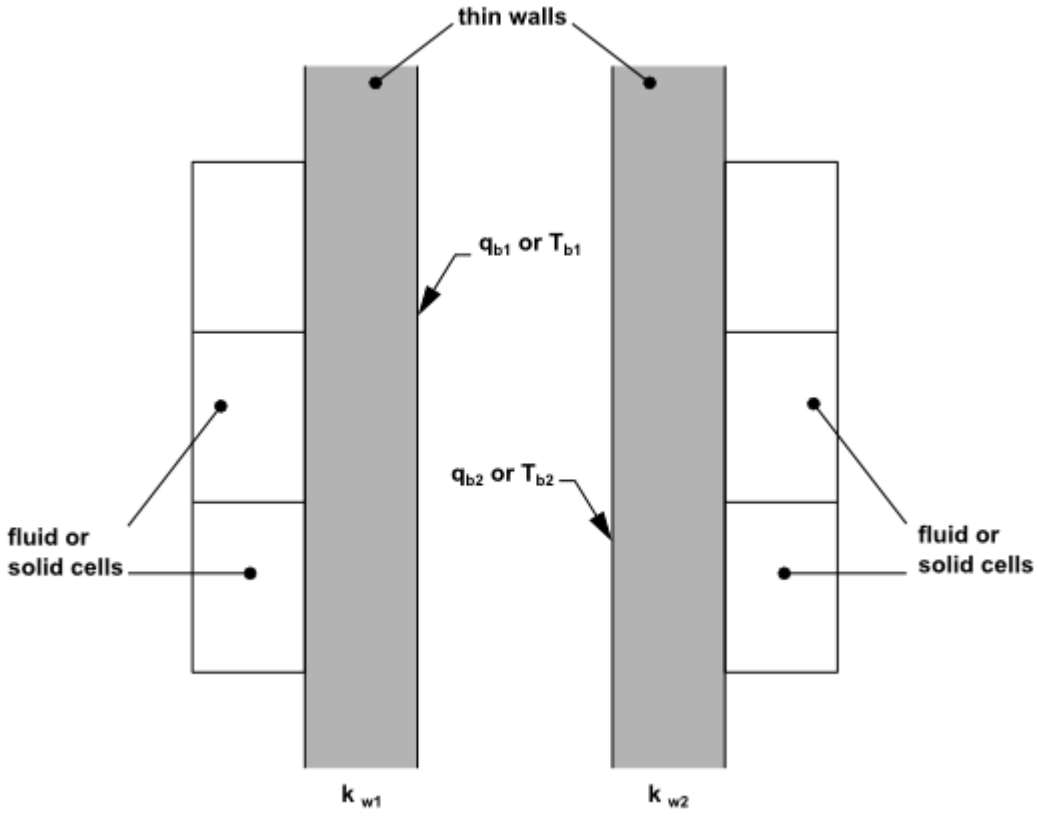
  – Results may differ for coupled walls that are intersecting boundaries. That is, you may see different results on a wall as compared to its wall-shadow depending on whether **Node Values** are enabled/disabled in this scenario.

- To uncouple the two sides of the wall and specify different thermal conditions on each one, choose **Temperature** or **Heat Flux** as the thermal condition type (**Convection** and **Radiation** are not applicable for two-sided walls); note that this uncoupling will not be effective if you have enabled shell conduction for the wall. The relationship between the wall and its shadow will be retained, so that you can couple them again at a later time, if desired. You will need to set the relevant parameters for the selected thermal condition, as described above. The two uncoupled walls can have different thicknesses, and are effectively insulated from one another. If you specify a nonzero wall thickness for the uncoupled walls, the thermal boundary conditions you set will be specified for each thin wall on the surface that is separated from the fluid / solid cells by the wall thickness, as shown in Figure 7.57: Uncoupled Thin Walls (p. 1473), where

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

T_{b1} is the **Temperature** (or q_{b1} is the **Heat Flux**) specified on one wall and T_{b2} is the **Temperature** (or q_{b2} is the **Heat Flux**) specified on the other wall. k_{w1} and k_{w2} are the thermal conductivities of the uncoupled thin walls. Note that the gap between the walls in Figure 7.57: Uncoupled Thin Walls (p. 1473) is not part of the model; it is included in the figure only to show where the thermal boundary condition for each uncoupled wall is applied.

Figure 7.57: Uncoupled Thin Walls



7.4.15.5.8.1. Orthogonality-Based Secondary Gradient Limiting at Coupled Two-Sided Walls

When cells near coupled two-sided walls have poor face orthogonality with respect to the wall, the tangential component (also known as the secondary gradient) of the discretized diffusive flux for the energy and user-defined scalar equations can become unbounded. This generates instabilities that can lead to solution divergence. When using the pressure-based solver, you have the option of limiting the secondary gradients in order to control the cross-diffusion source term. This control mechanism can improve the robustness of the discretization method while maintaining solution accuracy.

When this secondary gradient limiting is enabled, the diffusive flux approximation (described in Connectivity Macros in the *Fluent Customization Manual*) at the wall is modified so that a coefficient (ψ) controls the diffusion term:

$$D_f = \Gamma_f \frac{(\phi_1 - \phi_0)}{ds} \frac{\vec{A}\cdot\vec{A}}{\vec{A}\cdot\vec{e}_s} + \psi\Gamma_f \left(\overline{\nabla}\phi\cdot\vec{A} - \overline{\nabla}\phi\cdot\vec{e}_s \frac{\vec{A}\cdot\vec{A}}{\vec{A}\cdot\vec{e}_s} \right) \tag{7.121}$$

where the diffusion coefficient ψ is a function of face orthogonality—that is, the dot product between $\vec{A}$ (the area vector for the face, normal to the wall) and $\vec{e}_s$ (the unit vector along the line connecting the cell centroid (0) and face centroid (1)). Note that $0 \le \psi \le 1$.

Note the following limitations:

- It is only available with the pressure-based solver.
- It can only be applied to the energy calculation and/or user-defined scalars (UDS) on coupled two-sided walls (that is, wall / wall-shadow pairs with **Coupled** selected from the **Thermal Conditions** list in the **Thermal** tab).

To enable orthogonality-based secondary gradient limiting, use the following text commands:

- For the energy equation:

  `solve → set → advanced → secondary-gradient-limiting → energy?`

- For user-defined scalars:

  `solve → set → advanced → secondary-gradient-limiting → uds?`

When you enable secondary gradient limiting, by default it is applied to faces on the coupled two-sided walls using mesh quality limits from 0.7–0.9. You can shift the range if necessary by using the following text command; note that shifting this range closer to 1 will decrease the risk of divergence, but at the cost of accuracy.

`solve → set → advanced → secondary-gradient-limiting → mesh-quality-limits`

7.4.15.5.9. Boundary Advection for Solid Motion

If the wall is not between two cell zones but is adjacent only to a single solid cell zone that involves solid motion with a nonzero translational or rotational velocity, then the **Boundary Advection** option is available. If the velocity for the solid motion (specified in the **Solid Motion** tab in the **Solid** dialog box) is directed into or out of the domain to a significant degree on that wall, then you should consider enabling the **Boundary Advection** option. This option allows you to account for the advection of thermal energy into or out of the solid domain at a rate that is consistent with the velocity normal to the boundary, the temperature of the material, and the material properties.

As an alternative to setting this option for each individual wall, you can use the following text command to review all the walls and automatically enable the **Boundary Advection** option where needed. The option is enabled on every appropriate wall for which the average angle of the velocity of the solid is greater than 2 degrees.

`define → boundary-conditions → detect-boundary-advection`

For details on setting up the solid motion model for a domain, see Defining Zone Motion (p. 1328).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.15.5.10. Shell Conduction

To enable shell conduction for a wall, enable the **Shell Conduction** option in the **Wall** boundary condition dialog box. You can then click the **Edit...** button to open the **Shell Conduction Layers** dialog box, where you can define the properties of the single or multiple layers of the shell. Note that you must specify a nonzero wall thickness for every layer of the shell. When shell conduction is enabled, Ansys Fluent will compute heat conduction for the wall not only in the normal direction (which is always computed when the energy equation is solved), but also in the planar directions. The **Shell Conduction** option will appear in the **Wall** dialog box for all walls when solution of the energy equation is active (except for mapped interfaces). For information about how the thermal conditions are applied on a wall with shell conduction enabled, managing multiple shells, and postprocessing shell conduction walls, see Shell Conduction (p. 2182).

Ansys Fluent cases with shell conduction can be read in serial or parallel. Either a partitioned or an unpartitioned case file can be read in parallel (see Mesh Partitioning and Load Balancing (p. 4315) for more information on partitioning). After reading a case file in parallel, shell zones can be created on any wall.

To convert every wall with a finite thickness into a shell with a single action, the TUI command `define/boundary-conditions/modify-zones/create-all-shell-threads` can be used; each converted shell will have a single layer with the same thickness as the original thin wall (any pre-existing shells will not be modified). To disable shell conduction in every wall with a single action, the TUI command `define/boundary-conditions/modify-zones/delete-all-shells` can be used. These capabilities are available in both serial and parallel mode.

For more information on shell conduction, see Shell Conduction (p. 2182).

7.4.15.5.11. Heat Transfer Boundary Conditions Through System Coupling

System Coupling allows the input and output of thermal data from Ansys Fluent. When Fluent is coupled with another system in Workbench using System Coupling, you can select the **via System Coupling** option on the desired wall boundaries to receive thermal data through System Coupling service. Note that this option does not need to be selected to provide thermal data from Fluent.

For more details about setting up a simulation with System Coupling, see the Performing System Coupling Simulations Using Fluent (p. 4647) and the System Coupling User's Guide.

When thermal data is transferred into Fluent via System Coupling, the following variables are available:

- temperature
- heat flow (heat rate)
- heat transfer coefficient (also known as "convection coefficient")

When thermal data is transferred out of Fluent via System Coupling, the following variables are available:

- temperature

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- heat flow (heat rate)
- heat transfer coefficient (also known as "convection coefficient")
- near wall temperature (also known as "bulk temperature" or "ambient temperature")

For each data transfer, you specify the type of data transfer (the variables transferred) during the System Coupling setup.

As part of standard heat transfer boundary condition settings in Ansys Fluent, you can also specify the type of material, the thickness of the wall, and the heat generation rate in the wall. Select the material type in the **Material Name** drop-down list; if you want to check or modify the properties of the selected material, you can click **Edit...** to open the **Edit Material** dialog box (this dialog box contains just the properties of the selected material, and not the full contents of the standard **Create/Edit Materials** dialog box). You can specify the thickness in the **Wall Thickness** number-entry box and the heat generation rate in the **Heat Generation Rate** number-entry box.

Note:

A boundary will behave in the same way as an adiabatic boundary if the **via System Coupling** option is enabled on this boundary, and Ansys Fluent is not receiving data from the coupling data transfer. Fluent is not receiving coupling data if it is either not involved with a System Coupling simulation, or if the coupling is a one-way transfer with the Fluent analysis only providing data to the second solver.

7.4.15.5.12. Heat Transfer Boundary Conditions Across a Mapped Interface

Mapped interfaces provide a robust approach for modeling coupled walls between zones when the interface zones penetrate each other or have gaps between them (see Figure 7.58: 2D Interface with Penetration and Gaps (p. 1477)). For the interface wall boundary zones created as part of such interfaces, **via Mapped Interface** is automatically selected from the **Thermal Conditions** list in the **Thermal** tab of the **Wall** dialog box, in order to interpolate the thermal data.

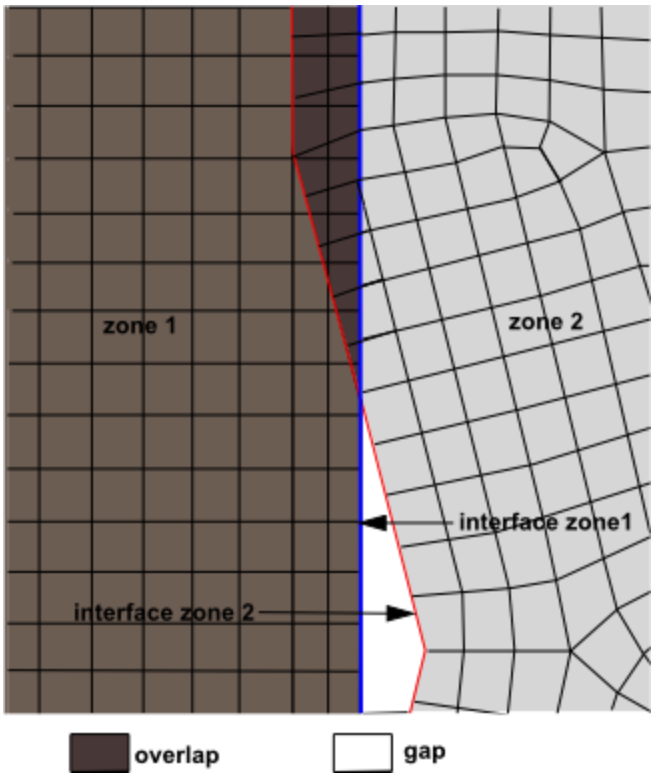
Important:

Note that thermal coupling calculations will not be performed at mapped interfaces for the Eulerian multiphase model.

For more details about mapped interfaces, see The Mapped Option (p. 1178) and Using a Non-Conformal Mesh in Ansys Fluent (p. 1185).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.58: 2D Interface with Penetration and Gaps



Such boundaries also allow you to specify the standard heat transfer boundary condition settings, such as the type of material, the thickness of the wall, and the heat generation rate in the wall. Select the material type in the **Material Name** drop-down list; if you want to check or modify the properties of the selected material, you can click **Edit...** to open the **Edit Material** dialog box (this dialog box contains just the properties of the selected material, and not the full contents of the standard **Create/Edit Materials** dialog box). You can specify the thickness in the **Wall Thickness** number-entry box and the heat generation rate in the **Heat Generation Rate** number-entry box. Note that the **Shell Conduction** option is not available with the **via Mapped Interface** thermal condition.

7.4.15.5.13. Temperature Jump for Rarefied Gases

In the rarefied regime, the parietal gas temperature may differ from the imposed wall temperature. For details, see Temperature Jump for Rarefied Gases (p. ?).

7.4.15.6. Species Boundary Conditions for Walls

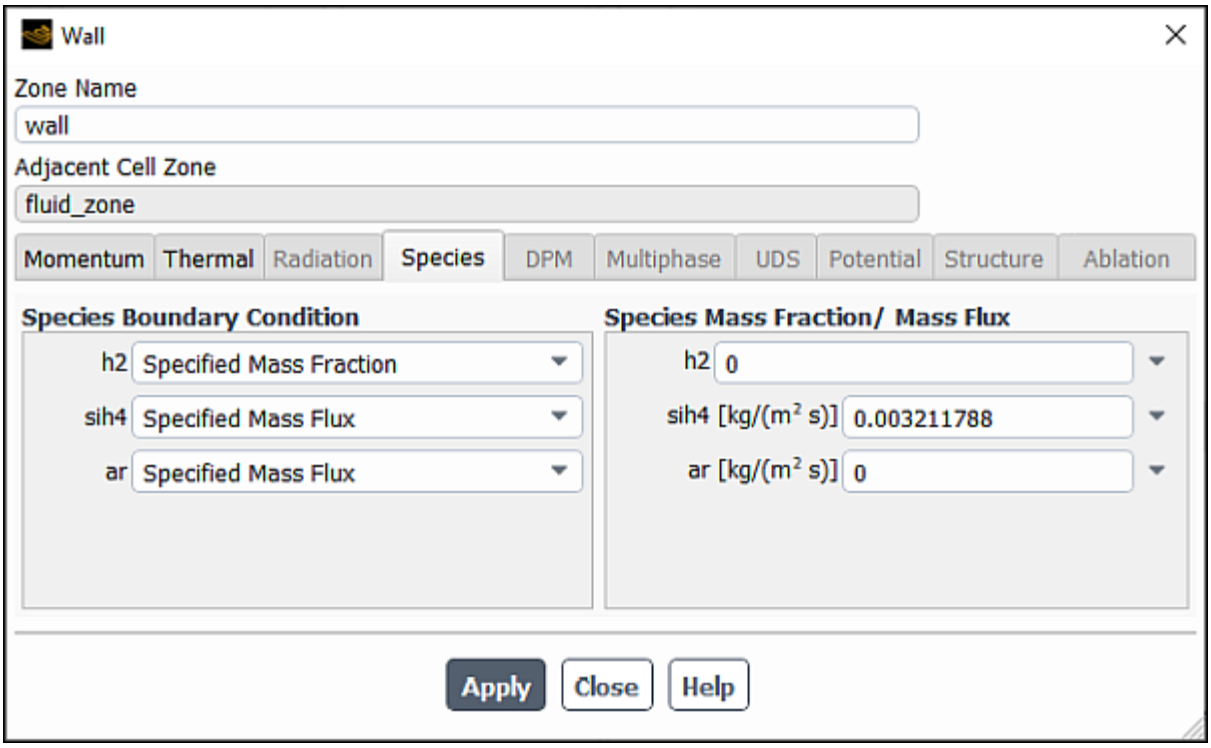
By default, a zero mass flux condition for all species is assumed at walls (except for species that participate in surface reactions). You can change the default settings and specify species mass fractions or non-zero mass fluxes at walls in your simulation. That is, both Dirichlet and Neumann boundary conditions such as those that are specified at inlets can be used at walls as well.

To change the default zero mass flux boundary condition for a species at the wall:

1. In the Wall Dialog Box (p. 5085), go to the **Species** tab to view the species boundary conditions for the wall (see Figure 7.59: The Wall Dialog Box for Species Boundary Condition Input (p. 1478)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.59: The Wall Dialog Box for Species Boundary Condition Input



2. In the **Species Boundary Condition** group box, you can define one of the following boundary conditions for the species at the wall:

   - If you want to specify a non-zero species mass flux, select **Specified Mass Flux** (default) in the drop-down list to the right of the species name. Then in the **Species Mass Fraction / Mass Flux** group box, enter its value for the species of interest.

   - Similarly, if you want to specify a species mass fraction, select **Specified Mass Fraction** in the drop-down list to the right of the species name. Then in the **Species Mass Fraction / Mass Flux** group box, enter its value for the species of interest.

The boundary condition type for each species is specified separately, so you can choose to use different methods for different species.

If you are modeling species transport with **Wall Surface** reactions enabled in the **Species Model** dialog box, you can alternatively enable a reaction mechanism at a wall by selecting the **Reaction** option and then selecting an available mechanism from the **Reaction Mechanisms** drop-down list. See Defining Zone-Based Reaction Mechanisms (p. 2387) for more information about defining reaction mechanisms.

You can also model unresolved surface washcoats, which greatly increase the catalytic surface area, by specifying the **Surface Area Washcoat Factor**. The surface washcoat increases the area available for surface reaction.

In transient simulations, if you want to postprocess the deposition of the solid species on the wall over time, you need to specify the following parameters:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Initial Deposition Thickness**: is the thickness of the solid species deposition on the wall at the start of the simulation.
- **Solid Species Density**: is the density of the solid species deposited on the wall. If more than one solid species exist in your simulation, then you need to specify the average solid species density.

Note that these two parameters are used for postprocessing purposes only and will not affect simulation results.

7.4.15.6.1. Partial Catalytic Boundary Conditions for Walls

In hypersonic flows, the recombination of atoms is usually dominant at the wall. The atoms that diffuse to the wall can either reflect from the wall or recombine into molecules, see Partial Catalytic Boundary Condition for Walls (p. ?).

7.4.15.7. Radiation Boundary Conditions for Walls

If you are using the gray P-1, DTRM, DO, surface-to-surface, or MC models, you will need to set the emissivity of the wall (**Internal Emissivity**) in the **Thermal** tab of the **Wall** dialog box. If you are using the Rosseland model you do not need to set the emissivity, because Ansys Fluent assumes the emissivity is 1.

For the non-gray P-1, DO, or MC models, specify a constant **Internal Emissivity** for each wavelength band in the **Radiation** tab of the **Wall** dialog box (the default value in each band is 1). Alternatively, you can specify the internal emissivity using a boundary condition parameter (see Creating a New Parameter (p. 1311)). If you are using the non-gray DO or MC models, you will also need to define the wall as opaque or semi-transparent in the **Radiation** tab. See Defining Boundary Conditions for Radiation (p. 2217) for details. If you are using the DO or MC models, you can enable a **Boundary Source**.

7.4.15.8. Discrete Phase Model (DPM) Boundary Conditions for Walls

If you are modeling a discrete phase of particles, you can set the fate of particle trajectories at the wall in the **DPM** section of the **Wall** dialog box. See Setting Boundary Conditions for the Discrete Phase (p. 2779) for details.

7.4.15.8.1. Wall Adhesion Contact Angle for VOF Model

If you are using the VOF model and you are modeling wall adhesion, you can specify the contact angle for each pair of phases at the wall in the **Momentum** tab of the **Wall** dialog box. See Steps for Setting Boundary Conditions (p. 2957) for details.

7.4.15.9. User-Defined Scalar (UDS) Boundary Conditions for Walls

If you have defined UDS transport equations in your model, you can specify boundary conditions for each equation in the **UDS** section of the **Wall** dialog box. See Setting Up UDS Equations in Ansys Fluent (p. 1739) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.15.10. Wall Film Conditions for Walls

If you are using the Eulerian Wall Film model (see Modeling Eulerian Wall Films (p. 3239) for details), you can set liquid film conditions at the wall in the **Wall Film** tab of the **Wall** dialog box. This tab is available only if you have enabled the Eulerian Wall Film model in the Models Task Page (p. 4696).

For details, see Setting Eulerian Wall Film Solution Controls (p. 3248).

7.4.15.11. Structural Model Conditions for Walls

For an intrinsic fluid-structure interaction (FSI) simulation, you can set the displacement boundary condition settings for walls that are adjacent to solid cell zones in the **Structure** tab of the **Wall** dialog box. This tab is only available when a model is selected in the Structural Model Dialog Box (p. 4869). For details, see Setting Up an Intrinsic Fluid-Structure Interaction (FSI) Simulation (p. 3229).

7.4.15.12. Default Settings at Wall Boundaries

The default thermal boundary condition is a fixed heat flux of zero. Walls are, by default, not moving.

7.4.15.13. Shear-Stress Calculation Procedure at Wall Boundaries

For no-slip wall conditions, Ansys Fluent uses the properties of the flow adjacent to the wall/fluid boundary to predict the shear stress on the fluid at the wall. In laminar flows this calculation simply depends on the velocity gradient at the wall, while in turbulent flows one of the approaches described in Near-Wall Treatments for Wall-Bounded Turbulent Flows in the Theory Guide is used.

For specified-shear walls, Ansys Fluent will compute the tangential velocity at the boundary.

If you are modeling inviscid flow with Ansys Fluent, all walls use a slip condition, so they are frictionless and exert no shear stress on the adjacent fluid.

7.4.15.13.1. Shear-Stress Calculation in Laminar Flow

In a laminar flow, the wall shear stress is defined by the normal gradient of the tangential velocity at the wall:

$$\vec{\tau}_w = \mu \frac{\partial \vec{v}}{\partial n} \tag{7.122}$$

When there is a steep velocity gradient at the wall, you must be sure that the mesh is sufficiently fine to accurately resolve the boundary layer. Guidelines for the appropriate placement of the near-wall node in laminar flows are provided in Mesh Element Distribution (p. 1149).

7.4.15.13.2. Shear-Stress Calculation in Turbulent Flows

Wall treatments for turbulent flows are described in Near-Wall Treatments for Wall-Bounded Turbulent Flows in the Theory Guide.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.15.14. Heat Transfer Calculations at Wall Boundaries

7.4.15.14.1. Temperature Boundary Conditions

When a fixed temperature condition is applied at the wall, the heat flux to the wall from a fluid cell is computed as

$$q = h_f\left(T_w - T_f\right) + q_{rad} \tag{7.123}$$

where

h_f = fluid-side local heat transfer coefficient

T_w = wall surface temperature

T_f = local fluid temperature

q_{rad} = radiative heat flux

Note that the fluid-side heat transfer coefficient is computed based on the local flow-field conditions (for example, turbulence level, temperature, and velocity profiles), as described by Equation 7.130 (p. 1483).

Heat transfer to the wall boundary from a solid cell is computed as

$$q = \frac{k_s}{\Delta n}\left(T_w - T_s\right) + q_{rad} \tag{7.124}$$

where

k_s = thermal conductivity of the solid

T_s = local solid temperature

Δn = distance between wall surface and the solid cell center

7.4.15.14.2. Heat Flux Boundary Conditions

When you define a heat flux boundary condition at a wall, you specify the heat flux at the wall surface. Ansys Fluent uses Equation 7.123 (p. 1481) and your input of heat flux to determine the wall surface temperature adjacent to a fluid cell as

$$T_w = \frac{q - q_{rad}}{h_f} + T_f \tag{7.125}$$

where, as noted above, the fluid-side heat transfer coefficient is computed based on the local flow-field conditions. When the wall borders a solid region, the wall surface temperature is computed as

$$T_w = \frac{\left(q - q_{rad}\right)\Delta n}{k_s} + T_s \tag{7.126}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.15.14.3. Convective Heat Transfer Boundary Conditions

When you specify a convective heat transfer coefficient boundary condition at a wall, Ansys Fluent uses your inputs of the external heat transfer coefficient and external heat sink temperature to compute the heat flux to the wall as

$$\begin{aligned} q &= h_f\left(T_w - T_f\right) + q_{rad} \\ &= h_{ext}\left(T_{ext} - T_w\right) \end{aligned} \tag{7.127}$$

where

h_{ext} = external heat transfer coefficient defined by you

T_{ext} = external heat-sink temperature defined by you

q_{rad} = radiative heat flux

Equation 7.127 (p. 1482) assumes a wall of zero-thickness.

7.4.15.14.4. External Radiation Boundary Conditions

When the external radiation boundary condition is used in Ansys Fluent, the heat flux to the wall is computed as

$$\begin{aligned} q &= h_f\left(T_w - T_f\right) + q_{rad} \\ &= \varepsilon_{ext}\sigma\left(T_\infty^4 - T_w^4\right) \end{aligned} \tag{7.128}$$

where

ε_{ext} = emissivity of the external wall surface defined by you

σ = Stefan-Boltzmann constant

T_w = surface temperature of the wall

T_∞ = temperature of the radiation source or sink on the exterior of the domain, defined by you

q_{rad} = radiative heat flux to the wall from within the domain

Equation 7.128 (p. 1482) assumes a wall of zero-thickness.

7.4.15.14.5. Combined External Convection and Radiation Boundary Conditions

When you choose the combined external heat transfer condition, the heat flux to the wall is computed as

$$\begin{aligned} q &= h_f\left(T_w - T_f\right) + q_{rad} \\ &= h_{ext}\left(T_{ext} - T_w\right) + \varepsilon_{ext}\sigma\left(T_\infty^4 - T_w^4\right) \end{aligned} \tag{7.129}$$

where the variables are as defined above. Equation 7.129 (p. 1482) assumes a wall of zero-thickness.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.15.14.6. Calculation of the Fluid-Side Heat Transfer Coefficient

In laminar flows, the fluid side heat transfer at walls is computed using Fourier's law applied at the walls. Ansys Fluent uses its discrete form:

$$q=k_f\left(\frac{\partial T}{\partial n}\right)_{wall} \tag{7.130}$$

where n is the local coordinate normal to the wall.

For turbulent flows, Ansys Fluent uses the law-of-the-wall for temperature derived using the analogy between heat and momentum transfer [84] (p. 5718). See Standard Wall Functions in the separate Theory Guide for details.

7.4.16. Perforated Wall Boundary Conditions

7.4.16.1. Overview and Limitations

Perforated walls can be used to introduce low-temperature air into a region operating under very high temperatures to provide efficient cooling of the walls. This is one of the most common techniques used in many industrial applications, such as combustor chambers.

Because perforated walls often have thousands of small holes, resolving every single hole is not practical due to the high computational cost. Ansys Fluent offers a model to account for the effects of air streams injected through the perforated holes.

Note the following limitations of the perforated wall model:

- The perforated wall model considers the effects of perforated holes only on:
  - velocity fields
  - temperature fields (if the energy equation is enabled)
  - species distribution (if the species transport model is enabled)
  - mixture fraction and progress variable distribution (if the PDF model is enabled)
- The perforated wall model is not supported with the density-based solver.
- When the Euler-Euler multiphase flow model is used, the perforated wall model is applied only to the primary phase.
- The effects of the perforated wall model on turbulence are not considered.

7.4.16.2. Modeling Concept

When modeling the effects of perforated holes in Ansys Fluent, solid walls are not meshed (except for cases that involve conjugate heat transfer as further described). Instead, a flow injection is imposed on the side where the flow is coming out through the perforated hole (hereinafter referred to as the injection side). In order to ensure the mass conservation, the same amount of mass flow is extracted at the side where the flow is coming into the perforated hole (hereinafter referred to as the extraction side).

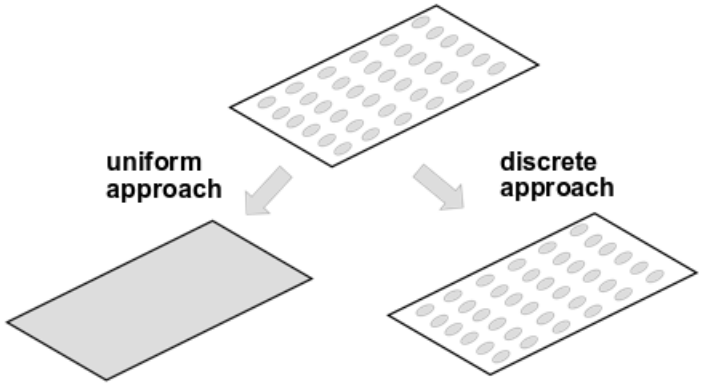
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.60: Modeling Perforated Walls

From the perspective of how the perforated walls are treated, there are two approaches to modeling the effects of perforated holes in the flow fields:

- Uniform approach
- Discrete approach

Figure 7.61: Uniform and Discrete Approaches



The uniform approach models the average effect of perforated walls on the flow field. Solid walls with the large number of the perforated holes are treated as having uniform injection conditions. The individual perforated hole is not resolved. Instead, the entire surface of the injection side wall has the same boundary condition of a prescribed mass flux. This approach is based on the rationale that in the preliminary design process, only the first-order effects of perforated walls on the flow field are of interest. This means that only the inviscid streamwise momentum flux and inviscid normal momentum flux due to injection are to be considered.

The discrete approach has more fidelity in modeling perforation holes effects. The individual perforated hole is resolved. In other words, the injections conditions are only imposed at the location of the perforated holes. Although this approach is able to resolve the flow fields at the individual perforated hole level, it does not imply that you need to actually mesh the holes during the meshing process. Instead, all the perforated holes can be marked via the perforated wall model capability in Ansys Fluent as described in Setting Perforated Walls (p. 1487).

Physically, if the perforated holes are fully resolved, the injection conditions at the exit of the holes are determined internally by the flow inside the perforated channel. More specifically, they are determined by the pressure difference between the extraction and injection sides. Therefore, the injection mass flow rate at the hole exit $\dot{m}$ (in kg/s) can be calculated by [14] (p. 5713):

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\dot{m}=\mathrm{A}C_D\sqrt{2\rho_e\left(p_e-p_i\right)} \tag{7.131}$$

where

A = effective area of the hole channel

C_D = discharging coefficient

ρ_e = density on the extraction side

p_e and p_i = pressures on the extraction side and injection side, respectively

In order to determine the injection mass flow rate, the value of the discharging coefficient C_D must be specified. In some practical scenarios, the product value of $\mathrm{A}C_D$ may be known instead of individual values of A and C_D. In Ansys Fluent, you can specify either C_D or $\mathrm{A}C_D$ as described in Setting Perforated Walls (p. 1487).

Equation 7.131 (p. 1485) assumes that the flow inside the perforated channel is incompressible. If the speed inside the perforated channel is very high, the compressible flow is more adequate to describe the flow behaviors under this circumstance. A compressible formulation for the injection condition is suggested as [11] (p. 5713):

$$\dot{m}=C_D A p_e\sqrt{\frac{2}{RT_e}\frac{\gamma}{\gamma-1}\left(p_{ratio}^{-\frac{2}{\gamma}}-p_{ratio}^{-\frac{\gamma+1}{\gamma}}\right)} \tag{7.132}$$

$$p_{ratio}=\min\left[\frac{p_e}{p_i},\left(\frac{\gamma+1}{2}\right)^{\frac{\gamma}{\gamma-1}}\right]$$

where

R = gas constant

T_e = temperature of the extraction side

γ = heat capacity ratio

Estimating injection condition based on Equation 7.131 (p. 1485) or Equation 7.132 (p. 1485) is referred to as the dynamic method in Ansys Fluent since the injection mass flow rate is estimated based on the local pressure field. Besides the dynamic injection method specification, you can directly specify the injection mass flow rate at the hole exit $\dot{m}$ (in kg/s). This is referred to as the static injection method in Ansys Fluent.

Once the injection mass flow rate is determined via either the dynamic or static injection method, a proper mass flux condition is imposed on the extraction side to ensure mass conservation. After the mass flux condition is determined on the extraction side, the amount of energy and species (or mixture fraction/progress variable) removed on the extraction side can be calculated. As a result, the same amount of energy and species is added on the injection side to ensure energy conservation and species mass conservation. For the momentum flux on the injection side, different options can be selected based on the application need.

Note that although the perforated wall model originally comes from the gas turbine combustor applications, it is also applicable to general problems if similar effects on the flow fields need to be accounted for.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For conjugate heat transfer cases, solid walls are meshed, while perforated holes are not meshed. The effects of perforated holes in the flow fields is modeled using the discrete approach. The injection mass flow rate at the hole exit is still calculated by Equation 7.131 (p. 1485). The heat transfer effects between the solid walls and the flow when the injection flow passes through the perforated holes can be accounted for as described below. An example of such a problem is conjugate heat transfer in a gas turbine combustor where low-temperature air passing through perforated holes is heated due to the higher-temperature solid walls.

The heat transfer between the solid walls and the injection flow q is calculated by:

$$q=h\pi DL\left(T_0-T_w\right) \tag{7.133}$$

where

D and L = diameter and length of the perforated hole, respectively

T_0 = flow temperature at the entrance of the perforated hole

T_w = solid wall temperature

h = convection heat transfer coefficient

h is calculated by Cho and Goldstein's correlation for short-length holes [11] (p. 5713):

$$Nu=\frac{hL}{k}=0.0397Re^{0.73}Pr^{0.33}\left(1+2.384\left(\frac{L}{D}\right)^{-0.621}\right) \tag{7.134}$$

where

Nu = Nusselt number

k = thermal conductivity of the fluid

Re and Pr = Reynolds number and Prandtl number, respectively

With Equation 7.133 (p. 1486), the injection flow temperature at the hole exit T_1 is calculated by:

$$T_1=T_0-\frac{q}{\dot{m}c_p} \tag{7.135}$$

where $\dot{m}$ is the injection mass flow rate given by Equation 7.131 (p. 1485), and c_p is the heat capacity of the fluid.

As the perforated holes are not meshed, the solid wall temperature T_w in Equation 7.133 (p. 1486) is estimated as follows. Given that the solid walls are relatively thin and have large thermal conductivity, the solid wall temperature is assumed to be linearly distributed in the direction of the orientation of the perforated holes. Then the analytical solution for the total heat transfer can be expressed as:

$$q=\left(\frac{h\pi D\left(T_0-T_{w0}\right)}{\tau}+\frac{h\pi D\left(T_{w1}-T_{w0}\right)}{L\tau^2}\right)\left(1-\exp\left(-\tau L\right)\right)-\frac{h\pi D\left(T_{w1}-T_{w0}\right)}{\tau} \tag{7.136}$$

with

$$\tau=\frac{h\pi D}{\dot{m}c_p} \tag{7.137}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In Equation 7.136 (p. 1486), T_{w0} and T_{w1} are the wall temperature at the entrance and exit of perforated holes, respectively.

7.4.16.3. Setting Perforated Walls

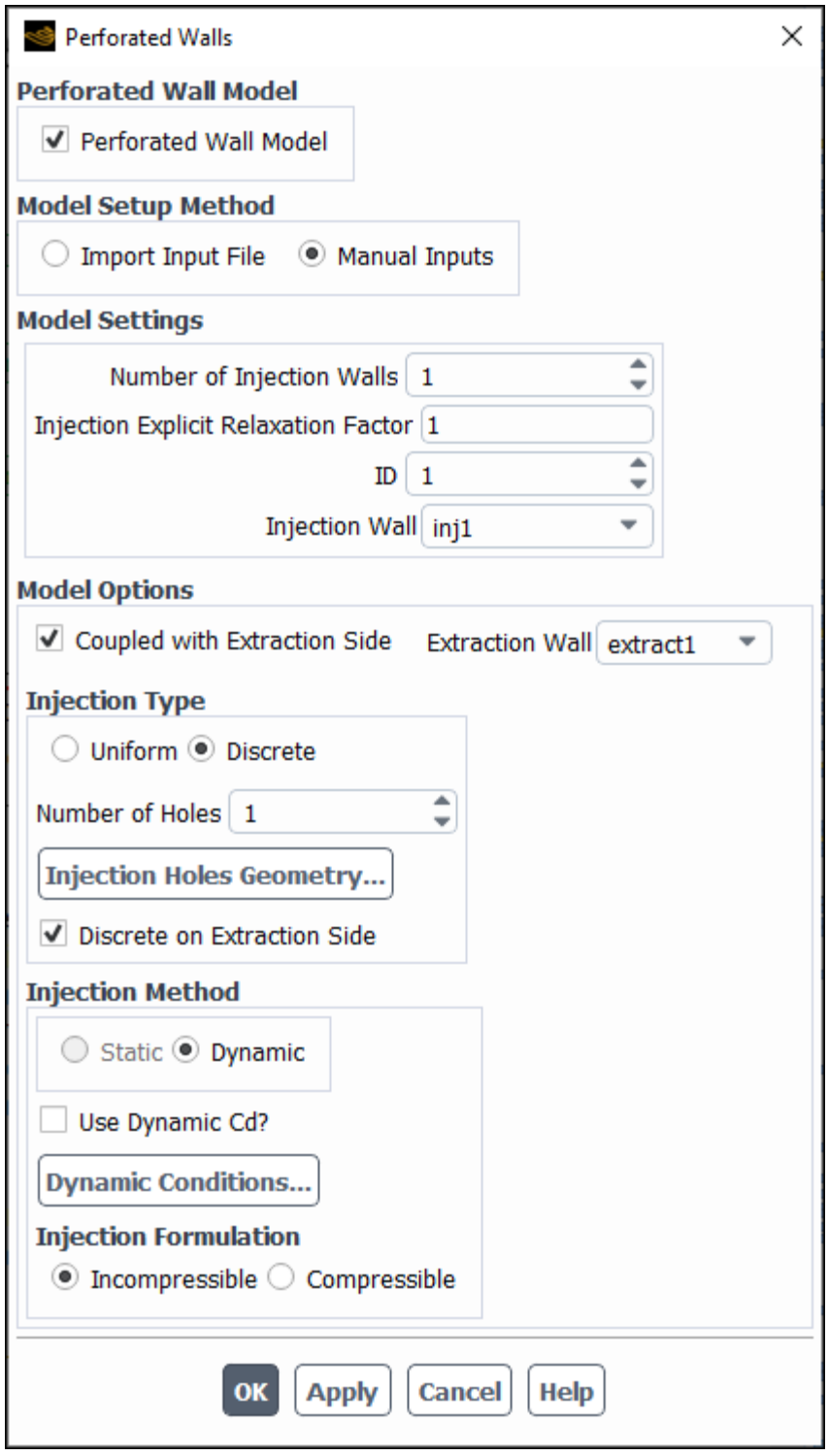
If you want to consider the effects of air streams injected through the perforated holes in your simulation, follow the below steps. Note that the perforated walls model setup is based on the injection side; the extraction side will be handled automatically by Ansys Fluent based on the selected model options.

1. In the **Boundary Conditions** task page, click the **Perforated Wallis...** button.

2. In the **Perforated Walls** dialog box that opens, enable the **Perforated Wall Model** option.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.62: The Perforated Walls Dialog Box



3. Select an appropriate **Model Setup Method** from the following options:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Import Input Files

allows you to read an input file that contains the selections of model options and values of the model parameters. This option is useful for cases where hundreds or even thousands of perforated holes need to be defined.

Manual Inputs

allows you to manually specify perforated wall options and parameters. When this option is selected, the **Perforated Walls** dialog box expands to reveal relevant model inputs.

4. If you have chosen **Import Input File**, use the **Select File** dialog box to read an input file for the perforated wall definition. See Perforated Wall File Format (p. 1495) for the details about input file format and an example of the perforated wall file.

   Once Ansys Fluent reads the file, the data entry fields in the **Injection Holes** and **Dynamic Setup** dialog boxes will be automatically populated with the appropriate information. You can review and edit these settings in each dialog box.

5. If you have chosen **Manual Inputs**, follow the steps outlined in Procedure for Manual Setup of Perforated Walls (p. 1489):

6. Click **OK** to close the **Perforated Walls** dialog box.

7.4.16.4. Procedure for Manual Setup of Perforated Walls

If you have chosen the **Manual Inputs** option in the **Perforated Walls** dialog box (**Model Setup Method** group box), you need to specify the following settings and options:

1. In the **Model Settings** group box (see Figure 7.62: The Perforated Walls Dialog Box (p. 1488)), specify the total number of walls where you want to define the perforated wall boundary condition in the **Number of Injection Walls** field. You will need to define boundary conditions for each wall separately.

2. In the **Injection Explicit Relaxation Factor** field, enter a value between 0 and 1 (default = 1).

   This parameter controls the update of the injection mass flow rate at each iteration. It is used to blend the injection mass flow rates calculated by Equation 7.131 (p. 1485) (for an incompressible flow) or Equation 7.132 (p. 1485) (for a compressible flow) at the current and previous iterations. If any convergence issues arise, you can try to lower this factor at the beginning of your run.

3. Set the **ID** of the injection wall. This number must be between 1 and the value specified in **Number of Injection Walls**.

4. From the **Injection Wall** drop-down list, select the name of the injection wall you are defining.

5. If you want to couple the injection and extraction walls, enable **Coupled with Extraction Side**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

With **Coupled with Extraction Side** enabled, Ansys Fluent automatically selects **Dynamic** in the **Injection Method** group box. Otherwise, Ansys Fluent uses the **Static** injection method.

Note:

In the current implementation, you cannot change the injection method (**Static** or **Dynamic**).

6. Select the name of the extraction wall that you want to couple with the injection wall from the **Extraction Wall** drop-down list (**Modeling Options** group box). When the flow is introduced on the injection wall, the same amount of mass is removed from the extraction wall.

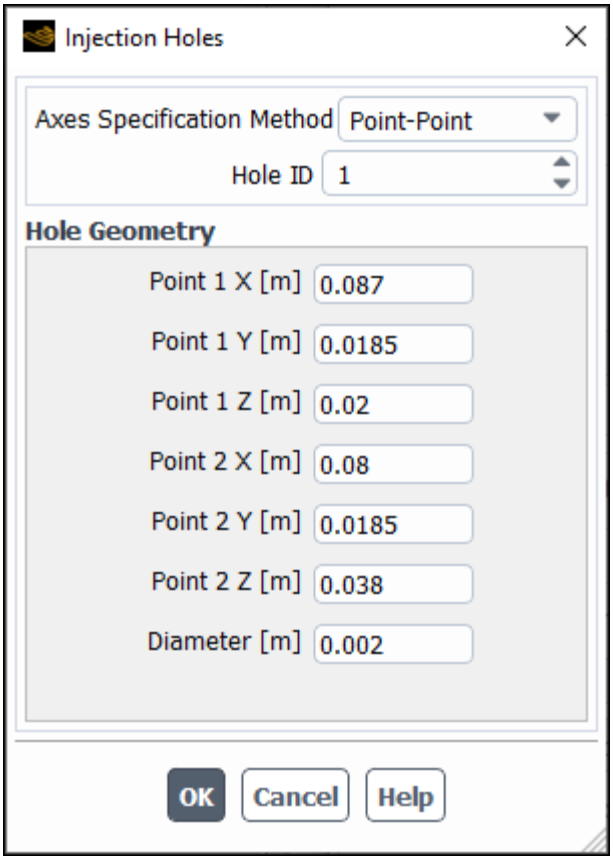
7. In the **Injection Type** group box select either **Uniform** or **Discrete** modeling approach. See Modeling Concept (p. 1483) for more information about these two approaches.

8. For the **Discrete** injection type, define the geometry of the injection holes.

   a. In the **Number of Holes** field, set the total number of perforated holes in the injection wall.

   b. Click the **Injection Holes Geometry...** button to specify the injection holes parameters in the **Injection Holes** dialog box that opens.

Figure 7.63: The Injection Holes Dialog Box



   i. Select the **Axes Specification Method** for defining perforated holes:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Point-Point

allows you to specify two arbitrary points on the cylinder axis.

Point-Direction

allows you to specify a point and a direction vector of the cylinder axis.

ii. For each hole with the ID set in the **Hole ID** field, you need to specify the following parameters in the **Hole Geometry** group box (depending on your selection of the **Axes Specification Method**):

- For the **Point-Point** axes specification method:
  - Coordinates of two points on the cylinder axis ((**Point 1 X**, **Point 1 Y**, **Point 1 Z**) and (**Point 2 X**, **Point 2 Y**, **Point 2 Z**))
  - **Diameter**
- For the **Point-Direction** axes specification method:
  - Coordinates of a point on the cylinder axis (**Point X**, **Point Y**, **Point Z**) and a direction vector (**Direction X**, **Direction Y**, **Direction Z**)
  - **Diameter**

Note:

Inputs for the Z coordinate will be displayed for 3D cases only.

Ansys Fluent will automatically find all intersections between the specified cylinders and the injection or extraction wall. Then each face involved in the intersections will be marked as either a hole exit or a hole inlet, respectively.

iii. Click **OK** to close the **Injection Holes** dialog box.

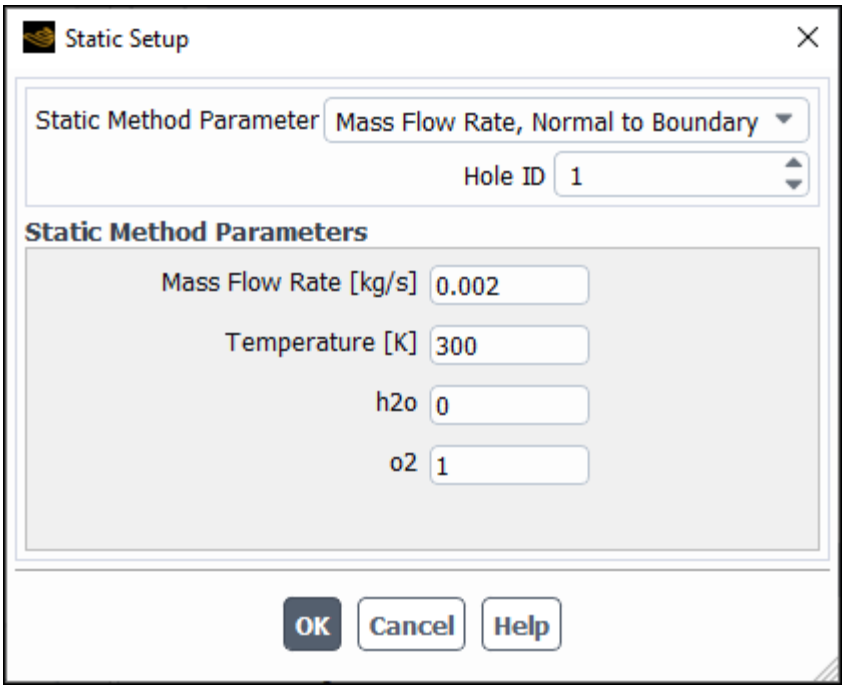
c. If you want to model the uniform condition on the extraction side, disable the **Discrete on Extraction Side** option (default) (**Injection Type** group box).

9. For the **Static** injection method, specify the static method settings.

a. In the **Injection Method** group box, click the **Static Conditions...** button to specify injection settings in the **Static Setup** dialog box that opens.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.64: The Static Setup Dialog Box (Uniform Injection)



b. Select the **Specification Method** from the following options:

Normal to Boundary

defines the injection direction by the normal to the surface of the injection wall.

Direction

defines the injection direction by the three components of the directional vector.

c. For each hole with the ID set in the **Hole ID** field, you need to specify the following parameters in the **Static Method Parameters** group box:

- **Mass Flow Rate**: Injection mass flow rate (kg/s)
- **Temperature**: Injection static temperature (K)
- Mass fractions of species that constitute the mixture material in the injection flow

  In the example shown in Figure 7.64: The Static Setup Dialog Box (Uniform Injection) (p. 1492), the mixture material contains H_2O, O_2, and N_2. In this case, the mixture fraction of **h2o** and **o2** must be specified. The mass fraction of N_2 is computed by subtracting the sum of the remaining mass fractions from unity.

- (**Direction** specification method only) **X Direction**, **Y Direction**, and **Z Direction** (3D only)

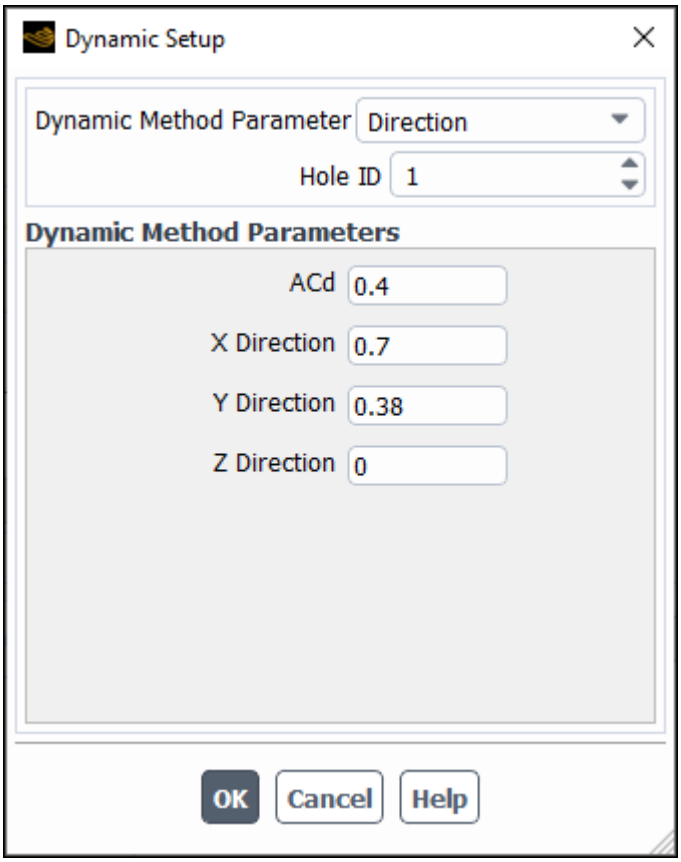
d. Click **OK** to close the **Static Setup** dialog box.

10. For the **Dynamic** injection method, specify the dynamic method settings.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. In the **Injection Method** group box, click the **Dynamic Conditions...** button to specify injection settings in the **Dynamic Setup** dialog box that opens.

Figure 7.65: The Dynamic Setup Dialog Box (Uniform Injection)



b. If you have selected the **Uniform** injection type, you need to specify the following:

i. Select the **Dynamic Method Parameter** from the following options:

Normal to Boundary

defines the injection direction by the normal to the surface of the injection wall.

Direction

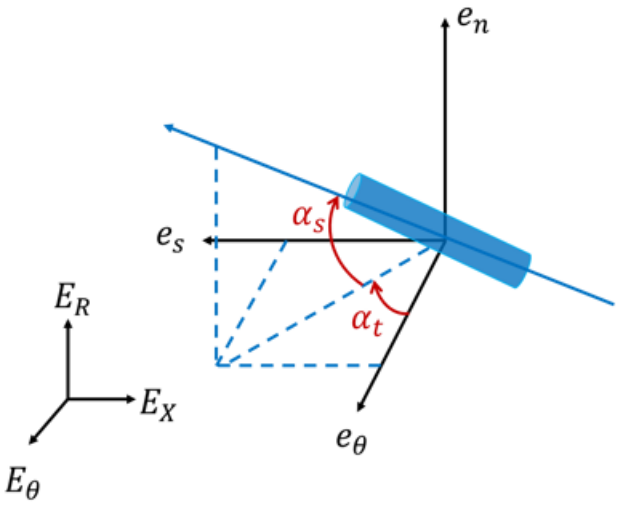
defines the injection direction by the three components of the directional vector.

Injection Angles

defines the injection direction by the surface and tangential angles. These are the angles of the cylinder axis α_s and α_t in the local cylindrical coordinate system (e_n, e_θ, e_s) as shown in Figure 7.66: Injection Surface and Tangential Angles (p. 1494).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.66: Injection Surface and Tangential Angles



The directions of the local cylindrical coordinate system are:

- e_n is the outward normal to the perforated wall surface
- e_θ has the same direction as the circumferential direction E_θ of the global cylindrical coordinate system (E_θ, E_X, E_R), where E_X and E_R are the axial and radial directions, respectively.
- e_s is the direction of the cross product e_s=e_θ×e_n.

ii. For each hole with the ID set in the **Hole ID** field, you need to specify the following parameters in the **Dynamic Method Parameters** group box (depending on your selection of the injection direction method):

- For all injection direction methods: **ACd** (the coefficient AC_D in Equation 7.131 (p. 1485) for an incompressible flow or Equation 7.132 (p. 1485) for a compressible flow)
- For the **Direction** method: Injection direction vector (**X Direction**, **Y Direction**, and **Z Direction**)
- For the **Injection Angles** method: Injection direction angles (**Surface Angle** and **Tangential Angle**)

iii. Click **OK** to close the **Dynamic Setup** dialog box.

c. If you have selected the **Discrete** injection type, you need to specify the following:

i. Select the **Dynamic Method Parameter** from the following options:

Cd

is C_D in Equation 7.131 (p. 1485) for an incompressible flow or Equation 7.132 (p. 1485) for a compressible flow

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ACd

Is AC_D in Equation 7.131 (p. 1485) for an incompressible flow or Equation 7.132 (p. 1485) for an compressible flow

ii. For each hole with the ID set in the **Hole ID** field, you need to enter the value for **Cd** (if you selected **Cd** for the **Dynamic Method Parameter**) or **ACd** (if you selected **ACd** for the **Dynamic Method Parameter**) in the **Dynamic Method Parameters** group box.

d. Click **OK** to close the **Dynamic Setup** dialog box.

e. Select the **Injection Formulation** (**Injection Method** group box) from the following options:

- **Incompressible**: Uses Equation 7.131 (p. 1485) to calculate the injection mass flow rate at the hole exit
- **Compressible**: Uses Equation 7.132 (p. 1485) to calculate the injection mass flow rate at the hole exit

f. (optional) By default, Ansys Fluent uses a constant value for C_D in Equation 7.131 (p. 1485). For the **Discrete** injection type with the **Discrete on Extraction Side** option, you can use a custom formulation for C_D by selecting the **Use Dynamic Cd?** option and hooking your `DEFINE_PERFORATED_CD` UDF to the code. See `DEFINE_PERFORATED_CD` in the *Fluent Customization Manual* for more information.

11. (cases with solid zone(s) only) If you want to simulate the effects of conjugate heat transfer between the solid walls and the injection flow, enable the **CHT Coupling** option. This option is available when the following options are enabled:

- **Coupled with Extraction Side** (**Model Options** group box)
- **Discrete** (**Injection Type** group box)
- **Discrete on Extraction Side** (**Injection Type** group box)

Note that the **Injection Wall** and **Extraction Wall** must be coupled, that is, they should be a wall/wall-shadow-pair.

See Modeling Concept (p. 1483) for more information about conjugate heat transfer coupling.

12. Repeat steps 3-11 for each wall ID where you want to define the perforated wall boundary condition.

7.4.16.5. Perforated Wall File Format

A perforated wall file is an ASCII file containing information for setting a perforated wall boundary condition on selected walls. The file can contain comment lines, which are lines that begin with either a forward slash (`/`), a semicolon (`;`), or an exclamation mark (`!`).

The following is an example of an input file that sets perforated wall boundary conditions on two walls for a 3D case:

```
// Sample Input File for Effusion Cooling Model
// total number of effusion walls (pairs)
```

```
2
// URF for mass flow calculations
0.6
//+++++++++++++++++++ First Pair ++++++++++++++++++++++++++
// Injection Wall Name
effusion-down:002
// Coupling Method
// Format: coupled [extraction wall name]
coupled effusion-up
// Injection Type
// "uniform" or "discrete" [cylinder method] [discrete on extraction side?]
discrete point-point ext-discrete
// Injection Method
// "dynamic" [mass flow rate formulation] ("incompressible" or "compressible")
dynamic compressible
// Model Inputs
// The header line starts with a '%', the subsequent lines define injection parameters
%   x1       y1       z1       x2       y2       z2   diameter    cd
0.010000 0.001000 0.010000 0.005000 0.001000 0.011000 0.000500 0.500000
0.010000 0.002000 0.010000 0.005000 0.002000 0.011000 0.000500 0.500000
...
0.025000 0.010000 0.010000 0.020000 0.010000 0.011000 0.000500 0.500000
//+++++++++++++++++++ Second Pair ++++++++++++++++++++++++++
// Injection Wall Name
effusion-down
// Coupling Method
// "coupled" [extraction wall name]
coupled effusion-up
// Injection Type
// "uniform" or "discrete" [cylinder method] [discrete on extraction side?]
discrete point-point ext-discrete
// Injection Method
// "dynamic" [mass flow rate formulation] ("incompressible" or "compressible")
dynamic compressible
// Model Inputs
// The header line starts with a '%', the subsequent lines define injection parameters
%   x1       y1       z1       x2       y2       z2   diameter    cd
0.032000 0.001000 0.010000 0.027000 0.001000 0.011000 0.000500 0.500000
0.032000 0.002000 0.010000 0.027000 0.002000 0.011000 0.000500 0.500000
...
0.042000 0.010000 0.010000 0.037000 0.010000 0.011000 0.000500 0.500000
```

The format of the perforated wall input file (excluding comment and blank lines) is as follows:

- The first line contains the number of the walls to which you want to assign the perforated wall boundary condition (a single integer number). You will need to define each perforated wall separately.
- Line 2 provides the injection explicit relaxation factor (a single float number between 0 and 1).
- Line 3 is the name of the injection wall. Lines that follow Line 3 contain parameters that define the perforated wall boundary condition for this wall.
- Line 4 contains the following keyword:
  - Model options: `coupled` or `uncoupled`.
  - (`coupled` injections only) The name of the coupled extraction wall.
- Line 5 contains the following keywords:
  - Injection type: `uniform` or `discrete`

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - (`discrete` injection type only) Perforated hole definition method: `point-point` or `point-direction`

  - (`uniform` injection type only) Injection direction definition method: `normal` (normal to boundary), `direction` (direction vector), or `angle` (injection angles)

  - Optional keyword `ext-discrete` for using the discrete condition on the extraction wall

  In the above example, Line 5 specifies the `discrete` injection type, the `point-point` hole definition method, and the `ext-discrete` condition on the extraction wall.

- Line 6 contains the following keywords:

  - Injection method (`static` or `dynamic`)

  - (optional, `dynamic` injection method only) `cd-profile` to use a UDF to calculate C_D in Equation 7.131 (p. 1485). If this keyword is not present, Ansys Fluent uses a constant value of C_D.

  - (`dynamic` injection method only) Mass flow rate formulation (`incompressible` or `compressible`) described in Equation 7.131 (p. 1485) and Equation 7.132 (p. 1485), respectively.

  - (optional, cases with solid zone(s) only) `cht-coupling` to consider the effects of heat transfer between the solid walls and the injection flow. This item is only valid for `coupled discrete` injections with the `dynamic` injection method.

- Line 7 begins with the percentage character (%) and contains the header information for the perforated wall inputs. The following keywords can be used in the header:

  ```
  x, y, z, vx, vy, vz, x1, y1, z1, x2, y2, z2, diameter, nx, ny, nz,
  alpha-s, alpha-t, mf, tem, fmean, pv, species names
  ```

  Note:

  The keyword and inputs for the Z coordinate are required for 3D cases only.

  The keywords may be provided in any order. However, the hole and injection data in the subsequent lines must follow the order you have defined in the header line.

  Depending on the selection of the injection type and dynamic conditions, different data must be provided as further described.

- The subsequent lines describe perforated hole parameters. Each line contains input data for each individual hole. The input values and keywords in each line must follow the order you have specified in the header (Line 7). The inputs are described below along with the keywords that must appear in the header line:

  - (`discrete` injection type only) Hole geometry information:

    → For the `point-point` hole definition method: the coordinates of two points on the hole axis and the diameter. The header keywords for these variables are:

    ```
    x1, y1, z1, x2, y2, z2, diameter
    ```

→ For the `point-direction` hole definition method: the coordinates of the point on the hole axis, the direction vector, and the diameter. The header keywords are:

```
x, y, z, vx, vy, vz, diameter
```

– (both uniform and discrete injection types) Coefficient for the injection mass flow rate:

→ For the `uniform` injection type: AC_D (in Equation 7.131 (p. 1485) or Equation 7.132 (p. 1485)). The header keyword is `acd`.

→ For the `discrete` injection type: either C_D or AC_D (in Equation 7.131 (p. 1485) or Equation 7.132 (p. 1485)). The header keywords are respectively: `cd` or `acd`.

– (`uniform` injection type only) Injection direction parameters:

→ For the `direction` injection direction definition method: the coordinates of the direction vector. The header keywords are: `nx`, `ny`, `nz`.

→ For the `angle` injection direction definition method: the surface angle and the tangential angle. The header keywords are: `alpha-s`, `alpha-t`.

– (`static` injection method only) Static injection parameters:

→ Mass flow rate of injection wall. The header keyword is `mf`.

→ (if the energy equation is enabled) Temperature of the injection flow. The header keyword is `tem`.

→ (if the species transport model is enabled): Species mass fractions. The header keywords are the species names present in the mixture material. You only need to specify non-zero values of species mass fractions in the injection flow, and the rest of the species in the mixture are automatically assumed to have zero mass fractions.

→ (if the PDF turbulent combustion model is enabled): the mean mixture fraction and/or progress variable. The header keywords are `fmean` and `pv`, respectively.

For example, for the uniform injection with the `angle` direction specification, each line must contain inputs for the following keywords:

```
acd alpha-s alpha-t
```

The number of input lines determines the number of perforated holes in the wall.

- If the perforated wall boundary condition is assigned to more than one wall in your case, provide information for each wall as described above.

7.4.16.6. Postprocessing for Perforated Walls

When the perforated wall model is enabled, you can generate graphical plots or alphanumeric reports of the following field variables. These quantities are available in the **Perforated Walls...** category:

- **Tagged Face**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Injection Hole Mass Flow Rate**
- **Injection Pressure**
- **Conjugate Heat Transfer**

See Alphabetical Listing of Field Variables and Their Definitions (p. 4205) for their definitions.

7.4.17. Symmetry Boundary Conditions

Symmetry boundary conditions are used when the physical geometry of interest, and the expected pattern of the flow/thermal solution, have mirror symmetry. They can also be used to model zero-shear slip walls in viscous flows. This section describes the treatment of the flow at symmetry planes and provides examples of the use of symmetry. You do not define any boundary conditions at symmetry boundaries, but you must take care to correctly define your symmetry boundary locations.

Important:

At the centerline of an axisymmetric geometry, you should use the axis boundary type rather than the symmetry boundary type, as illustrated in Figure 7.74: Use of an Axis Boundary as the Centerline in an Axisymmetric Geometry (p. 1504). See Axis Boundary Conditions (p. 1504) for details.

7.4.17.1. Examples of Symmetry Boundaries

Symmetry boundaries are used to reduce the extent of your computational model to a symmetric subsection of the overall physical system. Figure 7.67: Use of Symmetry to Model One Quarter of a 3D Duct (p. 1499) and Figure 7.68: Use of Symmetry to Model One Quarter of a Circular Cross-Section (p. 1500) illustrate two examples of symmetry boundary conditions used in this way.

Figure 7.67: Use of Symmetry to Model One Quarter of a 3D Duct

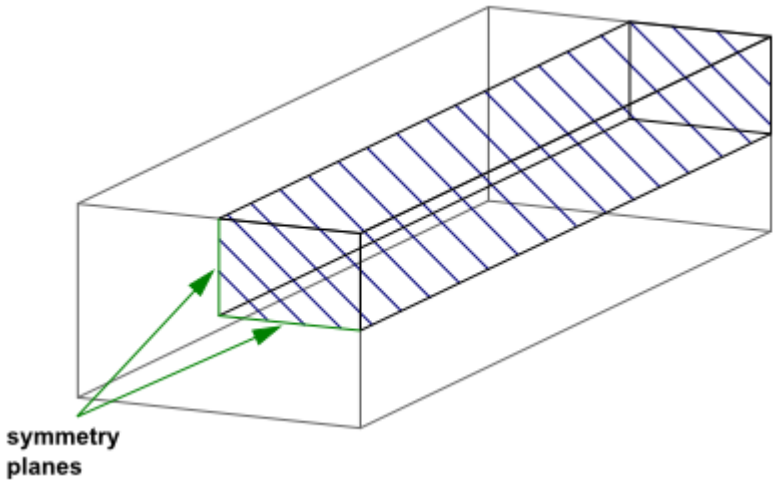


Figure 7.68: Use of Symmetry to Model One Quarter of a Circular Cross-Section

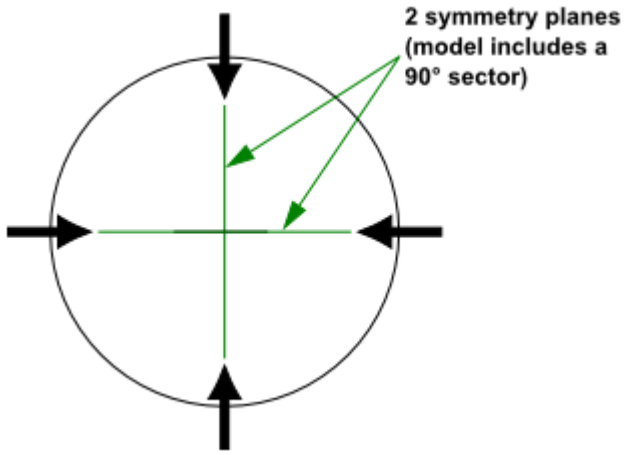
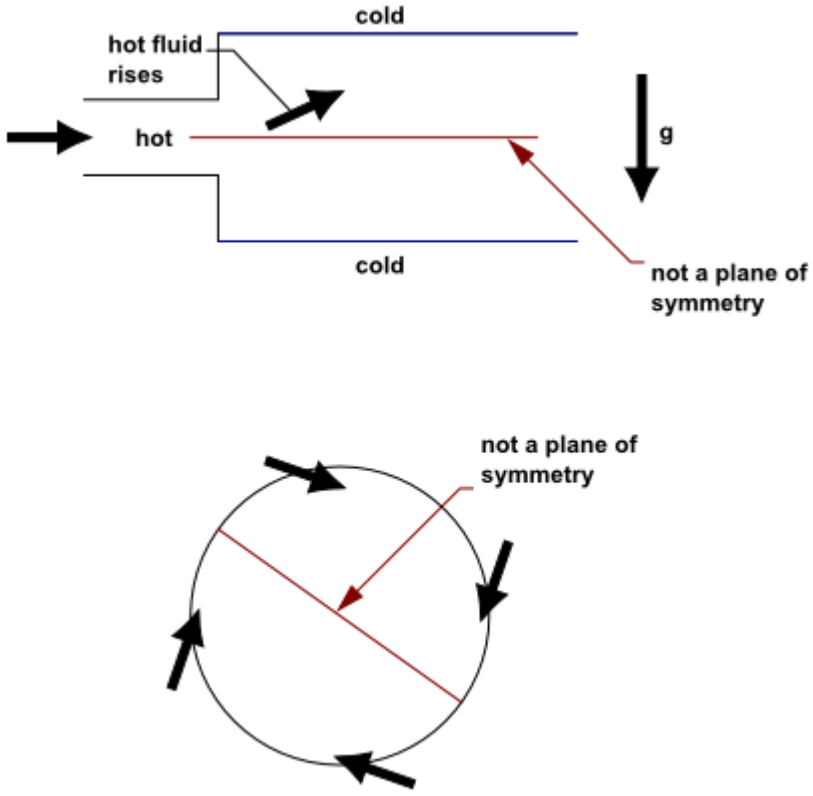


Figure 7.69: Inappropriate Use of Symmetry (p. 1500) illustrates two problems in which a symmetry plane would be *inappropriate*. In both examples, the problem geometry is symmetric but the flow itself does not obey the symmetry boundary conditions. In the first example, buoyancy creates an asymmetric flow. In the second, swirl in the flow creates a flow normal to the would-be symmetry plane. Note that this second example should be handled using rotationally periodic boundaries (as illustrated in Figure 7.70: Use of Periodic Boundaries to Define Swirling Flow in a Cylindrical Vessel (p. 1501)).

Figure 7.69: Inappropriate Use of Symmetry



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.17.2. Calculation Procedure at Symmetry Boundaries

Ansys Fluent assumes a zero flux of all quantities across a symmetry boundary. There is no convective flux across a symmetry plane: the normal velocity component at the symmetry plane is therefore zero. There is no diffusion flux across a symmetry plane: the normal gradients of all flow variables are therefore zero at the symmetry plane. The symmetry boundary condition can therefore be summarized as follows:

- zero normal velocity at a symmetry plane
- zero normal gradients of all variables at a symmetry plane

As stated above, these conditions determine a zero flux across the symmetry plane, which is required by the definition of symmetry. Since the shear stress is zero at a symmetry boundary, it can also be interpreted as a "slip" wall when used in viscous flow calculations.

7.4.18. Periodic Boundary Conditions

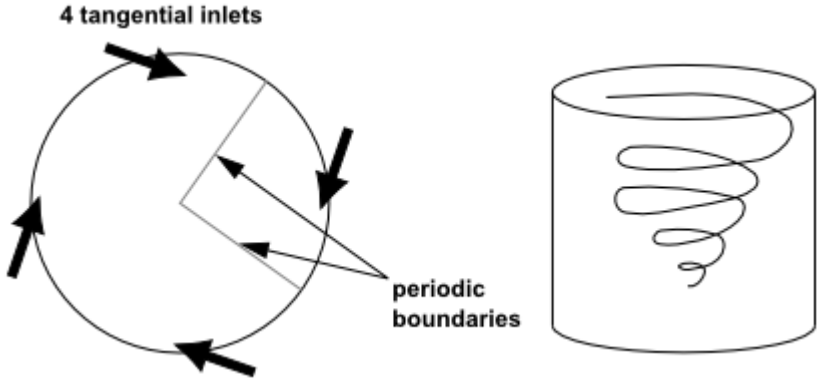
Periodic boundary conditions are used when the physical geometry of interest and the expected pattern of the flow/thermal solution have a periodically repeating nature. Two types of periodic conditions are available in Ansys Fluent. The first type does not allow a pressure drop across the periodic planes. (Note to FLUENT 4 users: This type of periodic boundary is referred to as a "cyclic" boundary in FLUENT 4.) The second type allows a pressure drop to occur across translationally periodic boundaries, enabling you to model "fully-developed" periodic flow. (In FLUENT 4 this is a "periodic" boundary.)

This section discusses the no-pressure-drop periodic boundary condition. A complete description of the fully-developed periodic flow modeling capability is provided in Periodic Flows (p. 1747).

7.4.18.1. Examples of Periodic Boundaries

Periodic boundary conditions are used when the flows across two opposite planes in your computational model are identical. Figure 7.70: Use of Periodic Boundaries to Define Swirling Flow in a Cylindrical Vessel (p. 1501) illustrates a typical application of periodic boundary conditions. In this example the flow entering the computational model through one periodic plane is identical to the flow exiting the domain through the opposite periodic plane. Periodic planes are always used in pairs as illustrated in this example.

Figure 7.70: Use of Periodic Boundaries to Define Swirling Flow in a Cylindrical Vessel



7.4.18.2. Inputs for Periodic Boundaries

For a periodic boundary without any pressure drop, there is only one input you need to consider: whether the geometry is rotationally or translationally periodic. (Additional inputs are required for a periodic flow with a periodic pressure drop. See Periodic Flows (p. 1747).)

Rotationally periodic boundaries are boundaries that form an included angle about the centerline of a rotationally symmetric geometry. Figure 7.70: Use of Periodic Boundaries to Define Swirling Flow in a Cylindrical Vessel (p. 1501) illustrates rotational periodicity. Translationally periodic boundaries are boundaries that form periodic planes in a rectilinear geometry. Figure 7.71: Example of Translational Periodicity - Physical Domain (p. 1502) illustrates translationally periodic boundaries.

Figure 7.71: Example of Translational Periodicity - Physical Domain

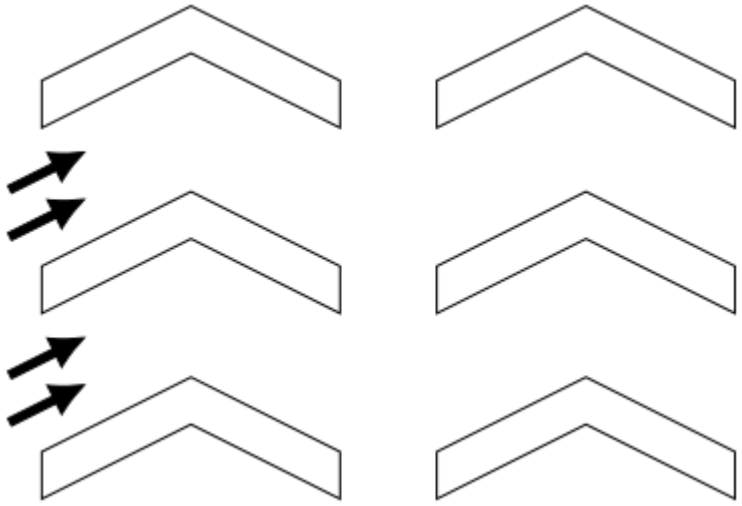
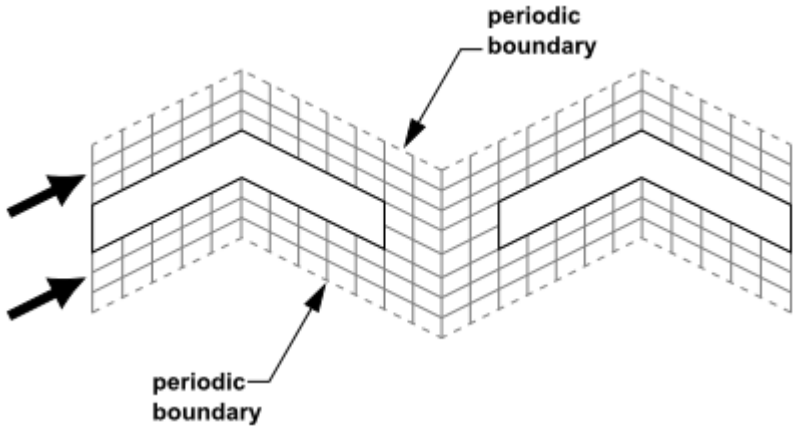


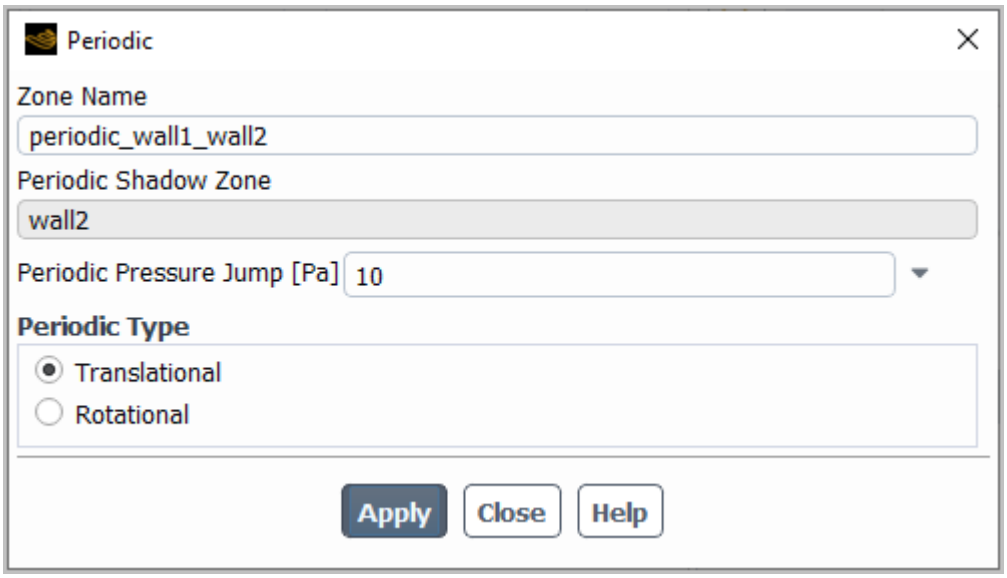
Figure 7.72: Example of Translational Periodicity - Modeled Domain



You will specify translational or rotational periodicity for a periodic boundary in the Periodic Dialog Box (p. 5041) (Figure 7.73: The Periodic Dialog Box (p. 1503)), which is opened from the **Boundary Conditions** task page (as described in Setting Cell Zone and Boundary Conditions (p. 1303)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.73: The Periodic Dialog Box



Note that there will be an additional item in the **Periodic** dialog box for the density-based solver, which allows you to specify the periodic pressure jump. See Periodic Flows (p. 1747) for details.

If the domain is rotationally periodic, select **Rotational** as the **Periodic Type**; if it is translationally periodic, select **Translational**. For rotationally periodic domains, the solver will automatically compute the angle through which the periodic zone is rotated. The axis used for this rotation is the axis of rotation specified for the adjacent cell zone.

Note that there is no need for the adjacent cell zone to be moving for you to use a rotationally periodic boundary. You could, for example, model pipe flow in 3D using a stationary reference frame with a pie-slice of the pipe; the sides of the slice would require rotational periodicity.

Click the **Check** button in the **Mesh** group box of the **Domain** tab (see Checking the Mesh (p. 1245)) to compute and display the minimum, maximum, and average rotational angles of all faces on periodic boundaries. If the difference between the minimum, maximum, and average values is not negligible, then there is a problem with the mesh: the mesh geometry is not periodic about the specified axis.

7.4.18.3. Default Settings at Periodic Boundaries

By default, all periodic boundaries are translational.

7.4.18.4. Calculation Procedure at Periodic Boundaries

Ansys Fluent treats the flow at a periodic boundary as though the opposing periodic plane is a direct neighbor to the cells adjacent to the first periodic boundary. Therefore, when calculating the flow through the periodic boundary adjacent to a fluid cell, the flow conditions at the fluid cell adjacent to the opposite periodic plane are used.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

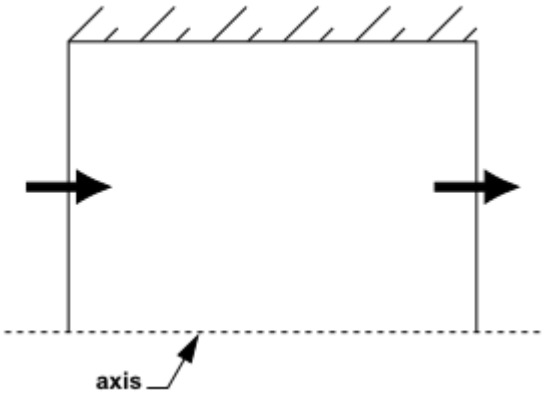
7.4.19. Axis Boundary Conditions

The axis boundary type must be used as the centerline of an axisymmetric geometry (see Figure 7.74: Use of an Axis Boundary as the Centerline in an Axisymmetric Geometry (p. 1504)). It can also be used for the centerline of a cylindrical-polar quadrilateral or hexahedral mesh (for example, a mesh created for a structured-mesh code such as FLUENT 4). You do not need to define any boundary conditions at axis boundaries.

Important:

When creating 2D axisymmetric geometry, the axis boundary must lie on the y=0 line.

Figure 7.74: Use of an Axis Boundary as the Centerline in an Axisymmetric Geometry



7.4.19.1. Calculation Procedure at Axis Boundaries

To determine the appropriate physical value for a particular variable at a point on the axis, Ansys Fluent uses the cell value in the adjacent cell.

7.4.20. Fan Boundary Conditions

The fan model is a lumped parameter model that can be used to determine the impact of a fan with known characteristics upon some larger flow field. The fan boundary type allows you to input an empirical fan curve that governs the relationship between head (pressure rise) and flow rate (velocity) across a fan element. You can also specify radial and tangential components of the fan swirl velocity. The fan model does not provide an accurate description of the detailed flow through the fan blades. Instead, it predicts the amount of flow through the fan. Fans may be used in conjunction with other flow sources, or as the sole source of flow in a simulation. In the latter case, the system flow rate is determined by the balance between losses in the system and the fan curve.

Ansys Fluent also provides a connection for a special user-defined fan model that updates the pressure jump function during the calculation. This feature is described in User-Defined Fan Model (p. 1556).

7.4.20.1. Limitations of Fan Boundary Conditions

The following are limitations when using fan boundary conditions:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- By default, fan boundaries are included in the solar flux calculation when using the Solar Load model with no option to disable.

7.4.20.2. Fan Equations

7.4.20.2.1. Modeling the Pressure Rise Across the Fan

A fan is considered to be infinitely thin, and the discontinuous pressure rise across it is specified as a function of the velocity through the fan. The relationship may be a constant, a polynomial, piecewise-linear, or piecewise-polynomial function, or a user-defined function.

In the case of a polynomial, the relationship is of the form

$$\Delta p = \sum_{n=1}^{N} f_n v^{n-1} \tag{7.138}$$

where Δp is the pressure jump, f_n are the pressure-jump polynomial coefficients, and v is the magnitude of the local fluid velocity normal to the fan.

Important:

The velocity v can be either positive or negative. You must be careful to model the fan so that a pressure rise occurs for forward flow through the fan.

You can, optionally, use the mass-averaged velocity normal to the fan to determine a single pressure-jump value for all faces in the fan zone.

7.4.20.2.2. Modeling the Fan Swirl Velocity

For three-dimensional problems, the values of the convected tangential and radial velocity fields can be imposed on the fan surface to generate swirl. These velocities can be specified as functions of the radial distance from the fan center. The relationships may be constant or polynomial functions, or user-defined functions.

Important:

You must use SI units for all fan swirl velocity inputs.

For the case of polynomial functions, the tangential and radial velocity components can be specified by the following equations:

$$U_\theta = \sum_{n=-1}^{N} f_n r^n; -1 \le N \le 6 \tag{7.139}$$

$$U_r = \sum_{n=-1}^{N} g_n r^n; -1 \le N \le 6 \tag{7.140}$$

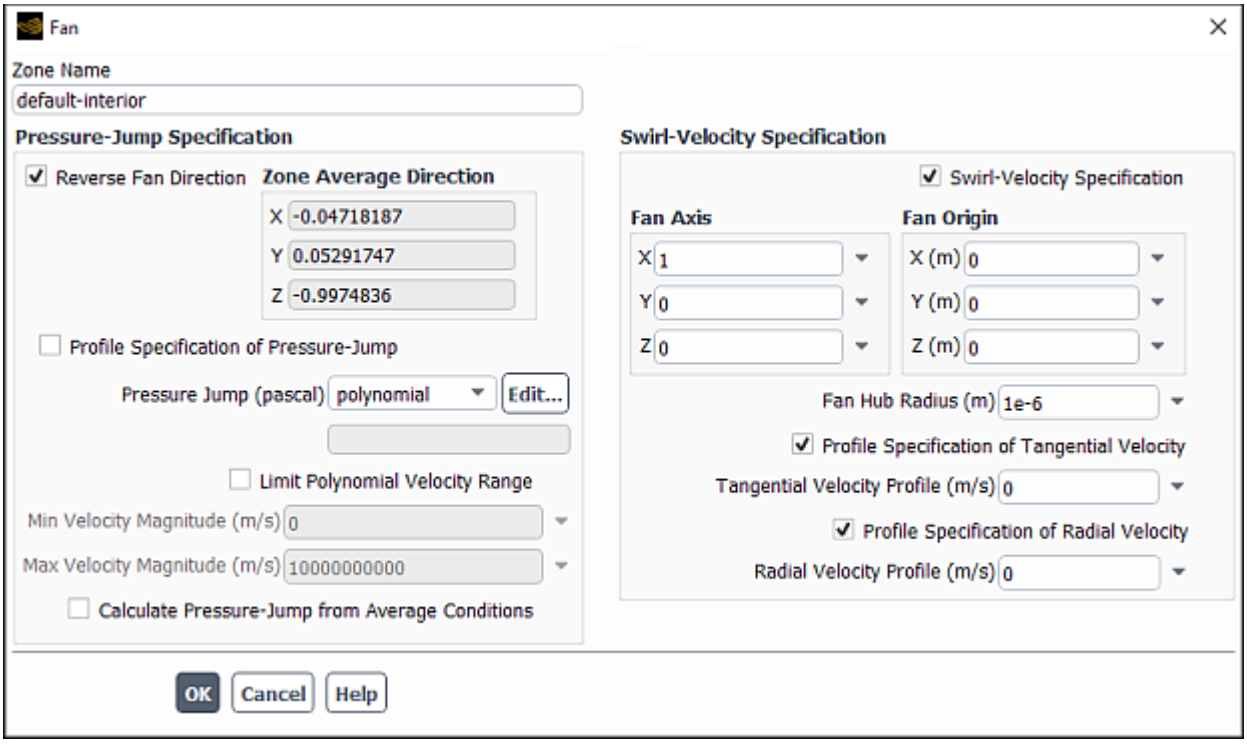
where U_θ and U_r are, respectively, the tangential and radial velocities on the fan surface in m/s, f_n and g_n are the tangential and radial velocity polynomial coefficients, and r is the distance to the fan center.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.20.3. User Inputs for Fans

Once the fan zone has been identified (in the **Boundary Conditions** task page), you will set all modeling inputs for the fan in the Fan Dialog Box (p. 5002) (Figure 7.75: The Fan Dialog Box (p. 1506)), which is opened from the Boundary Conditions Task Page (p. 4991) (as described in Setting Cell Zone and Boundary Conditions (p. 1303)).

Figure 7.75: The Fan Dialog Box



Inputs for a fan are as follows:

1. Identify the fan zone.
2. Define the pressure jump across the fan.
3. Define the discrete phase boundary condition for the fan (for discrete phase calculations).
4. Define the swirl velocity, if desired (3D only).

7.4.20.3.1. Identifying the Fan Zone

Since the fan is considered to be infinitely thin, it must be modeled as the interface between cells, rather than a cell zone. Therefore the fan zone is a type of internal face zone (where the faces are line segments in 2D or triangles/quadrilaterals in 3D). If, when you read your mesh into Ansys Fluent, the fan zone is identified as an **interior** zone, right-click the zone in the tree and use the **Type** sub-menu to change it to a **fan** zone (for more details, see Changing Cell and Boundary Zone Types (p. 1301)).

Once the interior zone has been changed to a fan zone, you can open the **Fan** dialog box and specify the pressure jump and, optionally, the swirl velocity.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.20.3.2. Defining the Pressure Jump

To define the pressure jump, you will specify a polynomial, piecewise-linear, or piecewise-polynomial function of velocity, a user-defined function, or a constant value. You should also check the **Zone Average Direction** vector to be sure that a pressure rise occurs for forward flow through the fan. The **Zone Average Direction**, calculated by the solver, is the face-averaged direction vector for the fan zone. If this vector is pointing in the direction you want the fan to blow, do not select **Reverse Fan Direction**; if it is pointing in the opposite direction, select **Reverse Fan Direction**.

7.4.20.3.2.1. Polynomial, Piecewise-Linear, or Piecewise-Polynomial Function

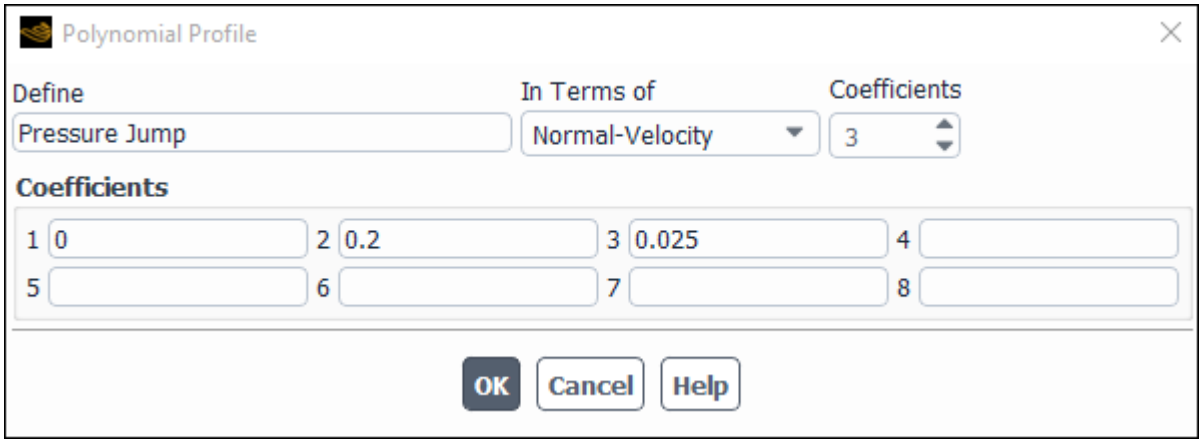
Follow these steps to set a polynomial, piecewise-linear, or piecewise-polynomial function for the pressure jump:

1. Check that the **Profile Specification of Pressure-Jump** option is off in the Fan Dialog Box (p. 5002).

2. Choose **polynomial**, **piecewise-linear**, or **piecewise-polynomial** in the drop-down list to the right of **Pressure-Jump**. (If the function type you want is already selected, you can click the **Edit...** button to open the dialog box where you will define the function.)

3. In the dialog box that appears for the definition of the **Pressure Jump** function (for example, Figure 7.76: Polynomial Profile Dialog Box for Pressure Jump Definition (p. 1507)), enter the appropriate values. These profile input dialog boxes are used the same way as the profile input dialog boxes for temperature-dependent properties. See Defining Properties Using Temperature-Dependent Functions (p. 1616) to find out how to use them.

   Figure 7.76: Polynomial Profile Dialog Box for Pressure Jump Definition



4. Set any of the optional parameters described below. (optional)

When you define the pressure jump using any of these types of functions, you can choose to limit the minimum and maximum velocity magnitudes used to calculate the pressure jump.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Enabling the **Limit Polynomial Velocity Range** option limits the pressure jump when a **Min Velocity Magnitude** and a **Max Velocity Magnitude** are specified.

Important:

The values corresponding to the **Min Velocity Magnitude** and the **Max Velocity Magnitude** do not limit the flow field velocity to this range. However, this range does limit the value of the pressure jump, which is a polynomial and a function of velocity, as seen in Equation 7.138 (p. 1505). If the calculated normal velocity magnitude exceeds the **Max Velocity Magnitude** that has been specified, then the pressure jump at the **Max Velocity Magnitude** value will be used. Similarly, if the calculated velocity is less than the specified **Min Velocity Magnitude**, the pressure jump at the **Min Velocity Magnitude** will be substituted for the pressure jump corresponding to the calculated velocity.

You also have the option to use the mass-averaged velocity normal to the fan to determine a single pressure-jump value for all faces in the fan zone. Turning on **Calculate Pressure-Jump from Average Conditions** enables this option.

7.4.20.3.2.2. Constant Value

To define a constant pressure jump, follow these steps:

1. Turn off the **Profile Specification of Pressure-Jump** option in the Fan Dialog Box (p. 5002).
2. Choose **constant** in the drop-down list to the right of **Pressure-Jump**.
3. Enter the value for Δp in the **Pressure-Jump** field.

You can follow the procedure below, if it is more convenient:

1. Turn on the **Profile Specification of Pressure-Jump** option.
2. Select **constant** in the drop-down list below **Pressure Jump Profile**, and enter the value for Δp in the **Pressure Jump Profile** field.

7.4.20.3.2.3. User-Defined Function or Profile

For a user-defined pressure-jump function or a function defined in a boundary profile file, you will follow these steps:

1. Turn on the **Profile Specification of Pressure-Jump** option.
2. Choose the appropriate function in the drop-down list below **Pressure Jump Profile**.

Important:

The profile must be written in ascending order with smallest pressure jump listed at the top. See Example: Determining the Pressure Jump Function (p. 1509).

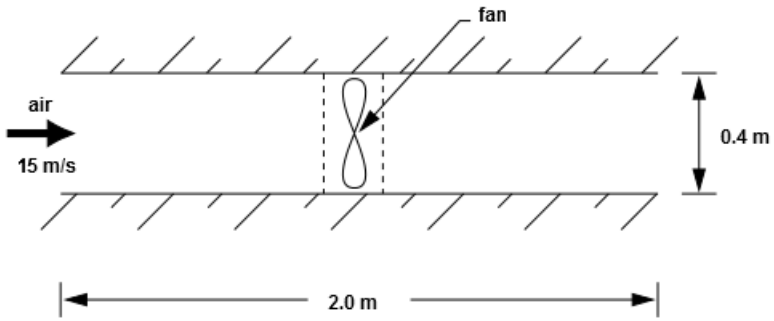
See the Fluent Customization Manual for information about user-defined functions, and Profiles (p. 1564) for details about profile files.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.20.3.2.4. Example: Determining the Pressure Jump Function

This example shows you how to determine the function for the pressure jump. Consider the simple two-dimensional duct flow illustrated in Figure 7.77: A Fan Located In a 2D Duct (p. 1509). Air at constant density enters the 2.0 m × 0.4 m duct with a velocity of 15 m/s. Centered in the duct is a fan.

Figure 7.77: A Fan Located In a 2D Duct



Assume that the fan characteristics are as follows when the fan is operating at 2000 rpm:

Q (m^3 /s)	Δp (Pa)
25	0.0
20	175
15	350
10	525
5	700
0	875

where Q is the flow through the fan and Δp is the pressure rise across the fan. The fan characteristics in this example follow a simple linear relationship between pressure rise and flow rate. To convert this into a relationship between pressure rise and velocity, the cross-sectional area of the fan must be known. In this example, assuming that the duct is 1.0 m deep, this area is 0.4 m^2, so that the corresponding velocity values are as follows:

v (m/s)	Δp (Pa)
62.5	0.0
50.0	175
37.5	350
25.0	525
12.5	700
0	875

The polynomial form of this relationship is the following equation for a line:

$$\Delta p = 875 - 14v \tag{7.141}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.20.3.3. Defining Discrete Phase Boundary Conditions for the Fan

If you are modeling a discrete phase of particles, you can set the fate of particle trajectories at the fan. See Setting Boundary Conditions for the Discrete Phase (p. 2779) for details.

7.4.20.3.4. Defining the Fan Swirl Velocity

If you want to set tangential and radial velocity fields on the fan surface to generate swirl in a 3D problem, follow these steps:

1. Turn on the **Swirl-Velocity Specification** option in the Fan Dialog Box (p. 5002).
2. Specify the fan's axis of rotation by defining the axis origin (**Fan Origin**) and direction vector (**Fan Axis**).
3. Set the value for the radius of the fan's hub (**Fan Hub Radius**). The default is 1×10^{-6} to avoid division by zero in the polynomial.
4. Set the tangential and radial velocity functions as polynomial functions of radial distance, constant values, or user-defined functions.

Important:

You must use SI units for all fan swirl velocity inputs.

7.4.20.3.4.1. Polynomial Function

To define a polynomial function for tangential or radial velocity, follow the steps below:

1. Check that the **Profile Specification of Tangential Velocity** or **Profile Specification of Radial Velocity** option is off in the Fan Dialog Box (p. 5002).
2. Enter the coefficients f_n in Equation 7.139 (p. 1505) or g_n in Equation 7.140 (p. 1505) in the **Tangential-** or **Radial-Velocity Polynomial Coefficients** field. Enter f_{-1} first, then f_0, and so on. Separate each coefficient by a blank space. Remember that the first coefficient is for $\frac{1}{r}$.

7.4.20.3.4.2. Constant Value

To define a constant tangential or radial velocity, the steps are as follows:

1. Turn on the **Profile Specification of Tangential Velocity** or **Profile Specification of Radial Velocity** option in the Fan Dialog Box (p. 5002).
2. Select **constant** in the drop-down list under **Tangential** or **Radial Velocity Profile**.
3. Enter the value for U_θ or U_r in the **Tangential** or **Radial Velocity Profile** field.

You can follow the procedure below, if it is more convenient:

1. Turn off the **Profile Specification of Tangential Velocity** or **Profile Specification of Radial Velocity** option in the Fan Dialog Box (p. 5002).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Enter the value for U_θ or U_r in the **Tangential-** or **Radial-Velocity Polynomial Coefficients** field.

7.4.20.3.4.3. User-Defined Function or Profile

For a user-defined tangential or radial velocity function or a function contained in a profile file, follow the procedure below:

1. Turn on the **Profile Specification of Tangential** or **Radial Velocity** option.

2. Choose the appropriate function from the drop-down list under **Tangential** or **Radial Velocity Profile**.

See the Fluent Customization Manual for information about user-defined functions, and Profiles (p. 1564) for details about profile files.

7.4.20.4. Postprocessing for Fans

7.4.20.4.1. Reporting the Pressure Rise Through the Fan

You can use the Surface Integrals Dialog Box (p. 5325) to report the pressure rise through the fan, as described in Surface Integration (p. 4150). There are two steps to this procedure:

1. Create a surface on each side of the fan zone. Use the Transform Surface Dialog Box (p. 5633) (as described in Transforming Surfaces (p. 3843)) to translate the fan zone slightly upstream and slightly downstream to create two new surfaces.

2. In the **Surface Integrals** dialog box, report the average **Static Pressure** just upstream and just downstream of the fan. You can then calculate the pressure rise through the fan.

7.4.20.4.2. Graphical Plots

Graphical reports of interest with fans are as follows:

- Contours or profiles of **Static Pressure** and **Static Temperature**.
- XY plots of **Static Pressure** and **Static Temperature** vs position.

Displaying Graphics (p. 3869) explains how to generate graphical displays of data.

Important:

When generating these plots, be sure to turn off the display of node values so that you can see the different values on each side of the fan. (If you display node values, the cell values on either side of the fan will be averaged to obtain a node value, and you will not see, for example, the pressure jump across the fan.)

7.4.21. Radiator Boundary Conditions

A lumped-parameter model for a heat exchange element (for example, a radiator or condenser), is available in Ansys Fluent. The radiator boundary type allows you to specify both the pressure drop and heat transfer coefficient as functions of the velocity normal to the radiator.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A more detailed heat exchanger model is also available in Ansys Fluent. See Modeling Heat Exchangers (p. 2279) for details.

7.4.21.1. Radiator Equations

7.4.21.1.1. Modeling the Pressure Loss Through a Radiator

A radiator is considered to be infinitely thin, and the pressure drop through the radiator is assumed to be proportional to the dynamic head of the fluid, with an empirically determined loss coefficient that you supply. That is, the pressure drop, Δp, varies with the normal component of velocity through the radiator, v, as follows:

$$\Delta p = k_L \frac{1}{2} \rho v^2 \tag{7.142}$$

where ρ is the fluid density, and k_L is the non-dimensional loss coefficient, which can be specified as a constant or as a polynomial, piecewise-linear, or piecewise-polynomial function.

In the case of a polynomial, the relationship is of the form

$$k_L = \sum_{n=1}^{N} r_n v^{n-1} \tag{7.143}$$

where r_n are polynomial coefficients and v is the magnitude of the local fluid velocity normal to the radiator.

7.4.21.1.2. Modeling the Heat Transfer Through a Radiator

The heat flux from the radiator to the surrounding fluid is given as

$$q = h\left(T_{ext} - T_{air,d}\right) \tag{7.144}$$

where q is the heat flux, $T_{air,d}$ is the temperature downstream of the heat exchanger (radiator), and T_{ext} is the radiator temperature. The convective heat transfer coefficient, h, can be specified as a constant or as a polynomial, piecewise-linear, or piecewise-polynomial function.

For a polynomial, the relationship is of the form

$$h = \sum_{n=0}^{N} h_n v^n \quad ; \quad 0 \le N \le 7 \tag{7.145}$$

where h_n are polynomial coefficients and v is the magnitude of the local fluid velocity normal to the radiator in m/s.

To define the actual heat flux $\left(q\right)$, specify a **Temperature** of absolute zero (0 K, 0 °R, –273.15 °C, –459.67 °F), and set the constant **Heat Flux** value.

To define the radiator temperature, enter the value for T_{ext} in the **Temperature** field. To define the **Heat-Transfer-Coefficient** $\left(h\right)$, you can specify a polynomial, piecewise-linear, or piecewise-polynomial function of velocity, or a constant value.

q (either the entered value or the value calculated using Equation 7.144 (p. 1512)) is integrated over the radiator surface area.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fluent does not allow you to specify a value less than absolute zero for the radiator temperature.

7.4.21.1.2.1. Calculating the Heat Transfer Coefficient

To model the thermal behavior of the radiator, you must supply an expression for the heat transfer coefficient, h, as a function of the fluid velocity through the radiator, v. To obtain this expression, consider the heat balance equation:

$$q=\frac{\dot{m}c_p\Delta T}{A}=h\left(T_{air,d}-T_{ext}\right) \tag{7.146}$$

where

q = heat flux (W/ m^2)

$\dot{m}$ = fluid mass flow rate (kg/s)

c_p = specific heat capacity of fluid (J/kg-K)

h = empirical heat transfer coefficient (W/ m^2 -K)

T_{ext} = external temperature (reference temperature for the liquid) (K)

$T_{air,d}$ = temperature downstream from the heat exchanger (K)

A = heat exchanger frontal area (m^2)

Equation 7.146 (p. 1513) can be rewritten as

$$q=\frac{\dot{m}c_p\left(T_{air,u}-T_{air,d}\right)}{A}=h\left(T_{air,d}-T_{ext}\right) \tag{7.147}$$

where $T_{air,u}$ is the upstream air temperature. The heat transfer coefficient, h, can therefore be computed as

$$h=\frac{\dot{m}c_p\left(T_{air,u}-T_{air,d}\right)}{A\left(T_{air,d}-T_{ext}\right)} \tag{7.148}$$

or, in terms of the fluid velocity,

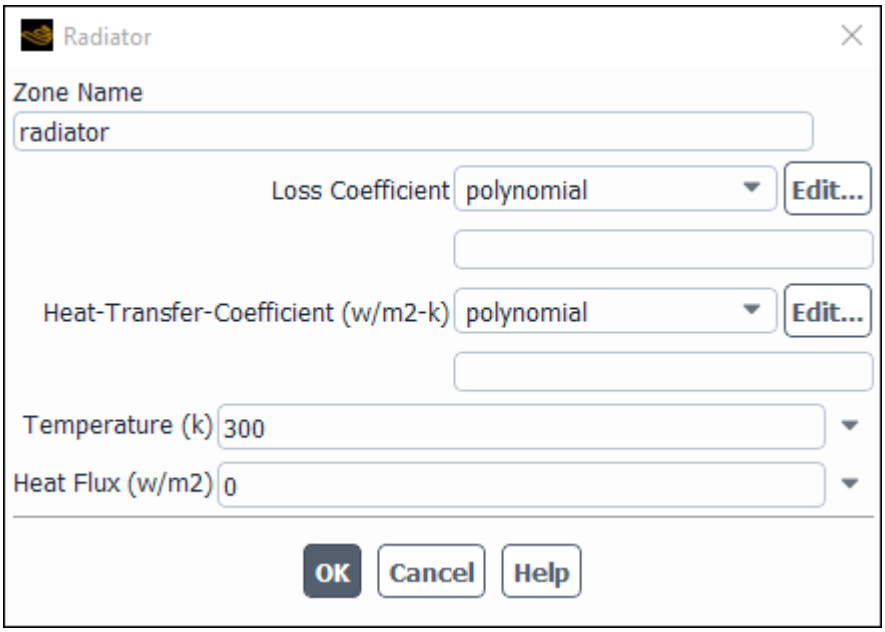
$$h=\frac{\rho vc_p\left(T_{air,u}-T_{air,d}\right)}{T_{air,d}-T_{ext}} \tag{7.149}$$

7.4.21.2. User Inputs for Radiators

Once the radiator zone has been identified (in the **Boundary Conditions** task page), you will set all modeling inputs for the radiator in the Radiator Dialog Box (p. 5069) (Figure 7.78: The Radiator Dialog Box (p. 1514)), which is opened from the Boundary Conditions Task Page (p. 4991) (as described in Setting Cell Zone and Boundary Conditions (p. 1303)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.78: The Radiator Dialog Box



The inputs for a radiator are as follows:

1. Identify the radiator zone.
2. Define the pressure loss coefficient.
3. Define either the heat flux or the heat transfer coefficient and radiator temperature.

   If defining the heat flux, specify the **Temperature** *as absolute zero.*
4. Define the discrete phase boundary condition for the radiator (for discrete phase calculations).

7.4.21.2.1. Identifying the Radiator Zone

Since the radiator is considered to be infinitely thin, it must be modeled as the interface between cells, rather than a cell zone. Therefore the radiator zone is a type of internal face zone (where the faces are line segments in 2D or triangles/quadrilaterals in 3D). If, when you read your mesh into Ansys Fluent, the radiator zone is identified as an **interior** zone, right-click the zone in the tree and use the **Type** sub-menu to change it to a **radiator** zone (for more details, see Changing Cell and Boundary Zone Types (p. 1301)).

Once the interior zone has been changed to a radiator zone, you can open the **Radiator** dialog box and specify the loss coefficient and heat flux information.

7.4.21.2.2. Defining the Pressure Loss Coefficient Function

To define the pressure loss coefficient k_L you can specify a polynomial, piecewise-linear, or piecewise-polynomial function of velocity, or a constant value.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

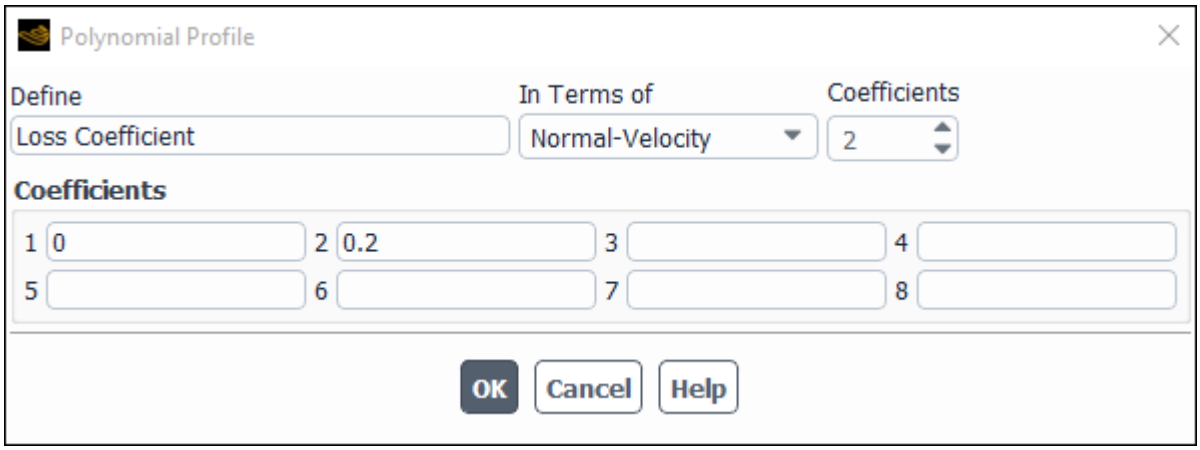
7.4.21.2.2.1. Polynomial, Piecewise-Linear, or Piecewise-Polynomial Function

Follow these steps to set a polynomial, piecewise-linear, or piecewise-polynomial function for the pressure loss coefficient:

1. Choose **polynomial**, **piecewise-linear**, or **piecewise-polynomial** in the drop-down list to the right of **Loss Coefficient**. (If the function type you want is already selected, you can click the **Edit...** button to open the dialog box where you will define the function.)

2. In the dialog box that appears for the definition of the **Loss Coefficient** function (for example, Figure 7.79: Polynomial Profile Dialog Box for Loss Coefficient Definition (p. 1515)), enter the appropriate values. These profile input dialog boxes are used the same way as the profile input dialog boxes for temperature-dependent properties. See Defining Properties Using Temperature-Dependent Functions (p. 1616) to find out how to use them.

Figure 7.79: Polynomial Profile Dialog Box for Loss Coefficient Definition



7.4.21.2.2.2. Constant Value

To define a constant loss coefficient, follow these steps:

1. Choose **constant** in the **Loss Coefficient** drop-down list.

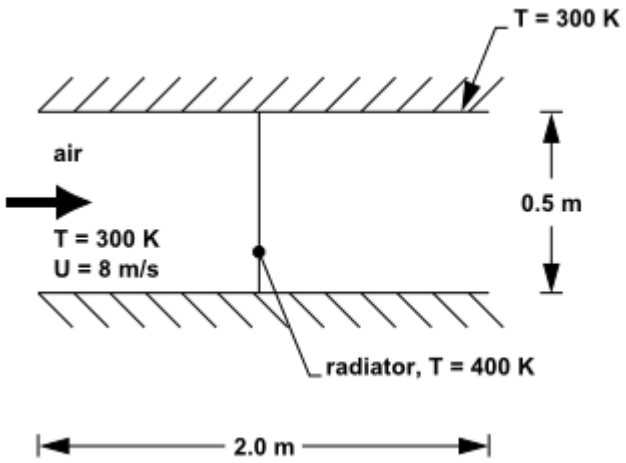
2. Enter the value for k_L in the **Loss Coefficient** field.

7.4.21.2.2.3. Example: Calculating the Loss Coefficient

This example shows you how to determine the loss coefficient function. Consider the simple two-dimensional duct flow of air through a water-cooled radiator, shown in Figure 7.80: A Simple Duct with a Radiator (p. 1516).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.80: A Simple Duct with a Radiator



The radiator characteristics must be known empirically. For this case, assume that the radiator to be modeled yields the test data shown in Table 7.2: Air-side Radiator Data (p. 1516), which was taken with a waterside flow rate of 7 kg/min and an inlet water temperature of 400.0 K. To compute the loss coefficient, it is helpful to construct a table with values of the dynamic head, $\frac{1}{2}\rho v^2$, as a function of pressure drop, Δp, and the ratio of these two values, k_L (from Equation 7.142 (p. 1512)). (The air density, defined in Figure 7.80: A Simple Duct with a Radiator (p. 1516), is 1.0 kg/ m^3.) The reduced data are shown in Table 7.3: Reduced Radiator Data (p. 1516).

Table 7.2: Air-side Radiator Data

Velocity (m/s)	Upstream Temp (K)	Downstream Temp (K)	Pressure Drop
5.0	300.0	330.0	75.0
10.0	300.0	322.5	250.0
15.0	300.0	320.0	450.0

Table 7.3: Reduced Radiator Data

v (m/s)	$\frac{1}{2}\rho v^2$ (Pa)	Δp (Pa)	k_L
5.0	12.5	75.0	6.0
10.0	50.0	250.0	5.0
15.0	112.5	450.0	4.0

The loss coefficient is a linear function of the velocity, decreasing as the velocity increases. The form of this relationship is

$$k_L = 7.0 - 0.2v \tag{7.150}$$

where v is now the absolute value of the velocity through the radiator.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.4.21.2.3. Defining the Heat Flux Parameters

As mentioned in Radiator Equations (p. 1512), you can either define the actual heat flux (q) in the **Heat Flux** field, or set the heat transfer coefficient and radiator temperature (h, T_{ext}). All inputs are in the Radiator Dialog Box (p. 5069).

To define the actual heat flux, specify a **Temperature** of `0` K, and set the constant **Heat Flux** value.

To define the radiator temperature, enter the value for T_{ext} in the **Temperature** field. To define the **Heat-Transfer-Coefficient**, you can specify a polynomial, piecewise-linear, or piecewise-polynomial function of velocity, or a constant value.

7.4.21.2.3.1. Polynomial, Piecewise-Linear, or Piecewise-Polynomial Function

Follow these steps to set a polynomial, piecewise-linear, or piecewise-polynomial function for the heat transfer coefficient:

1. Choose **polynomial**, **piecewise-linear**, or **piecewise-polynomial** in the drop-down list to the right of **Heat-Transfer-Coefficient**. (If the function type you want is already selected, you can click the **Edit...** button to open the dialog box where you will define the function.)

2. In the dialog box that appears for the definition of the **Heat-Transfer-Coefficient** function, enter the appropriate values. These profile input dialog boxes are used the same way as the profile input dialog boxes for temperature-dependent properties. See Defining Properties Using Temperature-Dependent Functions (p. 1616) to find out how to use them.

7.4.21.2.3.2. Constant Value

To define a constant heat transfer coefficient, follow these steps:

1. Choose **constant** in the **Heat-Transfer-Coefficient** drop-down list.

2. Enter the value for h in the **Heat-Transfer-Coefficient** field.

7.4.21.2.3.3. Example: Determining the Heat Transfer Coefficient Function

This example shows you how to determine the function for the heat transfer coefficient. Consider the simple two-dimensional duct flow of air through a water-cooled radiator, shown in Figure 7.80: A Simple Duct with a Radiator (p. 1516).

The data supplied in Table 7.2: Air-side Radiator Data (p. 1516) along with values for the air density (1.0 kg/ m^3) and specific heat (1000 J/kg-K) can be used to obtain the following values for the heat transfer coefficient h:

Velocity (m/s)	h (W/ m^2 -K)
5.0	2142.9
10.0	2903.2
15.0	3750.0

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The heat transfer coefficient obeys a second-order polynomial relationship (fit to the points in the table above) with the velocity, which is of the form

$$h = 1469.1 + 126.11v + 1.73v^2 \tag{7.151}$$

Note that the velocity v is assumed to be the absolute value of the velocity passing through the radiator.

7.4.21.2.4. Defining Discrete Phase Boundary Conditions for the Radiator

If you are modeling a discrete phase of particles, you can set the fate of particle trajectories at the radiator. See Setting Boundary Conditions for the Discrete Phase (p. 2779) for details.

7.4.21.3. Postprocessing for Radiators

7.4.21.3.1. Reporting the Radiator Pressure Drop

You can use the Surface Integrals Dialog Box (p. 5325) to report the pressure drop across the radiator, as described in Surface Integration (p. 4150). There are two steps to this procedure:

1. Create a surface on each side of the radiator zone. Use the Transform Surface Dialog Box (p. 5633) (as described in Transforming Surfaces (p. 3843)) to translate the radiator zone slightly upstream and slightly downstream to create two new surfaces.

2. In the **Surface Integrals** dialog box, report the average **Static Pressure** just upstream and just downstream of the radiator. You can then calculate the pressure drop across the radiator.

To check this value against the expected value based on Equation 7.142 (p. 1512), you can use the **Surface Integrals** dialog box to report the average normal velocity through the radiator. (If the radiator is not aligned with the x, y, or z axis, you will need to use the Custom Field Function Calculator Dialog Box (p. 5428) to generate a function for the velocity normal to the radiator.) Once you have the average normal velocity, you can use Equation 7.143 (p. 1512) to determine the loss coefficient and then Equation 7.142 (p. 1512) to calculate the expected pressure loss.

7.4.21.3.2. Reporting Heat Transfer in the Radiator

To determine the temperature rise across the radiator, follow the procedure outlined above for the pressure drop to generate surfaces upstream and downstream of the radiator. Then use the Surface Integrals Dialog Box (p. 5325) (as for the pressure drop report) to report the average **Static Temperature** on each surface. You can then calculate the temperature rise across the radiator.

7.4.21.3.3. Graphical Plots

Graphical reports of interest with radiators are as follows:

- Contours or profiles of **Static Pressure** and **Static Temperature**.
- XY plots of **Static Pressure** and **Static Temperature** vs position.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Displaying Graphics (p. 3869) explains how to generate graphical displays of data.

Important:

When generating these plots, be sure to turn off the display of node values so that you can see the different values on each side of the radiator. (If you display node values, the cell values on either side of the radiator will be averaged to obtain a node value, and you will not see, for example, the pressure loss across the radiator.)

7.4.22. Porous Jump Boundary Conditions

Porous jump conditions are used to model a thin "membrane" that has known velocity (pressure-drop) characteristics. It is essentially a 1D simplification of the porous media model available for cell zones. Examples of uses for the porous jump condition include modeling pressure drops through screens and filters, and modeling radiators when you are not concerned with heat transfer. This simpler model should be used whenever possible (instead of the full porous media model) because it is more robust and yields better convergence.

The thin porous medium has a finite thickness over which the pressure change is defined as a combination of Darcy's Law and an additional inertial loss term:

$$\Delta p = -\left(\frac{\mu}{\alpha} v + C_2 \frac{1}{2} \rho v^2\right) \Delta m \tag{7.152}$$

where μ is the laminar fluid viscosity, α is the permeability of the medium, C_2 is the pressure-jump coefficient, v is the velocity normal to the porous face, and Δm is the thickness of the medium. Appropriate values for α and C_2 can be calculated using the techniques described in User Inputs for Porous Media (p. 1343).

In a coupled analysis involving Fluent and System Coupling, data can be transferred to and from the porous jump boundary. For more about System Coupling, see Performing System Coupling Simulations Using Fluent (p. 4647). For more information about the data transferred, see Force transferred to System Coupling from a Porous Jump Boundary (p. 4656).

Note:

- The porous jump boundary condition considers only the momentum resistance. Diffusion processes are not considered.
- In some cases, the pressure-based coupled solver with porous jump boundary conditions may suffer from convergence issues that do not respond to changes in the coupled solver settings. This behavior depends on the specific flow configuration and porous jump boundary condition values. If convergence instability is observed using the coupled pressure-based solver, it is recommended that you change the pressure-velocity coupling to one of the segregated schemes.

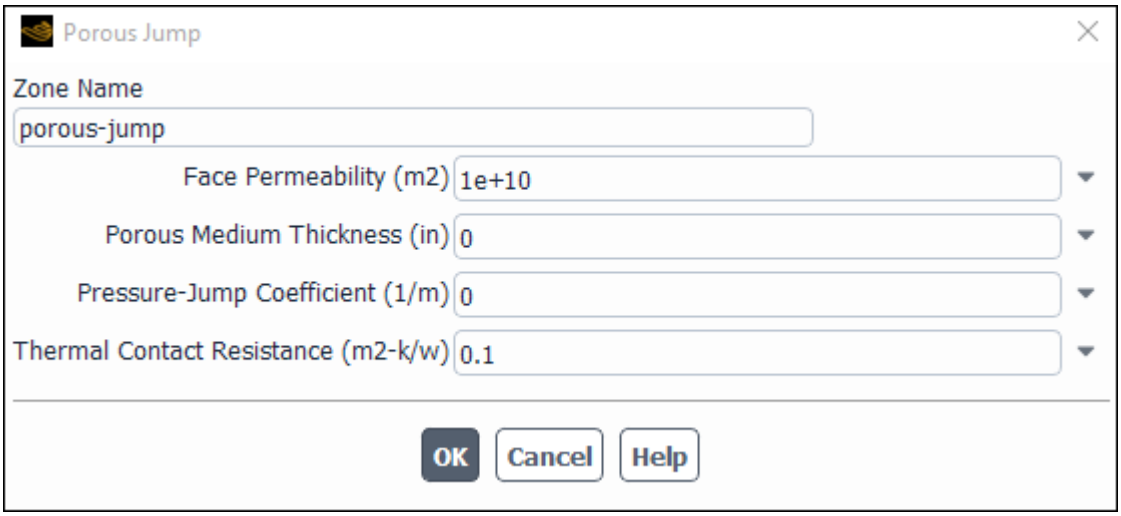
7.4.22.1. User Inputs for the Porous Jump Model

Once the porous jump zone has been identified (in the **Boundary Conditions** task page), you will set all modeling inputs for the porous jump in the Porous Jump Dialog Box (p. 5042) (Figure 7.81: The

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Porous Jump Dialog Box (p. 1520)), which is opened from the Boundary Conditions Task Page (p. 4991) (as described in Setting Cell Zone and Boundary Conditions (p. 1303)).

Figure 7.81: The Porous Jump Dialog Box



The inputs required for the porous jump model are as follows:

1. Identify the porous-jump zone.
2. Set the **Face Permeability** of the medium (α in Equation 7.152 (p. 1519)).
3. Set the **Porous Medium Thickness** (Δm).
4. Set the **Pressure-Jump Coefficient** (C_2).

> **Note:**
>
> The unit for the pressure-jump coefficient (C2) is the inverse of length. Should you want to define different units, you can do so by opening the **Units** dialog box and selecting **length-inverse** from the **Quantities** list.

5. Set the **Thermal Contact Resistance**.
6. Define the discrete phase boundary condition for the porous jump (for discrete phase calculations).
7. If you have enabled the solar load model, define the **Solar Boundary Conditions** for the porous jump. These settings allow you to define the boundary such that it acts as a semi-transparent surface (with respect to the solar radiation calculation only), and therefore allows a portion of the solar radiation to pass through it. See Solar Ray Tracing (p. 2261) for details.

7.4.22.1.1. Identifying the Porous Jump Zone

Since the porous jump model is a 1D simplification of the porous media model, the porous-jump zone must be modeled as the interface between cells, rather than a cell zone. Therefore the porous-jump zone is a type of internal face zone (where the faces are line segments in 2D or tri-

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

angles/quadrilaterals in 3D). If the porous-jump zone is not identified as such by default when you read in the mesh (that is, if it is identified as another type of internal face zone), you can right-click the zone in the tree and use the **Type** sub-menu to change it to a **porous-jump** zone (for more details, see Changing Cell and Boundary Zone Types (p. 1301)).

After the zone has been changed to a porous jump, you can open the Porous Jump Dialog Box (p. 5042) (as described in Setting Cell Zone and Boundary Conditions (p. 1303)) and specify the porous jump parameters listed above.

7.4.22.1.2. Defining Discrete Phase Boundary Conditions for the Porous Jump

If you are modeling a discrete phase of particles, you can set the fate of particle trajectories at the porous jump. See Setting Boundary Conditions for the Discrete Phase (p. 2779) for details.

7.4.22.2. Postprocessing for the Porous Jump

Postprocessing suggestions for a problem that includes a porous jump are the same as for porous media problems. See Postprocessing for Porous Media (p. 1371).

7.5. Editing Multiple Boundary Conditions at Once

You can edit multiple related boundaries (zones of the same type) at the same time to simplify the setup process for Ansys Fluent cases.

Using the **Multi Edit** dialog box, you can edit or apply selected settings only, to a set of boundaries without opening multiple different dialog boxes. The dialog box includes colored icons to indicate which setting(s) will be applied on the selected zones.

To access the **Multi Edit** dialog box:

1. Select the boundaries that you want to edit in the Outline View tree. *You can select multiple-boundaries by holding **Ctrl** while you left-click.*

   Alternatively, you can select the boundaries that you want to edit in the **Adjacent Face Zones** list of the Adjacency Dialog Box (p. 5390).

   Note:

   One-sided (or external) walls cannot be selected together with coupled (or internal) walls for multi-editing.

2. Right-click the highlighted boundaries and click **Multi Edit...**.

   While Figure 7.82: The Multi Edit Wall Settings Dialog Box (p. 1522) is only for performing a multi-edit of velocity inlets, the multi-edit functionality is available for the following boundary types:

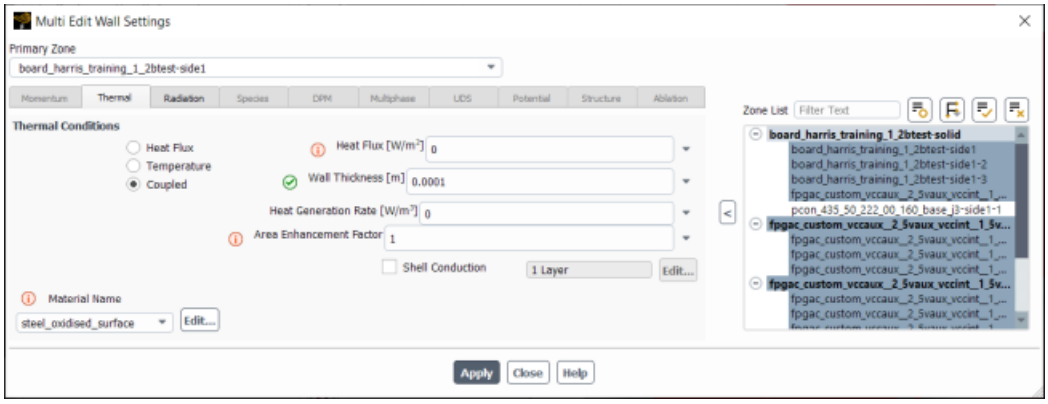
   - exhaust fans
   - inlet vents

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- intake fans
- mass flow inlets
- mass flow outlets
- outflows
- outlet vents
- pressure far fields
- pressure inlets
- pressure outlets
- velocity inlets
- walls

The setup for the multi-edit versions of these dialog boxes is the same as their non-multi-edit counterparts. The multi-edit functionality described in this section is consistent for all of the types.

Figure 7.82: The Multi Edit Wall Settings Dialog Box



The colored icons within the **Multi Edit** dialog box indicate the states of the various fields:

- ⓘ indicates that the value of this setting varies between the selected boundaries.
- ✓ indicates that the displayed value will be applied to the selected boundaries when you click **Apply**.

  *You can click an icon to change its state, which would be useful if you modify multiple settings, then realize you made a mistake on one of them. To rectify the mistake, you can simply click ✓ to remove it, indicating that the setting will not be applied when you click **Apply**.*

Note:

For any value entry or selection that results in changes to other fields, do not rely on just reverting the "primary field" (by clicking the ✓ icon) to revert the other affected fields.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If a field has a green check mark ( ) next to it, assume that the shown value will be applied on **Apply**, regardless of if you revert the initial change that led to the other field changes.

3. **Primary Zone** lists the zone that is used as the "baseline", to which all of the other selected zones are compared. You can change which zone is selected as the primary zone by:

   - Select a new primary zone from the **Primary Zone** drop-down list.
   - Deselect all of the zones in the **Zone** list and select the desired primary zone first when re-selecting which zones to use for the multi-edit functionality.

4. (Multiphase simulations only) Use the **Phase** drop-down list to specify boundary settings for each phase. *Note that you must click **Apply** before selecting a new phase for the settings to be applied.*

5. The **Zone List** shows the selected boundaries where the fields denoted with the icon will be applied.

   You can change which boundaries are highlighted in the **Zone List**, allowing you to selectively apply settings to a subset of the boundaries in the list. The settings are applied to the selected (highlighted) zones when you click **Apply**.

   You can sort the **Zone List** based on adjacency or by name by clicking and selecting **Adjacency** or **Name**.

Note:

Changes made to boundary settings using a "multi-edit" dialog box are also printed in the console and transcript (if one is being written).

7.6. Transient Cell Zone and Boundary Conditions

There are three ways you can specify transient cell zone and boundary conditions:

- transient profile with a format similar to the standard profiles described in Profiles (p. 1564)
- transient profile in a tabular format
- transient profile in comma separated values (CSV) format, as described in CSV Profiles (p. 1567).

Important:

For both methods, the cell zone or boundary condition will vary only in time; it must be spatially uniform. However, if the in-cylinder model is activated (In-Cylinder Settings (p. 1868)), then you have the option to use the crank angle instead of time. Crank angles can be included in transient tables as well as transient profiles, in a similar fashion to time. Examples of transient profiles and transient tables in crank angle can be found in the sections that follow.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For information about boundary profiles, refer to Reading and Writing Profile Files (p. 964).

7.6.1. Standard Transient Profiles

The format of the standard transient profile file (based on the profiles described in Profiles (p. 1564)) is

```
((profile-name transient n periodic?)
(field_name-1 a1 a2 a3 .... an)
(field_name-2 b1 b2 b3 .... bn)
.
.
.
.
(field_name-r r1 r2 r3 .... rn))
```

The profile name as well as the field names have to be shorter than 64 characters. One of the `field_name` s should be used for the `time` field, and the `time` field section *must* be in ascending order. n is the number of entries per field. The `periodic?` entry indicates whether or not the profile is time-periodic. Set it to `1` for a time-periodic profile, or `0` if the profile is not time-periodic.

An example is shown below:

```
((sampleprofile transient 3 0)
(time
1
2
3
)
(u
10
20
30
)
)
```

This example demonstrates the use of crank angle in a transient profile

```
((example transient 3 1)
(angle
0.000000e+00 1.800000e+02 3.600000e+02)
(temperature
3.000000e+02 5.000000e+02 3.000000e+02)
)
```

Important:

All quantities, including coordinate values, must be specified in SI units because Ansys Fluent does not perform unit conversion when reading profile files. Also, profile names must have all lowercase letters (for example, `name`). Uppercase letters in profile names are not acceptable.

You can read this file into Ansys Fluent using the Profiles Dialog Box (p. 4981) or the **File/Read/Profile...** ribbon tab item.

Setup → **Cell Zone Conditions** → **Profiles...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setup → **Boundary Conditions** → **Profiles...**

File → **Read** → **Profile...**

See Using Profiles (p. 1570) for details.

7.6.2. Tabular Transient Profiles

The format of the tabular transient profile file is

```
profile-name n_field n_data periodic?
field-name-1 field-name-2 field-name-3 .... field-name-n_field
v-1-1  v-2-1... ... ... ... v-n_field-1
v-1-2  v-2-2... ... ... ... v-n_field-2
.
.
.
.
.
v-1-n_data v-2-n_data ... ... ... ... v-n_field-n_data
```

The first field name (for example `field-name-1`) should be used for the `time` field, and the `time` field section, which represents the flow time, *must* be in ascending order. The `periodic?` entry indicates whether or not the profile is time-periodic. Set it to `1` for a time-periodic profile, or `0` if the profile is not time-periodic.

An example is shown below:

```
sampletabprofile 2 3 0
time u
1 10
2 20
3 30
```

This file defines the same transient profile as the standard profile example above.

If the periodicity is set to `1`, then `n_data` must be the number that closes one period.

An example is shown below:

```
periodtabprofile 2 4 1
time u
0 10
1 20
2 30
3 10
```

The following example uses crank angle instead of time:

```
example 2 3 1
angle temperature
0  300
```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
180 500
360 300
```

Important:

- All quantities, including coordinate values, must be specified in SI units because Ansys Fluent does not perform unit conversion when reading profile files. Also, profile names must have all lowercase letters (for example, `name`). Uppercase letters in profile names are not acceptable. When choosing the field names, spaces or parentheses should not be included.
- Some file formats may not be read into Fluent correctly (for example, UCS-2). If your file is not read correctly, copy the contents into a text editor (Notepad++/VIM/Notepad) and save it. This updated version of your profile is now readable in Fluent.

You can read this file into Ansys Fluent using the `read-transient-table` text command.

`file` → `read-transient-table`

After reading the table into Ansys Fluent, the profile will be listed in the Profiles Dialog Box (p. 4981) and can be used in the same way as a boundary profile. See Using Profiles (p. 1570) for details.

7.6.3. Profiles for Moving and Deforming Meshes

You can use profiles to specify position, velocity, and angular velocity. There are two different valid profile formats for moving and deforming mesh cases.

```
profile-name n_field n_data periodic?
field-name-1 field-name-2 field-name-3 .... field-name-n_field
v-1-1  v-2-1... ... ... ... v-n_field-1
v-1-2  v-2-2... ... ... ... v-n_field-2
.
.
.
.
.
v-1-n_data v-2-n_data ... ... ... ... v-n_field-n_data
```

and

```
((profile-name transient n periodic?)
(field_name-1 a1 a2 a3 .... an)
(field_name-2 b1 b2 b3 .... bn)
.
.
.
.
(field_name-r r1 r2 r3 .... rn))
```

You must use the appropriate variable nomenclature when writing your profile so that Fluent can properly interpret your inputs. For position, use `x`, `y`, and `z`. For specifying velocity, use `v_x`, `v_y`, and `v_z`. Similarly, for angular velocity, use `omega_x`, `omega_y`, and `omega_z`.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.7. Boundary Acoustic Wave Models

The standard boundary conditions, when imposed on the boundaries of an artificially truncated domain, result in the reflection of the outgoing pressure waves. As a consequence, the interior domain will contain spurious wave reflections. Many applications require precise control of the wave reflections from the domain boundaries to obtain accurate flow solutions. Ansys Fluent offers several boundary wave models that can be used to at the domain boundaries to control these spurious wave reflections.

The following boundary acoustic wave models are available:

- turbo-specific non-reflecting boundary condition (NRBC) (density-based solver only)
- general NRBC (pressure-based and density-based solvers)
- impedance boundary condition (pressure-based solver only)
- transparent flow forcing (pressure-based solver only)

In the density-based solver, the NRBC models are available for flows using the compressible ideal-gas law.

In the pressure-based solver, the general NRBC, impedance, and transient flow forcing models available for transient simulations of compressible flows (including the ideal-gas law, real gas law, species transport, and compressible mixture models).

Information about the available boundary acoustic wave models is provided in the following sections.

7.7.1. Turbo-Specific Non-Reflecting Boundary Conditions
7.7.2. General Non-Reflecting Boundary Conditions
7.7.3. Impedance Boundary Conditions
7.7.4. Transparent Flow Forcing Boundary Conditions

Note:

An alternative to the boundary acoustic wave models is to define sponge layers at boundary zones. Such sponge layers are based on the density, and are only available in a transient simulation for a compressible flow with the pressure-based solver. For details, see Sponge Layers (p. 2667).

7.7.1. Turbo-Specific Non-Reflecting Boundary Conditions

Information about turbo-specific NRBCs is provided in the following sections.

7.7.1.1. Overview
7.7.1.2. Limitations
7.7.1.3. Theory
7.7.1.4. Using Turbo-Specific Non-Reflecting Boundary Conditions

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.7.1.1. Overview

The standard pressure boundary conditions for compressible flow fix specific flow variables at the boundary (for example, static pressure at an outlet boundary). As a result, pressure waves incident on the boundary will reflect in an unphysical manner, leading to local errors. The effects are more pronounced for internal flow problems where boundaries are usually close to geometry inside the domain, such as compressor or turbine blade rows.

The turbo-specific non-reflecting boundary conditions permit waves to "pass" through the boundaries without spurious reflections. The method used in Ansys Fluent is based on the Fourier transformation of solution variables at the non-reflecting boundary [52] (p. 5716). Similar implementations have been investigated by other authors [102] (p. 5719) [135] (p. 5720). The solution is rearranged as a sum of terms corresponding to different frequencies, and their contributions are calculated independently. While the method was originally designed for axial turbomachinery, it has been extended for use with radial turbomachinery.

7.7.1.2. Limitations

Note the following limitations of turbo-specific NRBCs:

- They are available only with the density-based solver (explicit or implicit).
- The current implementation applies to steady compressible flows, with the density calculated using the ideal gas law.
- Inlet and outlet boundary conditions must be pressure inlets and outlets only.

  Important:

  Note that the pressure inlet boundaries must be set to the cylindrical coordinate flow specification method when turbo-specific NRBCs are used.

- Quad-mapped (structured) surface meshes must be used for inflow and outflow boundaries in a 3D geometry (that is, triangular or quad-paved surface meshes are not allowed). See Figure 7.83: Mesh and Prescribed Boundary Conditions in a 3D Axial Flow Problem (p. 1529) and Figure 7.84: Mesh and Prescribed Boundary Conditions in a 3D Radial Flow Problem (p. 1530) for examples.

  Important:

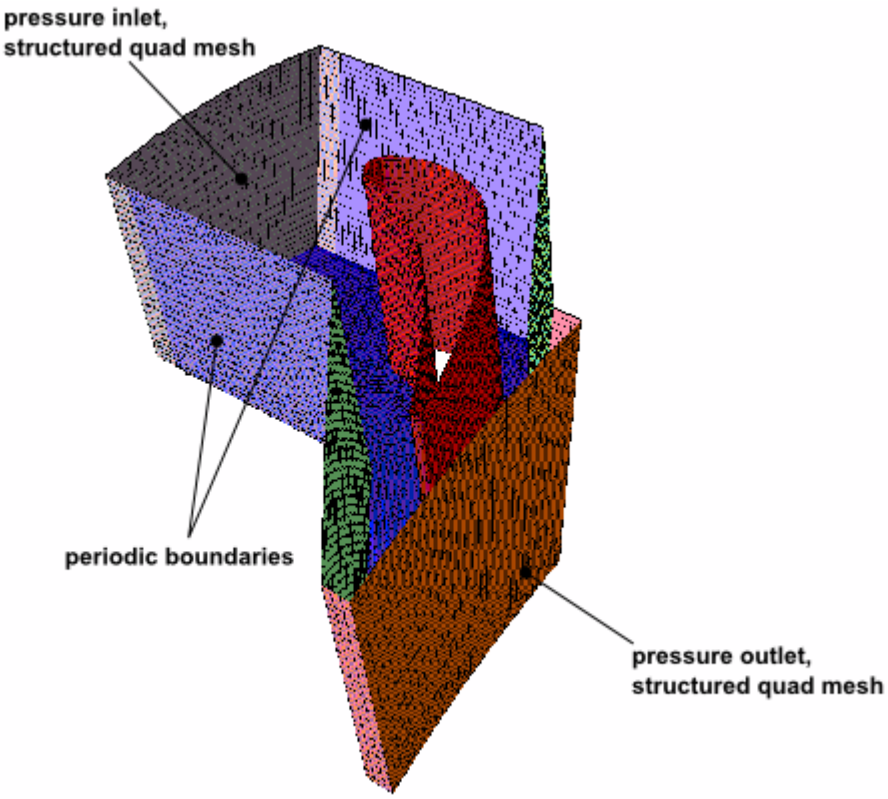
  Note that you may use unstructured meshes in 2D geometries (Figure 7.85: Mesh and Prescribed Boundary Conditions in a 2D Case (p. 1531)), and an unstructured mesh may be used away from the inlet and outlet boundaries in 3D geometries.

- The turbo-specific NRBC [52] (p. 5716) has been extended for use on 3D geometries [135] (p. 5720) by decoupling the tangential flow variations from the radial variations. This approximation works best for geometries with a blade pitch that is small compared to the radius of the geometry.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

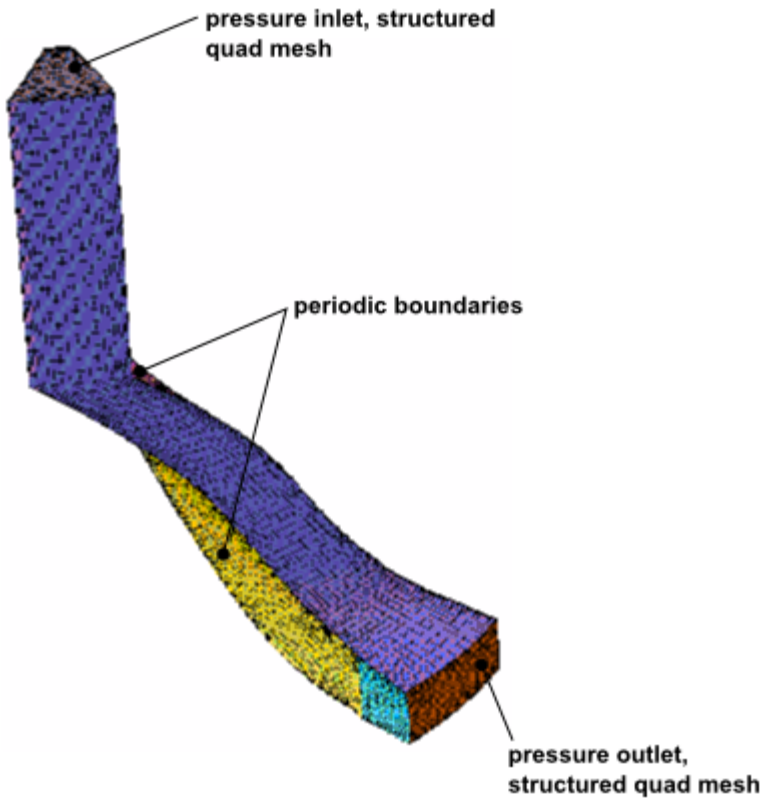
- Reverse flow on the inflow and outflow boundaries are not allowed. If strong reverse flow is present, then you should consider using the General NRBCs instead.
- NRBCs are not compatible with species transport models. They are mainly used to solve ideal-gas single-species flow.

Figure 7.83: Mesh and Prescribed Boundary Conditions in a 3D Axial Flow Problem



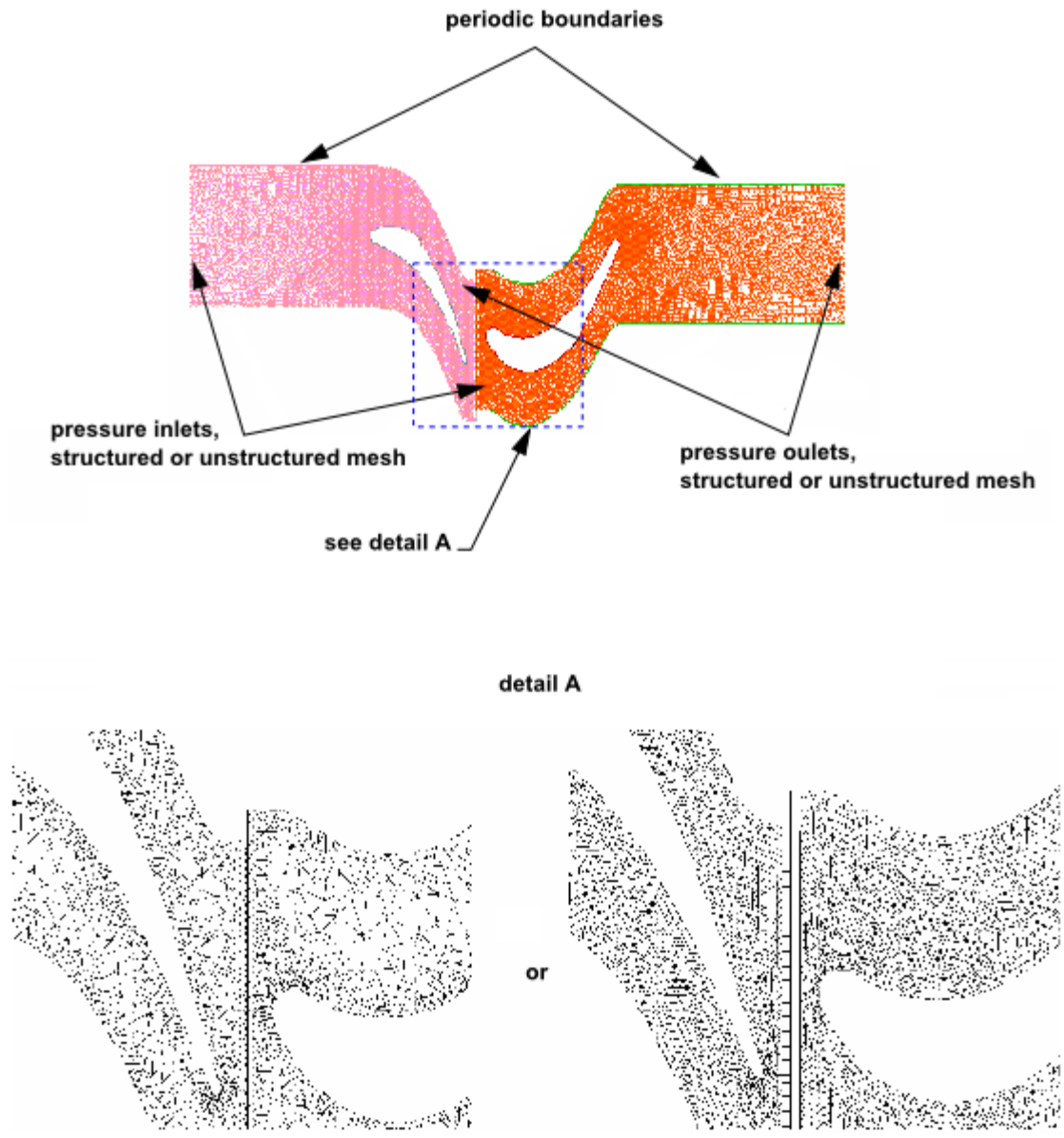
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.84: Mesh and Prescribed Boundary Conditions in a 3D Radial Flow Problem



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.85: Mesh and Prescribed Boundary Conditions in a 2D Case



7.7.1.3. Theory

Turbo-specific NRBCs are based on Fourier decomposition of solutions to the linearized Euler equations. The solution at the inlet and outlet boundaries is circumferentially decomposed into Fourier modes, with the 0th mode representing the average boundary value (which is to be imposed as a user input), and higher harmonics that are modified to eliminate reflections [135] (p. 5720).

7.7.1.3.1. Equations in Characteristic Variable Form

In order to treat individual waves, the linearized Euler equations are transformed to characteristic variable (C_i) form. If we first consider the 1D form of the linearized Euler equations, it can be shown that the characteristic variables C_i are related to the solution variables as follows:

$$\tilde{Q} = T^{-1} C \tag{7.153}$$

where

$$\tilde{Q}=\begin{Bmatrix}\tilde{\rho}\\ \tilde{u}_a\\ \tilde{u}_t\\ \tilde{u}_r\\ \tilde{p}\end{Bmatrix},T^{-1}=\begin{bmatrix}-\frac{1}{\overline{a}^2} & 0 & 0 & \frac{1}{2\overline{a}^2} & \frac{1}{2\overline{a}^2}\\ 0 & 0 & 0 & \frac{1}{2\overline{\rho}\,\overline{a}} & \frac{1}{2\overline{\rho}\,\overline{a}}\\ 0 & \frac{1}{\overline{\rho}\,\overline{a}} & 0 & 0 & 0\\ 0 & 0 & \frac{1}{\overline{\rho}\,\overline{a}} & 0 & 0\\ 0 & 0 & 0 & \frac{1}{2} & \frac{1}{2}\end{bmatrix},C=\begin{Bmatrix}C_1\\ C_2\\ C_3\\ C_4\\ C_5\end{Bmatrix} \tag{7.154}$$

where $\overline{a}$ is the average acoustic speed along a boundary zone, $\tilde{\rho}$, $\tilde{u}_a$, $\tilde{u}_t$, $\tilde{u}_r$, and $\tilde{p}$ represent perturbations from a uniform condition (for example, $\tilde{\rho}=\rho-\overline{\rho}$,$\tilde{p}=p-\overline{p}$, and so on).

Note that the analysis is performed using the cylindrical coordinate system. All overlined (averaged) flow field variables (for example, $\overline{\rho}$, $\overline{a}$) are intended to be averaged along the pitchwise direction.

In quasi-3D approaches [52] (p. 5716) [102] (p. 5719) [135] (p. 5720), a procedure is developed to determine the changes in the characteristic variables, denoted by δC_i, at the boundaries such that waves will not reflect. These changes in characteristic variables are determined as follows:

$$\delta C=T\,\delta Q \tag{7.155}$$

where

$$\delta C=\begin{Bmatrix}\delta C_1\\ \delta C_2\\ \delta C_3\\ \delta C_4\\ \delta C_5\end{Bmatrix},T=\begin{bmatrix}-\overline{a}^2 & 0 & 0 & 0 & 1\\ 0 & 0 & \overline{\rho}\,\overline{a} & 0 & 0\\ 0 & 0 & 0 & \overline{\rho}\,\overline{a} & 0\\ 0 & \overline{\rho}\,\overline{a} & 0 & 0 & 1\\ 0 & -\overline{\rho}\,\overline{a} & 0 & 0 & 1\end{bmatrix},\delta Q=\begin{Bmatrix}\delta\rho\\ \delta u_a\\ \delta u_t\\ \delta u_r\\ \delta p\end{Bmatrix} \tag{7.156}$$

The changes to the outgoing characteristics — one characteristic for subsonic inflow (δC_5), and four characteristics for subsonic outflow (δC_1, δC_2, δC_3, δC_4) — are determined from extrapolation of the flow field variables using Equation 7.155 (p. 1532).

The changes in the incoming characteristics — four characteristics for subsonic inflow (δC_1, δC_2, δC_3, δC_4), and one characteristic for subsonic outflow (δC_5) — are split into two components: average change along the boundary ($\delta\overline{C}_i$), and local changes in the characteristic variable due to harmonic variation along the boundary (δC_{iL}). The incoming characteristics are therefore given by

$$\delta C_{i_j}=\delta C_{iold_j}+\sigma\left(\delta C_{inew_j}-\delta C_{iold_j}\right) \tag{7.157}$$

$$\delta C_{inew_j}=\left(\delta\overline{C}_i+\delta C_{iL_j}\right) \tag{7.158}$$

where $i=1,2,3,4$ on the inlet boundary or $i=5$ on the outlet boundary, and $j=1,\ldots,N$ is the grid index in the pitchwise direction including the periodic point once. The under-relaxation factor σ has a default value of 0.75. Note that this method assumes a periodic solution in the pitchwise direction.

The flow is decomposed into mean and circumferential components using Fourier decomposition. The 0th Fourier mode corresponds to the average circumferential solution, and is treated according

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

to the standard 1D characteristic theory. The remaining parts of the solution are described by a sum of harmonics, and treated as 2D non-reflecting boundary conditions [52] (p. 5716).

7.7.1.3.2. Inlet Boundary

For subsonic inflow, there is one outgoing characteristic (δC_5) determined from Equation 7.155 (p. 1532), and four incoming characteristics (δC_1, δC_2, δC_3, δC_4) calculated using Equation 7.157 (p. 1532). The average changes in the incoming characteristics are computed from the requirement that the entropy (s), radial and tangential flow angles (α_r and α_t), and stagnation enthalpy (h_0) are specified. Note that in Ansys Fluent you can specify p_0 and T_0 at the inlet, from which s_{in} and $h_{0_{in}}$ are easily obtained. This is equivalent to forcing the following four residuals to be zero:

$$R_1 = \overline{\rho}\left(\overline{s} - s_{in}\right) \tag{7.159}$$

$$R_2 = \overline{\rho}\,\overline{a}\left(\overline{u}_t - \overline{u}_a \tan\alpha_t\right) \tag{7.160}$$

$$R_3 = \overline{\rho}\,\overline{a}\left(\overline{u}_r - \overline{u}_a \tan\alpha_r\right) \tag{7.161}$$

$$R_4 = \overline{\rho}\left(\overline{h}_0 - h_{0_{in}}\right) \tag{7.162}$$

where

$$s_{in} = \gamma \ln\left(T_{0_{in}}\right) - (\gamma - 1)\ln\left(p_{0_{in}}\right) \tag{7.163}$$

$$h_{0_{in}} = c_p T_{0_{in}} \tag{7.164}$$

The average characteristic is then obtained from residual linearization as follows (see also Figure 7.86: Prescribed Inlet Angles (p. 1534)

$$\begin{Bmatrix} \delta\overline{C}_1 \\ \delta\overline{C}_2 \\ \delta\overline{C}_3 \\ \delta\overline{C}_4 \end{Bmatrix} = \begin{bmatrix} -1 & 0 & 0 & 0 \\ \frac{\tan\alpha_t}{(1-\gamma)M} & \frac{M_1}{M} & \frac{-M_t}{M}\tan\alpha_t & \frac{\tan\alpha_t}{M} \\ \frac{\tan\alpha_r}{(\gamma-1)M} & \frac{M_t}{M}\tan\alpha_t & \frac{M_2}{M} & \frac{-\tan\alpha_r}{M} \\ \frac{2}{(\gamma-1)M} & 2\frac{M_t}{M} & 2\frac{M_r}{M} & \frac{-2}{M} \end{bmatrix} \begin{Bmatrix} R_1 \\ R_2 \\ R_3 \\ R_4 \end{Bmatrix} \tag{7.165}$$

where

$$M_a = \frac{\overline{u}_a}{\overline{a}} \tag{7.166}$$

$$M_t = \frac{\overline{u}_t}{\overline{a}} \tag{7.167}$$

$$M_r = \frac{\overline{u}_r}{\overline{a}} \tag{7.168}$$

and

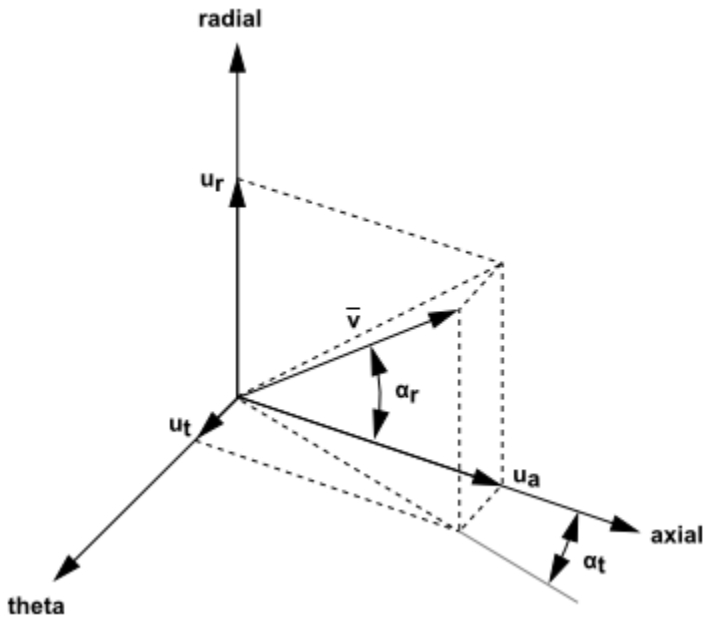
$$M = 1 + M_a - M_t \tan\alpha_t + M_r \tan\alpha_r \tag{7.169}$$

$$M_1 = -1 - M_a - M_r \tan\alpha_r \tag{7.170}$$

$$M_2 = -1 - M_a - M_t \tan\alpha_t \tag{7.171}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.86: Prescribed Inlet Angles



where

$$|v| = \sqrt{u_t^2 + u_r^2 + u_a^2} \tag{7.172}$$

$$e_t = \frac{u_t}{|v|} \tag{7.173}$$

$$e_r = \frac{u_r}{|v|} \tag{7.174}$$

$$e_a = \frac{u_a}{|v|} \tag{7.175}$$

$$\tan\alpha_t = \frac{e_t}{e_a} \tag{7.176}$$

$$\tan\alpha_r = \frac{e_r}{e_a} \tag{7.177}$$

To address the local characteristic changes at each j grid point along the inflow boundary, the following relations are developed [52] (p. 5716) [135] (p. 5720):

$$\begin{aligned}
\delta C_{1L_j} &= \overline{p}\left(s_j - \overline{s}\right) \\
\delta C_{2L_j} &= C'_{2_j} - \overline{\rho a}\left(u_{t_j} - \overline{u}_t\right) \\
\delta C_{3L_j} &= -\overline{\rho a}\left(u_{r_j} - \overline{u}_r\right) \\
\delta C_{4L_j} &= \frac{-2}{(1+M_{a_j})}\left(\frac{1}{\gamma-1}\delta C_{1L_j} + M_{t_j}\delta C_{2L_j} + M_{r_j}\delta C_{3L_j} + \overline{\rho}\left(h_{0_j} - \overline{h}_0\right)\right)
\end{aligned} \tag{7.178}$$

Note that the relation for the first and fourth local characteristics force the local entropy and stagnation enthalpy to match their average steady-state values.

The characteristic variable C'_{2_j} is computed from the inverse discrete Fourier transform of the second characteristic. The discrete Fourier transform of the second characteristic in turn is related to the discrete Fourier transform of the fifth characteristic. Hence, the characteristic variable C'_{2_j} is computed along the pitch as follows:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$C'_{2_j}=2\Re\left(\sum_{n=1}^{\frac{N}{2}-1}\hat{C}_{2_n}\exp\left(i2\pi n\frac{\theta_j-\theta_1}{\theta_N-\theta_1}\right)\right) \tag{7.179}$$

The Fourier coefficients C'_{2_n} are related to a set of equidistant distributed characteristic variables $C^*_{5_j}$ by the following [102] (p. 5719):

$$\hat{C}_{2_n}=\begin{cases}-\frac{\overline{u}_t+B}{N(\overline{a}+\overline{u}_a)}\sum_{j=1}^{N}C^*_{5_j}\exp\left(-i2\pi\frac{jn}{N}\right) & \beta>0\\ -\frac{\overline{u}_t+B}{\overline{a}+\overline{u}_a}C_{5_j} & \beta<0\end{cases} \tag{7.180}$$

where

$$B=\begin{cases}i\sqrt{\beta} & \beta>0\\ -sign(\overline{u}_t)\sqrt{|\beta|} & \beta<0\end{cases} \tag{7.181}$$

and

$$\beta=\overline{a}^2-\overline{u}_a^2-\overline{u}_t^2 \tag{7.182}$$

The set of equidistributed characteristic variables $C^*_{5_j}$ is computed from arbitrary distributed C_{5_j} by using a cubic spline for interpolation, where

$$C_{5_j}=-\overline{\rho}\,\overline{a}\left(u_{a_j}-\overline{u}_a\right)+\left(p_j-\overline{p}\right) \tag{7.183}$$

For supersonic inflow the user-prescribed static pressure ($p_{s_{in}}$) along with total pressure ($p_{0_{in}}$) and total temperature ($T_{0_{in}}$) are sufficient for determining the flow condition at the inlet.

7.7.1.3.3. Outlet Boundary

For subsonic outflow, there are four outgoing characteristics (δC_1, δC_2, δC_3, and δC_4) calculated using Equation 7.155 (p. 1532), and one incoming characteristic (δC_5) determined from Equation 7.157 (p. 1532). The average change in the incoming fifth characteristic is given by

$$\delta\overline{C}_5=-2\left(\overline{p}-p_{out}\right) \tag{7.184}$$

where $\overline{p}$ is the current averaged pressure at the exit plane and p_{out} is the desirable average exit pressure (this value is specified by you for single-blade calculations or obtained from the assigned profile for mixing-plane calculations). The local changes (δC_{5L_j}) are given by

$$\delta C_{5L_j}=C'_{5_j}+\overline{\rho}\,\overline{a}\left(u_{a_j}-\overline{u}_a\right)-\left(p_j-\overline{p}\right) \tag{7.185}$$

The characteristic variable C'_{5_j} is computed along the pitch as follows:

$$C'_{5_j}=2\Re\left(\sum_{n=1}^{\frac{N}{2}-1}\hat{C}_{5_n}\exp\left(i2\pi n\frac{\theta_j-\theta_1}{\theta_N-\theta_1}\right)\right) \tag{7.186}$$

The Fourier coefficients $\hat{C}_{5_n}$ are related to two sets of equidistantly distributed characteristic variables ($C^*_{2_j}$ and $C^*_{4_j}$, respectively) and given by the following [102] (p. 5719):

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\hat{C}_{5_n} = \begin{cases} \frac{A_2}{N}\sum_{j=1}^{N} C^*_{2_j} \exp\left(i2\pi\frac{jn}{N}\right) - \frac{A_4}{N}\sum_{j=1}^{N} C^*_{4_j} \exp\left(i2\pi\frac{jn}{N}\right) & \beta > 0 \\ A_2 C_{2_j} - A_4 C_{4_j} & \beta < 0 \end{cases} \tag{7.187}$$

where

$$A_2 = \frac{2\overline{u}_a}{B - \overline{u}_t} \tag{7.188}$$

$$A_4 = \frac{B + \overline{u}_t}{B - \overline{u}_t} \tag{7.189}$$

The two sets of equidistributed characteristic variables ($C^*_{2_j}$ and $C^*_{4_j}$) are computed from arbitrarily distributed C_{2_j} and C_{4_j} characteristics by using a cubic spline for interpolation, where

$$C_{2_j} = \overline{\rho}\,\overline{a}\left(u_{t_j} - \overline{u}_t\right) \tag{7.190}$$

$$C_{4_j} = \overline{\rho}\,\overline{a}\left(u_{a_j} - \overline{u}_a\right) + \left(p_j - \overline{p}\right) \tag{7.191}$$

For supersonic outflow all flow field variables are extrapolated from the interior.

7.7.1.3.4. Updated Flow Variables

Once the changes in the characteristics are determined on the inflow or outflow boundaries, the changes in the flow variables δQ can be obtained from Equation 7.155 (p. 1532). Therefore, the values of the flow variables at the boundary faces are as follows:

$$p_f = p_j + \delta p \tag{7.192}$$

$$u_{a_f} = u_{a_j} + \delta u_a \tag{7.193}$$

$$u_{t_f} = u_{t_j} + \delta u_t \tag{7.194}$$

$$u_{r_f} = u_{r_j} + \delta u_r \tag{7.195}$$

$$T_f = T_j + \delta T \tag{7.196}$$

7.7.1.4. Using Turbo-Specific Non-Reflecting Boundary Conditions

Important:

If you intend to use turbo-specific NRBCs in conjunction with the density-based implicit solver, it is recommended that you first converge the solution before turning on turbo-specific NRBCs, then converge it again with turbo-specific NRBCs turned on. If the solution is diverging, then you should lower the CFL number. These steps are necessary because only approximate flux Jacobians are used for the pressure-inlet and pressure-outlet boundaries when turbo-specific NRBCs are activated with the density-based implicit solver.

The procedure for using the turbo-specific NRBCs is as follows:

1. Turn on the turbo-specific NRBCs using the `non-reflecting-bc` text command:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

`define` → `boundary-conditions` → `non-reflecting-bc` → `turbo-specific-nrbc` → `enable?`

If you are not sure whether or not NRBCs are turned on, use the `show-status` text command.

2. Perform NRBC initialization using the `initialize` text command:

   `define` → `boundary-conditions` → `non-reflecting-bc` → `turbo-specific-nrbc` → `initialize`

   If the initialization is successful, a summary printout of the domain extent will be displayed. If the initialization is not successful, an error message will be displayed indicating the source of the problem. The initialization will set up the pressure-inlet and pressure-outlet boundaries for use with turbo-specific NRBCs.

   Important:

   Note that the pressure inlet boundaries must be set to the cylindrical coordinate flow specification method when turbo-specific NRBCs are used.

3. If necessary, modify the parameters in the `set/` submenu:

   `define` → `boundary-conditions` → `non-reflecting-bc` → `turbo-specific-nrbc` → `set`

   `under-relaxation`

   allows you to set the value of the under-relaxation factor σ in Equation 7.157 (p. 1532). The default value is 0.75.

   `discretization`

   allows you to set the discretization scheme. The default is to use higher-order reconstruction if available.

   `verbosity`

   allows you to control the amount of information printed to the console during an NRBC calculation.

   - `0`: silent
   - `1`: basic information (default)
   - `2`: detailed information (for debugging purposes only)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.7.1.4.1. Using the NRBCs with the Mixing-Plane Model

If you want to use the NRBCs with the mixing-plane model you must define the mixing plane interfaces as pressure-outlet and pressure-inlet zone type pairs.

Important:

Turbo-specific NRBCs should not be used with the mixing-plane model if reverse flow is present across the mixing-plane.

7.7.1.4.2. Using the NRBCs in Parallel Ansys Fluent

When the turbo-specific NRBCs are used in conjunction with the parallel solver, all cells in each boundary zone, where NRBCs will be applied, must be located or contained within a single partition. You can ensure this by manually partitioning the mesh (see Partitioning the Mesh Manually and Balancing the Load (p. 4319) for more information).

7.7.2. General Non-Reflecting Boundary Conditions

Information about general NRBCs is provided in the following sections.

7.7.2.1. Overview

7.7.2.2. Restrictions and Limitations

7.7.2.3. Theory

7.7.2.4. Using the General Non-Reflecting Boundary Condition

7.7.2.1. Overview

The general non-reflecting boundary conditions in Ansys Fluent are based on characteristic wave relations derived from the Euler equations. In the density-based solver, the non-reflecting boundary conditions are applied only on pressure-outlet boundary conditions. In the pressure-based solver they are applied on pressure-inlet, pressure-outlet, velocity-inlet and mass-flux boundary conditions. To obtain the primitive flow quantities (P, u, v, w, T), reformulated Euler equations are solved on the boundary of the domain in an algorithm similar to the flow equations applied to the interior of the domain.

Unlike the turbo-specific NRBC, the general NRBC method is not restricted by geometric constraints or mesh type. However, good cell skewness near the boundaries where the NRBCs can be applied for better convergence.

7.7.2.2. Restrictions and Limitations

Note the following restrictions and limitations on the general NRBCs:

- The general NRBC is not available if the target mass flow rate is activated in the pressure-outlet dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The general NRBC using the density-based solver is available only with compressible flow while using the ideal-gas law.

  Important:

  The general NRBC using the density-based solver should not be used with the wet steam or real gas models.

- The general NRBCs using the density-based solver are not compatible with species transport and mixture fraction transport models (for premixed and partially-premixed models). They are mainly used to solve ideal-gas single-component flow.
- The general NRBCs using the pressure-based solver are not compatible with steady-state cases or the Eulerian multiphase model. For VOF or mixture multiphase cases that involve compressible gases or compressible liquids, only pressure inlets and pressure outlets can enable the non-reflecting boundary condition option. Note that some small reflections may still occur from NRBC boundaries when secondary-phase volume fractions are nonzero at the boundaries.

7.7.2.3. Theory

General NRBCs are derived by first recasting the Euler equations in an orthogonal coordinate system (x_1, x_2, x_3) such that one of the coordinates, x_1, is normal to the boundary Figure 7.87: The Local Orthogonal Coordinate System onto which Euler Equations are Recasted for the General NRBC Method (p. 1540). The characteristic analysis [163] (p. 5722) [164] (p. 5722) is then used to modify terms corresponding to waves propagating in the x_1 normal direction. When doing so, a system of equations can be written to describe the wave propagation as follows:

$$\begin{aligned}
&\frac{\partial \rho}{\partial t} + d_1 + \frac{\partial m_2}{\partial x_2} + \frac{\partial m_3}{\partial x_3} = 0 \\
&\frac{\partial m_1}{\partial t} + U_1 d_1 + \rho d_3 + \frac{\partial (m_1 U_2)}{\partial x_2} + \frac{\partial (m_1 U_3)}{\partial x_3} = 0 \\
&\frac{\partial m_2}{\partial t} + U_2 d_1 + \rho d_4 + \frac{\partial (m_2 U_2)}{\partial x_2} + \frac{\partial (m_2 U_3)}{\partial x_3} + \frac{\partial P}{\partial x_2} = 0 \\
&\frac{\partial m_3}{\partial t} + U_3 d_1 + \rho d_5 + \frac{\partial (m_3 U_2)}{\partial x_2} + \frac{\partial (m_3 U_3)}{\partial x_3} + \frac{\partial P}{\partial x_3} = 0 \\
&\frac{\partial \rho E}{\partial t} + \frac{1}{2}|V|^2 d_1 + \frac{d_2}{(\gamma - 1)} + m_1 d_3 + m_2 d_4 + m_3 d_5 + \frac{\partial [(\rho E + P) U_2]}{\partial x_2} + \frac{\partial [(\rho E + P) U_3]}{\partial x_3} = 0
\end{aligned} \tag{7.197}$$

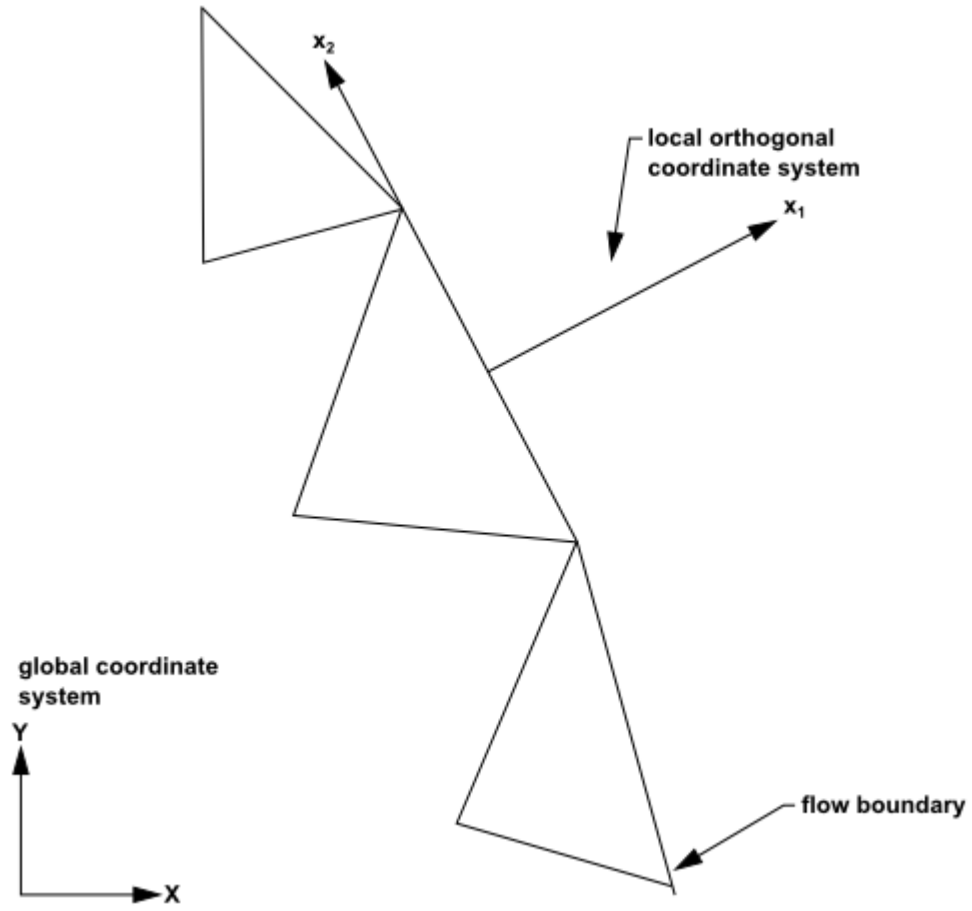
Where $m_1 = \rho U_1$, $m_2 = \rho U_2$ and $m_3 = \rho U_3$ and U_1, U_2 and U_3 are the velocity components in the coordinate system (x_1, x_2, x_3). The equations above are solved on non-reflecting boundaries, along with the interior governing flow equations, using similar time stepping algorithms to obtain the values of the primitive flow variables (P, u, v, w, T).

Important:

Note that a transformation between the local orthogonal coordinate system (x_1, x_2, x_3) and the global Cartesian system (X, Y, Z) must be defined on each face on the boundary to obtain the velocity components (u, v, w) in a global Cartesian system.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.87: The Local Orthogonal Coordinate System onto which Euler Equations are Recasted for the General NRBC Method



The d_i terms in the transformed Euler equations contain the outgoing and incoming characteristic wave amplitudes, L_i, and are defined as follows:

$$\begin{aligned} d_1 &= \frac{1}{c^2}\left[L_2 + \frac{1}{2}\left(L_5 + L_1\right)\right] \\ d_2 &= \frac{1}{2}\left(L_5 + L_1\right) \\ d_3 &= \frac{1}{2\rho c}\left(L_5 - L_1\right) \\ d_4 &= L_3 \\ d_5 &= L_4 \end{aligned} \tag{7.198}$$

From characteristic analyses, the wave amplitudes, L_i, are given by:

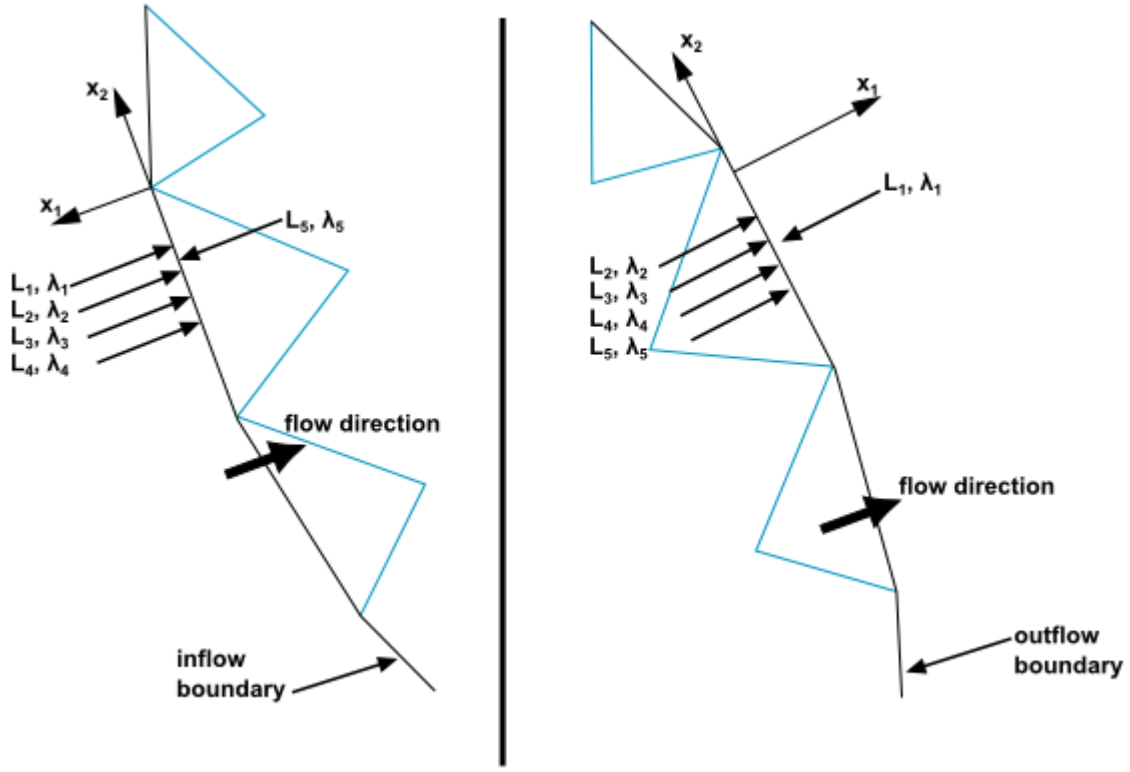
$$\begin{aligned} L_1 &= \lambda_1\left(\frac{\partial P}{\partial x_1} - \rho c \frac{\partial U_1}{\partial x_1}\right) \\ L_2 &= \lambda_2\left(c^2 \frac{\partial \rho}{\partial x_1} - \frac{\partial P}{\partial x_1}\right) \\ L_3 &= \lambda_3 \frac{\partial U_2}{\partial x_1} \\ L_4 &= \lambda_4 \frac{\partial U_3}{\partial x_1} \\ L_5 &= \lambda_5\left(\frac{\partial P}{\partial x_1} + \rho c \frac{\partial U_1}{\partial x_1}\right) \end{aligned} \tag{7.199}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The outgoing and incoming characteristic waves are associated with the characteristic velocities of the system (i.e eigenvalues), λ_i, as seen in Figure 7.88: Waves Leaving and Entering a Boundary Face on Inflow and Outflow Boundaries. The Wave Amplitudes are Shown with the Associated Eigenvalues for a Subsonic Flow Condition (p. 1541). These eigenvalues are given by:

$$\begin{aligned}\lambda_1&=U_1-c\\ \lambda_2&=\lambda_3=\lambda_4=U_1\\ \lambda_5&=U_1+c\end{aligned} \tag{7.200}$$

Figure 7.88: Waves Leaving and Entering a Boundary Face on Inflow and Outflow Boundaries. The Wave Amplitudes are Shown with the Associated Eigenvalues for a Subsonic Flow Condition



For subsonic flow leaving a boundary, four waves leave the domain (associated with positive eigenvalues λ_2, λ_3, λ_4, and λ_5) and one enters the domain (associated with negative eigenvalue λ_1). For subsonic flow entering a boundary, four waves enter the domain (associated with the negative eigenvalues λ_1, λ_2, λ_3, λ_4) and one leaves the domain (associated with a positive eigenvalue λ_5).

To solve Equation 7.197 (p. 1539) on a boundary, the amplitude of the incoming and outgoing waves must be first determined. The amplitude of outgoing waves are computed from Equation 7.199 (p. 1540) by using extrapolated values of flow derivatives $\frac{\partial P}{\partial x_1}$, $\frac{\partial \rho}{\partial x_1}$, $\frac{\partial U_1}{\partial x_1}$, $\frac{\partial U_2}{\partial x_1}$, and $\frac{\partial U_3}{\partial x_1}$ from inside the domain. The amplitude of the incoming waves are computed as follows. The amplitude of waves L_3, L_4, for tangential velocity components, is set to zero. The amplitude of the incoming pressure and entropy waves are computed from the Linear Relaxation Method (LRM) of Poinsot [120] (p. 5720) [121] (p. 5720). The LRM method sets the value of the incoming wave amplitude to be proportional to the differences between the local primitive variable on a boundary face and the imposed boundary value. The expression for the amplitudes depends on the boundary type as shown below:

1. **Pressure Outlet**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$L_1 = K(P - P_{exit}) \tag{7.201}$$

where P_{exit} is the imposed pressure at the exit boundary, K is the relaxation factor, and P is the local pressure value at the boundary.

In general, the desirable average pressure on a non-reflecting boundary can be either relaxed toward a pressure value at infinity or enforced to be equivalent to some desired pressure at the exit of the boundary.

In the case of a backflow, the pressure-based solver has two options for specifying pressure at the boundary; static pressure or total pressure. If the static pressure option is selected, the pressure at infinity is equal to the pressure defined in the boundary dialog box. If the total pressure option is selected, the pressure at infinity is a total pressure that is computed from the value in the dialog box. For the density-based solver, the total pressure option is used and no other options are available.

If you want the average pressure at the boundary to relax toward P at infinity (that is $P_{exit} = P_\infty$), the suggested K factor is given by:

$$K = \sigma_1 \left(1 - M_{max}^{\ 2}\right) \frac{c}{h} \tag{7.202}$$

where c is the acoustic speed, h is the domain size, M_{max} is the maximum Mach number in the domain, and σ_1 is the under-relaxation factor (default value is 0.15). On the other hand, if the desired average pressure at the boundary is to approach a specific imposed value at the boundary, then the K factor is given by:

$$K = \sigma_2 c \tag{7.203}$$

where the default value for σ_2 is 5.0

Important:

The desired average pressure option is only available for the density-based solver.

---

2. **Pressure Inlet**:

$$\begin{aligned} L_1 &= K(P - P_{exit}) \\ L_2 &= K\left[c^2\left(\rho - \rho_{exit}\right) - \left(P - P_{exit}\right)\right] \end{aligned} \tag{7.204}$$

where the imposed pressure, P_{exit}, and density, ρ_{exit}, are computed from the specified total pressure and total temperature.

3. **Mass Flux Boundary**:

$$\begin{aligned} L_1 &= -K\rho c(U_1 - U_{1,exit}) \\ L_2 &= Kc^2\left(\rho - \rho_{exit}\right) \end{aligned} \tag{7.205}$$

where the imposed exit velocity and density are computed from specific mass flux and total temperature.

4. **Velocity Inlet**:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$L_1 = -K\rho c(U_1 - U_{1,exit})$$
$$L_2 = Kc^2\left(\rho - \rho_{exit}\right) \tag{7.206}$$

where the imposed velocity, $U_{1,exit}$, is specified at the boundary and the imposed density is computed from the specified boundary temperature and extrapolated pressure.

7.7.2.4. Using the General Non-Reflecting Boundary Condition

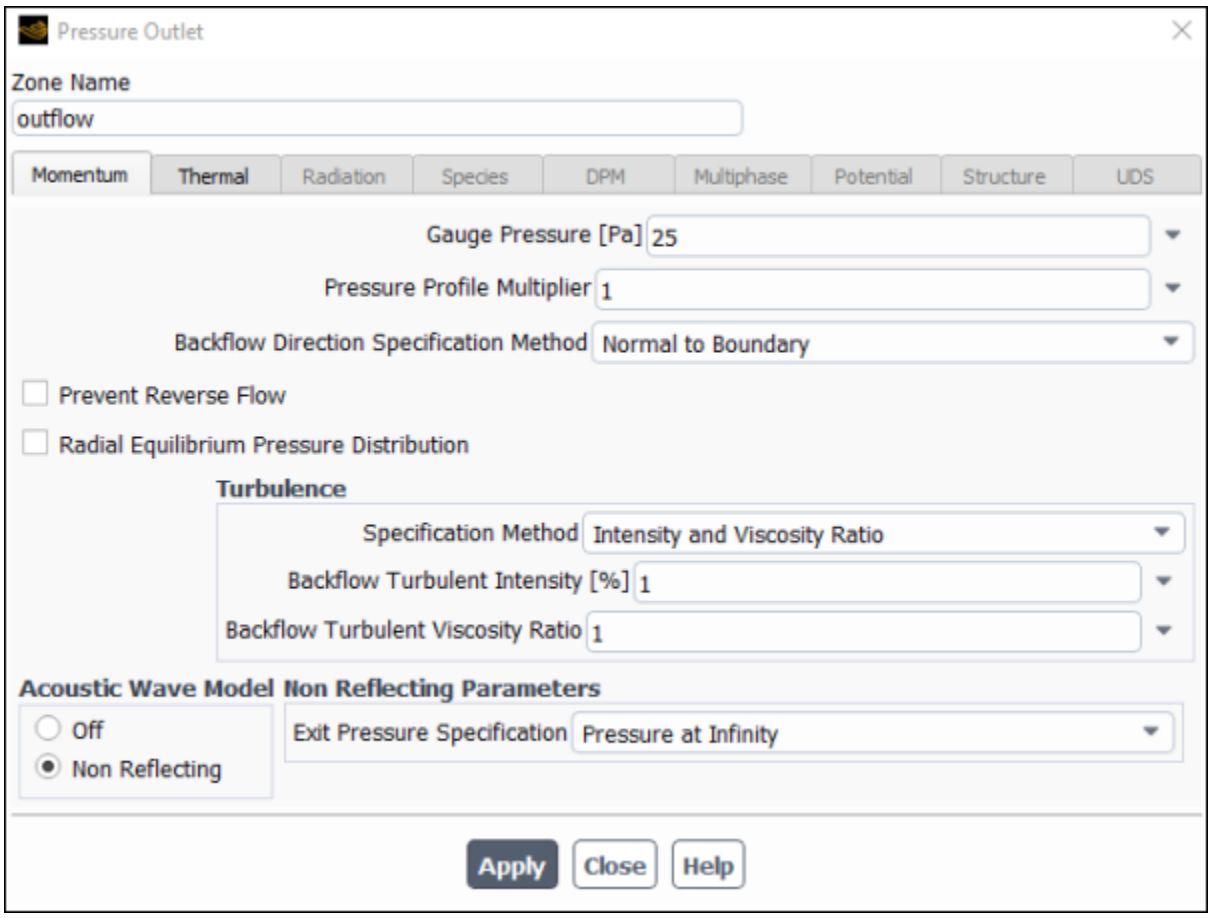
The general Non-Reflecting Boundary Condition is available for use in the **Pressure Outlet** dialog box when either the density-based (with ideal gas law) or pressure-based solvers are activated to solve for compressible flows. The general Non-Reflecting Boundary Condition is available for use in the **Pressure Inlet**, **Mass-Flow Inlet**, and **Velocity Inlet** dialog boxes when the pressure-based solver is activated to solve for compressible flows.

The example below shows you how to enable the general Non-Reflecting Boundary Condition in the **Pressure Outlet** dialog box. You would follow similar steps for the other boundary condition types mentioned above.

1. Select **pressure-outlet** from the **Boundary Conditions** task page and click the **Edit...** button.

2. In the **Pressure Outlet** dialog box, choose **Non Reflecting** under **Acoustic Wave Model**.

Figure 7.89: The Pressure Outlet Dialog Box With the Non-Reflecting Boundary Enabled



3. For the density-based solver, select one of the two **Exit Pressure Specification** options: **Pressure at Infinity** or **Average Boundary Pressure**. The pressure-based solver uses the **Pressure at Infinity** option and the **Exit Pressure Specification** drop-down list is not available.

   a. The **Pressure at Infinity** boundary is typically used in unsteady calculations or when the exit pressure value is imposed at infinity. The boundary is designed so that the pressure at the boundary relaxes toward the imposed pressure at infinity. The speed at which this relaxation takes place is controlled by the parameter, `sigma`, which can be adjusted in the TUI:

   `define` → `boundary-conditions` → `non-reflecting-bc` → `general-nrbc` → `set`

   In the `set/` submenu, you can set the `sigma` value. The default value for `sigma` is 0.15.

   b. The **Average Boundary Pressure** specification is usually used in steady-state calculations when you want to force the average pressure on the boundary to approach the exit pressure value. The matching of average exit pressure to the imposed average pressure is controlled by the parameter `sigma2` which can be adjusted in the TUI:

   `define` → `boundary-conditions` → `non-reflecting-bc` → `general-nrbc` → `set`

   In the `set/` submenu, you can set the `sigma2` value. The default value for `sigma2` is 5.0.

   Important:

   There is no guarantee that the `sigma2` value of 5.0 will force the average boundary pressure to match the specified exit pressure in all flow situations. In the case where the desired average boundary pressure has not been achieved, you can intervene to adjust the `sigma2` value so that the desired average pressure on the boundary is approached.

4. For the pressure-based solver, you can select one of two **Backflow Pressure Specification** options: **Static Pressure** or **Total Pressure**. For the density-based solver, the **Backflow Pressure Specification** is not shown and the **Total Pressure** option is used by default. Note that backflow is not allowed for pressure inlets, mass-flow inlets, or velocity inlets.

   Important:

   For the pressure-based solver, you should choose **Direction Vector** or **From Neighboring Cell** as the **Backflow Direction Specification Method** if the flow is tangential to the boundary. You should not select the **Normal to the Boundary** option for **Backflow Direction Specification Method** in this case because the face velocity components for this case will be computed from flux as zero during initialization. This initialization will cause the solver convergence to fail.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Usually, the solver can operate at higher CFL values without the NRBCs being turned on. Therefore, the best practice is to first achieve a good stable solution (not necessarily converged) before activating the non-reflecting boundary condition. In many flow situations, the CFL value must be reduced from the normal operation to keep the solution stable. This is particularly true with the density-based implicit solver since the boundary update is done in an explicit manner. A typical CFL value in the density-based implicit solver, with the NRBC activated, is 2.0 and 4.0 in the pressure-based solver.

7.7.3. Impedance Boundary Conditions

The impedance boundary condition (IBC) lies in between a traditional reflective boundary condition and a fully non-reflective boundary condition. It provides the ability to specify a partial reflection in the range from full-reflection to no-reflection. Impedance is a complex value; it is the reflection that changes the amplitude and the phase of the incoming wave. The use of impedance boundary conditions comes in cases where the flow in the simulation is highly influenced by reflected waves from objects outside the computational domain. In such cases the acoustic wave interaction from the larger domain can be modeled in the smaller domain through the use of impedance boundary conditions.

7.7.3.1. Restrictions and Limitations

The impedance boundary condition is available only in the pressure-based solver. It is incompatible with steady-state flow, multiphase, or compressible liquid models (**compressible-liquid** method for density).

7.7.3.2. Theory

Impedance specifies an acoustic resistance in the frequency domain. It is a characteristic of the properties of the media and specific geometry described by a ratio of the pressure perturbation to the normal velocity perturbation at the boundary (Blackstock [19] (p. 5714)). The specific impedance is calculated as follows:

$$\hat{Z}(\omega) = \frac{\hat{p}'}{\rho c \hat{u}'} \tag{7.207}$$

where apostrophe denotes acoustic perturbation and hat denotes quantity in the frequency domain.

Fluent is a time domain solver. It cannot use the impedance from the frequency domain directly. The above expression and all its variables have to be converted to the time domain. After the conversion the relation between pressure and normal velocity perturbations is expressed through a convolution integral.

$$p'(t) = \int_{-\infty}^{\infty} Z(\tau) u'(t-\tau) d\tau \tag{7.208}$$

If impedance, $Z(\tau)$, is unbounded in the time domain, then admittance is used (admittance is the inverse of impedance). Fluent uses the reflection coefficient instead of impedance/admittance, to uniformly treat unbounded cases (Fung [47] (p. 5715)). The reflection coefficient is a ratio between reflected and incoming wave amplitudes at the boundary. It is expressed through the impedance as:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\hat{R}(\omega)=\frac{\hat{Z}(\omega)-1}{\hat{Z}(\omega)+1} \tag{7.209}$$

Using a reflection coefficient, the relation between pressure and normal velocity perturbation is:

$$p'(t)-\rho c u'(t)=\int_{-\infty}^{\infty} R(\tau)\left[p'(t-\tau)+\rho c u'(t-\tau)\right]d\tau \tag{7.210}$$

The discretized form of this expression is used in Fluent to connect acoustic pressure and normal velocity. The computed acoustic perturbations are superimposed onto the pressure and velocity from non-reflecting boundary condition equations. The non-reflecting boundary condition equations provide mean flow values at the boundary, which drive the flow in the domain.

The data for the reflection coefficient are available in the frequency domain. As such they usually do not satisfy the causality and reality conditions. Fluent asks you to provide the reflection coefficient data in the form of a special approximation. This approximation is based on the system theory, which ensures that the reflection coefficient in the time domain will satisfy the above conditions (Fung [46] (p. 5715)). The reflection coefficient is represented as a sum of zero, first and second order systems. The zero system is described with a real value, the first order system is described with a real pole, and the second order system is described with a pair of complex conjugate poles. Introducing a system variable, $s=j\omega$ the complete approximation for the reflection coefficient is:

$$\hat{R}(s)=D+\sum_{k=1}^{N_1}\frac{A_k}{s+\lambda_k}+\sum_{l=1}^{N_2}\left(\frac{B_l+jC_l}{s+\alpha_l-j\beta_l}+\frac{B_l-jC_l}{s+\alpha_l+j\beta_l}\right) \tag{7.211}$$

where D is a real term, N_1 is a number of real poles, λ_k and A_k are real pole and its amplitude, N_2 is a number of complex conjugate pole pairs, α_l, β_l are real and imaginary part of the complex conjugate pole, B_l and C_l are real and imaginary part of the amplitude of the complex pole.

To obey the causality and reality conditions, real pole λ_k, real α_l and imaginary part β_l of the complex conjugate pole should be positive. The passivity condition requires that the absolute value of zero order term D be less than 1. The above restrictions are enforced in the user interface. In addition you should ensure that the absolute value of the reflection coefficient computed by this formula is less than 1.

The experimental data can be specific impedance, reflection coefficient, or absorption coefficient data, and can be obtained from measurements or from an acoustic solver. Processing these data using Fluent or a mathematics package will provide an approximation in terms of first and second order poles.

7.7.3.3. Using the Impedance Boundary Condition

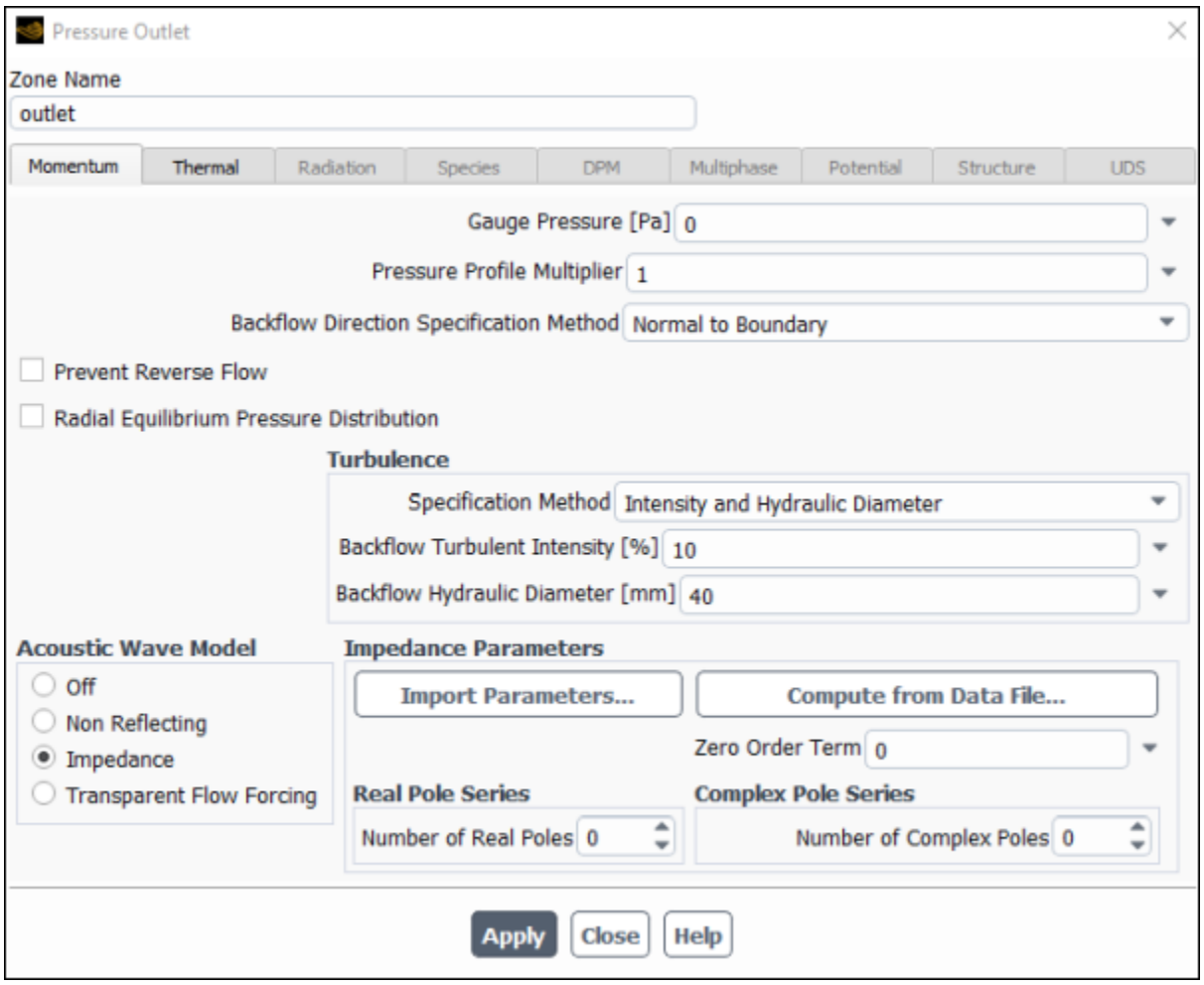
The example below shows you how to enable the impedance boundary condition (IBC) in the **Pressure Outlet** dialog box. Similarly, you can enable IBC in the **Pressure Inlet**, **Velocity Inlet**, and **Mass-Flow Inlet** dialog boxes for compressible flows with the pressure-based solver.

1. Select **pressure-outlet** from the **Boundary Condition** task page and click the **Edit...** button.

2. In the **Pressure Outlet** dialog box, choose **Impedance** under **Acoustic Wave Model**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The dialog box will expand to reveal **Impedance Parameters**.



3. Specify the data for the reflection coefficient in the **Impedance Parameters** group box. This can be done using one of the following methods:

   - If you have a pole / residue file that contains the terms of the approximation formula (Equation 7.211 (p. 1546)), click the **Import Parameters** button and select the file; the values will then be automatically entered in the various fields of the **Impedance Parameters** group box. For information about the proper format of this file as well as how to generate one from experimental impedance data in the frequency domain, see Calculating Impedance Parameters (p. 1548).

   - If you have a data file that contains experimental impedance data in the frequency domain, click the **Compute from Data File** button and use the **Impedance Data Fitting** dialog box that opens to compute the terms in the time domain and apply them to the various fields of the **Impedance Parameters** group box. For details, see Calculating Impedance Parameters (p. 1548).

   - Manually enter values in the various fields for the terms of the approximation formula (Equation 7.211 (p. 1546)):

     1. Specify the **Zero Order Term**, D.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Under **Real Pole Series**, set the **Number of Real Poles** and for each real pole specify the **Pole**, λ_k, and the **Amplitude**, A_k.

3. Under **Complex Pole Series**, set the **Number of Complex Poles** and for each complex pole pair, specify **Pole Real**, α_l, **Pole Imaginary**, β_l, **Amplitude Real**, B_l, and **Amplitude Imaginary**, C_l.

Important:

If flow is tangential to the boundary, then specify either **From Neighboring Cell** or **Direction Vector** for **Back flow Direction Specification Method**. Do not select **Normal to the boundary** because this computes the face velocity components from flux to zero values during initialization, which impairs solver convergence.

The IBC is implemented on top of the non-reflective boundary condition.

Note:

- The mean flow in the domain should be well established before enabling IBC.
- The cell acoustic Courant (CFL) number should be less than 1 in cells adjacent to the boundary where you define an impedance acoustic boundary condition. (For the definition of the **Cell Acoustic Courant Number**, refer to Field Function Definitions (p. 4167)).
- With the impedance acoustic boundary condition, you need to tighten the convergence criteria for continuity to 1e-4 in order to increase the number of coupling iterations between the impedance boundary condition calculation and the solver at each time step.

7.7.3.4. Calculating Impedance Parameters

You can use Fluent to read experimental impedance data in the frequency domain and then compute the terms needed for an approximation of the reflection coefficient as a series of poles / residues in the time domain. The calculation uses an algorithm based on the work of Gustavsen and Semlyen [58] (p. 5716). The terms of the pole / residue series can be directly applied to the various fields of the **Impedance Parameters** group box of a boundary condition dialog box, saved as a pole / residue file (which can be useful for setting up multiple boundary conditions and/or case files), and/or printed in the console; you can also write the fitted frequency / impedance data to an output file, which you can compare to the input data to evaluate how well it fits.

Note:

If you read experimental absorption coefficient data in the frequency domain, the terms of the pole / residue series that Fluent calculates may be different depending on which release or operating system you are using. Such differences will not affect the solution

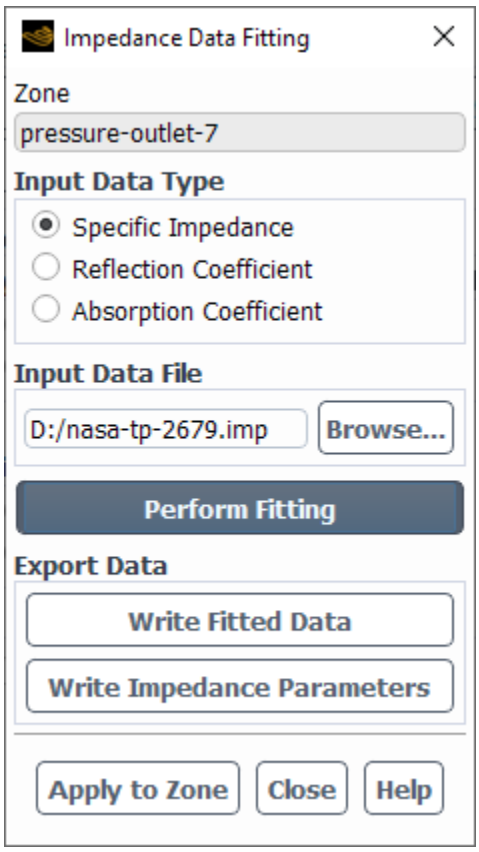
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

results, as only the curves of the fitted frequency / impedance data are used in the impedance boundary condition, and those curves will be consistent. You can verify this by comparing the output files of the fitted frequency / impedance data (see the steps that follow for details about writing such files).

To calculate and enter the impedance parameters, perform the following steps:

1. In the boundary condition dialog box (for example, the **Pressure Outlet** dialog box), click the **Compute from Data File** button in the **Impedance Parameters** group box, to open the **Impedance Data Fitting** dialog box.

   Figure 7.90: The Impedance Data Fitting Dialog Box



2. In the **Impedance Data Fitting** dialog box, make a selection from the **Input Data Type** for the experimental data in the frequency domain. The following types are available:

   - **Specific Impedance**

     This corresponds to Equation 7.207 (p. 1545).

   - **Reflection Coefficient**

     This corresponds to Equation 7.209 (p. 1546).

   - **Absorption Coefficient**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The absorption coefficient (α) is the ratio of the absorbed energy (E_a) to the incident energy (E_i):

$$\alpha = \frac{E_a}{E_i} \tag{7.212}$$

It relates to the reflection coefficient as follows:

$$|R| = \sqrt{1-\alpha} \tag{7.213}$$

3. Enter the name of the **Input Data File** or use the **Browse...** button to select it. The file must be in monitor format, with separate columns for the frequency and the data values. The following are examples:

   - For a specific impedance data file named `nasa-tp-2679.imp`:

```
"nasa-tp-2679"
"frequency" "impedance"
("frequency" "real" "imag")
5000.  0.41 -1.56
10000.  0.46 0.03
15000.  1.08 1.38
20000.  4.99 0.25
25000.  1.26 -1.53
30000.  0.69 -0.24
```

   - For a reflection coefficient data file named `dc-real-complex-rf.rfl`:

```
"dc real complex fitting"
"frequency" "reflection coefficient"
("frequency"  "real"  "imag")
150.0  0.6946227848755014 -0.08340940421826964
300.0  0.5785212268551001 -0.33760951170464315
450.0  0.3844647950039666 -0.36051768580559684
600.0  0.27881191269389494 -0.31875086372470873
750.0  0.22098593107397013 -0.2755872628668373
900.0  0.18674039023283218 -0.23974412499762482
1050.0  0.1650100694074955 -0.21097174097349292
1200.0  0.15043702854946334 -0.18781299172681953
1350.0  0.14022123873194448 -0.16894702736963293
1500.0  0.13279779044634915 -0.15336205026505353
1650.0  0.12724127578751093 -0.14031150077210966
1800.0  0.12297792604119287 -0.12924601721152512
1950.0  0.11963748299365515 -0.11975769596066593
2100.0  0.11697273089207115 -0.11153969759534471
2250.0  0.11481369319289189 -0.10435793597654594
2400.0  0.1130404841694425 -0.09803133050780145
2550.0  0.11156663966112866 -0.09241794300169018
2700.0  0.1103285600257371 -0.08740513783535145
2850.0  0.1092786384820483 -0.0829025079355948
3000.0  0.10838067800958853 -0.07883672147250274
```

   - For an absorption coefficient data file named `dc-real-complex-ab.abs`:

```
"dc-real-complex-rf.in fitting"
"frequency" "absorption coefficient"
("frequency"  "value")
150.0  0.5105420580197505
300.0  0.5513330076846182
450.0  0.722213819623932
600.0  0.8206618042147218
750.0  0.8752168788129335
900.0  0.9076507811848114
1050.0  0.9282626015047457
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
1200.0  0.9420949805798284
1350.0  0.9517949061514425
1500.0  0.9588448283910658
1650.0  0.9641223404870439
1800.0  0.9681718967415641
1950.0  0.9713449669211347
2100.0  0.9738762760879893
2250.0  0.9759272370541228
2400.0  0.9776117071776074
2550.0  0.9790118087260594
2700.0  0.9801879507226298
2850.0  0.9811853533494976
3000.0  0.9820383999816481
```

4. Click the **Perform Fitting** button to calculate the terms needed for an approximation of the reflection coefficient as a series of poles / residues in the time domain. The results are printed in the console (under `Zero Order Term`, `Number of Real Poles`, and so on, which correspond to the terms for the reflection coefficient according to the approximation formula Equation 7.211 (p. 1546)).

   Note that prior to clicking the **Perform Fitting** button, you have the option of revising a few additional settings for the fitting through the following text commands:

   - By default the algorithm will run 20 internal iterations; if this is inappropriate for your case, you can revise it using the following text command: `define/boundary-conditions/impedance-data-fitting/iterations`.
   - By default a convergence tolerance of 1e-6 is used for completing the iterative fitting procedure; if this is inappropriate for your case, you can revise it using the following text command: `define/boundary-conditions/impedance-data-fitting/convergence-tolerance`.
   - By default the minimum value of residues that are kept in the fitting is 1e-6. This residue check helps to eliminate parasitic poles. If the default value is inappropriate for your case, you can revise it using the following text command: `define/boundary-conditions/impedance-data-fitting/residue-tolerance`.
   - You can specify that messages are printed in the console as the fitting calculation progresses by entering the following text command: `define/boundary-conditions/impedance-data-fitting/verbosity 1`.

5. (optional) Click the **Write Fitted Data** button to write a file with fitted frequency / impedance data. The number of lines corresponds to the number of lines in the input data file, and there are separate columns for the frequency and the real and imaginary values. This file can be compared to the input data to evaluate how well it fits.

6. (optional) Click the **Write Impedance Parameters** button to write a pole / residue file with the impedance parameters, which can then be imported using the **Import Parameters** button of a boundary condition dialog box. Such a file can be useful when setting up multiple boundary conditions and/or case files. The following is an example of the format of a pole / residue file:

   ```
   Impedance parameters for nasa-tp-2679.imp:
   Zero Order Term
   6.227176322918e-01
   ```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
Number of Real Poles
 0
Number of Complex Conjugate Poles
 2
Pole Real           Pole Imaginary      Amplitude Real       Amplitude Imaginary
3.462578756313e+03  5.077495917952e+03  -5.463369722031e+03  -2.849664452670e+03
4.754179871958e+03  1.663686613863e+04  -1.614972036282e+03  -5.411609140947e+03
```

7. Click the **Apply to Zone** button to enter the results in the appropriate fields in the **Impedance Parameters** group box of the boundary condition dialog box.

8. Close the **Impedance Data Fitting** dialog box.

9. In the boundary condition dialog box, verify that the **Impedance Parameters** are appropriate and click the **Apply** button.

As an alternative to using the **Impedance Data Fitting** dialog box in a Fluent session, you can instead use the `impedance` utility to calculate the terms of the pole / residue series from a data file, print them, and optionally write them to a file. This utility has some additional functionality, which may be useful in some circumstances. To use the `impedance` utility, enter the following at a command prompt in a terminal or command window:

`utility impedance` [`--help`] [`-p` *P*] [`-f` *F*] [`-i` *I*] [`-t` *T*] [`-r` *R*] [`-y` *Y*] [`-m` *M*] [`-v` *V*] *I* [*R C*]

Note:

The items enclosed in square brackets are optional. Do not type the square brackets.

- `--help` (optional) prints the arguments available for the `impedance` utility.
- `-p` *P* (optional) specifies that the pole / residue mapping data is also written to an output file named *P*.
- `-f` *F* (optional) specifies that fitted frequency / impedance data is also written to an output file named *F*, with separate columns for the frequency and the real and imaginary values. This output file can be compared to the input data to evaluate how well it fits.
- `-i` *I* (optional) specifies that the algorithm uses a specified number (*I*) of internal iterations. By default, 20 iterations will be run.
- `-t` *T* (optional) sets the convergence tolerance (which is an accuracy that is used for completing the iterative fitting procedure) to *T*. By default, the tolerance is set to 1e-6.
- `-r` *R* (optional) sets the residue tolerance (that is, the minimum value of residues that are kept in the fitting) to *R*. This residue check helps to eliminate parasitic poles. By default, the residue tolerance is set to 1e-6.
- `-y` *Y* (optional) specifies the type of data file you are reading. *Y* can be `impedance` (for specific impedance data, which is the default), `reflection` (for reflection coefficient data), or `absorption` (for absorption coefficient data).
- `-m` *M* (optional) can be used with `-y absorption`, and specifies the phase of the absorption coefficient. *M* can be `zero` (the default) or `random`.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- `-v` *V* (optional) sets the verbosity of the progress messages. *V* can be `0` (the default, which prints no progress messages during the calculation) or `1` (which prints messages in the console as the fitting calculation progresses).
- *I* must be replaced with the name of the input file, which provides experimental data in monitor format with separate columns for the frequency and the data.
- *R* and *C* (optional) allow you to specify the number of poles (real and complex conjugate, respectively) that you want used during fitting. With this option, you must ensure that the numbers you enter are appropriate for the data you provide, such that:

$$n \geq 1+R+2C \tag{7.214}$$

  where *n* is the number of data points in your input file. When multiple combinations of values for *R* and *C* can satisfy the previous equation, you should ensure that the values you use yield the lowest possible RMS error (which is the root mean square difference between the fit and the input data, and is printed at the end of the calculation).

Note:

It is recommended that you omit *R* and *C*; when omitted, Fluent will automatically use numbers of poles that yield the lowest RMS error.

The following example demonstrates the use of the `impedance` utility from a command prompt, using the input file described previously.

```
utility impedance nasa-tp-2679.imp -p par.pr -f fitted.imp
```

The following will be printed:

```
command to execute: "python" "C:\ANSYS Inc\v242\fluent\fluent24.2.0\utility\impedance\impedance.pyc"
nasa-tp-2679.imp -p par.pr -f fitted.imp
Impedance parameters for nasa-tp-2679.imp:
Zero Order Term
6.227176322980e-01
Number of Real Poles
 0
Number of Complex Conjugate Poles
 2
Pole Real           Pole Imaginary      Amplitude Real       Amplitude Imaginary
3.462578756313e+04  5.077495917952e+04  -5.463369722031e+04  -2.849664452670e+04
4.754179871958e+04  1.663686613863e+05  -1.614972036282e+04  -5.411609140947e+04
```

The data printed in the terminal / command window and written in the output file named `par.pr` (under `Zero Order Term`, `Number of Real Poles`, and so on) is for the reflection coefficient (according to the approximation formula Equation 7.211 (p. 1546)) and can be imported or entered in the **Impedance Parameters** group box of a boundary condition dialog box (as described in Using the Impedance Boundary Condition (p. 1546)).

A frequency / impedance output file named `fitted.imp` will also be created, containing real and imaginary values for the frequencies defined in the input file.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.7.4. Transparent Flow Forcing Boundary Conditions

It is often desirable in simulations to cut a domain down to a smaller region to model only a part of the larger domain. The treatment at the boundaries created when doing so should be consistent with a slice through the larger domain. That is, incoming transients should be allowed to enter the computational domain and outgoing transients should leave the domain without reflections. Artificial numerical reflections at the boundaries can significantly affect the predicted flow, particularly if the geometry of the numerical domain has Eigen-frequencies that match the frequencies of physical waves propagating through the flow.

The transparent flow forcing treatment provides the capability to model incoming waves while allowing outgoing waves to pass through the boundaries without reflection. It is available in transient simulations of compressible flow using the pressure-based solver and can be applied for velocity inlets, mass-flow inlets, pressure inlets, and pressure outlets.

7.7.4.1. Restrictions and Limitations

The transparent flow forcing condition is available only in the pressure-based solver and it is incompatible with steady-state flow, multiphase, and compressible liquid models (those using the compressible-liquid method for density).

7.7.4.2. Theory

The transparent flow forcing boundary condition is implemented on top of the non-reflecting boundary condition. The non-reflecting part of the boundary condition drives the mean flow. The transient flow at the boundary is split into two waves [77] (p. 5717): incoming wave, $W_{in}\left(\vec{r},t\right)$, and outgoing wave, $W_{out}\left(\vec{r},t\right)$. In the acoustic limit, the intensities of the waves are expressed through pressure and normal velocity perturbations as:

$$W_{in}\left(\vec{r},t\right)=p'+zu'_n$$
$$W_{out}\left(\vec{r},t\right)=p'-zu'_n$$

where $z=\rho c$ is the impedance of the mean flow with density, ρ, and speed of sound, c. The outgoing wave intensity, $W_{out}\left(\vec{r},t\right)$, is computed internally while the incoming wave intensity, $W_{in}\left(\vec{r},t\right)$, is user-specified through a user-defined profile.

7.7.4.3. Using the Transparent Flow Forcing Boundary Condition

The transparent flow forcing (TFF) boundary condition is available at velocity inlets, mass-flow inlets, pressure inlets, and pressure outlets when simulating transient compressible flow using the pressure-based solver. The steps for using the transparent flow forcing boundary condition are as follows:

1. Create a profile for the incoming wave intensity on the boundary. The profile can be created either as a profile file as described in Profiles (p. 1564) or using the `DEFINE_PROFILE` macro as described in Fluent Customization Manual. In the profile definition, you will specify the wave intensity as a function of position and time. For the incoming wave, the pressure and normal velocity perturbations satisfy the relation:

$$p'=zu'_n$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Using this relation the incoming wave intensity at the boundary can be computed from any of the flow perturbation quantities.

- from acoustic pressure, p'

$$W_{in} = 2p'$$

- from acoustic velocity, u'_n

$$W_{in} = 2zu'_n$$

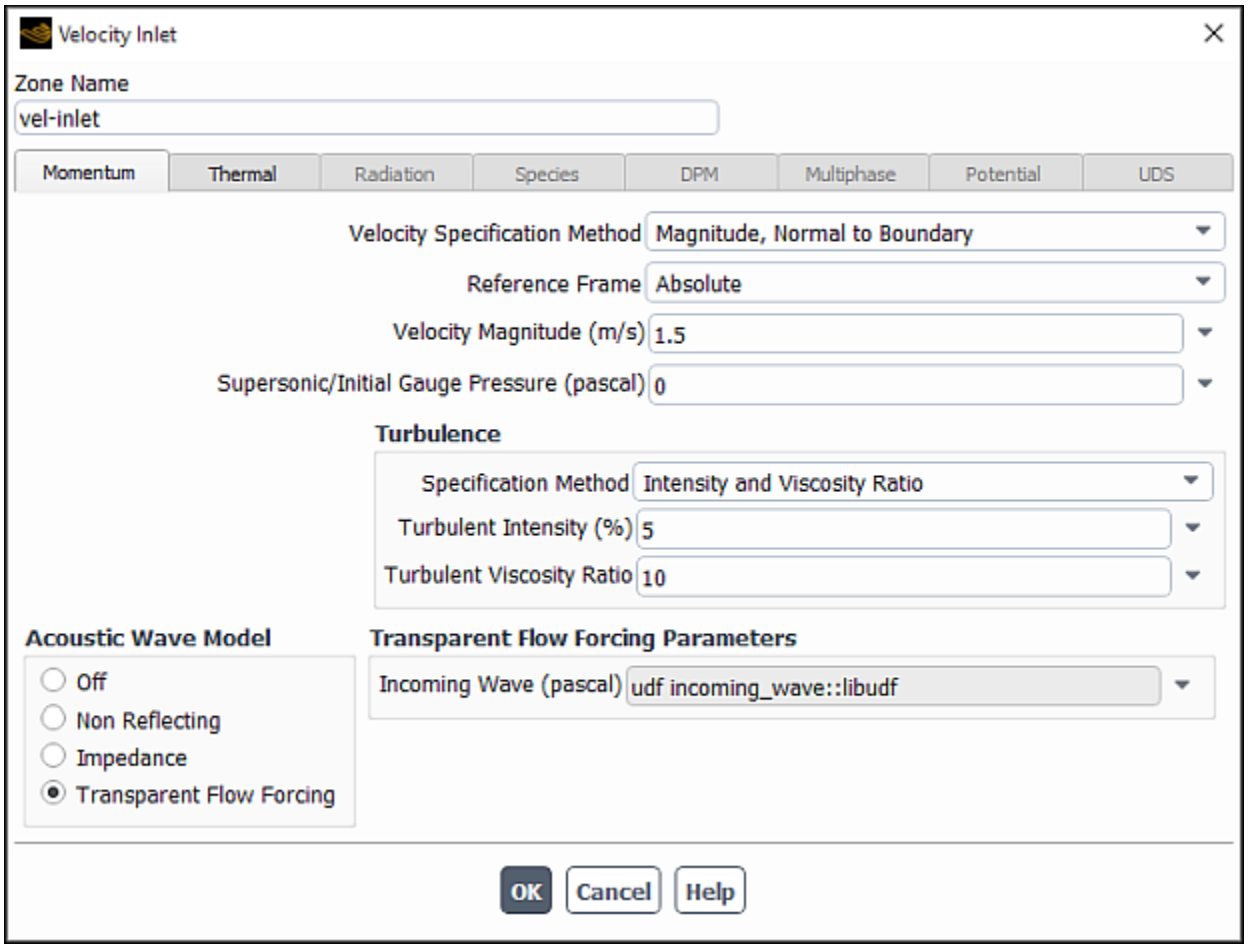
- from acoustic mass flux, m'_n

$$W_{in} = 2cm'_n$$

where c is the speed of sound and $z = \rho c$ is the acoustic impedance.

2. Open the boundary condition settings dialog box for the boundary zone of interest (Setting Cell Zone and Boundary Conditions (p. 1303)).
3. On the **Momentum** tab, select **Transparent Flow Forcing** under **Acoustic Wave Model**.
4. Under **Transparent Flow Forcing Parameters**, select your profile from the drop-down list next to **Incoming Wave**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Important:

- If flow is tangential to the boundary, then specify either **From Neighboring Cell** or **Direction Vector** for **Back flow Direction Specification Method**. Do not select **Normal to the boundary** because this computes the face velocity components from flux to zero values during initialization, which impairs solver convergence.
- The TFF condition is implemented on top of the non-reflective boundary condition. Choose a time step size that will not make the CFL number exceed a value of 1 in the cells adjacent to the TFF boundary.
- The mean flow in the domain should be well established before enabling TFF.

7.8. User-Defined Fan Model

The user-defined fan model in Ansys Fluent allows you to periodically regenerate a profile file that can be used to specify the characteristics of a fan, including pressure jump across the fan, and radial and swirling components of velocity generated by the fan.

For example, consider the calculation of the pressure jump across the fan. You can, through the standard interface, input a constant for the pressure jump, specify a polynomial that describes the

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

pressure jump as a function of axial velocity through the fan, or use a profile file that describes the pressure jump as a function of the axial velocity or location at the fan face. If you use a profile file, the same profile will be used consistently throughout the course of the solution. Suppose, however, that you want to change the profile as the flow field develops. This would require a periodic update to the profile file itself, based upon some instructions that you supply. The user-defined fan model is designed to help you do this.

To use this model, you need to generate an executable that reads a fan profile file that is written by Ansys Fluent, and writes out a modified one, which Ansys Fluent will then read. The source code for this executable can be written in any programming language (Fortran or C, for example). Your program will be called and executed automatically, according to inputs that you supply through the standard interface.

Information about the user-defined fan model is provided in the following sections.

7.8.1. Steps for Using the User-Defined Fan Model

7.8.2. Example of a User-Defined Fan

7.8.1. Steps for Using the User-Defined Fan Model

To make use of the user-defined fan model, follow the steps below.

1. In your model, identify one or more interior faces to represent one or more fan zones.

   Setup → **Boundary Conditions**

2. Input the name of your executable and the instructions for reading and writing profile files in the User-Defined Fan Model Dialog Box (p. 5647).

   User Defined → **Model Specific** → **Fan Model...**

3. Initialize the flow field and the profile files.

4. Enter the fan parameters using the standard Fan Dialog Box (p. 5002) (opened from the Boundary Conditions Task Page (p. 4991)).

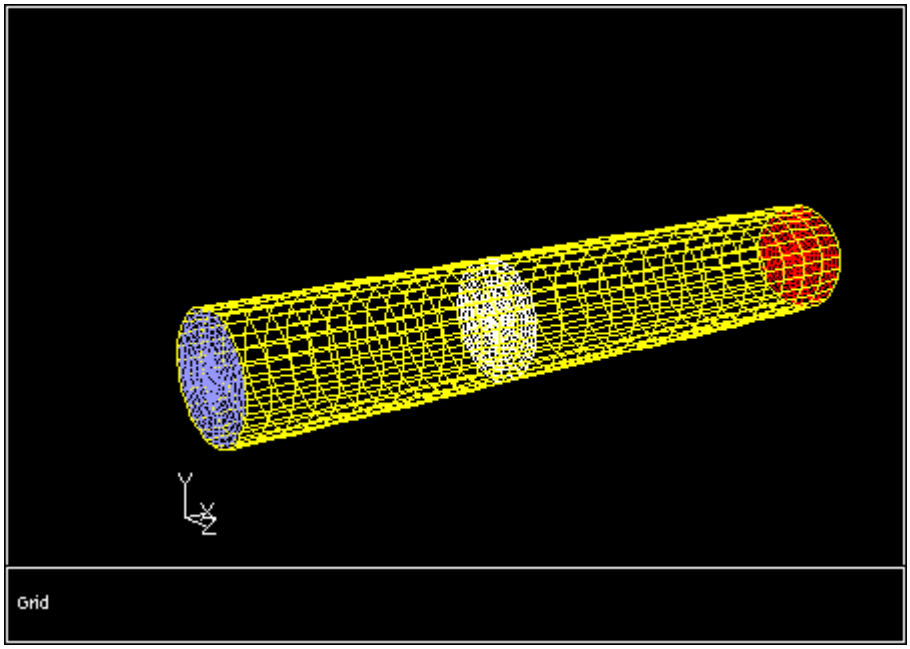
5. Perform the calculation.

7.8.2. Example of a User-Defined Fan

Usage of the user-defined fan model is best demonstrated by an example. With this in mind, consider the domain shown in Figure 7.91: The Inlet, Fan, and Pressure Outlet Zones for a Circular Fan Operating in a Cylindrical Domain (p. 1558). An inlet supplies air at 10 m/s to a cylindrical region, 1.25 m long and 0.2 m in diameter, surrounded by a symmetry boundary. At the center of the flow domain is a circular fan. A pressure outlet boundary is at the downstream end.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.91: The Inlet, Fan, and Pressure Outlet Zones for a Circular Fan Operating in a Cylindrical Domain



Solving this problem with the user-defined fan model will cause Ansys Fluent to periodically write out a radial profile file with the current solution variables at the fan face. These variables (static pressure, pressure jump, axial, radial, and swirling (tangential) velocity components) will represent averaged quantities over annular sections of the fan. The sizes of the annular regions are determined by the size of the fan and the number of radial points to be used in the profiles.

Once the profile file is written, Ansys Fluent will invoke an executable, which will perform the following tasks:

1. Read the profile file containing the current flow conditions at the fan.
2. Perform a calculation to compute new values for the pressure jump, radial velocity, and swirl velocity for the fan.
3. Write a new profile file that contains the results of these calculations.

Ansys Fluent will then read the new profile file and continue with the calculation.

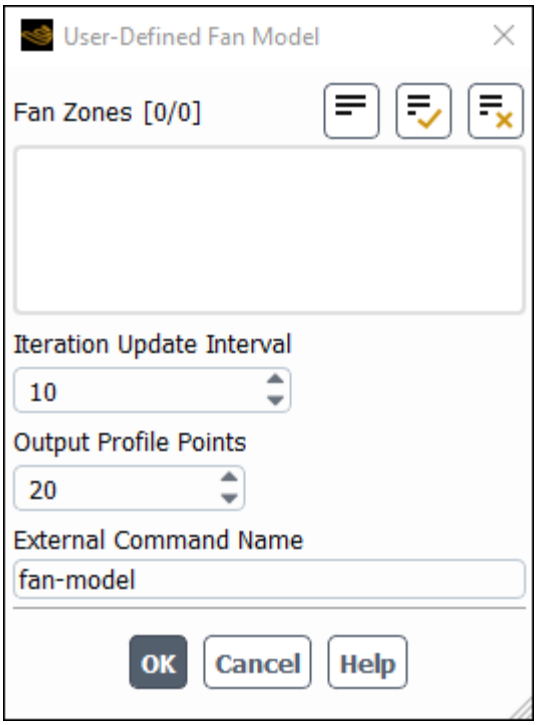
7.8.2.1. Setting the User-Defined Fan Parameters

Specification of the parameters for the user-defined fan begins in the User-Defined Fan Model Dialog Box (p. 5647) (Figure 7.92: The User-Defined Fan Model Dialog Box (p. 1559)).

User Defined → Model Specific → Fan Model...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.92: The User-Defined Fan Model Dialog Box



In this dialog box, you can select the fan zone(s) on which your executable will operate under **Fan Zones**. In this example, there is only one fan, **fan-8**. If you have multiple fan zones in a simulation, for which you have different profile specifications, you can select them all at this point. Your executable will be able to differentiate between the fan zones because the zone ID for each fan is included in the solution profile file. The executable will be invoked once for each zone, and separate profile files will be written for each.

The executable file will be called on to update the profile file periodically, based on the input for the **Iteration Update Interval**. An input of `10`, as shown in the dialog box, means that the fan executable in this example will act every 10 iterations to modify the profile file.

The number of points in the profile file to be written by Ansys Fluent is entered under **Output Profile Points**. This profile file can have the same or a different number of points as the one that is written by the external executable.

Finally, the name of the executable should be entered under **External Command Name**. In the current example, the name of the executable is `fan-model`.

Important:

If the executable is not located in your working directory, then you must type the complete path to the executable.

7.8.2.2. Sample User-Defined Fan Program

The executable file will be built from the Fortran program, `fantest.f`, which is shown below.

```
c
c  This program is invoked at intervals by Ansys Fluent to
```

```
c  read a profile-format file that contains radially
c  averaged data at a fan face, compute new pressure-jump
c  and swirl-velocity components, and write a new profile
c  file that will subsequently be read by Ansys Fluent to
c  update the fan conditions.
c
c  Usage: fantest input_profile output_profile
c
  integer npmax
  parameter (npmax = 900)
  integer inp         ! input: number of profile points
  integer iptype      ! input: profile type (0=radial, 1=point)
  real ir(npmax)      ! input: radial positions
  real ip(npmax)      ! input: pressure
  real idp(npmax)     ! input: pressure-jump
  real iva(npmax)     ! input: axial velocity
  real ivr(npmax)     ! input: radial velocity
  real ivt(npmax)     ! input: tangential velocity
  character*80 zoneid
  integer rfanprof    ! function to read a profile file
  integer status
c
  status = rfanprof(npmax,zoneid,iptype,
 $  inp,ir,ip,idp,iva,ivr,ivt)
  if (status.ne.0) then
    write(*,*) 'error reading input profile file'
  else
    do 10 i = 1, inp
      idp(i) = 200.0 - 10.0*iva(i)
      ivt(i) = 20.0*ir(i)
      ivr(i) = 0.0
 10  continue
  call wfanprof(6,zoneid,iptype,inp,ir,idp,ivr,ivt)
  endif
  stop
  end
```

After the variable declarations, which have comments on the right, the subroutine `rfanprof` is called to read the profile file, and pass the current values of the relevant variables (as defined in the declaration list) to `fantest`. A loop is done on the number of points in the profile to compute new values for:

- The pressure jump across the fan, `idp`, which in this example is a function of the axial velocity, `iva`.
- The swirling or tangential velocity, `ivt`, which in this example is proportional to the radial position, `ir`.
- The radial velocity, `ivr`, which in this example is set to zero.

After the loop, a new profile is written by the subroutine `wfanprof`, shown below. (For more information on profile file formats, see Profile File Formats (p. 1565).)

```
 subroutine wfanprof(unit,zoneid,ptype,n,r,dp,vr,vt)
c
c  writes an Ansys Fluent profile file for input by the
c  user fan model
c
  integer unit       ! output unit number
  character*80 zoneid
  integer ptype      ! profile type (0=radial, 1=point)
  integer n          ! number of points
  real r(n)          ! radial position
  real dp(n)         ! pressure jump
  real vr(n)         ! radial velocity
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
  real vt(n)         ! tangential velocity
  character*6 typenam
  if (ptype.eq.0) then
    typenam = 'radial'
  else
    typenam = 'point'
  endif
  write(unit,*) '((', zoneid(1:index(zoneid,'\0')-1), ' ',
 $  typenam, n, ')'
  write(unit,*) '(r'
  write(unit,100) r
  write(unit,*) ')'
  write(unit,*) '(pressure-jump'
  write(unit,100) dp
  write(unit,*) ')'
  write(unit,*) '(radial-velocity'
  write(unit,100) vr
  write(unit,*) ')'
  write(unit,*) '(tangential-velocity'
  write(unit,100) vt
  write(unit,*) ')'
 100 format(5(e15.8,1x))
  return
  end
```

This subroutine will write a profile file in either radial or point format, based on your input for the integer `ptype`. (See Profiles (p. 1564) for more details on the types of profile files that are available.) The names that you use for the various profiles are arbitrary. Once you have initialized the profile files, the names you use in `wfanprof` will appear as profile names in the Fan Dialog Box (p. 5002).

7.8.2.3. Initializing the Flow Field and Profile Files

The next step in the setup of the user-defined fan is to initialize (create) the profile files that will be used. To do this, first initialize the flow field with the Solution Initialization Task Page (p. 5179) (using the velocity inlet conditions, for example), and then type the command `(update-user-fans)` in the console window. (The parentheses are part of the command, and must be typed in.)

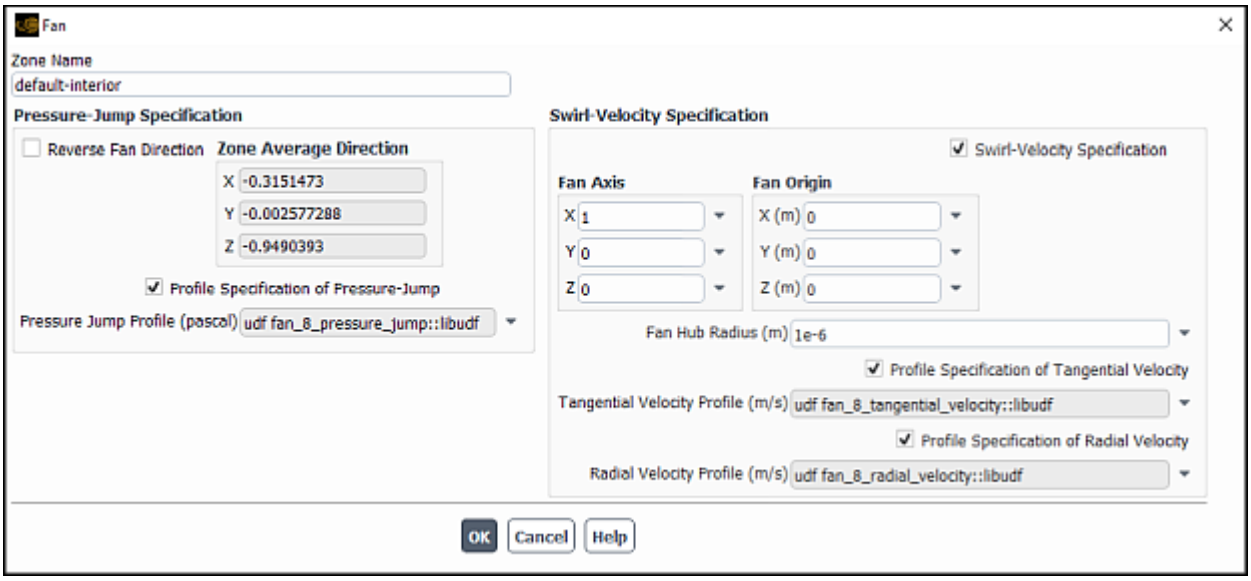
This will create the profile names that are given in the subroutine `wfanprof`.

7.8.2.4. Selecting the Profiles

Once the profile names have been established, you will need to visit the Fan Dialog Box (p. 5002) (Figure 7.93: The Fan Dialog Box (p. 1562)) to complete the problem setup. (See Fan Boundary Conditions (p. 1504) for general information on using the **Fan** dialog box.)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.93: The Fan Dialog Box



At this time, the **Fan Axis**, **Fan Origin**, and **Fan Hub Radius** can be entered, along with the choice of profiles for the calculation of pressure jump, tangential velocity, and radial velocity. With the profile options enabled, you can select the names of the profiles from the drop-down lists. In the dialog box above, the selected profiles are named **fan_8_pressure_jump**, **fan_8_tangential_velocity**, and **fan_8_radial_velocity**, corresponding to the names that were used in the subroutine `wfanprof`.

7.8.2.5. Performing the Calculation

The solution is now ready to run. As it begins to converge, the report in the console window shows that the profile files are being written and read every 10 iterations:

```
    iter   continuity   x-velocity   y-velocity   z-velocity          k
!      1   residual normalization factors changed (continuity
       1   1.0000e+00   1.0000e+00   1.0000e+00   1.0000e+00   1.0000e+00
!      2   residual normalization factors changed (continuity
       2   1.0000e+00   1.0000e+00   1.0000e+00   1.0000e+00   9.4933e-01
       3   6.8870e-01   7.2663e-01   7.3802e-01   7.5822e-01   6.1033e-01
       .       .            .            .            .            .
       .       .            .            .            .            .
       .       .            .            .            .            .
       9   2.1779e-01   9.8139e-02   3.0497e-01   2.9609e-01   2.8612e-01
 Writing "fan-8-out.prof"...
 Done.
 Reading "fan-8-in.prof"...

 Reading profile file...
      10   "fan-8"  radial-profile points, r, pressure-jump,
                          radial-velocity, tangential-velocity.

 Done.
      10   1.7612e-01   7.4618e-02   2.5194e-01   2.4538e-01   2.4569e-01
      11   1.6895e-01   8.3699e-02   2.0316e-01   2.0280e-01   2.1169e-01
       .       .            .            .            .            .
       .       .            .            .            .            .
       .       .            .            .            .            .
```

The file `fan-8-out.prof` is written out by Ansys Fluent and read by the executable `fantest`. It contains values for pressure, pressure jump, axial velocity, radial velocity, and tangential velocity

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

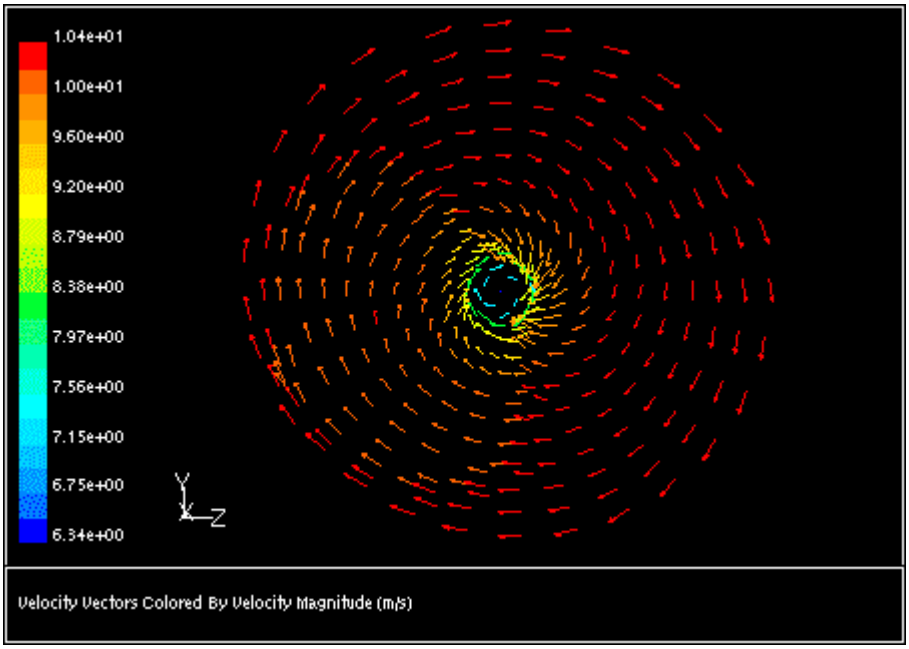
at 20 radial locations at the site of the fan. The file `fan-8-in.prof` is generated by `fantest` and contains updated values for pressure jump and radial and tangential velocity only. It is therefore a smaller file than `fan-8-out.prof`. The prefix for these files takes its name from the fan zone with which the profiles are associated. An example of the profile file `fan-8-in.prof` is shown below. This represents the last profile file to be written by `fantest` during the convergence history.

```
((fan-8 radial 10)
(r
0.24295786E-01 0.33130988E-01 0.41966137E-01 0.50801374E-01 0.59636571E-01
0.68471842E-01 0.77307090E-01 0.86142287E-01 0.94963484E-01 0.95353782E-01
)
(pressure-jump
0.10182057E+03 0.98394081E+02 0.97748657E+02 0.97787750E+02 0.97905228E+02
0.98020668E+02 0.98138817E+02 0.98264198E+02 0.98469681E+02 0.98478783E+02
)
(radial-velocity
0.00000000E+00 0.00000000E+00 0.00000000E+00 0.00000000E+00 0.00000000E+00
0.00000000E+00 0.00000000E+00 0.00000000E+00 0.00000000E+00 0.00000000E+00
)
(tangential-velocity
0.48591572E+00 0.66261977E+00 0.83932275E+00 0.10160275E+01 0.11927314E+01
0.13694369E+01 0.15461419E+01 0.17228458E+01 0.18992697E+01 0.19070756E+01
)
```

7.8.2.6. Results

A plot of the transverse velocity components at the site of the fan is shown in Figure 7.94: Transverse Velocities at the Site of the Fan (p. 1563). As expected, there is no radial component, and the tangential (swirling) component increases with radius.

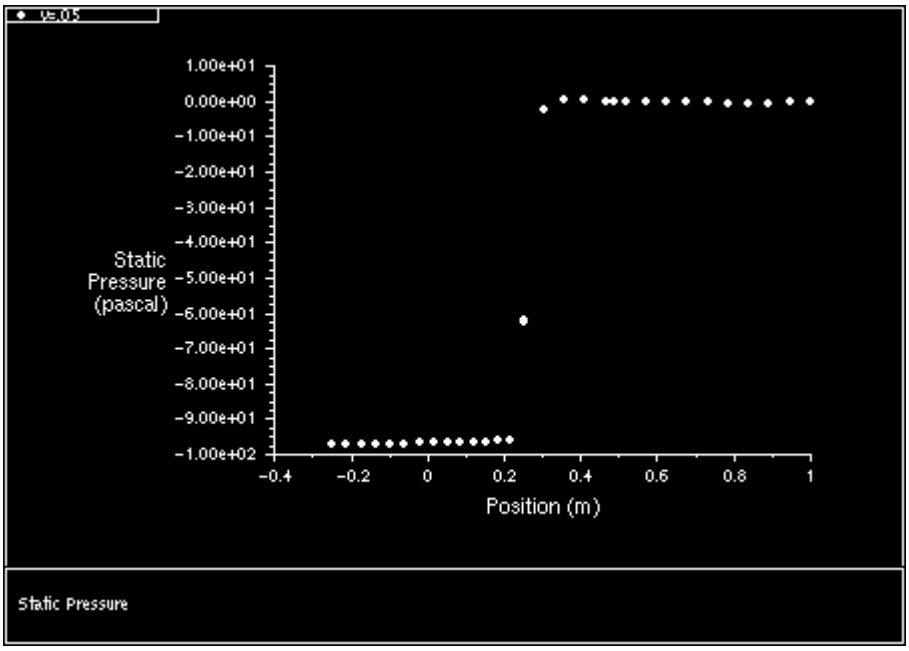
Figure 7.94: Transverse Velocities at the Site of the Fan



As a final check on the result, an XY plot of the static pressure as a function of x position is shown (Figure 7.95: Static Pressure Jump Across the Fan (p. 1564)). This XY plot is made on a line at y =0.05 m, or at about half the radius of the duct. According to the input file shown above, the pressure jump at the site of the fan should be approximately 97.8 Pa/m. Examination of the figure supports this finding.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.95: Static Pressure Jump Across the Fan



7.9. Profiles

Profiles can be boundary conditions, cell zone conditions, and initial conditions for discrete phases. Ansys Fluent provides a very flexible profile definition mechanism. This feature allows you to use experimental data, data calculated by an external program, or data written from a previous solution using the Write Profile Dialog Box (p. 4988) (as described in Reading and Writing Profile Files (p. 964)) as the boundary condition for a variable.

Information about profiles is presented in the following subsections:

7.9.1. Profile Specification Types

7.9.2. Profile File Formats

7.9.3. Using Profiles

7.9.1. Profile Specification Types

The following is a list of the six types of profiles that can be read into Ansys Fluent, as well as information about the interpolation method employed by Ansys Fluent for each type.

- Point profiles are specified by an unordered set of n points: $\left(x_i, y_i, v_i\right)$ for 2D problems or $\left(x_i, y_i, z_i, v_i\right)$ for 3D problems, where $1 \le i \le n$. Profiles written using the **Write Profile** dialog box and profiles of experimental data in random order are examples of point profiles.

  Ansys Fluent will interpolate the point cloud to obtain values at the boundary faces. The default interpolation method for the unstructured point data is zeroth order. That is, for each cell face at the boundary, the solver uses the value from the profile file located closest to the cell. Therefore, to get an accurate specification of an inlet profile using the default interpolation method, your

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

profile file should contain a sufficiently high point density. For information about other available interpolation methods for point profiles, see Using Profiles (p. 1570).

- Line profiles are specified for 2D problems by an ordered set of n points: $\left(x_i, y_i, v_i\right)$, where $1 \le i \le n$. Zeroth-order interpolation is performed between the points. An example of a line profile is a profile of data obtained from an external program that calculates a boundary-layer profile.

- Mesh profiles are specified for 3D problems by an m by n mesh of points: $\left(x_{ij}, y_{ij}, z_{ij}, v_{ij}\right)$, where $1 \le i \le m$ and $1 \le j \le n$. Zeroth-order interpolation is performed between the points. Examples of mesh profiles are profiles of data from a structured mesh solution and experimental data in a regular array.

- Radial profiles are specified for 2D and 3D problems by an ordered set of n points: $\left(r_i, v_i\right)$, where $1 \le i \le n$. The data in a radial profile are a function of radius only. Linear interpolation is performed between the points, which must be sorted in ascending order of the r field. The axis for the cylindrical coordinate system is determined as follows:

  – For 2D problems, it is the z -direction vector through (0,0).

  – For 2D axisymmetric problems, it is the x -direction vector through (0,0).

  – For 3D problems involving a swirling fan, it is the fan axis defined in the Fan Dialog Box (p. 5002) (unless you are using local cylindrical coordinates at the boundary, as described below).

  – For 3D problems without a swirling fan, it is the rotation axis of the adjacent fluid zone, as defined in the Fluid Dialog Box (p. 4961) (unless you are using local cylindrical coordinates at the boundary, as described below).

  – For 3D problems in which you are using local cylindrical coordinates to specify conditions at the boundary, it is the axis of the specified local coordinate system.

- Axial profiles are specified for 3D problems by an ordered set of n points: $\left(z_i, v_i\right)$, where $1 \le i \le n$. The data in an axial profile are a function of the axial direction. Linear interpolation is performed between the points, which must be sorted in ascending order of the z field.

- Transient profiles are specified for 2D and 3D profiles by an ordered set of n points: $\left(t_i, v_{0,i}, v_{1,i}, v_{2,i}, \ldots\right)$. Linear interpolations are done between the points which must be sorted in ascending order of the t (time or crank angle) field. Examples of transient profiles are transient cell zone and boundary conditions (see Transient Cell Zone and Boundary Conditions (p. 1523)) and point properties for particle injections (see Point Properties for Transient Injections (p. 2778)).

7.9.2. Profile File Formats

There are two profile format options:

7.9.2.1. Standard Profiles

7.9.2.2. CSV Profiles

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.9.2.1. Standard Profiles

The format of the profile files is fairly simple. The file can contain an arbitrary number of profiles. Each profile consists of a header that specifies the profile name, profile type (`point`, `line`, `mesh`, `radial`, or `axial`), and number of defining points, and is followed by an arbitrary number of named "fields". Some of these fields contain the coordinate points and the rest contain boundary data.

Important:

Keep profile names to 64 characters or less. If the profile name contains more than 64 characters, Fluent truncates the name to the first 64 characters.

All quantities, including coordinate values, must be specified in SI units because Ansys Fluent does not perform unit conversion when reading profile files.

Parentheses are used to delimit profiles and the fields within the profiles. Any combination of tabs, spaces, and newlines can be used to separate elements.

Important:

In the general format description below, " | " indicates that you should input only one of the items separated by |'s and " `...` " indicates a continuation of the list.

```
((profile1-name point|line|radial n)
 (field1-name a1 a2 ... an)
 (field2-name b1 b2 ... bn)
.
.
.
 (fieldf-name f1 f2 ... fn))

((profile2-name mesh m n)
  (field1-name  a11 a12 ... a1n
                a21 a22 ... a2n
                .
                .
                .
                am1 am2 ... amn)
.
.
.
(fieldf-name   f11  f12 ... f1n
               f21  f22 ... f2n
               .
               .
               .
               fm1 fm2 ... fmn))
```

Profile names must have all lowercase letters (for example, `name`). Uppercase letters in profile names are not acceptable. Each profile of type `point`, `line`, and `mesh` must contain fields with names `x`, `y`, and, for 3D, `z`. Each profile of type `radial` must contain a field with name `r`. Each profile of type `axial` must contain a field with name `z`. The rest of the names are arbitrary, but must be valid Scheme symbols. For compatibility with old-style profile files, if the profile type is missing, `point` is assumed.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.9.2.1.1. Example

A typical usage of a profile file is to specify the profile of the boundary layer at an inlet. For a compressible flow calculation, this will be done using profiles of total pressure, k, and ε. For an incompressible flow, it might be preferable to specify the inlet value of streamwise velocity, together with k and ε.

Below is an example of a profile file that does this:

```
((turb-prof point 8)
 (x
     4.00000E+00  4.00000E+00  4.00000E+00  4.00000E+00
     4.00000E+00  4.00000E+00  4.00000E+00  4.00000E+00 )
 (y
     1.06443E-03  3.19485E-03  5.33020E-03  7.47418E-03
     2.90494E-01  3.31222E-01  3.84519E-01  4.57471E-01 )
 (u
     5.47866E+00  6.59870E+00  7.05731E+00  7.40079E+00
     1.01674E+01  1.01656E+01  1.01637E+01  1.01616E+01 )
 (tke
     4.93228E-01  6.19247E-01  5.32680E-01  4.93642E-01
     6.89414E-03  6.89666E-03  6.90015E-03  6.90478E-03 )
 (eps
     1.27713E+02  6.04399E+01  3.31187E+01  2.21535E+01
     9.78365E-03  9.79056E-03  9.80001E-03  9.81265E-03 )
 )
```

7.9.2.2. CSV Profiles

CSV profiles are compatible with spreadsheets and are available for both reading and writing in Fluent. Many profile types can be written in CSV format, including:

- Radial
- Axial
- Point (X, Y, Z)
- Transient
- Transient-Periodic

If writing a profile using the **Write Profile** dialog box, you can save the profile in CSV format by saving with a `.csv` extension.

Files may also be written by other Ansys products including:

- Ansys Mechanical (**EXOPTION**)
- Ansys CFX (Profile Data Format)

Applications using the `.csv` profile format include:

- Aerodynamic Damping (Blade Flutter Analysis) (p. 1972)
- Blade Film Cooling for Gas Turbines (p. 2013)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Formatting Rules:

CSV profiles must be setup so they can be read and interpreted correctly in Fluent:

1. The profile file must contain the section identifiers shown in Table 7.4:**CSV Profile Section Identifiers** (p. 1568). Some of the identifiers are optional because not all profiles will contain these additional fields, and they are noted as such. The ordering presented in the table is the ordering that should be present in the file.

   Table 7.4: CSV Profile Section Identifiers

Section Identifiers	Remark
[Name]	The next line should have the profile name. *Note- if the profile name has space characters, they will be replaced with '-' (space character is not allowed).*
(Optional) [Parameters]	The next line should list the name of each parameter and its value. Parameters that have units should have the appropriate unit in brackets next to the value of the parameter.
(Optional) [Spatial Fields]	The next line should list the names and units of the Cartesian coordinate system. This indicates whether the profile is 1D, 2D, or 3D (with the number of values corresponding to the dimensions of the profile).
[Data]	Data values begin on the next line.
(Optional) [Faces]	The next lines provide the face-node connectivity that allows for previewing the profile as a surface.

2. After the `[Data]` identifier you must label the type of data that you are providing; refer to Table 7.5: Profile Types and the Corresponding Required Field Labels (p. 1568) for the correct labels. If Fluent does not find one of the expected labels, the profile read will fail.

   Table 7.5: Profile Types and the Corresponding Required Field Labels

Profile Type	Required Field Label
radial	radius
axial	axis
point	X, Y, Z
transient	time, angle
transient-periodic	time-periodic, angle-periodic

Important:

The following labels are reserved for X-axis functions: `x`, `y`, `z`, `r`, `time`, and `angle`. If your file contains fields labeled with any of these reserved names, then those fields

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

will be treated as X-axis functions. For fields to be considered as Y-axis functions, you must use names that are not reserved for X-axis functions.

3. If there is an unsupported section identifier, Fluent ignores the section and looks for the `[Name]` and `[Data]` section identifiers.

4. Multiple profiles may be included in a single file.

Important:

- Only use commas as value separators. Space characters should not be used as they will confuse external spreadsheet applications.
- If a profile contains only `Name` and `Data` fields, only SI values are supported. Quantities cannot be entered with units. However, if your profile contains any of the additional fields, then units must be provided and they are not required to be SI.

CSV Profile File Examples

A file containing one profile:

```
[Name]
outlet
[Data]
x,y,z,x-velocity
-0.00036448459,0.0068932362,3,0.5
0.0014999653,-0.0090896944,3,0.5
-0.0014358073,-0.0094413245,3,    0.5
0.0022810854,0.014916174,3,0.5
-0.0024638004,0.014873175,3,0.5
-0.0069573424,0.013364096,3,0.5
```

A file containing two profiles, the first one is spatial (steady-state) and the second is transient.

```
[Name]
outlet
[Data]
x,y,z,x-velocity
-0.00036448459,0.0068932362,3,0.05
0.0014999653,-0.0090896944,3,0.05

[Name]
transient-temperature
[Data]
time,temperature
1.1,300
1.2,350
1.3,400
```

A file containing one profile with all the optional fields

```
# Profile Description
[Name],,,,,,,,
example-profile-name,,,,,,,,
,,,,,,,,
[Parameters],,,,,,,,
example-parameter = value,,,,,,,,
```

```
example-parameter2 = value [unit],,,,,,,,
example-parameter3 = value [unit],,,,,,,,
,,,,,,,,
[Spatial Fields],,,,,,,,
Initial X [m], Initial Y [m], Initial Z [m],,,,,,
,,,,,,,,
[Data],,,,,,,,
Initial X [m], Initial Y [m], Initial Z [m], norm meshdisptot x [ ], norm meshdisptot y [ ], norm meshdisptot z
0.26249194,-0.15531575,-2.57E-02,-0.26383738,-0.27941432,-0.10469795,0.24519758,-1.56E-02,0.88077954
0.26336359,-0.15383309,-2.57E-02,-0.26204233,-0.28049517,-0.12765918,0.23875593,-1.19E-02,0.88023709
0.26452365,-0.15182963,-2.57E-02,-0.25952255,-0.28202344,-0.15844249,0.22989663,-6.86E-03,0.87890624
,,,,,,,,
[Faces],,,,,,,,
175,3045,3016,2790,,,,,
2790,3016,3001,2789,,,,,
2789,3001,2986,2788,,,,,
2788,2986,2971,2787,,,,,
```

7.9.3. Using Profiles

The procedure for using a profile to define a particular cell zone or boundary condition is outlined below.

1. Create a file that contains the desired profile, following the format described in Profile File Formats (p. 1565).

2. Read the profile using the **Read...** button in the Profiles Dialog Box (p. 4981) (Figure 7.96: The Profiles Dialog Box (p. 1572))

   or the **File/Read/Profile...** ribbon tab item.

   Setup → **Cell Zone Conditions** → **Profiles...**

   Setup → **Boundary Conditions** → **Profiles...**

   File → **Read** → **Profile...**

   Note that if you use the **Profiles** dialog box to read a file, and a profile in the file has the same name as an existing profile, the old profile will be overwritten.

3. If it is a point profile, you can choose the method of interpolation using the **Profiles** dialog box (Figure 7.96: The Profiles Dialog Box (p. 1572)):

   Setup → **Cell Zone Conditions** → **Profiles...**

   Setup → **Boundary Conditions** → **Profiles...**

   Select the point profile in the **Profile** selection list. Then select one of the three choices in the **Interpolation Method** list and click the **Apply** button. The three choices include:

   - **Constant**

     This method is zeroth-order interpolation. For each cell face at the boundary, the solver uses the value from the profile file located closest to the cell. Therefore, the accuracy of the interpol-

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ated profile will be affected by the density of the data points in your profile file. This is the default interpolation method for point profiles.

- **Inverse Distance**

  This method assigns a value to each cell face at the boundary based on weighted contributions from the values in the profile file. The weighting factor is inversely proportional to the distance between the profile point and the cell face center.

- **Least Squares**

  This method assigns values to the cell faces at the boundary through a first-order interpolation method that tries to minimizes the sum of the squares of the offsets (residuals) between the profile data points and the cell face centers. The least squares solution is found using Singular Value Decomposition (SVD).

Note:

The **Inverse Distance** and **Least Squares** profile interpolation methods are not applicable when a profile is attached to cell zones. When the cell face is outside the profile-covered zone, the Inverse Distance and Least Squares profile interpolation methods will be switched to the Constant interpolation method to improve the interpolation robustness.

For information about the interpolation methods employed for other profile types (that is, line, mesh, radial, or axial profiles), see Profile Specification Types (p. 1564).

4. You can attach a profile to a reference frame so that the profile will rotate according to the reference frame. This can be used to model the clocking effect of a rotor on a stator, for example. Under **Reference Frames**, select the reference frame to attach the chosen profile to.

5. In the boundary conditions dialog boxes (for example, the **Velocity Inlet** and **Pressure Inlet** dialog boxes), the fields defined in the profile file (and those defined in any other profile file that you have read in) will appear in the drop-down list to the right of or below each parameter for which profile specification is allowed. To use a particular profile, select it in the appropriate list.

6. Initialize the solution to interpolate the profile.

Note:

You can use a profile file or the `DEFINE_PROFILE` user-defined function to specify volumetric source terms. If you specify the source terms with a profile, you will not have access to the central coefficient of the equations solved in order to linearize the source term. You will need to use a user-defined function to do this.

For more information on UDFs, refer to the Fluent Customization Manual.

7.9.3.1. Checking and Deleting Profiles

Each profile file contains one or more profiles, and each profile has one or more fields defined in it. Once you have read in a profile file, you can check which fields are defined in each profile, and

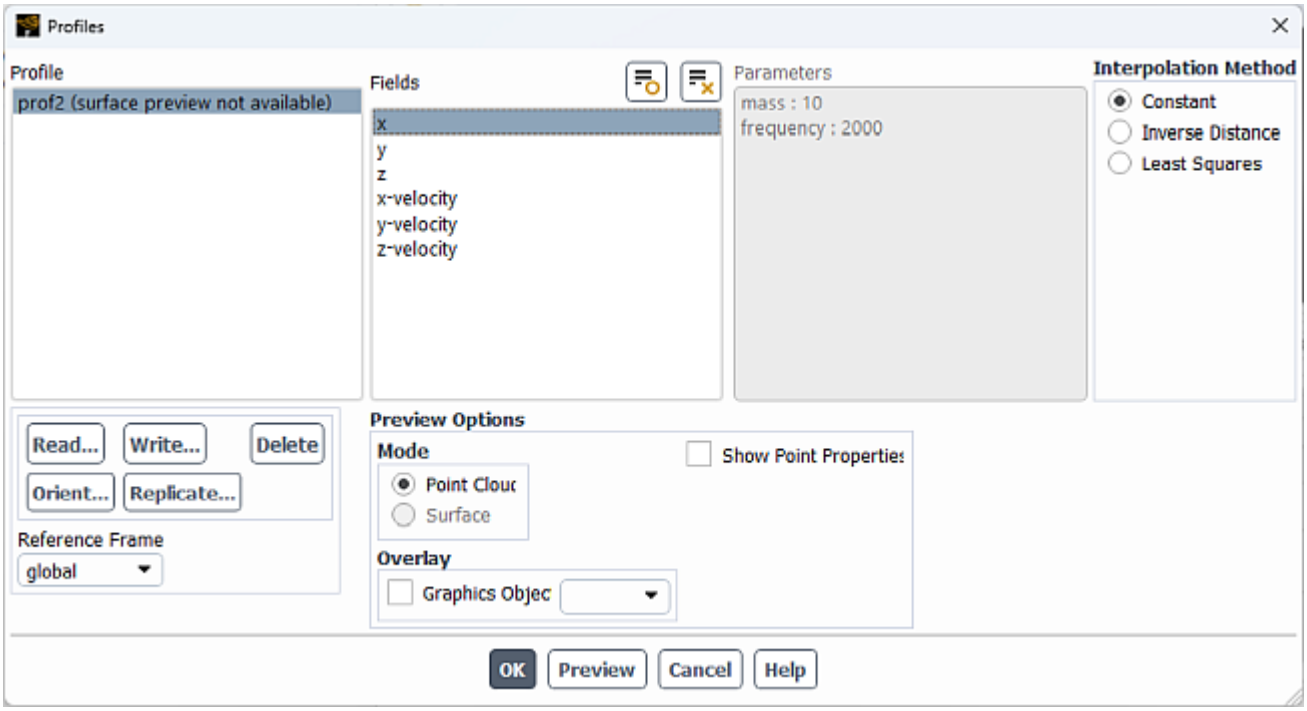
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

you can also delete a particular profile. These tasks are accomplished in the Profiles Dialog Box (p. 4981) (Figure 7.96: The Profiles Dialog Box (p. 1572)).

Setup → Cell Zone Conditions → Profiles...

Setup → Boundary Conditions → Profiles...

Figure 7.96: The Profiles Dialog Box



To check which fields are defined in a particular profile, select the profile name in the **Profile** list. The available fields in that file will be displayed in the **Fields** list. In Figure 7.96: The Profiles Dialog Box (p. 1572), the profile fields from the profile file of Example (p. 1567) are shown.

Any parameters defined within the profile will be listed under **Parameters** with it's accompanying value.

To delete a profile, select it in the **Profile** list and click the **Delete** button. When a profile is deleted, all fields defined in it will be removed from the **Fields** list.

Important:

If you are deleting a profile that is being used as an input for a boundary condition, a cell zone condition, or both, the input will be in a undefined/invalid state after deletion of the profile. To resolve this issue, you must manually update the inputs to a valid input type such as **constant**, **udf/profile**, or **expression**.

If you are deleting an existing profile to replace it with a new profile, use the following procedure:

1. Delete the existing profile.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Change the parameters defined by the existing profile to **Constant**.
3. Read the new profile and apply as desired.

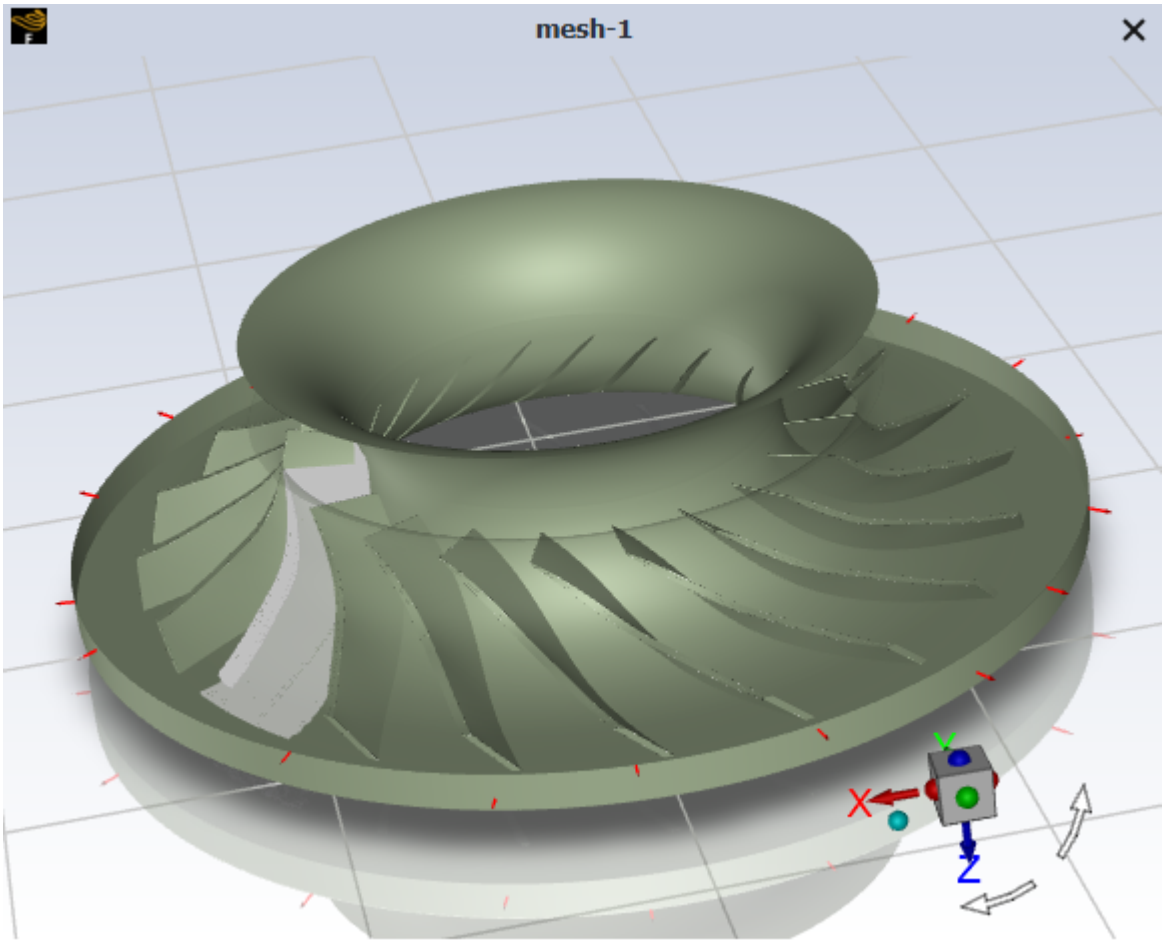
7.9.3.2. Viewing Profile Data

From within Figure 7.96: The Profiles Dialog Box (p. 1572) you can preview any profile or profile field as a cloud of points. Additionally, profiles containing node-connectivity data can also be previewed as a surface (profiles that cannot be previewed as a surface are identified in parenthesis in the **Profile** list).

With either type of profile preview, you can include a mesh or contour display by enabling **Graphics Object** in the **Overlay** group box and selecting the desired graphics object from the drop-down list. See Figure 7.97: Preview of Profile as a Surface (Gray) with Mesh Overlayed (Pastel Green) (p. 1573), Figure 7.98: Preview of Profile as a Point Cloud (Black) with Mesh Overlayed (Pastel Green) (p. 1574), and Figure 7.99: Preview of Profile Field as a Point Cloud (p. 1575) for examples.

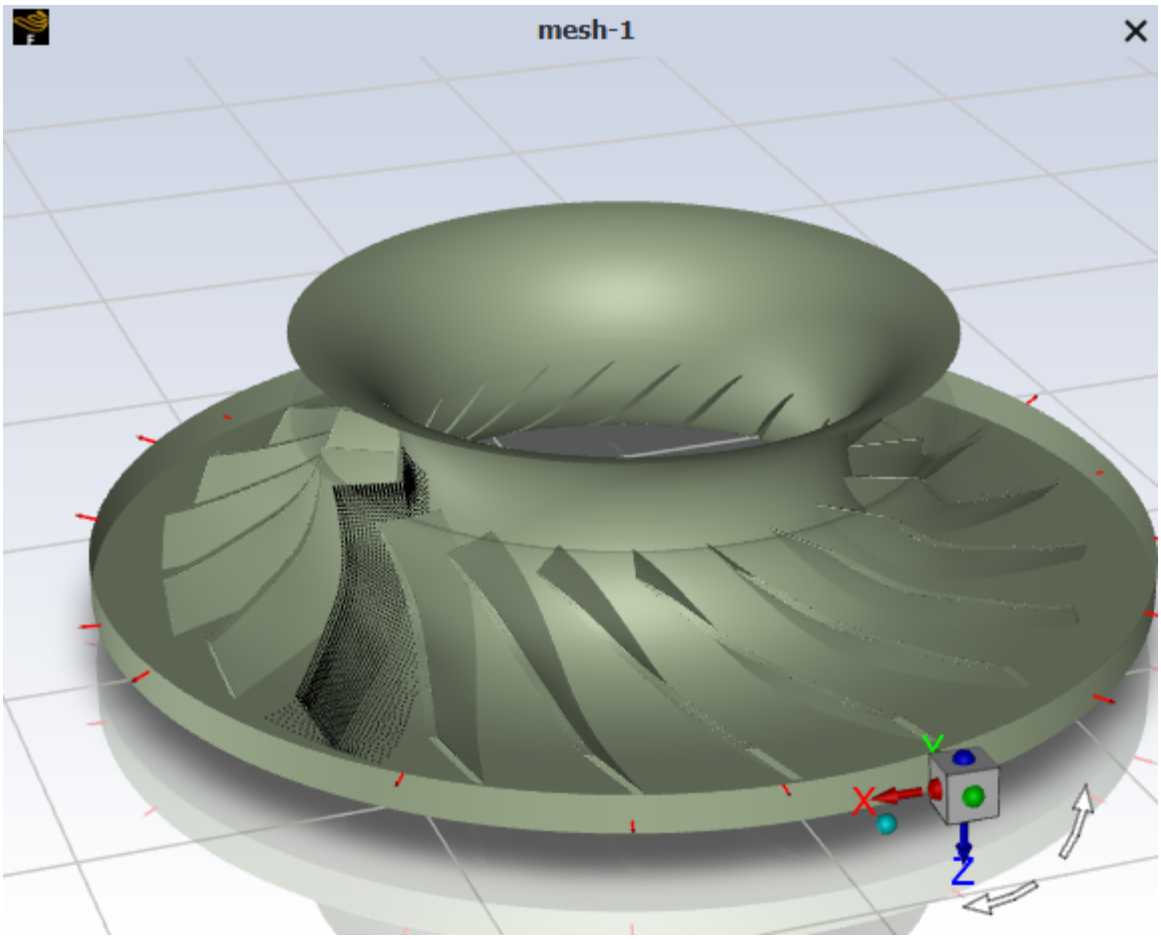
Controls for how the points appear for **Point Cloud** preview are available by enabling **Show Point Properties**.

Figure 7.97: Preview of Profile as a Surface (Gray) with Mesh Overlayed (Pastel Green)



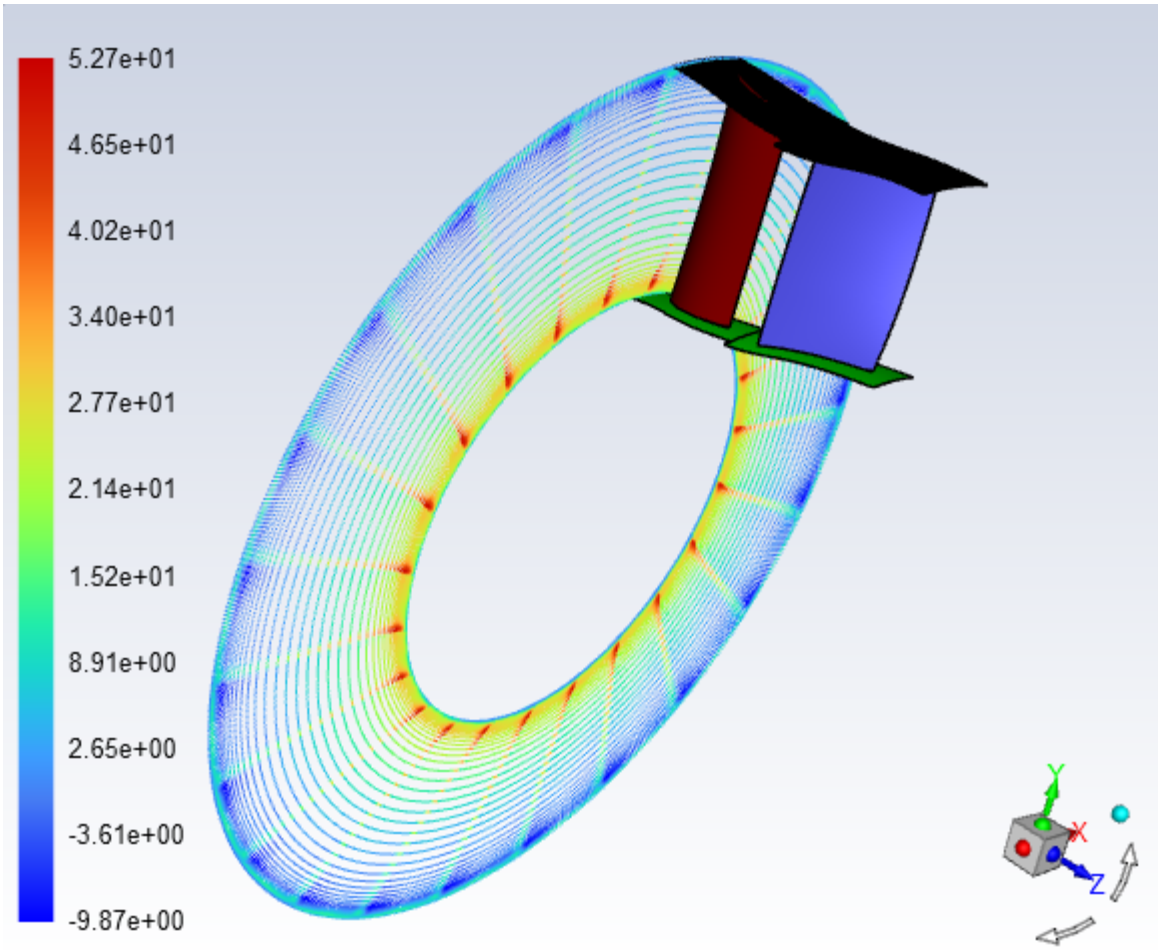
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.98: Preview of Profile as a Point Cloud (Black) with Mesh Overlayed (Pastel Green)



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.99: Preview of Profile Field as a Point Cloud



The **Plots** task page options allow you to generate XY plots of data related to profiles. You can plot the original data points from the profile file you have read into Ansys Fluent, or you can plot the values assigned to the cell faces on the boundary after the profile file has been interpolated. See XY Plots of Profiles (p. 4066) for the steps to generate these plots.

You have the additional option of viewing the parameters) using the **Plot** or the **Contours** options. Note that these display options do not allow you to plot the actual values of the cell faces (as is done with the **Interpolated Data** option), because they interpolate the values stored in the adjacent cells. To view the boundary condition parameters you must first read in the profile, save a boundary condition with a profile field selected as a parameter, and initialize the flow solution. Then you can view the surface data as follows:

- For 2D calculations, open the **Solution XY Plot** dialog box. Select the appropriate boundary zone in the **Surfaces** list, the variable of interest in the **Y Axis Function** drop-down list, and the desired **Plot Direction**. Ensure that the **Node Values** check button is turned on, and then click **Plot**. You should then see the profile plotted. If the data plotted does not agree with your specified profile, this means that there is an error in the profile file.

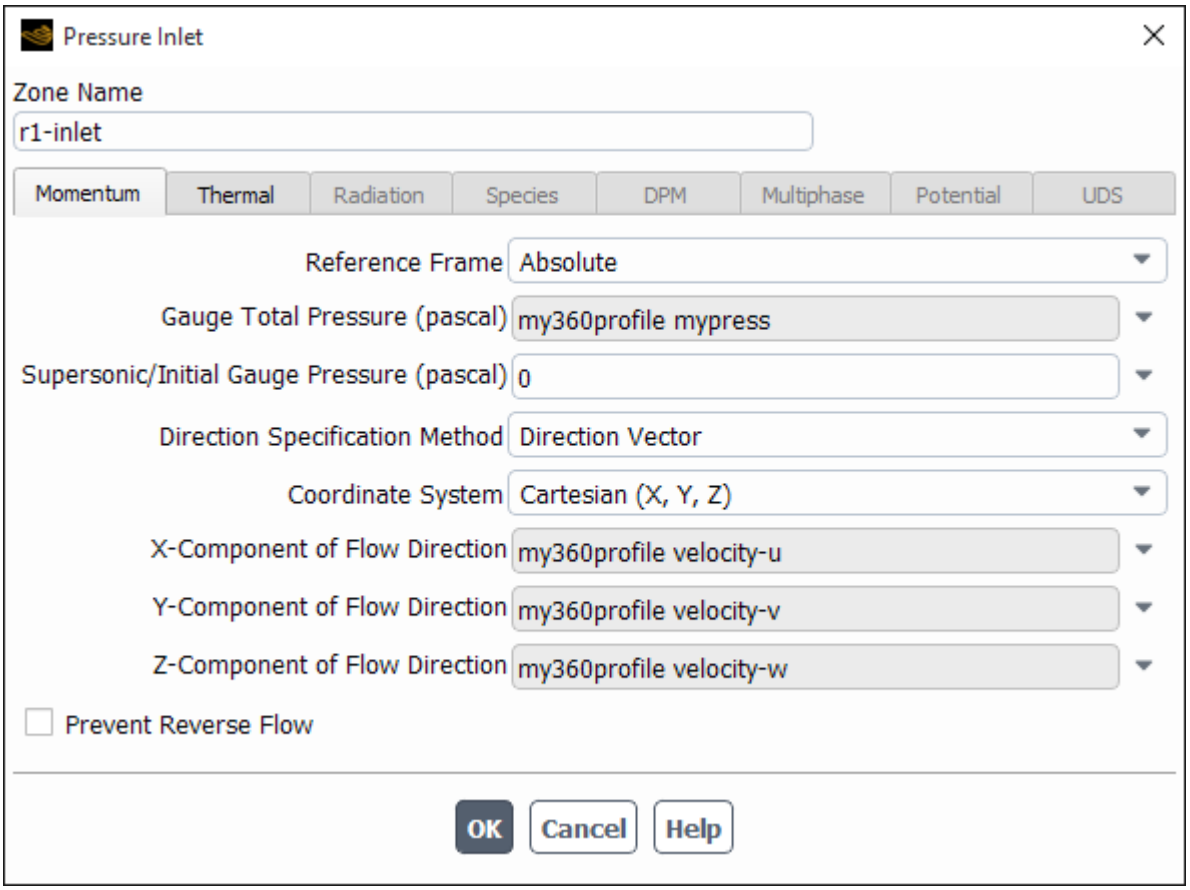
- For 3D calculations, use the **Contours** dialog box to display contours on the appropriate boundary zone surface. The **Node Values** check button must be turned on in order for you to view the profile data. If the data shown in the contour plot does not agree with your specified profile, this means that there is an error in the profile file.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.9.3.3. Example

In Figure 7.100: Example of Using Profiles as Boundary Conditions (p. 1576), profiles are used to specify the gauge total pressure and the x, y, z flow direction components at a pressure inlet boundary.

Figure 7.100: Example of Using Profiles as Boundary Conditions



7.9.3.4. Reorienting Profiles

For 3D cases only, Ansys Fluent allows you to change the orientation of an existing profile so that it can be used at a boundary positioned arbitrarily in space. This allows you, for example, to take experimental data for an inlet with one orientation and apply it to an inlet in your model that has a different spatial orientation. Note that Ansys Fluent assumes that the profile and the boundary are planar.

7.9.3.4.1. Steps for Changing the Profile Orientation

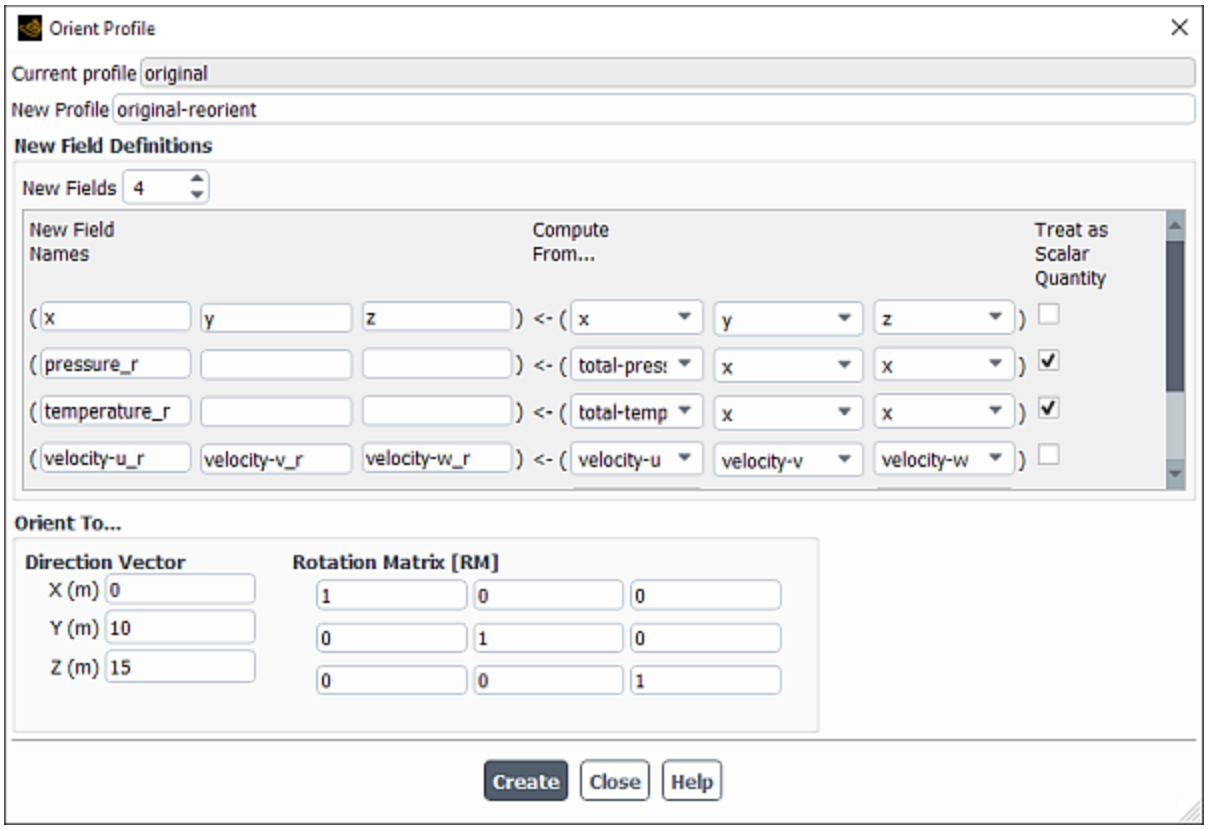
The procedure for orienting the profile data in the principal directions of a boundary is outlined below:

1. Define and read the profile as described in Using Profiles (p. 1570).

2. In the Profiles Dialog Box (p. 4981), select the profile in the **Profile** list, and then click the **Orient...** button. This will open the Orient Profile Dialog Box (p. 4986) (Figure 7.101: The Orient Profile Dialog Box (p. 1577)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.101: The Orient Profile Dialog Box



3. In the **Orient Profile** dialog box, enter the name of the new profile you want to create in the **New Profile** box.

4. Specify the number of fields you want to create using the up/down arrows next to the **New Fields** box. The number of new fields is equal to the number of vectors and scalars to be defined plus 1 (for the coordinates).

5. Define the coordinate field.

   a. Enter the names of the three coordinates (x, y, z) in the first row under **New Field Names**.

   Important:

   Ensure that the coordinates are named x, y, and z only. Do not use any other names or upper case letters in this field.

   b. Select the appropriate local coordinate fields for x, y, and z from the drop-down lists under **Compute From...**. (A selection of **0** indicates that the coordinate does not exist in the original profile; that is, the original profile was defined in 2D.)

6. Define the vector fields in the new profile.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. Enter the names of the 3 components in the directions of the coordinate axes of the boundary under **New Field Names**.

Important:

Do not use upper case letters in these fields.

b. Select the names of the 3 components of the vector in the local x, y, and z directions of the profile from the drop-down lists under **Compute From...**.

7. Define the scalar fields in the new profile.

a. Enter the name of the scalar in the first column under **New Field Names**.

Important:

Do not use upper case letters in these fields.

b. Click the button under **Treat as Scalar Quantity** in the same row.

c. Select the name of the scalar in the corresponding drop-down list under **Compute From...**.

8. Under **Orient To...**, specify the rotational matrix RM under the **Rotation Matrix [RM]**. The rotational matrix used here is based on Euler angles (γ, β, and α) that define an orthogonal system $x'y'z'$ as the result of the three successive rotations from the original system xyz. That is,

$$\begin{bmatrix} x' \\ y' \\ z' \end{bmatrix} = [RM] \begin{bmatrix} x \\ y \\ z \end{bmatrix} \tag{7.215}$$

$$RM = [C][B][A] \tag{7.216}$$

where C, B, and A are the successive rotations around the z, y, and x axes, respectively.

Rotation around the z axis:

$$C = \begin{bmatrix} \cos\gamma & -\sin\gamma & 0 \\ \sin\gamma & \cos\gamma & 0 \\ 0 & 0 & 1 \end{bmatrix} \tag{7.217}$$

Rotation around the y axis:

$$B = \begin{bmatrix} \cos\beta & 0 & \sin\beta \\ 0 & 1 & 0 \\ -\sin\beta & 0 & \cos\beta \end{bmatrix} \tag{7.218}$$

Rotation around the x axis:

$$A = \begin{bmatrix} 1 & 0 & 0 \\ 0 & \cos\alpha & -\sin\alpha \\ 0 & \sin\alpha & \cos\alpha \end{bmatrix} \tag{7.219}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9. Under **Orient To...**, specify the **Direction Vector**. The **Direction Vector** is the vector that translates a profile to the new position, and is defined between the centers of the profile fields.

   Important:

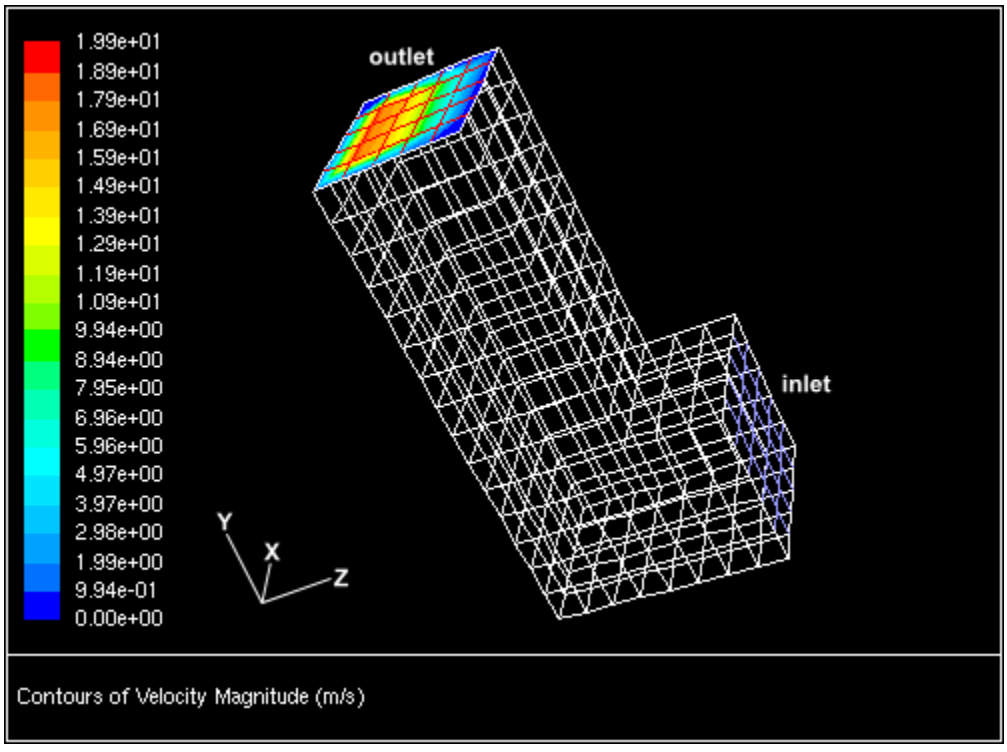
   Note that depending on your case, it may be necessary to perform only a rotation, only a translation, or a combination of a translation and a rotation.

10. Click the **Create** button in the **Orient Profile** dialog box, and your new profile will be created. Its name, which you entered in the **New Profile** box, will now appear in the **Profiles** dialog box and will be available for use at the desired boundary.

7.9.3.4.2. Profile Orienting Example

Consider the domain with a square inlet and outlet, shown in Figure 7.102: Scalar Profile at the Outlet (p. 1579). A scalar profile at the outlet is written out to a profile file. The purpose of this example is to impose this outlet profile on the inlet boundary via a 90 $^{\circ}$ rotation about the x axis. However, the rotation will locate the profile away from the inlet boundary. To align the profile to the inlet boundary, a translation via a directional vector must be performed.

Figure 7.102: Scalar Profile at the Outlet



The problem is shown schematically in Figure 7.103: Problem Specification (p. 1581). Φ_{out} is the scalar profile of the outlet. $\Phi_{out}^{'}$ is the image of the Φ_{out} rotated 90 $^{\circ}$ around the x axis. In this example, since $\gamma=\beta=0$, then $C=B=I$, where I is the identity matrix, and the rotation matrix is

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$RM=[C][B][A]=\begin{bmatrix}1 & 0 & 0\\ 0 & \cos 90^{\circ} & -\sin 90^{\circ}\\ 0 & \sin 90^{\circ} & \cos 90^{\circ}\end{bmatrix}=\begin{bmatrix}1 & 0 & 0\\ 0 & 0 & -1\\ 0 & 1 & 0\end{bmatrix} \tag{7.220}$$

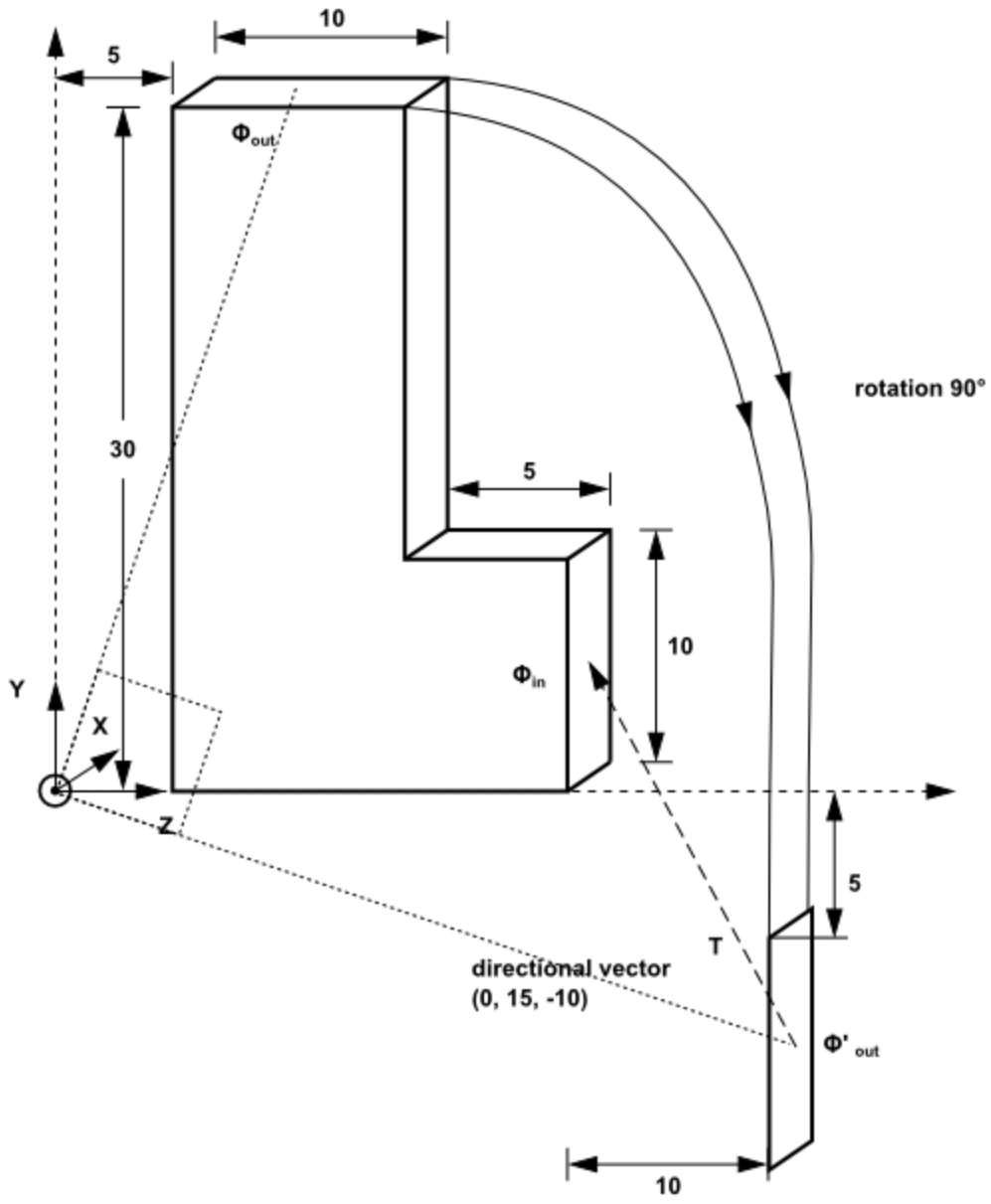
To overlay the outlet profile on the inlet boundary, a translation will be performed.

To overlay the outlet profile on the inlet boundary, a translation will be performed. The directional vector is the vector that translates Φ'_{out} to Φ_{in}. In this example, the directional vector is $(0,15,-10)^{T}$. The appropriate inputs for the **Orient Profile** dialog box are shown in Figure 7.101: The Orient Profile Dialog Box (p. 1577).

Note that if the profile being imposed on the inlet boundary was due to a rotation of -90 ° about the x axis, then the rotational matrix RM must be found for $\gamma=\beta=0$ and $\alpha=-90^{\circ}$, and a new directional vector must be found to align the profile to the boundary.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.103: Problem Specification



7.9.3.5. Replicating Profiles

In many turbomachinery cases, a profile can be obtained from an initial small-scale simulation (often one passage but sometimes more) and then applied to a larger, often full 360 degree, simulation. Replicating a profile enables you to copy the initial profile periodically for use in a larger simulation.

The following topics are discussed:

7.9.3.5.1. Steps for Replicating a Profile

7.9.3.5.2. Complex Mode Shape Profile Replication

7.9.3.5.1. Steps for Replicating a Profile

The procedure for replicating the profile data periodically is outlined below:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

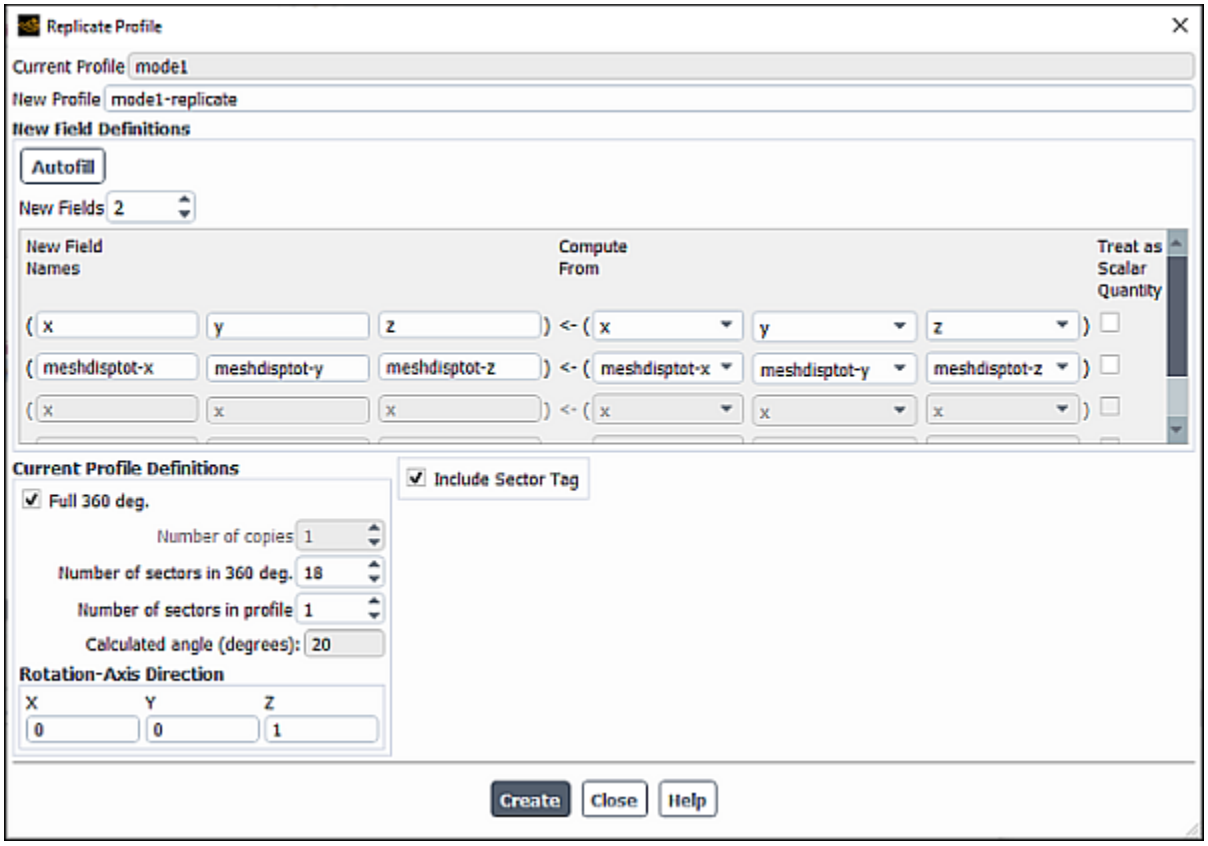
1. Define and read the profile as described in Using Profiles (p. 1570).

2. Open the **Profiles** dialog box.

   Physics → **Zones** → **Profiles...**

3. Select the profile to replicate and click **Replicate...** to open the **Replicate Profile** dialog box (Figure 7.104: The Replicate Profile Dialog Box (p. 1582)).

Figure 7.104: The Replicate Profile Dialog Box



   a. Enter a name in the **New Profile** field (for the expanded profile name).

   b. If the profile contains mesh displacement field variables (variables containing the `mesh-disp`, `meshdisptot` keyword), the **Autofill** button becomes available. Upon clicking **Autofill**, Fluent will automatically fill the dialog box based on the current profile. Otherwise, you can follow the steps below to specify profile replication manually.

   c. Specify the number of fields you want to create using the up/down arrows next to the **New Fields** box. The number of new fields is equal to the number of vectors and scalars to be defined plus 1 (including the coordinates).

   d. Define the coordinate field.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

i. Enter the names of the three coordinates (x, y, z) in the first row under **New Field Names**.

> **Important:**
>
> Ensure that the coordinates are named x, y, and z only. Do not use any other names or upper case letters in this field.

ii. Select the appropriate local coordinate fields for x, y, and z from the drop-down lists under **Compute From...**. (A selection of **0** indicates that the coordinate does not exist in the original profile; that is, the original profile was defined in 2D.)

e. Define the vector fields in the new profile.

i. Enter the names of the 3 components in the directions of the coordinate axes under **New Field Names**.

> **Important:**
>
> Do not use upper case letters in these fields.

ii. Select the names of the 3 components of the vector in the local coordinates of the profile from the drop-down lists under **Compute From...**.

f. Define the scalar fields in the new profile.

i. Enter the name of the scalar in the first column under **New Field Names**.

> **Important:**
>
> Do not use upper case letters in these fields.

ii. Select **Treat as Scalar Quantity** in the same row.

iii. Select the name of the scalar in the corresponding drop-down list under **Compute From...**.

4. Specify the replication settings in the **Current Profile Definitions** group box.

a. If you are creating a full 360 degree profile, select **Full 360 deg.**. Fluent automatically determines the number of copies of the original profile that are needed to make the full 360 degree profile based on the other information you specify.

b. If you did not select **Full 360 deg.**, you must specify the **Number of copies** to manually determine how many copies of the original profile are created.

c. The **Number of sectors in 360 deg.** is the number of periodic sections, based on the original profile, that are present in 360 degrees.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

d. The **Number of sectors in profile** is the number of periodic sections in the original profile.

e. The **Calculated angle** is for display only and is the angle occupied by the original profile. When using the **Full 360 deg.** option, you can determine the number of copies that are created by dividing 360 by the **Calculated angle**.

f. Specify the **Rotation-Axis Direction**. This is the **X**, **Y**, and **Z** components of the axis about which the selected profile is replicated.

5. Optionally, you can select **Include Sector Tag** and Fluent will append a sector tag column onto the replicated profile to indicate the original and replicated profiles. The numbering follows the right hand rule applied to the **Rotation-Axis Direction** that you have specified.

6. Click the **Create** button in the **Replicate Profile** dialog box, and your new profile will be created. Its name, which you entered in the **New Profile** box, will now appear in the **Profiles** dialog box and will be available for use at the desired boundary.

7.9.3.5.2. Complex Mode Shape Profile Replication

Complex mode shapes are composed of a real and an imaginary vector pair that cannot be replicated (expanded) separately in the same way as is done for other profiles with real-only vectors.

The process of complex mode shape replication is performed as follows:

1. The real and imaginary components are phase-shifted:

$$\begin{aligned} |\vec{X}| &= \sqrt{R^2 + I^2} \\ \phi &= 2k\pi \frac{ND}{N_{360}} + \arctan\left(\frac{I}{R}\right) \\ R^* &= |\vec{X}| \cos(\phi) \\ I^* &= |\vec{X}| \sin(\phi) \end{aligned} \tag{7.221}$$

2. The shifted components are then rotated to the passage k:

$$\begin{aligned} R^k &= R^* \cdot [M] \\ I^k &= I^* \cdot [M] \end{aligned} \tag{7.222}$$

Where the rotation matrix is $[M]$ and computed with a given rotation axis (u, v, w) as:

Note:

The rotation matrix is computed in the same way as for non-complex mode shape replication.

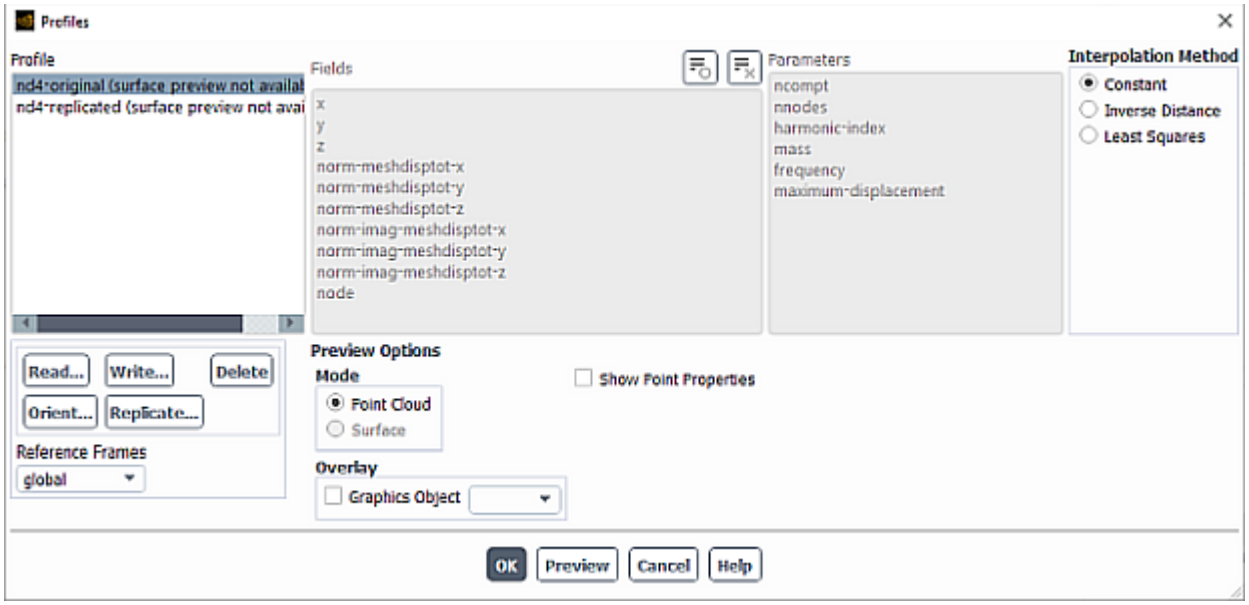
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$
\begin{aligned}
&\varphi = 2k\pi \frac{N_{component}}{N_{360}} \\
&M[0][0] = \cos(\varphi) + u*u*(1-\cos(\varphi)) \\
&M[0][1] = u*v*(1-\cos(\varphi)) - w*\sin(\varphi) \\
&M[0][2] = u*w*(1-\cos(\varphi)) - v*\sin(\varphi) \\
&M[1][0] = v*u*(1-\cos(\varphi)) - w*\sin(\varphi) \\
&M[1][1] = \cos(\varphi) + v*v*(1-\cos(\varphi)) \\
&M[1][2] = v*w*(1-\cos(\varphi)) - u*\sin(\varphi) \\
&M[2][0] = w*u*(1-\cos(\varphi)) - v*\sin(\varphi) \\
&M[2][1] = w*v*(1-\cos(\varphi)) - u*\sin(\varphi) \\
&M[2][2] = \cos(\varphi) + w*w*(1-\cos(\varphi))
\end{aligned}
\tag{7.223}
$$

7.9.3.5.2.1. Steps for Replicating a Complex Mode Shape Profile

After opening the **Replicate Profile** dialog box, as described in Replicating Profiles (p. 1581), Fluent detects the presence of a parameter called `harmonic index` as shown in Figure 7.105: The Profiles Dialog Box with Complex Mode Shape Profile (p. 1585). This parameter must be correctly defined for Fluent to accurately expand the complex mode shape. Note that the harmonic index parameter is automatically included in profiles generated in Ansys Mechanical for cases with Cyclic Symmetry. If the profile was created with a source outside of the Ansys environment, you are responsible for creating the parameter and setting it to the correct value.

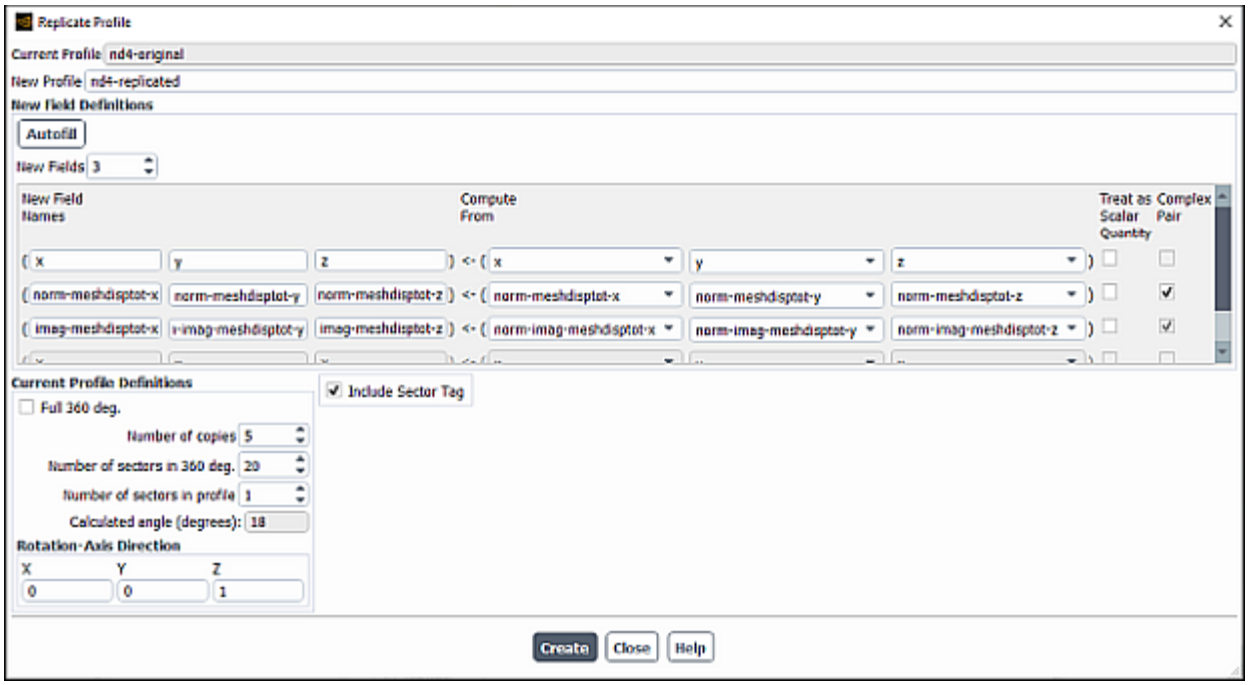
Figure 7.105: The Profiles Dialog Box with Complex Mode Shape Profile



With the presence of the `harmonic index` parameter, the **Replicate Profile** dialog box has an additional column of checkboxes named **Complex Pair**. Selecting the **Complex Pair** checkbox indicates the rows corresponding to the real and imaginary vector pairs of the complex mode shape, as shown in Figure 7.106: The Replicate Profile Dialog Box with Complex Pair (p. 1586).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.106: The Replicate Profile Dialog Box with Complex Pair



The workflow for expanding a complex mode shape is as follows:

1. Load a profile that contains a parameter named `harmonic index`. For details on loading a profile, see Using Profiles (p. 1570).

2. If the profile contains mesh displacement field variables (variables containing the `mesh-disp`, `meshdisptot` keyword), the **Autofill** button becomes available. Upon clicking **Autofill**, Fluent will automatically fill the dialog box based on the current profile. Otherwise, you can follow the steps below to specify profile replication manually.

3. Enter the real vector component of the complex mode shape into any row and select **Complex Pair**. This indicates to Fluent that it is the first part of the complex pair, and that the imaginary vector will immediately follow on the next row.

   The real and complex pairs can be defined in any new field row. However, the real vector component must be defined first, followed by the imaginary vector on the immediate row below the real vector.

4. When the **Complex Pair** checkbox is selected, the immediate following row will also be set in anticipation of the imaginary vector components being defined. Enter the imaginary vector components in the second row.

   Note:

   When the **Complex Pair** checkboxes are set, all other checkboxes in that column will be greyed out to prevent another pair from being defined. This is a current limitation that only one complex pair can be defined.

5. Complete the rest of the dialog box as described in Steps for Replicating a Profile (p. 1581)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.10. Coupling Boundary Conditions with GT-POWER

GT-POWER users can define time-dependent boundary conditions in Ansys Fluent based on information from GT-POWER. During the Ansys Fluent simulation, Ansys Fluent and GT-POWER are coupled together and information about the boundary conditions at each time step is transferred between them.

7.10.1. Requirements and Restrictions

7.10.2. User Inputs

7.10.3. Torque-Speed Coupling with GT-POWER

7.10.1. Requirements and Restrictions

Note the following requirements and restrictions for the GT-POWER coupling:

- The flow must be unsteady.
- The compressible ideal gas law must be used for density.
- Each boundary zone for which you plan to define conditions using GT-POWER must be a boundary of one of the following types:
  - velocity inlet
  - mass-flow inlet
  - pressure inlet
  - pressure outlet
  - wall (for details, see Torque-Speed Coupling with GT-POWER (p. 1591))

  Also, a maximum of 20 boundary zones can be coupled to GT-POWER.
- Fluent checks the area of the 1D coupled boundary as calculated by Fluent and GT-POWER to ensure the calculated areas differ by no more than 5%. When the difference is more than 5%, Fluent stops with an error message and displays the two areas.
- If a mass-flow inlet or pressure inlet is coupled to GT-POWER, you must select **Normal to Boundary** as the **Direction Specification Method** in the **Mass-Flow Inlet** or **Pressure Inlet** dialog box. For a velocity inlet, you must select **Magnitude, Normal to Boundary** as the **Velocity Specification Method** in the **Velocity Inlet** dialog box.
- The mass flow specification method in the **Mass-Flow Inlet** boundary has to always be **Mass Flux** and not **Mass Flow Rate** when coupling with GTPower.
- Boundary conditions for the following variables can be obtained from GT-POWER:
  - velocity
  - temperature
  - pressure

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - density
  - species mass fractions
  - k and ε (Note that it is recommended that you define these conditions in Ansys Fluent yourself, rather than using the data provided by GT-POWER, since the GT-POWER values are based on a 1D model.)

- Make sure that the material properties you set in Ansys Fluent are the same as those used in GT-POWER, so that the boundary conditions will be valid for your coupled simulation.
- If your model includes species, make sure that the name of each species in GT-POWER corresponds to the **Chemical Formula** for that species material in the **Materials** dialog box. Also, recall that Ansys Fluent can handle a maximum of 50 species.
- You can install the GT-POWER libraries in a directory other than the default location. If the GT-POWER libraries are loaded into a non-default location, you need to set the following environment variables:
  - `Fluent_GTIHOME` — the GTI installation directory where GT-POWER is installed
  - `Fluent_GTIVERSION` — the current version of the GTI installation

  Important:

  GTI is not backwards compatible.

7.10.2. User Inputs

The procedure for setting up the GT-POWER coupling in Ansys Fluent is presented below.

1. Read in the mesh file and define the models, materials, and boundary zone types (but *not* the actual boundary conditions), noting the requirements and restrictions listed in Requirements and Restrictions (p. 1587).
2. Specify the location of the GT-POWER data and have Ansys Fluent use them to generate user-defined functions for the relevant boundary conditions (using the 1D Simulation Library Dialog Box (p. 5385), shown in Figure 7.107: The 1D Simulation Library Dialog Box (p. 1589)).

   User Defined → **Model Specific** → **1D Coupling...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.107: The 1D Simulation Library Dialog Box

a. Select **GTpower** in the **1D Library** drop-down list.

b. Specify the name of the GT-POWER input file in the **1D Input File Name** field.

c. Click the **Start** button.

When you click **Start**, GT-POWER will start up and Ansys Fluent user-defined functions for each boundary in the input file will be generated.

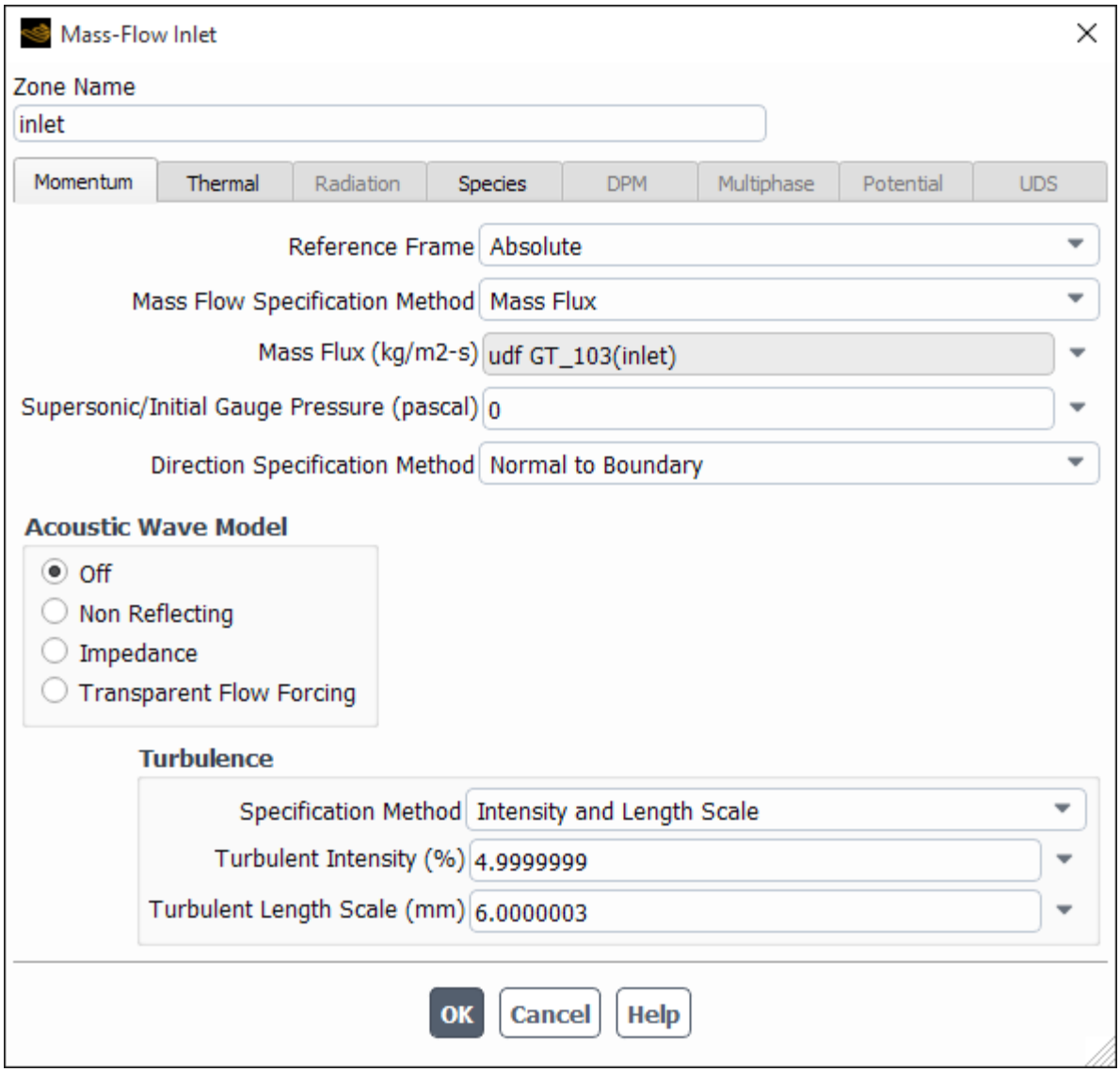
3. Set boundary conditions for all zones. For flow boundaries for which you are using GT-POWER data, select the appropriate UDFs as the conditions.

Important:

Note that you must select the same UDF for all conditions at a particular boundary zone (as shown, for example, in Figure 7.108: Using GT-POWER Data for Boundary Conditions (p. 1590)); this UDF contains all of the conditions at that boundary.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7.108: Using GT-POWER Data for Boundary Conditions



4. If you plan to continue the simulation at a later time, starting from the final data file of the current simulation, specify how often you want to have the case and data files saved automatically.

   Solution → Calculation Activities (Autosave Case/Data) → Edit...

   To use a GT-POWER restart file to restart an Ansys Fluent calculation, you must edit the GT-POWER input data file. See the GT-POWER User's Guide for instructions.

5. Continue the problem setup and calculate a solution in the usual manner.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7.10.3. Torque-Speed Coupling with GT-POWER

Torque-speed coupling with GT-POWER is a unique case in which Fluent receives shaft speed from GT-POWER and returns torque. The procedure differs from the typical coupling between GT-Power and Fluent, and is as follows:

Important:

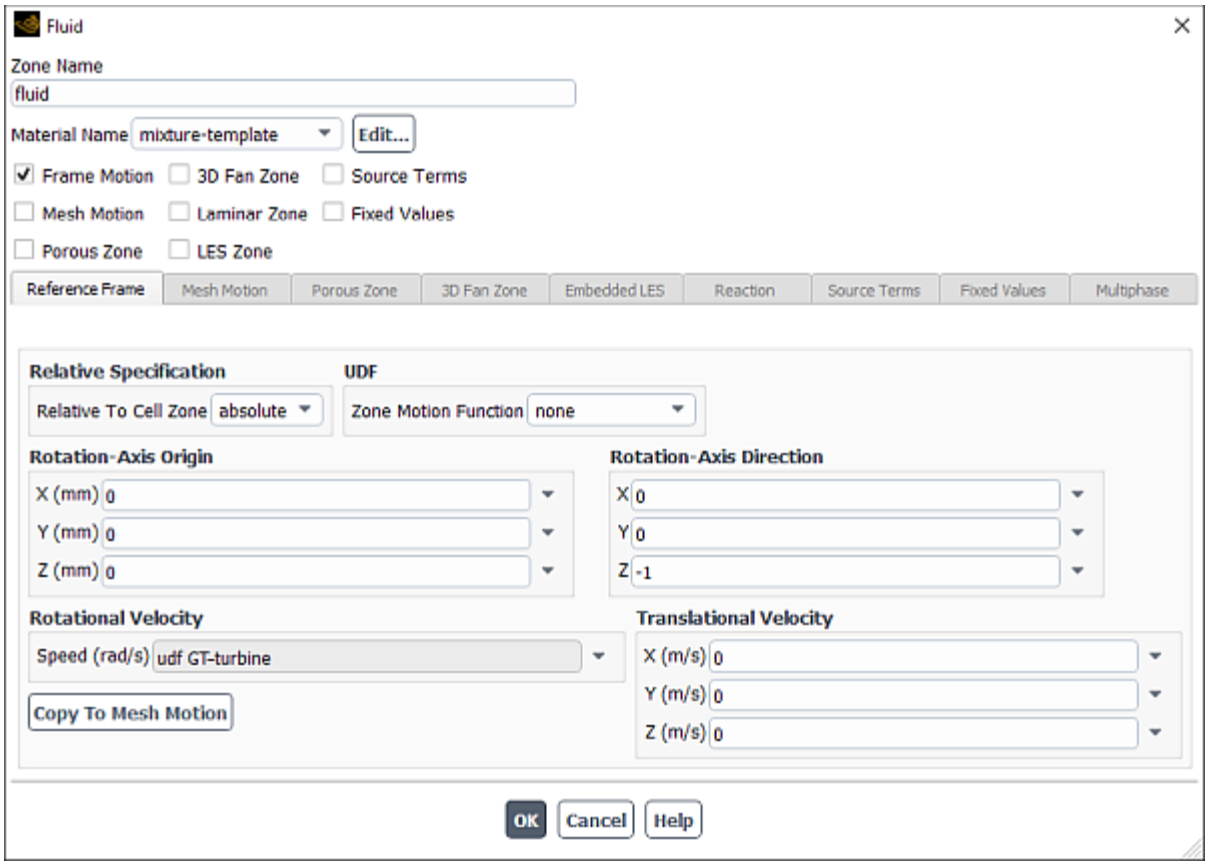
You must be using GT-POWER v2017.2 or greater to perform torque-speed coupling with Ansys Fluent.

---

1. Complete steps 1 and 2 in the above section on User Inputs (p. 1588). The `.dat.h5` file should have at least one turbo-machine exposed to complete the coupling.

2. Enable **Frame Motion** in the cell zone that is receiving shaft speed from GT-Power. You must also select the appropriate UDF for **Rotational Velocity**. The UDF must correspond to the turbo-machine exposed from GT-POWER ("turbine", in the example shown below).

   Figure 7.109: Cell Zone Conditions for Torque-Speed Coupling with GT-POWER



3. Specify a **Wall** boundary on which the torque is calculated and transferred to GT-POWER. The following requirements apply:

   - The wall boundary must be attached to the rotating cell zone from step 2, to receive the shaft speed from GT-POWER.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The name of the wall boundary must be the same as the turbo-machine it is coupled with ("turbine", in the case presented above).

During the solution, shaft speed is obtained from GT-POWER at every time-step, with Fluent calculating and returning torque within each time-step as well.

7.11. Coupling Boundary Conditions with WAVE

WAVE users can define time-dependent boundary conditions in Ansys Fluent based on information from WAVE. During the Ansys Fluent simulation, Ansys Fluent and WAVE are coupled together and information about the boundary conditions at each time step is transferred between them.

7.11.1. Requirements and Restrictions

7.11.2. User Inputs

7.11.1. Requirements and Restrictions

Note the following requirements and restrictions for the WAVE coupling:

- WAVE must be installed and licensed.
- There are always five species that must be modeled in Ansys Fluent just as they are defined in WAVE (F1, F2, F3, F4, and F5). It is recommended that realistic material properties be assigned to each of the five species.
- The flow must be unsteady.
- The compressible ideal gas law must be used for density.
- Each boundary zone for which you plan to define conditions using WAVE must be a flow boundary of one of the following types:
  - velocity inlet
  - mass-flow inlet
  - pressure inlet
  - pressure outlet

  Also, a maximum of 20 boundary zones can be coupled to WAVE.
- If a mass-flow inlet or pressure inlet is coupled to WAVE, you must select **Normal to Boundary** as the **Direction Specification Method** in the **Mass-Flow Inlet** or **Pressure Inlet** Dialog Box. For a velocity inlet, you must select **Magnitude, Normal to Boundary** as the **Velocity Specification Method** in the **Velocity Inlet** Dialog Box.
- Boundary conditions for the following variables can be obtained from WAVE:
  - velocity
  - temperature

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - pressure
  - density
  - species mass fractions
  - k and ε (Note that you are required to define these conditions in Ansys Fluent yourself, since WAVE does not calculate them.)

- Make sure that the material properties you set in Ansys Fluent are the same as those used in WAVE, so that the boundary conditions will be valid for your coupled simulation.
- If your model includes species, make sure that the name of each species in WAVE corresponds to the **Chemical Formula** for that species material in the **Create/Edit Materials** dialog box. Also, recall that Ansys Fluent can handle a maximum of 50 species.

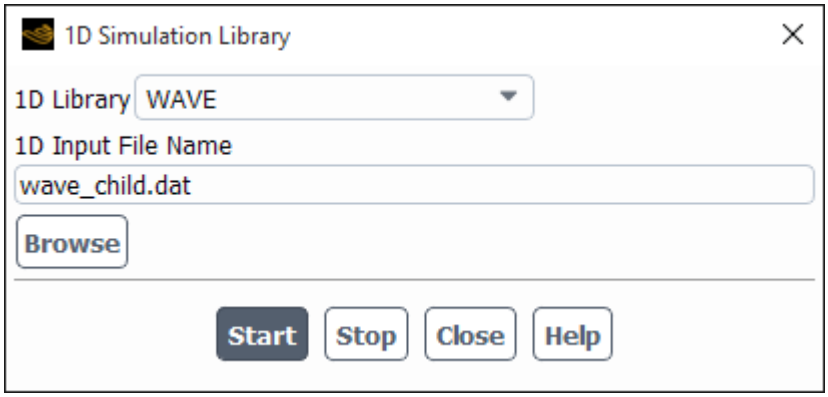
7.11.2. User Inputs

The procedure for setting up the WAVE coupling in Ansys Fluent is presented below.

1. Read in the mesh file and define the models, materials, and boundary zone types.
2. Specify the location of the WAVE data and have Ansys Fluent use them to generate user-defined functions for the relevant boundary conditions (using the 1D Simulation Library Dialog Box (p. 5385), shown in Figure 7.110: The 1D Simulation Library Dialog Box with WAVE Selected (p. 1593)).

   User Defined → Model Specific → 1D Coupling...

   Figure 7.110: The 1D Simulation Library Dialog Box with WAVE Selected

   a. Select **WAVE** in the **1D Library** drop-down list.

   b. Specify the name of the WAVE input file in the **1D Input File Name** field.

   c. Click the **Start** button.

   When you click **Start**, WAVE will start up and Ansys Fluent user-defined functions for each boundary in the input file will be generated.

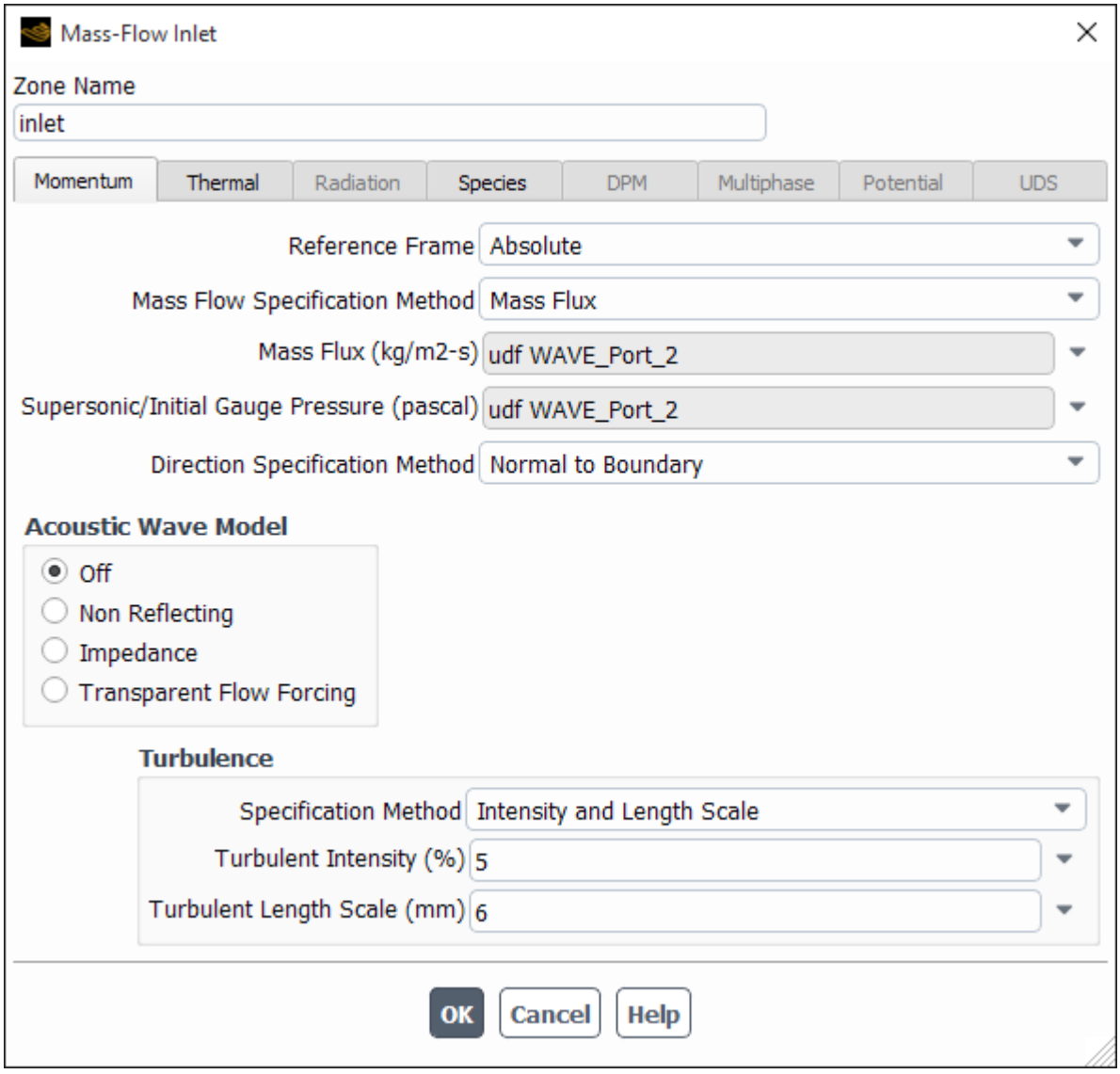
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Set boundary conditions for all zones. For flow boundaries for which you are using WAVE data, select the appropriate UDFs as the conditions.

   Important:

   Note that you must select the same UDF for all conditions at a particular boundary zone (as shown, for example, in Figure 7.111: Using WAVE Data for Boundary Conditions (p. 1594)); this UDF contains all of the conditions at that boundary.

Figure 7.111: Using WAVE Data for Boundary Conditions



4. If you plan to continue the simulation at a later time, restarting from the final data file of the current simulation, you need to instruct both Ansys Fluent and WAVE how often that you want to automatically save your data. You should instruct Ansys Fluent to automatically save case and data files at specified intervals using the autosave feature.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution → Calculation Activities → Autosave On

In addition, you should instruct WAVE as to how often it should generate its own restart files. See the WAVE User's Guide for instructions on this feature.

Important:

To use the restart feature, the time interval for writing data files must be set to the same value in both Ansys Fluent and WAVE. For example, if Ansys Fluent has set the autosave feature to 100, then WAVE must also set the restart file write frequency to 100 as well.

5. Continue the problem setup and calculate a solution in the usual manner.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 8: Physical Properties

This chapter describes how to define materials, the physical equations used to compute material properties, and the methods you can use for each property input. Each property is described in detail in the following sections. If you are using one of the general multiphase models (VOF, mixture, or Eulerian), see Defining the Phases (p. 2932) for information about how to define the individual phases and their material properties.

8.1. Defining Materials

8.2. Defining Properties Using Temperature-Dependent Functions

8.3. Density

8.4. Viscosity

8.5. Thermal Conductivity

8.6. User-Defined Scalar (UDS) Diffusivity

8.7. Specific Heat Capacity

8.8. Radiation Properties

8.9. Mass Diffusion Coefficients

8.10. Standard State Enthalpies

8.11. Standard State Entropies

8.12. Unburnt Thermal Diffusivity

8.13. Kinetic Theory Parameters

8.14. Operating Pressure

8.15. Using a Reference Pressure to Adjust the Gauge Pressure Field

8.16. Real Gas Models

8.1. Defining Materials

An important step in the setup of the model is to define the materials and their physical properties. Material properties are defined using either the Materials Task Page (p. 4883) or the **Materials** tree branch, where you can enter values for the properties that are relevant to the problem scope you have defined in the Models Task Page (p. 4696). These properties may include the following:

- density and/or molecular weights
- viscosity
- heat capacity
- thermal conductivity
- UDS diffusion coefficients

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- mass diffusion coefficients
- standard state enthalpies
- kinetic theory parameters

Properties may be temperature-dependent and/or composition-dependent, with temperature dependence based on a polynomial, piecewise-linear, or piecewise-polynomial function and individual component properties either defined by you or computed via kinetic theory.

The Materials Task Page (p. 4883) will show the properties that need to be defined for the active physical models. If any property you define requires the energy equation to be solved (for example, ideal gas law for density, temperature-dependent profile for viscosity), Ansys Fluent will automatically enable the energy equation. Then you have to define the thermal boundary conditions and other parameters yourself.

For additional information, see the following sections:

8.1.1. Physical Properties for Solid Materials
8.1.2. Material Types and Databases
8.1.3. Using the Create/Edit Materials Dialog Box
8.1.4. Using a User-Defined Materials Database

8.1.1. Physical Properties for Solid Materials

For solid materials, only density, thermal conductivity, and heat capacity are defined. If you are modeling semi-transparent media, then radiation properties are also defined. You can specify a constant value, a temperature-dependent function, or a user-defined function for thermal conductivity; a constant value or temperature-dependent function for heat capacity; and a constant value for density.

If you are using the pressure-based solver, density and heat capacity for a solid material are not required unless you are modeling transient flow or moving solid zones. Heat capacity will appear in the list of solid properties for steady flows as well. The value will be used just for postprocessing enthalpy; not in the calculation.

8.1.2. Material Types and Databases

In Ansys Fluent, you can define six types of materials: fluids, solids, mixtures, combusting-particles, droplet-particles, and inert-particles. Physical properties of fluids and solids are associated with named material; these materials are then assigned as boundary conditions for zones.

When you model species transport, define a mixture material, consisting of the various species involved in the problem. Properties will be defined for the mixture, as well as for the constituent species, which are fluid materials. The mixture material concept is discussed in detail in Mixture Materials (p. 2357). Combusting-particles, droplet-particles, and inert-particles are available for the discrete-phase model, as described in The Concept of Discrete-Phase Materials (p. 2814).

Ansys Fluent provides the following built-in databases:

- A built-in global database of approximately 675 predefined materials along with their properties and default values for each property. To define a material in the problem setup, you can copy

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

materials from this global (site-wide) database and use the default properties or define new materials by editing their properties. The Ansys Fluent materials database is located in the following file:

path `/ansys_inc/v242/fluent/fluent24.2.0/cortex/lib/propdb.scm`

where *path* is the directory in which you installed Ansys Fluent.

- A built-in database of more than 700 predefined solid materials with properties from the Ansys GRANTA Materials Data for Simulation (GRANTA MDS). A separate license is required for copying materials from the GRANTA MDS Database into the problem setup. Materials from the GRANTA MDS Database can be used in the same way as materials from the Ansys Fluent materials database.
- A GRANTA MDS license is held for 30 minutes or until the program is closed, whichever is the sooner. Subsequent accesses of MDS data resets the timer. When no MDS license is available, only the description of the selected material will be displayed, but not its properties. The copy operation will abort with an information message.

In addition to using the Ansys Fluent material database, you can also create your own database and materials, and use it to define the materials in your problem setup. See Using a User-Defined Materials Database (p. 1607) for information about creating and using user-defined custom material databases.

Important:

- All the materials in your local materials list will be saved in the case file (when you write one). The materials specified by you will be available to you if you read this case file into a new solver session.
- A material may co-exist as a pure substance fluid as well as a species in one or multiple mixture materials. The pure fluid and the species for each mixture are separate objects and may have independent property settings. *Therefore, editing the properties of the pure fluid will not modify the same properties for that fluid included in the mixture. If you intend to modify some material properties, you can make a copy of the pure fluid, and rename it. Using the copy in the mixture makes it easier to understand which version of the fluid is being modified.*

8.1.3. Using the Create/Edit Materials Dialog Box

The Create/Edit Materials Dialog Box (p. 4885) allows you to define the materials and their properties in your problem setup using either the **Fluent Database** or a **User-Defined Database**. It enables you to copy materials from a database, create new materials, and modify material properties.

These generic functions are described in this section. The inputs for temperature-dependent properties are explained in Defining Properties Using Temperature-Dependent Functions (p. 1616). The specific inputs for each material property are discussed in the remaining sections of this chapter.

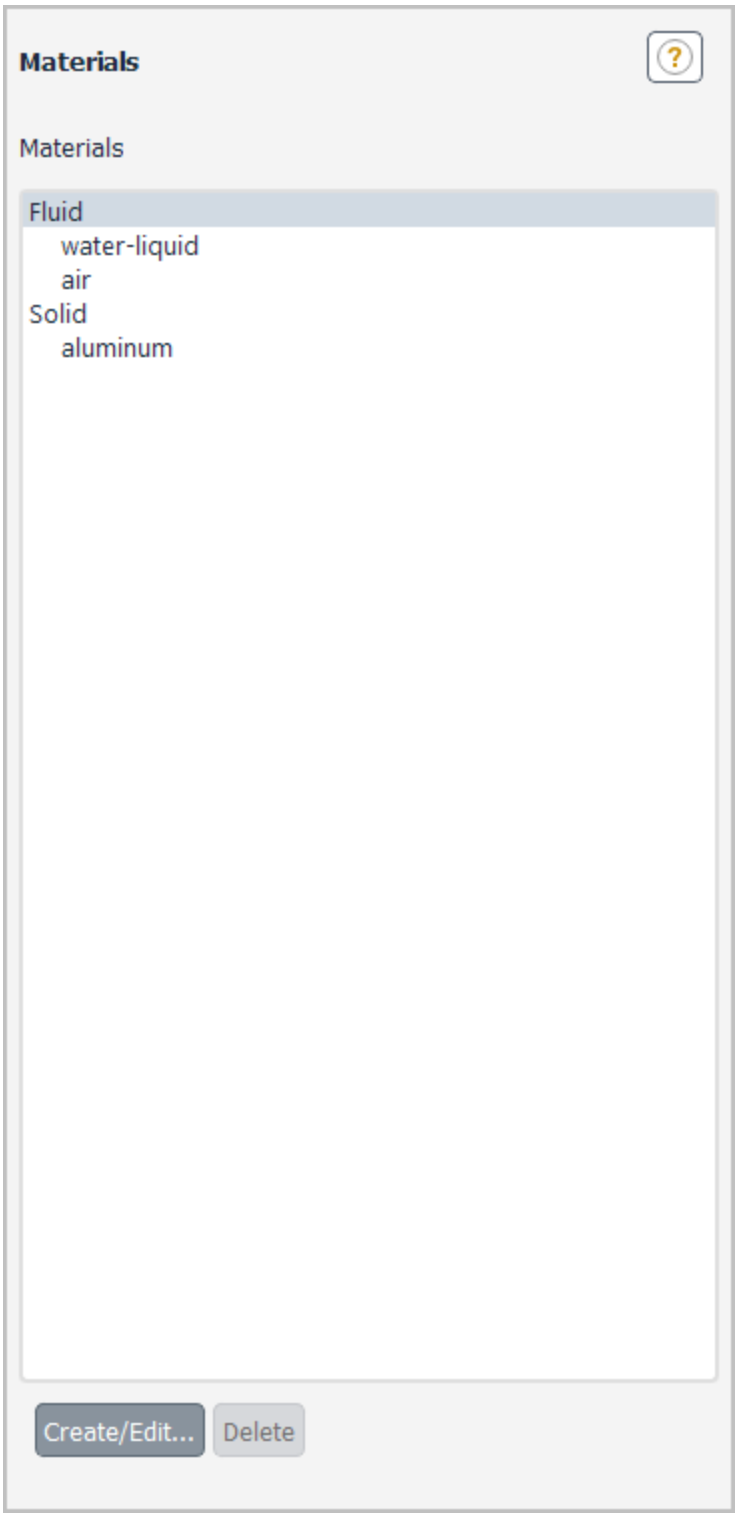
The Create/Edit Materials Dialog Box (p. 4885) can be accessed in one of two ways:

- Through the **Materials** task page.

Setup → **Materials**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 8.1: The Materials Task Page



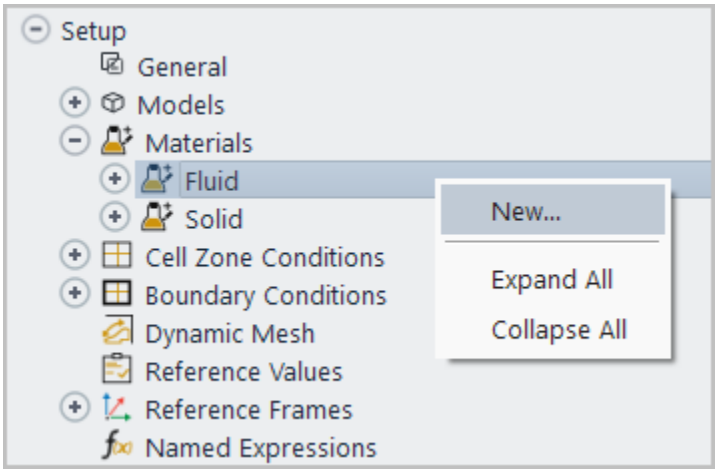
- Through the material types (**Fluid**, **Solid**, and so on) and defined in the case materials under **Materials** in the tree.

  For example, if you want to create a new fluid material you can access the **Create/Edit Materials** dialog box as follows:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setup → Materials → Fluid → New...

Figure 8.2: The Materials Branch in the Outline View



By default, your local materials list will include a single fluid material (air) and a single solid material (aluminum). If the fluid involved in your problem is air, you can use the default properties for air or modify the properties. If the fluid in your problem is water, you can either copy water from the Ansys Fluent database or create a new "water" material from scratch. If you copy water from the database, you can still make modifications to the properties of your local copy of water. See the following sections for detailed information on how to change material properties.

Mixture materials will not exist in your local list unless you have enabled species transport (see Modeling Species Transport and Finite-Rate Chemistry (p. 2355)). Similarly, inert, droplet, and combusting particle materials will not be available unless you have created a discrete phase injection of these particle types (see Modeling Discrete Phase (p. 2671)). When a mixture material is copied from the database, all of its constituent fluid materials (species) will automatically be copied over as well.

8.1.3.1. Modifying Properties of an Existing Material

Probably, the most common operation you will perform in the Create/Edit Materials Dialog Box (p. 4885) is the modification of properties for an existing material. The steps for this procedure are as follows:

1. In the Materials Task Page (p. 4883), select the material you want to modify and click the **Create/Edit...** button. (You can also right-click the material in the tree and select **Edit...** from the material submenu.)

2. In the Create/Edit Materials Dialog Box (p. 4885) select the type of material (**fluid**, **solid**, and so on) in the **Material Type** drop-down list.

3. Choose the material for which you want to modify properties, in the **Fluent Fluid Materials** drop-down list, **Fluent Solid Materials** list, or other similarly named list. The name of the list will be the same as the material type you selected in the previous step.

4. Make the required changes to the properties listed in the **Properties** section of the dialog box. You can use the scroll bar to the right of the **Properties** section to scroll through the listed items.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Click the **Change/Create** button to change the properties of the selected material to your new property settings.

To change the properties of an additional material, repeat the process described above. Click the **Change/Create** button after making changes to the properties for each material.

8.1.3.2. Renaming an Existing Material

Each material is identified by a name and a chemical formula (if one exists). You can change the name of a material, but not its chemical formula (unless you are creating a new material). The procedure for renaming a material is as follows:

1. In the Materials Task Page (p. 4883), select the material you want to rename and click the **Create/Edit...** button. (You can also right-click the material in the tree and select **Edit...** from the material submenu.)

2. In the Create/Edit Materials Dialog Box (p. 4885), choose the material for which you want to modify properties, in the **Fluent Fluid Materials** list, **Fluent Solid Materials** list, or other similarly named list. The name of the list will be the same as the material type you selected in the previous step.

3. Enter the new name in the **Name** field at the top of the Create/Edit Materials Dialog Box (p. 4885).

   Important:

   The maximum character length you can enter in the **Name** field is 29. If you enter a material name that is more than 29 characters long, Ansys Fluent will print an error message in the console window.

4. Click the **Change/Create** button.

   A Question Dialog Box (p. 931) will appear, asking you if the original material should be overwritten.

   If you are renaming the original material, click **Yes** to overwrite it. If you were creating a new material, click **No** to retain the original material.

To rename another material, repeat the process described above. Click the **Change/Create** button after renaming each material.

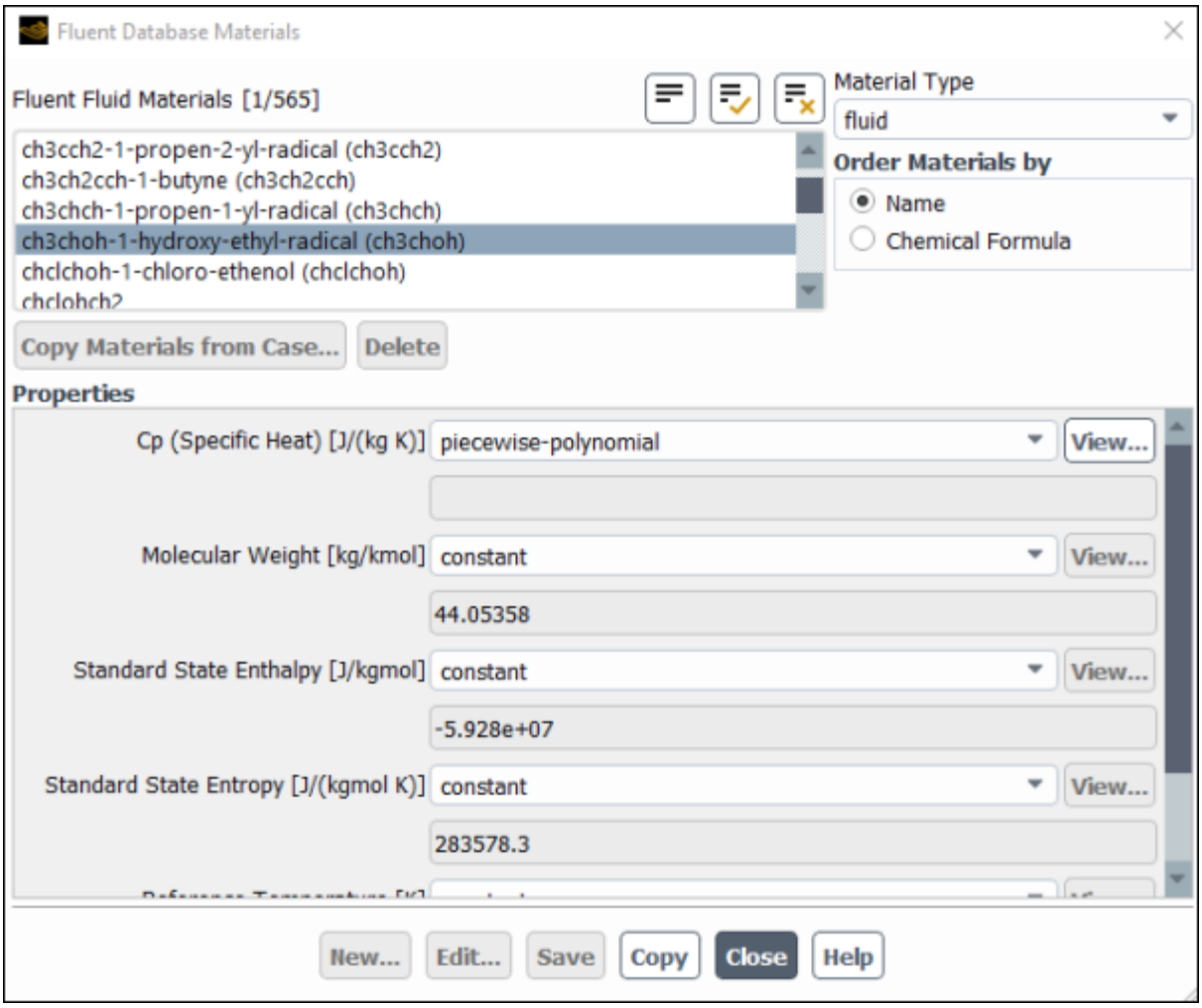
8.1.3.3. Copying Materials from the Ansys Fluent Database

The global (site-wide) materials database contains many commonly used fluid, solid, and mixture materials, with property data from several different sources [97] (p. 5718), [131] (p. 5720), [172] (p. 5723), [66] (p. 5716). To use one of these materials in your problem, copy it from the Ansys Fluent database to your local materials list. The procedure for copying a material is as follows:

1. Click the **Fluent Database...** button in the Create/Edit Materials Dialog Box (p. 4885) to open the Fluent Database Materials Dialog Box (p. 4897) (Figure 8.3: Fluent Database Materials Dialog Box (p. 1603)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 8.3: Fluent Database Materials Dialog Box



2. Select the type of material (**fluid**, **solid**, and so on) in the **Material Type** drop-down list.

3. In the **Fluent Fluid Materials** list, **Fluent Solid Materials** list, or other similarly named list, choose the materials you want to copy by clicking on them. The properties of the selected material will be displayed in the **Properties** area.

4. To check the material properties, use the scroll bar to the right of the **Properties** area to scroll through the listed items. For some properties, temperature-dependent functions are available in addition to the constant values. Select one of the function types in the drop-down list to the right of the property and the relevant parameters will be displayed. You cannot edit these values, but the dialog boxes in which they are displayed function in the same way as those used for setting temperature-dependent property functions (Defining Properties Using Temperature-Dependent Functions (p. 1616)).

   The inactive buttons in the Fluent Database Materials Dialog Box (p. 4897) are operations that are applicable only for a user-defined database. These operations will be available when you click the ***User-Defined Database...*** *button in the Create/Edit Materials Dialog Box (p. 4885).*

5. Click **Copy**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The materials and their properties will be downloaded from the database into your local list, and your copy of properties will now be displayed in the Materials Task Page (p. 4883) and in the tree under the appropriate material type.

6. Close the Fluent Database Materials Dialog Box (p. 4897).

After copying a material from the database, you can modify its properties or change its name, as described earlier in this section. The original material in the database will not be affected by any changes made to your local copy of the material.

8.1.3.4. Copying Materials from the Ansys GRANTA MDS Database

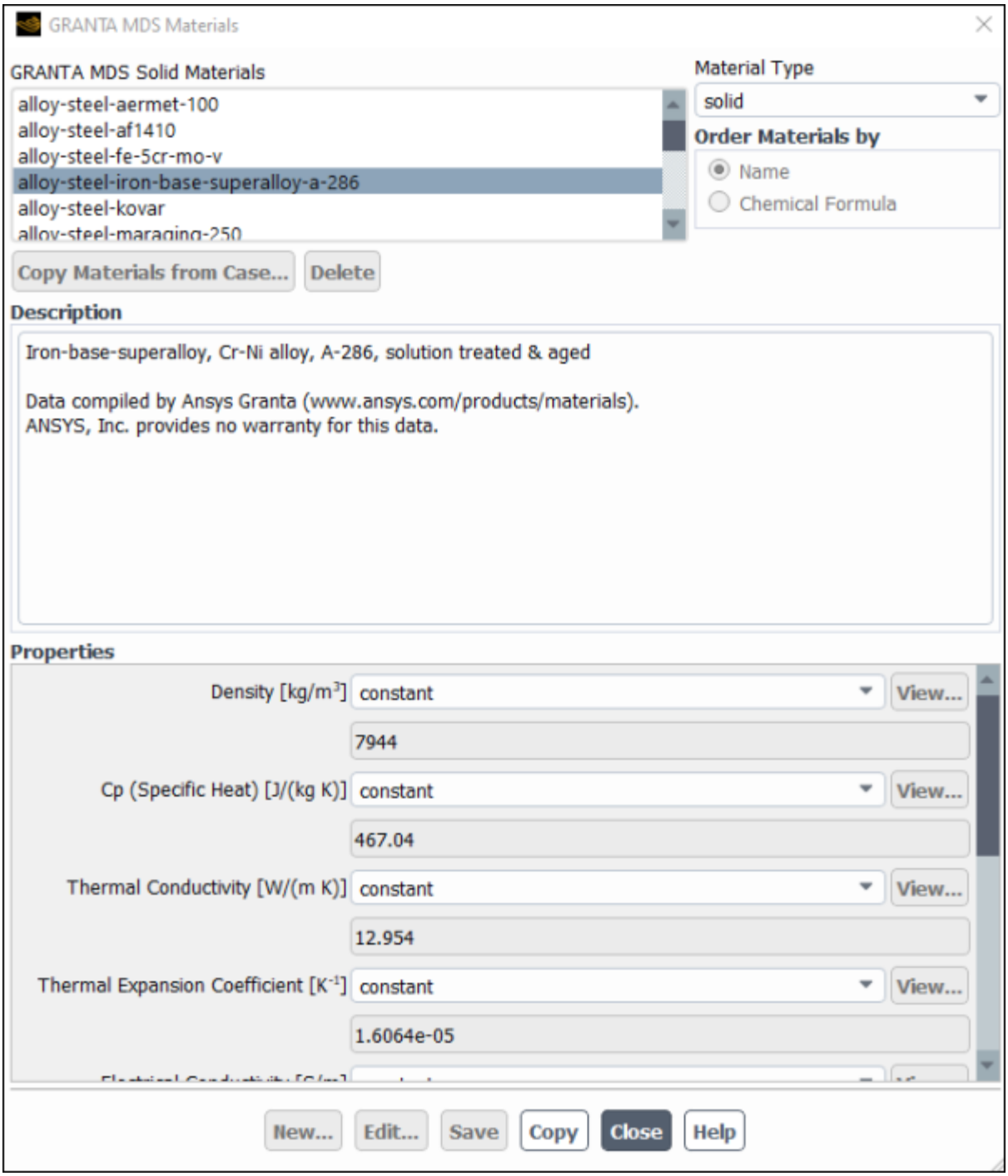
The procedure for copying a material is similar to that for the Ansys Fluent Database (Copying Materials from the Ansys Fluent Database (p. 1602)). Only the differences are highlighted here.

The global (site-wide) GRANTA MDS database contains many solid materials with property data from GRANTA. To use one of these materials, copy it from the Ansys GRANTA MDS database to your local materials list. Doing so requires a separate license.

1. Click the **GRANTA MDS Database...** button in the Create/Edit Materials Dialog Box (p. 4885) to open Figure 8.4: GRANTA MDS Materials Dialog Box (p. 1605).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 8.4: GRANTA MDS Materials Dialog Box



2. In the **GRANTA MDS Fluid Materials** list or **GRANTA MDS Solid Materials** list, choose the material you want to copy by clicking it (only one material can be selected at a time). A description of the selected material is displayed in the **Description** group box, and its properties are listed in the **Properties** group box.

   When no GRANTA license is available, only the description of the selected material will be displayed, but not its properties. The copy operation will abort with an information message.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The details of the remaining steps are covered in Copying Materials from the Ansys Fluent Database (p. 1602).

8.1.3.5. Creating a New Material

If the material you want to use is not available in the database, you can easily create a new material for the local list. This material will be available for use only for the current problem and will not be saved in the Ansys Fluent database. The procedure for creating a new material is as follows:

1. In the Materials Task Page (p. 4883), click the **Create/Edit...** button. (You can also right-click the corresponding material type in the tree (**Fluid**, **Solid**, and so on) and select **New...** from the material type submenu.)

2. Select the new material type (**fluid**, **solid**, and so on) in the **Material Type** drop-down list. It does not matter which material is selected in the **Fluent Fluid Materials**, **Fluent Solid Materials**, or other similarly named list.

3. Enter the new material name in the **Name** field.

   Important:

   - Ensure you change the name from the default, otherwise the updated properties will change the existing definition of the specified material instead of creating a new one.
   - The maximum character length you can enter in the **Name** field is 29. If you enter a material name that is more than 29 characters long, Ansys Fluent will print an error message in the console window.

4. Set the material's properties in the **Properties** area. If there are many properties listed, you may use the scroll bar to the right of the **Properties** area to scroll through the listed items.

5. Click the **Change/Create** button. A Question Dialog Box (p. 931) will appear, asking you if the original material should be overwritten.

   a. Click **No** to retain the original material and *add* your new material to the list. A dialog box will appear asking you to enter the chemical formula of your new material.

   b. Click **OK**, enter the formula if it is known. Else, leave the formula blank and click **OK**. Select the **Change/Create** button and answer the **Question**.

      The Materials Task Page (p. 4883) and the **Material** tree branch will be updated to show the new material name and chemical formula under the appropriate material type (**Fluid**, **Solid**, or others).

8.1.3.6. Saving Materials and Properties

All the materials and properties in your local list are saved in the case file when it is written. If you read this case file into a new solver session, all of your materials and properties will be available for use in the new session.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.1.3.7. Deleting a Material

If there are materials list that you no longer need, you can delete them in one of the following ways:

- Using the Materials Task Page (p. 4883).

  In the Materials Task Page (p. 4883), select the material to be deleted and click the **Delete** button at the bottom of the Materials Task Page (p. 4883).

- Using the tree.

  Right-click the material in the tree and select **Delete** from the material submenu.

- Using the Create/Edit Materials Dialog Box (p. 4885).

  1. From the **Material Type** drop-down list, select the type of material (**fluid**, **solid**, and so on).
  2. Select the material to be deleted from the **Fluent Fluid Materials** drop-down list, **Fluent Solid Materials** list, or other similarly named list. The list's name will be the same as the material type you selected in the previous step.
  3. Click the **Delete** button.

Deleting materials from your local list will have no effect on the materials contained in the global database.

8.1.3.8. Changing the Order of the Materials List

By default, the materials in your local list and those in the database are listed alphabetically by name (for example, **air**, **atomic-oxygen (o)**, **carbon-dioxide (co2)**). If you prefer to list them alphabetically by chemical formula, select the **Chemical Formula** option under **Order Materials By**. The example materials listed, will now be in the order of: **air**, **co2 (carbon-dioxide)**, **o (atomic-oxygen)**. To change back to the alphabetical listing by name, choose the **Name** option under **Order Materials By**.

You may specify the ordering method separately for the Create/Edit Materials Dialog Box (p. 4885) and Fluent Database Materials Dialog Box (p. 4897). For example, you can order the database materials list by name. Each dialog box has its own **Order Materials By** options.

8.1.4. Using a User-Defined Materials Database

In addition to the Fluent Database Materials Dialog Box (p. 4897), you can also use or create a user-defined materials database using the User-Defined Database Materials Dialog Box (p. 4901). You can browse and do the following:

- select from existing user-defined databases
- copy materials from a user-defined database
- create a new database, create new materials
- add them to the user-defined database

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- delete materials from the database
- copy materials from a case to a user-defined database
- view the database

The following sections will address each of these functionalities in detail.

8.1.4.1. Opening a User-Defined Database

In Ansys Fluent, you can open databases of custom materials saved as `.scm` files and use them to define the materials in your problem setup. The material data must be prepared in a specific format as shown in the examples that follow.

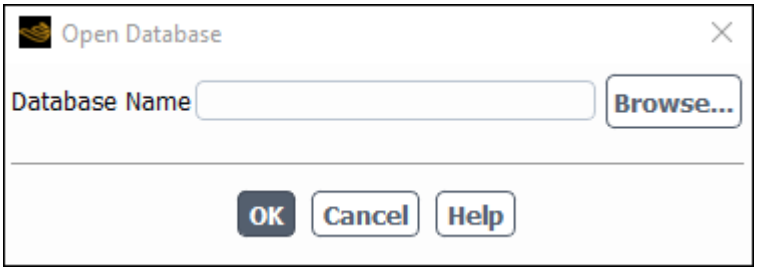
Examples:

The prescribed format for saving material properties information is shown here for air and aluminum. These files can be created in a text editor and saved with a `.scm` extension.

```
 ((air
 fluid
 (chemical-formula . #f)
 (density (constant . 1.225)
 (premixed-combustion 1.225 300))
 (specific-heat (constant . 1006.43))
 (thermal-conductivity (constant . 0.0242))
 (viscosity (constant . 1.7894e-05)
 (sutherland 1.7894e-05 273.11 110.56)
 (power-law 1.7894e-05 273.11 0.666))
 (molecular-weight (constant . 28.966))
 )
(aluminum
 solid
 (chemical-formula . al)
 (density (constant . 2719))
 (specific-heat (constant . 871))
 (thermal-conductivity (constant . 202.4))
 (formation-entropy (constant . 164448.08))
 ))
```

To select a user-defined database, click the **User-Defined Database...** button in the Create/Edit Materials Dialog Box (p. 4885). This will open the Open Database Dialog Box (p. 4901).

Figure 8.5: Open Database Dialog Box



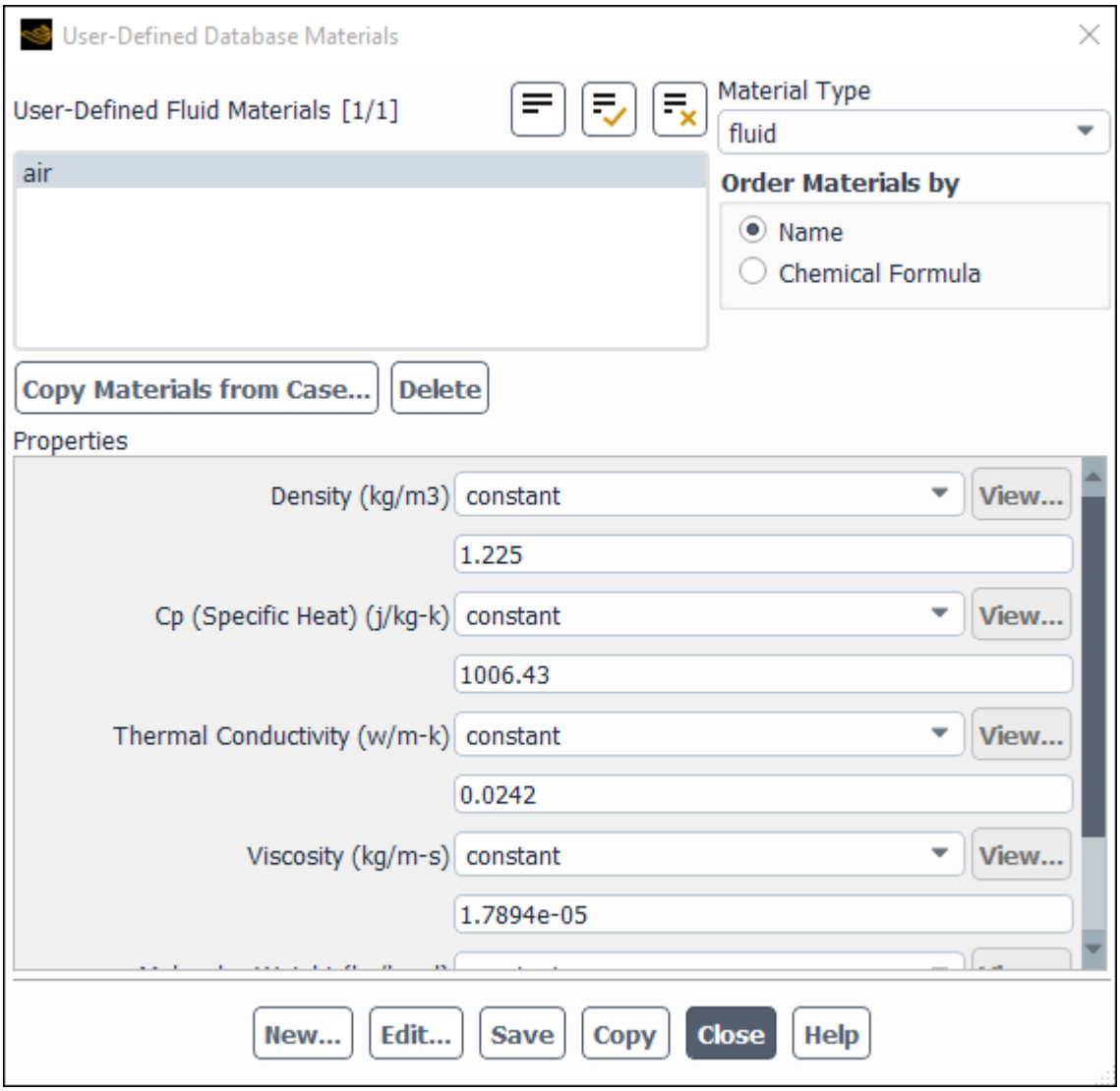
Click the **Browse...** button, select the database in The Select File Dialog Box (p. 931) that opens and click **OK**. Click **OK** in the Open Database Dialog Box (p. 4901) to open the User-Defined Database Materials Dialog Box (p. 4901).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.1.4.2. Viewing Materials in a User-Defined Database

When an existing user-defined database is opened, the materials present in the database are listed in the User-Defined Database Materials Dialog Box (p. 4901). You can select the material type in the **Material Type** drop-down list and the corresponding materials will appear in the **User-Defined Fluid Materials**, **User-Defined Solid Materials** or other similarly named list. The name of the list will be the same as the material type you selected.

Figure 8.6: User-Defined Database Materials Dialog Box



The properties of the selected material will appear in the **Properties** section of the dialog box. This dialog box is similar to the Fluent Database Materials Dialog Box (p. 4897) in function and operation.

8.1.4.3. Copying Materials from a User-Defined Database

The procedure for copying a material from a custom database is as follows:

1. In the Create/Edit Materials Dialog Box (p. 4885), click the **User-Defined Database...** button and open the database from which you want to copy the material.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. In the User-Defined Database Materials Dialog Box (p. 4901) of the selected database, select the type of material (**fluid**, **solid**, and so on) in the **Material Type** drop-down list.

3. In the **User-Defined Fluid Materials** list, **User-Defined Solid Materials** list, or other similarly named list (the list's name will be the same as the material type you selected in the previous step), choose the materials you want to copy by clicking on them. The properties are displayed in the **Properties** area.

4. If you want to check the material properties, use the scroll bar to the right of the **Properties** area to scroll through the listed items.

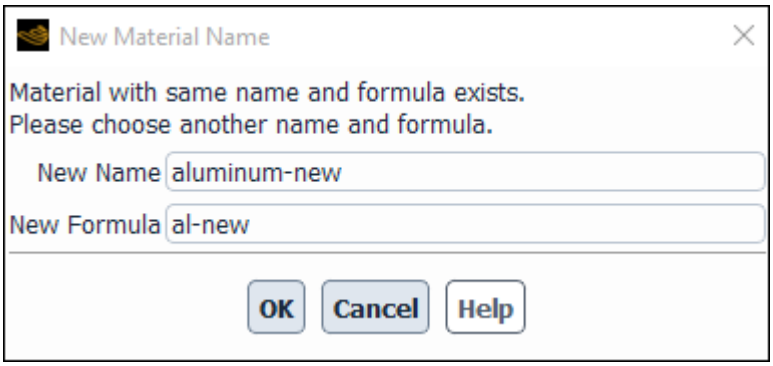
5. Click the **Copy** button.

   The selected materials and their properties will be copied from the database into your local list, and your copy of the properties will now be displayed in the Materials Task Page (p. 4883) and under the **Materials** tree branch.

   To copy all the materials from the database in one step, click the button (*) next to **User-Defined Materials** title and click **Copy**.*

   If a material with the same name is already defined in the case, Ansys Fluent will prompt you to enter a new name and formula in the New Material Name Dialog Box (p. 4906). Enter a new name and formula in the respective fields and click **OK** to make a local copy of the material.

   Figure 8.7: New Material Name Dialog Box



6. Close the User-Defined Database Materials Dialog Box (p. 4901).

After copying a material from the database, you may modify its properties or change its name, as described earlier in Using the **Create/Edit Materials** Dialog Box (p. 1599). The material in the database will not be affected by any changes you make to your local copy of the material.

8.1.4.4. Copying Materials from the Case to a User-Defined Database

The procedure for copying a material to a custom database is as follows:

1. In the Create/Edit Materials Dialog Box (p. 4885), click **User-Defined Database...**.

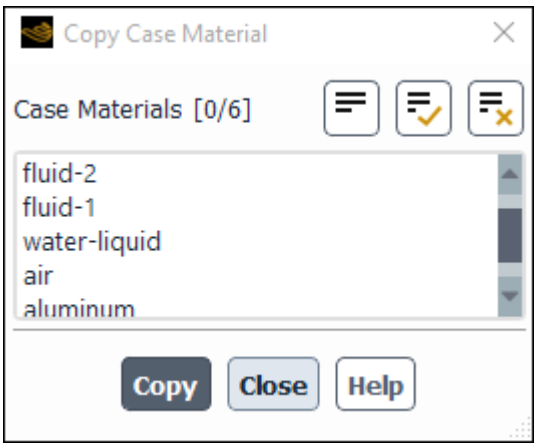
2. In the Open Database Dialog Box (p. 4901), select the database to which you want to copy the material. If you want to create a new database, enter the name of the new database in the

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Database Name field and click **OK**. A Question Dialog Box (p. 931) will ask you to confirm if you want to create a new file. Click **Yes** to confirm.

3. In the User-Defined Database Materials Dialog Box (p. 4901), click **Copy Materials From Case....** This will open the Copy Case Material Dialog Box (p. 4903).

Figure 8.8: Copy Case Material Dialog Box



a. In the Copy Case Material Dialog Box (p. 4903), select the materials that you want to copy.

*To select all the materials, click the shaded icon to the right of the **Case Materials** title. Clicking on the unshaded icon will deselect the selections in the list.*

b. Click **Copy** and close the dialog box.

> **Note:**
>
> Do not copy materials one by one. This will result in previously copied materials getting overwritten by the new ones. Instead, select all the materials to be copied and click **Copy**.

8.1.4.5. Modifying Properties of an Existing Material

You can modify the properties of an existing material and use the modified material in the problem setup and save the modified material to the materials database.

1. In the Create/Edit Materials Dialog Box (p. 4885), click the **User-Defined Database...** button and open the database that you want to use.

   a. In the User-Defined Database Materials Dialog Box (p. 4901) of the selected database, select the type of material (**fluid**, **solid**, and so on) in the **Material Type** drop-down list.

   b. In the **User-Defined Fluid Materials** list, **User-Defined Solid Materials** list, or other similarly named list (the name of the list will be the same as the material type you selected in the previous step), select the material to be modified.

   c. Click **Edit...** to open the Material Properties Dialog Box (p. 4904).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

i. In the **Materials Properties** list, select the property to be modified and click **Edit...** to open the Edit Property Methods Dialog Box (p. 4905).

ii. Select the method to be modified in the **Material Properties** list of the Edit Property Methods Dialog Box (p. 4905) and click **Edit...** under **Edit Properties**, in order to modify the properties.

iii. Make the changes in the corresponding method dialog box and click **OK**.

iv. Click **Apply** in the Material Properties Dialog Box (p. 4904).

d. To use the modified material in the problem setup, click **Copy** in the User-Defined Database Materials Dialog Box (p. 4901).

e. To save the modified material to the database, click **Save** and close the dialog box.

8.1.4.6. Creating a New Materials Database and Materials

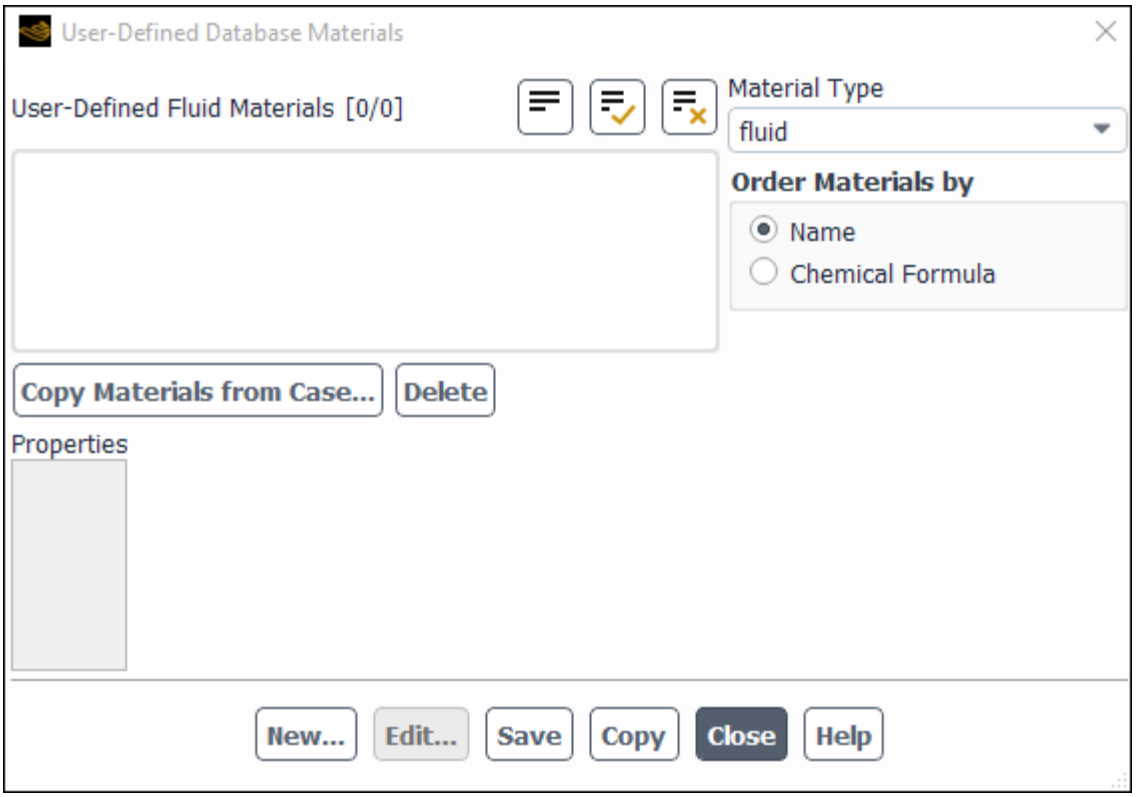
Using the Create/Edit Materials Dialog Box (p. 4885), you can create a new materials database, copy materials to this database, and also create new materials from scratch. The procedure for creating a new database and add new materials to the database is as follows:

1. In the Materials Task Page (p. 4883), click **User-Defined Database...**.
2. In the Open Database Dialog Box (p. 4901), enter the name of the database that you are creating and click **OK**.
3. A dialog box will appear asking you confirm the creation of a new file. Click **Yes** to confirm.

   This will open a blank User-Defined Database Materials Dialog Box (p. 4901) (Figure 8.9: User-Defined Database Materials Dialog Box: Blank (p. 1613)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

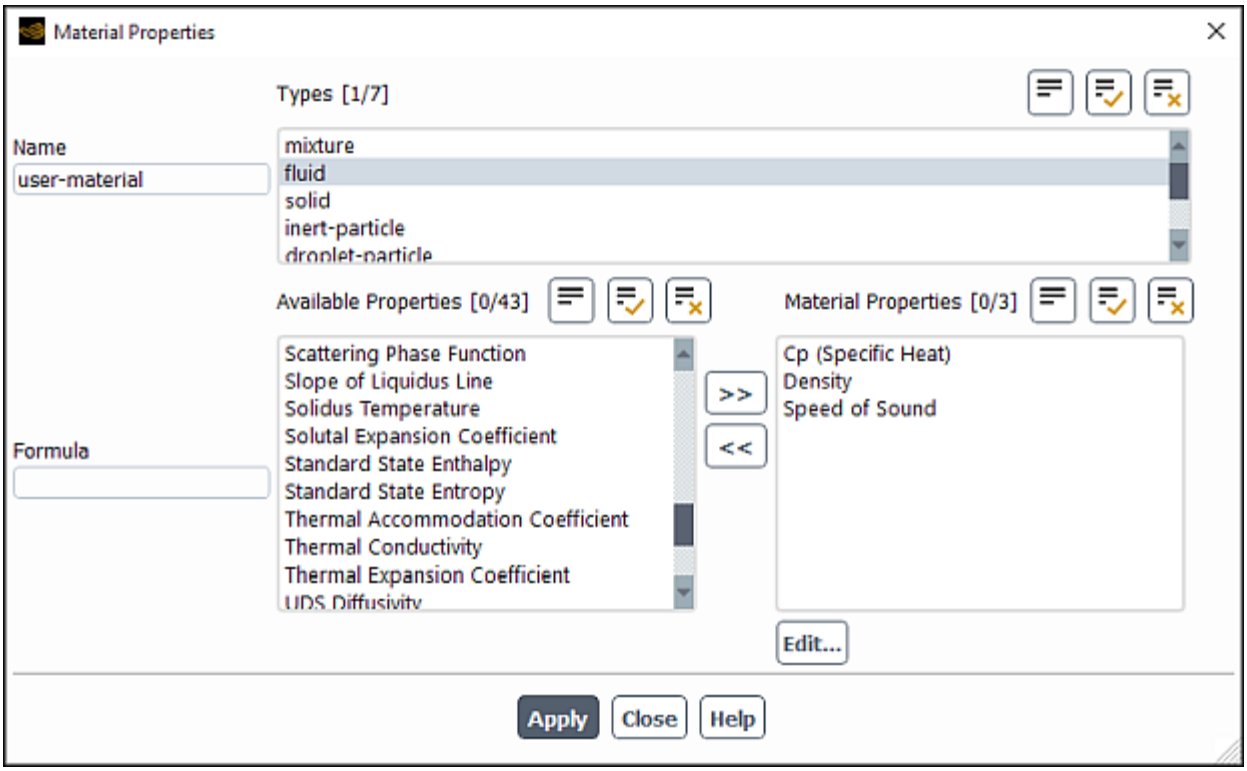
Figure 8.9: User-Defined Database Materials Dialog Box: Blank



4. Click **New...** in the User-Defined Database Materials Dialog Box (p. 4901). This will open a blank **Material Properties** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 8.10: Material Properties Dialog Box: Blank



a. In the Material Properties Dialog Box (p. 4904), under **Types**, select the material type. You can select from **fluid**, **solid**, **inert-particle**, **droplet-particle**, **combusting-particle**, and **mixture** materials.

b. Enter the name and formula (if required) of the material that you are creating in the **Name** and **Formula** fields.

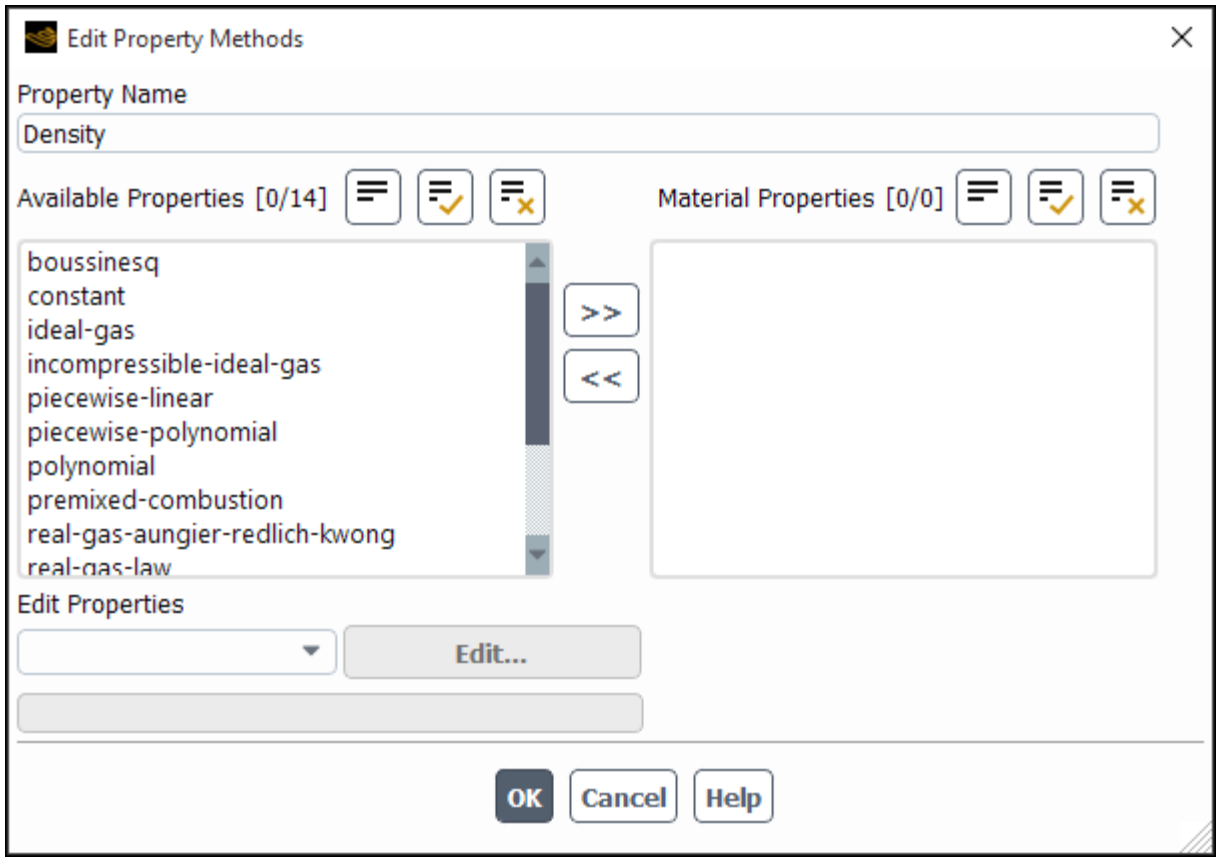
c. Depending on the type of material selected in the **Types** list, properties applicable to that material type will appear in the **Available Properties** list. Select the properties that are applicable for the material that you are defining by clicking on them.

d. Click the ≫ button to move these properties to the **Material Properties** list on the right and click **Apply**. You can use the ≪ button to move the property from the **Material Properties** list to the **Available Properties** list.

5. To edit the parameters that define a property, select the property in the **Material Properties** list and click **Edit...**. This opens the Edit Property Methods Dialog Box (p. 4905).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 8.11: Edit Property Methods Dialog Box



a. The methods that can be used to define the selected property are listed in the **Available Properties** list. You can select one or more methods and specify them for the material that you are defining, by selecting and moving them to the **Material Properties** list.

b. To modify each of these methods, you can select the method in the **Edit Properties** drop-down list and click **Edit...**. This will open the corresponding property dialog box, where you can modify the parameters used by the property method. Refer to Defining Properties Using Temperature-Dependent Functions (p. 1616) to Real Gas Models (p. 1681) for details of these properties, methods used to define the properties and the parameters for each method.

c. Click **OK** in the Edit Property Methods Dialog Box (p. 4905).

6. Click **Apply** in the Material Properties Dialog Box (p. 4904).

7. Click **Save** in the User-Defined Database Materials Dialog Box (p. 4901) to save the changes to the new materials database.

Similarly, you can also append new materials and click save to append these materials to the existing database.

8.1.4.7. Deleting Materials from a Database

To delete a material from a database, click the **User-Defined Database** button in the Open Database Dialog Box (p. 4901). Select the database and click **OK** in the Open Database Dialog Box (p. 4901). Select

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the **Material Type** and the materials that you want to delete in the **User-Defined Materials** list and click **Delete**. Click **Save** to save the database.

8.2. Defining Properties Using Temperature-Dependent Functions

Material properties can be defined as functions of temperature. For most properties, you can define a polynomial, piecewise-linear, or piecewise-polynomial function of temperature. For specific heat (Cp), you can also use a nasa-9-piecewise-polynomial function of temperature in extremely high-temperature applications. The definitions of these functions are as follows:

- polynomial:

$$\phi(T) = A_1 + A_2 T + A_3 T^2 + \ldots \tag{8.1}$$

- piecewise-linear:

$$\phi(T) = \phi_n + \frac{\phi_{n+1} - \phi_n}{T_{n+1} - T_n}(T - T_n) \tag{8.2}$$

  where $1 \le n \le N$ and N is the number of segments

- piecewise-polynomial:

$$\begin{aligned} &\text{for } T_{min,1} \le T < T_{max,1}: \phi(T) = A_1 + A_2 T + A_3 T^2 + \ldots \\ &\text{for } T_{min,2} \le T < T_{max,2}: \phi(T) = B_1 + B_2 T + B_3 T^2 + \ldots \end{aligned} \tag{8.3}$$

- nasa-9-piecewise-polynomial:

$$\begin{aligned} &\text{for } \quad T_{min,1} \le T < T_{max,1}: Cp(T) = A_1 T^{-2} + A_2 T^{-1} + A_3 + A_4 T + A_5 T^2 + A_6 T^3 + A_7 T^4 \\ &\text{for } \quad T_{min,2} \le T < T_{max,2}: Cp(T) = B_1 T^{-2} + B_2 T^{-1} + B_3 + B_4 T + B_5 T^2 + B_6 T^3 + B_7 T^4 \end{aligned} \tag{8.4}$$

In the equations above, ϕ is the property.

Important:

If you define a polynomial or piecewise-polynomial function of temperature, the temperature in the function is always in units of Kelvin or Rankine. If you use Celsius or Kelvin as the temperature unit, then polynomial coefficient values must be entered in terms of Kelvin. If you use Fahrenheit or Rankine as the temperature unit, enter the values in terms of Rankine.

Some properties have additional functions available and for some only a subset of these three functions can be used. See the section on the property in question to determine which temperature-dependent functions you can use.

For additional information, see the following sections:

8.2.1. Inputs for Polynomial Functions
8.2.2. Inputs for Piecewise-Linear Functions
8.2.3. Inputs for Piecewise-Polynomial Functions
8.2.4. Inputs for NASA-9-Piecewise-Polynomial Functions
8.2.5. Checking and Modifying Existing Profiles

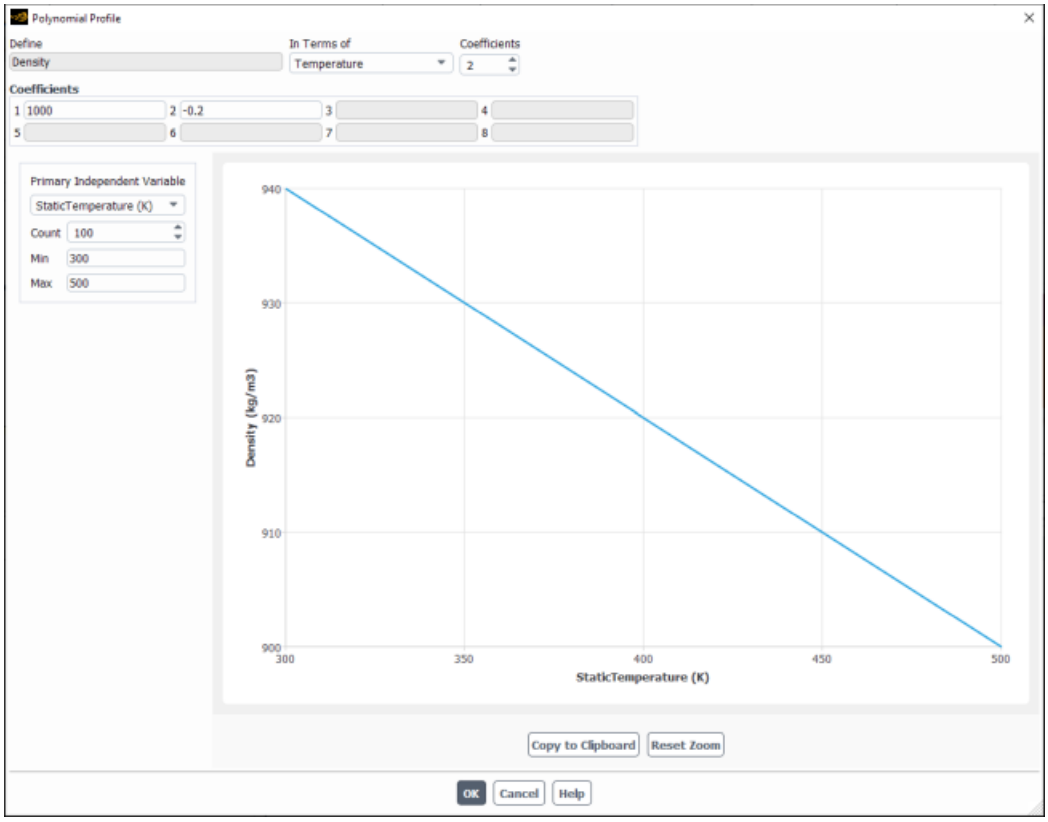
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.2.1. Inputs for Polynomial Functions

To define a polynomial function of temperature for a material property, do the following:

1. In the Create/Edit Materials Dialog Box (p. 4885), choose **polynomial** in the drop-down list to the right of the property name (for example, **Density**). The Polynomial Profile Dialog Box (p. 4906) (Figure 8.12: The Polynomial Profile Dialog Box (p. 1617)) will open automatically.

   Figure 8.12: The Polynomial Profile Dialog Box



   Since this is a modal dialog box, the solver will not allow you to do anything else until you perform the following steps.

   a. Specify the number of **Coefficients** up to 8 coefficients are available. The number of coefficients defines the order of the polynomial. The default of 1 defines a polynomial of order 0. The property will be constant and equal to the single coefficient A_1. An input of 2 defines a polynomial of order 1 and the property will vary linearly with temperature and so on.

   b. Define the coefficients. Coefficients **1**, **2**, **3**,... correspond to A_1, A_2, A_3,... in Equation 8.1 (p. 1616). The dialog box in Figure 8.12: The Polynomial Profile Dialog Box (p. 1617) shows the inputs for the following function:

$$\rho(T) = 1000 - 0.02T \tag{8.5}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. To visualize your profile, ensure the correct **Primary Independent Variable** is selected from the drop-down list, and enter a value for the **Min** and **Max**.

Important:

To ensure the robustness of simulations when using polynomials to specify material property, set the solution limits so that the values returned by the polynomial expression are physically valid. For example, when the density of a material is specified as a function of temperature using a polynomial expression, set the proper limits on the temperature range so that the density is always non-negative. For more information on setting solution limits see Setting Solution Limits (p. 3621).

Also, note the restriction on the units for temperature, as described in the previous section.

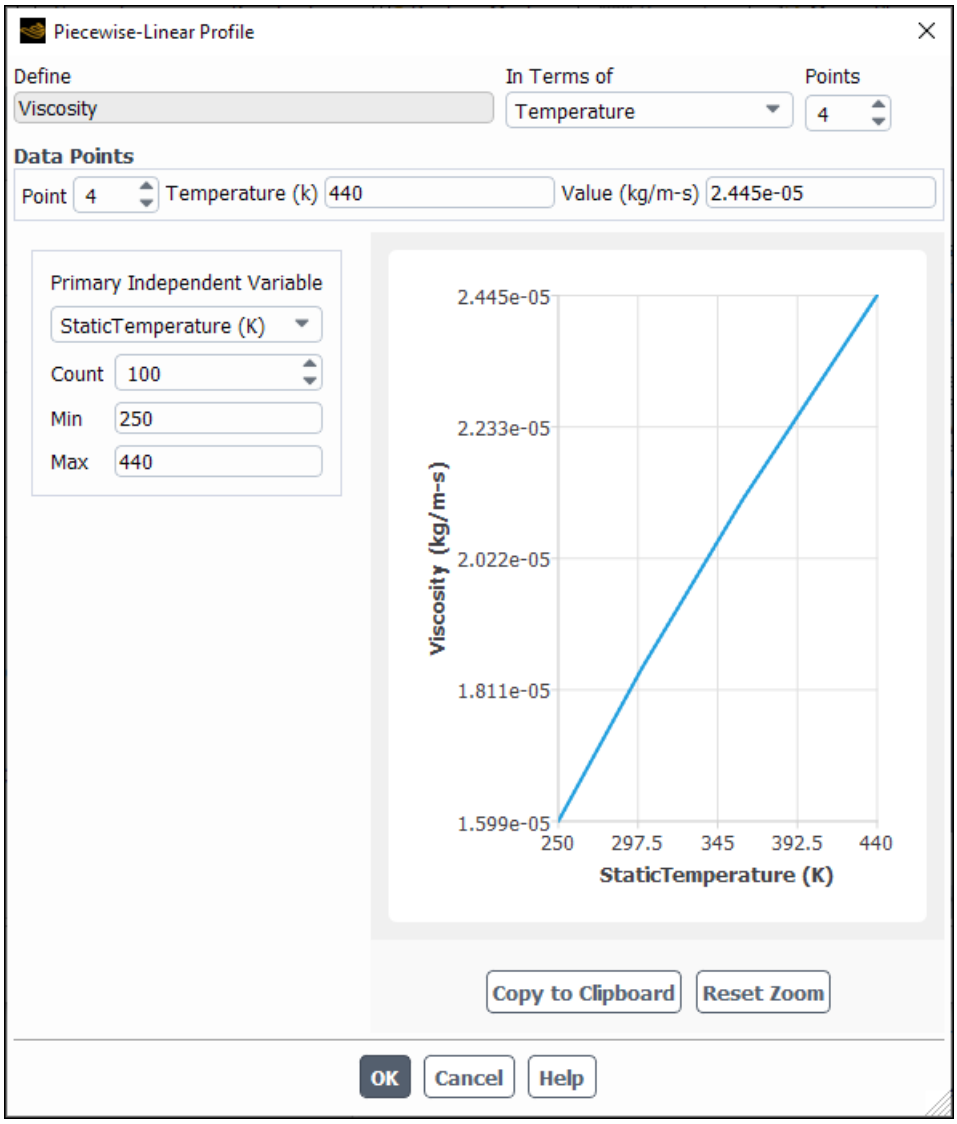
8.2.2. Inputs for Piecewise-Linear Functions

To define a piecewise-linear function of temperature for a material property, do the following:

1. In the Create/Edit Materials Dialog Box (p. 4885), choose **piecewise-linear** in the drop-down list to the right of the property name (for example, **Viscosity**). The Piecewise-Linear Profile Dialog Box (p. 4907) (Figure 8.13: The Piecewise-Linear Profile Dialog Box (p. 1619)) will open automatically.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 8.13: The Piecewise-Linear Profile Dialog Box



Since this is a modal dialog box, the solver will not allow you to do anything else until you perform the following steps.

a. Set the number of **Points** defining the piecewise distribution.

b. Under **Data Points**, enter the data pairs for each point. First enter the independent and dependent variable values for **Point** 1, then increase the **Point** number and enter the appropriate values for each additional pair of variables. The pairs of points must be supplied in the order of increasing value of temperature. The solver will *not* sort them for you. A maximum of 50 piecewise points can be defined for each property. The dialog box in Figure 8.13: The Piecewise-

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

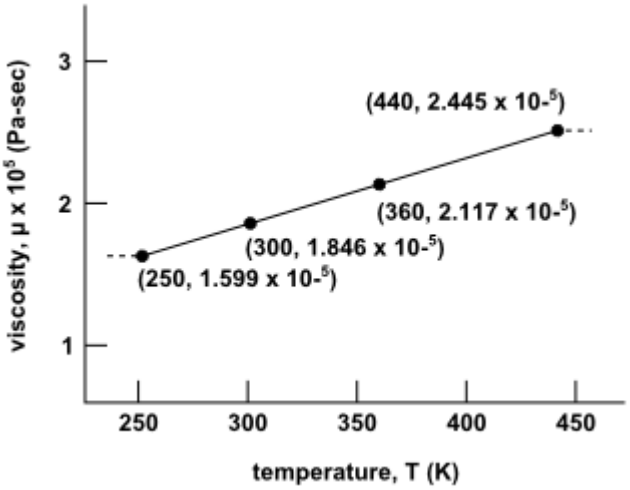
Linear Profile Dialog Box (p. 1619) shows the final inputs and a plot of the profile that is depicted in Figure 8.14: Piecewise-Linear Definition of Viscosity as a Function of Temperature (p. 1620).

Important:

If the temperature exceeds the maximum **Temperature** (T_{max}) you have specified for the profile, Ansys Fluent will use the **Value** corresponding to T_{max}. If the temperature falls below the minimum **Temperature** (T_{min}) specified for your profile, Ansys Fluent will use the **Value** corresponding to T_{min}.

c. To visualize your profile, ensure the correct **Primary Independent Variable** is selected from the drop-down list, and enter a value for the **Min** and **Max**.

Figure 8.14: Piecewise-Linear Definition of Viscosity as a Function of Temperature



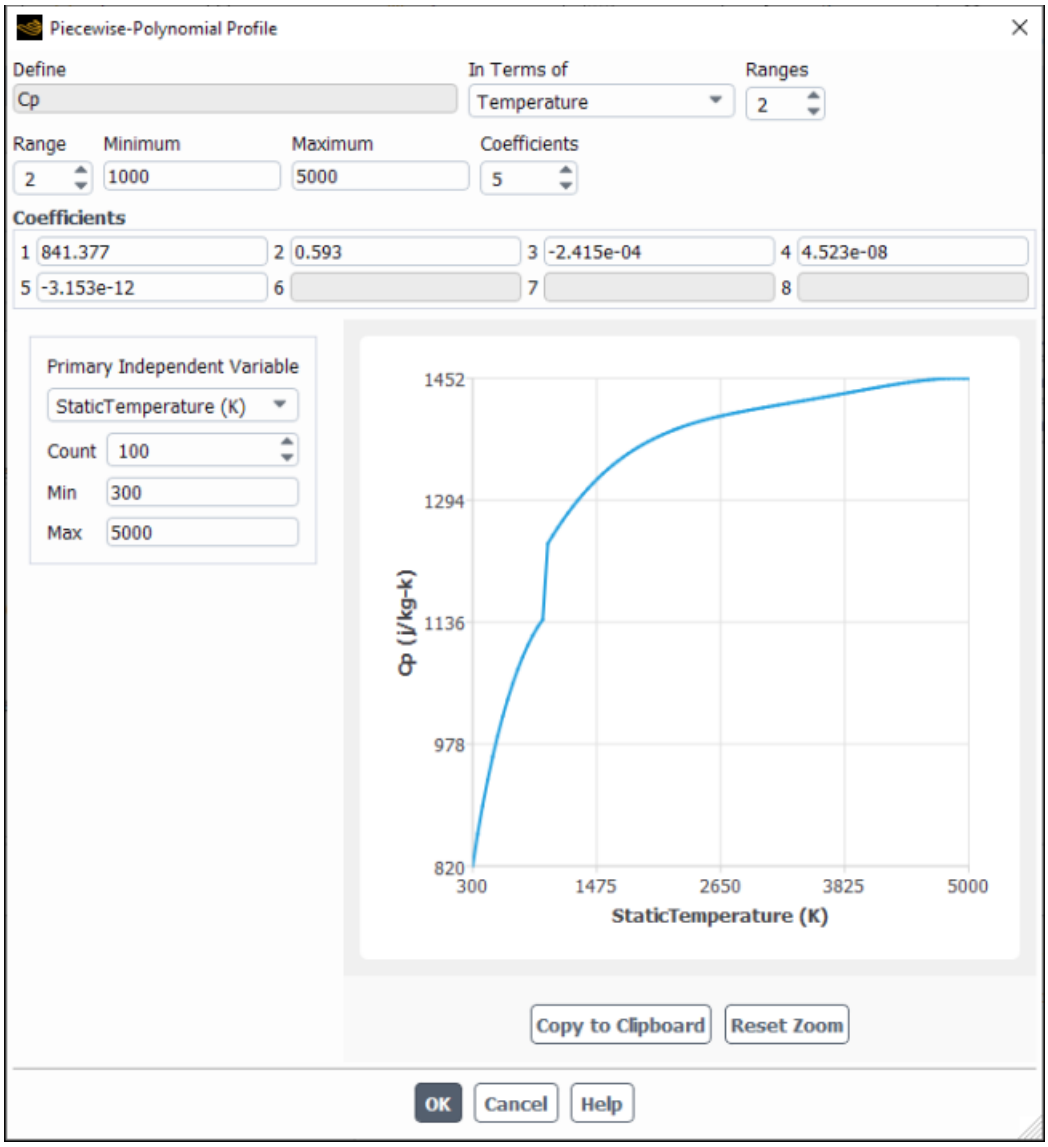
8.2.3. Inputs for Piecewise-Polynomial Functions

To define a piecewise-polynomial function of temperature for a material property, follow these steps:

1. In the Create/Edit Materials Dialog Box (p. 4885), choose **piecewise-polynomial** in the drop-down list to the right of the property name (for example, **Cp**). The Piecewise-Polynomial Profile Dialog Box (p. 4909) (Figure 8.15: The Piecewise-Polynomial Profile Dialog Box (p. 1621)) will open automatically. Since this is a modal dialog box, first perform the following steps.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 8.15: The Piecewise-Polynomial Profile Dialog Box



2. Specify the number of **Ranges**. For the example of Equation 8.6 (p. 1621), two ranges of temperatures are defined:

$$c_p(T) = \begin{cases} 429.929 + 1.784T - 1.966\times10^{-3}T^2 \\ \quad +1.297\times10^{-6}T^3 - 4.000\times10^{-10}T^4 & for\ 300 \le T < 1000 \\ 841.377 + 0.593T - 2.415\times10^{-4}T^2 \\ \quad +4.523\times10^{-8}T^3 - 3.153\times10^{-12}T^4 & for\ 1000 \le T < 5000 \end{cases} \tag{8.6}$$

   You may define up to three ranges. The ranges must be supplied in the order of increasing value of temperature. The solver will *not* sort them for you.

3. For the first range (**Range** = 1), specify the **Minimum** and **Maximum** temperatures, and the number of **Coefficients**. (Up to eight coefficients are available.) The number of coefficients defines the order of the polynomial. The default of 1 defines a polynomial of order 0. The property will

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

be constant and equal to the single coefficient A_1. An input of 2 defines a polynomial of order 1. The property will vary linearly with temperature and so on.

4. Define the coefficients. **Coefficients 1**, **2**, **3**,... correspond to A_1, A_2, A_3,... in Equation 8.3 (p. 1616). The dialog box in Figure 8.15: The Piecewise-Polynomial Profile Dialog Box (p. 1621) shows the inputs for the first range of Equation 8.6 (p. 1621).

5. Increase the value of **Range** and enter the **Minimum** and **Maximum** temperatures, number of **Coefficients**, and the **Coefficients** (B_1, B_2, B_3,...) for the next range. Repeat if there is a third range.

6. To visualize your profile, ensure the correct **Primary Independent Variable** is selected from the drop-down list, and enter a value for the **Min** and **Max**.

8.2.4. Inputs for NASA-9-Piecewise-Polynomial Functions

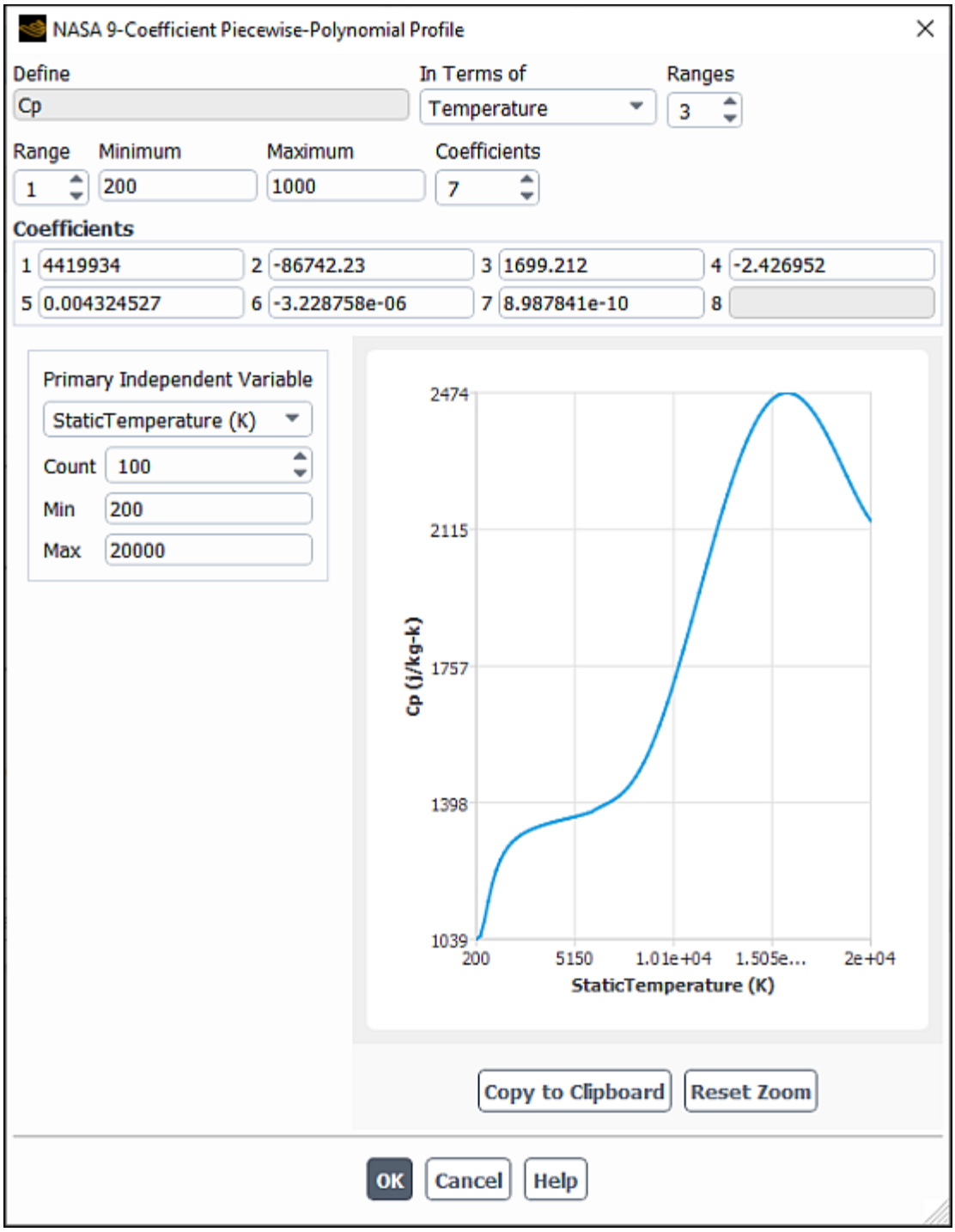
This function is available for specific heat only and is useful for hypersonic flows, and can be used with the two-temperature model, where piecewise-polynomial temperature profiles are not valid.

The procedure for defining a nasa-9-piecewise-polynomial function of temperature is similar to that for the piecewise-polynomial function:

1. In the Create/Edit Materials Dialog Box (p. 4885), choose **nasa-9-piecewise-polynomial** from the **Cp (Specific Heat)** drop-down list (**Properties** group box). The **NASA-9-Coefficient Piecewise-Polynomial Profile** dialog box opens automatically with **Cp** and **Temperature** displayed in the **Define** and **In Terms of** fields, respectively (see Figure 8.16: The NASA-9-Coefficient Piecewise-Polynomial Profile Dialog Box (p. 1623)). Note that because this is a modal dialog box, you must tend to it immediately.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 8.16: The NASA-9-Coefficient Piecewise-Polynomial Profile Dialog Box



2. Specify the number of **Ranges**.

   You may define up to eight ranges. The ranges must be supplied in the order of increasing value of temperature. The solver will *not* sort them for you.

3. For the first range (**Range** = 1) specify:

   - The **Minimum** and **Maximum** temperatures

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The number of **Coefficients**

  Up to eight coefficients are available

- **Coefficients 1**, **2**, **3**, and so on.

  Coefficients **1**, **2**, **3**,... correspond to A_1, A_2, A_3,..., respectively, in Equation 8.4 (p. 1616).

To visualize your profile, enter a value for the **Min** and **Max**. Note that **Static Temperature** is automatically selected from the **Primary Independent Variable** drop-down list. You can increase or decrease the value of **Count** to control the smoothness of the profile curve. *You can click and drag in the plot to change the zoom level.*

4. Increase the value of **Range** and enter the **Minimum** and **Maximum** temperatures, number of **Coefficients**, and their values (B_1, B_2, B_3, and so on) for the next range.

5. Repeat the previous step for every range if there are more than two ranges.

8.2.5. Checking and Modifying Existing Profiles

If you want to check or change the coefficients, data pairs, or ranges for a previously-defined profile, click the **Edit...** button to the right of the property name. The appropriate dialog box will open, and you can check or modify the inputs as desired.

Important:

In the Fluent Database Materials Dialog Box (p. 4897), you cannot edit the profiles, but you can examine them by clicking on the **View...** button (instead of the **Edit...** button.)

8.3. Density

Ansys Fluent provides several options for definition of the fluid density:

- constant density
- temperature and/or composition-dependent density
- pressure-dependent density

Each of these input options and the governing physical models are explained in the following sections. In all cases, you will define the **Density** in the Create/Edit Materials Dialog Box (p. 4885).

Setup → Materials

Note:

For transient, variable-density flows that use the pressure-based solver, you may reach convergence faster if you specify that the model equations are solved in an optimal order. For details, see Equation Order (p. 3600).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For additional information, see the following sections:

8.3.1. Defining Density for Various Flow Regimes
8.3.2. Input of Constant Density
8.3.3. Inputs for the Boussinesq Approximation
8.3.4. Compressible Liquid Density Method
8.3.5. Density as a Profile Function of Temperature
8.3.6. Incompressible Ideal Gas Law
8.3.7. Ideal Gas Law for Compressible Flows
8.3.8. Composition-Dependent Density for Multicomponent Mixtures

8.3.1. Defining Density for Various Flow Regimes

The selection of density in Ansys Fluent is very important. Set the density relationship based on your flow regime.

- For compressible flows, the ideal gas law is the appropriate density relationship.
- For incompressible flows, you may choose one of the following methods:
  - Constant density, if you do not want density to be a function of temperature.
  - The incompressible ideal gas law, when pressure variations are small enough that the flow is fully incompressible but you want to use the ideal gas law to express the relationship between density and temperature (for example, for a natural convection problem).
  - Density as a polynomial, piecewise-linear, or piecewise-polynomial function of temperature, when the density is a function of temperature only, as in a natural convection problem.
  - The Boussinesq model, for natural convection problems involving small changes in temperature.
  - The compressible liquid density method allows you to model compressible liquids under high pressures.

8.3.1.1. Mixing Density Relationships in Multiple-Zone Models

If your model has multiple fluid zones that use different materials, you should be aware of the following:

- For calculations with the pressure-based solver that do not use one of the general multiphase models (Solution Strategies for Multiphase Modeling (p. 3184)), the compressible ideal gas law cannot be mixed with any other density methods. This means that if the compressible ideal gas law is used for one material, it must be used for all materials.

  This restriction does not apply to the density-based solvers.

- There is only one specified operating pressure and one specified operating temperature. This means that if you are using the ideal gas law for more than one material, they will share the same operating pressure. If you are using the Boussinesq model for more than one material, they will share the same operating temperature.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.3.2. Input of Constant Density

If you want to define the density of the fluid as a constant, select **constant** in the **Density** drop-down list under **Properties** in the Create/Edit Materials Dialog Box (p. 4885). Enter the value of density for the material.

For the default fluid (air), the density is 1.225 kg/m^3.

8.3.3. Inputs for the Boussinesq Approximation

To enable the Boussinesq approximation for density, choose **boussinesq** from the **Density** drop-down list in the Create/Edit Materials Dialog Box (p. 4885) and specify a constant value for **Density**. You will also need to set the **Thermal Expansion Coefficient**, as well as relevant operating conditions, as described in The Boussinesq Model (p. 1388).

8.3.4. Compressible Liquid Density Method

The compressible liquid treatment enables you to model liquid compressibility under high pressure applications. Fluent models compressible liquids using the Tait equation of state, which establishes a nonlinear relationship between density and pressure under isothermal conditions.

The compressible liquid treatment also helps in reducing unphysical pressure spikes that appear in moving and dynamic mesh applications, especially during solid-fluid interactions.

The Tait equation can be represented in terms of pressure and density using the following relationship:

$$p = a + b\rho^n \tag{8.7}$$

where, a and b are coefficients that can be determined by assuming that the bulk modulus is a linear function of pressure. The values of coefficients a and b are based on the reference state values of pressure, density, and bulk modulus.

The simplified form of the Tait equation can be written as:

$$\left(\frac{\rho}{\rho_0}\right)^n = \frac{K}{K_0} \tag{8.8}$$

where,

$$K = K_0 + n\Delta p \tag{8.9}$$

and

$$\Delta p = p - p_0 \tag{8.10}$$

where,

p_0 = Reference liquid pressure (Absolute)

ρ_0 = Reference liquid density (Density at reference pressure, p_0)

K_0 = Reference bulk modulus (Bulk modulus at reference pressure, p_0)

n = Density exponent

p = Liquid pressure (Absolute)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ρ = Liquid density at pressure, p

K = Bulk modulus at pressure, p

The density ratio is limited to the range:

$$\rho_{r,min} < \frac{\rho}{\rho_0} < \rho_{r,max} \tag{8.11}$$

where,

$\rho_{r,min}$= Minimum density ratio limit, when $p<p_0$

$\rho_{r,max}$= Maximum density ratio limit, when $p>p_0$

The speed of sound, c, is calculated as:

$$c=\sqrt{\frac{K}{\rho}} \tag{8.12}$$

Note:

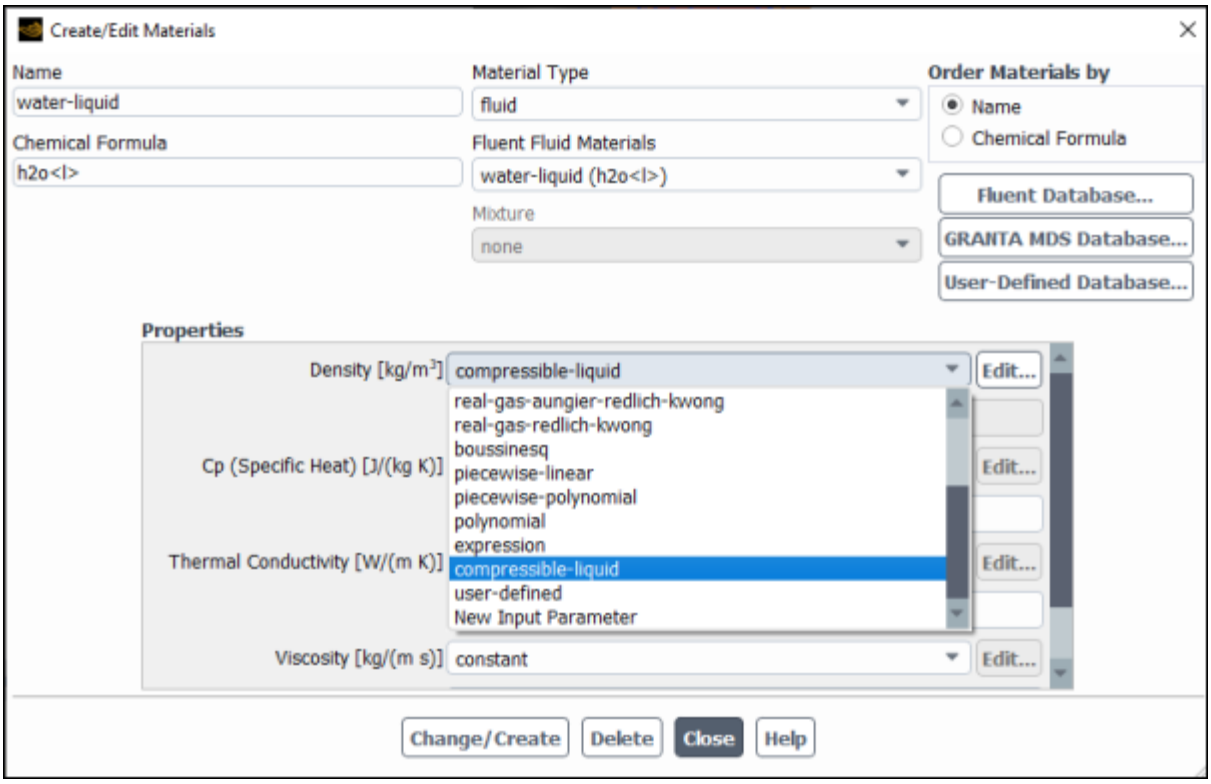
You can postprocess the speed of sound for compressible liquid materials. For multiphase models, sound-speed postprocessing is available at mixture and/or phase level, depending on the model selected. Sound-speed postprocessing is also enabled if you describe the speed of sound for a user-defined density.

8.3.4.1. Compressible Liquid Inputs

Select the `compressible-liquid` under **Density** in the **Create/Edit Materials** panel as seen in Figure 8.17: Compressible Liquid Materials Setting (p. 1628).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 8.17: Compressible Liquid Materials Setting

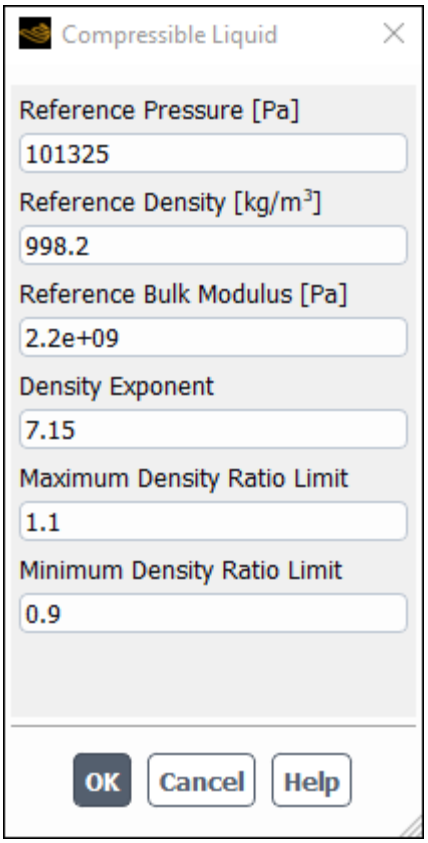


In the **Compressible Liquid** dialog box you need to specify values for the following settings:

- Reference Pressure
- Reference Density
- Reference Bulk Modulus
- Density Exponent
- Maximum Density Ratio Limit
- Minimum Density Ratio Limit

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 8.18: Compressible Liquid Density Settings Panel



Note:

- Ansys Fluent automatically fills user input values for certain materials based on open literature. If data is not available Fluent will fill in values of zero. Therefore, you must verify that suitable values are entered for your application.
- When Fluent automatically fills in values, the default value of the density exponent is set close to the value for water-liquid.
- The value for the density exponent is generally found by calibrating against data for density, pressure, bulk modulus, and speed of sound. If the value is unknown, it is recommended that you use a value of 1, which corresponds to a linear relationship between density and pressure.

You can set the compressible-liquid density method using the following text command:

`define` → `materials` → `change-create`

In the text command interface, set the `change Density?` option to `yes`, and set `new method [constant]` to `compressible-liquid` as shown below:

```
/define/materials> change-create

material-name> water-liquid
material name [water-liquid]
water-liquid is a fluid
```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
change Density? [no] yes

Density
methods: (constant ideal-gas incompressible-ideal-gas real-gas-soave-redlich-kwong real-gas-peng-robinson
real-gas-aungier-redlich-kwong real-gas-redlich-kwong boussinesq piecewise-linear piecewise-polynomial
polynomial compressible-liquid user-defined)
new method [constant] compressible-liquid
Reference Pressure (pascal) [101325]
Reference Density (kg/m3) [998.2000000000001]
Reference Bulk Modulus (pascal) [2200000000]
Density Exponent [7.15]
Maximum Density Ratio Limit [1.1]
Minimum Density Ratio Limit [0.9]
```

8.3.4.2. Compressible Liquid Density Method Availability

The compressible liquid density method is available with fluid materials or with components of a mixture material having the density method set to `volume-weighted-mixing-law` for both single and multiphase cases.

This method is not available with the density-based solver.

8.3.5. Density as a Profile Function of Temperature

If you are modeling a problem that involves heat transfer, you can define the density as a function of temperature. Three types of functions are available:

- piecewise-linear:

$$\rho(T)=\rho_n+\frac{\rho_{n+1}-\rho_n}{T_{n+1}-T_n}(T-T_n) \tag{8.13}$$

- piecewise-polynomial:

$$\begin{aligned}&\text{for } T_{min,1}\le T<T_{max,1}: \rho(T)=A_1+A_2T+A_3T^2+\ldots\\&\text{for } T_{min,2}\le T<T_{max,2}: \rho(T)=B_1+B_2T+B_3T^2+\ldots\end{aligned} \tag{8.14}$$

- polynomial:

$$\rho(T)=A_1+A_2T+A_3T^2+\ldots \tag{8.15}$$

For one of the these methods, select **piecewise-linear**, **piecewise-polynomial**, or **polynomial** in the **Density** drop-down list. You can enter the data pairs (T_n,ρ_n), ranges and coefficients, or coefficients that describe these functions using the Create/Edit Materials Dialog Box (p. 4885), as described in Defining Properties Using Temperature-Dependent Functions (p. 1616).

8.3.6. Incompressible Ideal Gas Law

In Ansys Fluent, if you choose to define the density using the ideal gas law for an incompressible flow, the solver will compute the density as

$$\rho=\frac{p_{op}}{\frac{R}{M_w}T} \tag{8.16}$$

where,

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

R = the universal gas constant

M_w = the molecular weight of the gas

p_{op} = the operating pressure

In this form, the density depends only on the operating pressure and not on the local relative pressure field.

8.3.6.1. Density Inputs for the Incompressible Ideal Gas Law

The inputs for the incompressible ideal gas law are as follows:

1. Enable the ideal gas law for an incompressible fluid by choosing **incompressible-ideal-gas** from the drop-down list to the right of **Density** in the Create/Edit Materials Dialog Box (p. 4885).

   Specify the incompressible ideal gas law individually for each material that you want to use it for. See Composition-Dependent Density for Multicomponent Mixtures (p. 1632) for information on specifying the incompressible ideal gas law for mixtures.

2. Set the operating pressure by defining the **Operating Pressure** in the Operating Conditions Dialog Box (p. 4978).

   Setup → **Cell Zone Conditions** → **Operating Conditions...**

   Important:

   By default, operating pressure is set to 101325 Pa. The input of the operating pressure is of great importance when you are computing density with the ideal gas law. See Operating Pressure (p. 1677) for recommendations on setting appropriate values for the operating pressure.

3. Set the molecular weight of the homogeneous or single-component fluid (if no chemical species transport equations are to be solved), or the molecular weights of each fluid material (species) in a multicomponent mixture. For each fluid material, enter the value of the **Molecular Weight** in the Create/Edit Materials Dialog Box (p. 4885).

8.3.7. Ideal Gas Law for Compressible Flows

For compressible flows, the gas law is as following:

$$\rho = \frac{p_{op} + p}{\frac{R}{M_w} T} \tag{8.17}$$

where,

p = the local relative (or gauge) pressure predicted by Ansys Fluent

p_{op} = the operating pressure

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.3.7.1. Density Inputs for the Ideal Gas Law for Compressible Flows

The inputs for the ideal gas law are as follows:

1. Enable the ideal gas law for a compressible fluid by choosing **ideal-gas** from the drop-down list to the right of **Density** in the Create/Edit Materials Dialog Box (p. 4885).

   Specify the ideal gas law individually for each material that you want to use it for. See Composition-Dependent Density for Multicomponent Mixtures (p. 1632) for information on specifying the ideal gas law for mixtures.

2. Set the operating pressure by defining the **Operating Pressure** in the Operating Conditions Dialog Box (p. 4978).

   Setup → **Cell Zone Conditions** → **Operating Conditions...**

   Important:

   The input of the operating pressure is of great importance when you are computing density with the ideal gas law. Equation 8.17 (p. 1631) notes that the operating pressure is added to the relative pressure field computed by the solver, yielding the absolute static pressure. See Operating Pressure (p. 1677) for recommendations on setting appropriate values for the operating pressure. By default, operating pressure is set to 101325 Pa.

3. Set the molecular weight of the homogeneous or single-component fluid (if no chemical species transport equations are to be solved), or the molecular weights of each fluid material (species) in a multicomponent mixture. For each fluid material, enter the value of the **Molecular Weight** in the Create/Edit Materials Dialog Box (p. 4885).

8.3.8. Composition-Dependent Density for Multicomponent Mixtures

If you are solving species transport equations, set properties for the mixture material and for the constituent fluids (species), as described in detail in Defining Properties for the Mixture and Its Constituent Species (p. 2374). To define a composition-dependent density for a mixture, do the following:

1. Select the density method:

   - For non-ideal-gas mixtures, you can select the **volume-weighted-mixing-law** method for the mixture material in the drop-down list to the right of **Density** in the Create/Edit Materials Dialog Box (p. 4885).

   - For compressible multicomponent mixtures where the individual species may be real fluids (using Real Gas Property (RGP) tables) and/or ideal gas, you can select the **compressible-volume-weighted** method for the mixture material in the drop-down list to the right of **Density** in the Create/Edit Materials Dialog Box (p. 4885). This option is recommended when the mixture contains

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

at least one RGP species. However, when all species are ideal gas, then you should use the **ideal-gas** method.

Note:

Prior to using this method, you need to create one or several materials for the species using RGP tables as described in Using Real Gas Property (RGP) Table Files (p. 1734).

The **compressible-volume-weighted** density method uses the same formula for density as the **volume-weighted-mixing-law**. The difference is the thermodynamic property algorithms used by the solver. That is, the compressible-volume-weighted method is intended for compressible species, including real gas, whereas the volume-weighted-mixing-law method is intended for incompressible species.

- If you are modeling compressible flow, you can select **ideal-gas** for the mixture material in the drop-down list to the right of **Density** in the Create/Edit Materials Dialog Box (p. 4885).
- If you are modeling incompressible flow using the ideal gas law, you can select **incompressible-ideal-gas** for the mixture material in the **Density** drop-down list in the Create/Edit Materials Dialog Box (p. 4885).
- If you have a user-defined function that you want to use to model the density, you can choose either the **user-defined** method or the **user-defined-mixing-law** method for the mixture material in the drop-down list.

The only difference between the **user-defined-mixing-law** and the **user-defined** option for specifying density, viscosity and thermal conductivity of mixture materials, is that with the **user-defined-mixing-law** option, the individual properties of the species materials can also be specified. (Note that only the constant, the polynomial methods and the user-defined methods are available.)

2. Define the remaining property methods for the mixture material as appropriate for your case.
3. Click **Change/Create**.
4. If you have selected **volume-weighted-mixing-law**, define the density for each of the fluid materials that make up the mixture. You may define constant or compressible-liquid or (if applicable) temperature-dependent densities for the individual species.
5. If you have selected **compressible-volume-weighted**, double-check settings for each of the species materials.
6. If you selected **user-defined-mixing-law**, define the density for each of the fluid materials that make up the mixture. You may define constant, or (if applicable) temperature-dependent densities, or user-defined densities for the individual species. More information on defining properties with user-defined functions can be found in the Fluent Customization Manual.

If you are modeling a non-ideal-gas mixture, Ansys Fluent will compute the mixture density as

$$\rho = \frac{1}{\sum_i \frac{Y_i}{\rho_i}} \tag{8.18}$$

where Y_i is the mass fraction and ρ_i is the density of species i.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For compressible flows, the gas law has the following form:

$$\rho = \frac{p_{op} + p}{RT \sum_i \frac{Y_i}{M_{w,i}}} \tag{8.19}$$

where,

p = the local relative (or gauge) pressure predicted by Ansys Fluent

R = the universal gas constant

Y_i = the mass fraction of species i

$M_{w,i}$ = the molecular weight of species i

p_{op} = the operating pressure

In Ansys Fluent, if you choose to define the density using the ideal gas law for an incompressible flow, the solver will compute the density as

$$\rho = \frac{p_{op}}{RT \sum_i \frac{Y_i}{M_{w,i}}} \tag{8.20}$$

where,

R = the universal gas constant

Y_i = the mass fraction of species i

$M_{w,i}$ = the molecular weight of species i

p_{op} = the operating pressure

8.4. Viscosity

Ansys Fluent provides several options for definition of the fluid viscosity:

- constant viscosity
- temperature-dependent and/or composition-dependent viscosity
- kinetic theory
- non-Newtonian viscosity
- user-defined function

Each of these input options and the governing physical models are detailed in this section. (User-defined functions are described in the Fluent Customization Manual). In all cases, define the **Viscosity** in the Create/Edit Materials Dialog Box (p. 4885).

Setup → Materials

Viscosities are input as dynamic viscosity (μ) in units of kg/m-s in SI units or lb_m/ft-s in British units. Ansys Fluent does not ask for input of the kinematic viscosity (ν).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For additional information, see the following sections:

8.4.1. Input of Constant Viscosity
8.4.2. Viscosity as a Function of Temperature
8.4.3. Defining the Viscosity Using Kinetic Theory
8.4.4. Defining Viscosity Using Gupta Curve Fits
8.4.5. Composition-Dependent Viscosity for Multicomponent Mixtures
8.4.6. Viscosity for Non-Newtonian Fluids

8.4.1. Input of Constant Viscosity

If you want to define the viscosity of your fluid as a constant, select **constant** in the **Viscosity** drop-down list in the Create/Edit Materials Dialog Box (p. 4885), and enter the value of viscosity for the fluid.

For the default fluid (air), the viscosity is 1.7894×10^{-5} kg/m-s.

8.4.2. Viscosity as a Function of Temperature

If you are modeling a problem that involves heat transfer, you can define the viscosity as a function of temperature. Five types of functions are available.

- piecewise-linear:

$$\mu(T)=\mu_n+\frac{\mu_{n+1}-\mu_n}{T_{n+1}-T_n}(T-T_n) \tag{8.21}$$

- piecewise-polynomial:

$$\begin{aligned}&\text{for } T_{min,1}\le T<T_{max,1}: \mu(T)=A_1+A_2T+A_3T^2+\ldots\\&\text{for } T_{min,2}\le T<T_{max,2}: \mu(T)=B_1+B_2T+B_3T^2+\ldots\end{aligned} \tag{8.22}$$

- polynomial:

$$\mu(T)=A_1+A_2T+A_3T^2+\ldots \tag{8.23}$$

- Sutherland's law
- power law

Important:

The power law described here is different from the non-Newtonian power law described in Viscosity for Non-Newtonian Fluids (p. 1639).

For one of the first three, select **piecewise-linear**, **piecewise-polynomial**, **polynomial** in the **Viscosity** drop-down list and then enter the data pairs (T_n,μ_n), ranges and coefficients, or coefficients that describe these functions Defining Properties Using Temperature-Dependent Functions (p. 1616). For Sutherland's law or the power law, choose **sutherland** or **power-law** respectively in the drop-down list and enter the parameters.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.4.2.1. Sutherland Viscosity Law

Sutherland's viscosity law resulted from a kinetic theory by Sutherland (1893) using an idealized intermolecular-force potential. The formula is specified using two or three coefficients.

Sutherland's law with two coefficients has the form

$$\mu=\frac{C_1 T^{3/2}}{T+C_2} \tag{8.24}$$

where,

μ = the viscosity in kg/m-s

T = the static temperature in K

C_1 and C_2 = the coefficients

For air at moderate temperatures and pressures, $C_1=1.458\times10^{-6}$ kg/m-s-$K^{1/2}$, and $C_2=110.4$ K.

Sutherland's law with three coefficients has the form

$$\mu=\mu_0\left(\frac{T}{T_0}\right)^{3/2}\frac{T_0+S}{T+S} \tag{8.25}$$

where,

μ = the viscosity in kg/m-s

T = the static temperature in K

μ_0 = reference value in kg/m-s

T_0 = reference temperature in K

S = an effective temperature in K (Sutherland constant)

For air at moderate temperatures and pressures, $\mu_0=1.716\times10^{-5}$ kg/m-s, T_0 = 273.11 K, and S = 110.56 K.

8.4.2.1.1. Inputs for Sutherland's Law

To use Sutherland's law, choose **sutherland** in the drop-down list to the right of **Viscosity**. The Sutherland Law Dialog Box (p. 4916) will open, and you can enter the coefficients as follows:

1. Select the **Two Coefficient Method** or the **Three Coefficient Method**.

   Important:

   Use SI units if you choose the two-coefficient method.

2. For the **Two Coefficient Method**, set **C1** and **C2**. For the **Three Coefficient Method**, set the **Reference Viscosity** μ_0, the **Reference Temperature** T_0, and the **Effective Temperature** S.

3. To visualize your profile, ensure the correct **Primary Independent Variable** is selected from the drop-down list, and enter a value for the **Min** and **Max**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.4.2.2. Power-Law Viscosity Law

Another common approximation for the viscosity of dilute gases is the power-law form. For dilute gases at moderate temperatures, this form is considered to be slightly less accurate than Sutherland's law.

A power-law viscosity law with two coefficients has the form

$$\mu = BT^n \tag{8.26}$$

where,

μ = the viscosity in kg/m-s

T = the static temperature in K

B = a dimensional coefficient

For air at moderate temperatures and pressures, $B = 4.093 \times 10^{-7}$, and $n = 2/3$.

A power-law viscosity law with three coefficients has the form

$$\mu = \mu_0 \left(\frac{T}{T_0} \right)^n \tag{8.27}$$

where,

μ = the viscosity in kg/m-s

T = the static temperature in K

T_0 = a reference value in K

μ_0 = a reference value in kg/m-s

For air at moderate temperatures and pressures, $\mu_0 = 1.716 \times 10^{-5}$ kg/m-s, $T_0 = 273$ K, and $n = 2/3$.

Important:

The non-Newtonian power law for viscosity is described in Viscosity for Non-Newtonian Fluids (p. 1639).

8.4.2.2.1. Inputs for the Power Law

To use the power law, choose **power-law** in the drop-down list to the right of **Viscosity**. The Power Law Dialog Box (p. 4917) will open, and you can enter the coefficients as follows:

1. Select the **Two Coefficient Method** or the **Three Coefficient Method**.

Important:

Note that you must use SI units if you choose the two-coefficient method.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. For the **Two Coefficient Method**, set **B** and the **Temperature Exponent** n. For the **Three Coefficient Method**, set the **Reference Viscosity** μ_0, the **Reference Temperature** T_0, and the **Temperature Exponent** n.

3. To visualize your profile, ensure the correct **Primary Independent Variable** is selected from the drop-down list, and enter a value for the **Min** and **Max**.

8.4.3. Defining the Viscosity Using Kinetic Theory

If you are using the ideal gas law (as described in Density (p. 1624)), you have the option to define the fluid viscosity using kinetic theory as

$$\mu = 2.67 \times 10^{-6} \frac{\sqrt{M_w T}}{\sigma^2 \Omega_\mu} \tag{8.28}$$

where,

μ is in units of kg/m-s,

T is in units of Kelvin

σ is in units of Angstroms

$\Omega_\mu = \Omega_\mu\left(T^*\right)$

M_w is the molecular weight

where

$$T^* = \frac{T}{\left(\varepsilon / k_B\right)} \tag{8.29}$$

The Lennard-Jones parameters, σ and ε / k_B, are inputs to the kinetic theory calculation that you supply by selecting **kinetic-theory** from the drop-down list to the right of **Viscosity** in the Create/Edit Materials Dialog Box (p. 4885). The solver will use these kinetic theory inputs in Equation 8.28 (p. 1638) to compute the fluid viscosity. See Kinetic Theory Parameters (p. 1676) for details about these inputs.

8.4.4. Defining Viscosity Using Gupta Curve Fits

When simulating hypersonic flows using the default one-temperature approach, Fluent offers Gupta curve fits that are applicable for air mixtures with temperatures up to 30 000 K. For details, see Modeling Transport Properties Using Gupta Curve Fits (p. ?).

8.4.5. Composition-Dependent Viscosity for Multicomponent Mixtures

If you are modeling a flow that includes more than one chemical species (multicomponent flow), you have the option to define a composition-dependent viscosity. (Note that you can also define the viscosity of the mixture as a constant value or a function of temperature.)

To define a composition-dependent viscosity for a mixture, follow these steps:

1. For the mixture material, choose **mass-weighted-mixing-law** or, if you are using the ideal gas law for density, **ideal-gas-mixing-law** in the drop-down list to the right of **Viscosity**. If you have a user-defined function that you want to use to model the viscosity, you can choose either the

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

user-defined method or the **user-defined-mixing-law** method for the mixture material in the drop-down list.

2. Click **Change/Create**.

3. Define the viscosity for each of the fluid materials that make up the mixture. You may define constant or (if applicable) temperature-dependent viscosities for the individual species. You may also use kinetic theory for the individual viscosities, or specify a non-Newtonian viscosity, if applicable.

4. If you selected **user-defined-mixing-law**, define the viscosity for each of the fluid materials that make up the mixture. You may define constant, or (if applicable) temperature-dependent viscosities, or user-defined viscosities for the individual species. More information on defining properties with user-defined functions can be found in the Fluent Customization Manual.

   The only difference between the **user-defined-mixing-law** and the **user-defined** option for specifying density, viscosity and thermal conductivity of mixture materials, is that with the **user-defined-mixing-law** option, the individual properties of the species materials can also be specified. (Note that only the constant, the polynomial methods and the user-defined methods are available.)

If you are using the ideal gas law, the solver will compute the mixture viscosity based on kinetic theory as

$$\mu = \sum_i \frac{X_i \mu_i}{\sum_j X_j \phi_{ij}} \tag{8.30}$$

where

$$\phi_{ij} = \frac{\left[1 + \left(\frac{\mu_i}{\mu_j}\right)^{1/2} \left(\frac{M_{w,j}}{M_{w,i}}\right)^{1/4}\right]^2}{\left[8\left(1 + \frac{M_{w,i}}{M_{w,j}}\right)\right]^{1/2}} \tag{8.31}$$

and X_i is the mole fraction of species i.

For non-ideal gas mixtures, the mixture viscosity is computed based on a simple mass fraction average of the pure species viscosities:

$$\mu = \sum_i Y_i \mu_i \tag{8.32}$$

8.4.6. Viscosity for Non-Newtonian Fluids

For incompressible Newtonian fluids, the shear stress is proportional to the rate-of-deformation tensor $\overline{\overline{D}}$:

$$\overline{\overline{\tau}} = \mu \overline{\overline{D}} \tag{8.33}$$

where $\overline{\overline{D}}$ is defined by

$$\overline{\overline{D}} = \left(\frac{\partial u_j}{\partial x_i} + \frac{\partial u_i}{\partial x_j}\right) \tag{8.34}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and μ is the viscosity, which is independent of $\overline{\overline{D}}$.

For some non-Newtonian fluids, the shear stress can similarly be written in terms of a non-Newtonian viscosity η:

$$\overline{\overline{\tau}}=\eta\left(\overline{\overline{D}}\right)\overline{\overline{D}} \tag{8.35}$$

In general, η is a function of all three invariants of the rate-of-deformation tensor $\overline{\overline{D}}$. However, in the non-Newtonian models available in Ansys Fluent, η is considered to be a function of the shear rate $\dot{\gamma}$ only. $\dot{\gamma}$ is related to the second invariant of $\overline{\overline{D}}$ and is defined as

$$\dot{\gamma}=\sqrt{\frac{1}{2}\overline{\overline{D}}:\overline{\overline{D}}} \tag{8.36}$$

8.4.6.1. Temperature Dependent Viscosity

If the flow is non-isothermal, then the temperature dependence on the viscosity can be included along with the shear rate dependence. In this case, the total viscosity consists of two parts and is calculated as

$$\mu=\eta\left(\dot{\gamma}\right)H\left(T\right) \tag{8.37}$$

where H(T) is the temperature dependence, known as the Arrhenius law.

$$H\left(T\right)=exp\left[\alpha\left(\frac{1}{T-T_0}-\frac{1}{T_\alpha-T_0}\right)\right] \tag{8.38}$$

where α is the ratio of the activation energy to the thermodynamic constant and T_α is a reference temperature for which H(T) = 1. T_0, which is the temperature shift, is set to 0 by default, and corresponds to the lowest temperature that is thermodynamically acceptable. Therefore T_α and T_0 are absolute temperatures. Temperature dependence is only included when the energy equation is enabled. Set the parameter α to 0 when you want temperature dependence to be ignored, even when the energy equation is solved.

Ansys Fluent provides four options for modeling non-Newtonian flows:

- power law
- Carreau model for pseudo-plastics
- Cross model
- Herschel-Bulkley model for Bingham plastics

Important:

- Note that the models listed above are not available when modeling turbulent flow.
- Note that the non-Newtonian power law described below is different from the power law described in Power-Law Viscosity Law (p. 1637).
- Non-Newtonian model based on single fluid formulation is available for the mixture model and it is recommended that this should be attached to the primary phase.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Appropriate values for the input parameters for these models can be found in the literature (for example, [160] (p. 5722)).

8.4.6.2. Power Law for Non-Newtonian Viscosity

If you choose **non-newtonian-power-law** in the drop-down list to the right of **Viscosity**, non-Newtonian flow will be modeled according to the following power law for the non-Newtonian viscosity:

$$\eta = k\dot{\gamma}^{n-1}H(T) \tag{8.39}$$

where k and n are input parameters. k is a measure of the average viscosity of the fluid (the consistency index); n is a measure of the deviation of the fluid from Newtonian (the power-law index). The value of n determines the class of the fluid:

$n=1 \rightarrow$ Newtonian fluid

$n>1 \rightarrow$ shear-thickening (dilatant fluids)

$n<1 \rightarrow$ shear-thinning (pseudo-plastics)

8.4.6.2.1. Inputs for the Non-Newtonian Power Law

To use the non-Newtonian power law, choose **non-newtonian-power-law** in the drop-down list to the right of **Viscosity**. The Non-Newtonian Power Law Dialog Box (p. 4919) will open, and you can choose between **Shear Rate Dependent** and **Shear Rate and Temperature Dependent**. Enter the **Consistency Index** k, **Power-Law Index** n, **Minimum** and **Maximum Viscosity Limit**, **Reference Temperature** T_{alpha}, and **Activation Energy/R**, α, which is the ratio of the activation energy to the thermodynamic constant.

8.4.6.3. The Carreau Model for Pseudo-Plastics

The power law model described in Equation 8.39 (p. 1641) results in a fluid viscosity that varies with shear rate. For $\dot{\gamma} \rightarrow 0$, $\eta \rightarrow \eta_0$, and for $\dot{\gamma} \rightarrow \infty$, $\eta \rightarrow \eta_\infty$, where η_0 and η_∞ are, respectively, the upper and lower limiting values of the fluid viscosity.

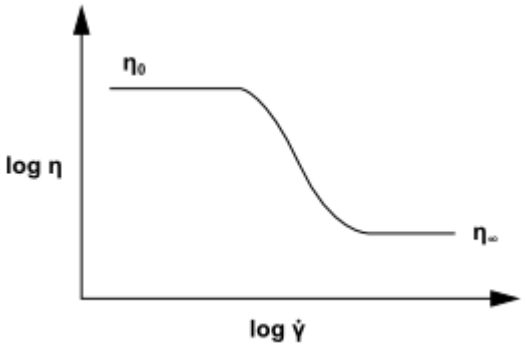
The Carreau model attempts to describe a wide range of fluids by the establishment of a curve-fit to piece together functions for both Newtonian and shear-thinning ($n < 1$) non-Newtonian laws. In the Carreau model, the viscosity is

$$\eta = H(T)\left(\eta_\infty + \left(\eta_0 - \eta_\infty\right)\left[1+\gamma^2\lambda^2\right]^{(n-1)/2}\right) \tag{8.40}$$

and the parameters n, λ, T_α, η_0, and η_∞ are dependent upon the fluid. λ is the time constant, n is the power-law index (as described above for the non-Newtonian power law), η_0 and η_∞ are, respectively, the zero- and infinite-shear viscosities, T_α is the reference temperature, and α is the ratio of the activation energy to thermodynamic constant. Figure 8.19: Variation of Viscosity with Shear Rate According to the Carreau Model (p. 1642) shows how viscosity is limited by η_0 and η_∞ at low and high shear rates.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

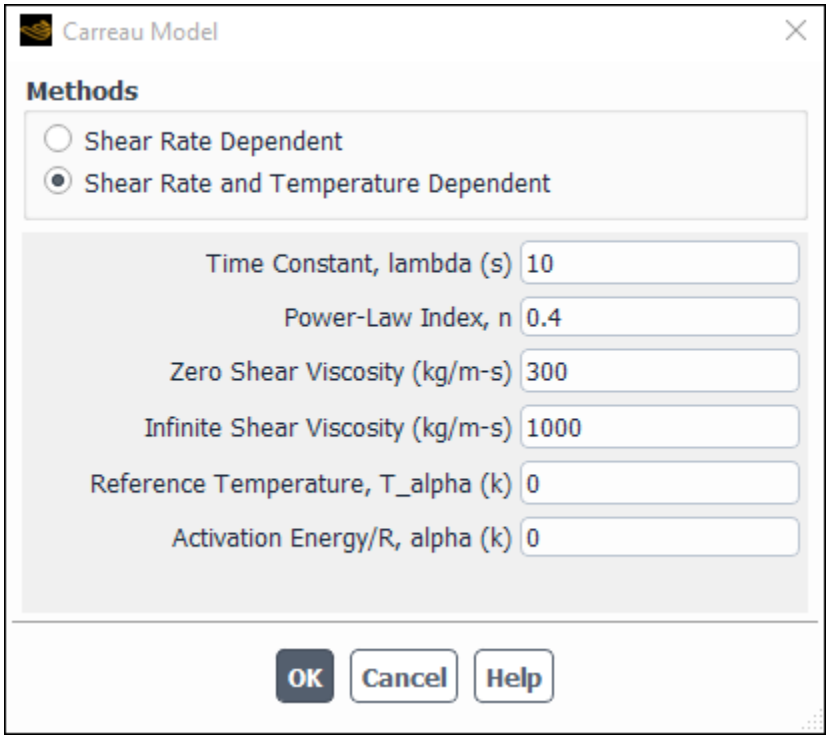
Figure 8.19: Variation of Viscosity with Shear Rate According to the Carreau Model



8.4.6.3.1. Inputs for the Carreau Model

To use the Carreau model, choose **carreau** in the drop-down list to the right of **Viscosity**. The Carreau Model Dialog Box (p. 4920) will open, and you can choose between **Shear Rate Dependent** and **Shear Rate and Temperature Dependent**. Enter the **Time Constant** λ, **Power-Law Index** n, **Reference Temperature** T_α, **Zero Shear Viscosity** η_0, **Infinite Shear Viscosity** η_∞, and **Activation Energy/R** α.

Figure 8.20: The Carreau Model Dialog Box



8.4.6.4. Cross Model

The Cross model for viscosity is:

$$\eta = H(T)\frac{\eta_0}{1+(\lambda\dot{\gamma})^{1-n}} \tag{8.41}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where,

η_0 = zero-shear-rate viscosity

λ = natural time (that is, inverse of the shear rate at which the fluid changes from Newtonian to power-law behavior)

n = power-law index

The Cross model is commonly used to describe the low-shear-rate behavior of the viscosity.

8.4.6.4.1. Inputs for the Cross Model

To use the Cross model, choose **cross** in the drop-down list to the right of **Viscosity**. The Cross Model Dialog Box (p. 4921) will open, and you can choose between **Shear Rate Dependent** and **Shear Rate and Temperature Dependent**. Enter the **Zero Shear Viscosity** η_0, **Time Constant** λ, **Power-Law Index** n, **Reference Temperature** T_α, and **Activation Energy/R**, α, which is the ratio of the activation energy to the thermodynamic constant.

8.4.6.5. Herschel-Bulkley Model for Bingham Plastics

The power law model described above is valid for fluids for which the shear stress is zero when the strain rate is zero. Bingham plastics are characterized by a nonzero shear stress when the strain rate is zero.

$$\bar{\bar{\tau}}=\bar{\bar{\tau}}_0+\eta\bar{\bar{D}} \tag{8.42}$$

where τ_0 is the yield stress:

- For $\tau<\tau_0$, the material remains rigid.
- For $\tau>\tau_0$, the material flows as a power-law fluid.

The Herschel-Bulkley model combines the effects of Bingham and power-law behavior in a fluid. For low strain rates ($\dot{\gamma}<\tau_0/\mu_0$), the "rigid" material acts like a very viscous fluid with viscosity μ_0. As the strain rate increases and the yield stress threshold, τ_0, is passed, the fluid behavior is described by a power law.

For $\dot{\gamma}>\dot{\gamma}_c$

$$\eta=\frac{\tau_0}{\dot{\gamma}}+k\dot{\gamma}^{n-1} \tag{8.43}$$

For $\dot{\gamma}<\dot{\gamma}_c$

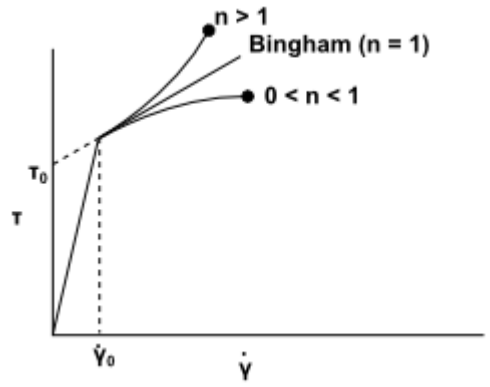
$$\eta=\frac{\tau_0\left(2-\frac{\dot{\gamma}}{\dot{\gamma}_c}\right)}{\dot{\gamma}_c}+k\left(\dot{\gamma}_c\right)^{n-1}\left[\left(2-n\right)+\left(n-1\right)\frac{\dot{\gamma}}{\dot{\gamma}_c}\right] \tag{8.44}$$

where k is the consistency index, and n is the power-law index.

Figure 8.21: Variation of Shear Stress with Shear Rate According to the Herschel-Bulkley Model (p. 1644) shows how shear stress (τ) varies with shear rate ($\dot{\gamma}$) for the Herschel-Bulkley model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 8.21: Variation of Shear Stress with Shear Rate According to the Herschel-Bulkley Model



If you choose the Herschel-Bulkley model for Bingham plastics, Equation 8.43 (p. 1643) will be used to determine the fluid viscosity.

The Herschel-Bulkley model is commonly used to describe materials such as concrete, mud, dough, and toothpaste, for which a constant viscosity after a critical shear stress is a reasonable assumption. In addition to the transition behavior between a flow and no-flow regime, the Herschel-Bulkley model can also exhibit a shear-thinning or shear-thickening behavior depending on the value of n.

8.4.6.5.1. Inputs for the Herschel-Bulkley Model

To use the Herschel-Bulkley model, choose **herschel-bulkley** in the drop-down list to the right of **Viscosity**. The Herschel-Bulkley Dialog Box (p. 4922) will open, and you can choose between **Shear Rate Dependent** and **Shear Rate and Temperature Dependent**. Enter the **Consistency Index** k, **Power-Law Index** n, **Yield Stress Threshold** τ_0, **Critical Shear Rate** $\dot{\gamma}_c$, **Reference Temperature** T_α, and the ratio of the activation energy to thermodynamic constant α, **Activation Energy/R**.

8.5. Thermal Conductivity

The thermal conductivity must be defined when heat transfer is active. You must define thermal conductivity when you are modeling energy and viscous flow.

Ansys Fluent provides several options for definition of the thermal conductivity:

- constant thermal conductivity
- temperature- and/or composition-dependent thermal conductivity
- kinetic theory
- anisotropic (anisotropic, biaxial, orthotropic, cylindrical orthotropic, principal axes and principal values, user-defined anisotropic) (for solid materials only)
- user-defined

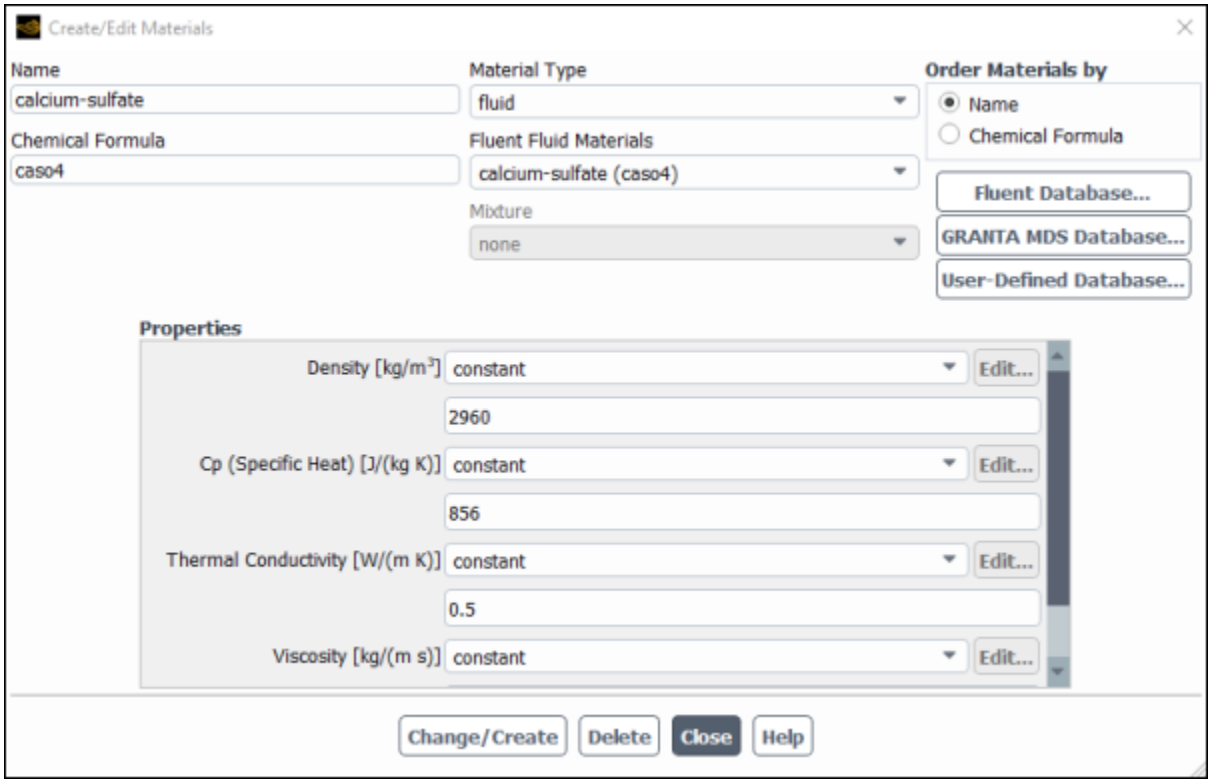
Each of these input options and the governing physical models are detailed in this section. User-defined functions (UDFs) are described in the Fluent Customization Manual.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In all cases, you will define the **Thermal Conductivity** in the Create/Edit Materials Dialog Box (p. 4885) (Figure 8.22: The Create/Edit Materials Dialog Box (p. 1645)).

Setup → Materials → Create/Edit...

Figure 8.22: The Create/Edit Materials Dialog Box



Thermal conductivity is defined in units of W/m-K in SI units or BTU/hr-ft-°R in British units.

For additional information, see the following sections:

8.5.1. Constant Thermal Conductivity
8.5.2. Thermal Conductivity as a Function of Temperature
8.5.3. Thermal Conductivity Using Kinetic Theory
8.5.4. Defining Thermal Conductivity Using Gupta Curve Fits
8.5.5. Composition-Dependent Thermal Conductivity for Multicomponent Mixtures
8.5.6. Anisotropic Thermal Conductivity for Solids

8.5.1. Constant Thermal Conductivity

If you want to define the thermal conductivity as a constant, check that **constant** is selected in the drop-down list to the right of **Thermal Conductivity** in the Create/Edit Materials Dialog Box (p. 4885) (Figure 8.22: The Create/Edit Materials Dialog Box (p. 1645)), and enter the value of thermal conductivity for the material.

For the default fluid (air), the thermal conductivity is 0.0242 W/m-K.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.5.2. Thermal Conductivity as a Function of Temperature

You can also choose to define the thermal conductivity as a function of temperature. Three types of functions are available:

- piecewise-linear:

$$k(T) = k_n + \frac{k_{n+1} - k_n}{T_{n+1} - T_n}(T - T_n) \tag{8.45}$$

- piecewise-polynomial:

$$\begin{aligned} &\text{for } T_{min,1} \le T < T_{max,1}: k(T) = A_1 + A_2 T + A_3 T^2 + \ldots \\ &\text{for } T_{min,2} \le T < T_{max,2}: k(T) = B_1 + B_2 T + B_3 T^2 + \ldots \end{aligned} \tag{8.46}$$

- polynomial:

$$k(T) = A_1 + A_2 T + A_3 T^2 + \ldots \tag{8.47}$$

You can input the data pairs (T_n, k_n), ranges and coefficients A_i and B_i, or coefficients A_i that describe these functions using the Create/Edit Materials Dialog Box (p. 4885), as described in Defining Properties Using Temperature-Dependent Functions (p. 1616).

8.5.3. Thermal Conductivity Using Kinetic Theory

If you are using the ideal gas law (as described in Density (p. 1624)), you have the option to define the thermal conductivity using kinetic theory as

$$k = \frac{15}{4}\frac{R}{M_w}\mu\left[\frac{4}{15}\frac{c_p M_w}{R} + \frac{1}{3}\right] \tag{8.48}$$

where R is the universal gas constant, M_w is the molecular weight, μ is the material's specified or computed viscosity, and c_p is the material's specified or computed specific heat capacity.

To enable the use of this equation for calculating thermal conductivity, select **kinetic-theory** from the drop-down list to the right of **Thermal Conductivity** in the Create/Edit Materials Dialog Box (p. 4885). The solver will use Equation 8.48 (p. 1646) to compute the thermal conductivity.

8.5.4. Defining Thermal Conductivity Using Gupta Curve Fits

When simulating hypersonic flows using the default one-temperature approach, Fluent offers Gupta curve fits that are applicable for air mixtures with temperatures up to 30 000 K. For details, see Modeling Transport Properties Using Gupta Curve Fits (p. ?).

8.5.5. Composition-Dependent Thermal Conductivity for Multicomponent Mixtures

If you are modeling a flow that includes more than one chemical species (multicomponent flow), you have the option to define a composition-dependent thermal conductivity. (Note that you can also define the thermal conductivity of the mixture as a constant value or a function of temperature, or using kinetic theory.)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To define a composition-dependent thermal conductivity for a mixture, follow these steps:

1. For the mixture material, choose **mass-weighted-mixing-law** or, if you are using the ideal gas law, **ideal-gas-mixing-law** in the drop-down list to the right of **Thermal Conductivity**. If you have a user-defined function that you want to use to model the thermal conductivity, you can choose either the **user-defined** method or the **user-defined-mixing-law** method for the mixture material in the drop-down list.

   The only difference between the **user-defined-mixing-law** and the **user-defined** option for specifying density, viscosity and thermal conductivity of mixture materials, is that with the **user-defined-mixing-law** option, the individual properties of the species materials can also be specified. Note that only the constant, the polynomial methods and the user-defined methods are available.

   Important:

   If you use **ideal-gas-mixing-law** for the thermal conductivity of a mixture, you must use **ideal-gas-mixing-law** or **mass-weighted-mixing-law** for viscosity, because these two viscosity specification methods are the only ones that allow specification of the component viscosities, which are used in the ideal gas law for thermal conductivity (Equation 8.49 (p. 1647)).

2. Click **Change/Create**.

3. Define the thermal conductivity for each of the fluid materials that make up the mixture. You may define constant or (if applicable) temperature-dependent thermal conductivities for the individual species. You may also use kinetic theory for the individual thermal conductivities, if applicable.

4. If you selected **user-defined-mixing-law**, define the thermal conductivity for each of the fluid materials that make up the mixture. You may define constant, or (if applicable) temperature-dependent thermal conductivities, or user-defined thermal conductivities for the individual species. More information about defining properties with user-defined functions can be found in the Fluent Customization Manual.

If you are using the ideal gas law, the solver will compute the mixture thermal conductivity based on kinetic theory as

$$k = \sum_i \frac{X_i k_i}{\sum_j X_j \phi_{ij}} \tag{8.49}$$

where

$$\phi_{ij} = \frac{\left[1 + \left(\frac{\mu_i}{\mu_j}\right)^{1/2} \left(\frac{M_{w,j}}{M_{w,i}}\right)^{1/4}\right]^2}{\left[8\left(1 + \frac{M_{w,i}}{M_{w,j}}\right)\right]^{1/2}} \tag{8.50}$$

and X_i is the mole fraction of species i.

For non-ideal gases, the mixture thermal conductivity is computed based on a simple mass fraction average of the pure species conductivities:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$k = \sum_i Y_i k_i \tag{8.51}$$

8.5.6. Anisotropic Thermal Conductivity for Solids

The anisotropic conductivity options in Ansys Fluent solves the conduction equation in solid zones and shells with the thermal conductivity specified as a matrix. The heat flux vector is written as

$$q_i = -k_{ij} \frac{\partial T}{\partial x_j} \tag{8.52}$$

The following options are available for defining anisotropic thermal conductivity in Ansys Fluent. These are discussed below.

- anisotropic
- biaxial (available for shells only)
- orthotropic
- cylindrical orthotropic
- principal axes and principal values
- user-defined anisotropic

Note:

When postprocessing solid material properties, the Thermal Conductivity will be displayed as zero if an anisotropic thermal conductivity model is being used for a solid material. However, additional postprocessing variables are available to display the components of the thermal conductivity matrix: **Thermal Conductivity XX | XY | XZ | YX | YY | YZ | ZX | ZY | ZZ**. *When orthotropic thermal conductivity is used, the thermal conductivity matrix is symmetric, and the following variables are not made available to avoid redundancy:* ***Thermal Conductivity YX | ZX | ZY***.

Important:

The anisotropic conductivity options are available only with the pressure-based solver; you cannot use them with the density-based solvers.

For cases that have solid zones that use an anisotropic thermal conductivity option, you can change the following settings: you can select the method used to calculate the heat flux (which can affect the calculation robustness and convergence rate); you can select the method used to calculate the temperature gradient (which can affect the simulation speed); and/or you can set the relaxation for the heat flux coefficient (which can improve convergence in certain cases). For details about changing these settings, see Settings for Anisotropic Solid Zones (p. 2179).

8.5.6.1. Anisotropic Thermal Conductivity

For anisotropic diffusion, the thermal conductivity matrix (Equation 8.52 (p. 1648)) is specified as

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$k_{ij}=k\hat{e}_{ij} \tag{8.53}$$

where k is the conductivity and $\hat{e}_{ij}$ is a matrix (2 × 2 for two dimensions and 3 × 3 for three-dimensional problems). Note that $\hat{e}_{ij}$ can be a non-symmetric matrix.

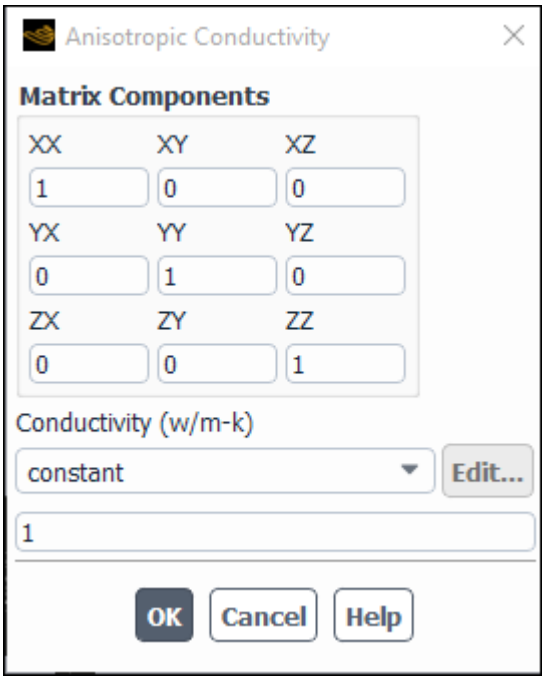
To define anisotropic thermal conductivity for a solid material, select **anisotropic** for **Thermal Conductivity** in the Create/Edit Materials Dialog Box (p. 4885) (Figure 8.22: The Create/Edit Materials Dialog Box (p. 1645)). Note the following:

- If the principal axes of your anisotropic material are not aligned with the global coordinate system of your simulation, it may be easier for you to use one of the following options: if the axes are orthogonal, you can use the **orthotropic** option, as described in Orthotropic Thermal Conductivity (p. 1650); if the axes are not orthogonal, you can use the **principal-axes-values** option, as described in Principal Axes and Principal Values (p. 1654). If you use the **anisotropic** option with unaligned axes, it is your responsibility to transform the thermal conductivity properties into the k_{ij} tensor aligned with the global coordinate system.

- The **anisotropic** option is appropriate when the components of the k_{ij} matrix are constants and do not vary independently; if this is not the case, you can use a UDF to define the matrix and select **user-defined-anisotropic-k**, as described in User-Defined Anisotropic Thermal Conductivity (p. 1656).

Selecting **anisotropic** will open the Anisotropic Conductivity Dialog Box (p. 4928) (Figure 8.23: The Anisotropic Conductivity Dialog Box (p. 1649)).

Figure 8.23: The Anisotropic Conductivity Dialog Box



In the Anisotropic Conductivity Dialog Box (p. 4928), enter the **Matrix Components** of matrix $\hat{e}_{ij}$ and then select the **Conductivity** (k in Equation 8.53 (p. 1649)) to be a **constant**, polynomial function of temperature (**polynomial**, **piecewise-linear**, **piecewise-polynomial**), or **user-defined** function.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

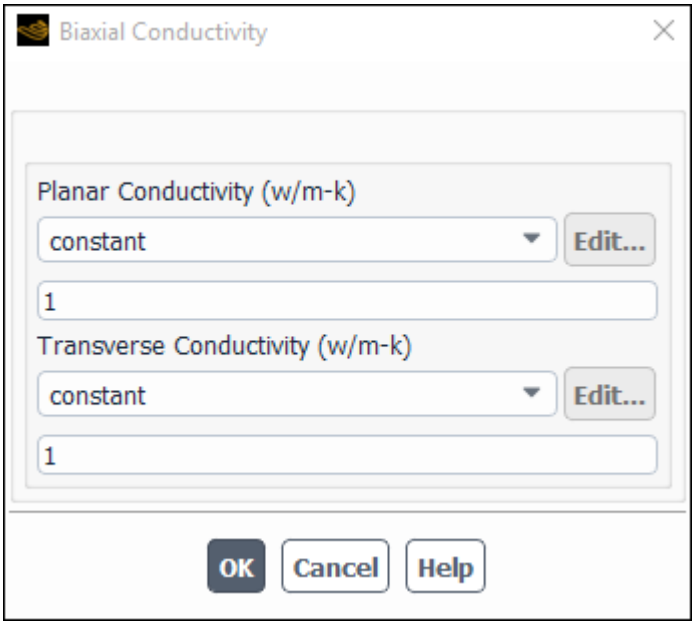
See Constant Thermal Conductivity (p. 1645) and Thermal Conductivity as a Function of Temperature (p. 1646) for details on constants and thermal polynomial functions.

When you select the **user-defined** option, the User-Defined Functions Dialog Box (p. 4915) will open, allowing you to hook a `DEFINE_PROPERTY` UDF *only* if you have previously loaded a compiled UDF library or interpreted the UDF. Otherwise, you will get an error message. Details about user-defined functions can be found in the Fluent Customization Manual.

8.5.6.2. Biaxial Thermal Conductivity

Biaxial thermal conductivity is applicable only for solid materials used for the wall shell conduction model. To define a biaxial thermal conductivity, select **biaxial** in the drop-down list for **Thermal Conductivity** in the Create/Edit Materials Dialog Box (p. 4885). This opens the Biaxial Conductivity Dialog Box (p. 4924) (Figure 8.24: The Biaxial Conductivity Dialog Box (p. 1650)).

Figure 8.24: The Biaxial Conductivity Dialog Box



In the Biaxial Conductivity Dialog Box (p. 4924), both the conductivity normal to the surface of the shell (**Transverse Conductivity**) and the conductivity parallel to the surface of the shell (**Planar Conductivity**) can be defined as **constant**, **polynomial**, **piecewise-linear**, or **piecewise-polynomial**. See Constant Thermal Conductivity (p. 1645) and Thermal Conductivity as a Function of Temperature (p. 1646) for details on these parameters. Note that the **Planar Conductivity** is isotropic. See Figure 16.10: The Conduction Manager Dialog Box (p. 2185).

8.5.6.3. Orthotropic Thermal Conductivity

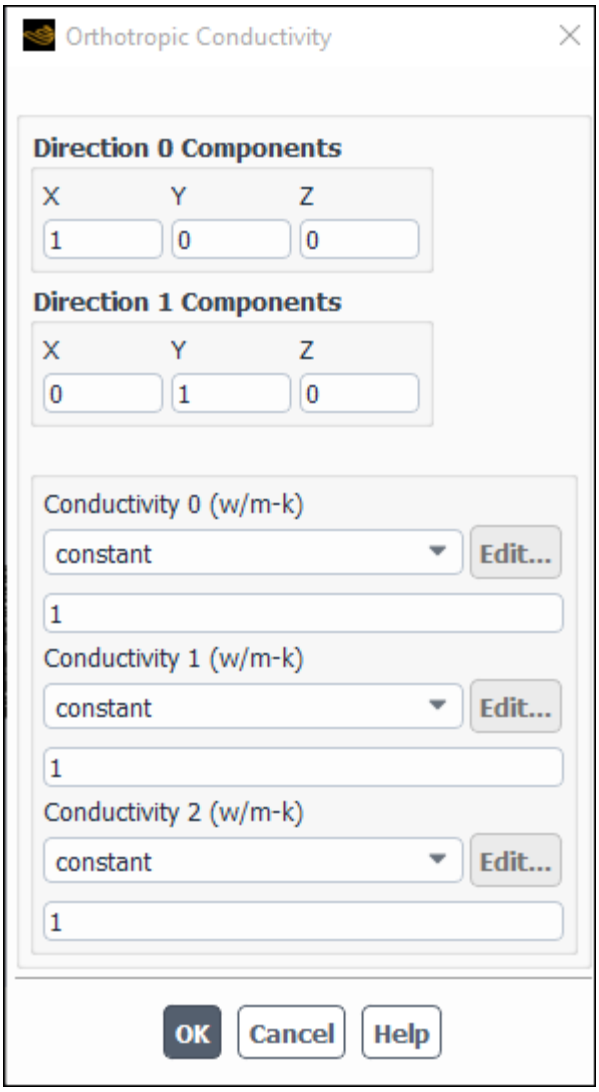
When the orthotropic thermal conductivity is used, the thermal conductivities $\left(k_\xi, k_\eta, k_\zeta\right)$ in the principal directions $\left(\hat{e}_\xi, \hat{e}_\eta, \hat{e}_\zeta\right)$ are specified. The conductivity matrix is then computed as

$$k_{ij} = k_\xi e_{\xi i} e_{\xi j} + k_\eta e_{\eta i} e_{\eta j} + k_\zeta e_{\zeta i} e_{\zeta j} \tag{8.54}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To define an orthotropic thermal conductivity in solids, select **orthotropic** in the drop-down list for **Thermal Conductivity** in the Create/Edit Materials Dialog Box (p. 4885). This opens the Orthotropic Conductivity Dialog Box (p. 4926) (Figure 8.25: The Orthotropic Conductivity Dialog Box (p. 1651)).

Figure 8.25: The Orthotropic Conductivity Dialog Box



Since the directions $\left(\hat{e}_\xi, \hat{e}_\eta, \hat{e}_\zeta\right)$ are mutually orthogonal, only the first two need to be specified for three-dimensional problems. $\hat{e}_\xi$ is defined using **X, Y, Z** under **Direction 0 Components**, and $\hat{e}_\eta$ is defined using **X, Y, Z** under **Direction 1 Components**. You can define **Conductivity 0** $\left(k_\xi\right)$, **Conductivity 1** $\left(k_\eta\right)$, and **Conductivity 2** $\left(k_\zeta\right)$ as **constant**, **polynomial**, **piecewise-linear**, **piecewise-polynomial** functions of temperature, **user-defined**, **expression**, or as a **New Input Parameter**. See Constant Thermal Conductivity (p. 1645) and Thermal Conductivity as a Function of Temperature (p. 1646) for details on constant and temperature profile functions.

When you select the **user-defined** option, the User-Defined Functions Dialog Box (p. 4915) will open, allowing you to hook a `DEFINE_PROPERTY` UDF *only* if you have previously loaded a compiled UDF library or interpreted the UDF. Otherwise, you will get an error message. More information about user-defined functions can be found in the Fluent Customization Manual.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When you select the **expression** option, the Expression Dialog Box (p. 5461) will open, allowing you to define an expression for each of the conductivity component fields. Refer to Fluent Expressions Language (p. 1037) for more information on defining expressions.

When you select the **New Input Parameter** option, the **Parameter Expression** dialog box will open as described in Defining and Viewing Parameters (p. 1309). Note that to specify an orthotropic conductivity as an input parameter though the TUI, you must first use the command `define/parameters/enable-in-TUI? yes`.

Important:

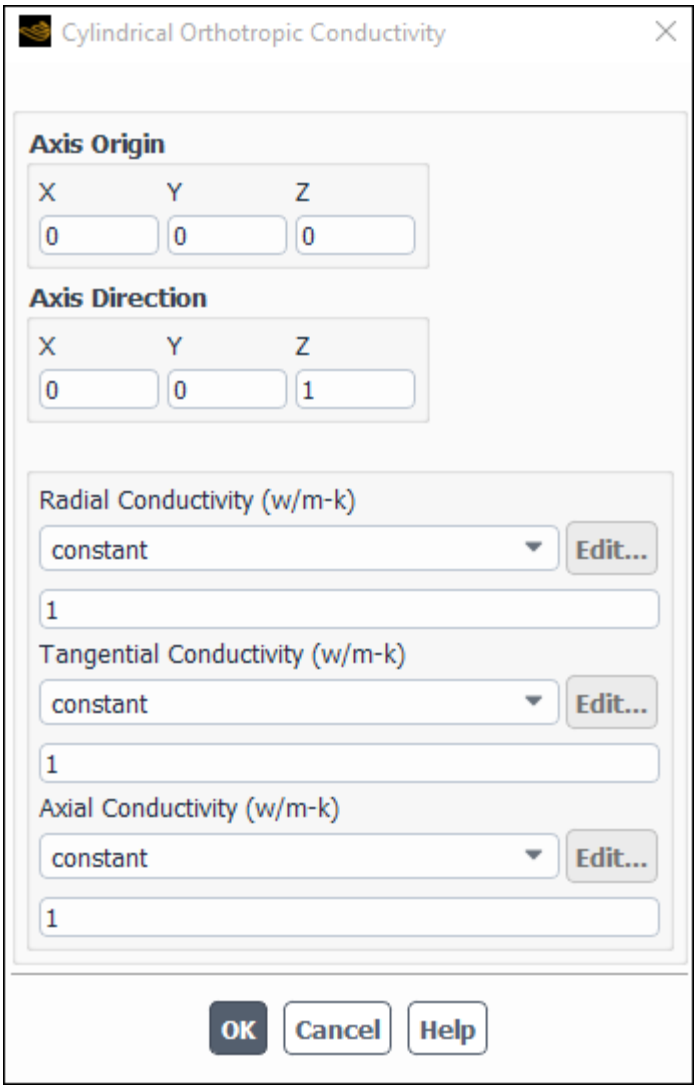
For two-dimensional problems, only the functions $\left(k_\xi, k_\eta\right)$ and the unit vector $\left(\hat{e}_\xi\right)$ need to be specified.

8.5.6.4. Cylindrical Orthotropic Thermal Conductivity

The orthotropic conductivity of solids can be specified in cylindrical coordinates. To define the orthotropic thermal conductivity in cylindrical coordinates, select **cyl-orthotropic** in the drop-down list for **Thermal Conductivity** in the Create/Edit Materials Dialog Box (p. 4885). This opens the Cylindrical Orthotropic Conductivity Dialog Box (p. 4924) (Figure 8.26: The Cylindrical Orthotropic Conductivity Dialog Box (p. 1653)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 8.26: The Cylindrical Orthotropic Conductivity Dialog Box



In three-dimensional cases, the origin and the direction of the cylindrical coordinate system must be specified along with the radial, tangential, and axial direction conductivities. In two-dimensional cases, the origin of the cylindrical coordinate system must be specified along with the radial and tangential direction conductivities. Note that in two-dimensional cases, the direction is always along the +Z axis. Ansys Fluent will automatically compute the anisotropic conductivity matrix at each cell from this input. The calculation is based on the location of the cell in the cylindrical coordinate system specified.

You can define the **Radial Conductivity**, **Tangential Conductivity**, and **Axial Conductivity** as **constant**, **polynomial**, **piecewise-linear**, **piecewise-polynomial**, or as user-defined functions of temperature. See Constant Thermal Conductivity (p. 1645) and Thermal Conductivity as a Function of Temperature (p. 1646) for details on constant and thermal profile functions.

When you select the **user-defined** option, the User-Defined Functions Dialog Box (p. 4915) will open, allowing you to hook a `DEFINE_PROPERTY` UDF *only* if you have previously loaded a compiled

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UDF library or interpreted the UDF. Otherwise, you will get an error message. More information about user-defined functions can be found in the Fluent Customization Manual.

Important:

For conductivity calculations near the wall, the cell next to the wall is chosen for computing the conductivity matrix instead of the wall itself.

8.5.6.5. Principal Axes and Principal Values

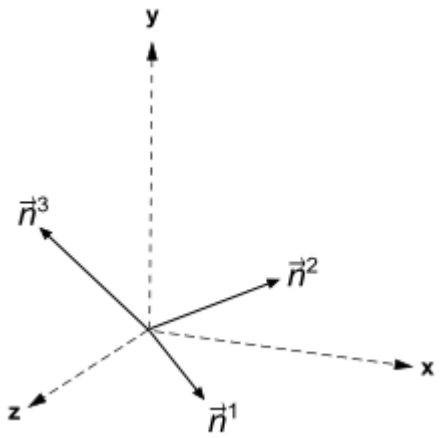
When the principal axes of your anisotropic material are not aligned with the global coordinate system of the simulation, you can define the thermal conductivity using the components of these axes ($\vec{n}^1$, $\vec{n}^2$, and for 3D problems, $\vec{n}^3$) and the associated principle values (K_1, K_2, and for 3D problems, K_3). The axes take the following form:

$$\begin{aligned}\vec{n}^1 &= n_1^1\hat{i} + n_2^1\hat{j} + n_3^1\hat{k} \\ \vec{n}^2 &= n_1^2\hat{i} + n_2^2\hat{j} + n_3^2\hat{k} \\ \vec{n}^3 &= n_1^3\hat{i} + n_2^3\hat{j} + n_3^3\hat{k}\end{aligned} \tag{8.55}$$

Note that while the axes must be linearly independent vectors, they do not have to be unit vectors or orthogonal.

An example of principal axes that are not aligned with the global coordinate system is shown in Figure 8.27: Unaligned Principal Axes (p. 1654).

Figure 8.27: Unaligned Principal Axes



Representing the thermal conductivity matrix (k_{ij} in Equation 8.52 (p. 1648)) in matrix notation ($[\mathbf{k}]$), the following equation is used by Ansys Fluent to transform the principal axes and principal values into the correct tensor form:

$$[\mathbf{k}] = [\mathbf{N}][\mathbf{D}][\mathbf{N}]^{-1} \tag{8.56}$$

The matrices in the previous equation are defined as follows:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

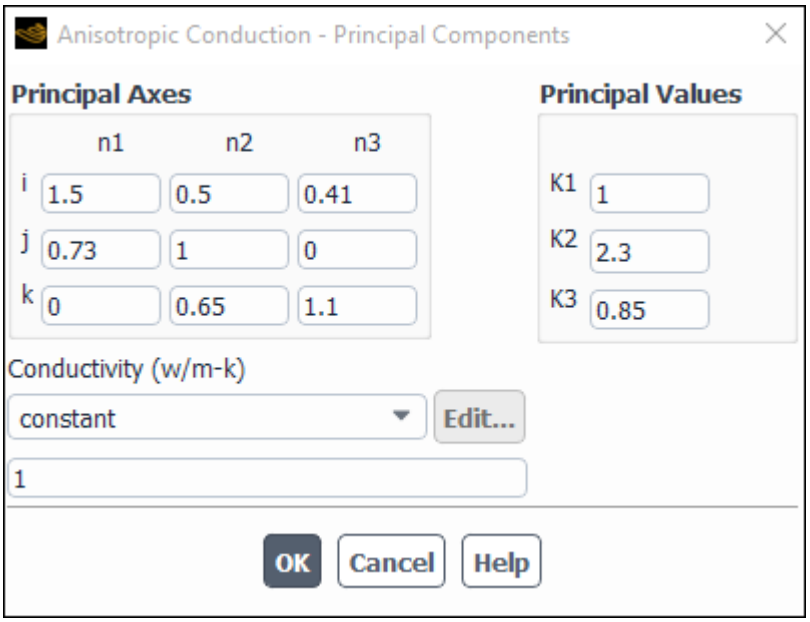
$$[\mathbf{N}] = \begin{bmatrix} n_1^1 & n_1^2 & n_1^3 \\ n_2^1 & n_2^2 & n_2^3 \\ n_3^1 & n_3^2 & n_3^3 \end{bmatrix} \tag{8.57}$$

$$[\mathbf{D}] = \begin{bmatrix} kK_1 & 0 & 0 \\ 0 & kK_2 & 0 \\ 0 & 0 & kK_3 \end{bmatrix} \tag{8.58}$$

where k is the conductivity.

To use principal axes and principal values to define the anisotropic thermal conductivity for your solid material, select **principal-axes-values** for **Thermal Conductivity** in the Create/Edit Materials Dialog Box (p. 4885) (Figure 8.22: The Create/Edit Materials Dialog Box (p. 1645)). This will open the Anisotropic Conduction - Principal Components Dialog Box (p. 4927) (Figure 8.28: The Anisotropic Conductivity - Principal Components Dialog Box (p. 1655)).

Figure 8.28: The Anisotropic Conductivity - Principal Components Dialog Box



In the **Principal Axes** group box of the Anisotropic Conduction - Principal Components Dialog Box (p. 4927), enter the **i**, **j**, and **k** components for each of the **n1**, **n2**, and **n3** vectors (Equation 8.55 (p. 1654)) that define the principal axes of your material in the global coordinate system; note that these must be linearly independent vectors. Then in the **Principal Values** group box, enter the **K1**, **K2**, and **K3** values that define the magnitudes of the principal values along the associated principal axes. Finally, make a selection from the **Conductivity** drop-down menu to specify k (in Equation 8.58 (p. 1655)) as a **constant**, polynomial function of temperature (**polynomial**, **piecewise-linear**, **piecewise-polynomial**), or **user-defined** function, and then define it using the number-entry box that becomes available or the dialog box that opens. See Constant Thermal Conductivity (p. 1645) and Thermal Conductivity as a Function of Temperature (p. 1646) for details on constants and thermal functions.

When you select the **user-defined** option, the User-Defined Functions Dialog Box (p. 4915) will open, allowing you to hook a `DEFINE_PROPERTY` UDF *only* if you have previously loaded a compiled

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UDF library or interpreted the UDF. Otherwise, you will get an error message. Details about user-defined functions can be found in the Fluent Customization Manual.

8.5.6.6. User-Defined Anisotropic Thermal Conductivity

You have the option of modeling anisotropic conduction for domains where the components of the thermal conductivity matrix (k_{ij} in Equation 8.53 (p. 1649)) are not constants and vary independently, perhaps as a function of space or because the domain has many anisotropic materials stitched together to form a composite. To use this option, you must first define the conductivity matrix using a UDF, as described in `DEFINE_ANISOTROPIC_CONDUCTIVITY` in the Fluent Customization Manual.

After you have interpreted or compiled the `DEFINE_ANISOTROPIC_CONDUCTIVITY` UDF, you must then select **user-defined-anisotropic-k** for **Thermal Conductivity** in the Create/Edit Materials Dialog Box (p. 4885). This will open the User-Defined Functions Dialog Box (p. 4915), which you can use to hook the UDF.

8.6. User-Defined Scalar (UDS) Diffusivity

There are two types of UDS diffusivity that you can specify in Ansys Fluent: isotropic and anisotropic. Diffusion is isotropic when it is the same in all directions. Isotropic diffusion coefficients can be specified in two ways: either as a single **user-defined** that applies to all UDS transport equations defined for your model; or on a per-scalar basis as **constants**, **polynomial** functions of temperature, or **user-defined** functions.

Diffusion is anisotropic when the diffusion coefficients are different in different directions. Anisotropic diffusion can be specified by a tensor diffusion coefficient matrix Γ (Equation 8.59 (p. 1656)) for each UDS (in both fluid and solid zones) in four different ways: general **anisotropic**, **orthotropic**, **cyl-orthotropic**, and **user-defined-anisotropic**. All UDS diffusivity parameters are set from the Create/Edit Materials Dialog Box (p. 4885) and are discussed below. Note that details about how to define and use UDFs in UDS transport equations is discussed in the Fluent Customization Manual.

The second-order diffusion term in the most general form is

$$\nabla \cdot \left(\Gamma \cdot \nabla \phi^k \right) \tag{8.59}$$

where Γ is a 3 × 3 tensor in 3D.

For additional information, see the following sections:

8.6.1. Isotropic Diffusion

8.6.2. Anisotropic Diffusion

8.6.3. User-Defined Anisotropic Diffusivity

8.6.1. Isotropic Diffusion

For isotropic diffusion, Γ in Equation 8.59 (p. 1656) is equal to a scalar Γ times the identity matrix and the equation reduces to

$$\nabla \cdot \left(\Gamma \nabla \phi^k \right) \tag{8.60}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

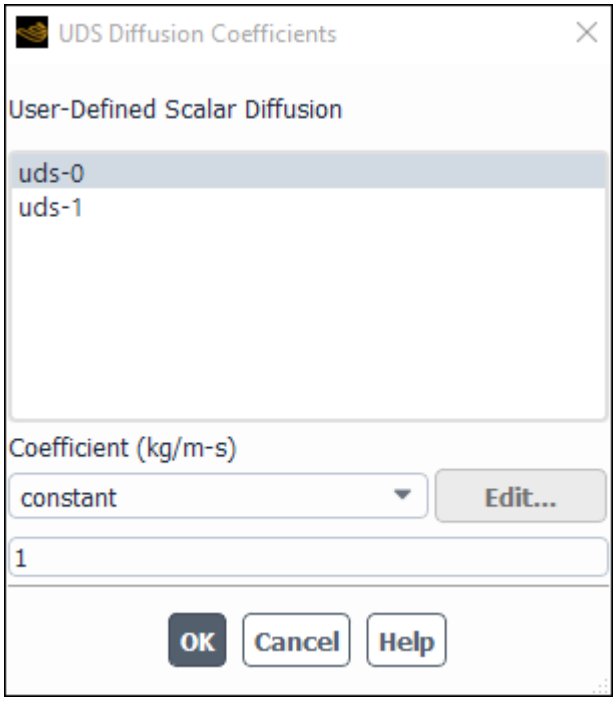
You can specify isotropic diffusivity as a single user-defined function that applies to all UDS transport equations. For this case, choose **user-defined** from the drop-down list for **UDS Diffusivity** in the Create/Edit Materials Dialog Box (p. 4885).

Setup → Materials

If you have previously loaded a compiled UDF library or have interpreted the UDF, then the User-Defined Functions Dialog Box (p. 4915) will open, allowing you to hook the `DEFINE_DIFFUSIVITY` UDF to Ansys Fluent. If no functions have been loaded, you will get an error message. More information about user-defined functions can be found in the Fluent Customization Manual.

Isotropic diffusion coefficients can also be defined on a per-scalar basis by selecting **defined-per-uds** from the drop-down list for **UDS Diffusivity** in the Create/Edit Materials Dialog Box (p. 4885). This will open the UDS Diffusion Coefficients Dialog Box (p. 4946) (Figure 8.29: The UDS Diffusion Coefficients Dialog Box (p. 1657)).

Figure 8.29: The UDS Diffusion Coefficients Dialog Box



In the UDS Diffusion Coefficients Dialog Box (p. 4946), select a scalar equation (for example, **uds-0**) and then choose a **constant**, **polynomial**, or **user-defined** function from the **Coefficient** drop-down list. For the default fluid (air), the constant diffusion coefficient is 1 kg/m-s. If you choose **polynomial**, the Polynomial Profile Dialog Box (p. 4906) will open and you can specify your coefficients as a function of temperature. See Inputs for Polynomial Functions (p. 1617) for details.

When you select the **user-defined** option, the User-Defined Functions Dialog Box (p. 4915) will open, allowing you to hook a `DEFINE_DIFFUSIVITY` UDF *only* if you have previously loaded a compiled UDF library or interpreted a UDF. Otherwise, you will get an error message. More information about user-defined functions can be found in the Fluent Customization Manual.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.6.2. Anisotropic Diffusion

You can specify anisotropic diffusion coefficients in both fluid and solid zones by defining the tensor diffusion coefficient matrix Γ (Equation 8.59 (p. 1656)) on a per-scalar basis. You can use anisotropic diffusivity for UDS scalar transport equations to model species transport equations in porous media and in solids where species diffusion shows anisotropic behavior.

Important:

- Note that the anisotropic diffusion options discussed in the following sections are available with the pressure-based solver and the density-based solvers.
- UDS diffusion coefficients can be postprocessed only in those cells that have isotropic diffusivity. In all other cells, the diffusion coefficient will be zero.

In all cases, you enable anisotropic diffusion by selecting **defined-per-uds** under **UDS Diffusivity** in the Create/Edit Materials Dialog Box (p. 4885). This will open the UDS Diffusion Coefficients Dialog Box (p. 4946) (Figure 8.29: The UDS Diffusion Coefficients Dialog Box (p. 1657)).

Setup → Materials

In the UDS Diffusion Coefficients Dialog Box (p. 4946), select a scalar equation (for example, **uds-0**) and then choose one of the following methods under **Coefficient** to specify the anisotropic diffusion coefficient. These methods are described in detail below.

- anisotropic
- orthotropic
- cylindrical orthotropic
- user-defined anisotropic

8.6.2.1. Anisotropic Diffusivity

For anisotropic diffusivity, you can specify Γ in Equation 8.59 (p. 1656) in the form $K\mathbf{\Gamma}$ where K is a constant $3x3$ matrix in 3D and Γ is a scalar multiplier.

The diffusion coefficient matrix is specified as

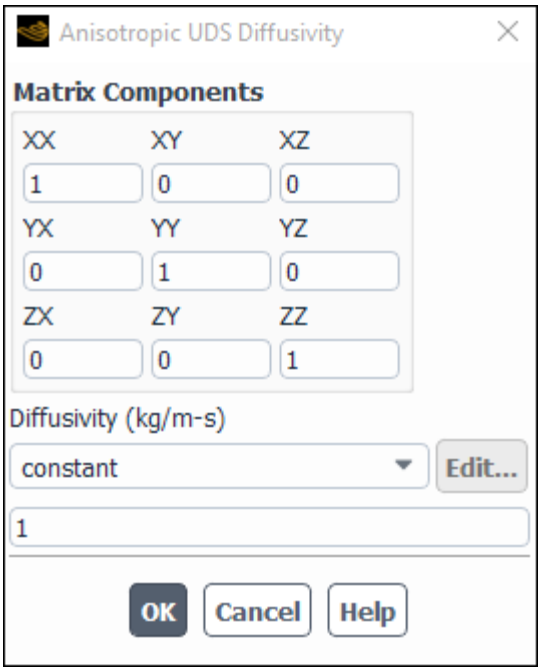
$$k_{ij}=k\hat{e}_{ij} \tag{8.61}$$

where k is the diffusivity and $\hat{e}_{ij}$ is a matrix (2 × 2 for two dimensions and 3 × 3 for three-dimensional problems). Note that $\hat{e}_{ij}$ can be a non-symmetric matrix.

To specify anisotropic diffusion coefficients, first select a scalar equation (for example, **uds-0**) from the **User-Defined Scalar Diffusion** list in the UDS Diffusion Coefficients Dialog Box (p. 4946) (Figure 8.29: The UDS Diffusion Coefficients Dialog Box (p. 1657)). Then choose **anisotropic** in the drop-down list under **Coefficient**. This will open the **Anisotropic UDS Diffusivity** dialog box (Figure 8.30: The Anisotropic UDS Diffusivity Dialog Box (p. 1659)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 8.30: The Anisotropic UDS Diffusivity Dialog Box



In the **Anisotropic UDS Diffusivity** dialog box, enter the **Matrix Components** and then select the **Diffusivity** to be a **constant**, polynomial function of temperature (**polynomial**, **piecewise-linear**, **piecewise-polynomial**), or **user-defined**. See Inputs for Polynomial Functions (p. 1617), Inputs for Piecewise-Linear Functions (p. 1618), and Inputs for Piecewise-Polynomial Functions (p. 1620) for details on polynomial temperature functions.

When you select the **user-defined** option, the User-Defined Functions Dialog Box (p. 4915) will open, allowing you to hook a `DEFINE_DIFFUSIVITY` UDF *only* if you have previously loaded a compiled UDF library or interpreted a UDF. Otherwise, you will get an error message. More information about user-defined functions can be found in the Fluent Customization Manual.

8.6.2.2. Orthotropic Diffusivity

For orthotropic diffusivity, you can specify Γ in Equation 8.59 (p. 1656) through 'principal' direction vectors and diffusion coefficients along these directions. Ansys Fluent, in turn, computes Γ from parameters that you supply. The principal directions are the same everywhere, but each of he directional diffusion coefficients can be specified as a constant, polynomial function of temperature, or through user-defined functions.

When orthotropic diffusivity is used, the diffusion coefficients $\left(k_\xi, k_\eta, k_\zeta\right)$ in the principal directions $\left(\hat{e}_\xi, \hat{e}_\eta, \hat{e}_\zeta\right)$ are specified. The diffusivity matrix is then computed as

$$k_{ij} = k_\xi e_{\xi i} e_{\xi j} + k_\eta e_{\eta i} e_{\eta j} + k_\zeta e_{\zeta i} e_{\zeta j} \tag{8.62}$$

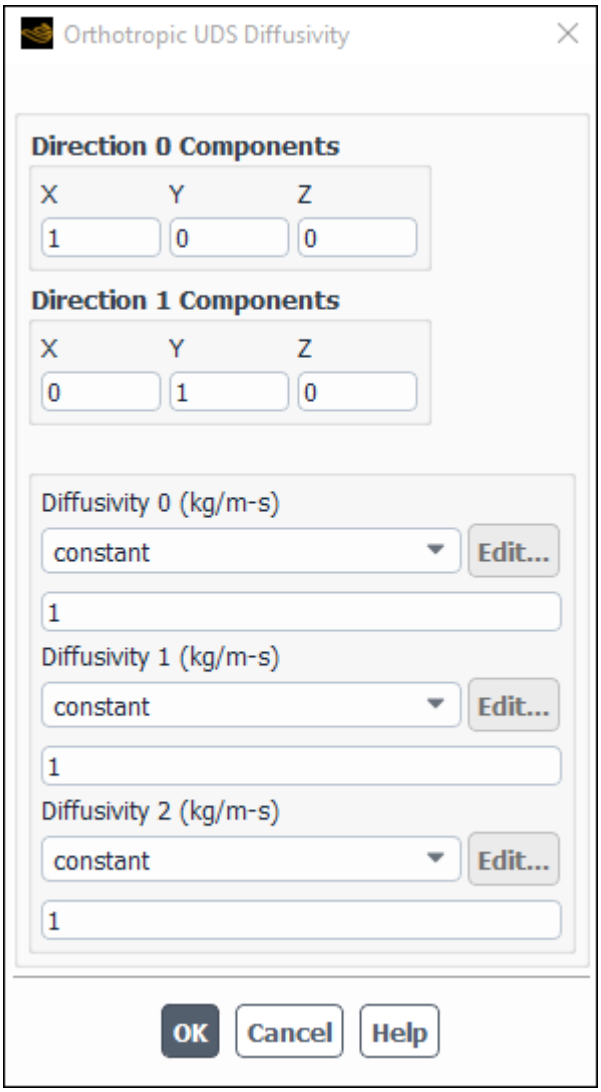
Important:

For two-dimensional problems, only the functions $\left(k_\xi, k_\eta\right)$ and the unit vector $\left(\hat{e}_\xi\right)$ need to be specified.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To specify orthotropic diffusion coefficients, first select a scalar equation (for example, **uds-0**) from the **User-Defined Scalar Diffusion** list in the UDS Diffusion Coefficients Dialog Box (p. 4946) (Figure 8.29: The UDS Diffusion Coefficients Dialog Box (p. 1657)). Then choose **orthotropic** in the drop-down list under **Coefficient**. This will open the **Orthotropic UDS Diffusivity** dialog box (Figure 8.31: The Orthotropic UDS Diffusivity Dialog Box (p. 1660)).

Figure 8.31: The Orthotropic UDS Diffusivity Dialog Box



Since the directions $\left(\hat{e}_\xi, \hat{e}_\eta, \hat{e}_\zeta\right)$ are mutually orthogonal, only the first two need to be specified for three-dimensional problems. $\hat{e}_\xi$ is defined using **X,Y,Z** under **Direction 0 Components**, and $\hat{e}_\eta$ is defined using **X,Y,Z** under **Direction 1 Components**. You can define **Diffusivity 0** $\left(k_\xi\right)$, **Diffusivity 1** $\left(k_\eta\right)$, and **Diffusivity 2** $\left(k_\zeta\right)$ as **constant**, **polynomial**, **piecewise-linear**, **piecewise-polynomial** functions of temperature, or **user-defined**. See Inputs for Polynomial Functions (p. 1617), Inputs for Piecewise-Linear Functions (p. 1618), and Inputs for Piecewise-Polynomial Functions (p. 1620) for details on polynomial temperature functions.

When you select the **user-defined** option, the User-Defined Functions Dialog Box (p. 4915) will open, allowing you to hook a `DEFINE_DIFFUSIVITY` UDF *only* if you have previously loaded a compiled

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

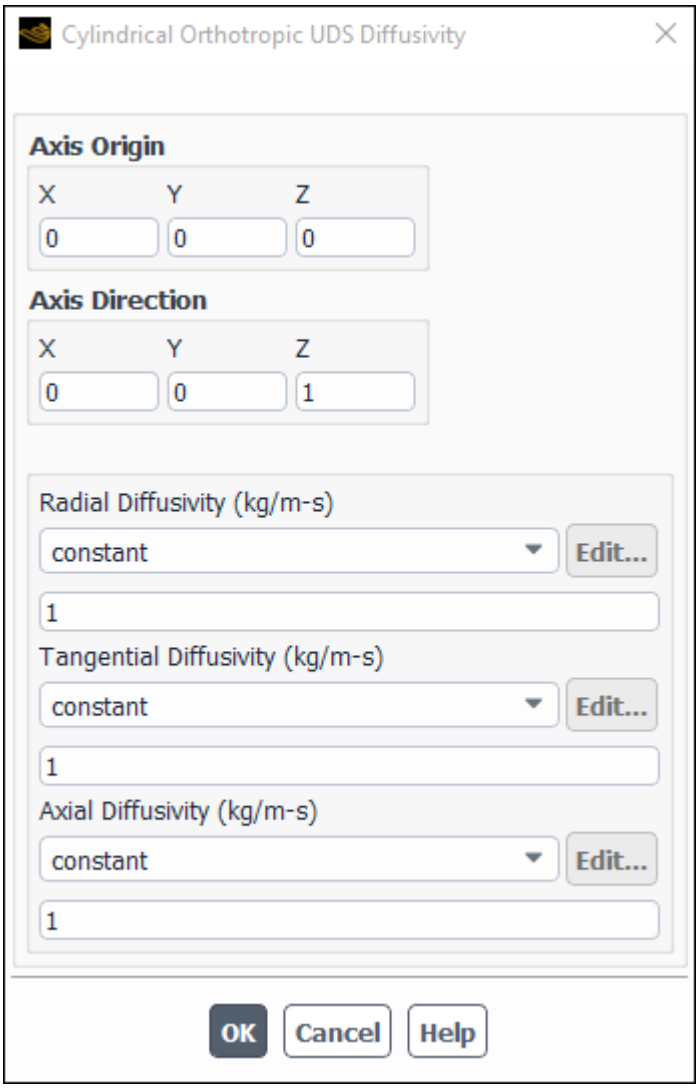
UDF library or interpreted a UDF. If no functions have been loaded, you will get an error message. More information about user-defined functions can be found in the Fluent Customization Manual.

8.6.2.3. Cylindrical Orthotropic Diffusivity

Orthotropic UDS diffusivity can also be specified on a per-scalar basis in cylindrical coordinates. This method is similar to orthotropic UDS diffusivity, except that the principal directions are specified as radial, tangential, and axial.

To specify cylindrical orthotropic diffusion coefficients, first select a scalar equation (for example, **uds-0**) from the **User-Defined Scalar Diffusion** list in the UDS Diffusion Coefficients Dialog Box (p. 4946) (Figure 8.29: The UDS Diffusion Coefficients Dialog Box (p. 1657)). Then choose **cyl-orthotropic** in the drop-down list under **Coefficient**. This will open the **Cylindrical Orthotropic UDS Diffusivity** dialog box (Figure 8.32: The Cylindrical Orthotropic UDS Diffusivity Dialog Box (p. 1661)).

Figure 8.32: The Cylindrical Orthotropic UDS Diffusivity Dialog Box



In three-dimensional cases, the origin and the direction of the cylindrical coordinate system must be specified along with the radial, tangential, and axial direction conductivities. In two-dimensional cases, the origin of the cylindrical coordinate system must be specified along with the radial and

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

tangential direction conductivities. Note that in two-dimensional cases, the direction is always along the +z axis. Ansys Fluent will automatically compute the anisotropic diffusivity matrix at each cell from this input. The calculation is based on the location of the cell in the cylindrical coordinate system specified.

You can define the **Radial Diffusivity**, **Tangential Diffusivity**, and **Axial Diffusivity** as **constant**, **polynomial**, **piecewise-linear**, **piecewise-polynomial**, or as **user-defined** functions of temperature, using the drop-down list below each of the diffusivities. See Inputs for Polynomial Functions (p. 1617), Inputs for Piecewise-Linear Functions (p. 1618), and Inputs for Piecewise-Polynomial Functions (p. 1620) for details on polynomial temperature functions.

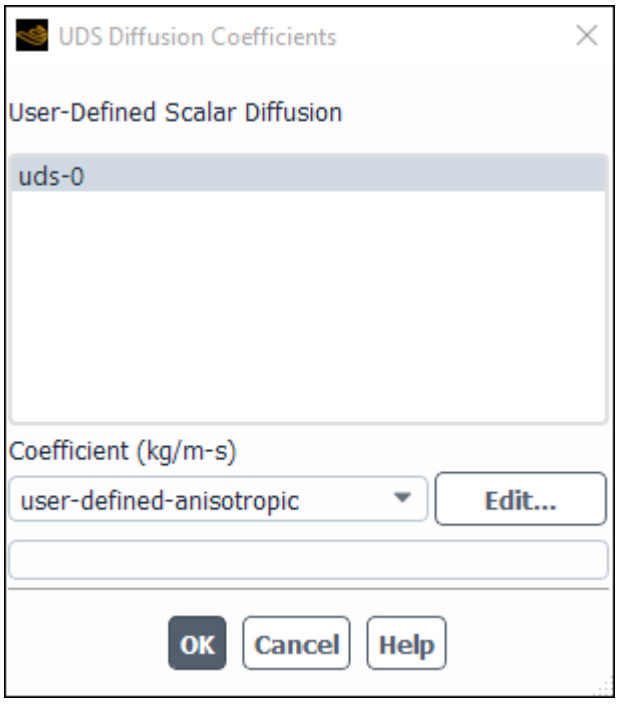
When you select the **user-defined** option, the User-Defined Functions Dialog Box (p. 4915) will open, allowing you to hook a `DEFINE_DIFFUSIVITY` UDF *only* if you have previously loaded a compiled UDF library or interpreted a UDF. If no functions have been loaded, you will get an error message. More information about user-defined functions can be found in the Fluent Customization Manual.

8.6.3. User-Defined Anisotropic Diffusivity

You can specify Γ in Equation 8.59 (p. 1656) on a per-scalar basis, directly, through user-defined functions (UDFs).

To specify a UDF for anisotropic diffusivity on a per-scalar basis, first select a scalar equation (for example, **uds-0**) from the **User-Defined Scalar Diffusion** list in the UDS Diffusion Coefficients Dialog Box (p. 4946) (Figure 8.33: The UDS Diffusion Coefficients Dialog Box (p. 1662)).

Figure 8.33: The UDS Diffusion Coefficients Dialog Box



Then choose **user-defined-anisotropic** in the drop-down list under **Coefficient**. The User-Defined Functions Dialog Box (p. 4915) will open, allowing you to hook a `DEFINE_ANISOTROPIC_DIFFUSIVITY` UDF *only* if you have previously loaded a compiled UDF library or interpreted a UDF. Otherwise,

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

you will get an error message. More information about user-defined functions can be found in the Fluent Customization Manual.

8.7. Specific Heat Capacity

The specific heat capacity must be defined when the energy equation is active. Ansys Fluent provides several options for definition of the heat capacity:

- constant heat capacity
- temperature- and/or composition-dependent heat capacity
- kinetic theory

Each of these input options and the governing physical models are detailed in this section. In all cases, you will define the **Cp** in the Create/Edit Materials Dialog Box (p. 4885).

Setup → **Materials**

Specific heat capacity is input in units of J/kg-K in SI units or BTU/lbm-°R in British units.

Important:

For combustion applications, a temperature-dependent specific heat is recommended.

For additional information, see the following sections:

8.7.1. Input of Constant Specific Heat Capacity
8.7.2. Specific Heat Capacity as a Function of Temperature
8.7.3. Defining Specific Heat Capacity Using Kinetic Theory
8.7.4. Specific Heat Capacity as a Function of Composition

8.7.1. Input of Constant Specific Heat Capacity

If you want to define the heat capacity as a constant, check that **constant** is selected in the drop-down list to the right of **Cp** in the Create/Edit Materials Dialog Box (p. 4885), and enter the value of heat capacity.

The specific heat for the default fluid (air) is 1006.43 J/kg-K.

8.7.2. Specific Heat Capacity as a Function of Temperature

You can also choose to define the specific heat capacity as a function of temperature. Three types of functions are available:

- piecewise-linear:

$$c_p(T) = c_{p_n} + \frac{c_{p_{n+1}} - c_{p_n}}{T_{n+1} - T_n}(T - T_n) \tag{8.63}$$

- piecewise-polynomial:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\text{for } T_{min,1} \le T < T_{max,1}: c_p(T) = A_1 + A_2 T + A_3 T^2 + \ldots$$
$$\text{for } T_{min,2} \le T < T_{max,2}: c_p(T) = B_1 + B_2 T + B_3 T^2 + \ldots \tag{8.64}$$

- nasa-9-piecewise-polynomial:

$$\text{for} \quad T_{min,1} \le T < T_{max,1}: Cp(T) = A_1 T^{-2} + A_2 T^{-1} + A_3 + A_4 T + A_5 T^2 + A_6 T^3 + A_7 T^4$$
$$\text{for} \quad T_{min,2} \le T < T_{max,2}: Cp(T) = B_1 T^{-2} + B_2 T^{-1} + B_3 + B_4 T + B_5 T^2 + B_6 T^3 + B_7 T^4 \tag{8.65}$$

- polynomial:

$$c_p(T) = A_1 + A_2 T + A_3 T^2 + \ldots \tag{8.66}$$

You can input the data pairs (T_n,c_{p_n}), ranges and coefficients A_i and B_i, or coefficients A_i that describe these functions using the Create/Edit Materials Dialog Box (p. 4885), as described in Defining Properties Using Temperature-Dependent Functions (p. 1616).

8.7.3. Defining Specific Heat Capacity Using Kinetic Theory

If you are using the ideal gas law (as described in Density (p. 1624)), you have the option to define the specific heat capacity using kinetic theory as

$$c_{p,i} = \frac{1}{2}\frac{R}{M_{w,i}}\left(f_i + 2\right) \tag{8.67}$$

where f_i is the number of modes of energy storage (degrees of freedom) for the gas species i that you can input by selecting **kinetic-theory** from the drop-down list to the right of **Cp** in the Create/Edit Materials Dialog Box (p. 4885). The solver will use your kinetic theory inputs in Equation 8.67 (p. 1664) to compute the specific heat capacity. See Kinetic Theory Parameters (p. 1676) for details about kinetic theory inputs.

8.7.4. Specific Heat Capacity as a Function of Composition

If you are modeling a flow that includes more than one chemical species (multicomponent flow), you have the option to define a composition-dependent specific heat capacity. You can also define the heat capacity of the mixture as a constant value or a function of temperature, or using kinetic theory.

To define a composition-dependent specific heat capacity for a mixture, follow these steps:

1. For the mixture material, choose **mixing-law** in the drop-down list to the right of **Cp**.
2. Click **Change/Create**.
3. Define the specific heat capacity for each of the fluid materials that make up the mixture. You may define constant or (if applicable) temperature-dependent heat capacities for the individual species. You may also use kinetic theory for the individual heat capacities, if applicable.

The solver will compute the mixture's specific heat capacity as a mass fraction average of the pure species heat capacities:

$$c_p = \sum_i Y_i c_{p,i} \tag{8.68}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.8. Radiation Properties

When you have activated one of the radiation models (except for the surface-to-surface model, which requires no additional properties), there will be additional properties for you to set in the Create/Edit Materials Dialog Box (p. 4885):

- For the P-1 model, you must set the radiation **Absorption Coefficient** and **Scattering Coefficient** (a and σ_s in Equation 5.57 in the Theory Guide).
- For the Rosseland model, set the **Absorption Coefficient** and **Scattering Coefficient** (a and σ_s in Equation 5.58 in the Theory Guide).
- For the DTRM, only the **Absorption Coefficient** is required (a in Equation 5.91 in the Theory Guide).
- For the DO model, set the **Absorption Coefficient** and the **Scattering Coefficient** (a and σ_s in Equation 5.98 in the Theory Guide). In addition, if you are modeling semi-transparent media, specify the **Refractive Index** (n_a or n_b in Equation 5.117 in the Theory Guide). With the DO model, you can specify radiation properties for solid materials, to be used when semi-transparent media are modeled.
- For the MC model, set the **Absorption Coefficient** and the **Scattering Coefficient** (a and σ_s in Equation 5.98 in the Theory Guide). In addition, if you are modeling semi-transparent media, specify the **Refractive Index** (n_a or n_b in Equation 5.117 in the Theory Guide). With the MC model, you can specify radiation properties for solid materials, to be used when semi-transparent media are modeled.

For additional information, see the following sections:

8.8.1. Absorption Coefficient

8.8.2. Scattering Coefficient

8.8.3. Refractive Index

8.8.4. Reporting the Radiation Properties

8.8.1. Absorption Coefficient

To define the absorption coefficient, you can specify a constant value, a temperature-dependent function (see Defining Properties Using Temperature-Dependent Functions (p. 1616)), a composition-dependent function, or a user-defined function. The absorbing and emitting parts of the radiative transfer equation (RTE), Equation 5.56 in the Theory Guide, is a function of the absorption coefficient. The absorbing or emitting effects depend on the chosen radiation model. If there are only absorption effects, then Lambert's Law of absorption applies

$$I = I_o \exp\left(-ax\right) \tag{8.69}$$

where I is the radiation intensity, a is the absorption coefficient, and x is the distance through the material.

If you are modeling non-gray radiation with the P-1, DO, or MC radiation models, you also have the option to specify a constant absorption coefficient in each of the gray bands. The absorption coefficient is requested in units of 1/length. Along with the scattering coefficient, it describes the change in radiation intensity per unit length along the path through the fluid medium. Absorption coefficients can be computed using tables of emissivity for CO_2 and H_2O, which are generally available in textbooks on radiation heat transfer.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.8.1.1. Inputs for a Constant Absorption Coefficient

To define a constant absorption coefficient, simply enter the value in the field next to **Absorption Coefficient** in the Create/Edit Materials Dialog Box (p. 4885). Select **constant** in the drop-down list first if it is not already selected.

8.8.1.2. Inputs for a Composition-Dependent Absorption Coefficient

Ansys Fluent also allows you to input a composition-dependent absorption coefficient, where the local value of a is a function of the local mass fractions of water vapor and carbon dioxide. This modeling option can be useful for the simulation of radiation in combustion applications. The variable-absorption-coefficient model used by Ansys Fluent is the weighted-sum-of-gray-gases model (WSGGM) described in Radiation in Combusting Flows in the Theory Guide. To activate it, first enable the species calculation and make sure that CO_2 and H_2O are present in the mixture. Next, select **wsggm-domain-based**, **wsggm-user-specified**, or **user-defined-wsggm** in the drop-down list to the right of **Absorption Coefficient** in the **Create/Edit Materials** dialog box. If you select **user-defined-wsggm**, the **User-Defined Functions** dialog box will open, allowing you to select a previously loaded compiled UDF library or a previously interpreted UDF (see Hooking `DEFINE_WSGGM_ABS_COEFF` UDFs in the Fluent Customization Manual). The WSGGM options differ in the method used to compute the path length, as described in the section that follows.

8.8.1.2.1. Path Length Inputs

When the WSGGM is used to compute the absorption coefficient, you can choose how path length, s, is defined for Equation 5.145 in the Theory Guide. See Radiation in Combusting Flows in the Theory Guide to determine which method is appropriate for your case.

You will select the path length method when you choose the property input method for **Absorption Coefficient**, as described previously.

- If you choose **wsggm-domain-based**, s is set equal to a mean beam length calculated by Ansys Fluent according to Equation 5.146 in the Theory Guide, which is an average dimension of the domain; no further inputs are required.
- If you choose **wsggm-user-specified**, s is set equal to a mean beam length that you enter for **Path Length** in the WSGGM User Specified Dialog Box (p. 4947).
- If you choose **user-defined-wsggm**, Ansys Fluent will initially compute the absorption coefficient in the same manner as described for the **wsggm-domain-based** option; however, you have the option of writing a user-defined function that customizes this calculated value. If the soot model is enabled, you can also use the UDF to customize the soot absorption coefficient computed by Ansys Fluent. See `DEFINE_WSGGM_ABS_COEFF` in the Fluent Customization Manual for further details.

8.8.1.2.1.1. Inputs for a Non-Gray Radiation Absorption Coefficient

If you are using the non-gray DO model (see The DO Model Equations of the Theory Guide and Defining Non-Gray Radiation for the DO Model (p. 2212)), the non-gray P-1 model (see The P-1 Model Equations of the Theory Guide and Setting Up the P-1 Model with Non-Gray Radiation (p. 2194)), or the non-gray MC model (see Monte Carlo (MC) Radiation Model Theory), you can specify a different constant absorption coefficient for each of the bands used by the gray-band model. Select **gray-band** from the **Absorption Coefficient** drop-down list in the **Cre-**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ate/Edit Materials dialog box and then define the absorption coefficient for each band in the Gray-Band Absorption Coefficient Dialog Box (p. 4948). (Note that you must complete this dialog box in order to proceed.)

8.8.1.2.1.2. Effect of Particles and Soot on the Absorption Coefficient

Ansys Fluent will include the effect of particles on the absorption coefficient if you have turned on the **Particle Radiation Interaction** option in the Discrete Phase Model Dialog Box (p. 4845) (only for the P-1 and DO radiation models).

If you are modeling soot formation and you want to include the effect of soot formation on the absorption coefficient, turn on the **Soot-Radiation Interaction** in the Soot Model Dialog Box (p. 4825). The soot effects can be included for any of the radiation models, as long as you are using the WSGGM to compute a composition-dependent absorption coefficient. Note that you can use the **user-defined-wsggm** option to customize the soot absorption coefficient calculated by Ansys Fluent, as described previously.

8.8.2. Scattering Coefficient

The scattering coefficient is, by default, set to zero, and it is assumed to be isotropic. You can specify a constant value, a temperature-dependent function (see Defining Properties Using Temperature-Dependent Functions (p. 1616)), or a user-defined function. You can also specify a non-isotropic phase function.

The scattering coefficient is requested in units of 1/length. Along with the absorption coefficient, it describes the change in radiation intensity per unit length along the path through the fluid medium. You may want to increase the scattering coefficient in combustion systems, where particulates may be present.

8.8.2.1. Inputs for a Constant Scattering Coefficient

To define a constant scattering coefficient, simply enter the value in the field next to **Scattering Coefficient** in the Create/Edit Materials Dialog Box (p. 4885). (Select **constant** in the drop-down list first if it is not already selected.)

8.8.2.2. Inputs for the Scattering Phase Function

Scattering is assumed to be isotropic, by default, but you can also specify a linear-anisotropic scattering function. If you are using the DO model, Delta-Eddington and user-defined scattering functions are also available.

8.8.2.2.1. Isotropic Phase Function

To model isotropic scattering, select **isotropic** in the **Scattering Phase Function** drop-down list. No further inputs are necessary. This is the default setting in Ansys Fluent.

8.8.2.2.2. Linear-Anisotropic Phase Function

To model anisotropic scattering, select **linear-anisotropic** in the **Scattering Phase Function** drop-down list and set the value of the phase function coefficient (C in Equation 5.58 in the Theory Guide).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.8.2.2.3. Delta-Eddington Phase Function

To use a Delta-Eddington phase function, select **delta-eddington** in the **Scattering Phase Function** drop-down list. This will open the Delta-Eddington Scattering Function Dialog Box (p. 4949), in which you can specify the **Forward Scattering Factor** and **Asymmetry Factor** (f and C in Equation 5.107 in the Theory Guide). Since this is a modal dialog box, you must tend to it immediately.

8.8.2.2.4. User-Defined Phase Function

To use a user-defined phase function, select **user-defined** in the **Scattering Phase Function** drop-down list. The user-defined function will contain specifications for Φ^* and f in Equation 5.108 in the Theory Guide. More information about user-defined functions can be found in the Fluent Customization Manual.

8.8.3. Refractive Index

The refractive index is the ratio of speed of light in the medium to the speed of light in vacuum. It is by default set to 1. You can specify a constant value in the field next to **Refractive Index**.

If you are using the non-gray DO model (see The DO Model Equations of the Theory Guide and Defining Non-Gray Radiation for the DO Model (p. 2212)), the non-gray P-1 model (see The P-1 Model Equations of the Theory Guide and Setting Up the P-1 Model with Non-Gray Radiation (p. 2194)), or the non-gray MC model (see Monte Carlo (MC) Radiation Model Theory), you can specify a different constant refractive index for each of the bands used by the gray-band model. Select **refractive-band** from the **Refractive Index** drop-down list in the **Create/Edit Materials** dialog box and then define the refractive index for each band in the Gray-Band Refractive Index Dialog Box (p. 4949). Note that because this is a modal dialog box, you must tend to it immediately.

8.8.4. Reporting the Radiation Properties

You can display the computed local values for a and σ_s using the **Absorption Coefficient** and **Scattering Coefficient** items in the **Radiation...** category of the variable selection drop-down list that appears in postprocessing dialog boxes. You will also find the **Refractive Index** in the **Radiation...** category.

8.9. Mass Diffusion Coefficients

For species transport calculations, you can model the diffusion of chemical species using the following methods:

- Fickian diffusion (default)

  The Fick's law approximation is adequate for most applications. See Mass Diffusion in Laminar Flows and Mass Diffusion in Turbulent Flows in the *Fluent Theory Guide* for more information about the theoretical background of the mass diffusion due to concentration gradients.

- Full multicomponent diffusion

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For some applications (for example, diffusion-dominated laminar flows such as chemical vapor deposition), the full multicomponent diffusion model is recommended. See Full Multicomponent Diffusion in the *Fluent Theory Guide* for more information about the theoretical background of the full multicomponent diffusion.

Important:

The full multicomponent diffusion model is enabled by selecting the **Full Multicomponent Diffusion** option in the Species Model Dialog Box (p. 4768) and is computationally expensive.

By default, the solver computes the species diffusion using:

- For laminar flows: Equation 7.2 in the *Fluent Theory Guide*
- For turbulent flows: Equation 7.4 in the *Fluent Theory Guide*

If you have enabled the **Full Multicomponent Diffusion** option in the Species Model Dialog Box (p. 4768), then the solver computes the diffusive mass flux using Equation 7.8 in the *Fluent Theory Guide*.

The inputs for the diffusion coefficient for species i in the mixture ($D_{i,m}$ in Equation 7.3 in the *Fluent Theory Guide*) or for the individual binary mass diffusion coefficients ($\mathcal{D}_{ij}$ in Equation 7.3 or Equation 7.6 in the *Fluent Theory Guide*) are described in Mass Diffusion Coefficient Inputs (p. 1669).

8.9.1. Mass Diffusion Coefficient Inputs

To specify the mass diffusion coefficients for the mixture, follow the steps below. The diffusion coefficients have units of m^2/s in SI units or ft^2/s in British units.

1. If you are modeling full multicomponent species transport described in Full Multicomponent Diffusion in the *Fluent Theory Guide*, enable the **Full Multicomponent Diffusion** option in the Species Model Dialog Box (p. 4768).
2. If you want to account for thermal diffusion, enable the **Thermal Diffusion** option in the **Species Model dialog** box.
3. Open the **Create/Edit Materials** dialog box for the mixture material.

   Setup → **Materials**
4. In the **Properties** group box, select the specification method for the mass diffusion coefficients from the **Mass Diffusivity** drop-down list. The following options are available:

   - **constant-dilute-appx**: Allows you to specify a constant value for all $D_{i,m}$ in the field below. Ansys Fluent automatically uses the Fickian diffusion (Mass Diffusion in Laminar Flows and Mass Diffusion in Turbulent Flows in the *Fluent Theory Guide*). This method is not available for the **Full Multicomponent Diffusion** option.

     See Constant Dilute Approximation Inputs (p. 1671).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **dilute-approx**: Allows you to define each $D_{i,m}$ as a constant or as a polynomial function of temperature (if heat transfer is enabled). Ansys Fluent automatically uses the Fickian diffusion (Mass Diffusion in Laminar Flows and Mass Diffusion in Turbulent Flows in the *Fluent Theory Guide*). This method is not available for the **Full Multicomponent Diffusion** option.

  See Dilute Approximation Inputs (p. 1671) for details.

- **unity-lewis-number**: No additional inputs are required. The mass diffusion coefficients $D_{i,m}$ are calculated under the assumption of unity Lewis number for all species in the mixture. This method is not available for the **Full Multicomponent Diffusion** option.

  The Lewis number is the ratio of thermal diffusivity (Prandtl number) to mass diffusivity (Schmidt number). The **unity-lewis-number** option is a reasonable approximation when flow is highly turbulent and the contribution of laminar diffusion is small.

- **kinetic-theory**: The solver uses kinetic theory to calculate the binary diffusion of species i in each species j, $\mathcal{D}_{ij}$.

  See Multicomponent Method Inputs (p. 1672) for details.

  For the theory behind this method, see Kinetic Theory Parameters for Diffusion Coefficients in the *Fluent Theory Guide*.

- **multicomponent**: Allows you to define the binary diffusion of species i in each species j, $\mathcal{D}_{ij}$ as a constant or a polynomial function of temperature.

  See Multicomponent Method Inputs (p. 1672) for details.

- **user-defined**: Allows you to define a single user-defined function that applies to all mass diffusion coefficients. This is done using the `DEFINE_DIFFUSIVITY` macro and is explained in the Fluent Customization Manual.

If you are modeling a dilute mixture where the mass fraction of all species is small compared to one dominant species, choose either the **constant-dilute-appx** or **dilute-approx** method to specify $D_{i,m}$. If you are modeling a non-dilute mixture, you may define the individual binary mass diffusion coefficients $\mathcal{D}_{ij}$. The solver will compute the diffusion of species i in the mixture using Equation 7.3 in the *Fluent Theory Guide*, unless you have enabled **Full Multicomponent Diffusion** the **Species Model** dialog box.. In the latter case, the solver will use the inputs for the individual binary mass diffusion coefficients $\mathcal{D}_{ij}$ to calculate the diffusive mass flux according to Equation 7.8 in the *Fluent Theory Guide*.

5. (turbulent flows only) Optionally, you can adjust the **Turbulent Schmidt Number** in the Viscous Model Dialog Box (p. 4715) (**Model Constants** group box). See Mass Diffusion Coefficient Inputs for Turbulent Flow (p. 1673) for more information.

6. If you have enable the **Thermal Diffusion** option in the **Species Model dialog** box, define the thermal diffusion coefficients as described in Thermal Diffusion Coefficient Inputs (p. 1674).

7. If you are modeling anisotropic species diffusion in porous media, configure appropriate settings as described in Anisotropic Species Diffusion (p. 1673).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.9.1.1. Constant Dilute Approximation Inputs

To use the constant dilute approximation method, follow these steps:

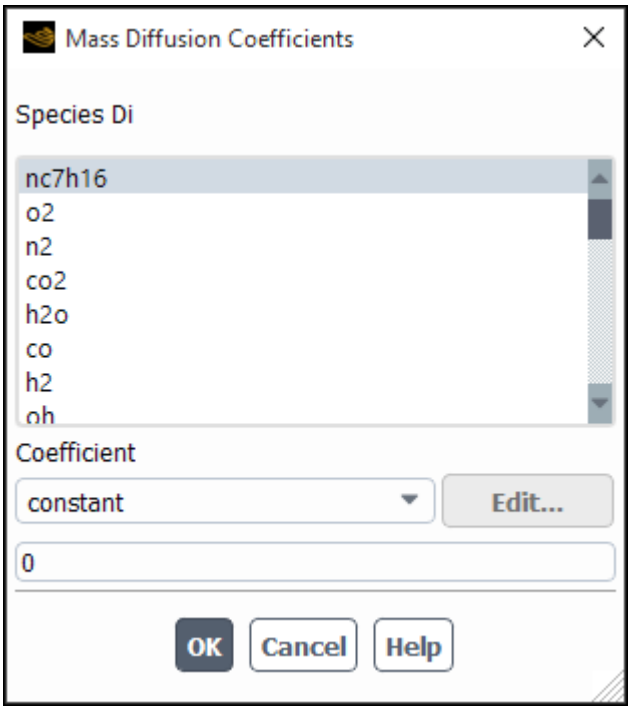
1. Select **constant-dilute-appx** in the drop-down list to the right of **Mass Diffusivity**.
2. Enter a single value of $D_{i,m}$. The same value will be used for the diffusion coefficient of each species in the mixture.

8.9.1.2. Dilute Approximation Inputs

To use the dilute approximation method, follow the steps below:

1. Select **dilute-approx** in the drop-down list to the right of **Mass Diffusivity**.
2. In the resulting Mass Diffusion Coefficients Dialog Box (p. 4944) (Figure 8.34: The Mass Diffusion Coefficients Dialog Box for Dilute Approximation (p. 1671)), select the species in the **Species Di** list for which you are going to define the mass diffusion coefficient.

   Figure 8.34: The Mass Diffusion Coefficients Dialog Box for Dilute Approximation



3. You can define $D_{i,m}$ for the selected species either as a constant value or (if heat transfer is active) as a polynomial function of temperature:
   - To define a constant diffusion coefficient, select **constant** (the default) in the drop-down list below **Coefficient**, and then enter the value in the field below the list.
   - To define a temperature-dependent diffusion coefficient, choose **polynomial** in the **Coefficient** drop-down list and then define the polynomial coefficients as described in Inputs for Polynomial Functions (p. 1617).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$D_{i,m}=A_1+A_2T+A_3T^2+\ldots \tag{8.70}$$

4. Repeat steps 2 and 3 until you have defined diffusion coefficients for all species in the **Species Di** list in the Mass Diffusion Coefficients Dialog Box (p. 4944).

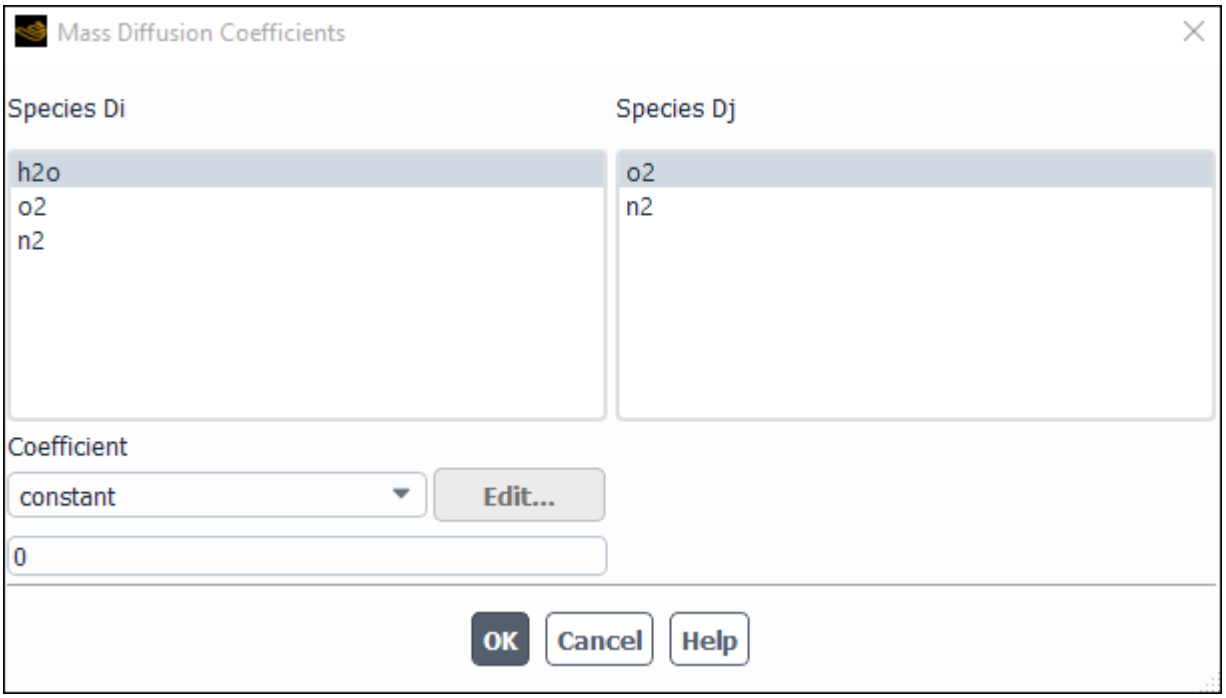
8.9.1.3. Multicomponent Method Inputs

To use the multicomponent method, and define constant or temperature-dependent diffusion coefficients, follow the steps below:

1. Select **multicomponent** in the drop-down list to the right of **Mass Diffusivity**.

2. In the resulting Mass Diffusion Coefficients Dialog Box (p. 4944) (Figure 8.35: The Mass Diffusion Coefficients Dialog Box for the Multicomponent Method (p. 1672)), select the species in the **Species Di** list and the **Species Dj** list for which you are going to define the mass diffusion coefficient D_{ij} for species i in species j.

Figure 8.35: The Mass Diffusion Coefficients Dialog Box for the Multicomponent Method



3. Define $\mathcal{D}_{ij}$ for the selected pair of species using one of the following methods: as a constant value or as a polynomial function of temperature (if heat transfer is active).

   - To define a constant diffusion coefficient, select **constant** (the default) from the**Coefficient** drop-down list, and then enter the value in the field below the list.

   - (only if heat transfer is enabled) To define a temperature-dependent diffusion coefficient, choose **polynomial** in the **Coefficient** drop-down list and then define the polynomial coefficients as described in Inputs for Polynomial Functions (p. 1617).

$$\mathcal{D}_{ij}=A_1+A_2T+A_3T^2+\ldots \tag{8.71}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Repeat steps 2 and 3 until you have defined diffusion coefficients for all pairs of species in the **Species Di** and **Species Dj** lists in the Mass Diffusion Coefficients Dialog Box (p. 4944).

To use the multicomponent method, and define the diffusion coefficient using kinetic theory (available only when the ideal gas law is used), follow these steps:

1. Choose **kinetic-theory** in the drop-down list to the right of **Mass Diffusivity**.
2. Click **Change/Create** after completing other property definitions for the mixture material.
3. Define the Lennard-Jones parameters, σ_i and $\left(\varepsilon / k_B\right)_i$, for each species (fluid material), as described in Kinetic Theory Parameters (p. 1676).

8.9.2. Mass Diffusion Coefficient Inputs for Turbulent Flow

When your flow is turbulent, you will define $D_{i,m}$ or $\mathcal{D}_{ij}$, as described for laminar flows in Mass Diffusion Coefficient Inputs (p. 1669), and you will also have the option to alter the default setting for the turbulent Schmidt number, Sc_t, as defined in Equation 7.5 in the *Fluent Theory Guide*.

Usually, in a turbulent flow, the mass diffusion is dominated by the turbulent transport as determined by the turbulent Schmidt number (Equation 7.5 in the *Fluent Theory Guide*). The turbulent Schmidt number measures the relative diffusion of momentum and mass due to turbulence and is on the order of unity in all turbulent flows. Because the turbulent Schmidt number is an empirical constant that is relatively insensitive to the molecular fluid properties, you will have little reason to alter the default value (0.7) for any species.

Should you want to modify the Schmidt number, enter a new value for **Turbulent Schmidt Number** in the Viscous Model Dialog Box (p. 4715) (**Model Constants** group box).

Setup → Models → Viscous → Edit...

Important:

Note that the full multicomponent diffusion model described in Full Multicomponent Diffusion in the *Fluent Theory Guide* is not recommended for turbulent flows.

8.9.3. Anisotropic Species Diffusion

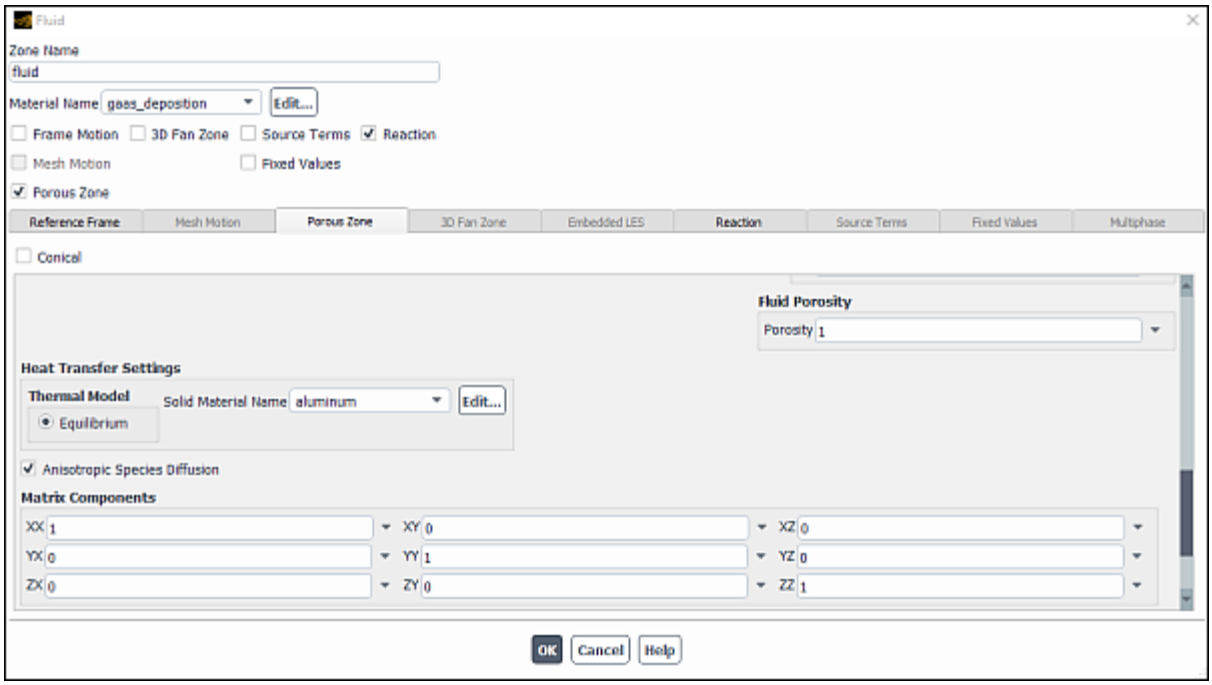
You can model anisotropic species diffusion in porous media. For the theory behind this modeling method, see Anisotropic Species Diffusion in the *Fluent Theory Guide*.

To account for anisotropic species diffusion in porous media simulations:

1. In the **Fluid** dialog box, enable **Porous Zone**.
2. In the **Porous Zone** tab, select the **Anisotropic Species Diffusion** check box (see Figure 8.36: Anisotropic Species Diffusion Matrix (p. 1674)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 8.36: Anisotropic Species Diffusion Matrix



3. Specify the **Matrix Components** for the anisotropic diffusion matrix $\mathbf{K}$.

   If you want to include the effect of porous media tortuosity, you can modify components of the anisotropic diffusion matrix accordingly.

   By default, $\mathbf{K}$ is the identity matrix that corresponds to isotropic diffusion.

8.9.4. Thermal Diffusion Coefficient Inputs

If you have enabled the **Thermal Diffusion** option in the Species Model Dialog Box (p. 4768), you can define the thermal diffusion coefficients in the Create/Edit Materials Dialog Box (p. 4885) as follows:

1. In the **Create/Edit Materials** dialog box for the mixture, select one of the following methods from the **Thermal Diffusion Coefficient** drop-down list:

   - Choose **kinetic-theory** to have Ansys Fluent compute the thermal diffusion coefficients using the empirically-based expression in Equation 7.17 in the *Fluent Theory Guide*. No further inputs are required for this option.

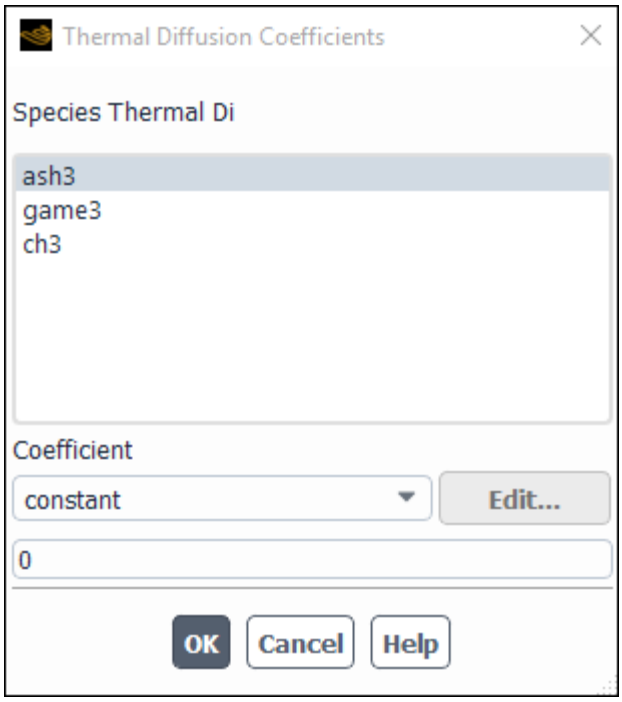
   - Choose **specified** to input the coefficient for each species in the Thermal Diffusion Coefficients Dialog Box (p. 4945) (Figure 8.37: The Thermal Diffusion Coefficients Dialog Box (p. 1675)) that opens automatically as further described.

   - Choose **user-defined** to use a user-defined function. More information about user-defined functions can be found in the Fluent Customization Manual.

2. (**specified** method only) For each species in the **Species Thermal Di** list of the **Thermal Diffusion Coefficients** dialog box, perform the following steps:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 8.37: The Thermal Diffusion Coefficients Dialog Box



a. Select the species in the **Species Thermal Di** list for which you are going to define the thermal diffusion coefficient.

b. Define $D_{T,i}$ for the selected species either as a constant value or as a polynomial function of temperature:

   - To define a constant diffusion coefficient, select **constant** (the default) in the drop-down list below **Coefficient**, and then enter the value in the field below the list.
   - To define a temperature-dependent diffusion coefficient, choose **polynomial** in the **Coefficient** drop-down list and then define the polynomial coefficients as described in Defining Properties Using Temperature-Dependent Functions (p. 1616).

For the theory behind this method, see Thermal Diffusion Coefficients in the *Fluent Theory Guide*.

8.10. Standard State Enthalpies

When you are solving a reacting flow using the finite-rate or eddy dissipation model, you must define the standard state enthalpy (also known as the formation enthalpy or heat of formation), h_j^0 for each species j. These inputs are used to define the mixture enthalpy as

$$H = \sum_j Y_j \left[\frac{h_j^0}{M_j} + \int_{T_{ref,j}}^{T} c_{p,j} dT \right] \tag{8.72}$$

where M_j is the molecular weight of the j^{th} species with units of kg/kmol and $T_{ref,j}$ is the reference temperature at which h_j^0 is defined. Standard state enthalpies are input in units of J/kg mol in SI units or in units of Btu/lb_mmol in British units.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For each species involved in the reaction (that is, each fluid material contained in the mixture material), you can set the **Standard State Enthalpy** and **Reference Temperature** in the Create/Edit Materials Dialog Box (p. 4885).

8.11. Standard State Entropies

If you are using the finite-rate model with reversible reactions (see Direct Use of Finite-Rate Kinetics (no TCI) in the Theory Guide), you must define the standard state entropy, s_j^0 for each species j. These inputs are used to define the mixture entropy as

$$S = \sum_j Y_j \left[\frac{s_j^0}{M_j} + \int_T^{T_{ref,j}} \frac{c_{p,j}}{T} dT \right] \tag{8.73}$$

where M_j is the molecular weight of the j^{th} species with units of kg/kmol and $T_{ref,j}$ is the reference temperature at which s_j^0 is defined. Standard state entropies are input in units of J/kmol-K in SI units or in units of Btu/lb_mmol-°R in British units.

For each species involved in the reaction (that is, each fluid material contained in the mixture material), you can set the **Standard State Entropy** and **Reference Temperature** in the Create/Edit Materials Dialog Box (p. 4885).

8.12. Unburnt Thermal Diffusivity

If you are modeling premixed combustion (see Modeling Premixed Combustion (p. 2536)), the fluid material in your domain should be assigned the properties of the unburnt mixture, including the thermal diffusivity (α in Equation 8.77 in the Theory Guide). α is defined as $k/\rho c_p$, and values at standard conditions can be found in combustion handbooks (for example, [83] (p. 5718)). To determine values at non-standard conditions, you must use a third-party 1D combustion program with detailed chemistry. You can set the **Unburnt Thermal Diffusivity** in the Create/Edit Materials Dialog Box (p. 4885).

8.13. Kinetic Theory Parameters

You may choose to define the following properties using kinetic theory when the ideal gas law is enabled:

- viscosity (for fluids)
- thermal conductivity (for fluids)
- specific heat capacity (for fluids)
- mass diffusion coefficients (for multi-species mixtures)

If you are using kinetic theory for a fluid's viscosity (Equation 8.28 (p. 1638)), you must input the kinetic theory parameters σ and ε/k_B for that fluid. These parameters are the Lennard-Jones parameters and are referred to by Ansys Fluent as the "characteristic length" and the "energy parameter" respectively.

- When kinetic theory is applied to calculation of a fluid's thermal conductivity only, no inputs are required.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To calculate specific heat of a fluid using kinetic theory (Equation 8.67 (p. 1664)), you need to enter the degrees of freedom for the fluid material.
- If you use kinetic theory to define a mixture material's mass diffusivity (Equation 7.10 in the *Fluent Theory Guide*), you must input σ_i and $\left(\varepsilon / k_B\right)_i$ for each chemical species i.

For additional information, see the following section:

8.13.1. Inputs for Kinetic Theory

8.13.1. Inputs for Kinetic Theory

The procedure for using kinetic theory is as follows:

1. Select **kinetic-theory** as the property specification method for the **Viscosity**, **Thermal Conductivity**, or heat capacity **Cp** of a fluid material, or for the **Mass Diffusivity** of a mixture material.
2. If the material for which you have selected the kinetic theory method for one or more properties is a fluid material, you must set the kinetic theory parameters for each of the constituent species (fluid materials).

   The parameters to be set are as follows:

   - **L-J Characteristic Length**
   - **L-J Energy Parameter**
   - **Degrees of Freedom** (only required if kinetic theory is used for specific heat)

   See the beginning of this section to find out which parameters are required to calculate each property using kinetic theory.

Characteristic length is defined in units of Angstroms. The energy parameter is defined in units of absolute temperature. Degrees of freedom is a dimensionless input. All kinetic theory materials can be found in the literature (for example, [65] (p. 5716)).

8.14. Operating Pressure

Specification of the operating pressure affects your calculation in different ways for different flow regimes. This section presents information about the operating pressure, its relevance for different cases, and how to set it correctly.

For additional information, see the following sections:

8.14.1. The Significance of Operating Pressure

8.14.2. Operating Pressure, Gauge Pressure, and Absolute Pressure

8.14.3. Setting the Operating Pressure

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.14.1. The Significance of Operating Pressure

Operating pressure is significant for incompressible ideal gas flows because it directly determines the density: the incompressible ideal gas law computes density as $\rho=\frac{p_{op}}{\frac{R}{M_w}T}$. You must therefore be sure to set the operating pressure appropriately.

In low-Mach-number compressible flow, the overall pressure drop is small compared to the absolute static pressure, and can be significantly affected by numerical roundoff. To understand why this is true, consider a compressible flow with $M<<1$. The pressure changes, Δp, are related to the dynamic head, $\frac{1}{2}\gamma pM^2$, where p is the static pressure and γ is the ratio of specific heats. This gives the simple relationship $\Delta p/\ p\sim M^2$, so that $\Delta p/\ p\rightarrow 0$ as $M\rightarrow 0$. Therefore, unless adequate precaution is taken, low-Mach-number flow calculations are very susceptible to roundoff error.

Operating pressure is significant for low-Mach-number compressible flows because of its role in avoiding roundoff error problems. Ensure that you set the operating pressure appropriately. You may want to specify a floating operating pressure instead of a constant operating pressure for low-Mach-number, time-dependent compressible flows with average pressure in the domain varying in time. See Floating Operating Pressure (p. 1763) for details.

Operating pressure is less significant for higher-Mach-number compressible flows. The pressure changes in such flows are much larger than those in low-Mach-number compressible flows, so there is no real problem with roundoff error and, therefore, no real need to use gauge pressure. In fact, it is common convention to use absolute pressures in such calculations. Since Ansys Fluent always uses gauge pressure, you can simply set the operating pressure to zero, making gauge and absolute pressures equivalent.

If the density is assumed constant or if it is derived from a profile function of temperature, the operating pressure is not used in the density calculation.

Note that the default operating pressure is 101325 Pa.

8.14.2. Operating Pressure, Gauge Pressure, and Absolute Pressure

Ansys Fluent avoids the problem of roundoff error (discussed in The Significance of Operating Pressure (p. 1678)) by subtracting the operating pressure (generally a large pressure roughly equal to the average absolute pressure in the flow) from the absolute pressure, and using the result (termed the gauge pressure). The relationship between the operating pressure, gauge pressure, and absolute pressure is shown below. The absolute pressure is simply the sum of the operating pressure and the gauge pressure:

$$p_{abs}=p_{op}+p_{gauge} \tag{8.74}$$

All pressures that you specify and all pressures computed or reported by Ansys Fluent are gauge pressures.

8.14.3. Setting the Operating Pressure

The criteria for choosing a suitable operating pressure are based on the Mach-number regime of the flow and the relationship that is used to determine density. For example, if you use the ideal gas law

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

in an incompressible flow calculation (for example, for a natural convection problem), you should use a value representative of the mean flow pressure.

To place this discussion in perspective, Table 8.1: Recommended Settings for Operating Pressure (p. 1679) shows the recommended approach for setting operating pressures. The default operating pressure is 101325 Pa.

Table 8.1: Recommended Settings for Operating Pressure

Density Relationship	Mach Number Regime	Operating Pressure
ideal gas law	$M > 0.1$	0 or ≈ mean flow pressure
ideal gas law	$M < 0.1$	≈ mean flow pressure
profile function of temperature	incompressible	not used
constant	incompressible	not used
incompressible ideal gas law	incompressible	≈ mean flow pressure

You will set the **Operating Pressure** in the Operating Conditions Dialog Box (p. 4978).

Setup → **Cell Zone Conditions** → **Operating Conditions...**

8.15. Using a Reference Pressure to Adjust the Gauge Pressure Field

For an incompressible flow or (for a steady-state case) a compressible flow in a cell zone or cell zone group that does not involve any pressure boundaries, Ansys Fluent can adjust the gauge pressure field after each iteration to keep it from floating. This is done using the pressure in the cell located at or nearest to the reference pressure location. The pressure value in this cell is subtracted from the entire gauge pressure field; as a result, the gauge pressure at the reference pressure cell is always zero.

Note:

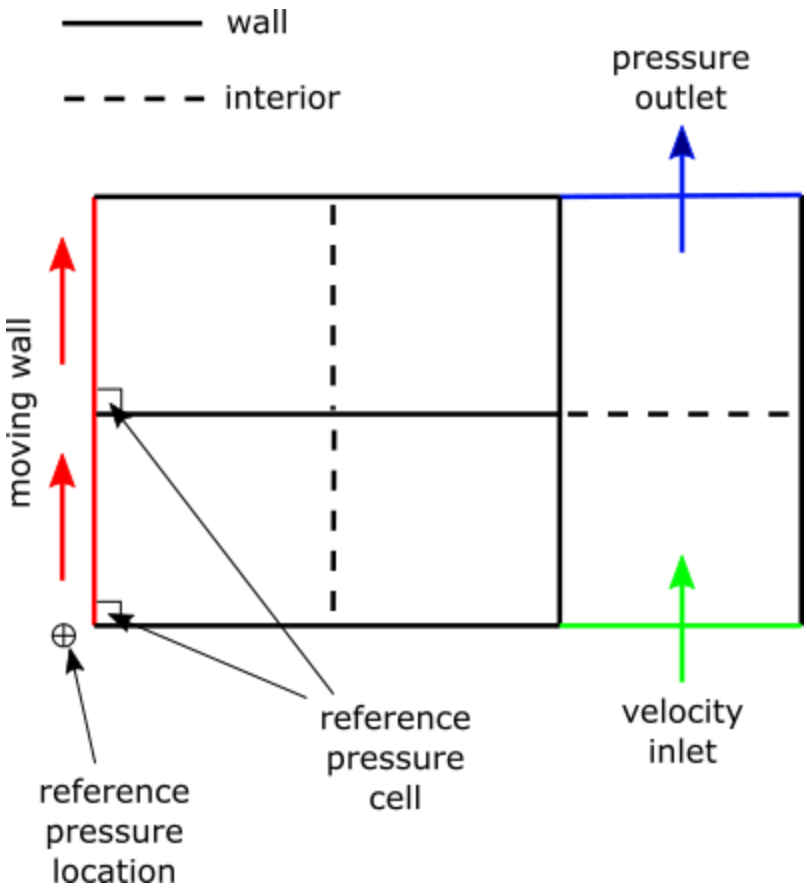
When you patch the pressure field for an incompressible flow or steady-state compressible flow, the reference pressure is subtracted in any closed domain after the patching. Therefore, the pressure field will be altered from your specified values.

By default, Fluent uses a reference pressure method that is designed for a mix of connected and disconnected fluid zones. Zones are considered connected if fluid can pass between them; this is the case when the zones are contiguous through boundary zones of type **interior**, **fan**, **porous-jump**, **radiator**, **rans-les-interface**, or **overset**, or share a mesh interface. A unique reference pressure cell will be used in each group of connected zones, and the presence of a pressure boundary in one group will not disqualify other groups from having the pressure adjusted.

An example of connected and disconnected fluid zones is shown in the following figure. Here, two lid-driven cavities are on the left, each consisting of a group of two cell zones enclosed by walls; there is also a channel on the right made up of two cell zones, which includes a pressure outlet. Fluent identifies reference pressure cells only in the left groups, and subtracts a unique value from the gauge pressure field within each of these groups. The gauge pressure field for the group on the right is not adjusted.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 8.38: Connected and Disconnected Cell Zones



The reference pressure cell of a particular group is the one that is at or closest to the reference pressure location. By default, this reference pressure location is defined as (0,0,0). There may be cases in which you might want to move the reference pressure location, perhaps locating it at a point where the absolute static pressure is known (for example, if you are planning to compare your results with experimental data). To change the location, enter new (**X**,**Y**,**Z**) coordinates for **Reference Pressure Location** in the Operating Conditions Dialog Box (p. 4978).

Setup → **Cell Zone Conditions** → **Operating Conditions...**

You can use the following text command to print the center coordinates of the reference pressure cells and the number of cell groups formed, the number of groups that will undergo pressure adjustment, and the cell zone IDs within each group:

`define` → `operating-conditions` → `used-ref-pressure-location`

Prior to Release 2020 R2, the default reference pressure method assumed that all of the cell zones are connected, which means that a single reference pressure cell is used in all of the zones, and the pressure adjustment is not performed anywhere if a pressure boundary exists in the case. For case files that were set up in such earlier releases, you can ensure that the reference pressure method accounts for connected and disconnected fluid zones (and thus always results in the most accurate static pressure calculations) by entering the following text command:

`define` → `operating-conditions` → `reference-pressure-method 2`

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.16. Real Gas Models

Some engineering problems involve fluids that do not behave as ideal gases. For example, at very high-pressure or very low-temperature conditions (for example, the flow of a refrigerant through a compressor) the flow cannot typically be modeled accurately using the ideal-gas assumption. Therefore, the real gas model allows you to solve accurately for the fluid flow and heat transfer problems where the working fluid behavior deviates from the ideal-gas assumption.

Ansys Fluent provides three real gas options for solving these types of flows:

- Cubic Equation of State Models (p. 1684)
- The NIST Real Gas Models (p. 1701)
- The User-Defined Real Gas Model (p. 1721)

All the models allow you to solve for either a single-species fluid flow or a multiple-species mixture fluid flow.

In addition, real gas property (RGP) table files can be used to incorporate material properties from external sources into an Ansys Fluent simulation.

For additional information, see the following sections:

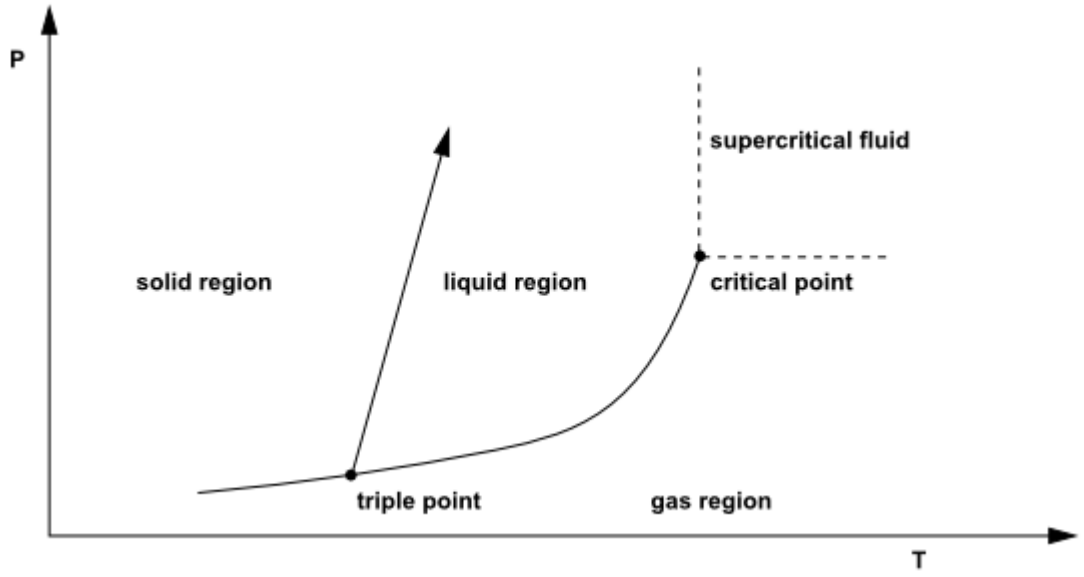
8.16.1. Introduction
8.16.2. Choosing a Real Gas Model
8.16.3. Cubic Equation of State Models
8.16.4. The NIST Real Gas Models
8.16.5. The User-Defined Real Gas Model
8.16.6. Using Real Gas Property (RGP) Table Files

8.16.1. Introduction

The states at which a pure material can exist can be graphically represented in diagrams of pressure vs. temperature (PT diagrams) and pressure vs. molecular or specific volume (PV diagrams). Homogeneous fluids are normally divided into two classes, liquids and gases. However the distinction cannot always be sharply drawn, because the two phases become indistinguishable at what is called the critical point. A typical pressure-temperature (PT) diagram of a pure material is shown in Figure 8.39: Typical PT Diagram of a Pure Material (p. 1682).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 8.39: Typical PT Diagram of a Pure Material



This figure shows the single phase regions, as well as the conditions of P and T where two phases coexist. Therefore the solid and the gas region are divided by the sublimation curve, the liquid and gas regions by the vaporization curve, and the solid and liquid regions by the fusion curve. The three curves meet at the triple point, where all three phases coexist in equilibrium. Although the fusion curve continues upward indefinitely, the vaporization curve terminates at the critical point. The coordinates of this point are called the critical pressure P_c and critical temperature T_c. These represent the highest temperature and pressure at which a pure material can exist in vapor-liquid equilibrium. At temperatures and pressures above the critical point, the physical property differences that differentiate the liquid phase from the gas phase become less defined. This reflects the fact that, at extremely high temperatures and pressures, the liquid and gaseous phases become indistinguishable. This new phase, which has some properties that are similar to a liquid and some properties that are similar to a gas, is called a supercritical fluid.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 8.40: Typical PV Diagram of a Pure Material

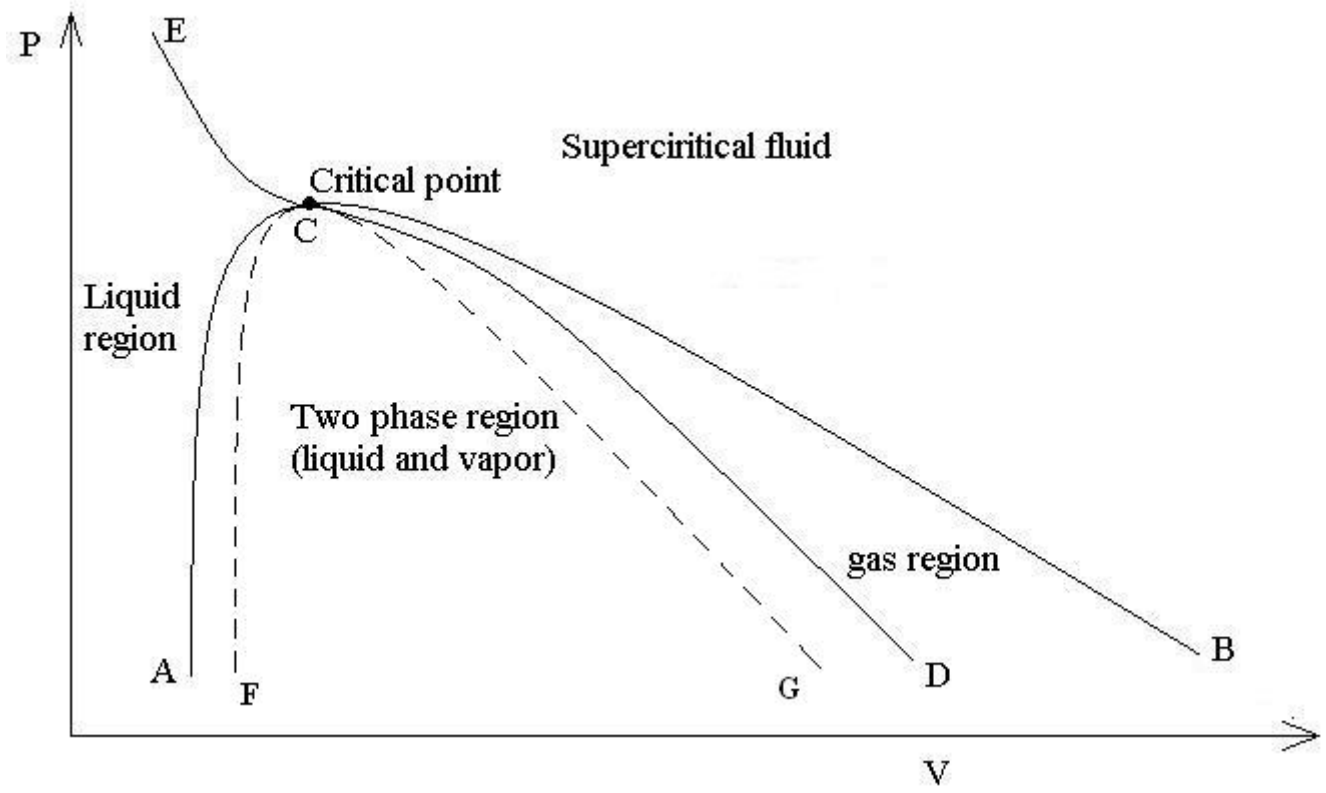


Figure 8.40: Typical PV Diagram of a Pure Material (p. 1683) presents a typical diagram of pressure versus molar or specific volume (PV diagram) of a pure material. The dome shaped curve ACD is called the saturation dome and separates the single phase regions in the diagram; curve AC represents the saturated liquid and curve CD the saturated vapor. The area under the saturation dome ACD is the two-phase region and represents all possible mixtures of vapor and liquid in equilibrium. Curve ECB is the critical isotherm and exhibits a horizontal inflection at point C at the top of the dome. This is the critical point. The specific volume corresponding to the critical point, is called the critical specific volume V_c. The conditions to the right of the critical isotherm ECB correspond to supercritical fluid. The dashed lines CF and CG in Figure 8.40: Typical PV Diagram of a Pure Material (p. 1683) represent the liquid and the vapor spinodal curves with the regions ACF and DCG between the saturation and spinodal lines representing the superheated liquid and supercooled vapor states, respectively. These states are called "metastable" because they exist temporarily in small local regions until phase change occurs. As an example, liquid metastable states may be formed in situations of fast depressurization, depending on the rate of depressurization and the disturbances that tend to make the liquid flash into vapor.

8.16.2. Choosing a Real Gas Model

The equation of state is the mathematical expression that relates pressure, molar or specific volume, and temperature for any pure homogeneous fluid in equilibrium states.

The simplest equation of state is the ideal gas law, which is approximately valid for the low pressure gas region of the PT and PV diagrams. Ideal gas behavior can be expected when

$$P/P_c \ll 1$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

or

$T/T_c > 2$ and $P/P_c < 1$

If your flow conditions correspond to either of those cases, you may use the ideal gas law in your simulation.

Another idealization, that of the incompressible fluid, can be employed for the low pressure region of the liquid phase. A constant density option is the appropriate selection in that case.

However, both of these approaches are not good approximations for flow conditions close to and beyond the critical point, where the fluid behavior cannot be described by the ideal gas, or the incompressible liquid assumptions. We refer to a fluid under those conditions as a real fluid, or a real gas and more complex relations are used for the determination of its physical and thermodynamic properties.

Ansys Fluent provides the following options for solving real fluid problems:

- The cubic equation of state models can be used to solve problems in the gas, liquid, and supercritical fluid regimes. The models are not available for the two-phase region under the phase dome. For further details see Cubic Equation of State Models (p. 1684).
- The NIST real gas model can be used to solve problems in the liquid, or gas and supercritical fluid regimes. The model does not allow modeling of the two-phase region. For further details see The NIST Real Gas Models (p. 1701).
- The user-defined real gas model allows you to solve problems in all regimes, as long as appropriate relationships are provided through the user-defined real gas functions. For further details see The User-Defined Real Gas Model (p. 1721).

The concepts presented in this section for pure materials are also extended to multicomponent mixtures with the introduction of appropriate composition-dependent parameters in the real gas equations of state and the material property models. All the real-gas modeling options above allow for either single-species or multicomponent flow modeling. In addition, you may solve reacting flow problems with the cubic equations of state models and the user-defined real gas functions.

8.16.3. Cubic Equation of State Models

8.16.3.1. Overview and Limitations

An equation of state is a thermodynamic equation, which provides a mathematical relationship between two or more state functions associated with the matter, such as its temperature, pressure, volume, or internal energy. One of the simplest equations of state for this purpose is the ideal gas law, which is roughly accurate for gases at low pressures and high temperatures. However, this equation becomes increasingly inaccurate at higher pressures and lower temperatures, and fails to predict condensation from a gas to a liquid.

Introduced in 1949, the Redlich-Kwong equation of state [122] (p. 5720) was a considerable improvement over other equations of that time. It is an analytic cubic equation of state and is still of interest primarily due to its relatively simple form. The original form is

$$P = \frac{RT}{V-b} - \frac{\alpha_0}{V(V+b)T_r^{0.5}} \tag{8.75}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where

P = absolute pressure (Pa)

R = universal gas constant

V = specific molar volume ($\mathrm{m}^3/\mathrm{kmol}$)

T = temperature (K)

T_r = reduced temperature $\frac{T}{T_c}$, where T_c is the critical temperature

α_0 and b are constants related directly to the fluid critical pressure and temperature

Many investigators have attempted to improve the accuracy of the Redlich-Kwong equation. Ansys Fluent has adopted the original form of the Redlich-Kwong equation, as well as the following modified forms:

- The Soave-Redlich-Kwong [152] (p. 5721) equation is a three-parameter equation of state, which can be applied for vapor, supercritical, and liquid property predictions. It has found wide acceptance mainly in the oil and gas industry and requires knowledge of the critical temperature, critical pressure, and acentric factor. It was developed by replacing the Redlich-Kwong attractive coefficient defined as $\alpha=\alpha_0/T_r^{0.5}$ with a two-parameter form $\alpha(T_r,\omega)$, where ω is the acentric factor.

- The Peng-Robinson equation [117] (p. 5719) is a three-parameter equation of state, also requiring the critical temperature, critical pressure and acentric factor parameters. It is thought to perform as well as the Soave-Redlich-Kwong equation, with an advantage in the prediction of the liquid densities.

- The Aungier-Redlich-Kwong [15] (p. 5713) equation provides improved predictions for vapor and supercritical fluids near the critical point, as well as for materials with a negative value of the acentric factor. The Aungier modified form is a four parameter equation and requires the critical specific volume in addition to the critical temperature, critical pressure and acentric factor.

The following limitations exist in Ansys Fluent for all cubic equation of state models:

- Pressure-inlets, mass flow-inlets, and pressure-outlets are the only inflow and outflow boundaries available for use with the real gas models.

- Non-reflecting boundary conditions are not compatible with the real gas models when using the density-based solver. If your model requires NRBC and a real gas model, you must use the pressure-based solver.

- The cubic equation of state real gas models are compatible with the Eulerian multiphase models.

- The cubic equation of state models are compatible with the Lagrangian Dispersed Phase Models. If you are modeling droplet or multicomponent particles, note that the current formulation does not take into account the near-critical point phenomena, which means that accurate results can be obtained for droplet temperatures below $T_{\lim}=0.9T_c$. See Using the Cubic Equation of State Models with the Lagrangian Dispersed Phase Models (p. 1698) for more details.

- The real gas models are not available with the premixed, inert, and composition PDF transport combustion models.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The cubic equation of state real gas models can be used with the following species models:
  - Species transport. Chemical reactions can be modeled with the finite rate and eddy dissipation models. Note that the **Dimension Reduction** model is not available with the real gas models.
  - Non-premixed model and partially-premixed model. Note that the **Compressibility Effects** option must be enabled in the **Species Model** dialog box in order for the real-gas models to be available. In addition the following restrictions apply when the non-premixed model is used together with a cubic equation of state real-gas model:
    → The empirical stream options cannot be used, as for the empirical species the critical properties cannot be defined.
    → Condensed species such as h2o<l> and c<s> are not supported and should be excluded from the PDF table, so ensure that you add all condensed species for your system in the excluded species list prior to the PDF table generation.
    The laminar flame speeds for the partially-premixed model are assumed to be the same as for ideal-gases.
- When the cubic equation of state models are applied for subcritical conditions near or under the phase dome, the two-phase flow is not modeled and either vapor or liquid state must be selected. Subcritical and supercritical states can co-exist in the same simulation.

8.16.3.2. Equation of State

The general form of pressure P for the cubic equation of state models is written as [15] (p. 5713) :

$$P=\frac{RT}{V-b+c}-\frac{\alpha}{\left(V^2+\delta V+\varepsilon\right)} \tag{8.76}$$

where

P = absolute pressure (Pa)

V = specific molar volume ($\mathrm{m}^3/\mathrm{kmol}$)

T = temperature (K)

R = universal gas constant

The coefficients α, b, c, δ, and ε are given for each equation of state as functions of the critical temperature T_c, critical pressure P_c, acentric factor ω and critical specific volume V_c. Note that the attractive coefficient α also has a temperature dependence, which varies for each equation of state model, and is commonly written as $\alpha=\alpha(T)$.

Redlich-Kwong Equation [128] (p. 5720):

$$\alpha(T)=\frac{\alpha_0}{\left(T/T_c\right)^{0.5}} \tag{8.77}$$

$$\alpha_0=\frac{0.42747R^2T_c^2}{P_c} \tag{8.78}$$

$$b=\frac{0.08664RT_c}{P_c} \tag{8.79}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The parameter δ is set equal to b, while c and ε are set to 0. The Redlich-Kwong equation is the simplest of the cubic equations of state in Ansys Fluent and requires two parameters only, the critical temperature T_c and the critical pressure P_c.

Soave-Redlich-Kwong Equation [152] (p. 5721):

$$\alpha(T)=\alpha_0[1+n(1-(T/T_c)^{0.5})]^2 \tag{8.80}$$

$$n=0.48+1.574\omega-0.176\omega^2 \tag{8.81}$$

α_0 and b are given by Equation 8.78 (p. 1686) and Equation 8.79 (p. 1686) respectively. As in the original Redlich-Kwong equation the parameter δ is set equal to b, while c and ε are set to 0. The Soave-Redlich-Kwong requires three parameters, the critical temperature T_c, the critical pressure P_c, and the acentric factor ω.

Peng-Robinson Equation [117] (p. 5719):

$$\alpha_0=\frac{0.457247R^2T_c^2}{P_c} \tag{8.82}$$

$$b=\frac{0.07780RT_c}{P_c} \tag{8.83}$$

The function $\alpha(T)$ is given by Equation 8.80 (p. 1687), with n provided in Equation 8.84 (p. 1687) as follows:

$$n=0.37464+1.54226\omega-0.26992\omega^2 \tag{8.84}$$

In the Peng-Robinson equation δ is set equal to 2b, ε is equal to $-b^2$, and c is set to 0.

Similar to the Soave-Redlich-Kwong equation, the Peng-Robinson equation is a three-parameter equation and requires the critical temperature T_c, the critical pressure P_c, and the acentric factor ω.

Aungier-Redlich-Kwong Equation [15] (p. 5713):

$$\alpha(T)=\alpha_0\left(\frac{T}{T_c}\right)^{-n} \tag{8.85}$$

$$n=0.4986+1.1735\omega+0.4754\omega^2 \tag{8.86}$$

α_0 and b are given by Equation 8.78 (p. 1686) and Equation 8.79 (p. 1686), respectively. As in the original Redlich-Kwong equation, the parameter δ is set equal to b and ε is set to 0. Parameter c is given by:

$$c=\frac{RT_c}{P_c+\frac{\alpha_0}{V_c(V_c+b)}}+b-V_c \tag{8.87}$$

The Aungier-Redlich-Kwong equation requires four parameters, namely the critical temperature T_c, the critical pressure P_c, the critical specific volume V_c, and the acentric factor ω.

8.16.3.3. Enthalpy, Entropy, and Specific Heat Calculations

Enthalpy, entropy, and specific heat are computed in terms of the relevant ideal gas properties and the departure functions. The departure function F_{dep} of any conceptual property F is defined as [122] (p. 5720)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$F_{dep}=F_{ideal}(T,P)-F(T,P) \tag{8.88}$$

where F_{ideal} is the value of the property as computed from the ideal gas relations. The departure function F_{dep} can be derived from basic thermodynamic relations and the equation of state.

Following the above definition, the enthalpy H for the equations of state models is given by the following equations [15] (p. 5713):

$$H=H_{ideal}-\frac{H_{dep}}{MW} \tag{8.89}$$

$$H_{dep}=-PV+RT-\frac{T\frac{\partial a}{\partial T}-a}{\Delta}ln\left[\frac{2V+\delta+\Delta}{2V+\delta-\Delta}\right] \tag{8.90}$$

$$\Delta=\left(\delta^2-4\varepsilon\right)^{0.5} \tag{8.91}$$

where,

H_{ideal} = ideal gas enthalpy at temperature T (J/kg)

H_{dep} = departure enthalpy (J/kmol)

MW = mean molecular weight (Kg/kmol)

P = pressure (Pa)

V = specific molar volume (m^3/kmol)

R = universal gas constant

α, δ, and ε are computed using Equation 8.77 (p. 1686)– Equation 8.87 (p. 1687) depending on the equation of state model

See Equation 8.76 (p. 1686) for a description of other coefficients.

Similarly, the departure internal energy U_{dep} can be shown to be

$$U_{dep}=-\frac{T\frac{\partial a}{\partial T}-a}{\Delta}ln\left[\frac{2V+\delta+\Delta}{2V+\delta-\Delta}\right] \tag{8.92}$$

where

U_{dep} = departure internal energy (J/kmol)

T = temperature (K)

V = specific molar volume (m^3/kmol)

Δ is given by Equation 8.91 (p. 1688)

α, δ, and ε are computed using Equation 8.77 (p. 1686) – Equation 8.87 (p. 1687) depending on the equation of state model

The specific heat c_p can be computed from the ideal specific heat $c_{p,ideal}$ and the departure specific heat at constant volume $c_{v,dep}$ as follows:

$$c_p=c_{p,ideal}-\frac{c_{p,dep}}{MW} \tag{8.93}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$c_{p,dep}=c_{v,dep}-R-T\frac{\left(\partial V/\partial T\right)^2}{\partial V/\partial P} \tag{8.94}$$

where

MW = mean molecular weight (kg/kmol)

P = pressure (Pa)

T = temperature (K)

V = specific molar volume (m^3/kmol)

R = universal gas constant

In Equation 8.94 (p. 1689) $c_{v,dep}$ is computed by differentiating the equation of departure internal energy (Equation 8.92 (p. 1688)) with respect to T, and the partial derivatives of the specific volume are computed by differentiating Equation 8.76 (p. 1686) appropriately.

The entropy S is computed by

$$S=S_{ideal,0}-\frac{S_{dep}}{MW} \tag{8.95}$$

$$S_{dep}=-Rln\left(\frac{V-b+c}{V_0}\right)-\frac{\frac{da}{dT}}{\Delta}ln\left[\frac{2V+\delta+\Delta}{2V+\delta-\Delta}\right] \tag{8.96}$$

where

$S_{ideal,0}$ = ideal gas entropy at temperature T and reference pressure (J/kg/K)

V_0 = ideal gas specific molar volume at temperature T and the reference pressure (m^3/kmol)

P = pressure (Pa)

T = temperature (K)

V = specific molar volume (m^3/kmol)

MW = mean molecular weight (kg/kmol)

Δ is given by Equation 8.91 (p. 1688) and α, δ, and ε are computed using Equation 8.77 (p. 1686) – Equation 8.87 (p. 1687), depending on the equation of state model. Note that the pressure term in Equation 8.95 (p. 1689) cancels out as both $S_{ideal,0}$ and V_0 are evaluated at the reference pressure.

8.16.3.4. Critical Constants for Pure Components

Equations describing real-gas properties require the knowledge of the critical constants for pure components and mixtures. These consist of the critical temperature (T_c), critical pressure (P_c), critical specific volume (V_c), and the acentric factor (ω).

Several critical constants for fluid materials in the Ansys Fluent property database `propdb.scm` have been compiled from a variety of sources available in the open literature [122] (p. 5720), [111] (p. 5719), [112] (p. 5719), [146] (p. 5721), [147] (p. 5721), [153] (p. 5721), [13] (p. 5713).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For those fluid materials, for which the critical properties have not been found in the open literature, these have been estimated using the commercially available software CRANIUM by Molecular Knowledge Systems Inc. [1] (p. 5713) : `http://www.molknow.com/Cranium/cranium.htm`

Critical property values for many hydrocarbon and nitrogenous radical species have been obtained from Tang and Brezinsky [159] (p. 5722). Where the critical properties for the radicals were not available in the literature, these were estimated using a modification of the Joback method [122] (p. 5720). This assumes that the radical site constitutes a distinct group with zero group contribution and utilizes the group contribution values for stable species.

The critical properties of coal volatiles have been estimated assuming that the volatiles can be approximated by a mixture of CO, CO_2, H_2, CH_4, and C_2H_6 in such a way, that the atom composition and the net calorific value of the volatiles is similar to that of the assumed mixture. The critical properties of the lignite and biomass volatiles have been assumed equal to those of formaldehyde. The critical properties of diesel and jet-a fuels have been set equal to those of decane. The critical properties of kerosene have been set equal to those of dodecane.

8.16.3.5. Calculations for Mixtures

For the computation of properties in real-gas mixtures, Ansys Fluent follows the so-called pseudo-critical method. According to this method, the behavior and properties of a real gas mixture will be the same as that of a pure component, to which appropriate critical constants are assigned. These mixture critical constants are functions of the mixture composition and pure component critical properties, and are sometimes called pseudocritical constants, because their values are generally expected to be different from the true mixture critical constants that may be determined experimentally. However for computational purposes they are the appropriate critical constant values for the mixture. According to the pseudocritical method, Ansys Fluent applies Equation 8.76 (p. 1686)– Equation 8.96 (p. 1689) also for mixtures, where the critical temperature T_c, critical pressure P_c, critical specific volume V_c, and acentric factor ω, are replaced by the corresponding mixture critical constants, critical temperature T_{cm}, critical pressure P_{cm}, critical specific volume V_{cm}, and acentric factor ω_m.

The following options are available in Ansys Fluent for the calculation of the mixture pseudocritical constants:

- The simplest rule for computing the pseudocritical constants C_{cm} for a real gas mixture is the mole fraction average [122] (p. 5720). This method is recommended for mixtures where the pure component critical properties for all components are not very different:

$$C_{cm} = \sum_{i=1}^{k} x_i C_{ci} \tag{8.97}$$

  where

  C_{cm} = mixture pseudocritical constant (temperature, pressure, specific volume or acentric factor)

  C_{ci} = critical constant of component i (temperature, pressure, specific volume or acentric factor)

  x_i = mole fraction of component i

  k = number of components in mixture

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- An alternative approach is based on the one-fluid van der Waals mixing rules as expressed in [128] (p. 5720). According to this approach, in order to apply the equation of state models to mixtures, the coefficients a and b in Equation 8.76 (p. 1686) are replaced by composition-dependent expressions as follows:

$$(\alpha_m)^{0.5} = \sum_i x_i (\alpha_i)^{0.5} \tag{8.98}$$

$$b_m = \sum_i x_i b_i \tag{8.99}$$

where

x_i = mole fraction of species i

With the appropriate expressions from Equation 8.77 (p. 1686)– Equation 8.87 (p. 1687) for each equation of state, and assuming a mixture acentric factor ω_m for the evaluation of parameter n in Equation 8.81 (p. 1687), Equation 8.84 (p. 1687), and Equation 8.86 (p. 1687), the mixing rules Equation 8.98 (p. 1691) and Equation 8.99 (p. 1691) can be rearranged to yield direct expressions of the mixture critical properties as functions of the mole fractions and the component critical properties [15] (p. 5713).

The resulting expressions for the mixture critical specific temperature T_{cm} are as follows:

– Soave-Redlich-Kwong and Peng-Robinson models

$$T_{cm} = \frac{\left[\sum_i \left(x_i \frac{T_{ci}}{P_{ci}^{0.5}}\right)\right]^2}{\sum_i \left(\frac{x_i T_{ci}}{P_{ci}}\right)} \tag{8.100}$$

– Redlich-Kwong model and Aungier-Redlich-Kwong model:

$$T_{cm}^{1+n} = \frac{\left[\sum_i \left(x_i \sqrt{\frac{T_{ci}^{(2+n)}}{P_{ci}}}\right)\right]^2}{\sum_i \left(\frac{x_i T_{ci}}{P_{ci}}\right)} \tag{8.101}$$

where for the Aungier-Redlich-Kwong model n is obtained from Equation 8.86 (p. 1687) as function of the mixture acentric factor ω_m. For the Redlich-Kwong model $n = 0.5$.

The mixture critical specific pressure P_{cm} is computed as:

$$P_{cm} = \frac{T_{cm}}{\sum_i \left(\frac{x_i T_{ci}}{P_{ci}}\right)} \tag{8.102}$$

The mixture critical specific volume V_{cm} is computed as:

$$V_{cm} = \sum_I \frac{x_i P_{ci} V_{ci}}{T_{ci}} \left(\frac{T_{cm}}{P_{cm}}\right) \tag{8.103}$$

The notation for Equation 8.100 (p. 1691) to Equation 8.103 (p. 1691) is

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

T_{cm} = mixture pseudocritical temperature (K)

P_{cm} = mixture pseudocritical pressure (Pa)

V_{cm} = mixture pseudocritical molar volume (m^3/kmol)

T_{ci} = critical temperature for component i (K)

P_{ci} = critical pressure for component i (Pa)

V_{ci} = critical molar volume for component i (m^3/kmol)

x_i = mole fraction for component i

8.16.3.5.1. Using the Cubic Equation of State Real Gas Models

For single or multicomponent flows, you will enable the cubic equation of state real gas models by selecting **real-gas-soave-redlich-kwong**, **real-gas-peng-robinson**, **real-gas-aungier-redlich-kwong**, or **real-gas-redlich-kwong** from the **Density** drop-down list in the **Create/Edit Materials** dialog box.

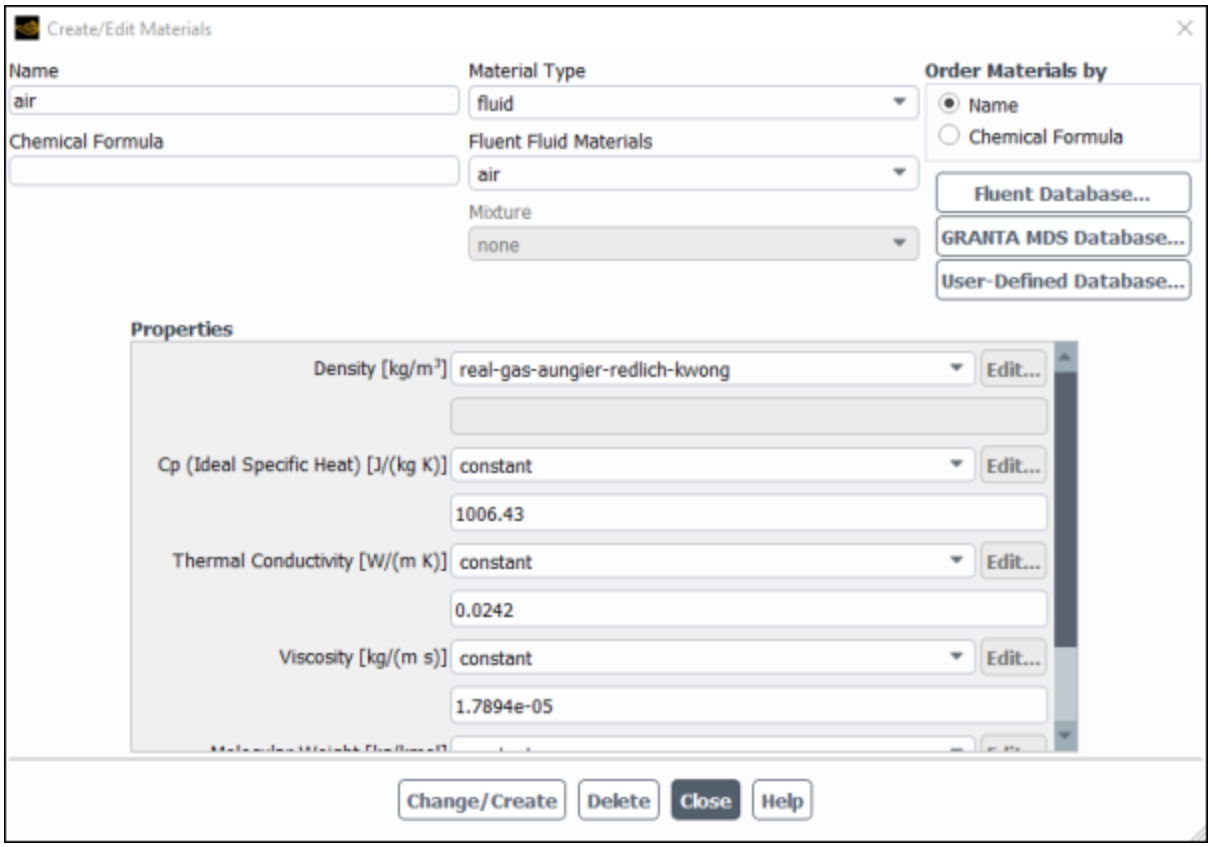
Setup → Materials → Create/Edit...

The required inputs for the cubic equation of state real gas models for single component flow and mixtures are described below.

Single Component Flow

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 8.41: The Cubic Equation of State Model for a Real-Gas Fluid



When any of the cubic equation of state models is enabled, enter the following material properties in the dialog box:

- ideal specific heat
- molecular weight
- standard state entropy
- reference temperature
- critical temperature
- critical pressure
- critical specific volume
- acentric factor

Important:

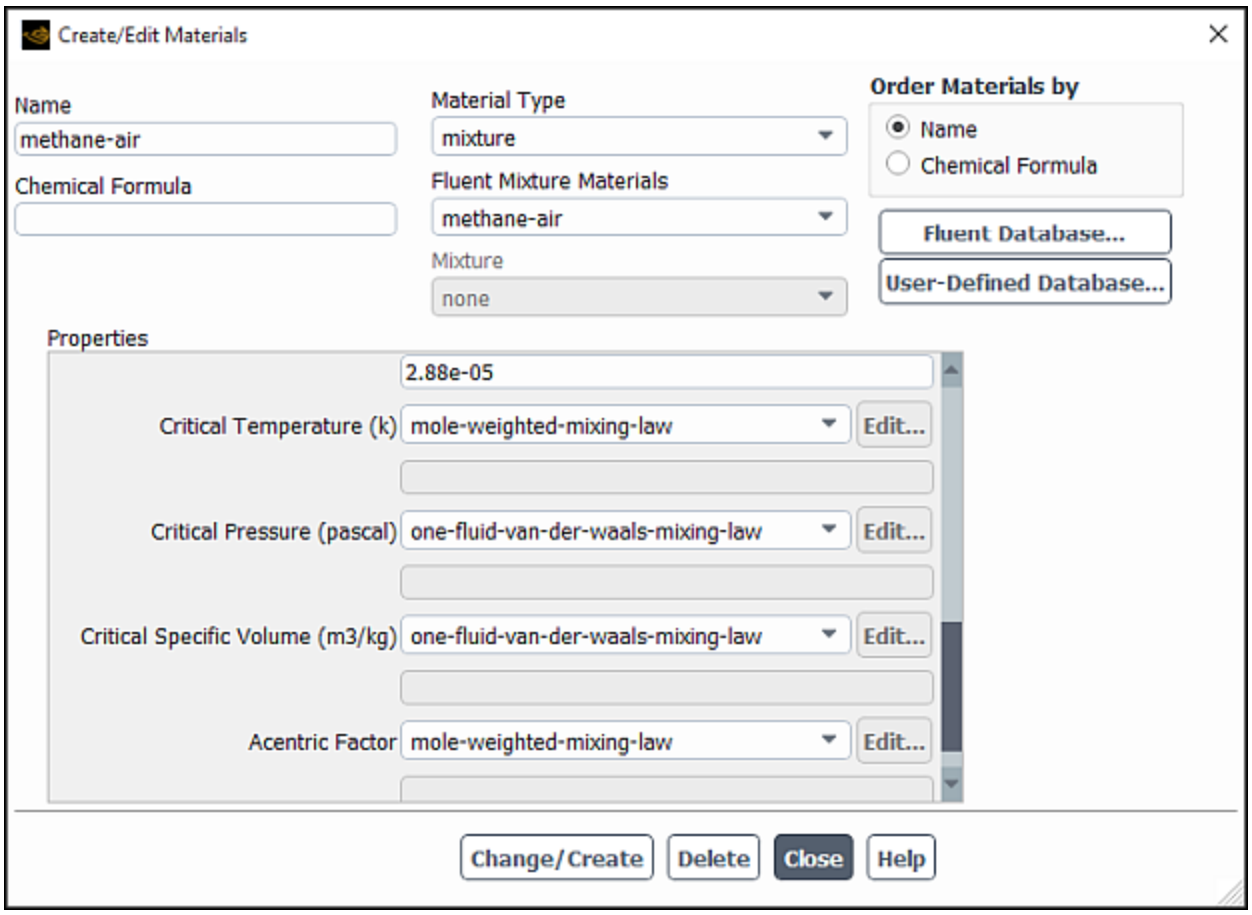
Your inputs for the specific heat in the **Materials** dialog box will now be used to compute the ideal property functions H_{ideal}, $c_{p,ideal}$, and $S_{ideal,0}$ in Equation 8.89 (p. 1688), Equation 8.93 (p. 1688), and Equation 8.95 (p. 1689), respectively. In Ansys Fluent the de-

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

parture properties will be computed and added to the ideal part, to yield the real gas specific heat, enthalpy, and entropy.

Mixtures

Figure 8.42: The Cubic Equation of State Model for a Real-Gas Mixture



When one of the cubic equation of state models is selected from the **Density** drop-down list, specify the following material properties for the mixture material in the dialog box:

- ideal specific heat
- critical temperature
- critical pressure
- critical specific volume
- acentric factor

You also need to enter the following material properties for each of the mixture components in the dialog box:

- molecular weight
- standard state entropy

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- reference temperature

When you are modeling a real-gas mixture, the following methods are available for the mixture critical constants:

- **constant**: defines a constant critical temperature, critical pressure, critical specific volume, or acentric factor for the mixture material.
- **mole-weighted-mixing-law**: applies Equation 8.97 (p. 1690) for critical temperature, critical pressure, critical specific volume, or acentric factor for the mixture material.
- **one-fluid-van-der-waals-mixing-law** applies Equation 8.100 (p. 1691) and Equation 8.101 (p. 1691) for the mixture critical temperature (depending on the real-gas model), Equation 8.102 (p. 1691) for the mixture critical pressure, and Equation 8.103 (p. 1691) for the mixture molar volume.

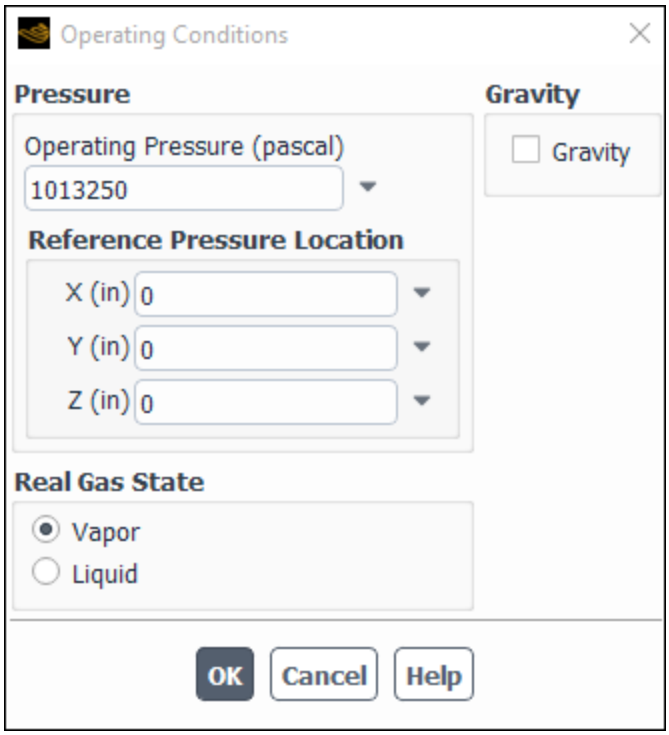
Important:

- Ensure to click the **Change/Create** button so that all the above mentioned properties are uploaded and are visible in the interface.
- If you have selected **mixing-law** for the mixture ideal specific heat you will also need to enter the ideal specific heat values for the individual mixture components. If you have not selected **constant** as the option for any of the critical properties, you must enter the corresponding pure component critical properties for the mixture components.

If the operating conditions in your model are in the subcritical regime, select **Vapor** or **Liquid** as the **Real Gas State** in the **Operating Conditions** dialog box (Figure 8.43: The Operating Conditions for a Real Gas State (p. 1696)). Alternatively, you can use the `define/operating-conditions/set-state` text command. Note that the default is **Vapor** and therefore **Vapor** is assumed if no changes are made to the **Real Gas State** settings. If the operating conditions in your model are entirely in the supercritical regime, this setting will have no effect.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 8.43: The Operating Conditions for a Real Gas State



In addition, you may specify the state for a specific fluid zone or a specific phase (if you are using multiphase modeling) by typing the following text command at the Ansys Fluent console prompt:

```
> define boundary-conditions modify-zones change-zone-state
Select a name/id from fluid zones list [(liquid-zone fluid-1)] liquid-zone
Set zone real-gas state:
  -1:use global setting
  0:liquid
  1:vapor
[-1]  0
```

8.16.3.5.2. Solution Strategies and Considerations for Cubic Equations of State Real Gas Models

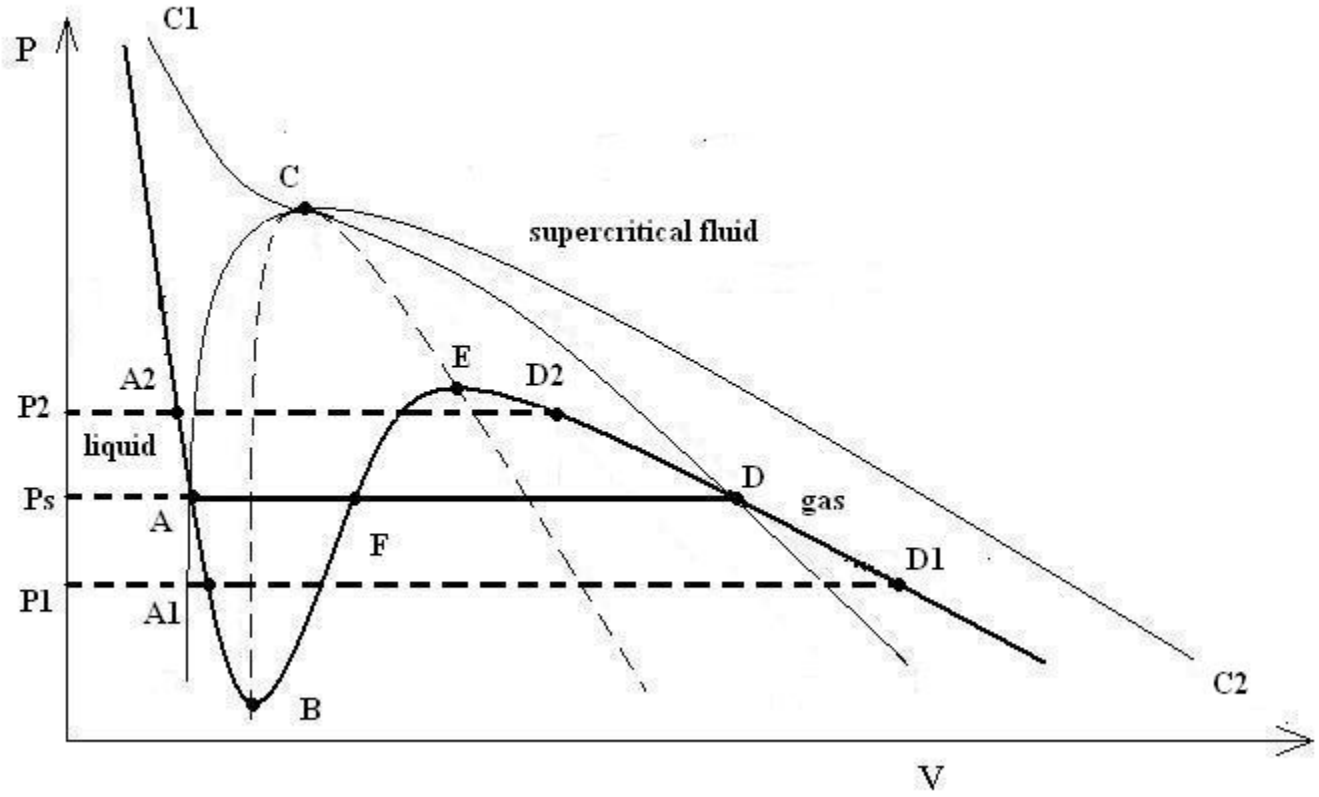
The flow modeling of real-gas flow is more complex and challenging than simple ideal-gas flow. Therefore, the solution might converge at a slower rate with real-gas flow than when running ideal-gas flow. It is recommended that you first attempt to converge your solution using first-order discretization then switch to second-order discretizations and re-iterate to convergence.

Special considerations apply if the operating regime in your model is fully or partly subcritical, where the physical state may be vapor, liquid, or a vapor/liquid mixture. In the cubic EOS real-gas model, the phase state is determined by the selection of the cubic root to calculate the molar-volume. This is illustrated in Figure 8.44: The PV Diagram for the Cubic Equation of State Real Gas Model (p. 1697), which shows a pressure versus molar volume (PV) diagram, with a sub-critical isotherm ABFED at temperature T calculated from a cubic EOS. Curve C1CC2 represents the critical isotherm and curve ACD the saturation dome. Points A, F, and D represent the three roots of the EOS at the saturation pressure Ps, where A corresponds to the molar volume of saturated liquid, D to the molar volume of saturated vapor and point F does not have a physical significance. Points A1 and D1 correspond to the liquid and vapor molar volumes at pressure P1, which is lower than the saturation pressure. The liquid and vapor molar volumes at pressure P2,

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

which is higher than the saturation pressure, are marked as A2 and D2 respectively. Points B and E, where the partial derivatives of pressure with respect to volume $\left(\partial P/\partial V\right)_T$ are 0, are called spinodal points. The loci of these points for all subcritical temperatures, the spinodal curve, is shown with the dashed curve BCE and sets the boundary beyond which the equation of state is no longer valid, because the local derivative of pressure with respect to volume becomes positive. State points inside the dome, up to the spinodal curve, are called "metastable" because normally they only exist temporarily in small local regions until phase change occurs.

Figure 8.44: The PV Diagram for the Cubic Equation of State Real Gas Model



It is important to realize that the current implementation of the cubic equations of state real gas model does not determine the saturation conditions and does not model the two-phase flow where liquid and vapor coexist. In the subcritical regime and for conditions where the cubic EOS has three roots, the phase state selection is controlled by your input of **Vapor** or **Liquid** for **Real Gas State** in the **Operating Conditions** dialog box (Figure 8.43: The Operating Conditions for a Real Gas State (p. 1696)). The default setting is **Vapor**, which corresponds to the state points to the right of the vapor spinodal. With reference to Figure 8.44: The PV Diagram for the Cubic Equation of State Real Gas Model (p. 1697), for the operating point at temperature T and pressure P1, if the **Real Gas State** is set to **Vapor**, the molar volume at point D1, which corresponds to superheated vapor, will be selected in the calculation. On the other hand, if you have selected **Liquid**, the molar volume of point A1 will be taken, and the corresponding state will be metastable super-heated liquid. Similarly for pressure P2, which is higher than the saturation pressure Ps, if you select **Liquid** for **Real Gas State**, the liquid molar volume at A2 will be computed, and if you select **Vapor** the molar volume will correspond to the subcooled vapor state point D2.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In case the calculations fall inside the saturation dome, the solver will not limit the solution, but if conditions are predicted that extend beyond the vapor spinodal curve a warning will be issued:

```
temperature is below the spinodal point in 12 cells on zone 3.
```

Cubic equations of state real gas models are not available for two-phase flows but can be applied to conditions of supercritical pressure $P>P_c$ and subcritical temperature $T<T_c$, where the fluid is in the liquid state, and the supercritical liquid co-exists with gas and supercritical fluid in the same simulation. In those cases, phase change may take place, without any of the flow conditions falling inside the saturation dome.

For multicomponent simulations, when **Diffusion Energy Source** is enabled in the **Species Model** dialog box, the species energy diffusion is by default suppressed in the supercritical liquid regime and across the gas-liquid boundary. The diffusion energy source can be included in the liquid regime using the `define/models/species/liquid-energy-diffusion?` text command.

8.16.3.5.3. Using the Cubic Equation of State Models with the Lagrangian Dispersed Phase Models

If your simulation contains a liquid stream, the appropriate modeling approach in the various operating condition regimes is as follows:

1. For $T>T_c$ a liquid phase does not exist.

2. In the region $P>P_c$, you can define the flow streams directly in the boundary conditions by setting the appropriate pressure and temperature and the properties will be computed directly by the cubic EOS.

3. For $P<P_c$ or $P>P_c$, a stream can exist in liquid phase when its temperature $T<T_c$. If you would like to model phase change in this regime, the following recommendations apply:

   - The DPM droplet and multicomponent models are adequate and recommended for the conditions away from the critical point. This regime can be defined as $T<0.9T_c$, where the liquid physical properties can be assumed independent of pressure.

   - The region $T_c>T>0.9T_c$ is characterized by near-critical-point phenomena, such as strong liquid density and specific heat dependence on both temperature and pressure. The DPM model can also be used in this regime, but you should be cautious, as it will not take into consideration the near-critical-point behavior. In addition, the applicability of the evaporation and boiling rate equations is questionable in this regime.

   - If the pressure in your model is above the critical point, it is recommended to use the **compressible-liquid** method for **Density** of the droplet material.

The temperature $T_{\lim}=0.9T_c$ gives an indication of the maximum droplet temperature for applicability of the DPM models for each droplet material. These temperature limits are listed in Table 8.2: Temperature Limits for Droplet Materials in Ansys Fluent Database prodb.scm (p. 1699) for many of the droplet materials in Ansys Fluent's `propdb.scm` materials database. For super-

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

critical pressures ($P>P_c$), the DPM models can still be used provided the droplet temperatures remain below the limit T_{lim}.

Table 8.2: Temperature Limits for Droplet Materials in Ansys Fluent Database prodb.scm

Material	Tc (K)	Pc (MPa)	Normal BP (K)	Tlim (K)
Argon	150.86	4.89	87.30	135.77
Benzene	562.00	4.89	353.00	505.80
Helium	5.30	0.23	4.20	4.77
Hydrogen	32.98	1.30	20.40	29.68
Methyl-alcohol	512.00	8.10	338.00	460.80
Heptanes	540.00	2.74	371.00	486.80
Hexane	507.00	3.02	342.00	456.30
Octane	569.00	2.49	399.00	512.10
Pentane	470.00	3.37	309.00	423.00
Nitrogen	126.00	3.40	77.40	113.40
Oxygen	154.00	5.04	90.20	138.60
Toluene	592.00	4.11	384.00	532.80
Water	647.00	22.00	373.00	582.30

For high pressure simulations the boiling point will be different from the normal boiling point, and for varying pressure applications the boiling point will vary with the droplet location in the domain.

When a cubic equation of state real gas model is enabled in a simulation that includes evaporating droplet particles, the boiling point is calculated from the vapor pressure data directly, as the temperature where the saturation vapor pressure equals the domain pressure. In addition, the latent heat of the evaporating or boiling droplet will vary with the droplet temperature.

The latent heat at temperature T_p is given by

$$H_{lat_T} = -\int_{T_p}^{T_{bp}} c_{p,g} dT + H_{lat} + \int_{T_p}^{T_{bp}} c_{p,p} dT \tag{8.104}$$

where T_{bp} is the normal boiling point and H_{lat} is the latent heat at the normal boiling point.

In the **Create/Edit Materials** dialog box you must enter the **Normal Boiling Point (NBP)** and the **Latent Heat at NBP** for the **droplet-particle Material Type**. These inputs will be used for calculating the **Latent Heat** at the reference temperature (see Equation 12.509) and the latent heat in the droplet energy balance during vaporization and boiling according to Equation 8.104 (p. 1699).

Important:

The constant property option is disabled for the **Saturation Vapor Pressure** property when a real gas model is used in the simulation. Also, ensure to enter

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the appropriate droplet saturation vapor pressure data to cover the complete pressure/temperature range in your model.

Finally when a cubic equation of state real-gas model is enabled in a simulation with droplet models, the condition for switching from the vaporization to the boiling law will be

$$P_{sat} > P \tag{8.105}$$

where P_{sat} is the saturation vapor pressure and P is the domain pressure. If $P_{sat} < P$ while in the boiling law, the model will switch back to the vaporization law. Under supercritical pressure conditions ($P > P_c$), the vaporization models are applicable at droplet temperatures below the critical point, and the switching condition from vaporization to boiling $P_{sat} > P_c$ is never met because the vapor pressure curve is defined only up to the critical point.

For the multicomponent droplets in the supercritical pressure regime, failure of the saturation temperature calculation algorithm may indicate that the droplet has approached the critical temperature, provided that you have entered accurate vapor pressure data for all components. In such a case, you can use the following text command:

```
define/models/dpm/options/treat-multicomponent-saturation-temperature-
failure?
```

```
Dump multicomponent particle mass if the saturation temperature
calculation fails? [no] y
```

When this option is enabled, Ansys Fluent dumps the particle mass into the continuous phase if the saturation temperature calculation fails.

8.16.3.5.4. Postprocessing the Cubic Equations of State Real Gas Model

All postprocessing functions properly report and display the current thermodynamic and transport properties of the selected real gas model. The thermodynamic and transport properties controlled by the cubic equations of state real gas model include the following:

- density
- enthalpy
- entropy
- sound speed
- specific heat
- any quantities that are derived from the properties listed above (for example, total quantities, ratio of specific heats)

In addition to the properties listed above, you can also report

- compressibility factor
- reduced temperature
- reduced pressure

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- spinodal temperature
- subcritical condition

If you are modeling a real-gas mixture you can report the composition-dependent mixture critical properties

- critical temperature
- critical pressure
- critical specific volume
- acentric factor

8.16.4. The NIST Real Gas Models

The NIST real gas models use the National Institute of Standards and Technology (NIST) Thermodynamic and Transport Properties of Refrigerants and Refrigerant Mixtures Database Version 10.0 (REFPROP v10.0) to evaluate thermodynamic and transport properties of approximately 152 fluids or a mixture of these fluids.

The REFPROP v10.0 database is a shared library that is dynamically loaded into the solver when you enable one of the NIST real gas models in an Ansys Fluent session. Once the NIST real gas model is activated, control of relevant property evaluations is relinquished to the REFPROP database. All postprocessing functions will properly report and display the current thermodynamic and transport properties of the real gas.

8.16.4.1. Limitations of the NIST Real Gas Models

The following limitations exist for the NIST real gas model:

- The NIST real gas model assumes that the fluid you will be using in your Ansys Fluent computation is superheated vapor, supercritical fluid, or liquid. Note that subcritical flow conditions, where vapor coexists with liquid in two-phase flow, are not supported. In addition, all fluid zones must contain the real gas; you cannot include a real gas and another fluid in the same problem.
- Pressure-inlet, mass flow-inlet, and pressure-outlet are the only inflow and outflow boundaries available for use with the real gas models.
- Non-reflecting boundary conditions are not compatible with the real gas models when using the density-based solver. If your model requires NRBC and a real gas model, you must use the pressure-based solver.
- The mixture flow does not permit chemical reactions with the NIST real gas model.
- The multicomponent NIST real gas models cannot be used with any of the multiphase models. You can, however, use a NIST material in single-component multiphase flows. The model is compatible with the Lagrangian Dispersed Phase Models only for the massless and inert particle types.
- You cannot modify material properties in the REFPROP database libraries, or add custom materials to the NIST real gas model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Only one pure fluid or one multiple-species mixture in a simulation may use the NIST real gas model. Multiple NIST materials may be created, but those participating in the simulation must all refer to a unique NIST fluid. This is a third-party limitation of the NIST/REFPROP module.

8.16.4.2. The REFPROP v10.0 Database

The NIST real gas model supports 152 fluids from the REFPROP database. These include pure fluids (with the file extension .FLD) and pseudo-fluids (with the file extension .PPF). The fluids that are supported by REFPROP v10.0 and used in the NIST real gas model are listed in Table 8.3: v10.0(p. 1702) (the corresponding property file name appears in parentheses, where it does not coincide with the fluid name).

The REFPROP v10.0 database employs accurate pure-fluid equations of state that are available from NIST. These equations are based on three models:

- modified Benedict-Webb-Rubin (MBWR) equation of state
- Helmholtz-energy equation of state
- extended corresponding states (ECS)

For a fluid that consists of a multispecies-mixture the thermodynamic properties are computed by employing mixing-rules applied to the Helmholtz energy of the mixture components.

Important:

The database does not include transport property models for the species marked with * in Table 8.3: v10.0(p. 1702). As a result the NIST real gas model with those species can only be used for modeling inviscid flow.

Table 8.3: v10.0

Short Name	Full Name	Fluid Data File
Butene	1-Butene	1BUTENE.FLD
1-Butyne	But-1-yne	1BUTYNE.FLD
1-Pentene	Pent-1-en	1PENTENE.FLD
3-Methylpentane	3-Methylpentane	3METHYLPENTANE.FLD
1,3-Butadiene	Buta-1,3-diene	13BUTADIENE.FLD
2,2-Dimethylbutan	2,2-Dimethylbutan	22DIMETHYLBUTANE.FLD
2,3-Dimethylbutan	2,3-Dimethylbutan	23DIMETHYLBUTANE.FLD
Acetone	Propanone	ACETONE.FLD
Acetylene	Ethyne	ACETYLENE.FLD
Ammonia	Ammonia	AMMONIA.FLD
Argon	Argon	ARGON.FLD
Benzene	Benzene	BENZENE.FLD
Butane	n-Butane	BUTANE.FLD
Methylcyclohexane	Methylcyclohexane	C1CC6.FLD

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Short Name	Full Name	Fluid Data File
cis-Butene	cis-2-Butene	C2BUTENE.FLD
Propylcyclohexane	n-Propylcyclohexane	C3CC6.FLD
Perfluorobutane	Decafluorobutane	C4F10.FLD
Perfluoropentane	Dodecafluoropentane	C5F12.FLD
Perfluorohexane	Tetradecafluorohexane	C6F14.FLD
Undecane	Undecane	C11.FLD
Dodecane	Dodecane	C12.FLD
Hexadecane	Hexadecane	C16.FLD
Docosane	Docosane	C22.FLD
R13I1	Trifluoroiodomethane	CF3I.FLD
Chlorine	Chlorine	CHLORINE.FLD
Chlorobenzene	Chlorobenzene	CHLOROBENZENE.FLD
Carbon monoxide	Carbon monoxide	CO.FLD
Carbon dioxide	Carbon dioxide	CO2.FLD
Carbonyl sulfide	Carbon oxide sulfide	COS.FLD
Cyclobutene	1-Cyclobutene	CYCLOBUTENE.FLD
Cyclohexane	Cyclohexane	CYCLOHEX.FLD
Cyclopentane	Cyclopentane	CYCLOPEN.FLD
Cyclopropane	Cyclopropane	CYCLOPRO.FLD
Deuterium	Deuterium	D2.FLD
Heavy water	Deuterium oxide	D2O.FLD
D4	Octamethylcyclotetrasiloxane	D4.FLD
D5	Decamethylcyclopentasiloxane	D5.FLD
D6	Dodecamethylcyclohexasiloxane	D6.FLD
Diethanolamine	2,2'-Iminodiethanol	DEA.FLD
Decane	Decane	DECANE.FLD
Diethyl ether	Diethyl ether	DEE.FLD
Dimethyl carbonate	Dimethyl ester carbonic acid	DMC.FLD
Dimethyl ether	Methoxymethane	DME.FLD
Ethylbenzene	Phenylethane	EBENZENE.FLD
Ethylene glycol	1,2-Ethandiol	EGLYCOL.FLD
Ethane	Ethane	ETHANE.FLD
Ethanol	Ethyl alcohol	ETHANOL.FLD
Ethylene	Ethene	ETHYLENE.FLD
Ethylene oxide	Ethylene oxide	ETHYLENEOXIDE.FLD
Ethylene oxide	Fluorine	FLUORINE.FLD
Hydrogen sulfide	Hydrogen sulfide	H2S.FLD
Hydrogen chloride	Hydrogen chloride	HCL.FLD

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Short Name	Full Name	Fluid Data File
Helium	Helium-4	HELIUM.FLD
Heptane	Heptane	HEPTANE.FLD
Hexane	Hexane	HEXANE.FLD
Hydrogen (normal)	Hydrogen (normal)	HYDROGEN.FLD
Isobutene	2-Methyl-1-propene	IBUTENE.FLD
Isohexane	2-Methylpentane	IHEXANE.FLD
Isooctane	2,2,4-Trimethylpentane	IOCTANE.FLD
Isopentane	2-Methylbutane	IPENTANE.FLD
Isobutane	2-Methylpropane	ISOBUTAN.FLD
Krypton	Krypton	KRYPTON.FLD
MD2M	Decamethyltetrasiloxane	MD2M.FLD
MD3M	Dodecamethylpentasiloxane	MD3M.FLD
MD4M	Tetradecamethylhexasiloxane	MD4M.FLD
MDM	Octamethyltrisiloxane	MDM.FLD
Monoethanolamine	Ethanolamine	MEA.FLD
Methane	Methane	METHANE.FLD
Methanol	Methanol	METHANOL.FLD
Methyl linoleate	Methyl (Z,Z)-9,12-octadecadienoate	MLINOLEA.FLD
Methyl linolenate	Methyl (Z,Z,Z)-9,12,15-octadecatrienoate	MLINOLEN.FLD
MM	Hexamethyldisiloxane	MM.FLD
Methyl oleate	Methyl cis-9-octadecenoate	MOLEATE.FLD
Methyl palmitate	Methyl hexadecanoate	MPALMITA.FLD
Methyl stearate	Methyl octadecanoate	MSTEARAT.FLD
m-Xylene	1,3-Dimethylbenzene	MXYLENE.FLD
Nitrous oxide	Dinitrogen monoxide	N2O.FLD
Neon	Neon	NEON.FLD
Neopentane	2,2-Dimethylpropane	NEOPENTN.FLD
Nitrogen trifluoride	Nitrogen trifluoride	NF3.FLD *
Nitrogen	Nitrogen	NITROGEN.FLD
Nonane	Nonane	NONANE.FLD
Novec 649, 1230	1,1,1,2,2,4,5,5,5-Nonafluoro-4-(trifluoromethyl)-3-pentanone	NOVEC649.FLD
Octane	Octane	OCTANE.FLD
Orthohydrogen	Orthohydrogen	ORTHOHYD.FLD *
Oxygen	Oxygen	OXYGEN.FLD
o-Xylene	1,2-Dimethylbenzene	OXYLENE.FLD
Parahydrogen	Parahydrogen	PARAHYD.FLD

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Short Name	Full Name	Fluid Data File
Pentane	Pentane	PENTANE.FLD
Propadiene	1,2-Propadiene	PROPADIENE.FLD
Propane	Propane	PROPANE.FLD
Propylene	Propene	PROPYLEN.FLD
Propylene oxide	1,2-Epoxypropane	PROPYLENEOXIDE.FLD
Propyne	Propyne	PROPYNE.FLD
p-Xylene	1,4-Dimethylbenzene	PXYLENE.FLD
R11	Trichlorofluoromethane	R11.FLD
R12	Dichlorodifluoromethane	R12.FLD
R13	Chlorotrifluoromethane	R13.FLD
R14	Tetrafluoromethane	R14.FLD
R21	Dichlorofluoromethane	R21.FLD
R22	Chlorodifluoromethane	R22.FLD
R23	Trifluoromethane	R23.FLD
R32	Difluoromethane	R32.FLD
R40	Methyl chloride	R40.FLD
R41	Fluoromethane	R41.FLD
R113	1,1,2-Trichloro-1,2,2-trifluoroethane	R113.FLD
R114	1,2-Dichloro-1,1,2,2-tetrafluoroethane	R114.FLD
R115	Chloropentafluoroethane	R115.FLD
R116	Hexafluoroethane	R116.FLD
R123	2,2-Dichloro-1,1,1-trifluoroethane	R123.FLD
R124	1-Chloro-1,2,2,2-tetrafluoroethane	R124.FLD
R125	Pentafluoroethane	R125.FLD
R134a	1,1,1,2-Tetrafluoroethane	R134A.FLD
R141b	1,1-Dichloro-1-fluoroethane	R141B.FLD
R142b	1-Chloro-1,1-difluoroethane	R142B.FLD
R143a	1,1,1-Trifluoroethane	R143A.FLD
Dichloroethane	1,2-Dichloroethane	R150.FLD
R152a	1,1-Difluoroethane	R152A.FLD
R161	Fluoroethane	R161.FLD
R218	Octafluoropropane	R218.FLD
R227ea	1,1,1,2,3,3,3-Heptafluoropropane	R227EA.FLD
R236ea	1,1,1,2,3,3-Hexafluoropropane	R236EA.FLD
R236fa	1,1,1,3,3,3-Hexafluoropropane	R236FA.FLD
R245ca	1,1,2,2,3-Pentafluoropropane	R245CA.FLD
R245fa	1,1,1,3,3-Pentafluoropropane	R245FA.FLD
R365mfc	1,1,1,3,3-Pentafluorobutane	R365MFC.FLD

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Short Name	Full Name	Fluid Data File
R1123	Trifluoroethylene	R1123.FLD
R1216	Hexafluoropropene	R1216.FLD
R1224yd(Z)	(Z)-1-Chloro-2,3,3,3-tetrafluoropropene	R1224YDZ.FLD
R1233zd(E)	trans-1-Chloro-3,3,3-trifluoro-1-propene	R1233ZDE.FLD
R1234yf	2,3,3,3-Tetrafluoroprop-1-ene	R1234YF.FLD
R1234ze(E	trans-1,3,3,3-Tetrafluoropropene	R1234ZEE.FLD
R1234ze(Z)	cis-1,3,3,3-Tetrafluoropropene	R1234ZEZ.FLD
R1243zf	3,3,3-Trifluoropropene	R1243ZF.FLD
R1336mzz(Z)	(Z)-1,1,1,4,4,4-Hexafluoro-2-butene	R1336MZZZ.FLD
RC318	Octafluorocyclobutane	RC318.FLD
RE143a	Methyl trifluoromethyl ether	RE143A.FLD
RE245cb2	Methyl-pentafluoroethyl-ether	RE245CB2.FLD
RE245fa2	2,2,2-Trifluoroethyl-difluoromethyl-ether	RE245FA2.FLD
RE347mcc (HFE-7000)	1,1,1,2,2,3,3-Heptafluoro-3-methoxypropane	RE347MCC.FLD
Sulfur hexafluoride	Sulfur hexafluoride	SF6.FLD
Sulfur dioxide	Sulfur dioxide	SO2.FLD
trans-Butene	trans-2-Butene	T2BUTENE.FLD
Toluene	Methylbenzene	TOLUENE.FLD
Vinyl chloride	Chloroethylene	VINYLCHLORIDE.FLD
Water	Water	WATER.FLD
Xenon	Xenon	XENON.FLD
Air (dry)	Nitrogen + oxygen + argon (0.7812/0.2096/0.0092)	AIR.PPF
R404A	44% R125/4% R134a/52% R143a	R404A.PPF *
R407C	23% R32/25% R125/52% R134a	R407C.PPF *
R410A	50% R32/50% R125	R410A.PPF *
R507A	50% R125/50% R143a	R507A.PPF *

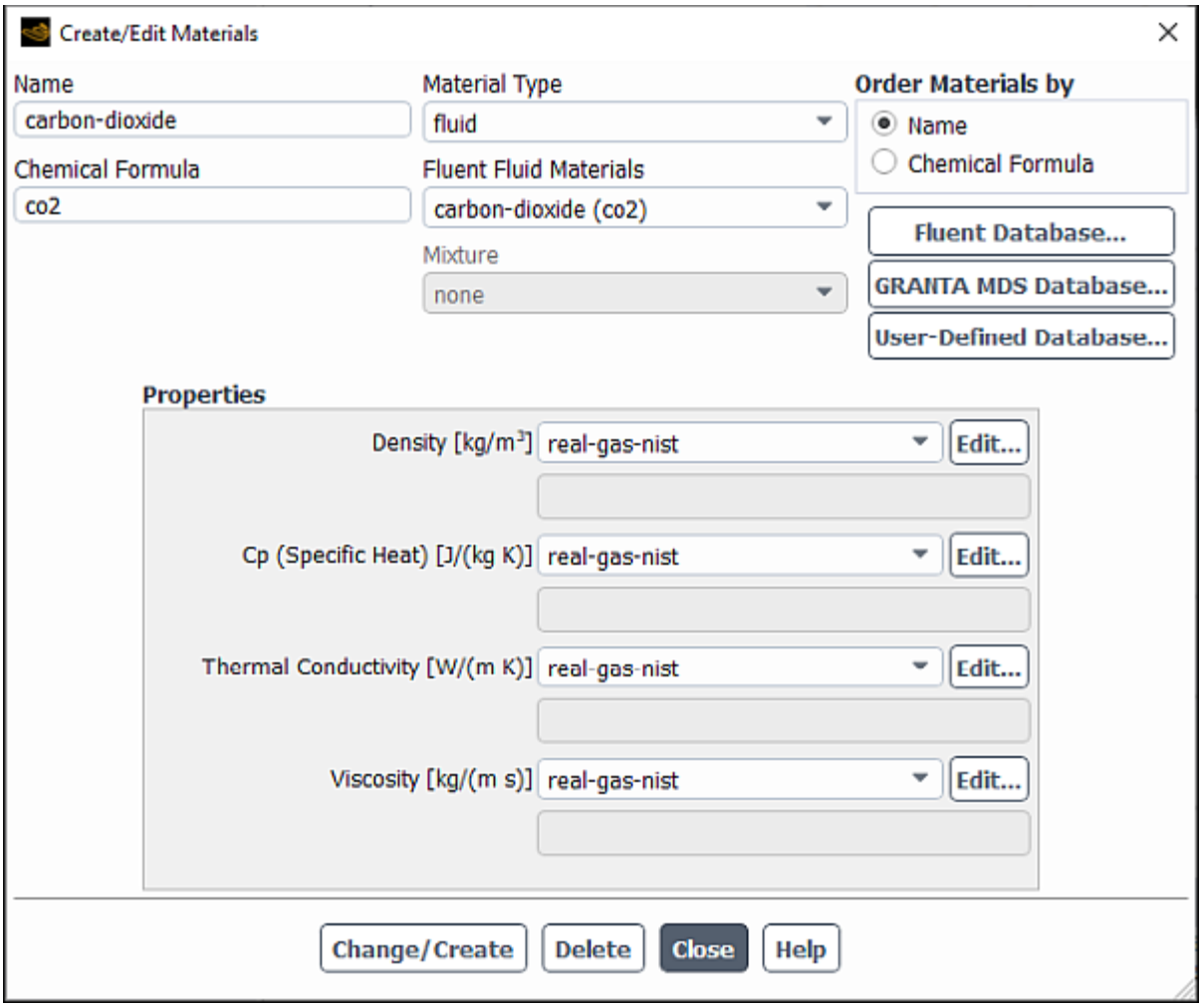
8.16.4.3. Using the NIST Real Gas Models

The NIST real gas model can be enabled for pure fluids by selecting `real-gas-nist` for **Density** on the **Create/Edit Materials** dialog box.

Physics → Materials → Create/Edit...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

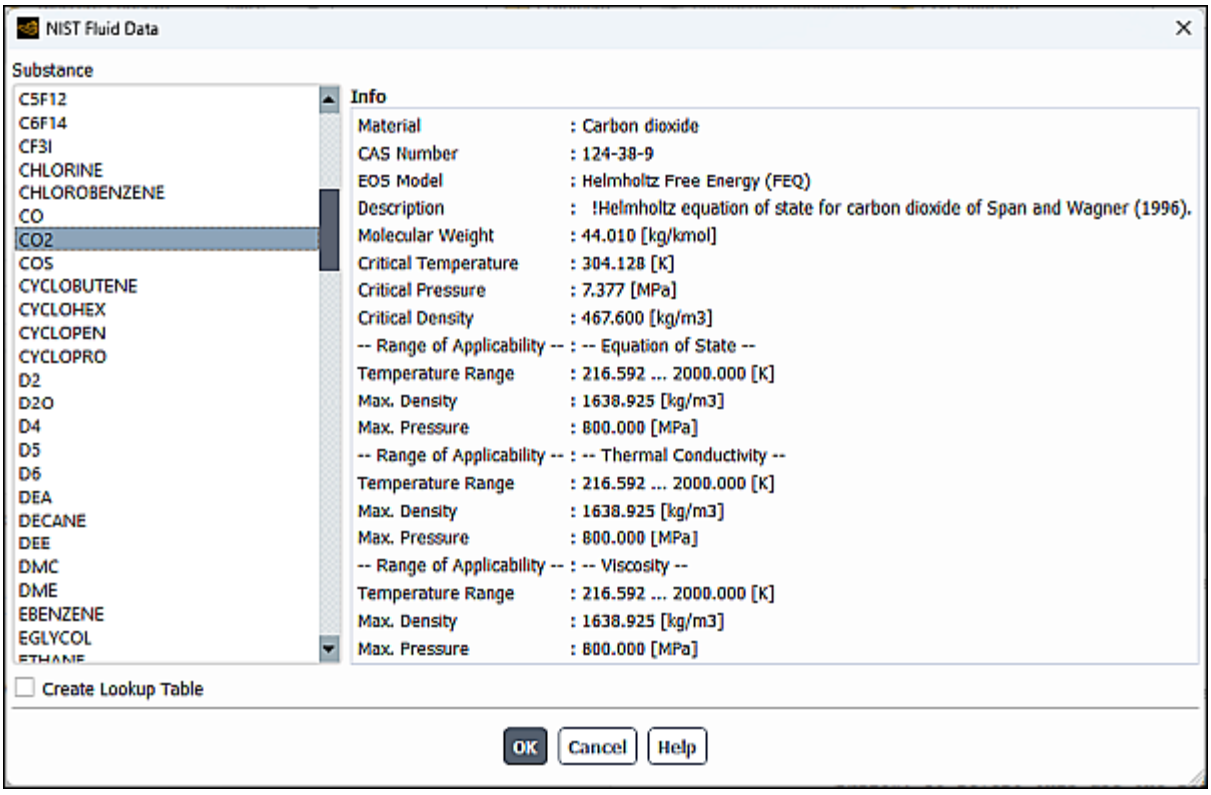
Figure 8.45: The Create/Edit Materials Dialog Box with NIST Model Enabled



When you select **Edit...** for **Density**, you will be prompted to select the appropriate NIST data from the **NIST Fluid Data** dialog box. After selecting a valid fluid, properties for the specified material, including its applicable pressure and temperature ranges, will be displayed.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

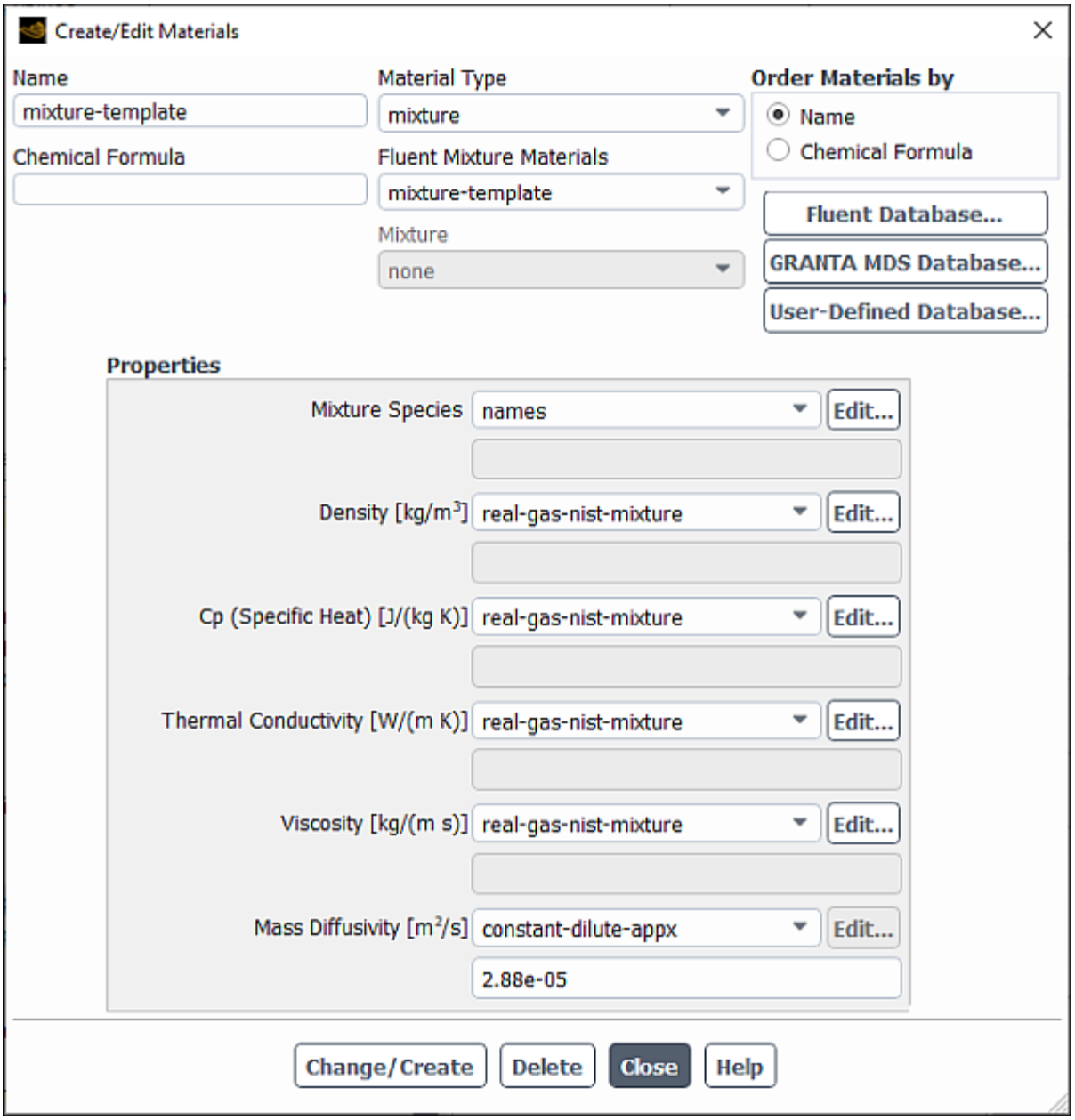
Figure 8.46: The NIST Fluid Data Dialog Box



For multiple-species mixtures, enable the **Species Transport** model and specify the number and names of species, as described in Enabling Species Transport and Reactions and Choosing the Mixture Material (p. 2359). Then on the **Create/Edit Materials** dialog box, select **real-gas-nist-mixture** for **Density**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 8.47: The Create/Edit Materials Dialog Box with NIST Mixture Model Enabled



Afterwards, you must select the appropriate NIST fluid data for each species as you would for a pure fluid.

Important:

- For mixture flows, not all combinations of species mixtures are allowed. This could be due to lack of data for one or more binary pairs. In such situations an error message generated by NIST will be returned and displayed on the Ansys Fluent console, and no real gas material is allowed to be created. In some combinations the mixing data will be estimated, a warning message will be displayed on the Ansys Fluent console and the mixture material allowed to be created.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

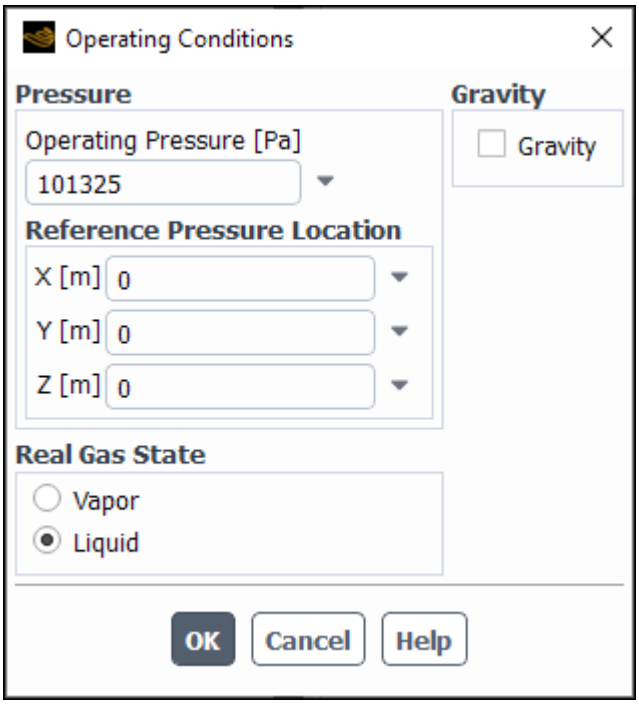
- Transport property calculations are not supported for mixtures that include water with molar concentration over 5%. If this limit is exceeded in your calculation, Ansys Fluent will issue a warning message. Transport equations are not available for the ammonia/water mixture and for mixtures with alcohols. As a result, the NIST real gas model with those mixtures can only be used for modeling inviscid flow.

Cp (Specific Heat), **Thermal Conductivity**, and **Viscosity** will be automatically set respectively to real-gas-nist (pure fluid) or real-gas-nist-mixture (multiple species mixture).

If you want to use NIST lookup tables in your simulation, you can create them as described in Creating NIST Look-up Tables (p. 1710).

If your case contains no NIST tables, and you would like to model flow in the liquid state, you can set the **Real Gas State** to liquid in the **Operating Conditions** dialog box.

Figure 8.48: Operating Conditions dialog Box with Liquid State Enabled



In addition, you may specify the state for specific fluid zones or a specific phase (if you are using the multiphase model) by typing the following text command in the Ansys Fluent console prompt:

```
> define boundary-conditions modify-zones change-zone-state
Select a name/id from fluid zones list [(liquid-zone fluid-1)] liquid-zone
Set zone real-gas state:
  -1:use global setting
  0:liquid
  1:vapor
[-1]  0
```

8.16.4.3.1. Creating NIST Look-up Tables

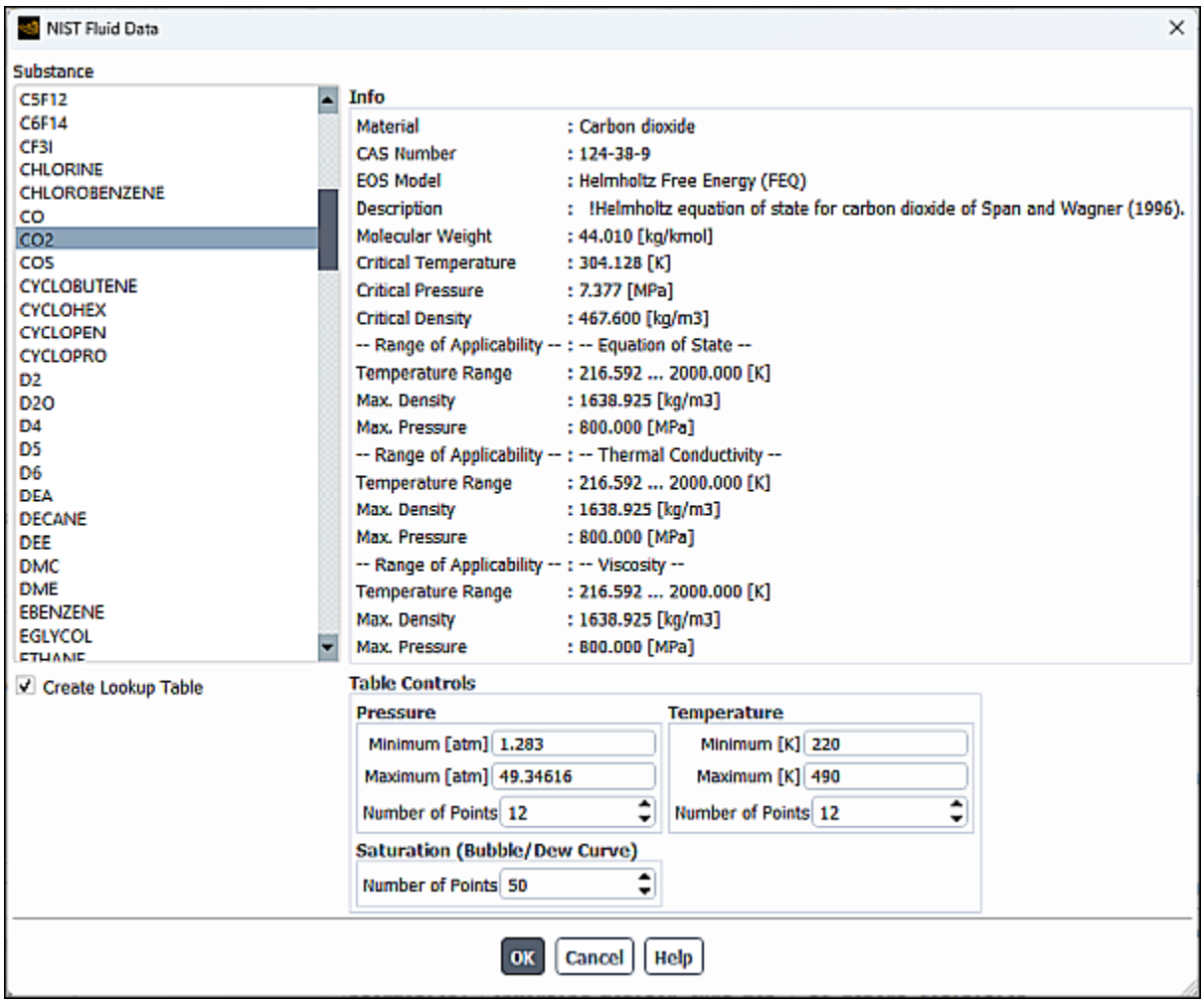
By default, the NIST functions are called directly each time the thermal properties are updated during an iteration. Alternatively, you can use NIST look-up tables in your simulation as described

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

below. Computing the look-up table reduces the computation time because the NIST functions do not need to be called each time the thermal properties are updated.

To create a full NIST look-up table, first enable the NIST real gas model as described in Using the NIST Real Gas Models (p. 1706), and then select **Create Lookup Table**.

Figure 8.49: The NIST Fluid Data Dialog Box With Lookup Table Enabled



For both single and multi-species real gas models, you need to specify:

- **Pressure** and **Temperature** bounds for the table.

  You must ensure that the temperature and pressure bounds you specify are strictly within the stated limits of applicability that are displayed when selecting the NIST fluid data.

- Number of pressure and temperature points.

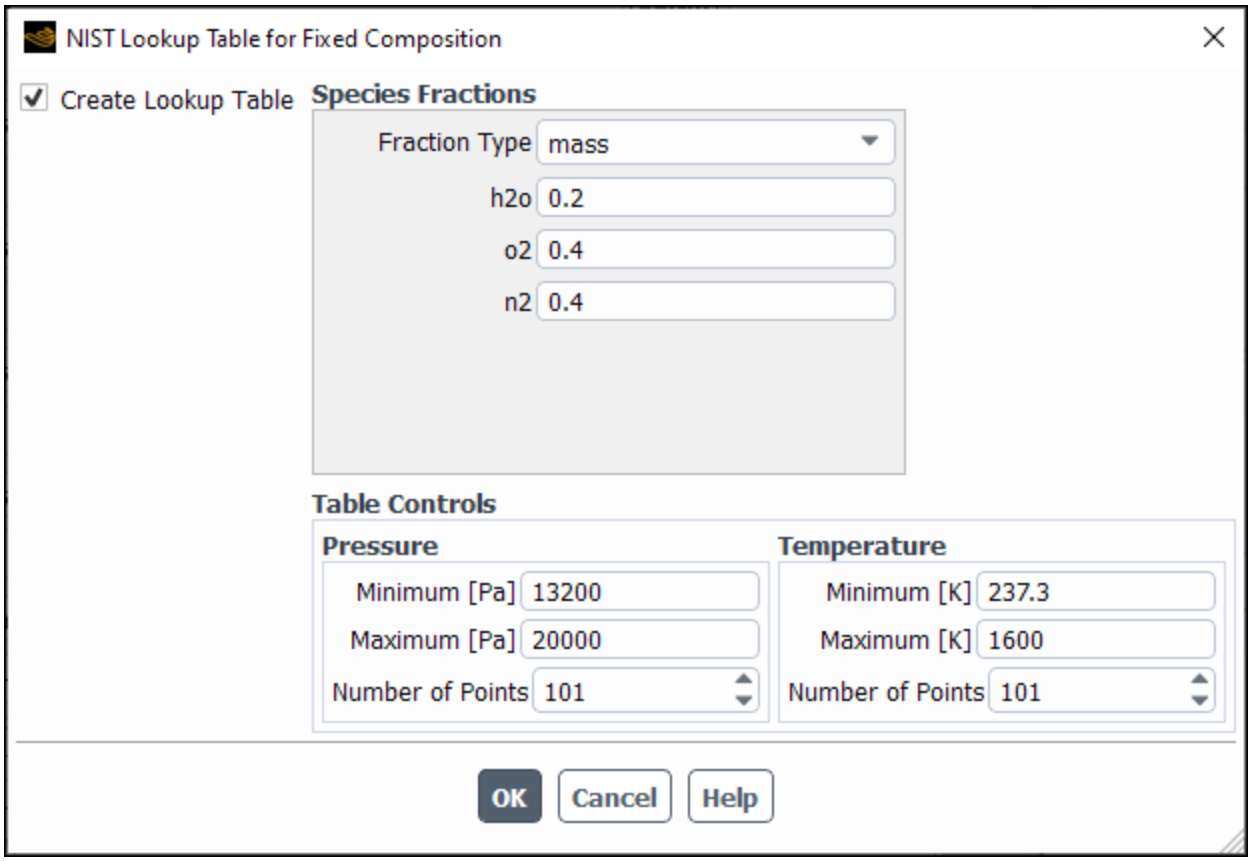
  The minimum number of pressure and temperature points is 12 and there is no upper limit. However, the larger the table, the greater the memory and computing time requirements. It is advised to perform test runs to balance the costs (memory and computing time) and accuracy.

For single-species, you also need to specify the number of points for **Saturation (Bubble/Dew Curve)** properties.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For multi-species, you also need to specify the mixture composition on a mass or mole basis. You must ensure that the sum of all constituent fractions add up to 1 when specifying mass/molar fractions for the components. The maximum number of species is 20.

Figure 8.50: The NIST Fluid Data Dialog Box For Fixed Composition



If properties are needed at conditions outside the limits you have specified for the table, Fluent automatically calls the original NIST model functions to calculate the thermal properties. If more than 10% of the property evaluations require computation outside the look-up table, Fluent prints a message indicating the percentage of calculations using direct computation. This can be used as an indicator to adjust the temperature and pressure ranges for the table to minimize the computational cost.

8.16.4.3.1.1. Capabilities and Limitations with NIST Look-up Tables

- For computation of saturation properties for the bubble and dew points for a single- or multiple-species, Fluent provides macros that you can use in UDFs. See NIST Real Gas Saturation Properties for more information.
- Look-up tables apply for compressible fluids with material properties that are a function of local pressure and temperature. The material properties covered by the table include fluids in the liquid and vapor phases. Flashing applications between liquid and vapor phases can be conducted using saturation conditions established for the fluid(s) using mass transfer models.
- Look-up tables and saturation tables are created for fixed fluid compositions. This means the look-up tables are based on the following assumptions:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - Mixture composition is constant.
  - No chemical reactions are involved.
  - No mixing of fluids of different compositions, including no mixing between phases.
- Look-up tables only apply to single-species/multi-species in single-phase flows
- The temperature and pressure bounds must be specified within the stated limits of applicability that are displayed when the NIST model is loaded.
- When specifying the look-up table size, it is important to pick temperature and pressure ranges that are large enough to satisfy the current flow conditions.
- If the temperature and pressure ranges you specify are too narrow, Fluent uses the NIST model to calculate thermal properties. The time saved by creating the table will be negated if the percentage of calculations performed outside of the table is greater than 10%. This percentage is printed, so you can use it as an indicator for adjusting the temperature and pressure range of the table.
- The look-up table is a structured table and may not be suitable for simulating flow behaviors near the critical point when higher resolution is required.

8.16.4.4. Legacy TUI for the NIST Real Gas Models

When you enable one of the NIST real gas models (pure fluid or multiple-species mixture) using the legacy TUI and select a valid material, Ansys Fluent's functionality remains the same as when you model fluid flow and heat transfer using an ideal gas, with the exception of the Create/Edit Materials Dialog Box (p. 4885).

8.16.4.4.1. Limitations of the Legacy TUI NIST Real Gas Models

When you enable one of the NIST real gas models (pure fluid or multiple-species mixture) through the legacy TUI, the following limitations apply in addition to those listed in Limitations of the NIST Real Gas Models (p. 1701):

- When you are using the NIST real gas models, the NIST materials defined in your simulation will appear in the Create/Edit Materials Dialog Box (p. 4885) either with the name **real-gas-*_name***, where ***_name*** is the NIST material selected for the single species model, or with the name **real-gas-mixture** for the multiple species model. The inputs for the properties calculated by the NIST functions are disabled. You can use the Create/Edit Materials Dialog Box (p. 4885) to define or modify:
  - Mass diffusivity property in the **real-gas-mixture** material if you are using the multiple-species NIST real gas model (note that the **kinetic-theory** option is not available)
  - Radiation properties if you are modeling radiation
  - Properties of materials other than the NIST **real-gas-mixture** or **real-gas-fluid** materials

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.16.4.4.2. Activating the NIST Real Gas Model

Activating one of the NIST real gas models is a two-step process. First you enable either the single- or multi-species NIST real gas model, and then you select the fluid material from the REFPROP database.

1. Enable the appropriate NIST real gas model by typing the following text command at the Ansys Fluent console prompt:

   - Pure fluid real gas model:

     ```
     > define/user-defined/real-gas-models/nist-real-gas-model
     use NIST real gas? [no] yes
     ```

   - Multi-species real gas model:

     ```
     > define/user-defined/real-gas-models/nist-multispecies-real-gas-model
     use multispecies NIST real gas? [no] yes
     ```

   The list of available pure fluid materials you can select from will be displayed.

2. Select material from the REFPROP database list:

   - Pure fluid real gas model:

     Enter the name of a single pure fluid that you want to investigate as follows:

     ```
     select real-gas data file [""]  methane.fld
     ```

   - Multi-species real gas model:

     Enter the number of desired species in the mixture:

     ```
     Number of species []  3
     ```

     followed by the name of each fluid in quotation marks selected from the list printed in the console:

     ```
     select real-gas data file [""] nitrogen.fld
     select real-gas data file [""] co2.fld
     select real-gas data file [""] methane.fld
     ```

     a. Upon selection of a valid pure or multi-species fluids, Ansys Fluent will load the relevant material data from a library of pure fluids supported by the REFPROP database, and report that it is opening the shared library (`librealgas.so`) where the compiled REFPROP

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

database source code is located. A list of properties for the specified material, including its applicable pressure and temperature ranges, will also be reported in the console.

Important:

- For mixture flows, not all combinations of species mixtures are allowed. This could be due to lack of data for one or more binary pairs. In such situations an error message generated by NIST will be returned and displayed on the Ansys Fluent console, and no real gas material is allowed to be created. In some combinations the mixing data will be estimated, a warning message will be displayed on the Ansys Fluent console and the mixture material allowed to be created.
- Transport property calculations are not supported for mixtures that include water with molar concentration over 5%. If this limit is exceeded in your calculation, Ansys Fluent will issue a warning message. Transport equations are not available for the ammonia/water mixture and for mixtures with alcohols. As a result, the NIST real gas model with those mixtures can only be used for modeling inviscid flow.

3. If you want to use NIST lookup tables in your simulation, you can create them as described in the following sections:
   - Creating Full NIST Look-up Tables (p. 1715) (single and multi-species materials with thermal properties as well as saturation data for a fixed multi-species composition)
   - Creating Binary Mixture Saturation Tables for Binary Mixtures (p. 1718) (binary mixtures only)
4. (cases with no NIST tables only) If you would like to model flow in the liquid state, you can use the `set-state` TUI command. Note that the default state is vapor, so if you do not go through this step, vapor is assumed. Also, if the flow conditions do not permit liquid to exist, a vapor calculation will be performed instead.

```
 > define/user-defined/real-gas-models/set-state
Select vapor state (else liquid)?[yes]
```

   In addition, you may specify the state for a specific fluid zone or a specific phase (if you are using the multiphase model) by typing the following text command at the Ansys Fluent console prompt:

```
> define boundary-conditions modify-zones change-zone-state
Select a name/id from fluid zones list [(liquid-zone fluid-1)] liquid-zone
Set zone real-gas state:
  -1:use global setting
  0:liquid
  1:vapor
[-1]  0
```

8.16.4.4.3. Creating Full NIST Look-up Tables

By default, the NIST functions are called directly each time the thermal properties are updated during iteration. Alternatively, you can use the NIST look-up tables in your simulation as described below. Computing the look-up table offers two advantages:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Reduction in computation time because the NIST functions do not need to be called each time the thermal properties are updated
- The look-up table created is applicable to both liquid and vapor phases

To create a full NIST look-up table, first enable the NIST real gas model as described in Activating the NIST Real Gas Model (p. 1714), and then follow these steps:

1. When prompted with `Create Full NIST LookUp Tables for multi components? [no]`, answer **`yes`**.
2. Specify parameters for the look-up table generation.

   For both single- and multi-species real gas models, you need to specify:

   - Pressure and temperature bounds for the table

     You must ensure that the temperature and pressure bounds you specify are strictly within the stated limits of applicability that are displayed when the NIST model is loaded.

   - Number of pressure, temperature, and saturation points, which determines the resolution of the property look-up table

     The minimum number of pressure and temperature points is 12 and there is no upper limit. However, the larger the table, the greater the memory and computing time requirements. Use a few test runs to strike the best balance of costs (memory and computing time) and accuracy.

   For multi-species, you will also need to specify:

   - Upper and lower molar density bounds for bubble and dew point calculations

     Molar density is used to build the saturation table. The default values work well for most compositions.

   - The mixture composition on a mass or mole basis

     You must ensure that the sum of all constituent fractions adds up to 1 when specifying mass/molar fractions for the components. The maximum number of species is 20.

   - The bubble points and dew points filenames and the directory where you want to store the files.

   Examples of inputs for the NIST look-up tables:

   - Pure fluid real gas model:

```
Min Pressure Value For NIST Table (pa) [13200]
Max Pressure Value For NIST Table (pa) [20000] 1000000
Min Temperature Value For NIST Table (k) [237.3]
Max Temperature Value For NIST Table (k) [500] 400
Number of Pressure Points For NIST Table [101]
Number of Temperature Points For NIST Table [101]
Number of Saturation Points For Bubble/Dew Curve [300]

 NIST table created! Properties for both liquid and vapor phases included!
 Tcrit = 3.391730e+02 pcrit = 3.617700e+06 Dcrit = 5.735823e+02 Ttrp = 1.725200e+02
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Multi-species real gas model:

```
Min Pressure Value For NIST Table (pa) [130000]
Max Pressure Value For NIST Table (pa) [5000000]
Min Temperature Value For NIST Table (k) [220] 180
Max Temperature Value For NIST Table (k) [490] 600
Number of Pressure Points For NIST Table [101]
Number of Temperature Points For NIST Table [101]
Please Select Mass Fractions [Y] or Molar Fractions [N] for Composition of Fluid
 [yes]
Composition of Fluid will be set in Mass Fractions!
co2.fld [0.4]
methane.fld [0.4] 0.6
Create NIST Saturation Curves for the same multi components?
[no] y
Number of Saturation Points For Bubble/Dew Curve [300] 22
Min Molar Density For Bubble and Dew Points Calculations (mol/l) [0.1]
Max Molar Density For Bubble and Dew Points Calculations (mol/l) [25]
Bubble Points File Name ["Bubble_PT.xy"]
Dew Points File Name ["Dew_PT.xy"]
Directory to store Saturation Files [""]
Start NIST Look-Up Table Calculations ...

Property Table... 83.3 percent completed!

NIST table for 2-components created! Properties for both liquid and vapor phases included!
End NIST Look-Up Table Calculations
Use the created NIST Lookup Table for thermal property calculations? [no] y
```

Once the table is created, you must confirm that you want to use the look-up table for thermal properties calculations when prompted.

```
Use the created NIST Lookup Table for thermal property calculations? [no] yes

 The created NIST Lookup Table will be used to calculate the thermal properties!
```

If properties are needed at conditions outside the limits you have specified for the table, Fluent will automatically call the original NIST model functions to calculate the thermal properties. If more than 10% of the property evaluations require computation outside the look-up table, Fluent will print a message indicating the percentage of calculations using direct computation. This can be used as an indicator to adjust the temperature and pressure ranges for the table to minimize the computational cost.

For multi-species real gas properties, the following tables will be created:

- fluid properties
- saturation conditions

Capabilities and Limitations on Using Multi-Species Full NIST look-up Tables

- The look-up table is not saved with the case file. If you restart Fluent or re-read case and data files into the current Fluent session, the table will be cleared and you must go through the steps above to re-create it before running a simulation.
- For computation of saturation properties for the bubble and dew points for a single- or multiple-species, Fluent provides macros that you can use in UDFs. See NIST Real Gas Saturation Properties in the *Fluent Customization Manual* for more information.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The tables apply for compressible fluids with material properties that are a function of local pressure and temperature. The material properties covered by the table include fluids in the liquid and vapor phases. The flashing applications between liquid and vapor phases can be conducted using saturation conditions established for the fluid(s) using mass transfer models.
- The property look-up tables and saturation tables are created for fixed fluid compositions. This means the look-up tables are based on the following assumptions:
  - Mixture composition is constant
  - No chemical reactions are involved
  - No mixing of fluids of different compositions, including no mixing between phases
- The look-up table only applies to single-species/multi-species in single-phase flows.
- The temperature and pressure bounds must be specified within the stated limits of applicability that are displayed when the NIST model is loaded.
- When specifying the table size, it is important to pick temperature and pressure ranges that are large enough to satisfy the current flow conditions.
- If the temperature and pressure ranges you specify are too narrow, Fluent uses the NIST model to calculate the thermal properties. The time saved by creating the table will be negated if the percentage of calculations performed outside of the table is greater than 10%. This percentage is printed, so you can use it as an indicator for adjusting the temperature and pressure range of the table.
- For multi-species table, the bubble and dew points are saved into files(Fluent plot file format) called `Bubble_PT.xy` and `Dew_PT.xy` in the working directory.
- The look-up table is a structured table and may not be suitable for simulating flow behaviors near the critical point when higher resolution is required.

8.16.4.4.4. Creating Binary Mixture Saturation Tables for Binary Mixtures

To create a saturation data look-up table for binary species mixtures (mixtures of two species only), first enable the NIST real gas model as described in Activating the NIST Real Gas Model (p. 1714) and then follow these steps:

1. When prompted with `Create Full NIST LookUp Tables for multi components? [no]`, answer **no**.
2. When prompted with `Create Saturation Table For Binary Mixture Only? [no]`, answer **yes**.

Important:

Not all binary mixtures can be computed from the Fluent built-in NIST model. The allowed combination of materials is determined by the current database of NIST that can be found in `hmx.bnc` in the Ansys Fluent installation directory.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Specify parameters for the binary saturation data look-up table generation:

   - The number of saturation curves
   - The number of points for each saturation curve

     The default number is 12. The actual number of points for each bubble and dew curve may be different than specified because the NIST model may not yield valid computation results for every density point for a given mass fraction.

   - Upper and lower molar density bounds for bubble and dew point calculations

     The molar density is used to build the saturation table. You should provide reasonable values for the minimum and maximum molar densities corresponding to gas phase at low pressure and high temperature, and liquid phase at high pressure and low temperature. The default values (as shown in the example below) are a good starting point. You may need to adjust these values if there is not enough data in the selected range.

   - The filenames for bubble points and dew points and the existing directory where you want to store the files.

   An example of inputs for the NIST binary look-up table:

   ```
   Number of Mass Fraction Points For Binary Mixture [12]
   Number of Saturation Points For Bubble/Dew Curve [12]
   Min Molar Density For Bubble and Dew Points Calculations (mol/l) [0.1]
   Max Molar Density For Bubble and Dew Points Calculations (mol/l) [25]
   Bubble Points File Name ["Bubble_PT.xy"]
   Dew Points File Name ["Dew_PT.xy"]
   Directory to store Saturation Files [""] out
   ```

   The lookup table builds multiple sets of the bubble and dew points for the binary mixture with mass fractions for the first component between 0 and 1. The second component mass fraction is equal to the first component mass fraction subtracted from unity. The number of saturation curves is determined by the number of mass fraction points that you specified.

   Once the lookup table is created, Fluent saves the bubble and dew points data into files and directory you specified.

NIST Binary Saturation Files

NIST binary saturation files store saturation data lookup table for bubble and dew points for a binary mixture with mass fractions between 0 and 1. They can be used for mass and heat transfer modeling involving phase changes.

The binary saturation file structure is as follows. The file contains several data sets for bubble/dew point curves for a series of mass fractions. Each data set consists of the following three parts:

1. Line 1: The mass fraction of the first species (a single float number between 0 and 1).
2. Line 2: The number of data pairs for saturation pressure and temperature.
3. The specified number of lines that follow line 2: Data pairs of saturation pressure and temperature. The last pressure-temperature pair corresponds to the critical condition.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The example below shows the saturation dew point data for the first two curves, both with 12 pressure-temperature pairs. The first data set is for 0 and the second is for a 0.083333 mass fraction of the first species. The pressure-temperature critical condition is (4.872200e+06, 3.053220e+02) for the first curve and (5.547305e+06, 2.947561e+02) for the second curve.

```
 0.000000
12
1.142108e+00 9.036800e+01
7.314020e+01 1.099093e+02
1.202771e+03 1.294505e+02
8.833626e+03 1.489918e+02
3.890130e+04 1.685331e+02
1.219542e+05 1.880744e+02
3.025174e+05 2.076156e+02
6.354397e+05 2.271569e+02
1.182515e+06 2.466982e+02
2.011733e+06 2.662395e+02
3.201942e+06 2.857807e+02
4.872200e+06 3.053220e+02

 0.083333
12
1.509489e+05 1.891238e+02
1.257388e+06 2.428513e+02
2.252097e+06 2.631615e+02
3.090473e+06 2.752623e+02
3.775539e+06 2.831498e+02
4.318958e+06 2.883822e+02
4.735586e+06 2.917723e+02
5.042345e+06 2.938262e+02
5.258091e+06 2.949048e+02
5.402834e+06 2.952901e+02
5.495209e+06 2.951982e+02
5.547305e+06 2.947561e+02

.....
.....
```

8.16.4.5. Solution Strategies and Considerations for NIST Real Gas Model Simulation

The flow modeling of NIST real-gas flow is much more complex and challenging than simple ideal-gas flow. Therefore, you should expect the solution to converge at much slower rate with real-gas flow than when running ideal-gas flow. Also due to the complexity of the equations used in property evaluations, converging a solution with the real-gas model is in general done at much lower Courant values when you are using the density-based solver, or at much lower under-relaxation values if you are using the pressure-based solver. It is recommended that you first attempt to converge your solution using first-order discretization, then switch to second-order discretizations and re-iterate to convergence.

The real-gas properties in NIST are defined within a limited/bounded range. It is important that the flow conditions you are prescribing fall within the range of the database. It is possible that you specify flow at a state that is physically valid but otherwise not defined in the database. In this situation the solution will diverge or immediately generate an error message on the Ansys Fluent console as soon as the state crosses the limit of the database. In some instances, the actual converged state is just within the bounded defined database but only transitory outside the range. In this situation the divergence can be avoided by lowering the Courant value or under-relaxation factors so a less aggressive convergence rate is adapted.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Finally, if you attempt to initialize the flow from an inlet flow conditions and an error message is generated from one of the property routines, then this is an indicator that the flow conditions you have specified is not defined within the range of the database.

8.16.4.5.1. Writing Your Case File

When you save your completed real gas model to a case file, the linkage to the shared library containing real gas properties will be saved to the case file (along with property data for the material you selected in the NIST real gas model). This information can be reported to the console by listing the material properties (using the command `/define/materials/list-properties <material-name>`) and viewed in the **Create/Edit Materials** dialog box by selecting **Edit...** for density.

8.16.4.5.2. Postprocessing

All postprocessing functions properly report and display the current thermodynamic and transport properties of the selected real gas model. The thermodynamic and transport properties controlled by the NIST real gas model include the following:

- density
- enthalpy
- entropy
- molecular weight
- molecular viscosity
- sound speed
- specific heat
- thermal conductivity In addition to the properties listed above, you can also report
- compressibility factor
- any quantities that are derived from the properties listed above (for example, total quantities, ratio of specific heats)

8.16.5. The User-Defined Real Gas Model

The user-defined real gas model (UDRGM) has been developed to allow you to write your own custom real gas model to fit your particular modeling needs. It also allows you to simulate a single-species flow, multiple-species mixture flow, multiphase flow, or volumetric reactions.

The following limitations exist for the UDRGM:

8.16.5.1. Limitations of the User-Defined Real Gas Model

- You cannot include more than one user-defined real gas material (fluid or mixture) in the same problem. However, you can use other materials together with real gas in your simulation.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- When you are using the UDRGM, the materials defined in your real gas UDF will appear in the Create/Edit Materials Dialog Box (p. 4885) with the name **real-gas-fluid** or **real-gas-mixture** and all physical and thermodynamic property inputs disabled. Use the Create/Edit Materials Dialog Box (p. 4885) to define or modify:
  - Mass diffusivity property in the real-gas-mixture material if you are modeling multicomponent flow.
  - Chemical reactions in the real-gas-mixture if you are modeling reacting flow.
  - Radiation properties for the real-gas-mixture or the real-gas-fluid materials if you are modeling radiation.
  - Properties of materials other than the user-defined **real-gas-mixture** or **real-gas-fluid** materials.

  Note:

  If you are using the UDRGM together with other materials from Ansys Fluent's property database in dispersed phase or multiphase calculations, take care to use the same reference temperature as in Ansys Fluent in your real-gas UDF. The reference temperature in Ansys Fluent is 298.15 K.

- Pressure-inlets, mass flow-inlets, and pressure-outlets are the only inflow and outflow boundaries available for use with the real gas models.
- Non-reflecting boundary conditions are not compatible with the real gas models when using the density-based solver. If your model requires NRBC and a real gas model, you must use the pressure-based solver.
- The UDRGM can be used with the Eulerian multiphase models.
- The UDRGM is compatible with the Lagrangian Dispersed Phase Models. Refer to Using the Cubic Equation of State Models with the Lagrangian Dispersed Phase Models (p. 1698) for guidelines and restrictions of this approach.
- The real gas models cannot be used with the non-premixed, partially premixed, and composition PDF transport combustion models. Chemical reactions can however be modeled with the finite rate and eddy dissipation models. Note that the **Dimension Reduction** model is not available with the real gas models.
- The UDRGM does not allow specification of site or solid species and consequently cannot be used with wall surface reactions.

The UDRGM requires a library of functions written in the C programming language. Moreover, there are certain coding requirements that need to be followed when writing these functions. Sample real gas function libraries are provided to assist you in writing your own UDRGM. When UDRGM functions are compiled, they will be grouped in a shared library that later will be loaded and linked with the Ansys Fluent executable. The procedure for using the UDRGM is defined as follows:

1. Define the real gas equation of state and all related thermodynamic and transport property equations.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Create a C source code file that conforms to the format defined in this section.

3. Start Ansys Fluent and set up your case file in the usual way.

4. Compile your UDRGM C library and build a shared library file (you can use the available compiled UDF utilities in either the graphical user interface or the text command interface).

5. Load your newly created UDRGM library via the text command menu:

   If a single-species UDRGM to be used, then the text command menu is:

   ```
   > define/user-defined/real-gas-models/user-defined-real-gas-model

     use user defined real gas? [no] yes
   ```

   On the other hand, if you are simulating multiple-species UDRGM flow, then the text command menu to use is:

   ```
   > define/user-defined/real-gas-models/user-defined-multispecies-real-gas-model
     use user multispecies defined real gas? [no] yes
   ```

   Upon activating the UDRGM, the function library will now supply the fluid material properties for your case.

6. You can simulate volumetric reactions with your real gas model using the Species Model Dialog Box (p. 4768), or the text interface (`define/models/species/volumetric-reactions?`).

   You can access the **Species Model** from the tree by going to **Setup**/**Models**/**Species** and double-clicking **Species**.

   In the Species Model Dialog Box (p. 4768)

   - Enable **Species Transport** under **Model**.
   - Enable **Volumetric** under **Reactions**.
   - Select the appropriate **Turbulence-Chemistry Interaction** option.
   - Set up the reaction by clicking the **Edit...** button for the **real-gas-mixture Mixture Material**.

   Important:

   Note that the fluid materials and their properties, appearing in the Create/Edit Materials Dialog Box (p. 4885), are the ones defined in your real gas UDF. You cannot modify the materials via this dialog box, however, you can set up the volumetric reaction. If you would like to modify the mixture materials and their properties, this should be done in the real gas UDF. The volumetric reactions for your real gas mixture are defined in the same way as for any Ansys Fluent mixture. For details, refer to Defining Reactions (p. 2379).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Alternatively, the chemical reactions can be set up using the `define/models/species` and `define/materials` text command.

Important:

Note that the chemical reactions should be activated after your real gas UDF has been built and loaded. It is also recommended to test and validate your real gas UDF, running the cold flow calculation prior to attempting to solve the reacting flow. Also, make sure that the applicability range of the real gas functions in your UDF fully covers the temperature and pressure range of the reacting flow calculation.

7. Run your calculation.

When using the UDRGM the robustness of the solver and the speed of flow convergence will largely depend on the complexity of the material properties you have defined in your UDF. It is important to understand the operational range of the property functions you are coding so you can simulate the flow within that range.

8.16.5.2. Writing the UDRGM C Function Library

Creating a UDRGM C function library is reasonably straightforward; however, your code must make use of specific function names and macros, which will be described in detail below. The basic library requirements are as follows:

- The code must contain the `udf.h` file inclusion directive at the beginning of the source code. This allows the definitions for `DEFINE` macros and other Ansys Fluent functions to be accessible during the compilation process.
- The code must include at least one of the UDF's `DEFINE` functions (that is `DEFINE_ON_DEMAND`) to be able to use the compiled UDFs utility (see the sample UDRGM codes provided below).
- Any values that are passed to the solver by the UDRGM or returned by the solver to the UDRGM are assumed to be in SI units.
- You must use the principal set of functions listed below in your UDRGM library. These functions are the mechanism by which your thermodynamic property data are transferred to the Ansys Fluent solver. Note that `ANYNAME` can be any string of alphanumeric characters, and allows you to provide unique names to your library functions.

Function inputs from the Ansys Fluent solver consist of one or more of the following variables:

T = Temperature, K

p = Pressure, Pa

ρ = Density, kg/m^3

Y_i[] = Species mass fraction

`cell` = Cell or face index, depending on whether the thread is a cell or face zone; `NULL_CELL` is passed when the function is called without a cell/face context

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

`thread` = Pointer to thread structure; `NULL` is passed when the function called without thread context

Important:

- Y_i[]: Ansys Fluent solver returns a value of `1.0` for Y_i[] in single-species flows. For multiple-species flows, Y_i[] is a vector array containing species mass fraction in an order defined by the user setup function.
- For temperature, density, pressure and species mass fractions the UDRGM code should use the values provided as functions arguments. The corresponding values in the solution arrays, which are accessible via cell index and thread pointer, may temporarily differ from these depending on algorithmic details, and the function arguments take precedence.

The UDRGM function names and argument lists, followed by a short description of the function, are as follows:

`void ANYNAME_Setup(Domain *domain, cxboolean vapor_phase, char *species_list, int (*messagefunc)(const char *format,...), void (*errorfunc)(const char *format, ...))`

performs model setup and initialization. Can be used to read data and parameters related to your UDRGM. When writing UDFs for multiple-species, use this function to specify the number of species and the name of the species as shown in the multiple-species example. The Boolean variable, `vapor_phase`, passes to your UDF the setting of the text-interface command `define/user-defined/real-gas-models/set-state`.

The `messagefunc` argument is a function that you can use for including diagnostic output in the transcript file, and the `errorfunc` argument is the equivalent of error reporting. The format string and additional arguments follow the same syntax as the `printf` C function.

`double ANYNAME_density(cell_t cell, Thread *thread, cxboolean vapor_phase, double T, double P, double yi[])`

returns the value of density as a function of phase, temperature, pressure and species mass-fraction if applicable. The Boolean variable `vapor_phase` passes to your UDF the setting of the text-interface command `define/user-defined/real-gas-models/set-state`, or, if applicable, the zone state set by the text-interface command `define/boundary-conditions/modify-zones/change-zone-state`.

Important:

Since this function is called numerous times during each solver iteration, it is important to make this function as numerically efficient as possible.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

double ANYNAME_specific_heat(cell_t cell, Thread *thread, double T, double Rho, double P, double yi[])

returns the real gas specific heat at constant pressure as a function of temperature, density, absolute pressure, and species mass-fraction if applicable.

double ANYNAME_enthalpy(cell_t cell, Thread *thread, double T, double Rho, double P, double yi[])

returns the enthalpy as a function of temperature, density, absolute pressure, and species mass-fraction if applicable.

double ANYNAME_entropy(cell_t cell, Thread *thread, double T, double Rho, double P, double yi[])

returns the entropy as a function of temperature, density, absolute pressure, and species mass-fraction if applicable.

double ANYNAME_mw(double yi[])

returns the fluid molecular weight.

double ANYNAME_speed_of_sound(cell_t cell, Thread *thread, double T, double Rho, double P, double yi[])

returns the value of speed of sound as a function of temperature, density, absolute pressure, and species mass-fraction if applicable.

double ANYNAME_viscosity(cell_t cell, Thread *thread, double T, double Rho, double P, double yi[])

returns the value of dynamic viscosity as a function of temperature, density, absolute pressure, and species mass-fraction if applicable.

double ANYNAME_thermal_conductivity(cell_t cell, Thread *thread, double T, double Rho, double P, double yi[])

returns the value of thermal conductivity as a function of temperature, density, absolute pressure, and species mass-fraction if applicable.

double ANYNAME_rho_t(cell_t cell, Thread *thread, double T, double Rho, double P, double yi[])

returns the value of $\frac{d\rho}{dT}$ at constant pressure as a function of temperature, density, absolute pressure, and species mass-fraction if applicable.

double ANYNAME_rho_p(cell_t cell, Thread *thread, double T, double Rho, double P, double yi[])

returns the value of $\frac{d\rho}{dp}$ at constant temperature as a function of temperature, density, absolute pressure, and species mass-fraction if applicable.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

double ANYNAME_enthalpy_t(cell_t cell, Thread *thread, double T, double Rho, double P, double yi[])

returns the value of $\frac{dh}{dT}$ at constant pressure as a function of temperature, density, absolute pressure, and species mass-fraction if applicable. Note that by definition $dh/\ dt==c_p$, so this function should simply return the specific heat value.

double ANYNAME_enthalpy_p(cell_t cell, Thread *thread, double T, double Rho, double P, double yi[])

returns the value of $\frac{dh}{dp}$ at constant temperature as a function of temperature, density, absolute pressure, and species mass-fraction if applicable.

double ANYNAME_enthalpy_prime(cell_t cell, Thread *thread, double T, double Rho, double P, double yi[], double hi[])

returns the value of the mixture enthalpy as a function of temperature, density, absolute pressure, and species mass fraction. In addition, your UDF must set the elements of the double array `hi[]` to the enthalpy of each species, in the same order as they are referenced in the mass fraction array `yi[]`. Note that the enthalpy in the function `enthalpy_prime` is defined as the sum of sensible enthalpy plus species formation enthalpy, and you should make sure that its computation is consistent with the sensible enthalpy function ANYNAME_`enthalpy`. The function ANYNAME_`enthalpy_prime` is required for the calculation of the heat of reactions, if chemical reactions are being simulated. If you are not solving reacting flows, the function ANYNAME_`enthalpy_prime` can simply be omitted.

At the end of the code you must define a structure of type RGAS_`Function` whose members are pointers to the principal functions listed above. The structure is of type RGAS_`Function` and its name is `RealGasFunctionList`.

Important:

It is imperative that the sequence of function pointers shown below be followed. Otherwise, your real gas model will not load properly into the Ansys Fluent code.

```
UDF_EXPORT RGAS_Functions RealGasFunctionList =
{
 ANYNAME_Setup,                  /* Setup initialize */
 ANYNAME_density,                /* density */
 ANYNAME_enthalpy,               /* sensible enthalpy */
 ANYNAME_entropy,                /* entropy */
 ANYNAME_specific_heat,          /* specific_heat */
 ANYNAME_mw,                     /* molecular_weight */
 ANYNAME_speed_of_sound,         /* speed_of_sound */
 ANYNAME_viscosity,              /* viscosity */
 ANYNAME_thermal_conductivity,   /* thermal_conductivity */
 ANYNAME_rho_t,                  /* drho/dT |const p */
 ANYNAME_rho_p,                  /* drho/dp |const T */
 ANYNAME_enthalpy_t,             /* dh/dT |const p */
 ANYNAME_enthalpy_p              /* dh/dp |const T */
 ANYNAME_enthalpy_prime          /* enthalpy */
};
```

If volumetric reactions are not being simulated, then the function ANYNAME_`enthalpy_prime` can be removed or ignored from the `RealGasFunctionList` structure described here.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The principal set of functions described are the only functions in the UDRGM that will be interacting directly with the Ansys Fluent code. In many cases, your model may require further functions that will be called from the principal function set. For example, when multiple-species real gas model UDFs are written, the principal functions will return the mixture thermodynamic properties based on some specified mixing-law. Therefore, you may want to add further functions that will return the thermodynamic properties for the individual species. These auxiliary functions will be called from the principal set of functions. See User-Defined Real Gas Models (UDRGM) in the Fluent Customization Manual for examples that clearly illustrate this strategy.

8.16.5.3. Compiling Your UDRGM C Functions and Building a Shared Library File

This section presents the steps you must follow to compile your UDRGM C code and build a shared library file. This process requires the use of a C compiler. For more details on compiler requirements, see Compilers in the *Fluent Customization Manual*. To use the UDRGM you must first build the UDRGM library by compiling your UDRGM C code and then load the library into the Ansys Fluent code. The UDRGM shared library is built in the same way that the Ansys Fluent executable itself is built. Internally, a script called `Makefile` is used to invoke the system C compiler to build an object code library that contains the native machine language translation of your higher-level C source code. This shared library is then loaded into Ansys Fluent (either at run time or automatically when a case file is read) by a process called *dynamic loading*. The object libraries are specific to the computer architecture being used, as well as to the particular version of the Ansys Fluent executable being run. The libraries must, therefore, be rebuilt any time Ansys Fluent is upgraded, when the computer's operating system level changes, or when the job is run on a different type of computer. The general procedure for compiling UDRGM C code is as follows:

- Place the UDRGM C code in the folder, that is, where your case file resides.
- Launch Ansys Fluent.
- Read your case file into Ansys Fluent.
- You can now compile your UDRGM C code and build a shared library file using either the graphical interface or the text command interface.

Important:

To build UDRGM library you will use the compiled UDF utilities. However, you will not use the UDF utilities to load the library. A separate loading area for the UDRGM library will be used.

8.16.5.3.1. Compiling the UDRGM Using the Graphical Interface

If the build is successful, then the compiled library will be placed in the appropriate architecture folder (for example, `win64/2d`). By default the library name is `libudf.so` (`libudf.dll` on Windows).

More information on compiled UDFs and building libraries using the Ansys Fluent graphical user interface can be found in the Fluent Customization Manual.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.16.5.3.2. Compiling the UDRGM Using the Text Interface

The UDRGM library can be compiled in the text command interface as follows:

- Select the menu item `define` → `user-defined` → `compiled-functions` .
- Select the `compile` option.
- Enter the compiled UDF library name.

  Important:

  The name given here is the name of the folder where the shared library (for example, `libudf`) will reside. For example, if you press **Enter** then a folder should exist with the name `libudf`, and this folder will contain a library file called `libudf`. If, however, you type a new library name such as `myrealgas`, then a folder called `myrealgas` will be created and it will contain the library `libudf`.

- Continue on with the procedure when prompted.
- Enter the C source file names.

  Important:

  Ideally you should place all of your functions into a single file. However, you can split them into separate files if desired.

- Enter the header file names, if applicable. If you do not have an extra header file then press **Enter** when prompted.

Ansys Fluent will then start compiling the UDRGM C code and put it in the appropriate architecture folder.

Example:

```
> define/user-defined/compiled-functions
  load OR compile ? [load]  compile

  Compiled UDF library name: ["libudf"] my_lib

   Make sure that UDF source files are in the folder
   that contains your case and data files. If you have
   an existing libudf folder, please remove this
   folder to ensure that latest files are used.
  Continue?[yes] RETURN

  Give C-Source file names:
  First file name: [""] my_c_file.c RETURN

  Next file name: [""] RETURN

  Give header file names:
  First file name: [""] my_header_file.h RETURN
```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.16.5.3.3. Loading the UDRGM Shared Library File

Load the UDRGM library:

- Go to the following menu item in the text command interface.

  `define` → `user-defined` → `real-gas-models`

- Select one of the following

  – `user-defined-real-gas-model` if you are modeling a single-species real gas fluid

  – `user-defined-multispecies-real-gas-model` if you are modeling a multiple-species fluid-mixture

- Turn on the real gas model.

  – For single-species:

  ```
  use user defined real gas? [no]   yes
  ```

  – For multiple-species:

  ```
  use multispecies user defined real gas? [no]   yes
  ```

  Ansys Fluent will ask for the location of the user-defined real gas library. You can enter either the name of the folder where the UDRGM shared library is called or the entire path to the UDRGM shared library.

  If the loading of the UDRGM library is successful you will see a message similar to the following:

  ```
  Opening user-defined realgas library "RealgasLibraryname"...
  Library "RealgasDirName/lnamd64/2d/libudf.so" opened
  Setting material "air" to a real-gas...
  Loading Real-RealGasPrefexLable Library:
  ```

8.16.5.4. UDRGM Example: Ideal Gas Equation of State

This section describes an example of a user-defined real gas model. You can use this example as the basis for your own UDRGM code. In this simple example, the standard ideal gas equation of state is used in the UDRGM. See User-Defined Real Gas Models (UDRGM) in the Fluent Customization Manual for more examples of UDRGM functions, including multi-species real gas and reacting real-gas examples.

p = pressure

T = temperature

C_p = specific heat

H = enthalpy

S = entropy

ρ = density

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c = speed of sound

R = universal gas constant/molecular weight

The ideal gas equation of state can be written in terms of pressure and temperature as

$$\rho=\frac{p}{RT} \tag{8.106}$$

The specific heat is defined to be constant C_p = 1006.42.

The enthalpy is, therefore, defined as

$$H=C_pT \tag{8.107}$$

and entropy is given by

$$S=C_p\log\left(\frac{T}{T_{ref}}\right)+R\log\left(\frac{p_{ref}}{p}\right) \tag{8.108}$$

where T_{ref} = 288.15 K and p_{ref} = 101325 Pa

The speed of sound is simply defined as

$$c=\sqrt{TC_p\frac{R}{\left(C_p-R\right)}} \tag{8.109}$$

The density derivatives are:

$$\left(\frac{d\rho}{dp}\right)_T=\frac{1}{RT} \tag{8.110}$$

$$\left(\frac{d\rho}{dT}\right)_p=-\frac{p}{RT^2}=-\frac{\rho}{T} \tag{8.111}$$

The enthalpy derivatives are:

$$\left(\frac{dH}{dT}\right)_p=C_p \tag{8.112}$$

$$\left(\frac{dH}{dp}\right)_T=\frac{C_p}{\rho R}\left[1-\frac{p}{\rho}\frac{d\rho}{dp}\right]=0 \tag{8.113}$$

When you enable the real gas model and load the library successfully into Ansys Fluent, you will be using the equation of state and other fluid properties from this library rather than the one built into the Ansys Fluent code.

8.16.5.4.1. Ideal Gas UDRGM Code Listing

```
/************************************************************************/
/* User Defined Real Gas Model :                                        */
/* For Ideal Gas Equation of State                                      */
/*                                                                      */
/************************************************************************/

#include "udf.h"
#include "stdio.h"
#include "ctype.h"
#include "stdarg.h"

#define MW 28.966   /* molec. wt. for single gas (Kg/Kmol) */
#define RGAS (UNIVERSAL_GAS_CONSTANT/MW)
```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
static int (*usersMessage)(const char *, ...);
static void (*usersError)(const char *, ...);


DEFINE_ON_DEMAND(I_do_nothing)
{
  /* This is a dummy function to allow us to use */
  /* the Compiled UDFs utility      */
}


void IDEAL_error(int err, char *f, char *msg)
{
  if (err)
    usersError("IDEAL_error (%d) from function: %s\n%s\n", err, f, msg);
}

void IDEAL_Setup(Domain *domain, cxboolean vapor_phase, char *filename,
                 int (*messagefunc)(const char *format, ...),
                 void (*errorfunc)(const char *format, ...))
{
  /* Use this function for any initialization or model setups*/
  usersMessage = messagefunc;
  usersError  = errorfunc;
  usersMessage("\nLoading Real-Ideal Library: %s\n", filename);
}

double IDEAL_density(cell_t cell, Thread *thread,
                     cxboolean vapor_phase, double Temp, double press, double yi[])
{
  double r = press / (RGAS * Temp); /* Density at Temp & press */
  return r;      /* (Kg/m^3) */
}

double IDEAL_specific_heat(cell_t cell, Thread *thread,
                           double Temp, double density, double P, double yi[])
{
  double cp = 1006.43;
  return cp;     /* (J/Kg/K) */
}

double IDEAL_enthalpy(cell_t cell, Thread *thread,
                      double Temp, double density, double P, double yi[])
{
  double h = Temp * IDEAL_specific_heat(cell, thread, Temp, density, P, yi);
  return h;      /* (J/Kg) */
}

#define TDatum 288.15
#define PDatum 1.01325e5

double IDEAL_entropy(cell_t cell, Thread *thread,
                     double Temp, double density, double P, double yi[])
{
  double s = IDEAL_specific_heat(cell, thread, Temp, density, P, yi) * log(fabs(Temp / TDatum)) +
             RGAS * log(fabs(PDatum / P));
  return s;      /* (J/Kg/K) */
}


double IDEAL_mw(double yi[])
{
  return MW;     /* (Kg/Kmol) */
}

double IDEAL_speed_of_sound(cell_t cell, Thread *thread,
                            double Temp, double density, double P, double yi[])
{
  double cp = IDEAL_specific_heat(cell, thread, Temp, density, P, yi);
  return sqrt(Temp * cp * RGAS / (cp - RGAS)); /* m/s */
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
}

double IDEAL_viscosity(cell_t cell, Thread *thread,
                       double Temp, double density, double P, double yi[])
{
  double mu = 1.7894e-05;
  return mu;     /* (Kg/m/s) */
}

double IDEAL_thermal_conductivity(cell_t cell, Thread *thread,
                                  double Temp, double density, double P,
                                  double yi[])
{
  double ktc = 0.0242;
  return ktc;      /* W/m/K */
}
double IDEAL_rho_t(cell_t cell, Thread *thread,
                   double Temp, double density, double P, double yi[])
{
  /* derivative of rho wrt. Temp at constant p */
  double rho_t = -density / Temp;
  return rho_t;     /* (Kg/m^3/K) */
}

double IDEAL_rho_p(cell_t cell, Thread *thread,
                   double Temp, double density, double P, double yi[])
{
  /* derivative of rho wrt. pressure at constant T */
  double rho_p = 1.0 / (RGAS * Temp);
  return rho_p;    /* (Kg/m^3/Pa) */
}

double IDEAL_enthalpy_t(cell_t cell, Thread *thread,
                        double Temp, double density, double P, double yi[])
{
  /* derivative of enthalpy wrt. Temp at constant p */
  return IDEAL_specific_heat(cell, thread, Temp, density, P, yi);
}

double IDEAL_enthalpy_p(cell_t cell, Thread *thread,
                        double Temp, double density, double P, double yi[])
{
  /* derivative of enthalpy wrt. pressure at constant T  */
  /* general form dh/dp|T = (1/rho)*[ 1 + (T/rho)*drho/dT|p] */
  /* but for ideal gas dh/dp = 0        */
  return 0.0 ;
}

UDF_EXPORT RGAS_Functions RealGasFunctionList =
{
  IDEAL_Setup,                   /* initialize */
  IDEAL_density,                 /* density */
  IDEAL_enthalpy,                /* enthalpy */
  IDEAL_entropy,                 /* entropy */
  IDEAL_specific_heat,           /* specific_heat */
  IDEAL_mw,                      /* molecular_weight */
  IDEAL_speed_of_sound,          /* speed_of_sound */
  IDEAL_viscosity,               /* viscosity */
  IDEAL_thermal_conductivity,    /* thermal_conductivity */
  IDEAL_rho_t,                   /* drho/dT |const p */
  IDEAL_rho_p,                   /* drho/dp |const T */
  IDEAL_enthalpy_t,              /* dh/dT |const p  */
  IDEAL_enthalpy_p               /* dh/dp |const T  */
};
/************************************************************/
```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8.16.5.5. Additional UDRGM Examples

You can find the following additional UDRGM examples in the Fluent Customization Manual:

- The Aungier Redlich Kwong equation of state for single component flow. See UDRGM Example: Redlich-Kwong Equation of State in the Fluent Customization Manual for details.
- A simple example of a multi-species real-gas model. See UDRGM Example: Multiple-Species Real Gas Model in the Fluent Customization Manual for details.
- A real gas model example with the Aungier Redlich Kwong equation of state, ideal gas mixing rules and volumetric reactions. See UDRGM Example: Real Gas Model with Volumetric Reactions in the Fluent Customization Manual for details.

8.16.6. Using Real Gas Property (RGP) Table Files

8.16.6.1. Overview

Real Gas Property (RGP) tables provide thermodynamic and transport material properties as functions of temperature and pressure. The material is modeled as a real-fluid equivalent of the cubic equations of state or NIST real gas models. For sub-critical conditions, the RGP tables may contain data for the liquid of vapor state, or both. RGP files often include data tables for both liquid and vapor physical states.

In `.rgp` files, the material property values are provided in tabular format as functions of temperature and pressure. As opposed to user-defined functions (UDFs), which are executed at solver run-time, when RGP tables are used, the potentially expensive property calculations are carried out in a pre-processing step, where you can deal with possible errors that may occur in property calculation. During the solution, the solver obtains material properties by table lookup, which is a computationally very efficient operation.

RGP table files support two-dimensional tables for material properties as functions of temperature and pressure.

8.16.6.2. Defining Material Properties Using RGP Tables

To import RGP tables into your simulation and use them for material property definition, follow these steps:

1. Read the RGP tables into your session as described in Reading Files in Tabular Format (p. 969).

   Upon importing the RGP file, table data set for each material contained in the file is automatically derived using the access key. The names of all RGB data sets appear in the **Tables** selection list in the **Table File Manage**r dialog box.

2. If necessary, create materials for which you want to use the RGP table data by doing one of the following:

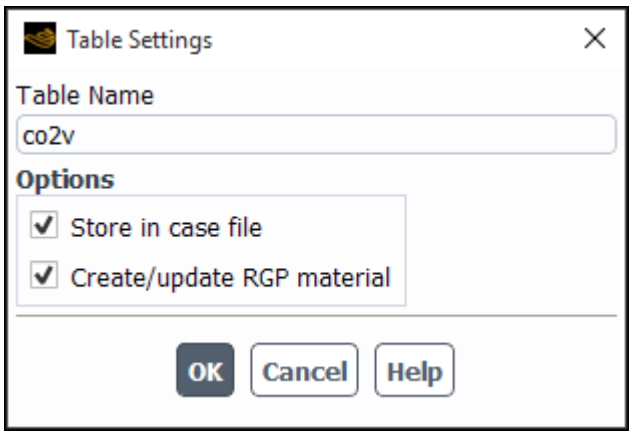
   - Create a material for each RGP table data set using the **Table Settings** dialog box:

     i. In the **Table File Manager** dialog box, select the name of the RGP table data set for which you want to create a material and click the **Settings...** button.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ii. In the **Table Settings** dialog box that opens, select **Create/update RGP material**.

Figure 8.51: The Table Settings Dialog Box



iii. Click **OK** to close the **Table Settings** dialog box.

A material with the same name as the RGP table data set is created or updated if already exists. This material is automatically set to use the RGP tables for all properties available from the RGP data.

iv. Repeat these steps for all RGP table data sets that you want to use in your simulation.

- Create the material in the usual way.

3. If you want to embed the RGP data for the selected material set into your case file that you can read back into a later session of Ansys Fluent, select **Store in case file** in the **Table Settings** dialog box. This allows sharing the case without the need to provide the RGP data in a separate file at the expense of increased case file size.

4. Click **OK** to close the **Table File Manager** dialog box.

5. In the **Create/Edit Materials** dialog box, select the **rgp-table** option from the drop-down list for **Density** and then click **Edit...** to select the appropriate RGP table in the **RGP Table Data Set** dialog box. **Cp (Specific Heat)**, **Thermal Conductivity** and **Viscosity** will be automatically set to the **rgp-table** option using the RGP table data set selected for **Density**.

8.16.6.3. Defining Saturation Properties via RGP Tables

You can use RGP tables to specify the following saturation properties:

- Cavitation model: **Vaporization Pressure**
- Evaporation-condensation model: **Saturation Temperature**
- Boiling model: **Saturation Temperature**
- Species-mass-transfer model, Raoult's law: **Saturation Pressure**

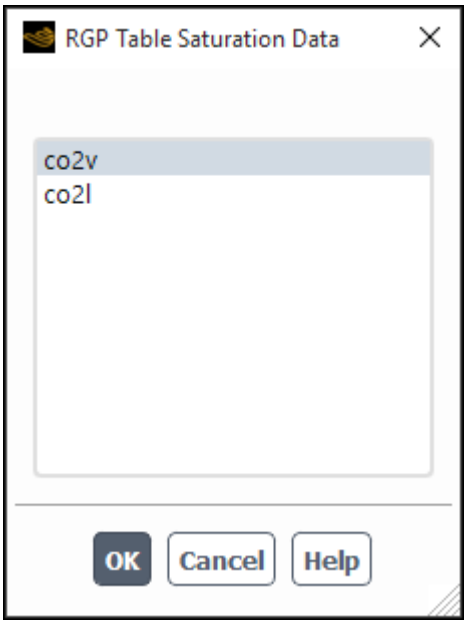
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Phase Interaction Forces: **Surface Tension Coefficient** (using temperature-dependent values at saturation conditions)
- Droplet/Particle evaporation (DPM): **Saturation Vapor Pressure**

To do this:

1. Read the RGP table into your case as described in section Reading Files in Tabular Format (p. 969).
2. In the dialog box for the selected model, select **rgp-table-sat** from the drop-down list for the appropriate saturation property.
3. In the **RGP Table Saturation Data** dialog box that opens, select the desired saturation table and click **OK**.

Figure 8.52: The RGP Table Saturation Data Dialog Box



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 9: Modeling Basic Fluid Flow

This chapter describes the basic physical models that Ansys Fluent provides for fluid flow and the commands for defining and using them. Models for flows in moving zones (including sliding and dynamic meshes) are explained in Modeling Flows with Moving Reference Frames (p. 1769), models for turbulence are described in Modeling Turbulence (p. 2061), and models for heat transfer (including radiation) are presented in Modeling Thermal Energy (p. 2163). An overview of modeling species transport and reacting flows is provided in Modeling Species Transport and Finite-Rate Chemistry (p. 2355), details about models for species transport and reacting flows are described in Modeling Species Transport and Finite-Rate Chemistry (p. 2355) – Modeling a Composition PDF Transport Problem (p. 2442), and models for pollutant formation are presented in Modeling Pollutant Formation (p. 2585). The discrete phase model is described in Modeling Discrete Phase (p. 2671), general multiphase models are described in Modeling Multiphase Flows (p. 2915), and the melting and solidification model is described in Modeling Solidification and Melting (p. 3219). For information on modeling porous media, porous jumps, and lumped parameter fans and radiators, see Cell Zone and Boundary Conditions (p. 1299).

The information in this chapter is presented in the following sections:

9.1. User-Defined Scalar (UDS) Transport Equations

9.2. Periodic Flows

9.3. Swirling and Rotating Flows

9.4. Compressible Flows

9.5. Inviscid Flows

9.1. User-Defined Scalar (UDS) Transport Equations

For additional information, see the following sections:

9.1.1. Introduction

9.1.2. UDS Theory

9.1.3. Setting Up UDS Equations in Ansys Fluent

9.1.1. Introduction

Ansys Fluent can solve the transport equation for an arbitrary, user-defined scalar (UDS) in the same way that it solves the transport equation for a scalar such as species mass fraction. Extra scalar transport equations may be needed in certain types of combustion applications or for example in plasma-enhanced surface reaction modeling. Ansys Fluent allows you to define additional scalar transport equations in your model in the User-Defined Scalars Dialog Box (p. 5654).

9.1.2. UDS Theory

UDS theory is described in the following sections:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.1.2.1. Single Phase Flow
9.1.2.2. Multiphase Flow

9.1.2.1. Single Phase Flow

For an arbitrary scalar ϕ_k, Ansys Fluent solves the equation

$$\frac{\partial \rho \phi_k}{\partial t} + \frac{\partial}{\partial x_i}\left(\rho u_i \phi_k - \Gamma_k \frac{\partial \phi_k}{\partial x_i}\right) = S_{\phi_k} \quad k = 1, \ldots, N \tag{9.1}$$

where Γ_k and S_{ϕ_k} are the diffusion coefficient and source term supplied by you for each of the N scalar equations. Note that Γ_k is defined as a tensor in the case of anisotropic diffusivity. The diffusion term is therefore $\nabla \cdot \left(\Gamma_k \cdot \phi_k\right)$

For isotropic diffusivity, Γ_k could be written as $\Gamma_k I$ where I is the identity matrix.

For the steady-state case, Ansys Fluent will solve one of the three following equations, depending on the method used to compute the convective flux:

- If convective flux is *not* to be computed, Ansys Fluent will solve the equation

$$-\frac{\partial}{\partial x_i}\left(\Gamma_k \frac{\partial \phi_k}{\partial x_i}\right) = S_{\phi_k} \quad k = 1, \ldots, N \tag{9.2}$$

  where Γ_k and S_{ϕ_k} are the diffusion coefficient and source term supplied by you for each of the N scalar equations.

- If convective flux is to be computed with mass flow rate, Ansys Fluent will solve the equation

$$\frac{\partial}{\partial x_i}\left(\rho u_i \phi_k - \Gamma_k \frac{\partial \phi_k}{\partial x_i}\right) = S_{\phi_k} \quad k = 1, \ldots, N \tag{9.3}$$

- It is also possible to specify a user-defined function to be used in the computation of convective flux. In this case, the user-defined mass flux is assumed to be of the form

$$F = \int_S \rho \vec{u} \cdot d\vec{S} \tag{9.4}$$

where $d\vec{S}$ is the face vector area.

Important:

User-defined scalars in solid zones do not take into account the convective term with moving reference frames.

9.1.2.2. Multiphase Flow

For multiphase flows, Ansys Fluent solves transport equations for two types of scalars: *per phase* and *mixture*. For an arbitrary k scalar in *phase-l*, denoted by ϕ_l^k, Ansys Fluent solves the transport equation inside the volume occupied by *phase-l*

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\frac{\partial \alpha_l \rho_l \phi_l^k}{\partial t} + \nabla \cdot \left(\alpha_l \rho_l \vec{u}_l \phi_l^k - \alpha_l \Gamma_l^k \nabla \phi_l^k \right) = S_l^k \; k = 1, \ldots, N \tag{9.5}$$

where α_l, ρ_l, and $\vec{u}_l$ are the volume fraction, physical density, and velocity of *phase-l*, respectively. Γ_l^k and S_l^k are the diffusion coefficient and source term, respectively, which you will need to specify. In this case, scalar ϕ_l^k is associated only with one phase (*phase-l*) and is considered an individual field variable of *phase-l*.

The mass flux for *phase-l* is defined as

$$F_l = \int_S \alpha_l \rho_l \vec{u}_l \cdot d\vec{S} \tag{9.6}$$

If the transport variable described by scalar ϕ_l^k represents the physical field that is shared between phases, or is considered the same for each phase, then you should consider this scalar as being associated with a mixture of phases, ϕ^k. In this case, the generic transport equation for the scalar is

$$\frac{\partial \rho_m \phi^k}{\partial t} + \nabla \cdot \left(\rho_m \vec{u}_m \phi^k - \Gamma_m^k \nabla \phi^k \right) = S^{k_m} \; k = 1, \ldots, N \tag{9.7}$$

where mixture density ρ_m, mixture velocity $\vec{u}_m$, and mixture diffusivity for the scalar k Γ_m^k are calculated according to

$$\rho_m = \sum_l \alpha_l \rho_l \tag{9.8}$$

$$\rho_m \vec{u}_m = \sum_l \alpha_l \rho_l \vec{u}_l \tag{9.9}$$

$$F_m = \int_S \rho_m \vec{u}_m \cdot d\vec{S} \tag{9.10}$$

$$\Gamma_m^k = \sum_l \alpha_l \Gamma_l^k \tag{9.11}$$

$$S_m^k = \sum_l S_l^k \tag{9.12}$$

To calculate mixture diffusivity, you must specify individual diffusivities for each material associated with individual phases.

Note that if the user-defined mass flux option is activated, then mass fluxes shown in Equation 9.6 (p. 1739) and Equation 9.10 (p. 1739) must be replaced in the corresponding scalar transport equations. For more information about the theoretical background of user-defined scalar transport equations, see User-Defined Scalar (UDS) Transport Equations in the Theory Guide.

9.1.3. Setting Up UDS Equations in Ansys Fluent

Ansys Fluent allows you to define up to 50 user-defined scalar (UDS) transport equations in your model. The general scalar transport equation, Equation 1.8 in the Theory Guide, is shown below with

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the four terms (transient, flux, diffusivity, source) that you can customize. (Equation 9.13 (p. 1740)). You will define a UDS transport equation by setting the parameters for these four terms.

$$\underbrace{\frac{\partial \rho \phi_k}{\partial t}}_{unsteady} + \frac{\partial}{\partial x_i}\left(\underbrace{F_i \phi_k}_{convection} - \Gamma_k \underbrace{\frac{\partial \phi_k}{\partial x_i}}_{diffusion}\right) = \underbrace{S_{\phi_k}}_{sources} \qquad k=1,\ldots,N_{scalars} \tag{9.13}$$

In addition, you can set boundary conditions for the variables within cells of a fluid or solid zone for a particular scalar equation. This is done by fixing the value of ϕ_k in Equation 9.13 (p. 1740). When ϕ_k is fixed in a given cell, the UDS scalar transport is not solved and the cell is not included when the residual sum is computed. Additionally, you can also specify custom boundary conditions in the mixture on all wall, inflow, and outflow boundaries on a per-scalar basis.

The procedures for setting up a user-defined scalar (UDS) equation for single-phase and multiphase flows are outlined below. Note that a significant difference between a UDS for a single-phase versus a multiphase application is that you must associate each UDS with its corresponding phase domain or mixture domain, depending on your application. If you supply UDFs for transient terms, convective fluxes, and sources, you must be aware that they are directly called from the phase or mixture domains, according to the scalar association settings.

When setting up a UDS, ensure that the scalar transport problem is well defined with proper Dirichlet/Neumann boundary conditions. Depending on the nature of the problem, the UDS equation may need to converge further than the default convergence criterion of 0.1. Set the convergence criterion according to your needs.

In some instances, the residuals may not reflect the quality of the solution. In these cases you should create monitors on the UDS and use these monitors to judge solution convergence. There are several controls that you can adjust to improve convergence, however some of these adjustments can increase the computation time. The following controls can help with convergence:

- Using a fixed cycle, such as **F-Cycle** in place of **Flexible** cycle in the **Advanced Solution Controls** dialog box.
- Using an additional stabilizer such as the Biconjugate gradient stabilization method.
- Using an ILU smoother.
- Increasing the pre- and post-sweeps available in the **Advanced Solution Controls** dialog box.
- Adjusting the under relaxation factors.

See the Fluent Customization Manual for information on using UDFs to define scalar quantities.

9.1.3.1. Single Phase Flow

1. Specify the number of UDS equations you require in the User-Defined Scalars Dialog Box (p. 5654) (Figure 9.1: The User-Defined Scalars Dialog Box (p. 1741)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parameters & Customization → **User Defined Scalars** **New...**

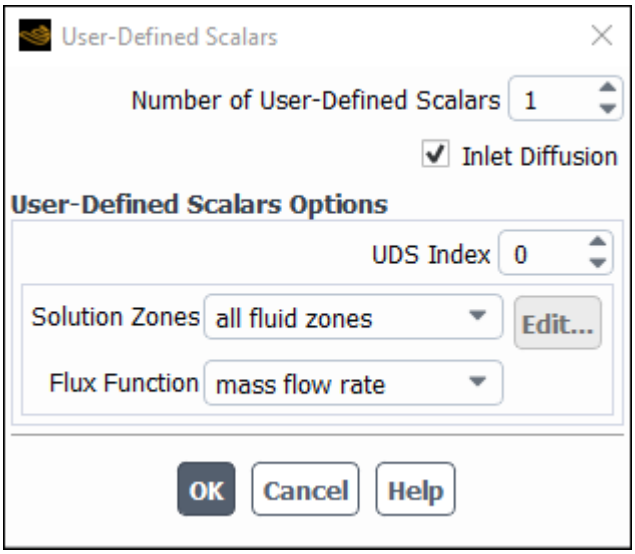
Important:

The maximum number of user-defined scalar transport equations you can define is 50. Ansys Fluent assigns numbers to the equations starting with 0.

Important:

Note that Ansys Fluent assigns a default name for each scalar equation (`User Scalar 0`, `User Scalar 1`, and so on). These labels will appear in graphics dialog boxes in Ansys Fluent. You can change them by means of a UDF. See the Fluent Customization Manual for details.

Figure 9.1: The User-Defined Scalars Dialog Box



2. Enable **Inlet Diffusion** if you want to include the diffusion term in the UDS transport equation for all inflow and outflow boundaries.

3. Set the first user-defined scalar equation parameters by making sure that the **UDS Index** is set to 0.

   a. Specify the **Solution Zones** you want the scalar equation to be solved in as **all fluid zones**, **all solid zones**, **all zones** (fluid and solid) or **selected zones**. If you choose **selected zones**, click the **Edit** button to view the list of zones you can select.

   b. Specify the **Flux Function** to be **none**, **mass flow rate**, or a user-defined function (UDF). The **Flux Function** determines how the convective flux is computed, which determines the equation that Ansys Fluent solves for the user-defined scalar. Selecting **none**, **mass flow rate**, or a user-defined function results in Ansys Fluent solving Equation 1.9, Equation 1.10, or Equation 1.11, respectively (in the Theory Guide). See the Fluent Customization Manual for details on flux UDFs.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. Specify the **Unsteady Function** to be **none**, **default**, or a user-defined function (UDF). Select **none** for a steady state solution and **default** if you want the transient term in Equation 1.8 in the Theory Guide). See the separate Fluent Customization Manual for details on unsteady UDFs.

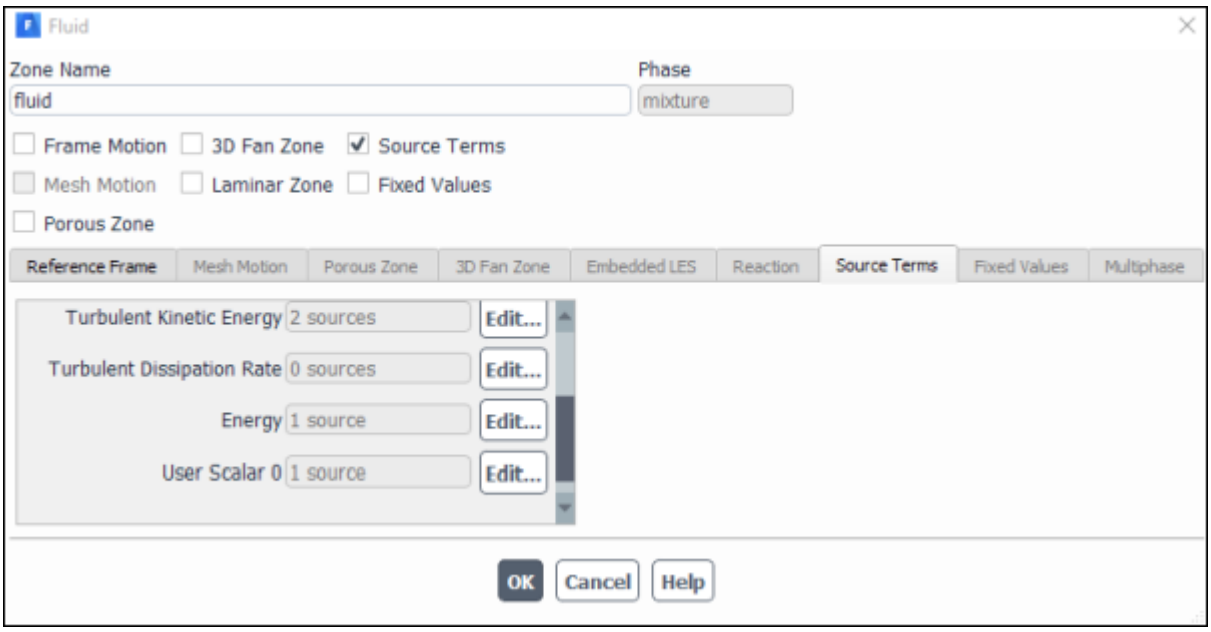
d. Repeat this process for each scalar equation by incrementing the **UDS Index**.

e. Click **OK** when all user scalar equations have been defined.

4. To specify source term(s) for each of the N UDS equations, enable the **Source Terms** option in the **Fluid** or **Solid** dialog box (Figure 9.2: The Fluid Dialog Box with Inputs for Source Terms for a User-Defined Scalar (p. 1742)) and click the **Source Terms** tab. The source parameters will be displayed.

Setup → Cell Zone Conditions

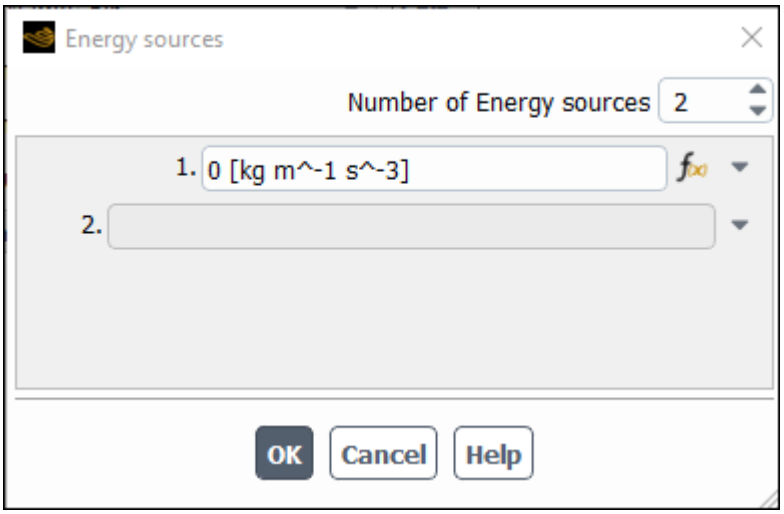
Figure 9.2: The Fluid Dialog Box with Inputs for Source Terms for a User-Defined Scalar



a. Specify the number of sources you require for each scalar equation by clicking on the **Edit...** button next to the scalar name (for example, **User Scalar 0**). This will open the **User Scalar 0 sources** dialog box (Figure 9.3: The User Scalar Sources Dialog Box (p. 1743)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 9.3: The User Scalar Sources Dialog Box



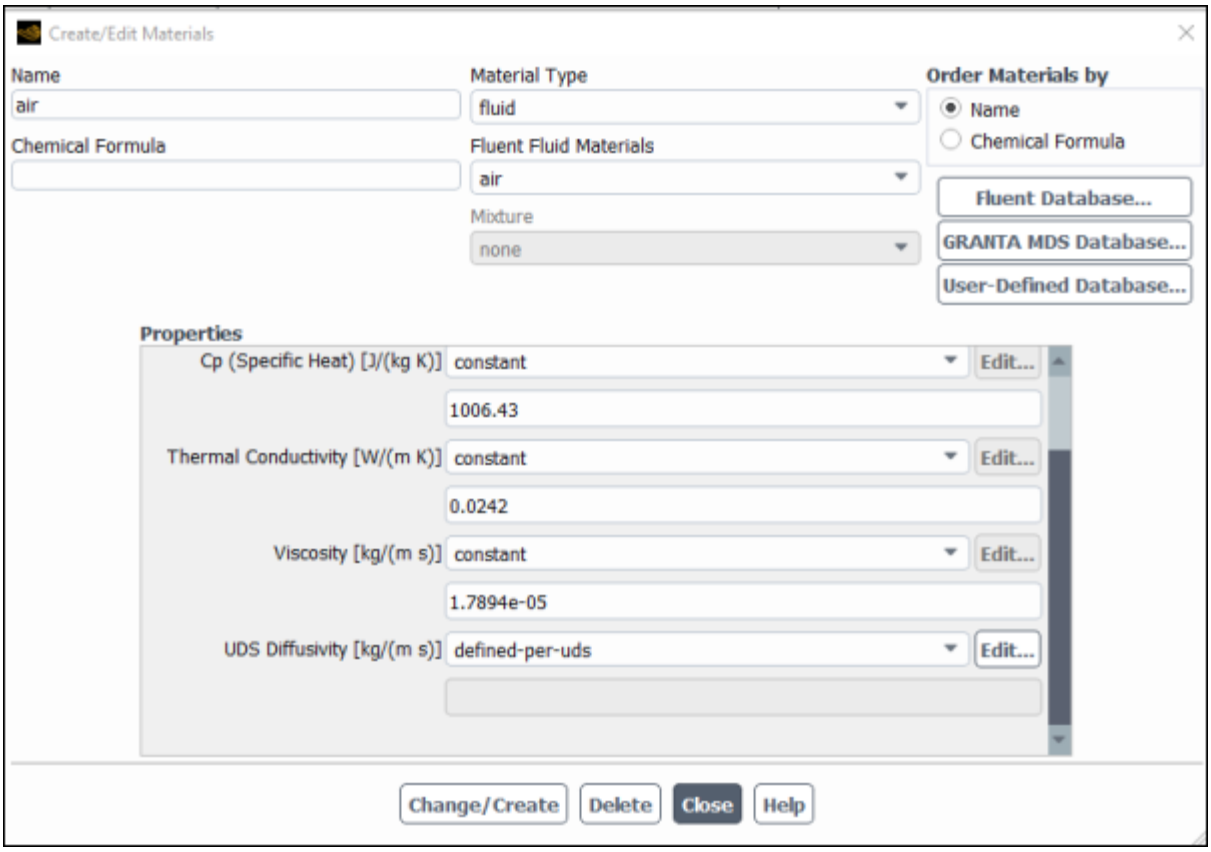
b. Specify the **Number of User Scalar Sources** for the scalar equation by incrementing the counter. Based on the value you have chosen, the sources will be added to the list in the dialog box. Specify each source to be **none**, **constant**, or a user-defined function (UDF). For details on defining a UDF scalar source, see the Fluent Customization Manual. Click **OK** when you have specified all scalar sources.

5. To specify diffusivity for each of the N UDS equations, display the Materials Task Page (p. 4883) (Figure 9.4: The Materials Dialog Box with Input for Diffusivity for UDS Equations (p. 1744)) and select either **defined-per-uds** (the default) or **user-defined** in the drop-down list for **UDS Diffusivity**.

Setup → **Materials** → **Create/Edit**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 9.4: The Materials Dialog Box with Input for Diffusivity for UDS Equations



See User-Defined Scalar (UDS) Diffusivity (p. 1656) for details on the different options available to you for defining diffusion coefficients.

6. To specify boundary conditions for the user-defined scalars on wall, inflow, and outflow boundaries, you can define a specific value or a specific flux for each scalar. A coupled boundary condition can be specified on two-sided walls for scalars that are to be solved in regions on both sides of the wall (that is, scalars solved in both **fluid and solid** zones).

   Setup → Boundary Conditions

   a. In the **UDS** tab under **User Defined Scalar Boundary Condition**, select either **Specified Flux** or **Specified Value** in the drop-down list next to each scalar (for example, **User Scalar 0**) for a boundary wall. For interior walls, select **Coupled Boundary** if the scalars are to be solved on both sides of a two-sided wall. Note that the **Coupled Boundary** option will only show up in the drop-down list if the scalar is defined in the **fluid and solid** zones in the User-Defined Scalars Dialog Box (p. 5654).

   b. Under **User Defined Scalar Boundary Value**, enter a constant value or select a user-defined function from the drop-down list for each scalar. If you select **Specified Flux**, your input will be the value of the flux at the boundary (that is, the negative of the term in parenthesis on the left-hand side of in the Theory Guide) dot [as in the dot product of] n [as in the vector, n], where n is the normal into the domain). If you select **Specified Value**, your input will be the value of the scalar itself at the boundary. See the Fluent Customization Manual for information on using UDFs for UDS boundary conditions.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Set the solution parameters in the **Solution Controls** task page, specify an initial value for each UDS (as you do for all other scalar transport equations), and calculate a solution.

8. Examine the results using the usual postprocessing tools. In each postprocessing dialog box, the list of field variables will include the **User Defined Scalars...** category, which contains the value of each UDS and its diffusion coefficient (Γ_k in Equation 1.8, Equation 1.9, Equation 1.10, or Equation 1.11 (in the Theory Guide):

   - **User Scalar-n**
   - **Diffusion Coef. of Scalar-n**

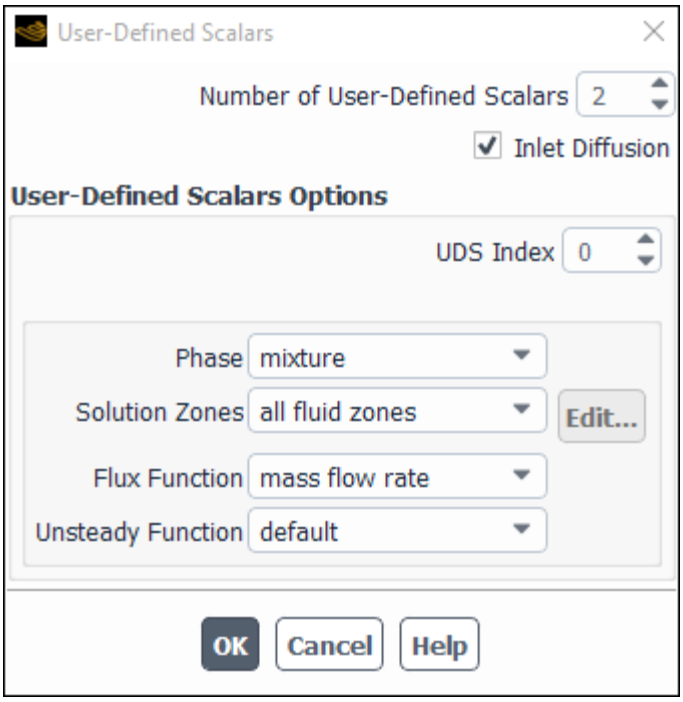
9.1.3.2. Multiphase Flow

1. Specify the number of scalars in the User-Defined Scalars Dialog Box (p. 5654) (Figure 9.5: The User-Defined Scalars Dialog Box for a Multiphase Flow (p. 1746)).

   Parameters & Customization → **User Defined Scalars** **New...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 9.5: The User-Defined Scalars Dialog Box for a Multiphase Flow



Important:

The maximum number of user-defined scalar transport equations you can define is 50. Ansys Fluent assigns numbers to the equations starting with 0. The default association type is set to `mixture` for all scalars.

Important:

Note that Ansys Fluent assigns a default name for each scalar equation (`User Scalar 0`, `User Scalar 1`, and so on). These labels will appear in graphics dialog boxes in Ansys Fluent. You can change them by means of a UDF. See the Fluent Customization Manual for details.

2. Keep the default **Inlet Diffusion** enabled if you want to include the diffusion term in the UDS transport equation for all inflow and outflow boundaries.

3. Set the first user-defined scalar equation parameters by making sure that the **UDS Index** is set to 0.

   a. Select the **Phase** you want the scalar equation solved in as a primary phase, secondary phase, or the mixture.

   b. Specify the **Solution Zones** you want the scalar equation to be solved in as **all fluid zones**, **all solid zones**, **all zones** (fluid and solid) or **selected zones**. If you choose **selected zones**, click the **Edit** button to view the list of zones you can select.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. Specify the **Flux Function** to **Unsteady Function** the same way as you would for a single phase flow (see above).

d. Repeat this process for each scalar equation by incrementing the **UDS Index**.

e. Click **OK** when all user scalar equations have been defined.

4. Specify source term(s) for each of the N UDS equations in the **Fluid** or **Solid** dialog box as described for a single phase flow (see above).

5. Specify boundary conditions for the user-defined scalars in the mixture on all wall, inflow, and outflow boundary as described for a single phase flow (see above).

6. Set the solution parameters, specify an initial value for each UDS (as you do for all other scalar transport equations), and calculate a solution.

9.2. Periodic Flows

Periodic flow occurs when the physical geometry of interest and the expected pattern of the flow/thermal solution have a periodically repeating nature. Two types of periodic flow can be modeled in Ansys Fluent. In the first type, no pressure drop occurs across the periodic planes. In the second type, a pressure drop occurs across translationally periodic boundaries, resulting in "fully-developed" or "streamwise-periodic" flow.

This section discusses streamwise-periodic flow. A description of no-pressure-drop periodic flow is provided in Periodic Boundary Conditions (p. 1501), and a description of streamwise-periodic heat transfer is provided in Modeling Periodic Heat Transfer (p. 2274).

Information about streamwise-periodic flow is presented in the following sections:

9.2.1. Overview and Limitations
9.2.2. User Inputs for the Pressure-Based Solver
9.2.3. User Inputs for the Density-Based Solvers
9.2.4. Monitoring the Value of the Pressure Gradient
9.2.5. Postprocessing for Streamwise-Periodic Flows

For more information about the theoretical background of periodic flows, see Periodic Flows in the Theory Guide.

9.2.1. Overview and Limitations

More information about periodic flows is presented in the following sections:

9.2.1.1. Overview
9.2.1.2. Limitations for Modeling Streamwise-Periodic Flow

9.2.1.1. Overview

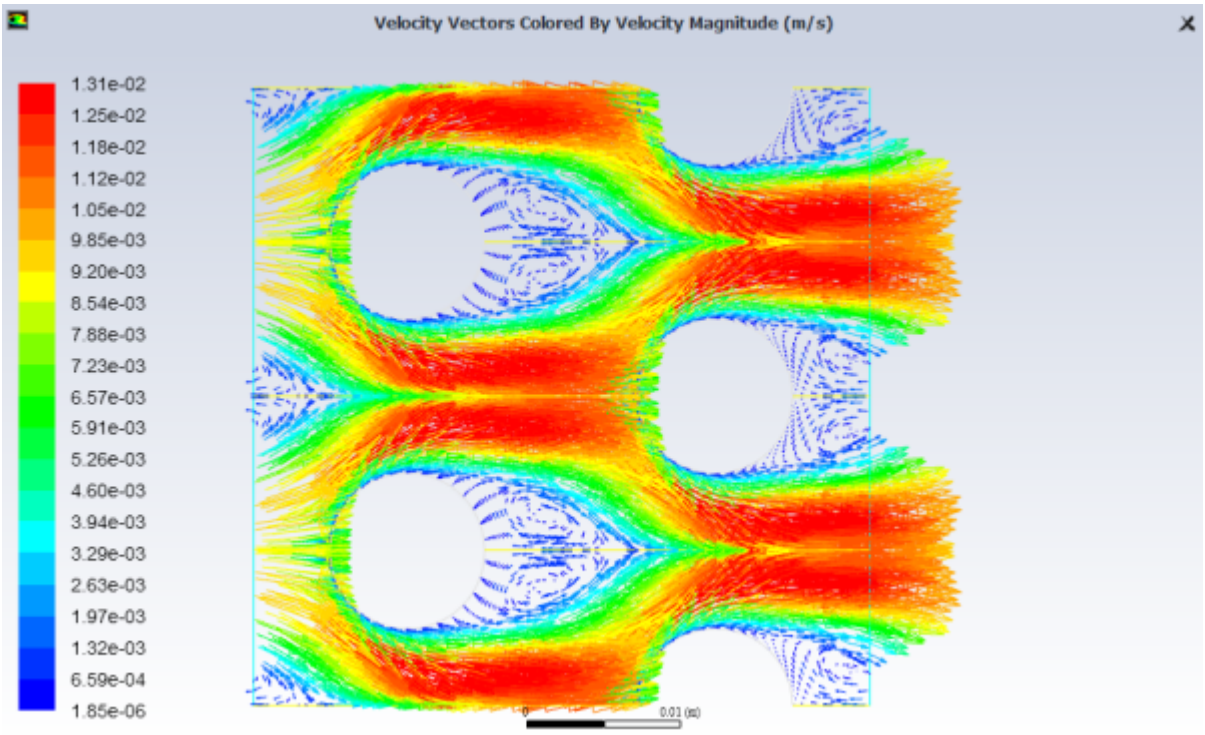
Ansys Fluent provides the ability to calculate streamwise-periodic—or "fully-developed"—fluid flow. These flows are encountered in a variety of applications, including flows in compact heat exchanger channels and flows across tube banks. In such flow configurations, the geometry varies in a repeating

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

manner along the direction of the flow, leading to a periodic fully-developed flow regime in which the flow pattern repeats in successive cycles. Other examples of streamwise-periodic flows include fully-developed flow in pipes and ducts. These periodic conditions are achieved after a sufficient entrance length, which depends on the flow Reynolds number and geometric configuration.

Streamwise-periodic flow conditions exist when the flow pattern repeats over some length L, with a constant pressure drop across each repeating module along the streamwise direction. Figure 9.6: Example of Periodic Flow in a 2D Heat Exchanger Geometry (p. 1748) depicts one example of a periodically repeating flow of this type that has been modeled by including a single representative module. Note that the figure shows the results plotted for the one periodic pair and two symmetries.

Figure 9.6: Example of Periodic Flow in a 2D Heat Exchanger Geometry



9.2.1.2. Limitations for Modeling Streamwise-Periodic Flow

The following limitations apply to modeling streamwise-periodic flow:

- The flow must be incompressible.
- When performing unsteady-state simulations with translational periodic boundary conditions, the specified pressure gradient is recommended.
- If one of the density-based solvers is used, you can specify only the pressure jump; for the pressure-based solver, you can specify either the pressure jump or the mass flow rate.
- No net mass addition through inlets/exits or extra source terms is allowed.
- Species can be modeled only if inlets/exits (without net mass addition) are included in the problem. Reacting flows are not permitted.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- When you specify a periodic mass-flow rate, Fluent will assume that the entire flow rate passes through one periodic continuous face zone only. It is not supported to define a mass-flow periodic condition for more than 1 periodic boundary condition (that is, it does not work for 2 periodic pairs).
- Steady particle tracks can be modeled only if the particles have a possibility to leave the domain without generating incomplete trajectories.
- While multiphase flow can be modeled with translational periodic boundary conditions, you cannot use the mass flow rate specification method. However, you can specify a constant pressure gradient.
- For the pressure-based solver, the periodic boundary condition with specified non-zero mass flow rate and/or pressure gradient cannot be used for variable density flows.
- For the density-based solver, the periodic pressure jump cannot be used when density is a function of pressure.
- When displaying contours of solution variables related to temperature and pressure (such as static temperature, enthalpy, and so on), each nodal value on the periodic boundary is calculated as the weighted average of the values in the cells contacting that node. Since no cells adjacent to the periodic shadow zone contribute to the average, contour values might differ between corresponding places on periodic zones and periodic shadow zones.

For periodic simulations that include heat transfer, see also the limitations described in Constraints for Periodic Heat Transfer Predictions (p. 2274).

9.2.2. User Inputs for the Pressure-Based Solver

Note:

The **Pressure Gradient** refers to the linear component of the calculated pressure gradient.

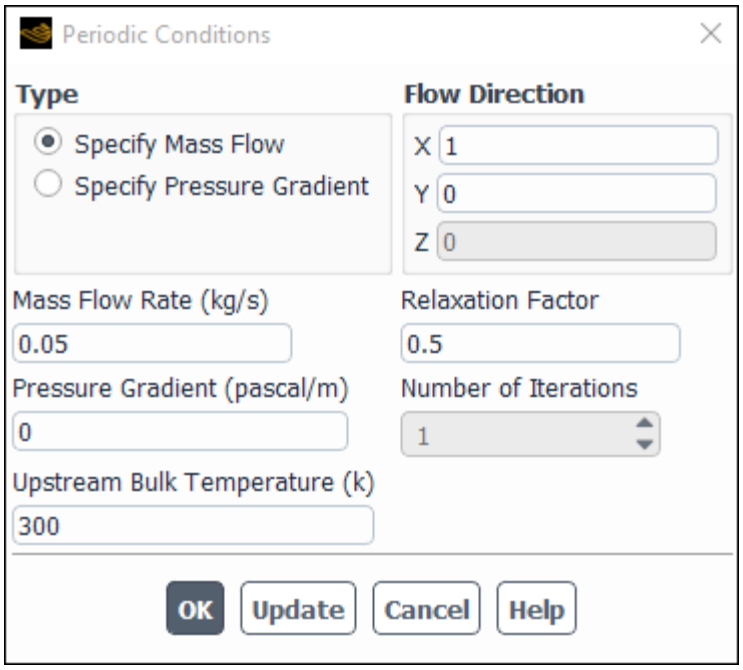
If you are using the pressure-based solver, in order to calculate a spatially periodic flow field with a specified mass flow rate or pressure derivative, you must first create a mesh with translationally periodic boundaries that are parallel to each other and equal in size. You can specify translational periodicity in the Periodic Conditions Dialog Box (p. 5104), as described in Periodic Boundary Conditions (p. 1501). (If you need to create periodic boundaries, see Creating Periodic Zones and Interfaces (p. 1269)).

In the Periodic Conditions Dialog Box (p. 5104) that is opened from the Boundary Conditions Task Page (p. 4991), you will complete the following inputs after the mesh has been read into Ansys Fluent (Figure 9.7: The Periodic Conditions Dialog Box (p. 1750)):

Setup → **Boundary Conditions** → **Periodic Conditions...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 9.7: The Periodic Conditions Dialog Box



1. Select either the specified mass flow rate (**Specify Mass Flow**) option or the specified pressure gradient (**Specify Pressure Gradient**) option. For most problems, the mass flow rate across the periodic boundary will be a known quantity; for others, the mass flow rate will be unknown, but the pressure gradient (β in Equation 1.22, in the Theory Guide) will be a known quantity.

2. Specify the mass flow rate and/or the pressure gradient (β in Equation 1.22, in the Theory Guide):

   - If you selected the **Specify Mass Flow** option, enter the desired value for the **Mass Flow Rate**. You can also specify an initial guess for the **Pressure Gradient**, but this is not required.

     Important:

     For axisymmetric problems, the mass flow rate is per 2π radians.

   - If you selected the **Specify Pressure Gradient** option, enter the desired value for **Pressure Gradient**.

3. Define the flow direction by setting the **X,Y,Z** (or **X,Y** in 2D) point under **Flow Direction**. The flow will move in the direction of the vector pointing from the origin to the specified point. The direction vector must be parallel to the periodic translation direction or its opposite.

4. If you chose in step 1 to specify the mass flow rate, set the parameters used for the calculation of β. These parameters are described in detail below.

After completing these inputs, you can solve the periodic velocity field to convergence.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.2.2.1. Setting Parameters for the Calculation of β

If you choose to specify the mass flow rate, Ansys Fluent must calculate the appropriate value of the pressure gradient β. You can control this calculation by specifying the **Relaxation Factor** and the **Number of Iterations**, and by supplying an initial guess for β. All of these inputs are entered in the Periodic Conditions Dialog Box (p. 5104).

The **Number of Iterations** sets the number of sub-iterations performed on the correction of β in the pressure correction equation. Because the value of β is not known a priori, it must be iterated on until the **Mass Flow Rate** that you have defined is achieved in the computational model. This correction of β occurs in the pressure correction step of the SIMPLE, SIMPLEC, or PISO algorithm. A correction to the current value of β is calculated based on the difference between the desired mass flow rate and the actual one. The sub-iterations referred to here are performed within the pressure correction step to improve the correction for β before the pressure correction equation is solved for the resulting pressure (and velocity) correction values. The default value of 2 sub-iterations should suffice in most problems, but can be increased to help speed convergence. The **Relaxation Factor** is an under-relaxation factor that controls convergence of this iteration process.

You can also speed up convergence of the periodic calculation by supplying an initial guess for β in the **Pressure Gradient** field. Note that the current value of β will be displayed in this field if you have performed any calculations. To update the **Pressure Gradient** field with the current value at any time, click the **Update** button.

9.2.3. User Inputs for the Density-Based Solvers

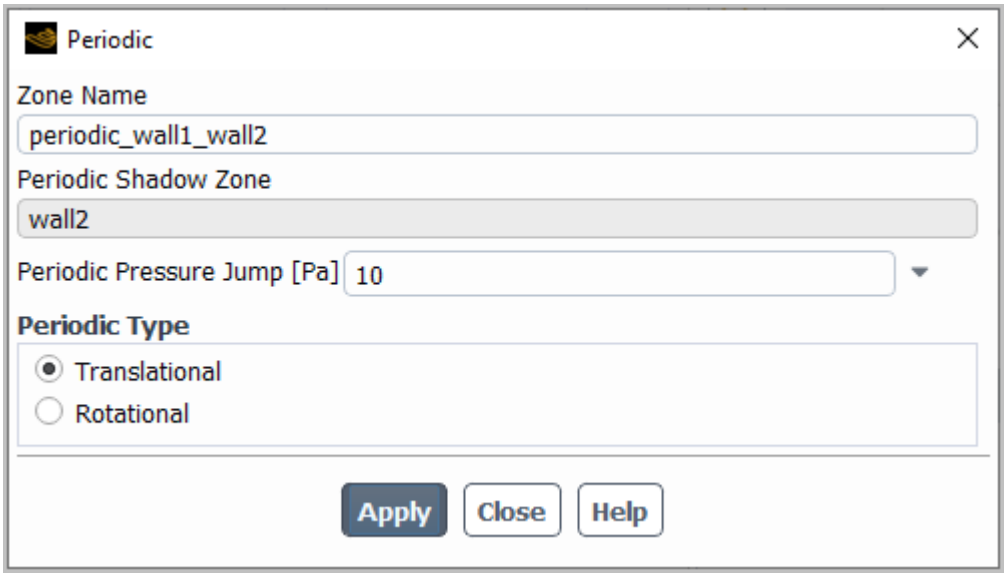
If you are using one of the density-based solvers, in order to calculate a spatially periodic flow field with a specified pressure jump, you must first create a mesh with translationally periodic boundaries that are parallel to each other and equal in size. (If you need to create periodic boundaries, see Creating Periodic Zones and Interfaces (p. 1269).)

Then, follow the steps below:

1. In the Periodic Dialog Box (p. 5041) (Figure 9.8: The Periodic Dialog Box (p. 1752)), which is opened from the **Boundary Conditions** task page, indicate that the periodicity is **Translational** (the default).

   Setup → **Boundary Conditions** → **Periodic** → **Edit...**

Figure 9.8: The Periodic Dialog Box



2. Also in the Periodic Dialog Box (p. 5041), set the **Periodic Pressure Jump** (Δp in Equation 1.21 in the Theory Guide).

After completing these inputs, you can solve the periodic velocity field to convergence.

9.2.4. Monitoring the Value of the Pressure Gradient

If you have specified the mass flow rate, you can monitor the value of the pressure gradient β during the calculation by creating a report plot that includes periodic pressure gradient, to ensure that you reach a converged solution. See Monitoring Statistics (p. 3695) for additional information.

9.2.5. Postprocessing for Streamwise-Periodic Flows

Note:

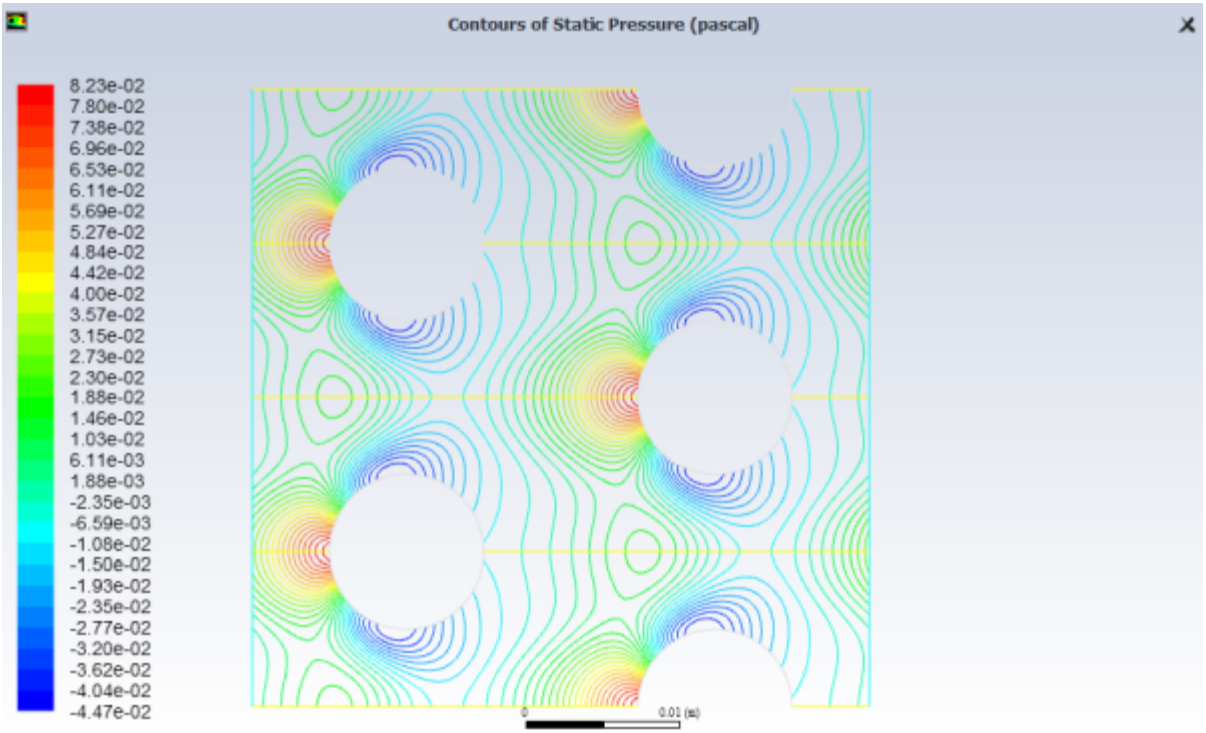
Contours of **Pressure Gradient** are represented by an auxiliary pressure field used by the solver and does not represent a real value.

For streamwise-periodic flows, the velocity field should be completely periodic. If a density-based solver is used to compute the periodic flow, only the linear component of the pressure field will be reported (which is not periodic). If the pressure-based solver is used, the pressure field reported will be the periodic pressure field $\tilde{p}(\vec{r})$ of Equation 1.22, in the Theory Guide. Figure 9.9: Periodic Pressure Field Predicted for Flow in a 2D Heat Exchanger Geometry (p. 1753) displays the periodic pressure field in the geometry of Figure 9.6: Example of Periodic Flow in a 2D Heat Exchanger Geometry (p. 1748).

If you specified a mass flow rate and had Ansys Fluent calculate the pressure gradient, you can check the pressure gradient in the streamwise direction (β) by looking at the current value for **Pressure Gradient** in the Periodic Conditions Dialog Box (p. 5104).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 9.9: Periodic Pressure Field Predicted for Flow in a 2D Heat Exchanger Geometry



9.3. Swirling and Rotating Flows

Many important engineering flows involve swirl or rotation and Ansys Fluent is well-equipped to model such flows. Swirling flows are common in combustion, with swirl introduced in burners and combustors in order to increase residence time and stabilize the flow pattern. Rotating flows are also encountered in turbomachinery, mixing tanks, and a variety of other applications.

Information about rotating and swirling flows is provided in the following subsections:

9.3.1. Overview of Swirling and Rotating Flows
9.3.2. Turbulence Modeling in Swirling Flows
9.3.3. Mesh Setup for Swirling and Rotating Flows
9.3.4. Modeling Axisymmetric Flows with Swirl or Rotation

For more information about the theoretical background of swirling and rotating flows, see Swirling and Rotating Flows in the Theory Guide.

When you begin the analysis of a rotating or swirling flow, it is essential that you classify your problem into one of the following five categories of flow:

- axisymmetric flows with swirl or rotation
- fully three-dimensional swirling or rotating flows
- flows requiring a moving reference frame
- flows requiring multiple moving reference frames or mixing planes

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- flows requiring sliding meshes

Modeling and solution procedures for the first two categories are presented in this section. The remaining three, which all involve "moving zones", are discussed in Modeling Flows with Moving Reference Frames (p. 1769).

9.3.1. Overview of Swirling and Rotating Flows

An overview of swirling and rotating flows is presented in the following sections:

9.3.1.1. Axisymmetric Flows with Swirl or Rotation

9.3.1.2. Three-Dimensional Swirling Flows

9.3.1.3. Flows Requiring a Moving Reference Frame

9.3.1.1. Axisymmetric Flows with Swirl or Rotation

Your problem may be axisymmetric with respect to geometry and flow conditions but still include swirl or rotation. In this case, you can model the flow in 2D (that is, solve the axisymmetric problem) and include the prediction of the circumferential (or swirl) velocity. It is important to note that while the assumption of axisymmetry implies that there are no circumferential gradients in the flow, there may still be nonzero swirl velocities.

9.3.1.1.1. Momentum Conservation Equation for Swirl Velocity

The tangential momentum equation for 2D swirling flows may be written as

$$\frac{\partial}{\partial t}\left(\rho w\right)+\frac{1}{r}\frac{\partial}{\partial x}\left(r\rho uw\right)+\frac{1}{r}\frac{\partial}{\partial r}\left(r\rho vw\right) = \frac{1}{r}\frac{\partial}{\partial x}\left[r\mu\frac{\partial w}{\partial x}\right] + \frac{1}{r^2}\frac{\partial}{\partial r}\left[r^3\mu\frac{\partial}{\partial r}\left(\frac{w}{r}\right)\right] - \rho\frac{vw}{r} \tag{9.14}$$

where x is the axial coordinate, r is the radial coordinate, u is the axial velocity, v is the radial velocity, and w is the swirl velocity.

9.3.1.2. Three-Dimensional Swirling Flows

When there are geometric changes and/or flow gradients in the circumferential direction, your swirling flow prediction requires a three-dimensional model. If you are planning a 3D Ansys Fluent model that includes swirl or rotation, you should be aware of the setup constraints listed in Coordinate System Restrictions (p. 1756). In addition, you might consider simplifications to the problem that might reduce it to an equivalent axisymmetric problem, especially for your initial modeling effort. Because of the complexity of swirling flows, an initial 2D study, in which you can quickly determine the effects of various modeling and design choices, can be very beneficial.

Important:

For 3D problems involving swirl or rotation, there are no special inputs required during the problem setup and no special solution procedures. Note, however, that you may want to use the cylindrical coordinate system for defining velocity-inlet boundary condition inputs, as described in Defining the Velocity (p. 1413). Also, you may find the gradual increase of the rotational speed (set as a wall or inlet boundary condition) helpful during

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the solution process. This is described for axisymmetric swirling flows in Improving Solution Stability by Gradually Increasing the Rotational or Swirl Speed (p. 1759).

9.3.1.3. Flows Requiring a Moving Reference Frame

If your flow involves a rotating boundary that moves through the fluid (for example, an impeller blade or a grooved or notched surface), you must use a moving reference frame to model the problem. Such applications are described in detail in Introduction (p. 1769). If you have more than one rotating boundary (for example, several impellers in a row), you can use multiple reference frames (described in The Multiple Reference Frame Model (p. 1779)) or mixing planes (described in Legacy Mixing Plane Model (p. 1966)).

9.3.2. Turbulence Modeling in Swirling Flows

If you are modeling turbulent flow with a significant amount of swirl (for example, cyclone flows, swirling jets), you should consider using one of Ansys Fluent's advanced turbulence models: the RNG k- ε model, realizable k- ε model, or Reynolds stress model. The appropriate choice depends on the strength of the swirl, which can be gauged by the swirl number. The swirl number is defined as the ratio of the axial flux of angular momentum to the axial flux of axial momentum:

$$S=\frac{\int rw\vec{v}\cdot d\vec{A}}{\overline{R}\int u\vec{v}\cdot d\vec{A}} \tag{9.15}$$

where $\overline{R}$ is the hydraulic radius.

For flows with weak to moderate swirl ($S<0.5$), both the RNG k- ε model and the realizable k- ε model yield appreciable improvements over the standard k- ε model. See RNG k-ε Model and Realizable k-ε Model Swirl Modification (p. 2136) for details about these models.

For highly swirling flows ($S>0.5$), the Reynolds stress model (RSM) is strongly recommended. The effects of strong turbulence anisotropy can be modeled rigorously only by the second-moment closure adopted in the RSM. See Reynolds Stress Model (RSM) Steps in Using a Turbulence Model (p. 2081) for details about this model.

For swirling flows encountered in devices such as cyclone separators and swirl combustors, near-wall turbulence modeling is quite often a secondary issue at most. The fidelity of the predictions in these cases is mainly determined by the accuracy of the turbulence model in the core region. However, in cases where walls actively participate in the generation of swirl (that is, where the secondary flows and vortical flows are generated by pressure gradients), non-equilibrium wall functions can often improve the predictions since they use a law of the wall for mean velocity sensitized to pressure gradients. See Near-Wall Treatments for Wall-Bounded Turbulent Flows in the Theory Guide for additional details about near-wall treatments for turbulence.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.3.3. Mesh Setup for Swirling and Rotating Flows

9.3.3.1. Coordinate System Restrictions

Recall that for an axisymmetric problem, the axis of rotation must be the x axis and the mesh must lie on or above the $y=0$ line.

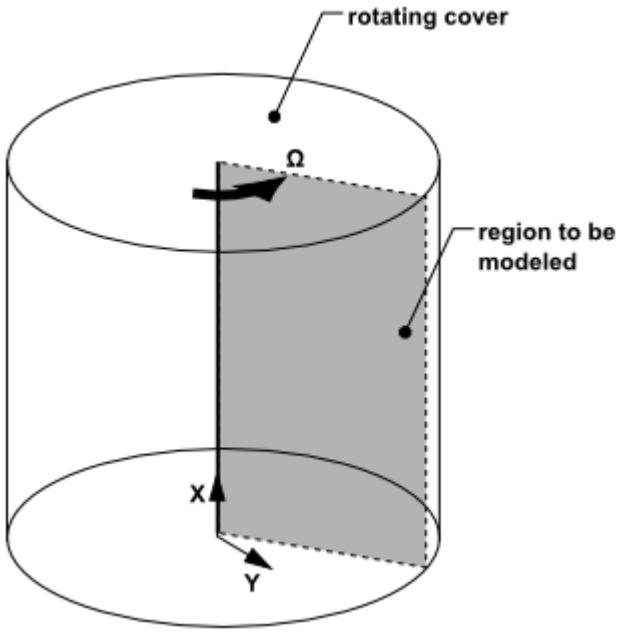
9.3.3.2. Mesh Sensitivity in Swirling and Rotating Flows

In addition to the setup constraint described above, you should be aware of the need for sufficient resolution in your mesh when solving flows that include swirl or rotation. Typically, rotating boundary layers may be very thin, and your Ansys Fluent model will require a very fine mesh near a rotating wall. In addition, swirling flows will often involve steep gradients in the circumferential velocity (for example, near the centerline of a free-vortex type flow), and therefore require a fine mesh for accurate resolution.

9.3.4. Modeling Axisymmetric Flows with Swirl or Rotation

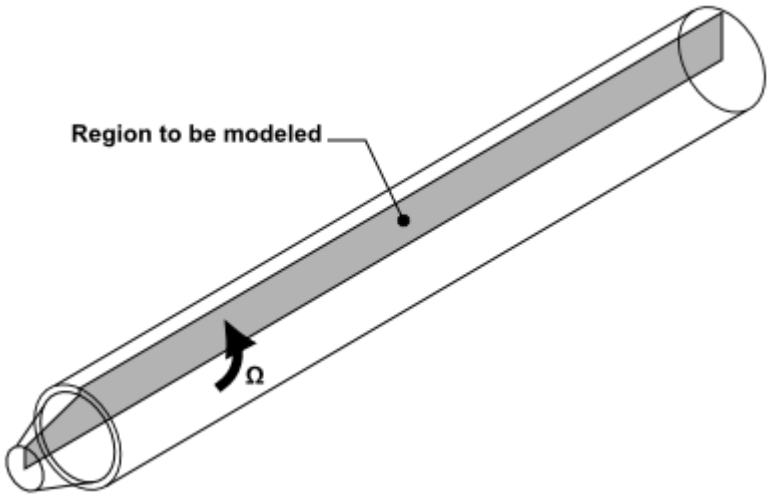
As discussed in Overview of Swirling and Rotating Flows (p. 1754), you can solve a 2D axisymmetric problem that includes the prediction of the circumferential or swirl velocity. The assumption of axisymmetry implies that there are no circumferential gradients in the flow, but that there may be nonzero circumferential velocities. Examples of axisymmetric flows involving swirl or rotation are depicted in Figure 9.10: Rotating Flow in a Cavity (p. 1756) and Figure 9.11: Swirling Flow in a Gas Burner (p. 1757).

Figure 9.10: Rotating Flow in a Cavity



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 9.11: Swirling Flow in a Gas Burner



9.3.4.1. Problem Setup for Axisymmetric Swirling Flows

For axisymmetric problems, you must perform the following steps during the problem setup procedure. (Only those steps relevant specifically to the setup of axisymmetric swirl/rotation are listed here. You must set up the rest of the problem as usual.)

1. Enable the solution of the momentum equation in the circumferential direction by turning on the **Axisymmetric Swirl** option for **Space** in the **General** task page.

   Setup → **General** → **Axisymmetric Swirl**

2. Define the rotational or swirling component of velocity, $r\Omega$, at inlets or walls.

   Setup → **Boundary Conditions**

   Important:

   Remember to use the axis boundary type for the axis of rotation.

The procedures for input of rotational velocities at inlets and at walls are described in detail in Defining the Velocity (p. 1413) and Velocity Conditions for Moving Walls (p. 1462).

9.3.4.2. Solution Strategies for Axisymmetric Swirling Flows

The difficulties associated with solving swirling and rotating flows are a result of the high degree of coupling between the momentum equations, which is introduced when the influence of the rotational terms is large. A high level of rotation introduces a large radial pressure gradient that drives the flow in the axial and radial directions. This, in turn, determines the distribution of the swirl or rotation in the field. This coupling may lead to instabilities in the solution process, and you may require special solution techniques in order to obtain a converged solution. Solution techniques that may be beneficial in swirling or rotating flow calculations include the following:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- (Pressure-based segregated solver only) Use the PRESTO! scheme (enabled in the **Pressure** list for **Spatial Discretization** in the Solution Methods Task Page (p. 5157)), which is well-suited for the steep pressure gradients involved in swirling flows.
- Ensure that the mesh is sufficiently refined to resolve large gradients in pressure and swirl velocity.
- (Pressure-based solver only) Change the under-relaxation parameters on the velocities, perhaps to 0.3–0.5 for the radial and axial velocities and 0.8–1.0 for swirl.
- (Pressure-based solver only) Use a sequential or step-by-step solution procedure, in which some equations are temporarily left inactive (see below).
- If necessary, start the calculations using a low rotational speed or inlet swirl velocity, increasing the rotation or swirl gradually in order to reach the final desired operating condition (see below).

See Using the Solver (p. 3571) for details on the procedures used to make these changes to the solution parameters. More details on the step-by-step procedure and on the gradual increase of the rotational speed are provided below.

9.3.4.2.1. Step-By-Step Solution Procedures for Axisymmetric Swirling Flows

Often, flows with a high degree of swirl or rotation will be easier to solve if you use the following step-by-step solution procedure, in which only selected equations are left active in each step. This approach allows you to establish the field of angular momentum, then leave it fixed while you update the velocity field, and then finally to couple the two fields by solving all equations simultaneously.

Important:

Since the density-based solvers solve all the flow equations simultaneously, the following procedure applies only to the pressure-based solver.

In this procedure, you will use the **Equations...** button in the Solution Controls Task Page (p. 5163) to turn individual transport equations on and off between calculations.

1. If your problem involves inflow/outflow, begin by solving the flow without rotation or swirl effects. That is, enable the **Axisymmetric** option instead of the **Axisymmetric Swirl** option in the General Task Page (p. 4684), and do not set any rotating boundary conditions. The resulting flow-field data can be used as a starting guess for the full problem.
2. Enable the **Axisymmetric Swirl** option and set all rotating/swirling boundary conditions.
3. Begin the prediction of the rotating/swirling flow by solving only the momentum equation describing the circumferential velocity. This is the **Swirl Velocity** listed in the **Equations** list in the Equations Dialog Box (p. 5166). Let the rotation "diffuse" throughout the flow field, based on your boundary condition inputs. In a turbulent flow simulation, you may also want to leave the turbulence equations active during this step. This step will establish the field of rotation throughout the domain.
4. Turn off the momentum equations describing the circumferential motion (**Swirl Velocity**). Leaving the velocity in the circumferential direction fixed, solve the momentum and continuity (pressure) equations (**Flow** in the **Equations** list in the Equations Dialog Box (p. 5166)) in the

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

other coordinate directions. This step will establish the axial and radial flows that are a result of the rotation in the field. Again, if your problem involves turbulent flow, you should leave the turbulence equations active during this calculation.

5. Turn on all of the equations simultaneously to obtain a fully coupled solution. Note the under-relaxation controls suggested above.

In addition to the steps above, you may want to simplify your calculation by solving isothermal flow before adding heat transfer or by solving laminar flow before adding a turbulence model. These two methods can be used for any of the solvers (that is, pressure-based or density-based).

9.3.4.2.2. Improving Solution Stability by Gradually Increasing the Rotational or Swirl Speed

Because the rotation or swirl defined by the boundary conditions can lead to large complex forces in the flow, your Ansys Fluent calculations will be less stable as the speed of rotation or degree of swirl increases. Hence, one of the most effective controls you can apply to the solution is to solve your rotating flow problem starting with a low rotational speed or swirl velocity and then slowly increase the magnitude up to the desired level. The procedure for accomplishing this is as follows:

1. Set up the problem using a low rotational speed or swirl velocity in your inputs for boundary conditions. The rotation or swirl in this first attempt might be selected as 10% of the actual operating conditions.

2. Solve the problem at these conditions, perhaps using the step-by-step solution strategy outlined above.

3. Save this initial solution data.

4. Modify your inputs (boundary conditions). Increase the speed of rotation, perhaps doubling it.

5. Restart the calculation using the solution data saved in step 3 as the initial solution for the new calculation. Save the new data.

6. Continue to increment the speed of rotation, following steps 4 and 5, until you reach the desired operating condition.

9.3.4.2.2.1. Postprocessing for Axisymmetric Swirling Flows

Reporting of results for axisymmetric swirling flows is the same as for other flows. The following additional variables are available for postprocessing when axisymmetric swirl is active:

- **Swirl Velocity** (in the **Velocity...** category)
- **Swirl-Wall Shear Stress** (in the **Wall Fluxes...** category)

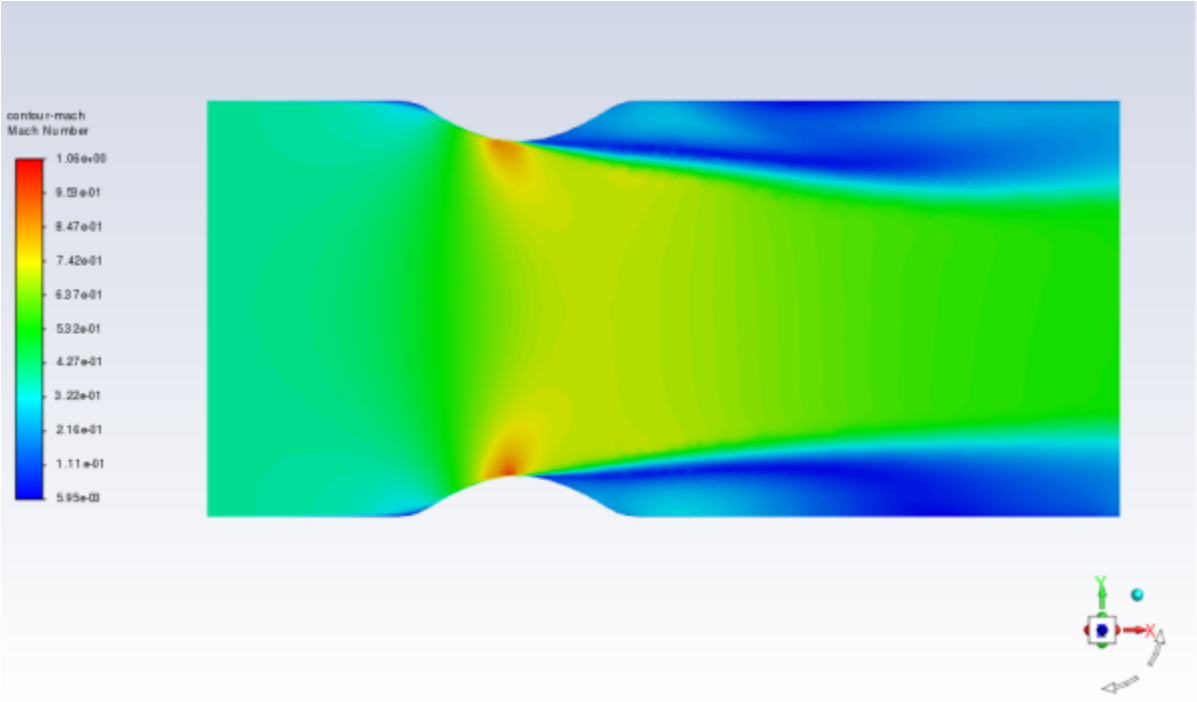
9.4. Compressible Flows

Compressibility effects are encountered in gas flows at high velocity and/or in which there are large pressure variations. When the flow velocity approaches or exceeds the speed of sound of the gas or

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

when the pressure change in the system ($\Delta p / p$) is large, the variation of the gas density with pressure has a significant impact on the flow velocity, pressure, and temperature. Compressible flows create a unique set of flow physics for which you must be aware of the special input requirements and solution techniques described in this section. Figure 9.12: Flow in a Converging-Diverging Nozzle (p. 1760) shows compressible flows computed using Ansys Fluent.

Figure 9.12: Flow in a Converging-Diverging Nozzle



Information about compressible flows is provided in the following subsections:

9.4.1. When to Use the Compressible Flow Model

9.4.2. Physics of Compressible Flows

9.4.3. Modeling Inputs for Compressible Flows

9.4.4. Floating Operating Pressure

9.4.5. Solution Strategies for Compressible Flows

9.4.6. Reporting of Results for Compressible Flows

For more information about the theoretical background of compressible flows, see Compressible Flows in the Theory Guide.

9.4.1. When to Use the Compressible Flow Model

Compressible flows can be characterized by the value of the Mach number:

$$M \equiv u / c \tag{9.16}$$

Here, c is the speed of sound in the gas:

$$c = \sqrt{\gamma RT} \tag{9.17}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and γ is the ratio of specific heats $\left(c_p / c_v\right)$.

When the Mach number is less than 1.0, the flow is termed subsonic. At Mach numbers much less than 1.0 (M 0.1 or so), compressibility effects are negligible and the variation of the gas density with pressure can safely be ignored in your flow modeling. As the Mach number approaches 1.0 (which is referred to as the transonic flow regime), compressibility effects become important. When the Mach number exceeds 1.0, the flow is termed supersonic, and may contain shocks and expansion fans which can impact the flow pattern significantly. Ansys Fluent provides a wide range of compressible flow modeling capabilities for subsonic, transonic, and supersonic flows.

9.4.2. Physics of Compressible Flows

Compressible flows are typically characterized by the total pressure p_0 and total temperature T_0 of the flow. For an ideal gas, these quantities can be related to the static pressure and temperature by the following:

$$\frac{p_0}{p} = exp\left(\frac{\int_T^{T_0} \frac{C_p}{T} dT}{R}\right) \tag{9.18}$$

For constant C_p, Equation 9.18 (p. 1761) reduces to

$$\begin{aligned} \frac{p_0}{p} &= \left(1+\frac{\gamma-1}{2}M^2\right)^{\gamma/(\gamma-1)} \\ \frac{T_0}{T} &= 1+\frac{\gamma-1}{2}M^2 \end{aligned} \tag{9.19}$$

These relationships describe the variation of the static pressure and temperature in the flow as the velocity (Mach number) changes under isentropic conditions. For example, given a pressure ratio from inlet to exit (total to static), Equation 9.19 (p. 1761) can be used to estimate the exit Mach number that would exist in a one-dimensional isentropic flow. For air, Equation 9.19 (p. 1761) p / p_0, of 0.5283. This choked flow condition will be established at the point of minimum flow area (for example, in the throat of a nozzle). In the subsequent area expansion the flow may either accelerate to a supersonic flow in which the pressure will continue to drop, or return to subsonic flow conditions, decelerating with a pressure rise. If a supersonic flow is exposed to an imposed pressure increase, a shock will occur, with a sudden pressure rise and deceleration accomplished across the shock.

9.4.2.1. Basic Equations for Compressible Flows

Compressible flows are described by the standard continuity and momentum equations solved by Ansys Fluent, and you do not need to enable any special physical models (other than the compressible treatment of density as detailed below). The energy equation solved by Ansys Fluent correctly incorporates the coupling between the flow velocity and the static temperature, and should be activated whenever you are solving a compressible flow. In addition, if you are using the pressure-based solver, you should enable the viscous dissipation terms in Equation 5.1 in the Theory Guide, which become important in high-Mach-number flows.

9.4.2.2. The Compressible Form of the Gas Law

For compressible flows, the ideal gas law is written in the following form:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\rho = \frac{p_{op} + p}{\frac{R}{M_w} T} \tag{9.20}$$

where p_{op} is the operating pressure defined in the Operating Conditions Dialog Box (p. 4978), p is the local static pressure relative to the operating pressure, R is the universal gas constant, and M_w is the molecular weight. The temperature, T, will be computed from the energy equation.

Some compressible flow problems involve fluids that do not behave as ideal gases. For example, flow under very high-pressure conditions cannot typically be modeled accurately using the ideal-gas assumption. Therefore, the real gas model described in Real Gas Models (p. 1681) should be used instead.

9.4.3. Modeling Inputs for Compressible Flows

To set up a compressible flow in Ansys Fluent, you must follow the steps listed below. (Only those steps relevant specifically to the setup of compressible flows are listed here. You must set up the rest of the problem as usual.)

1. Set the **Operating Pressure** in the Operating Conditions Dialog Box (p. 4978).

   Setup → **Boundary Conditions** → **Operating Conditions...**

   (You can think of p_{op} as the absolute static pressure at a point in the flow where you will define the gauge pressure p to be zero. See Operating Pressure (p. 1677) for guidelines on setting the operating pressure. For time-dependent compressible flows, you may want to specify a floating operating pressure instead of a constant operating pressure. See Floating Operating Pressure (p. 1763) for details.)

2. Enable the solution of the energy equation.

   Setup → **Models** → **Energy** **On**

3. (Pressure-based solver only) If you are modeling turbulent flow, enable the optional viscous dissipation terms in the energy equation by turning on **Viscous Heating** in the Viscous Model Dialog Box (p. 4715). Note that these terms can be important in high-speed flows.

   Setup → **Models** → **Viscous** → **Edit...**

   This step is not necessary if you are using one of the density-based solvers, because the density-based solvers always include the viscous dissipation terms in the energy equation.

4. Set the following items in the Create/Edit Materials Dialog Box (p. 4885):

   Setup → **Materials** → **Create/Edit...**

   a. Select **ideal-gas** in the drop-down list next to **Density**.

   b. Define all relevant properties (specific heat, molecular weight, thermal conductivity, and so on).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Set cell zone conditions and boundary conditions (using the Boundary Conditions Task Page (p. 4991) and Cell Zone Conditions Task Page (p. 4958)), being sure to choose a well-posed cell zone or boundary condition combination that is appropriate for the flow regime. See below for details. Recall that all inputs for pressure (either total pressure or static pressure) must be relative to the operating pressure, and the temperature inputs at inlets should be total (stagnation) temperatures, *not* static temperatures.

   Setup → Cell Zone Conditions

   Setup → Boundary Conditions

These inputs should ensure a well-posed compressible flow problem. You will also want to consider special solution parameter settings, as noted in Solution Strategies for Compressible Flows (p. 1765), before beginning the flow calculation.

9.4.3.1. Boundary Conditions for Compressible Flows

Well-posed inlet and exit boundary conditions for compressible flow are listed below:

- For flow inlets:
  - Pressure inlet: Inlet total temperature and total pressure and, for supersonic inlets, static pressure
  - Mass-flow inlet: Inlet mass flow and total temperature
- For flow exits:
  - Pressure outlet: Exit static pressure (ignored if flow is supersonic at the exit. All the information travels downstream in a supersonic region, hence the pressure at the outlet can be computed by directly extrapolating from the adjacent cell center [64] (p. 5716). Therefore, it is not meaningful to use the exit static pressure prescribed in the boundary conditions task page, and the exit static pressure is ignored).
  - Mass-flow outlet: Outlet mass flow

It is important to note that your boundary condition inputs for pressure (either total pressure or static pressure) must be in terms of gauge pressure — that is, pressure relative to the operating pressure defined in the Operating Conditions Dialog Box (p. 4978), as described above.

All temperature inputs at inlets should be total (stagnation) temperatures, *not* static temperatures.

9.4.4. Floating Operating Pressure

Ansys Fluent provides a "floating operating pressure" option to handle time-dependent compressible flows with a gradual increase in the absolute pressure in the domain. This option is desirable for slow subsonic flows with static pressure build-up, since it efficiently accounts for the slow changing of absolute pressure without using acoustic waves as the transport mechanism for the pressure build-up.

Examples of typical applications include the following:

- combustion or heating of a gas in a closed domain

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- pumping of a gas into a closed domain

9.4.4.1. Limitations

The floating operating pressure option should *not* be used for transonic or incompressible flows. In addition, it cannot be used if your model includes any pressure inlet, pressure outlet, exhaust fan, inlet vent, intake fan, outlet vent, or pressure far field boundaries.

9.4.4.2. Theory

The floating operating pressure option allows Ansys Fluent to calculate the pressure rise (or drop) from the integral mass balance, separately from the solution of the pressure correction equation. When this option is activated, the absolute pressure at each iteration can be expressed as

$$p_{abs} = p_{op,float} + p \tag{9.21}$$

where p is the pressure relative to the reference location, which in this case is in the cell with the minimum pressure value. Therefore the reference location itself is floating.

$p_{op,float}$ is referred to as the floating operating pressure, and is defined as

$$p_{op,float} = p^0_{op} + \Delta p_{op} \tag{9.22}$$

where p^0_{op} is the initial operating pressure and Δp_{op} is the pressure rise.

Including the pressure rise Δp_{op} in the floating operating pressure $p_{op,float}$, rather than in the pressure p, helps to prevent roundoff error. If the pressure rise were included in p, the calculation of the pressure gradient for the momentum equation would give an inexact balance due to precision limits for 32-bit real numbers.

9.4.4.3. Enabling Floating Operating Pressure

When time dependence is active, you can turn on the **Floating Operating Pressure** option in the Operating Conditions Dialog Box (p. 4978).

Setup → **Boundary Conditions** → **Operating Conditions...**

(Note that the inputs for **Reference Pressure Location** will disappear when you enable **Floating Operating Pressure**, since these inputs are no longer relevant.)

Important:

The floating operating pressure option should *not* be used for transonic flows or for incompressible flows. It is meaningful only for slow subsonic flows of ideal or real gases, when the characteristic time scale is much larger than the sonic time scale.

9.4.4.4. Setting the Initial Value for the Floating Operating Pressure

When the floating operating pressure option is enabled, you must specify a value for the **Initial Operating Pressure** in the Solution Initialization Task Page (p. 5179).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution → **Initialization**

This initial value is stored in the case file with all your other initial values.

9.4.4.5. Storage and Reporting of the Floating Operating Pressure

The current value of the floating operating pressure is stored in the data file. If you visit the Operating Conditions Dialog Box (p. 4978) after a number of time steps have been performed, the current value of the **Operating Pressure** will be displayed.

Note that the floating operating pressure will automatically be reset to the initial operating pressure if you reset the data (that is, start over at the first iteration of the first time step).

9.4.4.6. Monitoring Absolute Pressure

You can monitor the absolute pressure during the calculation using the Surface Report Definition Dialog Box (p. 5626). You can also generate graphical plots or alphanumeric reports of absolute pressure when your solution is complete. The **Absolute Pressure** variable is contained in the **Pressure...** category of the variable selection drop-down list that appears in postprocessing dialog boxes. See Field Function Definitions (p. 4167) for its definition.

9.4.5. Solution Strategies for Compressible Flows

The difficulties associated with solving compressible flows are a result of the high degree of coupling between the flow velocity, density, pressure, and energy. This coupling may lead to instabilities in the solution process and, therefore, may require special solution techniques in order to obtain a converged solution. In addition, the presence of shocks (discontinuities) in the flow introduces an additional stability problem during the calculation. Solution techniques that may be beneficial in compressible flow calculations include the following:

- (Pressure-based solver only) Initialize the flow to be near stagnation (that is velocity small but not zero, pressure to inlet total pressure, temperature to inlet total temperature). Turn off the energy equation for the first 50 iterations. Leave the energy under-relaxation at 1. Set the pressure under-relaxation to 0.4, and the momentum under-relaxation to 0.3. After the solution stabilizes and the energy equation has been turned on, increase the pressure under-relaxation to 0.7.

- Set reasonable limits for the temperature and pressure (in the Solution Limits Dialog Box (p. 5166)) to avoid solution divergence, especially at the start of the calculation. If Ansys Fluent prints messages about temperature or pressure being limited as the solution nears convergence, the high or low computed values may be physical, and you must change the limits to allow these values.

- If required, begin the calculations using a reduced pressure ratio at the boundaries, increasing the pressure ratio gradually in order to reach the final desired operating condition. If the Mach number is low, you can also consider starting the compressible flow calculation from an incompressible flow solution (although the incompressible flow solution can in some cases be a rather poor initial guess for the compressible calculation).

- In some cases, computing an inviscid solution as a starting point may be helpful.

See Using the Solver (p. 3571) for details on the procedures used to make these changes to the solution parameters.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9.4.6. Reporting of Results for Compressible Flows

You can display the results of your compressible flow calculations in the same manner that you would use for an incompressible flow. The variables listed below are of particular interest when you model compressible flow:

- **Total Temperature**
- **Total Pressure**
- **Mach Number**

These variables are contained in the variable selection drop-down list that appears in postprocessing dialog boxes. **Total Temperature** is in the **Temperature...** category, **Total Pressure** is in the **Pressure...** category, and **Mach Number** is in the **Velocity...** category. See Field Function Definitions (p. 4167) for their definitions.

9.5. Inviscid Flows

Inviscid flow analysis neglect the effect of viscosity on the flow and are appropriate for high-Reynolds-number applications where inertial forces tend to dominate viscous forces. One example for which an inviscid flow calculation is appropriate is an aerodynamic analysis of some high-speed projectile. In a case like this, the pressure forces on the body will dominate the viscous forces. Hence, an inviscid analysis will give you a quick estimate of the primary forces acting on the body. After the body shape has been modified to maximize the lift forces and minimize the drag forces, you can perform a viscous analysis to include the effects of the fluid viscosity and turbulent viscosity on the lift and drag forces.

Another area where inviscid flow analysis are routinely used is to provide a good initial solution for problems involving complicated flow physics and/or complicated flow geometry. In a case like this, the viscous forces are important, but in the early stages of the calculation the viscous terms in the momentum equations will be ignored. Once the calculation has been started and the residuals are decreasing, you can turn on the viscous terms (by enabling laminar or turbulent flow) and continue the solution to convergence. For some very complicated flows, this is the only way to get the calculation started.

Information about inviscid flows is provided in the following subsections:

9.5.1. Setting Up an Inviscid Flow Model
9.5.2. Solution Strategies for Inviscid Flows
9.5.3. Postprocessing for Inviscid Flows

For more information about the theoretical background of inviscid flows, see Inviscid Flows in the Theory Guide.

9.5.1. Setting Up an Inviscid Flow Model

For inviscid flow problems, you must perform the following steps during the problem setup procedure. (Only those steps relevant specifically to the setup of inviscid flow are listed here. You must set up the rest of the problem as usual.)

1. Enable the calculation of inviscid flow by selecting **Inviscid** in the Viscous Model Dialog Box (p. 4715).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setup → **Models** → **Viscous** → **Model** → **Inviscid**

2. Set boundary conditions and flow properties.

Setup → **Boundary Conditions**

Note:

Walls are assumed to be slip surfaces (the velocity is not equal to zero, unlike viscous flows) and therefore have a tangential velocity computed based on the solution of the governing equations.

Setup → **Materials**

3. Solve the problem and examine the results.

9.5.2. Solution Strategies for Inviscid Flows

Since inviscid flow problems will usually involve high-speed flow, you may have to reduce the under-relaxation factors for momentum (if you are using the pressure-based solver) or reduce the Courant number (if you are using the density-based solver), in order to get the solution started. Once the flow is started and the residuals are monotonically decreasing, you can start increasing the under-relaxation factors or Courant number back up to the default values.

Modifications to the under-relaxation factors and the Courant number can be made in the Solution Controls Task Page (p. 5163).

Solution → **Controls**

The solution strategies for compressible flows apply also to inviscid flows. See Solution Strategies for Compressible Flows (p. 1765) for details.

9.5.3. Postprocessing for Inviscid Flows

If you are interested in the lift and drag forces acting on your model, you can use the Force Reports Dialog Box (p. 5322) to compute them.

Results → **Reports** → **Forces** **Edit...**

See Forces on Boundaries (p. 4145) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 10: Modeling Flows with Moving Reference Frames

This chapter provides details about the moving reference frame capabilities in Ansys Fluent.

The information in this chapter is divided into the following sections:

10.1. Introduction
10.2. Flow in Single Moving Reference Frames (SRF)
10.3. Flow in Multiple Moving Reference Frames

10.1. Introduction

Ansys Fluent solves the equations of fluid flow and heat transfer, by default, in a stationary (or inertial) reference frame. However, there are many problems where it is advantageous to solve the equations in a moving (or non-inertial) reference frame. Such problems typically involve moving parts (such as rotating blades, impellers, and similar types of moving surfaces), and it is the flow around these moving parts that is of interest. In most cases, the moving parts render the problem unsteady when viewed from the stationary frame. With a moving reference frame, however, the flow around the moving part can (with certain restrictions) be modeled as a steady-state problem with respect to the moving frame.

Ansys Fluent's moving reference frame modeling capability allows you to model problems involving moving parts by allowing you to enable moving reference frames in selected cell zones. When a moving reference frame is activated, the equations of motion are modified to incorporate the additional acceleration terms that occur due to the transformation from the stationary to the moving reference frame. By solving these equations in a steady-state manner, the flow around the moving parts can be modeled.

For many problems, it may be possible to refer the entire computational domain to a single moving reference frame. This is known as the single reference frame (or SRF) approach. The use of the SRF approach is possible; provided the geometry meets certain requirements. For more complex geometries, it may not be possible to use a single reference frame. In such cases, you must break up the problem into multiple cell zones, with well-defined interfaces between the zones. This is the multiple reference frame (or MRF) approach and is discussed in The Multiple Reference Frame Model (p. 1779). If unsteady interaction between the stationary and moving parts is important, you can employ the sliding mesh approach to capture the transient behavior of the flow. The sliding meshing model will be discussed in Modeling Flows Using Sliding and Dynamic Meshes (p. 1797).

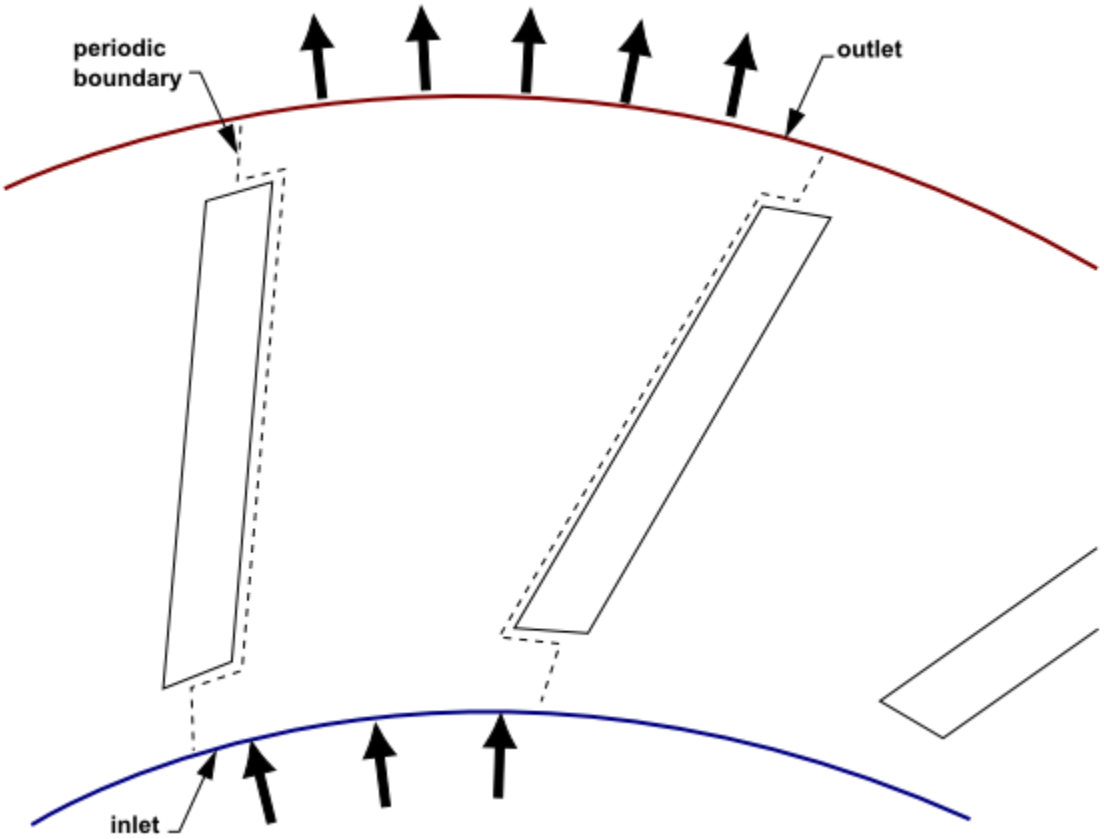
The principal reason for employing a moving reference frame is to render a problem that is unsteady in the stationary (inertial) frame, steady with respect to the moving frame. For a steadily moving frame (for example, the frame speed is constant), it is possible to transform the equations of fluid motion to the moving frame such that steady-state solutions are possible. By default, Ansys Fluent permits the activation of a moving reference frame with a steady speed. If the speed is not constant, the transformed equations will contain additional terms (see Relative Velocity Formulation in the Theory Guide). It should also be noted that you can run an unsteady simulation in a moving reference frame with constant

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

speed. This would be necessary if you wanted to simulate, for example, vortex shedding from a rotating fan blade. The unsteadiness in this case is due to a natural fluid instability (vortex generation) rather than induced from interaction with a stationary component.

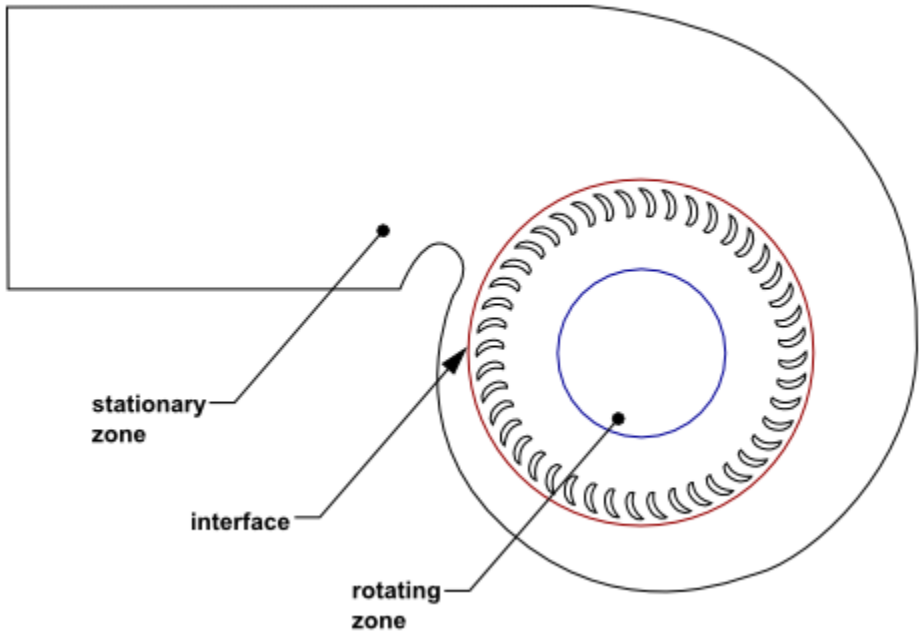
For more information about the equations for moving reference frames, see Equations for a Moving Reference Frame in the Theory Guide.

Figure 10.1: Single Component (Blower Wheel Blade Passage)



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.2: Multiple Component (Blower Wheel and Casing)



10.2. Flow in Single Moving Reference Frames (SRF)

Many problems permit the entire computational domain to be referred to a single moving reference frame (hence the name SRF modeling). In such cases, the equations given in Equations for a Moving Reference Frame are solved in all fluid cell zones. Steady-state solutions are possible in SRF models provided suitable boundary conditions are prescribed. In particular, wall boundaries must adhere to the following requirements:

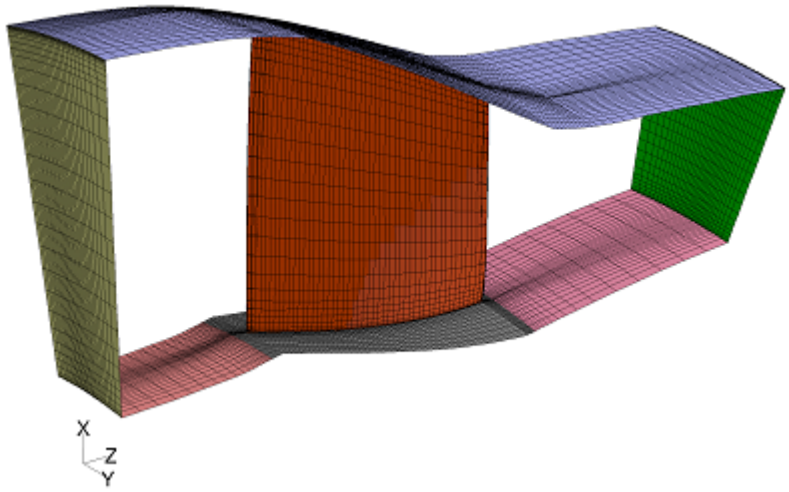
- Any walls that are moving with the reference frame can assume any shape. An example would be the blade surfaces associated with a pump impeller. The no slip condition is defined in the relative frame such that the relative velocity is zero on the moving walls.
- For a rotating problem, you can define walls that are non-moving with respect to the stationary coordinate system, but these walls must be surfaces of revolution about the axis of rotation. Here the no slip condition is defined such that the absolute velocity is zero on the walls. An example of this type of boundary would be a cylindrical wind tunnel wall that surrounds a rotating propeller.

Rotationally periodic boundaries may also be used, but the surface must be periodic about the axis of rotation. As an example, it is very common to model flow through a blade row of a turbomachine by assuming the flow to be rotationally periodic and using a periodic domain about a single blade. This permits good resolution of the flow around the blade without the expense of modeling all blades in the blade row (see Figure 10.3: Single Blade Model with Rotationally Periodic Boundaries (p. 1772)).

Flow boundary conditions in Ansys Fluent (inlets and outlets) can, in most cases, be prescribed in either the stationary or moving frames. For example, for a velocity inlet, one can specify either the relative velocity or absolute velocity, depending on which is more convenient. For additional information on these and other boundary conditions, see Setting Up a Single Moving Reference Frame Problem (p. 1772) and Cell Zone and Boundary Conditions (p. 1299).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.3: Single Blade Model with Rotationally Periodic Boundaries



10.2.1. Mesh Setup for a Single Moving Reference Frame

It is important to remember the following coordinate-system constraints when you are setting up a problem involving a moving reference frame for a rotating problem:

- For 2D problems, the axis of rotation must be parallel to the z axis.
- For 2D axisymmetric problems, the axis of rotation must be the x axis.
- For 3D geometries, you should generate the mesh with a specific origin and rotational axis in mind for the rotating cell zone. Usually it is convenient to use the origin of the global coordinate system (0,0,0) for the frame origin, and either the x, y, or z axis for the rotational axis; however, Ansys Fluent can accommodate an arbitrary origin and rotational axis.

With 3D rotating problems, it is also important to note that if you want to include walls that have zero velocity in the stationary frame, these walls must be a surface of revolution with respect to the axis of rotation. If the stationary walls are not surfaces of revolution, you must encapsulate the rotating parts with interface boundaries, thereby breaking your model up into multiple zones, and use the MRF model for a steady-state solution (see The Multiple Reference Frame Model (p. 1779)), or the sliding mesh model for unsteady interaction (see Modeling Flows Using Sliding and Dynamic Meshes (p. 1797)).

10.2.2. Setting Up a Single Moving Reference Frame Problem

To model a problem involving a single moving reference frame, follow the steps outlined below.

1. Select the **Velocity Formulation** to be used when solving: either **Relative** or **Absolute**. (See Choosing the Relative or Absolute Velocity Formulation (p. 1775) for details.)

   Setup → **General**

   (Note that this step is irrelevant if you are using one of the density-based solvers; these solvers always use an absolute velocity formulation.)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. For each cell zone in the domain, specify the translational velocity of the reference frame and/or the angular velocity (ω) of the reference frame and the axis about which it rotates.

   Setup → Cell Zone Conditions

   a. In the **Fluid** or **Solid** dialog box, specify the **Rotation-Axis Origin** and **Rotation-Axis Direction** for the frame motion in the **Reference Frame** tab, in order to define the axis of rotation.

   b. Also in the **Fluid** (Figure 10.4: The Fluid Dialog Box Displaying Frame Motion Inputs (p. 1774)) or **Solid** dialog box, enable the **Frame Motion** option. (Note that a solid zone cannot move at a different speed than an adjacent solid zone; for such a situation, you must instead use **Solid Motion**.)

      In the **Reference Frame** tab, set the **Speed** under **Rotational Velocity** and/or the **X**, **Y**, and **Z** components of the **Translational Velocity**. Note that the speed can be specified as a constant value or a transient profile. The transient profile may be in a file format, as described in Transient Cell Zone and Boundary Conditions (p. 1523), or a UDF macro, described in `DEFINE_TRANSIENT_PROFILE`. Specifying the individual velocities as either a profile or a UDF allows you to specify a specific input of the frame motion individually. However, you can also specify the frame motion inputs via a single user-defined function that uses the UDF macro `DEFINE_ZONE_MOTION`. This may prove to be quite convenient if you are modeling a more complicated motion of the moving reference frame, where the hooking of many different user-defined functions or profiles can be cumbersome.

      Note:

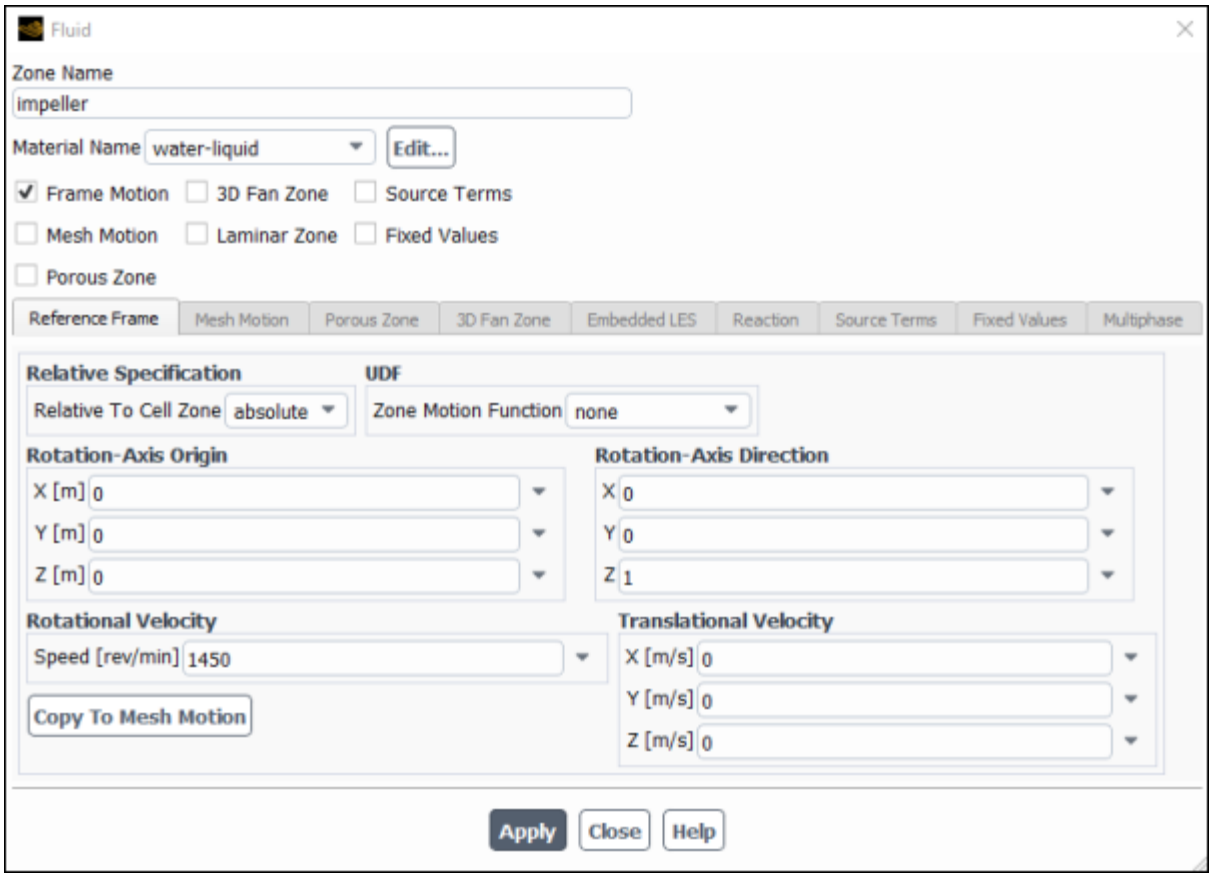
      If you decide to hook a UDF, you will no longer have access to the rotation axis origin and direction, or the velocities.

      Details about these inputs are presented in Inputs for Fluid Zones (p. 1321) and in Inputs for Solid Zones (p. 1327). Details about the zone motion UDF can be found in `DEFINE_ZONE_MOTION` in the Fluent Customization Manual.

   c. If you need to switch between the moving reference frame and moving mesh models, simply click the **Copy To Mesh Motion** for zones with a moving frame of reference and **Copy to Frame Motion** for zones with moving meshes to transfer motion variables, such as the axes, frame origin, and velocity components between the two models. The variables used for the origin, axis, and velocity components, as well as for the UDF `DEFINE_ZONE_MOTION` will be copied. This is particularly useful if you are doing a steady-state MRF simulation to obtain an initial solution for a transient Moving Mesh simulation in a turbomachine.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.4: The Fluid Dialog Box Displaying Frame Motion Inputs



Important:

Normally it is not necessary to enable the **Frame Motion** option for solid zones, as this is not required if you want to do a conjugate heat transfer problem where the solid and fluid zones are moving together. A case in which you would want to enable the **Frame Motion** option is if you are running an intrinsic fluid-structure-interaction (FSI) simulation, and you want the structural model to account for rotational forces when calculating the deformation of the solid cell zone (see Modeling Fluid-Structure Interaction (FSI) Within Fluent (p. 3227)).

3. Define the velocity boundary conditions at walls. You can choose to define either an absolute velocity or a velocity relative to the moving reference frame (that is, relative to the velocity of the adjacent cell zone specified in step 2).

   If the wall is moving at the speed of the moving frame (and hence stationary in the moving frame), it is convenient to specify a relative angular velocity of zero. Likewise, a wall that is stationary in the non-moving frame of reference should be given a velocity of zero in the absolute reference frame. Specifying the wall velocities in this manner obviates the need to modify these inputs later if a change is made in the velocity of the fluid zone.

   Details about these inputs are presented in Velocity Conditions for Moving Walls (p. 1462).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Define the boundary conditions at the inlets, as described in Boundary Conditions (p. 1392). For velocity inlets, you can choose to define either absolute velocities or velocities relative to the motion of the adjacent cell zone (specified in step 2). Likewise, the total pressure and flow direction can be prescribed in absolute or relative frames for pressure inlets.

   Details about these inputs are presented in Defining the Flow Direction (p. 1405) and Defining the Velocity (p. 1413).

5. (optional) For transient cases, after you have initialized or run the calculation you can use the **Moving Mesh Courant Number** field variable (in the **Velocity...** category) to guide your solution. This field variable is a non-dimensional value that indicates the number of cells that might be swept in a single time step due to a mesh motion.

10.2.2.1. Choosing the Relative or Absolute Velocity Formulation

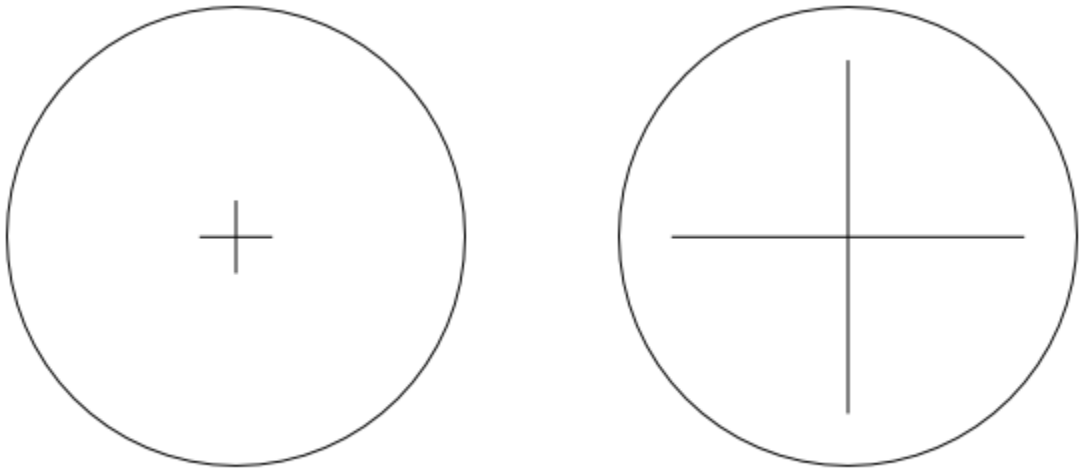
It is recommended that you use the velocity formulation that will result in most of the flow domain having the smallest velocities in that frame, thereby reducing the numerical diffusion in the solution and leading to a more accurate solution.

The absolute velocity formulation is preferred in applications where the flow in most of the domain is not moving (for example, a fan in a large room). The relative velocity formulation is appropriate when most of the fluid in the domain is moving, as in the case of a large impeller in a mixing tank.

10.2.2.1.1. Example

A problem with stationary outer walls and a rotating impeller can be solved in a single reference frame. The example is illustrated in Figure 10.5: Geometry with the Rotating Impeller (p. 1775).

Figure 10.5: Geometry with the Rotating Impeller



In case A, it is expected that only the flow near the impeller would be rotating and that much of the flow away from the impeller would have a low velocity magnitude in the absolute frame. Therefore, solving using the absolute velocity formulation is recommended. In case B, most of the flow is expected to be rotating with a velocity close to that of the impeller. Hence, the relative velocity formulation is appropriate.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In a situation between case A and case B, either of the formulations may be used.

Important:

- If the velocity formulation is switched during the solution process, Ansys Fluent will not transform the current solution to the other frame, which can lead to large jumps in residuals. If changing the frame is necessary, it is recommended that you first reinitialize, and then solve.
- When one of the density-based solution algorithms is used, the absolute formulation is always used; the relative velocity formulation is not available in the density-based solvers.

For velocity inlets, pressure inlets, mass-flow inlets, and walls, you may specify velocity in either the absolute or the relative frame, regardless of whether the absolute or relative velocity formulation is used in the computation. Note, that the inlet turbulence values are processed using the velocity values, which are converted to the Velocity Formulation selected in the General Task Page (p. 4684), see explanations in Estimating Turbulent Kinetic Energy from Turbulence Intensity (p. 1398) and in the introductory part of Inlet Boundary Conditions for Scale Resolving Simulations.

For pressure outlets, the specified static pressure is independent of frame. However, when there is backflow at a pressure outlet, the specified static pressure is used as the total pressure. For calculations using the absolute velocity formulation, the specified static pressure is used as the total pressure in the absolute frame; for the relative velocity formulation, the specified static pressure is assumed to be the total pressure in the relative frame. As for the flow direction, Ansys Fluent assumes the absolute velocity to be normal to the pressure outlet for the absolute velocity formulation; for the relative velocity formulation, it is the relative velocity that is assumed to be normal to the pressure outlet.

10.2.3. Solution Strategies for a Single Moving Reference Frame

The difficulties associated with solving flows in moving reference frames are similar to those discussed in Solution Strategies for Axisymmetric Swirling Flows (p. 1757). The primary issue you must confront is the high degree of coupling between the momentum equations when the influence of the rotational terms is large. A high degree of rotation introduces a large radial pressure gradient that drives the flow in the axial and radial directions, thereby setting up a distribution of the swirl or rotation in the field. This coupling may lead to instabilities in the solution process, and hence require special solution techniques to obtain a converged solution. Some techniques that may be beneficial include the following:

- (Pressure-based solver only) Consider switching the frame in which velocities are solved by changing the velocity formulation setting in the General Task Page (p. 4684). (See Choosing the Relative or Absolute Velocity Formulation (p. 1775) for details.)
- (Pressure-based segregated solver only) Use the PRESTO! scheme (enabled in the Solution Methods Task Page (p. 5157)), which is well-suited for the steep pressure gradients involved in rotating flows.
- Ensure that the mesh is sufficiently refined to resolve large gradients in pressure and swirl velocity.
- (Pressure-based, segregated solver only) Reduce the under-relaxation factors for the velocities, perhaps to 0.3–0.5 or lower, if necessary.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Begin the calculations using a low rotational speed, increasing the rotational speed gradually in order to reach the final desired operating condition.

See Using the Solver (p. 3571) for details on the procedures used to make these changes to the solution parameters.

10.2.3.1. Gradual Increase of the Rotational Speed to Improve Solution Stability

Because the rotation of the reference frame and the rotation defined via boundary conditions can lead to large complex forces in the flow, your Ansys Fluent calculations may be less stable as the speed of rotation (and hence the magnitude of these forces) increases. One of the most effective controls you can exert on the solution is to start with a low rotational speed and then slowly increase the rotation up to the desired level. The procedure you use to accomplish this is as follows:

1. Set up the problem using a low rotational speed in your inputs for boundary conditions and for the angular velocity of the reference frame. The rotational speed in this first attempt might be selected as 10% of the actual operating condition.

2. Solve the problem at these conditions.

3. Save this initial solution data.

4. Modify your inputs (that is, boundary conditions and angular velocity of the reference frame). Increase the speed of rotation, perhaps doubling it.

5. Restart or continue the calculation using the solution data saved in Step 3 as the initial guess for the new calculation. Save the new data.

6. Continue to increment the rotational speed, following Steps 4 and 5, until you reach the desired operating condition.

10.2.4. Postprocessing for a Single Moving Reference Frame

When you solve a problem in a moving reference frame, you can plot or report both absolute and relative velocities. For all velocity parameters (for example, **Velocity Magnitude** and **Mach Number**), corresponding relative values will be available for postprocessing (for example, **Relative Velocity Magnitude** and **Relative Mach Number**). These variables are contained in the **Velocity...** category of the variable selection drop-down list that appears in postprocessing dialog boxes. Relative values are also available for postprocessing of total pressure, total temperature, and any other parameters that include a dynamic contribution dependent on the reference frame (for example, **Relative Total Pressure**, **Relative Total Temperature**, **Rothalpy**).

When plotting velocity vectors, you can choose to plot vectors in the absolute frame (the default), or you can select **Relative Velocity** in the **Vectors of** drop-down list in the Vectors Dialog Box (p. 5655) to plot vectors in the moving frame. If you plot relative velocity vectors, you might want to color the vectors by relative velocity magnitude (by choosing **Relative Velocity Magnitude** in the **Color by** list); by default they will be colored by absolute velocity magnitude. Figure 10.6: Absolute Velocity Vectors (p. 1778) and Figure 10.7: Relative Velocity Vectors (p. 1778) show absolute and relative velocity vectors in a moving domain with a stationary outer wall.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.6: Absolute Velocity Vectors

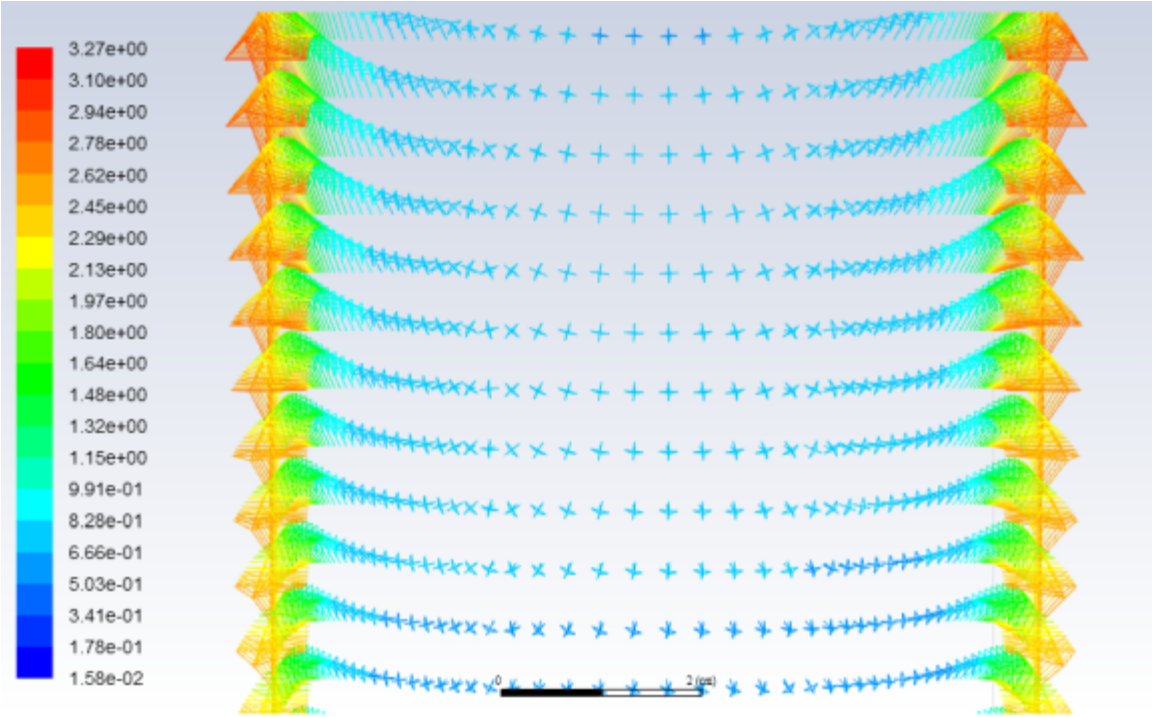
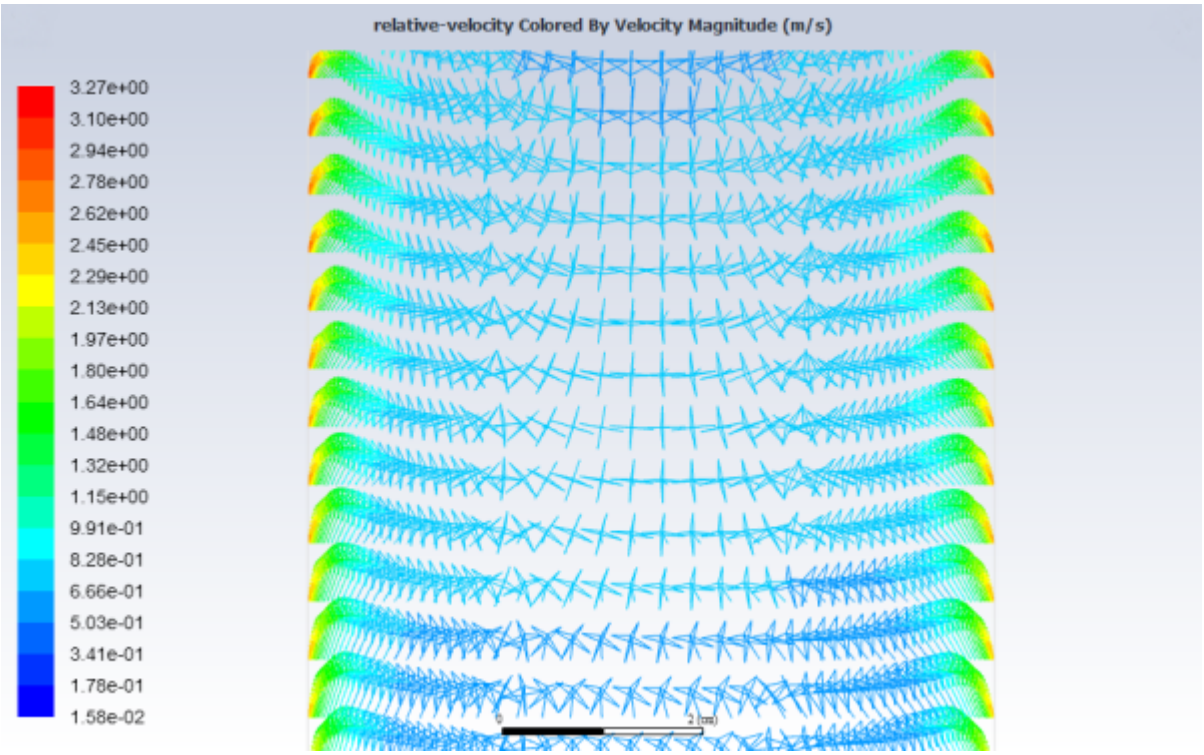


Figure 10.7: Relative Velocity Vectors



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.3. Flow in Multiple Moving Reference Frames

Many problems involve multiple moving parts or contain stationary surfaces which are not surfaces of revolution (and therefore cannot be used with the Single Reference Frame modeling approach). For these problems, you must break up the model into multiple fluid/solid cell zones, with interface boundaries separating the zones. Zones that contain the moving components can then be solved using the moving reference frame equations (Equations for a Moving Reference Frame in the Theory Guide), whereas stationary zones can be solved with the stationary frame equations.

- Multiple Reference Frame Model (MRF)
- Sliding Mesh Model (SMM)

The MRF is a steady-state approximation whereas the SMM approach is inherently unsteady due to the motion of the mesh with time. This approach is discussed in Modeling Flows Using Sliding and Dynamic Meshes (p. 1797).

For additional information, see the following sections:

10.3.1. The Multiple Reference Frame Model
10.3.2. Mesh Setup for a Multiple Moving Reference Frame
10.3.3. Setting Up a Multiple Moving Reference Frame Problem
10.3.4. Solution Strategies for MRF and Problems
10.3.5. Postprocessing for MRF Problems

10.3.1. The Multiple Reference Frame Model

Additional information about the MRF model is presented in the following sections:

10.3.1.1. Overview
10.3.1.2. Limitations

10.3.1.1. Overview

The MRF model [93] (p. 5718) is, perhaps, the simplest of the two approaches for multiple zones. It is a steady-state approximation in which individual cell zones can be assigned different rotational and/or translational speeds. The flow in each moving cell zone is solved using the moving reference frame equations (see Introduction (p. 1769)). If the zone is stationary (ω=0), the equations reduce to their stationary forms. At the interfaces between cell zones, a local reference frame transformation is performed to enable flow variables in one zone to be used to calculate fluxes at the boundary of the adjacent zone. For more information about the MRF interface formulation, see The MRF Interface Formulation in the Theory Guide.

It should be noted that the MRF approach does not account for the relative motion of a moving zone with respect to adjacent zones (which may be moving or stationary); the mesh remains fixed for the computation. This is analogous to freezing the motion of the moving part in a specific position and observing the instantaneous flowfield with the rotor in that position. Hence, the MRF is often referred to as the "frozen rotor approach."

While the MRF approach is clearly an approximation, it can provide a reasonable model of the flow for many applications. For example, the MRF model can be used for turbomachinery applications

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

in which rotor-stator interaction is relatively weak, and the flow is relatively uncomplicated at the interface between the moving and stationary zones. In mixing tanks, for example, since the impeller-baffle interactions are relatively weak, large-scale transient effects are not present and the MRF model can be used.

Another potential use of the MRF model is to compute a flow field that can be used as an initial condition for a transient sliding mesh calculation. This eliminates the need for a startup calculation. The multiple reference frame model should not be used, however, if it is necessary to actually simulate the transients that may occur in strong rotor-stator interactions, the sliding mesh model alone should be used (see Modeling Flows Using Sliding and Dynamic Meshes (p. 1797).

For more information about and examples of multiple moving reference frames, see The Multiple Reference Frame Model in the Theory Guide.

10.3.1.2. Limitations

The following limitations exist when using the MRF approach:

- The interfaces separating a moving region from adjacent regions must be oriented such that the component of the frame velocity normal to the boundary is zero. This means that for a translationally moving frame, the moving zone's boundaries must be parallel to the translational velocity vector. For rotating problems, the interfaces must be surfaces of revolution about the axis of rotation defined for the fluid zone. For the example shown Figure 2.4: Geometry with One Rotating Impeller (in the Theory Guide), this requires the dashed boundary to be circular (not square or any other shape).

- Strictly speaking, the use of multiple reference frames is meaningful only for steady flow. However, Ansys Fluent will allow you to solve an unsteady flow when multiple reference frames are being used. In this case, unsteady terms (as described in Temporal Discretization in the Theory Guide) are added to all the governing transport equations. You should carefully consider whether this will yield meaningful results for your application, because, for unsteady flows, a sliding mesh calculation will generally yield more meaningful results than an MRF calculation.

- By default, pathlines and particle trajectories drawn by Ansys Fluent use the velocity relative to the reference frame motion of the cell zone. However, for particle trajectories, you can change the reference frame by either selecting **Track in Absolute Frame** in the **Numerics** tab of the **Discrete Phase Model** dialog box or using the text command `define/models/dpm/options/track-in-absolute-frame`.

  Note:

  When a pathline or particle trajectory crosses a boundary between zones with different reference frame motion specifications, you may observe a sharp change in particle velocity as a result of the coordinate transformation.

  The particle injection velocities (specified in the Set Injection Properties Dialog Box (p. 5608)) are interpreted in the same way as tracking, that is, either relative to the local frame of reference's motion specification or in absolute coordinates.

- You cannot accurately model axisymmetric swirl in the presence of multiple reference frames using the relative velocity formulation. This is because the current implementation does not apply

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the transformation used in Equation 2.16 (in the Theory Guide) to the swirl velocity derivatives. For this situation, the absolute velocity formulation should be used.

- Translational and rotational velocities are assumed to be constant (time varying ω, v_t are not allowed).
- The relative velocity formulation cannot be used in combination with the MRF and mixture models. (For details, see Mixture Model Theory in the Theory Guide). For such cases, use the absolute velocity formulation instead.
- You must not have a single interface between reference frames where part of the interface is made up of a coupled two-sided wall, while another part is not coupled (that is, the normal interface treatment). In such cases, you must break the interface up into two interfaces: one that is a coupled interface, and the other that is a standard fluid-fluid interface. See Using a Non-Conformal Mesh in Ansys Fluent (p. 1185) for the steps involved in setting up a coupled interface.

Important:

You can switch from the MRF model to the sliding mesh model for a more robust and accurate solution, by using the `mesh/modify-zones/mrf-to-sliding-mesh` text command. See Using Sliding Meshes (p. 1804) for details on how to make this change in the fluid's boundary conditions.

10.3.2. Mesh Setup for a Multiple Moving Reference Frame

Two mesh setup methods are available. Choose the method that is appropriate for your model, noting the restrictions in Limitations (p. 1780).

- If the boundary between two zones that are in different reference frames is conformal (that is, the mesh node locations are identical at the boundary where the two zones meet), you can simply create the mesh as usual, with all cell zones contained in the same mesh file. A different cell zone should exist for each portion of the domain that is modeled in a different reference frame. Use an *interior* zone for the boundary between reference frames.
- If the boundary between two zones that are in different reference frames is non-conformal (that is, the mesh node locations are *not* identical at the boundary where the two zones meet), follow the non-conformal mesh setup procedure described in Using a Non-Conformal Mesh in Ansys Fluent (p. 1185).

10.3.3. Setting Up a Multiple Moving Reference Frame Problem

To learn more about setting up a multiple reference frame problem, see the following sections:

10.3.3.1. Setting Up Multiple Reference Frames

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.3.3.1. Setting Up Multiple Reference Frames

To model a problem involving multiple reference frames, perform the following:

Important:

The mesh-setup constraints for a moving reference frame listed in Mesh Setup for a Single Moving Reference Frame (p. 1772) apply to multiple reference frames as well.

---

1. Select the **Velocity Formulation** to be used in the General Task Page (p. 4684): either **Absolute** or **Relative**. (For details, see Choosing the Relative or Absolute Velocity Formulation (p. 1775).)

   Setup → **General**

   (Note that this step is irrelevant if you are using one of the density-based solution algorithms; these algorithms always use an absolute velocity formulation.)

2. For each cell zone in the domain, specify its translational velocity and/or its angular velocity (ω) and the axis about which it rotates.

   Setup → **Cell Zone Conditions**

   a. If the zone is moving, or if you plan to specify cylindrical velocity or flow-direction components at inlets to the zone, you must define the axis of rotation of the frame of reference. In the **Fluid** or **Solid** dialog box, specify the **Rotation-Axis Origin** and **Rotation-Axis Direction** under the **Reference Frame** tab.

   b. In the **Fluid** (Figure 10.4: The Fluid Dialog Box Displaying Frame Motion Inputs (p. 1774)) or **Solid** dialog box, enable the **Frame Motion** option. (Note that a solid zone cannot move at a different speed than an adjacent solid zone; for such a situation, you must instead use **Solid Motion**.)

      In the **Reference Frame** tab, set the **Speed** under **Rotational Velocity** and/or the **X**, **Y**, and **Z** components of the **Translational Velocity**. Note that the speed can be specified as a constant value or a transient profile. The transient profile may be in a file format, as described in Transient Cell Zone and Boundary Conditions (p. 1523), or a UDF macro, described in `DEFINE_TRANSIENT_PROFILE`. Specifying the individual velocities as either a profile or a UDF allows you to specify a specific input of the frame motion individually. However, you can also specify the frame motion inputs via a single user-defined function that uses the UDF macro `DEFINE_ZONE_MOTION`. This may prove to be quite convenient if you are modeling a more complicated motion of the moving reference frame, where the hooking of many different user-defined functions or profiles can be cumbersome.

      Note:

      If you decide to hook a UDF, then you will no longer have access to the rotation axis origin and direction, or the velocities.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Details about these inputs are presented in Inputs for Fluid Zones (p. 1321) and in Inputs for Solid Zones (p. 1327). Details about the zone motion UDF can be found in `DEFINE_ZONE_MOTION` in the Fluent Customization Manual.

c. To switch between the MRF and moving mesh models, click the **Copy To Mesh Motion** for zones with a moving frame of reference and **Copy to Frame Motion** for zones with moving meshes to transfer motion variables, such as the axes, frame origin, and velocity components between the two models.

The variables used for the origin, axis, and velocity components, as well as for the UDF `DEFINE_ZONE_MOTION` will be copied. This is particularly useful if you are doing a steady-state MRF simulation to obtain an initial solution for a transient Moving Mesh simulation in a turbomachine.

3. Define the velocity boundary conditions at walls. You can choose to define either an absolute velocity or a velocity relative to the velocity of the adjacent cell zone specified in step 2.

If the wall is moving at the speed of the moving frame (and hence stationary relative to the moving frame), it is convenient to specify a relative angular velocity of zero. Likewise, a wall that is stationary in the non-moving frame of reference should be given a velocity of zero in the absolute reference frame. Specifying the wall velocities in this manner obviates the need to modify these inputs later if a change is made in the rotational velocity of the fluid zone.

An example for which you would specify a relative velocity is as follows: If an impeller is defined as **wall-3** and the fluid region within the impeller's radius is defined as **fluid-5**, you would need to specify the angular velocity and axis of rotation for **fluid-5** and then assign **wall-3** a relative velocity of 0. If you later wanted to model a different angular velocity for the impeller, you would need to change only the angular velocity of the fluid region; you would not need to modify the wall velocity conditions.

Details about these inputs are presented in Velocity Conditions for Moving Walls (p. 1462).

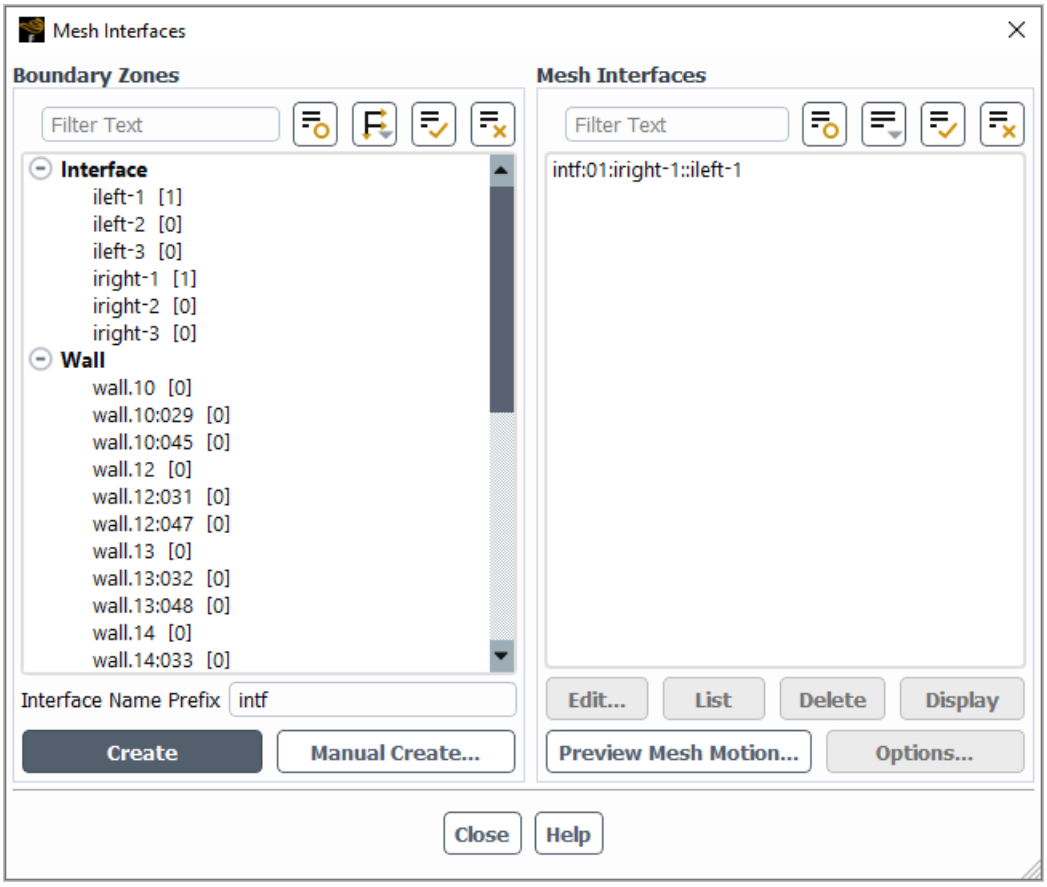
4. Define the boundary conditions at the inlets, as described in Boundary Conditions (p. 1392). For velocity inlets, you can choose to define either absolute velocities or velocities relative to the motion of the adjacent cell zone (specified in step 2). Likewise, the total pressure and flow direction can be prescribed in absolute or relative frames for pressure inlets.

Details about these inputs are presented in Defining the Flow Direction (p. 1405) and Defining the Velocity (p. 1413).

5. Define the mesh interfaces using the Mesh Interfaces Dialog Box (p. 5525) (Figure 13.10: The Mesh Interfaces Dialog Box (p. 1807)).

Setup → Mesh Interfaces → New...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



To learn how to use the **Mesh Interfaces** dialog box, see Using a Non-Conformal Mesh in Ansys Fluent (p. 1185).

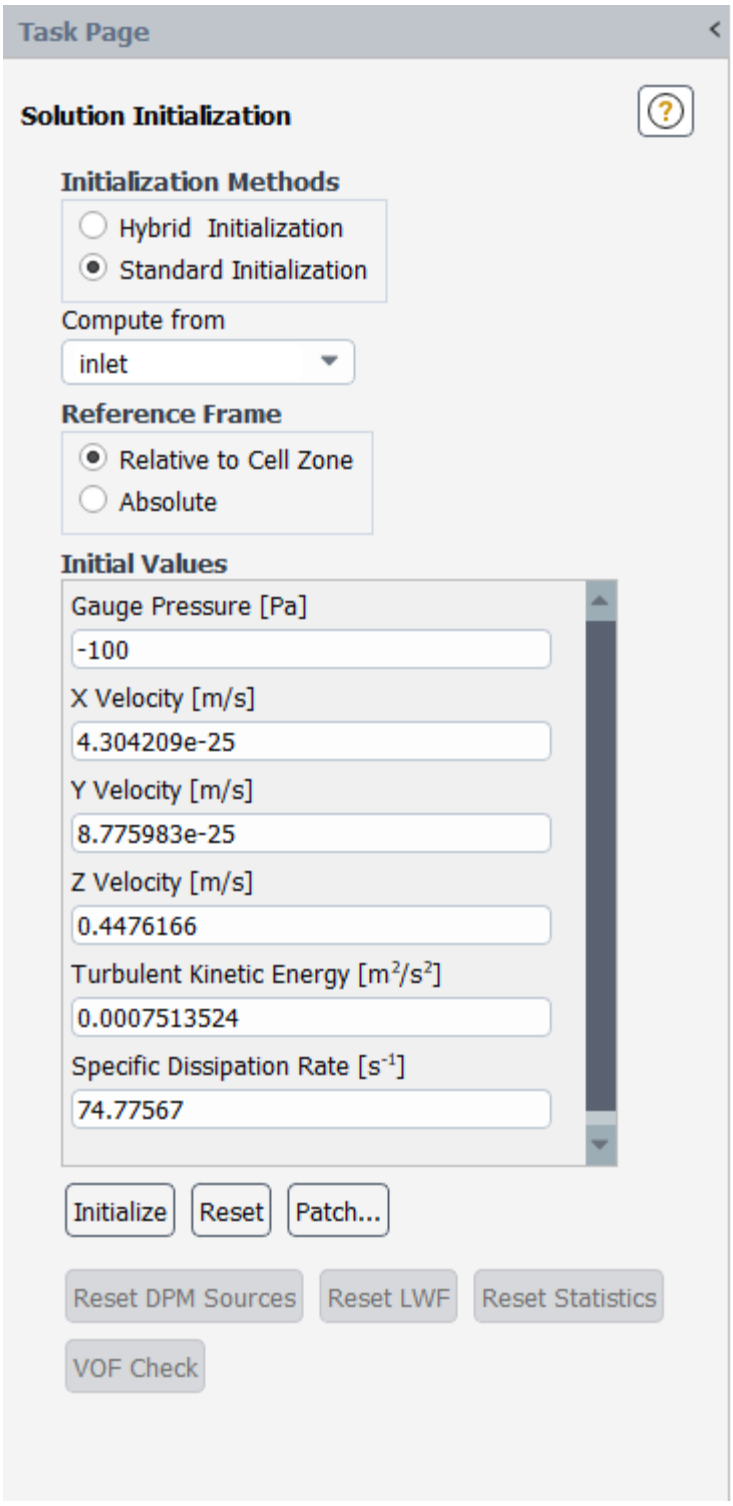
6. Initialize the solution using an absolute frame of reference (Figure 10.8: The Solution Initialization Task Page for Moving Reference Frames (p. 1785)).

   Solution → **Initialization**

   Select the **Absolute** option under **Reference Frame**. If the **Relative to Cell Zone** option is selected, the initial flow field can contain discontinuities, which can cause convergence problems in the first few iterations.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 10.8: The Solution Initialization Task Page for Moving Reference Frames



7. (optional) For transient cases, after you have initialized or run the calculation you can use the **Moving Mesh Courant Number** field variable (in the **Velocity...** category) to guide your solution. This field variable is a non-dimensional value that indicates the number of cells that might be swept in a single time step due to a mesh motion.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

10.3.4. Solution Strategies for MRF and Problems

For multiple moving reference frames, follow the guidelines presented in Solution Strategies for a Single Moving Reference Frame (p. 1776) for a single moving reference frame. Remember that with multiple zones, the possibility exists of interaction between moving and stationary components. This will manifest itself as poor or oscillatory convergence. In such cases, it is strongly recommend that the sliding mesh approach be used to compute the flowfield in order to resolve the unsteady interactions.

10.3.5. Postprocessing for MRF Problems

When you solve a problem using the multiple reference frame model, you can plot or report both absolute and relative velocities. For all velocity parameters (for example, **Velocity Magnitude** and **Mach Number**), corresponding relative values will be available for postprocessing (for example, **Relative Velocity Magnitude** and **Relative Mach Number**). These variables are contained in the **Velocity...** category of the variable selection drop-down list that appears in postprocessing dialog boxes. Relative values are also available for postprocessing of total pressure, total temperature, and any other parameters that include a dynamic contribution dependent on the reference frame (for example, **Relative Total Pressure**, **Relative Total Temperature**).

Important:

Relative velocities are relative to the translational/rotational velocity of the "reference zone" (specified in the **Reference Values** task page (see Reference Values Task Page (p. 5154))). The velocity of the reference zone is the velocity defined in the Fluid Dialog Box (p. 4961) for that zone.

When plotting velocity vectors, you can choose to plot vectors in the absolute frame (the default), or you can select **Relative Velocity** in the **Vectors of** drop-down list in the Vectors Dialog Box (p. 5655) to plot vectors relative to the translational/rotational velocity of the "reference zone" (specified in the Reference Values Task Page (p. 5154)). If you plot relative velocity vectors, you might want to color the vectors by relative velocity magnitude (by choosing **Relative Velocity Magnitude** in the **Color by** list); by default they will be colored by absolute velocity magnitude.

You can also generate a plot of circumferential averages in Ansys Fluent. This allows you to find the average value of a quantity at several different radial or axial positions in your model. Ansys Fluent computes the average of the quantity over a specified circumferential area, and then plots the average against the radial or axial coordinate. For more information on generating XY plots of circumferential averages, see XY Plots of Circumferential Averages (p. 4068).

For details about turbomachinery-specific postprocessing features, see Turbomachinery Postprocessing (p. 2029).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 11: Managing Motion Definitions

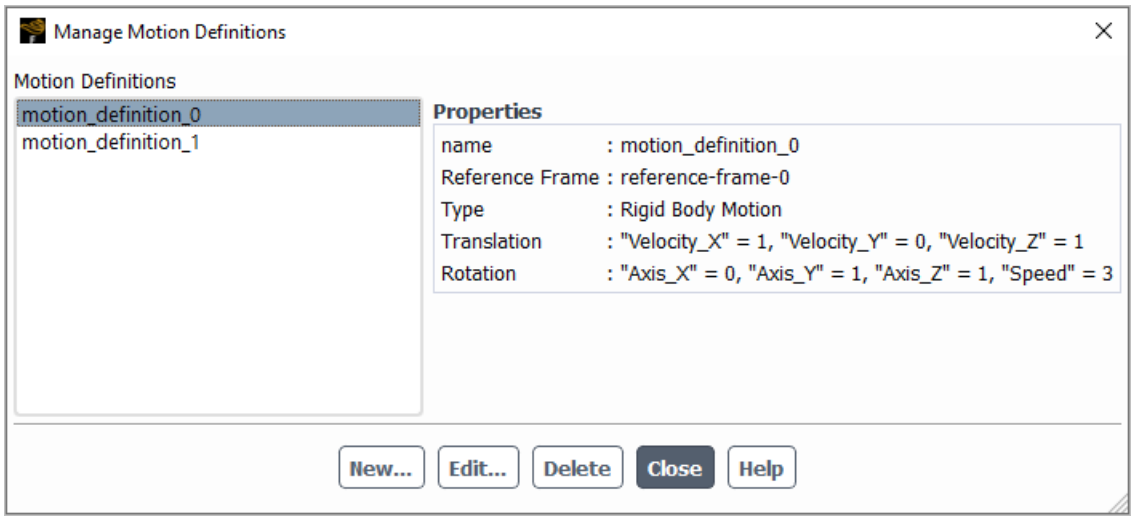
This chapter describes how to create and manage motion definitions, which can be useful for rigid body dynamic mesh zones (as described in Rigid Body Motion (p. 1906)). A motion definition is comprised of a name, reference frame, translation definition, and rotation definition. When defining the rigid body motion of the dynamic zone, using a motion definition may be advantageous, as it can be simpler to set up than the alternatives (that is, a profile or user-defined function) and allows you to use expressions to define the dynamic mesh motion.

To manage motion definitions, perform the following steps:

1. Open the **Manage Motion Definitions** dialog box from the **Outline View**.

   Setup → Motion Definitions → Manage...

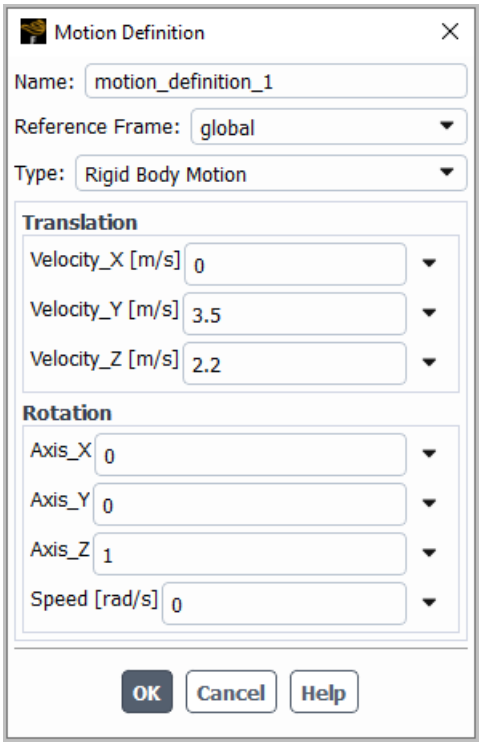
   Figure 11.1: The Manage Motion Definitions Dialog Box



2. Click the **New...** button in the **Manage Motion Definitions** dialog box to open the **Motion Definition** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 11.2: The Motion Definition Dialog Box



a. Enter a suitable **Name** for the definition.

b. Make a selection from the **Reference Frame** drop-down list to define the reference frame for the motion. You can use the default **global** reference frame, or select a local reference frame definition that you have previously set up, as described in Reference Frames (p. 1160).

c. Make a selection from the **Type** drop-down list to specify the kind of motion you are defining. The **Rigid Body Motion** selection is the only available option.

d. Define the translation velocity components in the coordinate directions using the fields in the **Translation** group box. You can define **Velocity_X**, **Velocity_Y**, and (for 3D cases) **Velocity_Z** components. These can be defined as a constant value, an expression, an input parameter, or a named expression.

e. Define the rotation using the fields in the **Rotation** group box. For 3D cases, you can define the rotation axis components in the coordinate directions through the **Axis_X**, **Axis_Y**, and **Axis_Z** fields; for 2D cases, the axis direction is the positive Z direction. Then you can provide a value for the **Speed**, which is defined using the right-hand rule with respect to the defined axis. Each of these fields can be defined as a constant value, an expression, an input parameter, or a named expression.

f. Click **OK** to save your settings and close the **Motion Definition** dialog box.

3. Repeat the previous step to add additional motion definitions.

4. After you have created motion definitions, you can use the **Manage Motion Definitions** dialog box to take further actions. If you select a definition from the **Motion Definitions** list, details are displayed under **Properties**. You can then take the following actions:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. To revise the selected definition, click the **Edit...** button and change the setup using the **Motion Definition** dialog box that opens.

b. To delete the selected definition, click the **Delete** button.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 12: Managing Auxiliary Geometry Definitions

This chapter describes how to create and manage auxiliary geometry definitions, which can be useful for the following:

- boundary adaption with the default polyhedral unstructured mesh adaption (PUMA) method (as described in Refining and Coarsening (p. 3777))

  When performing PUMA adaption on a surface, you can specify that the new nodes that are created are projected onto a geometry definition, so that the resulting mesh better conforms to the geometry shape. For more information about the concept of geometry-based adaption, see Geometry-Based Adaption in the Theory Guide.

- deforming dynamic mesh zones (as described in Deforming Motion (p. 1910))

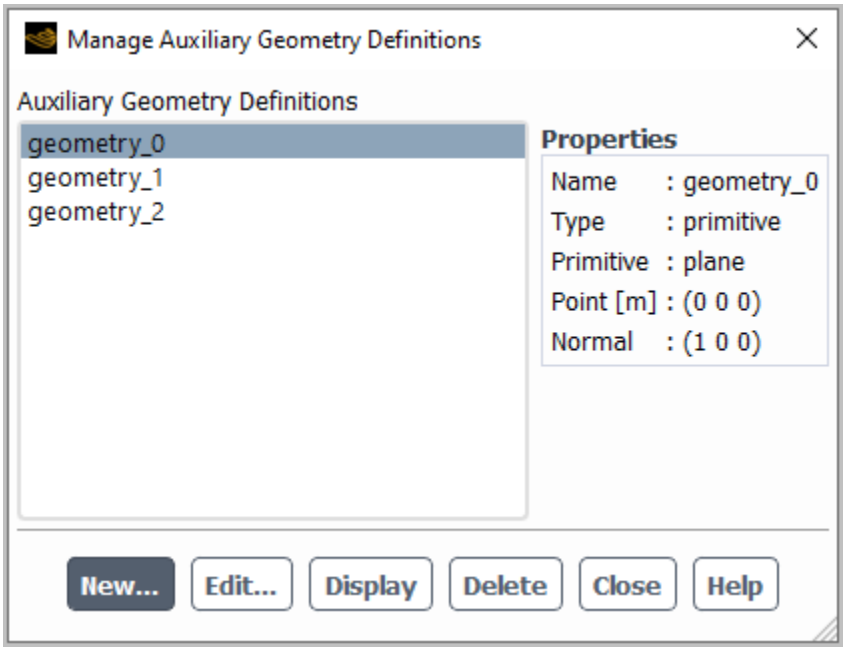
  You can specify that the nodes of the deforming mesh are projected onto an auxiliary geometry definition. This may simplify your setup, as the auxiliary geometry definitions provide shapes that are otherwise not available for deforming zones except through user-defined functions.

To manage auxiliary geometry, perform the following steps:

1. Open the **Manage Auxiliary Geometry Definitions** dialog box from the **Outline View**.

   Setup → Auxiliary Geometry Definitions → Manage...

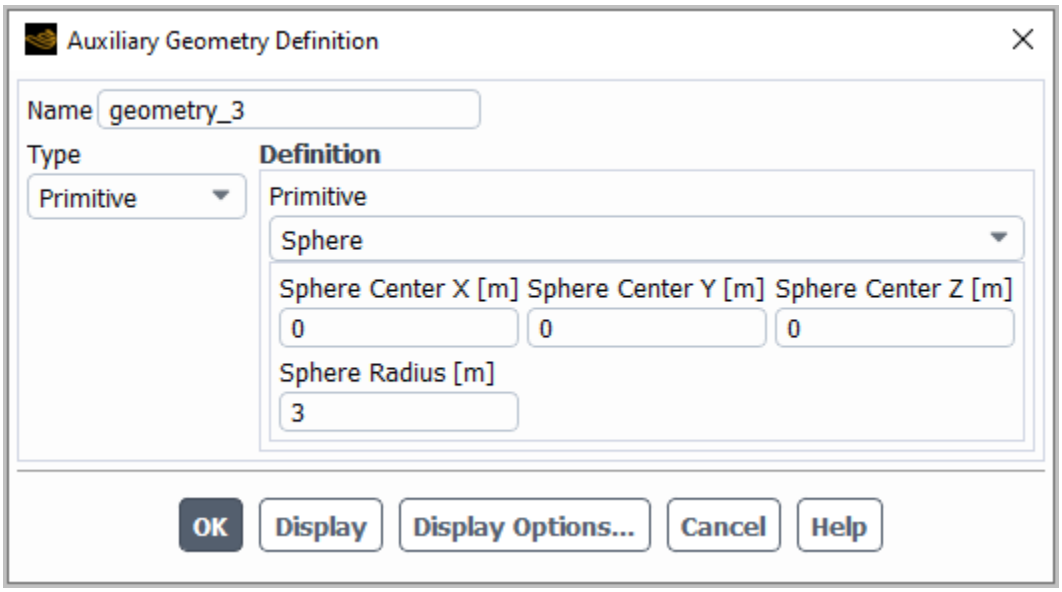
   Figure 12.1: The Manage Auxiliary Geometry Definitions Dialog Box



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Click the **New...** button in the **Manage Auxiliary Geometry Definitions** dialog box to open the **Auxiliary Geometry Definition** dialog box.

Figure 12.2: The Auxiliary Geometry Definition Dialog Box



a. Enter a suitable **Name** for the definition.

b. Make a selection from the **Type** drop-down list to specify how you will define the geometry. You have the following options:

- **Primitive**

  This allows you to make a selection from the **Primitive** drop-down list in the **Definition** group box. You can choose to define the geometry as a **Cone**, **Plane**, **Sphere**, **Frustum** (of a cone), or **Cylinder**. Then you must define the geometry using the other settings in the **Definition** group box.

Note:

– For the **Frustum** definition, the **Lower Radius** is defined for the end at the **Frustum Origin** coordinates, and the **Upper Radius** is defined for the opposite end.

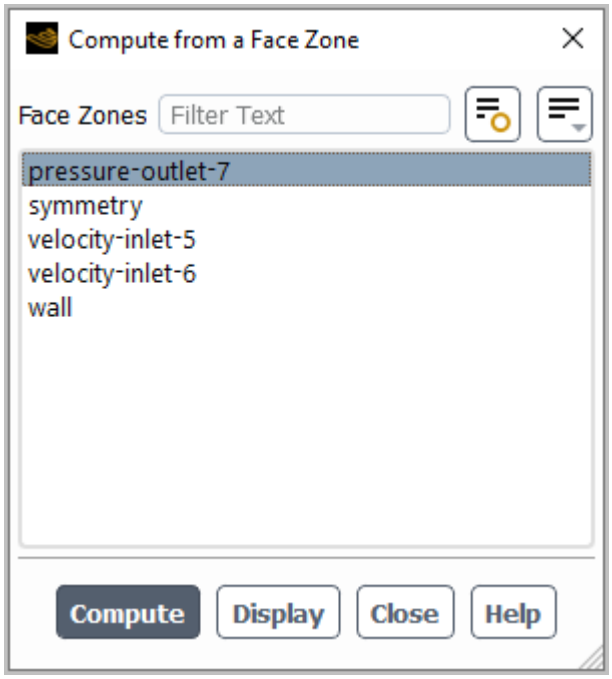
– The geometry definition for the **Frustum** and **Cylinder** only includes the curved surfaces, that is, it does not include the circular planar faces on the ends. To create a geometry definition for the ends, you would need to create additional definitions of **Plane** primitives.

– For the **Plane** definition, an easy way to define it is to click the **Compute from a Face Zone...** button; then in the **Compute from a Face Zone** dialog box that opens, select a face zone that is planar from the list and click the **Compute** button, and the fields in the **Auxiliary Geometry Definition** will be automatically filled. Note that after you select a zone in the **Compute from a Face Zone** dialog box,

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

you have the option of clicking the **Display** button to verify that you have selected the intended zone.

Figure 12.3: The Compute from a Face Zone Dialog Box



- **User-Defined**

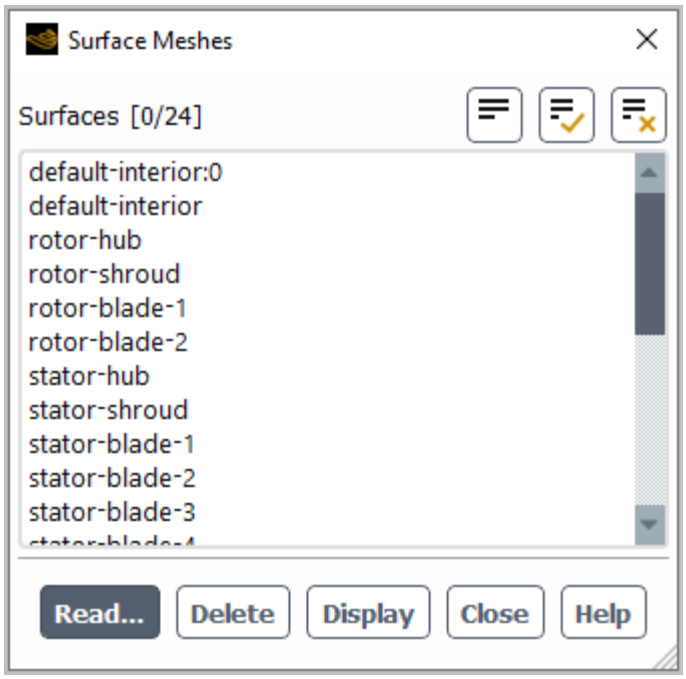
  This allows you to define the geometry by a user-defined function (UDF) that you have previously compiled. Select the UDF from the **Function** drop-down list in the **Definition** group box. Note that the UDF must use the `DEFINE_GEOMETRY` macro, as described in `DEFINE_GEOMETRY` in the *Fluent Customization Manual*.

- **Surface Mesh**

  This allows you to define the geometry using a surface zone from a separate mesh file. Click the **Surfaces Meshes...** button; then in the **Surfaces Meshes** dialog box that opens, click the **Read...** button, select a case, mesh, or STL file in the **Select File** dialog box that opens (being sure to select the units that the **Mesh was created in**), and close the **Surfaces Meshes** dialog box. Finally, specify the surface zone you want to define the geometry by making a selection from the **Background Mesh** drop-down in the **Auxiliary Geometry Definition** dialog box. For more information about reading surface meshes, see Reading Surface Mesh Files (p. 1159).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 12.4: The Surfaces Meshes Dialog Box



- **Reconstruction**

  This allows you to define the geometry using a reconstruction of the one or more boundary zones selected in the **Face zones** list in the **Definition** group box. This reconstruction represents the geometry by building a smooth background model based on the current node coordinates. If you want to maintain sharp feature edges where two boundary zones meet, you should ensure that the boundary zones are in separate geometry definitions (otherwise the edge may be smoothed / rounded in the reconstruction).

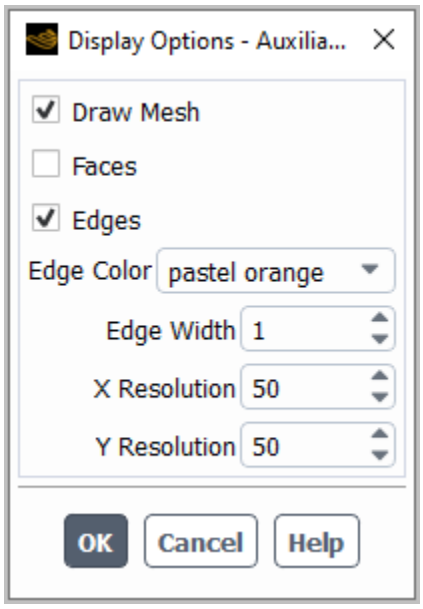
  Note:

  Auxiliary geometry definitions of the **Reconstruction** type cannot be used in a deforming dynamic mesh zone when remeshing is enabled.

c. You can verify that your geometry is appropriately defined by clicking the **Display** button and viewing it in the graphics window. If necessary, you can alter how it is displayed by clicking the **Display Options...** button and revising the setup in the **Display Options - Auxiliary Geometry Definitions** dialog box that opens.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 12.5: The Display Options - Auxiliary Geometry Definitions Dialog Box



The settings for the **Display Options - Auxiliary Geometry Definitions** dialog box are the following:

- **Draw Mesh**

  toggles the ability to draw the mesh at the same time that the geometry definition is displayed. Enabling this option opens the Mesh Display Dialog Box (p. 4688), which allows you to select the desired surface or zone meshes to be displayed.

- **Faces**

  enables the display of shaded faces for the geometry definition. The color of the faces are specified using the **Face Color** drop-down list.

- **Edges**

  enables the display of edges for the geometry definition. The color of the edges are specified using the **Edge Color** drop-down list, and the thickness of the edges is specified through the **Edge Thickness** field.

- **X Resolution** and **Y Resolution**

  These set the number of faces into which the curved surface of a primitive shape is discretized; higher values results in a higher resolution. The **X Resolution** defines the discretization along the length direction or (for a **Sphere**) the z-direction, while the **Y Resolution** defines it around the circumference or (for a **Sphere**) the longitude.

d. Click **OK** to save your settings and close the **Auxiliary Geometry Definition** dialog box.

3. Repeat the previous step to add additional auxiliary geometry definitions.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. After you have created auxiliary geometry definitions, you can use the **Manage Auxiliary Geometry Definitions** dialog box to take further actions. If you select a definition from the **Auxiliary Geometry Definitions** list, details are displayed under **Properties**. You can then take the following actions:

   a. To revise the selected definition, click the **Edit...** button and change the setup using the **Auxiliary Geometry Definition** dialog box that opens.

   b. To display the selected definition in the graphics window, click the **Display** button.

   c. To delete the selected definition, click the **Delete** button.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 13: Modeling Flows Using Sliding and Dynamic Meshes

This chapter describes the setup and use of the sliding and dynamic mesh models in Ansys Fluent. To learn more about the theory of sliding meshes in Ansys Fluent, see Sliding Mesh Theory in the Theory Guide. For more information about the theory behind dynamic meshes in Ansys Fluent, see Dynamic Mesh Theory in the Theory Guide.

Understanding and using the sliding and deforming mesh models is presented in the following sections:

13.1. Introduction

13.2. Sliding Mesh Examples

13.3. The Sliding Mesh Technique

13.4. Sliding Mesh Interface Shapes

13.5. Using Sliding Meshes

13.6. Using Dynamic Meshes

13.1. Introduction

The sliding mesh model allows you to set up a problem in which separate zones move relative to each other. The motion can be translational or rotational. The relative motion of stationary and moving components (for example, in a rotating machine) will give rise to transient interactions. Often, the transient solution that is sought in a sliding mesh simulation is time-periodic. That is, the transient solution repeats with a period related to the speeds of the moving domains.

The dynamic mesh model allows you to move the boundaries of a cell zone relative to other boundaries of the zone, and to adjust the mesh accordingly. The boundaries can move rigidly with respect to each other (that is, linear or rotational motion), and/or deform.

When deciding whether to use a sliding mesh versus a dynamic mesh, consider the following:

- Many problems could be solved by either approach.
- If the problem does not involve mesh deformation, the sliding mesh model is recommended, as it is simpler and more efficient.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The dynamic mesh method must be used if the mesh is deforming, or if the mesh motion is a function of the solution (for example, the six degrees of freedom solver).

Important:

Sliding mesh and dynamic mesh are incompatible if they are combined within the same cell zone or adjacent cell zones. If you want to combine these two models within the same case file, they must be separated by a non-moving cell zone.

Note:

By default, transient simulations that involve a rotating fluid zone utilize a better flow field predictor that can speed up the calculation in some flow simulations. You can deactivate the flow field predictor by using the command `solve/set/transient-controls/rotating-mesh-flow-predictor?` and answering `no`.

For examples of typical sliding mesh and dynamic mesh problems, see Introduction in the Theory Guide.

13.2. Sliding Mesh Examples

When a time-accurate solution (rather than a time-averaged solution) for rotor-stator interaction is desired, you must use the sliding mesh model to compute the unsteady flow field. The sliding mesh model is the most accurate method for simulating flows in multiple moving reference frames, but also the most computationally demanding.

Most often, the unsteady solution that is sought in a sliding mesh simulation is time-periodic. That is, the unsteady solution repeats with a period related to the speeds of the moving domains. However, you can model other types of transients, including translating sliding mesh zones (for example, two cars or trains passing in a tunnel, as shown in Figure 13.1: Two Passing Trains in a Tunnel (p. 1799)). Note that the sliding mesh can be modeled using periodic boundaries (Figure 13.2: Rotor-Stator Interaction (Stationary Guide Vanes with Rotating Blades) (p. 1799)) or a circular/cylindrical interface (Figure 13.3: Blower (p. 1800)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.1: Two Passing Trains in a Tunnel

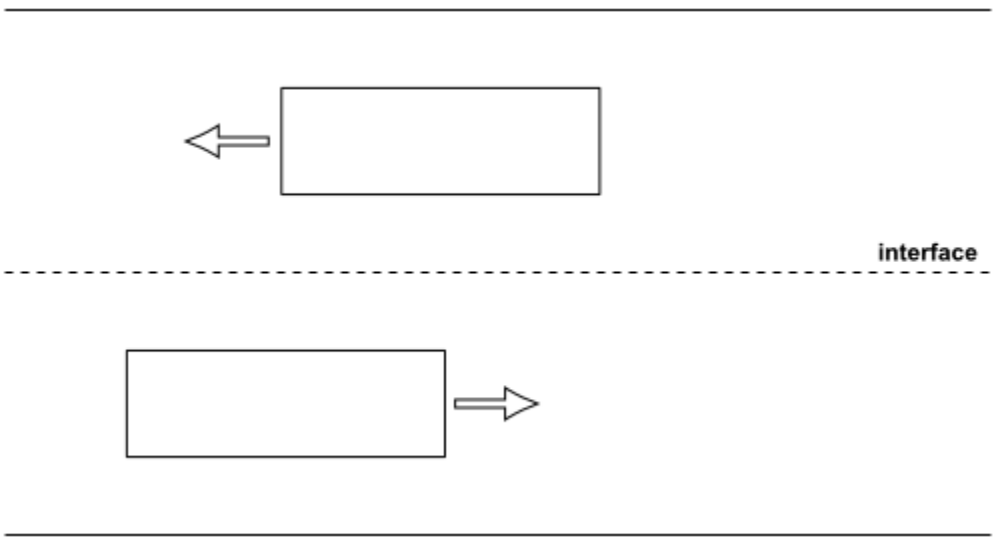


Figure 13.2: Rotor-Stator Interaction (Stationary Guide Vanes with Rotating Blades)

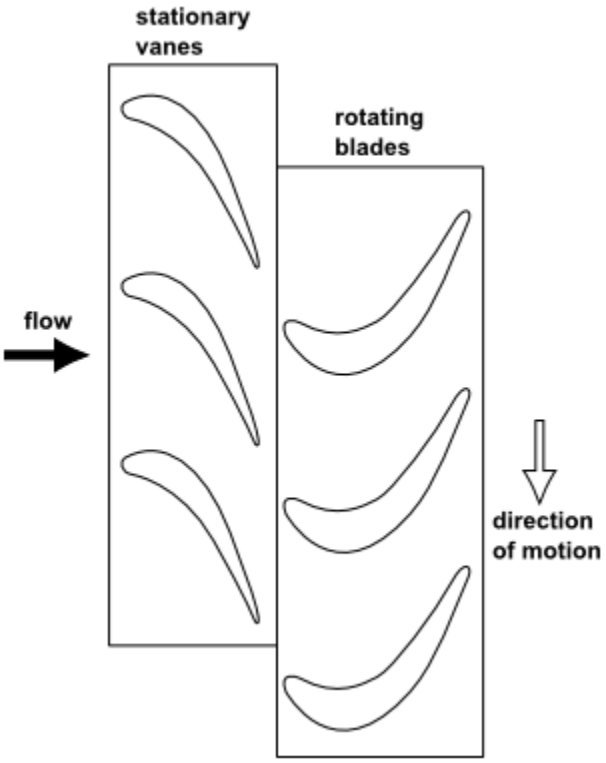
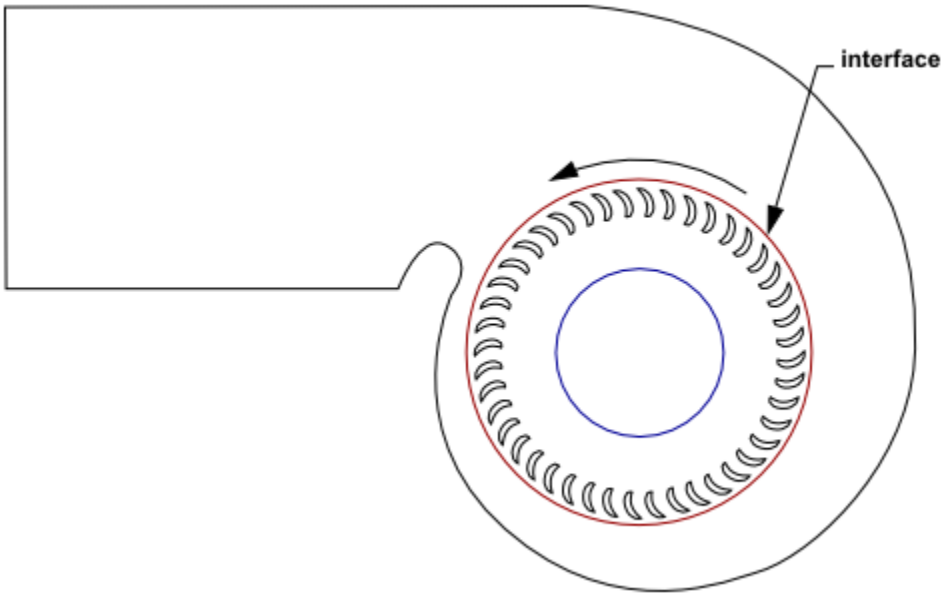


Figure 13.3: Blower



Note that for flow situations where there is no interaction between stationary and moving parts (for example, when there is only a rotor), it is not necessary to use a sliding mesh, and the moving reference frame model is recommended. (See Flow in a Moving Reference Frame for details.) If you are interested in a steady approximation of the interaction, you may use the multiple reference frame model or the mixing plane model, as described in The Multiple Reference Frame Model and The Mixing Plane Model in the Theory Guide.

13.3. The Sliding Mesh Technique

In the sliding mesh technique, two or more cell zones are used. (If you generate the mesh in each zone independently, you must merge the mesh files prior to starting the calculation, as described in Reading Multiple Mesh/Case/Data Files (p. 1156) in the User's Guide.) Each cell zone is bounded by at least one "interface zone" where it meets the opposing cell zone. The interface zones of adjacent cell zones are associated with one another to form a "mesh interface." The two cell zones will move relative to each other along the mesh interface.

During the calculation, the cell zones slide (that is, rotate or translate) relative to one another along the mesh interface in discrete steps. Figure 13.4: Initial Position of the Meshes (p. 1801) and Figure 13.5: Rotor Mesh Slides with Respect to the Stator (p. 1801) show the initial position of two meshes and their positions after some translation has occurred. Note that all non-conformal interfaces will be automatically updated (if necessary) by Ansys Fluent when the mesh is updated.

As the rotation or translation takes place, node alignment along the mesh interface is not required. Since the flow is inherently unsteady, a time-dependent solution procedure is required.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.4: Initial Position of the Meshes

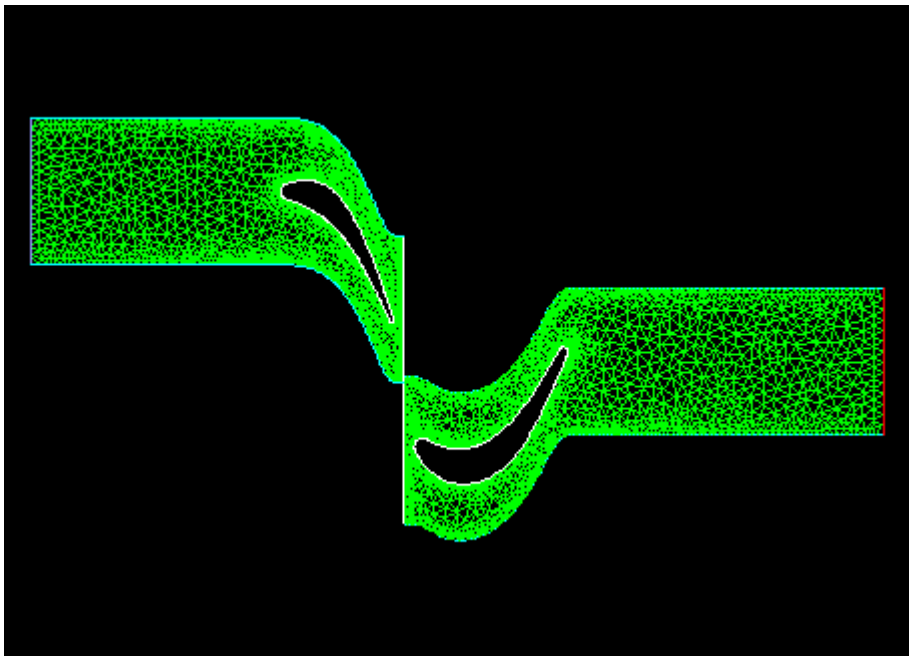
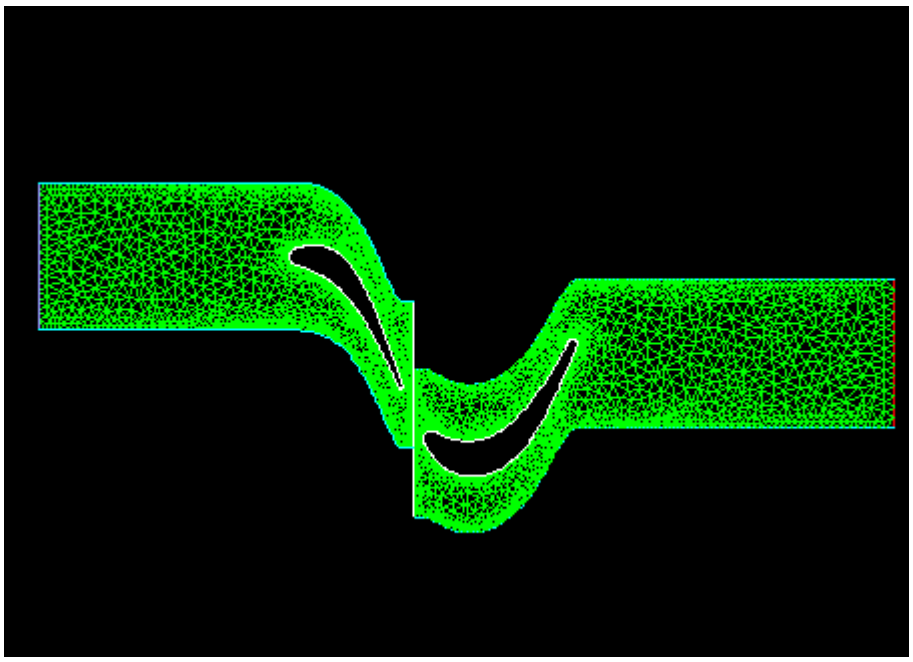


Figure 13.5: Rotor Mesh Slides with Respect to the Stator



13.4. Sliding Mesh Interface Shapes

The mesh interface and the associated interface zones can be any shape, provided that the two interface boundaries are based on the same geometry. Figure 13.6: 2D Linear Mesh Interface (p. 1802) shows an example with a linear mesh interface and Figure 13.7: 2D Circular-Arc Mesh Interface (p. 1802) shows a circular-arc mesh interface. (In both figures, the mesh interface is designated by a dashed line.)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.6: 2D Linear Mesh Interface

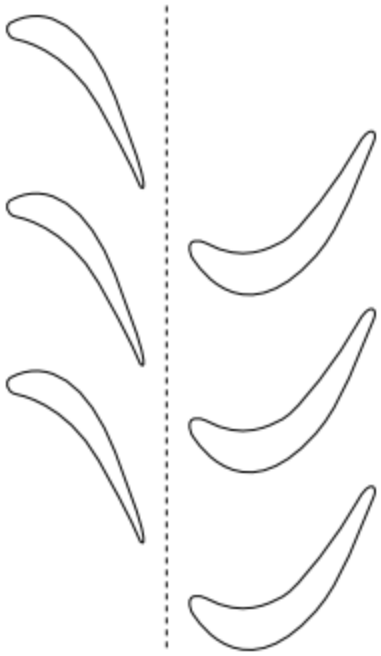
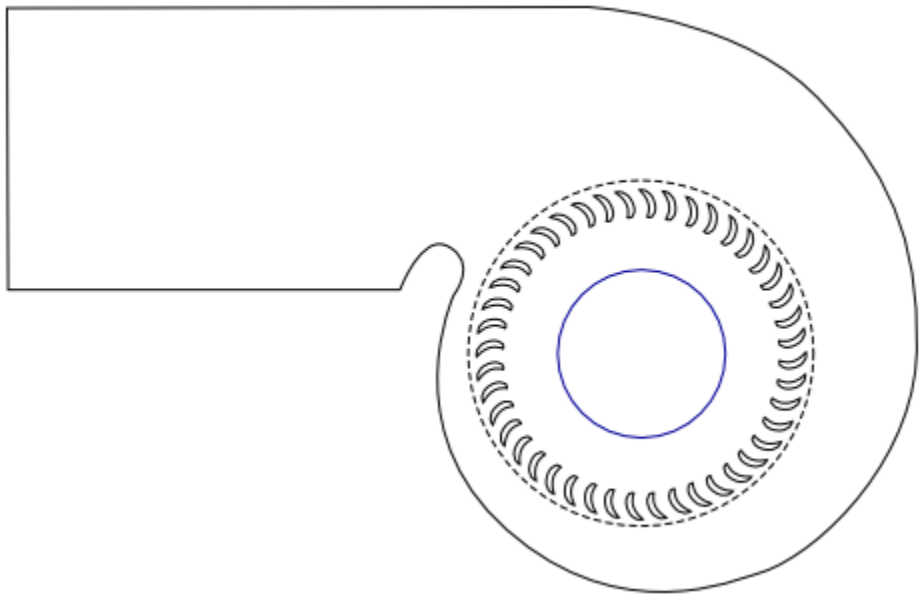


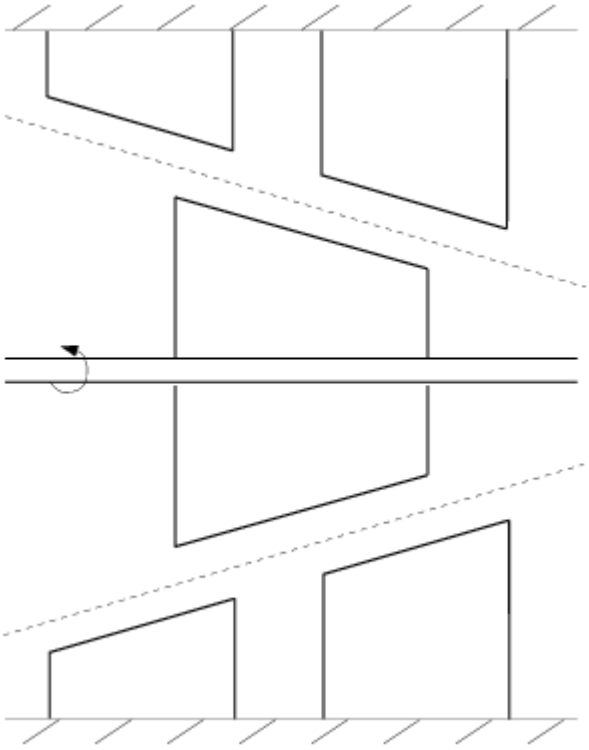
Figure 13.7: 2D Circular-Arc Mesh Interface



If Figure 13.6: 2D Linear Mesh Interface (p. 1802) was extruded to 3D, the resulting sliding interface would be a planar rectangle; if Figure 13.7: 2D Circular-Arc Mesh Interface (p. 1802) was extruded to 3D, the resulting interface would be a cylinder. Figure 13.8: 3D Conical Mesh Interface (p. 1803) shows an example that would use a conical mesh interface. (The slanted, dashed lines represent the intersection of the conical interface with a 2D plane.)

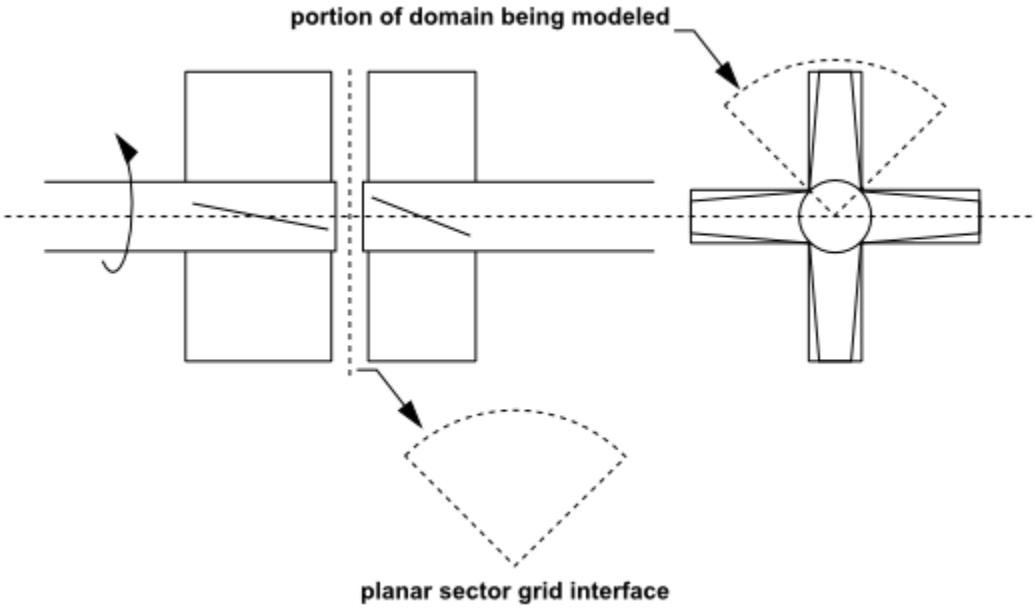
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.8: 3D Conical Mesh Interface



For an axial rotor/stator configuration, in which the rotating and stationary parts are aligned axially instead of being concentric (see Figure 13.9: 3D Planar-Sector Mesh Interface (p. 1803)), the interface will be a planar sector. This planar sector is a cross-section of the domain perpendicular to the axis of rotation at a position along the axis between the rotor and the stator.

Figure 13.9: 3D Planar-Sector Mesh Interface



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.5. Using Sliding Meshes

This section describes how to use sliding meshes, including restrictions and constraints, problem setup, solution strategies, and postprocessing.

13.5.1. Requirements, Constraints, and Considerations
13.5.2. Setting Up the Sliding Mesh Problem
13.5.3. Solution Strategies for Sliding Meshes
13.5.4. Postprocessing for Sliding Meshes

13.5.1. Requirements, Constraints, and Considerations

Before beginning the problem setup in Ansys Fluent, be sure that the mesh you have created meets the following requirements:

- The mesh must have a different cell zone for each portion of the domain that is sliding at a different speed.
- The mesh interface must be situated such that there is no motion normal to it.
- The mesh interface can be any shape (including a non-planar surface, in 3D), provided that the two interface boundaries are based on the same geometry. If there are sharp features in the mesh (for example, 90-degree angles), it is especially important that both sides of the interface closely follow that feature.
- If you create a single mesh with multiple cell zones, you must be sure that each cell zone has a distinct face zone on the sliding boundary. The face zones for two adjacent cell zones will have the same position and shape, but one will correspond to one cell zone and one to the other. (Note that it is also possible to create a separate mesh file for each of the cell zones, and then merge them as described in Reading Multiple Mesh/Case/Data Files (p. 1156).)
- If you are modeling a rotor/stator geometry using periodicity, the periodic angle of the mesh around the rotor blade(s) must be the same as that of the mesh around the stationary vane(s).
- All periodic zones must be correctly oriented (either rotational or translational) before you create the mesh interface.
- Note the following limitations if you want to use the periodic repeats option as part of the mesh interface:
  - The periodic repeats option is not supported if the interface zone is adjacent to a solid zone.
  - The edges of the second interface zone must be offset from the corresponding edges of the first interface zone by a uniform amount (either a uniform translational displacement or a uniform rotation angle).
  - Some portion of the two interface zones must overlap (that is, be spatially coincident).
  - The non-overlapping portions of the interface zones must have identical shape and dimensions at all times during the mesh motion.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  – One pair of conformal periodic zones must be adjacent to each of the interface zones. For example, when you calculate just one channel and blade of a fan, turbine, and so on, you must have conformal periodics on either side of the interface threads. This will not work with non-conformal periodics.

    Note that for 3D cases, you cannot have more than one pair of conformal periodic zones adjacent to each of the interface zones.

- You must not have a single sliding mesh interface where part of the interface is made up of a coupled two-sided wall, while another part is not coupled (that is, the normal interface treatment). In such cases, you must break the interface up into two interfaces: one that is a coupled interface, and the other that is a standard fluid-fluid interface. See Using a Non-Conformal Mesh in Ansys Fluent (p. 1185) for information about creating coupled interfaces.

- When using the sliding mesh technique, mass fluxes are interpolated from the sliding interface faces to the sliding boundary faces, which can affect the local mass balance at the sliding boundary. This can result in pressure field oscillations at the sliding boundary. These oscillations may be reduced by ensuring the surface mesh node distributions are similar on either side of the sliding boundaries. You can use the matching option (Matching Option (p. 1177)), where applicable, to reduce solution oscillations at the sliding boundaries.

- When using a sliding mesh to simulate flow through narrow gaps that open or close over time (such as in a valve or a rotary pump), you can use the gap model to block or decelerate the flow when the gaps are closed (as described in Controlling Flow in Narrow Gaps for Valves and Pumps (p. 1234)).

For details about these restrictions and general information about how the sliding mesh model works in Ansys Fluent, see The Sliding Mesh Technique (p. 1800).

13.5.2. Setting Up the Sliding Mesh Problem

The steps for setting up a sliding mesh problem are listed below. (Note that this procedure includes only those steps necessary for the sliding mesh model itself; you must set up other models, boundary conditions, and so on, as usual.)

1. Enable the appropriate option for modeling transient flow. (See Performing Time-Dependent Calculations (p. 3643) for details about the transient modeling capabilities in Ansys Fluent.)

   Setup → **General** **Analysis Type** → **Transient**

2. Set the cell zone conditions for the sliding motion:

   Setup → **Cell Zone Conditions**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the Fluid Dialog Box (p. 4961) or Solid Dialog Box (p. 4973) of each moving fluid or solid zone, enable the **Mesh Motion** option and set the translational and/or rotational velocity under the **Mesh Motion** tab.

Important:

Note that simultaneous translation and rotation can be modeled only if the rotation axis and the translation direction are the same (that is, the origin is fixed).

Note that the speed can be specified as a constant value or a transient profile. The transient profile may be in a file format, as described in Transient Cell Zone and Boundary Conditions (p. 1523), or a UDF macro, described in `DEFINE_TRANSIENT_PROFILE` in the Fluent Customization Manual. Specifying the individual velocities as either a profile or a UDF allows you to specify a single component of the sliding motion individually. However, you can also specify the sliding motion using a user-defined function. This may prove to be quite convenient if you are modeling a more complicated motion, where the hooking of many different user-defined functions or profiles can be cumbersome.

Note:

If you decide to hook a UDF, then you will no longer have access to the rotation axis origin and direction, or the velocities.

3. Set the boundary conditions for the sliding motion:

   Setup → **Boundary Conditions**

   Change the zone **Type** of the interface zones of adjacent cell zones to **interface** in the Boundary Conditions Task Page (p. 4991).

   By default, the velocity of a wall is set to zero relative to the motion of the adjacent mesh. For walls bounding a moving mesh this results in a “no-slip” condition in the reference frame of the mesh. Therefore, you need not modify the wall velocity boundary conditions unless the wall is stationary in the absolute frame, and therefore moving in the relative frame. See Velocity Conditions for Moving Walls (p. 1462) for details about wall motion.

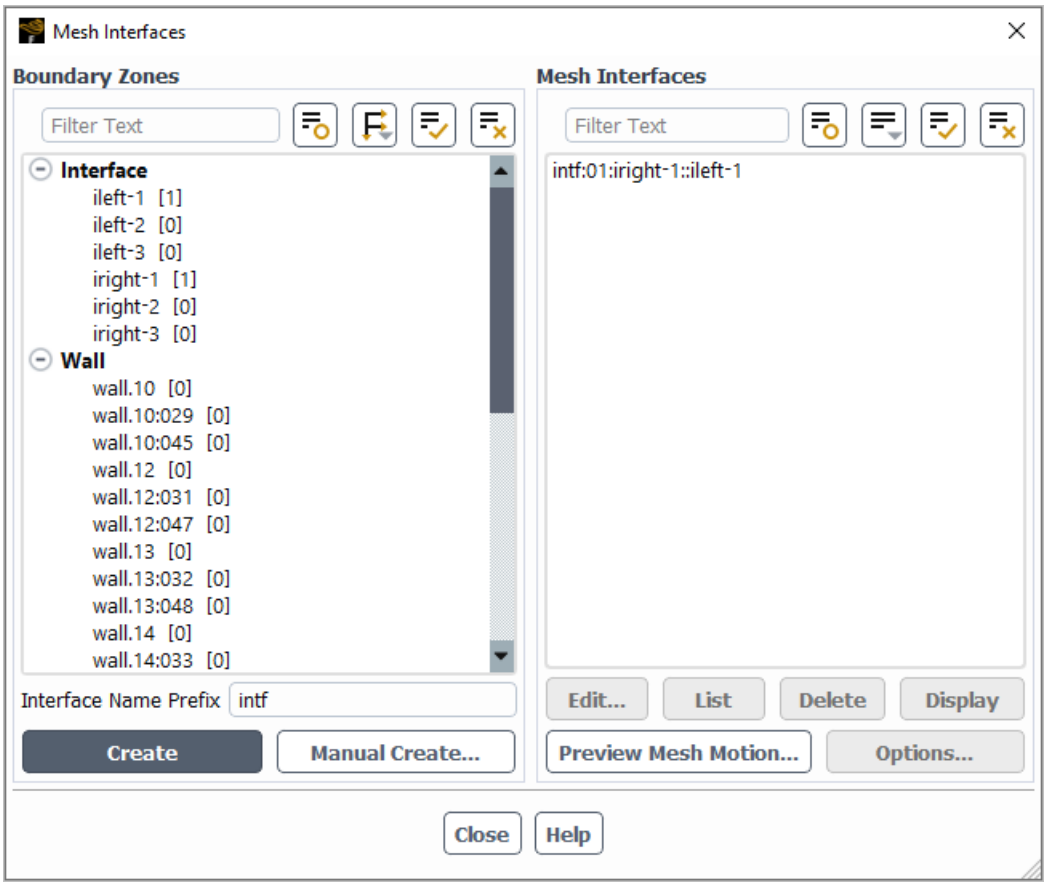
   See Cell Zone and Boundary Conditions (p. 1299) for details about inputting cell zone and boundary conditions.

4. Define the mesh interfaces using the Mesh Interfaces Dialog Box (p. 5525) (Figure 13.10: The Mesh Interfaces Dialog Box (p. 1807)).

   Setup → **Mesh Interfaces** → **New...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.10: The Mesh Interfaces Dialog Box



To learn how to use the **Mesh Interfaces** dialog box, see Using a Non-Conformal Mesh in Ansys Fluent (p. 1185). Note that the **Interface Options** that are relevant for sliding meshes are **Periodic Repeats**, **Coupled Wall**, and **Matching** (for more information on these options, see Non-Conformal Mesh Calculations (p. 1170)).

5. Preview the mesh motion using the **Zone Motion** dialog box, which can be opened from the **Run Calculation** task page.

   Solution → **Run Calculation** **Preview Mesh Motion...**

   For details about using the **Zone Motion** dialog box, see Previewing the Dynamic Mesh (p. 1922).

6. (optional) After you have initialized or run the calculation, you can use the **Moving Mesh Courant Number** field variable (in the **Velocity...** category) to guide your solution. This field variable is a non-dimensional value that indicates the number of cells that might be swept in a single time step due to a mesh motion.

Important:

When you have completed the problem setup, you should save an initial case file so that you can easily return to the original mesh position (that is, the positions before any sliding

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

occurs). The mesh position is stored in the case file, so case files that you save at different times during the transient calculation will contain meshes at different positions.

Important:

If you want to go from an MRF model setup to a sliding mesh setup, use the following text command:

`mesh` → `modify-zones` → `mrf-to-sliding-mesh`

To successfully switch from an MRF to a sliding mesh, you must provide the ID of the fluid zone. Ansys Fluent identifies all the interior boundary zones of this fluid zone, as well as any adjacent fluid zones. Ansys Fluent changes these interior boundary zones into coupled walls, and then decouples these walls and changes them to interface zones. Ansys Fluent then changes the cell zone condition of the fluid zone to **Moving Mesh** in the Fluid Dialog Box (p. 4961). You do not need to do this if you have already created a mesh interface. The sliding mesh solution tends to be more robust than the MRF solution.

Note that *all* of the interfaces between the cells zones must be interior boundaries. This means that this text command cannot be used for a thermal simulation with solid zones adjacent to the MRF zone (for example, an MRF zone in contact with a solid volute).

13.5.3. Solution Strategies for Sliding Meshes

You will begin the sliding mesh calculation by initializing the solution (as described in Initializing the Entire Flow Field Using Standard Initialization (p. 3628)) and then specifying the time step size and number of time steps in the Run Calculation Task Page (p. 5207), as for any other transient calculation. (See Performing Time-Dependent Calculations (p. 3643) for details about time-dependent solutions.) Note that the time step size in the initial case file is saved without clicking **Calculate**. Ansys Fluent will iterate on the current time step solution until satisfactory residual reduction is achieved, or the maximum number of iterations per time step is reached. When it advances to the next time step, the cell and wall zones will automatically be moved according to the specified translational or rotational velocities (as discussed in the previous section). The new interface-zone intersections will be computed automatically, and resultant interior/periodic/external boundary zones will be updated.

Note that you can run the MRF case using mesh interfaces with an appreciable loss of accuracy, doing so makes it easier to later on convert to a sliding mesh.

Note:

The sliding mesh can be initialized with an MRF solution (rather using Standard or Hybrid initializations).

It is recommended that you preview the sliding mesh motion (as described in Previewing the Dynamic Mesh (p. 1922)) before beginning your calculation. This can catch problems with the motion specification(s) before you begin the CFD calculation. Remember to save the case and initial data files before doing a mesh preview since the mesh position is altered once you do the preview. You can reread the initial condition case/data files to get back to the original mesh position.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.5.3.1. Saving Case and Data Files

Ansys Fluent's automatic saving of case and data files (see Automatic Saving of Case and Data Files (p. 958)) can be used with the sliding mesh model. This provides a convenient way for you to save results at successive time steps for later postprocessing.

Important:

You must save a case file each time you save a data file because the mesh position is stored in the case file. Since the mesh position changes with each time step, reading data for a given time step will require the case file at that time step so that the mesh will be in the proper position. You should also save your initial case file so that you can easily return to the mesh's original position to restart the solution if desired.

13.5.3.2. Time-Periodic Solutions

For some problems (for example, rotor-stator interactions), you may be interested in a time-periodic solution. That is, the startup transient behavior may not be of interest to you. Once this startup phase has passed, the flow will start to exhibit time-periodic behavior. If T is the period of unsteadiness, then for some flow property ϕ at a given point in the flow field:

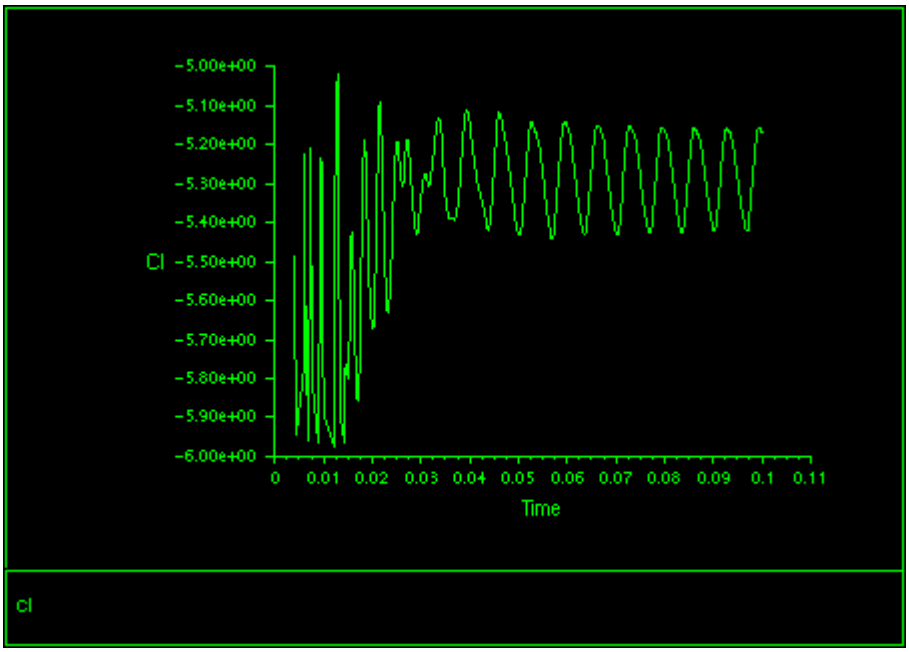
$$\phi(t)=\phi(t+NT)(N=1,2,3,\ldots) \tag{13.1}$$

For rotating problems, the period (in seconds) can be calculated by dividing the sector angle of the domain (in radians) by the rotor speed (in radians/sec): $T=\theta/\Omega$. For 2D rotor-stator problems, $T=P/v_b$, where P is the pitch and v_b is the blade speed. The number of time steps in a period can be determined by dividing the time period by the time step size. When the solution field does not change from one period to the next (for example, if the change is less than 5%), a time-periodic solution has been reached.

To determine how the solution changes from one period to the next, you must compare the solution at some point in the flow field over two periods. For example, if the time period is 10 seconds, you can compare the solution at a given point after 22 seconds with the solution after 32 seconds to see if a time-periodic solution has been reached. If not, you can continue the calculation for another period and compare the solutions after 32 and 42 seconds, and so on until you see little or no change from one period to the next. You can also track global quantities, such as lift and drag coefficients and mass flow, in the same manner. Figure 13.11: Lift Coefficient Plot for a Time-Periodic Solution (p. 1810) shows a lift coefficient plot for a time-periodic solution.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.11: Lift Coefficient Plot for a Time-Periodic Solution



The final time-periodic solution is independent of the size of the time steps taken during the initial stages of the solution procedure. You can therefore define a "large" time step size in the initial stages of the calculation, since you are not interested in a time-accurate solution for the startup phase of the flow. Starting out with a large time step size will allow the solution to become time-periodic more quickly. As the solution becomes time-periodic, however, you should reduce size of the time steps in order to achieve a time-accurate result.

Important:

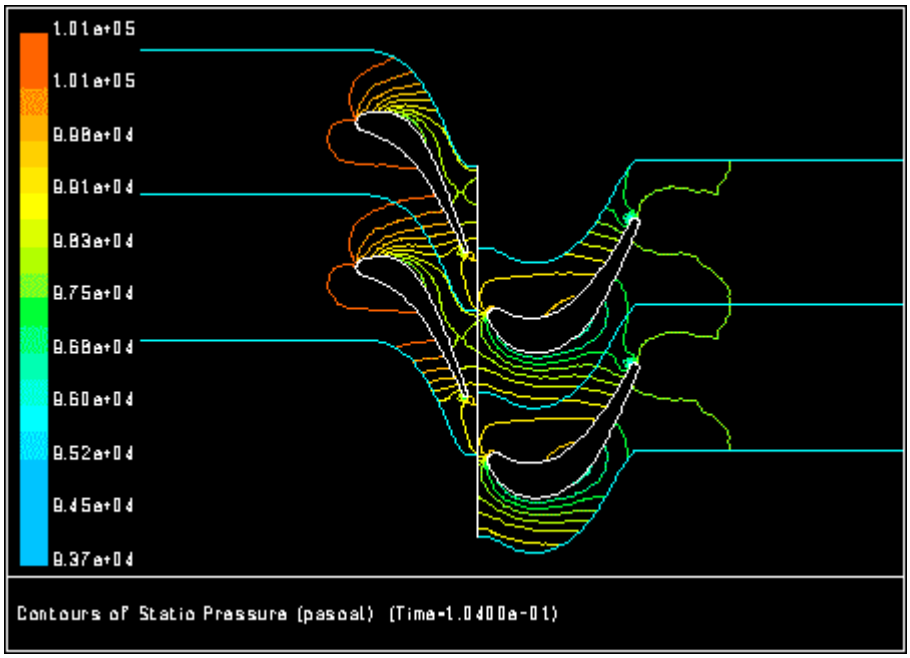
If you are solving with second-order time accuracy, the temporal accuracy of the solution will be affected if you change the time step size during the calculation. You may start out with a larger time step size, but you should not change the time step size by more than 20% during the solution process. You should not change the time step size at all during the last several periods to ensure that the solution has approached a time-periodic state.

13.5.4. Postprocessing for Sliding Meshes

Postprocessing for sliding mesh problems is the same as for other transient problems. You will read in the case and data file for the time of interest and display and report results as usual. For spatially-periodic problems, you may want to use periodic repeats (set in the Views Dialog Box (p. 5272), as described in Controlling the Display State and Modifying the View (p. 4025)) to display the geometry. Figure 13.12: Contours of Static Pressure for the Rotor-Stator Example (p. 1811) shows the flow field for the rotor-stator example in Figure 13.4: Initial Position of the Meshes (p. 1801)) at one instant in time, using 1 periodic repeat.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.12: Contours of Static Pressure for the Rotor-Stator Example



When displaying velocity vectors, note that absolute velocities (that is, velocities in the inertial, or laboratory, reference frame) are displayed by default. You may also choose to display relative velocities by selecting **Relative Velocity** in the **Vectors of** drop-down list in the Vectors Dialog Box (p. 5655). In this case, velocities relative to the translational/rotational velocity of the "reference zone" (specified in the Reference Values Task Page (p. 5154)) will be displayed. (The velocity of the reference zone is the velocity defined in the Fluid Dialog Box (p. 4961) for that zone.)

Note that you cannot create zone surfaces for the intersection boundaries (that is, the interior/periodic/external zones created from the intersection of the interface zones). You may instead create zone surfaces for the interface zones. Data displayed on these surfaces will be "one-sided". That is, nodes on the interface zones will "see" only the cells on one side of the mesh interface, and slight discontinuities may appear when you plot contour lines across the interface. Note also that, for non-planar interface shapes in 3D, you may see small gaps in your plots of filled contours. These discontinuities and gaps are only graphical in nature. The solution does not have these discontinuities or gaps. To eliminate these discontinuities for postprocessing purposes only, you can use the `define/mesh-interfaces/enforce-continuity-after-bc?` text command, which will ensure that continuity will take precedence over the boundary condition.

You can also generate a plot of circumferential averages in Ansys Fluent. This allows you to find the average value of a quantity at several different radial or axial positions in your model. Ansys Fluent computes the average of the quantity over a specified circumferential area, and then plots the average against the radial or axial coordinate. For more information on generating XY plots of circumferential averages, see XY Plots of Circumferential Averages (p. 4068).

Sliding mesh results can be analyzed by employing the time averaging (or RMSE averaging) option (**Data Sampling for Time Statistics**) in the **Run Calculation** task page. This will compute time averages for velocity, pressure, temperature, and turbulence. You must plan ahead since you have to engage the time averaging after the solution has become time-periodic and run for at least one period of the oscillating flow field.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.6. Using Dynamic Meshes

The steps for setting up a dynamic mesh problem are listed below. (Note that this procedure includes only those steps necessary for the dynamic mesh model itself; you must set up other models, cell zone conditions, boundary conditions, and so on, as usual.)

1. Enable the appropriate option for modeling transient or steady flow in the General Task Page (p. 4684).

   Setup → **General**

   If your problem involves a steady flow, see Steady-State Dynamic Mesh Applications (p. 1924) for important considerations.

2. Set cell zone conditions and boundary conditions as required in the Cell Zone Conditions Task Page (p. 4958) and Boundary Conditions Task Page (p. 4991).

   Setup → **Cell Zone Conditions**

   Setup → **Boundary Conditions**

   See Cell Zone and Boundary Conditions (p. 1299) for information about inputting conditions. The correct wall velocity is set up automatically when a dynamic zone is created for a wall zone and the motion attributes are specified, so you will not specify wall motion in the **Wall** dialog box. If you create a moving dynamic cell zone, then all wall boundaries adjacent to that cell zone will, by default, impose the correct (moving) boundary conditions, and it is not necessary to declare these wall zones as dynamic zones. Note that if a wall boundary mesh is moving because it belongs to an adjacent moving dynamic cell zone, but the physical boundary conditions are such that the wall is actually not moving, then you will have to declare this boundary as a dynamic zone and specify that the mesh motion is not included in the boundary conditions (see Specifying the Motion of Dynamic Zones (p. 1904)).

3. Enable the dynamic mesh model, and specify related parameters in the Dynamic Mesh Task Page (p. 5111).

   Setup → **Dynamic Mesh** → **Dynamic Mesh**

   See Setting Dynamic Mesh Modeling Parameters (p. 1814) for details.

4. Create the dynamic zones for your model, using the Dynamic Mesh Zones Dialog Box (p. 5140).

   Setup → **Dynamic Mesh** → **Create/Edit...**

   See Specifying the Motion of Dynamic Zones (p. 1904) for details.

5. You can display the motion of the moving zones with prescribed motion to verify the simulation setup.

   Setup → **Dynamic Mesh** → **Display Zone Motion...**

   See Previewing the Dynamic Mesh (p. 1922) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. If it is a transient simulation, define the events that will occur during the calculation.

   Setup → Dynamic Mesh → Events...

   See Defining Dynamic Mesh Events (p. 1894) for details.

7. Save the case and data.

   File → Write → Case & Data...

8. Preview your dynamic mesh setup (when the motion is a prescribed motion). See Steady-State Dynamic Mesh Applications (p. 1924) for previewing your steady-state dynamic mesh motion and refer to Previewing the Dynamic Mesh (p. 1922) for details.

   Setup → Dynamic Mesh → Preview Mesh Motion...

9. Specify the pressure-velocity coupling scheme. For transient flow calculations, the PISO algorithm is recommended, as it is the most efficient for such cases (see PISO (p. 3586) for details).

10. Use the automatic saving feature to specify the file name and frequency with which case and data files should be saved during the solution process.

    Solution → Calculation Activities → Autosave (Every) → Edit...

    See Automatic Saving of Case and Data Files (p. 958) for details about the use of this feature. This provides a convenient way for you to save results at successive time steps for later postprocessing.

    Important:

    You must save a case file each time you save a data file because the mesh position is stored in the case file. Since the mesh position changes with each time step, reading data for a given time step will require the case file at that time step so that the mesh will be in the proper position. You should also save your initial case file so that you can easily return to the mesh's original position to restart the solution if desired.

11. (optional) If you want to create a graphical animation of the mesh over time during the solution procedure, you can use the Calculation Activities Task Page (p. 5188) to set up the graphical displays that you want to use in the animation. See Animating the Solution (p. 3714) for details.

12. (optional) For transient cases, after you have initialized or run the calculation you can use the **Moving Mesh Courant Number** field variable (in the **Velocity...** category) to guide your solution. This field variable is a non-dimensional value that indicates the number of cells that might be swept in a single time step due to a mesh motion.

For additional information, see the following sections:

13.6.1. Setting Dynamic Mesh Modeling Parameters

13.6.2. Dynamic Mesh Update Methods

13.6.3. Feature Detection

13.6.4. In-Cylinder Settings

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.6.5. Six DOF Solver Settings
13.6.6. Implicit Update Settings
13.6.7. Contact Detection Settings
13.6.8. Defining Dynamic Mesh Events
13.6.9. Specifying the Motion of Dynamic Zones
13.6.10. Previewing the Dynamic Mesh
13.6.11. Steady-State Dynamic Mesh Applications

13.6.1. Setting Dynamic Mesh Modeling Parameters

To enable the dynamic mesh model, enable **Dynamic Mesh** in the Dynamic Mesh Task Page (p. 5111) (Figure 13.13: The Dynamic Mesh Task Page (p. 1815)).

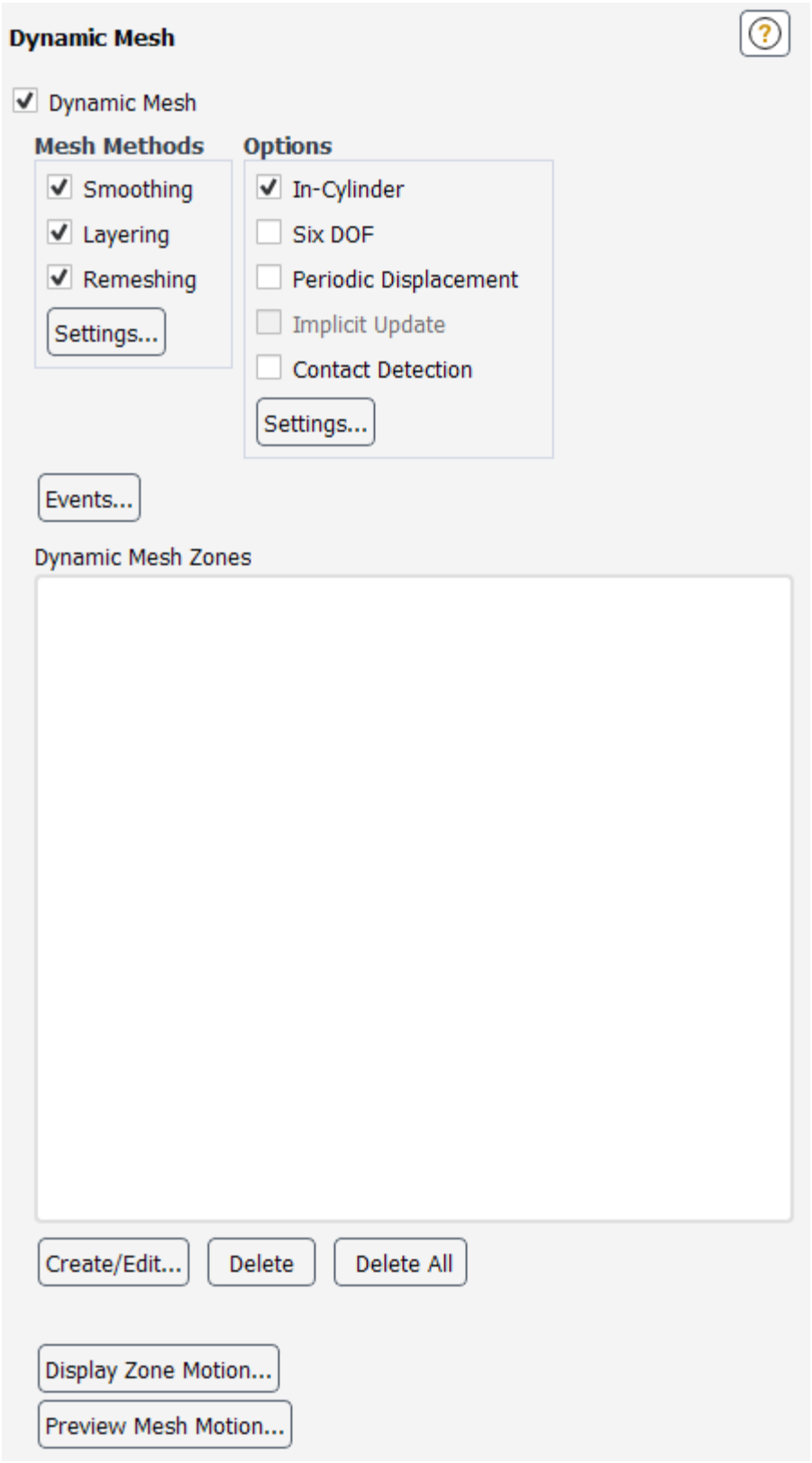
Setup → **Dynamic Mesh** → **Dynamic Mesh**

Then, enable the appropriate options in the **Options** group box. If you are modeling in-cylinder motion, enable the **In-Cylinder** option. If you are going to use the six degrees of freedom solver, then enable the **Six DOF** option. If you want to specify periodic displacement based on a mode shape for blade flutter analysis, select **Periodic Displacement**. If you want to have the dynamic mesh updated during a time step (as opposed to just at the beginning of a time step), then enable the **Implicit Update** option. More information about these options and the related settings can be found in In-Cylinder Settings (p. 1868), Six DOF Solver Settings (p. 1883), Defining the Periodic Displacement of the Blades (p. 1978), and Implicit Update Settings (p. 1888), respectively.

Next, you must select the appropriate mesh update methods in the **Mesh Methods** group box, and set the associated parameters, if relevant. See Dynamic Mesh Update Methods (p. 1816) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.13: The Dynamic Mesh Task Page



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.6.2. Dynamic Mesh Update Methods

Three groups of mesh motion methods are available in Ansys Fluent to update the volume mesh in the deforming regions subject to the motion defined at the boundaries:

- Smoothing Methods (p. 1816)
- Dynamic Layering (p. 1837)
- Remeshing (p. 1841)

Note that

- you can use Ansys Fluent's dynamic mesh models in conjunction with adaption, with the exception of dynamic layering (in almost all cases) and any remeshing method.
- the current default mesh adaption method, that Dynamic Mesh and adaption may lack robustness
- PUMA 3D is incompatible with Dynamic mesh with no plans of support in the near future

For more information on the available adaption methods, see Hanging Node Adaption and Polyhedral Unstructured Mesh Adaption in the Theory Guide.

Details on how to set up the various dynamic mesh update methods are provided in the sections that follow.

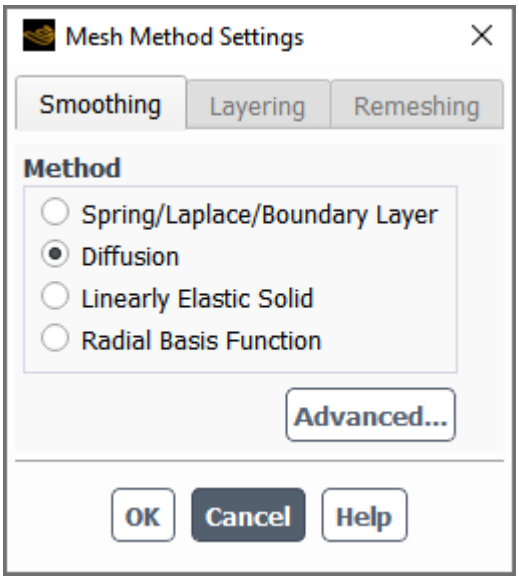
13.6.2.1. Smoothing Methods

When smoothing is used to adjust the mesh of a zone with a moving and/or deforming boundary, the interior nodes of the mesh move, but the number of nodes and their connectivity does not change. In this way, the interior nodes "absorb" the movement of the boundary. To enable smoothing, perform the following steps:

1. Enable the **Smoothing** option in the **Mesh Methods** group box of the Dynamic Mesh Task Page (p. 5111).
2. Click the **Settings...** button to open the **Mesh Method Settings** dialog box.

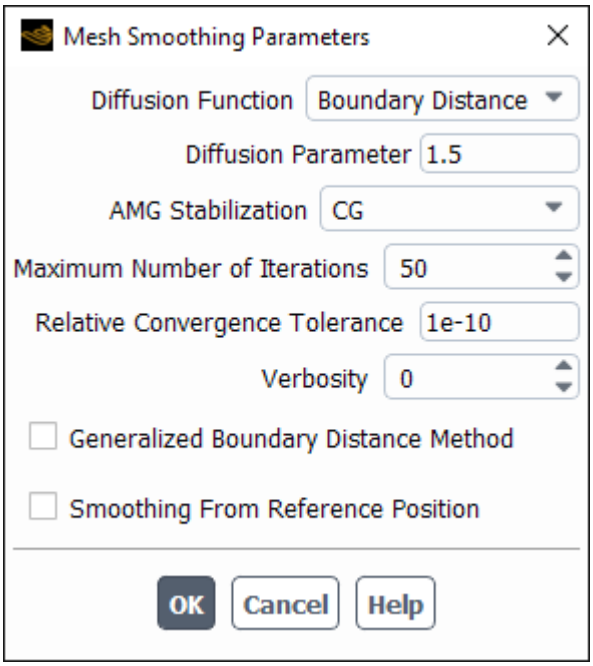
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.14: The Smoothing Tab of the Mesh Method Settings Dialog Box



3. If you want diffusion-based smoothing, select **Diffusion** from the **Method** list. Then click the **Advanced...** button and define the settings in the **Mesh Smoothing Parameters** dialog box. For details, see Diffusion-Based Smoothing (p. 1818).

Figure 13.15: The Mesh Smoothing Parameters Dialog Box



4. If you want spring-based smoothing, select **Spring/Laplace/Boundary Layer** from the **Method** list. Then click the **Advanced...** button and define the settings in the **Mesh Smoothing Parameters** dialog box. For details, see Spring-Based Smoothing (p. 1824).

5. If you want smoothing using the linearly elastic solid method, select **Linearly Elastic Solid** from the **Method** list. Then click the **Advanced...** button and define the settings in the **Mesh**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Smoothing Parameters dialog box. For details, see Linearly Elastic Solid Based Smoothing Method (p. 1828).

6. If you want smoothing that is based on a radial basis function interpolation, select **Radial Basis Function** from the **Method** list. Then click the **Advanced...** button and define the settings in the **Mesh Smoothing Parameters** dialog box. For details, see Radial Basis Function Smoothing (p. 1830).

7. If you selected **Diffusion**, **Linearly Elastic Solid**, or **Radial Basis Function** smoothing and the simulation involves periodic or quasi-periodic motion, you can specify that the smoothing uses a reference position. This option may improve the mesh quality consistency from cycle to cycle—see Smoothing from a Reference Position (p. 1832) for details.

8. If you plan to apply the 2.5D remeshing method (as described in 2.5D Surface Remeshing Method (p. 1857)), perform the following steps to set up Laplacian smoothing (as described in Laplacian Smoothing Method (p. 1833)).

   a. Select **Spring/Laplace/Boundary Layer** from the **Method** list

   b. Click the **Advanced...** button and define only the **Laplace Node Relaxation** and the **Maximum Number of Iterations** in the **Mesh Smoothing Parameters** group box (the other settings are not relevant).

9. If you plan to apply the boundary layer smoothing method (as described in Boundary Layer Smoothing Method (p. 1834)), select **Spring/Laplace/Boundary Layer** from the **Method** list.

13.6.2.1.1. Diffusion-Based Smoothing

For diffusion-based smoothing, the mesh motion is governed by the diffusion equation

$$\nabla \cdot \left(\gamma \nabla \vec{u} \right) = 0 \tag{13.2}$$

where $\vec{u}$ is the mesh displacement velocity. The boundary conditions for Equation 13.2 (p. 1818) are obtained from the user-prescribed or computed (six DOF) boundary motion. On deforming boundaries, the boundary conditions are such that the mesh motion is tangent to the boundary (that is, the normal velocity component vanishes). The Laplace equation Equation 13.2 (p. 1818) then describes how the prescribed boundary motion diffuses into the interior of the deforming mesh.

The diffusion coefficient γ in Equation 13.2 (p. 1818) can be used to control how the boundary motion affects the interior mesh motion. A constant coefficient means that the boundary motion diffuses uniformly throughout the mesh. With a nonuniform diffusion coefficient, mesh nodes in regions with high diffusivity tend to move together (that is, with less relative motion).

In Fluent, two different formulations for the diffusion coefficient γ are available for selection from the **Diffusion Function** drop-down list in the Mesh Smoothing Parameters Dialog Box (p. 5118). The first formulation allows you to have the diffusion coefficient be a function of the **Boundary Distance**, and is of the form

$$\gamma = \frac{1}{d^{\alpha}} \tag{13.3}$$

where d is a normalized boundary distance. The second formulation allows you to have the diffusion coefficient be a function of the **Cell Volume**, and is of the form

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\gamma = \frac{1}{V^{\alpha}} \tag{13.4}$$

where V is the normalized cell volume. In both Equation 13.3 (p. 1818) and Equation 13.4 (p. 1819), $\alpha \geq 0$ is a user input parameter. See Diffusivity Based on Boundary Distance (p. 1822) and Diffusivity Based on Cell Volume (p. 1823) for information about defining the diffusion coefficient.

Ansys Fluent uses different numerical methods to solve the vector Equation 13.2 (p. 1818), depending on the element types present in the mesh. In the absence of polyhedral elements or elements with hanging nodes (that is, meshes that have undergone hanging node adaption and some hexcore or CutCell meshes) the equation is solved using a finite element discretization and the displacement velocity, $\vec{u}$, is obtained directly at each mesh node. If the mesh contains polyhedral elements or hanging nodes, the equation is discretized using Ansys Fluent's standard finite volume method and the cell-centered solution for the displacement velocity, $\vec{u}$, is interpolated onto the nodes using inverse-distance-weighted averaging. The node positions are then updated according to:

$$\vec{x}_{new} = \vec{x}_{old} + \vec{u}\Delta t \tag{13.5}$$

The finite element discretization is generally superior, as the solution is obtained directly at the nodes and no interpolation step is necessary. The finite volume method can be enforced for meshes of all element types by executing the TUI command:

```
/define/dynamic-mesh/controls/smoothing-parameters/diffusion-fvm? yes
```

With finite element discretization, you can specify the solution method used by the linear solver for the diffusion-based smoothing calculations. By default, the **CG** (conjugate gradient) method is selected from the **AMG Stabilization** drop-down list. This method should be faster in this context than the other available methods, as it takes advantage of the symmetry of the matrix used in the linear systems of equations of finite-element-based mesh smoothing. If divergence is detected with CG, then the generalized minimal residual (GMRES) method will be used for an iteration as a fallback, and you will be informed in the console that a finite-element-based mesh smoothing coupled equation is being stabilized to enhance linear solver robustness. In rare cases, the CG method may result in the generation of negative volume cells; you may be able to avoid this by increasing the **Maximum Number of Iterations** from the default of 50 to a value in the range of 200–500.

If the CG method continues to generate negative volume cells with a higher number of iterations or if you repeatedly see console messages that say the GMRES fallback is being used, it is recommended that you select **GMRES** from the **AMG Stabilization** drop-down list. Note that the GMRES method is more robust than the CG method, especially for high-aspect-ratio meshes; but it is also more demanding in terms of memory usage and solver time.

Note that this drop-down list also allows you to select the **BCGSTAB** (bi-conjugate gradient stabilized) method; like the CG method, BCGSTAB falls back to the GMRES method for an iteration when divergence is detected.

Computationally, solving a PDE for mesh smoothing is generally more costly than spring-based smoothing. But it tends to produce better quality meshes than spring-based smoothing and often allows larger boundary deformations before breaking down. Figure 13.16: The Initial Mesh (p. 1820) and Figure 13.17: Valid Mesh After 45 Degree Rotation Using Diffusion-Based Smoothing (p. 1820) show a mesh before and after rotating the boundary by 45 degrees, using diffusion-based smoothing. With spring-based smoothing, the same mesh shows degenerated cells after a rotation

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

of 40 degrees (Figure 13.18: Degenerated Mesh After 40 Degree Rotation Using Spring-Based Smoothing (p. 1821)). (Note that radial basis function smoothing handles rotation better than both diffusion- and spring-based smoothing.)

Figure 13.16: The Initial Mesh

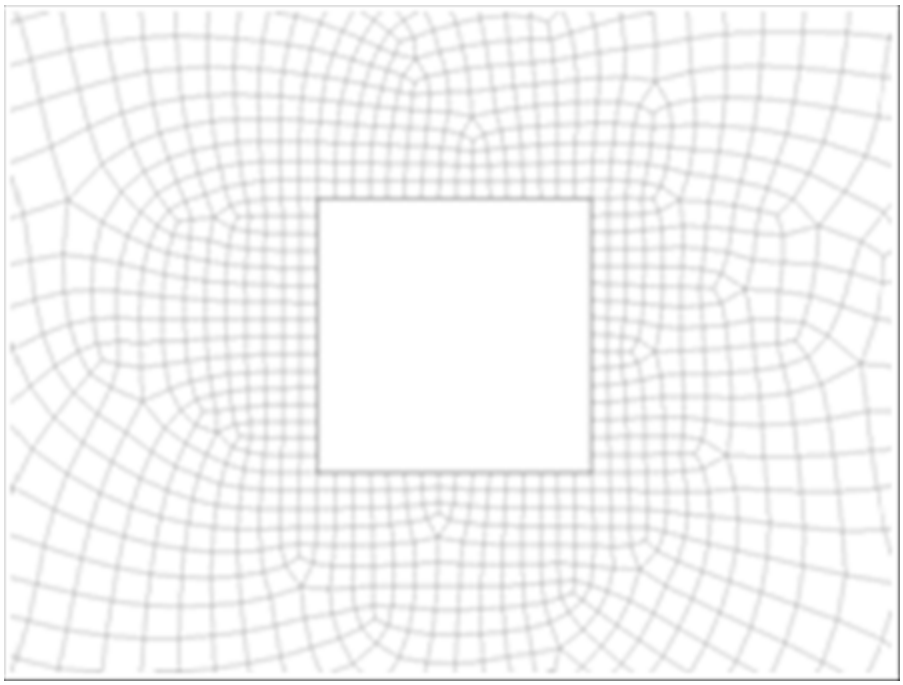
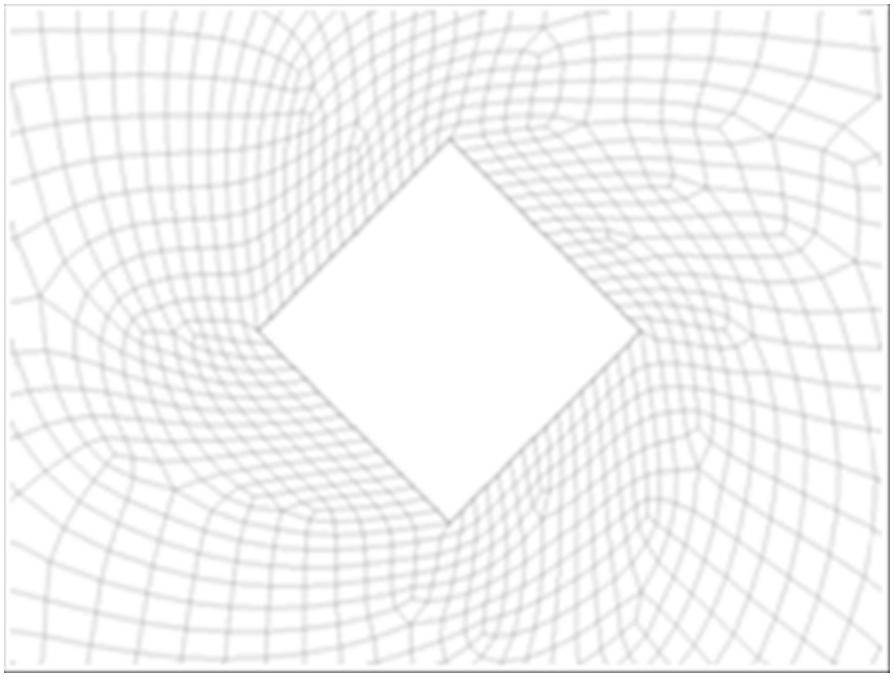
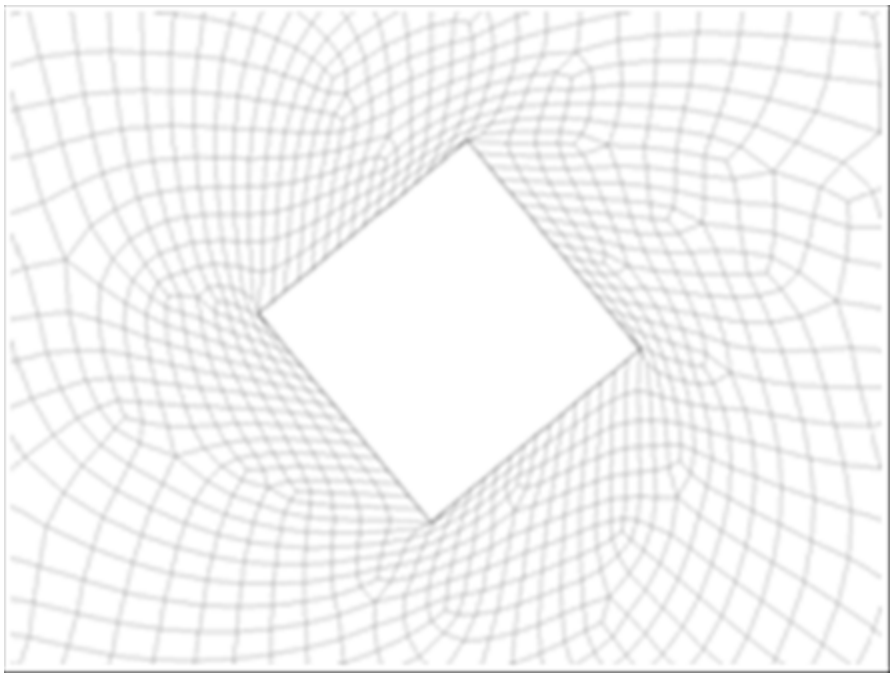


Figure 13.17: Valid Mesh After 45 Degree Rotation Using Diffusion-Based Smoothing



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.18: Degenerated Mesh After 40 Degree Rotation Using Spring-Based Smoothing



It should be noted that with diffusion-based smoothing the interior mesh motion is governed by the solution to Equation 13.2 (p. 1818) and the prescribed boundary motion, and not by mesh irregularities. Poor quality elements or mesh defects are not smoothed out by this method, but rather move together with the pre-computed (at the begin of each mesh update) displacement velocity $\vec{u}$.

It is also worth noting that the nature of the diffusion equation is such that the resulting solution (that is, the displacement velocity $\vec{u}$) depends on the dimensionality of the problem and the type of boundary motion prescribed. To illustrate the impact of the type of boundary motion, consider the goal of boundary-distance-based diffusion (Equation 13.3 (p. 1818)): to control which parts of the mesh absorb the boundary motion, so that you can preserve the mesh in the vicinity of the moving boundary (at the expense of the interior of the mesh). For more translational (piston-type) boundary motions, you can preserve a reasonably "thick" region of the mesh adjacent to the boundary (that is, multiple layers of cells); for rotational boundary motions, the rate of decay for the solution as you move away from the boundary is such that it can be difficult to preserve even a "thin" region. For this reason, mesh smoothing can handle translational boundary motions generally much better than rotational motions.

Although it should in most cases not be necessary, the accuracy of the solution to the diffusion equation governing the mesh motion can be controlled by defining the **Maximum Number of Iterations** and the **Relative Convergence Tolerance**.

You can enable printing of smoothing residuals in the console by entering 1 for the **Verbosity**.

For simulations with periodic or quasi-periodic motion, you can improve the mesh quality consistency by specifying that the diffusion-based smoothing uses a reference position, as described in Smoothing from a Reference Position (p. 1832).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

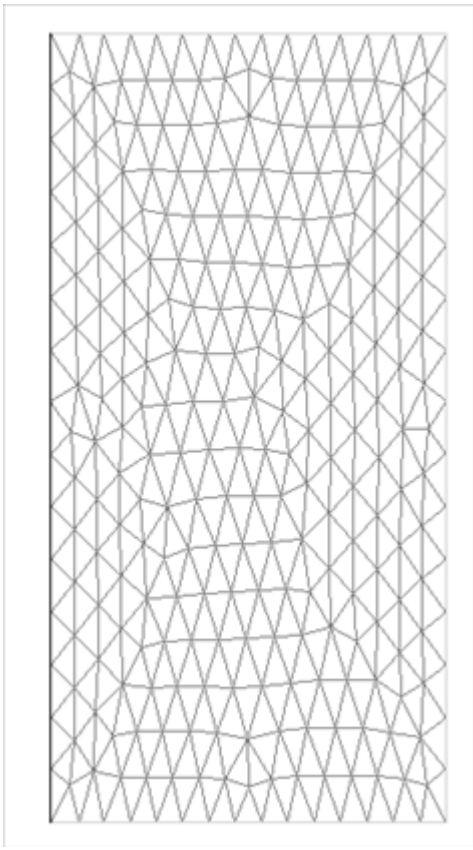
13.6.2.1.1.1. Diffusivity Based on Boundary Distance

Using boundary-distance-based diffusion (Equation 13.3 (p. 1818)) allows you to control how the boundary motion diffuses into the interior of the domain as a function of boundary distance. Decreasing the diffusivity away from the moving boundary causes those regions to absorb more of the mesh motion, and better preserves the mesh quality near the moving boundary. This is particularly helpful for a moving boundary that has pronounced geometrical features (such as sharp corners) along with a prescribed motion that is predominantly rotational.

You can manipulate the diffusion coefficient γ (in Equation 13.3 (p. 1818)) primarily by adjusting the **Diffusion Parameter** in the Mesh Smoothing Parameters Dialog Box (p. 5118) (α). A range of 0 to 2 has been shown to be of practical use. A value of 0 (the default value) specifies that $\gamma=1$ and yields a uniform diffusion of the boundary motion throughout the mesh. Higher values of α preserve larger regions of the mesh near the moving boundary, and cause the regions away from the moving boundary to absorb more of the motion.

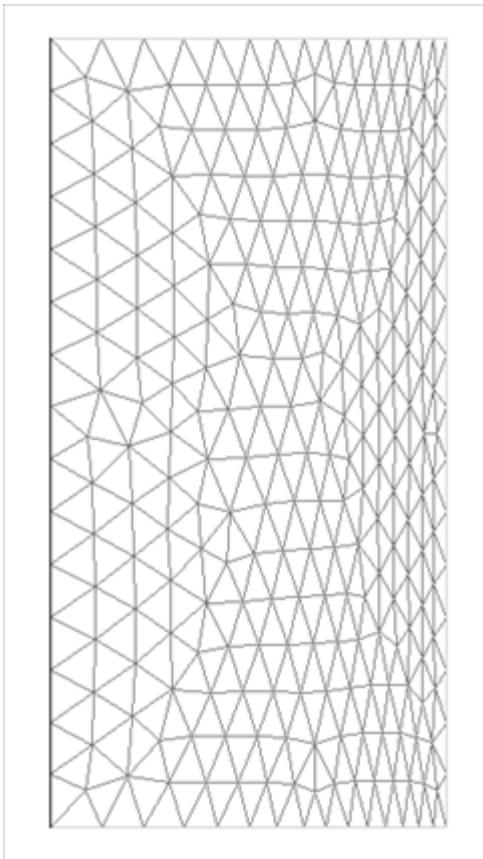
The following two figures illustrate the effect of the **Diffusion Parameter** α on the resulting mesh for a translational (piston-type) boundary motion, when the diffusivity is based on the boundary distance. In this example, an initially uniformly meshed square domain is deformed by moving the left boundary to the right.

Figure 13.19: Effect of Diffusion Parameter of 0 on Interior Node Motion



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.20: Effect of Diffusion Parameter of 1 on Interior Node Motion



For rotational boundary motions, a value of 1.5 for the **Diffusion Parameter** α is recommended as a good starting point.

Two different methods are available for the evaluation of the boundary distance d if boundary-distance-based diffusion is used. By default, Fluent uses the "standard" boundary distance in Equation 13.3 (p. 1818), which is the normalized distance to the nearest wall boundary; note that none of the other boundary types (for example, inlets, outlets, symmetry, and periodic boundaries) are considered. This method is the same as that which is used to evaluate the boundary distance for turbulence models. An example of this method is shown in Figure 13.20: Effect of Diffusion Parameter of 1 on Interior Node Motion (p. 1823), where only the left and right boundaries are walls. You have the option of enabling the **Generalized Boundary Distance Method** option instead, so that d is the normalized distance to the nearest boundary that is not declared as deforming, regardless of type. Both methods use the largest distance found in all deforming cell zones to normalize the value.

If the generalized boundary distance is used, an additional scalar equation for the boundary distance d will be solved as part of the solution of Equation 13.2 (p. 1818).

13.6.2.1.1.2. Diffusivity Based on Cell Volume

Using cell-volume-based diffusion (Equation 13.4 (p. 1819)) allows you to control how the boundary motion diffuses into the interior of the domain as a function of cell size. Decreasing the diffusivity in larger cells causes those cells to absorb more of mesh motion and therefore better preserves the cell quality of smaller cells.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can manipulate the diffusion coefficient γ (in Equation 13.3 (p. 1818)) by adjusting the **Diffusion Parameter** in the Mesh Smoothing Parameters Dialog Box (p. 5118) (α). A value of 0 (the default value) specifies that $\gamma=1$ and yields a uniform diffusion of the boundary motion throughout the mesh. Higher values of α result in larger cells absorbing more of the motion than smaller cells.

Note that the cell volume used in Equation 13.4 (p. 1819) is the local cell volume, normalized by the average cell volume of all deforming cell zones.

13.6.2.1.1.3. Applicability of the Diffusion-Based Smoothing Method

Diffusion-based mesh smoothing can be used to update any cell zone whose boundaries are moving or deforming. It is available for all element types, though it is not as robust for polyhedral elements as radial basis function smoothing.

Diffusion-based smoothing is computationally more expensive than spring-based smoothing, but likely results in better mesh quality (especially for non-tetrahedral / non-triangular cell zones, and for polyhedral cells in particular) and generally allows for larger boundary deformations before breaking down.

Similar to spring-based smoothing, diffusion-based mesh smoothing can handle translational boundary deformations much better than rotational motions. Rotational motions are best handled by radial basis function smoothing.

Diffusion-based smoothing is not compatible with the boundary layer smoothing method or the face region remeshing method. For more information about these methods, see Boundary Layer Smoothing Method (p. 1834) and Face Region Remeshing Method (p. 1852).

Unlike the radial basis function smoothing method, the diffusion-based smoothing method is not supported when the smoothing zone contains deforming conformal periodic boundaries and/or is a solid that has a moving wall.

13.6.2.1.2. Spring-Based Smoothing

For spring-based smoothing, the edges between any two mesh nodes are idealized as a network of interconnected springs. The initial spacings of the edges before any boundary motion constitute the equilibrium state of the mesh. A displacement at a given boundary node will generate a force proportional to the displacement along all the springs connected to the node. Using Hook's Law, the force on a mesh node can be written as

$$\vec{F}_i = \sum_j^{n_i} k_{ij} \left(\Delta\vec{x}_j - \Delta\vec{x}_i \right) \tag{13.6}$$

where $\Delta\vec{x}_i$ and $\Delta\vec{x}_j$ are the displacements of node i and its neighbor j, n_i is the number of neighboring nodes connected to node i, and k_{ij} is the spring constant (or stiffness) between node i and its neighbor j. The spring constant for the edge connecting nodes i and j is defined as

$$k_{ij} = \frac{k_{\text{fac}}}{\sqrt{\left|\vec{x}_i - \vec{x}_j\right|}} \tag{13.7}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where k_{fac} is the value you enter for **Spring Constant Factor** in the Mesh Smoothing Parameters Dialog Box (p. 5118).

At equilibrium, the net force on a node due to all the springs connected to the node must be zero. This condition results in an iterative equation such that

$$\Delta\vec{x}_i^{m+1} = \frac{\sum_j^{n_i} k_{ij}\Delta\vec{x}_j^{m}}{\sum_j^{n_i} k_{ij}} \tag{13.8}$$

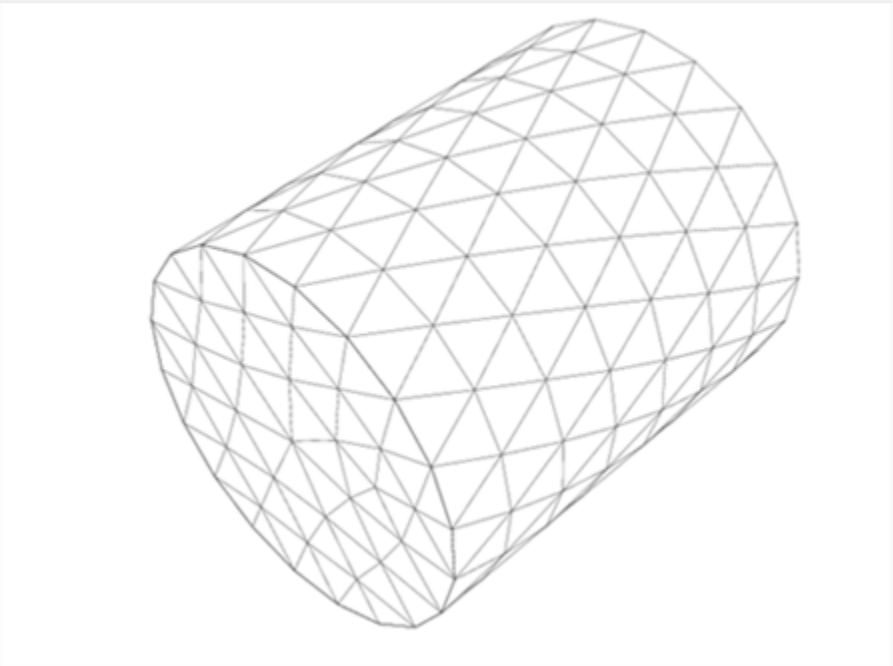
where m is the iteration number.

Since displacements are known at the boundaries (after boundary node positions have been updated), Equation 13.8 (p. 1825) is solved using a Jacobi sweep on all interior nodes. At convergence, the positions are updated such that

$$\vec{x}_i^{n+1} = \vec{x}_i^{n} + \Delta\vec{x}_i^{\text{converged}} \tag{13.9}$$

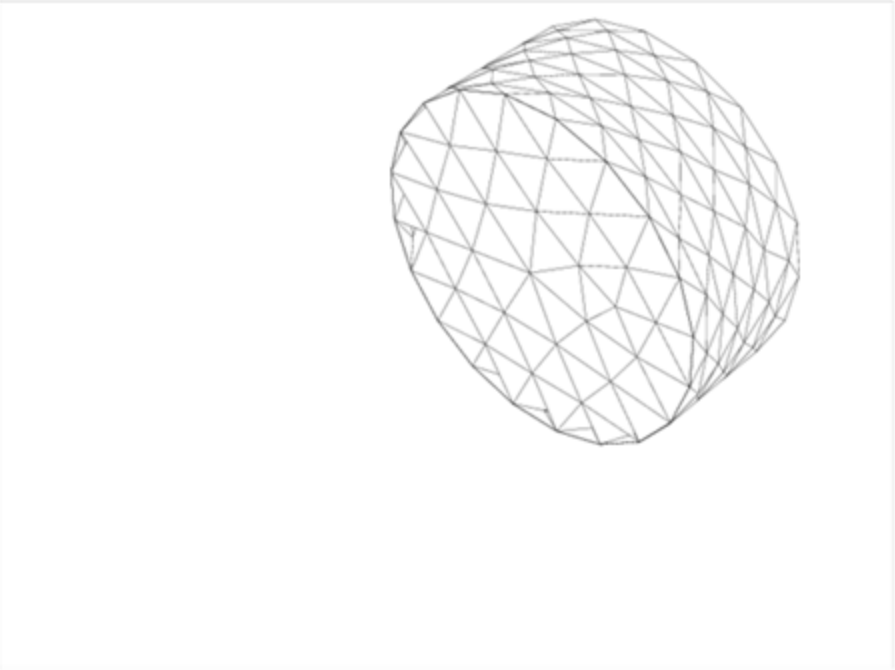
where $n+1$ and n are used to denote the positions at the next time step and the current time step, respectively. The spring-based smoothing is shown in Figure 13.21: Spring-Based Smoothing on Interior Nodes: Start (p. 1825) and Figure 13.22: Spring-Based Smoothing on Interior Nodes: End (p. 1826) for a cylindrical cell zone where one end of the cylinder is moving.

Figure 13.21: Spring-Based Smoothing on Interior Nodes: Start



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.22: Spring-Based Smoothing on Interior Nodes: End



You can control the spring stiffness by adjusting the value of the **Spring Constant Factor** between 0 and 1. A value of 0 indicates that there is no damping on the springs, and boundary node displacements have more influence on the motion of the interior nodes. A value of 1 imposes the default level of damping on the interior node displacements as determined by solving Equation 13.8 (p. 1825).

The effect of the **Spring Constant Factor** is illustrated in Figure 13.23: Interior Nodes Extend Beyond Boundary (Spring Constant Factor = 1) (p. 1827) and Figure 13.24: Interior Nodes Remain Within Boundary (Spring Constant Factor = 0) (p. 1827), which show the trailing edge of a NACA-0012 airfoil after a counter-clockwise rotation of 2.3° and the mesh is smoothed using the spring-based smoother but limited to 20 iterations. Degenerate cells (Figure 13.23: Interior Nodes Extend Beyond Boundary (Spring Constant Factor = 1) (p. 1827)) are created with the default value of 1 for the **Spring Constant Factor**, as the interior nodes extend beyond the moving boundary. However, the original mesh distribution (Figure 13.24: Interior Nodes Remain Within Boundary (Spring Constant Factor = 0) (p. 1827)) is recovered if the **Spring Constant Factor** is set to 0 (that is, no damping on the displacement of interior nodes near the airfoil surface).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.23: Interior Nodes Extend Beyond Boundary (Spring Constant Factor = 1)

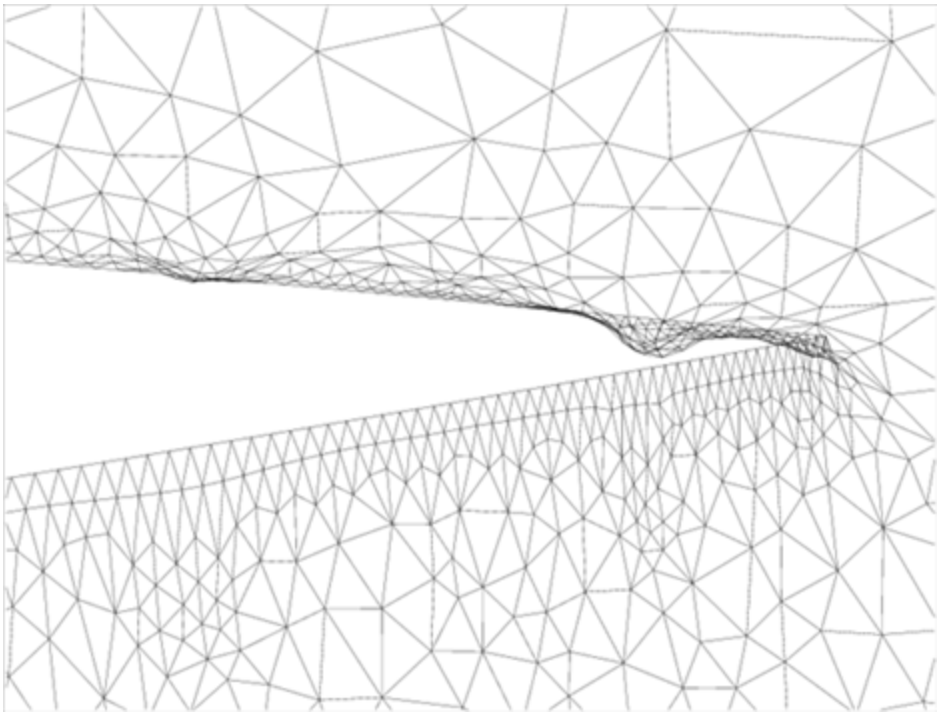
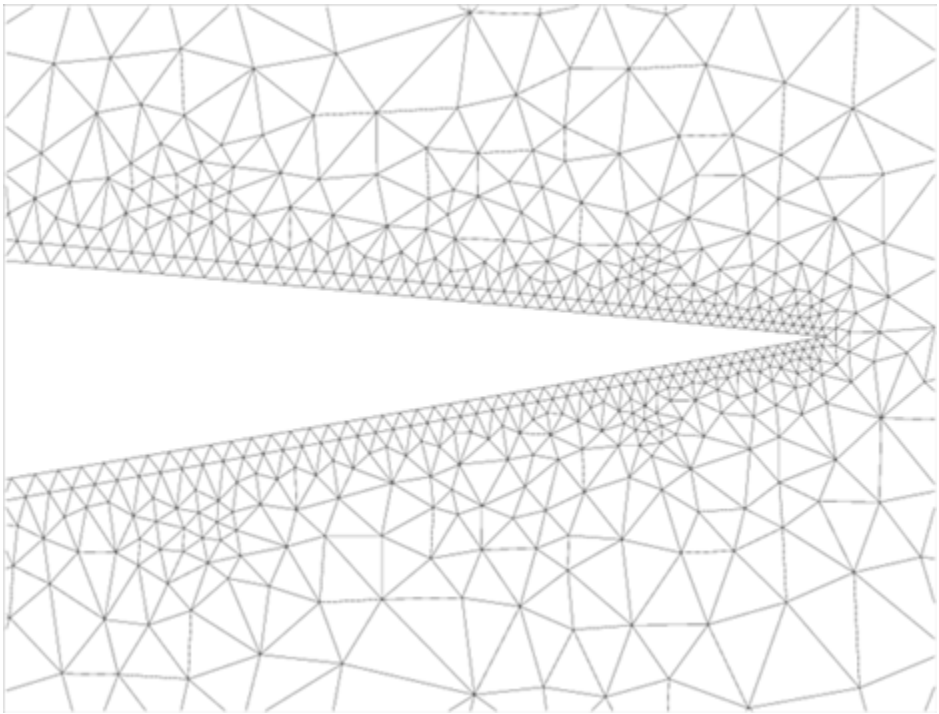


Figure 13.24: Interior Nodes Remain Within Boundary (Spring Constant Factor = 0)



You can control the solution of Equation 13.8 (p. 1825) using the values of **Convergence Tolerance** and **Maximum Number of Iterations**. Ansys Fluent solves Equation 13.8 (p. 1825) iteratively during each time step until one of the following criteria is met:

- The specified number of iterations has been performed.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The solution is converged for that time step:

$$\left(\frac{\Delta\vec{x}^{m}_{rms}}{\Delta\vec{x}^{1}_{rms}}\right) < \text{convergence tolerance} \tag{13.10}$$

where $\Delta\vec{x}^{1}_{rms}$ is the interior and deforming nodes RMS displacement at the first iteration.

13.6.2.1.2.1. Applicability of the Spring-Based Smoothing Method

You can use the spring-based smoothing method to update any cell or face zone whose boundary is moving or deforming.

For non-tetrahedral cell zones (non-triangular in 2D), the spring-based method can be used when the following conditions are met:

- The boundary of the cell zone moves predominantly in one direction (that is, no excessive anisotropic stretching or compression of the cell zone).
- The motion is predominantly normal to the boundary zone.

If these conditions are not met, the resulting cells may have high skewness values, since not all possible combinations of node pairs in non-tetrahedral cells (or non-triangular in 2D) are idealized as springs. Polyhedral cells are particularly likely to become highly skewed with spring-based smoothing (regardless of whether the previous conditions are met), and so the radial basis function smoothing method is generally recommended for polyhedra (see Radial Basis Function Smoothing (p. 1830)).

By default, spring-based smoothing is enabled for all cell zones. This is reflected in the Mesh Smoothing Parameters Dialog Box (p. 5118), where **All** is selected from the **Elements** list. If you want to disable spring-based smoothing for cell zones that are not entirely composed of either tetrahedral or triangular cells, you can do that by selecting **Tet in Tet Zones** in 3D (or **Tri in Tri Zones** in 2D).

If you have mixed element zones and you do not want spring-based smoothing on all element types, you can enable spring-based smoothing on only the tetrahedral or triangular cells by selecting **Tet in Mixed Zones** in 3D (or **Tri in Mixed Zones** in 2D). Selection of smoothing elements in the Mesh Smoothing Parameters Dialog Box (p. 5118) applies by default to all cell zones that undergo spring-based smoothing. In order to have more precise control, it is possible to overwrite this global selection on individual dynamic cell zones (see Deforming Motion (p. 1910)). This gives, for example, the flexibility to suppress smoothing in zones where dynamic layering (see Dynamic Layering (p. 1837)) is used and allows at the same time smoothing of non-simplex (that is not tetrahedral or triangular) elements in other zones.

Unlike the radial basis function smoothing method, the spring-based smoothing method is not supported when the smoothing zone contains deforming conformal periodic boundaries and/or is a solid that has a moving wall.

13.6.2.1.3. Linearly Elastic Solid Based Smoothing Method

With mesh smoothing based on the linearly elastic solid model, the mesh motion is governed by the following set of equations.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\begin{aligned}&\nabla\cdot\boldsymbol{\sigma}\left(\vec{y}\right)=\mathbf{0}\\&\boldsymbol{\sigma}\left(\vec{y}\right)=\lambda\left(tr\boldsymbol{\varepsilon}\left(\vec{y}\right)\right)\mathbf{I}+2\mu\boldsymbol{\varepsilon}\left(\vec{y}\right)\\&\boldsymbol{\varepsilon}\left(\vec{y}\right)=\tfrac{1}{2}\left(\nabla\vec{y}+\left(\nabla\vec{y}\right)^{T}\right)\end{aligned} \tag{13.11}$$

where $\boldsymbol{\sigma}$ is the stress tensor, $\boldsymbol{\varepsilon}$ is the strain tensor, and $\vec{y}$ is the mesh displacement. For the solution of Equation 13.11 (p. 1829) only the ratio between the shear modulus, μ, and Lamé's first parameter, λ, matters. This ratio is parameterized through Poisson's ratio (ν):

$$\nu=\frac{1}{2\left(1+{}_{\mu/\lambda}\right)} \tag{13.12}$$

You define this property through the **Poisson's Ratio** field in the Mesh Smoothing Parameters Dialog Box (p. 5118). Note that the linearly elastic solid mesh smoothing model supports constant material properties only. The permissible range for **Poisson's Ratio** is between -1.0 and 0.5.

The boundary conditions for Equation 13.11 (p. 1829) are obtained from the user-prescribed or computed (in the case of six DOF motion) boundary deformations. These imposed deformations are transferred into the interior of the deforming mesh as if the mesh was a linearly elastic solid with the given material properties. On deforming boundaries you can either specify a geometry along which the mesh can slide, or leave the geometry unspecified. If a geometry is specified for the deforming boundary, then the boundary conditions are such that the deformation normal to the boundary vanishes and the stress tangential to the boundary is zero. If the geometry of the deforming boundary is unspecified, then the deforming boundary can also deform in the normal direction and the boundary conditions are such that the traction is zero in all directions. See Deforming Motion (p. 1910) for details of how to specify geometry on deforming zones.

The linear system in Equation 13.11 (p. 1829) is solved using a finite element discretization and the mesh displacements for the interior and deforming boundary nodes are obtained directly at the nodes. The accuracy to which the linear system is solved can be controlled by the **Maximum Number of Iterations** and the **Relative Convergence Tolerance**.

You can specify the solution method used by the linear solver for the linearly elastic solid mesh smoothing calculations. By default, the **CG** (conjugate gradient) method is selected from the **AMG Stabilization** drop-down list. This method should be faster in this context than the other available methods, as it takes advantage of the symmetry of the matrix used in the linear systems of equations of finite-element-based mesh smoothing. If divergence is detected with CG, then the generalized minimal residual (GMRES) method will be used for an iteration as a fallback, and you will be informed in the console that a finite-element-based mesh smoothing coupled equation is being stabilized to enhance linear solver robustness. In rare cases, the CG method may result in the generation of negative volume cells; you may be able to avoid this by increasing the **Maximum Number of Iterations** from the default of 50 to a value in the range of 200–500.

If the CG method continues to generate negative volume cells with a higher number of inner iterations or if you repeatedly see console messages that say the GMRES fallback is being used, it is recommended that you select **GMRES** from the **AMG Stabilization** drop-down list. Note that the GMRES method is more robust than the CG method, especially for high-aspect-ratio meshes; but it is also more demanding in terms of memory usage and solver time.

Note that this drop-down list also allows you to select the **BCGSTAB** (bi-conjugate gradient stabilized) method; like the CG method, BCGSTAB falls back to the GMRES method for an iteration when divergence is detected.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can enable printing of smoothing residuals in the console by entering 1 for the **Verbosity**.

For simulations with periodic or quasi-periodic motion, you can improve the mesh quality consistency by specifying that the linearly elastic solid mesh smoothing uses a reference position, as described in Smoothing from a Reference Position (p. 1832).

13.6.2.1.3.1. Applicability of the Linearly Elastic Solid Based Smoothing Method

Most of the properties and limitations discussed for diffusion-based smoothing (Applicability of the Diffusion-Based Smoothing Method (p. 1824)) also apply to the linearly elastic solid model, particularly the mesh quality degradation for rotational motions. The linearly elastic solid model is computationally more expensive than diffusion-based smoothing, but for some meshes and mesh motions preserves the mesh quality better.

The current implementation with constant material properties can be a limitation compared with diffusion-based smoothing with non-uniform diffusivity. In cases with rotational boundary motion and sharp corners it may be advantageous to use radial basis function smoothing, or perhaps diffusion-based smoothing with boundary-distance-dependent diffusivity.

The linearly elastic solid smoothing method provides you the option to leave the geometry of deforming face zones unspecified. This option is not available for any other smoothing method except for the radial basis function method.

The linearly elastic solid smoothing method supports triangular and quadrilateral elements in 2D and tetrahedral, hexahedral, wedge, and pyramid cells in 3D. It cannot be applied if the deforming cell zone contains polyhedral cells or hanging nodes. In such cases radial basis function smoothing is recommended.

Linearly elastic solid smoothing is not compatible with the boundary layer smoothing method or the face region remeshing method. For more information about these methods, see Boundary Layer Smoothing Method (p. 1834) and Face Region Remeshing Method (p. 1852).

Unlike the radial basis function smoothing method, the linearly elastic solid smoothing method is not supported when the smoothing zone contains deforming conformal periodic boundaries and/or is a solid that has a moving wall.

13.6.2.1.4. Radial Basis Function Smoothing

When the smoothing is based on a radial basis function interpolation, Fluent solves for a displacement field that satisfies the displacements at boundaries with prescribed motion, and then interpolates the displacements onto all other nodes in the interior and at the deforming boundaries. As with the other available smoothing methods, deforming boundaries are boundaries where the nodes are allowed to slide tangentially.

Radial basis function smoothing has the following advantages:

- Compared to the other available smoothing methods, it is particularly well suited for motions that undergo rotation. On average, more rotational motion can be absorbed by the mesh with radial basis function smoothing before the mesh quality degenerates.
- Unlike linearly elastic solid based smoothing, it is available for adapted meshes with hanging nodes or polyhedra elements.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Unlike diffusion smoothing, it is robust for polyhedral elements.
- It is the only smoothing method that is supported when:
  - the smoothing zone contains deforming conformal periodic boundaries.
  - the smoothing zone is a solid that has a moving wall, as may be the case in a simulation that models ablation or erosion / accretion (see Thermal Conditions for Two-Sided Walls (p. 1472) and Particle Erosion Coupled with Dynamic Meshes (p. 2793), respectively). Note that the ablation or erosion / accretion must be set up on the fluid side of the coupled wall.

On deforming boundaries, you can either specify a geometry (such as a plane, cylinder, and so on) along which the mesh can slide, or leave the geometry unspecified so that the nodes are allowed to move in any direction. See Deforming Motion (p. 1910) for details of how to define the geometry for deforming zones. The ability to leave the geometry of deforming face zones unspecified is not available for any other smoothing method except for the radial basis function smoothing method and the linearly elastic solid smoothing method. Note that if you have a non-conformal interface where both of the interface boundaries are defined as deforming dynamic zones with unspecified geometry, then Fluent will ensure that the two interface zones stay connected (by projecting one of the interface zones onto the other); such behavior is only supported with radial basis function smoothing, and then only if the same cell zone is on either side of the non-conformal interface.

Advanced settings are available for radial basis function smoothing in the Mesh Smoothing Parameters Dialog Box (p. 5118). The accuracy to which the displacement field is solved can be controlled by the **Relative Convergence Tolerance**: if the smoothing produces negative volume cells, you can decrease this value to try to address the problem; if the solution is running well, you can increase the value to speed up the calculation. You can also set the **Verbosity** to 1 if you want to print the absolute tolerance computed (based on your specified relative tolerance) in the console during the calculation. If you want to limit the amount of the domain that is smoothed to only those cells that are close to the moving boundary zone(s), you can enable the **Local Smoothing** option (for details, see Local Smoothing with the Radial Basis Function Smoothing Method (p. 1832)). Finally, you can specify that the smoothing uses a reference position (as described in Smoothing from a Reference Position (p. 1832)), which can improve mesh quality consistency when performing many cycles of periodic or quasi-periodic motion for stationary or moving meshes.

If you want to smooth the entire mesh but would like to take advantage of the boundary layer smoothing that is done as part of the **Local Smoothing** option (and hence retain the shape of the boundary layers as much as possible), you can make sure that the **Local Smoothing** option is disabled and use the following text command:

`define → dynamic → controls → smoothing-parameters → smooth-boundary-layers-with-adjacent-zone?`

Note the following limitations:

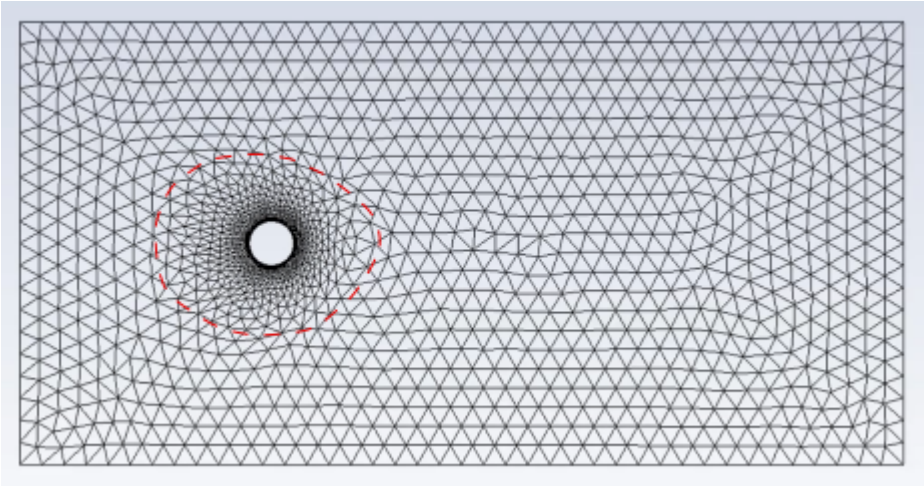
- The performance of radial basis function smoothing does not scale as well as the other smoothing methods when running distributed memory on a cluster.
- Radial basis function smoothing is not compatible with the boundary layer smoothing method. For more information about this method, see Boundary Layer Smoothing Method (p. 1834).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.6.2.1.4.1. Local Smoothing with the Radial Basis Function Smoothing Method

As mentioned in the previous section, you can enable the **Local Smoothing** option in the Mesh Smoothing Parameters Dialog Box (p. 5118) when using the radial basis function smoothing method, so that only the portion of the mesh adjacent to the moving boundary zone is smoothed. This ensures that the mesh away from the moving boundary remains undisturbed by the smoothing, and decreases the computational time required for smoothing. The following figure provides an example of such local smoothing: the circular boundary zone is moving to the right, and only those cells within the red dotted line are smoothed, while the rest of the domain is unaffected.

Figure 13.25: An Example of Where Local Smoothing is Applied



Note:

Depending on the amplitude of motion, it might be necessary to enable remeshing along with the local smoothing, as the motion is now absorbed in a smaller region. For details, see Remeshing (p. 1841).

If there are boundary layers adjacent to the moving zone, they are smoothed before and independently of the local smoothing region, ensuring that they retain their shape as much as possible. The portion of the mesh smoothed by the local smoothing is then evaluated starting from the edge of the boundary layers. When identifying which cells are boundary layers, Fluent searches for prismatic cells (that is, hexahedral cells or wedge or polyhedral cells that have the same number of nodes on the top and bottom faces relative to the boundary normal) that are in layers adjacent to a boundary zone.

13.6.2.1.5. Smoothing from a Reference Position

When the mesh smoothing is based on diffusion, the linearly elastic solid model, or a radial basis function interpolation, you have the option of specifying that the smoothing uses a reference position. This can be helpful when performing many cycles of periodic or quasi-periodic motion for stationary or moving meshes; for example, turbomachinery rotors with blade flutter. When you ensure that the smoothing is always done from the same reference position, the mesh quality may remain more consistent from cycle to cycle.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To use this feature, enable the **Smoothing From Reference Position** option in the Mesh Smoothing Parameters Dialog Box (p. 5118). Note this option is enabled by default during the calculation if the case includes a cell zone that has **Frame Motion** or **Mesh Motion** enabled with a nonzero rotational velocity and that is also defined as a deforming dynamic mesh zone. This option is not available if the **Layering** and/or **Remeshing** options are enabled in the **Dynamic Mesh** task page, or if automatic adaption criteria is activated in the **Manage Adaption Criteria** dialog box.

The reference position is saved the first time you perform smoothing after enabling this option. It is recommended that you do not save your case / data files in the legacy format (that is, with the `file/cff-file?` text command set to `no`): the reference position is not saved in this format, and so restarting from a `.cas` file will imply that a different reference position is used after restart. This still provides the benefit of consistent quality, but the resulting meshes might be slightly different from a continuous run. The impact of such slight mesh changes on the solution results will be small, except for cases with a high degree of mesh dependence; if you must use a legacy case file, it is recommended that you evaluate the original mesh to ensure that it is suitable for the problem.

13.6.2.1.6. Laplacian Smoothing Method

Laplacian smoothing is the most commonly used and the simplest mesh smoothing method. This method adjusts the location of each mesh vertex to the geometric center of its neighboring vertices. This method is computationally inexpensive but it does not guarantee an improvement on mesh quality, since repositioning a vertex by Laplacian smoothing can result in poor quality elements. To overcome this problem, Ansys Fluent only relocates the vertex to the geometric center of its neighboring vertices if and only if there is an improvement in the mesh quality (that is, the skewness has been improved).

This improved Laplacian smoothing can be enabled on deforming boundaries only (that is, the zone with triangular elements in 3D and zones with linear elements in 2D). The computation of the node positions works as follows:

$$\overrightarrow{\overline{x}}_i^m = \frac{\sum_j^{n_i} \vec{x}_i^m}{n_i} \tag{13.13}$$

where $\overrightarrow{\overline{x}}_i^m$ is the averaged node position of node i at iteration m, $\vec{x}_j^m$ is the node position of neighbor node of $\vec{x}_i^m$ at iteration m, and n_i is the number nodes neighboring node i. The new node position $\vec{x}_i^{m+1}$ is then computed as follows:

$$\vec{x}_i^{m+1} = \vec{x}_i^m (1-\beta) + \overrightarrow{\overline{x}}_i^m \beta \tag{13.14}$$

where β is the Laplace node relaxation factor.

This update only happens if the maximum skewness of all faces adjacent to $\vec{x}_i^{m+1}$ is improved in comparison to $\vec{x}_i^m$.

For details on applying Laplacian smoothing to either a cell zone (with 2.5D remeshing) or a face zone, see Smoothing Methods (p. 1816) or Deforming Motion (p. 1910), respectively.

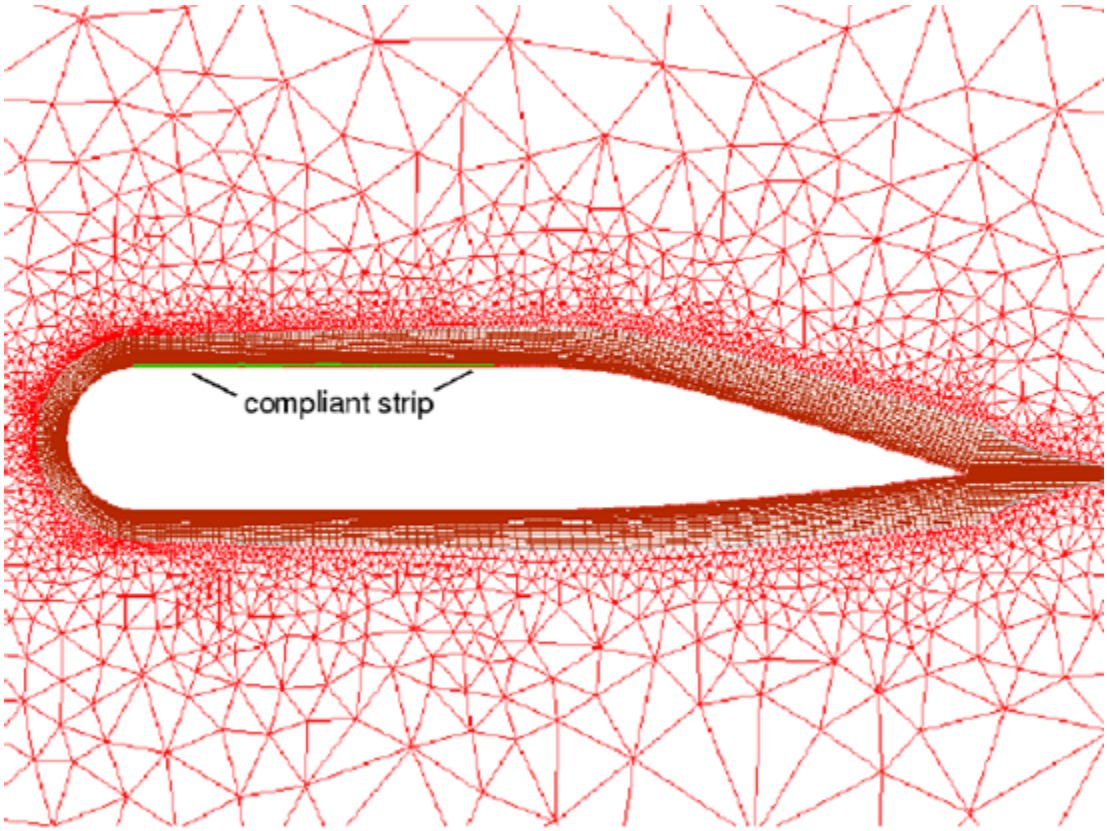
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.6.2.1.7. Boundary Layer Smoothing Method

The boundary layer smoothing method is used to deform the boundary layer mesh during a moving-deforming mesh simulation. For cases that apply mesh motion (either **Rigid Body** or **User-Defined**) to a face zone with adjacent boundary layers, the boundary layers can be made to deform accordingly by enabling **Deform Adjacent Boundary Layer with Zone** for the face zone in the Dynamic Mesh Zones Dialog Box (p. 5140). With boundary layer smoothing enabled, the nodal coordinates of each cell in the boundary layer zone are updated with the same displacement vector as the corresponding nodes on the underlying face zone. The boundary layer smoothing method can be applied to boundary layer zones of all mesh types (that is, wedges and hexahedra in 3D, quadrilaterals in 2D).

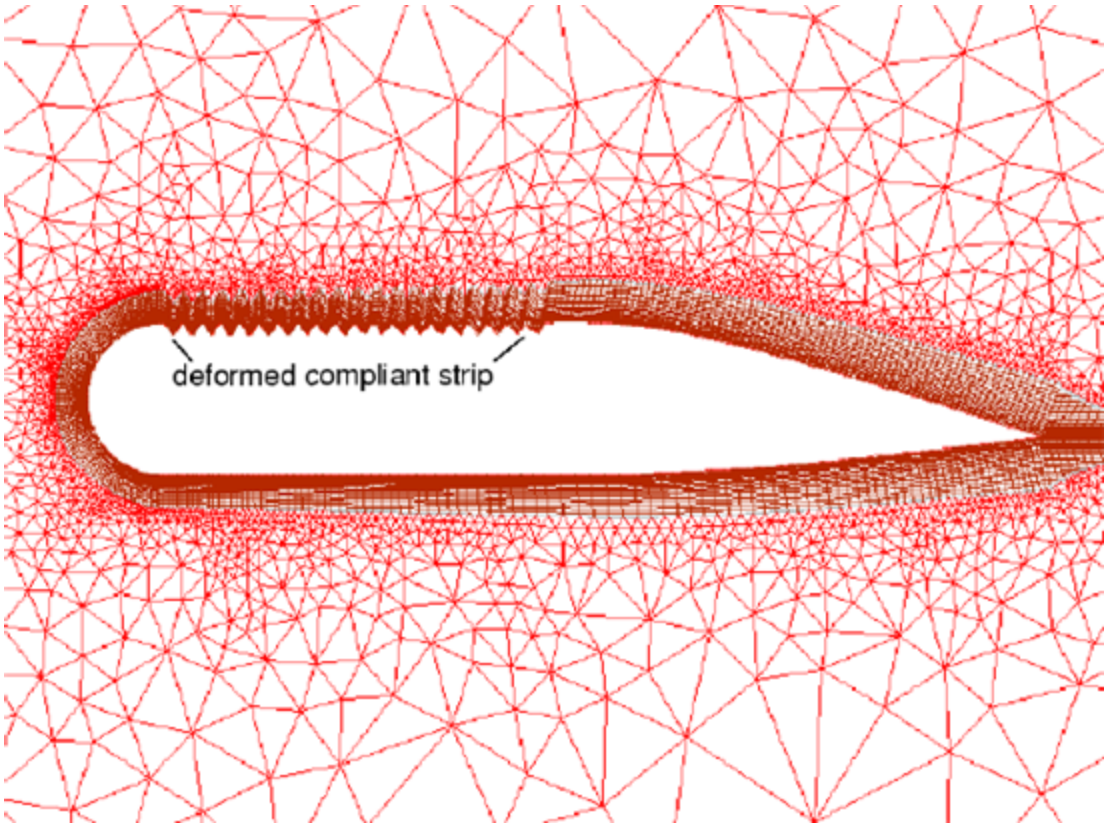
Consider the example below, where a UDF of the form `DEFINE_GRID_MOTION` provides the moving-deforming mesh model with the locations of the nodes located on the compliant strip on an idealized airfoil. The node motion varies sinusoidally in time and space (compare Figure 13.26: The Undeformed Mesh (p. 1834) with Figure 13.27: The Deformed Mesh (p. 1835)).

Figure 13.26: The Undeformed Mesh



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

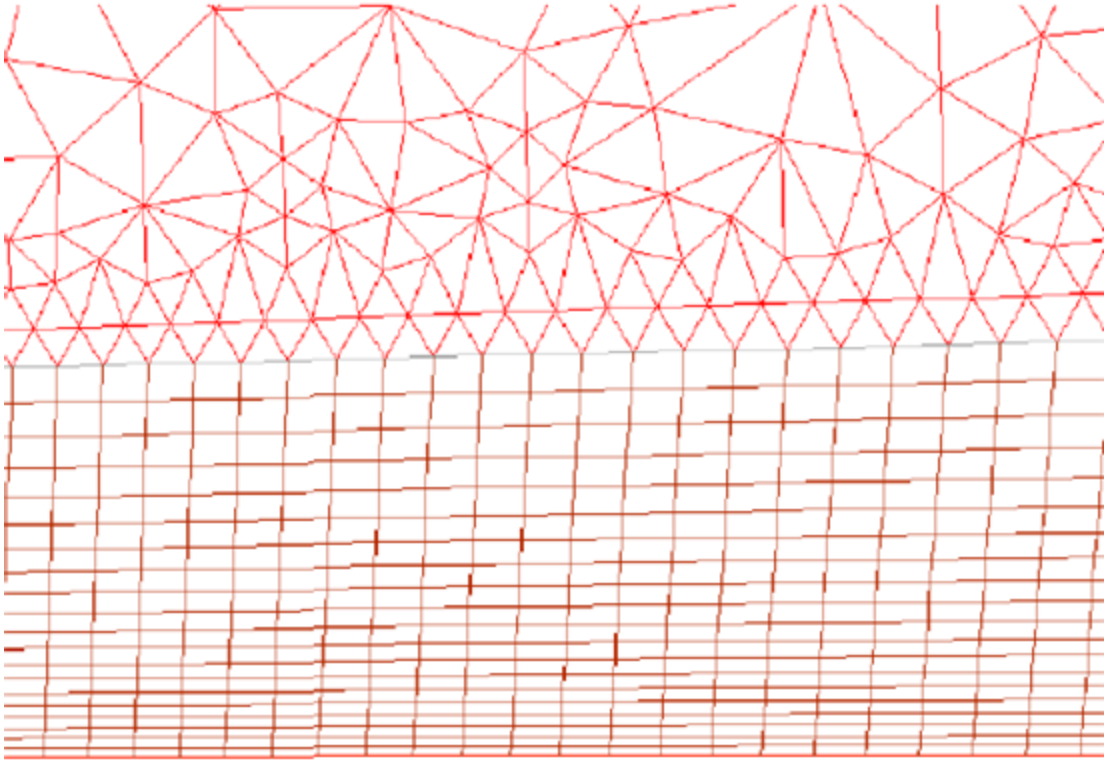
Figure 13.27: The Deformed Mesh



As a result of the boundary layer smoothing, the cells adjacent to the deforming wall are also deformed in order to preserve the quality of boundary layer zone. If you compare Figure 13.28: Zooming into the Undeformed Compliant Strip (p. 1836) with Figure 13.29: Zooming into the Deformed Compliant Strip with Boundary Layer Smoothing Applied (p. 1837), you can see how the boundary layer cells have been deformed according to the motion of the nodes on the compliant strip.

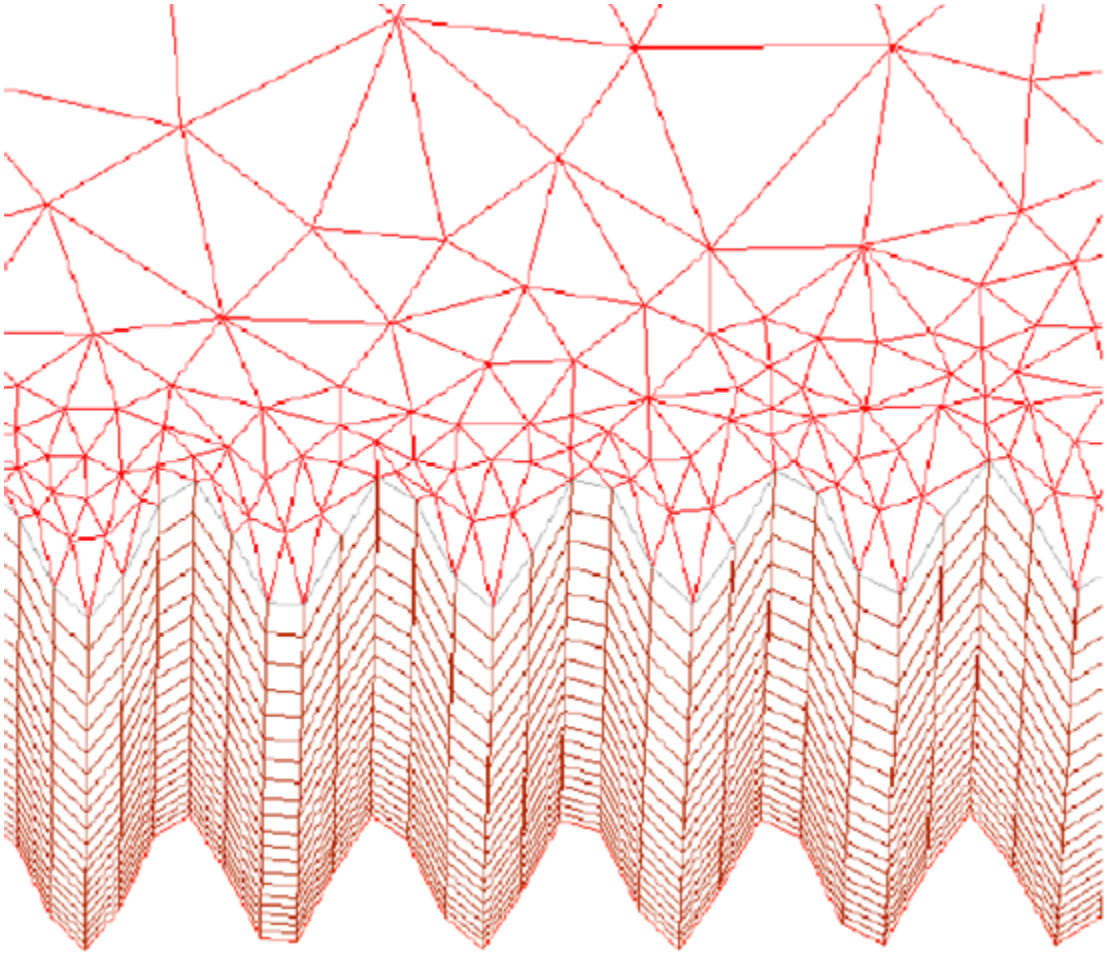
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.28: Zooming into the Undeformed Compliant Strip



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.29: Zooming into the Deformed Compliant Strip with Boundary Layer Smoothing Applied



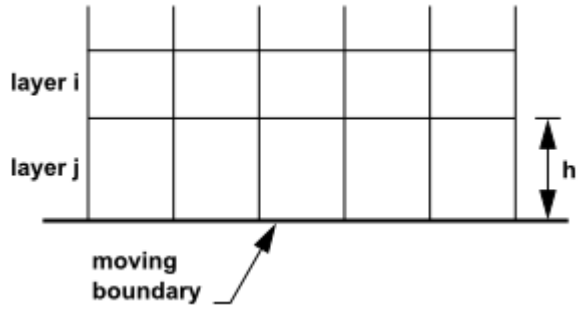
Typically, the boundary layer smoothing method preserves the height of the boundary layer cells adjacent to the deformed face zone. However, note that this approach is primarily intended for translational motion. If the faces undergo substantial rotation, the boundary layer cells may become skewed. See Smoothing Methods (p. 1816) and Specifying Boundary Layer Deformation Smoothing (p. 1915) for details about enabling smoothing and defining a moving and deforming boundary layer, respectively. Note that boundary layer smoothing is compatible with spring-based smoothing only. It cannot be used with diffusion-based smoothing, linearly elastic solid smoothing, or radial basis function smoothing. Also note that the boundary layer smoothing method will work whether or not you have segregated the boundary layer elements into a separate cell zone.

13.6.2.2. Dynamic Layering

In prismatic (polyhedral, hexahedral, and/or wedge in 3D, or quadrilateral in 2D) mesh zones, you can use dynamic layering to add or remove layers of cells adjacent to a moving boundary, based on the height of the layer adjacent to the moving surface. The dynamic mesh model in Ansys Fluent allows you to specify an ideal layer height on each moving boundary. The layer of cells adjacent to the moving boundary (layer j in Figure 13.30: Dynamic Layering (p. 1838)) is split or merged with the layer of cells next to it (layer i in Figure 13.30: Dynamic Layering (p. 1838)) based on the height (h) of the cells in layer j.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.30: Dynamic Layering



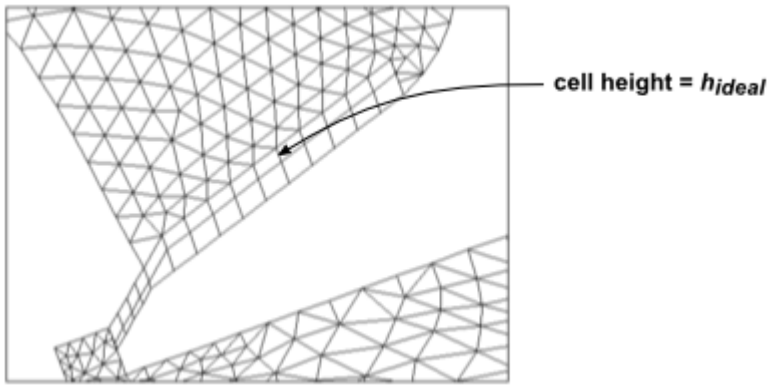
If the cells in layer j are expanding, the cell heights are allowed to increase until

$$h_{min} > (1+\alpha_s)h_{ideal} \tag{13.15}$$

where h_{min} is the minimum cell height of cell layer j, h_{ideal} is the ideal cell height, and α_s is the layer split factor. Note that Ansys Fluent allows you to define h_{ideal} as either a constant value or a value that varies as a function of time or crank angle. When the condition in Equation 13.15 (p. 1838) is met, the cells are split based on the specified layering option. This option can be height based or ratio based.

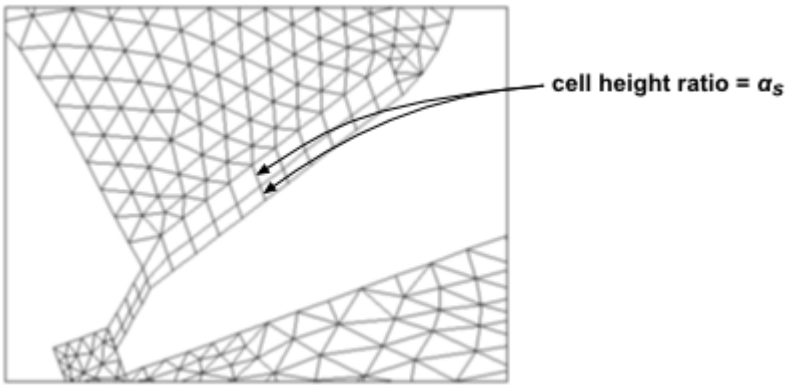
With the height-based option, the cells are split to create a layer of cells with constant height h_{ideal} and a layer of cells of height $h-h_{ideal}$. With the ratio-based option the cells are split such that, locally, the ratio of the new cell heights to old cell heights is exactly α_s everywhere. Figure 13.31: Results of Splitting Layer with the Height-Based Option (p. 1838) and Figure 13.32: Results of Splitting Layer with the Ratio-Based Option (p. 1839) show the result of splitting a layer of cells above a valve geometry using the height-based and ratio-based option.

Figure 13.31: Results of Splitting Layer with the Height-Based Option



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.32: Results of Splitting Layer with the Ratio-Based Option



If the cells in layer j are being compressed, they can be compressed until

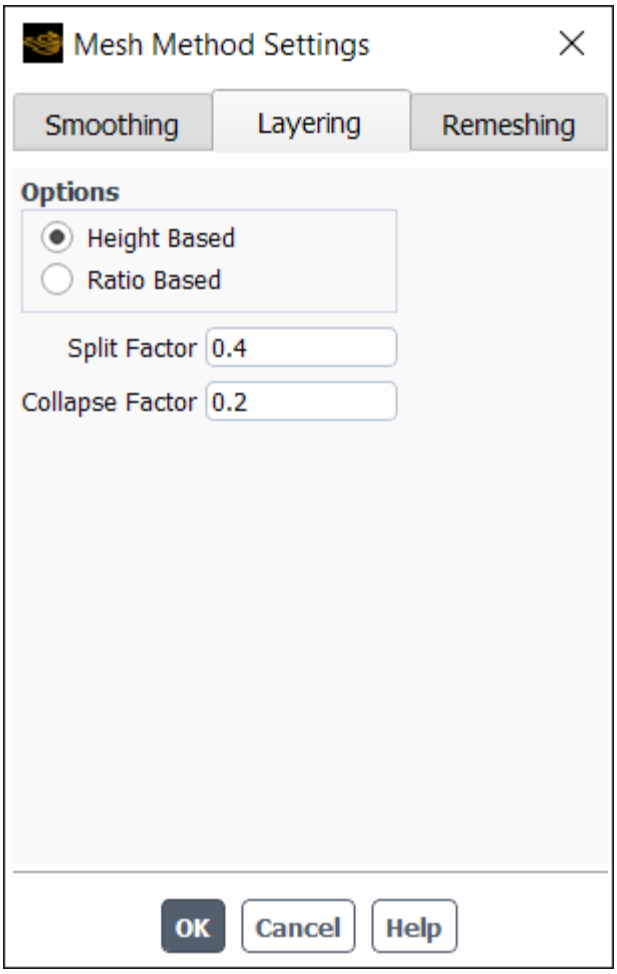
$$h_{min} < \alpha_c h_{ideal} \tag{13.16}$$

where α_c is the layer collapse factor. When this condition is met, the compressed layer of cells is merged into the layer of cells above the compressed layer in Figure 13.30: Dynamic Layering (p. 1838); that is, the cells in layer j are merged with those in layer i.

To enable dynamic layering, enable the **Layering** option under **Mesh Methods** in the Dynamic Mesh Task Page (p. 5111) (Figure 13.33: The Layering Tab in the Mesh Method Settings Dialog Box (p. 1840)). The layering control is specified in the **Layering** tab, which can be displayed by clicking **Settings...**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.33: The Layering Tab in the Mesh Method Settings Dialog Box



You can control how a cell layer is split by specifying either **Height Based** or **Ratio Based** under **Options**. Note that for **Height Based**, the height of the cells in a particular new layer will be constant, but you can choose to have this height vary from layer to layer as a function of time or crank angle when you specify the **Cell Height** in the **Dynamic Mesh Zones** dialog box (see Specifying the Motion of Dynamic Zones (p. 1904) for further details).

The **Split Factor** and **Collapse Factor** (α_s in Equation 13.15 (p. 1838) and α_c in Equation 13.16 (p. 1839), respectively) are the factors that determine when a layer of prismatic cells (polyhedra, hexahedra, or wedges in 3D, or quadrilaterals in 2D) that is next to a moving boundary is split or merged with the adjacent cell layer, respectively.

13.6.2.2.1. Applicability of the Dynamic Layering Method

You can use the dynamic layering method to split or merge cells adjacent to any moving boundary provided the following conditions are met:

- All cells adjacent to the moving face zone are prismatic and either polyhedra, hexahedra, or wedges (quadrilaterals in 2D), even though the cell zone may contain mixed cell shapes.
- The cell layers must be completely bounded by one-sided face zones, except when sliding interfaces are used (see Applicability of the Face Region Remeshing Method (p. 1856)).

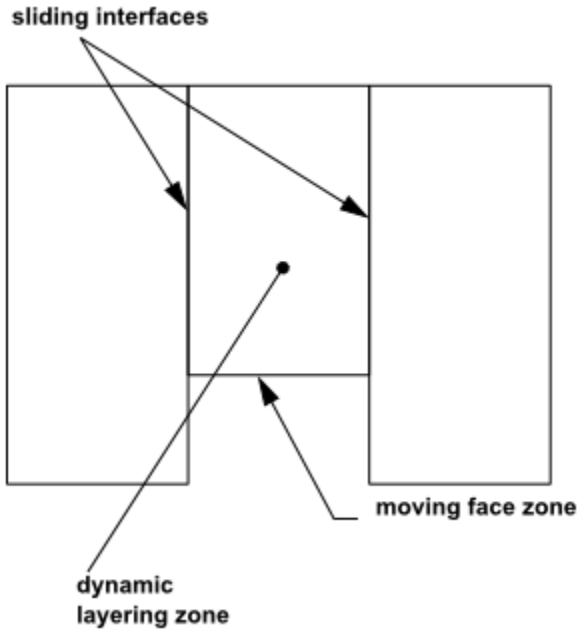
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Note that you cannot use the dynamic layering method in conjunction with adaption in almost all cases. For more information on the available adaption methods, see Hanging Node Adaption and Polyhedral Unstructured Mesh Adaption in the Theory Guide.

If the moving boundary is an internal zone, cells on both sides (possibly with different ideal cell layer heights) of the internal zone are considered for dynamic layering.

If you want to use dynamic layering on cells adjacent to a moving wall that do not span from boundary to boundary, you must separate those cells that are involved in the dynamic layering and use the sliding interfaces capability in Ansys Fluent to transition from the deforming cells to the adjacent non-deforming cells (see Figure 13.34: Use of Sliding Interfaces to Transition Between Adjacent Cell Zones and the Dynamic Layering Cell Zone (p. 1841)). For a moving interior face, the zones must be separated such that they are either expanding or collapsing on the same side. No one zone can consist of both expanding and collapsing layers.

Figure 13.34: Use of Sliding Interfaces to Transition Between Adjacent Cell Zones and the Dynamic Layering Cell Zone



13.6.2.3. Remeshing

When the boundary displacement is large compared to the local cell sizes, the cell quality can deteriorate or the cells can become degenerate if only mesh smoothing is used. This will invalidate the mesh (for example, result in negative cell volumes) and consequently, will lead to convergence problems when the solution is updated to the next time step.

To circumvent this problem, Ansys Fluent can agglomerate cells that violate the skewness or size criteria and locally remesh the agglomerated cells or faces. If the new cells or faces satisfy the skewness criterion, the mesh is locally updated with the new cells (with the solution interpolated from the old cells). Otherwise, the new cells are discarded and the old cells are retained.

When defining the remeshing settings for your dynamic zones, it is recommended that you use unified remeshing (the default), which applies an algorithm that combines aspects of a variety of remeshing methods and by default attempts to maintain the initial mesh size distribution even as

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the mesh moves. This simplifies the setup and can provide increased robustness, especially for parallel simulations. It should be noted that such unified remeshing is applied to triangular or tetrahedral cells and can produce wedge cells in 3D boundary layer meshes. As an alternative, you can use methods-based remeshing, which allows you to selectively enable the following options, each of which is suitable for particular kinds of cell types:

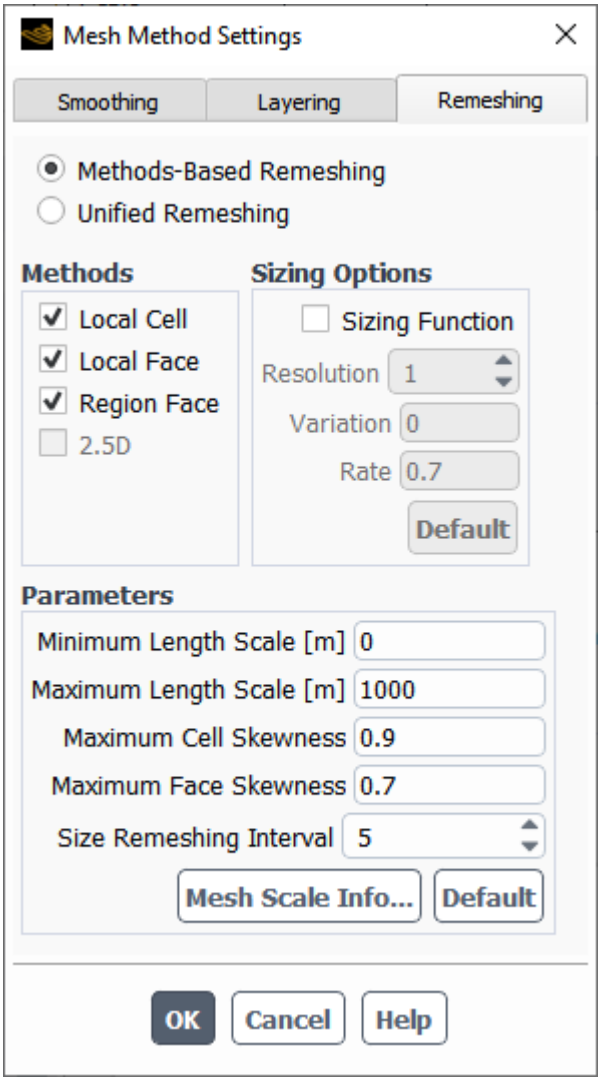
- The local cell remeshing method only affects triangular and tetrahedral cell types in the mesh (that is, in mixed cell zones the non-triangular/tetrahedral cells are skipped).
- The local face remeshing method is available in 3D only and can remesh tetrahedral cells and wedge cells in boundary layer meshes.
- The zone remeshing method replaces all cell types with triangular and tetrahedral cells (in 2D and 3D domains, respectively), and can remesh and produce wedge cells in 3D boundary layer meshes.
- The face region remeshing method is applied to triangular cells in 2D, and tetrahedral cells in 3D. In 3D domains, the face region remeshing method can also remesh and produce wedge cells in 3D boundary layer meshes.
- The 2.5D remeshing method only works on hexagonal meshes or wedge cells extruded from triangular surface elements.

To apply remeshing, enable the **Remeshing** option under **Mesh Methods** in the Dynamic Mesh Task Page (p. 5111) (Figure 13.13: The Dynamic Mesh Task Page (p. 1815)). Click the **Settings...** button to open the **Mesh Method Settings** dialog box, where you can select either **Methods-Based Remeshing** or **Unified Remeshing** in the **Remeshing** tab (Figure 13.35: The Remeshing Tab in the Mesh Method Settings Dialog Box for Methods-Based Remeshing (p. 1843)) and revise the options / parameters or the advanced settings.

For **Methods-Based Remeshing**, you can view the vital statistics of your mesh by clicking the **Mesh Scale Info...** button at the bottom of the Mesh Method Settings Dialog Box (p. 5114). This button opens the Mesh Scale Info Dialog Box (p. 5121), where you can view the minimum and maximum length scale values, as well as the maximum cell and face skewness values.

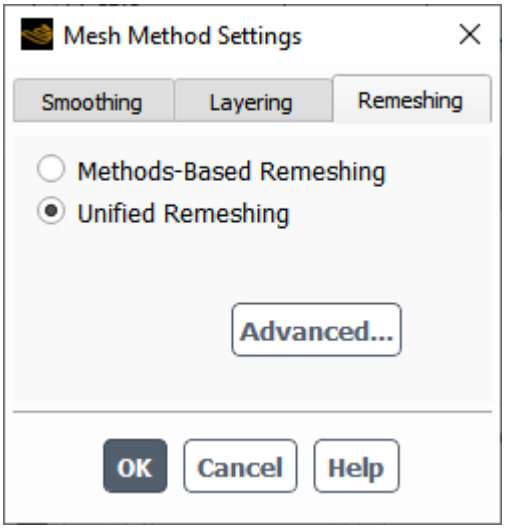
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.35: The Remeshing Tab in the Mesh Method Settings Dialog Box for Methods-Based Remeshing



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.36: The Remeshing Tab in the Mesh Method Settings Dialog Box for Unified Remeshing



For details, see the following sections:

13.6.2.3.1. Methods-Based Remeshing

13.6.2.3.2. Unified Remeshing

13.6.2.3.1. Methods-Based Remeshing

When **Methods-Based Remeshing** is selected in the **Remeshing** tab of the **Mesh Method Settings** dialog box, you can selectively enable the following methods:

13.6.2.3.1.1. Local Remeshing Method

13.6.2.3.1.2. Cell Zone Remeshing Method

13.6.2.3.1.3. Face Region Remeshing Method

13.6.2.3.1.4. 2.5D Surface Remeshing Method

13.6.2.3.1.1. Local Remeshing Method

Using the local remeshing method (that is, local cell remeshing, with or without local face remeshing), Ansys Fluent marks cells based on cell skewness and minimum and maximum length scales as well as an optional sizing function.

Ansys Fluent evaluates each cell and marks it for remeshing if it meets one or more of the following criteria:

- It has a skewness that is greater than a specified maximum skewness.
- It is smaller than a specified minimum length scale.
- It is larger than a specified maximum length scale.
- Its height does not meet the specified length scale (at moving face zones, for example, above a moving piston).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If local remeshing is not able to reduce the maximum cell skewness sufficiently, then the cell zone remeshing method is used to automatically remesh all of the cells in the cell zone, as well as the faces of all adjacent deforming dynamic face zones (see Cell Zone Remeshing Method (p. 1852) for details). The maximum allowable cell skewness is set to be 0.98 by default. The cell zone remeshing method gives the mesher more flexibility to create a new mesh of better quality than the local cell remeshing method. The automatic remeshing of cell zones can be disabled using the following text command:

`define` → `dynamic-mesh` → `controls` → `remeshing-parameter` → `zone-remeshing`

13.6.2.3.1.1.1. Local Cell Remeshing Method

As previously mentioned, in local cell remeshing, Ansys Fluent agglomerates cells based on skewness, size, and height (adjacent moving face zones) prior to the movement of the boundary. The size criteria are specified with **Minimum Length Scale** and **Maximum Length Scale**. Cells with length scales below the minimum length scale and above the maximum length scale are marked for remeshing. The value of **Maximum Cell Skewness** indicates the desired skewness of the mesh. By default, the **Maximum Cell Skewness** is set to 0.9 for 3D simulations and 0.7 for 2D simulations. Cells with skewness above the maximum skewness are marked for remeshing.

The marking of cells based on skewness is done at every time step when the local remeshing method is enabled. However, marking based on size and height is performed between the specified **Size Remeshing Interval** since the change in cell size distribution is typically small over one time step.

Note that you should edit the fields in the **Parameters** group box, as the initial values are unlikely to produce your desired outcome. Clicking **Default** will define values that are based on your particular mesh and that should represent a reasonable starting point, and you can then adjust them as needed. After you click the **Default** button, it will then change to be a **Reset** button, which can be used to return to the prior values.

By default, Ansys Fluent replaces the agglomerated cells only if the quality of the remeshed cells has improved.

Note that you cannot use the local cell remeshing method in conjunction with adaption. For more information on the available adaption methods, see Hanging Node Adaption and Polyhedral Unstructured Mesh Adaption in the Theory Guide.

13.6.2.3.1.1.2. Local Face Remeshing Method

The local face remeshing method only applies to 3D geometries. You can apply this method to the boundaries of deforming or user-defined zone types, so that Ansys Fluent will mark faces (and adjacent cells) based on the face skewness, and then remesh them.

Local face remeshing also allows the remeshing of wedge cells in boundary layers at those boundaries. The detection of boundary layers (as well as the wedge element height distribution and number of layers) is automatic and does not require your input.

To apply local face remeshing, perform the following steps:

1. Enable the **Local Face** option in the **Remeshing** tab of the Mesh Method Settings Dialog Box (p. 5114), and set the **Maximum Face Skewness** to an appropriate global value.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. For each deforming or user-defined boundary zone on which you want local face remeshing applied, enable the **Local** option in the **Meshing Options** tab of the Dynamic Mesh Zones Dialog Box (p. 5140). You can also revise the **Maximum Skewness** in this tab if you don't want to use the global setting for a particular zone. Note that for deforming zones, the **Remeshing** option must be enabled—which is the default condition—in order to access the **Local** option, and the **Global Settings** option must be disabled to revise the **Maximum Skewness**. For further details on the zone setup, see Deforming Motion (p. 1910) and/or User-Defined Motion (p. 1913).

13.6.2.3.1.1.2.1. Applicability of the Local Face Remeshing Method

When you enable local face remeshing for a boundary, the faces can be remeshed only if the following conditions are met:

- The faces are triangular.
- The agglomerated faces do not span multiple zones or feature edges.

Note that you cannot use the local face remeshing method in conjunction with adaption. For more information on the available adaption methods, see Hanging Node Adaption and Polyhedral Unstructured Mesh Adaption in the Theory Guide.

13.6.2.3.1.1.3. Local Remeshing Based on Sizing Function

Instead of marking cells based on minimum and maximum length scales, Ansys Fluent also marks cells based on the size distribution generated by the sizing function if the **Sizing Function** option is enabled.

Cells can be marked using sizing functions only with the following remeshing methods:

- local cell remeshing
- 2.5D surface remeshing (as described in 2.5D Surface Remeshing Method (p. 1857))

Figure 13.38: Mesh at the End of a Dynamic Mesh Simulation With Sizing Functions (p. 1847) demonstrates the advantages of using sizing functions for local remeshing.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.37: Mesh at the End of a Dynamic Mesh Simulation Without Sizing Functions

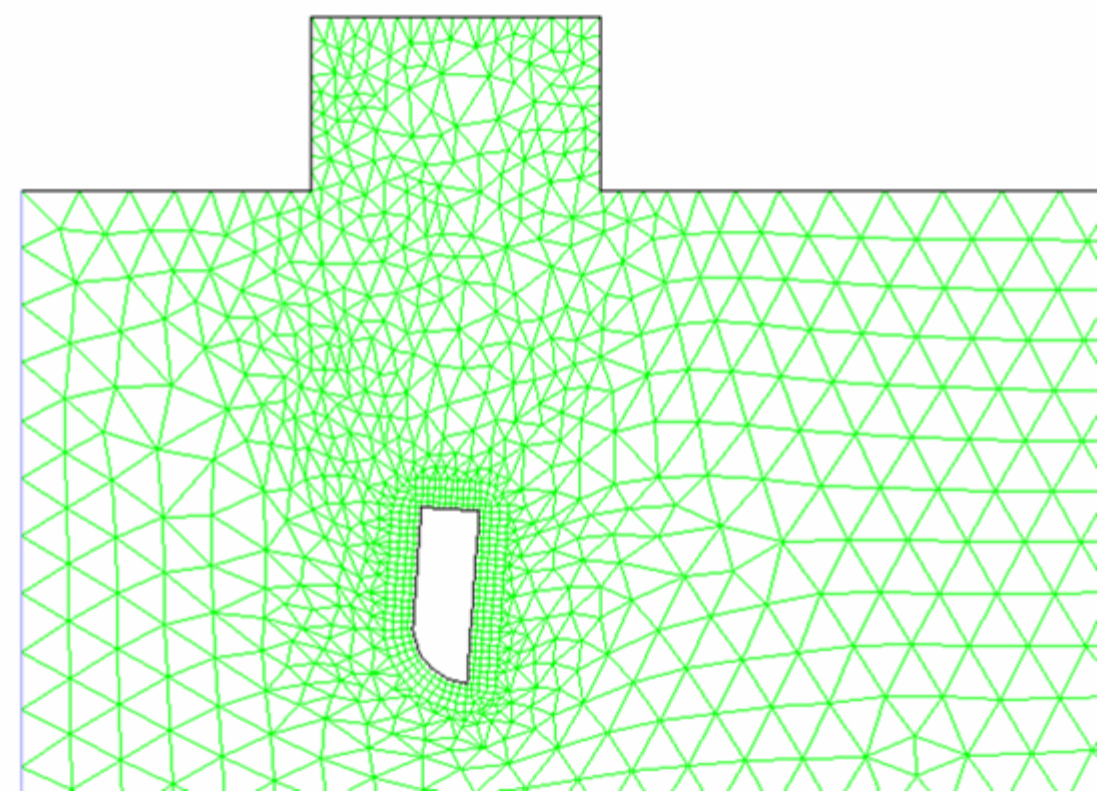
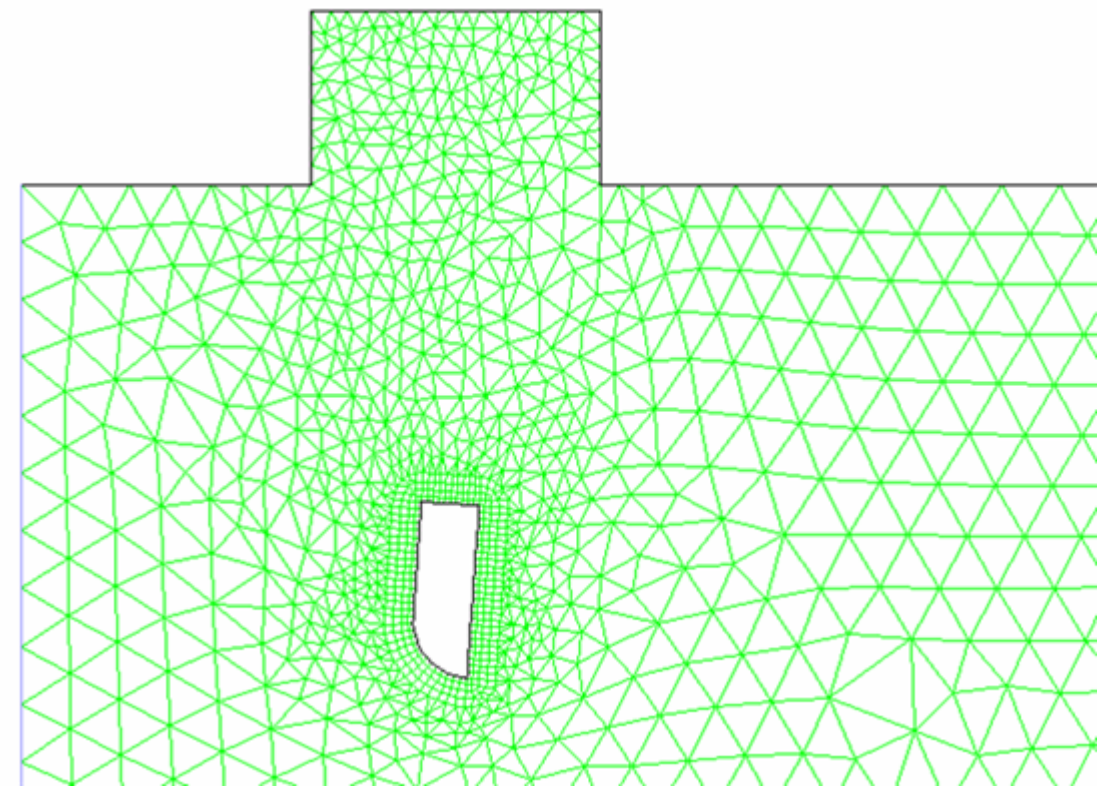


Figure 13.38: Mesh at the End of a Dynamic Mesh Simulation With Sizing Functions

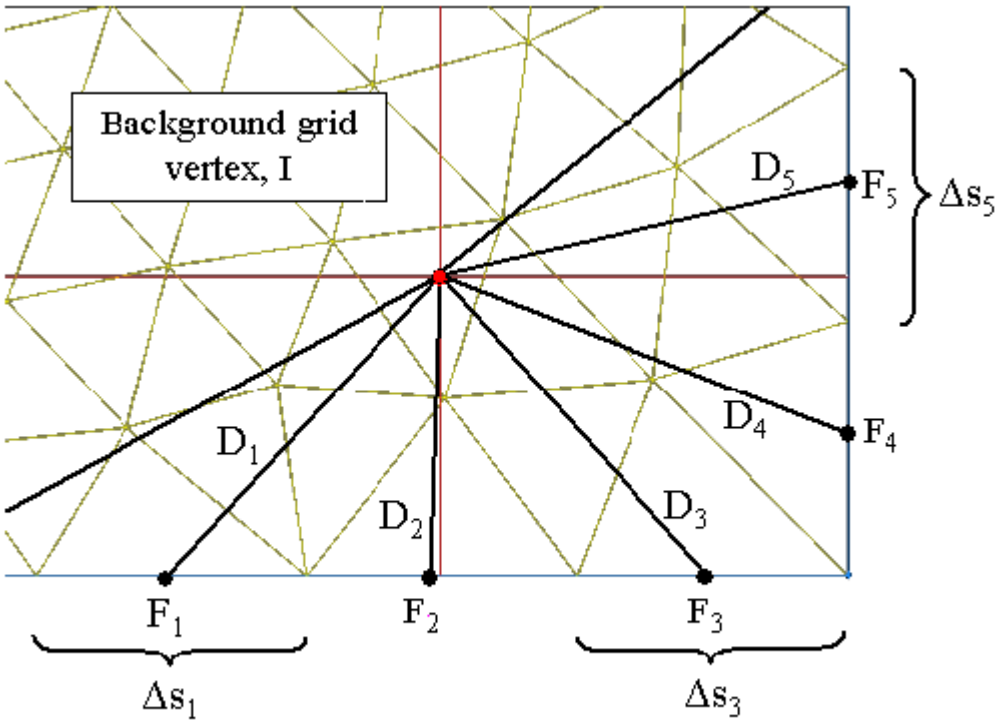


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In determining the sizing function, Ansys Fluent draws a bounding box around the zone that is approximately twice the size of the zone, and locates the shortest feature length within each fluid zone. Ansys Fluent then subdivides the bounding box based on the shortest feature length and the sizing function **Resolution** that you specify. This allows Ansys Fluent to create a background mesh.

You control the resolution of the background mesh and a background mesh is created for each fluid zone. The shortest feature length is determined by shrinking a second box around the object, and then selecting the shortest edge on that box. The sizing function is evaluated at the vertex of each individual background mesh.

Figure 13.39: Sizing Function Determination at Background Mesh Vertex I



As seen in Figure 13.39: Sizing Function Determination at Background Mesh Vertex I (p. 1848), the local value of the sizing function SF_I is defined by

$$SF_I = \left(\frac{\Sigma \frac{1}{D_J} \Delta s_J}{\Sigma \frac{1}{D_J}} \right) \tag{13.17}$$

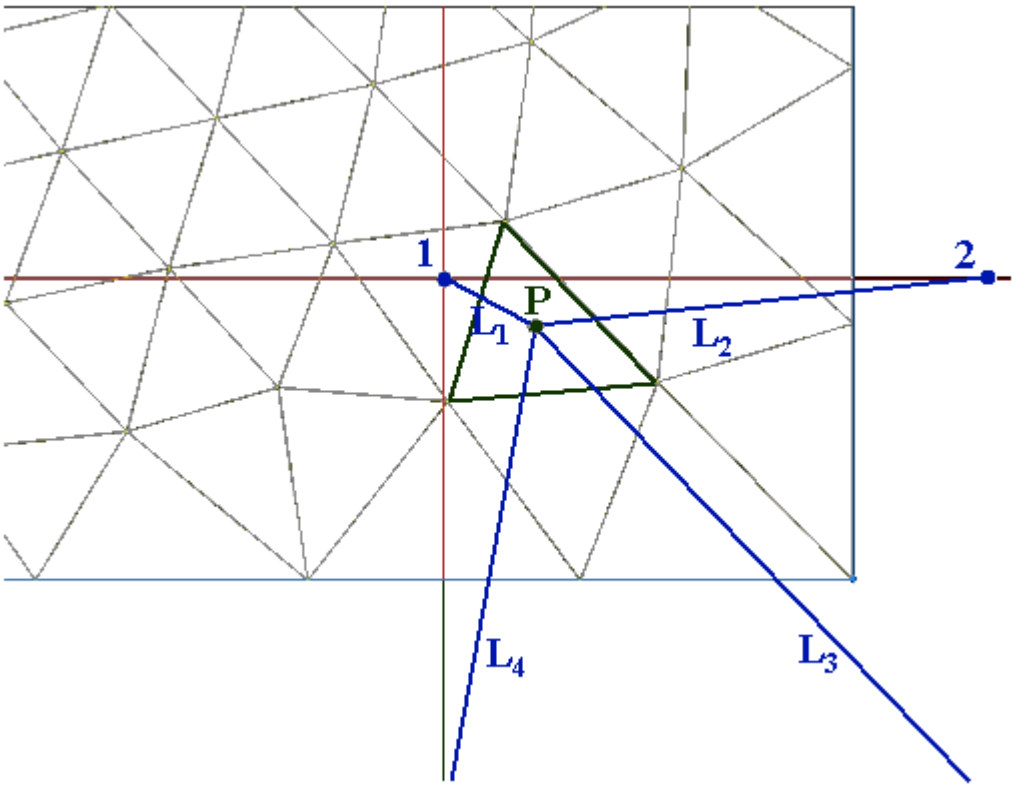
where D_J is the distance from vertex I on the background mesh to the centroid of boundary cell J and Δs_J is the mesh size (length) of boundary cell J.

The sizing function is then smoothed using Laplacian smoothing. Ansys Fluent then interpolates the value of the sizing function by calculating the distance L_I from a given cell centroid P to the background mesh vertices that surround the cell (see Figure 13.40: Interpolating the Value of the Sizing Function (p. 1849)). The intermediate value of the sizing function $size_b$ at the centroid is computed from

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$size_b = \left(\frac{\Sigma SF_I \frac{1}{L_I}}{\Sigma \frac{1}{L_I}} \right) \tag{13.18}$$

Figure 13.40: Interpolating the Value of the Sizing Function

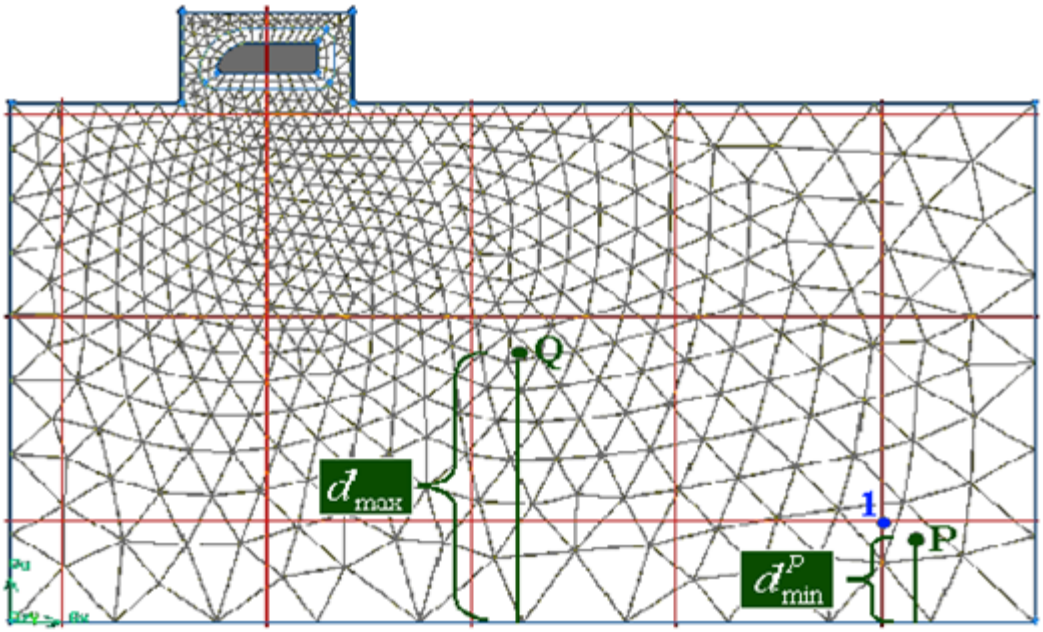


Next, a single point Q is located within the domain (see Figure 13.41: Determining the Normalized Distance (p. 1850)) that has the largest distance d_{max} to the nearest boundary to it. The normalized distance d_b for the given centroid P is given by

$$d_b = \frac{d^P_{min}}{d_{max}} \tag{13.19}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.41: Determining the Normalized Distance



Using the parameters α and β (the sizing function **Variation** and the sizing function **Rate**, respectively), you can write the final value $size_P$ of the sizing function at point P as

$$size_P = size_b \times \left(1 + \alpha \times d_b^{1+2\beta}\right) = size_b \times \gamma \tag{13.20}$$

where $size_b$ is the intermediate value of the sizing function at the cell centroid.

Note that α is the sizing function *variation*. Positive values mean that the cell size increases as you move away from the boundary. Since the maximum value of d_b is one, the maximum cell size becomes

$$size_{P,max} = size_b \times \left(1 + \alpha\right) = size_b \times \gamma_{max} \tag{13.21}$$

therefore, α is really a measure of the maximum cell size.

The factor γ is computed from

$$\gamma = 1 + \alpha d_b^{1+2\beta} \quad \text{if } \alpha > 0 \tag{13.22}$$

$$\gamma = 1 + \alpha d_b^{\frac{1}{1-\beta}} \quad \text{if } \alpha < 0 \tag{13.23}$$

You can use sizing function **Variation** (or α) to control how large or small an interior cell can be with respect to its closest boundary cell. α ranges from -1 to ∞, an α value of 0.5 indicates that the interior cell size can be, at most, 1.5 the size of the closest boundary cell. Conversely, an α value of -0.5 indicates that the cell size interior of the boundary can be half of that at the closest boundary cell. A value of 0 indicates a constant size distribution away from the boundary.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can use the sizing function **Rate** (or β) to control how rapidly the cell size varies from the boundary. The value of β should be specified such that $-0.99<\beta<+0.99$. A positive value indicates a slower transition from the boundary to the specified sizing function **Variation** value. Conversely, a negative value indicates a faster transition from the boundary to the sizing function **Variation** value. A value of 0 indicates a linear variation of cell size away from the boundary.

You can also control the **Resolution** of the sizing function. This determines the size of the background bins used to evaluate the size distribution with respect to the shortest feature length of the current mesh. By default, the sizing function **Resolution** is 3 in 2D problems, and 1 in 3D problems.

Note that if you edit the **Resolution**, **Variation**, and/or **Rate**, clicking **Default** will return these fields to the default values based on the current mesh. The **Default** button will then change to be a **Reset** button, which can be used to return to the prior values.

In summary, the sizing function is a distance-weighted average of all mesh sizes on all boundary faces (both stationary and moving boundaries). The sizing function is based on the sizes of the boundary cells, with the size computed from the cell volume by assuming a perfect (equilateral) triangle in 2D and a perfect tetrahedron in 3D. You can control the size distribution by specifying the sizing function **Variation** and the sizing function **Rate**. If you have enabled the **Sizing Function** option, Ansys Fluent will agglomerate a cell if

$$size \notin \left[\frac{4}{5}\gamma size_b, \frac{5}{4}\gamma size_b\right] \tag{13.24}$$

where γ is a factor defined by Equation 13.22 (p. 1850) and Equation 13.23 (p. 1850).

Note that the sizing function is only used for marking cells *before* remeshing. The sizing function is not used to govern the size of the cell during remeshing.

For steady-state applications (see Steady-State Dynamic Mesh Applications (p. 1924)), you can instruct Ansys Fluent to perform a second round of cell marking and agglomeration after the boundary has moved, based on skewness criteria. The intent is to further improve the mesh quality through additional local remeshing. This optional feature works in conjunction with the **Mesh Method Settings** dialog box (Figure 13.35: The Remeshing Tab in the Mesh Method Settings Dialog Box for Methods-Based Remeshing (p. 1843)), and operates according to the skewness parameters you set in this dialog box. The sizing function parameters are not considered during this additional remeshing. Note that enabling this option will increase the time required to update the mesh during the solution.

Note:

If you want to employ additional local remeshing, first make sure that you have enabled the **Remeshing** option in the Dynamic Mesh Task Page (p. 5111).

To enable the additional round of local remeshing, use the following text command:

`define → dynamic-mesh → controls → remeshing-parameter → remeshing-after-moving?`

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.6.2.3.1.2. Cell Zone Remeshing Method

The cell zone remeshing method allows for the remeshing of the complete cell zone, and provides the option to also remesh the faces of all adjacent deforming dynamic face zones. This remeshing method is enabled by default when local cell remeshing is enabled, and is performed automatically if the local cell remeshing does not produce an acceptable mesh (see Local Remeshing Method (p. 1844) for the acceptability criteria). Cell zone remeshing can also be manually invoked, using the following text command:

`define` → `dynamic-mesh` → `actions` → `remesh-cell-zone`

The cell zone remeshing method is available for triangular cells in 2D meshes and tetrahedral cells in 3D meshes. For 3D meshes, the method also allows the remeshing of wedge cells in boundary layers. The detection of the boundary layers (as well as the wedge element height distribution and number of layers) is automatically performed by default, and allows for different layer parameters on each boundary zone. You can manually specify the parameters by entering nonzero values for first height, growth rate, and number of layers via the text commands available in the `define/dynamic-mesh/controls/remeshing-parameters/prism-layer-parameters` menu, although generally it is not necessary to do this. When you enter them manually, the boundary layer parameters are global: they apply to every prism layer detected in the remeshing zone.

Note that it is necessary to enable the dynamic mesh model and the remeshing method in order to attain access to the prism layer controls, even if the cell zone is remeshed manually and not as part of a dynamic mesh update.

13.6.2.3.1.2.1. Limitations of the Cell Zone Remeshing Method

Zone remeshing has the following limitations:

- Only triangular, tetrahedral, and wedge cell types (when the wedge cells are part of a 3D boundary layer mesh) are remeshed.
- Zones with hanging nodes cannot be remeshed.

13.6.2.3.1.3. Face Region Remeshing Method

The face region remeshing method allows for the remeshing of those triangular faces (in 3D meshes) and linear faces (in 2D meshes) that are on a deforming face zone and adjacent to a moving face zone (see layer j in Figure 13.42: Expanding Cylinder Before Region Face Remeshing (p. 1853)). Ansys Fluent marks the faces based on minimum and maximum length scales, and then remeshes the faces and the associated cells to produce a very regular mesh on the deforming boundary. Although primarily designed for in-cylinder type configurations, where the remeshing region is located where cylinder walls meet the moving piston, face region remeshing can be used for all applications where a moving dynamic zone abuts deforming dynamic face zones.

For 3D simulations, Ansys Fluent allows face region remeshing with symmetric boundary conditions and across multiple face zones. The remeshing can preserve features not only between the different deforming face zones, but also within a face zone. For more information on feature preservation, see Feature Detection (p. 1868). Ansys Fluent also allows face region remeshing of tetrahedral cell zones that contain wedge cells in boundary layers, as described in the section that follows.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To begin marking the faces for face region remeshing, Ansys Fluent identifies the nodes at the intersection of a moving dynamic zone and the adjacent deforming zones. Ansys Fluent then analyzes the height of the faces on the deforming zones that are in the range of the identified nodes, and then remeshes the faces depending on the specified maximum or minimum length scale.

Consider the simple tetrahedral mesh of a cylinder that has a moving end wall (see Figure 13.42: Expanding Cylinder Before Region Face Remeshing (p. 1853)). The faces that are subject to remeshing are in layer j of the side wall. If the faces in layer j are expanding, the expansion continues until the height h reaches the maximum length scale, and then the layer is remeshed to form 2 layers of elements (see Figure 13.43: Expanding Cylinder After Region Face Remeshing (p. 1854)). Conversely, if the faces of layer j are contracting, the contraction continues until h reaches the minimum length scale, and then layer j and the neighboring layer of faces (layer i) on the deforming zone are remeshed to form a single layer.

Figure 13.42: Expanding Cylinder Before Region Face Remeshing

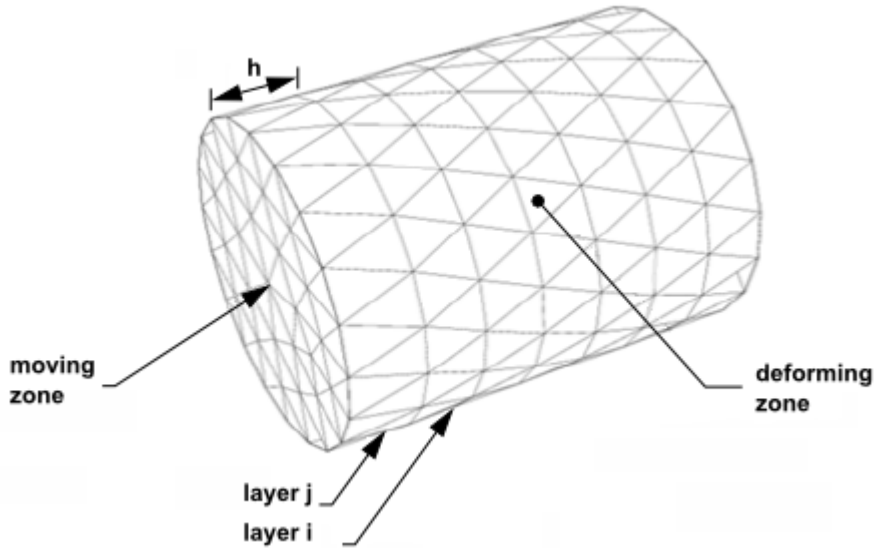
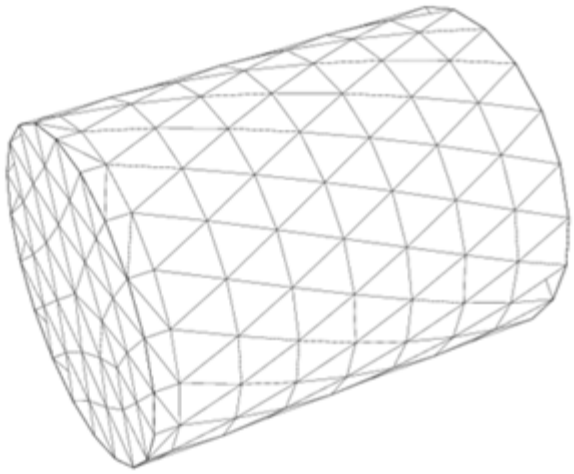


Figure 13.43: Expanding Cylinder After Region Face Remeshing



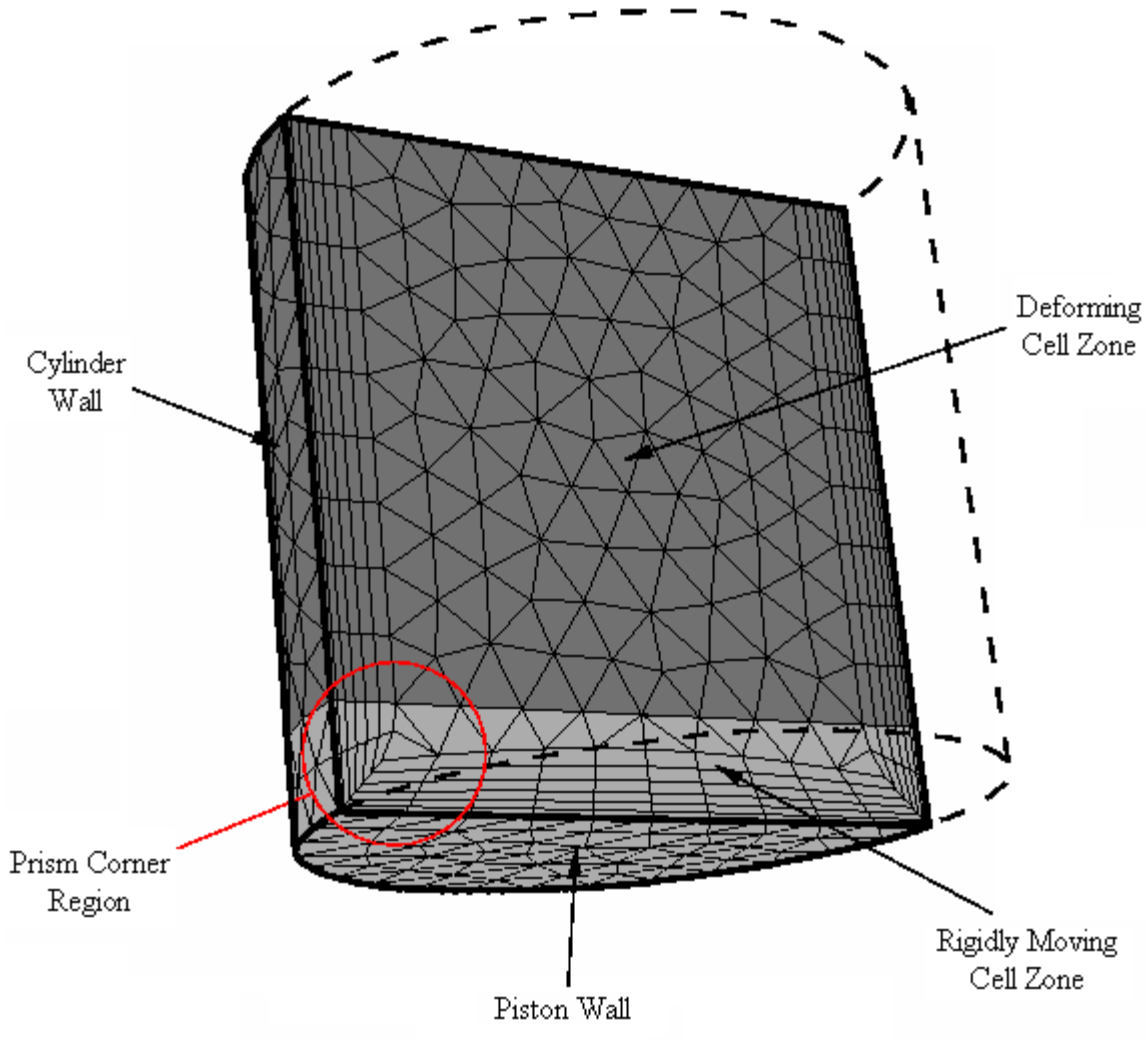
13.6.2.3.1.3.1. Face Region Remeshing with Wedge Cells in Prism Layers

In 3D simulations, the face region remeshing method can be applied on meshes that have wedge cells along the deforming face zones. When remeshing the faces on the deforming face zones, the associated wedge cells are remeshed as well. The layer parameters are based on the existing mesh by default; you have the option of manually setting these parameters, as described later in this section.

If the motion of the face zone is large compared to the height of the adjacent deforming face zones, it is recommended that you decompose the mesh volume in such a way as to create a dynamic cell zone that moves as a rigid body in between the moving face zone and the deforming dynamic cell zone on which face region remeshing is applied. Consider Figure 13.44: Volume Decomposition for Prism Layers (p. 1855), which displays only half of an in-cylinder mesh. The rigidly moving cell zone encapsulates the prism layers on the moving piston so that the layers are not remeshed, and therefore the risk of generating degenerate cells during the mesh motion update is reduced.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.44: Volume Decomposition for Prism Layers

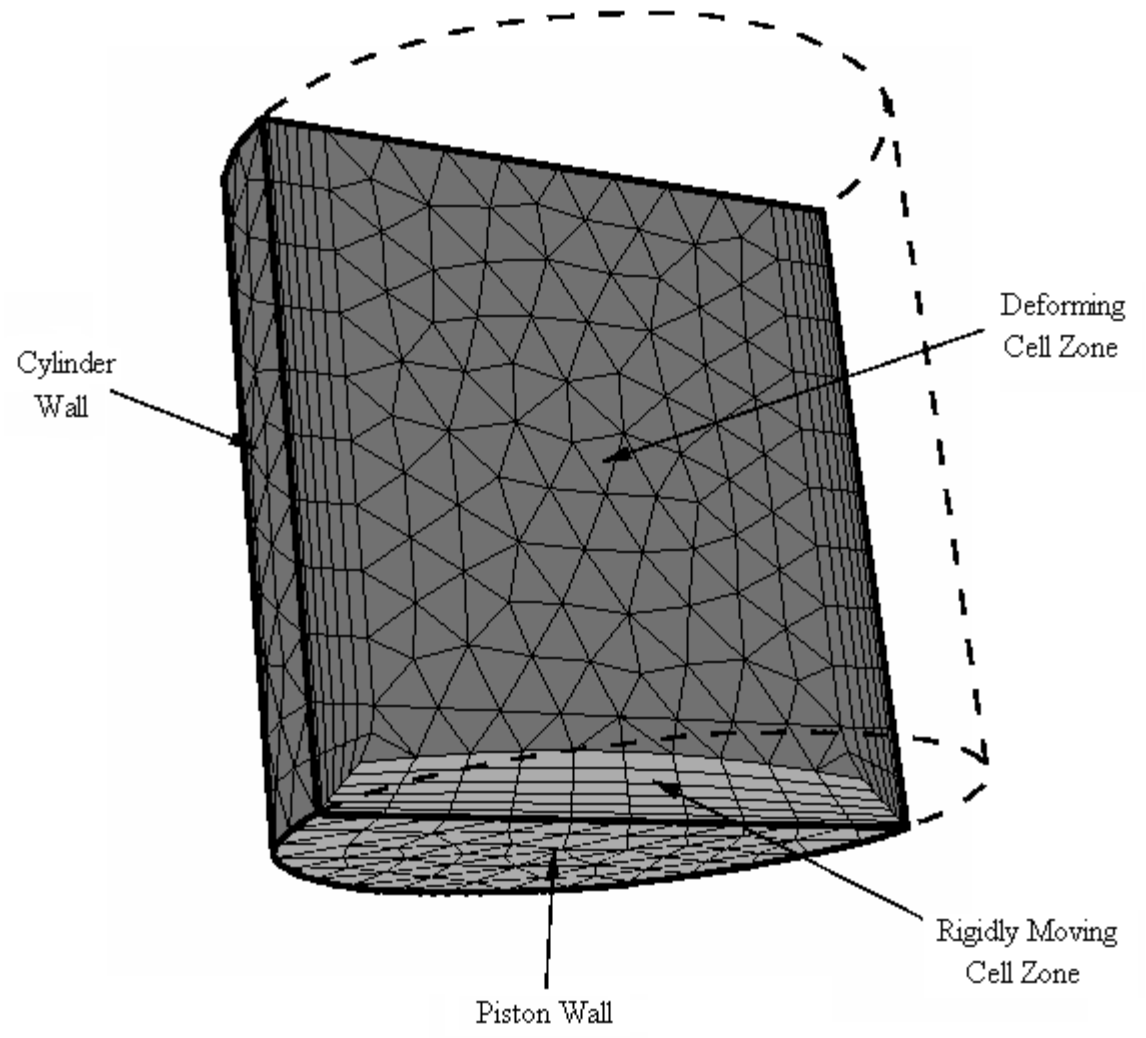


It is preferable (and even mandatory, if the mesh is a half model with a symmetry plane) to decompose the volume such that the "corner" region of the prism layers (shown in the previous figure) exists entirely within the rigidly moving zone. This allows for the largest deformations without risking degenerate elements, because the prism normals of the remeshed cells are uniformly perpendicular to the faces undergoing remeshing.

If the range of motion does not allow you to encapsulate the entire corner region of the prism layers in a rigidly moving zone, it is recommended that you encapsulate the "base" of the prism layers (shown in Figure 13.45: Volume Decomposition for the Base of the Prism Layers (p. 1856)) and move these cells with a rigid body motion. Although this is less ideal than encapsulating the corner region, it does reduce the risk of degenerate mesh elements.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.45: Volume Decomposition for the Base of the Prism Layers



For piston-type applications that contain prism layers, a reasonable rule of thumb is that you should decompose the mesh volume if the piston motion is more than half the cylinder height. If you decide not to decompose the volume at all, you must at the very least enable the **Deform Adjacent Boundary Layer with Zone** option in the **Meshing Options** tab of the **Dynamic Mesh Zone** dialog box when setting up the moving face zone (see Rigid Body Motion (p. 1906) for details). In any case, it is recommended that you always preview the mesh motion over the complete simulation time, to make sure that you will have a valid mesh at each time step.

The prism layer parameters (that is, element height, growth rate, and number of layers) are extracted automatically from the mesh and do not generally require your input. To prevent the prism parameters from drifting due to repeated remeshing, the prism parameters for first height, growth rate, and number of layers can be entered manually, using the text commands available in the `define/dynamic-mesh/controls/remeshing-parameters/prism-layer-parameters` menu.

13.6.2.3.1.3.2. Applicability of the Face Region Remeshing Method

Note the following limitations associated with face region remeshing:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- You can use the face region remeshing method only in cell zones that contain triangular cells (in 2D) or tetrahedral cells, with or without prism layers (in 3D). For 3D meshes, the faces on the deforming boundaries that border the moving face must all be triangular.
- The face region remeshing method is not compatible with diffusion-based smoothing or linearly elastic solid smoothing.
- You cannot use the face region remeshing method in conjunction with adaption. For more information on the available adaption methods, see Hanging Node Adaption and Polyhedral Unstructured Mesh Adaption in the Theory Guide.

13.6.2.3.1.4. 2.5D Surface Remeshing Method

The 2.5D surface remeshing method only applies to extruded 3D geometries and is similar to local remeshing in two dimensions on a triangular surface mesh (not a mixed zone). Faces on a deforming boundary are marked for remeshing based on face skewness, minimum and maximum length scale; the 2.5D remeshing method also gives you the option of marking cells using sizing functions, as described in Local Remeshing Based on Sizing Function (p. 1846).

Figure 13.46: Close-Up of 2.5D Extruded Flow Meter Pump Geometry Before Remeshing and Laplacian Smoothing

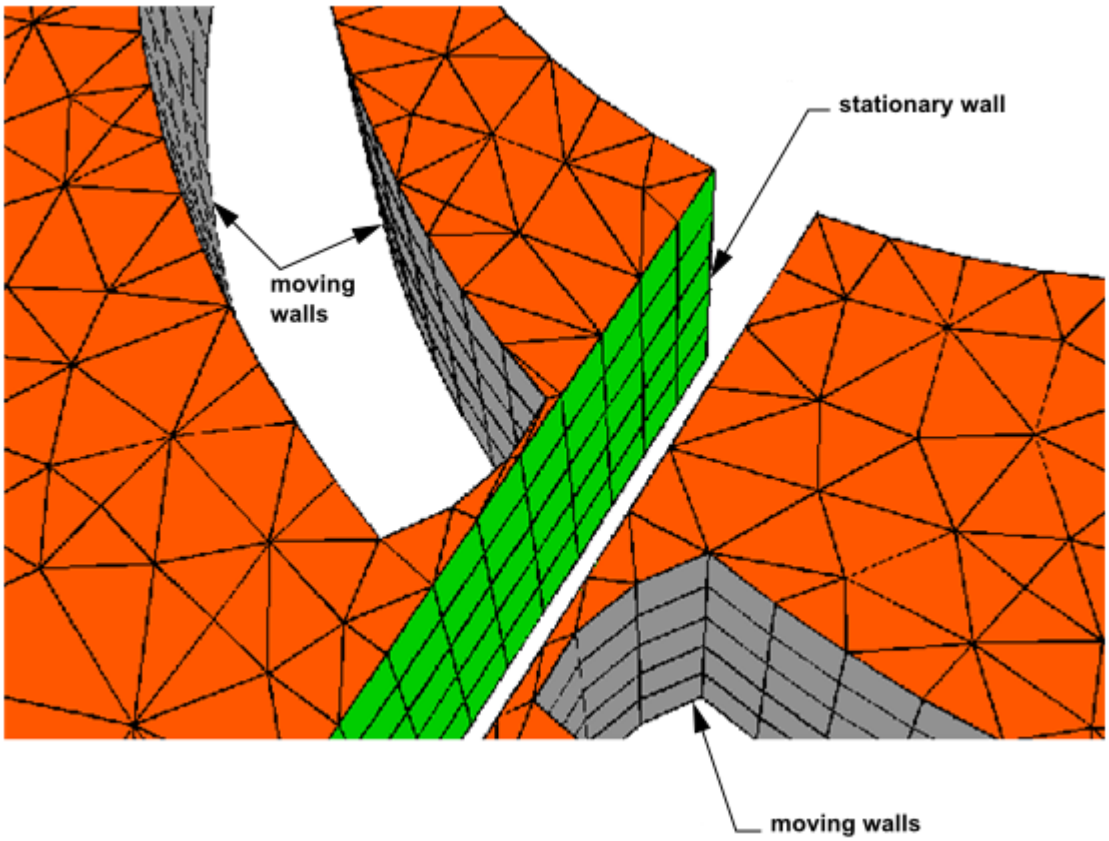
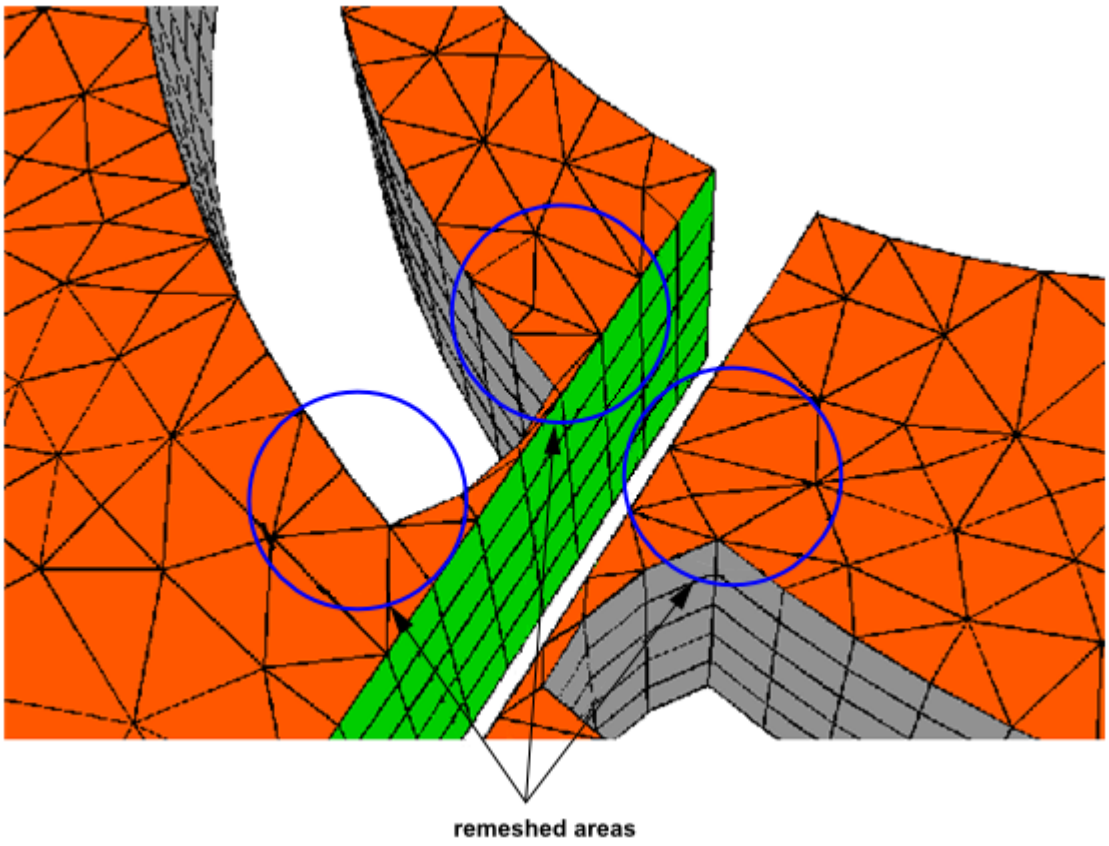


Figure 13.47: Close-Up of 2.5D Extruded Flow Meter Pump Geometry After Remeshing and Laplacian Smoothing



13.6.2.3.1.4.1. Applicability of the 2.5D Surface Remeshing Method

The following applies to the 2.5D surface remeshing method:

- Triangular faces get remeshed based on marking.
- Extruded wedges get remeshed based on the remeshing of the triangular face. Only extruded regions get remeshed, not mixed regions.
- The 2.5D remeshing method does not support remeshing or moving nodes on the perimeter of the extruded zone(s).
- The only smoothing method that is compatible with 2.5D remeshing is Laplacian smoothing.
- Note that you cannot use the 2.5D surface remeshing method in conjunction with adaption. For more information on the available adaption methods, see Hanging Node Adaption and Polyhedral Unstructured Mesh Adaption in the Theory Guide.
- The extrusion must be along a straight line normal to the deforming zone and the cross-section of the extruded/coopered mesh must be constant along the extrusion direction. For more information about the 2.5D model, see Using the 2.5D Model (p. 1859).
- Periodics are not supported at the extruded zones.

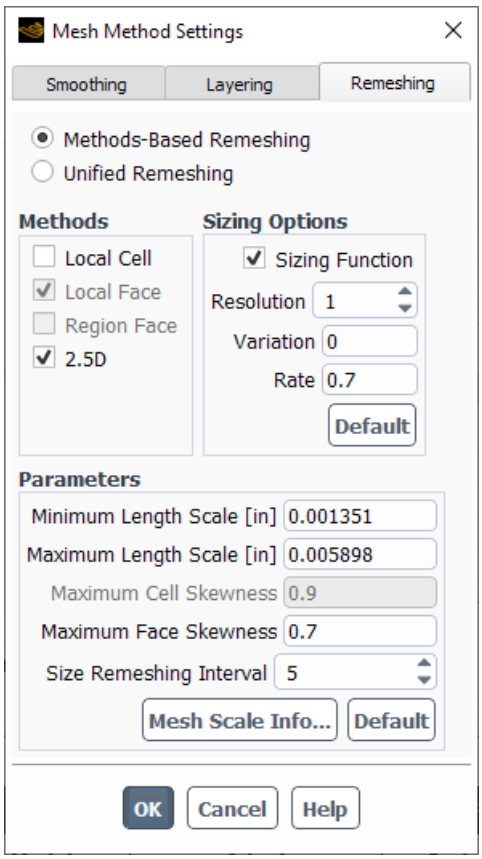
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.6.2.3.1.4.2. Using the 2.5D Model

For 3D simulations only, you can select the **2.5D** model under the **Remeshing** tab in the Mesh Method Settings Dialog Box (p. 5114). This model allows for a specific subset of remeshing techniques.

The 2.5D mesh essentially is a 2D triangular mesh which is expanded, or extruded, along the normal axis of the specific dynamic zone that you are interested in modeling. The triangular surface mesh is remeshed and smoothed on one side, and the changes are then extruded to the opposite side. Rigid body motion is applied to the moving face zones, while the triangular extrusion surface is assigned to a deforming zone with remeshing and smoothing enabled. The opposite side of the triangular mesh is assigned to be a deforming zone as well, with only smoothing enabled, as in Figure 13.49: 2.5D Extruded Gear Pump Geometry (p. 1860).

Figure 13.48: The Remeshing Tab for the 2.5D Model

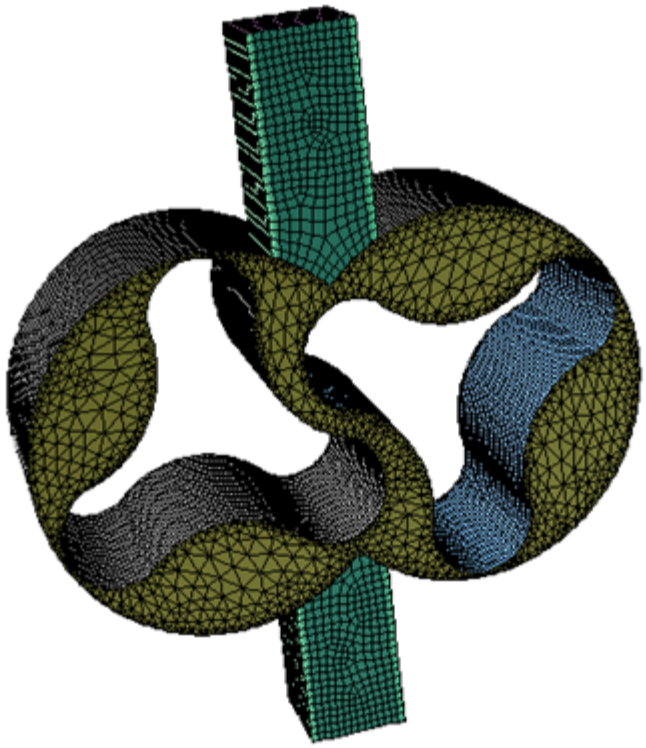


For more information on setting smoothing and remeshing parameters, see Dynamic Mesh Update Methods (p. 1816).

The 2.5D model only applies to mappable (that is, extrudable) mesh geometries such as pumps, as in Figure 13.49: 2.5D Extruded Gear Pump Geometry (p. 1860). Only the aspects of the geometry that represent the "moving parts" must be extruded in the mesh.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.49: 2.5D Extruded Gear Pump Geometry



Important:

You must only apply smoothing to the opposite side of the extruded mesh, since Ansys Fluent requires the geometry information for the dynamic zone. Ansys Fluent projects the nodes back to its geometry after the extrusion. Without this geometry information, the dynamic zones tends to lose its integrity.

Important:

In parallel, a partition method that partitions perpendicular to the extrusion surface should be used. For example, if the normal of the extrusion surface points in the X-direction then Cartesian-Y or Cartesian-Z would be the perfect partition methods.

The 2.5D model is used in combination with a `DEFINE_GRID_MOTION` UDF. (See Hooking `DEFINE_GRID_MOTION` UDFs in the Fluent Customization Manual for information about hooking this UDF.)

This UDF is associated with the extrusion surface that is adjacent to the cell zone, in turn applying the same deformation to the entire cell zone. This approach is particularly useful when modeling gear pumps that are predominantly extruded hexahedral meshes. For more information about this UDF, contact your support engineer.

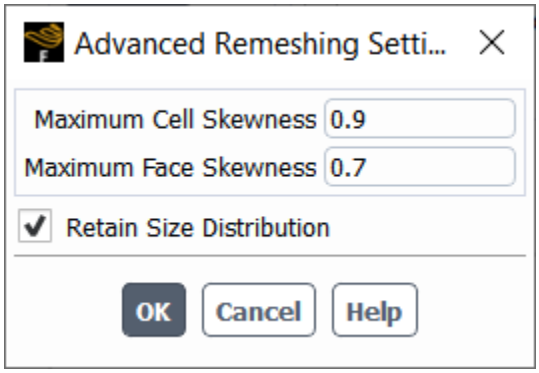
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.6.2.3.2. Unified Remeshing

When **Unified Remeshing** is selected in the **Remeshing** tab of the **Mesh Method Settings** dialog box, the mesh quality is improved using an algorithm that combines aspects of a variety of remeshing methods and by default attempts to maintain the initial mesh size distribution even as the mesh moves. **Unified Remeshing** simplifies the remeshing setup and can provide increased robustness compared to other methods, especially for parallel simulations.

If the default settings of the **Unified Remeshing** method do not yield a desirable mesh distribution for your simulation, you can attempt to improve it by clicking the **Advanced...** button and using the **Advanced Remeshing Settings** dialog box that opens. The value of the **Maximum Cell Skewness** sets the threshold for remeshing cells based on skewness, and by default is set to 0.9 for 3D simulations and 0.7 for 2D simulations. For 3D cases, faces (and adjacent cells) on the boundaries of deforming or user-defined zone types are remeshed based on the **Maximum Face Skewness**.

Figure 13.50: The Advanced Remeshing Settings Dialog Box



If necessary, you can disable the **Retain Size Distribution** option, so that algorithm does not attempt to respect the initial mesh distribution by using local size criteria, but instead marks cells for remeshing when their length scales are outside of the minimum and maximum length scale values of the cell zone in the initial mesh. Should these zonal values be unsuccessful, you can use the following text commands to specify other appropriate length scale values:

`define` → `dynamic-mesh` → `controls` → `remeshing-parameters` → `length-min`

and

`define` → `dynamic-mesh` → `controls` → `remeshing-parameters` → `length-max`

The mesh is evaluated at every time step, and the remeshing is applied only if the quality of the cells / faces has improved.

Note:

- Unified remeshing is applied to triangular or tetrahedral cells, and can produce wedge cells in 3D boundary layer meshes.
- You cannot use unified remeshing in conjunction with adaption. For more information on the available adaption methods, see Hanging Node Adaption and Polyhedral Unstructured Mesh Adaption in the Theory Guide.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You have the option of creating additional controls that can affect the size distribution of the cells throughout the mesh or just the wedge cells in boundary layers by using the `define/dynamic-mesh/controls/remeshing-parameters/sizing-controls` or `define/dynamic-mesh/controls/remeshing-parameters/prism-controls/` text command menus, respectively. It is generally not necessary to create such controls, but they may be helpful if you have strict sizing requirements or want more information when troubleshooting size-related issues. Note that any sizing controls and/or prism controls created in the meshing mode as part a **Watertight Geometry** meshing guided workflow (as described in Using the Watertight Geometry Meshing Guided Workflows (p. 293)) are automatically imported and used as part the **Unified Remeshing**. For further details, see the following sections:

13.6.2.3.2.1. Sizing Controls

13.6.2.3.2.2. Prism Controls

13.6.2.3.2.1. Sizing Controls

When the **Retain Size Distribution** option is enabled in the **Advanced Remeshing Settings** dialog box, you can use one or more sizing controls to modify the algorithm that attempts to respect the initial mesh distribution during unified remeshing. Each sizing control definition is applied to one or more boundary zones, and then affects the size of the cells throughout the mesh based on their distance from those boundary zone(s) and your settings in the definition. Any sizing controls created in the meshing mode as part a **Watertight Geometry** meshing guided workflow (as described in Using the Watertight Geometry Meshing Guided Workflows (p. 293)) are automatically imported and used; you can choose to edit and/or delete these, and/or create new controls.

> **Warning:**
>
> Sizing controls should be used with caution by experienced users only, as they could make your simulation less robust if used incorrectly.

To create and manage sizing controls, perform the following steps:

1. Define a sizing controls definition using the following text command:

   `define` → `dynamic-mesh` → `controls` → `remeshing-parameters` → `sizing-controls` → `add`

   After being prompted for a name, you can enter the following to complete the definition:

   - `zones`

     This specifies all of the boundary zones on which the sizing controls definition is applied.

   - `growth-rate`

     This sets the growth rate of the sizing controls definition.

   - `motion`

     This determines whether the evaluation of the sizing of the cells that result from the sizing control definition is updated when zones move: `auto` specifies that Fluent updates

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the evaluation during motion; and `static` specifies that the evaluation is not updated during motion.

- `type`

  This specifies how the sizing is affected by the selected boundary zones: `auto` specifies that the default size distribution (rather than the initial size distribution in your selected boundary zones) is used, along with your specified `max-length` and `min-length` values; `soft` specifies that the maximum length scale of your selected boundary zones is used, along with your specified `min-length` value; and `meshed` specifies that the maximum and minimum length scales of your selected boundary zones are used, in order to respect their initial size distribution. This setting is only relevant if you have more than one sizing controls definition.

- `max-length`

  This sets a maximum length threshold that is used when the `type` is set to `auto` or `soft`.

- `min-length`

  This sets a minimum length threshold that is used when the `type` is set to `auto`.

- `interval`

  This sets the interval (in time steps) at which the evaluation of the sizing of the cells that result from the sizing control definition is updated. This setting is only relevant when `auto` is selected for the `motion` setting. By default, the `interval` is set to `10` time steps. This setting does not typically need to be adjusted: you could decrease it if you are using large time steps (which is not recommended), and you could increase it if it negatively impacts the calculation speed.

- `name`

  You can use this to revise the name you entered when you created the sizing controls definition.

When the definition is complete, enter `q` to return to the text command menu.

2. You can repeat the previous step if you would like additional sizing controls definitions.

3. You can perform the following actions as needed prior to running the calculation:

   - To edit an existing sizing controls definition, use the following text command:

     `define` → `dynamic-mesh` → `controls` → `remeshing-parameters` → `sizing-controls` → `edit`

   - To delete an existing sizing controls definition, use the following text command:

     `define` → `dynamic-mesh` → `controls` → `remeshing-parameters` → `sizing-controls` → `delete`

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To print a list of the existing sizing controls definitions in the console, use the following text command:

  `define` → `dynamic-mesh` → `controls` → `remeshing-parameters` → `sizing-controls` → `list`

- To print the properties of an existing sizing controls definition in the console, use the following text command:

  `define` → `dynamic-mesh` → `controls` → `remeshing-parameters` → `sizing-controls` → `list-properties`

13.6.2.3.2.2. Prism Controls

When the **Retain Size Distribution** option is enabled in the **Advanced Remeshing Settings** dialog box, you can use one or more prism controls to modify the algorithm that attempts to respect the initial mesh distribution during unified remeshing. Each prism control definition is applied to one or more boundary zones, and then affects the height distribution and number of layers of the wedge cells in the adjacent boundary layers. Any prism controls created in the meshing mode as part a **Watertight Geometry** meshing guided workflow (as described in Using the Watertight Geometry Meshing Guided Workflows (p. 293)) are automatically imported and used; you can choose to edit and/or delete these, and/or create new controls.

> **Warning:**
>
> Prism controls should be used with caution by experienced users only, as they could make your simulation less robust if used incorrectly.

To create and manage prism controls, perform the following steps:

1. Define a prism controls definition using the following text command:

   `define` → `dynamic-mesh` → `controls` → `remeshing-parameters` → `prism-controls` → `add`

   After being prompted for a name, you can enter the following to complete the definition:

   - `zones`

     This specifies all of the boundary zones on which this prism controls definition is applied.

   - `nlayers`

     This sets the number of layers of wedge cells in the boundary layer adjacent to the specified `zones`.

   - `growth-method`

     This specifies the method used to determine the increase in height of the wedge cell layers beyond the first layer. The only available option is `geometric`, so that the height of each layer is the height of the previous layer multiplied by the `rate`.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- `rate`

  This sets the coefficient for the `growth-method` used to determine the increase in height of the wedge cell layers beyond the first layer.

- `offset-type`

  This allows you to select from the following types, in order to determine how the first layer of wedge cells (adjacent to the `zones`) are generated during remeshing:

  – `uniform`

    This specifies that every new node (child) of the first layer of wedge cells is the same distance away from its parent node (that is, the corresponding node on the boundary zone, from which the direction vector is pointing). This distance is defined by the `first-height`. The `uniform` type is selected by default.

  – `aspect-ratio`

    This specifies that the first layer of wedge cells that are extruded from the `zones` are defined using an aspect ratio, that is, the ratio of the local wedge base length (as averaged at the nodes) to the local wedge layer height. This aspect ratio is defined by the `first-aspect-ratio`. Note that for a uniform mesh, the heights of the cells in the first layer will be roughly the same, while for a varying mesh, these heights will vary.

  – `smooth-transition`

    This specifies that the last layer of wedge cells that are extruded from the `zones` are defined using a transition rate, that is, the ratio of the local wedge cell height (as averaged at the nodes) to the height of the local tetrahedral cell that will be generated adjacent to the last layer. This transition rate / ratio is defined by the `transition-rate`. This offset type allows you to easily provide a smooth transition for the cell volumes from the wedge cell layers to the cells deeper within the mesh. After the height of the last wedge cell layer is calculated, the height of the first layer is automatically computed according to your specified growth `rate` and `nlayers`. Note that for a uniform mesh, the heights of the cells in the first layer will be roughly the same, while for a varying mesh, these heights will vary.

  These offset types are similar to the offset method types available when adding boundary layers in a **Watertight Geometry** guided workflow in the meshing mode (as described in Adding Boundary Layers (p. 340)). For further details about how the `uniform` and `aspect-ratio` types affect the placement of the nodes during remeshing, see Offset Distances (p. 765).

- `first-height`

  This sets the height of the first layer of wedge cells in the boundary layer adjacent to the specified `zones`. It is used when `uniform` is selected for the `offset-type` (which is the default).

- `first-aspect-ratio`

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This sets the aspect ratio for the individual wedge cells in the first layer of cells adjacent to the specified `zones`. It is defined as the ratio of the local wedge base length (as averaged at the nodes) to the local wedge layer height. It is used when `aspect-ratio` is selected for the `offset-type`.

- `transition-rate`

  This sets the ratio of the local wedge cell height (as averaged at the nodes) in the last wedge cell layer to the height of the local tetrahedral cell that will be generated adjacent to the last layer (deeper within the mesh). It is used when `smooth-transition` is selected for the `offset-type`.

- `name`

  You can use this to revise the name you entered when you created the prism controls definition.

When the definition is complete, enter `q` to return to the text command menu.

2. You can repeat the previous step if you would like additional prism controls definitions.

3. You can perform the following actions as needed prior to running the calculation:

   - To edit an existing prism controls definition, use the following text command:

     `define` → `dynamic-mesh` → `controls` → `remeshing-parameters` → `prism-controls` → `edit`

   - To delete an existing prism controls definition, use the following text command:

     `define` → `dynamic-mesh` → `controls` → `remeshing-parameters` → `prism-controls` → `delete`

   - To print a list of the existing prism controls definitions in the console, use the following text command:

     `define` → `dynamic-mesh` → `controls` → `remeshing-parameters` → `prism-controls` → `list`

   - To print the properties of an existing prism controls definition in the console, use the following text command:

     `define` → `dynamic-mesh` → `controls` → `remeshing-parameters` → `prism-controls` → `list-properties`

13.6.2.4. Volume Mesh Update Procedure

The volume mesh is updated automatically based on the methods described in Dynamic Mesh Update Methods (p. 1816). Ansys Fluent decides which method to use for a particular zone based on which model is enabled and the shape of the cells in the zone. For example, if the boundaries of a tetrahedral cell zone are moving, the mesh smoothing and local remeshing methods will be used to update the volume mesh in this zone. If the zone consists of prismatic (hexahedral and/or wedge)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

cells, then the dynamic layer method will be used to determine where and when to insert and remove cell layers. On extruded prism zones, the 2.5D surface meshing method will be used.

Depending on which model is enabled, Ansys Fluent automatically determines which method to use by visiting the adjacent cell zones and setting appropriate flags for the volume mesh update methods to be used. If you specify the motion for a cell zone, Ansys Fluent will visit all of the neighboring cell zones and set the flags appropriately. If you specify the motion of a boundary zone, Ansys Fluent will analyze only the adjacent cell zones. If a cell zone does not have any moving boundaries, then no volume mesh update method will be applied to the zone.

Important:

Note that as a result of the remeshing procedures, updated meshes may be slightly different when dynamic meshes are used in parallel Ansys Fluent, and therefore very small differences may arise in the solutions.

Important:

Note that if your dynamic mesh model consists of numerous shell conduction zones, the mesh update may be very time consuming because all shells are deleted and recreated during the mesh update.

13.6.2.5. Transient Considerations for Remeshing and Layering

The second order in time transient formulation can be used with remeshing, layering, and smoothing cases. In such cases, second order in time accuracy is preserved during layering events and/or mesh smoothing. However, during remeshing events, the time advancement accuracy reverts to first order. Therefore, it is recommended that you only use second order in time for cases with very low remeshing frequency, that is, for cases where remeshing occurs every several time steps or more. If remeshing occurs every time step, the method reduces to a first order formulation. For cases that need very frequent remeshing, it is recommended that you retain the first order in time formulation. For details, see Dynamic Mesh Theory in the *Fluent Theory Guide*.

The switch to first order happens automatically when the solver detects a topology change as a result of remeshing within a time step. You will not see a change in the user interface, however you can monitor the switch between first and second order in time by using the following text command to enable the verbosity with dynamic meshes:

`define` → `dynamic-mesh` → `transient-settings` → `verbosity`

A message showing the active time formulation will print to the console when verbosity is set to `1`.

Note:

For cases with second order in time with dynamic mesh enabled, any mesh swapping and manual mesh manipulation will trigger a switch to first order in time.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

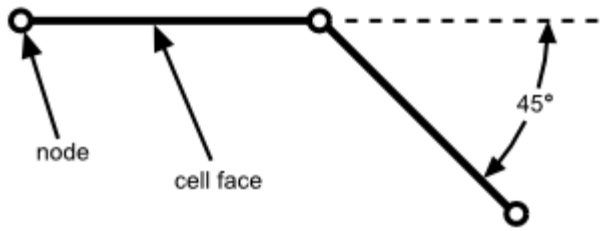
13.6.3. Feature Detection

The remeshing and/or smoothing of boundary zones may result in the chamfering of corners where cell faces meet. However, there may be some corners (identified by a particular angular range) that are important for your flow, and that you do not want chamfered. For 3D cases, you can specify that Ansys Fluent detects such corners (or "features") and preserves them. Such features may be at the juncture of different boundary zones, or they may be on a single non-planar boundary zone.

In the **Geometry Definition** tab of the Dynamic Mesh Zones Dialog Box (p. 5140) for any **Definition** type, you can indicate whether you want to preserve features by enabling **Feature Detection** under **Feature Options** and specifying a threshold value for the **Feature Angle**. A value of 180 degrees (the default) for the **Feature Angle** means that none of the corners must be preserved, whereas a value of 0 means that all of the corners must be preserved.

To illustrate the setting of the **Feature Angle**, see Figure 13.51: Cross Section of a 3D Corner (p. 1868). If you set the **Feature Angle** to 40 degrees, this corner would be guaranteed to be preserved. If you set it to 50 degrees, this corner could be chamfered.

Figure 13.51: Cross Section of a 3D Corner



13.6.3.1. Applicability of Feature Detection

The following items are applicable for use with feature detection:

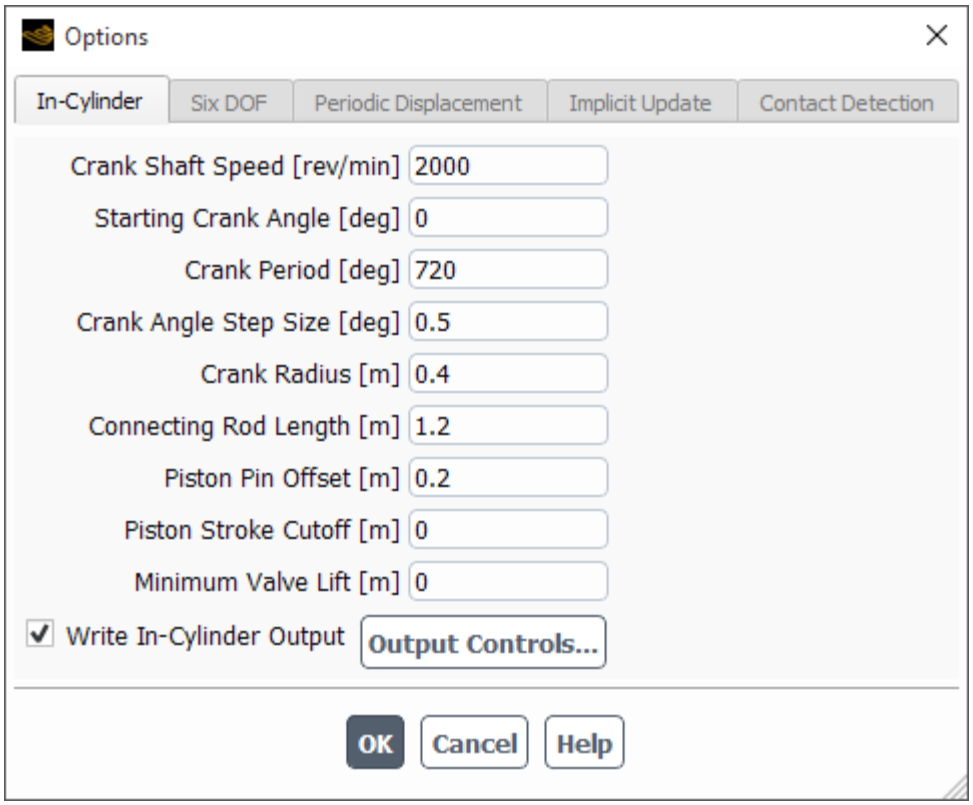
- Feature remeshing is only possible with face region remeshing.
- Features are preserved by local face remeshing, that is, there is no local face remeshing across features.
- Smoothing methods preserve features, that is, nodes at feature edges are not allowed to be smoothed.

13.6.4. In-Cylinder Settings

You can enable the **In-Cylinder** option in the Dynamic Mesh Task Page (p. 5111) (Figure 13.13: The Dynamic Mesh Task Page (p. 1815)) for transient problems. Then click the **Settings...** button in the **Options** group box to open the Options Dialog Box (p. 5121). Click the **In-Cylinder** tab and specify the **Crank Shaft Speed**, the **Starting Crank Angle**, and the **Crank Period**, which are used to convert between flow time and crank angle. You must also specify the time step size to use for advancing the solution in terms of crank angle in **Crank Angle Step Size**. By default, Ansys Fluent assumes a **Crank Angle Step Size** of 0.5 degrees.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

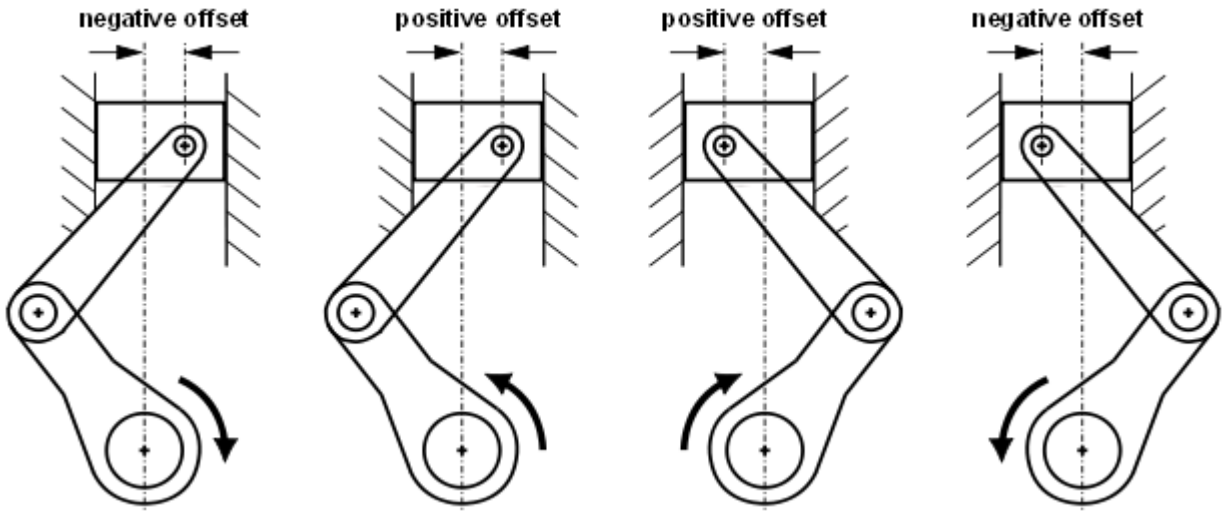
Figure 13.52: The In-Cylinder Tab of the Options Dialog Box



Ansys Fluent provides a built-in function that can define the location of the piston as a function of crank angle. This function is named ****piston-full****, and is selected from the **Motion UDF/Profile** drop-down list in the **Dynamic Mesh Zones** dialog box as part of rigid body motion (see Rigid Body Motion (p. 1906) for details). If you plan to specify the piston motion using this function, you must specify the **Crank Radius** (that is, the distance between the crank center and the center of the crank pin) and the **Connecting Rod Length**. You also have the option of entering a value for the **Piston Pin Offset** for cases when the piston pin is offset perpendicularly from the plane defined by the crank shaft axis and the direction of motion of the piston. The sign of this offset can be positive or negative, and is determined based on the geometry and the direction of rotation of the crank shaft (as shown in Figure 13.53: Determining the Sign of the Piston Pin Offset (p. 1870)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.53: Determining the Sign of the Piston Pin Offset



When the ****piston-full**** function is used, the piston location is calculated according to the following equation:

$$p_s=\sqrt{(r_c+L)^2-x_{\text{offset}}^2}-r_c\cos(\theta_c)-\sqrt{L^2-(r_c\sin(\theta_c)+x_{\text{offset}})^2} \tag{13.25}$$

where p_s is the piston location, r_c the crank radius, L is the connecting rod length, x_{offset} is the piston pin offset, and θ_c is the current crank angle.

The piston location p_s is always 0 at top-dead-center (TDC), that is, when the crank pin is perfectly aligned between the piston pin and the center of rotation of the crank shaft. TDC occurs when the crank angle is 0° when there is no piston pin offset, and prior to the crank angle reaching 0° when there is a positive piston pin offset. The piston location is a positive value at bottom-dead-center (BDC), that is, when the crank shaft is perfectly aligned between the piston pin and the crank pin. The value of p_s at BDC is equal to $2r_c$ when there is no piston pin offset, and greater than $2r_c$ when there is a nonzero piston pin offset (positive or negative).

The current crank angle θ_c is calculated from

$$\theta_c=\theta_s+t\Omega_{\text{shaft}} \tag{13.26}$$

where θ_s is the **Starting Crank Angle** and Ω_{shaft} is the **Crank Shaft Speed**.

The **Piston Stroke Cutoff** and **Minimum Valve Lift** values are used to control the actual values of the valve lift and piston stroke such that

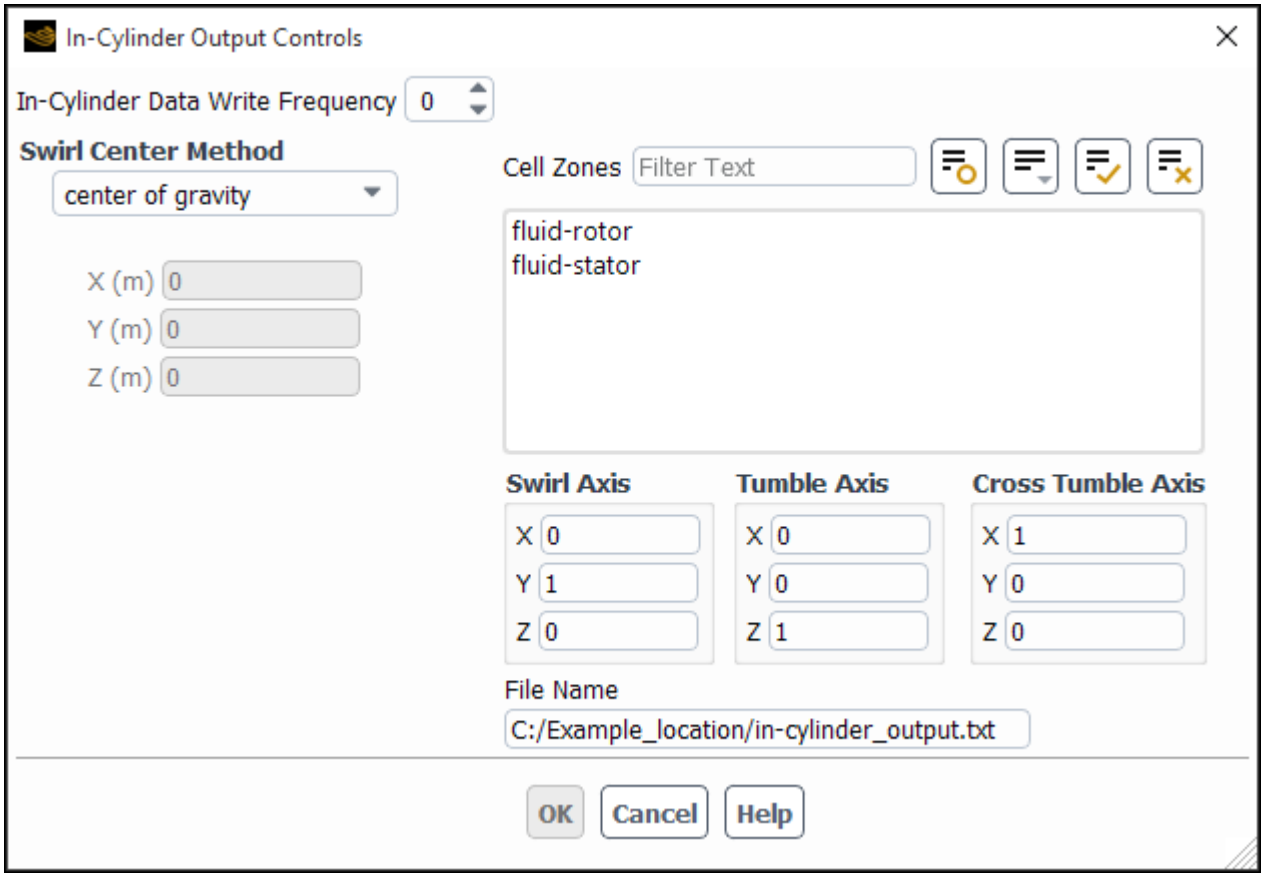
$$\begin{aligned} v_{lift}&=\max\left(v_{lift}^c, v_{lift}^{min}\right)\\ p_s&=\min\left(p_s^c, p_s^{min}\right)\end{aligned} \tag{13.27}$$

where v_{lift}^c is the valve lift computed from the appropriate valve profiles, v_{lift}^{min} is the **Minimum Valve Lift**, p_s^c is the stroke calculated from Equation 13.25 (p. 1870), and p_s^{min} is the **Piston Stroke Cutoff**. (See Defining Motion/Geometry Attributes of Mesh Zones (p. 1877) on how the **Piston Stroke Cutoff** is used to control the onset of layering in the cylinder chamber.)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Enable the **Write In-Cylinder Output** option then click the **Output Controls...** button if you want to specify specific output parameters. The In-Cylinder Output Controls Dialog Box (p. 5125) will open, where you can specify various quantities needed for the calculation of swirl and tumble along with the frequency of writing the output to the chosen file. Swirl is used to describe circulation about the cylinder axis. Tumble flow circulates around an axis perpendicular to the cylinder axis, orthogonal to swirl flow.

Figure 13.54: The In-Cylinder Output Controls Dialog Box



The following list describes the In-Cylinder Output Controls Dialog Box (p. 5125).

In-Cylinder Data Write Frequency

is an integer entry specifying the interval in number of time-steps. Make sure that a value other than 0 is used for the frequency, in order to allow you to complete your setup.

Swirl Center Method

is a drop-down list that allows you to select the method to calculate the swirl center. The list contains **center of gravity** and **fixed**, with **center of gravity** being the default value.

center of gravity

option calculates the swirl center inside the code and is used as the center of gravity of the chosen cell zones.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

fixed

option enables you to specify a swirl center in the entries below the drop-down list.

In addition to these two options, you can chose to use your own compiled UDF to calculate the swirl center.

For details on using a dynamic mesh UDF, see the Fluent Customization Manual for information on user-defined functions.

Cell Zones

is a list that displays the names of all existing cell zones in the case files. You can select only the zones relevant for the swirl and tumble calculations.

Swirl Axis

specifies the swirl axis with three entries for the directional components. By default, **X**, **Y**, **Z** = **0**, **1**, **0**.

Tumble Axis

specifies the directional components of **Tumble Axis** in **X**, **Y**, **Z** directions. By default, **X**, **Y**, **Z** = **0**, **0**, **1**. This applies only in 3D.

Cross Tumble Axis

specifies the directional components of **Cross Tumble Axis** in **X**, **Y**, **Z** directions. By default, **X**, **Y**, **Z** = **1**, **0**, **0**. This applies only in 3D.

File Name

specifies the name of the **In-Cylinder** output file. By default, the file name contains the name of the case file appended with a `.txt` extension.

The **In-Cylinder** specific output controls can also be controlled using the TUI as follows:

```
 Go to
 define/dynamic-mesh/controls/in-cylinder-output?
Enable in-cylinder output?[no] yes
Output Write Frequency[0] 10
Cell zone name/id(1)[()] 2
Cell zone name/id(1)[()]
File Name[''/nfs/devvault/data9/ic-sp-output.txt'']
Swirl Center Method: (fixed cg user-defined)
 Option[cg]
 Swirl Axis x[0]
 Swirl Axis y[1]
 Swirl Axis z[0]
 Tumble Axis x[0]
 Tumble Axis y[0]
 Tumble Axis z[1]
 Cross Tumble Axis x[1]
 Cross Tumble Axis y[0]
 Cross Tumble Axis z[0]
```

If you select **fixed** as the choice at **Swirl Center Method** then you will be prompted to enter the swirl center as follows:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
Swirl Center(x) (mm) [0]
Swirl Center(y) (mm) [0]
Swirl Center(z) (mm) [0]
```

If a swirl center method UDF has been compiled already and loaded into UDF then you can choose **user-defined** as the swirl center method option, in such a case the following is the sequence of prompts.

```
Swirl Center UDF[] swirl_udf::libudf
```

If the name of the UDF library is **libudf** then you can omit this and enter in the swirl center `UDF[]swirl_udf`, otherwise the name of the UDF followed by the UDF library name with symbol:: in between, should be entered.

By filling up the various entries that are needed in the In-Cylinder Output Controls Dialog Box (p. 5125) and pressing the **OK** button, the swirl and tumble calculations will be written at the chosen frequency to the chosen file while doing the solution run. Details of the quantities written to the file are as follows:

CA = Crank Angle

m = Mass of the entire fluid contained in the selected cell zones

L = Angular momentum vector of fluid mass contained in selected cell zones with respect to the swirl center

$|\vec{L}|$ = Magnitude of angular momentum of fluid

$\vec{sa}$ = Swirl Axis

$\vec{ta}$ = Tumble Axis

$\vec{cta}$ = Cross Tumble Axis

I $_{sa}$ = Moment of inertia of the fluid mass about Swirl axis

I $_{ta}$ = Moment of inertia of the fluid mass about Tumble Axis

I $_{cta}$ = Moment of inertia of the fluid mass about Cross Tumble Axis

$\cdot$ = Dot product between two vectors

Altogether, the previous quantities are combined to yield eight columns of data in the output file, as shown in the figure that follows:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.55: Sample Output File Showing Various Quantities

CA	(L . sa)	(L . ta)	(L . cta)	|L|	I_{sa}	I_{ta}	I_{cta}
350.00	0.0000e+00	0.0000e+00	0.0000e+00	0.0000e+00	1.1474e-07	7.0881e-08	6.8234e-08
365.00	6.4910e-07	7.6332e-07	-2.3923e-08	1.0023e-06	9.9355e-08	6.1951e-08	5.9120e-08
380.00	-1.6684e-06	-2.3585e-06	-7.8704e-08	2.8900e-06	1.5460e-07	9.2601e-08	9.2951e-08
395.00	-3.0125e-05	8.1780e-06	2.9730e-06	3.1357e-05	2.9557e-07	1.7709e-07	1.8031e-07
410.00	-8.9259e-05	1.7637e-05	-1.3191e-05	9.1936e-05	4.9396e-07	3.1077e-07	3.1660e-07
425.00	-2.1336e-04	2.4657e-05	-4.6908e-05	2.1984e-04	7.3845e-07	5.1056e-07	5.1811e-07
440.00	-3.9555e-04	6.6114e-05	-1.0156e-04	4.1370e-04	1.0084e-06	7.8920e-07	7.9931e-07
455.00	-6.0621e-04	1.2651e-04	-1.7990e-04	6.4487e-04	1.2833e-06	1.1499e-06	1.1623e-06
470.00	-8.1472e-04	2.0109e-04	-2.6251e-04	8.7927e-04	1.5486e-06	1.5784e-06	1.5913e-06
485.00	-9.9456e-04	2.7342e-04	-3.6243e-04	1.0933e-03	1.7921e-06	2.0409e-06	2.0526e-06
500.00	-1.1160e-03	2.9711e-04	-4.5477e-04	1.2412e-03	2.0003e-06	2.4850e-06	2.4945e-06

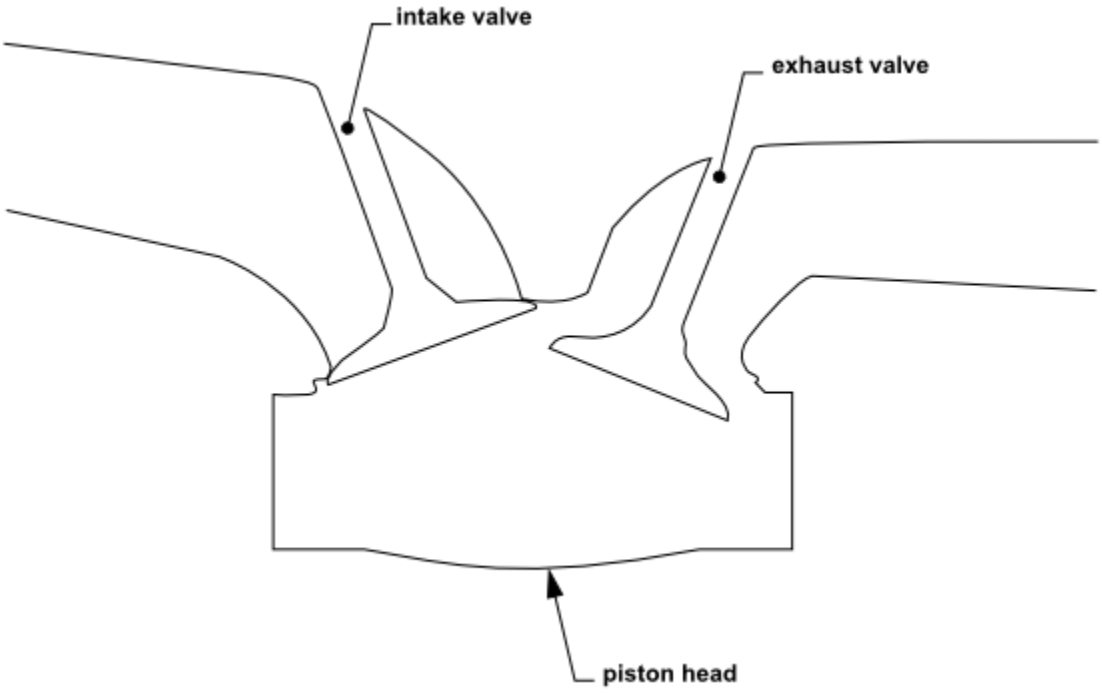
13.6.4.1. Using the In-Cylinder Option

This section describes the problem setup procedure for an in-cylinder dynamic mesh simulation.

13.6.4.1.1. Overview

Consider the 2D in-cylinder example shown in Figure 13.56: A 2D In-Cylinder Geometry (p. 1874) for a typical pent-roof engine.

Figure 13.56: A 2D In-Cylinder Geometry



In setting up the dynamic mesh model for an in-cylinder problem, you must consider the following issues:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- how to provide the proper mesh topology for the volume mesh update methods (smoothing, dynamic layering, and local or zonal remeshing)
- how to define the motion attributes and geometry for the valve and piston surfaces
- how to address the opening and closing of the intake and exhaust valves
- how to specify the sequence of events that controls the in-cylinder simulation

13.6.4.1.2. Defining the Mesh Topology

Ansys Fluent requires that you provide an initial volume mesh with the appropriate mesh topology such that the various mesh update methods described in Dynamic Mesh Update Methods (p. 1816) can be used to automatically update the dynamic mesh. However, Ansys Fluent does not require you to set up all in-cylinder problems using the same mesh topology. When you generate the mesh for your in-cylinder problem, you must consider the various mesh regions that you can identify as moving, deforming, or stationary, and generate these mesh regions with the appropriate cell shape.

The mesh topology for the example problem in Figure 13.56: A 2D In-Cylinder Geometry (p. 1874) is shown in Figure 13.57: Mesh Topology Showing the Various Mesh Regions (p. 1875), and the corresponding volume mesh is shown in Figure 13.58: Mesh Associated With the Chosen Topology (p. 1876).

Figure 13.57: Mesh Topology Showing the Various Mesh Regions

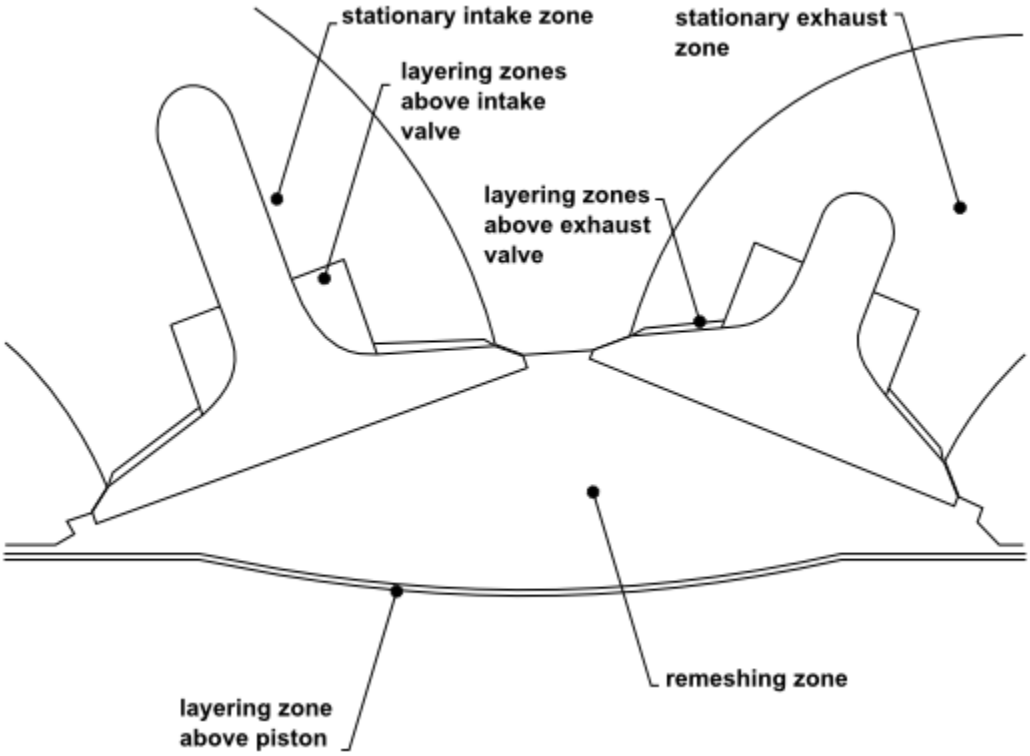
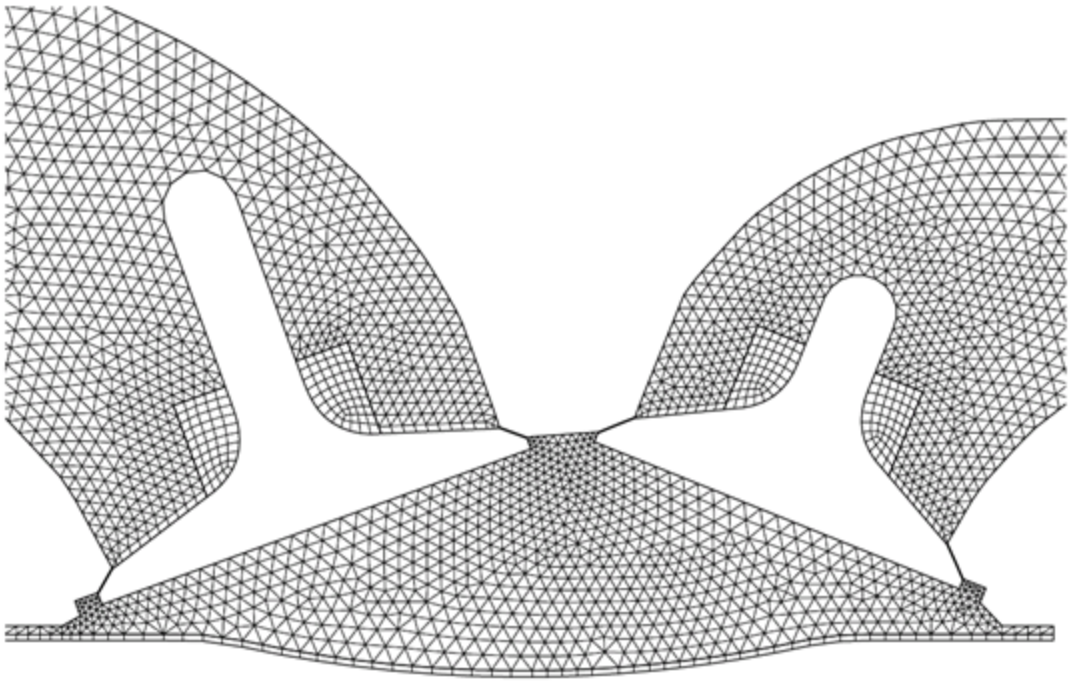


Figure 13.58: Mesh Associated With the Chosen Topology



Because of the rectilinear motion of the moving surfaces, you can use dynamic layering zones to represent the mesh regions swept out by the moving surfaces. These regions are the regions above the top surfaces of the intake and exhaust valves and above the piston head surface, and must be meshed with prismatic cells that are quadrilateral in 2D or polyhedral, hexahedral, or wedge cells in 3D (as required by the dynamic layering method).

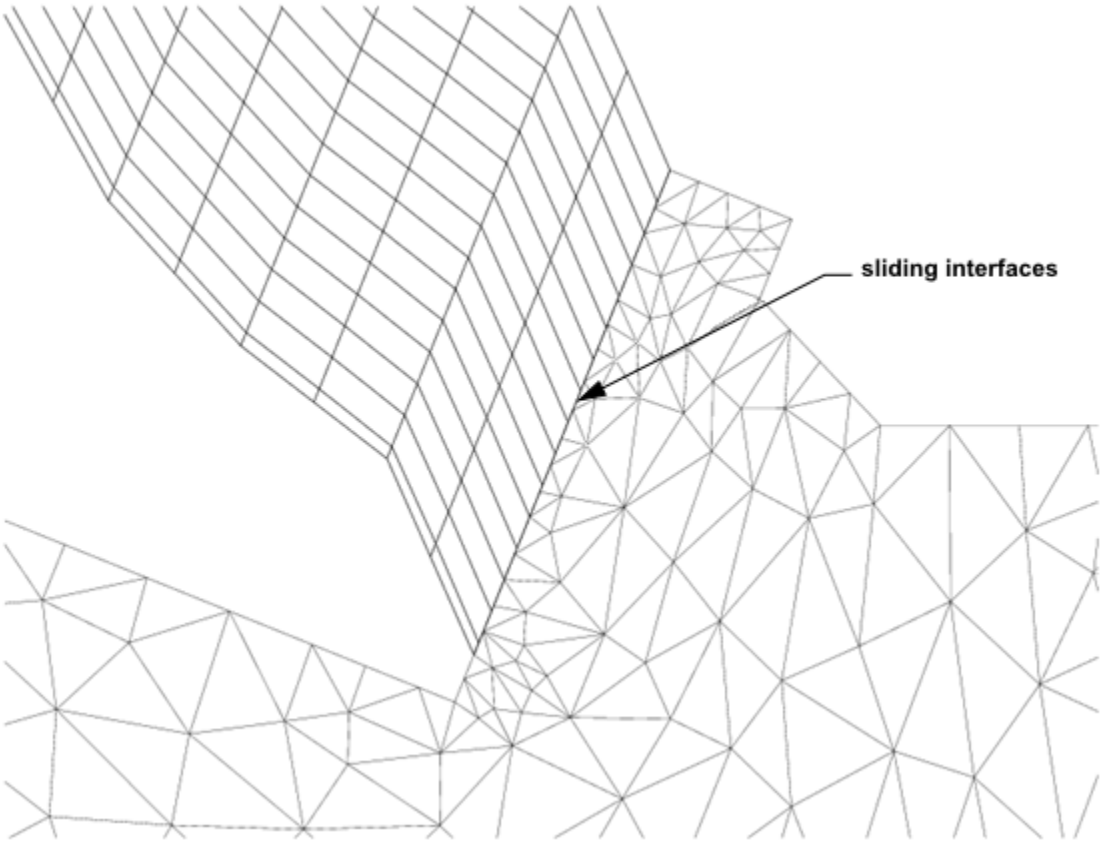
For the chamber region, you must define a remeshing zone (triangular cells) to accommodate the various positions of the valves in the course of the simulation. In this region, the motion of the boundaries (valves and piston surfaces) is propagated to the interior nodes through smoothing. If the cell quality violates any of the remeshing criteria that you have specified, Ansys Fluent will automatically agglomerate these cells and remesh them. Furthermore, Ansys Fluent will also remesh the deforming faces (based on the minimum and maximum length scale that you have specified) on the cylinder walls as well as those on the sliding interfaces used to connect the chamber cell zone to the layering zones above the valve surfaces.

For the intake and exhaust port regions, you can use either triangular or quadrilateral cell zones because these zones are not moving or deforming. Ansys Fluent will automatically mark these regions as stationary zones and will not apply any mesh motion method on these cell zones.

The dynamic layering regions above the piston and valves are conformal with the adjacent cell zone in the chamber and ports, respectively, so you do not have to use sliding interfaces to connect these cell zones together. However, you must use sliding interfaces to connect the dynamic layering regions above the valves and the remeshing region in the chamber. This is shown in Figure 13.59: The Use of Sliding Interfaces to Connect the Exhaust Valve Layering Zone to the Remeshing Zone (p. 1877) with the exhaust valve almost at full extension. Notice that cells on the chamber side of the interface zone are remeshed (that is, split or merged) as the interface zone opens and closes because of the motion of the exhaust valve.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.59: The Use of Sliding Interfaces to Connect the Exhaust Valve Layering Zone to the Remeshing Zone



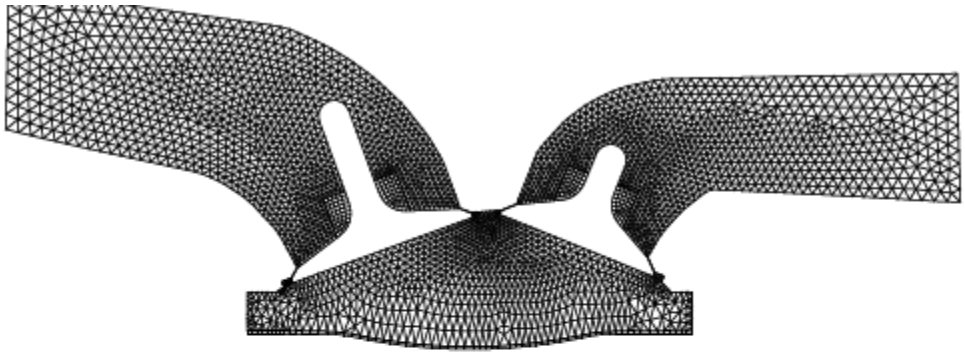
13.6.4.1.3. Defining Motion/Geometry Attributes of Mesh Zones

As the piston moves down from the TDC to the BDC position, you must expand the remeshing region such that it can accommodate the valves when they are fully extended. To accomplish this, you must specify the dynamic layering zone adjacent to the piston surface to move with the piston until some specified distance from the TDC position. Beyond this cutoff distance, the motion of the layering zone is stopped and the piston wall is allowed to continue to the BDC position. Because there is relative motion between the piston head surface and the now non-moving dynamic layering zone, cell layers will be added when the ideal layer height criteria is violated. Figure 13.60: Mesh Sequence 1 (p. 1878) to Figure 13.65: Mesh Sequence 6 (p. 1880) show the sequence of meshes before and after the onset of cell layering when the motion in the layering zone above the piston surface is stopped (shown with $\Delta\theta = 5°$).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.60: Mesh Sequence 1

Figure 13.61: Mesh Sequence 2



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.62: Mesh Sequence 3

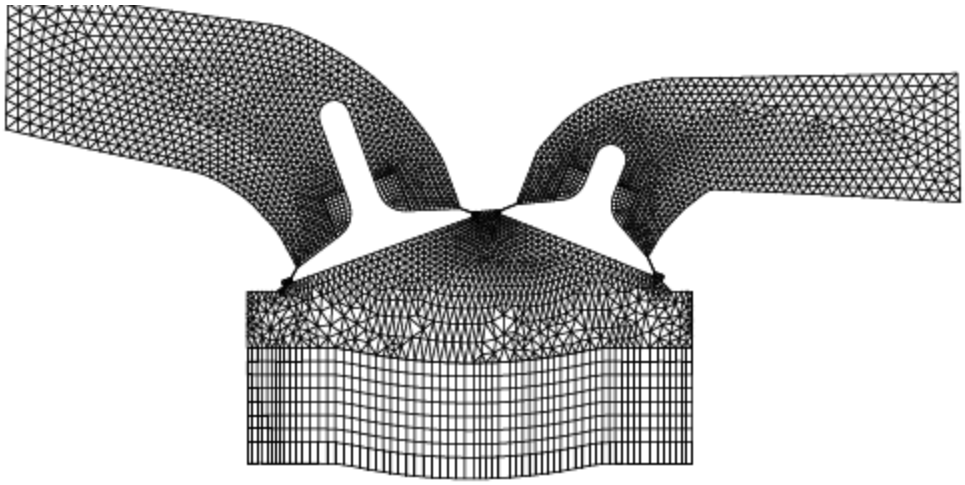
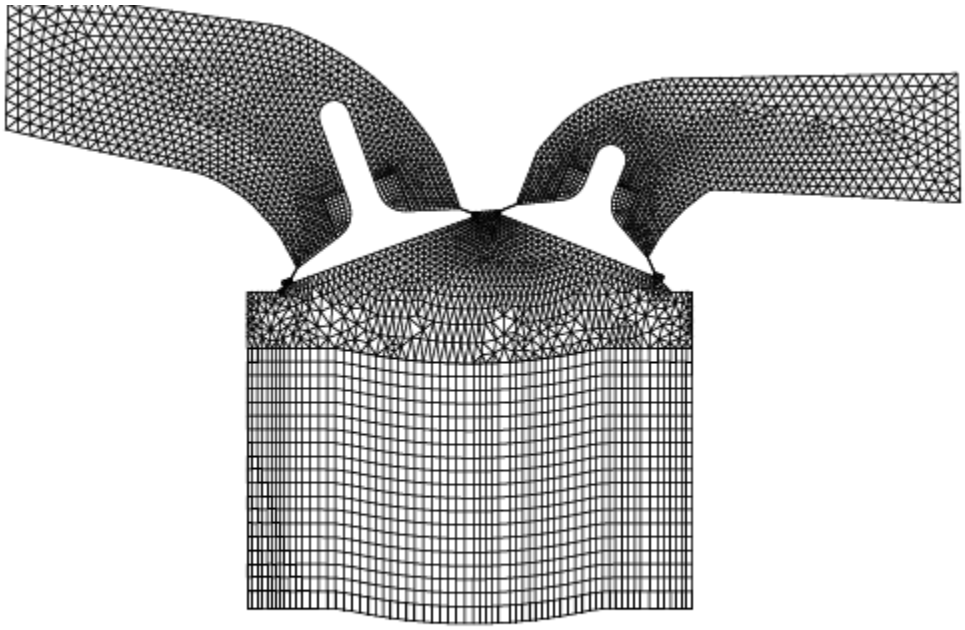


Figure 13.63: Mesh Sequence 4



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.64: Mesh Sequence 5

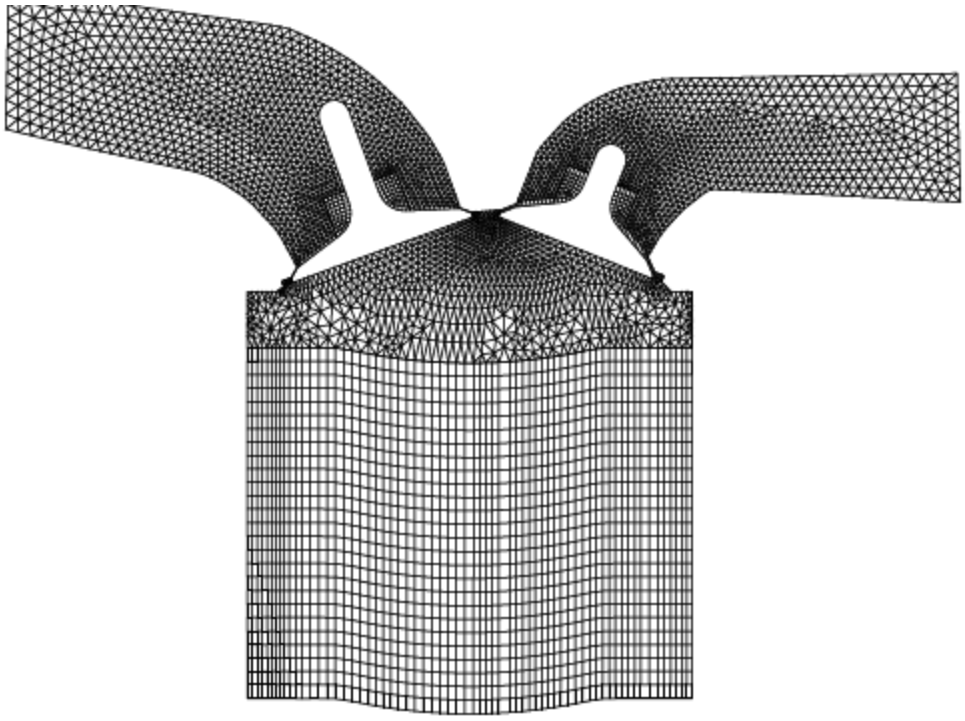
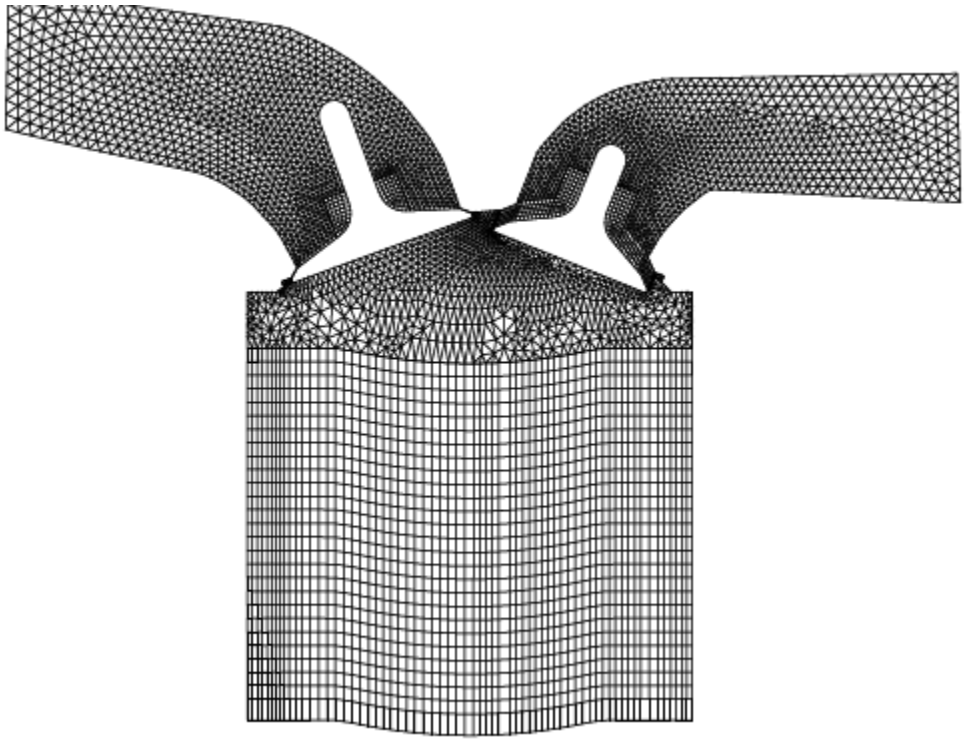


Figure 13.65: Mesh Sequence 6

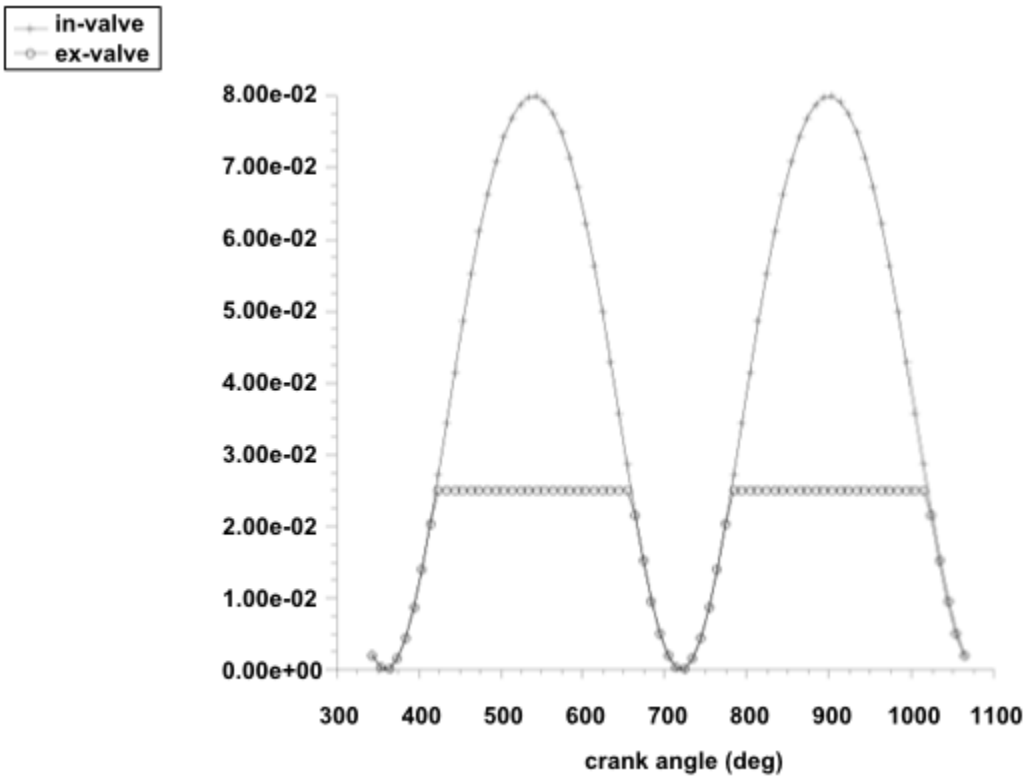


Ansys Fluent provides built-in functions to handle the full piston motion and the limited piston motion for the dynamic layering zone above the piston surface. When you define the motion

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

attribute of the dynamic layering zone above the piston surface, you must use the limited piston motion function (****piston-limit**** in the **Motion UDF/Profile** field in the Dynamic Mesh Zones Dialog Box (p. 5140)). Note that you must define the parameters used by these functions before you can use them. In the current example, the crank radius is 40 mm and the connecting rod length is 140 mm. The piston stroke cutoff is assumed to happen at 25 mm from TDC position. The lift as a function of crank angle between 344° and 1064° is shown in Figure 13.66: Piston Position (m) as a Function of Crank Angle (deg) (p. 1881) for both limited and full piston motion.

Figure 13.66: Piston Position (m) as a Function of Crank Angle (deg)



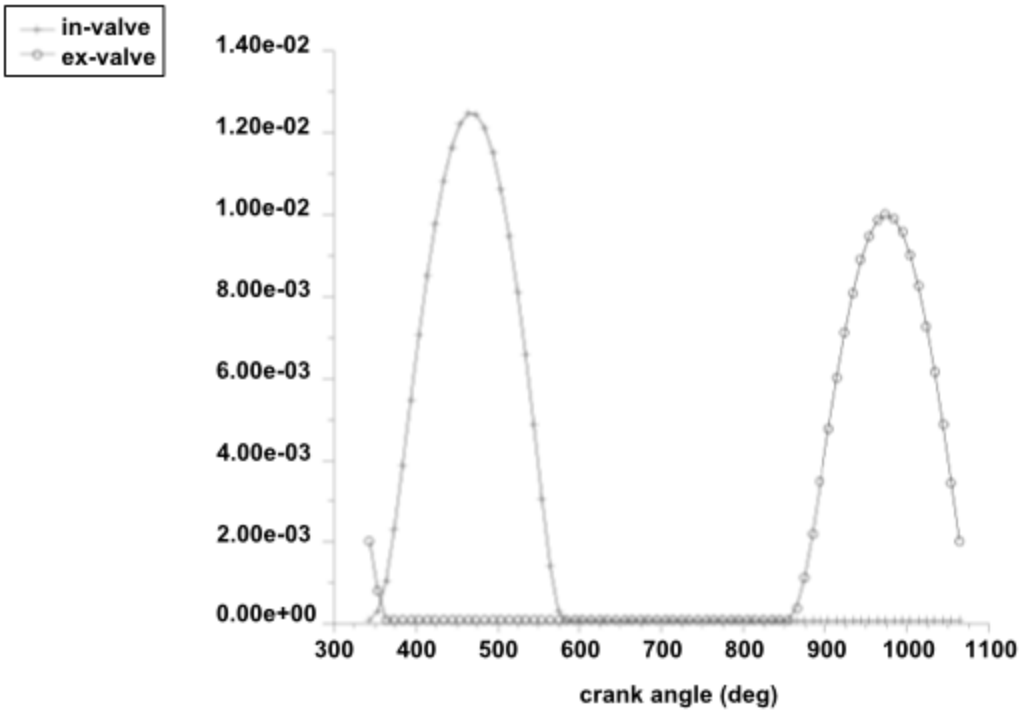
To define the motion of the valves, you must use profiles that describe the variation of valve lift with crank angle. Ansys Fluent expects certain profile fields to be used to define the lift and the crank angle. For example, consider the following simplified profile definition:

```
((ex-valve 5 point)
(angle 0  180 270 360  720)
(lift 0.05 0.05 1.8 0.05 0.05))

((in-valve 5 point)
(angle 0  355  440 540 720)
(lift 0.05 0.05 2.0 0.05 0.05))
```

Ansys Fluent expects the `angle` and `lift` fields to define the crank angle and lift variations, respectively. The angle must be specified in degrees and the lift values must be in meters. The actual valve lift profiles that you will use for the current example are shown in Figure 13.67: Intake and Exhaust Valve Lift (m) as a Function of Crank Angle (deg) (p. 1882). Notice that there is an overlapped period where both the intake and exhaust valves are open.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.67: Intake and Exhaust Valve Lift (m) as a Function of Crank Angle (deg)



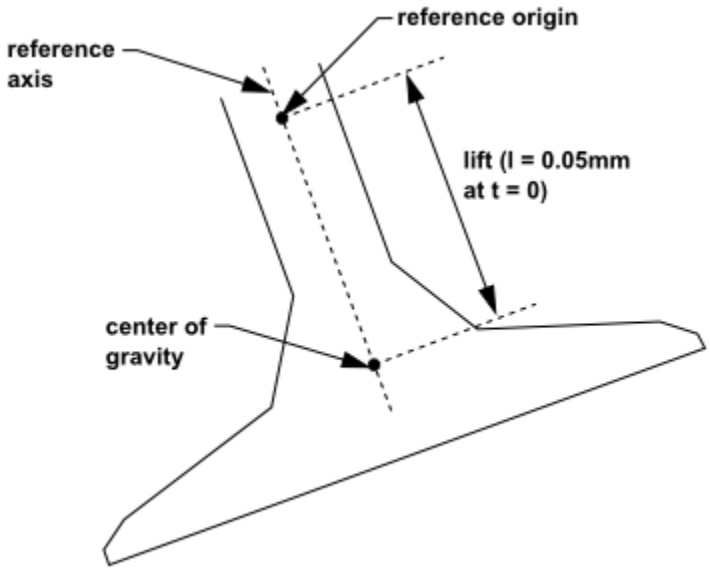
The valve lift profiles and the built-in functions will describe how each surface moves as a function of crank angle with respect to some reference point. For example, the valve lift is zero when the valve is fully closed and the valve lift is maximum when it is fully open. In order to move the surfaces, Ansys Fluent requires that you specify the direction of motion for each surface. Ansys Fluent will then update the "center of gravity" of each surface such that

$$\vec{x}=\vec{x}_{ref}-l\vec{e}_{axis} \tag{13.28}$$

where $\vec{x}_{ref}$ is some reference position, $\vec{e}_{axis}$ is the unit vector in the direction of motion, and l is either the valve or the piston distance with respect to the reference position $\vec{x}_{ref}$. Note that the unit vector of the direction of motion is specified to point in the negative direction. For example, the correct intake valve axis for this example is $\left(-0.3421, 0.9397\right)$, as shown in Figure 13.68: Definition of Valve Zone Attributes (Intake Valve) (p. 1883).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.68: Definition of Valve Zone Attributes (Intake Valve)



13.6.4.1.4. Defining Valve Opening and Closure

Ansys Fluent assumes that once you have set up the mesh topology, the mesh topology is unchanged throughout the entire simulation. Therefore, Ansys Fluent does not allow you to completely close the valves such that the cells between the valve and the valve seat become degenerate (flat cells) when these surfaces come in contact (removing these flat cells would require the creation of new boundary face zones). To prevent the collapse, you must define a minimum valve lift and Ansys Fluent will automatically stop the motion of the valve when the valve lift is smaller than the minimum valve lift value. The minimum valve lift value can be specified in the **In-Cylinder** tab of the Options Dialog Box (p. 5121). For the current example, a minimum valve lift value of 0.1 mm is assumed.

When the valve position is smaller than the minimum valve lift value, it is normal practice to assume that the valve is closed. The actual closing of the valves is accomplished by deleting the sliding interfaces that connect the chamber cell zone to the dynamic layering zones on the valves. The interface zones are then converted to walls to close off the "gaps" between the valves and the valve seats.

The valve opening is achieved by the reverse process. When the valve lift has reached beyond the minimum valve lift value, the valve is assumed to be open and you can redefine the sliding interfaces such that the chamber zone is now connected to the dynamic layering zones above the valves.

13.6.5. Six DOF Solver Settings

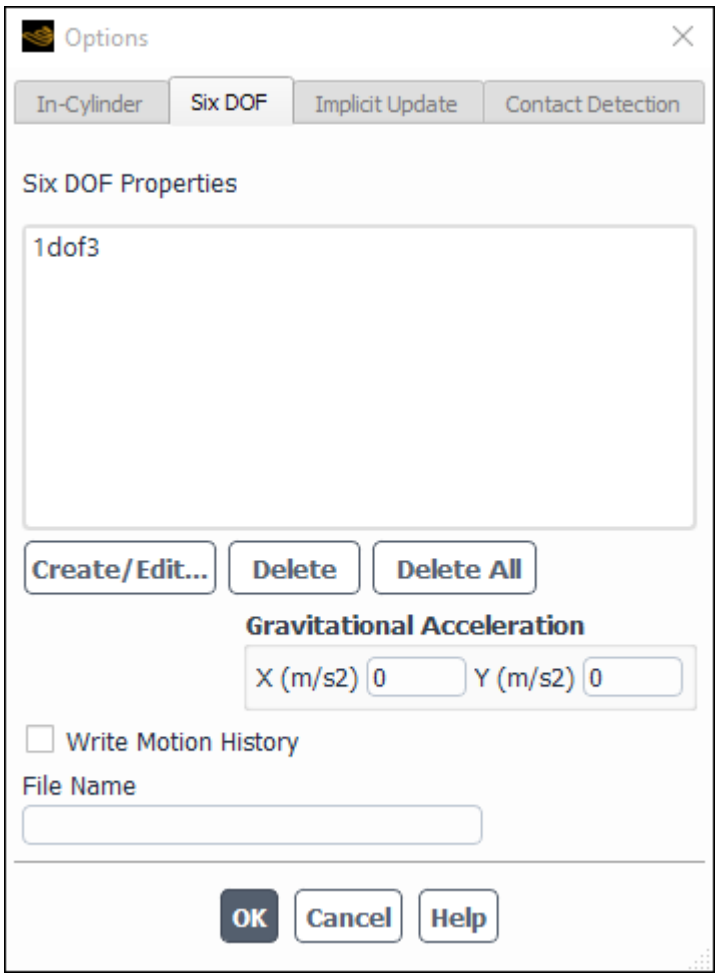
Ansys Fluent's six degrees of freedom (six DOF) solver computes external forces and moments (such as aerodynamic and gravitational forces and moments) on objects that undergo rigid body motion. These forces are computed by numerical integration of pressure and shear stress over the object's surfaces. Additional load forces can be added, such as injector forces, thrust (propulsive) forces, and moments produced by a coil spring. This technique, along with the Ansys Fluent solver and the use of dynamic meshes, can be readily applied to many useful applications, such as store separation [140] (p. 5721), [151] (p. 5721). Note that the objects may be composed of multiple zones.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To use the six degrees of freedom solver for your transient dynamic mesh simulation, perform the following steps:

1. Enable the **Six DOF** option in the **Options** group box of the Dynamic Mesh Task Page (p. 5111) (Figure 13.13: The Dynamic Mesh Task Page (p. 1815)) and click the **Settings...** button. The **Options** dialog box will open, where you can click the **Six DOF** tab (Figure 13.69: The Six DOF Tab of the Options Dialog Box (p. 1884)).

   Figure 13.69: The Six DOF Tab of the Options Dialog Box



   This tab allows you to perform the following actions:

   - You can create and manage sets of six DOF properties for rigid body motion by using the list and buttons under **Six DOF Properties**; otherwise, you will have to create one or more user-defined functions. See Setting Rigid Body Motion Attributes for the Six DOF Solver (p. 1885) for details.

   - You can specify the gravitational acceleration for the *x*, *y*, and *z* directions either in this dialog box, or in the **Operating Conditions** dialog box.

   - You can keep track of an object's motion history by enabling the **Write Motion History** option and entering a **File Name**. A single motion history file will be generated for each moving object, which can be used to display zone motion for postprocessing your results. Note that the file name you specify will be appended with the name of the set of six DOF properties / UDF, as

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

well as the extension `.6dof`; you can change the extension by specifying your own as part of the **File Name**.

2. If the mesh motion of your six DOF simulation depends on the fluid flow, it is beneficial to enable implicit mesh updating (as described in Implicit Update Settings (p. 1888)). If the implicit mesh update is not sufficient to stabilize the solver due to the fluid motion, solution stabilization may be necessary. The solution stabilization settings are set in the **Solver Options** tab of the **Dynamic Mesh Zones** dialog box for boundary zones (Solution Stabilization for Dynamic Mesh Boundary Zones (p. 1918)).

3. Define the rigid-body zone(s) that make up each moving object by using the Dynamic Mesh Zones Dialog Box (p. 5140), as described in Rigid Body Motion (p. 1906).

Setup → **Dynamic Mesh** → **Create/Edit...**

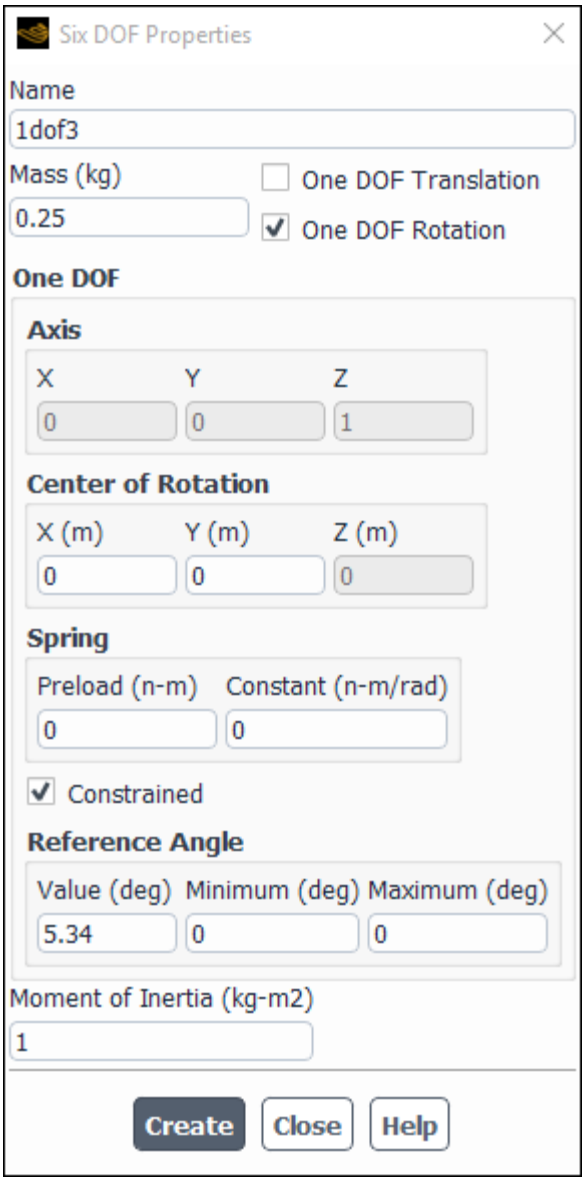
13.6.5.1. Setting Rigid Body Motion Attributes for the Six DOF Solver

As part of the six DOF solver settings, you must provide either a set of properties or a user-defined function (UDF) for each moving object, in order to define its overall mass, rotational inertia, constraints, loading from sources other than aerodynamics and gravity, and so on. Each object needs exactly one set of properties / UDF, regardless of how many zones make up the object. Note that if you have multiple identical objects, you can create an initial set of properties / UDF, and then make copies with unique names.

If the motion of the object is complex or if only half of the object is modeled using a symmetry zone, then you will need to use a UDF (see `DEFINE_SDOF_PROPERTIES` in the *Fluent Customization Manual* for the details); otherwise, it is easier to use a set of properties. Such sets are created using the **Six DOF Properties** dialog box, which is opened by clicking the **Create/Edit...** button in the **Six DOF** tab of the Options Dialog Box (p. 5121).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.70: The Six DOF Properties Dialog Box



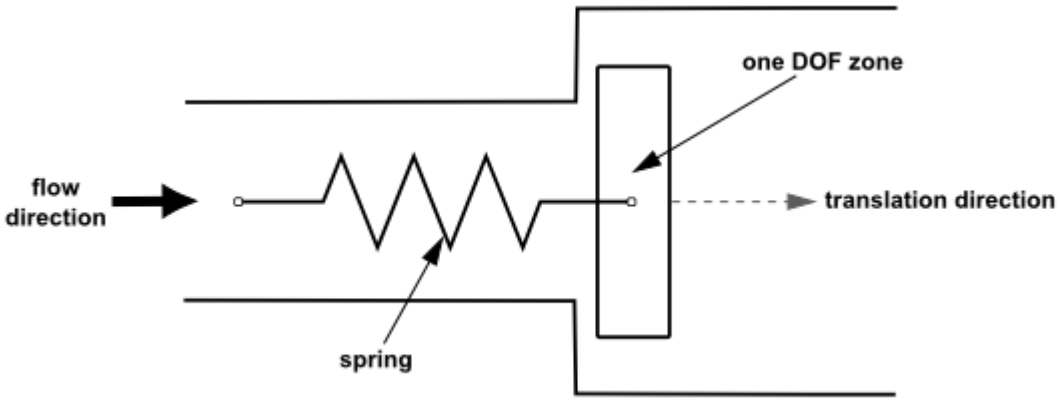
In the **Six DOF Properties** dialog box, you will need to provide a **Name** for the set of properties, define the other fields in the dialog box, and click **Create**.

For six DOF motion, the only other fields that must be defined are the **Mass** and the **Inertia Tensor** components. Note that the inertia tensor components are defined relative to the **Center of Gravity** and **Rigid Body Orientation** settings defined in the Dynamic Mesh Zones Dialog Box (p. 5140); the latter settings allow you to use a coordinate system that is local (rather than global), if that is more convenient.

If you want to specify that the dynamic object be limited to one DOF motion (that is, either a simple translation or a simple rotation), enable the **One DOF Translation** or **One DOF Rotation** option and define the settings in the **One DOF** group box. The following figure shows an example of a one DOF translation:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.71: A Check Valve with One DOF Translation



Note the following about one DOF motion settings:

- For rotation, you will define the **Mass** of the dynamic object, in case the **Center of Rotation** is different than the **Center of Gravity** defined in the Dynamic Mesh Zones Dialog Box (p. 5140) (which may be the case in a swinging pendulum, for example), so that Fluent can calculate the moment due to gravity.

  For rotation, you will also define the **Moment of Inertia**, as calculated around the **Center of Rotation** and **Axis**. Fluent calculates the moment of fluid forces about the **Center of Rotation**. Note that this definition of the **Moment of Inertia** is different from a six DOF case that does not use **One DOF Rotation**, where you would calculate the **Inertia Tensor** relative to the **Center of Gravity** and **Rigid Body Orientation** settings defined in the Dynamic Mesh Zones Dialog Box (p. 5140), as noted previously.

- For 2D cases with rotation, the axis will be either the positive or negative *z* direction, depending on the sign of the value you enter for **Z** in the **Axis** group box.

- You have the option of defining a Hooke's law **Spring** that exerts forces / torques on the dynamic object. It is a compression / extension spring for translations, and it is a torsion spring for rotations.

  For example: for a translation, the specified **Constant** will be multiplied by the distance of the dynamic object from its initial location. This force will be combined with a **Preload** value that is constantly applied to the dynamic object, in order to produce the net spring loading. Note that the **Preload** can be a positive or negative value, and is defined relative to the **Direction**.

- You can limit the range of motion of the dynamic object by enabling the **Constrained** option and defining the settings in the **Reference Point** group box (for translation) or the **Reference Angle** group box (for rotation).

  For example: for a translation, you could enter 2 inches for the **Location**, in order to assign a coordinate value to a point located on the dynamic object in its initial position. Then you would define the range of motion allowed relative to that coordinate: you would enter `1.2` and 2 inches for the **Minimum** and **Maximum**, respectively, if you wanted the object to only be allowed to move .8 inches in the direction opposite to the **Direction** vector. The value of the **Location** is arbitrary, and allows you to define the range using values that are most convenient for your problem.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The initial position / orientation of the object (which is used to calculate the spring loading and/or range of motion) is established when you create the dynamic zone(s) using the Dynamic Mesh Zones Dialog Box (p. 5140).

13.6.6. Implicit Update Settings

For transient problems, you can enable implicit mesh updating when you want to have the dynamic mesh updated during a time step (as opposed to just at the beginning of a time step). This capability is beneficial only for applications in which the mesh motion depends on the flow field (for example, cases that use the six DOF solver or involve fluid-structure interaction). For such applications, having the mesh motion updated within the time step based on the converging flow solution results in a stronger coupling between the flow solution and the mesh motion, and leads to a more robust solver run. Implicit mesh updating allows you to run simulations that otherwise could not be solved or would require an unreasonably small time step size.

Note that implicit mesh updating cannot be used with the following:

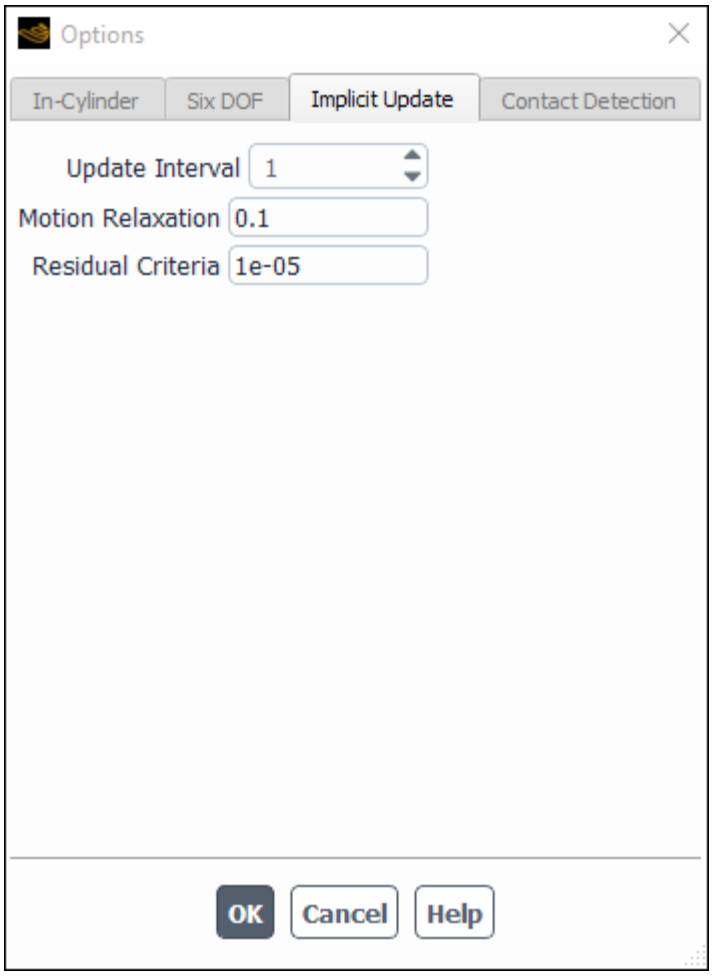
- the density-based solver when **Explicit** is selected from the **Transient Formulation** drop-down menu in the **Solution Methods** task page
- steady-state solutions
- in-cylinder applications

To enable implicit mesh updating, perform the following steps:

1. Enable **Implicit Update** in the **Options** group box of the Dynamic Mesh Task Page (p. 5111) (Figure 13.13: The Dynamic Mesh Task Page (p. 1815)).
2. Click the **Settings...** button to open the Options Dialog Box (p. 5121).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.72: The Implicit Update Tab of the Options Dialog Box



Click the **Implicit Update** tab and enter values for the settings.

a. Enter a value for **Update Interval**, in order to specify the frequency in iterations at which the mesh will be updated within a time step.

b. Enter a value (within the range of 0 to 1) for **Motion Relaxation**, in order to define the relaxation of the motion (that is, displacement of the nodes) during the mesh update. The relaxation of the displacements is defined by the following equation:

$$x_k = \omega(x_{computed,k}) + (1-\omega)x_{k-1} \tag{13.29}$$

where x_k is the node position at iteration k (within a time step), $x_{computed,k}$ is the computed node position (based on the flow field), and ω is the motion relaxation.

Note:

The nodal positions returned from the UDF / profile will *not* be relaxed according to Equation 13.29 (p. 1889) if you select **User-Defined** from the **Type** list when setting up your dynamic mesh zones in the Dynamic Mesh Zones Dialog Box (p. 5140).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. Enter a value for **Residual Criteria**, in order to set the relative residual threshold that is used to check the motion convergence. The residual criteria is applied to a relative residual. Ansys Fluent scales the difference between the motion in iteration k and iteration $k-1$ by the motion computed at the beginning of the time step. If this relative motion difference is smaller than the residual criteria, the mesh motion is considered converged.

3. If you are using a UDF to compute the motion, make sure that the UDF uses the current flow field during each call to compute the motion (that is, no previously stored information should be used). This is necessary, as the UDF will be called each time the mesh is updated — which can be several times within a time step, depending on what you entered for the **Update Interval**.

4. After you run the simulation, make sure that the motion (and consequently, the solution) is properly converged. If the motion requires more iterations to converge than the flow field, a warning will be displayed in the console during the iteration process. Note that the maximum number of iterations per time step (defined in the **Run Calculation** task page) is respected by the mesh motion convergence check.

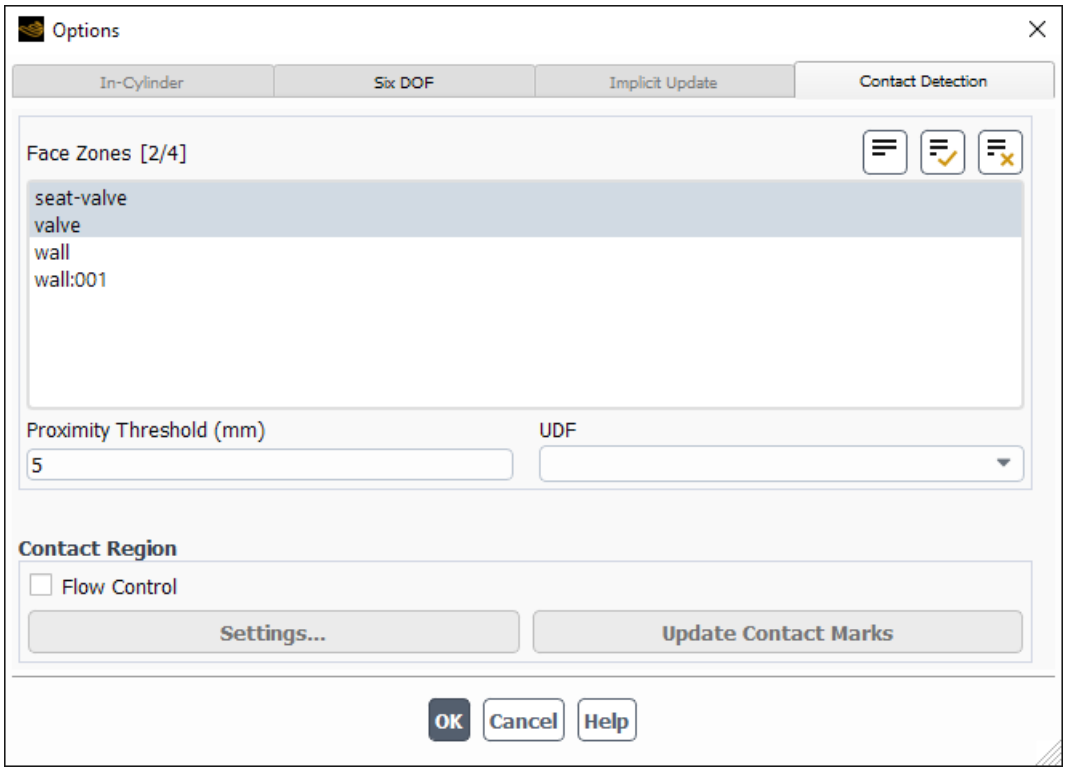
13.6.7. Contact Detection Settings

Contact detection is used to detect if the computed mesh motion will result in contact of a moving surface with other surrounding surfaces. If there is contact within a specified tolerance, then the mesh motion of the moving zone can be constrained using nodal contact information within user-defined functions (See Example 2 under DEFINE_CONTACT in the Fluent Customization Manual) and/or the flow can be controlled in the contact region.

To enable contact detection in your dynamic mesh simulation, select the **Contact Detection** check box in the **Options** group of the **Dynamic Mesh** task page. Click the **Settings...** button to open the **Options** dialog box and select the **Contact Detection** tab (see Figure 13.73: The Contact Detection Tab of the Options Dialog Box (p. 1891)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.73: The Contact Detection Tab of the Options Dialog Box



You can select the face zones that will be involved in the contact detection process from the **Face Zones** list. While all eligible face zones in the **Face Zones** list are selected by default, you can choose to exclude certain zones for efficiency. Only walls and dynamic zones of type **Rigid Body** or **User-Defined** can be selected for contact detection.

From the **UDF** drop-down list, you can specify a user-defined function that will be invoked when contact has been detected. For more information about contact UDFs, see `DEFINE_CONTACT` in the Fluent Customization Manual.

You must specify a **Proximity Threshold** value in order to enable the contact detection process. When the distance between face zones falls below this threshold value, the UDF specified in the **UDF** drop-down list is invoked.

You can also apply flow control in the contact region. Flow control is applied to the cells that lie within the **Proximity Threshold** distance of the specified face zones. There are two methods available for flow control: one based on contact zones (Flow Control Using Contact Zones (p. 1892)), and another based on contact marks (Flow Control Using Contact Marks (p. 1892)). To use one of these methods, enable the **Flow Control** option in the **Contact Region** group box and click the **Settings...** button to open the **Flow Control Settings** dialog box, where you can complete the setup (as described in the following sections).

Note:

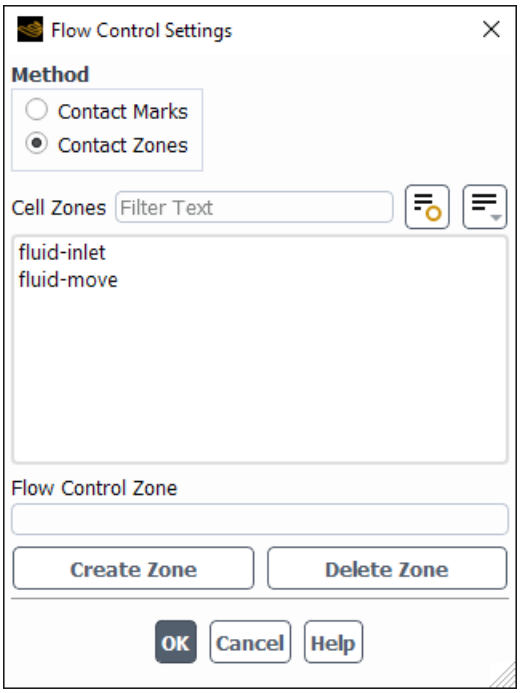
When simulating the complete blockage of flow between moving surfaces, it is recommended that you use the gap model rather than the **Flow Control** option (and the two are not compatible with each other). For details, see Controlling Flow in Narrow Gaps for Valves and Pumps (p. 1234).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.6.7.1. Flow Control Using Contact Zones

You can select **Contact Zones** from the **Method** list in the **Flow Control Settings** dialog box. The **Contact Zones** method for controlling flow allows you to modify the flow in the contact region by creating a separate cell zone containing the marked cells. The flow in this cell zone may be controlled using the available cell zone conditions. By default, the zone is defined to have porous resistance. This method is fully compatible with all solver options.

Figure 13.74: The Flow Control Settings Dialog Box with Contact Zones



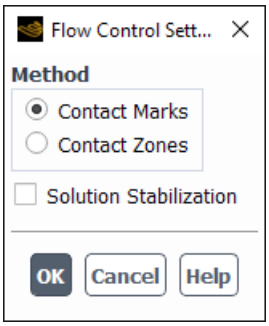
To complete the setup, you must create one or more flow control zones. Select a fluid cell zone from the **Cell Zones** list, specify a new name in the **Flow Control Zone** text box, and click the **Create Zone** button. A new cell zone will be created that is based on the selection from the **Cell Zones** list, only with the **Porous Zone** option enabled. The **Fluids** dialog box will open for this new zone, allowing you to revise physical properties (for example, inertial and/or viscous porous resistance) so that it better represents the restricted flow. During the simulation, after cells in the contact region have been separated into new zones, the physical properties specified in the respective flow control zone are copied to the newly separated cell zone.

13.6.7.2. Flow Control Using Contact Marks

You can select **Contact Marks** from the **Method** list in the **Flow Control Settings** dialog box. The **Contact Marks** method applies a zero mass flux numerical boundary condition on the boundary of the marked contact region that blocks the flow.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.75: The Flow Control Settings Dialog Box with Contact Marks



When the **Contact Marks** method is selected, you have the option of enabling **Solution Stabilization** to improve the stability of the simulation. **Solution Stabilization** applies solution control to stabilize the solution in the vicinity of the contact region during contact engagement, evolution, and disengagement. **Solution Stabilization** involves additional iterations within the time step, during which solution convergence checks are suspended.

Note that the following limitations apply when using the **Contact Marks** method:

- It is limited to the pressure-based solver.
- The use of the **Green-Gauss Node Based** gradient method and/or the **QUICK** discretization scheme is not supported.
- The Non-Iterative Time Advancement (NITA) scheme is not supported.
- Boundary zones of type **outflow** are not recommended.

13.6.7.2.1. Selecting Parameters for Flow Control

For transient flows with moving face zones, the marked contact region is not fixed and could change during every time step when there is mesh motion. With the **Contact Zones** method, this involves deletion and recreation of the cell zone used to apply flow control; this is a computationally expensive operation. Furthermore, while the porous sources are effective in providing resistance to flow, they do not achieve complete flow blockage (which may desirable, for example, in the case of modeling the closing of a valve). Consequently, the **Contact Zones** method may be the recommended method for flow control only under the following conditions: complete flow blocking is not required; the flexibility of cell zone conditions to modify the flow is sufficient; and the contact region is not undergoing rapid changes.

The **Contact Marks** method applies a numerical zero mass flux boundary condition at the boundary faces of the region marked for contact, such that the cells in the contact region no longer participate in the solution. There are a few key advantages of this method that are important to note:

- Marking is computationally more efficient than zone separation.
- The flow is completely blocked at the contact region boundary and there is no flow leakage. This is a realistic flow condition for valves in the closed position.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- It can reduce computational cost in narrow gaps by reducing the need for a high cell count to resolve high solution gradients in the flow between walls in close proximity (for example, a valve opening and closing).
- It can avoid the difficulties of meshing and/or remeshing in narrow passages, especially with moving deforming meshes.

While there are clear advantages, there are some important considerations when using the **Contact Marks** method. The imposition of the zero mass flux condition on arbitrary interior boundary faces can introduce large gradients in the flow in the vicinity of the contact region boundary. Here, **Solution Stabilization** is recommended for cases where the engagement, evolution, and disengagement of contact can be rather abrupt and the maintaining of solver stability poses a challenge. The use of high order term relaxation and a reduced time step size (for compressible flows) may also be beneficial for improving convergence behavior with the **Contact Marks** method.

13.6.7.2.2. Modifying and Displaying Contact Cell Marks

The extents of the contact region are determined by the definition of the proximity threshold distance, the selection of face zones participating in contact detection, and the local mesh size, and this in turn influences the final solution. Contact marks are established during initialization if the proximity threshold distance is satisfied for both steady-state and transient simulations. For transient simulations, subsequently contact marks are updated every time step.

Note that the **Update Contact Marks** button is available in the **Options** dialog box when the **Contact Marks** method is selected for flow control. This provides the ability to update the contact marks immediately without having to reinitialize the solution. That way, you can instantly modify the contact region based on varying the parameters of the contact detection. You can remove or introduce flow blocking after obtaining a solution initially with or without flow blocking, respectively. This option is particularly useful in the process of generating realistic initial conditions for modeling transient flows using flow blocking based on **Contact Marks**.

By default, the contact cell marks are not rendered. You can render the contact cell marks by entering the following text command:

`define` → `dynamic-mesh` → `controls` → `contact-parameters` → `render-contact-cells?`

Then you can display the marked cells by selecting **Cell Info...** and **Contact Cell Mark** from the **Contours of** drop-down lists, and postprocess other field variables on those cells. Note that only cell values are available for **Contact Cell Mark**, so you must ensure that the **Node Values** option is disabled in the **Contours** dialog box.

13.6.8. Defining Dynamic Mesh Events

If you are simulating a transient flow, you can use the events in Ansys Fluent to control the timing of specific events during the course of the simulation. With in-cylinder flows for example, you may want to open the exhaust valve (represented by a pair of deforming sliding interfaces) by creating an event to create the sliding interfaces at some crank angle. You can also use dynamic mesh events to control when to suspend the motion of a face or cell zone by creating the appropriate events based on the crank angle or time. Note that in-cylinder flows are crank angle-based, whereas all other flows are time-based.

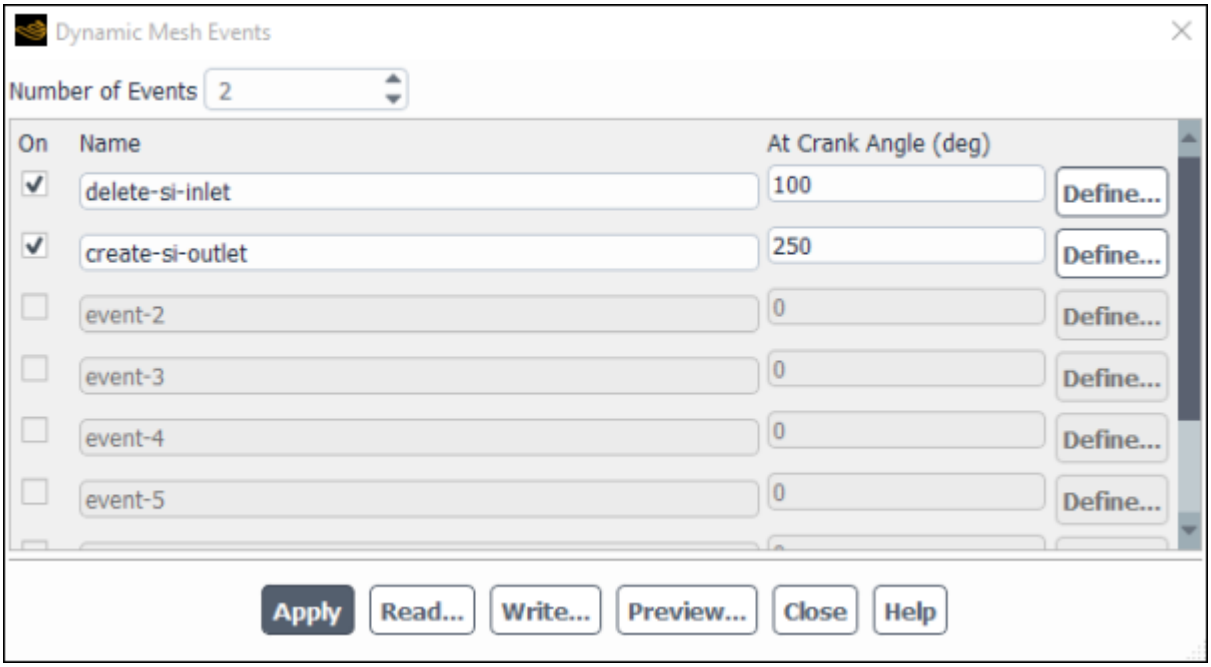
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.6.8.1. Procedure for Defining Events

You can define the events using the Dynamic Mesh Events Dialog Box (p. 5135) (Figure 13.76: The Dynamic Mesh Events Dialog Box (p. 1895)).

Setup → Dynamic Mesh → Events...

Figure 13.76: The Dynamic Mesh Events Dialog Box



The procedure for defining events is as follows:

1. Increase the **Number of Events** value to the number of events you want to specify. As this value is increased, additional event entries in the dialog box will become editable.

2. Enable the check box next to the first event and enter a name for the event under the **Name** heading.

3. Specify either the time or the crank angle at which you want the event to occur.

   For in-cylinder flows, specify the crank angle at which you want the event to occur under **At Crank Angle**.

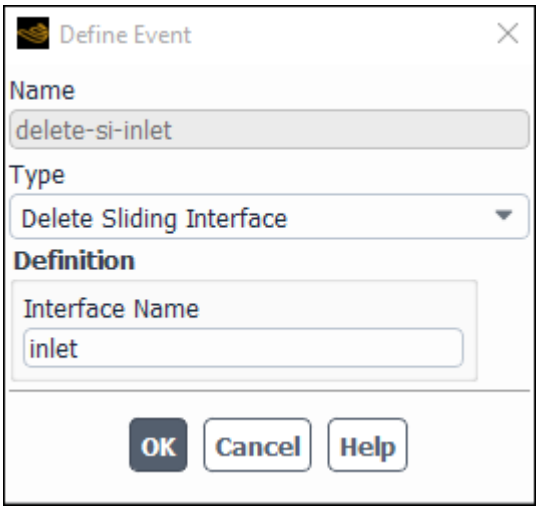
   For non-in-cylinder flows, specify the time (in seconds) at which you want the event to occur under **At Time**.

   It is not necessary to specify the events in order of increasing time or crank angle, but it may be easier to keep track of events if you specify them in the order of increasing time or angle.

4. Click the **Define...** button to open the Define Event Dialog Box (p. 5137) (Figure 13.77: The Define Event Dialog Box (p. 1896)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.77: The Define Event Dialog Box



5. In the Define Event Dialog Box (p. 5137), choose the type of event by selecting **Change Zone Type**, **Copy Zone BC**, **Activate Cell Zone**, **Deactivate Cell Zone**, **Create Sliding Interface**, **Delete Sliding Interface**, **Change Motion Attribute**, **Change Time Step Size**, **Change Under-Relaxation Factors**, **Insert Boundary Zone Layer**, **Remove Boundary Zone Layer**, **Insert Interior Zone Layer**, **Remove Interior Zone Layer**, **Insert Cell Layer**, **Remove Cell Layer**, **Execute Command**, **Replace Mesh**, **Inert EGR Reset**, or **Diesel Unsteady Flamelet Reset** in the **Type** drop-down list. These event types and their definitions are described later in this section.

6. Repeat steps 2–5 for the other events, if relevant.

7. Click **Apply** in the Dynamic Mesh Events Dialog Box (p. 5135) after you finish defining all events.

8. To play the events to check that they are defined correctly, click the **Preview...** button in the Dynamic Mesh Events Dialog Box (p. 5135). This displays the Events Preview Dialog Box (p. 5139).

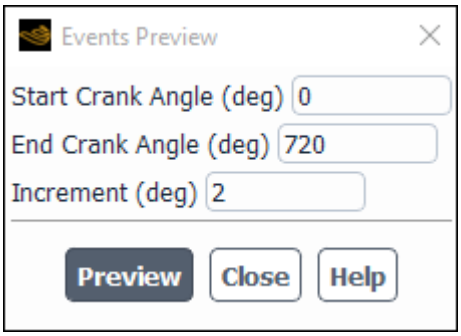
   For in-cylinder flows, you use the Events Preview Dialog Box (p. 5139) (Figure 13.78: The Events Preview Dialog Box for In-Cylinder Flows (p. 1897)), to enter the crank angles at which you want to start and end the playback in the **Start Crank Angle** and **End Crank Angle** fields, respectively.

   For non-in-cylinder flows, you use the Events Preview Dialog Box (p. 5139) to enter the time at which you want to start and end the playback in the **Start Time** and **End Time** fields, respectively.

   Specify the size of the step to take during the playback in the **Increment** field. Click **Preview** to play back the events. Ansys Fluent will play the events at the time (or crank angle in the case of in-cylinder flows) specified for each event and report when each event occurs in the text (console) window.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.78: The Events Preview Dialog Box for In-Cylinder Flows



For in-cylinder simulations, you must specify the events for one complete engine cycle. In the subsequent cycles, the events are executed whenever

$$\theta_{event} = \theta_c \pm n\theta_{period} \tag{13.30}$$

where θ_{event} is the event crank angle, θ_c is the current crank angle calculated from Equation 13.26 (p. 1870), θ_{period} is the crank angle period for one cycle, and n is some integer.

As an example, for in-cylinder simulations, you are not required to specify the event crank angle to correspond exactly to the current crank angle calculated from Equation 13.26 (p. 1870). Ansys Fluent will execute an event if the current crank angle is between $\pm 0.5\Delta\theta$ where $\Delta\theta$ is the equivalent change in crank angle for the time step. For example, if the event preview is executed between crank angle of 340° and 1060° (crank period is 720°) using an increment of 1°, Ansys Fluent will report the following in the text window.

```
Execute Event: open-in-valve-left (defined at: 353.10, current angle: 353.00)
Execute Event: open-in-valve-right (defined at: 353.00, current angle: 353.00)
Execute Event: close-ex-valve-right (defined at: 355.60, current angle: 356.00)
Execute Event: close-ex-valve-left (defined at: 357.80, current angle: 358.00)
Execute Event: close-in-valve-left (defined at: 571.60, current angle: 572.00)
Execute Event: close-in-valve-right (defined at: 571.80, current angle: 572.00)
Execute Event: open-ex-valve-right (defined at: 137.10, current angle: 857.00)
Execute Event: open-ex-valve-left (defined at: 139.00, current angle: 859.00)
```

Notice that events defined at 137.10° and 139° are executed at 857° and 859°, respectively, because they satisfy the condition of Equation 13.30 (p. 1897).

13.6.8.2. Defining Events for In-Cylinder Applications

Ansys Fluent will automatically limit the valve lift values depending on the specified minimum valve lift value. However, the conversion of the sliding interface zones to walls (and vice versa) is accomplished via the in-cylinder events (see Defining Dynamic Mesh Events (p. 1894)). For example, if the exhaust valve closes at -5° before TDC position, you must define a **Delete Sliding Interface** event at the crank angle of -5°. You must define similar events for the intake valve opening (using the **Create Sliding Interface** event), the intake valve closing (**Delete Sliding Interface** event), and the exhaust valve opening (**Create Sliding Interface** event) at the respective crank angles.

For the current example, the exhaust valve is assumed to be open between 131° and 371° and the intake valve is open between at 345° and 584°.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.6.8.2.1. Events

Each of the available events is described below.

13.6.8.2.2. Changing the Zone Type

You can change the type of a zone to be a wall, or an interface, interior, fluid, or solid zone during your simulation. To change the type of a zone, select **Change Zone Type** in the **Type** drop-down list in the Define Event Dialog Box (p. 5137) (Figure 13.77: The Define Event Dialog Box (p. 1896)). Select the zone(s) that you want to change in the **Zone** list, and then select the new zone type in the **New Zone Type** drop-down list.

13.6.8.2.3. Copying Zone Boundary Conditions

You can copy boundary conditions from one zone to other zones during your simulation. If, for example, you have changed an inlet zone to type wall with the **Change Zone Type** event, you can set the boundary conditions of the new zone type by simply copying the boundary conditions from a known zone with the corresponding zone type.

To copy boundary conditions from one zone to another, select **Copy Zone BC** in the **Type** drop-down list in the Define Event Dialog Box (p. 5137) (Figure 13.77: The Define Event Dialog Box (p. 1896)). In the **From Zone** drop-down list, select the zone that has the conditions you want to copy. In the **To Zone(s)** list, select the zone or zones to which you want to copy the conditions.

Ansys Fluent will set *all* of the boundary conditions for the zones selected in the **To Zone(s)** list to be the same as the conditions for the zone selected in the **From Zone** list. (You cannot copy a subset of the conditions, such as only the thermal conditions.)

Note that you cannot copy conditions from external walls to internal (that is, two-sided) walls, or vice versa, if the energy equation is being solved, since the thermal conditions for external and internal walls are different.

13.6.8.2.4. Activating a Cell Zone

To activate a cell zone, select **Activate Cell Zone** in the **Type** drop-down list in the Define Event Dialog Box (p. 5137) (Figure 13.77: The Define Event Dialog Box (p. 1896)), then select the zone that you want to activate in the **Zone(s)** list. For more information, see Replacing, Deleting, Deactivating, and Activating Zones (p. 1275).

13.6.8.2.5. Deactivating a Cell Zone

To deactivate a cell zone, select **Deactivate Cell Zone** in the **Type** drop-down list in the Define Event Dialog Box (p. 5137) (Figure 13.77: The Define Event Dialog Box (p. 1896)), then select the zone that you want to deactivate in the **Zone(s)** list.

Only deactivated zones can be activated. When a zone is deactivated, Ansys Fluent skips the zone during the calculations. For more information, see Replacing, Deleting, Deactivating, and Activating Zones (p. 1275).

13.6.8.2.6. Creating a Sliding Interface

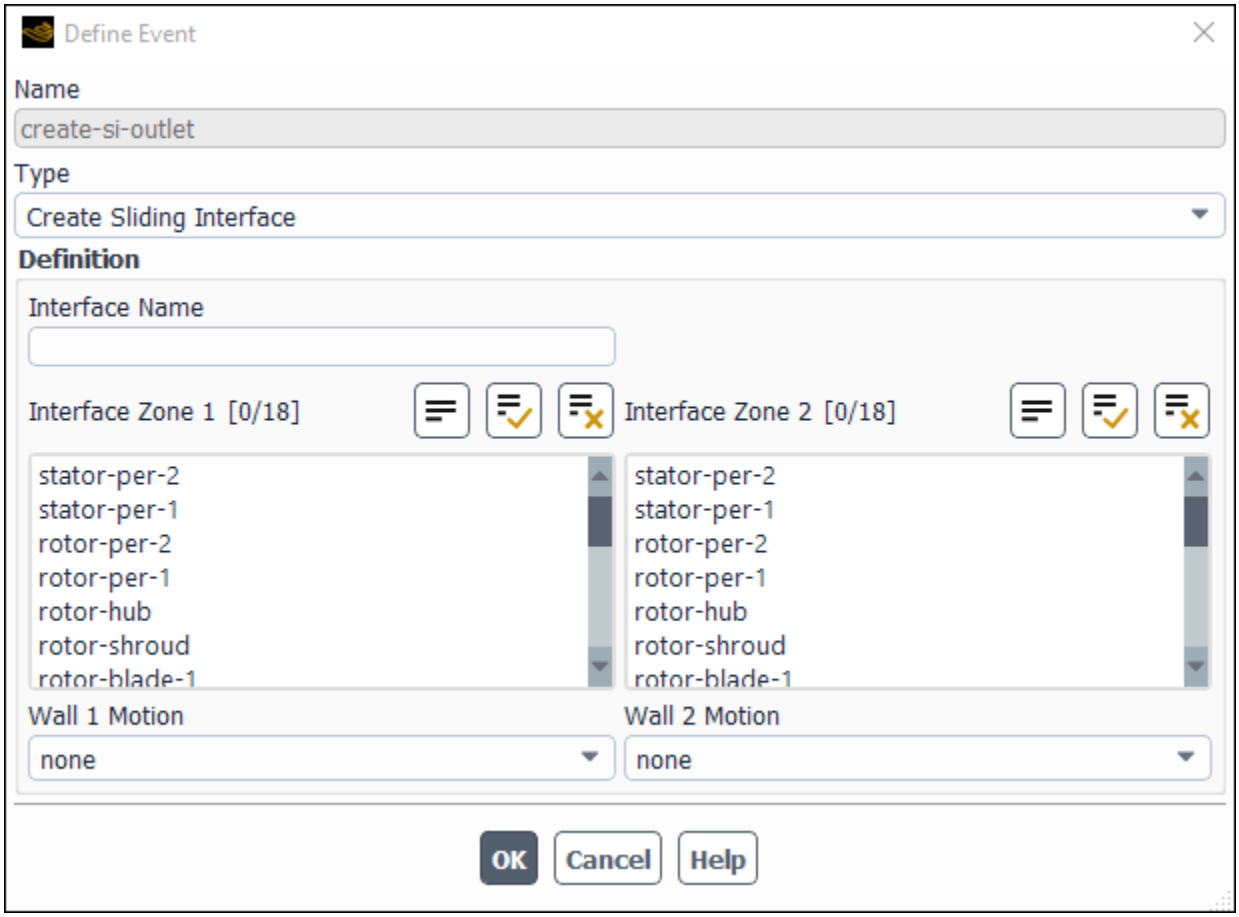
To create a sliding interface during your simulation, select **Create Sliding Interface** in the **Type** drop-down list in the Define Event Dialog Box (p. 5137) (Figure 13.79: The Define Event Dialog Box

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

for the Creating Sliding Interface Option (p. 1899)). Enter a name for the sliding interface in the **Interface Name** field. Select the zones on either side of the interface in the **Interface Zone 1** and **Interface Zone 2** drop-down lists.

You have the option to select any number of zones listed under each of the interface zones. Ansys Fluent calculates intersections between all possible combinations of the left and right side of the interfaces, allowing you more flexibility in terms of creating zones and defining the interfaces.

Figure 13.79: The Define Event Dialog Box for the Creating Sliding Interface Option



Important:

If Ansys Fluent finds another interface with the same name as defined in the event, then the old interface will be deleted and a new one created as defined in the dynamic mesh event.

If the interface zones that you selected above do not overlap each other completely, the non-overlapped regions on each interface zones are put into separate wall zones by Ansys Fluent. If these wall zones (that is, non-overlapped regions) have motion attributes associated with them, their motion can only be specified by copying the motion from another dynamic zone by selecting the appropriate dynamic zones in the **Wall 1 Motion** and **Wall 2 Motion** drop-down lists, respectively.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that you do not have to change the boundary type from wall to interface. When the **Create Sliding Interface** event is executed, Ansys Fluent will automatically change the boundary type of the face zones selected in **Interface Zone 1** and **Interface Zone 2** to type interface before the sliding interface is created.

13.6.8.2.7. Deleting a Sliding Interface

To delete a sliding interface that has been created earlier in your simulation, select **Delete Sliding Interface** in the **Type** drop-down list in the Define Event Dialog Box (p. 5137) (Figure 13.77: The Define Event Dialog Box (p. 1896)). Enter the name of the sliding interface to be deleted in the **Interface Name** field.

As with the **Create Sliding Interface** event, Ansys Fluent will automatically change the corresponding interface zones to wall. However, you may want to use the **Copy Zone BC** event to set any boundary conditions that are not the default conditions that Ansys Fluent assumes.

13.6.8.2.8. Changing the Motion Attribute of a Dynamic Zone

To change the motion attribute of a dynamic zone during your in-cylinder calculation, select **Change Motion Attribute** in the **Type** drop-down list in the Define Event Dialog Box (p. 5137) (Figure 13.77: The Define Event Dialog Box (p. 1896)). Select the **Attribute** (**smoothing** or **remeshing**) and set the appropriate **Status** (**enable** or **disable**). Select the corresponding dynamic zones for which you want to change the motion attributes in the **Dynamic Zones** list.

The **smoothing** attribute is used to enable or disable smoothing of nodes on selected deforming face zones and the **remeshing** attribute is used to enable and disable face remeshing on selected deforming face zones.

13.6.8.2.9. Changing the Time Step Size

To change the time step size at some point during the simulation, select **Change Time Step Size** in the **Type** drop-down list in the Define Event Dialog Box (p. 5137). Specify the new physical time step size by entering the new **Time Step Size** in seconds.

For in-cylinder simulations, specify the new physical time step size by entering the new **Crank Angle Step Size** value in degrees. The physical time step size is calculated from

$$\Delta t = \frac{\Delta \theta_c}{\Omega_{shaft}} \tag{13.31}$$

13.6.8.2.10. Changing the Under-Relaxation Factor

To change one or more under-relaxation factors, select **Change Under-Relaxation Factor** in the **Type** drop-down list in the Define Event Dialog Box (p. 5137) (Figure 13.77: The Define Event Dialog Box (p. 1896)). Select the under-relaxation factor that you want to change, and assign a new value to it in the **Under-Relaxation Factors** list. For more information on setting under-relaxation factors, see Setting Under-Relaxation Factors (p. 3589).

13.6.8.2.11. Inserting a Boundary Zone Layer

To insert a new cell zone layer as a separate cell zone adjacent to a boundary, select **Insert Boundary Zone Layer** in the **Type** drop-down list in the Define Event Dialog Box (p. 5137). Specify the **Base Dynamic Zone**, from which the layer of cells is to be created, and the **Side Dynamic**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Zone, which represents the deforming face zone adjacent to the **Base Dynamic Zone** before the layer is inserted. The new cell zone will inherit the boundary conditions of the cell zone adjacent to the **Base Dynamic Zone** before the layer is inserted.

Note that a new cell layer can be inserted only from a one-sided **Base Dynamic Zone**. You cannot insert a new cell layer from an interior face zone.

Figure 13.80: Boundary Zone Before Insertion (p. 1901) and Figure 13.81: Boundary Zone After Insertion (p. 1901) illustrate the insertion of a boundary zone layer. In both figures, the circular face at the top of the cylinder is the base dynamic zone.

Figure 13.80: Boundary Zone Before Insertion

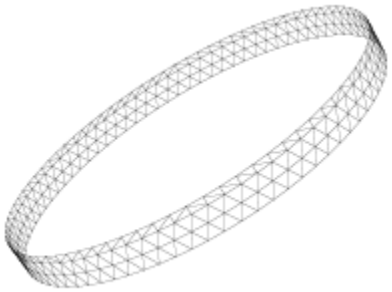


Figure 13.81: Boundary Zone After Insertion

13.6.8.2.12. Removing a Boundary Zone Layer

To remove the cell zone layer inserted using the **Insert Boundary Zone Layer** event, select **Remove Boundary Zone Layer** in the **Type** drop-down list in the Define Event Dialog Box (p. 5137). Specify the same **Base Dynamic Zone** that you used when you defined the insert boundary layer event.

Note that a cell layer can be removed only from a one-sided **Base Dynamic Zone**.

13.6.8.2.13. Inserting an Interior Zone Layer

To insert a new zone layer as a separate cell zone adjacent to the internal side of a boundary, select **Insert Interior Zone Layer** in the **Type** drop-down list in the Define Event Dialog Box (p. 5137). Specify the **Base Dynamic Zone** and the **Side Dynamic Zone** as described in the **Insert Boundary Zone Layer** event. You also must specify the names of the new interior face zones

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

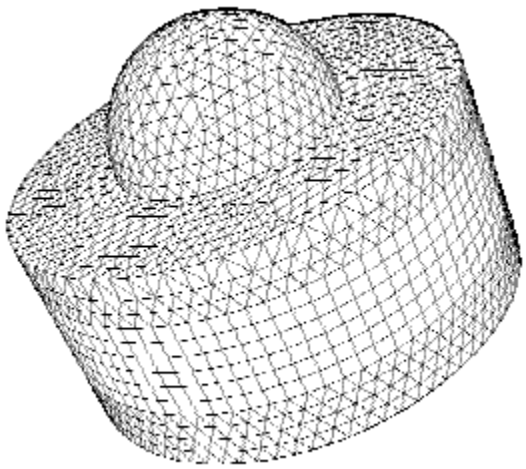
(**Internal Zone 1 Name** and **Internal Zone 2 Name**) that will be created after the cell zone layer is created by Ansys Fluent.

Ansys Fluent inserts the interior cell layer by splitting the cell zone adjacent to the **Base Dynamic Zone** with a plane. The position of the plane and the normal direction of the plane are implicitly defined by the cylinder origin and cylinder axis of the **Side Dynamic Zone**.

Figure 13.82: Interior Zone Before Insertion (p. 1902) and Figure 13.83: Interior Zone After Insertion (p. 1902) illustrate the insertion of an interior zone layer.

Figure 13.82: Interior Zone Before Insertion

Figure 13.83: Interior Zone After Insertion



13.6.8.2.14. Removing an Interior Zone Layer

To remove the zone layer inserted using the **Insert Interior Zone Layer** event, select **Remove Interior Zone Layer** in the **Type** drop-down list in the Define Event Dialog Box (p. 5137). Specify the same **Internal Zone 1 Name** and **Internal Zone 2 Name** that you used to define the **Insert Interior Zone Layer** event.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.6.8.2.15. Inserting a Cell Layer

To manually insert a new cell layer to the existing cell zone, select **Insert Cell Layer** in the **Type** drop-down list in the Define Event Dialog Box (p. 5137). Specify the **Adjacent Dynamic Face Zone** and the **Direction Parameter**. This can only work on zones that are suited for layering (see Applicability of the Dynamic Layering Method (p. 1840)).

13.6.8.2.16. Removing a Cell Layer

To manually remove a cell layer from an existing cell zone, select **Remove Cell Layer** in the **Type** drop-down list in the Define Event Dialog Box (p. 5137). Specify the **Adjacent Dynamic Face Zone** and the **Direction Parameter**. This can only work on zones that are suited for layering (see Applicability of the Dynamic Layering Method (p. 1840)).

13.6.8.2.17. Executing a Command

To execute a command, select **Execute Command** in the **Type** drop-down list in the Define Event Dialog Box (p. 5137) (Figure 13.77: The Define Event Dialog Box (p. 1896)). A command can be a series of text or Scheme commands, or a macro you have defined (or will define) using the Define Macro Dialog Box (p. 5203) (see Defining Macros (p. 3703)). Enter the series of commands or the name of the macro in the **Command** text-entry box.

Important:

If the command to be executed involves saving a file, see Saving Files During the Calculation (p. 3705) for important information.

13.6.8.2.18. Replacing the Mesh

To replace the mesh and interpolate existing data onto the new mesh during your simulation, select **Replace Mesh** from the **Type** drop-down list in the **Define Event** dialog box (Figure 13.77: The Define Event Dialog Box (p. 1896)). Then, specify the replacement mesh under **Mesh File**. Enable **Interpolate Data Across Zones** if necessary (see Replacing the Mesh (p. 1279) for details).

13.6.8.2.19. Resetting Inert EGR

To convert burnt gases at the end of the cycle to inert for the next cycle, select **Inert EGR Reset** from the **Type** drop-down list in the **Define Event** dialog box (Figure 13.77: The Define Event Dialog Box (p. 1896)). Specify the **Zone(s)**. For further details, see Resetting Inert EGR (p. 2525).

13.6.8.2.20. Diesel Unsteady Flamelet Reset

To simulate multiple-cycle internal combustion engines using diesel unsteady flamelets, select **Diesel Unsteady Flamelet Reset** from the **Type** drop-down list in the **Define Event** dialog box (Figure 13.77: The Define Event Dialog Box (p. 1896)). Specify the **Zone(s)**. This event is only applicable to and available with diesel unsteady flamelets with two or more flamelets. For further details, see Resetting Diesel Unsteady Flamelets (p. 2487). Note that **Diesel Unsteady Flamelet Reset** is generally preferable to **Inert EGR Reset** since an additional transport equation is avoided.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.6.8.3. Exporting and Importing Events

If you want to save the events you have defined to a file, click **Write...** in the Dynamic Mesh Events Dialog Box (p. 5135) and specify the **Event File** in The Select File Dialog Box (p. 931).

To read the events back into Ansys Fluent, click **Read...** in the Dynamic Mesh Events Dialog Box (p. 5135) and specify the **Event File** in The Select File Dialog Box (p. 931).

13.6.9. Specifying the Motion of Dynamic Zones

You must define the motion of the dynamic zones in your model. If the zone is a rigid body, you can use a motion definition, profile, or user-defined function (UDF) to define the motion of the rigid body, or use the six DOF solver. If the zone is a deforming zone, you can define the geometry and the parameters that control the face or zone remeshing, if applicable. For a zone that is deforming and moving at the same time, you can use a user-defined function to define the geometry and motion of the zone as they change with time.

13.6.9.1. General Procedure

You will specify the motion of the dynamic zones in your model using the **Dynamic Mesh Zones** dialog box

Setup → **Dynamic Mesh** → **Create/Edit...**

Details about specifying different types of motion are provided in this section.

13.6.9.1.1. Creating a Dynamic Zone

When you have completed the specification of a dynamic zone, click **Create** in the Dynamic Mesh Zones Dialog Box (p. 5140) to complete the specification and add the zone to the **Dynamic Mesh Zones** list.

13.6.9.1.2. Modifying a Dynamic Zone

If you want to make a change to the specification of a dynamic zone, double-click the zone in the **Dynamic Mesh Zones** list, change the specification, and then click **Create** in the Dynamic Mesh Zones Dialog Box (p. 5140) to update the specification.

13.6.9.1.3. Checking the Center of Gravity

If a dynamic zone has rigid body motion, you can view its current position and orientation of the center of gravity (with respect to initial data) by double-clicking the zone in the **Dynamic Mesh Zones** list and viewing the values under **Center of Gravity Location** and **Rigid Body Orientation**.

13.6.9.1.4. Deleting a Dynamic Zone

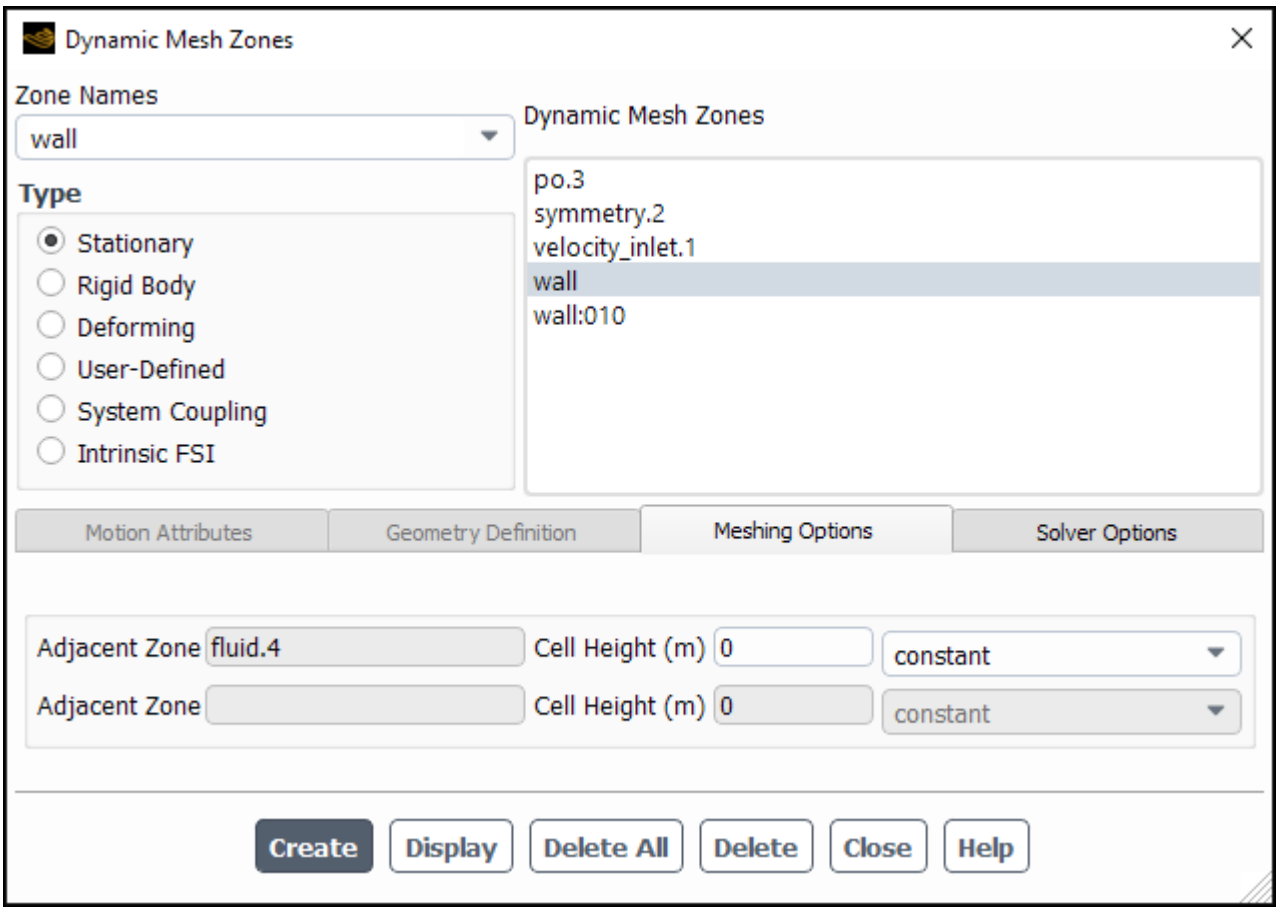
To delete a dynamic zone that you have specified, select the zone in the **Dynamic Mesh Zones** list, and click **Delete** or **Delete All**. The zone or zones will be removed from the **Dynamic Mesh Zones** list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.6.9.2. Stationary Zones

By default, if no motion (moving or deforming) attributes are assigned to a face or cell zone, then the zone is not considered when updating the mesh to the next time step. However, there are cases where an explicit declaration of a stationary zone is required. For example, if a cell zone is assigned some solid body motion, the positions of all nodes belonging to the cell zone will be updated even though some of the nodes may also be part of a non-moving boundary zone. An explicit declaration of a stationary zone excludes the nodes on these zones when updating the node positions.

Figure 13.84: The Dynamic Mesh Zones Dialog Box for a Stationary Zone



To define a stationary zone in your model, follow the steps below.

1. Select the stationary zone in the **Zone Names** drop-down list.

2. Select **Stationary** under **Type**.

3. If the stationary zone is a face zone, then define the **Cell Height** in the **Meshing Options** tab for any **Adjacent Zone** that is involved in dynamic layering. It does not influence the cell size for local or global remeshing. The **Cell Height** specifies the ideal height (h_{ideal} in Equation 13.15 (p. 1838) and Equation 13.16 (p. 1839) of the adjacent cells. Make a selection in the **Cell Height** drop-down menu to specify this value as either a **constant** or a compiled user-defined function.

   If you select the **constant** option, enter a value in the **Cell Height** text-entry box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you choose to use a compiled user-defined function to define an ideal cell height that varies as a function of time or crank angle, you must first define a `DEFINE_DYNAMIC_ZONE_PROPERTY` UDF. After you have compiled the UDF source file, built a shared library, and loaded it into Ansys Fluent, the name of the UDF library will be available for selection in the **Cell Height** drop-down list.

Refer to the Fluent Customization Manual for information about UDFs.

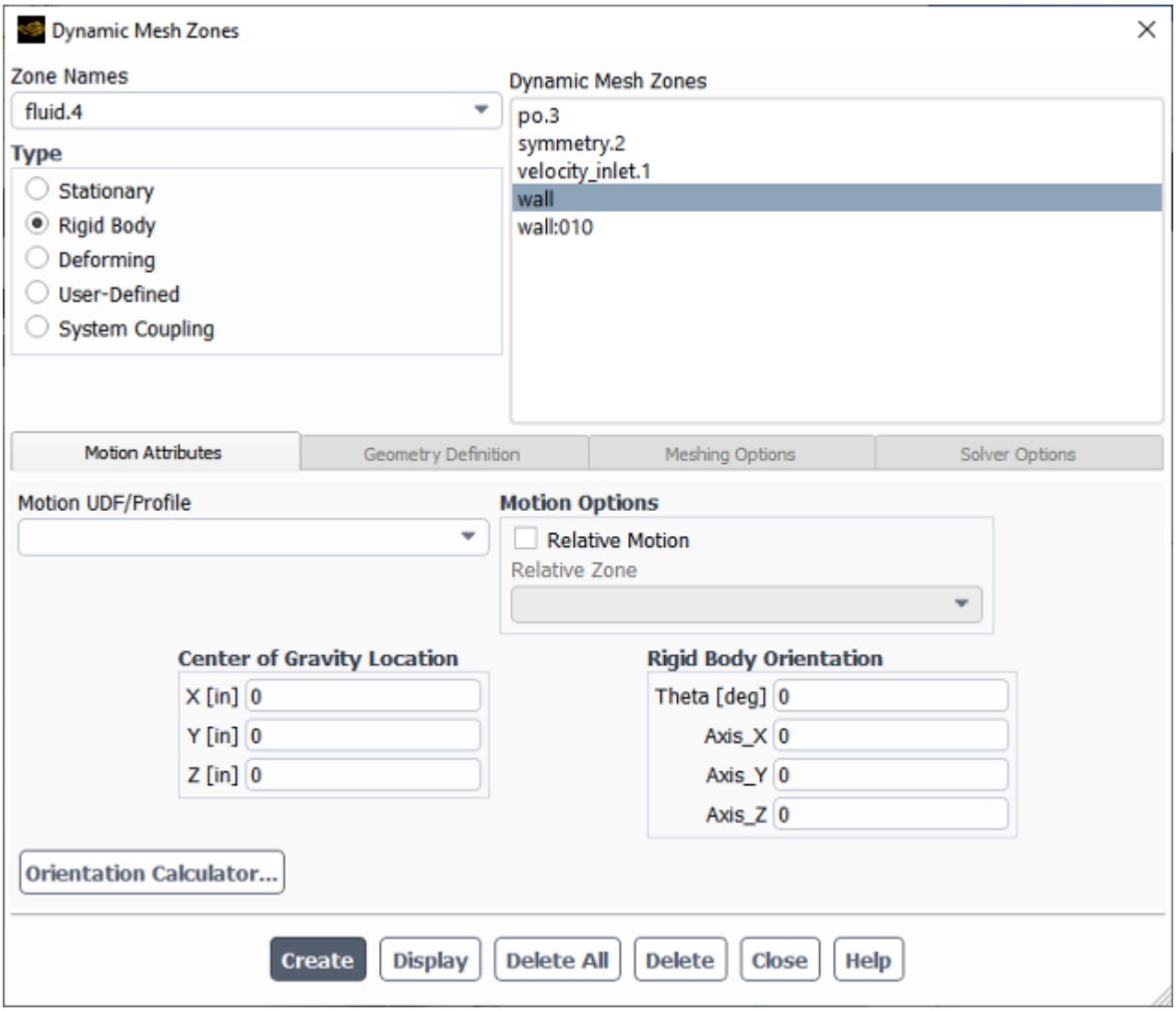
4. For cases with strong fluid-structure interaction, solution stabilization may help convergence for boundary zones. Solution stabilization can be set in the **Solver Options** tab. See Solution Stabilization for Dynamic Mesh Boundary Zones (p. 1918) for details.

5. Click **Create**.

13.6.9.3. Rigid Body Motion

To define a rigid-body zone in your model, follow the steps below.

Figure 13.85: The Dynamic Mesh Zones Dialog Box for a Rigid Body Motion



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the rigid body zone in the **Zone Names** drop-down list.

2. Select the **Rigid Body** option under **Type**.

3. If you want to specify the motion of the rigid body zone using a motion definition, profile, or user-defined function, then make a selection from the **Motion UDF/Profile** drop-down list in the **Motion Attributes** tab. Note that the names of motion definitions are preceded by **motion::** in this list. See Managing Motion Definitions (p. 1787) for information about motion definitions, see Profiles (p. 1564) and Solid-Body Kinematics (p. 1919) for information on profiles, and see the Fluent Customization Manual for information on user-defined functions. Using a motion definition may be advantageous, as it can be simpler to set up and allows you to use expressions to define the rigid body motion.

4. If you enabled the **In-Cylinder** option in the Dynamic Mesh Task Page (p. 5111), Ansys Fluent provides built-in functions in the **Motion UDF/Profile** drop-down list that can be useful for defining the rigid body motion of a piston. If you would like the motion of the piston to be a function of crank angle (that is, governed by Equation 13.25 (p. 1870)), then you should select ****piston-full****. For further information, see In-Cylinder Settings (p. 1868).

5. If you enabled the **Six DOF** option in the Dynamic Mesh Task Page (p. 5111), make sure that **On** is enabled in the **Six DOF** group box in the **Motion Attributes** tab (see Figure 13.87: The Dynamic Mesh Zones Dialog Box for a Rigid Body Motion Using the Six DOF Solver (p. 1909)), to ensure that the six DOF solver is being used. Then make a selection from the **Six DOF UDF/Properties** drop-down list; for details on creating the selections available in this list, Setting Rigid Body Motion Attributes for the Six DOF Solver (p. 1885).

   Note that the **Passive** option in the **Six DOF** group box is used when you do not want the forces and moments on the zone to be taken into consideration.

6. Specify the initial location of the center of gravity for the rigid body by entering coordinates in the **Center of Gravity Location** group box.

7. If you want a rigid body to move relative to another rigid body, then you must enable **Relative Motion** and specify the **Relative Zone**.

   Note:

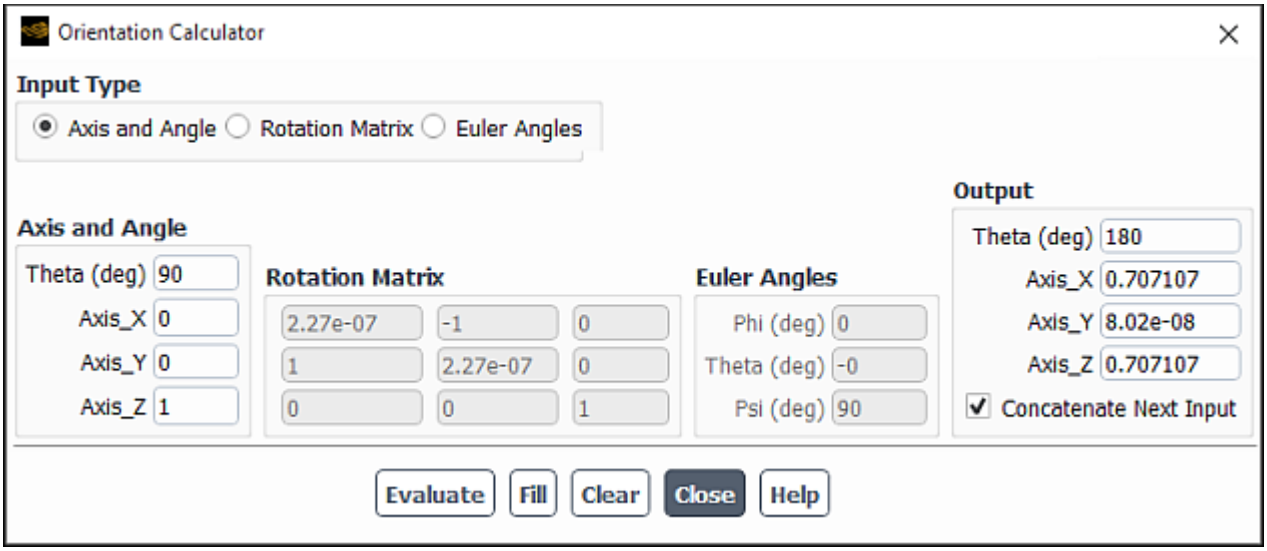
   You must create the relative rigid body zone before creating a rigid body that is dependent on that zone, if you are setting up the motion using the graphical user interface (GUI). The order in which you specify the zones does not matter if you set up the motion using the text user interface (TUI).

8. Specify the orientation of the rigid body in **Rigid Body Orientation**. You can consider the orientation of the rigid body as the rotation that it would take to rotate the global coordinate system into your desired rigid body coordinate system. For most cases, you will use these fields as a way to keep track of the object's current orientation, as Ansys Fluent will update your initial rigid body orientation values as the calculation progresses. The rigid body orientation is used, if needed, by the six DOF solver to evaluate the external load forces and moments (see `DEFINE_SDOF_PROPERTIES` in the *Fluent Customization Manual*), as well as when evaluating the relative motion of a dynamic zone with respect to another (see `DEFINE_CG_MOTION` in the *Fluent Customization Manual*).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The orientation is expressed in the axis angle format. Per the Euler Theorem, any set of rotations around a fixed point can be expressed via a single rotation about a specific axis. If you know the orientation of the rigid body in another format or as a set of consecutive rotations, you can use the **Orientation Calculator** dialog box and evaluate the equivalent axis angle format. To open this dialog box, click the **Orientation Calculator...** button; then you can perform the following steps:

Figure 13.86: Orientation Calculator Dialog Box



a. Enter a value, either in **Axis and Angle**, **Rotation Matrix**, or **Euler Angle** format, and click **Evaluate** to see the value in the output format.

   The calculator also converts the entered values into the other two "grayed-out" input formats.

   Note:

   There are several possible Euler angle representations. Fluent uses a vector defined by ϕ, θ, and ψ where: ψ is the rotation about the original Z axis, θ is the rotation about the rotated Y axis, and ϕ is the rotation about the rotated X axis.

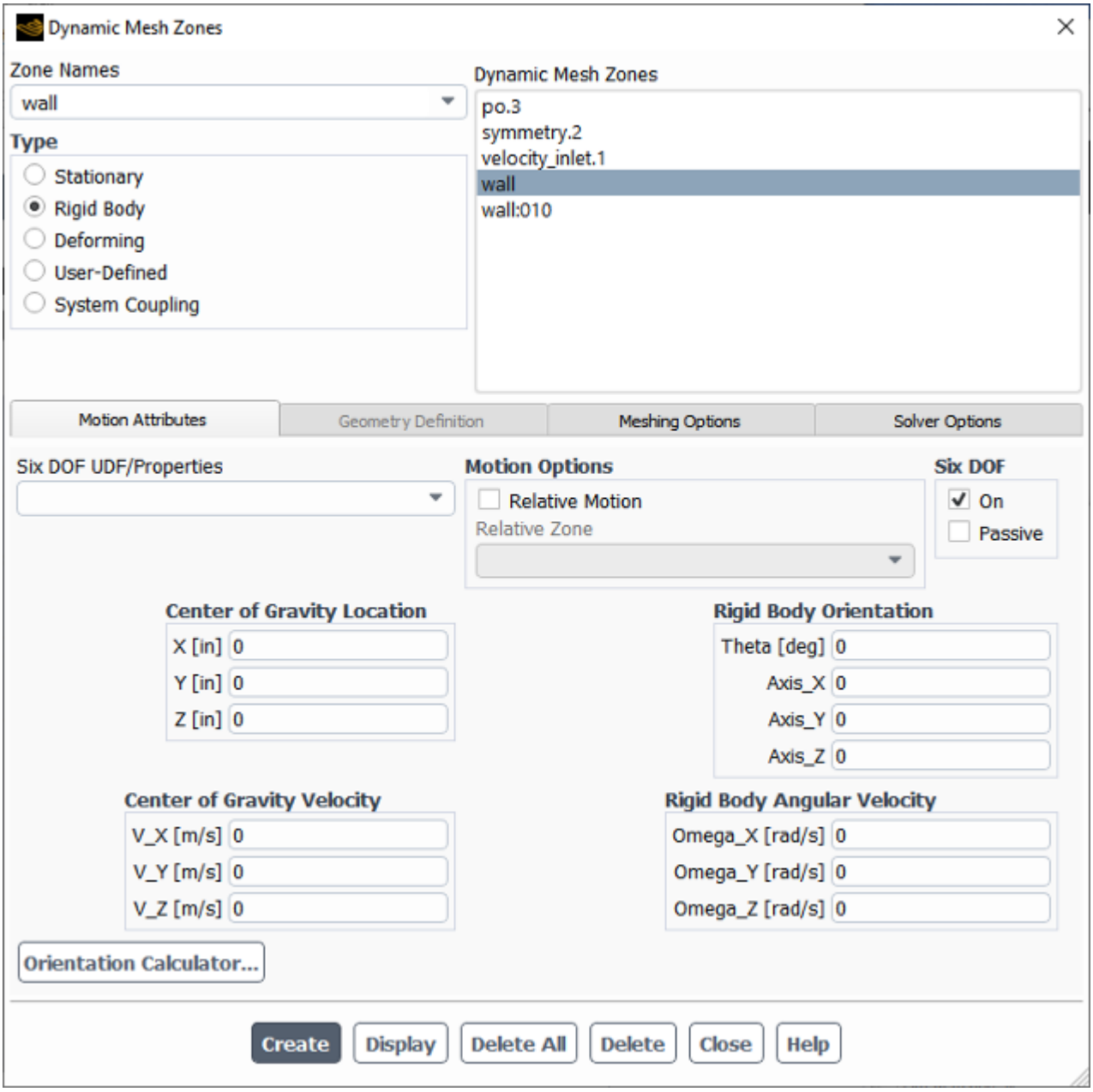
b. Enable **Concatenate Next Input** if you want the next input to be concatenated to the previous output.

c. Click **Fill** to use the current **Output** as your **Rigid Body Orientation** in the **Dynamic Mesh Zones** dialog box.

9. When using the six DOF solver, specify the velocity of the center of gravity with respect to the global coordinate system by entering the velocity of the center of gravity in **Center of Gravity Velocity**. Also, specify the angular velocity of the rigid body with respect to the global coordinate system by entering the angular velocity of the rigid body in **Rigid Body Angular Velocity**. Note that these fields are ignored for one DOF rotation and translation, respectively.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.87: The Dynamic Mesh Zones Dialog Box for a Rigid Body Motion Using the Six DOF Solver



10. If you are solving an in-cylinder problem, specify the direction of the reference axis of the valves or piston in **Valve/Piston Axis**.

    The current valve lift or piston stroke is automatically updated in **Lift/Stroke** when you click **Create** based on the parameters you have specified earlier when you first invoke the in-cylinder option.

11. By default, the boundary mesh motion is taken into consideration when imposing the physical boundary conditions, even if the boundary moves because of a moving adjacent cell zone and no dynamic zone has been created for the boundary. If this is not the desired behavior and you instead want to exclude the mesh motion from contributing to the boundary conditions, then

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

you need to enable the **Exclude Mesh Motion in Boundary Conditions** option for that boundary zone in the **Motion Attributes** tab.

12. If the rigid body zone is a face zone, specify the **Cell Height** for each **Adjacent Zone** in the **Meshing Options** tab. The **Cell Height** is the ideal cell height (h_{ideal} in Equation 13.15 (p. 1838) and Equation 13.16 (p. 1839) that is used by Ansys Fluent to determine when the prismatic layer next to the rigid body should be split or merged with the layer next to it. If the adjacent zone is tetrahedral or triangular, the ideal height is used by Ansys Fluent to determine if adjacent cells must be agglomerated for local remeshing. Make a selection in the **Cell Height** drop-down menu to specify this value as either a **constant** or a compiled user-defined function.

    If you select the **constant** option, enter a value in the **Cell Height** text-entry box.

    If you choose to use a compiled user-defined function to define an ideal cell height that varies as a function of time or crank angle, you must first define a `DEFINE_DYNAMIC_ZONE_PROPERTY` UDF. After you have compiled the UDF source file, built a shared library, and loaded it into Ansys Fluent, the name of the UDF library will be available for selection in the **Cell Height** drop-down list.

    Refer to the Fluent Customization Manual for information about UDFs.

13. If the dynamic zone is a face zone with an adjacent boundary layer mesh, you must apply boundary layer smoothing using the **Deform Adjacent Boundary Layer with Zone** option in the **Meshing Options** tab if you want the boundary layer to move with the moving face zone. For example, this option is necessary if you are applying face region remeshing with prism layers and you have not decomposed the mesh volume (see Face Region Remeshing with Wedge Cells in Prism Layers (p. 1854)). In such circumstances, the **Deform Adjacent Boundary Layer with Zone** option ensures that the base prism elements shown in Figure 13.45: Volume Decomposition for the Base of the Prism Layers (p. 1856) will move rigidly with the piston.

    Note that the boundary layer smoothing method is primarily intended for translational motion of the face zone. If significant rotation is applied to the face zone, the boundary layer cells may become skewed. For more information refer to Boundary Layer Smoothing Method (p. 1834).

14. For cases with strong fluid-structure interaction, solution stabilization may be necessary for boundary zones if the use of implicit mesh updating (described in Implicit Update Settings (p. 1888)) is insufficient to stabilize the solver due to the fluid motion. Solution stabilization options can be set in the **Solver Options** tab. See Solution Stabilization for Dynamic Mesh Boundary Zones (p. 1918) for details.

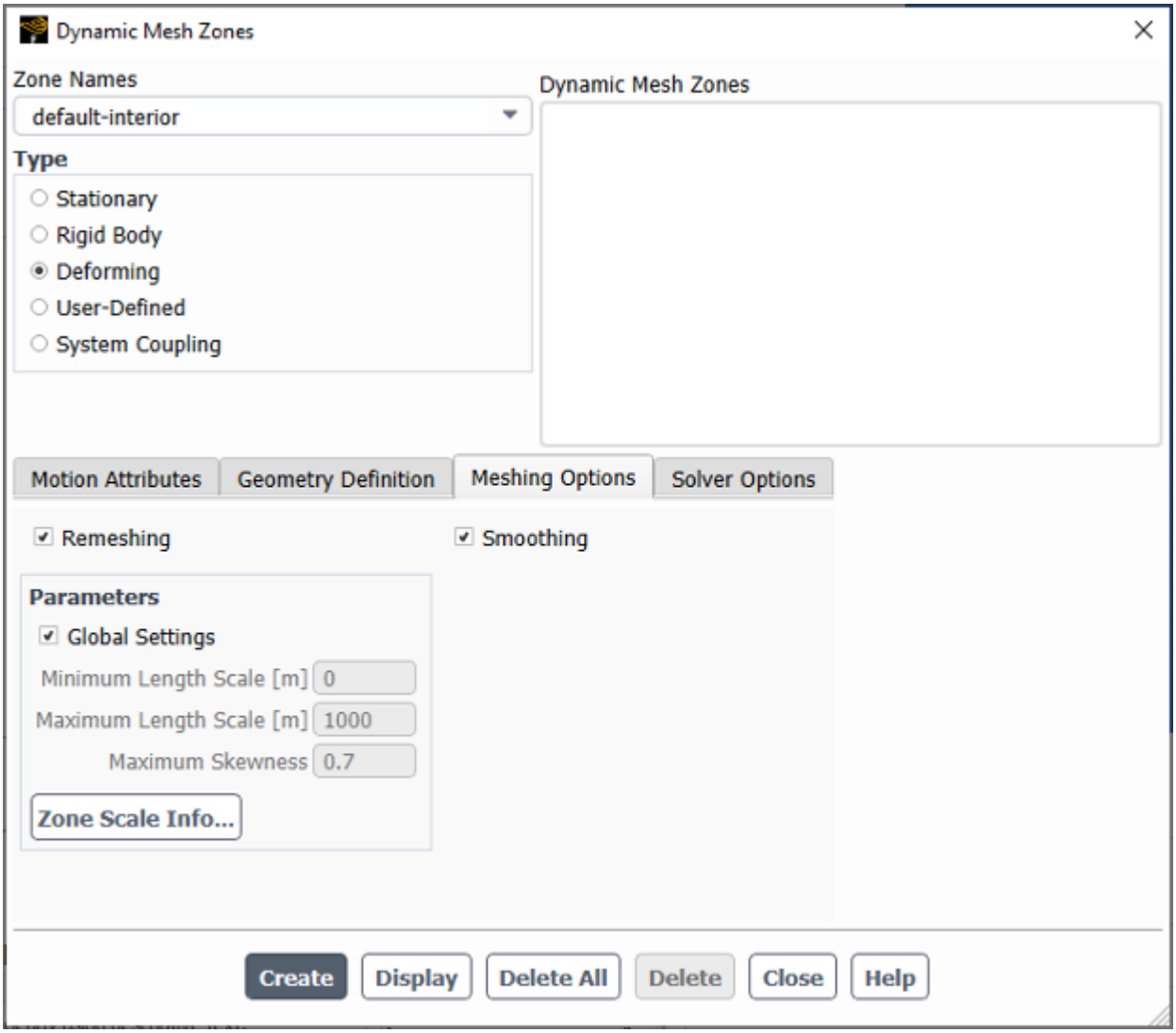
15. Click **Create**.

13.6.9.4. Deforming Motion

To define a deforming zone in your model, follow the steps below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.88: The Dynamic Mesh Zones Dialog Box for a Deforming Motion with Cell Zone Options



1. Select the deforming zone in the **Zone Names** drop-down list.

2. Select the **Deforming** option under **Type**.

3. By default, the boundary mesh motion is not taken into consideration when imposing the physical boundary conditions. If this is not the desired behavior and you instead want to include the mesh motion in the boundary conditions, then you need to disable the **Exclude Mesh Motion in Boundary Conditions** option for that boundary zone in the **Motion Attributes** tab. For example, you might want to disable this option when using the linearly elastic solid mesh smoothing method with the unspecified geometry option to not only smooth the mesh but to also propagate the motion from another boundary.

4. Specify the geometry of the deforming zone in the **Geometry Definition** tab.

   - If no geometry is available, select **faceted** in the **Definition** drop-down list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If the geometry is a plane, select **plane** in the **Definition** drop-down list. To define the plane, enter the position of a point on the plane in **Point on Plane** and the plane normal in **Plane Normal**.
- If the geometry is a cylinder, select **cylinder** in the **Definition** drop-down list. To define the cylinder, enter the **Cylinder Radius**, the **Cylinder Origin** and the **Cylinder Axis**.
- If the geometry is unspecified and mesh motion normal to the boundary is permissible, select **unspecified** in the **Definition** drop-down list. This option is only available if the linearly elastic solid mesh smoothing method or the radial basis function smoothing method is enabled (see Linearly Elastic Solid Based Smoothing Method (p. 1828) or Radial Basis Function Smoothing (p. 1830), respectively). Note that with the radial basis function smoothing method, if you have a non-conformal interface where both of the interface boundaries are deforming dynamic zones with unspecified geometry, then Fluent will ensure that the two interface zones stay connected (by projecting one of the interface zones onto the other); such behavior is only supported if the same cell zone is on either side of the non-conformal interface.
- If the geometry is described by a user-defined function, select **user-defined** in the **Definition** drop-down list and the appropriate user-defined functions in the **Geometry UDF** drop-down list. See the Fluent Customization Manual for information on user-defined functions.
- If the geometry is defined by an auxiliary geometry definition that you have created previously (for details, see Managing Auxiliary Geometry Definitions (p. 1791)), select it in the **Definition** drop-down list. The names of auxiliary geometry definitions are preceded by **geom::** in this list, in order to identify them.

  Note:

  Auxiliary geometry definitions of the **Reconstruction** type are not available for selection when **Remeshing** is enabled in the **Mesh Methods** list in the **Dynamic Mesh** task page.

  For 3D simulations, Ansys Fluent allows you to preserve features on the boundary zones that are important for the flow. Such features may be at the juncture of different boundary zones, or they may be on a single non-planar boundary zone. For any geometry definition (**faceted**, **plane**, **cylinder**, or **user-defined**), you can indicate whether you want to preserve features of a specific angular range by enabling **Feature Detection** under **Feature Options** and setting a threshold value for the **Feature Angle** in degrees. For more information (including how the **Feature Angle** is defined), see Feature Detection (p. 1868).

  When available, the geometry information is used to project nodes on the deforming zone after remeshing the face zone, or if nodes are moved through smoothing.

5. For deforming boundary and cell zones, specify the appropriate remeshing parameters in the **Meshing Options** tab.
   - You can locally enable **Remeshing** and define the settings in the group box under this option, including the following:

     Under **Parameters**, enabling the **Global Settings** option ensures that settings defined in the Mesh Method Settings Dialog Box (p. 5114) are used for this dynamic zone. Disabling this option

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

allows you to revise the **Minimum Length Scale**, **Maximum Length Scale** and/or **Maximum Skewness** locally. When setting these values, you should use the vital statistics of your zone found in the Zone Scale Info Dialog Box (p. 5150), which can be opened by clicking the **Zone Scale Info...** button.

For boundary zones, you may have additional **Options**: if you selected **Methods-Based Remeshing** in the Mesh Method Settings Dialog Box (p. 5114), you can specify the remeshing methods applied to this dynamic zone, including **Region** and **Local**.

- You can locally enable **Smoothing** and, if **Spring/Laplace/Boundary Layer** is selected for smoothing in the Mesh Method Settings Dialog Box (p. 5114), define the settings in the group box under this option, including the following:

  If you selected a cell zone, enabling the **Global Settings** option ensures that the element type defined in the Mesh Smoothing Parameters Dialog Box (p. 5118) is used for this dynamic zone. Disabling it allows you to select a different element type locally.

  If you selected a boundary zone, you can specify whether the smoothing method is spring based or Laplacian.

6. For cases with strong fluid-structure interaction, solution stabilization may help convergence for boundary zones. Solution stabilization can be set in the **Solver Options** tab. See Solution Stabilization for Dynamic Mesh Boundary Zones (p. 1918) for details.

7. Click **Create**.

13.6.9.5. User-Defined Motion

For a zone that is deforming and/or moving, you can define the position of each node on the general deforming / moving zone using a user-defined function (UDF). To define a moving / deforming zone, follow the steps below.

1. Select the moving and deforming zone in the **Zone Names** drop-down list.

2. Select the **User-Defined** option under **Type**.

3. In the **Motion Attributes** tab, select the user-defined function that defines the geometry and motion of the zone from the **Mesh Motion UDF/Profile** drop-down list. See the Fluent Customization Manual for information on user-defined functions used to specify user-defined motion.

4. By default, the boundary mesh motion is taken into consideration when imposing the physical boundary conditions. If this behavior is not desired and you instead want to exclude the mesh motion from contributing to the boundary conditions, then you need to enable the **Exclude Mesh Motion in Boundary Conditions** option for that boundary zone in the **Motion Attributes** tab. The following are examples of when you might want to exclude such motion:

   - You have created a dynamic zone for a cell zone that moves, but one or more of the boundaries of that cell zone should be treated as stationary.

   - You are using a `DEFINE_GRID_MOTION` UDF that defines nodal motion on a boundary that, for example, includes both rigid body motion (which should contribute to the boundary condition) and smoothing (which should not). With such UDFs, you must perform the following steps for that boundary:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– Enable the **Exclude Mesh Motion in Boundary Conditions** option.

– Define the rigid body motion in the **Momentum** tab of the **Wall** dialog box.

5. If you want the movement / deformation to be relative to the rigid body motion of a cell zone, then you can enable **Relative Motion** and select the cell zone from the **Relative Zone** list.

Note:

You must specify the movement / deformation in your UDF using `NODE_COORD_NEST`, rather than specify the final coordinates (as you would if you were NOT specifying it relative to a cell zone). For additional information, see `DEFINE_GRID_MOTION` in the *Fluent Customization Manual*.

You must create the relative rigid body zone before creating the user-defined zone if you are setting up the zones using the graphical user interface (GUI). The order in which you create the zones does not matter if you set up the zones using the text user interface (TUI).

---

6. For a face zone, you can specify the **Cell Height** in the **Meshing Options** tab for any **Adjacent Zone** that is involved in local remeshing or dynamic layering. The **Cell Height** specifies the ideal height (h_{ideal} in Equation 13.15 (p. 1838) and Equation 13.16 (p. 1839) of the adjacent cells. Make a selection in the **Cell Height** drop-down menu to specify this value as either a **constant** or a compiled user-defined function.

   If you select the **constant** option, enter a value in the **Cell Height** text-entry box.

   If you choose to use a compiled user-defined function to define an ideal cell height that varies as a function of time or crank angle, you must first define a `DEFINE_DYNAMIC_ZONE_PROPERTY` UDF. After you have compiled the UDF source file, built a shared library, and loaded it into Ansys Fluent, the name of the UDF library will be available for selection in the **Cell Height** drop-down list.

   Refer to the separate Fluent Customization Manual for information about UDFs.

7. If the dynamic zone is a face zone with an adjacent boundary layer mesh, and you want to use the boundary layer smoothing method (as described in Boundary Layer Smoothing Method (p. 1834)), enable the **Deform Adjacent Boundary Layer with Zone** option in the **Meshing Options** tab.

8. For a 3D boundary zone, you can allow local face remeshing by enabling the **Local** option (under **Remeshing Method**) in the **Meshing Options** tab. For details, see Local Face Remeshing Method (p. 1845). Enabling this option allows you to do the following:

   - You can revise the **Maximum Skewness** allowed for faces if you want to specify a local value for this dynamic zone that differs from the one you specified globally in the Mesh Method Settings Dialog Box (p. 5114). Note that the global setting may still be used if Ansys Fluent determines that your local setting is unreasonable.

   - You can preserve features, not only at the juncture of different boundary zones, but also on a single non-planar boundary zone. In the **Geometry Definition** tab, you can indicate

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

whether you want to include features of a specific angular range by enabling **Feature Detection** under **Feature Options** and setting a threshold value for the **Feature Angle** in degrees. For more information (including how the **Feature Angle** is defined), see Feature Detection (p. 1868).

9. For cases with strong fluid-structure interaction, solution stabilization may help convergence for boundary zones. Solution stabilization can be set in the **Solver Options** tab. See Solution Stabilization for Dynamic Mesh Boundary Zones (p. 1918) for details.

10. Click **Create**.

11. If this zone is part of a mesh interface and only one side undergoes user-defined motion, then by default that motion will be transferred to the other side in order to maintain the coupling. You can change the method by which the displacement of the passive nodes is calculated or disable this transfer altogether, as described in Transferring Motion Across a Mesh Interface (p. 1205).

13.6.9.5.1. Specifying Boundary Layer Deformation Smoothing

For a boundary layer that deforms according to the adjacent face zone, the zone that is deforming and moving is usually defined using a user-defined function (UDF), as described in User-Defined Motion (p. 1913). To define a moving and deforming boundary layer, perform the steps that follow.

If the boundary layer borders a face zone that is only moving and is not deforming, you should consider applying rigid body motion to the face zone (see Rigid Body Motion (p. 1906)) rather than user-defined motion, as rigid body motion usually involves a simpler definition.

1. Set up the moving and deforming zone.

   a. Select the moving and deforming zone in the **Zone Names** drop-down list.

   b. Select **User-Defined** from the **Type** list.

   c. In the **Motion Attributes** tab, select the user-defined function that defines the geometry and motion of the zone from the **Mesh Motion UDF/Profile** drop-down list.

   d. In the **Meshing Options** tab, enable the **Deform Adjacent Boundary Layer with Zone** option.

   e. Click **Create**.

2. Set up a deforming dynamic zone for the fluid zone that contains the boundary layer.

   a. Select the fluid zone that contains the boundary layer from the **Zone Names** drop-down list.

   b. Select **Deforming** from the **Type** list.

   c. In the **Meshing Options** tab, ensure that **Remeshing** and **Smoothing** are enabled.

   d. Click **Create**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. If the fluid zone set up in the step 2 consists entirely of the boundary layer elements, set up a deforming dynamic zone for the neighboring fluid zone. This step is necessary because the deforming boundary layer will deform the adjacent cells.

   a. Select the fluid zone that neighbors the boundary layer zone from the **Zone Names** drop-down list.

   b. Select **Deforming** from **Type** list.

   c. In the **Meshing Options** tab, enable **Smoothing** and **Remeshing** in the **Methods** group box.

   d. Click **Create**.

13.6.9.6. System Coupling Motion

For a zone that is involved in a system coupling, the motion is defined by the application that Ansys Fluent is coupled with on this zone. For more details about setting up a simulation with system coupling see Performing System Coupling Simulations Using Fluent (p. 4647) and the System Coupling User's Guide. To define a system coupling region, follow the steps below.

1. Select the moving and deforming zone in the **Zone Names** drop-down list.

2. Select the **System Coupling** option under **Type**.

3. You must specify the **Cell Height** in the **Meshing Options** tab for any **Adjacent Zone** that is involved in local remeshing or dynamic layering. The **Cell Height** specifies the ideal height (h_{ideal} in Equation 13.15 (p. 1838) and Equation 13.16 (p. 1839) of the adjacent cells. Make a selection in the **Cell Height** drop-down menu to specify this value as either a **constant** or a compiled user-defined function.

   If you select the **constant** option, enter a value in the **Cell Height** text-entry box.

   If you choose to use a compiled user-defined function to define an ideal cell height that varies as a function of time or crank angle, you must first define a `DEFINE_DYNAMIC_ZONE_PROPERTY` UDF. After you have compiled the UDF source file, built a shared library, and loaded it into Ansys Fluent, the name of the UDF library will be available for selection in the **Cell Height** drop-down list.

   Refer to the separate Fluent Customization Manual for information about UDFs.

4. For cases with strong fluid-structure interaction, solution stabilization may be necessary to achieve convergence for boundary zones undergoing system coupling motion. Solution stabilization options can be set in the **Solver Options** tab. See Solution Stabilization for Dynamic Mesh Boundary Zones (p. 1918) for details.

5. Click **Create**.

6. If this zone is part of a mesh interface and only one side undergoes system coupling motion, then by default that motion will be transferred to the other side in order to maintain the coupling. You can change the method by which the displacement of the passive nodes is calcu-

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

lated or disable this transfer altogether, as described in Transferring Motion Across a Mesh Interface (p. 1205).

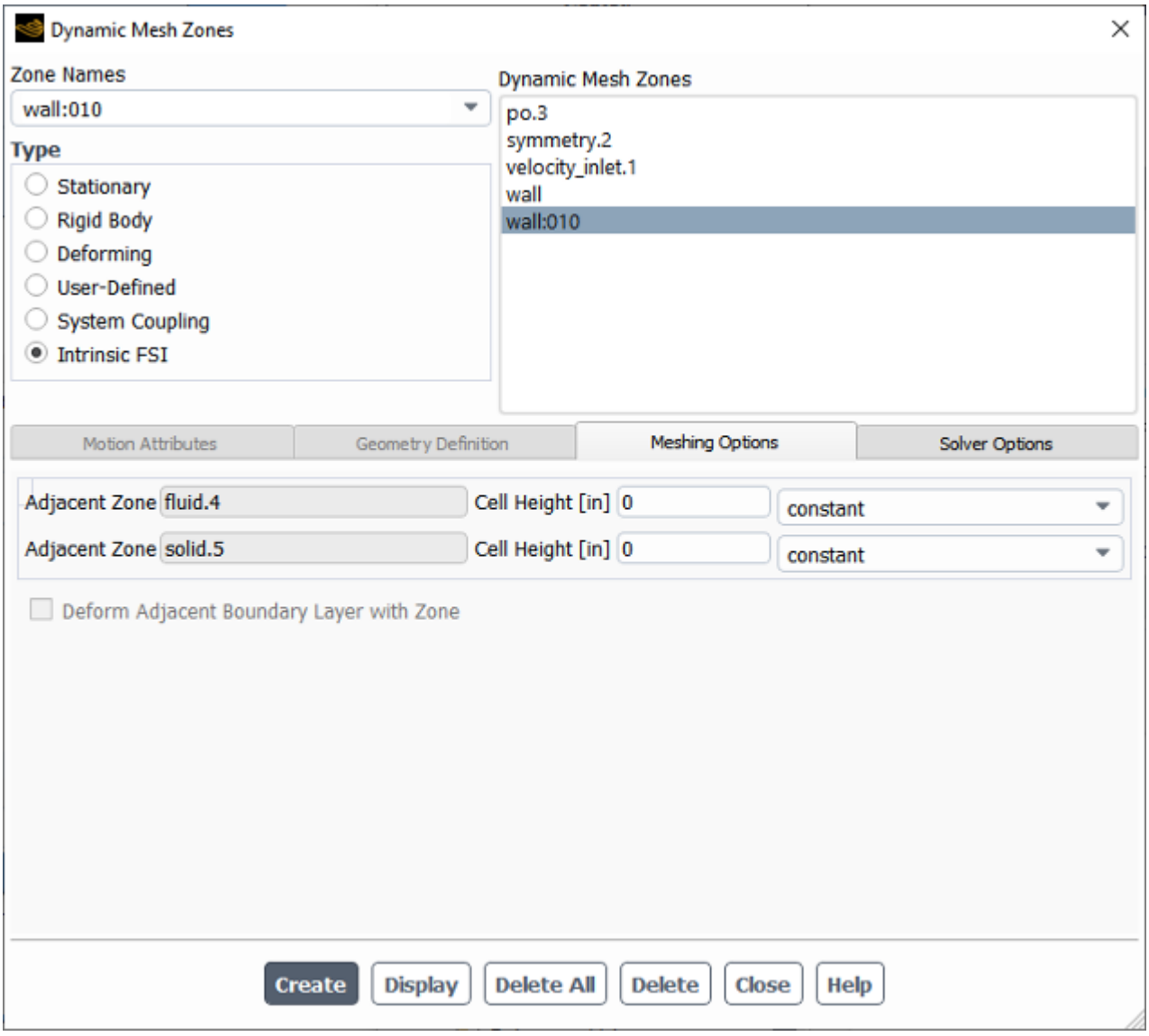
Note:

If the **System Coupling** option is enabled, and Ansys Fluent is not involved with a system coupling simulation, then this zone type behaves in the same way as a stationary zone.

13.6.9.7. Intrinsic FSI Motion

For a two-way intrinsic fluid-structure interaction (FSI) simulation, you can specify that a wall zone between a fluid and solid cell zone deforms according to the deformation of the adjacent solid zone by performing the following steps. For more details about setting up an intrinsic FSI simulation, see Setting Up an Intrinsic Fluid-Structure Interaction (FSI) Simulation (p. 3229).

Figure 13.89: The Dynamic Mesh Zones Dialog Box for an Intrinsic FSI Zone



1. Select a wall zone that is between a fluid and solid cell zone in the **Zone Names** drop-down list. Note that this zone must be the side of a two-sided wall (that is, the wall or wall-shadow) that is immediately adjacent to the fluid cell zone (as indicated by the **Adjacent Cell Zone** field in the **Wall** dialog box).

2. Select **Intrinsic FSI** from the **Type** list. Note that this type is only available when you have selected a model in the Structural Model Dialog Box (p. 4869).

3. For a face zone, you can specify the **Cell Height** in the **Meshing Options** tab for any **Adjacent Zone** that is involved in local remeshing or dynamic layering. The **Cell Height** specifies the ideal height (h_{ideal} in Equation 13.15 (p. 1838) and Equation 13.16 (p. 1839) of the adjacent cells. Make a selection in the **Cell Height** drop-down menu to specify this value as either a **constant** or a compiled user-defined function.

   If you select the **constant** option, enter a value in the **Cell Height** text-entry box.

   If you choose to use a compiled user-defined function to define an ideal cell height that varies as a function of time or crank angle, you must first define a `DEFINE_DYNAMIC_ZONE_PROPERTY` UDF. After you have compiled the UDF source file, built a shared library, and loaded it into Ansys Fluent, the name of the UDF library will be available for selection in the **Cell Height** drop-down list.

   Refer to the separate Fluent Customization Manual for information about UDFs.

4. If the dynamic zone is a face zone with an adjacent boundary layer mesh, and you want to use the boundary layer smoothing method (as described in Boundary Layer Smoothing Method (p. 1834)), enable the **Deform Adjacent Boundary Layer with Zone** option in the **Meshing Options** tab.

5. For cases with strong fluid-structure interaction, solution stabilization may help convergence for boundary zones. Solution stabilization can be set in the **Solver Options** tab. See Solution Stabilization for Dynamic Mesh Boundary Zones (p. 1918) for details.

6. Click **Create**.

13.6.9.8. Solution Stabilization for Dynamic Mesh Boundary Zones

For cases with strong fluid-structure interaction, stabilization is achieved through a boundary source coefficient introduced in the continuity equation, designed to improve the diagonal dominance of the matrix system in the cells adjacent to dynamic mesh boundary zones. Two methods for this boundary source coefficient are available:

- volume-based

  This method uses the cell volume V to re-scale the diagonal entry of the linear matrix system corresponding to the discretized continuity equation (a_{ij}) as $a_{ij,s}$:

  $$a_{ij,s}=a_{ij}+KV \quad \text{if } i=j \forall i,j \in \{1,2,...,n\} \tag{13.32}$$

- coefficient-based

  This method directly re-scales the diagonal entry of the linear matrix system corresponding to the discretized continuity equation (a_{ij}) as $a_{ij,s}$:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$a_{ij,s}=a_{ij}+Ka_{ij} \quad \text{if } i=j \forall i,j \in \{1,2,...,n\} \tag{13.33}$$

where K is the scale factor and n is the number of cells. Note that the value you choose for K will only affect the rate of convergence, and will not affect the converged solution.

Note:

While solution stabilization is available for all types of dynamic mesh boundary zones, it is more commonly needed for rigid body motion with the six degrees of freedom (six DOF) solver or for system coupling motion. It is not recommended for boundary zones of type interior, as it can adversely affect convergence.

Solution stabilization can be enabled for a boundary zone by opening the **Solver Options** tab in the Dynamic Mesh Zones Dialog Box (p. 5140), enabling the **Solution Stabilization** option, and then defining the settings under **Stabilization Parameters**. Available settings include the **Scale Factor** (K) and the **Method**, which can be either **volume-based** or **coefficient-based** (default). The **Scale Factor** must be set to a nonzero value for the stabilization to have an effect.

The solver option settings can also be defined using the `define/dynamic-mesh/zones/create` text command, and a summary of these settings can be printed in the console for each dynamic zone by using the `define/dynamic-mesh/zones/list` text command.

13.6.9.9. Solid-Body Kinematics

Ansys Fluent uses solid-body kinematics if the motion is prescribed based on the position and orientation of the center of gravity of a moving object. This is applicable to both cell and face zones.

The motion of the solid-body can be specified either as a profile or as a user-defined function (UDF). A profile may be defined by the following profile fields:

- time (time)
- crank angle (angle) (in-cylinder flows only)
- position (x, y, z)
- linear velocity (v_x, v_y, v_z)
- angular velocity (ω_x, ω_y, ω_z)
- orientation (θ_x, θ_y, θ_z)

For in-cylinder simulations, the velocity profiles for valves can be expressed as a function of crank angle instead of time. In addition, transient boundary condition profiles can also be expressed as a function of crank angle instead of time. For more information about transient profiles, see Transient Cell Zone and Boundary Conditions (p. 1523).

Below are two examples of a profile format:

```
((movement_linear 3 point)
(time
0  1  2 )
(x
```

```
2  3  4 )
(v_y
0  -5  0 )
)

((movement_angular 3 point)
(time
0  1  2 )
(omega_x
2  3  4 )
)
```

For in-cylinder flows, crank angles can be included in transient tables as well as transient profiles, in a similar fashion to time. An example of a transient table using (crank) angle is as follows:

```
example 2 3 1
angle temperature
0  300
180 500
360 300
```

An example of a transient profile using (crank) angle is as follows:

```
((example transient 3 1)
(angle
0.000000e+00 1.800000e+02 3.600000e+02)
(temperature
3.000000e+02 5.000000e+02 3.000000e+02)
)
```

In addition to the motion description, you must also specify the starting location of the center of gravity and orientation of the solid body. In 2D cases (and 3D cases that do not use the six DOF solver), Ansys Fluent automatically updates the center of gravity position and orientation at every time step such that

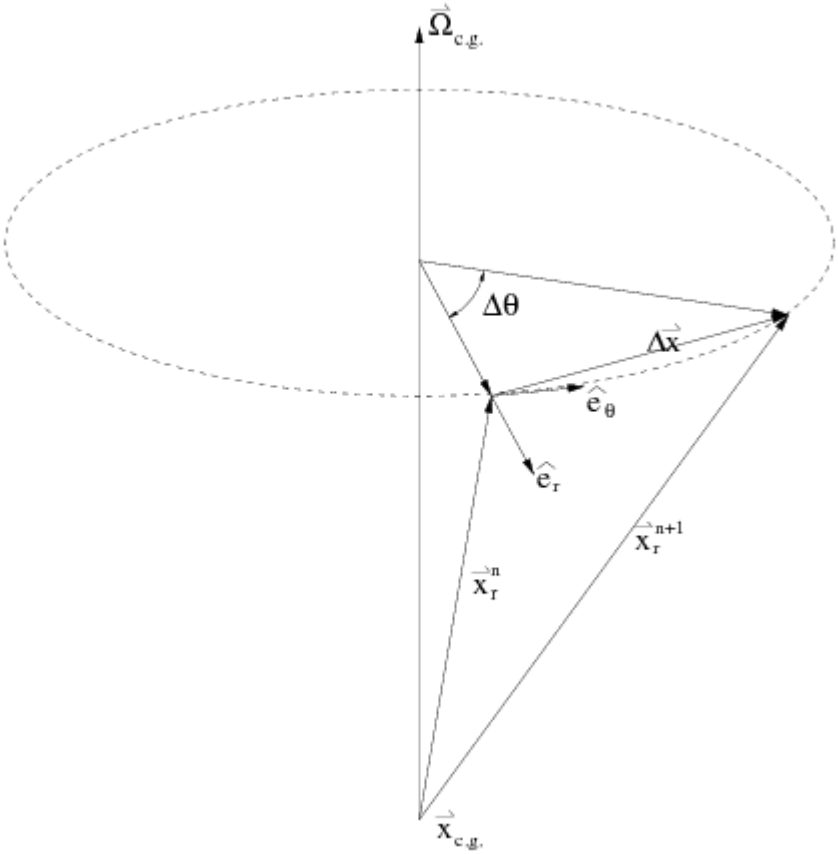
$$\begin{aligned}\vec{x}_{c.g.}^{\,n+1} &= \vec{x}_{c.g.}^{\,n} + \vec{v}_{c.g.}\Delta t \\ \vec{\theta}_{c.g.}^{\,n+1} &= \vec{\theta}_{c.g.}^{\,n} + \vec{\Omega}_{c.g.}\Delta t\end{aligned} \tag{13.34}$$

where $\vec{x}_{c.g.}$ and $\vec{\theta}_{c.g.}$ are the position and orientation of the center of gravity, $\vec{v}_{c.g.}$ and $\vec{\Omega}_{c.g.}$ are the linear and angular velocities of the center of gravity. 3D, six DOF cases use a more complex form of Equation 13.34 (p. 1920) when updating θ.

Typically, $\vec{\theta}$ is chosen to be an appropriate set of Euler angles. In this case, the solid-body motion must be specified using a user-defined function (`DEFINE_CG_MOTION`).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.90: Solid Body Rotation Coordinates



The position vectors on the solid body are updated based on rotation about the instantaneous angular velocity vector $\vec{\Omega}_{c.g.}$. For a finite rotation angle $\Delta\theta = |\vec{\Omega}_{c.g.}|\Delta t$, the final position of a vector $\vec{x}_r$ on the solid body with respect to $\vec{x}_{c.g.}$ can be expressed as (See Figure 13.90: Solid Body Rotation Coordinates (p. 1921))

$$\vec{x}_r^{n+1} = \vec{x}_r^{n} + \Delta\vec{x} \tag{13.35}$$

where $\Delta\vec{x}$ can be shown to be

$$\Delta\vec{x} = |\vec{x}_r^{n} - \vec{x}_{c.g.}|\left[\sin(\Delta\theta)\hat{e}_\theta + (\cos(\Delta\theta) - 1)\,\hat{e}_r\right] \tag{13.36}$$

The unit vectors $\hat{e}_\theta$ and $\vec{e}_r$ are defined as

$$\hat{e}_\theta = \frac{\vec{\Omega}_{c.g.} \times \vec{x}_r}{|\vec{\Omega}_{c.g.} \times \vec{x}_r|} \tag{13.37}$$

$$\hat{e}_r = \frac{\hat{e}_\theta \times \vec{\Omega}_{c.g.}}{|\hat{e}_\theta \times \vec{\Omega}_{c.g.}|} \tag{13.38}$$

If the solid body is also translating with $\vec{v}_{c.g.}$, the $n+1$ position vector on the solid body can be expressed as

$$\vec{x}^{n+1} = \vec{x}_{c.g.}^{n} + \vec{v}_{c.g.}\Delta t + \vec{x}_r^{n+1} \tag{13.39}$$

where $\vec{x}_r^{n+1}$ is given by Equation 13.35 (p. 1921).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

13.6.10. Previewing the Dynamic Mesh

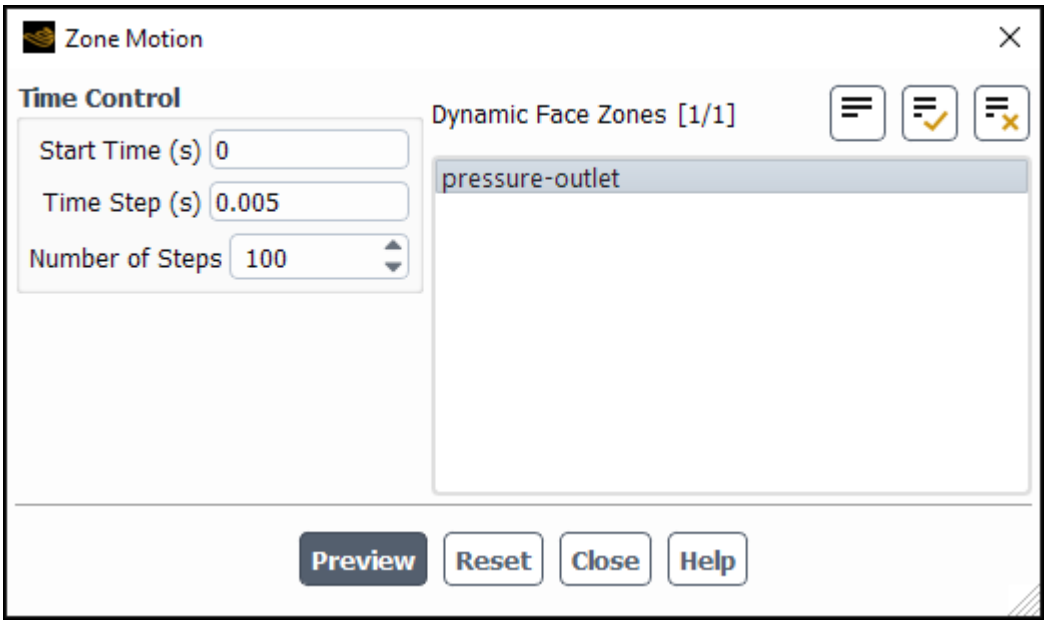
When you have specified the mesh update methods and their associated parameters, and you have defined the motion of dynamic zones, as described in Specifying the Motion of Dynamic Zones (p. 1904), you can preview the motion of the mesh or the zone as it changes with time before you start your simulation. The same dynamic zone or mesh motion will be executed when you start your simulation.

13.6.10.1. Previewing Zone Motion

You can preview the motion of zones with **Rigid Body** or **User-Defined** motion using the Zone Motion Dialog Box (p. 5150) (Figure 13.91: The Zone Motion Dialog Box (p. 1922)).

Setup → **Dynamic Mesh** → **Display Zone Motion...**

Figure 13.91: The Zone Motion Dialog Box



The zone motion preview only updates the graphical representation (in the graphics window) of the zones that you have selected in the **Dynamic Face Zones** list. Only zones that have been specified with either **User-Defined** or **Rigid Body** motion type are available for zone motion preview. To use the **Zone Motion** preview:

1. In the Zone Motion Dialog Box (p. 5150), select the face zones for which you want to preview the motion from the **Dynamic Face Zones** list. The **Dynamic Face Zones** list displays zones that have either **User-Defined** or **Rigid Body** motion specified. By default, all such zones are selected.

2. Enter the **Start Time**, **Time Step**, and **Number of Steps** under **Time Control**.

3. Click the **Preview** button to preview the zone motion. This positions the mesh according to the specified **Start Time**, and then integrates the position of the selected surfaces in time. The zone positions at the specified **Start Time** can be previewed without any subsequent motion by entering 0 for the **Number of Steps**.

4. Click **Reset** to restore the mesh to its initial state.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Previewing the zone motion can also be used as a postprocessor for six DOF simulations (see Six DOF Solver Settings (p. 1883)).

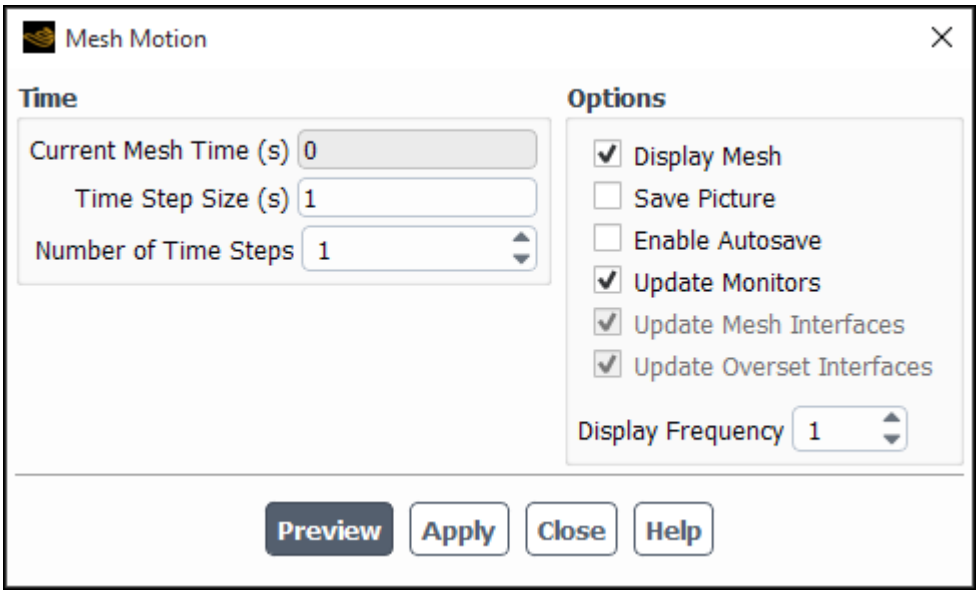
13.6.10.2. Previewing Mesh Motion

The mesh motion preview is different from the zone motion described above in that the mesh connectivity is changed in mesh motion.

To preview the dynamic mesh of a transient case, you can use the Mesh Motion Dialog Box (p. 5151) (Figure 13.92: The Mesh Motion Dialog Box (p. 1923))

Setup → Dynamic Mesh → Preview Mesh Motion...

Figure 13.92: The Mesh Motion Dialog Box



The procedure is as follows:

1. Save the case file.

   File → Write → Case...

   Important:

   Note that the mesh motion will actually update the node locations as well as the connectivity of the mesh, so you must be sure to save your case file before doing the dynamic mesh motion. Once you have advanced the mesh by a certain number of time steps, you will not be able to recover the previous status of the mesh, other than by reloading the appropriate Ansys Fluent case file.

2. Specify the **Number of Time Steps** and the size of each time step (**Time Step Size**). The current time will be displayed in the **Current Mesh Time** field after the dynamic mesh has been advanced the specified number of steps.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that if you turned on the in-cylinder option, the **Time Step Size** is automatically calculated from the **Crank Angle Step Size** and the **Crank Shaft Speed** that you have specified in the **In-Cylinder** tab of the Options Dialog Box (p. 5121).

3. To view the dynamic mesh in the graphics window, enable the **Display Mesh** option. In addition, you can control the frequency at which Ansys Fluent should display an updated mesh in the **Display Frequency** field. To save a picture file of the mesh each time Ansys Fluent updates it during the preview, turn on the **Save Picture** option. This opens the Save Picture Dialog Box (p. 5254) (see Saving Picture Files (p. 1019)).

4. Turn on **Enable Autosave** to use the automatic saving feature to specify the file name and frequency with which case and data files should be saved during the solution process. This opens the Autosave Case During Mesh Motion Preview Dialog Box (p. 5153).

   See Automatic Saving of Case and Data Files (p. 958) for details about the use of this feature. This provides a convenient way for you to save results at successive time steps for later postprocessing.

5. Enable the **Update Mesh Interfaces** option to update the interface at every time step.

6. Enable the **Update Overset Interfaces** option to update the overset interface at every time step.

7. Use the **Update Monitors** option to disable the processing of monitors and computation activities during mesh motion preview. This allows you to set up monitors and computational activities before running mesh motion preview without creating monitor files during the mesh motion preview.

8. Click **Preview** to start the preview. Ansys Fluent will update the dynamic mesh by moving and deforming the face and cell zones that you have specified as dynamic zones. Click **Apply** to save your settings for mesh motion.

During the preview, information about the dynamic mesh will be displayed in the console window for each time step. Note that for the in-cylinder option, the reported `Maximum Cell Skew` is calculated only from zones undergoing remeshing. This ensures that you can always ascertain whether the skewness is increasing in the deforming zones. To report the maximum skewness of a cell from *any* zone, you can click the **Report Quality** button in the General Task Page (p. 4684).

Setup → **General** → **Report Quality**

13.6.11. Steady-State Dynamic Mesh Applications

While many dynamic mesh problems are transient, you can use dynamic meshes for steady-state applications as well. Some examples of steady-state applications include: checking the valve application after reaching a steady-state valve position; or after a fluid-structure interface application has reached a steady-state solution.

There are no differences in the meshing aspect between steady-state cases and transient cases. Furthermore, setting up a steady-state simulation is similar to setting up a transient case, described in Using Dynamic Meshes (p. 1812). However, there are a few differences that you should note:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- A `CG_MOTION` UDF is needed to specify the motion of the boundary: a transient profile used in transient cases cannot be used in steady-state cases.
- The `dtime` passed to the `CG_MOTION` UDF is 1 by default: if a displacement of 1 mm is needed to move the boundary, you can specify the velocity to be 1e-3 m/s.
- Dynamic mesh parameters can be different since an interpolation error is no longer a concern.
- If you have enabled local remeshing for your steady-state application, you can instruct Ansys Fluent to perform additional remeshing after the boundary has moved. This additional remeshing is based on skewness criteria, and can further increase the quality of your mesh. See Dynamic Mesh Update Methods (p. 1816) for further details.

The mesh must be manually updated through journal files or execute commands. To update the mesh, you can use the Mesh Motion Dialog Box (p. 5151).

Solution → Run Calculation → Update Dynamic Mesh...

Alternatively, you can use the following text command:

`solve → mesh-motion`

which can also be used as an execute command in the Execute Commands Dialog Box (p. 5201):

Note:

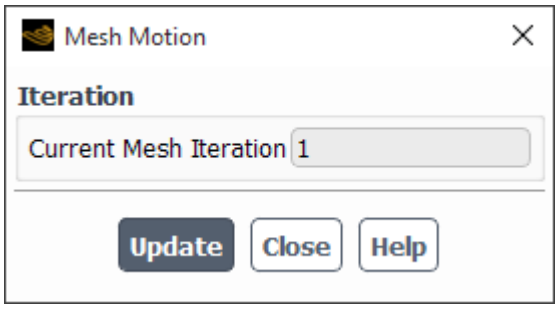
When you are solving a problem using the six DOF solver, you may need to modify the pseudo time step size in order for the solver to converge. It is also recommended that you ensure that the steady-state solution is reasonably well converged before performing the steady-state dynamic mesh update.

Solution → Calculation Activities → Execute Commands New...

You can display dynamic mesh statistics (such as minimum and maximum volumes and maximum cell and face skewness) by clicking the **Update** button in the Mesh Motion Dialog Box (p. 5151) (Figure 13.93: The Mesh Motion Dialog Box for Steady-State Dynamic Meshes (p. 1926)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.93: The Mesh Motion Dialog Box for Steady-State Dynamic Meshes



Important:

The following options are not available for steady-state applications:

- **In-Cylinder**
- **Implicit Update**

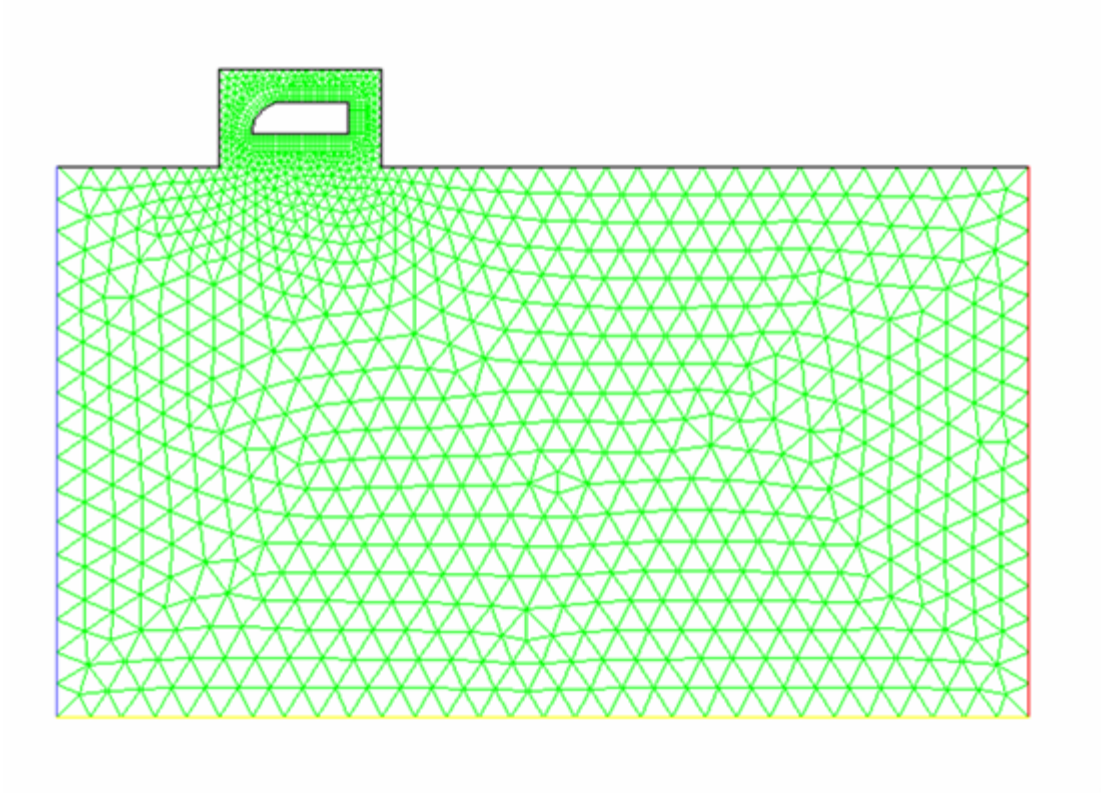
Also, the **Dynamic Mesh Events** dialog box is not available for steady-state applications.

13.6.11.1. An Example of Steady-State Dynamic Mesh Usage

Consider a rescue drop case shown in Figure 13.94: Initial Object Position (p. 1927). The object can be moved in any position in the steady-state solver, after which steady-state analyses can be performed at different object positions.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.94: Initial Object Position



The dynamic mesh parameters setup is identical for the steady-state and transient cases, which is described in Setting Dynamic Mesh Modeling Parameters (p. 1814). When setting up the dynamic zones, the procedures are similar to those described in Specifying the Motion of Dynamic Zones (p. 1904), except that the UDF selected from the **Motion UDF/Profile** drop-down list is different. In steady-state cases the `dtime` passed to the UDF is by default 1. So, in this example, the object will move 50 mm each time the following UDF is executed:

```
#include "udf.h"

DEFINE_CG_MOTION(pod,dt,vel,omega,time,dtime)

{
 NV_S(vel,=,0);
 NV_S(omega,=,0);
 vel[1] = -50e-3;
}
```

The resulting mesh is shown in Figure 13.96: Final Object Position After 40 Executions (p. 1928).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 13.95: The Mesh Motion Dialog Box After 40 Updates

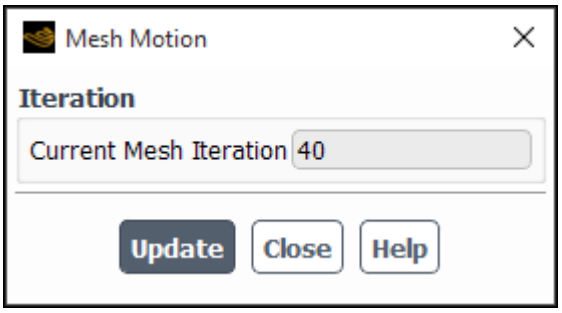
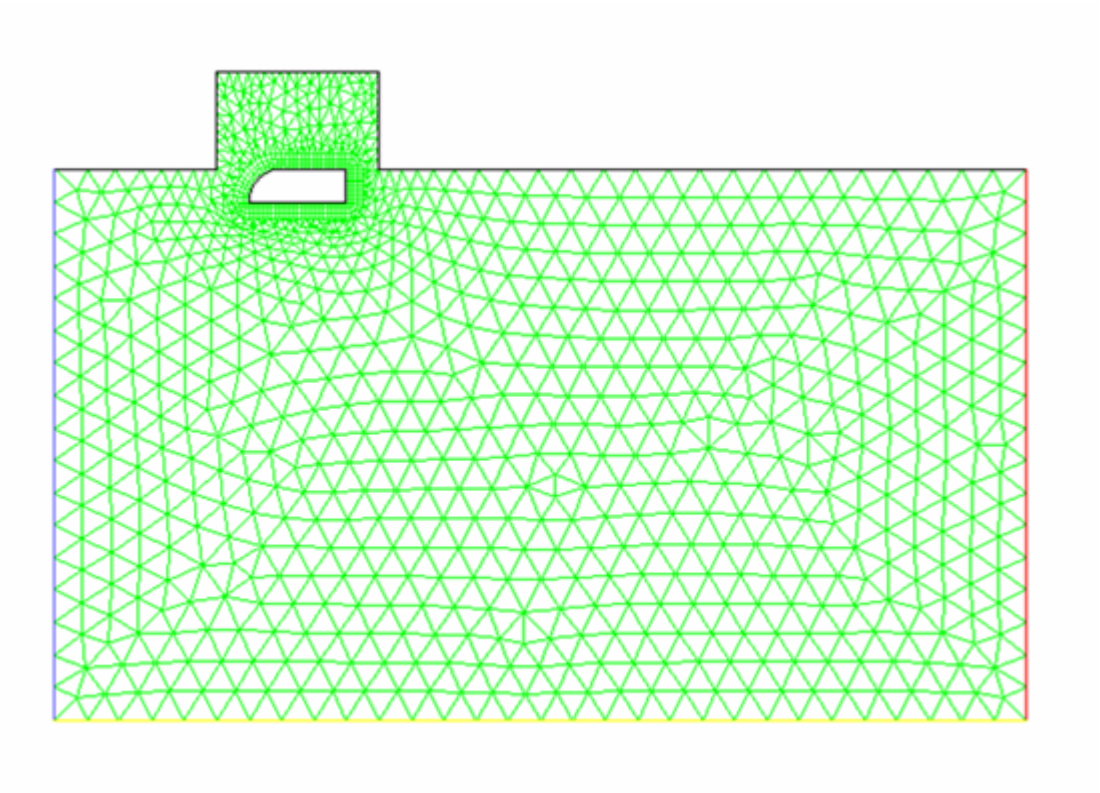


Figure 13.96: Final Object Position After 40 Executions



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 14: Modeling Turbomachinery Flows

This chapter provides details about the turbomachinery capabilities in Ansys Fluent.

The information in this chapter is divided into the following sections:

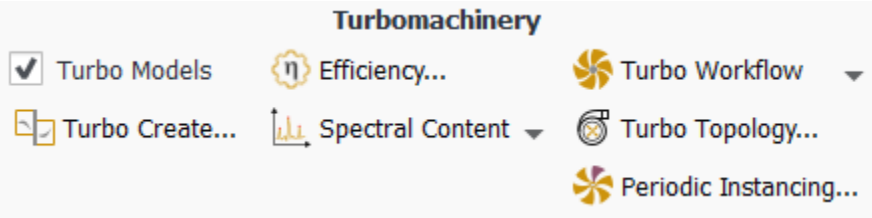
14.1. Using the Turbomachinery Guided Workflow
14.2. Frozen Gust / Inlet Disturbance Flow Modeling
14.3. Blade Row Interaction Modeling
14.4. Aerodynamic Damping (Blade Flutter Analysis)
14.5. Phase-lag Method
14.6. Non-equilibrium Wet Steam Model for Steam Turbines
14.7. Blade Film Cooling for Gas Turbines
14.8. Turbomachinery Description
14.9. Turbomachinery-Specific Numerics
14.10. Turbomachinery Postprocessing

14.1. Using the Turbomachinery Guided Workflow

The **Turbo Workflow** provides you with the basic steps to defining and refining the setup and solution of your turbomachinery simulation within Fluent.

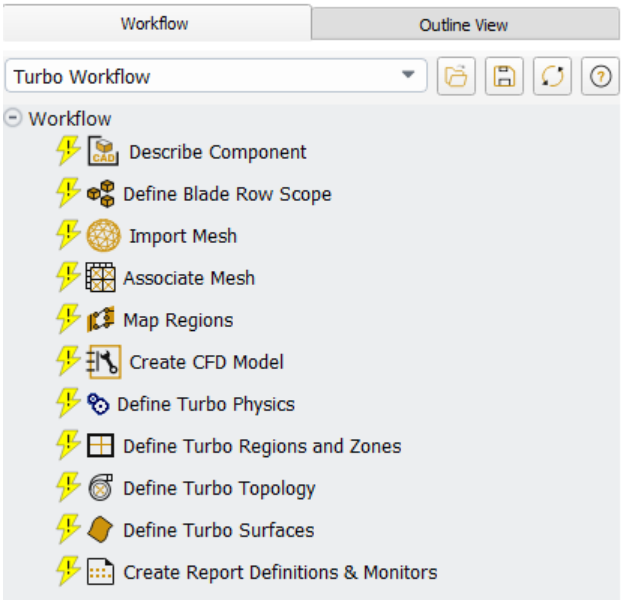
You can access the turbomachinery guided workflow by selecting the **Enable Workflow** option in the **Turbo Workflow** drop-down menu in the **Turbomachinery** category of the **Domain** Ribbon.

Domain → Turbomachinery → Turbo Workflow → Enable Workflow



This exposes a Workflow tab, alongside the Outline View, where you can use the **Turbo Workflow** to begin your work. The **Turbo Workflow** provides you with a series of turbomachinery-specific workflow tasks. Your goal is to progress through each task in order to more easily analyze your turbomachinery problem and generate an accurate simulation.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Once you have completed the steps in the **Turbo Workflow**, you can review your simulation settings in the **Outline View**, and/or proceed on to calculate a solution and analyze the results of your turbomachinery simulation.

Note:

The workflow is similar to the Fluent guided meshing workflows (Getting Started with the Fluent Guided Workflows (p. 277)). See Understanding Task States (p. 284), Operating on Tasks (p. 285), Grouping Tasks (p. 285), or Editing Tasks (p. 286) for more information about working with and understanding tasks in the Fluent workflows.

Additional options are available under the **Turbo Workflow** drop-down menu in the Ribbon:

- **Enable Workflow**: exposes the **Turbo Workflow** in the **Workflow** tab.
- **Hide Workflow**: removes the workflow from the graphical user interface.
- **Show Workflow**: exposes the workflow tab in the graphical user interface.
- **Disable Workflow**: removes the workflow tab and workflow-related information, while retaining settings performed in Fluent so far. Fluent will prompt you to make sure that you want to proceed.

This section describes the various tasks that are provided when using the **Turbo Workflow**:

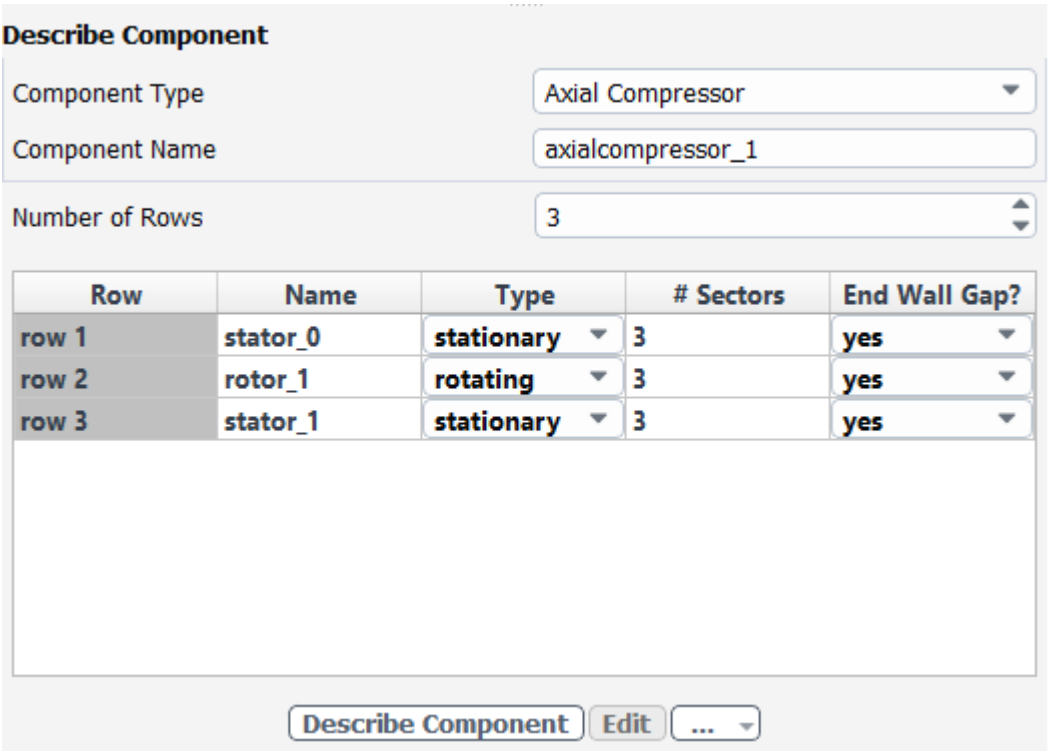
14.1.1. Describing the Components of the Turbo Machine
14.1.2. Defining Your Blade Row Scope
14.1.3. Importing Your Mesh
14.1.4. Associating Your Mesh
14.1.5. Mapping Your Regions
14.1.6. Creating the CFD Model
14.1.7. Defining the Turbomachinery Physics
14.1.8. Defining the Turbomachinery Regions and Zones

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.1.9. Defining the Turbomachinery Topology
14.1.10. Defining Turbomachinery Surfaces
14.1.11. Creating Turbomachinery Report Definitions and Monitors
14.1.12. Text Command List for the Turbo Workflow
14.1.13. Editing Tasks in the Turbo Workflow
14.1.14. Saving and Loading Turbo Workflows
14.1.15. Applying Preferences to the Turbo Workflow

14.1.1. Describing the Components of the Turbo Machine

A turbomachine could have one or more turbomachine components, and each component has a unique set of information (number of rows, row type, and number of blades). Use the **Describe Component** task to describe a single component of the turbomachine system that you want to simulate. For example, you might want to simulate a full wheel axial compressor blade/row assembly where you have 26 internal guide vanes, 23 rotor blades, and 30 stator blades.



1. Specify the **Component Type**. You can have one of two types of turbomachinery components:

   - **Axial Compressor** or **Axial Turbine**

     Axial components that are described by the number of rows (between 1 and 99) and the X, Y or Z axis of rotation. Each row could be either a stationary row (for example, inlet guide vanes (IGV), nozzle guide vane (NGV), stators) or a rotating row (for example, rotors). In addition to specifying the name and type for each row, you also need to define the number of blades for each row.

   - **Radial Compressor** or **Radial Turbine**

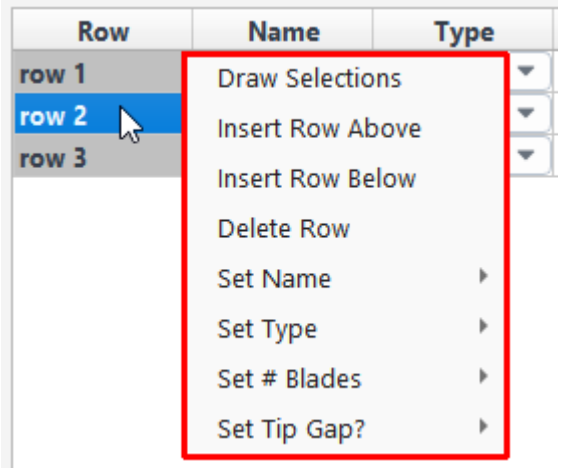
Similar to axial components, radial components are described by the number of rows (between 1 and 99) and the X, Y or Z axis of rotation. Each row could be either a stationary row (for example, inlet guide vanes (IGV), nozzle guide vane (NGV), stators) or a rotating row (for example, rotors). In addition to specifying the name and type for each row, you also need to define the number of blades for each row.

2. Specify the **Component Name**, or keep the default value.
3. Specify the **Number of Rows**, or keep the default value.
4. For each row:
   - Specify a **Name** or use the default name.
   - Specify the **Type** of motion as either **stationary** or **rotating**.
   - Specify the number of blade sectors (**# Sectors**), or keep the default value(s).
   - Use the **Tip Gap?** field to determine whether or not you need to allow for any spacing between the blade and the hub/shroud, or keep the default value(s). Non-conformal mesh interfaces are often placed here, so identifying this as **yes** will allow the non-conformal interface to be generated automatically.

     By default, this option is set to **Yes** for rotor rows and **No** for stator rows. The tip gap is considered a hub gap when this option is set to **Yes** for stator rows.

Note:

Read-only table cells (such as **Name** in this task) are indicated in gray, however, you can multiply-select and edit cells in the table using the context menu.



This allows you to draw the selected item(s) in the graphics window, insert a row above or below a single selected row, remove a single selected row, change the name of a single row, or create a name for a group of rows, change the type, the number of blades, and the tip gap presence. Note that inserting a row before or after another row always copies the properties and settings of the current row to the inserted row.

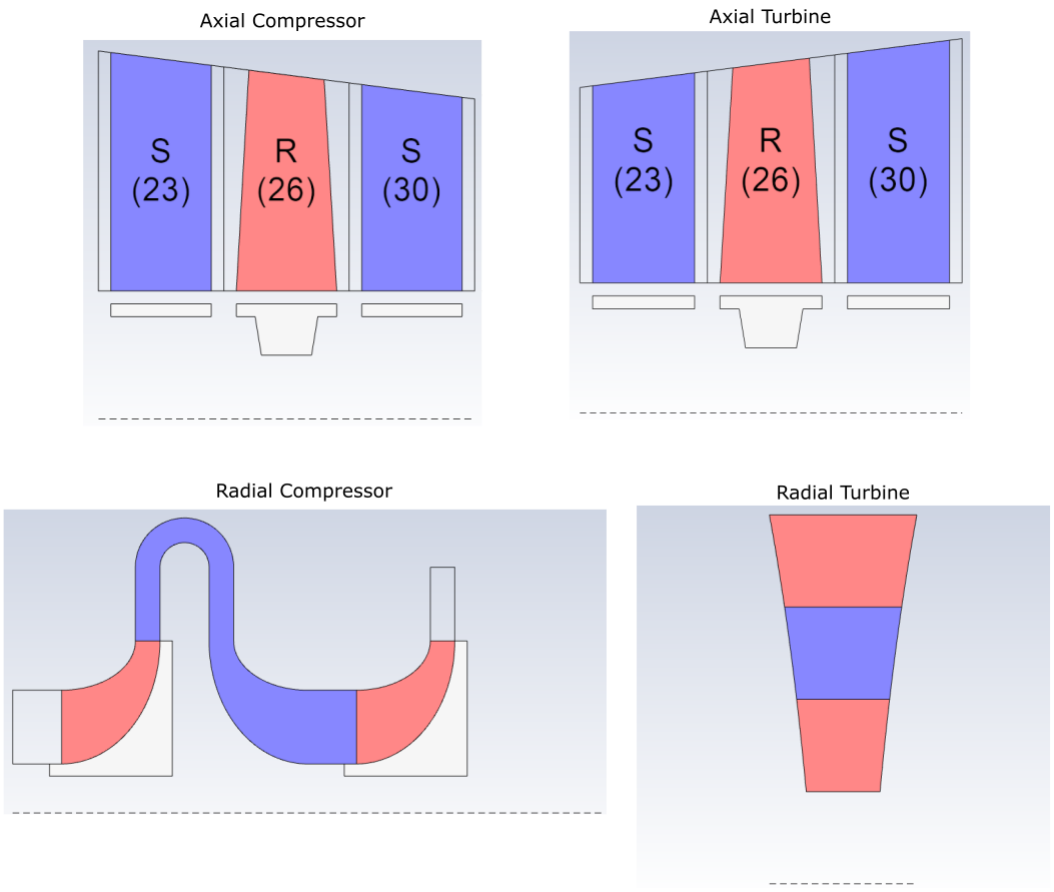
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Once your selections are made, click the **Describe Component** button to apply your component description settings.

   If you need to make adjustments to any of your settings in this task, click **Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

   While describing your turbomachine, a 2D graphical schematic representation is made available in the graphics window for a single component (Figure 14.1: Examples of Schematics for Various Multi-Row Component Types (p. 1933)):

   Figure 14.1: Examples of Schematics for Various Multi-Row Component Types

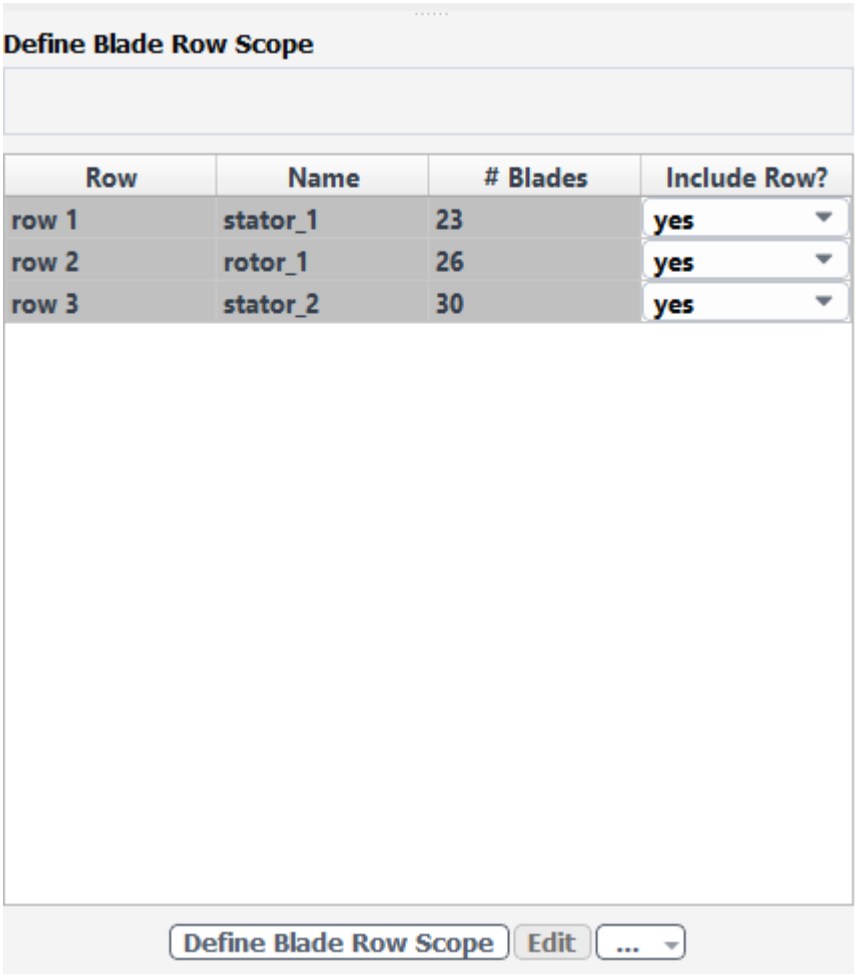


6. Proceed to the next step in the workflow.

14.1.2. Defining Your Blade Row Scope

Use the **Define Blade Row Scope** task to specify which rows to include in your turbomachinery analysis. You must define a single contiguous set of rows. Indicate whether or not to include or exclude the row. You can elect to keep all or just a subset of the turbomachine system that you defined in the previous task.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



1. Review the information in the table, and make any changes as needed.
2. Once your selections are made, click the **Define Blade Row Scope** button to apply your settings.

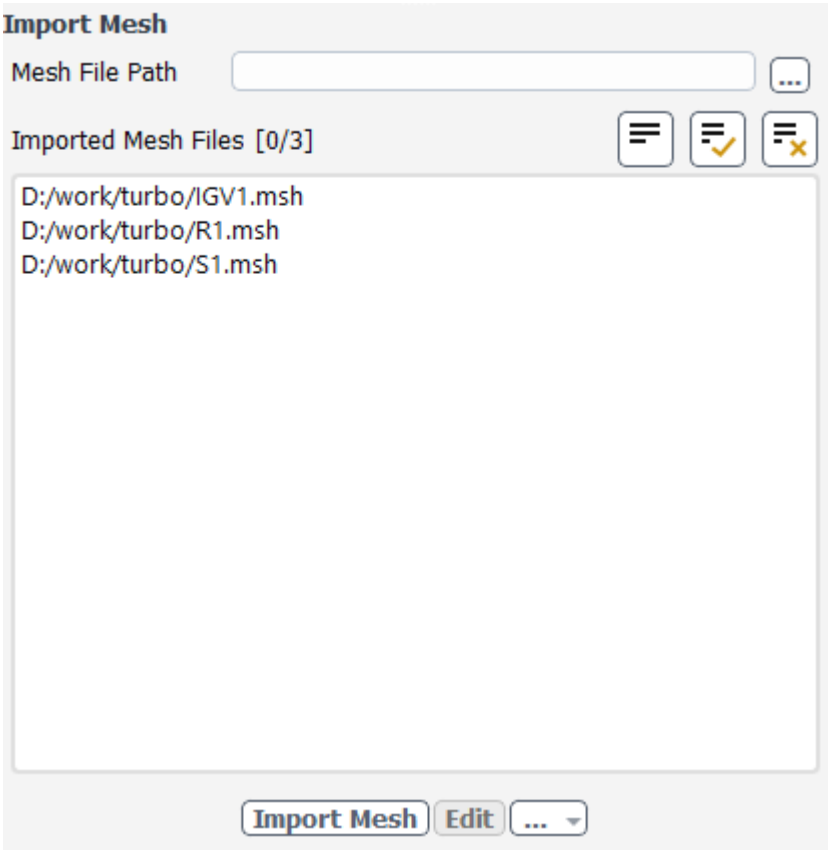
   If you need to make adjustments to any of your settings in this task, click **Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

   The graphics window will display a basic 2D schematic for each component scope. For radial and axial components, the row color is grayed out if it is not included in the analysis.
3. Proceed to the next step in the workflow.

14.1.3. Importing Your Mesh

Use the **Import Mesh** task to supply the mesh for the rows you wish to include in the CFD model. You can import a single mesh file that includes all of the rows/passages/blades, or import multiple mesh files that represent each passage/blade.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

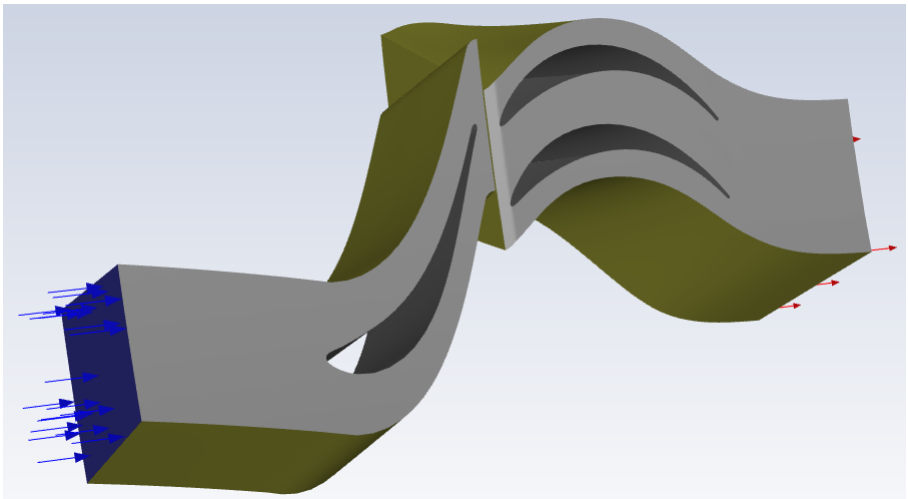


1. For the **Mesh File Path** field, provide a path and a name for the mesh or case file that you want to load into the task, or use the (**...**) button to browse for a specific file. Each file may contain several cell zones.

   Standard Ansys mesh file types are supported, including `.msh`, `.msh.h5`, `.def`, `.cgns`, and `.gtm`.

Note:

The workflow also supports importing meshes with multiple blade(s) and/or splitter(s) in a particular cell zone.

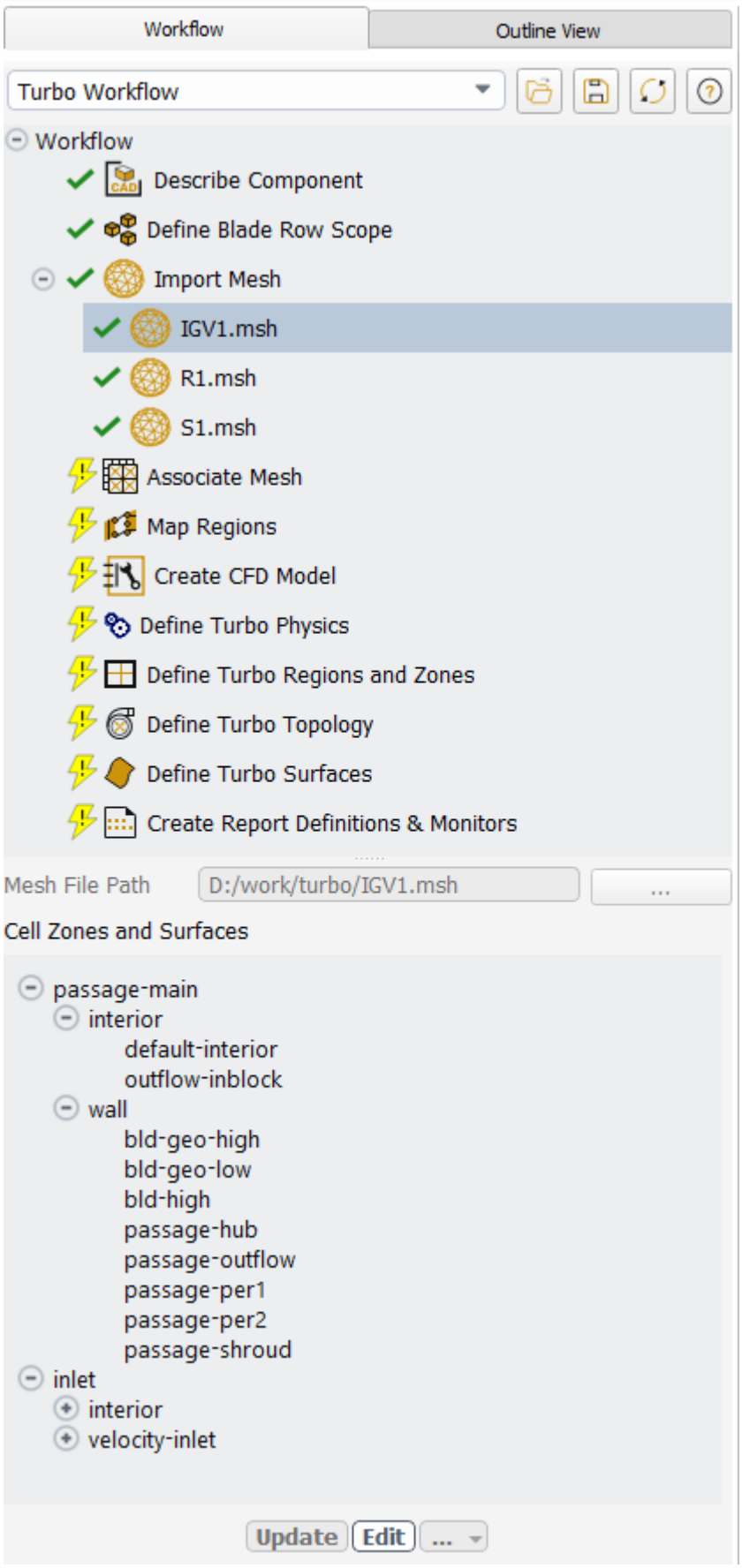


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Once your selections are made, click the **Import Mesh** button and proceed to the next task.

   For each loaded file, a child task will be created which shows the full path to the file. The child task also provides a tree view displaying the cell zones and their associated face zones.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



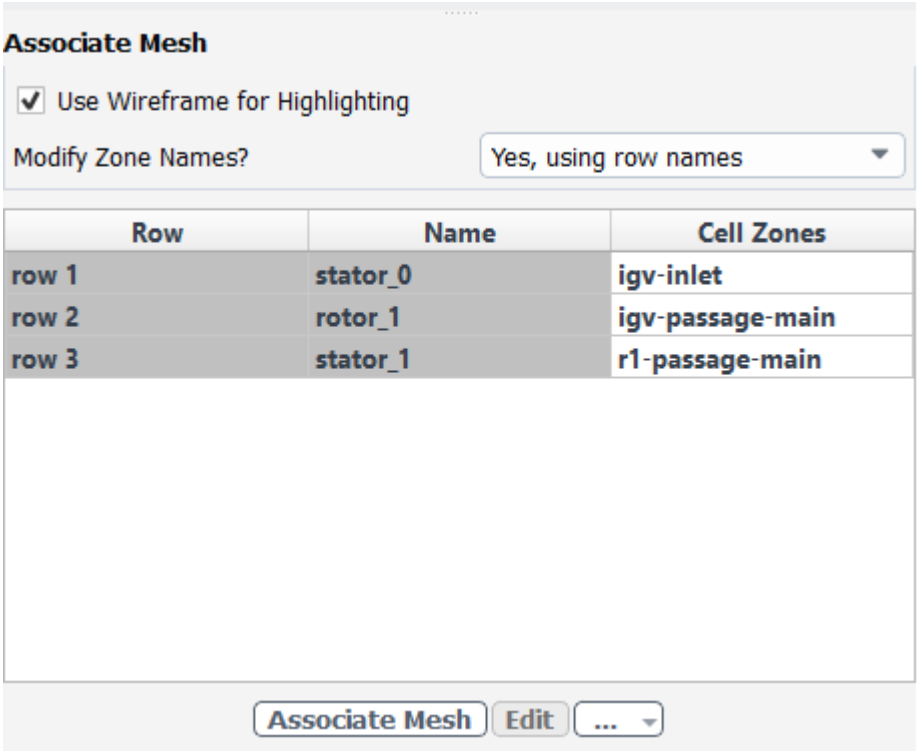
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you need to make adjustments to any of your settings in this task, click **Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes. If a new mesh or case file is specified, the old file will be unloaded and the new file will be imported and displayed.

3. Proceed to the next step in the workflow.

14.1.4. Associating Your Mesh

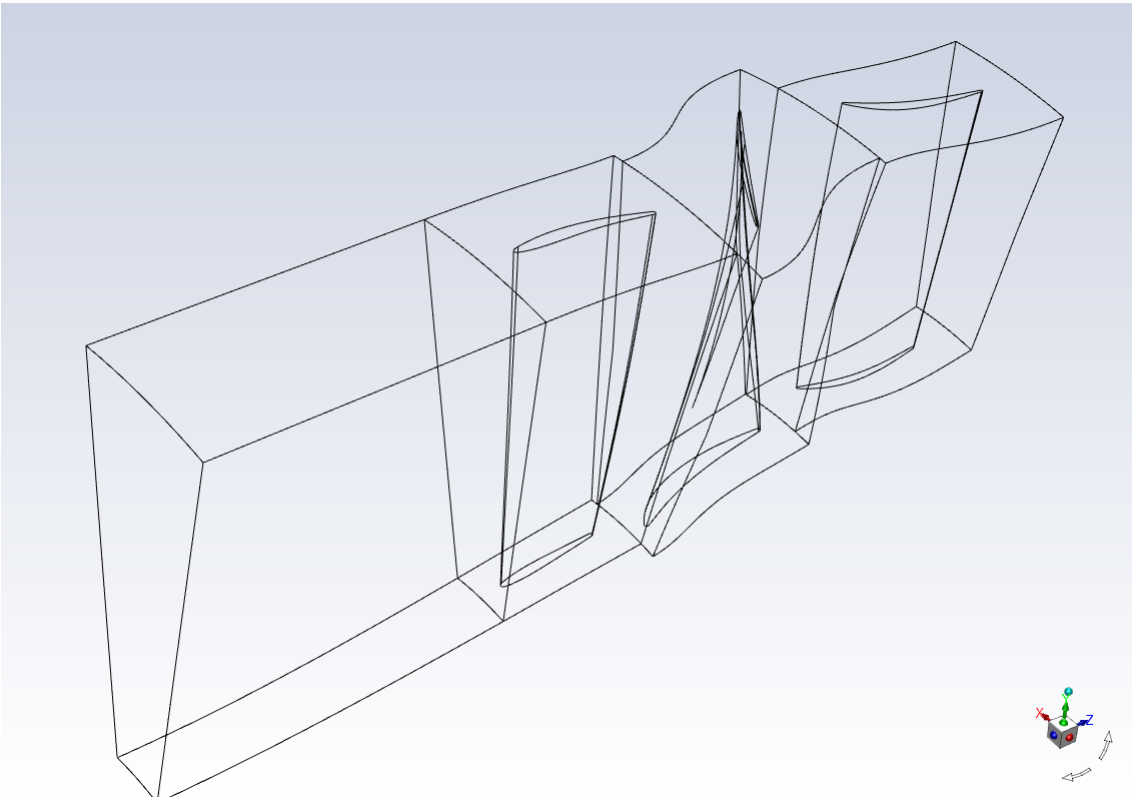
Use the **Associate Mesh** task to associate and map each cell zone from the imported mesh file(s) with the corresponding row of the turbomachine.



This will be helpful in later tasks where Fluent copies and stitches cell zones to create the full CFD model.

1. By default, the **Use Wireframe for Highlighting** option is selected so that the imported geometries are displayed in wireframe. Arrows are also used to indicate flow direction.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Use the **Modify Zone Names?** field to adjust how your zones names appear once this task is completed, depending on your preferences. When set to **Yes, using row names**, this field will change the associated cell (or face) zone name according to the corresponding **Name**. When set to **Yes, using row numbers**, this field will change the associated cell (or face) zone name according to the corresponding **Row** number. You can also choose **No** to keep the zone names as they are.

   Say, for instance, the zone is called `s1-inflow-passage` and it is located in the table where its **Row** is designated as `row3` and its **Name** is designated as `stator_1`.

   - For cell zones, **Yes, using row names** modifies the cell zone name entirely. For instance, the cell zone `s1-inflow-passage` will be modified to `stator_1`. When there are multiple zones, a numerical designation is added at the end of the name, such as `stator_1.1` for example.

     Likewise, if you select **Yes, using row numbers**, then the zone name will be changed to `row03:s1-inflow-passage`.

   - For face zones, **Yes, using row names** modifies the face zone name by adding the row name as a prefix. For instance, the face zone `s1-inflow-passage` will be modified to `stator_1:s1-inflow-passage`. When there are multiple zones, a numerical designation is added at the end of the name, such as `stator_1.1:s1-inflow-passage` for example.

     Likewise, if you select **Yes, using row numbers**, then the zone name will be changed to `row03:s1-inflow-passage`.

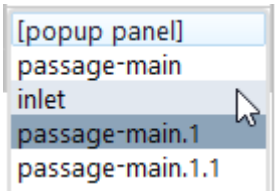
3. In the table, ensure that the selected **Cell Zone** has been properly assigned.

   You are expected to assign one or more cell zones to each row. This information will be used for later tasks to copy and/or transform cell zones for each row and create the CFD model.

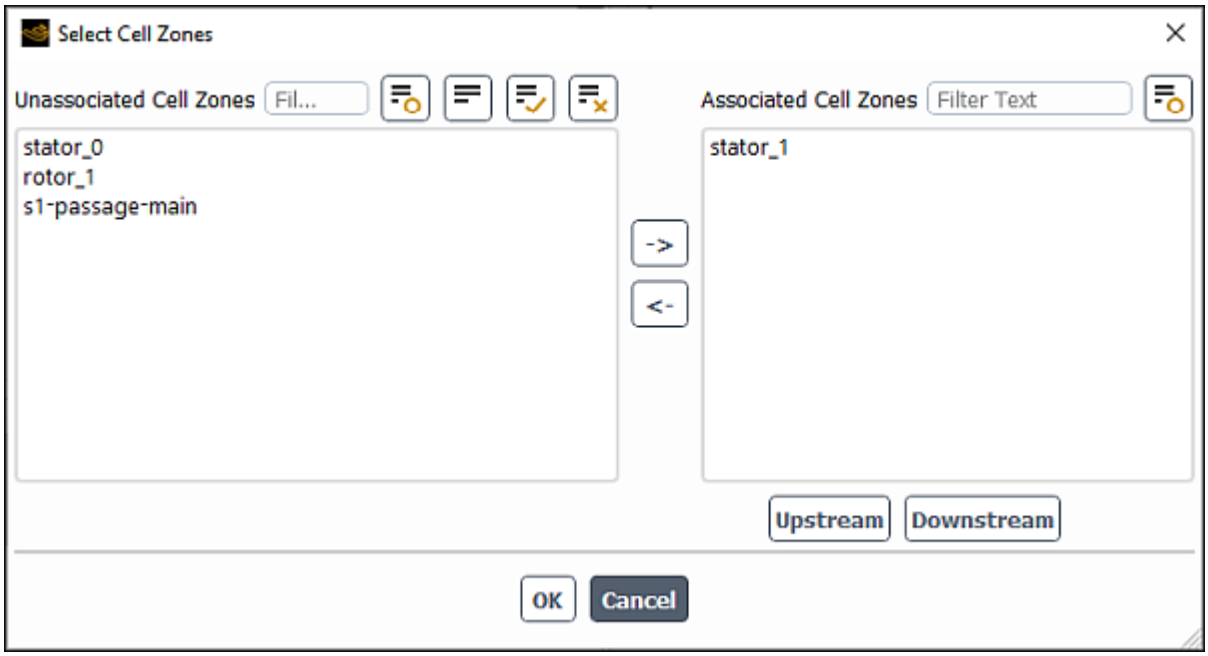
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. If any cell zone association changes are still required, click the corresponding table cell to make the change.

   Use the cell's drop-down menu to choose from a limited list of potential cell zones.

   To choose from a potentially larger selection of cell zones, select the **[popup panel]** option to open the **Select Cell Zones** dialog, where all available associated and unassociated cell zones are listed. This dialog also allows you to filter and sort potentially large lists of cell zones. Associated zones can be assigned to be upstream or downstream as needed.

   Make your selections in the dialog and click **OK**, or click **Cancel** to dismiss the dialog.

5. Once your selections are made, click the **Associate Mesh** button and proceed to the next task.

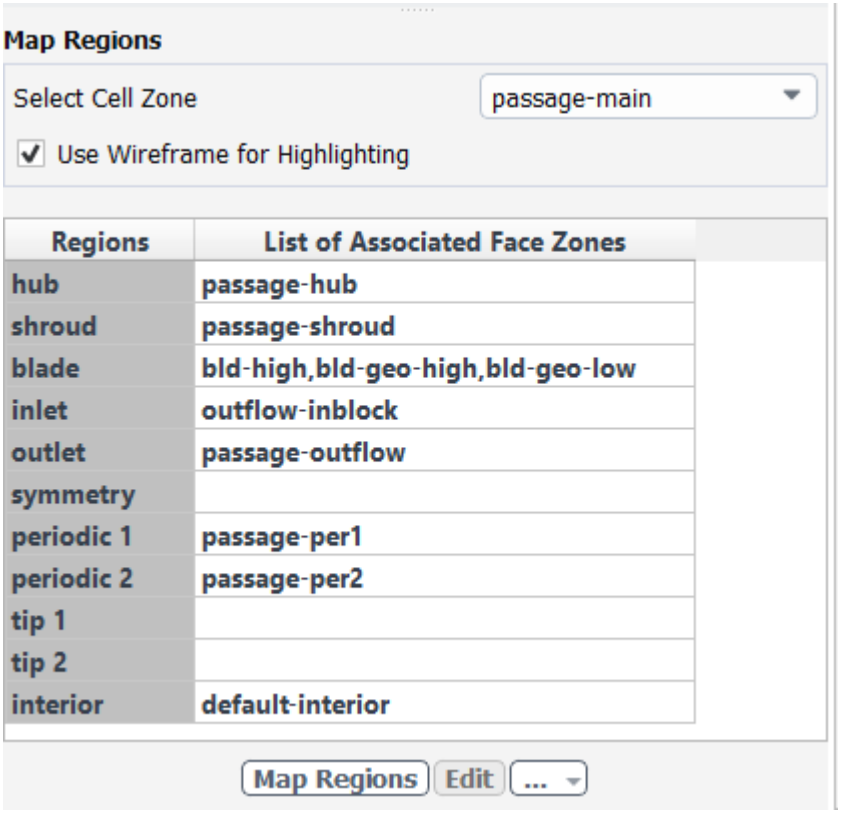
   If you need to make adjustments to any of your settings in this task, click **Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

6. Proceed to the next step in the workflow.

14.1.5. Mapping Your Regions

For each original cell zone, use the **Map Regions** task to associate and map the corresponding face zones to ensure that the face zones correspond to the correct blade passage region.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



This information will be used while defining boundary conditions and the turbo topology in Fluent.

1. Use the **Select Cell Zone** field to select a particular zone for mapping its regions.
2. In the table provided, for each of the listed **Regions**, ensure the proper face zone(s) is selected under the **List of Associated Face Zones**.

Note:

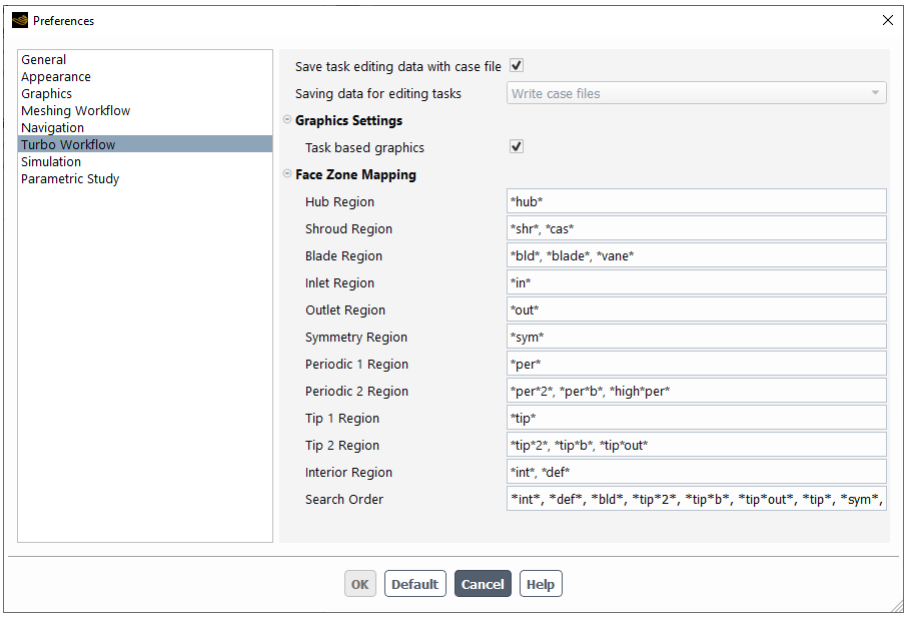
As you hover over items in the table, they are highlighted in the graphics window.

Once this task is updated, upon subsequent visits to this task, periodic zones are no longer highlighted as other surfaces are, since they will have been merged into different periodic zones. These highlighted periodic zones are visible in the **Define Turbo Topology** task, however, if you need to highlight them in this task, you will have to first undo your work in this task.

Note:

Face zone mapping and assignment associations are based on those defined in the Fluent **Preferences** dialog (under the **File** menu), in the **Turbo Workflow** category.

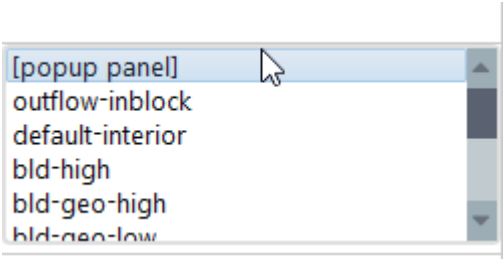
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can edit these associations according to your specific simulation requirements.

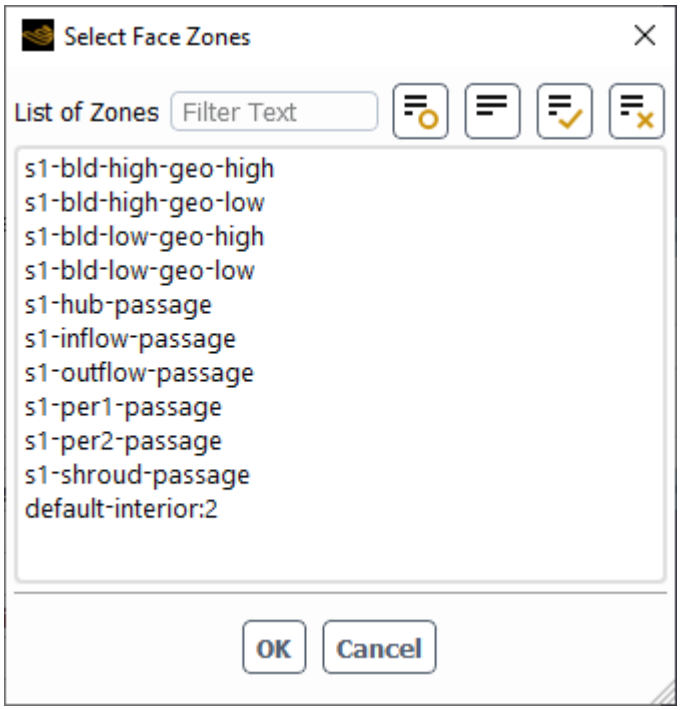
3. If any face zone association changes are still required, click the corresponding table cell to make the change.

   Use the cell's drop-down menu to choose from a limited list of potential face zones.



   To choose from a potentially larger selection of face zones, select the **[popup panel]** option to open the **Select Face Zones** dialog, where all available face zones are listed. This dialog also allows you to filter and sort potentially large lists of face zones.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Make your selections in the dialog and click **OK**, or click **Cancel** to dismiss the dialog.

4. Once your selections are made, click the **Map Regions** button and proceed to the next task.

   For each selected cell zone, a child task will be created that shows the mapped face zones for the corresponding region.

   If you need to make adjustments to any of your settings in this task, click **Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

   Note:

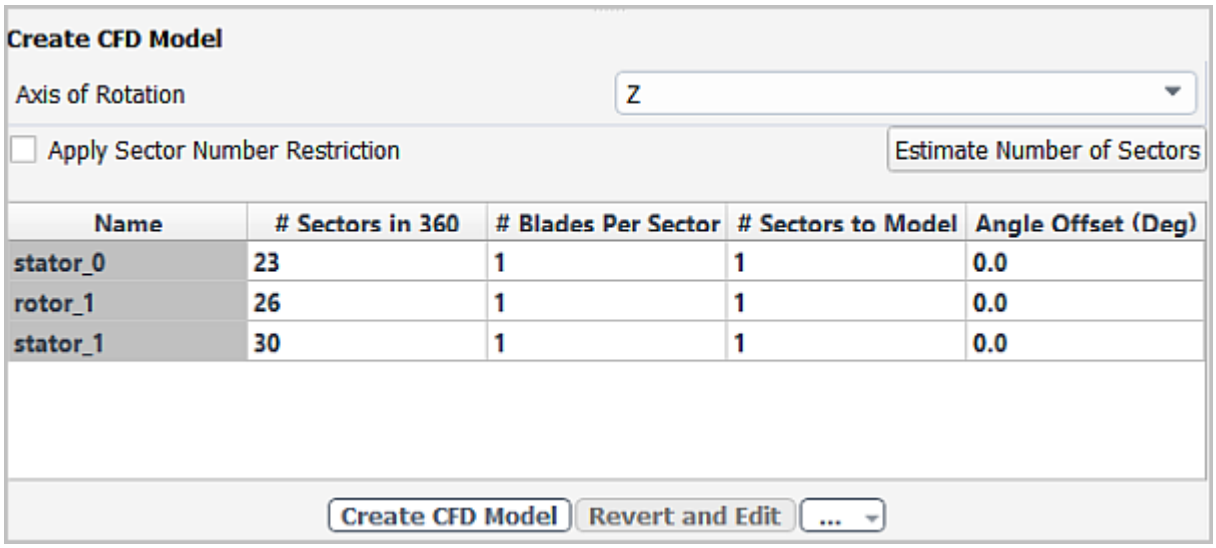
   If you have completed the **Define Turbo Regions and Zones** task, and need to edit the **Map Regions** task, you must first **Revert and Edit** the **Create CFD Model** task.

5. Proceed to the next step in the workflow.

14.1.6. Creating the CFD Model

Use the **Create CFD Model** task to formally create the 3D model of the turbomachine based on your inputs for the previous tasks. Use the graphics window to visually verify and ensure the CFD model is constructed properly.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



1. Specify the **Axis of Rotation** for the generated CFD turbomachine geometry.
2. Use the **Apply Sector Number Restriction** field to indicate whether you need to restrict the number of blade sectors to model. This will restrict the number of model blade sectors to factors of the number of actual blades.
3. Use the **Estimate Number of Sectors** button to have the workflow provide an estimation of the number of blade sectors, based on your imported geometry.

Note:

Upon importing your mesh, the number of sectors can now be estimated based on the axis of rotation and the angle between the two periodic surfaces. This value, however, may differ from the value provided earlier in the **Describe Component** task where you might not have confidently known the number of blades for a specific component.

If the estimated number of sectors differs from the value provided in the **Describe Component** task, you should make the changes persistent and make the necessary changes in the **Describe Component** task accordingly.

---

4. If necessary, specify the **Number Sectors in 360** to indicate the number of blade sectors for a full 360 degree geometry.
5. Specify the number of blades in a given sector for the CFD model (**# Blades Per Sector**).
6. Specify the number of modeled passages per row by specifying the number of sectors in the CFD model (**# Sectors to Model**) and/or any **Angle Offset** for each row, around the specified axis of rotation. For instance, you can choose to create a small or large portion of the full turbomachine, or the entire turbomachine, as needed.

   In each case, once a value is entered, a preview of your changes appears in the graphics window without necessarily updating the task.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Once your selections are made, click the **Create CFD Model** button and proceed to the next task.

   Note:

   Once this task is updated, zone names are appended with the appropriate sector number (even for single sectors), such that `passage`, for example, becomes `passage_01`, `passage_02`, and so on.

   If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

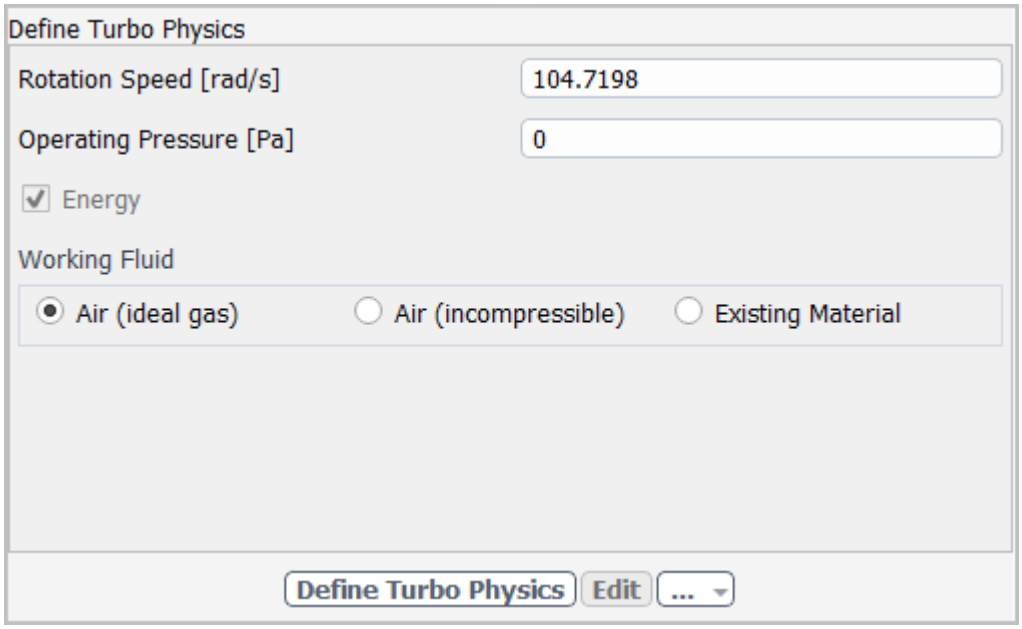
   Note:

   If you have completed the **Define Turbo Regions and Zones** task, and need to edit the **Map Regions** task, you must first **Revert and Edit** the **Create CFD Model** task.

8. Proceed to the next step in the workflow.

14.1.7. Defining the Turbomachinery Physics

Use the **Define Turbo Physics** task to specify the overall physics conditions of the turbomachinery simulation, such as the rotational speed, pressure, and the material assignment for the working fluid.



1. Specify the **Rotation Speed**, or keep the default value.
2. Specify the **Operating Pressure**, or keep the default value.
3. Specify whether **Energy**, or temperature effects, need to be considered.
4. Specify the **Working Fluid** as either **Air (ideal gas)**, **Air (incompressible)**, or an **Existing Material**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you select **Air (incompressible)**, then you can also provide the **Air Density**, or keep the default value.
- If you select **Existing Material**, then you can choose to use the currently available default fluid material (air) from the **Existing Fluid Material** drop-down list, or use the **Create/Edit...** button to create additional materials or edit existing material(s).

5. Once your selections are made, click the **Define Turbo Physics** button and proceed to the next task.

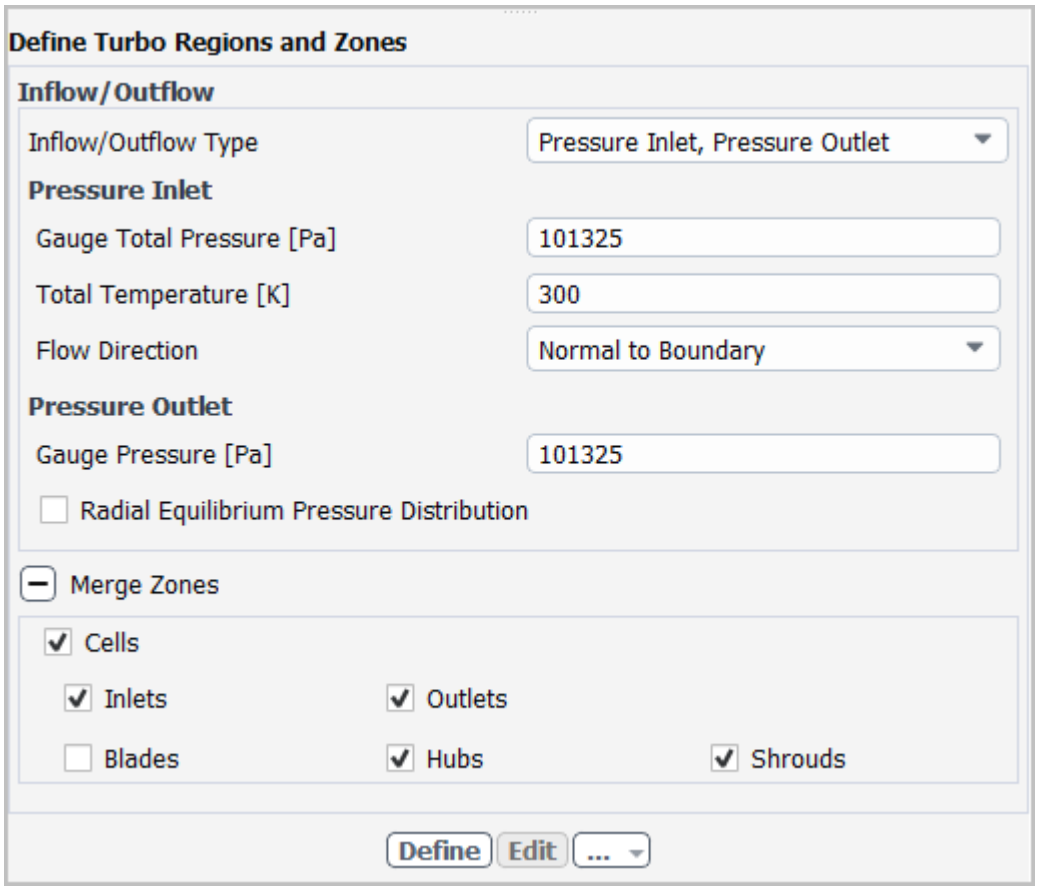
   If you need to make adjustments to any of your settings in this task, click **Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

6. Proceed to the next step in the workflow.

14.1.8. Defining the Turbomachinery Regions and Zones

Use the **Define Turbo Regions and Zones** task to specify the overall regional or zonal operating conditions of the turbomachinery simulation. In this task, you can specify the type of flow and various pressure specifications. You can also determine if/how cell zones are to be merged. By default, zones are merged to simplify boundary condition input, and based on the target turbo model requirements.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Note:

While this task uses basic default values for turbomachinery-related properties, if you require advanced settings (such as profile support), you can access and edit them accordingly directly through the Outline View.

1. Specify the **Inlet/Outlet Type** where inlet and outlet conditions are defined by combinations of pressure and mass flow specifications. Choice include

   - **Pressure Inlet, Pressure Outlet** - where both inlet and outlet conditions are specified through pressure values. If you choose this option, then you must specify the following as well:

     – Under **Pressure Inlet**:

       → Specify the **Gauge Static Pressure** at the inlet, or keep the default value.

       → Specify the **Total Temperature** at the inlet, or keep the default value.

       → Specify the **Flow Direction** at the inlet. Choices include the default **Normal to Boundary**, or you can specify the **Cartesian** choices of **X**, **Y**, and **Z** directions, or the **Cylindrical** choices of **Radial**, **Tangential**, and **Axial** directions

     – Under **Pressure Outlet**:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

→ Specify the **Gauge Pressure** at the outlet, or keep the default value.

→ Enable the **Radial Equilibrium Pressure Distribution** option at the outlet if your simulation requires it.

- **Pressure Inlet, Mass Flow Outlet** - where the inlet conditions are specified through pressure values, and the outlet conditions are specified through mass flow values.
  - Under **Pressure Inlet**:

    → Specify the **Gauge Static Pressure** at the inlet, or keep the default value.

    → Specify the **Total Temperature** at the inlet, or keep the default value.

    → Specify the **Flow Direction** at the inlet. Choices include the default **Normal to Boundary**, or you can specify the **Cartesian** choices of **X**, **Y**, and **Z** directions, or the **Cylindrical** choices of **Radial**, **Tangential**, and **Axial** directions

  - Under **Mass Flow Outlet**:

    → Specify the **Mass Flow Rate** at the outlet, or keep the default value.

    → Indicate how the outlet mass flow rate is provided, whether by **Machine**, **Sector**, or **Passage**.

- **Mass Flow Inlet, Pressure Outlet** - where the inlet conditions are specified through mass flow values, and the outlet conditions are specified through pressure values.
  - Under **Mass Flow Inlet**:

    → Specify the **Mass Flow Rate** at the inlet, or keep the default value.

    → Indicate how the inlet mass flow rate is provided, whether by **Machine**, **Sector**, or **Passage**.

    → Specify the **Flow Direction** at the inlet. Choices include the default **Normal to Boundary**, or you can specify the **Cartesian** choices of **X**, **Y**, and **Z** directions, or the **Cylindrical** choices of **Radial**, **Tangential**, and **Axial** directions

  - Under **Pressure Outlet**:

    → Specify the **Gauge Pressure** at the outlet, or keep the default value.

    → Enable the **Radial Equilibrium Pressure Distribution** option at the outlet if your simulation requires it.

2. Under **Merge Zones**, indicate how zones in the same row are to be merged (if at all). By default, all cells are merged as well as the inlets, outlets, hubs, and shrouds. You can deselect options or choose to not merge cells at all.

   So, if you have added multiple passages, this task merges the zones into a singular zone. If you have created two blade passages, for example, instead of having two separate hub surfaces, you

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

can have only one hub surface that consists of the two merged together hub surfaces, making it easier to set up boundary conditions.

3. Once your selections are made, click the **Define** button and proceed to the next task.

   If you need to make adjustments to any of your settings in this task, click **Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

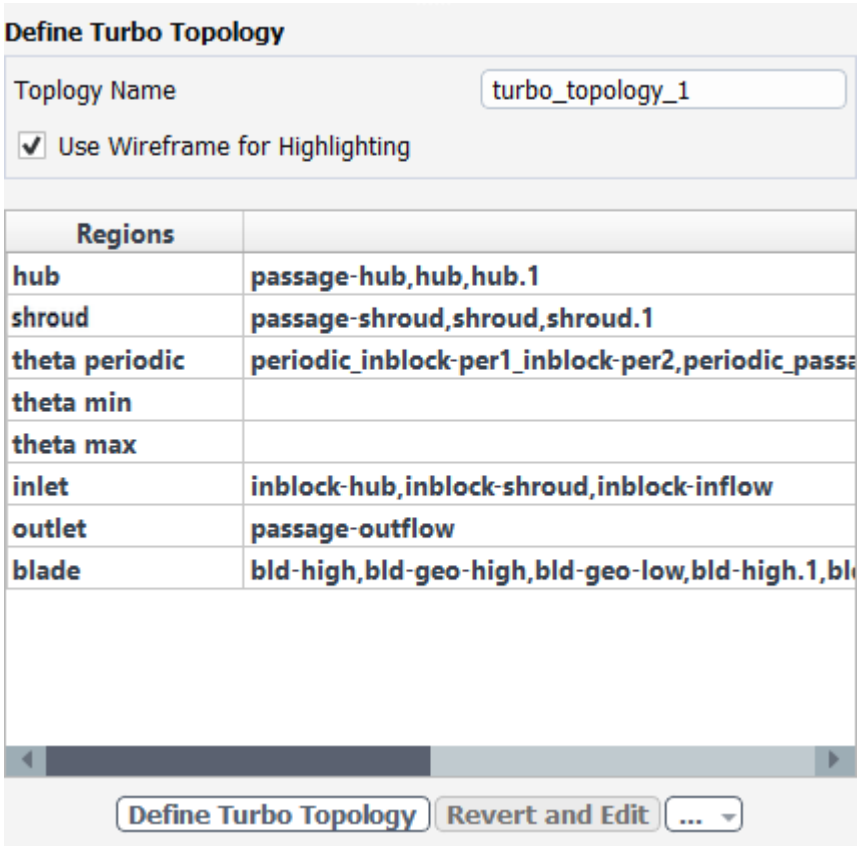
   Note:

   If you have completed the **Define Turbo Regions and Zones** task, and need to edit the **Map Regions** task, you must first **Revert and Edit** the **Create CFD Model** task.

4. Proceed to the next step in the workflow.

14.1.9. Defining the Turbomachinery Topology

Use the **Define Turbo Topology** task to associate topological zones (the machine main path inflow and outflow, the hub and shroud, and the side boundaries) with their regions so that Ansys Fluent can compute the coordinates of the turbo topology.



1. Specify a **Topology Name**, or keep the default value.

2. By default, the **Use Wireframe for Highlighting** option is selected so that the imported geometries are displayed in wireframe. Arrows are also used to indicate flow direction.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Use the table to review and/or assign topological zones to the list of regions provided. This table gives you an overall list of your simulation's regions and the zones assigned to them.

4. Once your selections are made, click the **Define Turbo Topology** button and proceed to the next task.

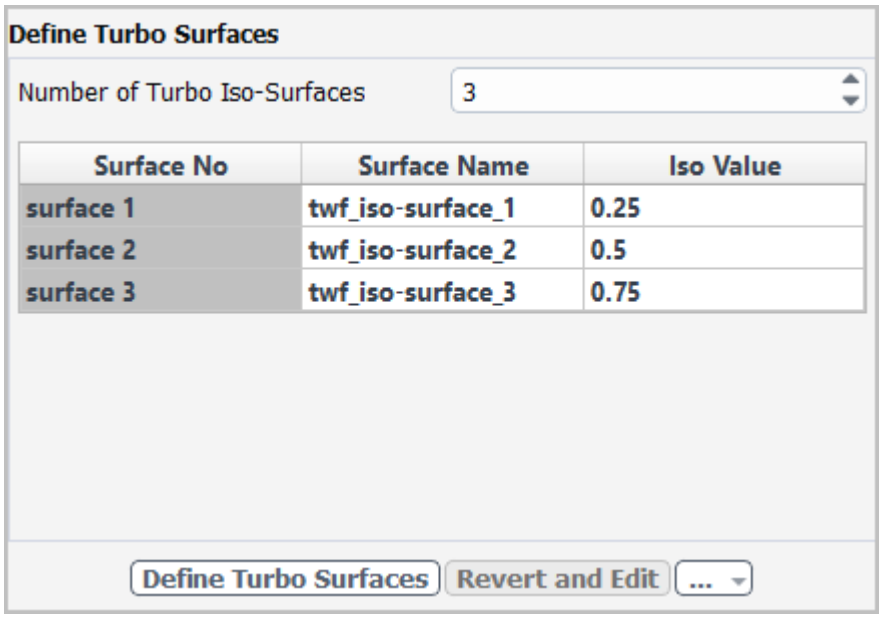
   A separate child task will be created that shows the details of the particular turbo topology. Additional topologies can also be created using this task.

   If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

5. Proceed to the next step in the workflow.

14.1.10. Defining Turbomachinery Surfaces

Use the **Define Turbo Surfaces** task to create turbo surfaces at span-wise locations that will be used in solution post processing. You can define three surfaces by default, but you can create as many or as few as you need.



1. Specify the **Number of Turbo Iso-Surfaces**, or keep the default value of `3`.

2. Specify an **Iso-Value** for **surface 1**, or keep the default value of `0.25`.

3. Specify an **Iso-Value** for **surface 2**, or keep the default value of `0.5`.

4. Specify an **Iso-Value** for **surface 3**, or keep the default value of `0.75`.

5. Repeat as needed, depending on the number of iso-surfaces.

6. Click the **Define Turbo Surfaces** button to generate a list of turbo surfaces.

   If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Proceed to the next step in the workflow.

14.1.11. Creating Turbomachinery Report Definitions and Monitors

Use the **Create Report Definitions and Monitors** task to have the workflow automatically create turbomachinery-related performance parameters such as flow rate, pressure-ratio, efficiency, and so on. In addition, you can use this task to have the workflow create contours for any iso-surfaces you created in the **Define Turbo Surfaces** task.

1. Determine whether or not you want contour plots to be automatically created for the surfaces you defined in the **Define Turbo Surfaces** task.

2. Click the **Create** button to generate the contour plots, if any, and a list of turbomachinery-related report definitions and monitors.

   If you need to make adjustments to any of your settings in this task, click **Revert and Edit**, make your changes and click **Update**, or click **Cancel** to cancel your changes.

3. Once the workflow is completed, you can review various monitors and contours in the Outline View and you can proceed to run the turbo calculation.

   In addition, you can make use of simulation reports within Ansys Fluent to further enhance your turbomachinery analysis. See Using Simulation Reports (p. 4359) for details.

14.1.12. Text Command List for the Turbo Workflow

Commands related to using the Turbo Workflow through the text command interface can be found in `turbo-workflow/`

14.1.13. Editing Tasks in the Turbo Workflow

When a task is complete and up-to-date, all controls within that task are disabled. If you need to make any changes to any of your settings within the task, enable the task's controls by clicking the **Edit** button within the task (or click **Edit** in the context menu from the workflow tree). Once you have made your changes, clicking **Update** restores the task's state (causing all subsequent up-to-date tasks to then become out-of-date) and updates the current task with its new changes. Downstream tasks will remain out-of-date until you update them manually. If you decide to select **Cancel**, then all changes are discarded and the task's state will remain up-to-date.

Some tasks include zone or label selection lists where you have made a selection. To edit such tasks, enable the task's controls by clicking the **Revert and Edit** button (or click **Revert and Edit** in the context menu), thereby reverting to the task's previous state, and causing all subsequent up-to-date tasks to then become out-of-date. Once you have made your changes, clicking **Update** then updates the current task with its new values. If you decide to select **Cancel**, then all changes are discarded and the task's state is not restored. The current task and all downstream tasks will remain out-of-date until you update them manually.

Some tasks include one or more fields whose label includes unit information, such that, from an already completed workflow session, you can return to the beginning of the workflow, change the unit designation and update all subsequent tasks, and the numerical values will be automatically updated corresponding to the new unit settings.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.1.14. Saving and Loading Turbo Workflows

As you progress through using workflows, you can save them at any time using the **Save Workflow** button ( ). When you save the workflow itself, you save the tasks, sub-tasks, and their settings, as your own custom workflow template for future use. You can also load any saved workflow using the **Load Workflow** button ( ). If you make a mistake in setting up your workflow, you can delete the task (using the **Delete** command in the context menu for the selected task), or even reset the entire workflow (using the **Reset Workflow** button ( )).

It should be noted that when you write the mesh file (**File > Write > Mesh...**), Fluent saves the current workflow along with any mesh information within the saved mesh file.

While saving the mesh file, Fluent also locally copies relevant task editing data from the current workflow session's temporary folder into a separate workflow-specific folder (called, for example, *my_mesh_file* `_workflow_files`). This workflow-specific folder contains task editing data, making it easier to use the workflow when reverting and editing tasks, in another Fluent session. Use the corresponding **Save task editing data with mesh file** preference to enable or disable this behavior (see Applying Preferences to the Turbo Workflow (p. 1952) for more information).

Note:

Workflows are saved as workflow template files (using the `*.wft` file extension). The `*.wft` files include the version number of the current release (for example, for the 2024 R2, release, the files are internally designated with `"version": "24.2"`.

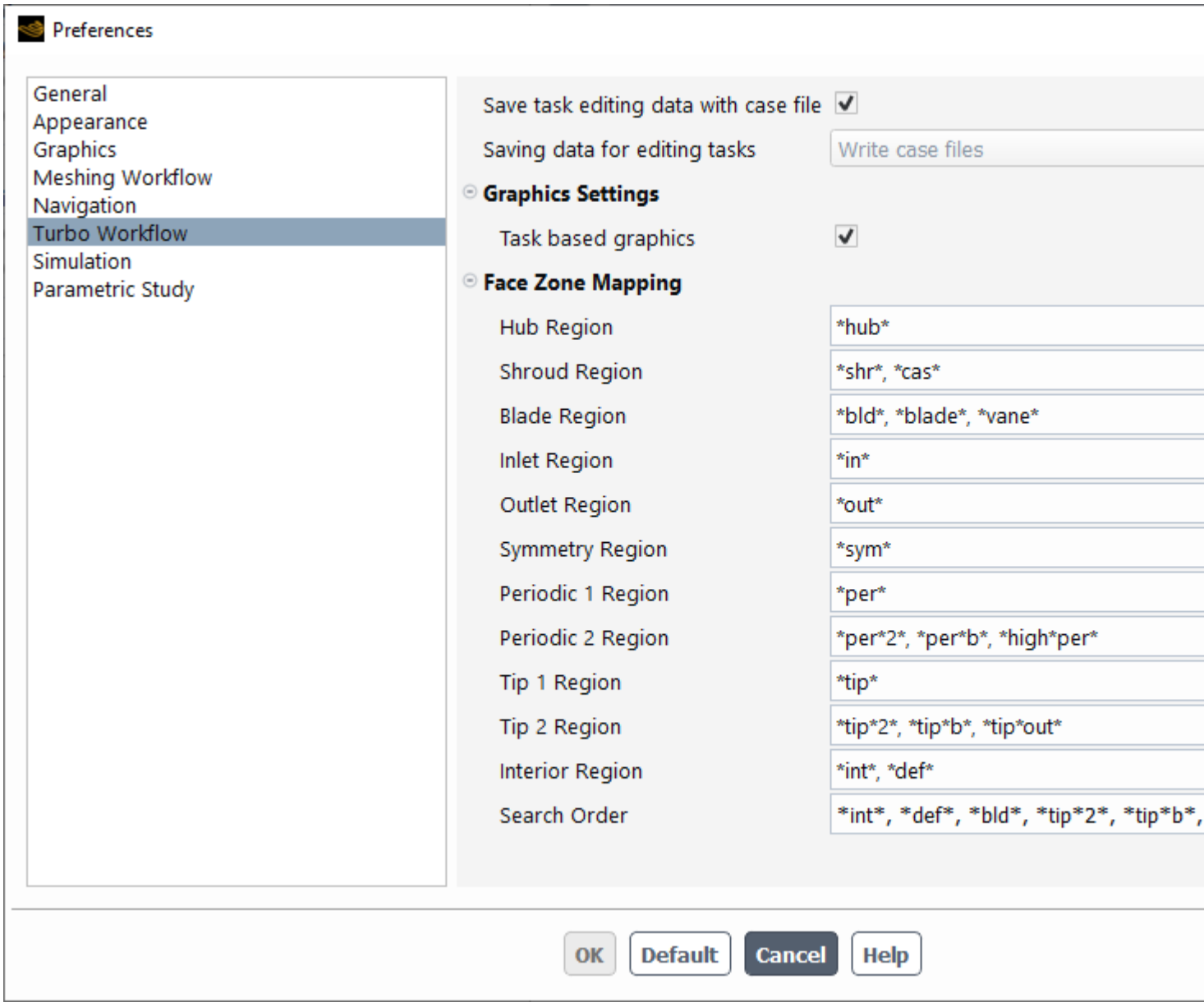
Additionally, all default settings in the workflow files or journals are model-based, therefore, default settings will change when the workflow file(s) are applied to a new model. If, however, you have explicitly changed any setting values, then those changes will be preserved as user settings across various models.

14.1.15. Applying Preferences to the Turbo Workflow

You can access global preferences for the Turbo Workflow in Fluent through the **Preferences** dialog, under the **Turbo Workflow** category.

File → **Preferences...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



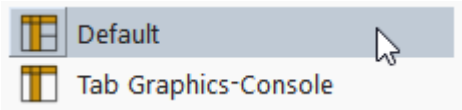
The following categories/preferences are most useful and applicable when using the Turbo Workflow:

- **General** category:
  - When the **Dockable Workflow Editor** preference is enabled, and a new session started, a task's properties can be separated into the **Workflow Editor**, that can then be relocated (undocked) and resized elsewhere in the interface. This is especially useful for workflows with many tasks and/or an abundance of properties in one or more tasks..

    You can relocate the Workflow Editor from its default location to any location in the Fluent interface. To return the editor to its default location, use the **Default** view from the list of view options.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 14.2: Setting the Default Workflow Editor View



- **Turbo Workflow** category
  - Use the **Save task editing data with case file** option to ensure that workflow-specific task editing data is saved when you save the case file. This data will be saved in a corresponding workflow-specific folder alongside the case file (see Saving and Loading Turbo Workflows (p. 1952) for more information). When this option is enabled, the **Saving data for editing tasks** preference is automatically set to **Write case files**.
  - For **Saving data for editing tasks**, you can determine how case files are managed when you edit your workflow tasks. Choices include:

    → Use the **Write case files** option to save a case file into a temporary folder while editing case-related tasks within the workflow.

    - When this option is enabled (the default), Fluent creates a case file for use when a completed case-related task must be edited.
    - When this option is disabled, Fluent will not create a case file, and when the completed case-related task must be edited, Fluent automatically updates each previous task in order to generate the required case file.

      Disabling this preference is recommended if you do not need to edit your workflow tasks once they have been defined.

    → Use **Do not save data** to not save a case file while editing a task.

Note:

If the **Saving data for editing tasks** option is set to **Do not save data**, and you are reverting any changes to a particular task, then updating that task subsequently updates all previous tasks.

Whether the **Saving data for editing tasks** option is set to **Write case files**, case information is saved for a particular task when the task is completed. When you choose to edit any settings to a completed task, the saved case information corresponding to that task is retrieved and the task's settings are restored accordingly.

See Editing Tasks in the Turbo Workflow (p. 1951) for more information about reverting and updating tasks.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

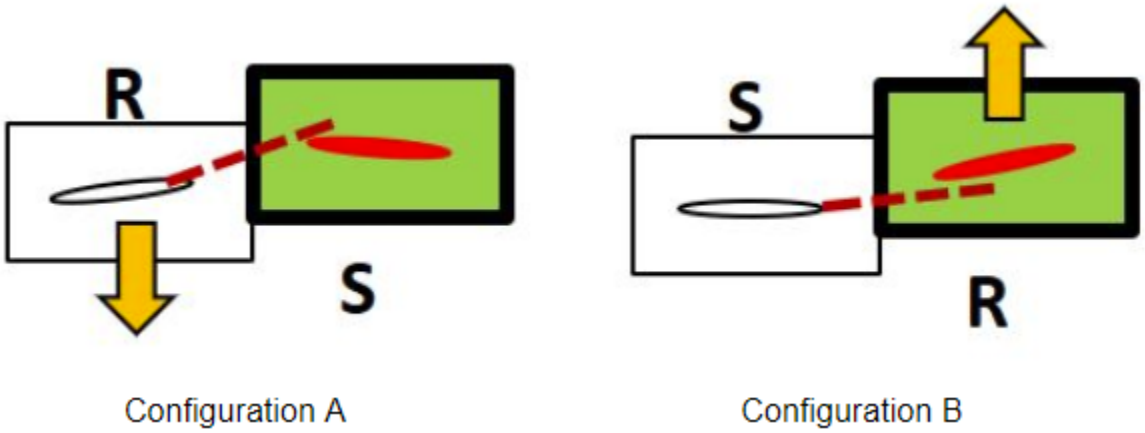
14.2. Frozen Gust / Inlet Disturbance Flow Modeling

To reduce blade row simulation cost in turbomachinery you can model blade row coupling (rotor / stator interaction) using a technique known as frozen gust or inlet disturbance flow modeling. In this modeling strategy, the wake of the upstream row is obtained usually from a steady-state simulation and imposed as an inlet profile on the downstream blade row. The same can be done with the potential field from the downstream row on an upstream row. By doing this, the blade row interaction problem is reduced to a single row with imposed inlet and/or outlet profiles.

There are two possible variations of this flow problem. See Figure 14.3: Frozen Gust variations (p. 1955).

- Wake from a rotor imposed on stator row (configuration A).
- Wake from a stator imposed on rotor row (configuration B).

Figure 14.3: Frozen Gust variations



The common workflow for the frozen gust / inlet disturbance modeling approach is as follows:

1. Perform a steady-state simulation of the blade row interactions for the rotor / stator configuration of interest using either the mixing plane or frozen-rotor method. See The Multiple Reference Frame Model (p. 1779) or Legacy Mixing Plane Model (p. 1966).
2. Extract the wake profile from the neighboring row (wake of the upstream row or potential field of the downstream row). See Reading and Writing Profile Files (p. 964).
3. Expand the wake profile to full wheel using the **Replicate Profile** dialog box. See Replicating Profiles (p. 1581).
4. Apply the expanded profile to the boundary of interest, typically an inlet, on the rotor or stator depending on the configuration.
5. The final step is dependent on which configuration you are modeling.

   For rotor wake imposed on stator passage (configuration A):

   The profile must be linked to a reference frame that is rotating with the rotor. See Using Profiles (p. 1570).

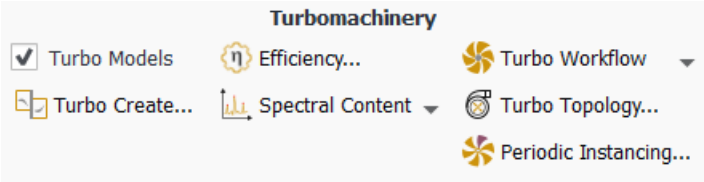
   For stator wake imposed on a rotor passage (configuration B):

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The motion of the rotor with respect to the stationary stator wake profile can be modeled using the standard **Mesh Motion** feature in Fluent, where the rotation of the rotor and the axis of rotation are specified in the fluid cell zone conditions. See Defining Zone Motion (p. 1323).

14.3. Blade Row Interaction Modeling

In Ansys Fluent, the modeling of flow in rotating and stationary components of turbomachines is possible with the use of a **General Turbo Interface** (GTI). The GTI provides a consistent workflow for connecting rotors to stators of different topology from full-wheel models to periodic-sectors to sectors with pitch-change of any pitch-ratio. A GTI can also be used to connect secondary flow paths such as cavities, injections, and bleeds to the main flow path of a turbomachine. GTIs are available in steady and transient simulations, and can be accessed for creation/editing by first selecting **Turbo Models** under **Domain** → **Turbomachinery**.



You can then click **Turbo Create...** from the following location:

- **Turbomachinery** ribbon under the **Domain** tab.

  Domain → **Turbomachinery** → **Turbo Create...**

Creating and editing a GTI is discussed in detail in Creating and Editing General Turbo Interfaces (p. 1962).

In most turbomachines, the number of blades in the rotor rows and the stator rows are not equal. Therefore, the adjacent rotor and stator blade row passages have a pitch-change. The most accurate simulations necessitate full-domain modeling or whenever pitchwise periodicity can be achieved (such as in periodic-sectors models). However, full-domain modeling is computationally demanding (in memory and CPU requirements). A pitch-change capable interface will alleviate these requirements by using one to a few blades per row to model the flow interaction between the rotor and stator blade passages. There are currently three types of pitch-change interfaces under GTI: **Pitch-Scale** (PS), **No Pitch-Scale** (NPS), and **Mixing Plane** (MP). These are discussed in Pitch-Change Models (p. 1957).

The PS and the NPS interface types provide two modeling techniques in handling pitch-change when using Frozen-Rotor (FR) model (a modeling strategy based on Multiple Moving Reference Frame) or Transient Rotor-Stator (TRS) model (a model based on Sliding Mesh Technique).

Note:

In Fluent, the combination of GTI selection and Fluid zone motion specification you choose determines if the interface connecting the rotor to the stator is a Frozen-Rotor (FR), Transient Rotor-Stator (TRS), or Mixing-Plane (MP).

The basic rules are, GTIs straddling a fluid zone with zone motion specified as Mesh Motion are considered to be Transient Rotor-Stator interfaces, with pitch-change type PS or NPS specified during the GTI selection. On the other hand, GTI straddling a fluid zone with zone

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

motion specified as Frame Motion can either be a Mixing-Plane interface or considered to be a Frozen-Rotor interface of the PS or the NPS pitch-change type.

The interface is implicitly defined as a Frozen Rotor or a Transient Rotor-Stator based on the Fluid zone motion specification. Therefore, you do not explicitly make interface selection called a Frozen-Rotor or a Transient Rotor-Stator.

14.3.1. Pitch-Change Models

There are three types of pitch-change interfaces:

14.3.1.1. Pitch-Scale interface

14.3.1.2. No Pitch-Scale interface

14.3.1.3. Mixing-Plane interface

Current compatibilities and limitations of the GTI and pitch-change options are:

- Available for steady-state and transient simulations only.
- Pitch-Scale and No Pitch-Scale options are available with the pressure-based coupled solver, density-based implicit solver, and SIMPLEC pressure based segregated solver (not available with other pressure-based segregated solvers or density-based explicit solver).
- Mixing Plane option is available with the pressure-based coupled solver and density-based implicit solver only (not available with pressure-based segregated solvers or density-based explicit solver).
- Available in relative velocity formulation in steady-state simulation with the pressure-based coupled solver
- Available with all Species models.
- Available with User-Defined Scalars.
- Applies to 3D rotational turbomachines with rotor-stator configuration, such as axial or radial compressors and turbines, or torque converters.
- Not available in 2D configuration or 3D translational problems (such as linear cascade).
- Not available with the following models:
  - Multiphase (except for Wet Steam)
  - Radiation
  - DPM
  - Acoustics
- Repartition is not supported for GTI.
- Migration is not supported for GTI.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.3.1.1. Pitch-Scale interface

You can enable the **Pitch-Scale** interface from the **Create/Edit Turbo Interfaces** dialog box (Figure 14.5: Create/Edit Turbo Interfaces Dialog Box (p. 1963)).

1. Select **General Turbo Interface** under **Interface Options**.
2. Select **Pitch-Scale** under **Pitch-Change Types**.

The **Pitch-Scale** interface maintains connectivity between blade passages by stretching or compressing the solution flux profiles on both sides of the interfaces. Pitch-scaling is valid for 3D rotational turbomachines, such as radial or axial turbines and compressors. It is also used for connecting a secondary flow-path to the main passage flow-path. The pitch-scale interface is available when the pitch-ratio between the connected passages is less than two. For connecting passages with pitch-ratio higher than two or full-wheel interfaces, the **No Pitch-Scale** interface should be used since it can handle any pitch-ratio.

Note:

If you select the **Pitch-Scale** option for passages with pitch-ratio higher than two, Fluent will print a warning message informing you that you cannot use this pitch change option and will recommend that you use the **No Pitch-Scale** option. You can switch from **Pitch-Scale** to **No Pitch-Scale** after interface creation by editing the interface.

Note:

In transient simulations, the **Pitch-Scale** option allows for transient interactions to take place between the connecting passages, but the exact frequency content of the interaction will not be preserved due to the periodic approximation made in this model. Therefore, this modeling strategy can be used to improve aerodynamic performance but not to predict accurate frequency content between the connected blade for a pitch-ratio not equal to one.

The **Pitch-Scale** option is available for use as a Frozen Rotor model if the fluid zone motion is set to Frame Motion or as a Transient Rotor-Stator model if the fluid zone motion is set to Mesh Motion.

14.3.1.2. No Pitch-Scale interface

You can enable the **No Pitch-Scale** interface from the **Create/Edit Turbo Interfaces** dialog box (Figure 14.5: Create/Edit Turbo Interfaces Dialog Box (p. 1963)).

1. Select **General Turbo Interface** under **Interface Options**.
2. Select **No Pitch-Scale** under **Pitch-Change Types**.

The **No Pitch-Scale** interface will maintain connectivity between blade row passages by creating virtual copies of the smaller passage flow profiles. The **No Pitch-Scale** interface is valid for 3D rotational turbomachines, such as radial or axial turbines and compressors. The **No Pitch-Scale** interface can be used for connecting a secondary flow-path to the main passage flow-path. The **No Pitch-Scale** interface is available for any pitch-ratio, including connecting a sector to a 360-degree interface or connecting full-wheel interfaces.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When the **No Pitch-Scale** interface is used between two zones with a large pitch difference, and if either Frame Motion or no motion is specified for the fluid zones, then total pressure loss across the interface can occur. To reduce the total pressure loss, an expert option is available via the following console command:

```
define/turbo-model/general-turbo-interface-settings/expert/nps-minimize-po-loss
```

Note:

You can switch from **Pitch-Scale** to **No Pitch-Scale** after interface creation if the pitch-ratio of the connected passages is less than two.

Note:

In transient simulations, the **No Pitch-Scale** option will allow for transient interactions to take place between the connecting passages, but the exact frequency content of the interaction will not be preserved due to the periodic approximation made in this model. Therefore, this modeling strategy can be used to improve aerodynamic performance due to the enhanced flow interaction captured over steady-state mixing-plane modeling but not to predict accurate frequency content between the connected blades for pitch-ratio not equal to one. The **No Pitch-Scale** interface can be used for asymmetric flow modeling or inlet distortion flow modeling.

Note:

When the **No Pitch-Scale** interface is used to connect a sector to a 360-degree interface, the pitch of a sector should result in an integer number of passages in the 360-degree machine.

The **No Pitch-Scale** option is available for use as a Frozen Rotor model if the fluid zone motion is set to Frame Motion or as a Transient Rotor-Stator model if the fluid zone motion is set to Mesh Motion.

14.3.1.3. Mixing-Plane interface

You can enable the **Mixing-Plane** interface from the **Create/Edit Turbo Interfaces** dialog box (Figure 14.5: Create/Edit Turbo Interfaces Dialog Box (p. 1963)).

1. Select **General Turbo Interface** under **Interface Options**.
2. Select **Mixing Plane** under **Pitch-Change Types**.

The mixing plane model in Fluent provides an alternative to the Frozen Rotor (multiple reference frame) and the Transient Rotor-Stator (sliding mesh models) models for simulating flow through domains with one or more regions in relative motion. In this implementation, a non-conformal interface capable of handling pitch-change topology is used to provide for the flow interaction between the blade rows. A pitchwise averaging procedure to mix out the flow variation between the blade rows is applied to the interfaces. The implicit nature of this implementation should provide a much more robust mixing process as well as fast convergence. This **Mixing-Plane** interface is an

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

alternate and enhanced implementation of the profile-exchange base Mixing Plane model described in Legacy Mixing Plane Model (p. 1966).

As discussed in The Multiple Reference Frame Model (p. 1779), the MRF model and the Frozen Rotor model are applicable when the flow at the interface between adjacent moving/stationary zones is nearly uniform ("mixed out"). If the flow at this interface is not uniform, the Frozen Rotor model may not provide a physically meaningful solution. The Transient Rotor-Stator model (sliding mesh model, see Modeling Flows Using Sliding and Dynamic Meshes (p. 1797)) may be appropriate for such cases, but in many situations it is not practical to employ a Transient Rotor-Stator model. For example, in a multistage turbomachine, if the number of blades is different for each blade row, a large number of blade passages is required in order to maintain circumferential periodicity. Moreover, sliding mesh calculations are necessarily unsteady, and therefore require significantly more computation to achieve a final, time-periodic solution. For situations where using the sliding mesh model is not feasible, the mixing plane model can be a cost-effective alternative.

In the mixing plane approach, each fluid zone is treated as a steady-state problem. Flow-field data from adjacent zones are passed as spatially averaged or "mixed" at the mixing plane interface. This mixing removes any unsteadiness that would arise due to circumferential variations in the passage-to-passage flow field (for example, wakes, shock waves, separated flow), therefore yielding a steady-state result. Despite the simplifications inherent in the mixing plane model, the resulting solutions can provide reasonable approximations of the time-averaged flow field.

The mixing plane model creates either radial or axial bands depending on the machine configuration (radial bands on axial machines and axial bands on radial machines). By default, a maximum of 40 equal width bands are requested for creating the interface bands between the upstream and downstream zone of the interface. The mixing plane model performs viability checks to remove bands with zero or very few faces in order to obtain proper pitchwise averaging. Therefore, while 40 bands are requested, the viability check may result in much fewer bands depending on the mesh density of the upstream and downstream zones of the mixing-plane interface.

After a mixing plane has been created, you can edit the model settings using the TUI command: `/define/turbo-model/general-turbo-interfaces-settings/mixing-plane-model-settings`

A List of General Turbo Interface text commands can be found under `define/` in the *Fluent Text Command List*.

Note:

The **Mixing Plane** interface is available in transient simulations if the specified zone motion for the fluid zone straddling the mixing-plane is defined as Frame Motion. Fluent will not allow you to create a **Mixing Plane** interface on a fluid zone specified with Mesh Motion, and print a warning message.

If you have a case set up using the legacy profile-exchange based mixing-plane model and want instead to replace it with the current interface based mixing-plane model you must do the following:

1. Delete the legacy mixing plane in the legacy Mixing Planes dialog box.

2. Change the upstream and downstream boundaries to be of type interface.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Follow the steps mentioned previously to enable the implicit mixing plane model from the **Create/Edit Turbo Interfaces** dialog box.

14.3.2. Modeling an Ensemble of Blades Per Row using GTI

In some modeling situations, you may need to model more than one blade per row and connect the blade ensemble via GTI in steady or transient simulations. There are restrictions on what can be modeled in Fluent using GTI (**Pitch-Scale** or **No Pitch-Scale**) and an ensemble of blades.

If the pitch-ratio of each ensemble (defined as 360 degrees divided by the pitch of ensemble) on both sides of the interface is an integer then both **Pitch-Scale** and **No Pitch-Scale** interfaces can be used in a **Frame Motion** or **Mesh Motion** simulation. However, if the pitch-ratio is a fraction on either side of the interface then only **Frame Motion** is allowed. In other words, transient-rotor-stator modeling of an ensemble can be performed only for ensembles with a pitch-ratio that is an integer.

For **Mixing Plane** interfaces, an ensemble can have a fraction or integer pitch-ratio because the **Mixing Plane** interface is available only with **Frame Motion** between the rotating and stationary zone.

Note:

Frame Motion is available in both steady-state and transient simulation, while **Mesh Motion** is available only in transient simulation.

Figure 14.4: Ensemble Blade Rows with a GTI Interface (p. 1962) shows multi-blade passage rows with a GTI between the rotor and IGV. The pitch-ratio of the rotor ensemble is a fraction while the pitch-ratio of the IGV ensemble is an integer (see Table 14.1: Pitch Ratio of Ensemble Blade Rows with GTI (p. 1962)). In this example, you could only use the **Pitch-Scale** or **No Pitch-Scale** interface using **Frame Motion**, in either steady or transient simulation. This configuration is not suitable to model a transient-rotor-stator simulation where **Mesh Motion** must be specified for the rotor as the ensemble pitch-ratio is non-integer.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 14.4: Ensemble Blade Rows with a GTI Interface

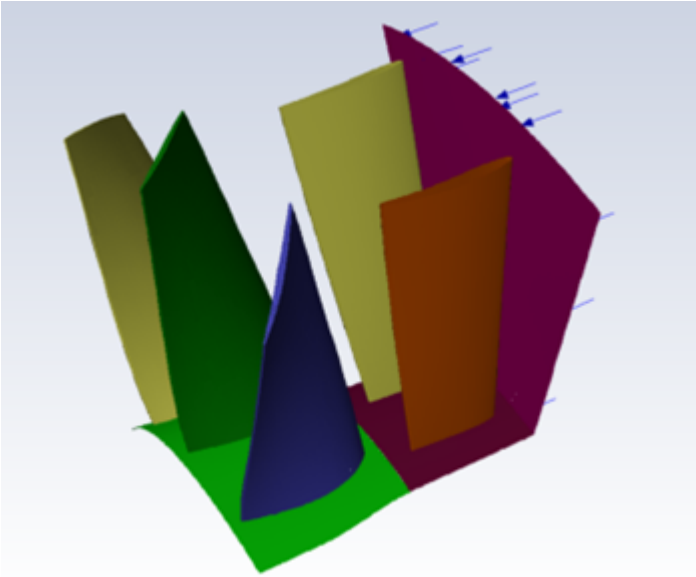
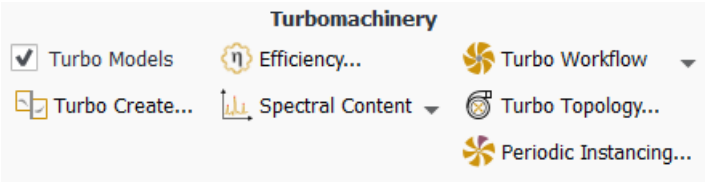


Table 14.1: Pitch Ratio of Ensemble Blade Rows with GTI

Blade Row	# Blades in 360 deg.	# Blades in model	Pitch of ensemble deg.	Ensemble pitch-ratio (360/Pitch of ensemble)
IGV	20	2	36	10
Rotor	16	3	67.5	5.333

14.3.3. Creating and Editing General Turbo Interfaces

Most of the turbomachinery related interfaces required for blade row interaction can now be created from the **Create/Edit Turbo Interfaces** (Figure 14.5: Create/Edit Turbo Interfaces Dialog Box (p. 1963)) panel. To access the **Create/Edit Turbo Interfaces** dialog box, you must first select **Turbo Models** from the ribbon under **Domain** → **Turbomachinery**.

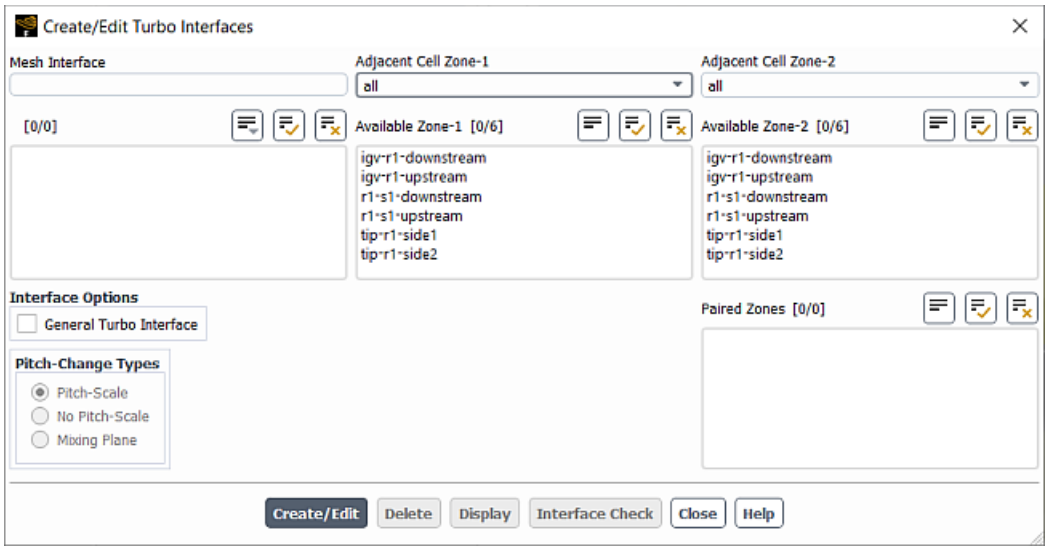


Afterwards you can open the **Create/Edit Turbo Interfaces** dialog box from the ribbon:

Domain → **Turbomachinery** → **Turbo Create...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 14.5: Create/Edit Turbo Interfaces Dialog Box

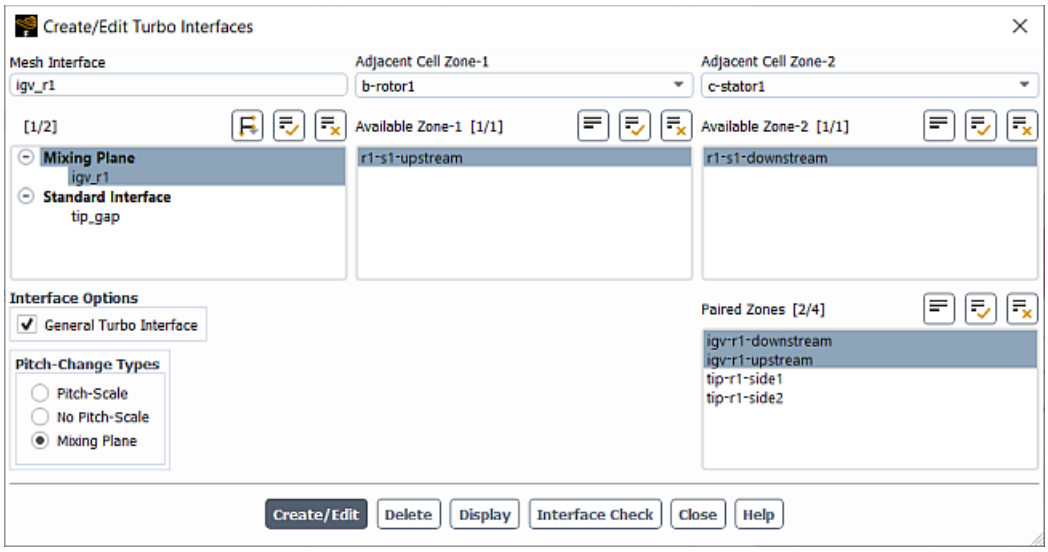


1. Enter a name for the interface in the **Mesh Interface** text-entry box.

2. Specify the interface zones that make up the mesh interface by selecting a zone in the **Available Zone-1** list and the appropriate pair in the **Available Zone-2** list.

   You can filter the available zones using the **Adjacent Cell Zone-1** and **Adjacent Cell Zone-2** drop down lists. When you select a cell zone, the list of available interface zones are reduced to only those that are adjacent to that cell zone and currently not defined in another interface. Once the available zones are used in an interface, the zones will move to the **Paired Zones** list.

   Figure 14.6: Create/Edit Turbo Interfaces Dialog Box with Adjacent Zone Filtering



3. Enable the desired **Interface Options**:

   - You can create a non-turbo, non-periodic interface (for example, a tip gap interface) by leaving all **Interface Options** unselected.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Enable **General Turbo Interface** to connect two turbo zones.

  The following **Pitch-Change Types** are available:

  – **Pitch-Scale**

  – **No Pitch-Scale**

  – **Mixing Plane**

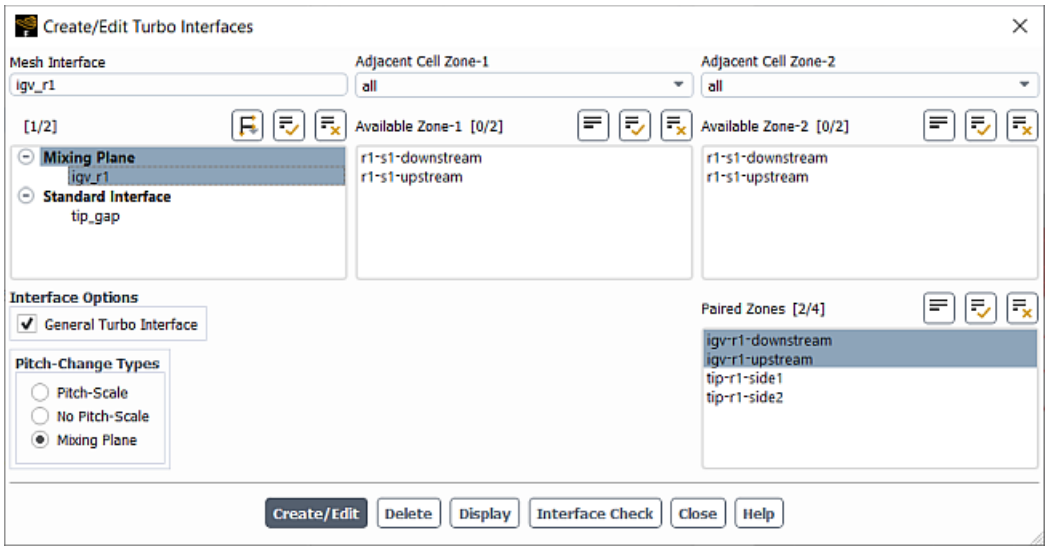
  See Pitch-Change Models (p. 1957) for more information on these options.

4. Click **Create/Edit...** to create a new mesh interface. The newly created interface appears in the list under **Mesh Interface**.

   Highlighting an interface in the list will also highlight the appropriate interface zones in the **Paired Zones** list. You can use the button to group the already created mesh interfaces by name or by type. Grouping by type as shown below can be helpful to quickly identify a GTI interface.

   You can also multi-select already created Turbo Interfaces to edit, delete, display, and interface check multiple interfaces simultaneously. Note that you can only convert multi-selected turbo interfaces to a single Pitch-Change type.

Figure 14.7: Create/Edit Turbo Interfaces Dialog Box for a Specified Interface



Note:

Whenever you delete and recreate a turbo interface during a solver run or if you are setting up a turbo interface for an initial run, you must initialize the solution before solving.

5. The buttons at the bottom of the dialog box allow you to take further actions, such as the following:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you create an incorrect mesh interface, you can select it in the selection list and click the **Delete** button to delete it. Any additional zones that were created when the interface was created will also be deleted.
- You can click the **Display** button to display interface zones or mesh interfaces in the graphics window. Displaying is particularly useful if you want to check the location of the interface zones prior to setting up a mesh interface.
- The **Interface Check** button displays the following information about the selected interface(s) in the console:
  - Interface name
  - Interface options

    Displays the Pitch-Change Type for the selected interface.
  - Blade-row-interaction configuration

    Displays the interface configuration as sector-to-sector, 360-to-sector, or 360-to-360.
  - Interface zone pitch
  - Pitch ratio

    Displays the ratio of larger pitch/smaller pitch.
  - Zone/side names
  - Zone/side area
  - Total intersection percentage

    Typically, this number should be close to 100% to indicate full interface connectivity.
  - Periodic boundary touching zone/side check

    This check informs whether a shared edge exists between the periodic boundaries and a zone/side forming the GTI interface for a blade row sector. A failure suggests you need to correct the geometry so that the periodic boundary touches the side/zone forming the GTI interface.
  - Ghost link check

    This check examines whether an appropriate link exists between a ghost cell and its associated real cell. A failure means the Fluent solver may not run successfully.

    Note:

    A ghost cell is the translated/rotated copy of a real cell in the domain. Fluent uses the ghost cell method to transfer flow variables between two different cell zones with the General Turbo Interface or in the phase-lag method for the same cell zone in the circumferential direction. A ghost link is the link between the ghost cell and its associated real cell.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– Face ordering check

This check inspects whether the faces on ghost cells are created in the same order as on the real cells.

An example of an **Interface Check** is shown below:

```
-------------------------------------------------------------------------------------------------
 Interface Name:     igv-r1-ps
 Interface Options:  Turbo - Pitch-Scale - Blade row interaction
 Blade-row-interaction configuration : sector-to-sector
 Interface zone1 pitch(deg) : 13.846154, Interface zone2 pitch(deg) : 15.652174
 Pitch ratio :   1.1304
-------------------------------------------------------------------------------------------------
                         Side-1 name (id): igv-r1-upstream (12)
                              Side-1 area: 2.79350456e-03
                         Side-2 name (id): igv-r1-downstream (28)
                              Side-2 area: 3.15787308e-03
                 Side1 intersection total:  99.9996%
                 Side2 intersection total:  99.9993%
 Checking ghost cell for a sliding interface: igv-r1-ps
 for thread 12, adj. thread0 14, adj. thread1 16
 for thread 28, adj. thread0 30, adj. thread1 31
Periodic boundary touching igv-r1-upstream check...successful.
Periodic boundary touching igv-r1-downstream check...successful.
Ghost link check...successful.
Face ordering check...successful.
```

14.3.4. Legacy Mixing Plane Model

This mixing-plane model is based on profile boundary exchange. It can be accessed from the TUI command:

`/define/turbo-model/legacy-models/legacy-mixing-plane>`

It is advised that you utilize the updated mixing plane model from the General Turbo Interface described in Blade Row Interaction Modeling (p. 1956).

Note:

The legacy mixing plane model is only available if you have deselected **Enable** under **Turbo Model**.

14.3.4.1. Limitations

Note the following limitations of the mixing plane model:

- The LES turbulence model cannot be used with the mixing plane model.
- The models for species transport and combustion cannot be used with the mixing plane model.
- The VOF multiphase model cannot be used with the mixing plane model.
- The discrete phase model cannot be used with the mixing plane model for coupled flows. Non-coupled computations can be done, but you should note that the particles leave the domain of the mixing plane.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can find more information about the following topics in the Theory Guide:

Rotor and Stator Domains
The Mixing Plane Concept
Choosing an Averaging Method
Mixing Plane Algorithm of Ansys Fluent
Mass Conservation
Swirl Conservation
Total Enthalpy Conservation

14.3.4.2. Setting Up the Legacy Mixing Plane Model

The model inputs for mixing planes are presented in this section. Only those steps relevant specifically to the setup of a mixing plane problem are listed here. Note that the use of wall and periodic boundaries in a mixing plane model is consistent with their use when the model is not active.

1. Select the **Absolute** or **Relative Velocity Formulation** in the General Task Page (p. 4684), when the pressure-based solver is enabled.

   Setup → **General**

   Important:

   When the density-based solver is enabled, only the **Absolute Velocity Formulation** can be used with the mixing plane model.

2. For each cell zone in the domain, specify its angular velocity (ω) and the axis about which it rotates.

   Setup → **Cell Zone Conditions**

   a. If the zone is rotating, or if you plan to specify cylindrical-velocity or flow-direction components at inlets to the zone, you must define the axis of rotation for the frame of reference. In the **Fluid** dialog box or Solid Dialog Box (p. 4973), specify the **Rotation-Axis Origin** and **Rotation-Axis Direction** under the **Reference Frame** tab.

   b. In the **Fluid** (Figure 10.4: The Fluid Dialog Box Displaying Frame Motion Inputs (p. 1774)) or **Solid** dialog box, enable the **Frame Motion** option and then set the **Speed** under **Rotational Velocity** and/or the **X**, **Y**, and **Z** components of the **Translational Velocity** in the expanded portion of the dialog box under the **Reference Frame** tab.

   Important:

   It is important to define the axis of rotation for the cell zones on *both* sides of the mixing plane interface, including the stationary zone.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Define the velocity boundary conditions at walls, as described in step 3 of Setting Up Multiple Reference Frames (p. 1782).

4. Define the boundary conditions at the inlets, as described in Boundary Conditions (p. 1392). For velocity inlets, you can choose to define either absolute velocities or velocities relative to the motion of the adjacent cell zone (specified in step 2). For pressure inlets and mass-flow inlets, the specification of the flow direction and total pressure will always be absolute, because the absolute velocity formulation is always used for mixing plane calculations. For a mass-flow inlet, you do not need to specify the mass flow rate or mass flux. Ansys Fluent will automatically select the **Mass Flux with Average Mass Flux** specification method and set the correct values when you create the mixing plane, as described in More About Mass Flux and Average Mass Flux (p. 1420).

   Details about these inputs are presented in Defining the Flow Direction (p. 1405), Defining the Velocity (p. 1413), and Inputs at Mass-Flow Inlet Boundaries (p. 1418).

   Important:

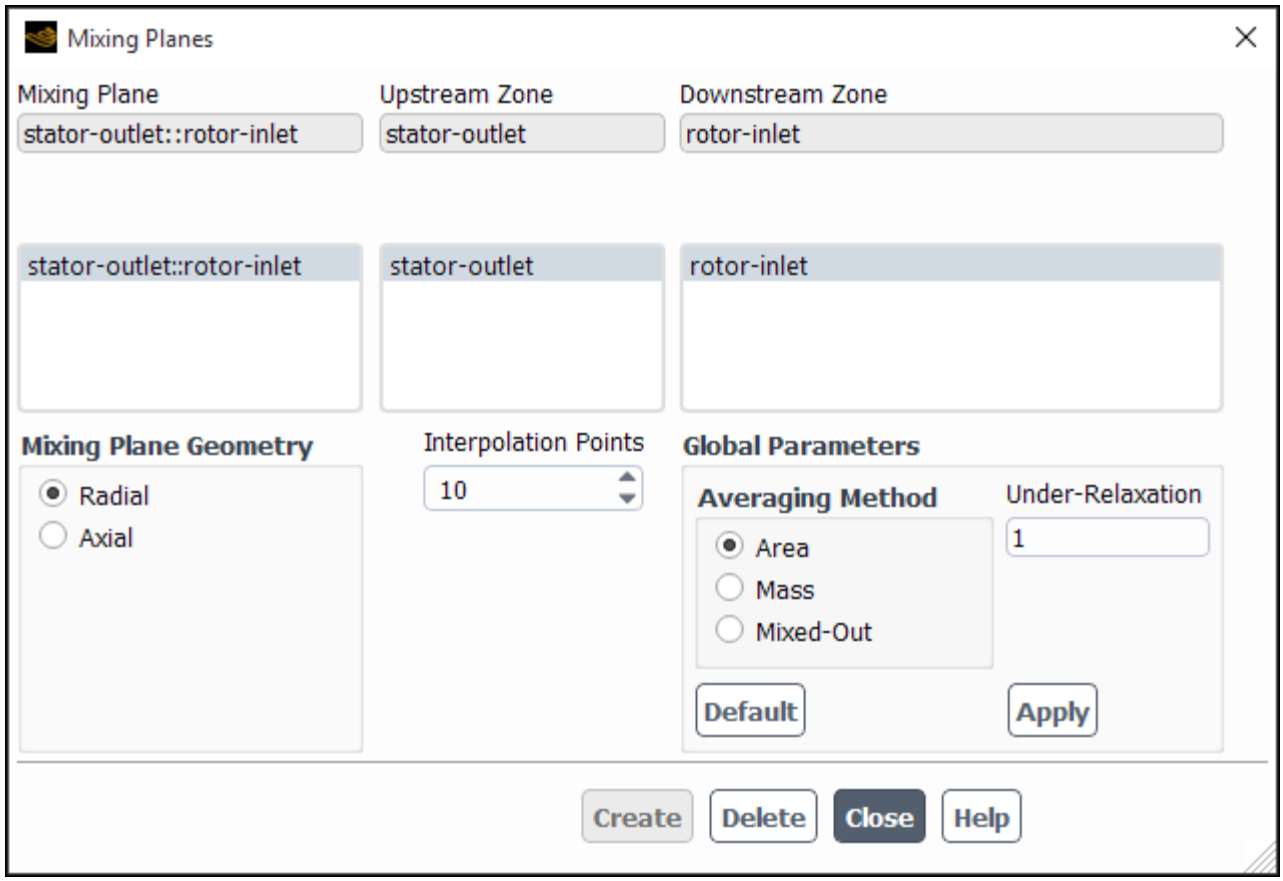
   Note that the outlet boundary zone at the mixing plane interface must be defined as a pressure outlet, and the inlet boundary zone at the mixing plane interface must be defined as either a velocity inlet (incompressible flow only), a pressure inlet, or a mass-flow inlet. The overall inlet and exit boundary conditions can be any suitable combination permitted by the solver (for example, velocity inlet, pressure inlet, or mass-flow inlet; pressure outlet). Remember, however, that if mass conservation across the mixing plane is important, you must use a mass-flow inlet as the downstream boundary; strict mass conservation is *not* maintained across the mixing plane when you use a velocity inlet or pressure inlet.

5. Define the mixing planes in the Mixing Planes Dialog Box (p. 5538) (Figure 14.8: The Mixing Planes Dialog Box (p. 1969)) accessed only from the TUI:

   ```
   /define/turbo-model/legacy-models/legacy-mixing-plane/mixing-plane-gui
   ```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 14.8: The Mixing Planes Dialog Box



a. Specify the two zones that make up the mixing plane by selecting an upstream zone in the **Upstream Zone** list and a downstream zone in the **Downstream Zone** list. It is essential that the correct pairs be chosen from these lists (that is, that the boundary zones selected lie on the mixing plane interface). You can check this by displaying the mesh.

Setup → **General** → **Display...**

b. (3D only) Indicate the geometry of the mixing plane interface by choosing one of the options under **Mixing Plane Geometry**.

A **Radial** geometry signifies that information at the mixing plane interface is to be circumferentially averaged into profiles that vary in the radial direction, for example, $p(r)$, $T(r)$. This is the case for axial-flow machines, for example.

An **Axial** geometry signifies that circumferentially averaged profiles are to be constructed that vary in the axial direction, for example, $p(x)$, $T(x)$. This is the situation for a radial-flow device.

Important:

Note that the radial direction is normal to the rotation axis for the fluid zone and the axial direction is parallel to the rotation axis.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. (3D only) Set the number of **Interpolation Points**. This is the number of radial or axial locations used in constructing the boundary profiles for circumferential averaging. You should choose a number that approximately corresponds to the resolution of the surface mesh in the radial or axial direction. Note that while you can use more points if you want, the resolution of the boundary profile will only be as fine as the resolution of the surface mesh itself.

   In 2D the flow data is averaged over the entire interface to create a profile consisting of a single data point. For this reason you do not need to set the number of **Interpolation Points** or select a **Mixing Plane Geometry** in 2D.

d. Set the **Global Parameters** for the mixing plane.

   i. Select the **Averaging Method**. The **Area** averaging method is the default method. For detailed information about each of the **Area**, **Mass**, or **Mixed-Out** options, see Choosing an Averaging Method in the Theory Guide.

   ii. Set the **Under-Relaxation** parameter. It is sometimes desirable to under-relax the changes in boundary values at mixing planes as these may change very rapidly during the early iterations of the solution and cause the calculation to diverge. The changes can be relaxed by specifying an under-relaxation less than 1. The new boundary profile values are then computed using

$$\phi_{new} = \phi_{old} + \alpha\left(\phi_{calculated} - \phi_{old}\right) \tag{14.1}$$

   where α is the under-relaxation factor. Once the flow field is established, the value of α can be increased.

   iii. Click **Apply** to set the **Global Parameters**. If the **Default** button is visible to the right of the **Apply** button, clicking the **Default** button will return **Global Parameters** back to their default values. The **Default** button will then change to be a **Reset** button. Clicking the **Reset** button will change the **Global Parameters** back to the values that were last applied.

e. Click **Create** to create a new mixing plane. Ansys Fluent will name the mixing plane by combining the names of the zones selected as the **Upstream Zone** and **Downstream Zone** and enter the new mixing plane in the **Mixing Plane** list.

   If you create an incorrect mixing plane, you can select it in the **Mixing Plane** list and click the **Delete** button to delete it.

14.3.4.2.1. Modeling Options

There are two options available for use with the mixing plane model: a fixed pressure level for incompressible flows, and the swirl conservation described in Swirl Conservation in the Theory Guide.

14.3.4.2.1.1. Fixing the Pressure Level for an Incompressible Flow

For certain turbomachinery configurations, such as a torque converter, there is no fixed-pressure boundary when the mixing plane model is used. The mixing plane model is usually used to model the three interfaces that connect the components of the torque converter. In this configuration, the pressure is no longer fixed. As a result, the pressure may float unbounded, making it difficult to obtain a converged solution.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To resolve this problem, Ansys Fluent offers an option for fixing the pressure level. When this option is enabled, Ansys Fluent will adjust the gauge pressure field after each iteration by subtracting from it the pressure value in the cell closest to the **Reference Pressure Location** in the Operating Conditions Dialog Box (p. 4978).

Important:

This option is available only for incompressible flows calculated using the pressure-based solver.

To enable the fixed pressure option, use the `fix-pressure-level` text command:

```
/define/turbo-model/legacy-models/legacy-mixing-plane/set/fix-pressure-level
```

14.3.4.2.1.2. Conserving Swirl Across the Mixing Plane

Conservation of swirl is important for applications such as torque converters (Swirl Conservation in the Theory Guide). If you want to enable swirl conservation across the mixing plane, you can use the commands in the `conserve-swirl` text menu:

```
/define/turbo-model/legacy-models/legacy-mixing-plane/set/conserve-swirl
```

To turn on swirl conservation, use the `enable?` text command. Once the option is turned on, you can ask the solver to report information about the swirl conservation during the calculation. If you turn on `verbosity?`, Ansys Fluent will report for every iteration the zone ID for the zone on which the swirl conservation is active, the upstream and downstream swirl integration per zone area, and the ratio of upstream to downstream swirl integration before and after the correction.

To obtain a report of the swirl integration at every pressure inlet, pressure outlet, velocity inlet, and mass-flow inlet in the domain, use the `report-swirl-integration` command. You can use this information to determine the torque acting on each component of the turbomachinery according to Equation 2.22 (in the Theory Guide).

14.3.4.2.1.3. Conserving Total Enthalpy Across the Mixing Plane

One of the options available in the mixing plane model is to conserve total enthalpy across the mixing plane. This is a desirable feature because global parameters such as efficiency are directly related to the change in total enthalpy across a blade row or stage.

The procedure for ensuring conservation of total enthalpy simply involves adjusting the downstream total temperature profile such that the integrated total enthalpy matches the upstream integrated total enthalpy.

If you want to enable total enthalpy conservation, you can use the commands in the `conserve-total-enthalpy` text menu:

```
/define/turbo-model/legacy-models/legacy-mixing-plane/set/conserve-total-enthalpy
```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To turn on total enthalpy conservation, use the `enable?` text command. Once the option is turned on, you can ask the solver to report information about the total enthalpy conservation during the calculation. If you turn on `verbosity?`, Ansys Fluent will report at every iteration the zone ID for the zone on which the total enthalpy conservation is active, the upstream and downstream heat flux, and the ratio of upstream to downstream heat flux.

14.3.4.3. Solution Strategies for Mixing Plane Problems

It should be emphasized that the mixing plane model is a reasonable approximation so long as there is no significant reverse flow in the vicinity of the mixing plane. If significant reverse flow occurs, the mixing plane will not be a satisfactory model of the actual flow. In a numerical simulation, reverse flow often occurs during the early stages of the computation even though the flow at convergence is not reversed. Therefore, it is helpful in these situations to first obtain a provisional solution using *fixed* conditions at the rotor-stator interface. The mixing plane model can then be enabled and the solution run to convergence.

If you are using the mass or mixed-out averaging method and you are experiencing convergence problems in the presence of severe reverse flow, initialize your solution using the default area-averaging method, then switch to mass or mixed-out averaging after the reverse flow dies out.

Under-relaxing the changes in the mixing plane boundary values can also help in troublesome situations. In many cases, setting the under-relaxation factor to a value less than `1` can be helpful. Once the flow field is established, you can gradually increase the under-relaxation factor.

14.4. Aerodynamic Damping (Blade Flutter Analysis)

The forced vibration analysis in turbomachinery (a.k.a. Forced Response Analysis) is essential for predictions of blade structural failures. An important ingredient in this analysis is the prediction of the blade aerodynamic damping value from blade flutter analysis. The aerodynamic damping value is directly impacted by the severity and characteristic of the vibration and the nature of the blade response.

Blade flutter usually occurs at the natural frequency of the blade-disc assembly. This natural frequency can be determined, along with blade displacements (mode-shapes), using Modal analysis in Ansys Mechanical (or other third party FEM software) prior to performing a flow analysis on the blade. In general, a rotor-disc assembly can have many vibration modes. However, only a limited number of these modes contribute to flutter and only these modes are considered in a blade flutter simulation setup.

Ansys Fluent offers two methods for the calculation of the aerodynamic damping value: the Travelling Wave Method and the Influence Coefficient Method.

The following topics are discussed:

14.4.1. Common Settings for a Blade Flutter Case
14.4.2. Traveling Wave Method (TWM)
14.4.3. Influence Coefficient Method (ICM)
14.4.4. Common Postprocessing for a Blade Flutter Case

14.4.1. Common Settings for a Blade Flutter Case

A typical process to set up a blade flutter simulation follows:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Set-up a steady-state case and run it to get a solution that can be used later for initialization purposes.
2. Read the mode-shape profile file.

   For details, see Reading the Mode Shapes (p. 1973).
3. Switch the simulation to transient.
4. Configure **Run Calculation** settings.

   Solution → **Run Calculation** → **Run Calculation...**

   For details, see Configuring Run Calculation Settings (p. 1976).
5. Set up dynamic mesh zones that impose blade flutter and other movement specifications on the boundaries.

   a. Enable **Dynamic Mesh** for the domain as described in Turning on Dynamic Mesh (p. 1978).

   b. On the **Periodic Displacement** tab, define periodic displacement settings as described in Defining the Periodic Displacement of the Blades (p. 1978).

   c. Assign proper motion specifications to every boundary as described in Creating and Applying Dynamic Mesh Zones (p. 1986).

   For details, see Using Dynamic Mesh Zones in a Blade Flutter Simulation (p. 1977).

   Important:

   It may be necessary to initialize the flow field immediately after specifying the dynamic mesh zones and before specifying any Report Definitions, otherwise you maybe receive an error message. This is due to data required by the Report Definitions only getting interpolated to the blades when the flow is initialized.
6. Set up solution monitors for aerodynamic damping.

   Solution → **Reports** → **Definitions** → **New** → **Aeromechanics Report**

   There are separate reports available for the Travelling Wave Method and the Influence Coefficient Method.

   For details, see Common Postprocessing for a Blade Flutter Case (p. 2003).

14.4.1.1. Reading the Mode Shapes

The blade displacements (mode shapes) for a blade flutter case are encoded in a mode shape profile file, which can be obtained by performing a modal analysis in Ansys Mechanical or third-party FEM software.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For more information on how to export mode shape profiles from Ansys Mechanical, see the following APDL commands:

- **EXPROFILE** - for exporting the mode shape profile.
- **EXOPTION** - for including face connectivity in the profile export.

A mode shape profile file from Ansys Mechanical has a `.csv` file extension and can be read as you would a regular boundary profile:

File → Read → Profile...

For details on reading profiles, see Reading Profile Files (p. 964).

The format expected by Fluent is a file that lists:

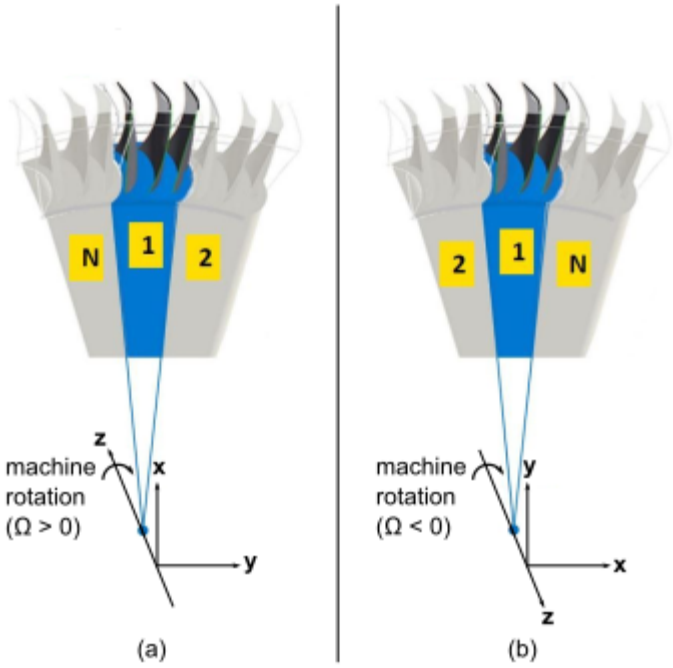
- [Name] - The name of the mode shape profile.
- [Parameters] - Parameters for various data that can be used during run setup. For example, the frequency of the mode shape is reported by Ansys Mechanical in the profile, which can then be used in the blade displacement definition. For details, see Defining the Periodic Displacement of the Blades (p. 1978).
- [Spatial Fields] - The mesh nodal coordinate system.
- [Data] - The normalized or non-normalized displacement values with real only (or with real and imaginary) components.
- [Faces] - Contains face connectivity information which allows you to preview the profile in the graphics window as a surface, which can be useful to ensure the profile is oriented correctly. To do this, click **Preview** on Figure 7.96: The Profiles Dialog Box (p. 1572). For details on reorienting a profile, see Reorienting Profiles (p. 1576).

Passage number (or sector tag) information is also required. This field is added as an additional column in the [Data] section of the profile. It is automatically generated when the profile is replicated from its original single passage modal shape calculation. For details on replicating a profile, see Replicating Profiles (p. 1581).

The passage number is independent of machine rotation and starts with a value of one for the original passage and increases around the wheel in a direction established using the right hand rule and the global coordinate frame, as shown in figure Figure 14.9: Passage Numbering from Profile Expansion (p. 1975).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 14.9: Passage Numbering from Profile Expansion



The machine rotation can be specified either as cell zone **Frame Motion** or **Mesh Motion** as described in Defining Zone Motion (p. 1323). In the case of **Mesh Motion**, extra care must be taken to ensure the profile is aligned with the zone mesh throughout the simulation. Therefore, a local reference frame must be defined and set with the same rotation speed and direction as the zone. For details on specifying a local reference frame, see Creating and Using Reference Frames (p. 1161).

Important:

When a combination of cell zone Mesh Motion and Periodic Displacement with profiles is used, Fluent checks if a local reference frame is set and tracking the zone (that is, same rotation and direction as the cell zone). If this requirement is not met, Fluent prompts you to create a local reference frame and link it to the profile that is being used in the Periodic Displacement.

Here is an abbreviated example of a valid mode shape profile file (with "..." representing omitted similar lines):

```
[Name]
modeshape1

[Parameters]
Ncompt = 3
Nnodes = 9080
Mass = 3.64422308E-002 [kg]
Frequency = 1075.7960 [Hz]
Maximum Displacement = 19.161390 [m]

[Spatial Fields]
Initial X [m], Initial Y [m], Initial Z [m]

[Data]
Initial X [m], Initial Y [m], Initial Z [m], norm meshdisptot x [ ], norm meshdisptot y [ ], norm meshdisptot z
```

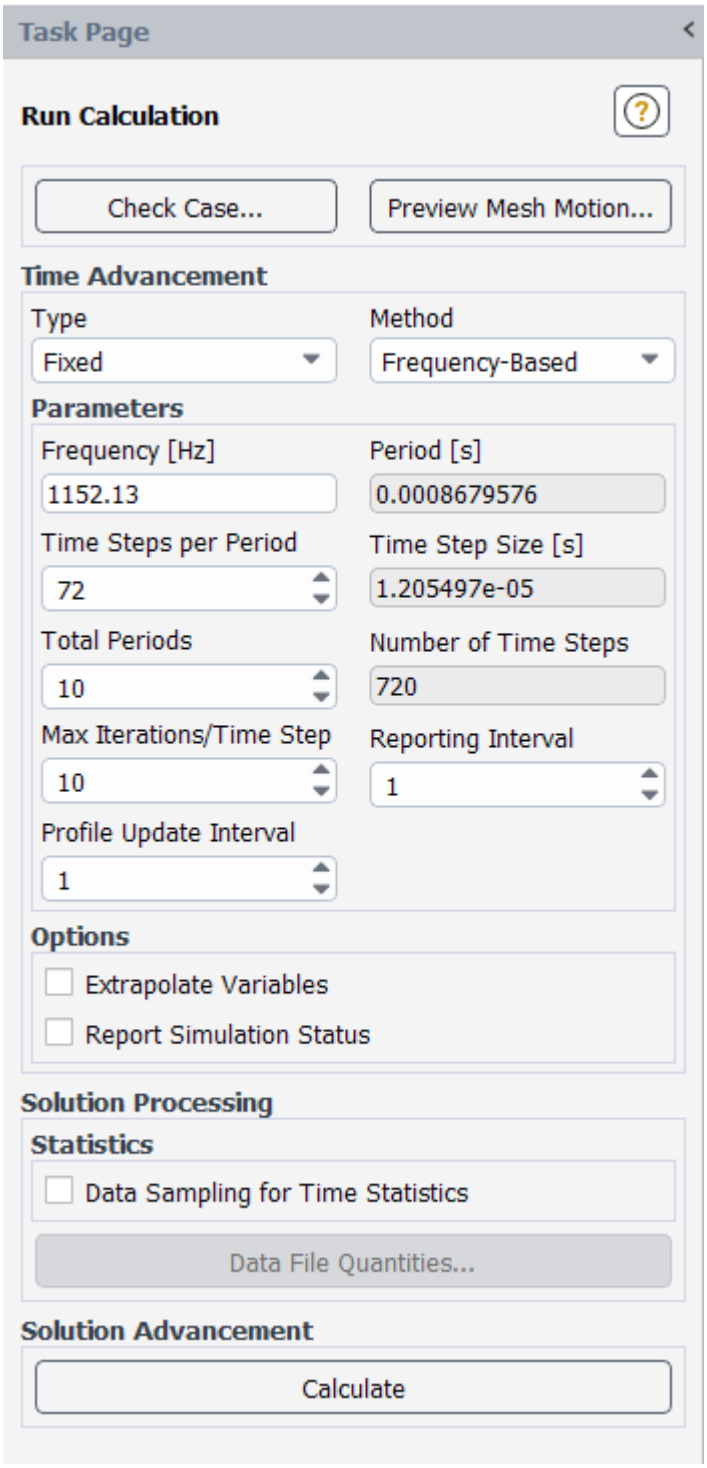
```
 0.24326721     ,  2.31312828E-002,  3.40935170E-002,  6.64623727E-002, -0.29486747     ,  0.26849515
 0.24326396     ,  2.31793349E-002,  3.40879478E-002,  6.70239889E-002, -0.29487824     ,  0.26807998
 0.25071740     , -2.49000213E-002,  6.40013147E-003, -0.11782411     , -0.61531118     ,  0.77716603
 0.25072379     , -2.48944600E-002,  6.37840505E-003, -0.11739263     , -0.61562871     ,  0.77721180
 0.25073487     , -2.48733125E-002,  6.34503139E-003, -0.11655654     , -0.61613393     ,  0.77716960
...
[Faces]
         127,          128,          377,          378
         706,          455,          456,          705
         128,          375,          376,          377
         705,          456,          457,          458
         127,          410,          129,          128
         706,          705,          704,          423
         415,          410,          127,          126
         418,          707,          706,          423
...
```

14.4.1.2. Configuring Run Calculation Settings

For the **Time Advancement** settings in the **Run Calculation** task page:

1. Set **Type** to `Fixed`.
2. Set **Method** to `Frequency-Based`.
3. Set **Frequency** to the frequency used for periodic-displacement mesh motion.
4. Set **Time Steps per Period** to determine the time step size for integration.
5. Set **Total Periods** and **Max Iterations/Time Step** as appropriate.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



For details on this task page, see Run Calculation Task Page (p. 5207).

14.4.1.3. Using Dynamic Mesh Zones in a Blade Flutter Simulation

To apply blade flutter, you can set up dynamic mesh zones and apply them to the boundaries. The dynamic mesh zones that are applied to the wall surfaces are driven according to periodic displacement settings that involve the Aerodynamic Damping option.

The following topics are discussed:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.4.1.3.1. Turning on Dynamic Mesh
14.4.1.3.2. Defining the Periodic Displacement of the Blades
14.4.1.3.3. Creating and Applying Dynamic Mesh Zones

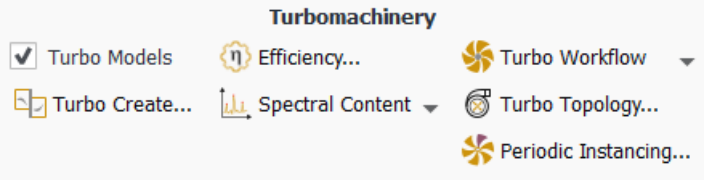
14.4.1.3.1. Turning on Dynamic Mesh

Blade flutter is applied to the blades using Dynamic Mesh. You must enable Dynamic Mesh before you can define the Periodic Displacement for the blades.

To turn on Dynamic Mesh:

1. Enable **Turbo Models** for the domain.

   Domain → **Turbomachinery** → **Turbo Models**

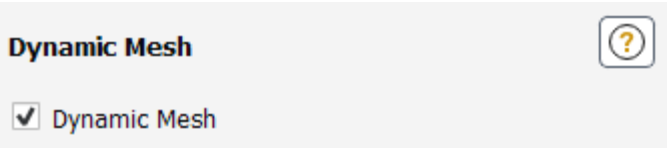
   

   When **Turbo Models** is enabled, the Dynamic Mesh feature reveals an option for Periodic Displacement.

2. Enable **Dynamic Mesh**.

   Domain → **Mesh Models** → **Dynamic Mesh...**

3. In the **Dynamic Mesh** task page, select **Dynamic Mesh**.

14.4.1.3.2. Defining the Periodic Displacement of the Blades

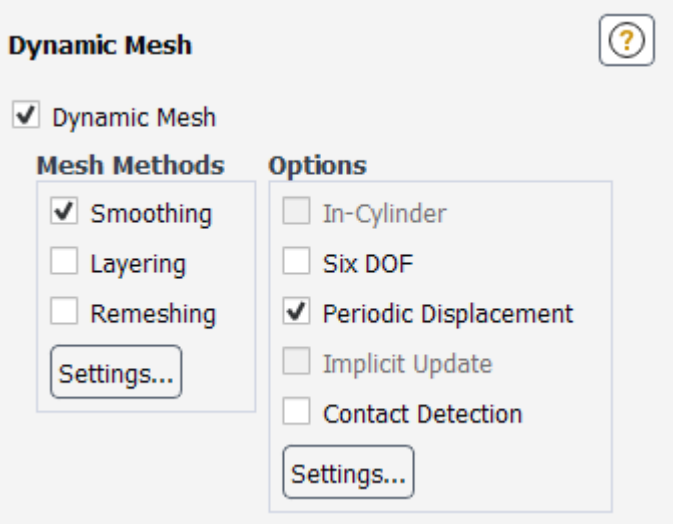
With Dynamic Mesh turned on, you are ready to define the Periodic Displacement for the blades. When Periodic Displacement is set, Fluent automatically turns on the Mesh Smoothing method. The recommended Mesh Smoothing Methods are, in order of preference:

- Linear Elastic Method (with a Poisson's ratio of 0.45)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

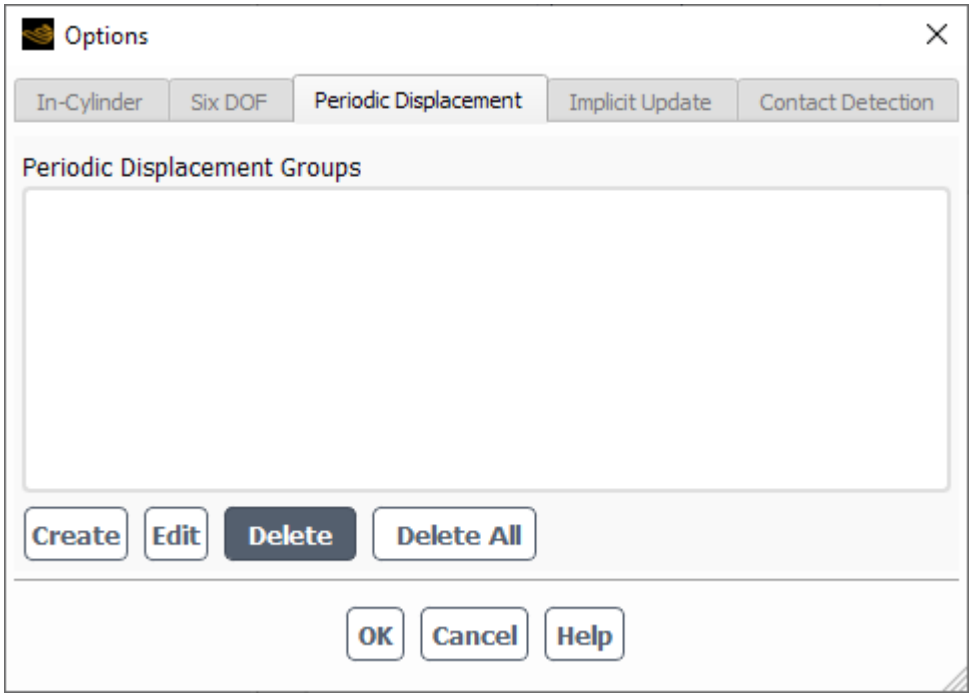
- Diffusion Method (with diffusion parameter of 1.5)

1. On the **Dynamic Mesh** task page, under **Options**, select **Periodic Displacement**, and click **Settings**.



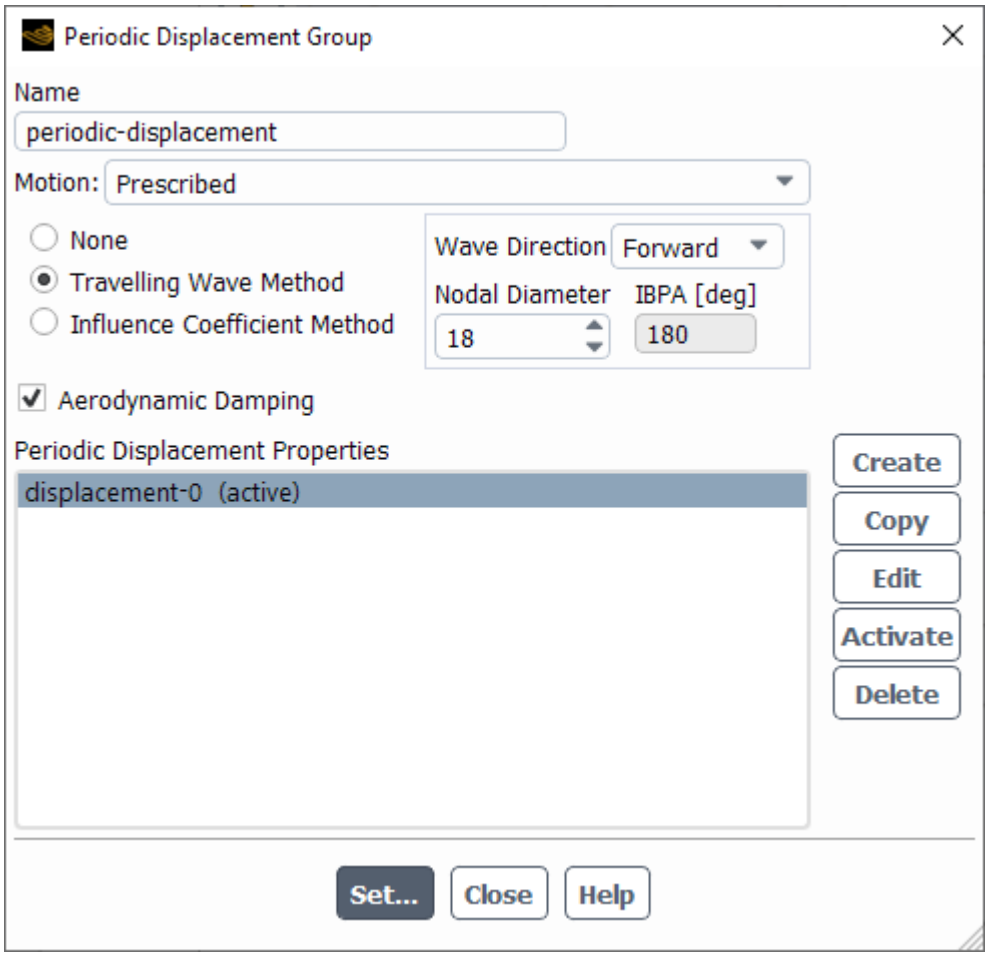
2. In the **Options** dialog box, on the **Periodic Displacement** tab:

   a. To create a new **Periodic Displacement Group**, click **Create**. You can create as many groups as needed. This allows you to quickly run different mode shapes by having different displacements activated under each group. Existing **Periodic Displacement Groups** can be modified by clicking **Edit** or deleted with **Delete** or **Delete All**.



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. Clicking **Create** opens the Periodic Displacement Group dialog where you can assign a name to group and specify some setting common to that group. Clicking **Set...** saves the settings for the group.



c. If you are editing an existing group, a dialog box appears for the selected group where you can specify some settings related to the motion of the blade.

d. The **Motion** can currently only be set as `Prescribed` with no wave (**None**) or one of the following options:

- **Travelling Wave Method**. For details, see Traveling Wave Method (TWM) (p. 1990).
- **Influence Coefficient Method**. For details, see Influence Coefficient Method (ICM) (p. 1997).

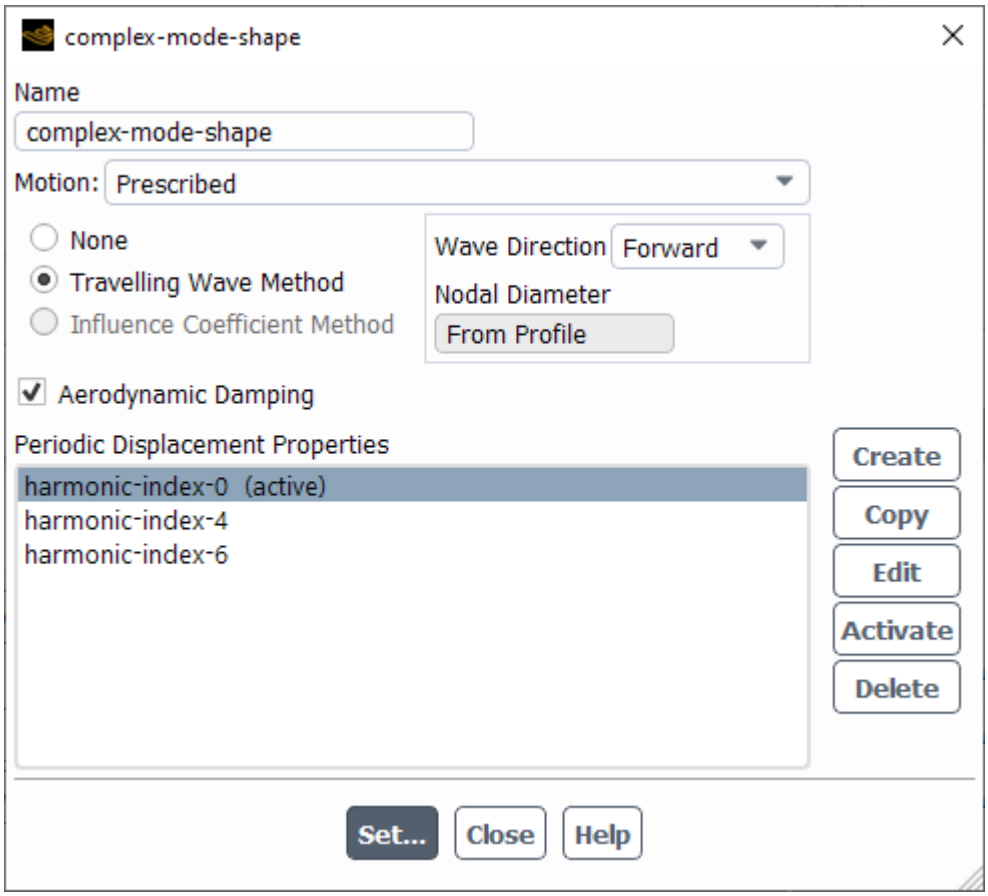
e. Select **Aerodynamic Damping**.

This will enable monitoring of the damping factor in postprocessing. **Aerodynamic Damping** is automatically enabled for new groups. See TWM Method Post-processing (p. 1996) for more details.

f. Create one or more periodic displacements under **Periodic Displacement Properties**. All currently defined periodic displacements will appear under each **Periodic Displace-**

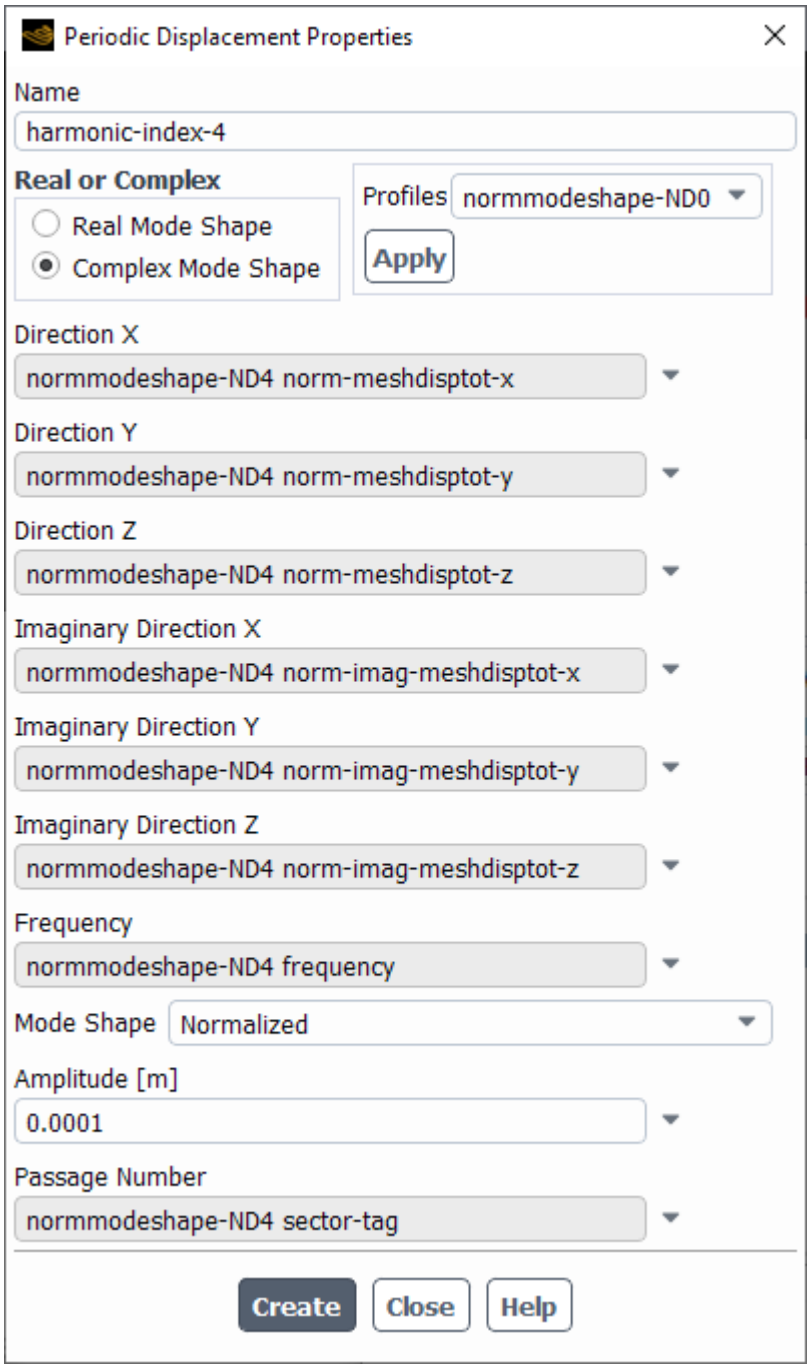
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ment Group, however you must activate one periodic displacement for each group. After selecting the displacement you want to make active for a particular group, click **Activate**. The `(active)` suffix will appear next to the currently active displacement.



g. You can also copy an existing displacement by selecting the original displacement you want and clicking **Copy.** The copied displacement will have the same settings as the original, but with the default displacement name (which can be changed at any time). To specify a new displacement, click **Create** and a dialog box will appear to specify the displacement.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



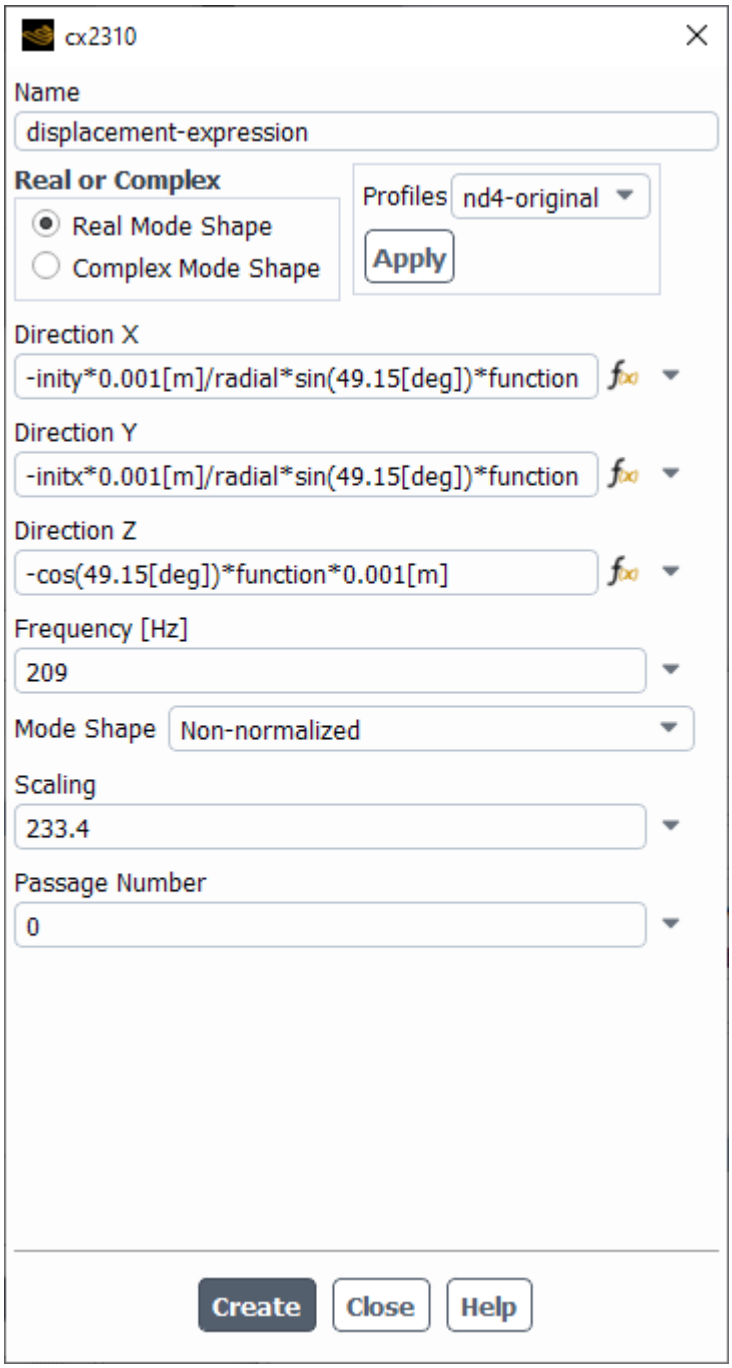
1. In the **Profiles** group box, all loaded profiles that contain the mesh displacement field variables (variables containing the meshdisp, meshdisptot keyword) can be selected. After selecting a profile from the drop-down list and clicking **Apply**, the dialog box will be filled based on the profile. Parameter values present in the profile such as mode shape frequency, and amplitude, if present, are also automatically applied to their respective fields. Otherwise, you can follow the steps below to specify a periodic displacement manually.

2. Specify the mode shape as **Real** or **Complex**.

3. Specify the displacement components (real and imaginary, if applicable) by setting **Direction X**, **Direction Y**, **Direction Z** (and **Imaginary Direction X**, **Y**, and **Z**, if ap-

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

plicable) using the mesh displacement variables from the `.csv` file (as described in Reading the Mode Shapes (p. 1973)), via UDF, or using an expression. For details on specifying the displacement components using a UDF, see `DEFINE_PERDISP_MOTION` in the *Fluent Customization Manual*. An example for specifying the displacement components with an expression is shown below:



The expression for **Direction X** is broken down in the following table:

Function:	Equivalent Expression:
$\frac{-Init_y*0.001[m]}{radial}*\sin(49.15^{\circ})*function$	`-inity*0.001[m]/radial*sin(49.15[deg])*function`
$radial=\sqrt{Init_x+Init_y}$	`sqrt(initx^2+inity^2)`
$function=0.063*radial[m^{-1}]-5.94\cdot10^{-3}$	`6.3*10^-2*radial*1[m^-1]-5.94*10^-3`
$Init_x=x-XTotalMeshDisplacement$	`x-XTotalMeshDisplacement`
$Init_y=y-YTotalMeshDisplacement$	`y-YTotalMeshDisplacement`

4. Set **Frequency** to the mode shape frequency.

   This frequency is among the data that is deleted from the original profile file (as output by Mechanical) while creating a profile file suitable for loading into Fluent.

5. Set whether the mode shape is **Normalized** or **Non-normalized**.

6. Set **Amplitude** (for normalized mode shapes) or **Scaling** (for non-normalized mode shapes).

   For normalized mode shapes, Fluent requires the **Amplitude** value. The **Amplitude** multiplies the provided normalized (maximum amplitude of 1.0) dimensionless modal displacement values by the specified amplitude in order to obtain the maximum modal displacement values. The recommended starting value for the **Amplitude** is 0.05% of the blade chord length.

   For non-normalized profiles, Fluent requires the **Scaling** value. The **Scaling** parameter scales the displacement vector by the value specified in the **Scaling** field. The recommended starting **Scaling** value is 0.5.

   It is recommended that you preview the mesh motion with the recommended settings of **Amplitude** or **Scaling** factor and **Mesh Methods** using the desired time step size and at least for the same number steps in one vibration period. The mesh preview feature is located at the bottom of the **Dynamic Mesh** task page.

   Domain → **Mesh Models** → **Dynamic Mesh...** → **Preview Mesh Motion...**

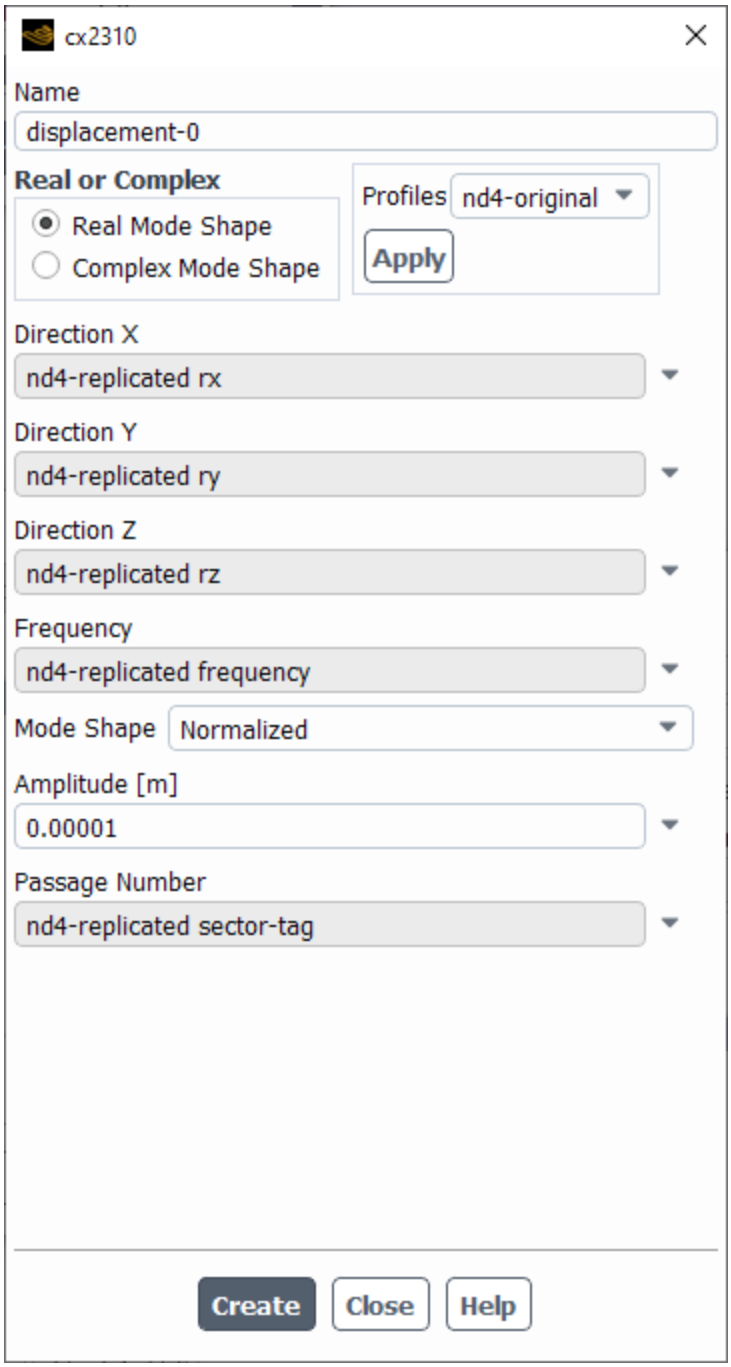
7. Set **Passage Number** to the sector tag variable from the `.csv` data. For more details, see Reading the Mode Shapes (p. 1973).

   This provides an integer that identifies each blade. In the case of the Travelling Wave Method, this is required for the proper specification of the IBPA. In the case of the Influence Coefficient Method, this blade numbering is a requirement of the aerodynamic damping calculation.

An example of a Periodic Displacement specification from a real mode shape is shown below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

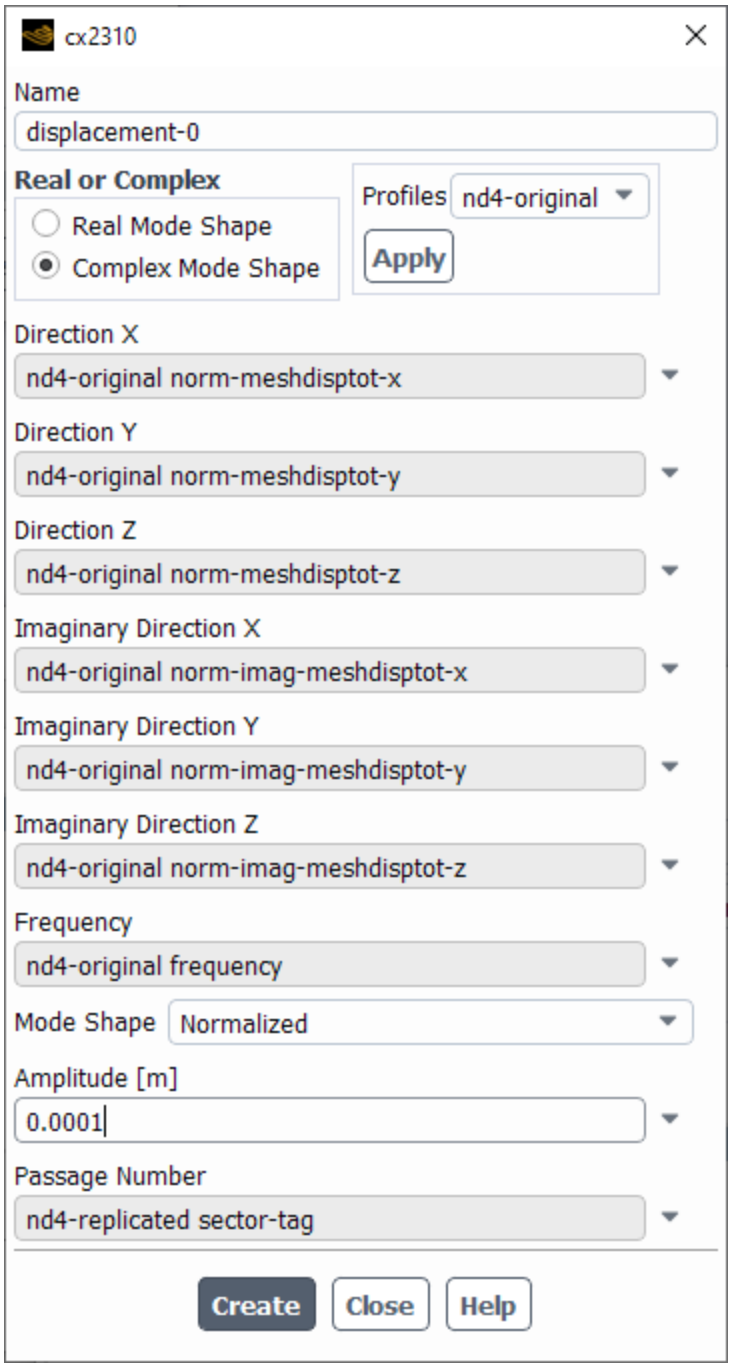
Figure 14.10: Periodic Displacement Example for Real Only Mode Shape



An example of a Periodic Displacement specification from a complex mode shape with frequency from the profile parameter is shown below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 14.11: Periodic Displacement Example for a Complex Mode Shape



14.4.1.3.3. Creating and Applying Dynamic Mesh Zones

With Periodic Displacement defined for the blades, you are ready to specify the motion of the blades and other boundaries. To do this, you need to create and apply dynamic mesh zones.

To create a dynamic mesh zone:

1. On the **Dynamic Mesh** task page, click **Create/Edit...**.

   The **Dynamic Mesh Zones** dialog box appears.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

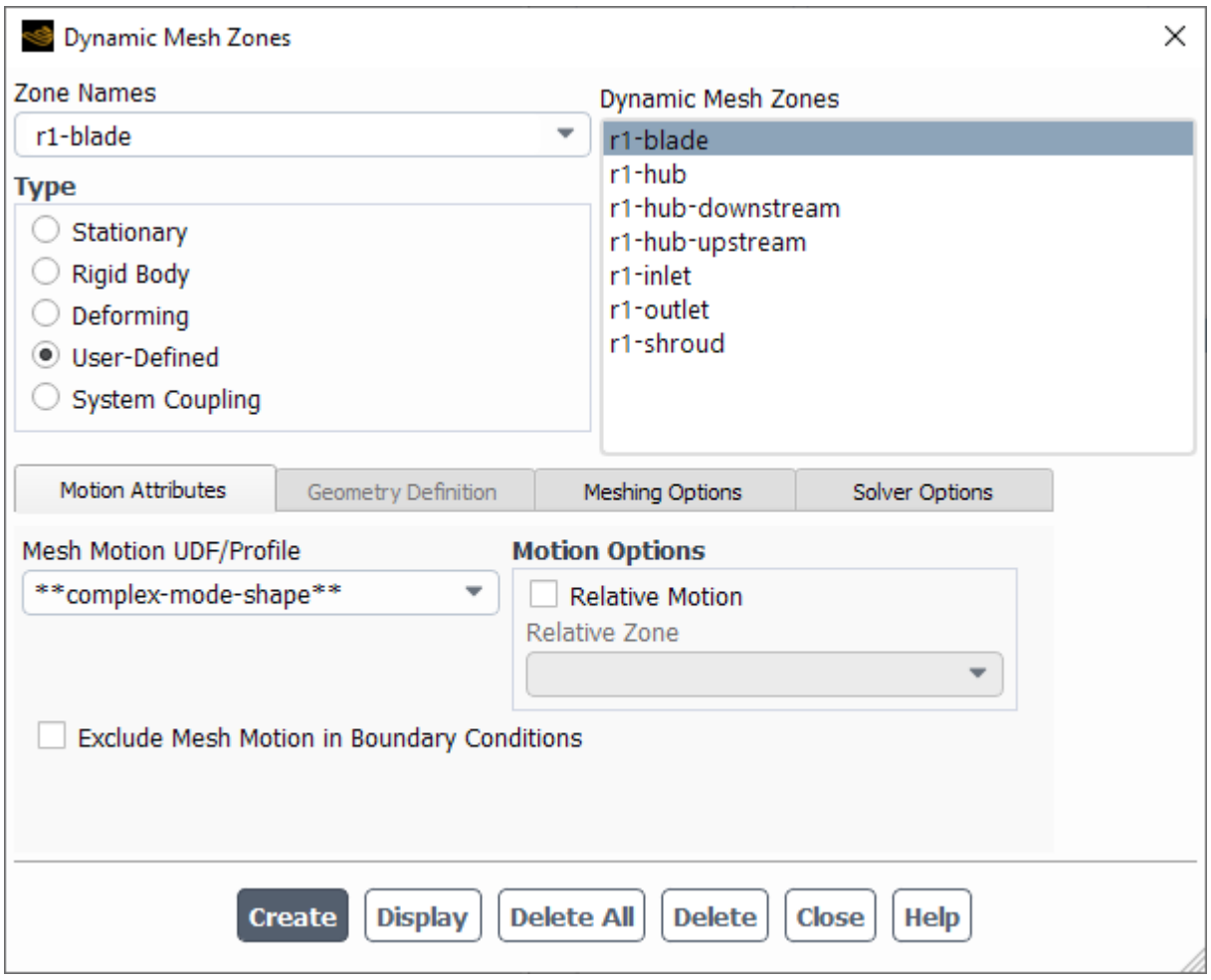
2. Choose a name for the mesh zone under **Zone Names**.
3. Configure settings for the mesh zone according to the type of boundary, as detailed below.
4. Click **Create**.

Details for the required dynamic mesh zones:

- **Blades**

  To impose vibration on the blades:

  1. In the **Dynamic Mesh Zones** dialog box, set **Type** to **User-Defined**.
  2. On the **Motion Attributes** tab, set **Mesh Motion UDF/Profile** to a Periodic Displacement Group specified early as described in Defining the Periodic Displacement of the Blades (p. 1978).



- **Shroud**

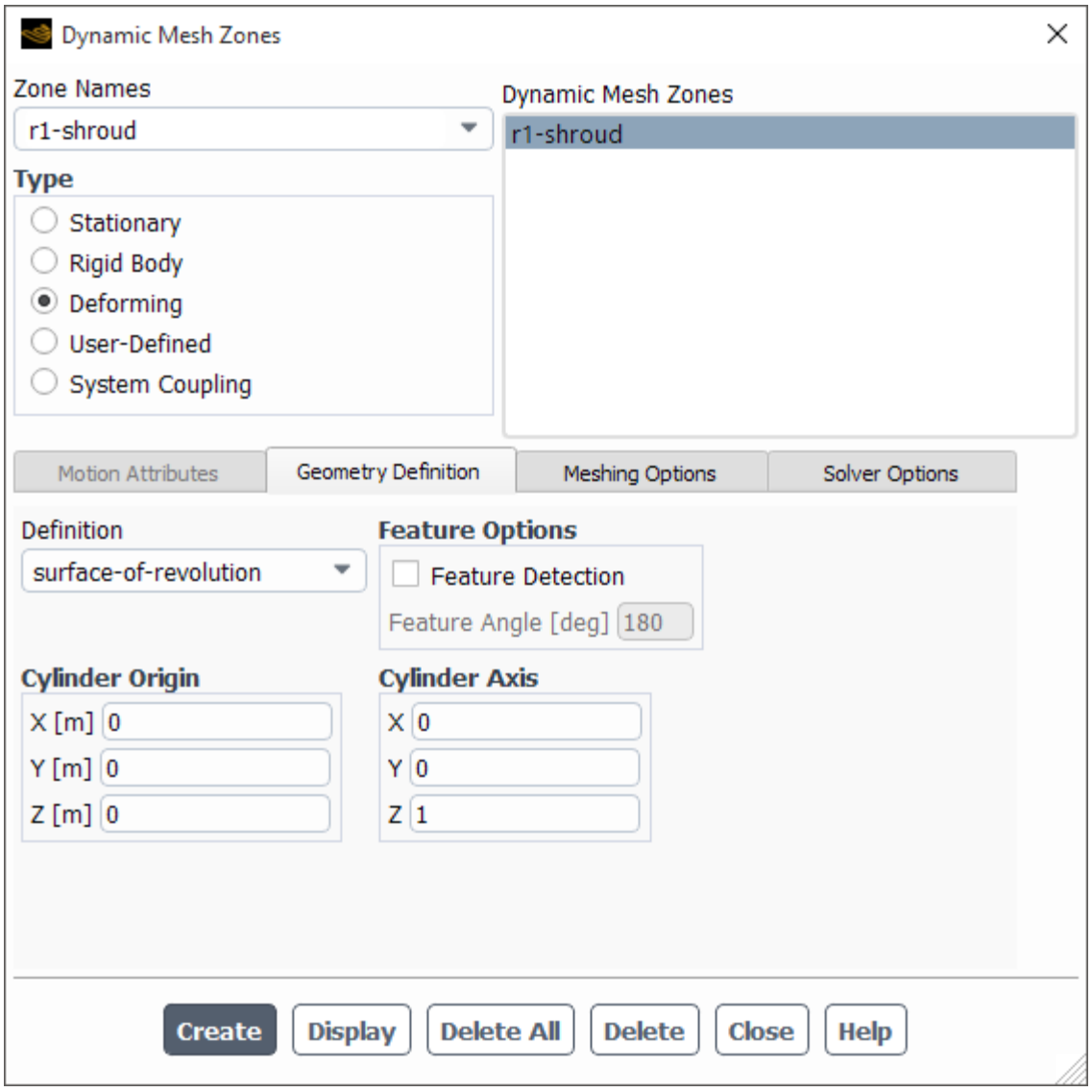
  For cases that involve blade tip gaps, it is beneficial to allow the mesh surface across the gap from the blade tip to slide freely along a surface of revolution.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To do this:

1. In the **Dynamic Mesh Zones** dialog box, set **Type** to **Deforming**.
2. On the **Geometry Definition** tab, set **Definition** to `surface-of-revolution`.

   This option prevents the mesh elements in the gap from becoming skewed by allowing the mesh nodes on the shroud to "slide" as the mesh follows the motion on the blade tip, while preventing the gap thickness from changing.
3. Ensure that the **Cylinder Axis** components (X, Y, Z) indicate the correct direction for the axis. These components are not automatically set according to your simulation.



Note:

Currently, tip gap with interfaces are not supported with the Periodic Displacement method. A tip gap without interfaces can be accommodated with the proper specification of the shroud boundary condition.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Hub**

  Typically, the hub is set as stationary. However, for cases that involve complex mode shapes, the hub surface may have to be added to the list of surfaces that undergo the periodic displacement motion. For such a case, you can set the hub in the same way as was done for the blades:

  1. In the **Dynamic Mesh Zones** dialog box, set **Type** to **User-Defined**.
  2. On the **Motion Attributes** tab, set **Mesh Motion UDF/Profile** to a Periodic Displacement Group specified early as described in Defining the Periodic Displacement of the Blades (p. 1978).

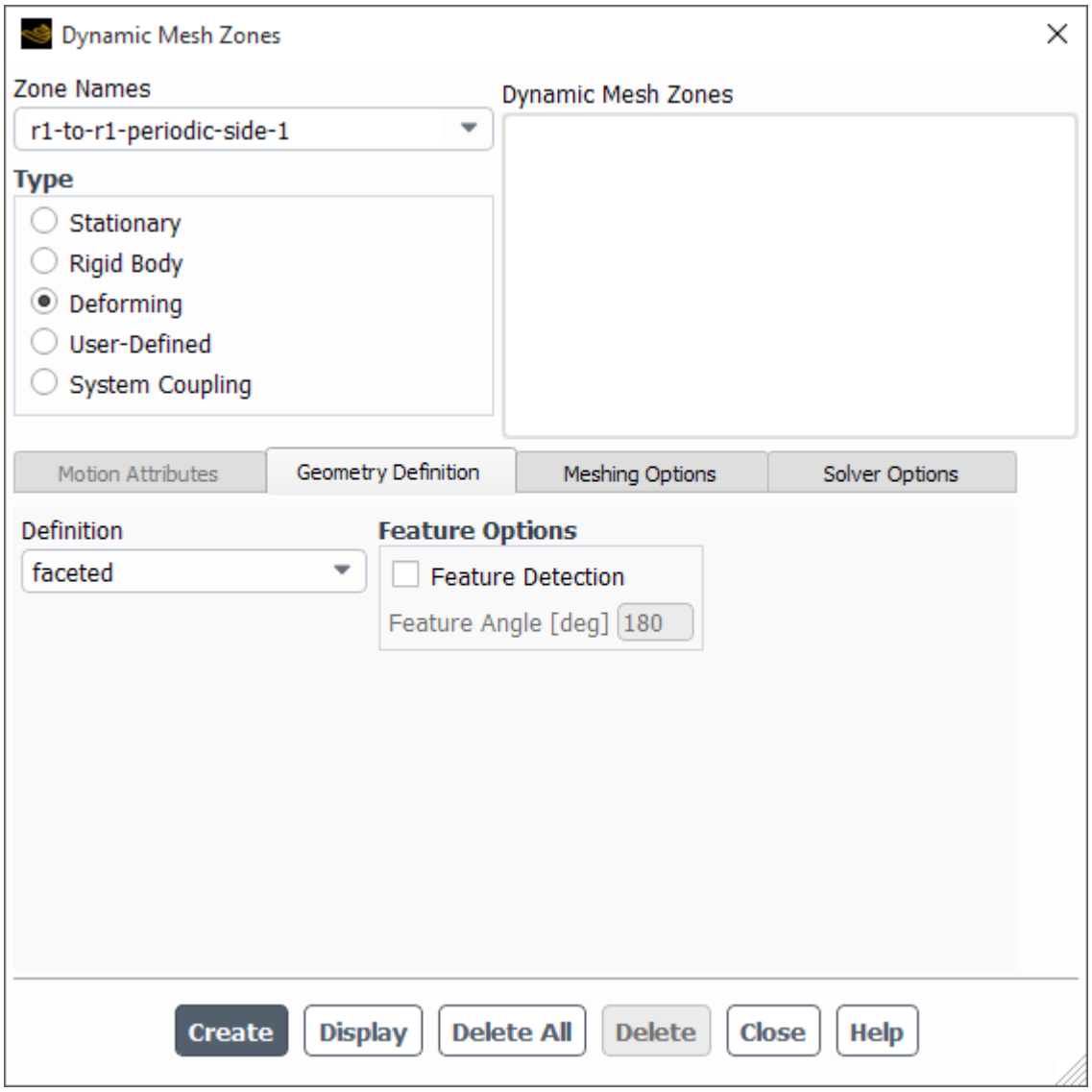
- **Rotational Periodic Boundaries**

  Typically, rotational periodic boundaries are set as stationary.

  For cases where the rotational periodic boundaries are connected to other surfaces that undergo periodic displacements, it is important to allow the periodic boundaries to move and match the displacement, especially on the shared nodes. This is common in cases involving complex mode shapes. In such cases, the **Deforming** type with the **Faceted** definition is recommended.

  1. In the **Dynamic Mesh Zones** dialog box, set **Type** to **Deforming**.
  2. On the **Geometry Definition** tab, set **Definition** to `faceted`.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If no motion is expected on the rotational periodic boundaries, you may set the **Type** to **Stationary**.

- **Other Boundaries**

  For the other boundaries, set **Type** to **Stationary**. Note that if boundaries are left unspecified, they will automatically be defined as **Stationary**.

For details on dynamic meshes, see Modeling Flows Using Sliding and Dynamic Meshes (p. 1797).

14.4.2. Traveling Wave Method (TWM)

To predict the aerodynamic damping, the TWM is employed in modeling the blade flutter analysis. In TWM, the mesh displacement is prescribed on all the blades in the model. The blades are assumed to vibrate with a frequency equivalent to the natural frequency of a blade and with a phase shift between each blade known also as the Inter-blade Phase Angle (IBPA). The IBPA is dependent on the nodal diameter. This vibration pattern behavior results in a traveling wave. A Forward Traveling Wave (FTW) or Backward Traveling Wave (BTW) pattern is the result of lead or lag in the phase of motion

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

from one blade to the next. A FTW is defined as a wave pattern that travels in the direction that the machine rotates, while a BTW is defined as a wave pattern that travels in the direction opposite that the machine rotates.

For blade flutter calculations, it is important to consider the aerodynamic effects of adjacent blades. For a rotor-disc assembly consisting of N_{BL} blades, there is a finite number of disc nodal diameters. At nodal diameter 0 ($ND=0$), all the blades vibrate in phase with an Interblade Phase Angle (IBPA) of 0. However, for any other nodal diameter values, each blade will be out of phase with respect to the adjacent blades by some finite IBPA value. The IBPA can be computed from the nodal diameter:

$$IBPA=\frac{2\pi ND}{N_{\mathrm{BL}}}$$

where $0\leq ND\leq\frac{N_{\mathrm{BL}}}{2}$ for an even number of blades and $0\leq ND\leq\left(N_{\mathrm{BL}}-1\right)/2$ for an odd number of blades.

The main objective of a blade flutter analysis is to obtain the aerodynamic damping, or work per vibration cycle. Once a transient periodic solution is achieved, you can use the energy balance method to perform the damping calculation. This calculation yields a measure of the system stability at each combination of nodal diameter and blade vibration frequency. The work per vibration cycle can be computed as:

$$W_{cycle}=\int_{t_0}^{t_0+T}\int_{A}p\vec{v}\cdot\hat{n}\,dAdt \tag{14.2}$$

Where:

- $T=2\pi/\omega$ is the period of one vibration cycle.
- t_0 is the time at the start of the vibration cycle.
- $\omega=2\pi/T$ is the vibration angular frequency.
- p is the fluid pressure.
- v is the velocity of the blade due to imposed vibrational displacements.
- A is the surface area (in this case, the surface of the blade).
- $\hat{n}$ is the unit normal surface vector.

Note:

Positive values for work indicate that the vibration is damped (for the combination of nodal diameter and blade vibration frequency being studied), whereas negative values indicate that the vibration is undamped/excited.

Typically, when using the TWM/Energy method to obtain aerodynamic damping, you need two simulations for each nodal diameter possible in the rotor assembly: one for a FTW and one for a BTW.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can use aerodynamic damping monitors to monitor aerodynamic damping during a run. The monitor value can be calculated using a normalized version of Equation 14.2 (p. 1991).

There are three possible setup configurations when modeling Aerodynamic Damping using TWM:

1. Use a full-wheel model where all the blades in the row are included. With this model topology, the entire range of nodal diameters is modeled using a single problem setup. However, the computational cost is expensive due to the size of the mesh.

2. Use a periodic sector for specific nodal diameters (periodicity must be achieved). This is a much more economical modeling strategy but the disadvantage is that multiple configurations are needed to cover the entire nodal diameter range. Also, it is not always possible to find a valid periodic sector if there is an odd number of blades in the row.

3. Use a single passage model and apply a Fourier-based phase-lag boundary condition. With this modeling option, a single setup is used to cover the entire nodal diameter and at the same time, the computational cost is much less than the periodic-sector or full-wheel model due to solving the flow problem on a single passage only. The additional problem setup required for an aerodynamic damping flow problem using a phase-lag boundary condition is described in Phase-lag Method (p. 2003).

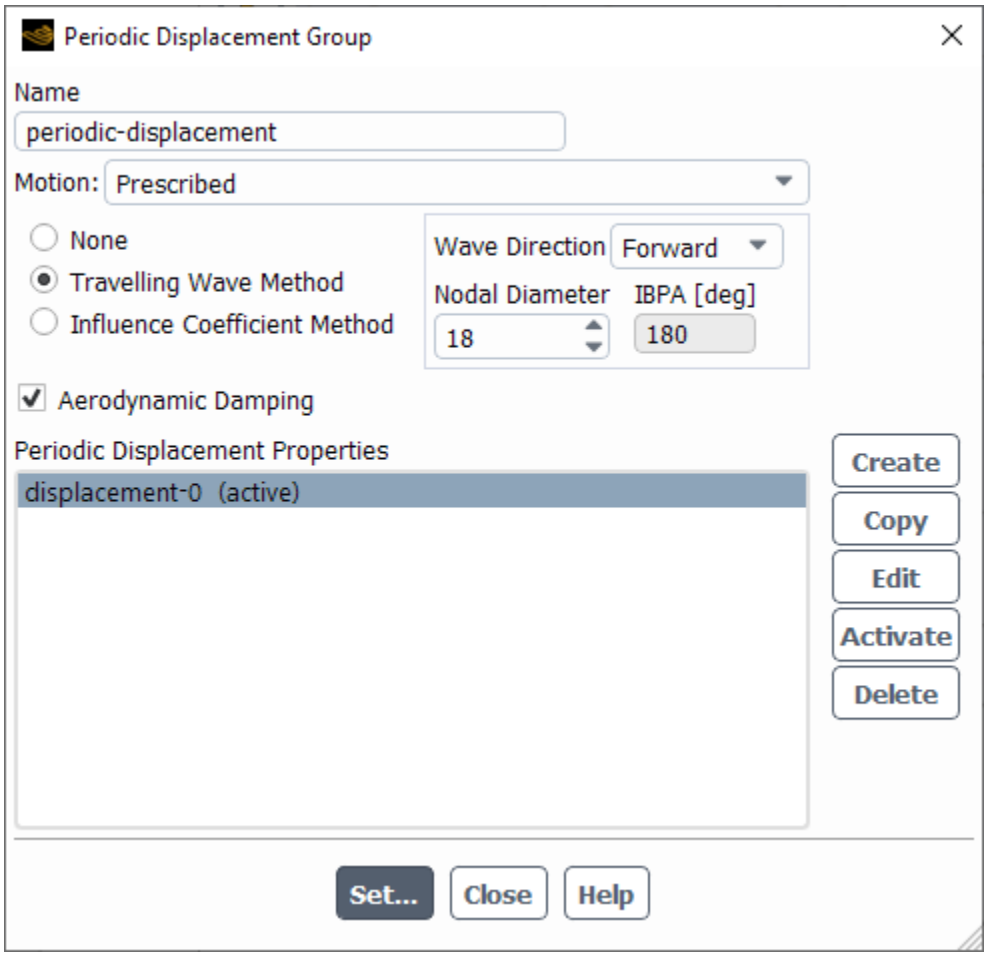
14.4.2.1. Setup Specific to the TWM Method

In addition to the steps outlined in Common Settings for a Blade Flutter Case (p. 1972), there are some settings that are unique to the TWM method. These settings are outlined below.

1. Refer to Using Dynamic Mesh Zones in a Blade Flutter Simulation (p. 1977) for steps on setup of the Dynamic Mesh and Periodic Displacement.

2. On the Periodic Displacement Group dialog box, select **Travelling Wave Method** and **Aerodynamic Damping**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



With the **Traveling Wave Method**, additional settings appear for controlling the displacement wave:

- **Wave Direction**: `Forward` or `Backward`

  A forward traveling wave moves in the direction of rotor rotation.

  A backward traveling wave moves in the direction opposite of rotor rotation.

- **Nodal Diameter**:

  The vibration of the blades with **Interblade Phase Angle** (IBPA) results in a traveling wave pattern. For complex mode shapes that are generated with cyclic symmetry, the mode shape profile is already phase-shifted so the IBPA is not needed. For real only mode shapes, you must supply information about the IBPA.

  For real only mode shapes, the corresponding IBPA is shown to the right of the **Nodal Diameter** setting. The IBPA is the blade pitch multiplied by the nodal diameter.

  The blade pitch is determined by the number of blades, which you can set using the Turbomachinery Description console command. For details, see Turbomachinery Description (p. 2026).

  For complex mode shapes, the IBPA is embedded in the mode shape profile. Therefore, the **Nodal Diameter** setting field is replaced by the non-editable option **From Profile**. To simulate

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a different Nodal Diameter, you must obtain another mode shape associated with the desired Nodal Diameter.

Note:

- Depending on the combination of Nodal Diameter and the total number of blades in the machine, you might have the option to model a periodic sector of the wheel instead of the full wheel.
- If splitter blades are involved, then for the purpose of calculating the magnitude of the nodal diameter, the blade pitch should be measured from one main blade to the next.

3. Create a periodic displacement for the group and complete the remainder of the setup as described in Common Settings for a Blade Flutter Case (p. 1972).

14.4.2.2. Visualizing and Exporting Blade Flutter Harmonics with TWM

As a prerequisite to the options discussed in this section, you must ensure that the turbomachine description has been previously specified. This is done automatically if you set up your case using the turbomachine workflow (as described in Using the Turbomachinery Guided Workflow (p. 1929)) or manually with the command `/define/turbo-model/create-turbomachine-description` (as described in Turbomachinery Description (p. 2026)).

You have the option to compute the following:

1. Fourier coefficients of pressure, velocity components, and temperature based on the blade flutter frequency of the simulation.

   Computing the Fourier coefficients is enabled from the **Spectral Content** tab in the Turbomachinery ribbon or from the `define/turbo-model` menu of the console text commands. For details, see General Fourier Coefficient Postprocessing for Turbomachinery Cases (p. 2035).

2. Real and imaginary harmonic pressure loads on surfaces based on the blade flutter frequency of the simulation. This computation is enabled using the following text command:

   ```
   define/turbo-model/blade-flutter-harmonics/enable-harmonic-exports?
   ```

   By default, the pressure loads will be computed for all boundaries, however, you can also specify which boundaries to use for the calculation.

At the conclusion of the run you can write the harmonic exports to a `.csv` file using the following command:

```
define/turbo-model/blade-flutter-harmonics/write-harmonic-exports?
```

The results from the aerodynamic damping analysis for the specified nodal diameter and harmonic index (both written in the file in the parameters section) at each surface chosen for harmonic export will be written to a separate file. The file name will be the surface name appended to a root file name that you are prompted for.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Since face connectivity is automatically written into the .csv file, these files can be read into Fluent (or CFD-Post) to visualize the export surfaces as well as the real and imaginary first harmonic of pressure. To do this, after reading the .csv file, click **Preview** on Figure 7.96: The Profiles Dialog Box (p. 1572). For details on reading profiles, see Reading Profile Files (p. 964).

The harmonic export .csv file is then read into Ansys Mechanical or a third-party FEM software for a forced response analysis. The file has the following format (with "..." representing omitted similar lines):

- File identification header - Information for documentation and traceability including Fluent version number, date generated, and case file.
- [Name] - Name of the profile.
- [Parameters] - Various parameters required by Ansys Mechanical.
- [Spatial Fields] - Mesh nodal coordinate system.
- [Data] - Nodal surface coordinates and real/imaginary pressure harmonics.
- [Faces] - Face/node connectivity data for visualization.

```
## ANSYS Fluent 22.2.0
##
## Generated on: Thu Nov 11 17:47:36 2021
##
## Generated from case: STCF-11_notipgap_TRS_64tspp_s1r.cas.h5
##
[Name]
blade-1

[Parameters]
Ncompt = 1
Nnodes = 3192
Rotation Axis From   = 0.0000000 [m], 0.0000000 [m], 0.0000000 [m]
Rotation Axis To     = 0.0000000 [m], 0.0000000 [m], 1.0000000 [m]
Rotating Speed       = 0.0000 [s^-1 rad]
Frequency            = 209.00000 [Hz]
Nodal Diameter       = -10
Mode Multiplier      = 2.33400000E+02
Travelling Wave Flag = 1

[Spatial Fields]
Initial X, Initial Y, Initial Z

[Data]
Initial X [m], Initial Y [m], Initial Z [m], Passage Number [ ], Real Pressure [kg m^-1 s^-2], Imaginary Pressu
 1.48709608E-01, -6.25026289E-02, -2.83326356E-02,  1.00000000E+00, -1.58392046E+02,  2.60064686E+01,
 1.48834760E-01, -6.22039797E-02, -2.81875705E-02,  1.00000000E+00,  9.35186279E+01,  4.60809849E+01,
 1.48281814E-01, -6.20267769E-02, -2.81889840E-02,  1.00000000E+00,  6.91992309E+01,  5.80998775E+01,
 1.48157155E-01, -6.23239746E-02, -2.83328554E-02,  1.00000000E+00, -1.87231559E+02,  5.86862670E+01,
 1.51736136E-01, -6.24118879E-02, -2.75272228E-02,  1.00000000E+00,  3.36947961E+02, -4.61095015E+01,
...
[Faces]
      2925,      2926,      2924,      2922
      2923,      2925,      2922,      2919
      2920,      2923,      2919,      2915
      2916,      2920,      2915,      2910
      2911,      2916,      2910,      2904
...
```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.4.2.3. TWM Method Post-processing

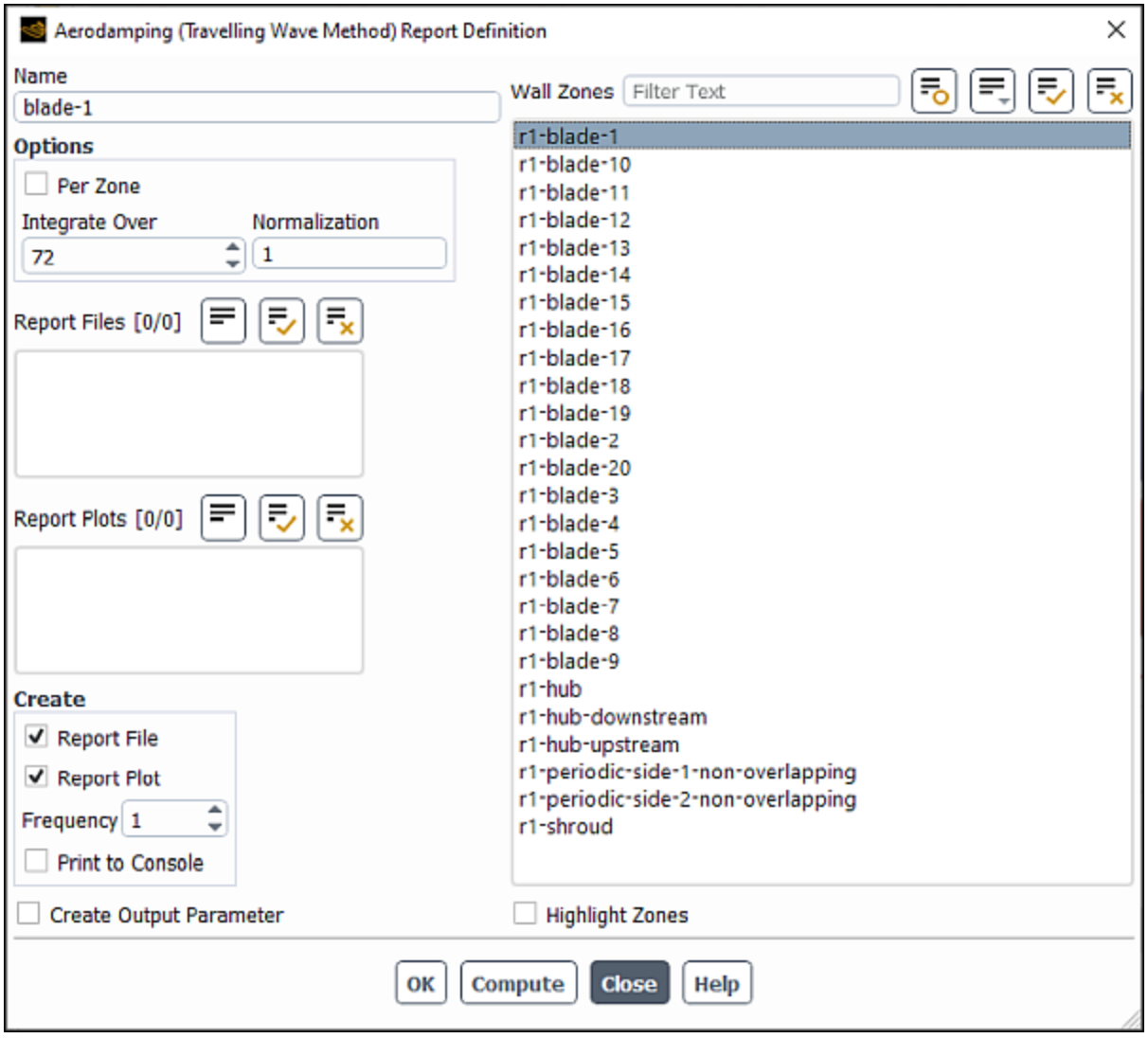
There is a report definition is available when using the TWM method to monitor the aerodyanmic damping value: **Aerodamping (Travelling Wave Method)** report definition. As a pre-requisite, you must have **Travelling Wave Method)** and **Aerodynamic Damping** selected as described in Defining the Periodic Displacement of the Blades (p. 1978).

1. Start a new **Aerodamping (Travelling Wave Method)** report definition from the ribbon:

   Solution → Reports → Definitions → New → Aeromechanics Report → Aerodamping (Travelling Wave Method)...

   The **Aerodamping (Travelling Wave Method)** dialog box appears.

   Figure 14.12: Aerodamping (Travelling Wave Method) Report Definition Dialog Box



2. If you are modeling multiple (main) blades, select **Per Zone** so that the damping value is calculated for each dynamic mesh zone (which encompasses one main blade and possibly accompanying splitter blades).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Set **Integrate Over** to the number of time steps over which to integrate when calculating the damping value; a suggested value is the number of time steps per period.

   The number of time steps per period is shown in the **Run Calculation** task page.

   Note:

   If you change the number of time steps per period in the Run Calculation settings, you probably need to make a corresponding change to the **Integrate Over** setting in the monitor definition.

You can also display contours of the TWM aeromechanics related variables (under **Contours of** type `Aeromechanics`).

- The Wall Work Density is the work performed on the wall by the fluid per unit of area. It is calculated as:

$$wall\ work\ density = pressure \times \left(\Delta\vec{X} \cdot \vec{A}\right)[J/m^2] \tag{14.3}$$

  where $\Delta\vec{X} = \vec{X} - \vec{X}_{N-1}$ is the relative mesh displacement and $\vec{A}$ is the area unit vector.

- The Wall Power Density is the power performed on the wall by the fluid per unit of area. It is calculated as:

$$wall\ power\ density = pressure \times \left(\frac{\Delta\vec{X}}{\Delta t} \cdot \vec{A}\right)[W/m^2] \tag{14.4}$$

  where $\frac{\Delta\vec{X}}{\Delta t}$ is the mesh velocity and $\vec{A}$ is the area unit vector.

14.4.3. Influence Coefficient Method (ICM)

This method provides a convenient way to calculate the complete range of aerodynamic damping values for all possible nodal diameters supported by the geometry in one Fluent run for a given a real mode shape. In addition, when using this method, you can also calculate and/or visualize the generalized/modal force on any or all blade surfaces involved in the calculation.

The key steps to this calculation are to model a large enough series of consecutively positioned blades and to only oscillate the middle blade while all others are kept stationary. The method relies on the linear superposition principle. The unsteady aerodynamic influences on the middle (reference) blade from all oscillating blades in the blade row are equivalent to the sum of unsteady aerodynamic influence of the middle blade oscillating on itself and on all other stationary blades in the blade row. This permits us to calculate the first harmonic of pressure on the middle(reference) blade in a travelling wave method (TWM) calculation as the sum of the first harmonic of pressure from the oscillating blade itself and its neighboring stationary blades.

In the following equations, n represents the blade number.

Using superposition principles, we can write the pressure field on a blade at a given inter-blade phase angle as a sum over all the contributions of the n neighboring blades.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$p(t,\beta)=\sum_n p_n(t)e^{-in\beta} \tag{14.5}$$

From Fourier theory, we can write both the first harmonic of the pressure field and the periodic displacement vector/mode shape as $A_{1,p}$, $B_{1,p}$, $B_{1,p}$, $B_{1,x}$. Then we can express the general aerodynamic damping equation

$$\delta(\beta)=-\oiint \vec{ds}\cdot\int_0^T dt\left\{\sum_n p_n(t)e^{-in\beta}\right\}\frac{d\vec{x}}{dt}(t)\Bigg|_R \tag{14.6}$$

$$\delta(\beta)=-\pi\sum_n\left[\vec{F}_{REAL}\big|_n\cos(n\beta)-\vec{F}_{IMAG}\big|_n\sin(n\beta)\right] \tag{14.7}$$

where the generalized/modal forces can be written as:

$$\vec{F}_{REAL}=\oiint_n \vec{dS}\cdot\vec{B}_{1,x}A_{1,p} \tag{14.8}$$

$$\vec{F}_{IMAG}=-\oiint_n \vec{dS}\cdot\vec{B}_{1,x}B_{1,p} \tag{14.9}$$

Note:

For sign convention:

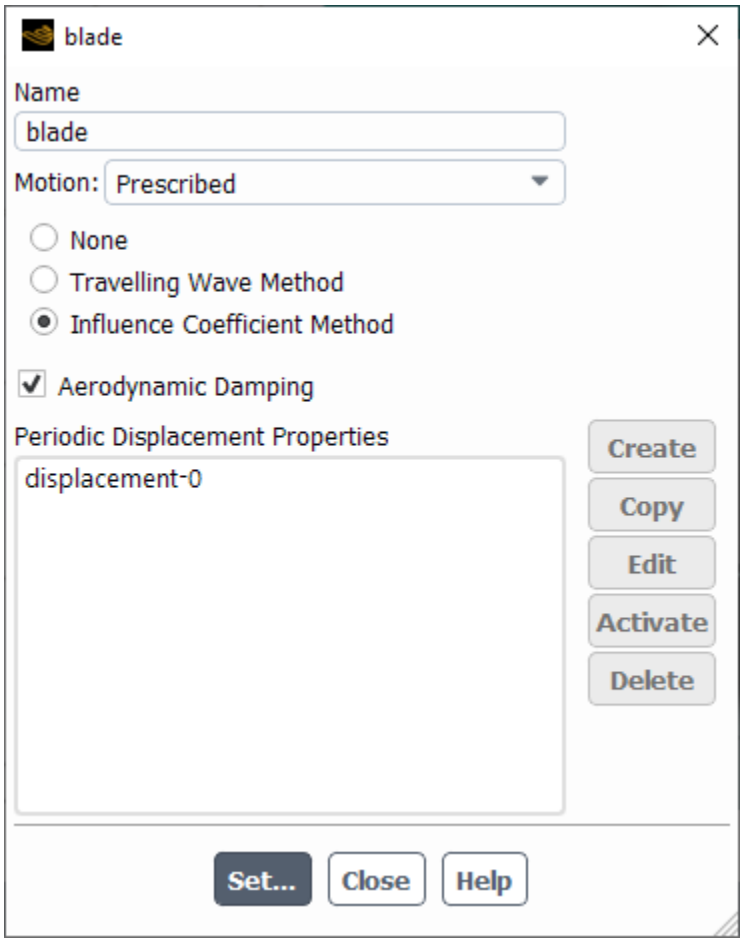
1. The Imaginary part of the pressure force is the opposite sign to the B Fourier coefficient that is accumulated.
2. Since ICM is for real modes, the B Fourier coefficient for the mesh motion, which can also be expressed as the mode shape, does not need to flip signs (it is just keeping track of the phase and results from using a sin wave for motion that equals 0 at time=0)

14.4.3.1. Setup Specific to the ICM Method

In addition to the steps outlined in Common Settings for a Blade Flutter Case (p. 1972), there are some settings that are unique to the ICM method. These settings are outlined below.

1. Refer to Using Dynamic Mesh Zones in a Blade Flutter Simulation (p. 1977) for steps on setup of the Dynamic Mesh and Periodic Displacement.
2. Specify settings for the periodic displacement group. You must select **Influence Coefficient Method** and **Aerodynamic Damping**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Create a periodic displacement for the group and complete the remainder of the setup as described in Common Settings for a Blade Flutter Case (p. 1972).

14.4.3.1.1. Limitations of the ICM Method

1. The ICM method is only supported for real mode shapes.
2. The mode shape must be read into Fluent and expanded to cover all blades that will be involved in the calculation.
3. The blade row modeled must have an odd number of blades. Although only the middle blade will undergo periodic motion at the specified mode shape, this requirement automatically handled in Fluent and all the blades need only to be assigned to the same dynamic mesh displacement group.

14.4.3.2. ICM Method Post-processing

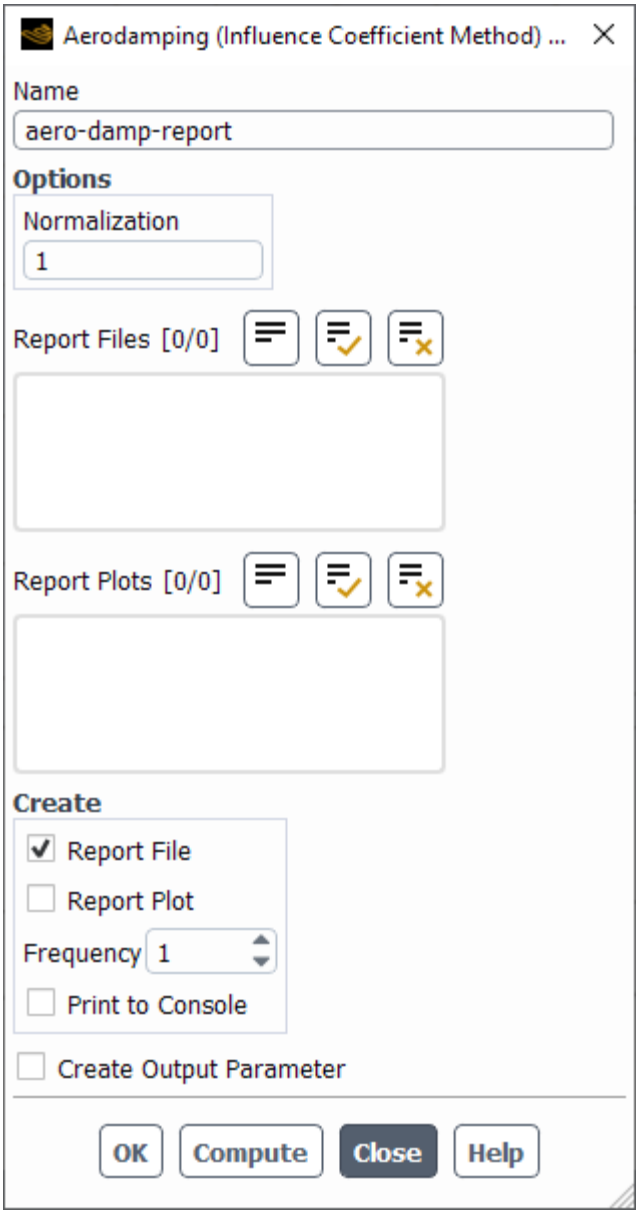
There are two reports available when using the ICM method: **Aerodamping (Influence Coefficient Method)** and **Generalized/Modal Force** report definitions. As a pre-requisite, you must have **Influence Coefficient Method** and **Aerodynamic Damping** selected as described in Defining the Periodic Displacement of the Blades (p. 1978). They are available under:

Solution → **Reports** → **Definitions** → **New** → **Aeromechanics Report**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Aerodamping (Influence Coefficient Method)

The **Aerodamping (Influence Coefficient Method)** report definition is used to output the aerodynamic damping values for each nodal diameter. You need to only input the **Normalization** factor before computing.



The aerodynamic damping values are reported in the console for each nodal diameter.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

	Aerodamping
-18	0.0063262946
-17	0.0063765402
-16	0.0063330763
-15	0.0062036116
-14	0.0059998617
-13	0.0057363682
-12	0.0054289799
-11	0.0050931355
-10	0.004742178
-9	0.0043860092
-8	0.0040304242
-7	0.003677395
-6	0.0033263659
-5	0.0029763284
-4	0.0026281445
-3	0.0022864363
-2	0.0019604375
-1	0.0016635194
0	0.001411541
1	0.0012205501
2	0.0011045217
3	0.0010737038
4	0.0011338237
5	0.0012860489
6	0.0015273671
7	0.001851028
8	0.002246845
9	0.0027013703
10	0.0031981153
11	0.0037180102
12	0.0042402038
13	0.0047431701
14	0.0052059868
15	0.0056096212
16	0.00593809
17	0.0061794107
18	0.0063262946
Net	0.13989081

A TUI command is provided to convert the aerodamping file exported by the **Aerodamping (Influence Coefficient Method)** report definition so that the results can be displayed as Aerodamping value vs Nodal diameter.

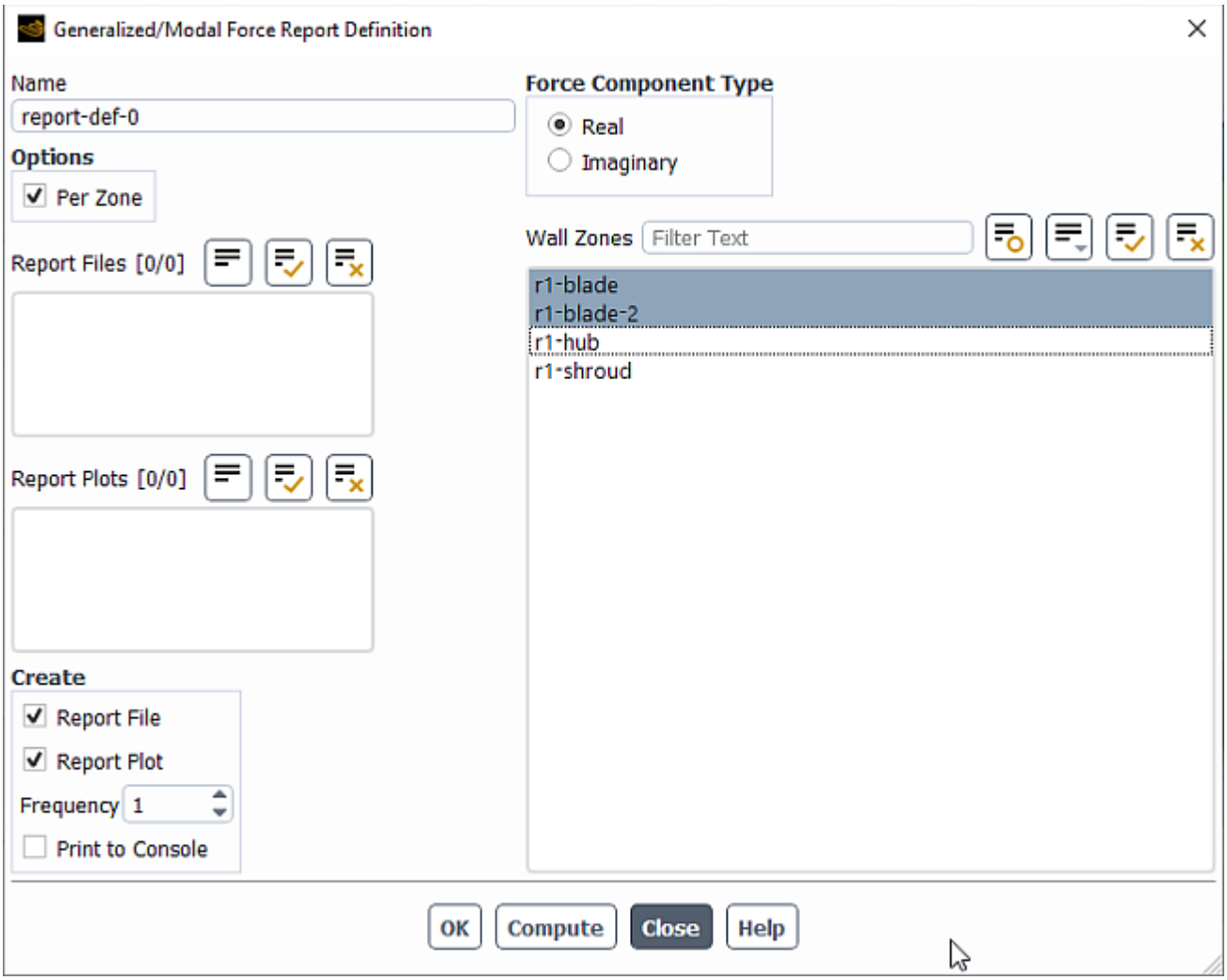
```
define/turbo-model/blade-flutter-harmonics/write-aerodamping-vs-nodal-diameter?
```

For details on creating plots from a file, see Creating an XY Plot From Multiple Data Sources (Including Files) (p. 4064).

Generalized/Modal Force

The **Generalized/Modal Force** report is used to output force on the blade. Under **Force Component Type**, you can choose to output **Real** or **Imaginary** force on the blade. Under **Options**, you can select **Per Zone** to conveniently output the force on each blade separately.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



With **Per Zone** selected, the force will be output for each blade as shown below:

```
                      Real Force                 [N]
------------------------------- -------------------
                           blade       3.0745982e-07
                       blade:018       6.2127746e-07
                       blade:027      -4.5349657e-07
                       blade:036      -1.2844985e-05
                       blade:045       0.00037964468
                       blade:054       -0.0023619392
                       blade:063        0.0011740335
                       blade:072      -9.0678331e-05
                       blade:081       1.0989858e-05
                       blade:090       1.8116493e-06
                       blade:099       -1.073464e-07
                --------------- -------------------
                             Net       -0.0008986149
```

Visualizing Generalized/Modal Force Components

When creating **Contours**, there are field variables available under the **Aeromechanics...** section to display the force on the blades.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Real Wall Generalized/Modal Force per Area**

  Imaginary Wall Generalized/Modal Force per Area

14.4.4. Common Postprocessing for a Blade Flutter Case

The main goal of a blade flutter simulation is to determine whether the aerodynamic damping value is positive or negative.

The aerodynamic damping is calculated as described in Equation 14.2 (p. 1991).

The aerodynamic damping value can be viewed as a report definition monitor or computed in the console using report definitions for the respective method:

- For the Travelling Wave Method, see TWM Method Post-processing (p. 1996).
- For the Influence Coefficient Method, see ICM Method Post-processing (p. 1999).

Additionally, there are various contour plots that might be of interest:

- Display mode shapes of the blade periodic displacement (under **Contours of** type `Mesh`).

  The interpolated real mode shape profile components (X,Y,Z) are stored in the following variables, respectively: `X Periodic Displacement`, `Y Periodic Displacement` and `Z Periodic Displacement`.

  Likewise, the interpolated imaginary mode shape profile components (X,Y,Z) are stored in the following variables, respectively: `X Imaginary Periodic Displacement`, `Y Imaginary Periodic Displacement`, and `Z Imaginary Periodic Displacement`.

- Display the periodic displacement mesh motion variables (under **Contours of** type `Mesh`).

  The Relative Mesh Displacement is calculated as the difference between the current time step and the previous time step mesh node position. It is stored in the following variables: `X Relative Mesh Displacement`, `Y Relative Mesh Displacement`, and `Z Relative Mesh Displacement`.

  The Total Mesh Displacement is calculated as the difference between the current time step and the reference (undeformed) mesh node position. It is stored in the following variables: `X Total Mesh Displacement`, `Y Total Mesh Displacement`, and `Z Total Mesh Displacement`.

14.5. Phase-lag Method

Turbomachinery geometries and simulations are governed by rotational periodicities. This periodicity can be taken advantage of to reduce the size of the modeled problem by using periodic interfaces. However, when the periodicity of the flow imposed does not match the periodicity of the geometry, normal periodic interfaces are not reliable. To be able to still benefit from using a reduced geometry, the Phase-lag boundary condition is introduced. This method usually allows you to limit the computational model to a single blade passage by applying a Phase-lag boundary condition on the boundaries where there would normally be periodic boundary conditions. The Phase-lag method can be used to apply periodic conditions that are phase-shifted in time to account for unequal pitches between adjacent

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

blade rows (for transient-rotor-stator cases), to account for inequality between signal and domain pitches (for boundary disturbance cases), or to account for the inter-blade phase angle (for blade flutter cases).

14.5.1. Phase-lag Theory

The method aims to directly "impose" a phase shift between the S1 and S2 boundaries (as shown in Figure 14.13: Inlet Disturbance Configuration (p. 2004)) for the primitive variables of the problem. A direct method of applying the phase shift boundary method was proposed by Erdos [41] (p. 5715), but this method requires storage of the signal for a complete period on all the S1/S2 boundaries. The phase lag method makes use of temporal Fourier series decomposition (Equation 14.10 (p. 2004)) of the signal on both sides of the phase-lag interface to avoid storing the signal at each time step at the same boundaries. This method was initially introduced by L. He [61] (p. 5716).

$$\tilde{f}(t) = \sum_{n=-M}^{M} A_n \exp^{-j\omega n t} \tag{14.10}$$

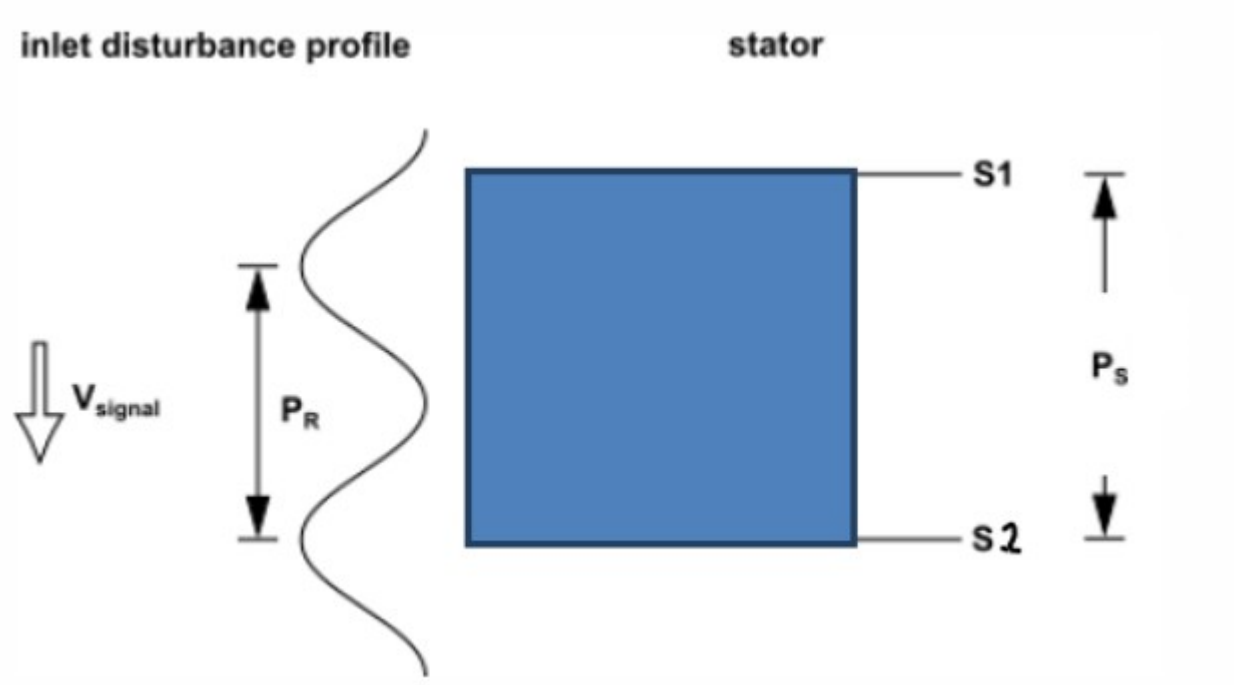
In this method, the signal is decomposed into harmonics of the fundamental frequencies $\omega / 2\pi$ present in each passage as in Equation 14.10 (p. 2004). For example, in typical TRS problems or boundary disturbance problems, this fundamental frequency is the inverse of the blade passing period T_{bpp}.

$$T_{bpp} = \frac{2\pi}{\omega} \tag{14.11}$$

The Phase-lag boundary approach converges very rapidly to a quasi-periodic state.

In Figure 14.13: Inlet Disturbance Configuration (p. 2004), an inlet disturbance configuration and how the phase shift/time shift is applied to the phase lag interfaces S1/S2 is illustrated.

Figure 14.13: Inlet Disturbance Configuration



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In that case, the phase shift between the two sides of the phase lag interface S1/S2 can be calculated as:

$$\phi = -2\pi \frac{P_s}{P_R} \tag{14.12}$$

and the equivalent time shift is:

$$\Delta T = \frac{P_R - P_s}{V_{signal}} \tag{14.13}$$

The signal on S1 is equal to the signal on S2 phase shifted by ϕ. The signal on S2 is equal to the signal on S1 phase shifted by $-\phi$.

The phase lag method theoretically can be used in cases involving combinations of:

- Rotational flow boundary disturbances
- Blade flutter TWM (travelling wave method)
- Transient-rotor-stator interactions

14.5.2. Phase-lag Capabilities and Limitations

The Phase lag method is supported for inlet/outlet disturbances (which can include multiple simultaneous disturbances) and for blade flutter (TWM) cases.

For DBNS solutions, phase-lag is applied only to the following primitive variables: static pressure, temperature, velocity components, turbulent kinetic energy, turbulent dissipation rate (ε), and specific dissipation rate (ω). For PBNS solutions, in addition to the above variables, phase-lag is applied as well to the density, static enthalpy and all derivatives and reconstructed gradients involved in the solution method chosen.

The current feature matrix is restricted to a subset of all the currently available models available in Fluent. The following models are officially supported:

- K-epsilon, k-omega, and SST turbulence models.
- Non-Equilibrium Condensation (Wet Steam) Model.

14.5.3. Using the Phase-lag Method

The basic procedure for setting up a transient turbomachinery simulation is outlined in Using the Turbomachinery Guided Workflow (p. 1929). The following are instructions for setting up a simulation with the Phase-lag method.

14.5.3.1. Pre-requisites and Recommendations
14.5.3.2. Creating the Phase-lag Interface
14.5.3.3. Phase-lag Spectral Description
14.5.3.4. Initializing and Running Phase-lag Simulations
14.5.3.5.21.2.3.6. Fourier Coefficient Postprocessing for Inlet/Outlet Disturbance and Blade Flutter Use-Cases

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.5.3.1. Pre-requisites and Recommendations

1. Available only for transient simulations.

2. Enable **Turbo Models**(/define/turbo-model/enable-turbo-model? y or from the ribbon).

   Domain → Turbomachinery → Turbo Models

3. A phase-lag periodic interface in the pitch-wise direction is required on the outer periodic boundaries.

4. Ensure that no disturbance signal has a speed of zero relative to the component on which that signal acts.

5. The Turbomachine Description is required. This is automatically completed when using the Turbomachine Workflow (Using the Turbomachinery Guided Workflow (p. 1929)) or you can manually complete the Turbomachine Description from the console (/define/turbo-model/create-turbomachine-description). For details, see Turbomachinery Description (p. 2026).

6. It is recommended to use either the `Frequency-Based` or `Period-Based` **Time Advancement Method** with a sufficiently small time-step to run the phase-lag method. A recommended time-step size is to select the **Frequency**/**Period** to match the disturbance and then ensure a sufficiently high **Time Steps per Period** such as 40.

7. Defining extra spectral settings for the Phase-lag method from the console:

   `/define/turbo-mode/phase-lag-extra-settings`

   Provides the option of changing defaults for spectral content creation for the Phase-lag method. In most cases, the default values are sufficient and need not be changed. The following options are available:

   - Fourier coefficients relaxation:

     Under-relaxation constant to be applied to Fourier coefficient accumulation. Current default is 0.7 but a lower value could be used to improve robustness if required.

   - Number of initialization cycles:

     Number of periods to run the simulation using the “periodic like BC” before starting Phase-lag. Permits delaying the onset of Phase-lag to have a better initial guesses for the Fourier coefficients for improved robustness at start-up.

   - Use the default number of Fourier harmonics for each fundamental frequency?

     Uses the default number of harmonics for each fundamental base frequency. Alternatively, you must provide a list of harmonics for each fundamental frequency in the simulation. [default to y]

   - Enter the default number of Fourier harmonics of each fundamental frequency:

     The number of harmonics to be applied as default to all base frequencies in the simulation [defaults to 10].

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

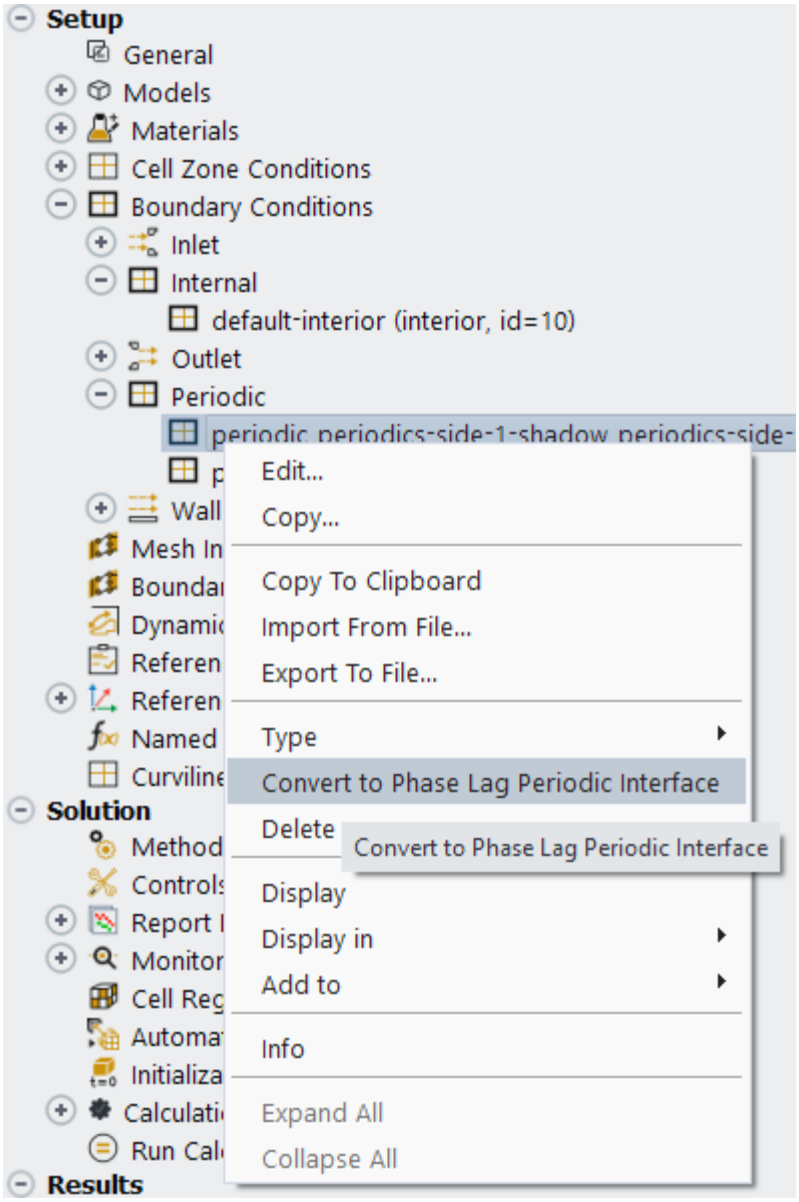
8. When solving a simulation that involves the Phase-lag method, the use of double precision is strongly recommended for accuracy and robustness.

14.5.3.2. Creating the Phase-lag Interface

The phase-lag interface can be created through two separate methods or the equivalent console commands.

1. Right-click on a periodic boundary to convert to a phase-lag interface by selecting **Convert to Phase Lag Periodic Interface** in the menu.

Figure 14.14: Converting Periodic Boundary to Phase Lag Periodic Interface

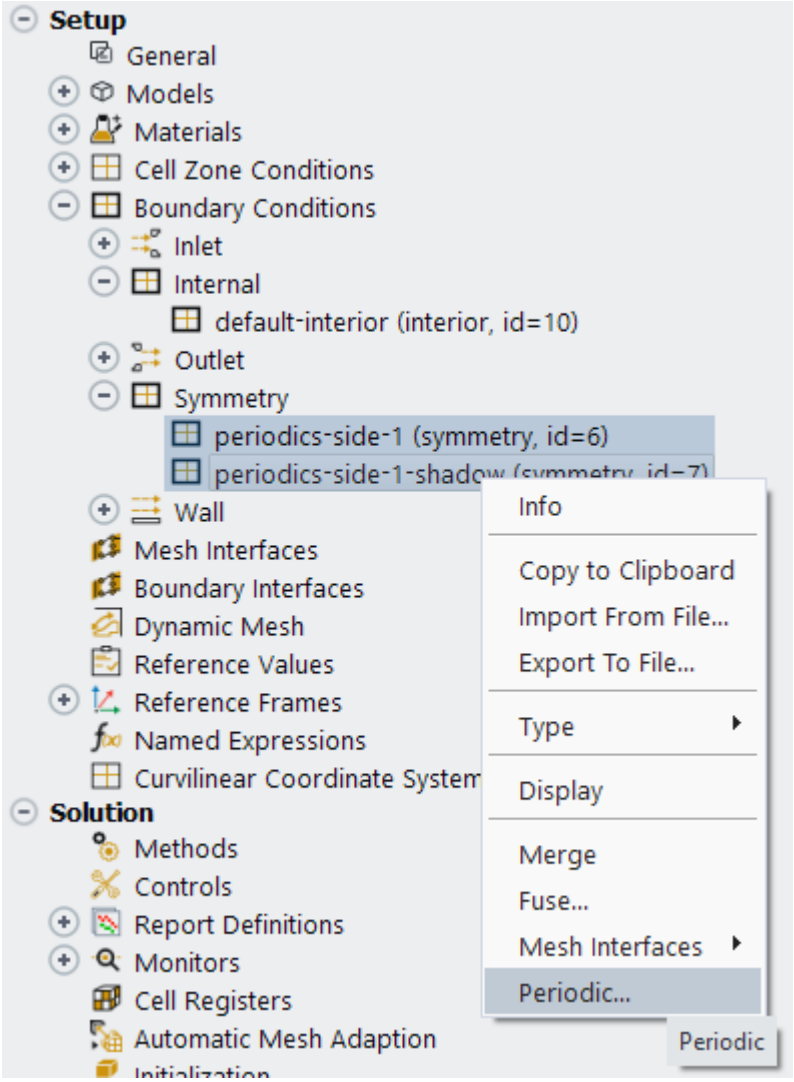


2. Multi-select the two boundaries that will make up the phase-lag interface, right-click, and select **Periodic...** from the menu as shown in Figure 14.15: Creating Phase Lag Periodic Interface from

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

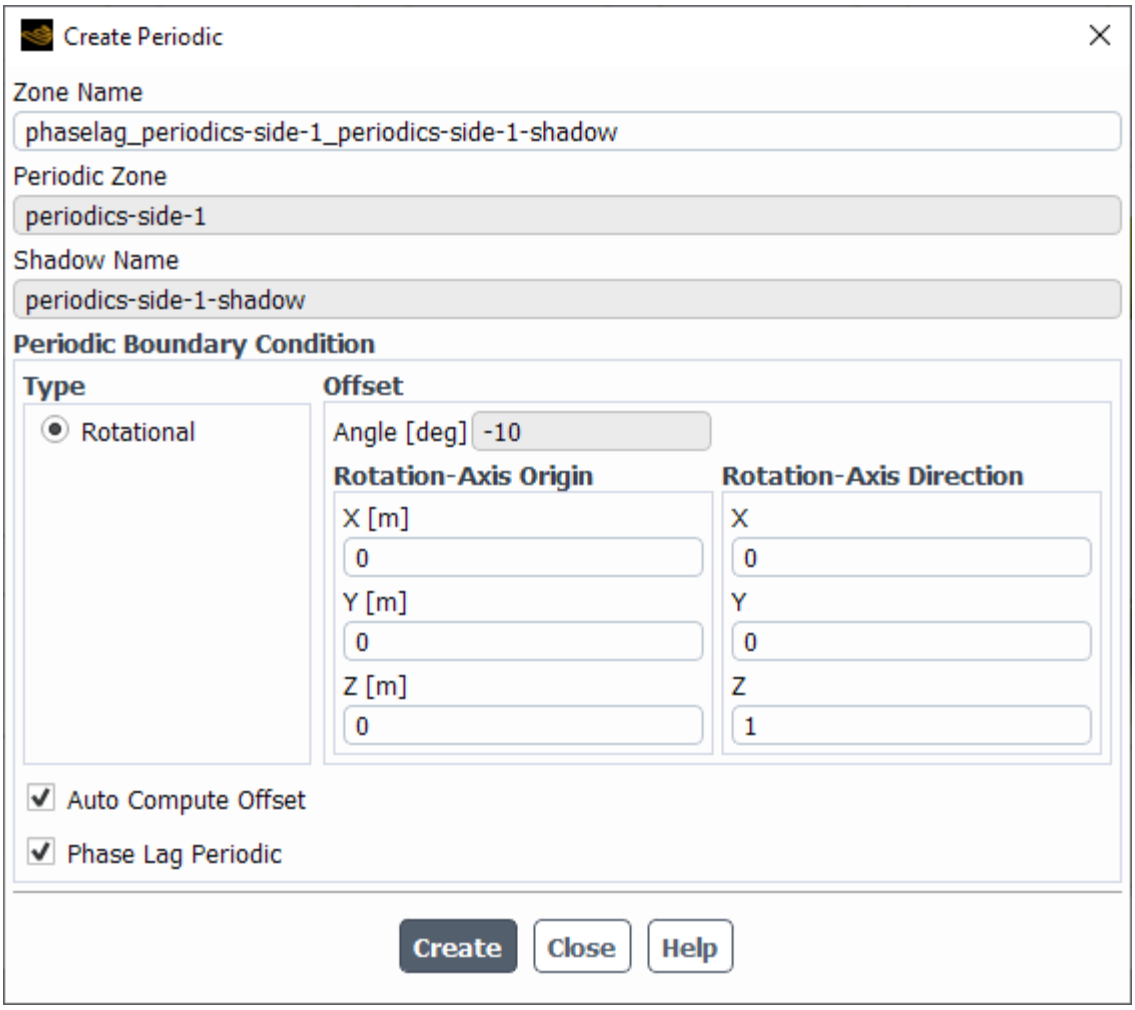
Boundaries (p. 2008). Then select **Phase Lag Periodic** at the bottom of the dialog box as shown in Figure 14.16: Create Periodic Dialog Box for a Phase-lag Interface (p. 2009).

Figure 14.15: Creating Phase Lag Periodic Interface from Boundaries



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 14.16: Create Periodic Dialog Box for a Phase-lag Interface



From the console, there are equivalent commands for creating a phase-lag interface from a periodic interface or from boundaries.

- For creating a phase-lag interface by converting a previously defined periodic interface:

  ```
  /define/turbo-model/make-phaselag-from-periodic
  ```

- For creating a phase-lag interface by specifying the two sides (boundaries) of the phase-lag Interface:

  ```
  /define/turbo-model/make-phaselag-from-boundaries
  ```

In either case, Fluent prints the following message if the phase-lag interface was successfully created:

```
Successfully created a phase-lag interface
```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.5.3.3. Phase-lag Spectral Description

The setup of the phase-lag spectral content for both use-cases of blade flutter and inlet/outlet disturbance occurs automatically once the setup acquires enough information to fully specify the phase-lag parameters.

14.5.3.3.1. Inlet/Outlet Disturbance Specific Phase-lag Method Workflow

Once the phase-lag interface has been created, the phase-lag spectral description needs to be set. For the inlet/outlet disturbance use case, it will be automatically calculated if the profile contains the parameter `NumberSectors360`, which represents the equivalent number of blades in 360 degrees of the disturbance. There should be one parameter per disturbance in the profile `.csv` file.

If there is more than one disturbance in the profile, then you need to append the disturbance number to make the parameter unique. (for example, `NumberSectors360_2` for the second disturbance). From these parameters the equivalent frequency and phase-lag angle will be calculated automatically, and no further input is needed. If successful, Fluent prints the following to the console.

```
Successfully created a phase-lag spectral description
Successfully initialized the phase-lag global spectral data structures
```

In cases where this parameter is not present in the profile `.csv` file or in cases where the disturbance(s) are defined by UDF, the phase-lag spectral description will need to be entered using the following text command in the console:

```
/define/turbo-model/create-phaselag-spectral-content
```

It prompts for all the required input to specify the frequencies and phase-shift values associated with each disturbance of the simulation in each blade row. If you chose to not use the default number of Fourier harmonics for each fundamental frequency in the specifying spectral options step, you are prompted to list the desired harmonics for each frequency in the simulation. The frequencies can be specified as disturbances where it will be computed using the input number of sectors in the 360-degree machine and a reference frame or directly input as a real value in Hertz and the attached reference frame. See below for sample dialog from a session:

```
/define/turbo-model/create-phaselag-spectral-content
Add disturbances/frequencies manually? [no] yes
List of all rows: (row-1)
Enter the list of rows to add disturbances/frequencies: (1) [row-1]
Enter the number of disturbances/frequencies for row "row-1": [0] 2
For disturbance/frequency #1:
--> Enter type (1= blade passing disturbance, 2= real frequency in [Hz], 3= blade flutter disturbance): [-1]
--> Enter the reference frame associated with this disturbance/frequency Available: ("rotating" "global")
Reference Frame: [] rotating
--> Enter the number of sectors in 360 deg. associated with this disturbance: [0] 23
For disturbance/frequency #2:
--> Enter type (1= disturbance, 2=real frequency in [Hz]): [-1] 2
--> Enter the reference frame associated with this disturbance/frequency Available: ("rotating" "global")
Reference Frame: [] rotating
--> Enter the base frequency: (in [Hz]) [0] 7695
```

Again, the same message is reported to the console if successful.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.5.3.3.2. Blade Flutter (TWM) Specific Phase-lag Workflow

For the blade flutter use-case, the spectral description is populated after the boundaries are specified for the dynamic mesh problem. When the vibrating blade(s) are selected, the attached mode shape is automatically queried for the frequency.The harmonic content is then populated and phase-lag angle calculated. To confirm this, Fluent prints the following to the console.

```
Successfully created a phase-lag spectral description
Successfully initialized the phase-lag global spectral data structures
```

To manually create the phase-lag spectral desription, you can use the following text command in the console:

`/define/turbo-model/create-phaselag-spectral-content`

The prompt list is similar but shorter than the inlet/outlet disturbance use-case because many of the spectral parameters can be automatically computed from the existing blade flutter setup.

```
/define/turbo-model/create-phaselag-spectral-content
Add disturbances/frequencies manually? [no] yes
List of all rows: (row-1)
Enter the list of rows to add disturbances/frequencies: (1) [row-1]
Enter the number of disturbances/frequencies for row "row-1": [0] 1
For disturbance/frequency #1:
--> Enter type (1= blade passing disturbance, 2= real frequency in [Hz], 3= blade flutter disturbance): [-1]
```

Again, the same message is reported to the console if successful.

The rest of the setup for a blade flutter case, can be found in Aerodynamic Damping (Blade Flutter Analysis) (p. 1972).

14.5.3.3.3. Additional Commands for Phase-lag Spectral Descriptions

There are additional commands for listing, appending, and deleting the phase-lag spectral content.

`/define/turbo-model/list-phaselag-state`

`/define/turbo-model/append-phaselag-spectral-content`

`/define/turbo-model/delete-phaselag-spectral-content`

For convenience, in addition to listing the phase-lag spectral content, the `list-phaselag-state` command lists the turbomachine description and graphics spectral content to give a better overview of the setup.

14.5.3.4. Initializing and Running Phase-lag Simulations

The suggested workflows to initialize Phase-lag simulations are as follows:

1. Steady state solution → switch to transient → modify case with appropriate phase-lag settings

   This method usually results in the best convergence.

2. Read interpolated results from a steady-state solution into an existing transient phase-lag case.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Initialization from inlet → hybrid initialization → transient phase-lag setup

Important:

After setting up a case with a phase-lag interface and then directly opening a `.dat` file from the equivalent Steady-State solution, the result is an error as the ghost-cells created for the phase-lag case will not have an equivalent in the steady state solution. Following one of the above workflows is recommended

14.5.3.5.21.2.3.6. Fourier Coefficient Postprocessing for Inlet/Outlet Disturbance and Blade Flutter Use-Cases

Refer to General Fourier Coefficient Postprocessing for Turbomachinery Cases (p. 2035) on how to enable and setup the simulation such that Fourier coefficients are available for postprocessing.

14.6. Non-equilibrium Wet Steam Model for Steam Turbines

Modeling wet steam is very important in the analysis and design of steam turbines. The increase in steam turbine exit wetness can cause severe erosion to the turbine blades at the low-pressure stages, and a reduction in aerodynamic efficiency of the turbine stages operating in the wet steam region.

You can choose the wet steam model (see Wet Steam Model Theory in the Theory Guide) by opening the **Multiphase Model** dialog box and selecting the **Wet Steam** option.

See Setting Up the Wet Steam Model (p. 3173) for information on available options.

Currently supported for use with Non-equilibrium Wet Steam Model include:

- Both pressure-based and density-based solvers.
- Phase-lag Method.

Physics → **Models** → **Multiphase**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 14.17: The Multiphase Model Dialog Box with the Wet Steam Model Selected (Pressure-Based)

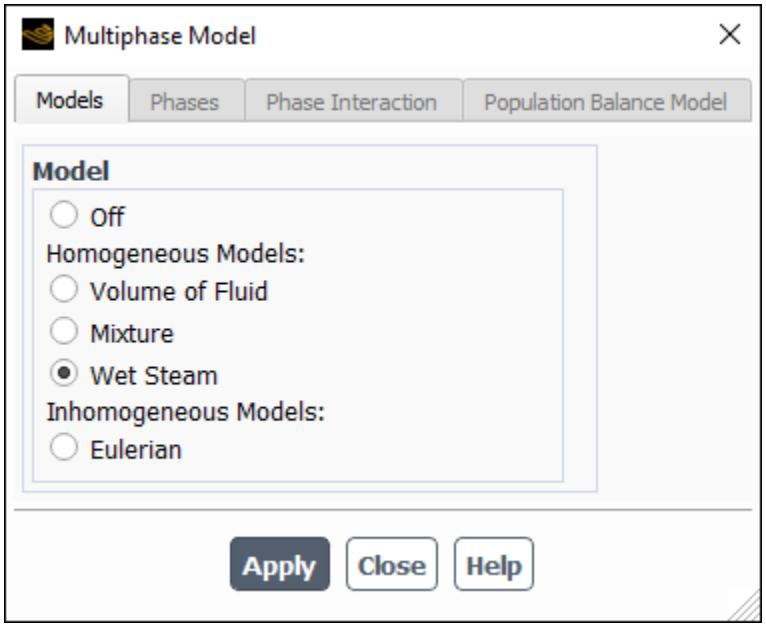
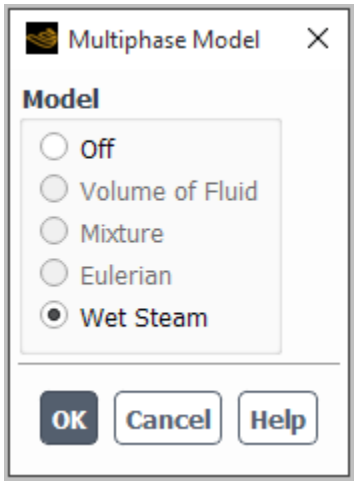


Figure 14.18: The Multiphase Model Dialog Box with the Wet Steam Model Selected (Density-Based)

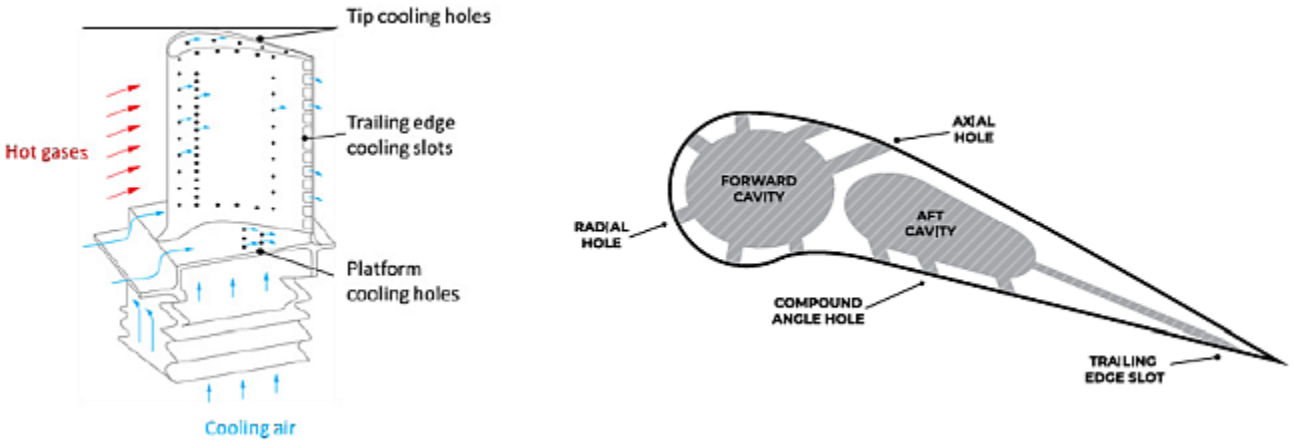


14.7. Blade Film Cooling for Gas Turbines

The cooling of the turbine blades within a gas turbine engine is usually achieved via the film-cooling approach. This is where cool gas is injected into the boundary layer of the flow over the blade from lots of small holes on its surface. This presents a problem for CFD modeling, as it can be both difficult and time-consuming to create and mesh a geometry which adequately resolves all the holes.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 14.19: Schematic Diagrams of a Cooled Turbine Blade



Within Ansys Fluent, you can easily create models for Blade Film Cooling (BFC) using the **Boundary Interface** feature. This involves taking a standard blade model setup, and creating virtual holes on an existing boundary mesh. The holes can then be treated as an inlet or outlet, and with the associated boundary condition settings, the simulation can be solved in the usual way. The holes can also be moved, or have their size and shape changed, without the need for costly remeshing.

The steps for setting up blade film cooling are discussed in the following sections:

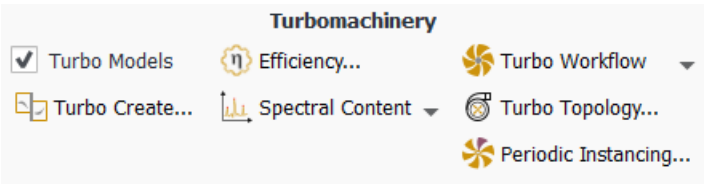
14.7.1. Specifying the Virtual Hole Geometry
14.7.2. Specifying the Boundary Interface
14.7.3. Boundary Interface Definitions
14.7.4. Specifying Overlap Boundary Conditions
14.7.5. Limitations for Boundary Interfaces and their Geometries

14.7.1. Specifying the Virtual Hole Geometry

Steps for creating a virtual Boundary Interface to perform Blade Film Cooling are outlined below:

1. Enable **Turbo Models** for the domain.

   Domain → Turbomachinery → Turbo Models

   Once enabled, **Boundary Interfaces** appears in the outline view.

2. Import hole center locations from a profile in `.csv` format. See Profiles (p. 1564) for details.

   Each data row in the profile represents an individual hole (x,y,z coordinates) and additional columns can be used to provide flow data for each hole. The Interpolation Method should remain the default **constant**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Reference Frame** specified for the profile controls how the hole positions behave during the simulation. For steady cases, the frame should be set to `global`. For transient cases, the chosen reference frame depends on the motion of the cell zone to which the holes are connected.

- If the cell zone is stationary (for example, a stator row), then the **Reference Frame** should be set to `global`.
- If the cell zone is moving (for example, a rotor), then selecting `global` for **Reference Frame** keeps the holes stationary as the rotor moves past them. Such an arrangement would be appropriate for shroud cooling. However, if the holes are intended to cool the moving rotor blade, then the **Reference Frame** should be set such that it that tracks the rotor Cell Zone (for details on creating a Reference Frame, see Creating and Using Reference Frames (p. 1161)). In this configuration, the holes will be stationary within the rotating frame and move position with the rotor.

Note:

The hole coordinates do not have to be defined precisely on the mesh surface, and a tolerance is used to locate the holes on the boundary. However, if the hole location is too far from the surface, it will not be located and is ignored.

An example of a hole definition profile is shown below:

```
[Name]
Rotor Pressure Surface Cooling

[Parameters]
InjectionTemperature = 420 [K]
MyParameter = 333 [s^-1]
MyParameter2 = 44.44 []
Diameter = 3.4 [mm]

[Spatial Fields]
x  y  z

[Data]        Velocity in Stn  Velocity in Stn  Velocity in Stn   Mass Flow Some Variable  Sector
x [m]  y [m]     z [m]   Frame u [m s^-1] Frame v [m s^-1] Frame w [m s^-1]  [kg s^-1]  [kg s^-1]     Tag [
-2.50E-01  1.08E-03  2.22E-01     2.63E+01        -9.87E+01   5.09E+01  3.71E-04     12.12         1
-2.61E-01  1.06E-03  2.22E-01     1.08E+01        -1.03E+02   5.32E+01  3.71E-04     12.12         1
-2.72E-01  1.04E-03  2.22E-01    -6.59E+00        -1.06E+02   4.60E+01  3.71E-04     12.12         1
-2.83E-01  1.03E-03  2.22E-01    -2.97E+01        -1.10E+02   4.41E+01  3.71E-04     12.12         1
-2.94E-01  1.01E-03  2.22E-01     2.07E+00        -1.12E+02   3.32E+01  3.87E-04     12.12         1
#-- End of profile Rotor Pressure Surface Cooling--

[Name]
Rotor Suction Surface Cooling

[Parameters]
InjectionTemperature = 420 [K]
MyParameter = 333 [s^-1]
MyParameter2 = 44.44 []
Diameter = 3.4 [mm]

[Spatial Fields]
x  y  z

[Data]        Velocity in Stn  Velocity in Stn  Velocity in Stn   Mass Flow Some Variable  Sector
x [m]  y [m]     z [m]   Frame u [m s^-1] Frame v [m s^-1] Frame w [m s^-1]  [kg s^-1]  [kg s^-1]     Tag [
-2.97E-01  -1.22E-03  2.25E-01     2.63E+01        -9.87E+01   5.09E+01  3.71E-04     12.12         1
-2.86E-01  -1.24E-03  2.22E-01     1.08E+01        -1.03E+02   5.32E+01  3.71E-04     12.12         1
-2.75E-01  -1.26E-03  2.22E-01    -6.59E+00        -1.06E+02   4.60E+01  3.71E-04     12.12         1
-2.63E-01  -1.29E-03  2.22E-01    -2.97E+01        -1.10E+02   4.41E+01  3.71E-04     12.12         1
```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

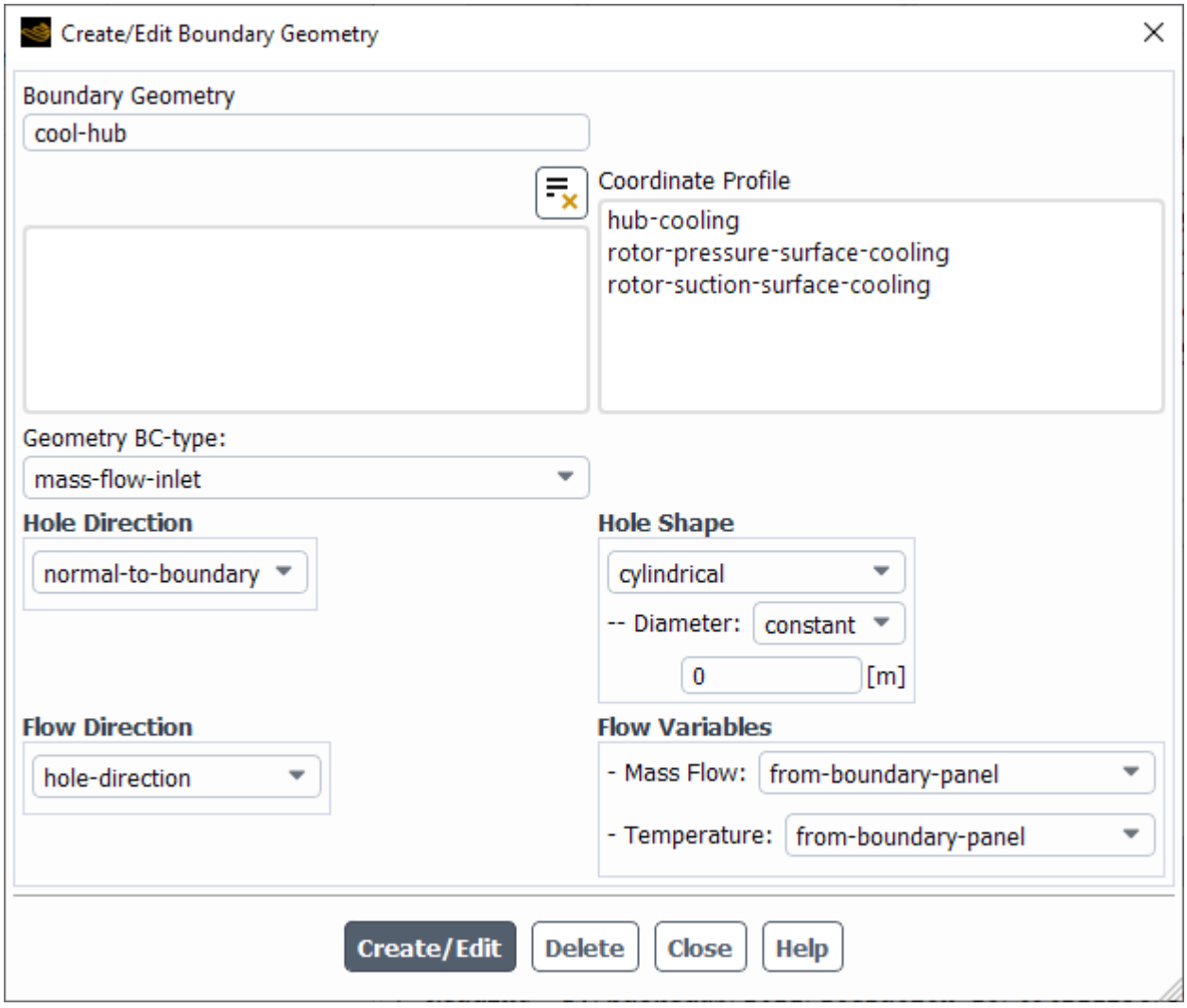
```
-2.52E-01  -1.32E-03  2.22E-01     2.07E+00        -1.12E+02   3.32E+01  3.87E-04     12.12          1
#-- End of profile Rotor Pressure Surface Cooling--
```

Note:

Multiple hole definition profiles can included in one `.csv` file as shown in the example.

3. Create a virtual geometry to represent the holes from the **Create/Edit Boundary Geometry** dialog box.

Setup → Boundary Interfaces → Create/Edit Geometry



1. Under **Boundary Geometry**, enter a name for the virtual hole geometry.

2. Under **Coordinate Profile**, select a profile to define the hole center locations. The hole center profile must be defined in `.csv` file format.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Under **Geometry BC-type**, select the type of boundary for the holes. The options are `mass-flow-inlet`, `mass-flow-outlet`, `pressure-inlet`, or `wall`. In most cases for blade film cooling, you will specify the holes as mass flow inlets, although for a bleed you would require an outlet.

4. Under **Hole Direction Specification**, select one of the following options:

   - `normal-to-boundary`

     The holes are oriented normal to the surface based on their location.

   - `cartesian`

     The holes are oriented by specifying a **X**, **Y**, and **Z Direction Component**. Using a `constant`direction component orients all holes in the same direction. If you want individual holes to be oriented in different directions, select `profile` and choose the appropriate **Profile Variable** to assign the direction components for each individual hole based on data from the profile. You can also select a profile Parameter which will apply the constant value to a **Direction Component** for all holes.

   - `cylindrical`

     The holes are oriented by specifying a **Radial**, **Tangential**, and **Axial Direction Component**. The axial direction will be defined by the Reference Frame properties of the parent cell zone. Using a `constant` direction component orients all holes in the same direction. If you want individual holes to be oriented in different directions, select `profile` and choose the appropriate **Profile Variable** to assign the direction components for each individual hole based on data from the profile. You can also select a profile Parameter which will apply the constant value to a **Direction Component** for all holes.

5. Under **Flow Direction Specification**, you have the option of specifying the flow direction independently from the hole direction. You have the following options:

   - `hole-direction`

     The flow is aligned with the hole direction.

   - `from-boundary-panel`

     The flow direction is taken from the overlap boundary condition specification. For details, see Specifying Overlap Boundary Conditions (p. 2024).

   - `normal-to-boundary`

     The flow is oriented normal to the surface regardless of hole direction.

   - `cartesian`

     The flow is oriented by specifying a **X**, **Y**, and **Z Component of Flow**. Using a `constant` **Component of Flow** orients the flow in the same direction for each hole. If you want individual holes to have different flow directions, select `profile` and choose the appropriate **Profile Variable** to assign the flow direction for each individual hole based on data from the profile. You can also select a profile Parameter which will apply the constant value to a **Component of Flow** for all holes.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- `cylindrical`

  The flow is oriented by specifying a **Radial**, **Tangential**, and **Axial Component of Flow**. The axial direction will be defined by the Reference Frame properties of the parent cell zone. Using a `constant`**Component of Flow** orients the flow in the same direction for each hole. If you want individual holes to have different flow directions, select `profile` and choose the appropriate **Profile Variable** to assign the flow direction for each individual hole based on data from the profile. You can also select a profile Parameter which will apply the constant value to a **Component of Flow** for all holes.

6. Under **Hole Shape**, select `cylindrical` or `rectangle` and specify the relevant dimensions.

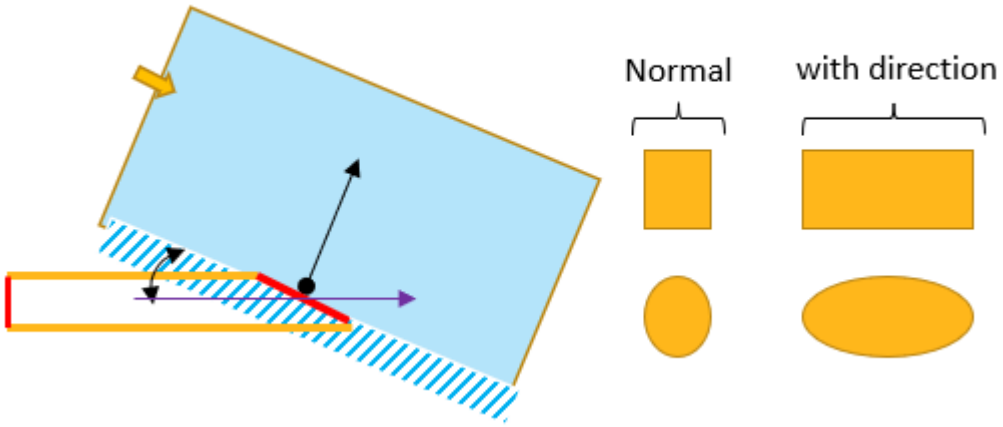
   - For cylindrical holes, specify the **Diameter** as `constant` if each hole is the same size or select `profile` and choose the appropriate **Profile Variable** to assign the diameter for each individual hole based on data from the profile. You can also select a profile Parameter which will apply the constant value to all holes.

   - For rectangular holes, specify the **Length** and **Width** as `constant` if each hole is the same size or select `profile` and choose the appropriate **Profile Variable** to assign the width or length for each individual hole based on data from the profile. You can also select a profile Parameter which will apply the constant value to all holes.

Note:

For both cylindrical and rectangular hole shapes, the orientation of the hole is based on the projection of the specified hole direction onto the local underlying boundary surface. If the specified hole direction is not normal to the boundary, then the hole shape is modified to account for this as if the hole were "drilled" into the boundary in the specified direction. As such, a cylindrical hole definition can result in an elliptical shape on the boundary, and similarly a rectangle defined with equal width and length can result in a non-square hole, as shown in Figure 14.20: Hole Shape Dependency from Hole Direction (p. 2018).

Figure 14.20: Hole Shape Dependency from Hole Direction



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Under **Flow Variables**, there exist additional settings depending on the physics present in the simulation. If **Energy** is enabled, and the **Geometry BC-type** is an inlet or wall, then a **Temperature** specification is required. If the **Geometry BC-type** is an inlet or an outlet, then a **Mass Flow** rate or **Total Pressure** specification is required.

   For **Mass Flow**, the following options are available:

   - `from-boundary-panel`

     The mass flow is taken from the overlap boundary condition specification. For details, see Specifying Overlap Boundary Conditions (p. 2024).

   - `profile`

     If you want individual holes to have different mass flow rates, select `profile` and choose the appropriate **Profile Variable** to assign the mass flow rate for each individual hole based on data from the profile. You can also select a profile Parameter which will apply the constant value to all holes.

   - `constant-per-hole`

     A single value for the mass flow rate is applied to each hole.

   - `total-mass-flow`

     This value corresponds to the total mass flow for all the located holes, which is distributed equally between those holes.

   - `total-for-all-sectors`

     This value corresponds to the total mass flow of all the located holes, assuming a full 360 degree wheel were being modeled. The specified mass flow is distributed proportionately to any modeled sectors, and then equally between the holes within those sectors.

   For **Temperature**, the following options are available:

   - `from-boundary-panel`

     The temperature is taken from the overlap boundary condition specification. For details, see Specifying Overlap Boundary Conditions (p. 2024).

   - `profile`

     If you want individual holes to have different temperatures, select `profile` and choose the appropriate **Profile Variable** to assign the temperature for each individual hole based on data from the profile. You can also select a profile Parameter which will apply the constant value to all holes.

   - `radial-distribution`

     Allows for a temperature variation within each individual hole. The temperature for each face within a hole is defined by the equation:

$$T = kT_{ref}\left(A + B\frac{r^2}{R^2}\right) \tag{14.14}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Where A and B are user-defined constants and T_{ref} is the Reference Temperature, which can be defined as `constant` under **Reference Value** or select `profile` to use hole-specific values for Reference Temperature as defined from the input profile.

The resultant temperature variation is quadratic, based on the radial position of the hole face centers from the input hole center, compared to the input hole radius. The value of k is evaluated automatically by Fluent, to ensure the enthalpy flux for each hole is approximately the same as if the hole were specified with a uniform temperature equal to the Reference Temperature. This helps to reduce the sensitivity of the model to the mesh resolution of the holes themselves.

For a pressure-inlet, a **Total Pressure** and **Loss Coefficient** specification is required.

For total pressure, the following options are available:

- `from-boundary-panel`

  The total pressure is taken from the overlap boundary condition specification. For details, see Specifying Overlap Boundary Conditions (p. 2024).

- `profile`

  If you want individual holes to have different total pressure specifications, select `profile` and choose the appropriate **Profile Variable** to assign the total pressure for each individual hole based on data from the profile. You can also select a profile Parameter which will apply the constant value to all holes.

The **Loss Coefficient** reproduces the pressure head loss due to the hole drilling itself. For details on the loss coefficient, see Specifying the Loss Coefficient (p. 1431). The following options are available:

- `from-boundary-panel`

  The loss coefficient is taken from the overlap boundary condition specification. For details, see Specifying Overlap Boundary Conditions (p. 2024).

- `constant`

  A single, constant value is applied to all holes.

- `factor-and-length-ratio`

  Two further values are required for an internal calculation of the loss coefficient. Both are constant for all holes: A Darcy **Friction Factor** and an assumed **Length-to-Diameter Ratio** for the hole drilling.

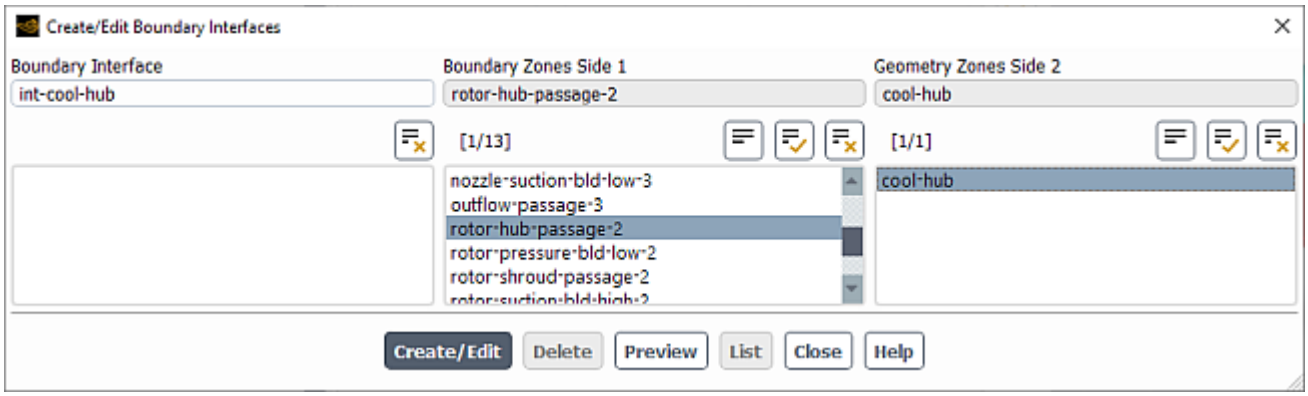
8. Once fully defined, click **Create/Edit**. Note that existing geometry definitions can be modified and you can define multiple geometries if needed. It is often useful to group holes into separate geometries according to their location: for example, blade pressure side, blade suction side, hub, shroud, and so on.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.7.2. Specifying the Boundary Interface

After the hole geometry has been defined, you must create the Boundary Interface between the geometry and an existing boundary. To do this, right-click **Boundary Interfaces** in the outline view and select **New** to open the **Create/Edit Boundary Interfaces** dialog box.

Setup → Boundary Interfaces → New



1. Under **Boundary Interface**, enter a name for the interface.
2. Under **Boundary Zones Side 1**, select the boundary that the holes will be located on.
3. Under **Geometry Zones Side 2**, select the boundary geometry that was created in Specifying the Virtual Hole Geometry (p. 2014).
4. Click **Preview** and the selected Boundary Zones will be displayed, along with symbols indicating the proposed locations of the holes as implied by the selected Geometry Zones.

   Note that the symbols do not indicate the actual size and shape of the holes; they are only markers. The default symbols and colors are used but these can be changed using the following console command for each Geometry: `/define/virtual-boundary/hole-geometry/edit/[geometry-name]/preview`.
5. Click **Create/Edit** and the boundary interface will appear in the list. You can also select previous defined boundary interfaces to edit or delete.

Once created, Fluent intersects the selected boundaries with the selected geometries, resulting in one overlap boundary for each geometry and separate non-overlap boundaries for each of the underlying boundary zone locations, as shown in Figure 14.21: Creation of Overlap and Non-overlap Surfaces (p. 2022). The new overlap boundaries will automatically be created as mass-flow inlets or outlets, or an inlet vent (for pressure), depending on the type defined within the geometry.

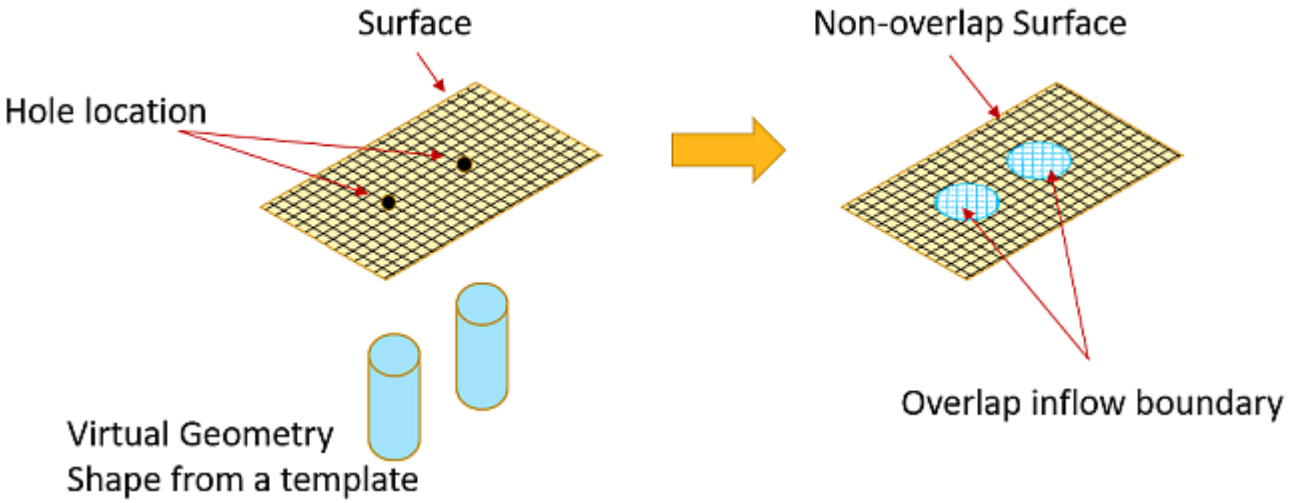
Note:

If a `profile` option, or any option other than `constant` or `from-boundary-panel` is selected for **Flow Direction Specification** or within **Flow Variables** for the relevant **Geometry Zone**, then Fluent creates a special internal profile that is only valid for this overlap boundary. It has a name beginning with `vbr` and also contains the overlap boundary name. Fluent then assigns this internal profile to the relevant boundary conditions

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

on the overlap boundary, corresponding to the Boundary Geometry specification. The internal profile will not be visible or usable anywhere but on the appropriate overlap boundary.

Figure 14.21: Creation of Overlap and Non-overlap Surfaces



14.7.2.1. Hole Locations for Rotationally Periodic Interface Pairs

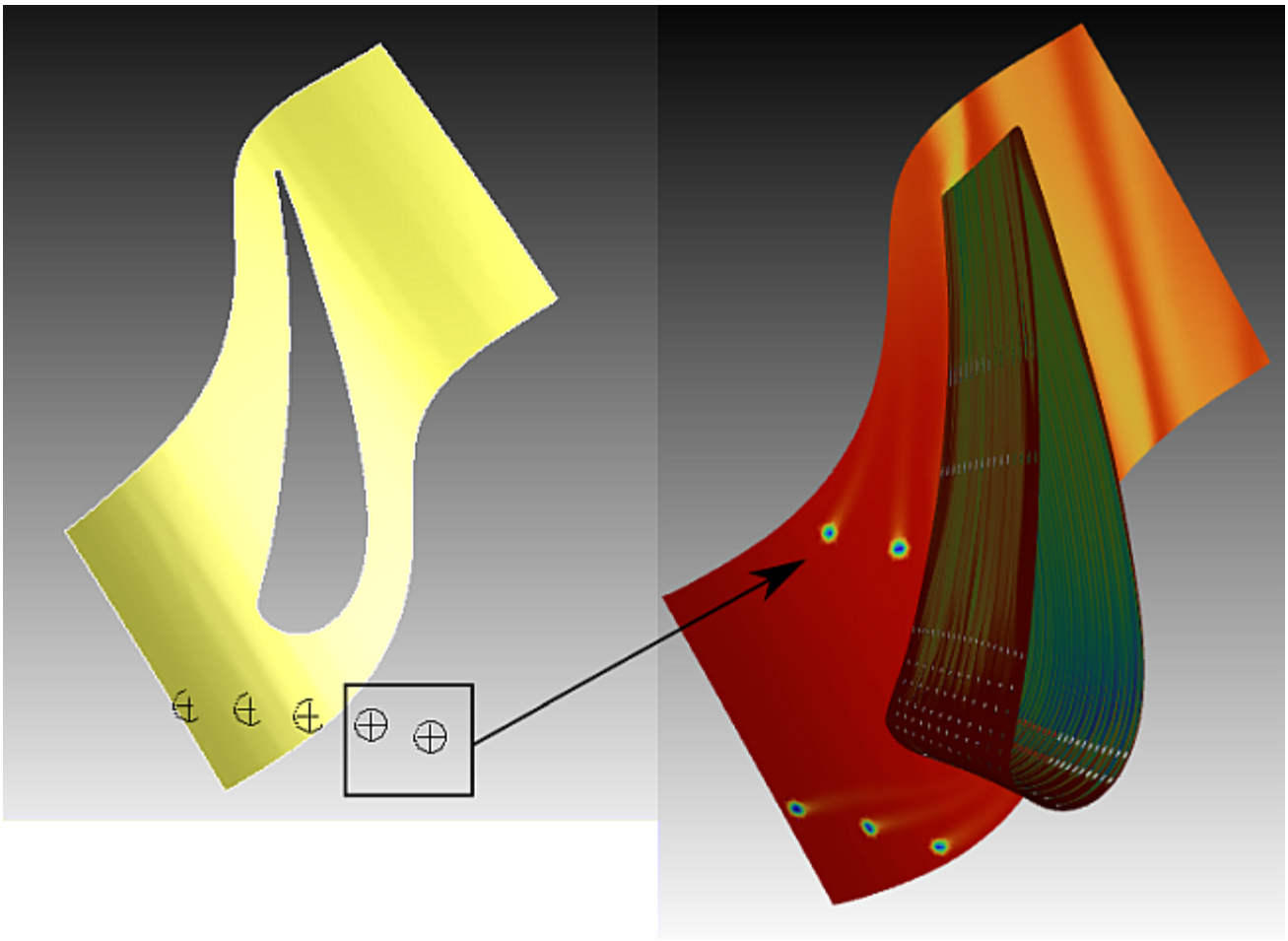
In the configuration where you have a rotationally periodic model with hole locations specified for a Boundary Zone that fall outside of that Boundary Zone and the periodic boundaries, then Fluent will attempt to rotate the hole locations until they overlap with the Boundary Zone as shown in Figure 14.22: Holes Outside the Parent Cell Zone Rotated to Overlap with Mesh (p. 2023)

This is triggered automatically when a Boundary Interface is created using a Boundary Zone that is attached to a parent Cell Zone that has an axis vector defined and at least on rotationally periodic interface pair. Any hole locations specified within the Boundary Geometry that are not initially overlapping with the parent Cell Zone mesh will be rotated (using the pitch angle defined by the periodic pair) until they do overlap. This rotation moves the hole location from just outside one periodic boundary, to just inside the other periodic boundary, as if it were located in an adjacent passage. If, even after successive rotation, the hole cannot be located within the mesh, then that hole will remain undefined.

This automated feature is designed to lower the sensitivity of the positioning of the rotationally periodic boundary pairs, in relation to hole locations.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 14.22: Holes Outside the Parent Cell Zone Rotated to Overlap with Mesh



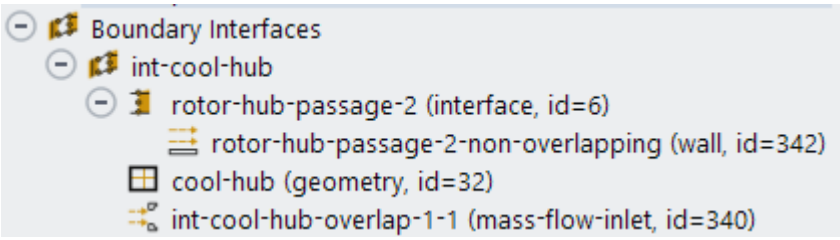
Additionally, if a hole is large enough to touch or cross one of the periodic boundaries, then the portion of the hole outside that boundary will be transformed so that it appears inside the other periodic boundary. The complete hole will then appear to span the periodic pair.

14.7.3. Boundary Interface Definitions

After a Boundary Interface has been created, you can expand its entry in the outline view to see all of the components that make up that boundary interface.

Setup → Boundary Interfaces → [Name]

In this example, expanding the Boundary Interface **`int-cool-hub`** reveals the following:



- Boundary zone (**`rotor-hub-passage-2`**)

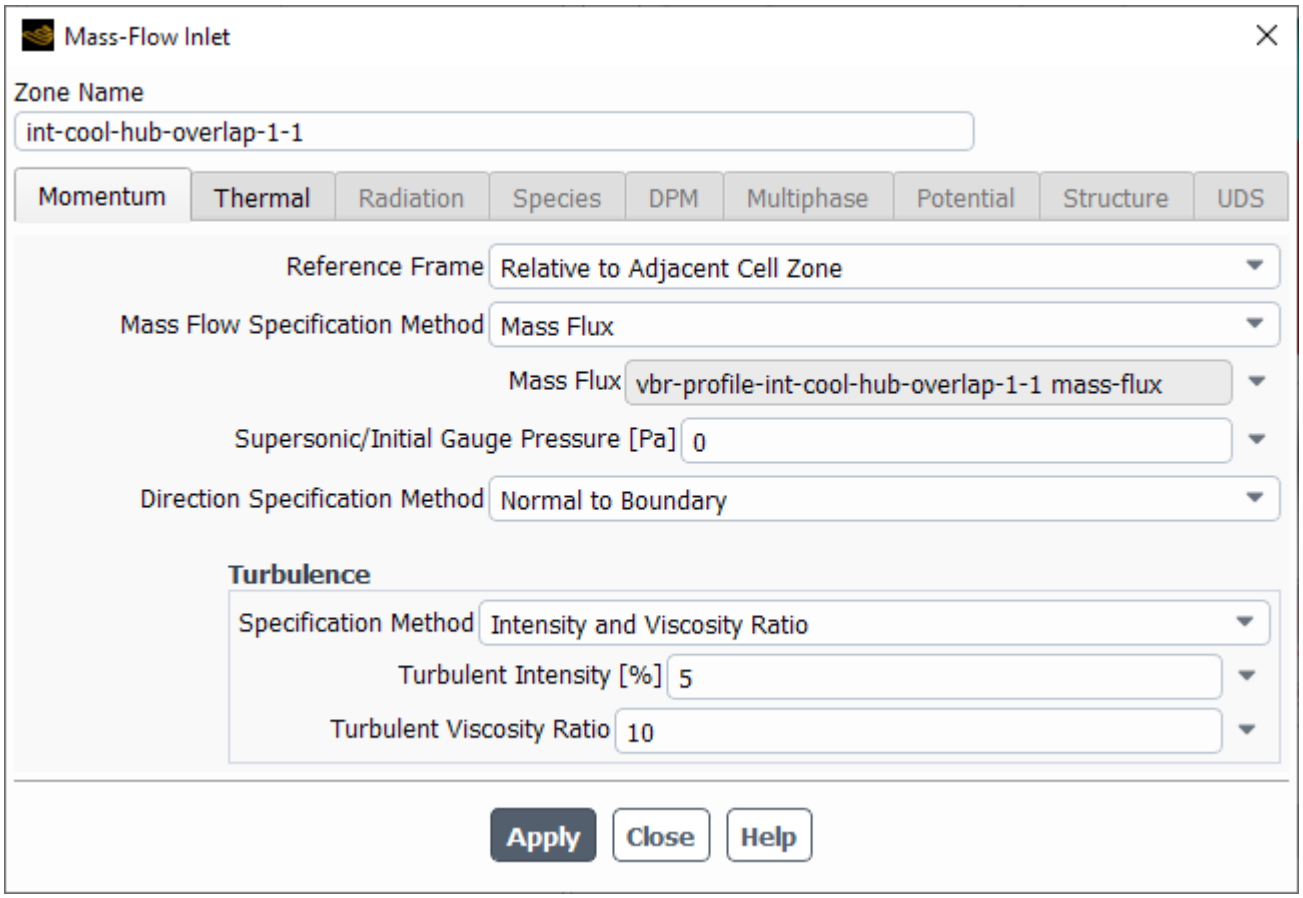
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Non-overlap zone (**rotor-hub-passage-2-non-overlapping**)
- Geometry (**cool-hub**)
- Overlap zone (**int-cool-hub-overlap-1-1**)

You can right-click the overlap or non-overlap zone and select **Display** to view the zone in the graphics viewer. The non-overlap zone displays the non-intersected faces and the overlap zone displays the intersected mesh, revealing the exact hole shapes and sizes.

14.7.4. Specifying Overlap Boundary Conditions

Once the Boundary Interface is created, you may need to specify additional boundary conditions at the new overlap boundary.



1. The boundary type will already be set based on the original geometry definition (Mass-Flow Inlet, Mass-Flow Outlet, or Wall).
2. The overlap boundary will have some of its options automatically set by Fluent, depending on the associated Boundary Geometry specification. For example, if you selected `profile` or `constant-per-hole` for the relevant Boundary Geometry, then Fluent will automatically assign the relevant internal profile to the **Mass Flux** specification. However, if the `from-boundary-panel` option is chosen then you must manually specify the mass flow (or mass flux) directly. This value then corresponds to the total mass flow for all the located holes which is then distributed between those holes based on area-weighting.

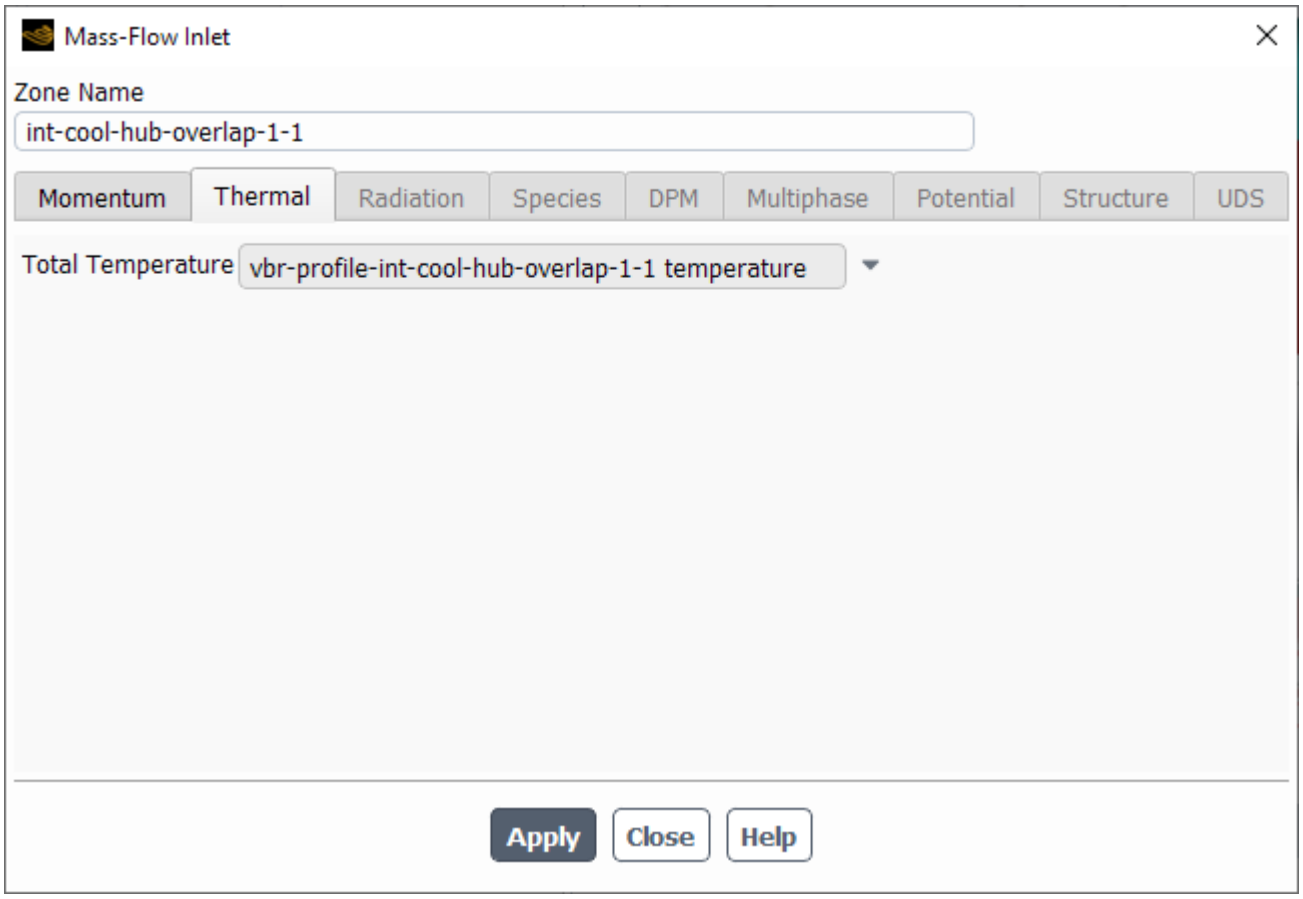
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Verify the flow direction for each hole. The **Direction Specification Method**, **Coordinate System**, and the **Flow Direction** components should all have been set automatically according to the Boundary Geometry specification. Note that if Fluent has not set these options, and instead you specify the flow direction components as constant values, then each hole will have the same flow direction.

4. Set appropriate values for other inflow quantities, where necessary, such as turbulence or temperature (shown below). For a quantity such as temperature, hole-specific data can be used by selecting the `profile` option for **Temperature** on the relevant Boundary Geometry, and choosing an additional data column from the original geometry definition profile.

Important:

You are still able to manually edit the overlap Boundary Condition and change any automatically generated settings. However, this can lead to inconsistency with the Boundary Geometry specification. It is generally recommended that you change any overlap Boundary Condition indirectly by editing the relevant geometry (see Specifying the Virtual Hole Geometry (p. 2014)).



14.7.5. Limitations for Boundary Interfaces and their Geometries

- Only available for boundaries, and not volumes. Flow injection is supported at a Boundary Interface only, which must refer to an existing boundary definition.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Holes must have their size and shape defined such that they do not overlap with another hole.
- Hole centers must be located within, or close to, a Cell Zone mesh. If any are not, then a warning will be printed that those holes will not be included and they will not contribute to fluid flow.
- Flow boundary conditions are currently specified as constant for each hole, and cannot vary across an individual hole.
- Reported boundary fluxes will show the specified mass flow for each hole overlap boundary, as well as the total mass flow for all geometries intersecting the original underlying boundary. However, the Net Results will not include this duplication and should still tend to zero across all boundaries for a converged solution.
- When modeling Blade Film Cooling on a moving rotor blade in a transient simulation, where the Boundary Interface Geometry references a profile defined in a local reference frame rotating with the rotor, an incorrect Mass Flow will be applied to the holes unless the `from-boundary-panel` option has been chosen for Mass Flow on the geometry. Also, Fluent may become unstable if the rotation angle becomes greater than one pitch.

  Even if the `from-boundary-panel` option has been selected, the hole positions may not be updated correctly with the rotor position if more than one Boundary Interface is defined.

14.8. Turbomachinery Description

Many turbomachinery models and/or features require some information about the turbomachine that is being modeled such as the number of modeled blade rows, the number of modeled blade passages per row (or sectors), and the number of blades per full wheel. This information is used in the setup of computational methods or postprocessing functionalities. There are two method to described a turbomachine:

- Using the **Turbo Workflow** (as described in Using the Turbomachinery Guided Workflow (p. 1929)) automatically defines the turbomachine description.
- If the Turbo Workflow is not used, you will need to enter this information manually using the following console commands:

`/define/turbo-models/create-turbomachine-description`

Describes the conceptual/abstract machine and CFD model setup and is required as a preliminary step to the following features:

- Harmonic export (as described in Visualizing and Exporting Blade Flutter Harmonics with TWM (p. 1994))
- Harmonic postprocessing (as described in General Fourier Coefficient Postprocessing for Turbomachinery Cases (p. 2035))

When this command is activated, Fluent prompts for the number of rows that comprise the turbomachine model, and for each row, you need to specify a name (or default is used), the cell IDs that define each row, the number of sectors in the 360-degree wheel, and the number of modeled sectors in the CFD mesh model.

`/define/turbo-models/list-turbomachine-description`

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Writes the current state of the turbomachine description to the console as a result of using the Turbo Workflow or manually entering it using the `create-turbomachine-description` command.

`/define/turbo-models/delete-turbomachine-description`

This command deletes a previously defined turbomachine description.

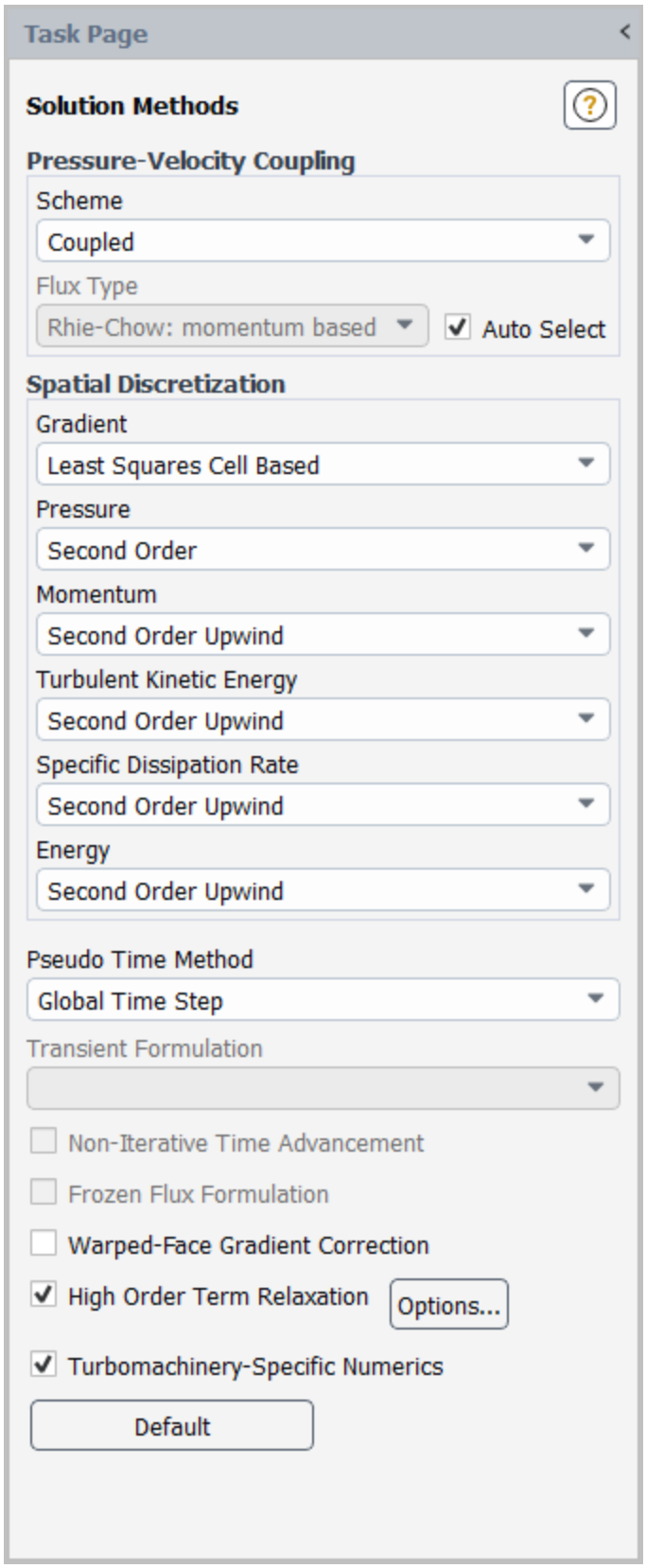
14.9. Turbomachinery-Specific Numerics

When the **Turbo Models** option is enabled with the pressure-based solver for a steady calculation, you have the option of using turbomachinery-specific numerics (TSN), that is, built-in customized numerics settings that can help to achieve and accelerate the convergence for turbomachinery flows. TSN is only available with the **Pseudo Time Method** (Performing Calculations with a Pseudo Time Method (p. 3673)) set to the **Global Time Step** option (Global Time Step Method Settings (p. 3676)).

To use TSN, enable the **Turbomachinery-Specific Numerics** option in the Solution Methods Task Page (p. 5157):

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 14.23: The Turbomachinery-Specific Numerics Option in the Solution Methods Task Page



TSN activates built-in solver customizations, along with the following optional solver settings that can improve convergence for turbomachinery flows:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- High Order Term Relaxation (for details, see High Order Term Relaxation (HOTR) (p. 3578))
  - Using **Convection Only** for the HOTR **Type**, with the **Relaxation Factor** set to 0.1 for all variables
- The **Apply Limiter Filter** option, with modified parameters (for details, see Selecting Gradient Limiters (p. 3626))

Disabling TSN will restore these options to their default values.

Note:

TSN is especially helpful for achieving faster and robust convergence when you have selected **Automatic** for the **Time Step Method** (in the Run Calculation Task Page (p. 5207)) and are using a larger **Time Scale Factor**, along with the default pseudo time method settings. If you encounter difficulties with achieving convergence using TSN, it is recommended that you disable TSN and reduce the pseudo time explicit relaxation for momentum (in the Solution Controls Task Page (p. 5163)) from the default value of 0.5 to 0.1. This has been shown to achieve convergence when the default settings and/or TSN are not effective, albeit at a much slower rate.

14.10. Turbomachinery Postprocessing

In addition to the many graphics tools already discussed, Ansys Fluent also provides turbomachinery-specific postprocessing features that can be accessed once you have defined the topology of the problem. Information on postprocessing for turbomachinery applications is provided in the following sections:

14.10.1. Defining the Turbomachinery Topology
14.10.2. Contours and Vectors Visualization for Turbomachinery
14.10.3. General Fourier Coefficient Postprocessing for Turbomachinery Cases
14.10.4. Circumferential-Averaged Profile Extraction
14.10.5. Turbo Post
14.10.6. Calculating Turbomachine Performance

14.10.1. Defining the Turbomachinery Topology

In order to establish the turbomachinery-specific coordinate system used in subsequent postprocessing functions, Ansys Fluent requires you to define the topology of the flow domain. The procedure for defining the topology is described further in this section, along with details about the boundary types.

The current implementation of the turbomachinery topology definition for postprocessing is no longer limited to one row of blades at a time. If your geometry contains multiple rows of blades, you can define all turbomachinery topologies simultaneously. You can name and/or manage all topologies

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and perform various turbomachinery postprocessing tasks on a single topology or on all topologies at once.

Important:

The turbo coordinates can only be generated properly if the correct rotation axis is specified in the boundary conditions dialog box for the fluid zone (see Specifying the Rotation Axis (p. 1323)).

To define the turbomachinery topology in Ansys Fluent, you will use the Turbo Topology Dialog Box (p. 5644) (Figure 14.24: The Turbo Topology Dialog Box (p. 2030)).

Domain → Turbomachinery → Turbo Topology...

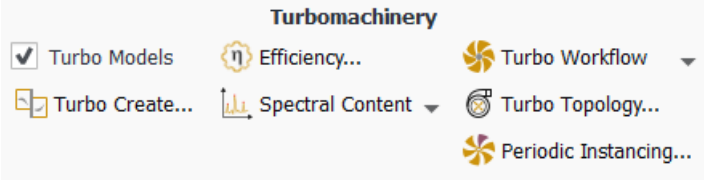
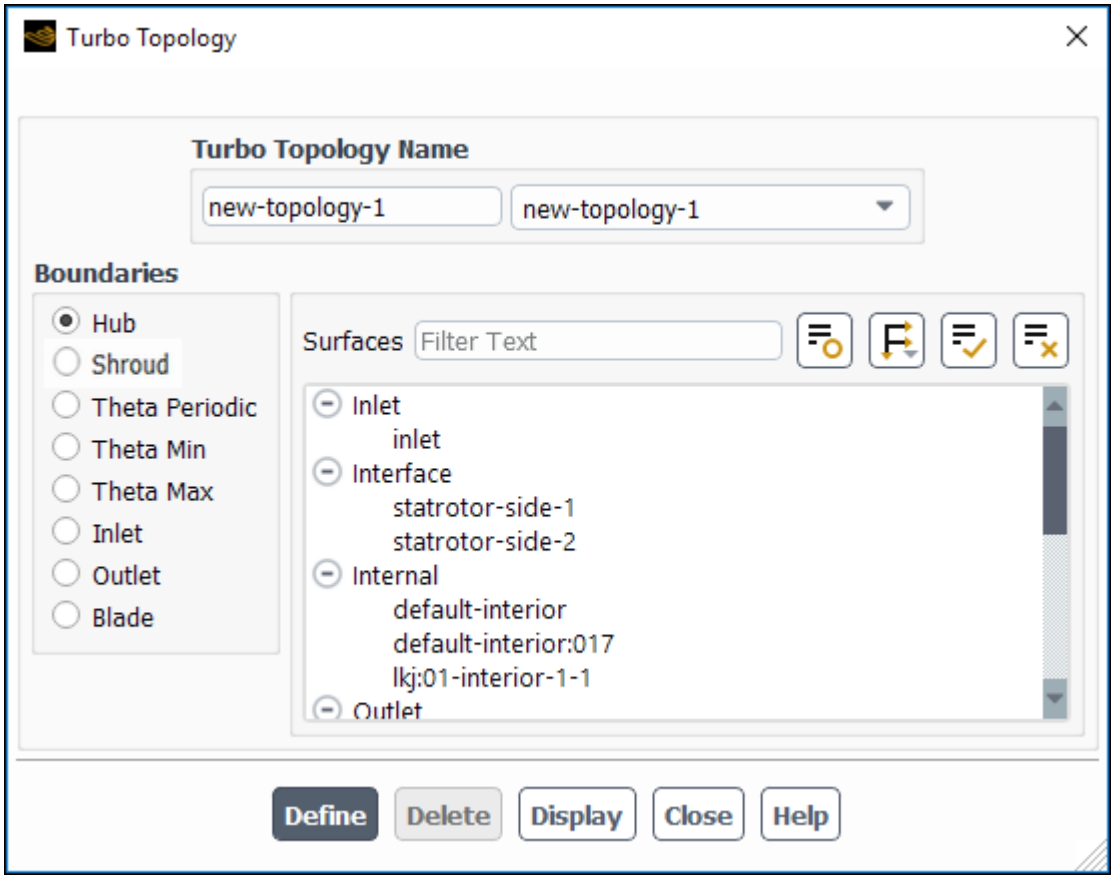


Figure 14.24: The Turbo Topology Dialog Box



The procedure for defining topology for your turbomachinery application are as follows:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select a boundary type under **Boundaries** (for example, **Hub** in Figure 14.24: The Turbo Topology Dialog Box (p. 2030)). The boundary types are described in detail in Boundary Types (p. 2032).

2. In the **Surfaces** list, choose the surface(s) that represent the boundary type you selected in step 1.

   If you want to select several surfaces of the same type, click and select **Surface Type** under **Group By**, which organizes the surfaces in a tree view that is grouped by surface type.

   Another shortcut is to use the **Filter Text** entry box to filter the **Surfaces** list to show only the surfaces that match the pattern you enter. For additional information on using the **Filter Text** entry box, see Filter Text Entry Boxes (p. 926).

3. Repeat the steps 1 and 2 for all the boundary types that are relevant for your model.

   Important:

   For a complete turbo topology definition the surfaces defined as inlet, outlet, hub, shroud, periodic, theta min, and theta max (if available) should form a closed domain.

4. Enter a name in the **Turbo Topology Name** field or keep the default name.

5. Click **Define** to complete the definition of the boundaries.

   Ansys Fluent will inform you that the turbomachinery postprocessing functions have been activated.

6. Specify a position vector that is defined as θ=0. This position vector should be outside the domain, for example, if your domain lies in the first and second quadrant, specify negative y-axis as the zero θ line. This will ensure that there is no discontinuity in angular coordinates within the domain. This can be done using the `display/set/zero-angle-dir` command.

   Default zero θ=0 line is +y-axis. If this axis passes through the domain, you should define the zero θ line, so as to satisfy above criteria.

7. To view a defined topology, select the topology from the **Turbo Topology Name** drop-down list and click **Display**. The defined topology is shown in the active graphics window. This enables you to visually check the boundaries to ensure that you have defined them correctly.

8. To edit a defined topology, select the topology from the **Turbo Topology Name** drop-down list, make the appropriate changes and click **Modify**.

9. To remove a defined topology, select the topology from the **Turbo Topology Name** drop-down list and click **Delete**.

   Important:

   Note that the topology setup that you define will be saved to the case file when you save the current model. Thus, if you read this case back into Ansys Fluent, you do not need to set up the topology again.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

However, use of a boundary condition file to set the turbo topology for two similar cases may not work properly. In that case you need to set the turbo topology manually.

Note:

From Fluent 2020 R1 onwards, turbo topology TUI access can be found at `define/turbo-model/turbo-topology`.

14.10.1.1. Boundary Types

The boundaries for the turbomachinery topology are as follows (see Figure 14.25: Turbomachinery Boundary Types (p. 2033)):

Hub

is the wall zone(s) forming the lower boundary of the flow passage (generally toward the axis of rotation of the machine).

, Shroud

is the wall zone(s) forming the upper boundary of the flow passage (away from the axis of rotation of the machine).

Theta Periodic

is the periodic boundary zone(s) on the circumferential boundaries of the flow passage.

Theta Min

and **Theta Max** are the wall zones at the minimum and maximum angular (θ) positions on a circumferential boundary.

Inlet

is the inlet zone(s) through which the flow enters the passage.

Outlet

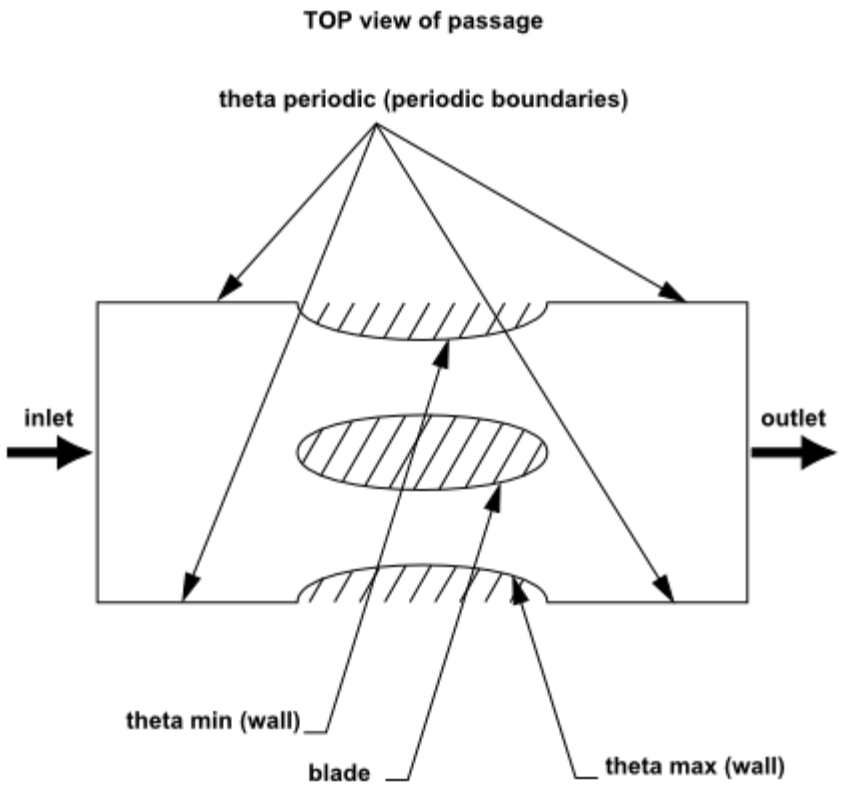
is the outlet zone(s) through which the flow exits the passage.

Blade

is the wall zone(s) that defines the blade(s) (if any). Note that these zones cannot be attached to the circumferential boundaries. For this situation, use **Theta Min** and **Theta Max** to define the blade.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 14.25: Turbomachinery Boundary Types



14.10.1.2. Turbomachinery-Specific Variables

After completing the turbo topology definition detailed in Defining the Turbomachinery Topology (p. 2029), the following turbomachinery-specific variables are available for postprocessing in Ansys Fluent:

- **Meridional Coordinate**
- **Abs Meridional Coordinate**
- **Spanwise Coordinate**
- **Abs (H-C) Spanwise Coordinate**
- **Abs (C-H) Spanwise Coordinate**
- **Pitchwise Coordinate**
- **Abs Pitchwise Coordinate**

These variables are contained in the **Mesh...** category of the variable selection drop-down list. See Field Function Definitions (p. 4167) for their definitions.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.10.2. Contours and Vectors Visualization for Turbomachinery

Turbomachinery postprocessing commonly requires viewing contour and vector plots on turbo surfaces. As a prerequisite, you must first follow the instructions in Defining the Turbomachinery Topology (p. 2029) to enable the Turbomachinery-Specific Variables (p. 2033) that are used when creating a turbo surface.

The following topics are discussed:

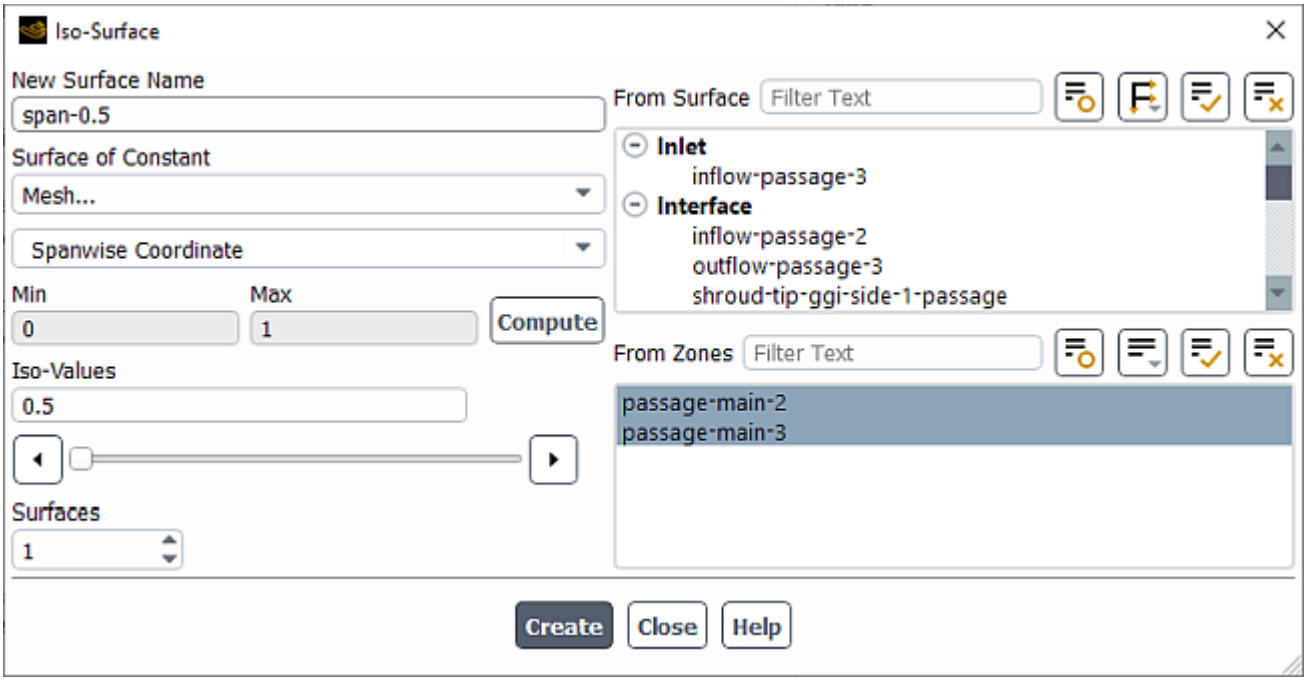
14.10.2.1. Creating Turbo Surfaces

14.10.2.2. Periodic Instancing

14.10.2.1. Creating Turbo Surfaces

Turbo surfaces are created using the Iso-Surface Dialog Box (p. 5501). The Turbomachinery-Specific Variables (p. 2033) are available under the **Mesh...** category for the **Surface of Constant** drop-down list. For details on creating iso-surfaces, see Iso-surfaces (p. 3838).

Figure 14.26: Iso-Surface Dialog Box for Turbo Surfaces



1. Enter a name for the turbo surface under **New Surface Name.**
2. Under **Surface of Constant**, select **Mesh...** and the appropriate turbo-specific variable (for example, Spanwise Coordinate).
3. Under **From Zones**, select the zone to create the turbo surface in.
4. Click **Compute**.
5. Specify **Iso-Values** or use the slide tool.
6. Click **Create**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The turbo surfaces will be used to display contours or velocity vectors. It is recommended that you create an iso-surface for each zone separately.

14.10.2.2. Periodic Instancing

In turbomachinery analysis, it is common that only a portion of the machine is modeled. Therefore, in order to visualize the periodic repeating pattern of a turbomachine, you can use **Periodic Instancing** to replicate passages around an axis.

Periodic Instancing is accessed from the **Turbomachinery** ribbon:

Domain → **Turbomachinery** → **Periodic Instancing...**

For more information on Periodic Instancing, see Mirroring and Periodic Repeats (p. 4036) and Periodic Instancing Dialog Box (p. 5276).

14.10.3. General Fourier Coefficient Postprocessing for Turbomachinery Cases

Fluent provides a post-processing Fourier coefficient visualization capability that is available for all transient cases when **Turbo Models** is enabled. This feature supports the accumulation of the following flow variables: static pressure, temperature, velocity components, turbulent kinetic energy, turbulent dissipation rate (ε), and specific dissipation rate (ω).

14.10.3.1. Fourier Coefficient Postprocessing Theory

The real representation of Fourier coefficients is used and the following $A_{i,n}$ and $B_{i,n}$ are provided in the list of variables available to postprocess. This representation allows for multiple (M) base frequencies to be captured (ω_i), each having its own harmonic content represented by n_i. The Fourier coefficient accumulation method used is based on the moving averages. Its equivalent closed period integration form is given below where the integration period T_c must be the common period between all frequencies captured.

Fourier coefficient definitions

$$A_0 = \frac{1}{T_c}\int_0^{T_c} dt\, f(t)$$
$$A_{i,n_i} = \frac{1}{T_c}\int_0^{T_c} dt\, f(t)\cos(n_i\omega_i t) \tag{14.15}$$
$$B_{i,n_i} = \frac{1}{T_c}\int_0^{T_c} dt\, f(t)\sin(n_i\omega_i t)$$

The reconstructed signal $\hat{f}(t)$ can then be written as:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\hat{f}(t)=A_0+\sum_{i=1}^{M}\left(\sum_{n=1}^{N_i}A_{i,n_i}\cos(n_i\omega_i t)+\sum_{n=1}^{N_i}B_{i,n_i}\sin(n_i\omega_i t)\right) \tag{14.16}$$

and is provided as a Fourier related variable.

Additional variables available for post-processing are as follows:

$$\text{Fourier Reconstruction Error}=\left|f-\hat{f}\right| \tag{14.17}$$

$$\text{Fourier DNorm Timestep Delta}\approx dA_0^2+\frac{1}{2}\sum_{i}^{n}(dA_i^2+dB_i^2) \tag{14.18}$$

where for example dA_i is the time step change in coefficient A_i.

14.10.3.2. Fourier Coefficient Postprocessing Pre-requisites

Utilizing General Fourier Coefficient Postprocessing requires the following pre-requisites.

1. **Turbo Model** must be enabled.

   Domain → **Turbomachinery** → **Turbo Models**

2. **Turbomachinery Description** must be complete. For details, see Turbomachinery Description (p. 2026).

3. The **Run Calculation** parameters must be specified using a **Fixed** time advancement method. It is recommended you use either the **Frequency-Based** or **Period-Based** time advancement method and it is required to use a sufficiently small timestep to run the phase-lag method. To accomplish this, you should specify the frequency or period to match the disturbance and then ensure a sufficiently high **Time Steps per Period** (such as 40). For details, see Configuring Run Calculation Settings (p. 1976).

14.10.3.3. Fourier Coefficient Postprocessing Use-Cases

There are two distinct scenarios for general Fourier coefficient postprocessing:

1. General cases which include periodic sector or full-wheel simulations for which there are three use-cases: Inlet/outlet disturbance, blade flutter, and transient-rotor-stator

   This can be used to help visualize the state of the harmonic content of any transient-periodic turbomachinery simulation.

   To make this implementation more general, separate controls are provided to define the spectral description of the postprocessing Fourier coefficients so that you can have different spectral content for each if desired.

2. Phase-lag single passage cases for which there are two use-cases: Inlet/outlet disturbance and blade flutter

   Although these cell-zone-based Fourier coefficients are not strictly the same as those used in the Phase-lag boundary condition, they provide a good approximation of their state if the same spectral content is used as in the Phase-lag boundary. If the Phase-lag boundary was previously

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

defined the first prompt when using the text command asks if the same spectral settings are to be used for the post-processing Fourier coefficients. The manual TUI method of enabling this is given below:

```
/define/turbo-models/create-graphics-spectral-content
Copy from phase lag spectral description? [yes]
```

More details on the setup of this option and on the more general user-defined methods of defining graphical spectral content are given below in Creating Graphics Spectral Content (p. 2037).

14.10.3.4. Creating Graphics Spectral Content

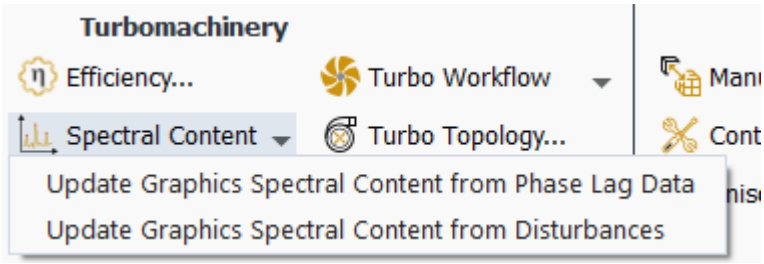
There are two methods for creating spectral content for Fourier coefficient postprocessing/

1. From the **Turbomachinery** Ribbon

   Domain → Turbomachinery → Spectral Content

   There are two entries in the drop down menu as shown in Figure 14.27: Options for Spectral Content in the Turbomachinery Ribbon (p. 2037).

   Figure 14.27: Options for Spectral Content in the Turbomachinery Ribbon

   - **Update Graphics Spectral Content from Phase Lag Data** - copies the spectral content from the phase-lag spectral content to the graphics spectral content identically. This will be the most used method to provide the spectral content for the post-processing Fourier coefficients.
   - **Update Graphics Spectral Content from Disturbances** - Derive the spectral content from the disturbances that are defined. For inlet/outlet type disturbances, the frequency is derived from the `NumberSectors360` parameters provided in any inlet or outlet boundary profile files. For details, see Inlet/Outlet Disturbance Specific Phase-lag Method Workflow (p. 2010). For blade flutter use cases, the frequency associated to the mode-shape that is attached to the fluttering blade is used.

2. In cases that don't comply with the ribbon option, you have the option to manually create graphics spectral content using the following text command:

   `/define/turbo-model/create-graphics-spectral-content`

   This commands prompts for all the required input to specify the frequencies associated with each disturbance of the simulation in each blade row. If you chose to not use the default number of Fourier harmonics for each fundamental frequency in the specifying spectral options step, you are prompted here to list the desired harmonics for each frequency

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

in the simulation. The frequencies can be specified as disturbances where it will be computed using the input number of sectors in the 360-degree machine and a reference frame. A sample session is shown below:

```
/define/turbo-model/create-graphics-spectral-content
Add disturbances/frequencies manually? [no] yes
List of all rows: (row-1)
Enter the list of rows to add disturbances/frequencies: (1) [row-1]
Enter the number of disturbances/frequencies for row "row-1": [0] 1
For disturbance/frequency #1:

--> Enter type (1= blade passing disturbance, 2= real frequency in [Hz], 3= blade flutter disturbance): [-1
--> Enter the reference frame associated with this disturbance/frequency Available: ("rotating" "global")

Reference Frame: [] rotating

--> Enter the number of sectors in 360 deg. associated with this disturbance: [0] 19
Calculate modal influence coefficients? [no] no
```

14.10.3.5. Extra Settings

There are additional settings found under the console command `/define/turbo-model/graphics-extra-settings` for changing defaults associated with spectral content creation of the graphics/post-processing Fourier coefficients. In most cases, the default values are sufficient and need not be changed. A sample dialog is shown below.

```
/define/turbo-model> graphics-extra-settings
Fourier coefficients relaxation [1] 1
Number of initialization cycles [0] 0
Use the default number of Fourier harmonics for each fundamental frequency? [yes] yes
Enter the default number of Fourier harmonics of each fundamental frequency: [10]
```

The following options are available:

- `Fourier coefficients relaxation`

  Under-relaxation constant to be applied to Fourier coefficient accumulation. Current default is 1.0 but a lower value could be used to improve robustness if required.

- `Number of initialization cycles`

  Number of periods to run the simulation using a “periodic like boundary condition” before starting Phase-lag. Permits delaying the onset of Phase-lag to have a better initial guess for the Fourier coefficients which improves robustness at start-up.

- `Use the default number of Fourier harmonics for each fundamental frequency?`

  By default, uses the default number of harmonics for each fundamental base frequency. Alternatively, you can specify a list of harmonics for each fundamental frequency in the simulation.

- `Enter the default number of Fourier harmonics of each fundamental frequency`

  A harmonic number to be applied as default to all the base frequencies in the simulation [defaults to 10].

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the case is multi-row, you can specify a frequency selection algorithm option. This provides an easier way to select the fundamental base frequency for each row.

```
Enter the multi-row frequency selection algorithm option
(1- nearest neighbors, 2- extended neighbors, 3- user-selected)
```

1. `Nearest neighbors`

   Frequencies related to the component immediately upstream and downstream of the current component.

2. `Extended neighbors`

   All possible frequency contributions (all rotors see the frequencies from all the stators and vice-versa).

3. `User selected`

   You are prompted for the base frequency to include in each blade row.

There are also additional commands to list, append, and delete postprocessing spectral content:

`/define/turbo-model/list-graphics-spectral-content`

`/define/turbo-model/append-graphics-spectral-content`

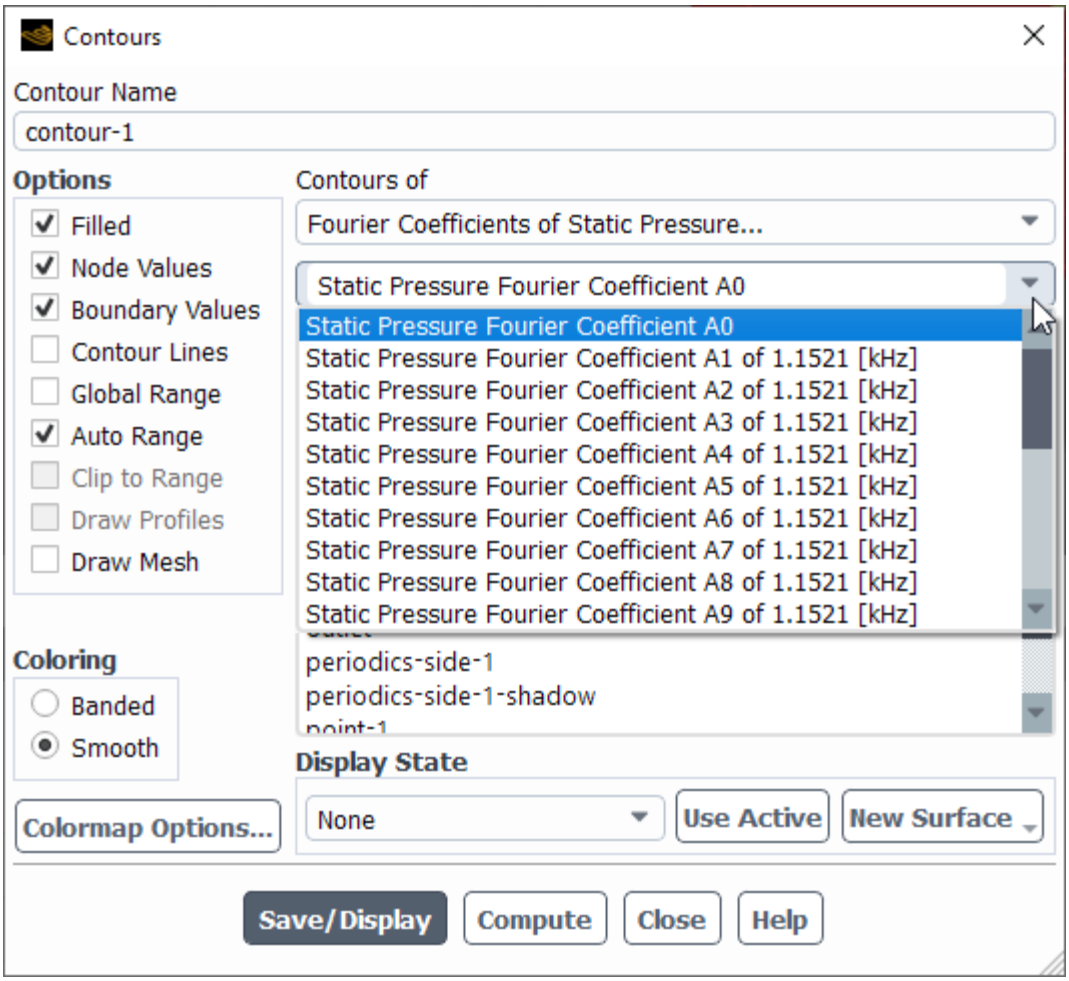
`/define/turbo-model/delete-graphics-spectral-content`

14.10.3.6. Graphical Postprocessing of Fourier Coefficients

When Fourier coefficients are defined, they will be available for postprocessing. They are grouped by flow variable and labeled to reflect the mode and frequency.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 14.28: Fourier Coefficients Available for Graphical Postprocessing



14.10.4. Circumferential-Averaged Profile Extraction

With **Turbo Models** enabled, you have the option to write circumferential-averaged profiles. Circumferential-averaged profiles are available for flow variables on various surfaces including inlet, outlet, interfaces, planes, iso-surfaces, and iso-clips. The determination of the profile type (radial or axial) is internally based on the specific type of machine. For more details, see Writing Circumferential-Averaged Profiles (p. 967).

14.10.5. Turbo Post

After the turbo topology has been defined, as described in Defining the Turbomachinery Topology (p. 2029), you can access various **Turbo Post** applications.

Results → Model Specific → Turbo Post

The following topics are discussed:

14.10.5.1. Generating Reports of Turbomachinery Data
14.10.5.2. Displaying Turbomachinery Averaged Contours
14.10.5.3. Displaying Turbomachinery 2D Contours

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.10.5.4. Generating Averaged XY Plots of Turbomachinery Solution Data
14.10.5.5. Globally Setting the Turbomachinery Topology

14.10.5.1. Generating Reports of Turbomachinery Data

Once you have defined your turbomachinery topologies, as described in Defining the Turbomachinery Topology (p. 2029), you can report a number of turbomachinery quantities, including mass flow, swirl number, torque, and efficiencies.

To report turbomachinery quantities in Ansys Fluent, you will use the Turbo Report Dialog Box (p. 5641) (Figure 14.29: The Turbo Report Dialog Box (p. 2041)).

Results → Model Specific → Turbo Post → Report...

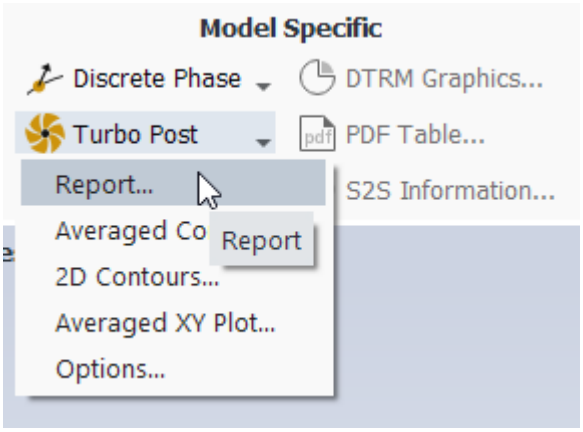
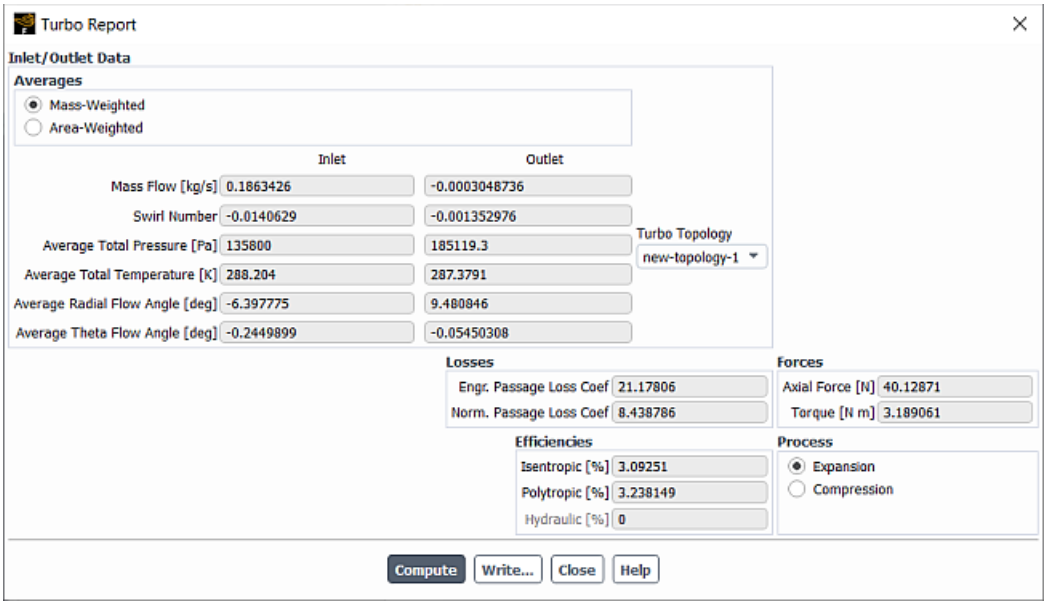


Figure 14.29: The Turbo Report Dialog Box



The procedure for using this dialog box is as follows:

1. Under **Averages**, specify whether you want to report **Mass-Weighted** or **Area-Weighted** averages.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Under **Turbo Topology**, specify a predefined turbomachinery topology from the drop-down list.

3. Under **Process**, specify whether to compute the efficiency for the flow **Expansion** (for example, in a turbine) or **Compression** (for example, in a pump or compressor).

4. Click **Compute**. Ansys Fluent will compute the turbomachinery quantities as described below, and display their values.

5. If you want to save the reported values to a file, click **Write...** and specify a name for the file in The Select File Dialog Box (p. 931).

14.10.5.1.1. Computing Turbomachinery Quantities

14.10.5.1.1.1. Mass Flow

The mass flow rate through a surface is defined as follows:

$$\dot{m}=\int_A \left(\rho\vec{v}\cdot\hat{n}\right)dA \tag{14.19}$$

where

A = is the area of the inlet or outlet

$\vec{v}$ = the velocity vector

ρ = the fluid density

$\hat{n}$ = a unit vector normal to the surface

14.10.5.1.1.2. Swirl Number

The swirl number is defined as follows:

$$SW=\frac{\int_S rv_\theta\left(\vec{v}\cdot\hat{n}\right)\,dS}{\overline{r}\int_S v_z\left(\vec{v}\cdot\hat{n}\right)\,dS} \tag{14.20}$$

where

r = the radial coordinate (specifically, the radial distance from the axis of rotation)

v_θ = the tangential velocity

$\vec{v}$ = the velocity vector

$\hat{n}$ = a unit vector normal to the surface,

S = the inlet or outlet

$\overline{r}=\frac{1}{S}\int_S rdS$

14.10.5.1.1.3. Average Total Pressure

The area-averaged total pressure is defined as follows:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\overline{p}_t = \frac{\int_A p_t dA}{A} \tag{14.21}$$

where p_t is the total pressure and A is the area of the inlet or outlet.

The mass-averaged total pressure is defined as follows:

$$\overline{p}_t = \frac{\int_A \left(\rho p_t |\vec{v} \cdot \hat{n}|\right) dA}{\int_A \left(\rho |\vec{v} \cdot \hat{n}|\right) dA} \tag{14.22}$$

where,

p_t = the total pressure
A = the area of the inlet or outlet
$\vec{v}$ = the velocity vector
ρ = the fluid density
$\hat{n}$ = a unit vector normal to the surface

14.10.5.1.1.4. Average Total Temperature

The area-averaged total temperature is defined as follows:

$$\overline{T}_t = \frac{\int_A T_t dA}{A} \tag{14.23}$$

where T_t is the total temperature and A is the area of the inlet or outlet.

The mass-averaged total temperature is defined as follows:

$$\overline{T}_t = \frac{\int_A \left(\rho T_t |\vec{v} \cdot \hat{n}|\right) dA}{\int_A \left(\rho |\vec{v} \cdot \hat{n}|\right) dA} \tag{14.24}$$

where,

T_t = the total temperature
A = the area of the inlet or outlet
$\vec{v}$ = the velocity vector
ρ = the fluid density
$\hat{n}$ = a unit vector normal to the surface

14.10.5.1.1.5. Average Flow Angles

The area-averaged flow angles are defined as follows:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\overline{\alpha}_r = \tan^{-1}\left(\frac{\int_A v_\theta dA}{\int_A v_z dA}\right) \tag{14.25}$$

in the radial direction, and

$$\overline{\alpha}_\theta = \tan^{-1}\left(\frac{\int_A v_r dA}{\int_A v_z dA}\right) \tag{14.26}$$

in the tangential direction, where v_z, v_r, and v_θ represent the axial, radial, and tangential velocities, respectively.

The mass-averaged flow angles are defined as follows:

$$\overline{\alpha}_{r,m} = \tan^{-1}\left(\frac{\int_A (\rho v_r) dA}{\int_A (\rho v_z) dA}\right) \tag{14.27}$$

in the radial direction, and

$$\overline{\alpha}_{\theta,m} = \tan^{-1}\left(\frac{\int_A (\rho v_\theta) dA}{\int_A (\rho v_z) dA}\right) \tag{14.28}$$

in the tangential direction.

14.10.5.1.1.6. Passage Loss Coefficient

The engineering loss coefficient is defined as follows:

$$K_L = \frac{\overline{p}_{t,i} - \overline{p}_{t,o}}{\frac{1}{2}\rho \overline{v}_i^2} \tag{14.29}$$

where,

$\overline{p}_{t,i}$ = the mass-averaged total pressure at the inlet

$\overline{p}_{t,o}$ = the mass-averaged total pressure at the outlet

ρ = the density of the fluid

$\overline{v}_i$ = the mass-averaged velocity magnitude at the inlet

The normalized loss coefficient is defined as follows:

$$K_{L,n} = \frac{\overline{p}_{t,i} - \overline{p}_{t,o}}{\overline{p}_{t,i} - \overline{p}_{s,o}} \tag{14.30}$$

where $\overline{p}_{s,o}$ is the mass-averaged static pressure at the outlet.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14.10.5.1.1.7. Axial Force

The axial force on the rotating parts is defined as follows:

$$F_a = \left(\int_S \left(\bar{\bar{\tau}} \cdot \hat{n} \right) dS \right) \cdot \hat{a} \tag{14.31}$$

where,

S = the surfaces comprising all rotating parts

$\bar{\bar{\tau}}$ = the total stress tensor (pressure and viscous stresses)

$\hat{n}$ = a unit vector normal to the surface

$\hat{a}$ = a unit vector parallel to the axis of rotation

14.10.5.1.1.8. Torque

The torque on the rotating parts is defined as follows:

$$T = \left(\int_S \left(\vec{r} \times \left(\bar{\bar{\tau}} \cdot \hat{n} \right) \right) dS \right) \cdot \hat{a} \tag{14.32}$$

where,

S = the surfaces comprising all rotating parts

$\bar{\bar{\tau}}$ = the total stress tensor

$\hat{n}$ = a unit vector normal to the surface

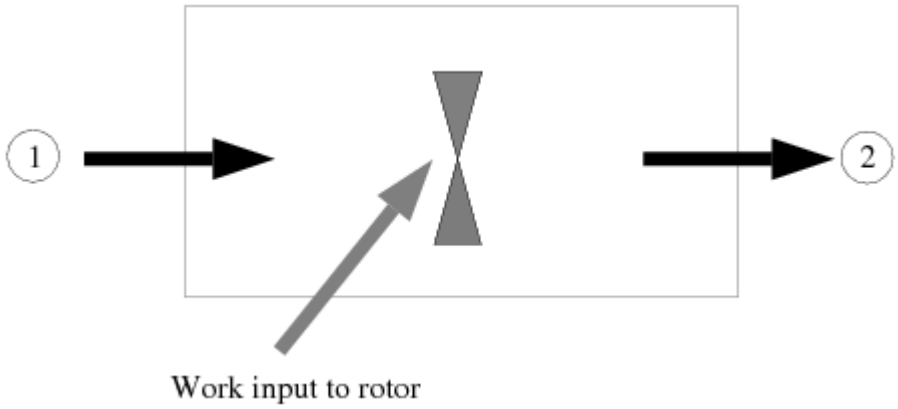
$\vec{r}$ = the position vector

$\hat{a}$ = a unit vector parallel to the axis of rotation

14.10.5.1.1.9. Efficiencies for Pumps and Compressors

The definitions of the efficiencies for compressible and incompressible flows in pumps and compressors are described in this section. Efficiencies for turbines are described later in this section. Consider a pumping or compression device operating between states 1 and 2 as illustrated in Figure 14.30: Pump or Compressor (p. 2045). Work input to the device is required to achieve a specified compression of the working fluid.

Figure 14.30: Pump or Compressor



Assuming that the processes are steady state, steady flow, and that the mass flow rates are equal at the inlet and outlet of the device (no film cooling, bleed air removal, and so on), the efficiencies for incompressible and compressible flows are as described in the following subsections.

14.10.5.1.1.9.1. Incompressible Flows

For devices such as liquid pumps and fans (2D boundaries or 3D cell zones) at low speeds, the working fluid can be treated as incompressible. The efficiency of a pumping process with an incompressible working fluid is defined as the ratio of the head rise achieved by the fluid to the power supplied to the rotor/impeller. This can be expressed as follows:

$$\eta = \frac{Q\left(p_{t2} - p_{t1}\right)}{T\omega} \tag{14.33}$$

where,

Q = volumetric flow rate

p_t = total pressure

T = net torque acting on the rotor/impeller

ω = rotational speed

This definition is sometimes called the "hydraulic efficiency". Often, other efficiencies are included to account for flow leakage (volumetric efficiency) and mechanical losses along the transmission system between the rotor and the machine providing the power for the rotor/impeller (mechanical efficiency). Incorporating these losses then yields a total efficiency for the system.

14.10.5.1.1.9.2. Compressible Flows

For gas compressors that operate at high speeds and high pressure ratios, the compressibility of the working fluid must be taken into account. The efficiency of a compression process with a compressible working fluid is defined as the ratio of the work required for an ideal (reversible) compression process to the actual work input. This assumes the compression process occurs between states 1 and 2 *for a given pressure ratio*. In most cases, the pressure ratio is the *total pressure at state 2* divided by the *total pressure at state 1*. If the process is also adiabatic, then the ideal state at 2 is the *isentropic* state.

From the foregoing definition, the efficiency for an adiabatic compression process can be written as

$$\eta_c = \frac{h_{t2,i} - h_{t1}}{h_{t2} - h_{t1}} \tag{14.34}$$

where,

h_{t1} = total enthalpy at 1

h_{t2} = actual total enthalpy at 2

$h_{t2,i}$ = isentropic total enthalpy at 2

If the specific heat is constant, Equation 14.34 (p. 2046) can also be expressed as

$$\eta_c = \frac{T_{t2,i} - T_{t1}}{T_{t2} - T_{t1}} \tag{14.35}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where,

T_{t1} = total temperature at 1

T_{t2} = actual total temperature at 2

$T_{t2,i}$ = isentropic total temperature at 2

Using the isentropic relation

$$\frac{T_{t2,i}}{T_{t1}} = \left(\frac{p_{t2}}{p_{t1}} \right)^{\frac{\gamma-1}{\gamma}} \tag{14.36}$$

where γ is the ratio of specific heats specified in the Reference Values Task Page (p. 5154).

The efficiency can be written in the compact form

$$\eta_c = \frac{T_{t1}\left[\left(\frac{p_{t2}}{p_{t1}} \right)^{\frac{\gamma-1}{\gamma}} - 1 \right]}{T_{t2} - T_{t1}} \tag{14.37}$$

Note that this definition requires data only for the *actual* states 1 and 2.

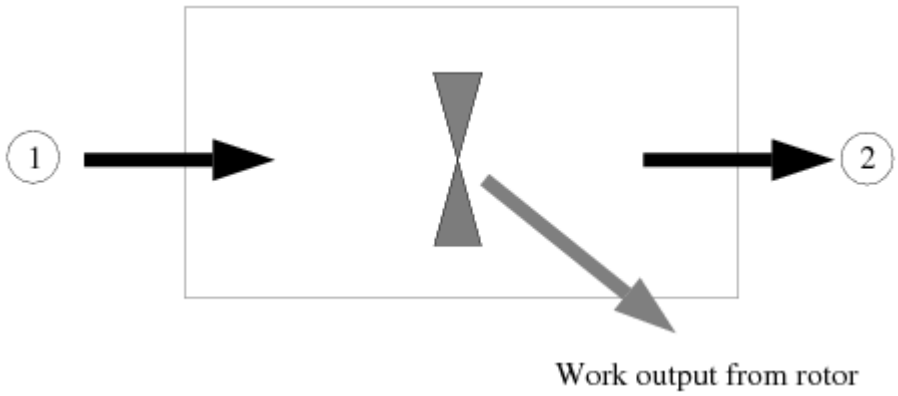
Compressor designers also make use of the polytropic efficiency when comparing one compressor with another. The polytropic efficiency is defined as follows:

$$\eta_{c,p} = \frac{\frac{\gamma-1}{\gamma} \ln\left(\frac{p_{t2}}{p_{t1}} \right)}{\ln\left(\frac{T_{t2}}{T_{t1}} \right)} \tag{14.38}$$

14.10.5.1.1.10. Efficiencies for Turbines

Consider a turbine operating between states 1 and 2 in Figure 14.31: Turbine (p. 2047). Work is extracted from the working fluid as it expands through the turbine. Assuming that the processes are steady state, steady flow, and that the mass flow rates are equal at the inlet and outlet of the device (no film cooling, bleed air removal, and so on), turbine efficiencies for incompressible and compressible flows are as described below.

Figure 14.31: Turbine



14.10.5.1.1.10.1. Incompressible Flows

The efficiency of a turbine with an incompressible working fluid is defined as the ratio of the work delivered to the rotor to the energy available from the fluid stream. This ratio can be expressed as follows:

$$\eta = \frac{T\omega}{Q\left(p_{t1} - p_{t2}\right)} \tag{14.39}$$

where,

Q = volumetric flow rate

p_t = total pressure

T = net torque acting on the rotor/impeller

ω = rotational speed

Note the similarity between this definition and the definition of incompressible compression efficiency (Equation 14.33 (p. 2046)). As with hydraulic pumps and compressors, other efficiencies (for example, volumetric and mechanical efficiencies) can be defined to account for other losses in the system.

14.10.5.1.1.10.2. Compressible Flows

For high-speed gas turbines operating at large expansion pressure ratios, compressibility must be accounted for. The efficiency of an expansion process with a compressible working fluid is defined as the ratio of the actual work extracted from the fluid to the work extracted from an ideal (reversible) process. This assumes that the expansion process occurs between states 1 and 2 *for a given pressure ratio*. In contrast to the compression process, the pressure ratio for expansion is the *total pressure at state 1* divided by the *total pressure at state 2*. If the process is also adiabatic, then the ideal state at 2 is the *isentropic* state.

From the foregoing definition, the efficiency for an adiabatic expansion process through a turbine can be written as

$$\eta_c = \frac{h_{t1} - h_{t2}}{h_{t1} - h_{t2,i}} \tag{14.40}$$

where

h_{t1} = total enthalpy at 1

h_{t2} = actual total enthalpy at 2

$h_{t2,i}$ = isentropic total enthalpy at 2

If the specific heat is constant, Equation 14.40 (p. 2048) can also be expressed as

$$\eta_e = \frac{T_{t1} - T_{t2}}{T_{t1} - T_{t2,i}} \tag{14.41}$$

where

T_{t1} = total temperature at 1

T_{t2} = actual total temperature at 2

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$T_{t2,i}$ = isentropic total temperature at 2

Using the isentropic relation

$$\frac{T_{t1}}{T_{t2,i}} = \left(\frac{p_{t1}}{p_{t2}}\right)^{\frac{\gamma-1}{\gamma}} \tag{14.42}$$

the expansion efficiency can be written in the compact form

$$\eta_e = \frac{T_{t1} - T_{t2}}{T_{t1}\left[1 - \left(\frac{p_{t2}}{p_{t1}}\right)^{\frac{\gamma-1}{\gamma}}\right]} \tag{14.43}$$

Note that this definition requires data only for the *actual* states 1 and 2.

As with compressors, one may also define a polytropic efficiency for turbines. The polytropic efficiency is defined as follows:

$$\eta_{e,p} = \frac{\ln\left(\frac{T_{t1}}{T_{t2}}\right)}{\frac{\gamma-1}{\gamma}\ln\left(\frac{p_{t1}}{p_{t2}}\right)} \tag{14.44}$$

14.10.5.2. Displaying Turbomachinery Averaged Contours

Turbo averaged contours are generated as projections of the values of a variable averaged in the circumferential direction and visualized on an r- z plane. A sample plot is shown in Figure 14.33: Turbo Averaged Filled Contours of Static Pressure (p. 2051).

14.10.5.2.1. Steps for Generating Turbomachinery Averaged Contour Plots

You can display contours using the Turbo Averaged Contours Dialog Box (p. 5637) (Figure 14.32: The Turbo Averaged Contours Dialog Box (p. 2050)).

Results → Model Specific → Turbo Post → Averaged Contours...

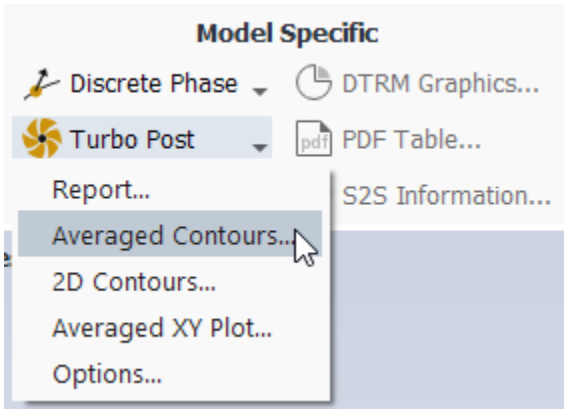
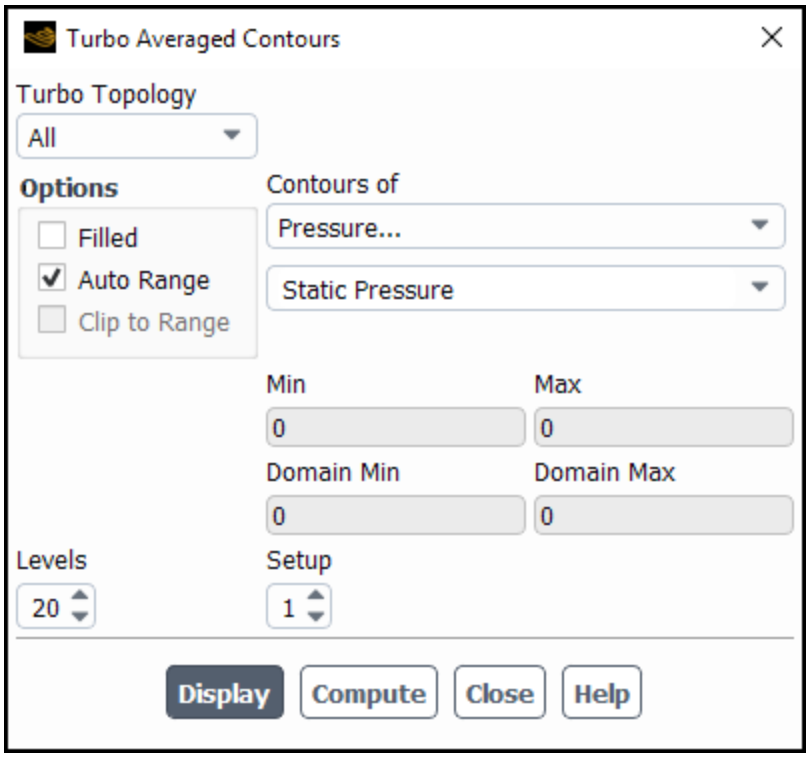


Figure 14.32: The Turbo Averaged Contours Dialog Box



The basic steps for generating a turbo averaged contour plot are as follows:

1. Select **All** or a specific predefined turbomachinery topology from the **Turbo Topology** drop-down list.

2. Select the variable or function to be displayed in the **Contours of** drop-down list. First select the desired category in the upper list; you may then select a related quantity in the lower list. (See Turbomachinery-Specific Variables (p. 2033) for a list of turbomachinery-specific variables, and see Field Function Definitions (p. 4167) for an explanation of the variables in the list.)

3. Specify the number of contours in the **Levels** field. The maximum number of levels allowed is 100.

4. Set any of the options described below.

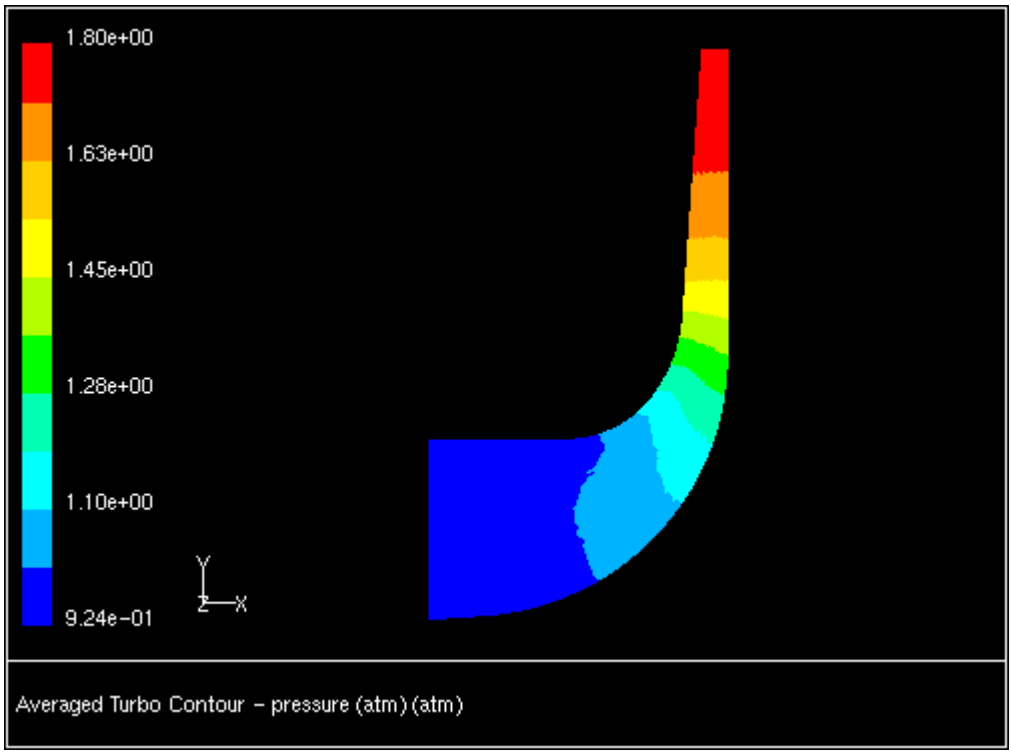
5. Click **Display** to display the specified contours in the active graphics window.

The resulting display will include the specified number of contours of the selected variable, with the magnitude on each one determined by equally incrementing between the values shown in the **Min** and **Max** fields.

Note that the **Min** and **Max** values displayed in the dialog box are the minimum and maximum averaged values. These limits will in general be different from the global **Domain Min** and **Domain Max**, which are also displayed for your reference (see Figure 14.32: The Turbo Averaged Contours Dialog Box (p. 2050)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 14.33: Turbo Averaged Filled Contours of Static Pressure



14.10.5.2.2. Contour Plot Options

The options mentioned in the procedure above include drawing color-filled contours (instead of line contours), specifying a range of values to be contoured, and storing the contour plot settings. These options are the same as those in the standard Contours Dialog Box (p. 5412). See Contour and Profile Plot Options (p. 3901) for details about using them.

14.10.5.3. Displaying Turbomachinery 2D Contours

In postprocessing a turbomachinery solution, it is often desirable to display contours on surfaces of constant spanwise coordinate, and then project these contours onto a plane. This permits easier evaluation of the contours, especially for surfaces that are highly three-dimensional.

14.10.5.3.1. Steps for Generating Turbo 2D Contour Plots

You can display contours using the Turbo 2D Contours Dialog Box (p. 5635) (Figure 14.34: The Turbo 2D Contours Dialog Box (p. 2052)).

Results → Model Specific → Turbo Post → 2D Contours...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

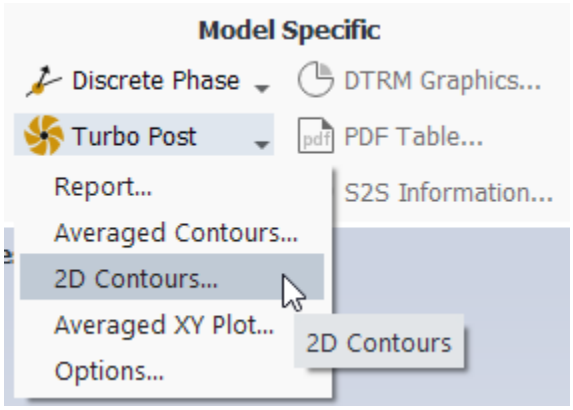
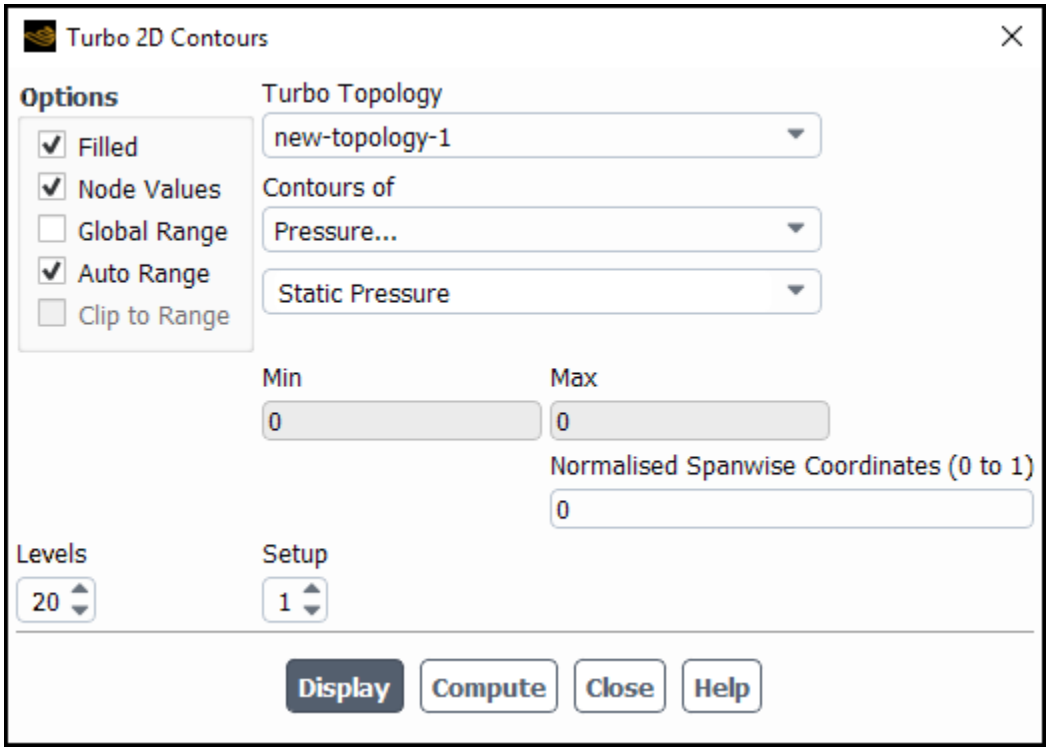


Figure 14.34: The Turbo 2D Contours Dialog Box



The basic steps for generating a turbo 2D contour plot are as follows:

1. Specify a specific predefined turbomachinery topology using the **Turbo Topology** drop-down list.

2. Enter a value for the **Normalized Spanwise Coordinates (0 to 1)** for the spanwise surface you want to create.

   Important:

   If shroud and hub are the curved surfaces, the isosurface very close to them may contain void spaces as Ansys Fluent displays only a plane cut surface.

3. Select the variable or function to be displayed in the **Contours of** drop-down list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

First select the desired category in the upper list; you may then select a related quantity in the lower list. See Turbomachinery-Specific Variables (p. 2033) for a list of turbomachinery-specific variables, and see Field Function Definitions (p. 4167) for an explanation of the variables in the list.

4. Specify the number of contours in the **Levels** field. The maximum number of levels allowed is 100.

5. Click **Display** to display the specified contours in the active graphics window.

The resulting display will include the specified number of contours of the selected variable, with the magnitude on each one determined by equally incrementing between the values shown in the **Min** and **Max** fields.

14.10.5.3.2. Contour Plot Options

Depending on the type of contour plot you want to display, select appropriate choice under **Options**. These options are the same as those in the standard Contours Dialog Box (p. 5412). See Contour and Profile Plot Options (p. 3901) for details about using them.

14.10.5.4. Generating Averaged XY Plots of Turbomachinery Solution Data

When comparing numerical solutions of turbomachinery problems to experimental data, it is often useful to plot circumferentially averaged quantities in the spanwise and meridional directions. This section describes how to do this in Ansys Fluent.

14.10.5.4.1. Steps for Generating Turbo Averaged XY Plots

To create an XY plot of circumferentially averaged solution data, you will use the Turbo Averaged XY Plot Dialog Box (p. 5639) (Figure 14.35: The Turbo Averaged XY Plot Dialog Box (p. 2054)).

Results → Model Specific → Turbo Post → Averaged XY Plot...

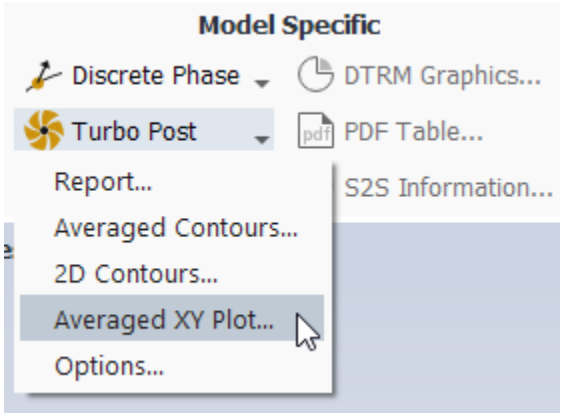
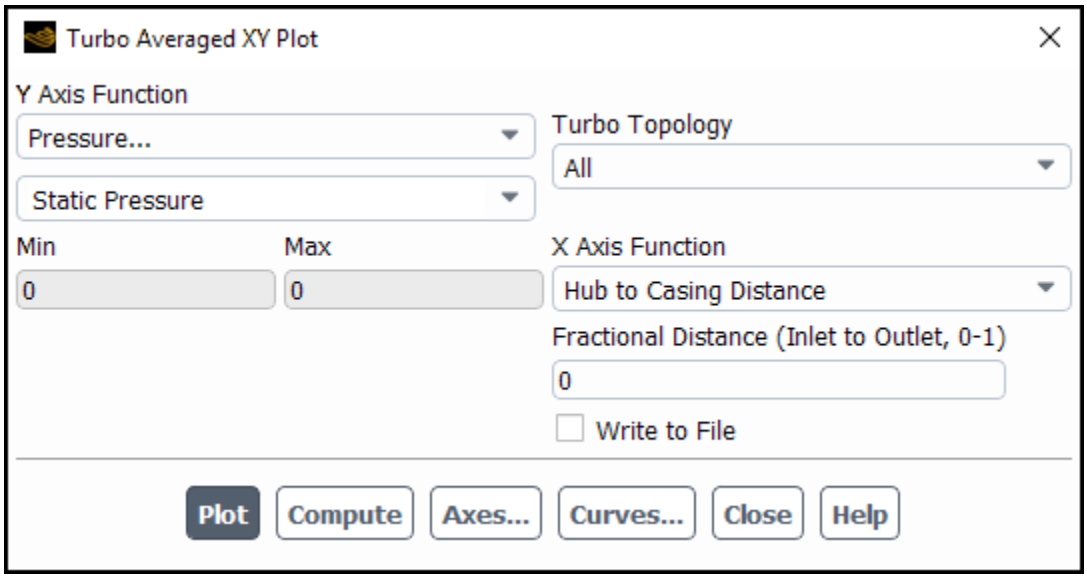


Figure 14.35: The Turbo Averaged XY Plot Dialog Box



The procedure for generating a turbo averaged XY plot are as follows:

1. Select the variable or function to be plotted in the **Y Axis Function** drop-down list. First select the desired category in the upper list; you may then select a related quantity in the lower list. (See Turbomachinery-Specific Variables (p. 2033) for a list of turbomachinery-specific variables, and see Field Function Definitions (p. 4167) for an explanation of the variables in the list.)

2. Select **All** or a specific predefined turbomachinery topology from the **Turbo Topology** drop-down list.

3. Select the variable or function to be plotted in the **X Axis Function** drop-down list. The choices are **Hub to Casing Distance** and **Meridional Distance**.

4. Specify the desired value in the **Fractional Distance** field. The definition of the fractional distance depends on your selection of **X Axis Function**:

   - If you select **Hub to Casing Distance**, the fractional distance will be **Inlet to Outlet**.
   - If you select **Meridional Distance**, the fractional distance will be **Hub to Casing**.

5. (optional) Modify the attributes of the axes or curves as described in Modifying Axis Attributes (p. 4074) and Modifying Curve Attributes (p. 4076).

6. Click **Plot** to generate the XY plot in the active graphics window.

   Note that you can use any of the mouse buttons to annotate the XY plot (see Adding Text to the Graphics Window (p. 3961)).

If you want to write the XY data to a file, follow these steps instead of Step 5 above:

1. Enable the **Write to File** option. The **Plot** button changes to a **Write...** button.

2. Click **Write...**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. In The Select File Dialog Box (p. 931), specify a name for the plot file and save it.

14.10.5.5. Globally Setting the Turbomachinery Topology

In some cases, that is, isosurface creation, Ansys Fluent enables you to globally set the current turbomachinery topology for your model using the Turbo Options Dialog Box (p. 5641) (Figure 14.36: The Turbo Options Dialog Box (p. 2055)).

Results → Model Specific → Turbo Post → Options...

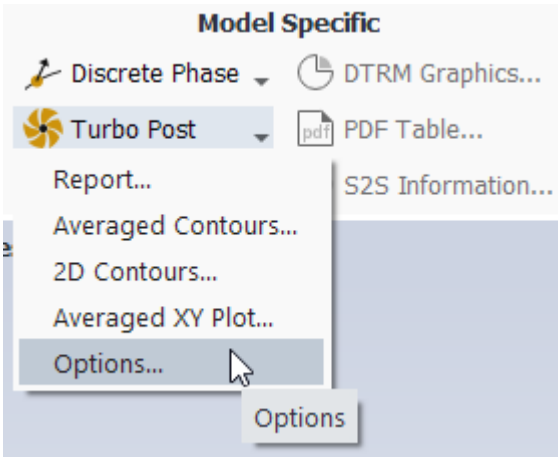
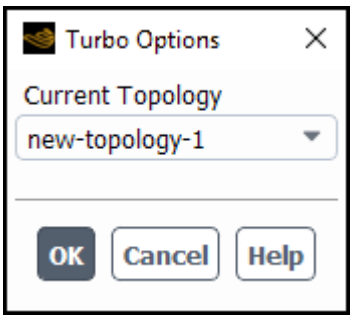


Figure 14.36: The Turbo Options Dialog Box



To set the current topology, select a topology from the **Current Topology** drop-down list and select **OK**.

14.10.6. Calculating Turbomachine Performance

An additional tool is available to calculate performance characteristics of a turbomachine. These characteristics are defined using **Named Expressions** and include efficiency, total pressure and total temperature ratios, as well as mass flow rates at the inlet and outlet.

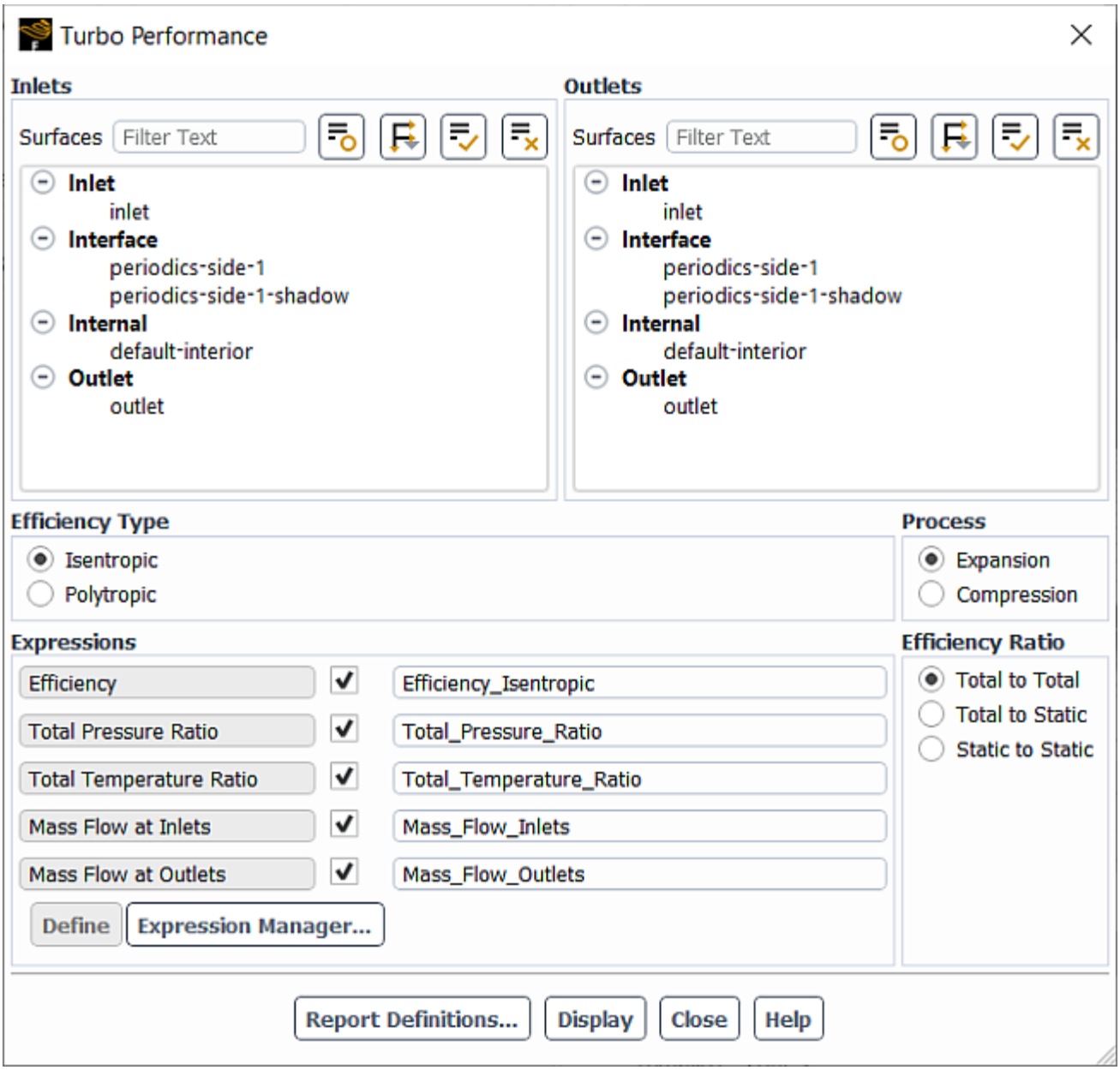
14.10.6.1. Turbo Performance

You can define and evaluate performance characteristics for a turbomachine using the **Turbo Performance** dialog box (Figure 14.37: The Turbo Performance Dialog Box (p. 2056)) accessed from:

Domain → Turbomachinery → Turbo Performance...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 14.37: The Turbo Performance Dialog Box



1. Select locations for the **Inlets** and **Outlets** using the corresponding surface lists. These lists are filtered to include only the surfaces of supported types.

2. Under **Efficiency Type**, select whether to compute the **Isentropic** or **Polytropic** efficiency.

3. Under **Process**, select whether to compute the performance characteristics for an **Expansion** or **Compression** process.

4. Under **Efficiency Ratio**, you can select which values are used when computing the turbomachine efficiency: **Total to Total**, **Total to Static**, or **Static to Static**.

5. You can define expressions for performance characteristics based on the dialog box input. Select the performance characteristics for which to define an expression. Optionally edit the default names (an integer number will be appended to each name automatically) and click

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Define to create expressions for each performance characteristic. All available expressions can be viewed and/or edited by clicking **Expression Manager...**. For more details, see Calculating Efficiency using Named Expressions (p. 2057), Calculating Total Pressure and Total Temperature Ratios using Named Expressions and Calculating Mass Flow Rates using Named Expressions.

6. Click **Report Definitions...** to open a dialog box which allows you to monitor, print, write, and/or plot the values of the expressions defined in the previous step. For more information, refer to Expression Report Definition (p. 4121).

7. Click **Display** to visualize the selected surfaces.

You can also access this tool by using the following console command and following the prompts:

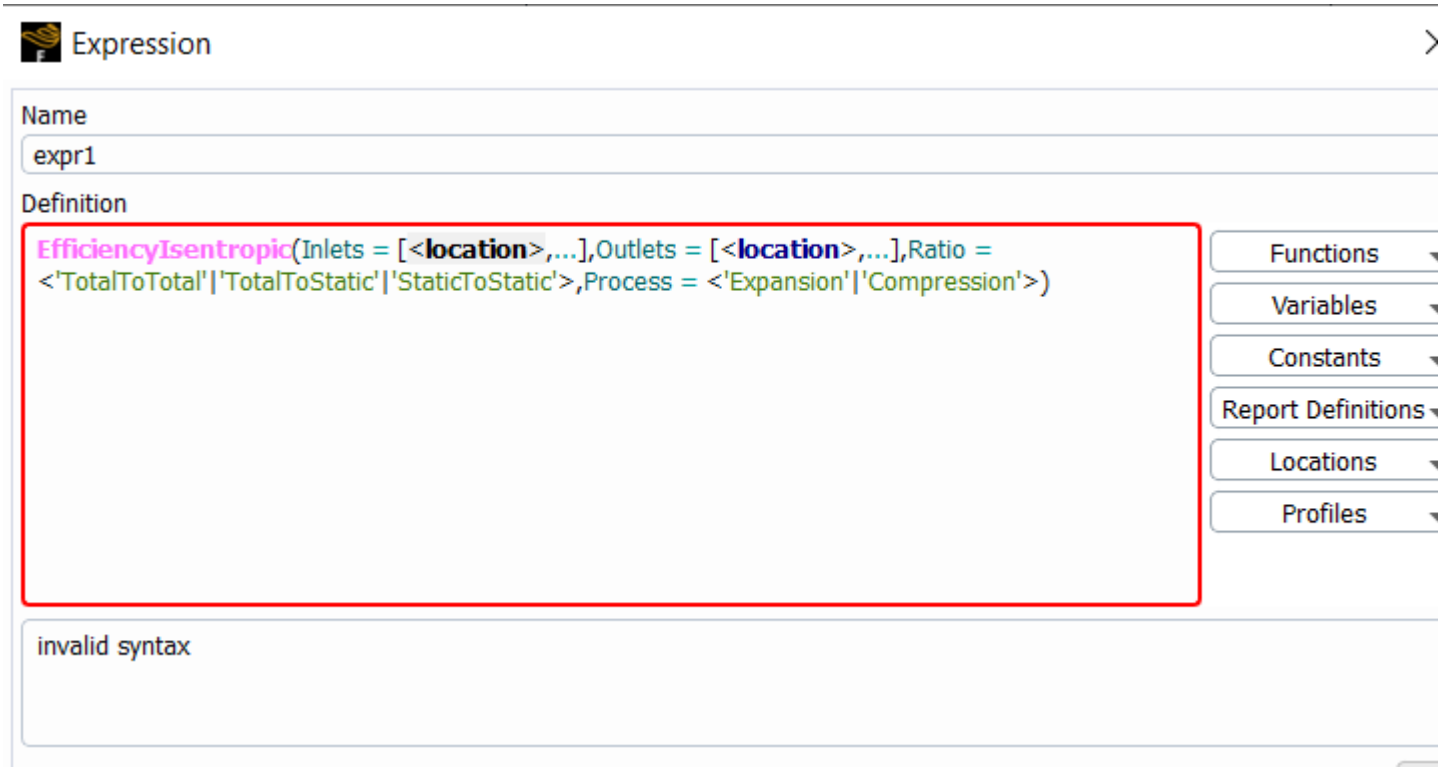
```
/define/turbo-model/performance
```

14.10.6.2. Calculating Efficiency using Named Expressions

You can compute the values of isentropic and polytropic efficiencies for a turbomachine using **Named Expressions** as shown in Figure 14.38: Expression Dialog Box with Efficiency Calculation (p. 2057).

→ **User Defined** → **User Defined** → **Named Expressions** → **New...**

Figure 14.38: Expression Dialog Box with Efficiency Calculation



Functions for both efficiency types are listed under **Functions** → **Reduction** as `EfficiencyIsentropic` and `EfficiencyPolytropic`.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Permeable boundaries of various types (for example, inlets, outlets, interfaces), as well as cut-planes and iso-surfaces, can be chosen as locations. Also, groups of surfaces for postprocessing is supported.

For the argument `Inlets`, you can select one or more locations of mixed types. However, multiple locations will still be treated as the single-stream inlet with properties averaged over all selected locations. This averaged inlet can be considered as the starting point for efficiency calculation. Similarly, the end point will be determined by locations selected for the argument `Outlets`. Options for the argument `Process(Expansion and Compression)` specify whether the flow expands or compresses when moving from the inlet to the outlet. Options for the argument `Ratio` (`TotaltoTotal`, `TotaltoStatic` and `StatictoStatic`) determines the values used in computing the efficiency.

14.10.6.2.1. Limitations

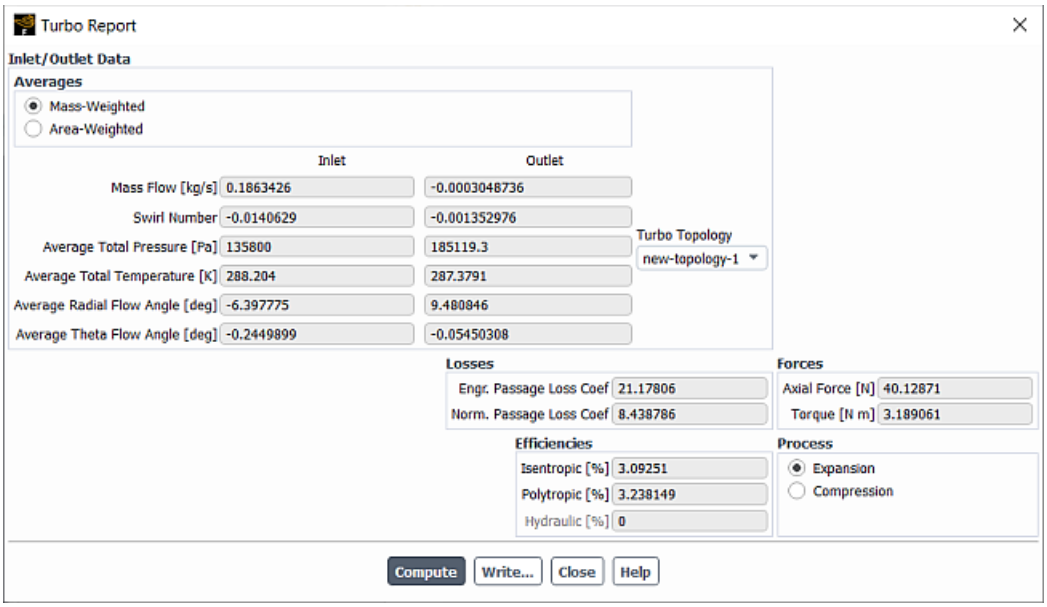
- Efficiency can be evaluated only for compressible flows of ideal gas, single-component real gas, and condensing water vapor (Wet Steam model).
- Only the following locations are supported: Pressure Inlet, Pressure Outlet, Mass-Flow Inlet, Mass-Flow Outlet, Velocity Inlet, Pressure-far-field, Inlet Vent, Outlet Vent, Intake Fan, Exhaust Fan, Interior, Interface, Cut-plane, Iso-surface, Groups of listed surfaces.
- Multiple inlet or outlet locations are reduced to a single stream with flow properties averaged over all selected.

14.10.6.3. Calculating Efficiency using Turbo Report

After you have defined Turbo Topology (as described in Defining the Turbomachinery Topology (p. 2029)), an additional option is available to compute efficiency using the **Turbo Report** dialog box as described in Generating Reports of Turbomachinery Data (p. 2041).

→ **Results** → **Model Specific** → **Turbo Post** → **Report...**

Figure 14.39: Turbo Report Dialog Box



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

By default, the **Turbo Report** dialog box computes the Total to Total values of isentropic and polytropic efficiencies assuming constant values of the specific heat capacity Cp and heat capacity ratio γ. You can switch to the more general equations which are implemented in the `EfficiencyIsentropic` and `EfficiencyPolytropic` functions, by using the following console command:

```
/report/efficiency/use-in-turbo-report?
```

Note that for an ideal gas with constant Cp value, the two methods give nearly identical results. However, for other cases, only the methods from the `EfficiencyIsentropic` and `EfficiencyPolytropic` functions are able to provide appropriate efficiency values.

14.10.6.4. Calculating Total Pressure and Total Temperature Ratios using Named Expressions

The **Turbo Performance** tool uses the `MassFlowAveAbs` function to compute the values of total pressure and total temperatures ratios. **Named Expressions** for these values are defined as follows:

```
MassFlowAveAbs(Field,['location_1'])/MassFlowAveAbs(Field,['location_2'])
```

where the first argument `Field` is `TotalPressure` or `TotalTemperature`. For the **Expansion** process, the second argument `['location_1']` is the list of inlet zones and `['location_2']` is the list of outlet zones selected in the dialog box. For the **Compression** process, the locations are reversed with `['location_1']` being the list of outlet zones and `['location_2']` being the list of inlet zones selected in the dialog box.

The `MassFlowAveAbs` function is listed under **Functions** → **Reduction** in the **Expression** dialog box. For more information on this and other functions from the Named Expressions library, refer to Fluent Expressions Language (p. 1037).

14.10.6.5. Calculating Mass Flow Rates using Named Expressions

The **Turbo Performance** tool uses the `MassFlowInt` function to compute mass flow at inlets and outlets. **Named Expressions** are defined as follows:

```
MassFlowInt(1, ['locations'])
```

The argument `['locations']` is the list of inlet or outlet zones selected in the dialog box for the definition of the corresponding expression.

The `MassFlowInt` function is listed under **Functions** → **Reduction** in the **Expression** dialog box. For more information on this and other functions from the Named Expressions library, refer to Fluent Expressions Language (p. 1037).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 15: Modeling Turbulence

This chapter provides details about how to use the turbulence models available in Ansys Fluent.

Information about turbulence modeling theory is presented in Turbulence in the Theory Guide. Information about using the turbulence models can be found in the following sections:

15.1. Introduction
15.2. Choosing a Turbulence Model
15.3. Steps in Using a Turbulence Model
15.4. Setting Up the Spalart-Allmaras Model
15.5. Setting Up the k-ε Model
15.6. Setting Up the k-ω Model
15.7. Setting Up the Transition k-kl-ω Model
15.8. Setting Up the Transition SST Model
15.9. Setting Up the Algebraic or Intermittency Transition Model
15.10. Setting Up the Reynolds Stress Model
15.11. Setting Up Scale-Adaptive Simulation (SAS) Modeling
15.12. Setting Up the Detached Eddy Simulation Model
15.13. Setting Up the Large Eddy Simulation Model
15.14. Model Constants
15.15. Setting Up the Embedded Large Eddy Simulation (ELES) Model
15.16. Setup Options for All Turbulence Modeling
15.17. Defining Turbulence Boundary Conditions
15.18. Providing an Initial Guess for k and ε (or k and ω)
15.19. Solution Strategies for Turbulent Flow Simulations
15.20. Postprocessing for Turbulent Flows

15.1. Introduction

Turbulence is the three-dimensional unsteady random motion observed in fluids at moderate to high Reynolds numbers. As technical flows are typically based on fluids of low viscosity, almost all technical flows are turbulent. Many quantities of technical interest depend on turbulence, including:

- Mixing of momentum, energy and species
- Heat transfer
- Pressure losses and efficiency

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Forces on aerodynamic bodies

While turbulence is, in principle, described by the Navier-Stokes equations, it is not feasible in most situations to resolve the wide range of scales in time and space by Direct Numerical Simulation (DNS) as the CPU requirements would by far exceed the available computing power for any foreseeable future. For this reason, averaging procedures have to be applied to the Navier-Stokes equations to filter out all, or at least, parts of the turbulent spectrum. The most widely applied averaging procedure is Reynolds-averaging (which, for all practical purposes is time-averaging) of the equations, resulting in the Reynolds-Averaged Navier-Stokes (RANS) equations. By this process, all turbulent structures are eliminated from the flow and a smooth variation of the averaged velocity and pressure fields can be obtained. However, the averaging process introduces additional unknown terms into the transport equations (Reynolds Stresses and Fluxes) that need to be provided by suitable turbulence models (turbulence closures). The quality of the simulation can depend crucially on the selected turbulence model and it is important to make the proper model choice as well as to provide a suitable numerical grid for the selected model. An alternative to RANS are Scale-Resolving Simulation (SRS) models. With SRS methods, at least a portion of the turbulent spectrum is resolved in at least a part of the flow domain. The most well-known such method is Large Eddy Simulation (LES), but many new hybrids (models between RANS and LES) are appearing. As all SRS methods require time-resolved simulations where the time step size is relatively small, it is important to understand that these methods are substantially more computationally expensive than RANS simulations.

Ansys Fluent provides the following choices of turbulence models:

- Spalart-Allmaras model
- k-ε models
  - Standard k-ε model
  - Renormalization-group (RNG) k-ε model
  - Realizable k-ε model
- k-ω models
  - Standard k-ω model
  - Baseline (BSL) k-ω model
  - Shear-stress transport (SST) k-ω model (enabled by default)
  - Generalized k-ω (GEKO) model
- v^2–f model (add-on)
- Transition k-kl-ω model
- Transition SST model
- Reynolds stress models (RSM)
  - Linear pressure-strain RSM
  - Quadratic pressure-strain RSM

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - Stress-Omega RSM
  - Stress-BSL RSM
- Scale-Adaptive Simulation (SAS) model, which can be used in combination with one of the following ω-based URANS models:
  - SST k-ω model
  - Standard k-ω model
  - BSL k-ω model
  - Transition SST model
  - ω-based Reynolds stress models (RSM)
- Detached eddy simulation (DES) model, which includes one of the following RANS models.
  - Spalart-Allmaras RANS model
  - Realizable k-ε RANS model
  - SST k-ω RANS model
  - BSL k-ω RANS model
  - Transition SST model
- Shielded Detached Eddy Simulation (SDES) model, which includes one of the following RANS models.
  - SST k-ω RANS model
  - BSL k-ω RANS model
  - Transition SST model
- Stress-Blended Eddy Simulation (SBES) model, which includes one of the following RANS models.
  - SST k-ω RANS model
  - BSL k-ω RANS model
  - Transition SST model
- Large eddy simulation (LES) model, which includes one of the following subgrid-scale models.
  - Smagorinsky-Lilly subgrid-scale model (with or without dynamic stress enabled)
  - WALE subgrid-scale model
  - Dynamic kinetic energy subgrid-scale model
  - Wall-Modeled LES (WMLES)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– Wall-Modeled LES S-Ω (WMLES S-Ω)

15.2. Choosing a Turbulence Model

It is an unfortunate fact that no single turbulence model is universally accepted as being superior for all classes of problems. The choice of turbulence model will depend on considerations such as the physics of the flow, the established practice for a specific class of problem, the level of accuracy required, the available computational resources, and the amount of time available for the simulation. To make the most appropriate choice of model for your application, you need to understand the capabilities and limitations of the various options.

The default model, k-ω SST, is adequate for most engineering flows, however, you may need to choose a different model depending on your simulation needs. You can select an alternative model as the default using the **Flow Model** drop-down list in the **Simulation** branch of **Preferences** (**File** > **Preferences...**).

Note:

The default turbulence model will be selected if you load a mesh file into the Fluent solver or if you load/create a mesh in Fluent Meshing and switch to solution mode.

It will not affect existing case files.

This section gives an overview of issues related to the turbulence models provided in Ansys Fluent. The computational effort and cost in terms of CPU time and memory of the individual models is discussed. While it is impossible to state categorically which model is best for a specific application, general guidelines are presented to help you choose the appropriate turbulence model for the flow you want to model.

Note:

For information on turbulence model compatibility with other Fluent models, see Appendix A: Ansys Fluent Model Compatibility (p. 5667).

For additional information, see the following sections:

15.2.1. Reynolds Averaged Navier-Stokes (RANS) Turbulence Models

15.2.2. Scale-Resolving Simulation (SRS) Models

15.2.3. Grid Resolution SRS Models

15.2.4. Numerics Settings for SRS Models

15.2.5. Model Hierarchy

15.2.1. Reynolds Averaged Navier-Stokes (RANS) Turbulence Models

RANS models (Reynolds (Ensemble) Averaging in the Theory Guide) offer the most economic approach for computing complex turbulent industrial flows. Typical examples of such models are the k-ε or the k-ω models in their different forms. These models simplify the problem to the solution of two additional transport equations and introduce an Eddy-Viscosity (turbulent viscosity) to compute the Reynolds Stresses. More complex RANS models are available that solve an individual equation for

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

each of the six independent Reynolds Stresses directly (Reynolds Stress Models – RSM) plus a scale equation (ε-equation or ω-equation). RANS models are suitable for many engineering applications and typically provide the level of accuracy required. Since none of the models is universal, you have to decide which model is the most suitable for a given applications.

15.2.1.1. Spalart-Allmaras One-Equation Model

The Spalart-Allmaras model is a relatively simple one-equation model that solves a modeled transport equation for the kinematic eddy (turbulent) viscosity. The Spalart-Allmaras model was designed specifically for aeronautics and aerospace applications involving wall-bounded flows and has been shown to give good results for boundary layers subjected to adverse pressure gradients. It is also gaining popularity in turbomachinery applications. Do not use the model as a general purpose model, as it is not well calibrated for free shear flows (large errors for example, jet flows).

The Spalart-Allmaras model has been extended within Ansys Fluent with a y^+-insensitive wall treatment, which automatically blends all solution variables from their viscous sublayer formulation to the corresponding logarithmic layer values depending on y^+. The blending is calibrated to also cover intermediate y^+ values in the buffer layer $(1<y^+<30)$

15.2.1.2. k-ε Models

Two-equation models are historically the most widely used turbulence models in industrial CFD. They solve two transport equations and model the Reynolds Stresses using the Eddy Viscosity approach. The standard k-ε model in Ansys Fluent falls within this class of models and has become the workhorse of practical engineering flow calculations in the time since it was proposed by Launder and Spalding [84] (p. 5718). Robustness, economy, and reasonable accuracy for a wide range of turbulent flows explain its popularity in industrial flow and heat transfer simulations.

The draw-back of some k-ε models is their insensitivity to adverse pressure gradients and boundary layer separation. They typically predict a delayed and reduced separation relative to observations. This can result in overly optimistic design evaluations for flows that separate from smooth surfaces (for example, aerodynamic bodies, diffusers). The k-ε model is therefore not widely used in external aerodynamics.

In Ansys Fluent, the use of the Realizable k-ε model is recommended relative to other variants of the k-ε family. You should use the k-ε model in combination with either the Enhanced Wall Treatment (EWT-ε) or the Menter-Lechner near-wall treatment. For details, refer to Enhanced Wall Treatment ε-Equation (EWT-ε) or Menter-Lechner ε-Equation (ML-ε) in the Theory Guide, respectively. For cases where the flow separates under adverse pressure gradients from smooth surfaces (airfoils, and so on), k-ε models are generally not recommended.

15.2.1.3. k-ω Models

The ω-equation offers several advantages relative to the ε-equation. The most prominent one is that the equation can be integrated without additional terms through the viscous sublayer. This makes the formulation of a robust y^+-insensitive treatment relatively straightforward. Refer to y+-Insensitive Near-Wall Treatment for ω-based Turbulence Models in the Theory Guide for details. Furthermore, k-ω models are typically better at predicting adverse pressure gradient boundary layer flows and separation. The downside of the standard ω-equation is a relatively strong sensitivity

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

of the solution depending on the freestream values of k and ω outside the shear layer. The use of the standard k-ω model is, for this reason, not generally recommended in Ansys Fluent.

The BSL and SST k-ω models have been designed to avoid the freestream sensitivity of the standard k-ω model, by combining elements of the ω-equation and the ε-equation. In addition, the SST model has been calibrated to accurately compute flow separation from smooth surfaces. Within the k-ω model family, it is therefore recommended to use either the BSL or SST model. These models are some of the most widely used models for aerodynamic flows. They are typically somewhat more accurate in predicting the details of the wall boundary layer characteristics than the Spalart-Allmaras model.

The BSL and SST models (like all other ω-equation based models) use a y^+-insensitive wall treatment. For details on the available options, see y+-Insensitive Near-Wall Treatment for ω-based Turbulence Models in the *Fluent Theory Guide*.

Note that the turbulence default option as of Fluent 2020 R1, the k-ω SST model, provides low levels of eddy viscosity (compared to k-ε) and hence can predict physical phenomenon such as separation more accurately, although this may make it less stable in some cases and some solver settings may need to be updated for proper convergence.

For the k-ω models, so-called low Reynolds number terms (low Re) have been proposed by Wilcox. These are available in Ansys Fluent as an option. It is important to point out that these terms are not required for integrating the equations through the viscous sublayer. Their main influence lies in mimicking laminar-turbulent transition processes. However, this functionality is not widely calibrated and for wall-boundary layer transition, the combination of the SST model with the Transition SST model (Transition SST Model in the Theory Guide), Intermittency Transition Model (Intermittency Transition Model in the Theory Guide), or the Transition k-kl-ω model (k-kl-ω Transition Model in the Theory Guide) is more reliable. The use of the low-Re terms is therefore not encouraged.

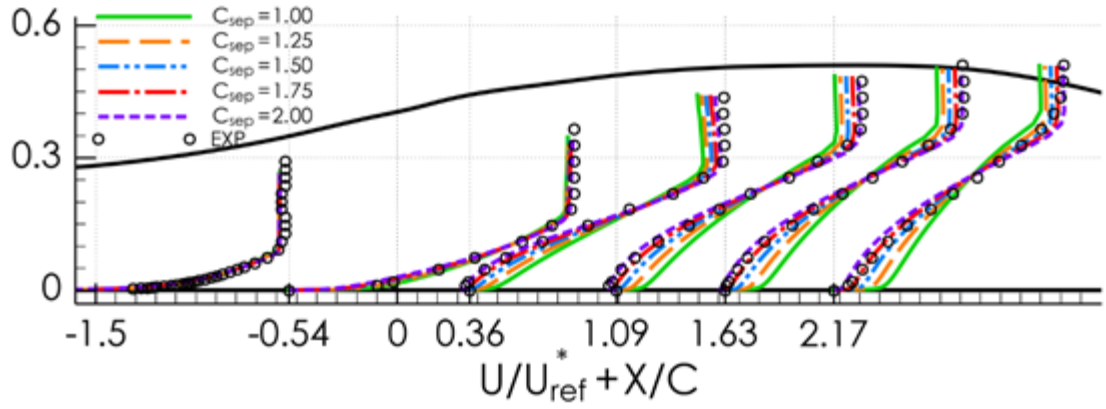
15.2.1.4. Generalized k-ω (GEKO) Model

The goal of the GEKO model is to offer a single model with enough flexibility to cover a wide range of applications. Although most applications are already covered by the default settings, the model provides four free parameters that can be adjusted for specific types of applications, without negative impact on the basic calibration of the model. This is a powerful tool for model optimization, but requires a proper understanding of the impact of these coefficients to avoid mistuning. It is important to emphasize that the model has strong defaults, so you can also apply the model without any fine-tuning and that you should make sure that any tuning is supported by high-quality experimental data.

In most cases, you need to only adjust C_{SEP} to change the GEKO model's sensitivity to boundary layer separation. The figure below shows the impact of this parameter for a diffuser flow [39] (p. 5715). With C_{SEP}=1, the model behaves essentially the same as the k–ε model (no separation) and with C_{SEP}=1.75, close to perfect agreement with the data is achieved (like the SST model [99] (p. 5718)). Larger values for C_{SEP} can be required (for example, when simulating high-lift airfoils, where even the SST model tends to under-estimate separation strength).

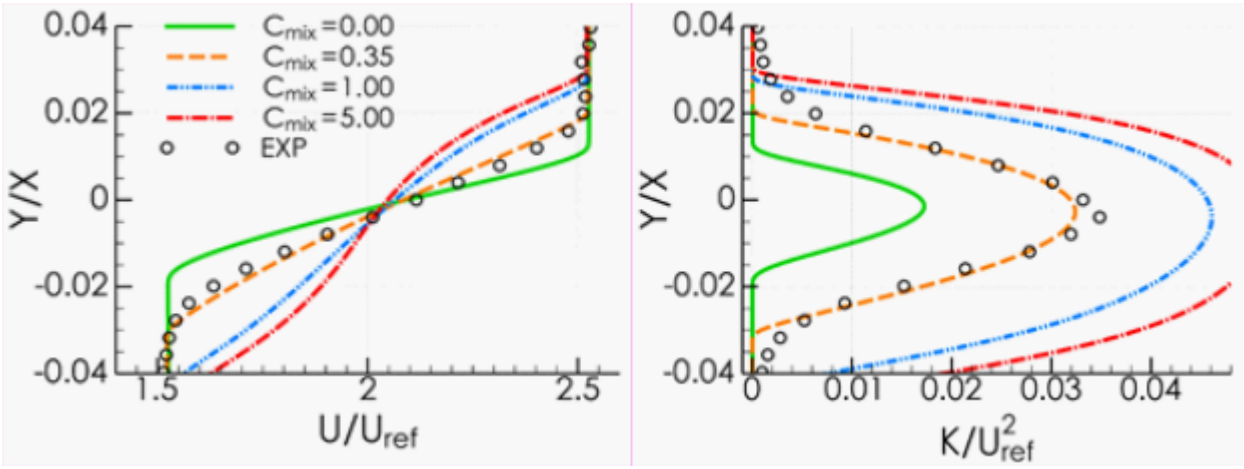
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.1: Velocity Profiles for Axi-symmetric Diffuser Flow (Case CS0 – Driver). Impact of Variation of C_{SEP}



The figure below shows variations of C_{MIX} for a free mixing layer (C_{SEP}=2.0) [18] (p. 5714). Increasing C_{MIX} leads to increased turbulence levels and larger spreading rates.

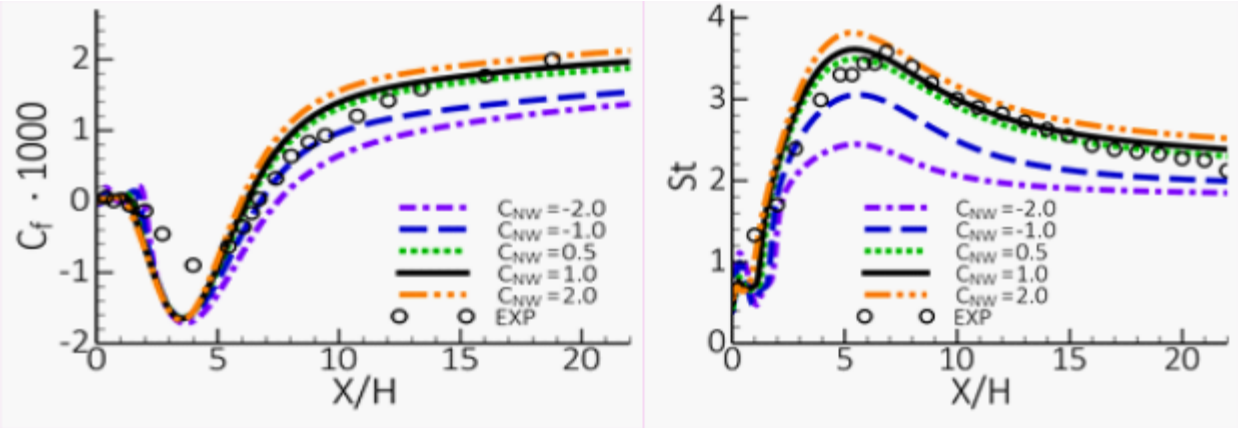
Figure 15.2: Impact of Changes in C_{MIX} on Free Mixing Layer. Left: Velocity Profiles, Right: Turbulence Kinetic Energy Profiles



The impact of C_{NW} is shown in the figure below for the reattachment region of a backward-facing step [166] (p. 5722). Both the wall shear stress coefficient C_f and the Stanton number for heat transfer, S_t, are strongly affected. Increasing C_{NW} leads to higher C_f and S_t numbers in this non-equilibrium region. In most cases, there is no need to change C_{NW} from its default value.

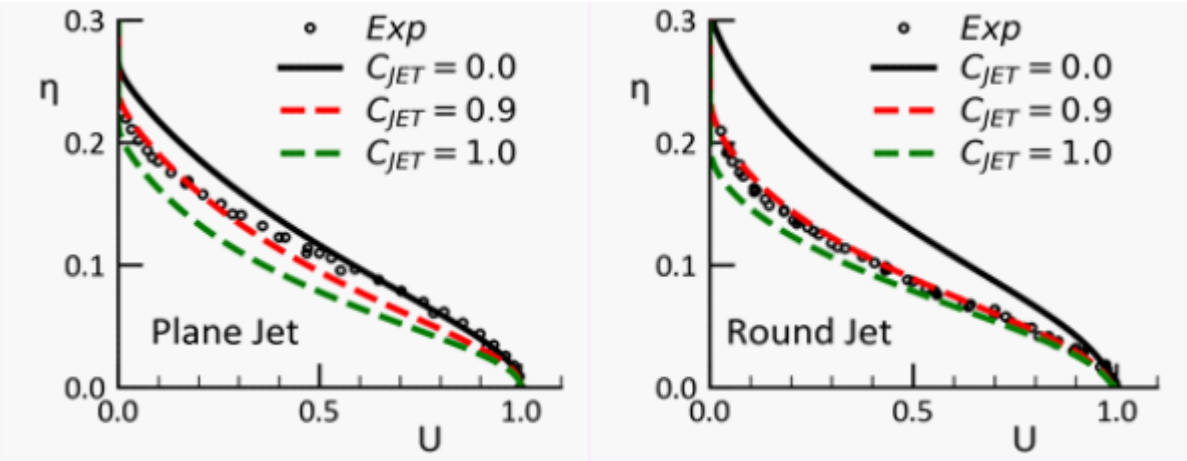
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.3: Impact of Changes in C_{NW} on Backward Facing Jet with Heat Transfer. Left: Wall Shear Stress Coefficient, C_f, Right: Wall Heat Transfer Coefficient, S_t



The figure below shows the influence of C_{JET} for plane [23] (p. 5714) and round jet flows [174] (p. 5723)(C_{SEP}=2, C_{MIX}=0.35, C_{NW}=0.5). Increasing C_{JET} (while C_{MIX} is active) reduces the spreading rate for jet flows without changing the spreading rate of mixing layers. C_{JET} is a sub-model of C_{MIX} and therefore loses its effectiveness in cases where C_{MIX} is reduced.

Figure 15.4: Impact of Changes in C_{JET} on Free Jet Flows. Left: Plane Jet, Right: Round Jet



Note that the region where C_{MIX} is active is defined by the blending function F_{Blend}. In regions where F_{Blend}=1 (near walls) there is no effect when changing C_{MIX}. F_{Blend} can be visualized in Postprocessing. It can also be augmented by the modification of expert parameters (see Generalized k-ω (GEKO) Model in the *Fluent Theory Guide*) or by over-writing the function via a UDF (see DEFINE_KW_GEKO Coefficients and Blending Function in the *Fluent Customization Manual*).

The GEKO model (like all other ω-equation based models) uses a y^+-insensitive wall treatment. For details on the available options, see y+-Insensitive Near-Wall Treatment for ω-based Turbulence Models in the *Fluent Theory Guide*.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15.2.1.5. Reynold Stress Models

Reynolds Stress models (RSM) include several effects that are not easily handled by Eddy-Viscosity models. The most important effect is the stabilization of turbulence due to strong rotation and streamline curvature (as observed for example, in cyclone flows). RSM on the other hand often demand a significant increase in computing time partly due to the additional equations but mostly due to reduced convergence. This additional effort is not always justified by increased accuracy. Their usage is therefore not generally recommended and should be restricted to flows for which their superiority has been established, especially flow with strong swirl and rotation. If wall boundary layers are important, the combination of RSM with the ω- or BSL-equation is more accurate than the combination with the ε-equation. The combination of RSM with BSL removes the free-stream sensitivity observed with the ω-equation in the same way as for two-equation models.

Ansys Fluent also offers an algebraic Reynolds Stress Model (EARSM) which represents an extension of the standard two-equation models and is therefore a good compromise between capturing additional flow effects and computational costs.

15.2.1.6. Laminar-Turbulent Transition Models

The following four models for transition prediction are available in Ansys Fluent. Three of them can be combined with the scale-resolving methods described in Scale-Resolving Simulation (SRS) Models (p. 2073).

- Transition SST model (also known as the γ-Re_θ model)

  This model can be combined with Scale-Adaptive Simulation, Detached Eddy Simulation, Stress-Blended Eddy Simulation, and Shielded Detached Eddy Simulation.

- Intermittency Transition model (also known as the γ model)

  It is available for BSL, SST, GEKO, Scale-Adaptive Simulation with BSL / SST / GEKO, Detached Eddy Simulation with BSL / SST / GEKO, Stress-Blended Eddy Simulation with BSL / SST / GEKO, and Shielded Detached Eddy Simulation with BSL / SST / GEKO.

- Algebraic Intermittency Transition Model (also known as the Alg-γ model)

  It is available for BSL, SST, GEKO, Scale-Adaptive Simulation with BSL / SST / GEKO, Detached Eddy Simulation with BSL / SST / GEKO, Stress-Blended Eddy Simulation with BSL / SST / GEKO, and Shielded Detached Eddy Simulation with BSL / SST / GEKO.

- Transition k-kl-ω model

For many test cases, the four models produce similar results. Due to their combination with the SST model, the Transition SST model, the Intermittency Transition model, and the Alg-γ model are recommended over the Transition k-kl-ω model. The Transition SST model is not Galilean invariant, and should therefore not be applied to surfaces that move relative to the coordinate system for which the velocity field is computed; furthermore, it is not suitable for fully developed pipe / channel flows where no freestream is present. For such cases, the Intermittency Transition model or the Alg-γ should be used instead (though you may need to adjust the Intermittency Transition model for pipe / channel flows by modifying the underlying correlations). Among the availble models, only the Intermittency Transition model is capable of accounting for crossflow instability.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The first three models belong to the class of "Local-Correlation based Transition Models" (LCTM). They model the transitional processes through inclusion of transition correlations. The performance of the models is similar for the generic test cases used in their calibration. However, since laminar-turbulent transition is a complex and highly sensitive process, the models can produce different results for complex cases.

The Transition SST model (γ-Re_θ model) solves two additional transport equations (in addition to the underlying turbulence model). The Intermittency Transition model solves only one additional transport equation and the Alg-γ model solves none. Avoiding additional transport equations saves computational effort and was the main motivation for the development of these models.

The Transition SST model is the most widely used model, especially in Aeronautics, where it has become a standard RANS transition model in the last decade. However, the Intermittency Transition model has also gained acceptance due to its simpler formulation and its Galilean invariance. The Alg-γ model is the latest addition and is less established. However, it has also shown strength in industrial simulations where transition is often defined by geometric features and/or laminar bubbles. This model is also Galilean invariant and can be applied in any coordinate system. The optimal choice depends on the application and it is recommended to run cases with all three models and compare which model provides the best ratio of computational cost versus accuracy.

The Transition k-kl-ω model is not being actively developed but has also found useful applications. Since it is only combined with the standard k-ω model it is not recommended as the default model.

When using these models, note the following:

- These models are only applicable to wall-bounded flows. Like all other engineering transition models, they are not applicable to transition in free shear flows. They will predict free shear flows as fully turbulent.
- These models have not been calibrated in combination with other physical effects that affect the source terms of the turbulence model, such as:
  - buoyancy
  - multiphase turbulence
- No special calibration has been performed for the combination of the Transition SST, Intermittency Transition, or Algebraic Transition models with scale-resolving methods.

  Note that the activation of the LES term in the freestream in any hybrid RANS-LES model can affect the decay of the freestream turbulence, which in turn affects the transition location.
- If you plan to include rough walls, it is recommended that you use the γ-Re_θ transition model (that is, the Transition SST model).
- Proper mesh refinement and specification of inlet turbulence levels is crucial for accurate transition prediction.
- Transition models are not recommended with tet/prism meshes.
- In general, there is some additional effort required during the mesh generation phase because a low-Re mesh with sufficient streamwise resolution is needed to accurately resolve the transition

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

region. The expansion factor in wall-normal direction should be below 1.15 to ensure sufficient resolution in the center of the boundary layer. Furthermore, in regions where laminar separation occurs, additional mesh refinement is necessary in order to properly capture the rapid transition due to the separation bubble.

- The decay of turbulence from the inlet to the leading edge of the device should always be estimated before running a solution as this can have a large effect on the predicted transition location. Physically correct values for the turbulence intensity (Tu) should be achieved near the location of transition. This is consistent with physics, as transition is highly dependent on the freestream turbulence levels outside the boundary layer.

15.2.1.7. Curvature Correction for the Spalart-Allmaras and Two-Equation Models

One weakness of the eddy-viscosity models is that these models are insensitive to streamline curvature and system rotations, which play an important role in many turbulent flow applications. To sensitize the standard eddy-viscosity models to these curvature effects, you can use a modified turbulence production term. For more information, see Curvature Correction for the Spalart-Allmaras and Two-Equation Models in the *Fluent Theory Guide*.

15.2.1.8. Corner Flow Correction

Eddy-viscosity models have a tendency to over-predict corner flow separation zones. A simplified quadratic non-linear algebraic extension for the Spalart-Allmaras and two-equation models based on the transport equation of the specific dissipation rate ω has been implemented that drives momentum into the corner and can thus reduce corner separation effects. For more information, see Corner Flow Correction in the *Fluent Theory Guide*.

15.2.1.9. Production Limiters for Two-Equation Models

A disadvantage of standard two-equation turbulence models is the excessive generation of the turbulence energy, G_k, in the vicinity of stagnation points. In order to avoid the buildup of turbulent kinetic energy in the stagnation regions, the production term in the turbulence equations can be limited by one of the two formulations described in Production Limiters for Two-Equation Models in the *Fluent Theory Guide*.

15.2.1.10. Model Enhancements

There are many model enhancements available for turbulence models. While such enhancements can improve simulations in some cases, they can also have detrimental effects. The general recommendation is therefore to use them with caution.

As noted above, the use of low-Re terms in combination with the ω-equation is not generally recommended.

Another model enhancement that should be used with care is the compressibility effects of Sarkar ([136] (p. 5720)). It can improve the prediction of free shear layers at high Mach numbers, but has also shown a pronounced negative impact on wall boundary layers. It is therefore not generally recommended. The compressibility effects option is available in the Viscous Model Dialog Box (p. 4715) when the compressible form of the ideal gas law or the real-gas model is enabled. This option is

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

disabled by default (though this was not the case in version 14.5 and earlier) and it is recommended that you keep this option disabled for cases not involving free shear flows.

Buoyancy has a pronounced effect on turbulence (Effects of Buoyancy on Turbulence in the Theory Guide). The use of the source term in the k-equation is therefore recommended for buoyancy affected flows. The source term in the ε-equation and ω-equation is much less established and the term should be enabled with care.

15.2.1.11. Wall Treatment for RANS Models

It is recommended that you use a y^+-insensitive wall treatment for all models for which it is available (Spalart-Allmaras, ε-equation and ω-equation). It provides the most consistent wall shear stress and wall heat transfer predictions with the least sensitivity to y^+ values. (See an Overview of the Spalart-Allmaras model, Enhanced Wall Treatment ε-Equation (EWT-ε), Menter-Lechner ε-Equation (ML-ε), and y+-Insensitive Near-Wall Treatment for ω-based Turbulence Models in the Theory Guide for more information.)

When Wall Functions are used, it is necessary to avoid grids with fine near wall spacing. It is recommended that you use $y^+>30$ in the entire domain. The application of Wall Functions is, however, not generally recommended as they do not allow a systematic refinement of the near wall grid. Wall Functions are especially damaging for flows at low to medium Reynolds numbers (Re~10^4-10^6), as the assumption of an extended logarithmic layer is not valid in these cases. In case that Wall Functions are desired, the option of Scalable Wall Functions (Scalable Wall Functions in the Theory Guide) avoids the grid restrictions and can be run on fine meshes.

15.2.1.12. Grid Resolution for RANS Models

Grid generation has a strong impact on model accuracy. There are many considerations that have to be followed when generating high quality CFD grids. From a turbulence modeling standpoint, the most important one is that the relevant shear layers should be covered by at least ~10 cells normal to the layer. Below this resolution, the model will not be able to provide its calibrated performance. Especially for free shear flows, whose location is not known during grid generation, this is a requirement that is hard to achieve. Nevertheless, you should be aware that for lower resolution, the model performance can degrade.

For wall bounded flows, a structured mesh in wall-normal direction is highly recommended. The structured portion of the mesh should cover the entire boundary layer and extend beyond the boundary layer thickness to avoid restricting the growth of the boundary layer. Advanced turbulence models for wall boundary layers like the Spalart-Allmaras model and the SST model will only provide improved results to other models if a minimum of 10 or more structured (hex or prism) cells are located inside the boundary layer. In addition, one should ensure that the prism layer covers the wall boundary layer entirely. Note that these are not specific requirements of these models, but are general requirements for wall boundary layer simulations.

Both ε-based and ω-based models offer y^+-insensitive wall treatment options (see Enhanced Wall Treatment ε-Equation (EWT-ε), Menter-Lechner ε-Equation (ML-ε), and y+-Insensitive Near-Wall Treatment for ω-based Turbulence Models in the Theory Guide for more information), which make the models relatively insensitive to the y^+-value of the wall cell. Generally speaking, it is more important to ensure that the boundary layer is covered with sufficient cells, then to achieve a certain y^+ criterion. However, for simulations with high accuracy demands on the wall boundary layer (especially for heat transfer predictions), near-wall meshes with y^+~1 are recommended. When wall

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

functions are used, it is essential to avoid meshes with y^+ values lower than ~30 as the wall shear stress and the wall heat transfer can and will seriously deteriorate under such conditions. For this reason, the usage of y^+-insensitive wall treatments (to be selected for ε-equation and default for ω-equation based models) is recommended.

For transition models (see k-kl-ω Transition Model, Transition SST Model, and Intermittency Transition Model in the Theory Guide), more stringent grid resolution requirements apply than for standard RANS models, as transition modeling requires the resolution of the thin laminar boundary layer upstream of the transition location. For this reason, a low-Re mesh ($y^+<1$) with sufficient streamwise resolution is needed to accurately resolve the transition region. The expansion ratio of the grid normal to the wall should not exceed 1.1. Furthermore, in regions where laminar separation occurs, additional mesh refinement is necessary, in order to properly capture the rapid transition due to the separation bubble. Finally, the decay of turbulence from the inlet to the leading edge of the device should always be estimated before running a solution as this can have a large effect on the predicted transition location.

15.2.2. Scale-Resolving Simulation (SRS) Models

The alternative to RANS models are models that resolve at least a portion of the turbulence for at least a portion of the flow domain. Such models are generally termed 'Scale-Resolving'.

15.2.2.1. Large Eddy Simulation (LES)

The most widely known SRS modeling concept is Large Eddy Simulation (LES). It is based on the approach of resolving large turbulent structures in space and time down to the grid limit everywhere in the flow. However, while widely used in the academic community, LES had very limited impact on industrial simulations. The reason lies in the excessively high resolution requirements for wall boundary layers. Near the wall, the largest scales in the turbulent spectrum are nevertheless geometrically very small and require a very fine grid and a small time step size. In addition, unlike RANS, the grid cannot only be refined in the wall normal direction, but also must resolve turbulence in the wall parallel plane. This can only be achieved for flows at very low Reynolds number and on very small geometric scales (the extent of the LES domain cannot be much larger than 10-100 times the boundary layer thickness parallel to the wall). For this reason the use of LES is only recommended for flows where wall boundary layers are not relevant and need not be resolved or for flows where the boundary layers are laminar due to the low Reynolds number. In such cases, the most balanced LES model is the WALE model (see Wall-Adapting Local Eddy-Viscosity (WALE) Model in the Theory Guide). It offers a good compromise between model complexity and generality. It also allows computing laminar shear (boundary) layers without any model impact.

An extension to LES is Wall-Modeled LES (WMLES). It allows the LES computation of wall bounded flows at higher Reynolds number without the large increase in grid resolution required for conventional LES at high Reynolds numbers. For WMLES, the grid resolution is largely independent of the Reynolds number with respect to the grid cells required per boundary layer volume. An enhanced version of WMLES called WMLES S-Ω is also available. WMLES does not provide zero eddy-viscosity for flows with constant shear. Therefore, it does not allow the computation of transitional effects, and can produce overly large eddy-viscosities in separating shear layers. The enhanced WMLES S-Ω formulation overcomes these deficiencies. See Algebraic Wall-Modeled LES Model (WMLES) in the Theory Guide for further details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15.2.2.2. Hybrid RANS-LES Models

In order to avoid the high resolution requirements of LES, numerous hybrid models have been developed in recent years. These are models that combine certain elements of RANS and LES approaches in a way that allows for the simulation of high Reynolds number flows. With hybrid models, the attached wall boundary layers are typically covered by the RANS part of the model, while large detached regions are handled in 'LES' mode, meaning with a partial resolution of the turbulent spectrum in space and time. Hybrid models rely on a strong enough flow instability to generate turbulent structures in the separated zone. This is typically the case for flows behind bluff bodies, where URANS models predict single-mode periodic vortex shedding. Hybrid models allow these vortices to generate smaller eddies down to the available grid limit. Figure 15.5: Illustration of SST-URANS vs. SST-SAS Models (p. 2074) shows a typical scenario: While the application of a standard RANS model in unsteady mode results in a single frequency vortex shedding (left), the application of hybrid models allows a break-up of the large structures into smaller scales. This is beneficial for predicting the correct mixing behind the body or to extract spectral information (for example, for acoustic simulations). At the same time, the wall boundary layers are covered by the RANS part of the hybrid model avoiding the excessive resolution requirements of LES.

Figure 15.5: Illustration of SST-URANS vs. SST-SAS Models

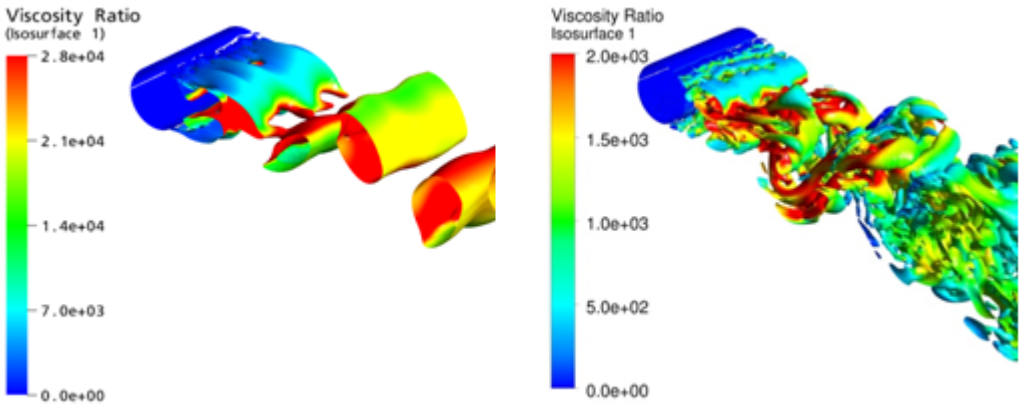


Figure 15.5: Illustration of SST-URANS vs. SST-SAS Models (p. 2074) shows a circular cylinder in a cross flow at $\mathrm{Re}=3.6\times10^6$. The left hand side illustrates the SST-URANS model, while the right-hand side illustrates the SST-SAS model. The isosurface of $Q=S^2-\Omega^2$ is colored according to the eddy viscosity ratio μ_t/μ (note that the scale in the right-hand side figure is smaller by a factor of 14).

15.2.2.2.1. Scale-Adaptive Simulation (SAS)

The SAS modeling approach (see Scale-Adaptive Simulation (SAS) Model in the Theory Guide) as proposed by Menter et al. ([100] (p. 5718) [101] (p. 5718)) is based on the introduction of the von Karman length scale, L_{vK}, into the turbulence equations (for the BSL and SST models, it enters into the ω-equation). L_{vK} is defined as the ratio of the first divided by the second derivative of the velocity vector (times the von Karman constant κ=0.41):

$$L_{vK}=\kappa\left|\frac{U'}{U''}\right|; \qquad U''=\sqrt{\frac{\partial^2 U_i}{\partial x_k^2}\frac{\partial^2 U_i}{\partial x_j^2}}\ ;$$
$$U'=S=\sqrt{2\cdot S_{ij}S_{ij}}\ ; \quad S_{ij}=\frac{1}{2}\left(\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i}\right) \tag{15.1}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The inclusion of this term allows the model to adjust its length scale to already resolved scales in the flow and thereby provide a low enough eddy viscosity to allow the model to operate in 'LES' mode.

The SAS approach has the advantage that the RANS part of the model is unaffected by the grid spacing and will therefore not allow a deterioration of model accuracy as seen in DES in regions of refined grid but insufficient flow instability. However, in cases where the flow instability is not strong enough, the SAS will remain in RANS mode and will not produce unsteady structures. While this is often a sign that the RANS model is still reasonably capable of handling the flow, it is a limitation if unsteady information is required (for example, in acoustics). In such cases, the internal interface option of the ELES implementation can be applied to convert modeled turbulence into resolved turbulence (see Internal Interface Without LES Zone in the Theory Guide).

15.2.2.2.2. Detached Eddy Simulation (DES)

The DES model (see Detached Eddy Simulation (DES) in the Theory Guide) achieves the switch between RANS and LES by a comparison of the turbulent length scale L_t with the grid spacing $\Delta_{\max}$. The model selects the minimum of both and thereby switches between RANS and LES mode by replacing ε in the k-equation by:

$$\varepsilon = \frac{k^{3/2}}{L_t} \rightarrow \varepsilon = \frac{k^{3/2}}{\min\left(L_t, c_{DES}\Delta_{\max}\right)} \tag{15.2}$$

Once the model selects the grid spacing as the minimum, the model is operating in 'LES' mode.

The grid spacing enters explicitly into the DES model. This can affect the RANS solution in regions, where the grid is between RANS and LES resolution (so-called 'gray zones' in DES) and/or where the flow instability is not strong enough to generate LES structures. Another issue to consider with DES is the problem of 'grid-induced-separation' (GIS). It occurs if the grid for an attached wall boundary layer flow is refined to a point where the DES limiter becomes active and affects the RANS solution. However, in such situations, the flow instability is not strong enough to balance the reduced RANS content by resolved turbulence. This will typically result in an artificial flow separation at the location of grid refinement. It typically happens if $\Delta_{\max} < \delta$ (δ being the boundary layer thickness). Remedies for this situation have been proposed by Menter et al. who recommended using the F1 blending function of the SST-DES model to shield the boundary layers from the DES limiter. Later, alternative blending functions for the same purpose have been proposed by Spalart et al. ([154] (p. 5721))—resulting in the terminology Delayed DES (DDES). The DDES model as originally proposed for the Spalart-Allmaras model provided limited protection against GIS for two-equation models such as BSL, SST, and k-ε. Therefore, the DDES function has been re-calibrated for the BSL, SST, and k-ε models and is now the recommended choice and the default setting when using these models.

A further refinement is provided by the Improved DDES (IDDES) formulation of Strelets et al. ([143] (p. 5721)), which extends the LES zone of the model to the outer part of wall boundary layers. This allows the simulation of wall boundary layers in Wall-Modeled LES (WMLES) mode. In this model, the IDDES model is applied like a LES model, typically with the specification of unsteady inlet conditions. The grid resolution requirements for WMLES are much less stringent than for LES.

All of the above shielding function variants are available in Ansys Fluent. For the Spalart-Allmaras model, the original DDES shielding function is used. For the k-ε, BSL, and SST models, the DDES function has been re-calibrated for better boundary layer protection. The re-calibrated DDES

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

function is the default selection and is recommended over the use of the F1 or F2 functions for the BSL / SST model. Despite the potential difficulties in the application of hybrid methods, they have the potential to greatly expand the usage of Scale-Resolving Simulation models for engineering applications, as they avoid the excessive resolution required by LES for wall boundary layers. IDDES in WMLES mode is an advanced option and should only be used if you are familiar with the original literature and the grid requirements for this model.

The DES and DDES models in Ansys Fluent have been superseded by the SDES and SBES models described in Shielded Detached Eddy Simulation (SDES) and Stress-Blended Eddy Simulation (SBES) (p. 2076).

15.2.2.2.3. Shielded Detached Eddy Simulation (SDES) and Stress-Blended Eddy Simulation (SBES)

The Shielded Detached Eddy Simulation (SDES) model and the Stress-Blended Eddy Simulation (SBES) model are hybrid RANS-LES models with an improved shielding function compared to DDES / IDDES. They provide strong shielding of the RANS boundary layer, faster "transition" from RANS to LES in separating shear layers, and (for SBES) explicit selection of the LES model. For implementation details, see Shielded Detached Eddy Simulation (SDES) and Stress-Blended Eddy Simulation (SBES) in the Theory Guide; for setup details, see Including the SDES or SBES Model with RANS Models (p. 2140).

These models are recommended over the older DES / DDES model formulations. The SBES model is considered the optimal selection, as it offers the clearest distinction between RANS and LES zones.

15.2.2.3. Zonal Modeling and Embedded LES (ELES)

As pointed out in the previous sections, hybrid models rely on flow instabilities to generate turbulent structures in large separated regions without the explicit introduction of unsteadiness through the boundary conditions. However, there are situations, where such instabilities are not present or are not reliable to serve this purpose. In such cases, it is desirable to apply RANS and the LES models in predefined zones and provide clearly defined interfaces between them. At these interfaces, the modeled turbulent kinetic energy from the upstream RANS model is converted explicitly to resolved scales at an internal boundary to the LES zone. The LES zone can then be limited to the region of interest where unsteady results are required. ELES is available in Ansys Fluent and allows the combination of most RANS model with classical LES models. It is important to emphasize that in this mode, a full LES resolution is required within the LES zone. In the LES zone, using the WALE model is recommended (see Wall-Adapting Local Eddy-Viscosity (WALE) Model in the Theory Guide).

15.2.3. Grid Resolution SRS Models

Grid Resolution SRS models are discussed in the following sections:

15.2.3.1. Wall Boundary Layers

15.2.3.2. Free Shear Flows

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15.2.3.1. Wall Boundary Layers

For wall boundary layers, it is important to distinguish if they are computed in RANS or wall-resolved or Wall-Modeled LES (WMLES) mode. Only the IDDES and the SBES model can be run in WMLES mode reliably.

In the case of RANS mode, the requirements are the same as for any RANS model. For a wall-resolved LES, it is typically recommended to use a mesh with a grid spacing scaling with $\Delta x^+ \approx 40, \Delta y^+ \approx 1, \Delta z^+ \approx 20$ where x is the streamwise, y the wall normal and z the spanwise direction (for example, channel flow). However, in complex applications, the distinction between streamwise and spanwise direction is not feasible and then $\Delta x^+ \approx 20, \Delta y^+ \approx 1, \Delta z^+ \approx 20$ would be required. This scaling demonstrates the strong Reynolds number dependency of the LES approach for wall bounded flows.

For the IDDES and SBES models in WMLES mode, the above requirements can be relaxed. The grid spacing no longer scale with the wall friction, but with the boundary layer thickness δ. It is recommended that you use $\Delta x \approx \Delta z \approx (0.05-0.1) \cdot \delta$. The wall normal resolution should be like for a finely resolved RANS simulation, meaning a near wall resolution of $\Delta y^+ \approx 1$ and around 30 nodes inside the boundary layer.

Note that octree-based meshes (see Generating Rapid Octree Meshes (p. 828)) are close to being uniform in the streamwise, spanwise, and wall normal directions inside the boundary layer, and so are suitable for Wall-Function LES (WFLES). A minimum mesh resolution for boundary layers would be 5x5x5 cells per boundary layer volume.

15.2.3.2. Free Shear Flows

For free shear flows, it is difficult to provide general recommendations, as there are many different flow scenarios. The current recommendation is therefore based on the most common (and most frequent) free shear flow – a turbulent mixing layer. It will not necessarily apply to other free shear flows like jets and you are advised to perform tests as to the optimal resolution of your specific flow.

For free shear flows and SRS models, one should aim for uniform isotropic cells (all edges have similar length). The shear layer should be covered by ~10-20 cells.

15.2.4. Numerics Settings for SRS Models

For Scale-Resolving Simulations (SRS), specific discretization and solver settings are required to achieve optimal accuracy with minimal numerical effort. The recommendations given below should be considered as a starting point for your specific flow application and are not generic, but problem dependent. The recommendations are based on incompressible, single phase flow without chemical reactions or other complex additional physics. In case your simulation features additional complex physical effects, it requires adjusting the recommended solver settings accordingly. In most cases, this will mean that a higher effort must be invested into the coupling of the equations (for example smaller time step size, reduced under-relaxation, higher iteration count, smaller residuals), in order to avoid a decoupling of different physical phenomena.

For Scale-Resolving Simulations, optimal numerics settings are essential for achieving accurate results in an acceptable time frame. The reason is that at the SRS models operate at the resolution limit of the provided grid where the smallest scales are of the order of the grid spacing and the time resolution.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Numerics settings therefore have to be chosen to provide an optimal balance between accuracy and robustness.

It is generally recommended to initialize the solution from a (reasonably) converged RANS simulation.

15.2.4.1. Time Discretization

For Scale-Resolving Simulations, the resolution of the turbulent structures in time is essential for the success of the simulation. This is, to the largest extent, defined by the selected time step size. As the SRS model is operating at the grid limit, you should select a time step size that ensures a Courant-Friedrichs-Levy (CFL) number of

$$CFL=\frac{U\Delta t}{\Delta x}<1 \tag{15.3}$$

The CFL number is computed by the solver and can be checked based on an initial RANS simulation, for example. It is important to emphasize that this is not a numerical limit and that the solver will be able to sustain much larger CFL numbers. In complex applications, there will always be limited regions of fine cells or large velocities and you should not restrict the CFL number based on such zones. The recommendation of CFL =1 should be applied in the main SRS region in combination with a uniform isotropic grid. It is recommended that you vary the time step size for each type of application and explore its optimal value. This can substantially save on computing costs.

The time derivative should be computed by the Second Order Implicit option. (See Inputs for Time-Dependent Problems (p. 3645) for guidelines on setting solution parameters for transient calculations in general.)

15.2.4.2. Spatial Discretization

For SRS models, it is important to minimize the numerical dissipation of the scheme in order to avoid damping of the smallest scales by numerical dissipation. For the pressure-based solver, the choice for spatial discretization is between the Central Differencing (CD) scheme (see Central-Differencing Scheme in the Theory Guide) and the Bounded Central Differencing scheme (BCD) (see Bounded Central Differencing Scheme in the Theory Guide); for the density-based solver, BCD is available with the Roe-FDS or AUSM flux type for the discretization of the flow equations, but CD is not. The Central Differencing scheme is the least dissipative and provides the highest resolution accuracy for the smallest scales. Especially for aero-acoustics simulations, where the spectral content at higher frequencies can be important, this is a desirable feature. However, Central Differencing schemes are prone to solution oscillations (checker boarding) in the velocity field. When using the Central Differencing scheme, it is therefore important to provide a high quality mesh (no mesh jumps, isotropic cells and high resolution in critical zones) and to avoid a large time step size (the CFL number should be smaller than 1 in the main SRS region). It is recommended that you visually monitor the solution regularly in order to avoid wasting computational resources. In case oscillations appear, the choice is to: improve the mesh; reduce the time step size; or to switch to the slightly more dissipative, but also more robust Bounded Central Differencing scheme. In many complex applications, the Bounded Central Differencing Scheme is the numerics option of choice. It typically provides sufficiently low dissipation to allow the turbulent structures to evolve, but, at the same time, is robust enough to handle non-optimal grids as they are typically encountered in industrial simulations. In addition, the Bounded Central Differencing scheme is also suitable for hybrid methods like SAS, DES, SDES, and SBES, and will provide stable solutions in RANS regions, with highly stretched grids and with a CFL number larger than 1. For ELES, the numerical scheme in the LES zone can be selected independently from the settings in the RANS zone. The RANS region can

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

then be computed with standard higher-order upwind schemes and the LES zone can be covered by either the Central Differencing scheme or the Bounded Central Differencing scheme.

For the gradient calculation, it is recommended that you select the Least Squares Cell Based option, or the Green-Gauss Node Based option, to ensure a second-order interpolation on non-orthogonal grids.

For pressure interpolation, it is recommended that you use the second-order scheme, or the body-force-weighted scheme. Due to its higher dissipation, the PRESTO! scheme (see Pressure Interpolation Schemes in the Theory Guide) can result in a delayed formation or damping of turbulent structures. It is therefore not recommended for SRS.

15.2.4.3. Iterative Scheme

The selection of the iterative scheme will mostly affect the computational costs as the cost per iteration between these methods is rather different. However, recommendations are not straightforward, as the higher cost per iteration of a scheme can be offset by faster convergence within the time step.

The fastest scheme is the Non-Iterative Time Advancement (NITA) scheme (see Non-Iterative Time-Advancement Scheme in the Theory Guide as well as Setting Solution Controls for the Non-Iterative Solver (p. 3592)). This scheme typically works well for limited LES zones and high quality meshes. It is also important to use a small time step size that results in a CFL number below 1. For the NITA scheme, all explicit under-relaxation factors are, by default, set equal to one. With NITA, it has slight advantages to use the fractional step method if there is no involvement of more complex physics, when otherwise the PISO scheme can be more beneficial. Note that with the LES model, you can enable an accelerated time marching option, which enables a modified NITA scheme and other setting changes that can further speed up the simulation; for details, see Setting Solution Controls for the Non-Iterative Solver (p. 3592).

In case the application is too complex for the NITA scheme to provide a solution, the iterative SIMPLEC (SIMPLE vs. SIMPLEC (p. 3586)) or PISO (PISO (p. 3586)) schemes are the next possible option. They should be preferred relative to the SIMPLE scheme, as they show faster convergence per time step and can be run with more aggressive under-relaxation (higher values equal to 1 or close to 1). If such settings prevent convergence within the time step, then check if your time step size is small enough for maintaining CFL numbers below 1 in the SRS region – if not, then try reducing Δt. If this is not possible, or does not lead to satisfactory convergence, then reduce the under-relaxation factors to values between the default settings and 1. For skewed meshes or meshes with problematic quality, the reduction of explicit relaxation factor for pressure correction down to 0.7 from 1 can be very helpful.

15.2.4.3.1. Convergence Control

The convergence criterion within each time step will strongly affect solution costs, as a low criterion will result in an increased number of iterations. It is not possible to provide general recommendations, as the required residual depends on the application. The most relevant residual in the SRS models is the continuity residual. It should be converged by approximately 2 orders of magnitude per time step for a CFL number ~ 1. This should be achievable with around 5-10 maximum iterations per time step for flows involving no other physical models. Be sure to check the impact of convergence on your solution to ensure that the residual is reduced to a level consistent with your problem.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For simulations with small CFL values, the **Extrapolate Variables** option (available in the Run Calculation Task Page (p. 5207)) can be very beneficial. This can substantially reduce the number of iterations within the time step up to 40% when the convergence control is based on the convergence criteria.

In case your simulation requires the combination of additional physical models, such as combustion or multiphase, a reduction of the under-relaxation factors might be required. In this case, the number of maximum iterations per time step can be increased to achieve the desired residual reduction.

For hybrid RANS/LES simulations, such as the SAS, DES, SDES, and SBES models, there can be situations where the RANS portion of the flow limits convergence (for example, due to poor grid quality). In such cases, the application of the coupled solver should be considered. For the coupled pressure-based solver, each iteration is typically more expensive than for the SIMPLEC algorithm, but this is at least partly offset by faster convergence. The recommendations concerning residuals are similar to the SIMPLEC method, but the maximal number of iterations can be reduced to values as low as 2-5 for highly unstable flows, and 5-10 for more sensible flows and acoustics simulations.

15.2.5. Model Hierarchy

As discussed, turbulence modeling is a balance between accuracy and cost. The recommendation is to use RANS models (such as the default k-ω SST model) as much as possible and as long as they provide the accuracy required for the simulation. RANS models will remain the workhorse of turbulence modeling for many years to come. Within the RANS family, eddy-viscosity models are typically sufficient for most engineering flow simulations. The application of RSM is only recommended for flows that are known to systematically benefit from their usage and justify the increase in computing power.

In cases where steady RANS or URANS models cannot provide the accuracy or unsteady information required, it is recommended that you switch to the SAS approach. It is relatively forgiving in terms of the grid resolution and will not deteriorate the results in case of insufficient resolution in the unsteady zone. The SAS model will only provide scale-resolution if a strong flow instability is present. Visual inspection (using isosurfaces of the Q criterion) will quickly allow a judgment if the model provides sufficient unsteadiness and resolution relative to the grid spacing. In such cases, the internal interface option of the ELES implementation can be applied to convert modeled turbulence into resolved turbulence (see Internal Interface Without LES Zone in the Theory Guide).

DDES (DES is not recommended) models can allow the formation of unsteadiness even for cases where SAS remains stable. DDES does require a more carefully crafted grid in the LES zone due to the DES grid influence on the RANS solution. For the DES-BSL / SST model, use the default DDES blending function for shielding.

SDES and SBES have an improved shielding function compared to DDES / IDDES. They provide strong shielding of the RANS boundary layer and demonstrate a fast "transition" between RANS and LES in separating shear layers. The SBES model formulation in combination with the WALE LES model is recommended over older models like DES, DDES, and IDDES.

ELES is recommended in cases where limited zones with high accuracy requirements are embedded inside a larger RANS zone. At the interface between the RANS and the LES zone, unsteady turbulence is generated either by the Vortex Method or the Spectral Synthesizer. For wall bounded flows, the ELES method requires a very fine near wall resolution in the LES zone.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Pure LES should only be applied to free shear flows (for example, combustion chambers without wall influence) or to very limited domains using meshes with fine LES near wall resolution. It is important to be aware of the strong increase of grid resolution requirements with Reynolds number for wall bounded flows. For wall boundary layers at higher Reynolds numbers, the IDDES model can be run in WMLES mode – meaning with specified unsteady inlet conditions.

15.3. Steps in Using a Turbulence Model

Your Ansys Fluent model includes the k-ω SST turbulence model by default. You need to choose the appropriate model and options, and supply turbulent boundary conditions. These inputs are described in this section.

The procedure for setting up a turbulent flow problem is described below. (Note that this procedure includes only those steps necessary for the turbulence model itself; you will need to define the other settings—for example, other models, boundary conditions—as usual.)

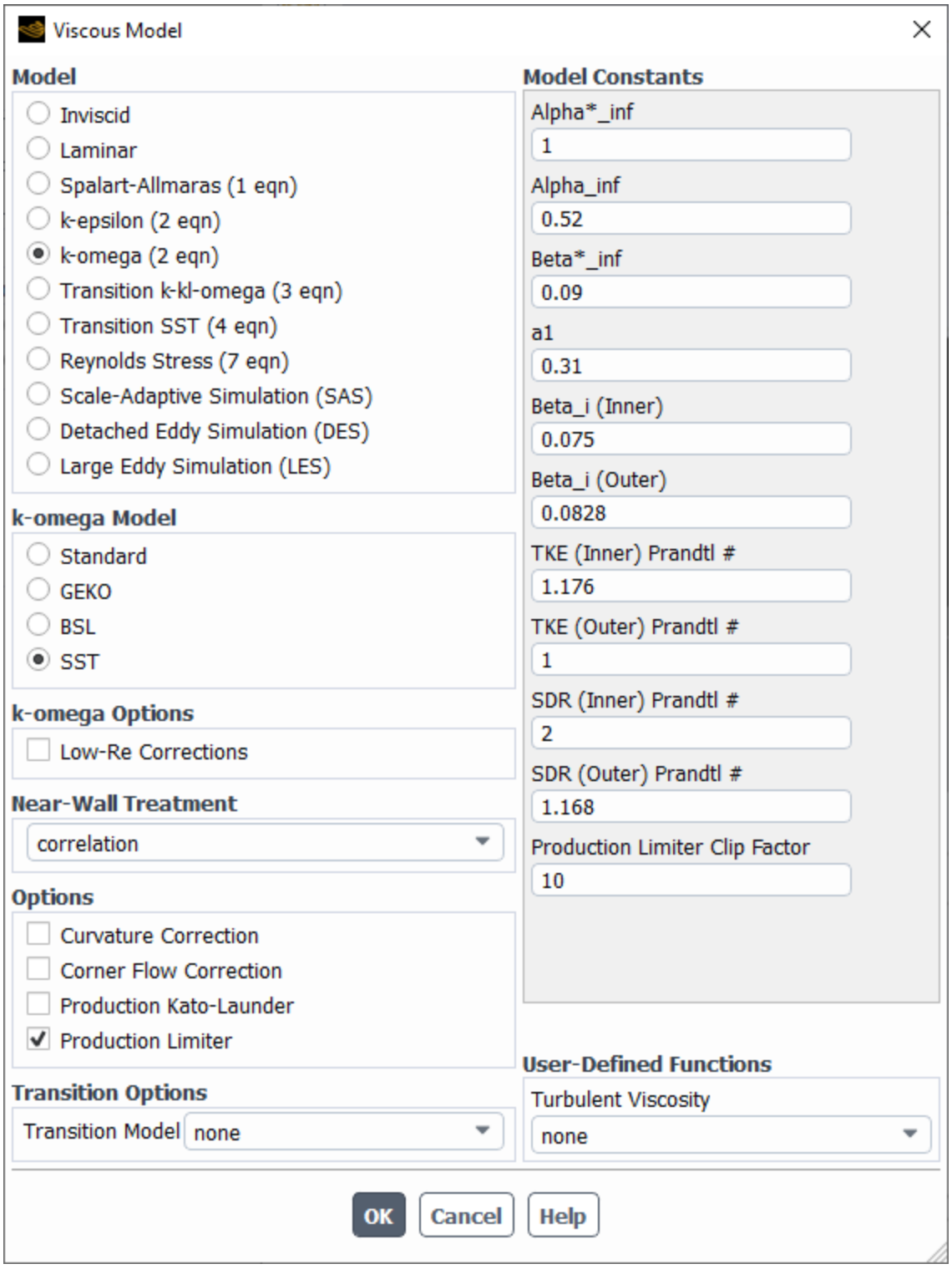
1. Define the turbulence model settings.

   a. Open the Viscous Model Dialog Box (p. 4715) (Figure 15.6: The Viscous Model Dialog Box (p. 2082)) by right-clicking **Viscous** in the tree (under **Setup/Models**) and clicking **Edit...** in the menu that opens.

      Setup → **Models** → **Viscous** **Edit...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.6: The Viscous Model Dialog Box



b. Select either **Spalart-Allmaras**, **k-epsilon**, **k-omega**, **Transition k-kl-omega**, **Transition SST**, **Reynolds Stress**, **Scale-Adaptive Simulation (SAS)**, **Detached Eddy Simulation (DES)**, or **Large Eddy Simulation (LES)** from the **Model** list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you choose the **k-epsilon** model, select either **Standard**, **RNG**, or **Realizable** from the **k-epsilon Model** list. If you choose the **k-omega** model, select **Standard**, **GEKO**, **BSL**, or **SST** from the **k-omega Model** list.

Important:

The **Large Eddy Simulation (LES)** model is available only for 3D cases.

c. If the flow involves walls, and you are using one of the k-ε models or the ε-based Reynolds stress models, choose one of the following options from the **Near-Wall Treatment** list in the **Viscous Model** dialog box:

- **Standard Wall Functions**
- **Scalable Wall Functions**
- **Non-Equilibrium Wall Functions**
- **Enhanced Wall Treatment**
- **Menter-Lechner** (for k-ε models only)
- **User-Defined Wall Functions**

For more information about these near-wall options, see Near-Wall Treatments for Wall-Bounded Turbulent Flows in the Theory Guide. By default, the standard wall function is enabled.

For more information about the automatically defined near-wall treatment for the Spalart-Allmaras model, see Wall Boundary Conditions in the Theory Guide.

For more information about the automatically defined near-wall treatment for the k-ω model, see y+-Insensitive Near-Wall Treatment for ω-based Turbulence Models in the *Fluent Theory Guide*.

For LES model, the following options from the Near-Wall Treatment dropdown list in the Viscous Model dialog box are available:

- **Kader blending Wall Functions** (current default option)
- **Harmonic blending Wall Functions based on r+** (recommended option)
- **Werner-Wengle Wall Treatment**

For more information about the near-wall treatments for the LES model, see LES Near-Wall Treatment in the *Fluent Theory Guide*.

d. Enable the other appropriate turbulence modeling options in the **Viscous Model** dialog box, as appropriate. See Setup Options for All Turbulence Modeling (p. 2128)

2. Specify the boundary conditions for the solution variables in the appropriate boundary condition dialog boxes. The boundary condition dialog boxes can be opened by right-clicking the boundary name in the tree (under **Setup/Boundary Conditions**) and clicking **Edit...** in the menu that opens; alternatively, you can open them from the **Boundary Conditions** task page:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setup → Boundary Conditions

See Defining Turbulence Boundary Conditions (p. 2143) for details.

3. Specify the initial guess for the solution variables in the **Solution Initialization** task page.

Solution → Initialization

See Providing an Initial Guess for k and ε (or k and ω) (p. 2147) for details. Note that Reynolds stresses are automatically initialized using k, and therefore need not be initialized explicitly.

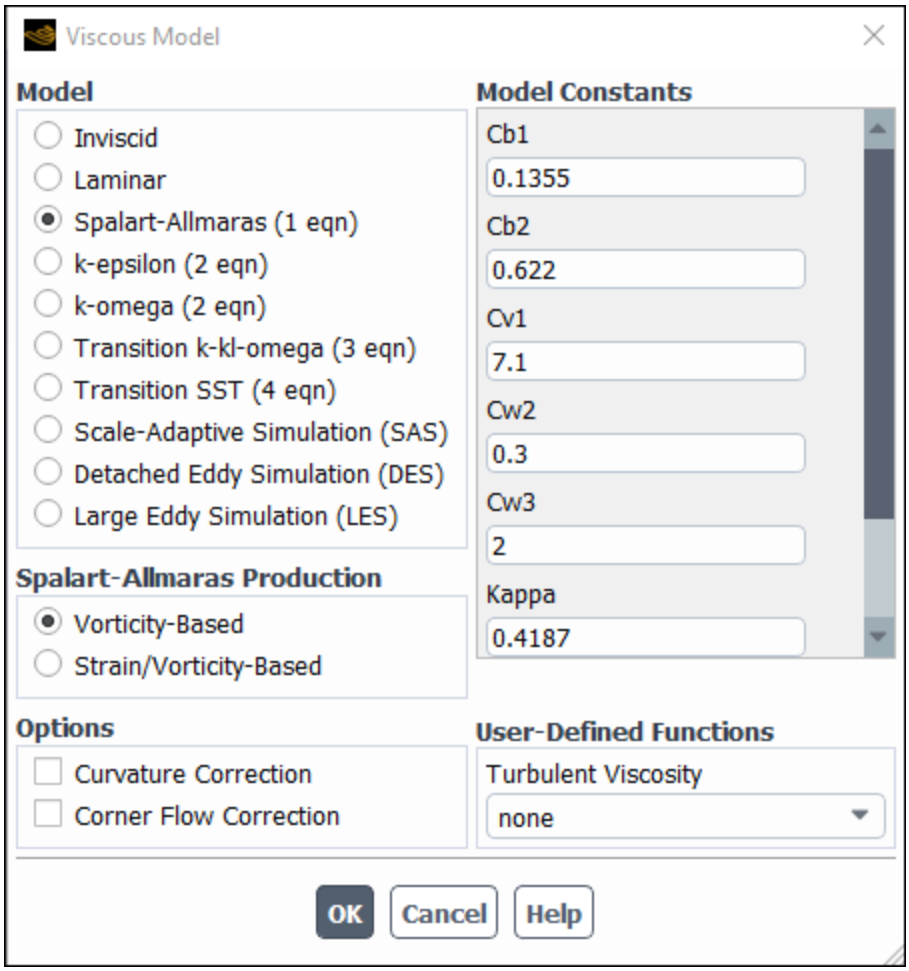
15.4. Setting Up the Spalart-Allmaras Model

If you choose the Spalart-Allmaras model, the following options are available:

- vorticity-based production (Vorticity- and Strain/Vorticity-Based Production (p. 2135))
- strain/vorticity-based production (Vorticity- and Strain/Vorticity-Based Production (p. 2135))
- viscous heating (always enabled for the density-based solvers) (Including the Viscous Heating Effects (p. 2129))
- curvature correction (Including the Curvature Correction for the Spalart-Allmaras and Two-Equation Turbulence Models (p. 2131))

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.7: The Viscous Model Dialog Box Displaying the Spalart-Allmaras Production



15.5. Setting Up the k-ε Model

For additional information, see the following sections:

15.5.1. Setting Up the Standard or Realizable k-ε Model
15.5.2. Setting Up the RNG k-ε Model

15.5.1. Setting Up the Standard or Realizable k-ε Model

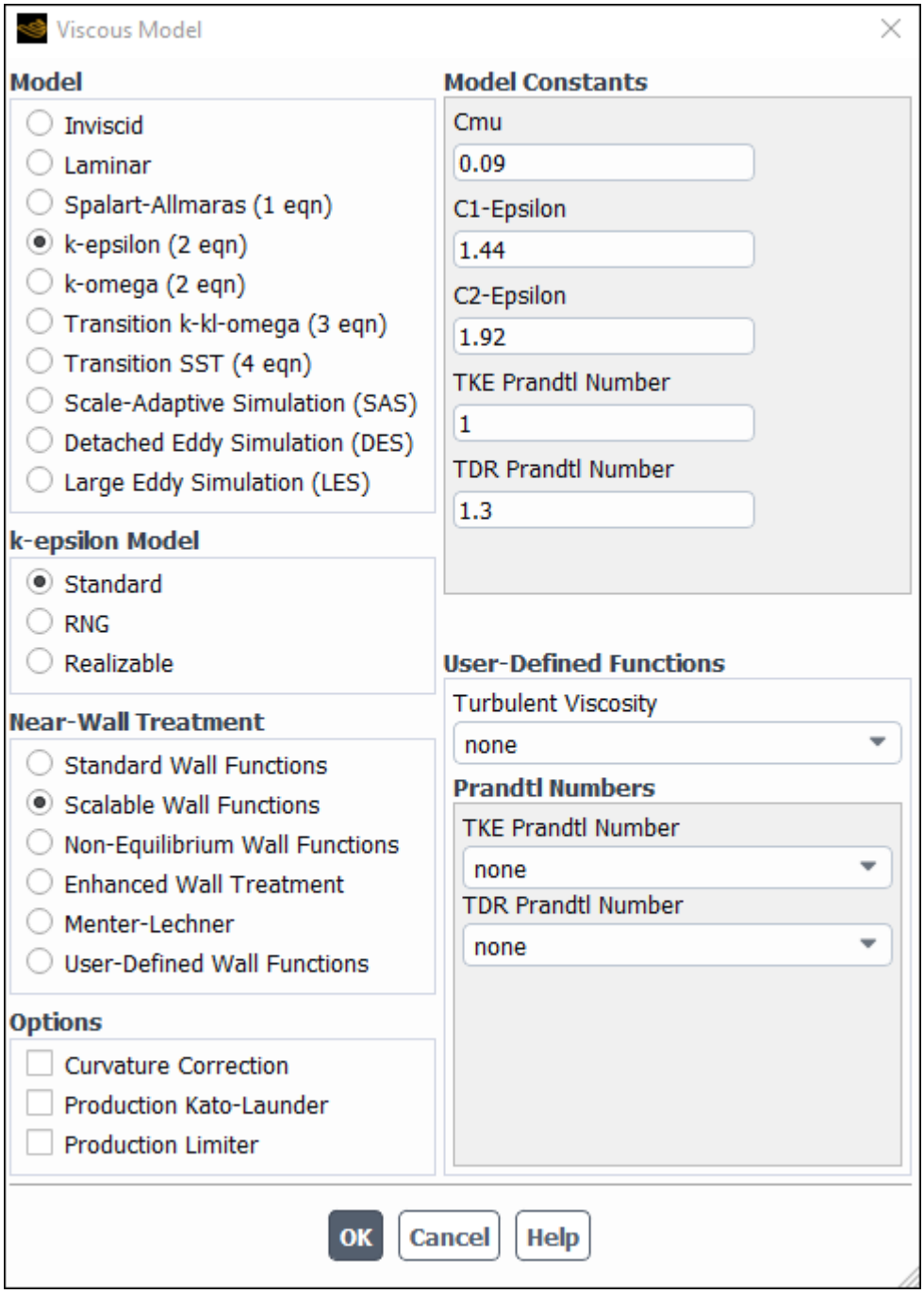
If you choose the standard k-ε model or the realizable k-ε model, the following options are available:

- viscous heating (always enabled for the density-based solvers) (Including the Viscous Heating Effects (p. 2129))
- inclusion of buoyancy effects on ε (see Effects of Buoyancy on Turbulence in the k-ε Models in the Theory Guide)
- inclusion of curvature correction (Including the Curvature Correction for the Spalart-Allmaras and Two-Equation Turbulence Models (p. 2131))
- inclusion of compressibility effects (Including the Compressibility Effects Option (p. 2134))

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- inclusion of production limiters (Including Production Limiters for Two-Equation Models (p. 2134))

Figure 15.8: The Viscous Model Dialog Box Displaying the Standard k-ε Model



For all k-ε models, one of the following near-wall treatments must be selected (see Near-Wall Treatments for Wall-Bounded Turbulent Flows in the Theory Guide):

- standard wall functions
- scalable wall functions
- non-equilibrium wall functions

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- enhanced wall treatment
- Menter-Lechner
- user-defined wall functions

If you choose the enhanced wall treatment, the following options are available:

- pressure gradient effects (Including Pressure Gradient Effects (p. 2137))
- thermal effects (Including Thermal Effects (p. 2137))

If you select user-defined wall functions for the near-wall treatment, hook your UDF using the **Law of the Wall** drop-down list that appears at the bottom of the **User-Defined Functions** group box in the **Viscous Model** dialog box.

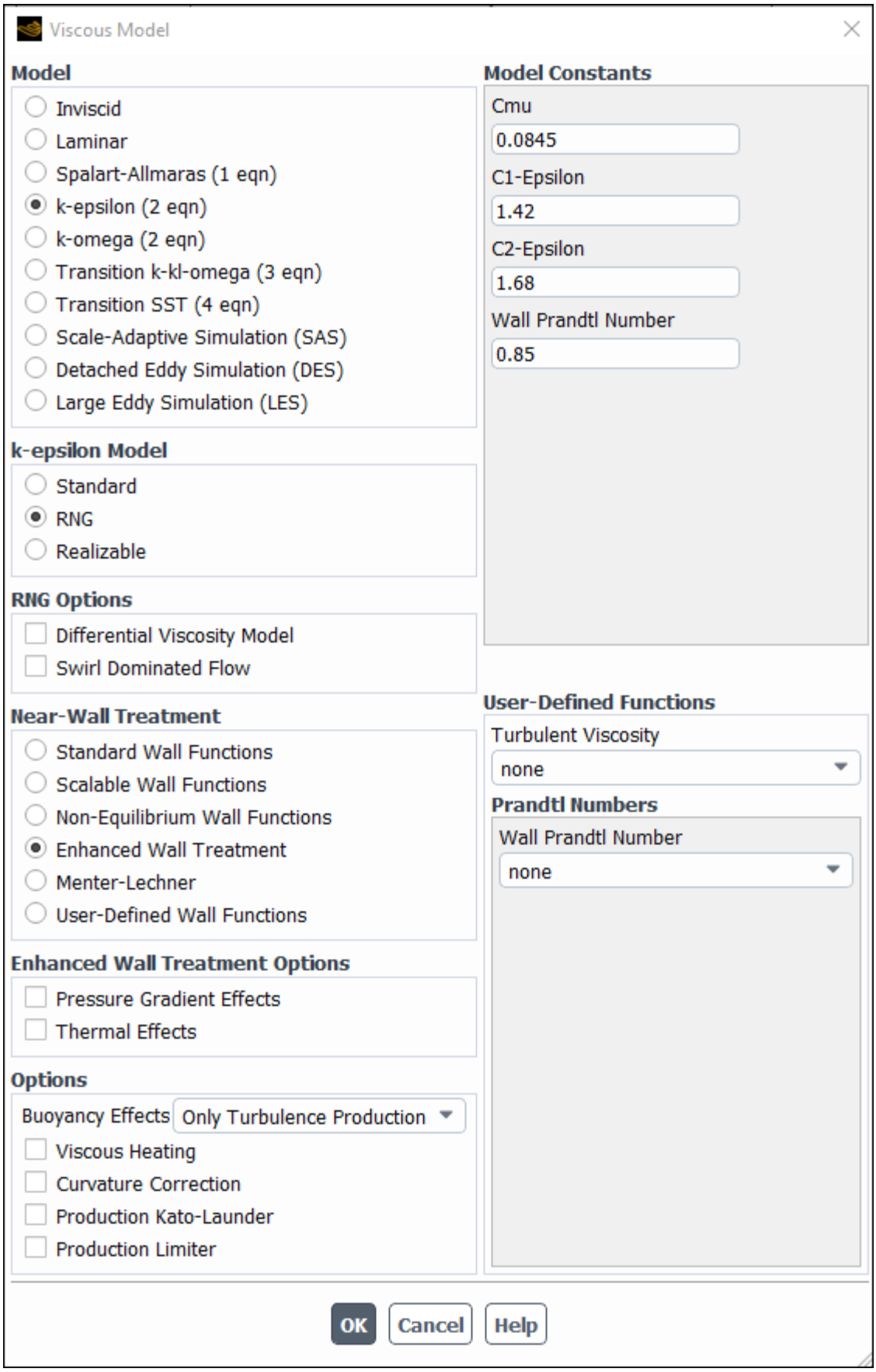
15.5.2. Setting Up the RNG k-ε Model

If you choose the RNG k-ε model, the following options are available:

- differential viscosity model (Differential Viscosity Modification (p. 2135))
- swirl modification (Swirl Modification (p. 2136))
- viscous heating (always enabled for the density-based solvers) (Including the Viscous Heating Effects (p. 2129))
- inclusion of buoyancy effects on ε (see Effects of Buoyancy on Turbulence in the k-ε Models in the Theory Guide)
- inclusion of curvature correction (Including the Curvature Correction for the Spalart-Allmaras and Two-Equation Turbulence Models (p. 2131))
- inclusion of compressibility effects (Including the Compressibility Effects Option (p. 2134))
- inclusion of production limiters (Including Production Limiters for Two-Equation Models (p. 2134))

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.9: The Viscous Model Dialog Box Displaying the RNG k-ε Model



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For all k-ε models, one of the following near-wall treatments must be selected (see Near-Wall Treatments for Wall-Bounded Turbulent Flows in the Theory Guide):

- standard wall functions
- scalable wall functions
- non-equilibrium wall functions
- enhanced wall treatment
- Menter-Lechner
- user-defined wall functions

If you choose the enhanced wall treatment, the following options are available:

- pressure gradient effects (Including Pressure Gradient Effects (p. 2137))
- thermal effects (Including Thermal Effects (p. 2137))

If you select user-defined wall functions for the near-wall treatment, hook your UDF using the **Law of the Wall** drop-down list that appears at the bottom of the **User-Defined Functions** group box in the **Viscous Model** dialog box.

15.6. Setting Up the k-ω Model

For additional information, see the following sections:

15.6.1. Setting Up the Standard k-ω Model
15.6.2. Setting up the Generalized k-ω (GEKO) Model
15.6.3. Setting Up the Baseline (BSL) k-ω Model
15.6.4. Setting Up the Shear-Stress Transport k-ω Model
15.6.5. Setting up the WJ-BSL-EARSM Model

15.6.1. Setting Up the Standard k-ω Model

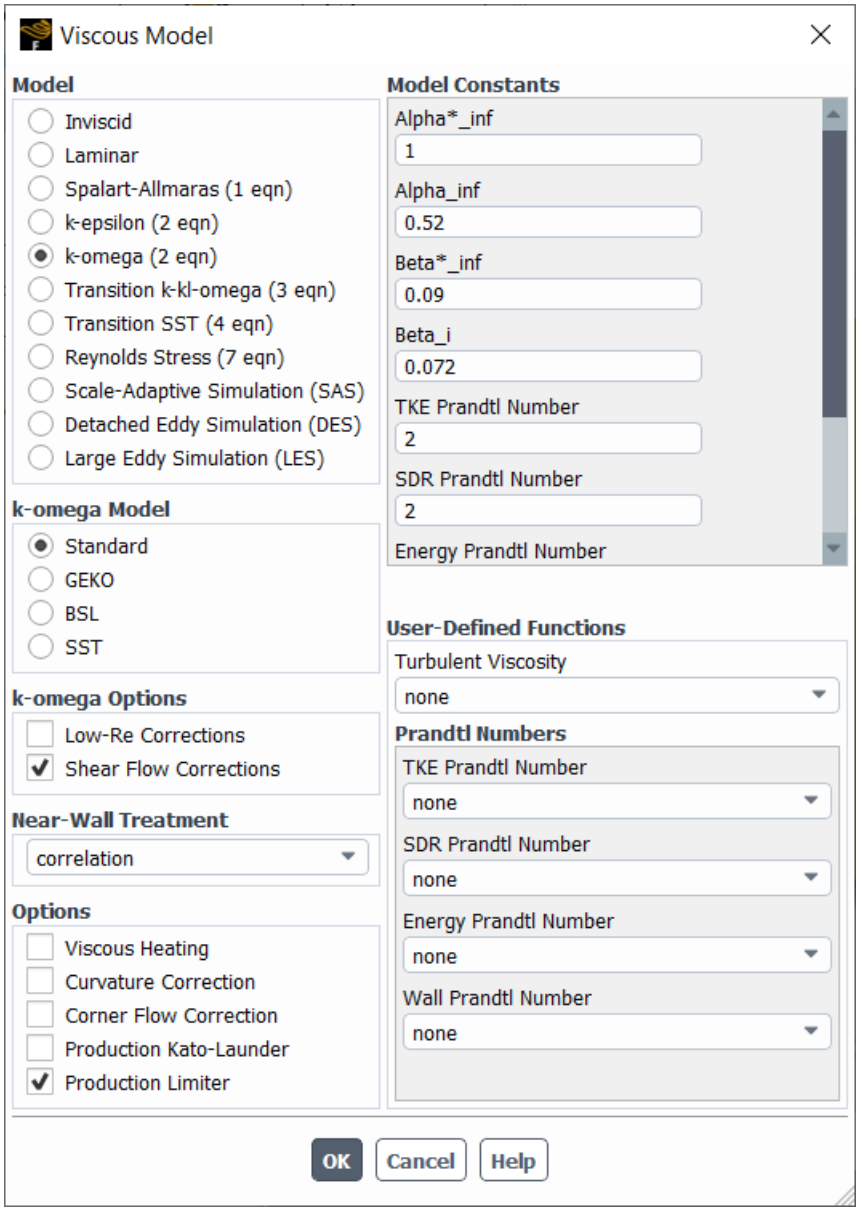
If you choose the standard k-ω model, the following options are available:

- low-Re corrections (Low-Re Corrections (p. 2136))
- shear flow corrections (Shear Flow Corrections (p. 2136))
- turbulence damping (available with the VOF and Mixture models and the Eulerian multiphase model when using the Multi-Fluid VOF model) (Turbulence Damping (p. 2136))
- near wall treatment (y+-Insensitive Near-Wall Treatment for ω-based Turbulence Models in the *Fluent Theory Guide*)
- viscous heating (always enabled for the density-based solvers) (Including the Viscous Heating Effects (p. 2129))

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- inclusion of curvature correction (Including the Curvature Correction for the Spalart-Allmaras and Two-Equation Turbulence Models (p. 2131))
- inclusion of compressibility effects (Including the Compressibility Effects Option (p. 2134))
- inclusion of production limiters (Including Production Limiters for Two-Equation Models (p. 2134))
- a scale-resolving simulation option: Scale-Adaptive Simulation (Setting Up Scale-Adaptive Simulation (SAS) Modeling (p. 2109))

Figure 15.10: The Viscous Model Dialog Box Displaying the Standard k-ω Model



15.6.2. Setting up the Generalized k-ω (GEKO) Model

To enable the GEKO turbulence model, you must select the **k-omega (2 eqn)** model on the Viscous Model dialog box and then select **GEKO** in the **k-omega Model** group box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you choose the GEKO k-ω model, the following options are available:

- turbulence damping (available with the VOF and Mixture models and the Eulerian multiphase model when using the Multi-Fluid VOF model) (Turbulence Damping (p. 2136))
- near wall treatment (y+-Insensitive Near-Wall Treatment for ω-based Turbulence Models in the *Fluent Theory Guide*)
- viscous heating (always enabled for the density-based solvers) (Including the Viscous Heating Effects (p. 2129))
- inclusion of curvature correction (Including the Curvature Correction for the Spalart-Allmaras and Two-Equation Turbulence Models (p. 2131))
- inclusion of compressibility effects (Including the Compressibility Effects Option (p. 2134))
- inclusion of production limiters (Including Production Limiters for Two-Equation Models (p. 2134))
- inclusion of the Intermittency Transition model
- scale-resolving simulation options: either Scale-Adaptive Simulation (Setting Up Scale-Adaptive Simulation (SAS) Modeling (p. 2109)), Detached Eddy Simulation (Setting Up DES with the Transition SST Model (p. 2120)), Stress-Blended Eddy Simulation (Including the SDES or SBES Model with RANS Models (p. 2140)), or Shielded Detached Eddy Simulation (Including the SDES or SBES Model with RANS Models (p. 2140)).

If you intend to combine the GEKO model with a Reynolds Stress Model, you must first select the Reynolds Stress Model based on BSL equation (Stress-BSL). You can then enable the **GEKO** option in the **Options** group box of the Viscous Model dialog box.

In most cases, you would only tune the coefficients C_{SEP} and C_{MIX}.

1. Optimize the coefficient C_{SEP} for the separation characteristics of wall boundary layers, which in most applications has the strongest impact on performance.
2. Adjust C_{MIX} to affect the mixing behavior of free shear flows (free mixing layers, etc.).

For more details regarding these coefficients, see Generalized k-ω (GEKO) Model (p. 2066) and Generalized k-ω (GEKO) Model in the *Fluent Theory Guide*.

As these coefficients are both available through expressions and UDF, you can also adjust them differently in different parts of the domain. The other two parameters can, in most cases, be left to their default settings.

By default, the full model is enabled and gives access to all model coefficients and the blending function in the **GEKO Options** group box (Figure 15.11: The Viscous Model Dialog Box with GEKO Options for the Full Model (p. 2093)). For the wall distance free model option, only a subset of model coefficients can be used.

The corresponding text command to enable the GEKO turbulence model is

```
/define/models/viscous/kw-geko?
```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For combination with Reynolds stress models you must first enable the corresponding Reynolds Stress Model and then use the TUI command.

```
/define/models/viscous/rsm-or-earsm-geko-option?
```

After GEKO is enabled, all model coefficients and the blending function are available in the submenu `geko-options`:

```
/define/models/viscous/geko-options/wall-distance-free?

/define/models/viscous/geko-options/csep

/define/models/viscous/geko-options/cnw

/define/models/viscous/geko-options/cmix

/define/models/viscous/geko-options/cjet

/define/models/viscous/geko-options/blending-function
```

The auxiliary coefficients are also available as TUI commands.

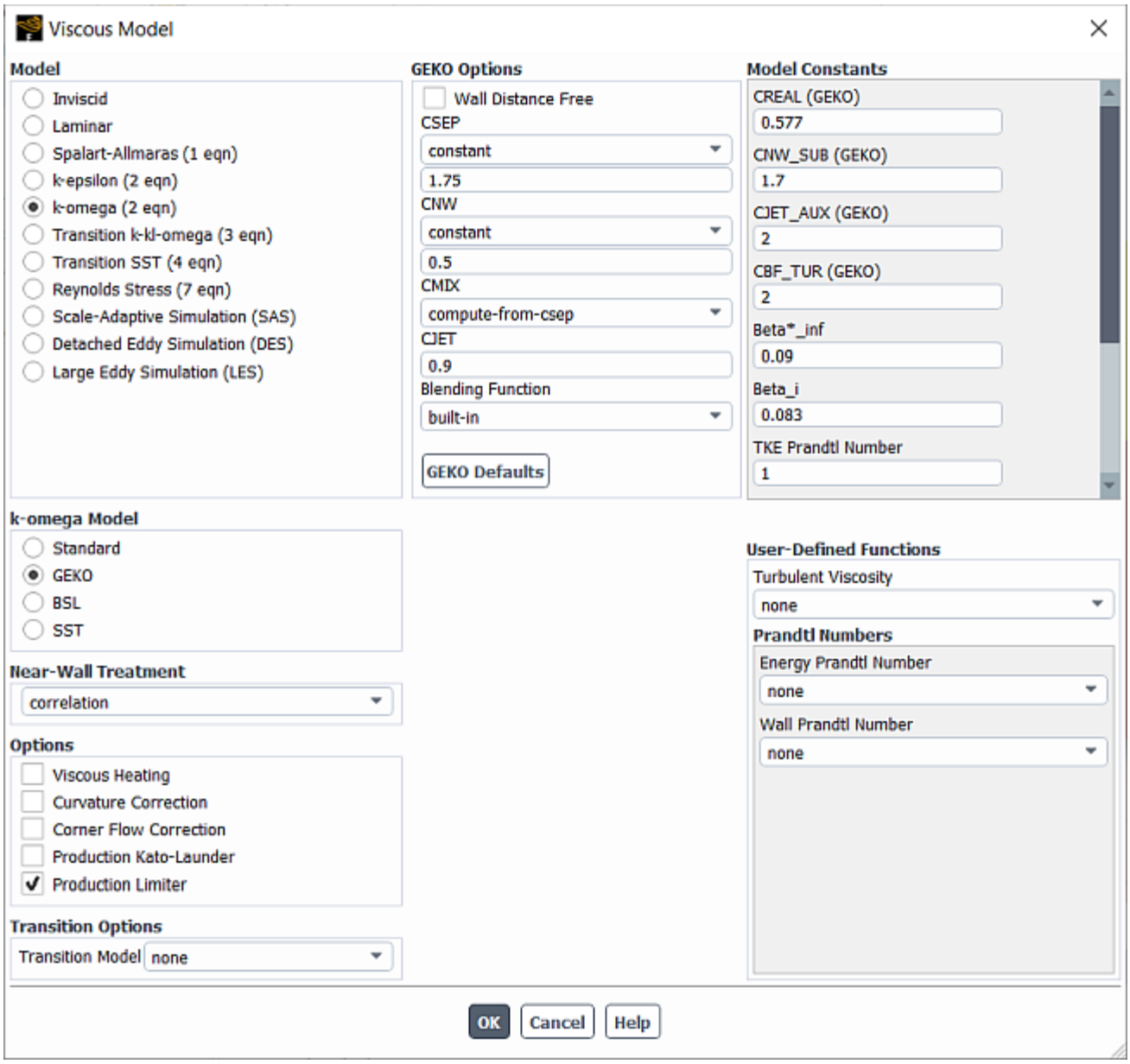
```
/define/models/viscous/geko-options/creal

/define/models/viscous/geko-options/cnw_sub

/define/models/viscous/geko-options/cjet_aux

/define/models/viscous/geko-options/cbf_lam

/define/models/viscous/geko-options/cbf_tur
```

Furthermore, it is possible to reset these model parameters to default values by clicking **GEKO Defaults** or the TUI command:

```
/define/models/viscous/geko-options/geko-defaults
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.11: The Viscous Model Dialog Box with GEKO Options for the Full Model



15.6.3. Setting Up the Baseline (BSL) k-ω Model

If you choose the baseline (BSL) k-ω model, the following options are available:

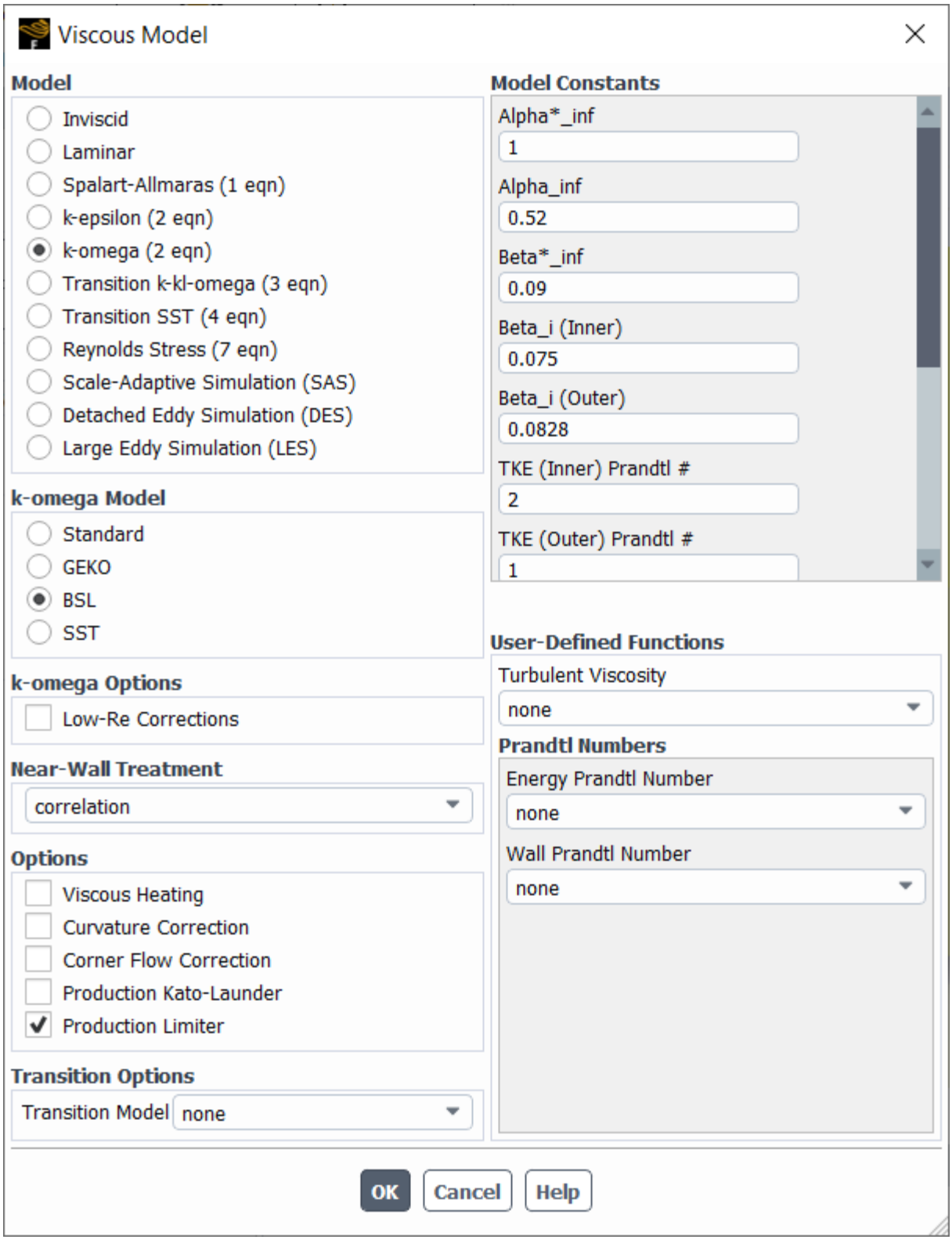
- low-Re corrections (Low-Re Corrections (p. 2136))
- turbulence damping (available with the VOF and Mixture models and the Eulerian multiphase model when using the Multi-Fluid VOF model) (Turbulence Damping (p. 2136))
- near wall treatment (y+-Insensitive Near-Wall Treatment for ω-based Turbulence Models in the *Fluent Theory Guide*)
- viscous heating (always enabled for the density-based solvers) (Including the Viscous Heating Effects (p. 2129))

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- inclusion of curvature correction (Including the Curvature Correction for the Spalart-Allmaras and Two-Equation Turbulence Models (p. 2131))
- inclusion of compressibility effects (Including the Compressibility Effects Option (p. 2134))
- inclusion of production limiters (Including Production Limiters for Two-Equation Models (p. 2134))
- inclusion of the Intermittency Transition model
- scale-resolving simulation options: either Scale-Adaptive Simulation (Setting Up Scale-Adaptive Simulation (SAS) Modeling (p. 2109)), Detached Eddy Simulation (Setting Up DES with the Transition SST Model (p. 2120)), Stress-Blended Eddy Simulation (Including the SDES or SBES Model with RANS Models (p. 2140)), or Shielded Detached Eddy Simulation (Including the SDES or SBES Model with RANS Models (p. 2140)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.12: The Viscous Model Dialog Box Displaying the BSL k-ω Model



15.6.4. Setting Up the Shear-Stress Transport k-ω Model

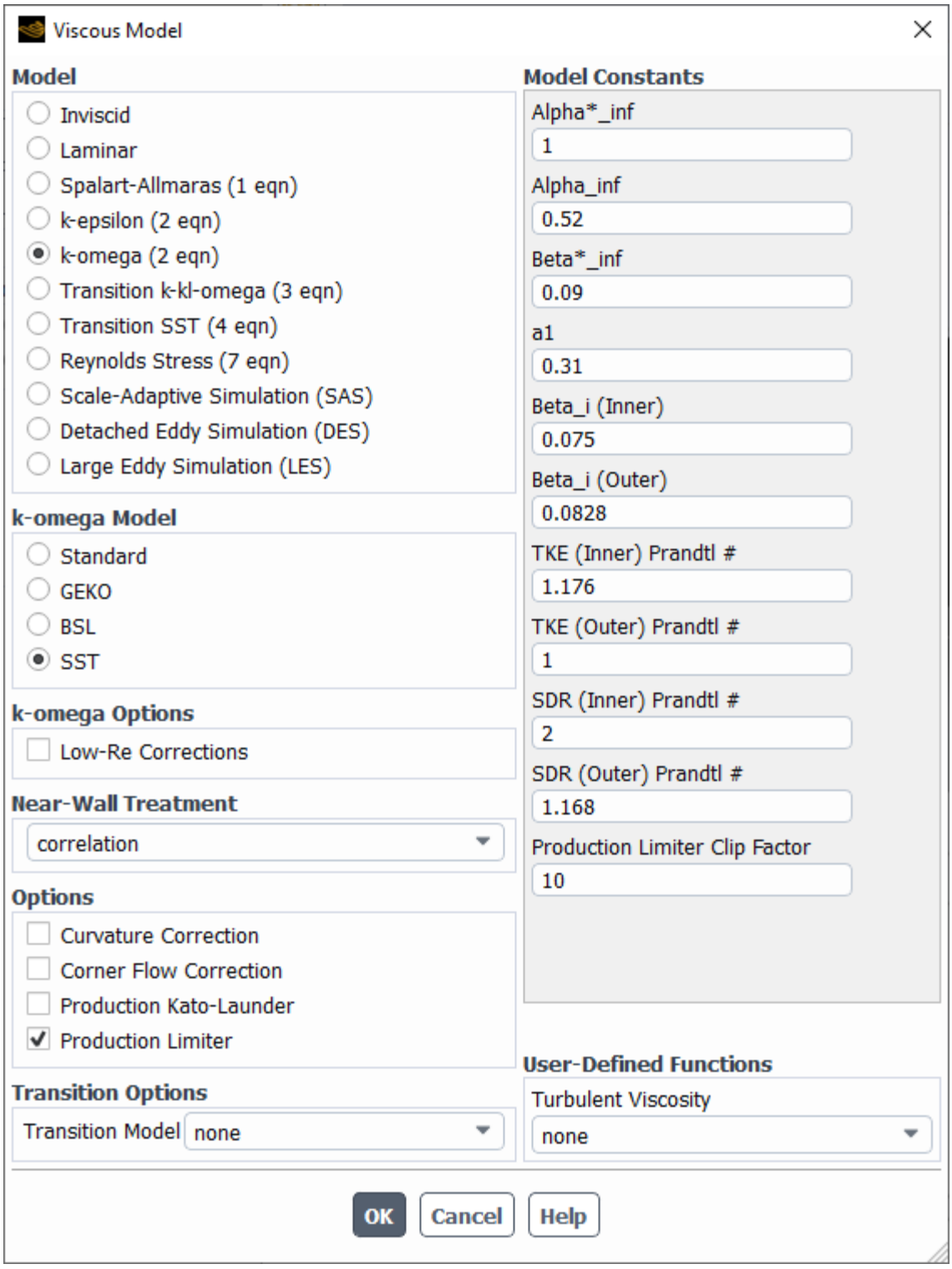
If you choose the shear-stress transport k-ω model, the following options are available:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- low-Re corrections (Low-Re Corrections (p. 2136))
- turbulence damping (available with the VOF and Mixture models and the Eulerian multiphase model when using the Multi-Fluid VOF model) (Turbulence Damping (p. 2136))
- near wall treatment (y+-Insensitive Near-Wall Treatment for ω-based Turbulence Models in the *Fluent Theory Guide*)
- viscous heating (always enabled for the density-based solvers) (Including the Viscous Heating Effects (p. 2129))
- inclusion of curvature correction (Including the Curvature Correction for the Spalart-Allmaras and Two-Equation Turbulence Models (p. 2131))
- inclusion of compressibility effects (Including the Compressibility Effects Option (p. 2134))
- inclusion of production limiters (Including Production Limiters for Two-Equation Models (p. 2134))
- inclusion of the Intermittency Transition model
- scale-resolving simulation options: Stress-Blended Eddy Simulation or Shielded Detached Eddy Simulation (Including the SDES or SBES Model with RANS Models (p. 2140)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.13: The Viscous Model Dialog Box Displaying the SST k-ω Model



15.6.5. Setting up the WJ-BSL-EARSM Model

If you choose the **WJ-BSL-EARSM** k-ω model, the following options are available:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- near wall treatment (y+-Insensitive Near-Wall Treatment for ω-based Turbulence Models in the *Fluent Theory Guide*)
- viscous heating (always enabled for the density-based solvers) (Including the Viscous Heating Effects (p. 2129))
- inclusion of production limiters (Including Production Limiters for Two-Equation Models (p. 2134))
- enabling the GEKO model in combination with the WJ-BSL-EARSM by selecting **GEKO** under **Options**.(Setting up the Generalized k-ω (GEKO) Model (p. 2090))
- inclusion of curvature correction (Including the Curvature Correction for the Spalart-Allmaras and Two-Equation Turbulence Models (p. 2131))

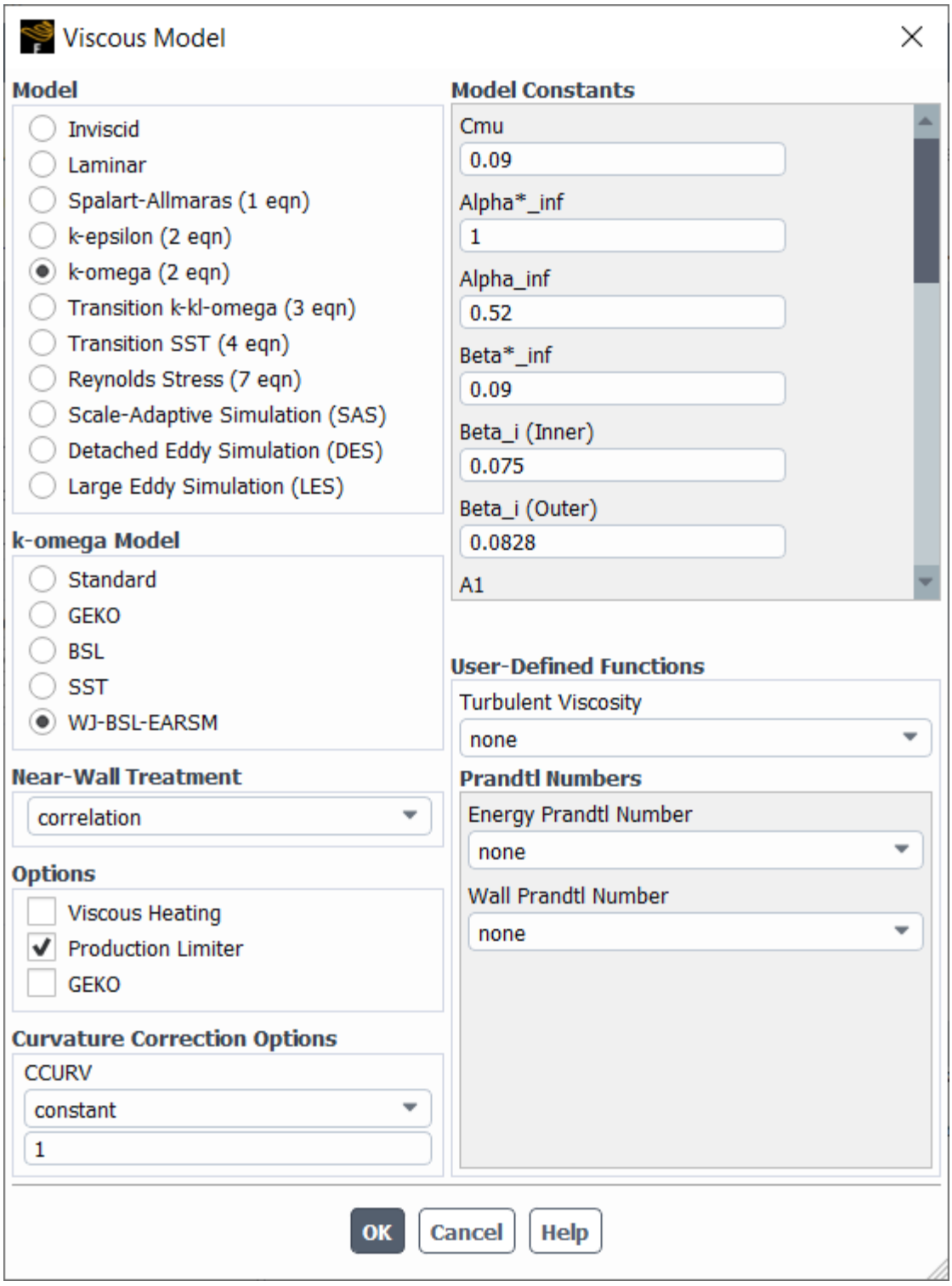
The model is available when the **k-omega** model is selected. The **WJ-BSL-EARSM** option becomes available under **k-omega Model**, as shown in Figure 15.14: The Viscous Model Dialog Box With WJ-BSL-EARSM Enabled (p. 2099).

Note:

It is only available for 3D cases, and is not supported with the gap model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.14: The Viscous Model Dialog Box With WJ-BSL-EARSM Enabled



The **WJ-BSL-EARSM** is an extension of the Baseline (**BSL**) k-ω model and allows to capture the following flow effects:

- Anisotropy of Reynolds Stresses
- Secondary Flows

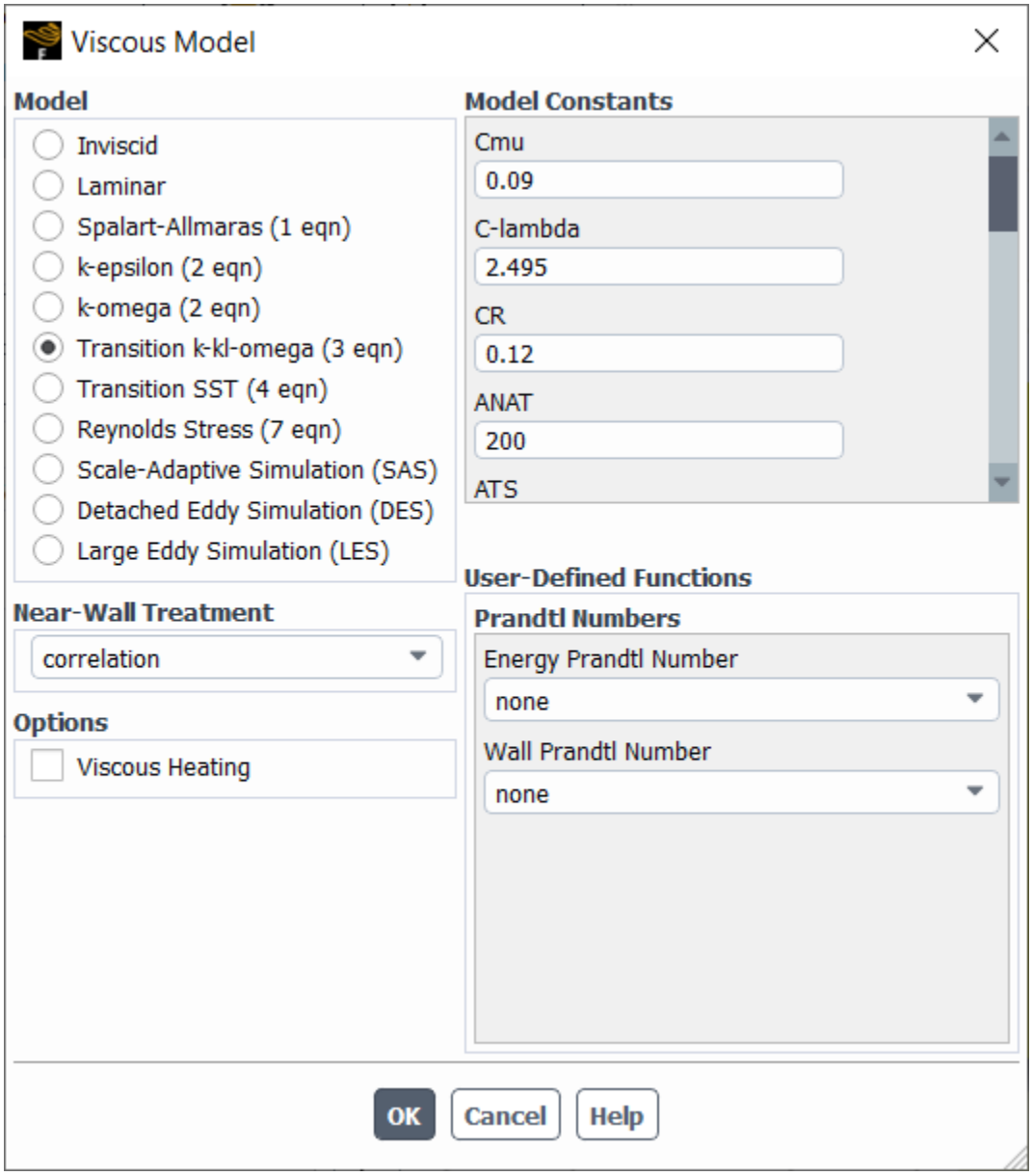
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15.7. Setting Up the Transition k-kl-ω Model

If you choose the Transition k-kl-ω model, the following options are available:

- near wall treatment (y+-Insensitive Near-Wall Treatment for ω-based Turbulence Models in the *Fluent Theory Guide*)
- viscous heating (always enabled for the density-based solvers) (Including the Viscous Heating Effects (p. 2129))

Figure 15.15: The Viscous Model Dialog Box for the Transition k-kl-ω Model



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

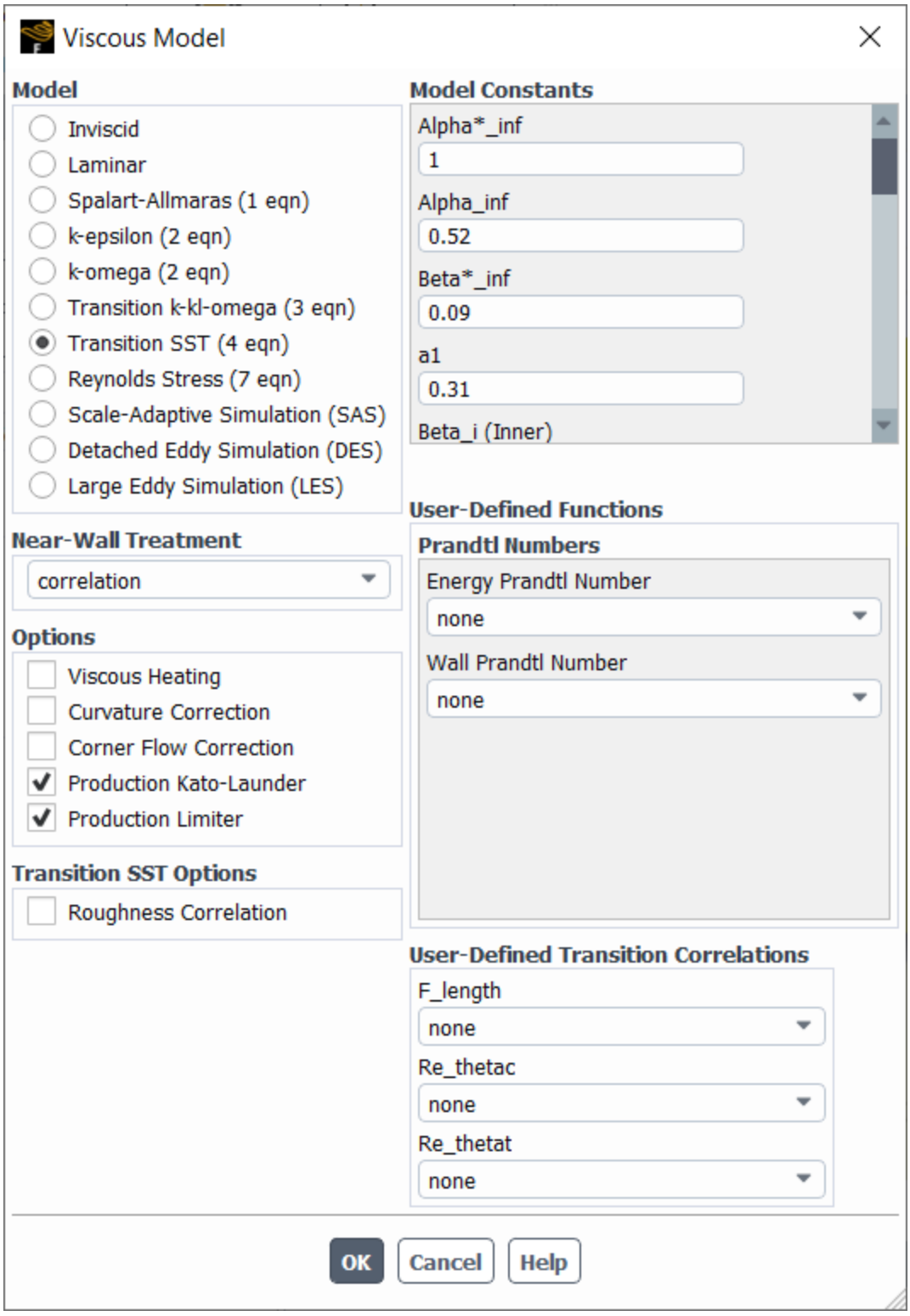
15.8. Setting Up the Transition SST Model

If you choose the Transition SST model (also known as the γ-Re_θ model), the following options are available:

- roughness correlation (Transition SST and Rough Walls in the Theory Guide)
- near wall treatment (y+-Insensitive Near-Wall Treatment for ω-based Turbulence Models in the *Fluent Theory Guide*)
- viscous heating (always enabled for the density-based solvers) (Including the Viscous Heating Effects (p. 2129))
- inclusion of curvature correction (Including the Curvature Correction for the Spalart-Allmaras and Two-Equation Turbulence Models (p. 2131))
- inclusion of production limiters (Including Production Limiters for Two-Equation Models (p. 2134))
- scale-resolving simulation options: either Scale-Adaptive Simulation (Setting Up Scale-Adaptive Simulation (SAS) Modeling (p. 2109)), Detached Eddy Simulation (Setting Up DES with the Transition SST Model (p. 2120)), Stress-Blended Eddy Simulation (Including the SDES or SBES Model with RANS Models (p. 2140)), or Shielded Detached Eddy Simulation (Including the SDES or SBES Model with RANS Models (p. 2140)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.16: The Viscous Model Dialog Box for the Transition SST Model



You can customize your transition correlations, which are used in conjunction with the Transition SST model. The user-defined functions that you can hook are:

- transition length function (**F_length**).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- critical momentum thickness Reynolds number (**Re_thetac**).
- transition onset momentum thickness Reynolds number (**Re_thetat**).

For detailed information about the transition correlation UDFs, see DEFINE_TRANS UDFs in the Fluent Customization Manual.

15.9. Setting Up the Algebraic or Intermittency Transition Model

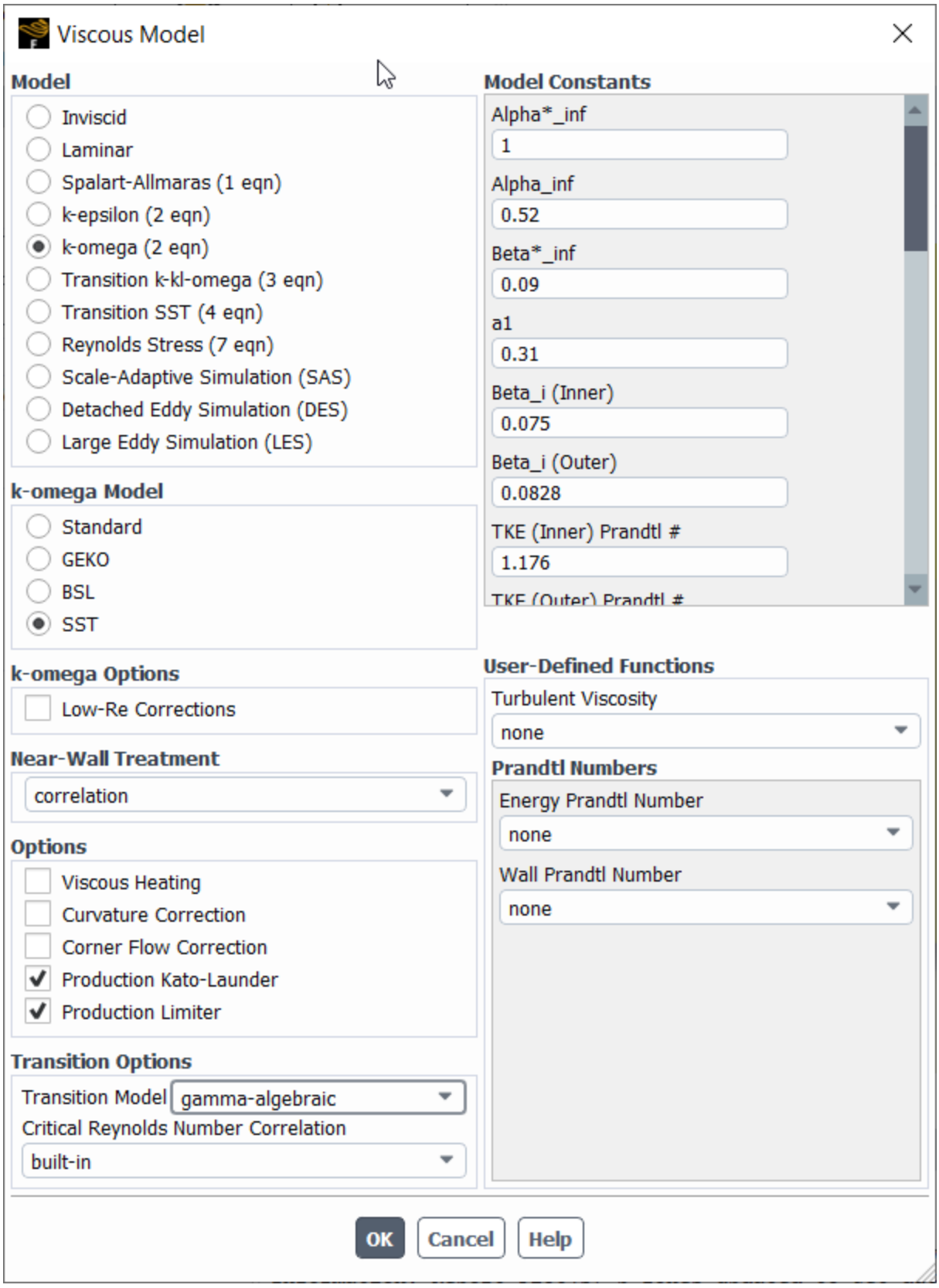
The Algebraic or Intermittency Transition models are available as **Transition Options** for the following turbulence models:

- BSL k-ω model
- SST k-ω model
- GEKO k-ω model
- Scale-Adaptive Simulation with BSL / SST
- Detached Eddy Simulation with BSL / SST
- Shielded Detached Eddy Simulation (SDES) with BSL / SST
- Stress-Blended Eddy Simulation (SBES) with BSL / SST

To apply it in combination with one of these turbulence models, you must enable a **Transition Model** from the **Transition Options** group box in the **Viscous Model** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.17: Transition Option enabled in Combination with the SST k-ω Model



By default, no transition model is active. Currently, you have the following options:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The **gamma-algebraic** model (also known as the Algebraic Transition Model). For more details see, Algebraic Transition Model in the *Fluent Theory Guide*.

  Additionally, you have the option to specify your own **Critical Reynolds Number Correlation** instead of using the built-in recipe.

- The **gamma-transport-eqn** model (also known as the Intermittency Transition Model). For more details see, Intermittency Transition Model in the *Fluent Theory Guide*.

  Additionally, you have the option to enable **Include Crossflow Transition** which includes the effects of crossflow instability.

Note that for the combination a model with the algebraic or intermittency transition model, the y^+-insensitive wall treatment of the underlying turbulence model is used. For details on the available options, see y+-Insensitive Near-Wall Treatment for ω-based Turbulence Models in the *Fluent Theory Guide*.

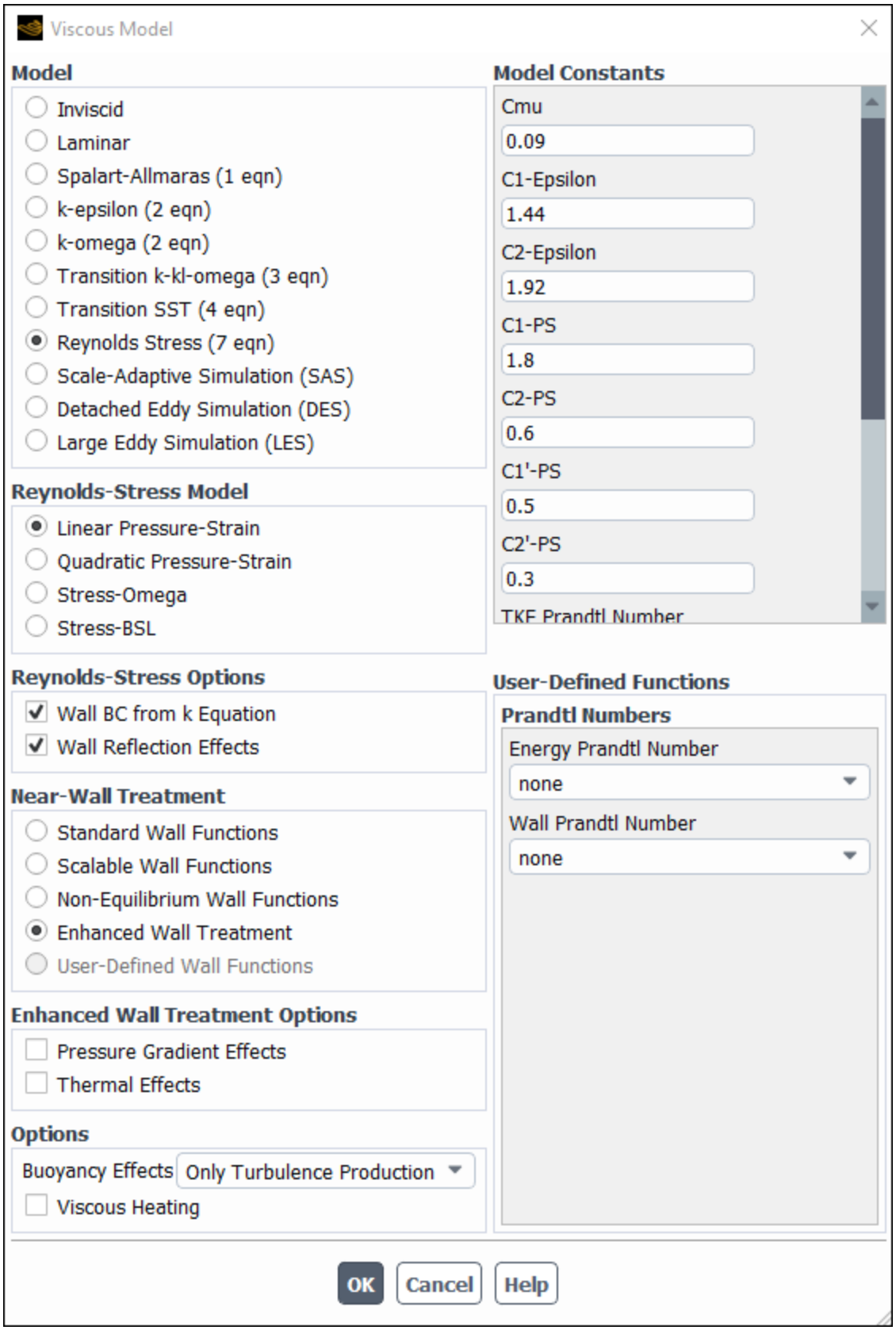
15.10. Setting Up the Reynolds Stress Model

If you choose the RSM, the following submodels are available:

- ε-based Reynolds stress models:
  - linear pressure-strain model (see Linear Pressure-Strain Model in the Theory Guide)
  - quadratic pressure-strain model (see Quadratic Pressure-Strain Model in the Theory Guide)
- ω-based Reynolds stress models:
  - Stress-Omega (see Stress-Omega Model in the Theory Guide)
  - Stress-BSL (see Stress-BSL Model in the Theory Guide)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.18: The Viscous Model Dialog Box Displaying the Reynolds Stress Model Options



The following options are specific to the Reynolds-stress models based on the ε-equation:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- wall boundary conditions for the Reynolds stresses from the k equation (Solving the k Equation to Obtain Wall Boundary Conditions (p. 2137)) for the linear and quadratic pressure-strain models
- wall reflection effects on Reynolds stresses (Including the Wall Reflection Term (p. 2137)) for the linear pressure-strain model
- near-wall treatments (see Near-Wall Treatments for Wall-Bounded Turbulent Flows in the Theory Guide):
  - standard wall functions
  - scalable wall functions
  - non-equilibrium wall functions
  - enhanced wall treatment (only for the linear pressure-strain model)

If you choose the enhanced wall treatment, the following options are available:

- pressure gradient effects (Including Pressure Gradient Effects (p. 2137))
- thermal effects (Including Thermal Effects (p. 2137))

Both the Reynolds stress models based on the ω- (**Stress-Omega**) and BSL-equation (**Stress-BSL**) use a y^+-insensitive wall treatment. For details on the available options, see y+-Insensitive Near-Wall Treatment for ω-based Turbulence Models in the *Fluent Theory Guide*. A low Re-number (LRN) mesh is recommended for these models.

For **Stress-Omega**, you do have the option of selecting any or all of the following $k-\omega$ options:

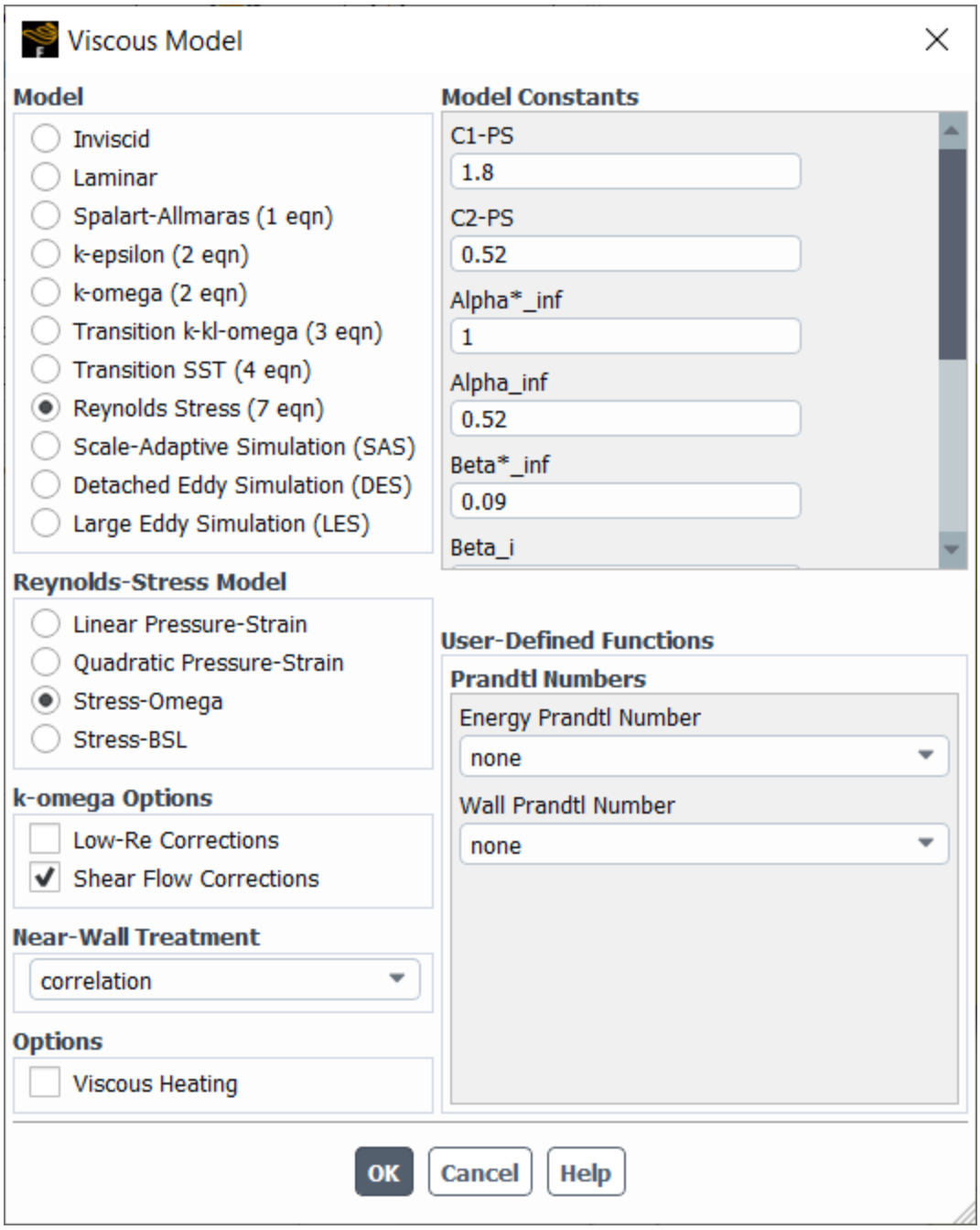
- low-Re corrections (Low-Re Corrections (p. 2136))
- shear flow corrections (Shear Flow Corrections (p. 2136))

For **Stress-BSL**, you have the option of combining the **GEKO** model with the **Stress-BSL** model. For details on the GEKO model, see Setting up the Generalized k-ω (GEKO) Model (p. 2090).

For both **Stress-Omega** and **Stress-BSL**, the following scale-resolving simulation option is available: Scale-Adaptive Simulation (Setting Up Scale-Adaptive Simulation (SAS) Modeling (p. 2109)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.19: The Viscous Model Dialog Box Displaying the Stress-Omega Model Options



Other options that are available for both ε- and ω-based Reynolds stress models depending on your case setup include:

- viscous heating (always enabled for the density-based solvers) (Including the Viscous Heating Effects (p. 2129))
- inclusion of compressibility effects (Including the Compressibility Effects Option (p. 2134))

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- inclusion of buoyancy effects on ε (see Effects of Buoyancy on Turbulence in the k-ε Models in the Theory Guide)

Note:

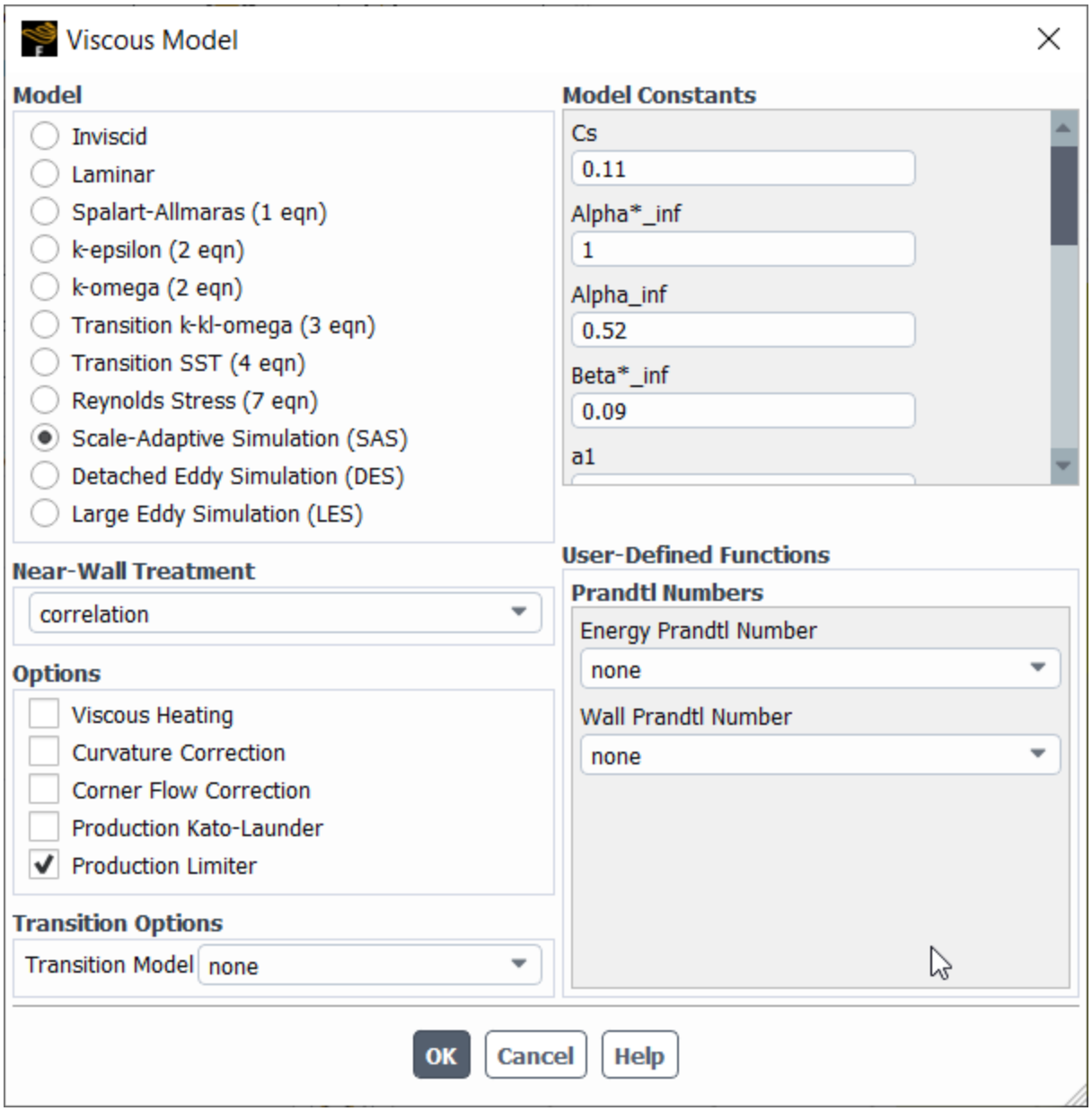
The ω-based Reynolds stress models are not compatible with the Eulerian multiphase model.

15.11. Setting Up Scale-Adaptive Simulation (SAS) Modeling

Scale-Adaptive Simulation (SAS) modeling is an approach for the simulation of unsteady turbulent flows, and can be applied in combination with most ω-based URANS turbulence models. To apply it in combination with the SST k-ω turbulence model, you can simply select **Scale-Adaptive Simulation (SAS)** from the **Model** list in the **Viscous Model** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

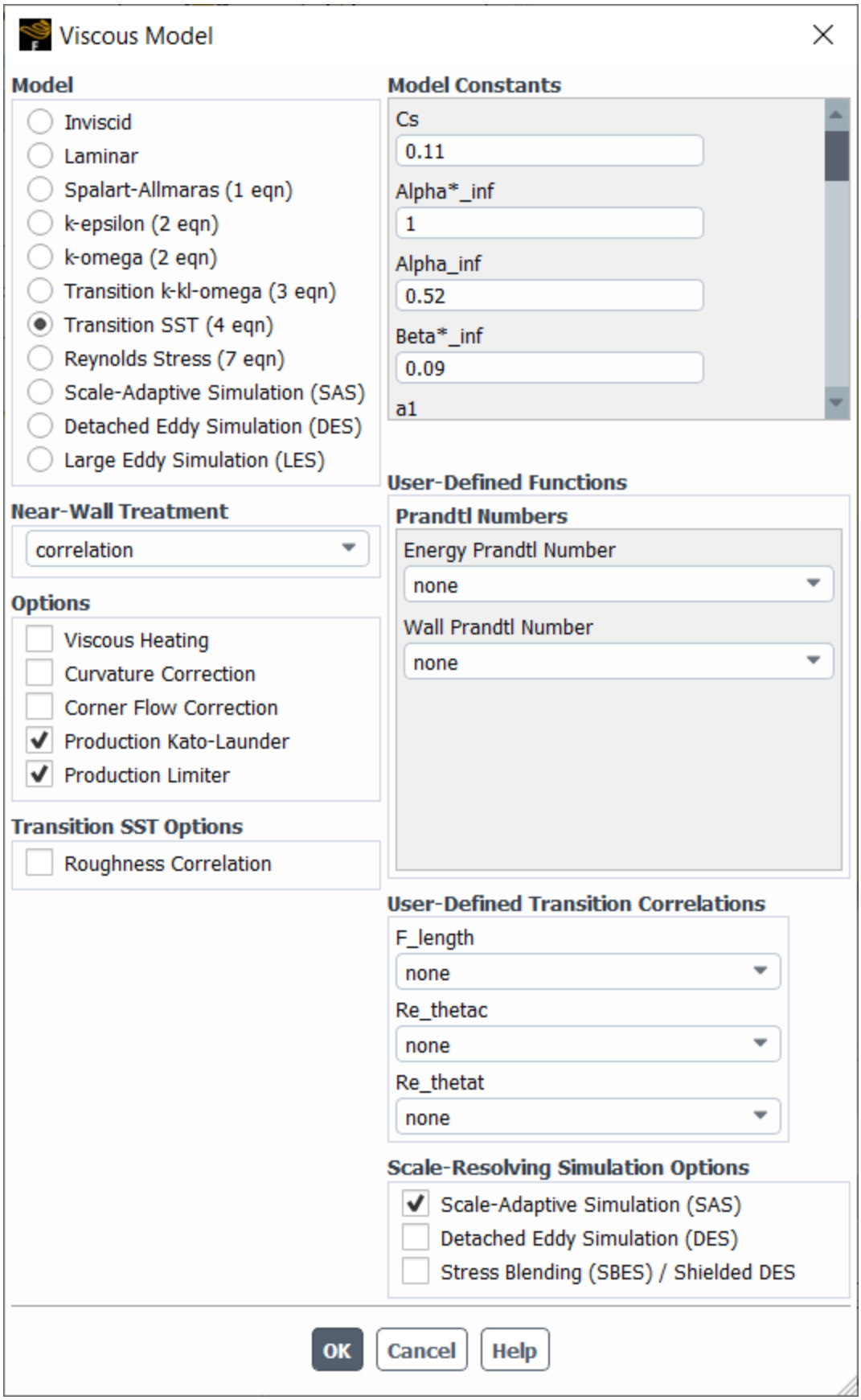
Figure 15.20: Scale-Adaptive Simulation (SAS) in Combination with the SST Turbulence Model



You can also apply SAS in combination with the following ω-based URANS models: the Standard k-ω model, the BSL k-ω model, the Generalized k-w (GEKO) model, the Transition SST model, the Explicit Algebraic Reynolds Stress (WJ-BSL-EARSM) model, and the ω-based Reynolds stress models (RSM). Simply select the appropriate **Model** and then enable the **Scale-Adaptive Simulation (SAS)** option in the **Scale-Resolving Simulation Options** group box. For an example, see Figure 15.21: Scale-Adaptive Simulation (SAS) in Combination with the Transition SST Model (p. 2111).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.21: Scale-Adaptive Simulation (SAS) in Combination with the Transition SST Model



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that for SAS modeling, the **Bounded Central Differencing** scheme (available in the **Solution Methods** task page) is recommended for momentum discretization, and is the default setting.

Furthermore, the y^+-insensitive wall treatment of the underlying turbulence model is used. For details on the available options, see y+-Insensitive Near-Wall Treatment for ω-based Turbulence Models in the *Fluent Theory Guide*.

15.12. Setting Up the Detached Eddy Simulation Model

When using the Detached Eddy Simulation (DES) model, the following underlying RANS turbulence models can be used:

- Spalart-Allmaras
- Realizable k-ε
- SST k-ω
- BSL k-ω
- Transition SST

Note that for the DES model, the **Bounded Central Differencing** scheme (available in the **Solution Methods** task page) is recommended for momentum discretization, and is the default setting.

For additional information, see the following sections:

15.12.1. Setting Up DES with the Spalart-Allmaras Model
15.12.2. Setting Up DES with the Realizable k-ε Model
15.12.3. Setting Up DES with the SST k-ω Model
15.12.4. Setting Up DES with the BSL k-ω Model
15.12.5. Setting Up DES with the Transition SST Model

15.12.1. Setting Up DES with the Spalart-Allmaras Model

To set up a Detached Eddy Simulation with the Spalart-Allmaras model, select **Detached Eddy Simulation (DES)** from the **Model** list in the **Viscous Model** dialog box and then select **Spalart-Allmaras** from the **RANS Model** list.

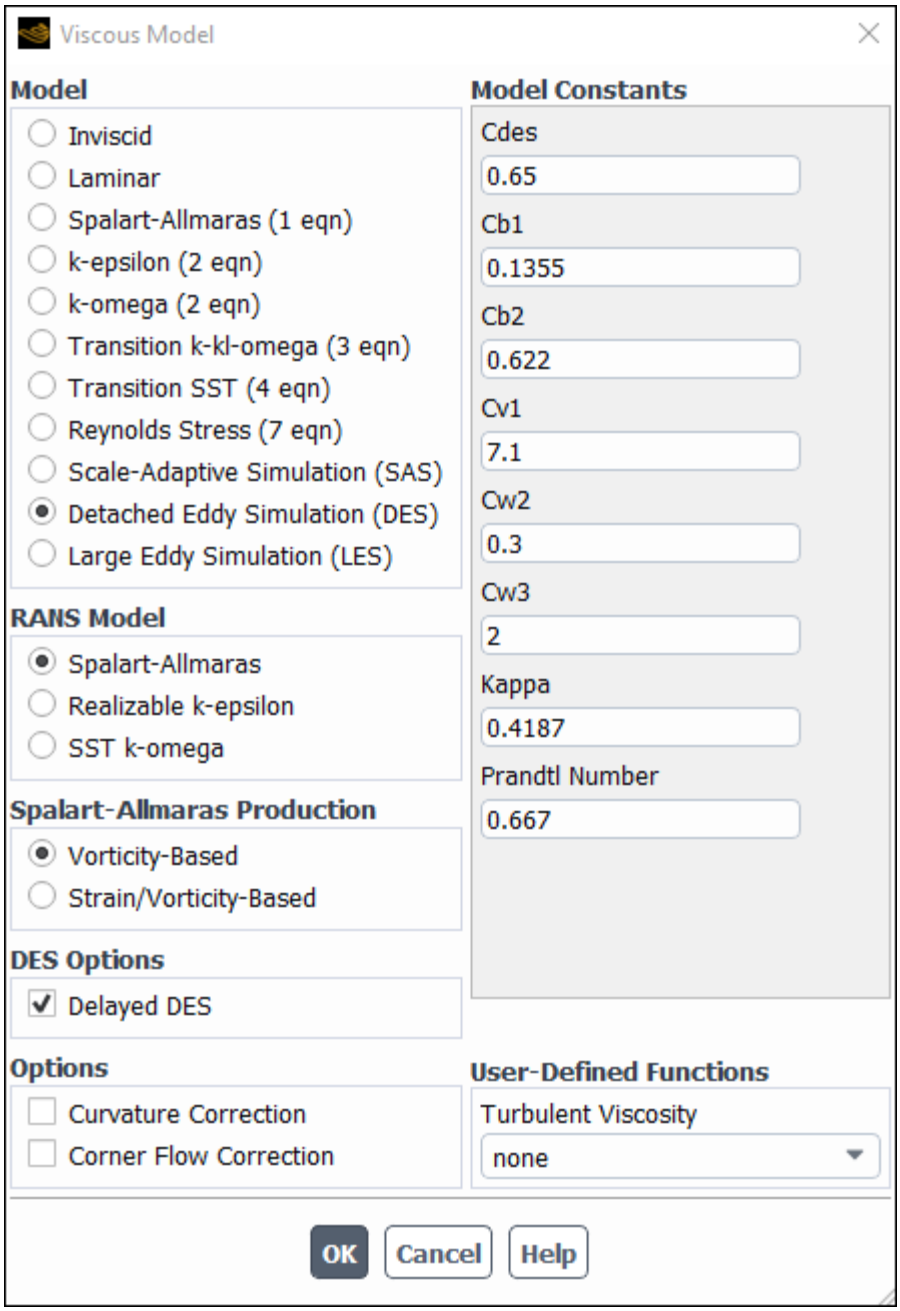
Ansys Fluent uses Equation 4.271 (in the Theory Guide) to compute the value of the length scale $\tilde{d}$ for the Spalart-Allmaras model. By default, the empirical constant C_{des} is set to 0.65. You can change its value in the **Cdes** field under **Model Constants**. The following options are available for this model:

- vorticity-based production (Vorticity- and Strain/Vorticity-Based Production (p. 2135))
- strain/vorticity-based production (Vorticity- and Strain/Vorticity-Based Production (p. 2135))
- delayed DES (Delayed Detached Eddy Simulation (DDES) (p. 2135))
- viscous heating (always enabled for the density-based solvers) (Including the Viscous Heating Effects (p. 2129))

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- inclusion of curvature correction (Including the Curvature Correction for the Spalart-Allmaras and Two-Equation Turbulence Models (p. 2131))

Figure 15.22: The Viscous Model Dialog Box Displaying Options for DES with the Spalart-Allmaras Model



Additionally, you can perform the following DES-specific function by using the `/define/models/viscous/detached-eddy-simulation?` text command:

- Modify only the length scales that appear in the destruction term in ν_t equation (the default is to modify all length scales within the ν_t equation)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15.12.2. Setting Up DES with the Realizable k-ε Model

To set up a Detached Eddy Simulation with the Realizable k-ε model, select **Detached Eddy Simulation (DES)** from the **Model** list in the **Viscous Model** dialog box and then select **Realizable k-epsilon** from the **RANS Model** list.

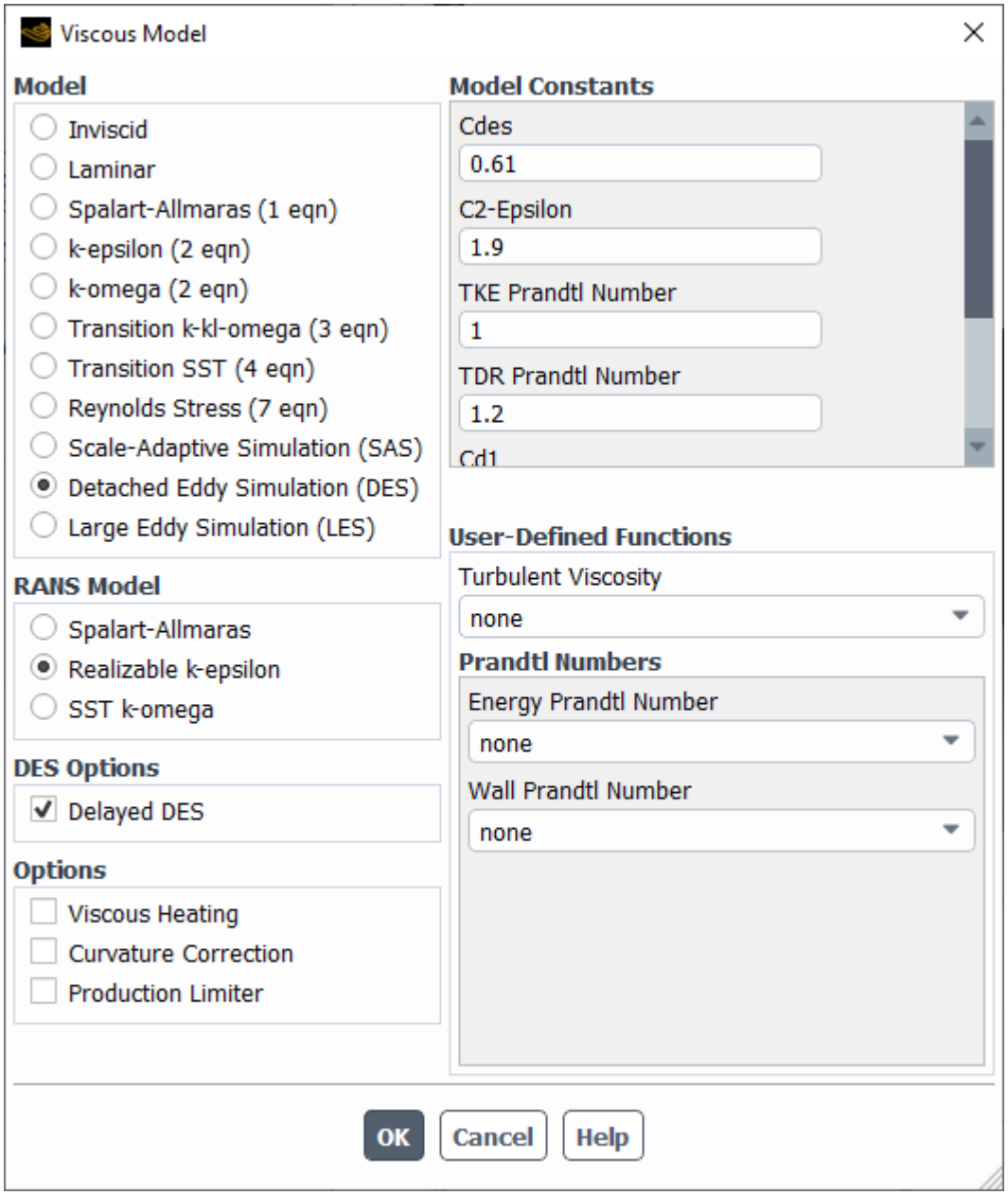
The model-specific options for DES with the Realizable k-ε model are the following:

- delayed DES (Delayed Detached Eddy Simulation (DDES) (p. 2135))
- viscous heating (always enabled for the density-based solvers) (Including the Viscous Heating Effects (p. 2129))
- inclusion of compressibility effects (Including the Compressibility Effects Option (p. 2134))

The model constant C_{des} is set to 0.61 for the Realizable k-ε model (see DES with the Realizable k-ε Model in the Theory Guide). The enhanced wall treatment (EWT-ε) is always enabled for DES with the Realizable k-ε model (see Enhanced Wall Treatment ε-Equation (EWT-ε) in the Theory Guide).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.23: The Viscous Model Dialog Box Displaying Options for DES with the Realizable k-ε Model



15.12.3. Setting Up DES with the SST k-ω Model

To set up a Detached Eddy Simulation with the SST k-ω model, select **Detached Eddy Simulation (DES)** from the **Model** list in the **Viscous Model** dialog box and then select **SST k-omega** from the **RANS Model** list.

The model-specific options that you can select for DES with the SST k-ω model are the following:

- low-Re corrections k-ω option (Low-Re Corrections (p. 2136))
- near wall treatment (y+-Insensitive Near-Wall Treatment for ω-based Turbulence Models in the *Fluent Theory Guide*)

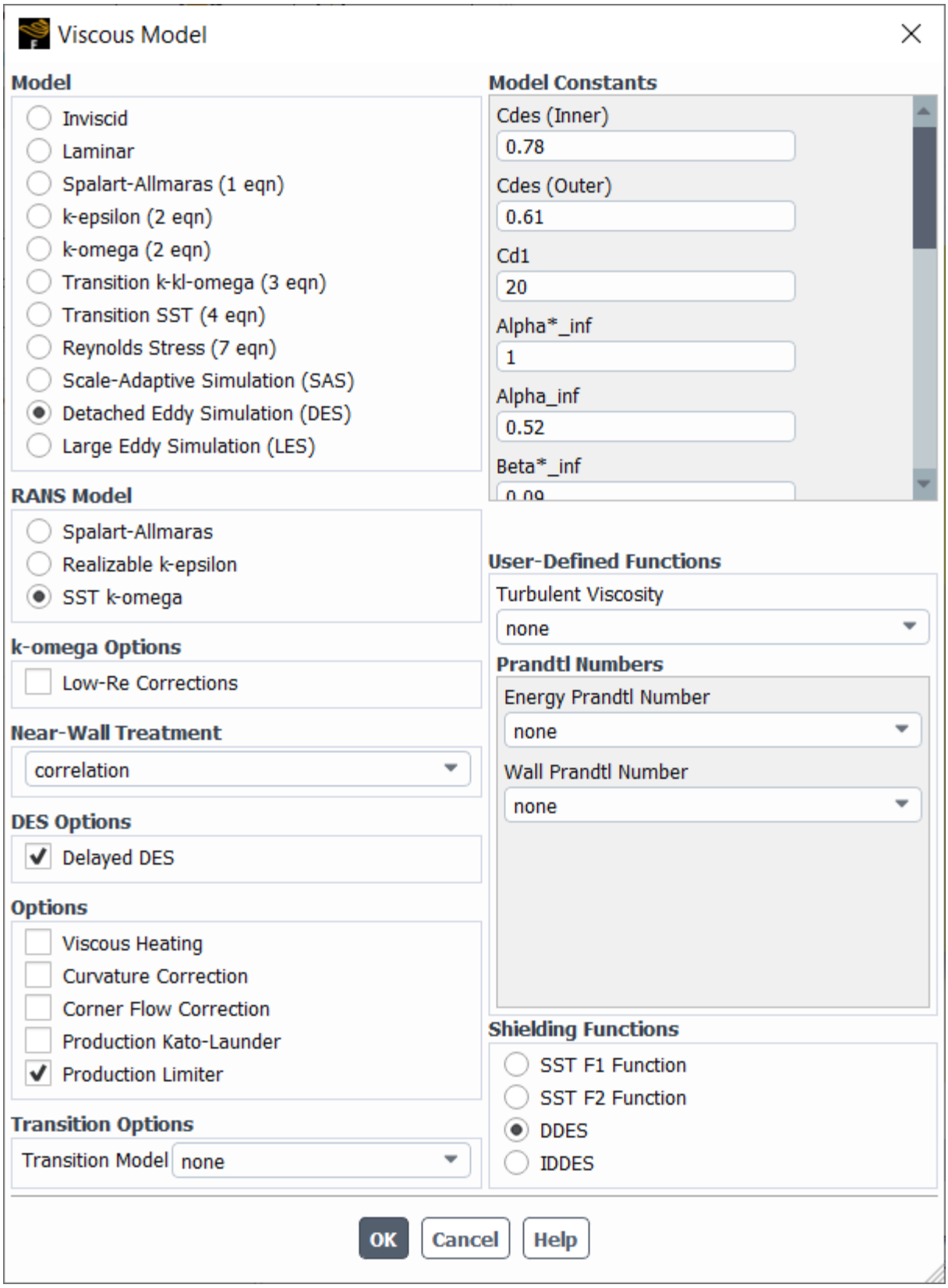
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- delayed DES (Delayed Detached Eddy Simulation (DDES) (p. 2135))
- viscous heating (always enabled for the density-based solvers) (Including the Viscous Heating Effects (p. 2129))
- inclusion of curvature correction (Including the Curvature Correction for the Spalart-Allmaras and Two-Equation Turbulence Models (p. 2131))
- inclusion of production limiters (Including Production Limiters for Two-Equation Models (p. 2134))
- inclusion of the Intermittency Transition Model
- Shielding functions—F1, F2, DDES (Delayed DES) and IDDES (Improved Delayed DES)—are available when the delayed DES option is enabled (Shielding Functions for the BSL / SST / Transition SST Detached Eddy Simulation Model (p. 2143))

The model constant C_{des} is set to 0.61 for the SST k-ω RANS model (see DES with the BSL or SST k-ω Model in the Theory Guide).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.24: The Viscous Model Dialog Box Displaying Options for DES with the SST k-ω Model



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15.12.4. Setting Up DES with the BSL k-ω Model

To set up a Detached Eddy Simulation with the BSL k-ω model for a transient case, select **k-omega** from the **Model** list in the **Viscous Model** dialog box, select **BSL** from the **k-omega Model** list, and enable the **Detached Eddy Simulation (DES)** option in the **Scale-Resolving Simulation Options** group box.

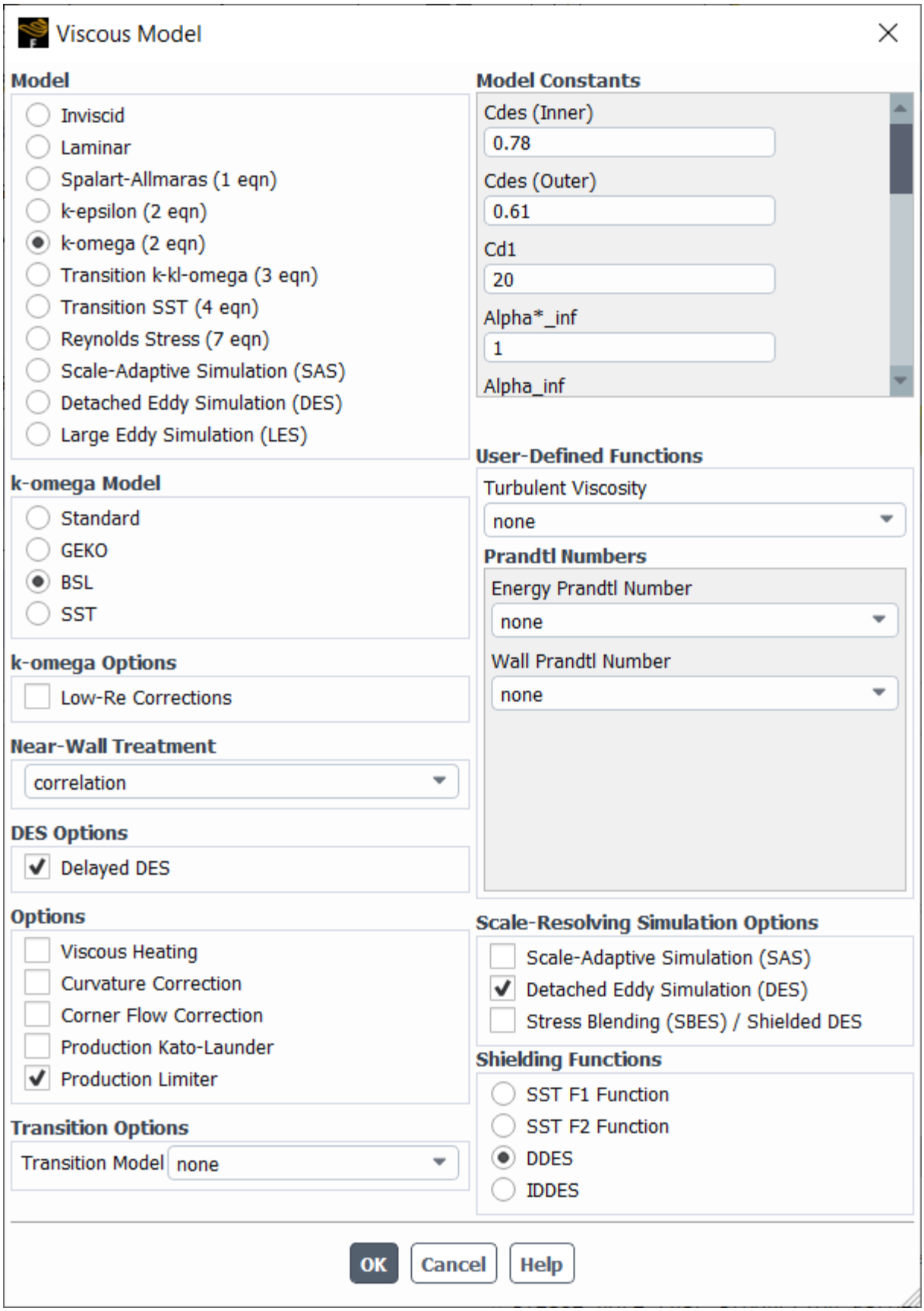
The model-specific options that you can select for DES with the BSL k-ω model are the following:

- low-Re corrections k-ω option (Low-Re Corrections (p. 2136))
- near wall treatment (y+-Insensitive Near-Wall Treatment for ω-based Turbulence Models in the *Fluent Theory Guide*)
- delayed DES (Delayed Detached Eddy Simulation (DDES) (p. 2135))
- viscous heating (always enabled for the density-based solvers) (Including the Viscous Heating Effects (p. 2129))
- inclusion of curvature correction (Including the Curvature Correction for the Spalart-Allmaras and Two-Equation Turbulence Models (p. 2131))
- inclusion of compressibility effects (Including the Compressibility Effects Option (p. 2134))
- inclusion of production limiters (Including Production Limiters for Two-Equation Models (p. 2134))
- inclusion of the Intermittency Transition Model
- Shielding functions—F1, F2, DDES (Delayed DES) and IDDES (Improved Delayed DES)—are available when the delayed DES option is enabled (Shielding Functions for the BSL / SST / Transition SST Detached Eddy Simulation Model (p. 2143))

The model constant C_{des} is set to 0.61 for the BSL k-ω RANS model (see DES with the BSL or SST k-ω Model in the Theory Guide).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.25: The Viscous Model Dialog Box Displaying Options for DES with the BSL k-ω Model



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

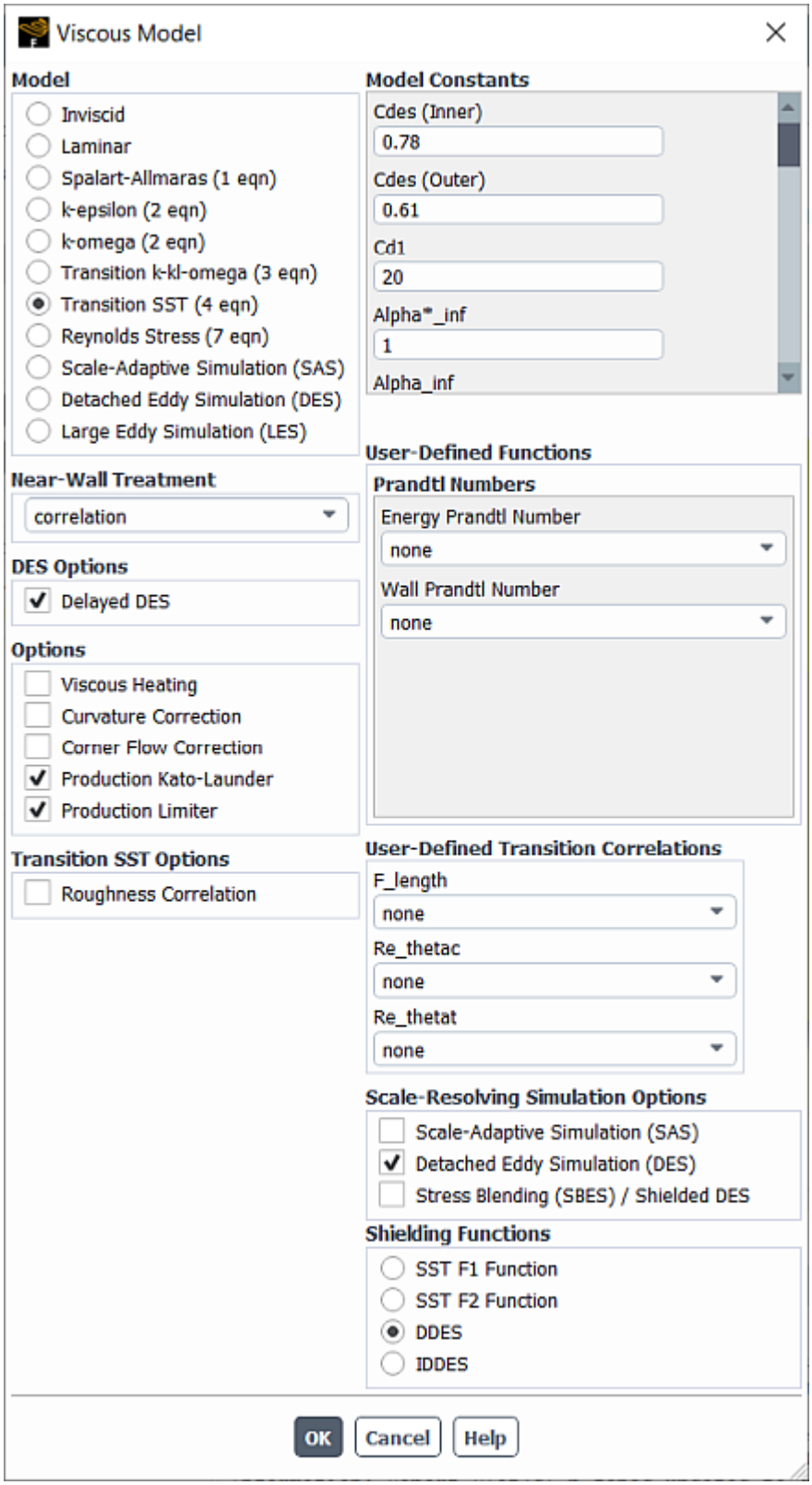
15.12.5. Setting Up DES with the Transition SST Model

To set up a Detached Eddy Simulation with the Transition SST model for a transient case, select **Transition SST** from the **Model** list in the **Viscous Model** dialog box and then enable the **Detached Eddy Simulation (DES)** option in the **Scale-Resolving Simulation Options** group box.

For DES with the Transition SST model, you can choose to enable the **Delayed DES** option in the **DES Options** group box (see Delayed Detached Eddy Simulation (DDES) (p. 2135) for details). When this option is enabled you can make a selection from the **Shielding Functions** list, as described in Shielding Functions for the BSL / SST / Transition SST Detached Eddy Simulation Model (p. 2143).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.26: The Viscous Model Dialog Box Displaying Options for DES with the Transition SST Model



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that the y+-insensitive wall treatment of the underlying turbulence model is used. For details on the available options, see y+-Insensitive Near-Wall Treatment for ω-based Turbulence Models in the *Fluent Theory Guide*.

15.13. Setting Up the Large Eddy Simulation Model

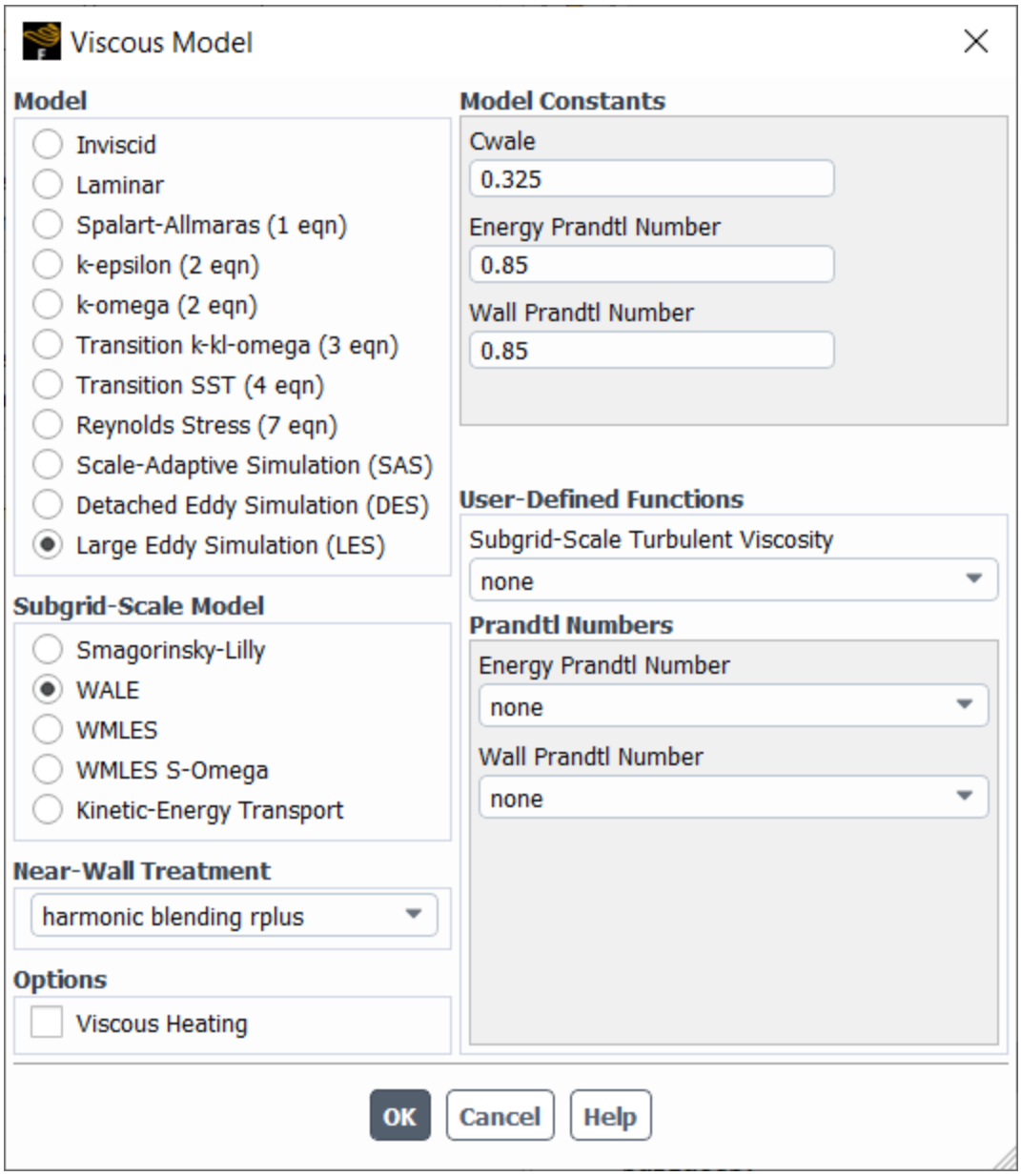
If you choose the LES model, the following subgrid-scale submodels are available (Subgrid-Scale Model (p. 2138)):

- **Smagorinsky-Lilly**
- **WALE**
- **WMLES**
- **WMLES S-Omega**
- **Kinetic-Energy Transport**

The default **Subgrid-Scale Model** for the LES model is the **WALE** model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.27: The Viscous Model Dialog Box Displaying the Large Eddy Simulation Model Options



The LES options that are available for the **Smagorinsky-Lilly** submodel are

- **Dynamic Stress**
- **Dynamic Energy Flux** (available only when the **Dynamic Stress** option is enabled)
- **Dynamic Scalar Flux** (available only when the **Dynamic Stress** option is enabled)

The LES options that are available when the **Kinetic-Energy Transport** submodel is selected are **Dynamic Energy Flux** and **Dynamic Scalar Flux**.

The **Dynamic Fvar** option is available for all of the subgrid-scale submodels when **Non-Premixed Combustion** or **Partially Premixed Combustion** is selected in the Species Model Dialog Box (p. 4768). This option enables the dynamic mixture fraction variance model. See The Non-Premixed Model for LES in the Theory Guide for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following **Near-Wall Treatment** options are available:

- **Kader blending Wall Functions** (current default option)
- **Harmonic blending Wall Functions based on r+** (recommended option)

  The corresponding text command to activate is: `define/models/viscous/near-wall-treatment/harmonic-blend-rplus-wf?`

- **Werner-Wengle Wall Treatment**

  The corresponding text command to activate is: `define/models/viscous/near-wall-treatment/werner-wengle-wall-fn?`

For more information about the near-wall treatments for the LES model, see LES Near-Wall Treatment in the *Fluent Theory Guide*.

15.14. Model Constants

It is also possible to modify the **Model Constants**, but this is not necessary for most applications. For more information about the constants, see Spalart-Allmaras Model through Large Eddy Simulation (LES) Model (in the Theory Guide). Note that **C1-PS** and **C2-PS** are the constants C_1 and C_2 in the linear pressure-strain approximation of Equation 4.231 and Equation 4.232 (in the Theory Guide), and **C1'-PS** and **C2'-PS** are the constants $C_1^{'}$ and $C_2^{'}$ in Equation 4.233 (in the Theory Guide). **C1-SSG-PS**, **C1'-SSG-PS**, **C2-SSG-PS**, **C3-SSG-PS**, **C3'-SSG-PS**, **C4-SSG-PS**, and **C5-SSG-PS** are the constants C_1, C_1^*, C_2, C_3, C_3^*, C_4, and C_5 in the quadratic pressure-strain approximation of Equation 4.242 (in the Theory Guide).

15.15. Setting Up the Embedded Large Eddy Simulation (ELES) Model

As described in Embedded Large Eddy Simulation (ELES) in the Theory Guide, the Embedded Large Eddy Simulation model is used when modeling a smaller embedded LES zone within a larger RANS computational domain. Recall that the interface between the upstream RANS zone and the LES zone must be defined (by assigning the interface to a velocity fluctuation algorithm). In addition, the interface between the LES zone and the downstream RANS zone must be considered.

This section describes how to set up the Embedded Large Eddy Simulation model.

1. In the **Viscous Model** dialog box, enable any RANS model (for example, k-ε, k-ω), or you can select DES, SAS, SDES, or SBES. The only RANS model not compatible with ELES is the Spalart-Allmaras model, as a one-equation model cannot provide the required turbulent length scale to the interface method.

   If a RANS model is selected, the model is applied globally to the computational domain, however, values for turbulence variables are frozen within the ELES region. The frozen state of the ELES zone is used to determine the flow conditions (for k-ε, k-ω, and so on) at the downstream LES-RANS zone interface. Note that this approach requires a fairly well converged global RANS solution to start with.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If DES, SAS, SDES, or SBES is used in the outer zone, these models are not frozen, but run in the background in the ELES region, obtaining proper flow conditions (for k-ε, k-ω, etc.) at the downstream LES-RANS zone interface.

2. For the specified fluid cell zone, enable **LES Zone** in the **Fluid** dialog box (Figure 15.28: Specifying an ELES Zone in the Fluid Dialog Box (p. 2126)). This enables the **Embedded LES** tab in the **Fluid** dialog box.

   Note:

   When DES, SAS, SDES, or SBES is used for the global model, this and the next step can, but need not, be skipped. (Remember the general limitations of DES in free, wall-independent flows.) If you choose to skip these steps, then proceed with step 4.

   The **ELES Zone** option will only appear in the **Fluid** dialog box if you select the **Transient Solver**. This must be done manually for RANS models.

3. In the **Embedded LES** tab, you can then specify the **Embedded Subgrid-Scale Model**, and the **Momentum Spatial Discretization**.

   For the **Embedded Subgrid-Scale Model**, the following subgrid-scale submodels are available (Subgrid-Scale Model (p. 2138)):

   - **Smagorinsky-Lilly**
   - **WALE**
   - **Dynamic Smagorinsky** (which is the same as the LES model with **Smagorinsky-Lilly** selected and the **Dynamic Stress** option enabled)
   - **WMLES**
   - **WMLES S-Omega**

   The default **Embedded-Subgrid Scale Model** for the ELES model is the **WALE** model.

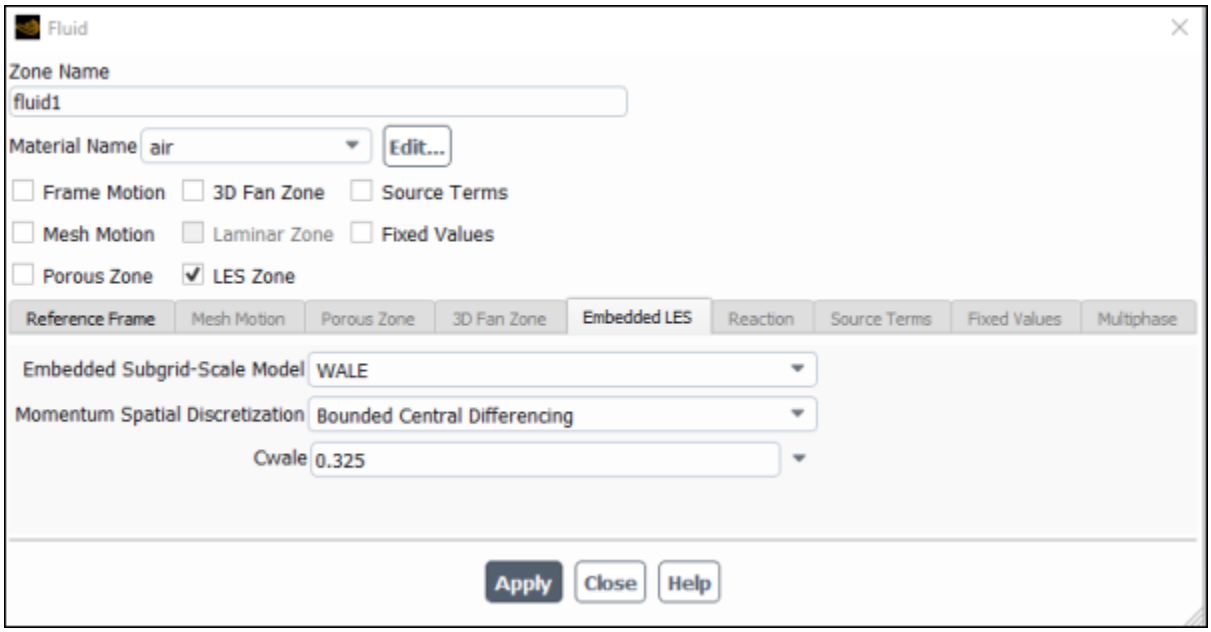
   When the WALE model is selected, you can specify a value for **Cwale** to define C_w (see Wall-Adapting Local Eddy-Viscosity (WALE) Model in the Theory Guide for details). Likewise, when the Smagorinsky-Lilly model is selected, you can specify a value for **Cs** to define C_s (see Smagorinsky-Lilly Model in the Theory Guide for details).

   For the **Momentum Spatial Discretization**, the following options are available:

   - **Bounded Central Differencing**
   - **Central Differencing**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

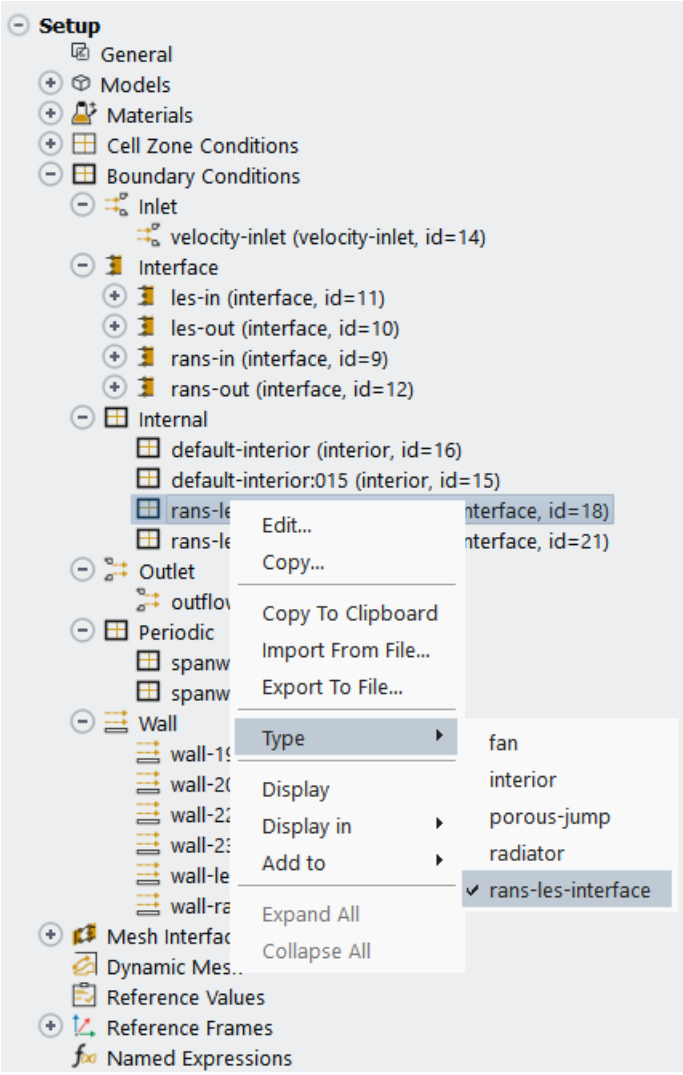
Figure 15.28: Specifying an ELES Zone in the Fluid Dialog Box



4. Select an appropriate interior interface and designate it as the RANS-LES interface, by right-clicking the boundary zone name in the tree (under **Setup/Boundary Conditions**) and selecting **rans-les-interface** in the **Type** sub-menu.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.29: Specifying the RANS/LES Interface



The **RANS/LES Interface** dialog box (Figure 15.30: The RANS/LES Interface Dialog Box (p. 2128)) will open, allowing you to assign a **Zone Name**, as well as the **Fluctuating Velocity Algorithm** and the **Number Of Vortices**. The options for the **Fluctuating Velocity Algorithm** are:

- **No Perturbations**
- **Spectral Synthesizer**
- **Vortex Method**

For more information about these options, refer to Inlet Boundary Conditions for Scale Resolving Simulations in the Theory Guide.

Note that the **Number Of Vortices** is the amount of vortices that the selected method distributes randomly over the face zone and uses to generate turbulent fluctuations. The value should be

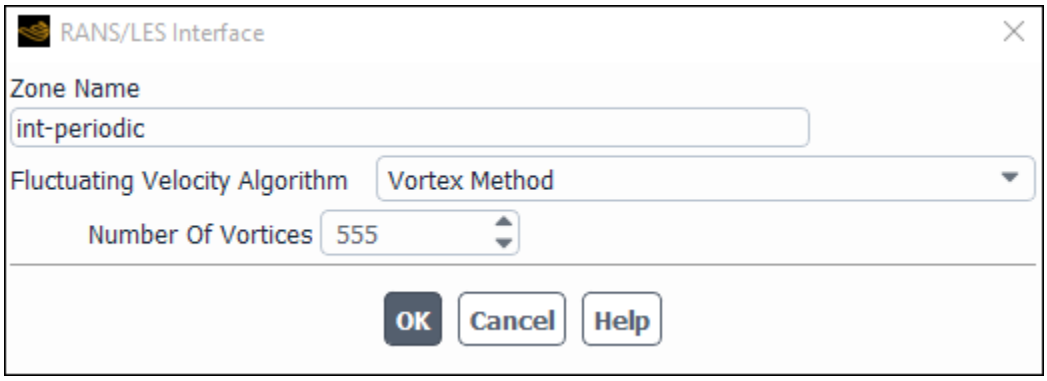
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

large enough to make sure there are no spots on the face zone that are unaffected by any vortex. Large numbers may slightly increase the CPU effort, but will not impair the results.

Note:

The minimum advised number of vortices is approximately a quarter of the number of cell faces at the LES side of the interface.

Figure 15.30: The RANS/LES Interface Dialog Box



If the RANS/LES interface is a non-conformal mesh interface, you can find the name of the interface interior zone that you want to change into a **rans-les-interface** zone by using the **List** button in the Mesh Interfaces Dialog Box (p. 5525).

Important:

It is recommended that the RANS-LES interface be situated in a region where there is no backflow.

15.16. Setup Options for All Turbulence Modeling

For more information about the various options available for the turbulence models, see Spalart-Allmaras Model through Large Eddy Simulation (LES) Model (in the Theory Guide). Instructions for activating these options are provided here.

For additional information, see the following sections:

15.16.1. Including the Viscous Heating Effects
15.16.2. Including Buoyancy Effects on Turbulence
15.16.3. Including the Curvature Correction for the Spalart-Allmaras and Two-Equation Turbulence Models
15.16.4. Including Corner Flow Correction
15.16.5. Including the Compressibility Effects Option
15.16.6. Including Production Limiters for Two-Equation Models
15.16.7. Vorticity- and Strain/Vorticity-Based Production
15.16.8. Delayed Detached Eddy Simulation (DDES)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15.16.9. Differential Viscosity Modification
15.16.10. Swirl Modification
15.16.11. Low-Re Corrections
15.16.12. Shear Flow Corrections
15.16.13. Turbulence Damping
15.16.14. Including Pressure Gradient Effects
15.16.15. Including Thermal Effects
15.16.16. Including the Wall Reflection Term
15.16.17. Solving the k Equation to Obtain Wall Boundary Conditions
15.16.18. Quadratic Pressure-Strain Model
15.16.19. Stress-Omega and Stress-BSL Models
15.16.20. Subgrid-Scale Model
15.16.21. Customizing the Turbulent Viscosity
15.16.22. Customizing the Turbulent Prandtl and Schmidt Numbers
15.16.23. Modeling Turbulence with Non-Newtonian Fluids
15.16.24. Including Scale-Adaptive Simulation with ω-Based URANS Models
15.16.25. Including Detached Eddy Simulation with the Transition SST Model
15.16.26. Including the SDES or SBES Model with RANS Models
15.16.27. Shielding Functions for the BSL / SST / Transition SST Detached Eddy Simulation Model

15.16.1. Including the Viscous Heating Effects

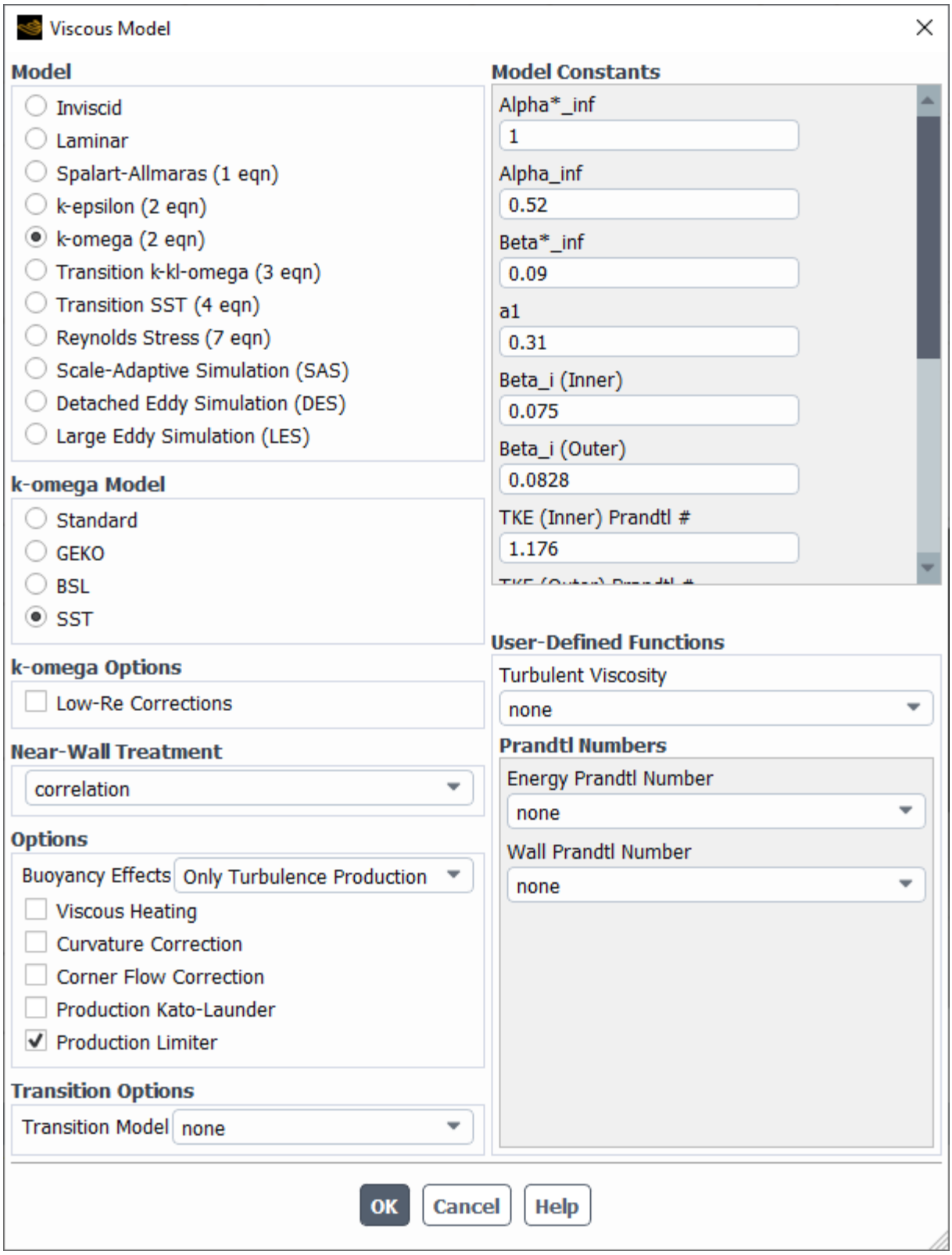
For information about including viscous heating effects in your model, see Inclusion of the Viscous Dissipation Terms in the Theory Guide and Solving Heat Transfer Problems (p. 2164).

15.16.2. Including Buoyancy Effects on Turbulence

If you specify a nonzero gravity force (in the Operating Conditions Dialog Box (p. 4978)), and you are modeling a non-isothermal flow, buoyancy effects can be included in the transport equations of the specified turbulence model as shown in Figure 15.31: SST Model with the **Buoyancy Effects: Only Turbulence Production** Option Enabled (p. 2130).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.31: SST Model with the Buoyancy Effects: Only Turbulence Production Option Enabled



The following options are available:

- **Buoyancy Effects: Off**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This option neglects buoyancy effects on turbulence.

- **Buoyancy Effects: Only Turbulence Production**

  This option is the default setting and includes buoyancy effects in the transport equations of the turbulent kinetic energy or Reynolds stresses. For turbulence models based on the transport equation of the specific dissipation rate, ω, this also includes the part in the ω-equation which represents the contribution from the k-equation (see Effects of Buoyancy on Turbulence in the k-ω Models in the *Fluent Theory Guide*).

- **Buoyancy Effects: Full**

  This option activates the full buoyancy model. For turbulence models based on the dissipation rate ε, it also includes the buoyancy term in the ε-equation. For models based on the specific dissipation rate ω, it includes the part in the ω-equation which represents the contribution from the ε-equation as well as that from the k-equation (see Effects of Buoyancy on Turbulence in the k-ω Models in the *Fluent Theory Guide*).

Buoyancy effects can be included in the following turbulence models:

- Standard, RNG and Realizable k-ε Models
- Standard, BSL and SST k-ω Models (also in combination with SRS methods or the Intermittency Transition Model)
- Generalized k-ω Model (also in combination with SRS methods or the Intermittency Transition Model)
- Transition SST
- Reynolds Stress Models

15.16.3. Including the Curvature Correction for the Spalart-Allmaras and Two-Equation Turbulence Models

Eddy-viscosity models display an insensitivity to streamline curvature and system rotation, which play a significant role in many turbulent flows of practical interest, as described in Curvature Correction for the Spalart-Allmaras and Two-Equation Models in the Theory Guide. A modification to the turbulence production term is available to sensitize the following standard eddy-viscosity models to the effects of streamline curvature and system rotation:

- Spalart-Allmaras one-equation model
- Standard, RNG, and Realizable (k-ε) models
- Standard (k-ω), BSL, SST, and Transition SST
- Scale-Adaptive Simulation (SAS)
- Detached Eddy Simulation with BSL (DES-BSL), with SST (DES-SST), with Spalart-Allmaras (DES-SA), and with Realizable (k-ε) model (DES-rke).
- Shielded Detached Eddy Simulation (SDES)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Stress-Blended Eddy Simulation (SBES)

Note that both the RNG and Realizable (k-ε) turbulence models already have their own terms to include rotational or swirl effects (see RNG Swirl Modification and Modeling the Turbulent Viscosity in the Theory Guide for more information). The curvature correction option should therefore be used with caution for these two models and is offered mainly for completeness for RNG and Realizable (k-ε).

Enable the **Curvature Correction** option under **Options** in the Viscous Model Dialog Box (p. 4715).

You can specify a value for **CCURV** under **Curvature Correction Options** to influence the strength of the curvature correction if needed for a specific flow. You have the option of specifying a constant, expression, or UDF.

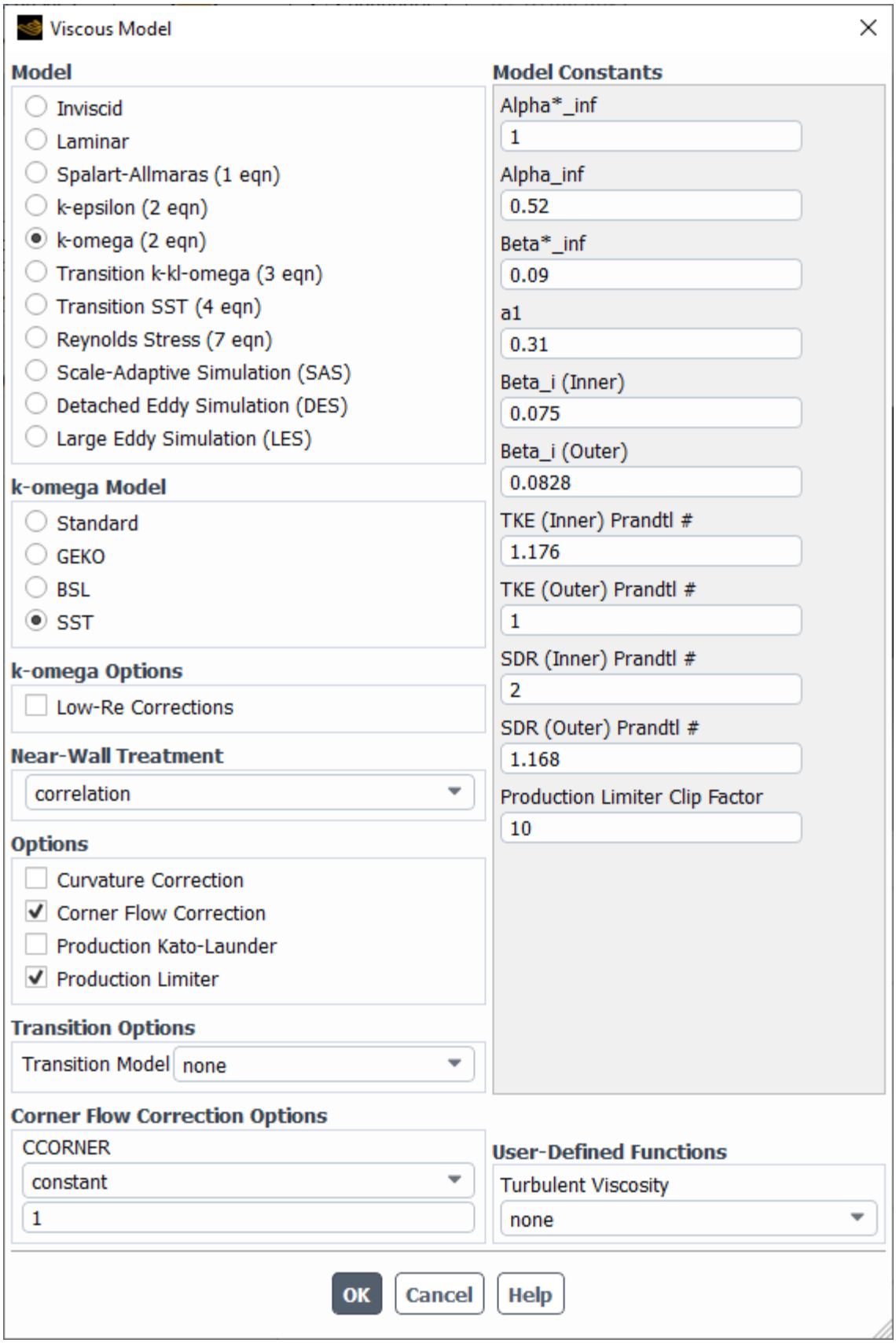
15.16.4. Including Corner Flow Correction

Turbulent flows through rectangular channels, pipes of non-circular cross-section, and wing-body-junction type geometries exhibit secondary flows in the plane normal to the main flow direction into the corner along the bisector. To account for these secondary flows, you can enable **Corner Flow Correction** in the **Options** group box of the **Viscous Model** dialog box when you have selected one of the following turbulence models:

- Spalart-Allmaras (SA,SA-DES)
- Standard k-ω model (and combination of this model with SAS)
- BSL, SST, GEKO, and Transition-SST models (and combination of these models with SAS, DES, and SBES)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.32: The Viscous Model Dialog Box with Corner Flow Correction option enabled



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Afterwards, the **Corner Flow Correction Options** group box appears and you can set the coefficient **CCORNER** as a constant value, expression, or UDF.

For a detailed description of this option, see Corner Flow Correction in the *Fluent Theory Guide*.

For an example of specifying this option with a UDF, see `DEFINE_CORNER_FLOW_CORRECTION_CCORNER` in the *Fluent Customization Manual*.

15.16.5. Including the Compressibility Effects Option

The **Compressibility Effects** option is available under **Options** in the Viscous Model Dialog Box (p. 4715) for most ε-based models, ω-based models, and Reynolds stress models when the compressible form of the ideal gas law or the real-gas model is enabled. This option can improve the prediction of free shear layers at high Mach numbers. See Model Enhancements (p. 2071) for recommendations on when to use this option. For details about how this correction is implemented, see Effects of Compressibility on Turbulence in the k-ε Models and Compressibility Effects in the Theory Guide.

15.16.6. Including Production Limiters for Two-Equation Models

A disadvantage of standard two-equation turbulence models is the excessive generation of the turbulence energy, G_k, in the vicinity of stagnation points. In order to avoid the buildup of turbulent kinetic energy in the stagnation regions, two formulations are available in order to limit the production term in the turbulence kinetic energy equations:

- Production Limiter (for details, see Equation 4.436 in the *Fluent Theory Guide*)
- Production Kato-Launder (for details, see Equation 4.440 in the *Fluent Theory Guide*)

Both these formulations can be accessed under **Options** in the **Viscous Model** dialog box. The **Production Limiter** model coefficient C_{lim} is called the **Production Limiter Clip Factor** and has a default value of 10. This value can be modified under **Model Constants** once the **Production Limiter** formulation is enabled.

You can use the following text commands to set the formulations:

- Production Limiter

  ```
  /define/models/viscous/turbulence-expert/production-limiter?[yes]
  ```

- Production Kato-Launder

  ```
  /define/models/viscous/turbulence-expert/kato-launder-model?[yes]
  ```

The **Production Limiter** is available for the following turbulence models:

- Standard, RNG, and Realizable (k-ε) models
- Standard (k-ω), BSL, SST, and Transition SST
- Scale-Adaptive Simulation (SAS)
- Detached Eddy Simulation with BSL (DES-BSL), with SST (DES-SST), and with Realizable (k-ε) models (DES-rke)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Shielded Detached Eddy Simulation (SDES)
- Stress-Blended Eddy Simulation (SBES)

By default, this limiter is enabled for all turbulence models based on the ω equation.

The **Production Kato-Launder** formulation for the production term is available for the following turbulence models:

- Standard, and RNG (k-ε) models
- Standard (k-ω), BSL, SST, and Transition SST
- Scale-Adaptive Simulation (SAS)
- Detached Eddy Simulation with BSL (DES-BSL) and with SST (DES-SST)
- Shielded Detached Eddy Simulation (SDES)
- Stress-Blended Eddy Simulation (SBES)

The Kato-Launder formulation is enabled by default for the Transition SST model only.

15.16.7. Vorticity- and Strain/Vorticity-Based Production

For the Spalart-Allmaras model, you can choose either **Vorticity-Based Production** or **Strain/Vorticity-Based Production** under **Spalart-Allmaras Production** in the Viscous Model Dialog Box (p. 4715). If you choose **Vorticity-Based Production**, Ansys Fluent will compute the value of the deformation tensor S using Equation 4.22 (in the Theory Guide); if you choose **Strain/Vorticity-Based Production**, it uses Equation 4.24 (in the Theory Guide).

(These options will not appear unless you have enabled the Spalart-Allmaras model.)

15.16.8. Delayed Detached Eddy Simulation (DDES)

The **Delayed DES** option is recommended when using the DES model. This option preserves the RANS model throughout the boundary layer. For more information, see Detached Eddy Simulation (DES) in the Theory Guide.

15.16.9. Differential Viscosity Modification

The RNG turbulence model in Ansys Fluent has an option of using a differential formula for the effective viscosity μ_{eff} (Equation 4.44 in the Theory Guide) to account for the low-Reynolds-number effects.

To enable this option, the **Differential Viscosity Model** option under **RNG Options** in the Viscous Model Dialog Box (p. 4715) must be enabled.

Important:

This option appears when you have enabled the RNG k-ε model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15.16.10. Swirl Modification

After you have chosen the RNG model, the swirl modification takes effect, by default, for all three-dimensional flows and axisymmetric flows with swirl. The default swirl constant (α_s in Equation 4.46 in the Theory Guide) is set to 0.07, which works well for weakly to moderately swirling flows. However, for strongly swirling flows, you may need to use a larger swirl constant.

To change the value of the swirl constant, you must first enable the **Swirl Dominated Flow** option under **RNG Options** in the Viscous Model Dialog Box (p. 4715).

Important:

This option will not appear unless you have enabled the RNG k-ε model.

15.16.11. Low-Re Corrections

If any of the k-ω models or the Stress-Omega RSM is used, you may enable a low-Reynolds-number correction to the turbulent viscosity by enabling the **Low-Re Corrections** option under **k-omega Options** in the **Viscous Model** dialog box. By default, this option is not enabled, and the damping coefficient (α^* in Equation 4.75 in the Theory Guide) is equal to 1.

15.16.12. Shear Flow Corrections

In the standard k-ω model and the Stress-Omega RSM, you also have the option of including corrections to improve the accuracy in predicting free shear flows. The **Shear Flow Corrections** option under the **k-omega Options** is enabled by default in the **Viscous Model** dialog box, as these corrections are included in the standard k-ω model [173] (p. 5723). When this option is enabled, Ansys Fluent will calculate f_β^* using Equation 4.85 (in the Theory Guide) and f_β using Equation 4.92 (in the Theory Guide). If this option is disabled, f_β^* and f_β will be set equal to 1.

15.16.13. Turbulence Damping

For cases that use the VOF or Mixture multiphase models, or the Eulerian multiphase model with the Multi-Fluid VOF option selected, you can enable **Turbulence Damping** (in the **Turbulence Damping Options** group box). This option is required for better resolution of velocity gradients in the vicinity of fluid-fluid interface. After enabling **Turbulence Damping**, you can specify the **Damping Factor**, which by default is set to 10. For a theoretical discussion about turbulence damping, refer to Turbulence Damping in the *Fluent Theory Guide*.

Note:

Incorporation of turbulence damping is recommended only for sharp and sharp/dispersed interface modeling types.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15.16.14. Including Pressure Gradient Effects

If the enhanced wall treatment is used, you may include the effects of pressure gradients by enabling the **Pressure Gradient Effects** option under the **Enhanced Wall Treatment Options**. When this option is enabled, Ansys Fluent will include the coefficient α in Equation 4.372 (in the Theory Guide).

15.16.15. Including Thermal Effects

If the enhanced wall treatment is used, you may include thermal effects by enabling the **Thermal Effects** option under **Enhanced Wall Treatment Options**. When this option is enabled, Ansys Fluent will include the coefficient β in Equation 4.372 (in the Theory Guide). γ will also be included in Equation 4.372 when the **Thermal Effects** option is enabled if the ideal gas law is selected for the fluid density in the **Create/Edit Materials** dialog box.

15.16.16. Including the Wall Reflection Term

If the RSM is used with the default model for linear pressure-strain, Ansys Fluent will, by default, include the wall-reflection effects in the pressure-strain term. That is, Ansys Fluent will calculate $\phi_{ij,w}$ using Equation 4.233 (in the Theory Guide) and include it in Equation 4.230 (in the Theory Guide). For the quadratic pressure-strain model, Stress-Omega model, and Stress-BSL model, the wall-reflection effects are not required and are not included.

Important:

The empirical constants and the function f used in the calculation of $\phi_{ij,w}$ are calibrated for simple canonical flows such as channel flows and flat-plate boundary layers involving a single wall. If the flow involves multiple walls and the wall has significant curvature (for example, an axisymmetric pipe or curvilinear duct), the inclusion of the wall-reflection term in Equation 4.233 (in the Theory Guide) may not improve the accuracy of the RSM predictions. In such cases, you can disable the wall-reflection effects by turning off the **Wall Reflection Effects** under **Reynolds-Stress Options** in the Viscous Model Dialog Box (p. 4715).

15.16.17. Solving the k Equation to Obtain Wall Boundary Conditions

For the ε-based Reynolds stress models, by default Ansys Fluent uses the explicit setting of boundary conditions for the Reynolds stresses near the walls, with the values computed with Equation 4.260 (in the Theory Guide). The turbulent kinetic energy, k, is calculated by solving the k equation obtained by summing Equation 4.226 (in the Theory Guide) for normal stresses. To disable this option and use the wall boundary conditions given in Equation 4.261 (in the Theory Guide), turn off the **Wall BC from k Equation** under the **Reynolds-Stress Options** in the **Viscous Model** dialog box.

Important:

This option will not appear unless you have enabled the **Reynolds Stress** model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15.16.18. Quadratic Pressure-Strain Model

To use the quadratic pressure-strain model described in Quadratic Pressure-Strain Model (in the Theory Guide), enable the **Quadratic Pressure-Strain Model** option under **Reynolds-Stress Options** in the Viscous Model Dialog Box (p. 4715). (This option will not appear unless you have enabled the RSM.) The following options are not available when the **Quadratic Pressure-Strain Model** is enabled:

- **Wall Reflection Effects** under **Reynolds-Stress Options**
- **Enhanced Wall Treatment** under **Near-Wall Treatment**

15.16.19. Stress-Omega and Stress-BSL Models

To use the Stress-Omega model (described in Stress-Omega Model) or the Stress-BSL model (described in Stress-BSL Model), select **Stress-Omega** or **Stress-BSL**, respectively, from the **Reynolds-Stress Model** list in the Viscous Model Dialog Box (p. 4715). (These options will not appear unless you have enabled the RSM.) The following options are not available when the **Stress-Omega** or the **Stress-BSL** model is selected:

- **Wall BC from k Equation** under **Reynolds-Stress Options**
- **Wall Reflection Effects** under **Reynolds-Stress Options**
- **Standard Wall Functions** under **Near-Wall Treatment**
- **Scalable Wall Functions** under **Near-Wall Treatment**
- **Non-Equilibrium Wall Functions** under **Near-Wall Treatment**
- **Enhanced Wall Treatment** under **Near-Wall Treatment**

Instead, the following options are available:

- **Low-Re Corrections** under **k-omega Options** (for **Stress-Omega** only)
- **Shear Flow Corrections** under **k-omega Options** (for **Stress-Omega** only)
- **Scale-Adaptive Simulation (SAS)** under **Scale-Resolving Simulation Options**

15.16.20. Subgrid-Scale Model

If you have selected the **Large Eddy Simulation** model, you will be able to choose one of the subgrid-scale models described in Subgrid-Scale Models (in the Theory Guide). You can choose from the **Smagorinsky-Lilly**, **WALE**, **WMLES**, **WMLES S-Omega**, or **Kinetic-Energy Transport** subgrid-scale models. Note that **Dynamic Stress** is an option available with the **Smagorinsky-Lilly** model (and when it is enabled, this model is referred to as the "dynamic Smagorinsky" model), while the **Kinetic-Energy Transport** model is always run as a dynamic model. The **Dynamic Fvar** option is available

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

for all of the subgrid-scale models when **Non-Premixed Combustion** or **Partially Premixed Combustion** is selected in the Species Model Dialog Box (p. 4768).

Important:

These options will not appear unless you have enabled the LES model.

15.16.21. Customizing the Turbulent Viscosity

If you are using the Spalart-Allmaras, k-ε, k-ω, DES, or LES models, a UDF can be used to customize the turbulent viscosity. This option will enable you to modify μ_t in the case of the Spalart-Allmaras, k-ε, and k-ω models, and incorporate completely new subgrid models in the case of the LES model. SBES with BSL / SST allows the usage of a UDF in order to provide a custom formulation for μ_t in the RANS region, but not for the subgrid model in the LES region. More information about UDFs can be found in the Fluent Customization Manual.

In the Viscous Model Dialog Box (p. 4715), under **User-Defined Functions**, select the appropriate user-defined function in the **Turbulent Viscosity** drop-down list. For the LES model, select the appropriate UDF in the **Subgrid-Scale Turbulent Viscosity** drop-down list.

15.16.22. Customizing the Turbulent Prandtl and Schmidt Numbers

You can use UDFs to customize the turbulent Prandtl and Schmidt numbers. For the standard and realizable k-ε models and the standard k-ω model, your UDF can calculate the TKE Prandtl number (σ_k) and either the TDR or SDR Prandtl number (σ_ε or σ_ω), depending on whether you are using the k-ε or k-ω model. All turbulence models allow you to use UDFs for the turbulent Prandtl number at the wall (Pr_t in Equation 4.343 in the Theory Guide), and all turbulent models except for the RNG k-ε model allow you to use UDFs for the turbulent Prandtl number for energy (Pr_t in, for example, Equation 4.64 or Equation 4.251 in the Theory Guide), as well as the turbulent Schmidt number (Sc_t, in Equation 7.4 in the Theory Guide) if species transport has been enabled. Note that UDFs cannot be used for the turbulent Prandtl number for energy or the turbulent Schmidt number if you have enabled **Dynamic Energy Flux** or **Dynamic Scalar Flux**, respectively, as part of the LES model. More information about UDFs can be found in the Fluent Customization Manual.

In the Viscous Model Dialog Box (p. 4715), under **User-Defined Functions**, select the appropriate UDF from the drop-down lists in the **Prandtl Numbers** or **Prandtl and Schmidt Numbers** group box. The options will depend on the turbulence model you have selected (as noted previously), and include: **TKE Prandtl Number**, **TDR Prandtl Number**, **SDR Prandtl Number**, **Energy Prandtl Number**, **Wall Prandtl Number**, and **Turbulent Schmidt Number**. Note that the availability of these drop-down lists can be affected by your settings for the energy equation and the multiphase and combustion models.

15.16.23. Modeling Turbulence with Non-Newtonian Fluids

If the turbulent flow involves non-Newtonian fluids, you can use the `define/models/viscous/turbulence-expert/turb-non-newtonian?` text command to enable the selection of non-Newtonian options for the material viscosity. See Viscosity for Non-Newtonian Fluids (p. 1639) for details about these options.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15.16.24. Including Scale-Adaptive Simulation with ω-Based URANS Models

You have the option of including Scale-Adaptive Simulation (SAS) with most ω-based URANS models. For details about SAS and how to include it, see Scale-Adaptive Simulation (SAS) (p. 2074) and Setting Up Scale-Adaptive Simulation (SAS) Modeling (p. 2109), respectively.

15.16.25. Including Detached Eddy Simulation with the Transition SST Model

You have the option of including Detached Eddy Simulation (DES) with the Transition SST model. For details about DES and how to include it, see Detached Eddy Simulation (DES) (p. 2075) and Setting Up DES with the Transition SST Model (p. 2120), respectively.

15.16.26. Including the SDES or SBES Model with RANS Models

To enable a hybrid RANS-LES model with an improved shielding function compared to DDES / IDDES, you can enable the **Stress Blending (SBES) / Shielded DES** option in the **Scale-Resolving Simulation Options** group box of the **Viscous Model** dialog box. This option is available for 3D transient cases when you have selected one of the following models (see Figure 15.33: The Viscous Model Dialog Box with the SBES Options for ω-based RANS models (p. 2142)):

- Generalized k-ω (GEKO) model
- Baseline (BSL) k-ω model
- Shear-Stress Transport (SST) k-ω model
- Transition SST model
- Realizable k-ε model with scalable wall functions or enhanced wall treatment.

You can then select one of the following from the **Hybrid Model** list:

- **SDES** to apply only a shielding function (Shielded Detached Eddy Simulation (SDES) in the *Fluent Theory Guide*)
- **SBES** to apply a shielding function with stress blending (Stress-Blended Eddy Simulation (SBES) in the *Fluent Theory Guide*)
- **SBES with User-Defined Function** to define your own blending function zonally for cases where the division between RANS and LES is clear from the geometry and the flow physics (`DEFINE_SBES_BF` in the *Fluent Customization Manual*)

If you select **SBES** or **SBES with User-Defined Function** for the **Hybrid Model**, you can also select one of the following subgrid-scale (LES) models:

- **Smagorinsky-Lilly** model
- **Dynamic Smagorinsky**-Lilly model
- Wall-Adapting Local Eddy-Viscosity (**WALE**) model

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Algebraic Wall-Modeled LES S-Ω (**WMLES S-Omega**) model

The **WALE** model is recommended, as it provides the lowest eddy viscosity in 2D flow regions and thereby allows the flow to quickly develop 3D turbulence structures in separating shear layers. The combination of SBES with the **WMLES S-Omega** formulation is likely not a useful combination, as the SBES-WALE model combination can by itself act in a WMLES mode.

If you select **SBES** or **SBES with User-Defined Function** for the **Hybrid Model**, you can also specify an integer value (between 1 and 10) for **Update Interval k-omega**, which determines how often the k and ω equations are computed during a run. This option is not available for the Realizable k-ε model.

In many cases, the global time step size required for the LES portion of the calculation is unnecessarily small for the RANS portion. For this reason, it is not required that the k-ω part of the SBES model is updated after every time step. Reducing the frequency at which the k-ω equations are solved by specifying an update interval can positively impact the solution time, with minimal impact on the solution or convergence. An optimal value for the parameter is 5. Note that the momentum equations as well as the LES eddy-viscosity portion of the SBES model are updated at every time step. The parameter only reduces the number of updates of the k and ω equations, as well as that of the blending function.

For example, if you specify an update interval of 5, the solver would update the k-ω portion every 5th timestep, while still updating the LES eddy viscosity portion every timestep.

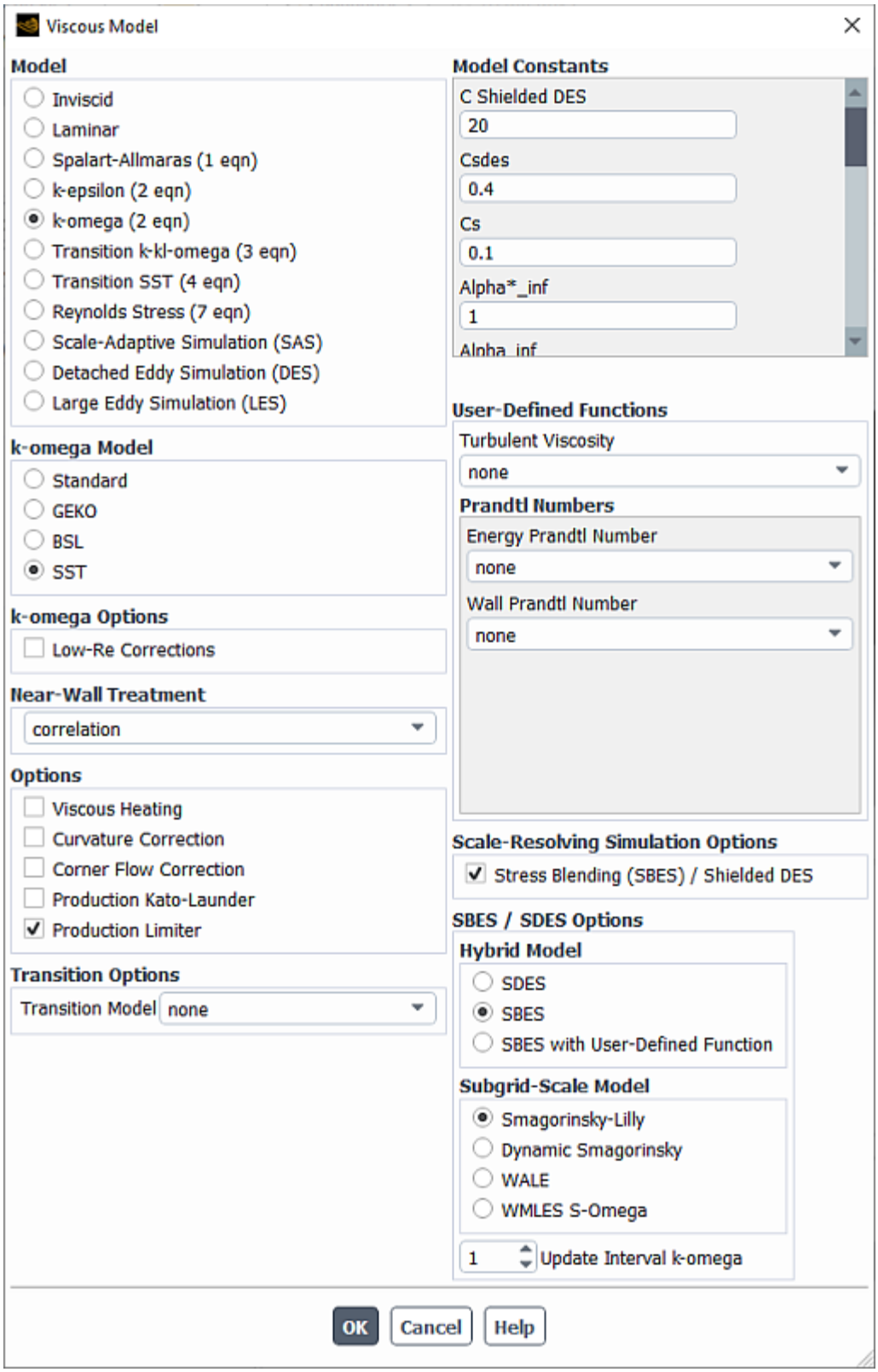
Note:

The **Update Interval k-omega** functionality has the following limitations:

- Only available for RANS models based on the ω-equation.
- Not compatible with multiphase flow.
- Not compatible with adaptive time-stepping.
- The time scheme for the k and ω equations are automatically set to first order in cases of variable density flow or moving meshes. The time scheme for other equations is not affected by this.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.33: The Viscous Model Dialog Box with the SBES Options for ω-based RANS models



Note:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For the SDES and SBES models, the **Bounded Central Differencing** scheme (available in the **Solution Methods** task page) is recommended for momentum discretization, and is the default setting.

15.16.27. Shielding Functions for the BSL / SST / Transition SST Detached Eddy Simulation Model

For the DES model with the BSL k-ω model, SST k-ω model, or the Transition SST model, the shielding (or blending) functions F1 and F2 are defined in Equation 4.108 and Equation 4.119 in the Theory Guide, respectively. Note that the F2 function is more conservative than the F1 function. In addition, the shielding functions DDES (Delayed DES) and IDDES (Improved Delayed DES), are described in DES with the BSL or SST k-ω Model and Improved Delayed Detached Eddy Simulation (IDDES) in the Theory Guide. DDES is recommended and is used as the default.

15.17. Defining Turbulence Boundary Conditions

For additional information, see the following sections:

15.17.1. Wall Roughness Effects

15.17.2. The Spalart-Allmaras Model

15.17.3. k-ε Models and k-ω Models

15.17.4. Reynolds Stress Model

15.17.5. Scale Resolving Simulations

15.17.1. Wall Roughness Effects

You may want to include the effects of the wall roughness on selected wall boundaries as part of your turbulent simulation. In such cases, you can specify the roughness parameters (roughness height and roughness constant) in the dialog boxes of the corresponding wall boundaries (see Wall Roughness Effects in Turbulent Wall-Bounded Flows (p. 1458)).

Note:

Rough walls cannot be used together with the following model combinations:

- an ε-equation model with enhanced wall treatment or the Menter-Lechner near-wall treatment

  Note that the following are the relevant ε-equation models:

  – all of the k-ε models (that is, standard, RNG, and realizable)

  – the Reynolds stress model with the **Linear Pressure-Strain** model selected

  – the detached eddy simulation (DES) model with the **Realizable k-epsilon** option selected

- the Reynolds stress model with the **Stress-Omega** or **Stress-BSL** model selected

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- the transition k-kl-ω model
- the large eddy simulation (LES) model

15.17.2. The Spalart-Allmaras Model

When you are modeling turbulent flows in Ansys Fluent using the Spalart-Allmaras model, you must provide the boundary conditions for $\tilde{\nu}$ in addition to other mean solution variables. The boundary conditions for $\tilde{\nu}$ at the walls are internally taken care of by Ansys Fluent, which obviates the need for your inputs. The boundary condition input for $\tilde{\nu}$, which you must enter in Ansys Fluent, is the one at inlet boundaries (for example, velocity inlets, pressure inlets). In many situations, it is important to specify correct or realistic boundary conditions at the inlets, because the inlet turbulence can significantly affect the downstream flow.

15.17.3. k-ε Models and k-ω Models

When you are modeling turbulent flows in Ansys Fluent using one of the k-ε models or one of the k-ω models, you must provide the boundary conditions for k and ε (or k and ω) in addition to other mean solution variables. The boundary conditions for k and ε (or k and ω) at the walls are internally taken care of by Ansys Fluent, which obviates the need for your inputs. The boundary condition inputs for k and ε (or k and ω), which you must enter in Ansys Fluent, are the ones at inlet boundaries (for example, velocity inlets, pressure inlets). In many situations, it is important to specify correct or realistic boundary conditions at the inlets, because the inlet turbulence can significantly affect the downstream flow.

See Determining Turbulence Parameters (p. 1395) for details about specifying the boundary conditions for k and ε (or k and ω) at the inlets.

Additionally, you can control whether or not to set the turbulent viscosity to zero within a laminar zone. If the fluid zone in question is laminar, the text command `define/ boundary-conditions/fluid` will contain an option called `Set Turbulent Viscosity to zero within laminar zone?`. By setting this option to `yes`, Ansys Fluent will set both the production term in the turbulence transport equation and μ_t to zero. The same defaults are applied when the **Laminar Zone** option is enabled in a **Fluid** cell zone condition dialog box. See Specifying a Laminar Zone (p. 1322) for details about laminar zones.

Important:

Note that the laminar zone feature is also available for the Spalart-Allmaras and Reynolds stress models.

15.17.4. Reynolds Stress Model

The specification of turbulent boundary conditions for the RSM is the same as for the other turbulence models for all boundaries except at boundaries where flow enters the domain. Additional input methods are available for these boundaries and are described here.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When you choose to use the RSM, the default inlet boundary condition inputs required are identical to those required when the k-ε model is active. You can input the turbulence quantities using any of the turbulence specification methods described in Determining Turbulence Parameters (p. 1395). Ansys Fluent then uses the specified turbulence quantities to derive the Reynolds stresses at the inlet from the assumption of isotropy of turbulence:

$$\overline{u_i'^2} = \frac{2}{3}k \quad (i=1,2,3) \tag{15.4}$$

$$\overline{u'_i u'_j}=0.0 \tag{15.5}$$

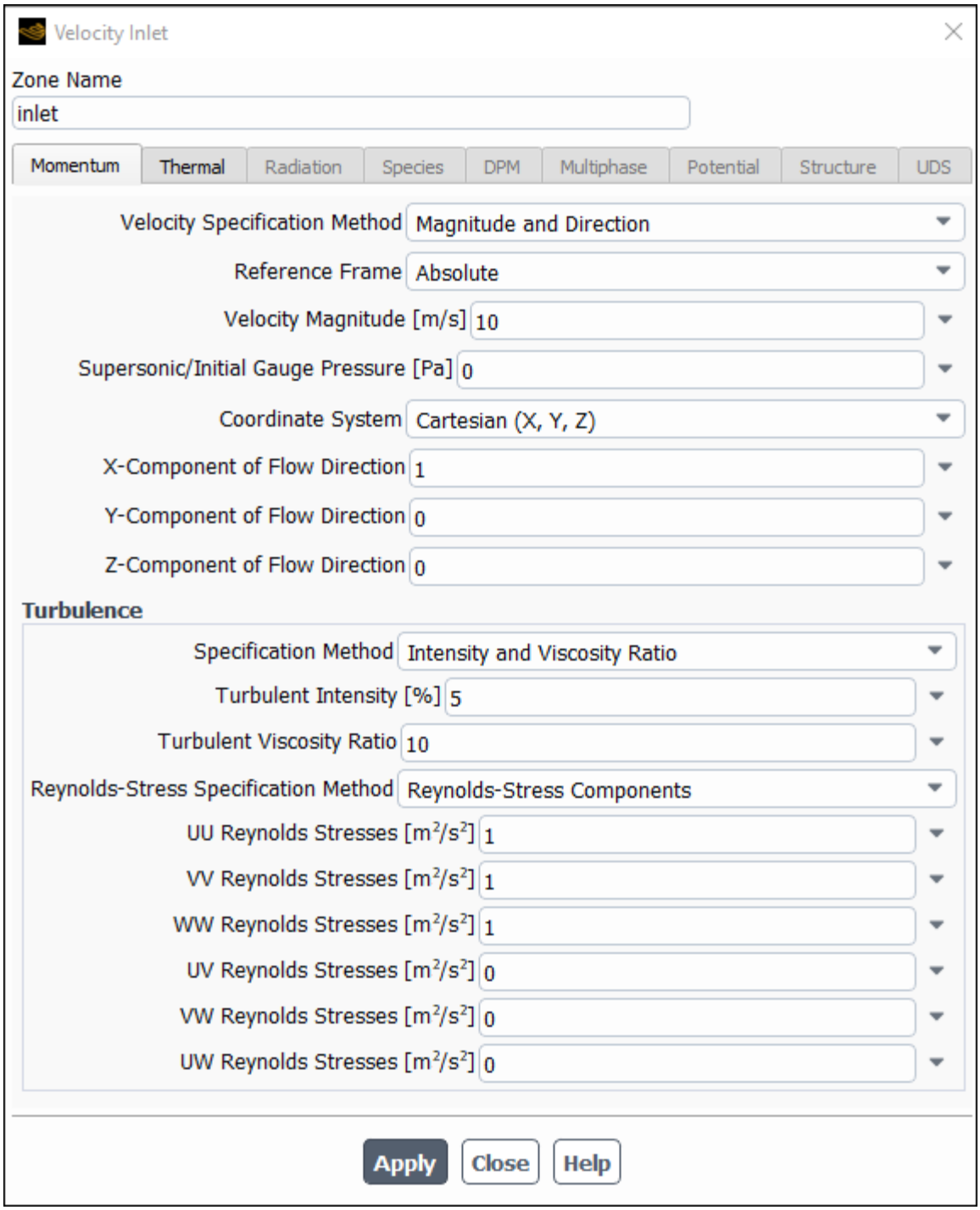
where $\overline{u_i'^2}$ is the normal Reynolds stress component in each direction. The boundary condition for ε is determined in the same manner as for the k-ε turbulence models (see Determining Turbulence Parameters (p. 1395)). To use this method, you will select **K or Turbulent Intensity** as the **Reynolds-Stress Specification Method** in the appropriate boundary condition dialog box.

Alternately, you can directly specify the Reynolds stresses by selecting the **Reynolds-Stress Components** as the **Reynolds-Stress Specification Method** in the boundary condition dialog box. When this option is selected, you should input the Reynolds stresses directly.

You can set the Reynolds stresses by using constant values, profile functions of coordinates (see Profiles (p. 1564)), or user-defined functions (in the Fluent Customization Manual).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 15.34: Specifying Inlet Boundary Conditions for the Reynolds Stresses



15.17.5. Scale Resolving Simulations

In Scale Resolving Simulations, it is possible to specify unsteady fluctuating velocity at velocity and pressure inlets to generate realistic turbulent content. The specified velocity and turbulent parameters are used to generate the fluctuations. For details, see Inlet Boundary Conditions for Scale Resolving Simulations in the *Fluent Theory Guide*.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15.18. Providing an Initial Guess for k and ε (or k and ω)

For flows using one of the k-ε models, one of the k-ω models, or the RSM, the converged solutions or (for unsteady calculations) the solutions after a sufficiently long time has elapsed should be independent of the initial values for k and ε (or k and ω). For better convergence, however, it is beneficial to use a reasonable initial guess for k and ε (or k and ω).

In general, it is recommended that you start from a suitable state of turbulence. When using a y^+-insensitive wall treatment, it is critically important to specify suitable turbulence fields. Guidelines are provided below:

- If you were able to specify reasonable boundary conditions at the inlet, it may be a good idea to compute the initial values for k and ε (or k and ω) in the whole domain from these boundary values. See Initializing the Solution (p. 3627) for details.
- For more complex flows (for example, flows with multiple inlets with different conditions) it may be better to specify the initial values in terms of **Turbulent Intensity** and **Turbulent Viscosity Ratio**, which can both be specified at the inlet(s). For **Turbulent Intensity**, T_μ, typical values lie in the range of 1-10%. For **Turbulent Viscosity Ratio**, typical values lie in the range of 1-100.
  - The value of k can then be computed from the **Turbulent Intensity** and the characteristic mean velocity magnitude of your problem (note that in the following calculation for k, T_μ must be given as a fraction):

$$k=1.5\left(T_\mu u_{avg}\right)^2 \tag{15.6}$$

  - You should specify an initial guess for ε so that the resulting eddy viscosity ($C_\mu \frac{\rho k^2}{\varepsilon}$) is sufficiently large in comparison to the molecular viscosity. Suitable levels of turbulent viscosity are typically in the range of 1-100 times larger than the molecular viscosity. From this, you can compute ε:

$$\varepsilon=\frac{\rho C_\mu k^2}{\mu_T}=\frac{\rho C_\mu k^2}{\mu\left(\frac{\mu_T}{\mu}\right)} \tag{15.7}$$

    where $\frac{\mu_T}{\mu}$ is the **Turbulent Viscosity Ratio** which can be prescribed at the inlet and then used for the domain initialization.

  - Similarly, for the specific dissipation rate, ω:

$$\omega=\frac{\rho k}{\mu\left(\frac{\mu_T}{\mu}\right)} \tag{15.8}$$

Note that, for the RSM, Reynolds stresses are initialized automatically using Equation 15.4 (p. 2145) and Equation 15.5 (p. 2145).

15.19. Solution Strategies for Turbulent Flow Simulations

Compared to laminar flows, simulations of turbulent flows are more challenging in many ways. For the Reynolds-averaged approach, additional equations are solved for the turbulence quantities. Since the equations for mean quantities and the turbulent quantities (μ_t, k, ε, ω, or the Reynolds stresses) are

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

strongly coupled in a highly nonlinear fashion, it takes more computational effort to obtain a converged turbulent solution than to obtain a converged laminar solution. The LES model, while embodying a simpler, algebraic model for the subgrid-scale viscosity, requires a transient solution on a very fine mesh.

The fidelity of the results for turbulent flows is largely determined by the turbulence model being used. Here are some guidelines that can enhance the quality of your turbulent flow simulations.

For additional information, see the following sections:

15.19.1. Mesh Generation

15.19.2. Accuracy

15.19.3. Convergence

15.19.4. RSM-Specific Solution Strategies

15.19.5. LES-Specific Solution Strategies

15.19.1. Mesh Generation

The following are suggestions to follow when generating the mesh for use in your turbulent flow simulation:

- First imagine the flow under consideration, then identify the main flow features expected in the flow using your physical intuition or any data for a similar flow situation. Generate a mesh that can resolve the major features that you expect.
- If the flow is wall-bounded and the wall is expected to affect the flow significantly, you should take additional care when generating the mesh. See Grid Resolution for RANS Models (p. 2072) and Wall Boundary Layers (p. 2077) for guidelines.

15.19.2. Accuracy

The suggestions below are provided to help you obtain better accuracy in your results:

- Use the turbulence model that is better suited for the salient features you expect to see in the flow (see Choosing a Turbulence Model (p. 2064)).
- Because the mean quantities have larger gradients in turbulent flows than in laminar flows, it is recommended that you use high-order schemes for the convection terms. This is especially true if you employ a triangular or tetrahedral mesh. Note that excessive numerical diffusion adversely affects the solution accuracy, even with the most elaborate turbulence model.
- In some flow situations involving inlet boundaries, the flow downstream of the inlet is dictated by the boundary conditions at the inlet. In such cases, you should exercise care to make sure that reasonably realistic boundary values are specified.

15.19.3. Convergence

The suggestions below are provided to help you enhance convergence for turbulent flow calculations:

- Starting with excessively crude initial guesses for mean and turbulence quantities may cause the solution to diverge. A safe approach is to start your calculation using conservative (small) under-

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

relaxation parameters and (for the density-based solvers) a conservative Courant number, and increase them gradually as the iterations proceed and the solution begins to settle down.

- It is also helpful for faster convergence to start with reasonable initial guesses for the k and ε (or k and ω) fields. Particularly when a y^+-insensitive wall treatment is used, it is important to start with a sufficiently developed turbulence field, as recommended in Providing an Initial Guess for k and ε (or k and ω) (p. 2147), to avoid the need for an excessive number of iterations to develop the turbulence field.

- When you are using the RNG k-ε model, an approach that might help you achieve better convergence is to obtain a solution with the standard k-ε model before switching to the RNG model. Due to the additional nonlinearities in the RNG model, lower under-relaxation factors and (for the density-based solvers) a lower Courant number might also be necessary.

Note that when you use the enhanced wall treatment (EWT-ε), you may sometimes find during the calculation that the residual for ε is reported to be zero. This happens when your flow is such that Re_y is less than 200 in the entire flow domain, and ε is obtained from the algebraic formula (Equation 4.366 in the Theory Guide) instead of from its transport equation.

15.19.4. RSM-Specific Solution Strategies

Using the RSM creates a high degree of coupling between the momentum equations and the turbulent stresses in the flow, and therefore the calculation can be more prone to stability and convergence difficulties than with the k-ε models. When you use the RSM, therefore, you may need to adopt special solution strategies in order to obtain a converged solution. The following strategies are generally recommended:

- Begin the calculations using the standard k-ε model. Turn on the RSM and use the k-ε solution data as a starting point for the RSM calculation.

- Use low under-relaxation factors (0.2 to 0.3) and (for the density-based solvers) a low Courant number for highly swirling flows or highly complex flows. In these cases, you may need to reduce the under-relaxation factors both for the velocities and for all of the stresses.

Instructions for setting these solution parameters are provided below. If you are applying the RSM to prediction of a highly swirling flow, you will want to consider the solution strategies discussed in Swirling and Rotating Flows (p. 1753) as well.

15.19.4.1. Under-Relaxation of the Reynolds Stresses

Ansys Fluent applies under-relaxation to the Reynolds stresses. You can set under-relaxation factors using the Solution Controls Task Page (p. 5163).

Solution → **Controls**

The default settings of 0.5 are recommended for most cases. You may be able to increase these settings and speed up the convergence when the RSM solution begins to converge.

In some situations, when poor convergence is observed one might facilitate the convergence rates by modifying some of the under-relaxation values whilst leaving the others unchanged. This might be a more successful approach than the simple scaling of all under-relaxation values.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

15.19.4.2. Disabling Calculation Updates of the Reynolds Stresses

In some instances, you may want to let the current Reynolds stress field remain fixed, skipping the solution of the Reynolds transport equations while solving the other transport equations. You can enable/disable all Reynolds stress equations in the **Equations** dialog box.

Solution → **Controls** **Equations...**

15.19.4.3. Residual Reporting for the RSM

When you use the RSM for turbulence, Ansys Fluent reports the equation residuals for the individual Reynolds stress transport equations. You can apply the usual convergence criteria to the Reynolds stress residuals: normalized residuals in the range of 10^{-3} usually indicate a practically-converged solution. However, you may need to apply tighter convergence criteria (below 10^{-4}) to ensure full convergence.

15.19.5. LES-Specific Solution Strategies

Large eddy simulation involves running a transient solution from some initial condition, on an appropriately fine mesh, using an appropriate time step size. The solution must be run long enough to become independent of the initial condition and to enable the statistics of the flow field to be determined.

The following are suggestions to follow when running a large eddy simulation:

1. Start by running a steady-state flow simulation using a Reynolds-averaged turbulence model such as standard k-ε, k-ω, or even RSM. Run until the flow field is reasonably converged and then use the `solve/initialize/init-turb-vel-fluctuations` text command to generate the instantaneous velocity field out of the steady-state RANS results. This command must be executed before LES is enabled. This option is available for all RANS-based models except the Spalart-Allmaras model, and it will create a much more realistic initial field for the LES run. Additionally, it will help in reducing the time needed for the LES simulation to reach a statistically stable mode. This step is optional.

2. When you enable LES, Ansys Fluent will automatically turn on the unsteady solver option and choose the second-order implicit formulation. You will need to set the appropriate time step size and all the needed solution parameters. (See Inputs for Time-Dependent Problems (p. 3645) for guidelines on setting solution parameters for transient calculations in general.) For the pressure-based solver, the bounded central differencing spatial discretization scheme will be automatically selected for momentum equations; both the bounded central-differencing and pure central-differencing schemes are available for all equations when running LES simulations. For the density-based solver, the bounded central differencing scheme can be used for the flow equations, though it is not selected by default.

3. Run LES until the flow becomes statistically steady. The best way to see if the flow is fully developed and statistically steady is to monitor forces and solution variables (for example, velocity components or pressure) at selected locations in the flow.

4. Zero out the initial statistics using the `solve/initialize/init-flow-statistics` text command. Before you restart the solution, enable **Data Sampling for Time Statistics** in the Run

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Calculation Task Page (p. 5207), as described in Inputs for Time-Dependent Problems (p. 3645). With this option enabled, Ansys Fluent will gather data for time statistics while performing a large eddy simulation. You can set the **Sampling Interval** such that **Data Sampling for Time Statistics** can be performed at the specified frequency. When **Data Sampling for Time Statistics** is enabled, the statistics collected at each sampling interval can be postprocessed and you can then view both the mean and the root-mean-square-error (RMSE) values in Ansys Fluent. The **Sampled Time** displays the time period over which data has been sampled for the postprocessing of the mean and RMSE values. As long as the time step size has been constant, dividing this by the time step size yields the number of data sets that have been collected. If the time step size is varied, every contribution of data sets sampled is automatically weighted by the current time step size.

5. Continue until you get statistically stable data. The duration of the simulation can be determined beforehand by estimating the mean flow residence time in the solution domain (L/U, where L is the characteristic length of the solution domain and U is a characteristic mean flow velocity). The simulation should be run for at least a few mean flow residence times.

Instructions for setting the solution parameters for LES are provided below.

15.19.5.1. Temporal Discretization

Ansys Fluent provides both first-order and second-order temporal discretizations. For LES, the second-order discretization is recommended.

Solution → **Methods**

15.19.5.2. Spatial Discretization

Overly diffusive schemes such as the first-order upwind scheme should be avoided, because they may unduly damp out the energy of the resolved eddies. The central-differencing based schemes are recommended for all equations when you use the LES model. Ansys Fluent provides two central-differencing based schemes: *pure* central-differencing and *bounded* central-differencing. The bounded scheme is the default option for momentum discretization when you select LES, SAS, DES, SDES, or SBES.

Solution → **Methods**

15.20. Postprocessing for Turbulent Flows

Ansys Fluent provides postprocessing options for displaying, plotting, and reporting various turbulence quantities, which include the main solution variables and other auxiliary quantities.

Turbulence quantities that can be reported for the Spalart-Allmaras model are as follows:

- **Modified Turbulent Viscosity**
- **Turbulent Viscosity**
- **Effective Viscosity**
- **Turbulent Viscosity Ratio**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Effective Thermal Conductivity**
- **Effective Prandtl Number**
- **Wall Yplus**
- **Curvature Correction Function fr** (only when the curvature correction is enabled)

Turbulence quantities that can be reported for the DES model in combination with the Spalart-Allmaras model are as follows:

- all of the quantities available for the Spalart-Allmaras model
- **DES TKE Dissipation Multiplier**
- **Q Criterion Normalized** (listed under **Velocity...**)
- **Q Criterion Raw** (listed under **Velocity...**)
- **Lambda 2 Criterion** (listed under **Velocity...**)

Note:

For a Detached Eddy Simulation with the Spalart-Allmaras model, the **DES TKE Dissipation Multiplier** represents the function f_d in Equation 4.272.

Turbulence quantities that can be reported for the k-ε models are as follows:

- **Turbulent Kinetic Energy (k)**
- **Turbulent Intensity**
- **Turbulent Dissipation Rate (Epsilon)**
- **Production of k**
- **Turbulent Viscosity**
- **Effective Viscosity**
- **Turbulent Viscosity Ratio**
- **Effective Thermal Conductivity**
- **Effective Prandtl Number**
- **Wall Ystar**
- **Wall Yplus**
- **Turbulent Reynolds Number (Re_y)** (only when the enhanced wall treatment is used for the near-wall treatment)
- **Curvature Correction Function fr** (only when the curvature correction is enabled)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Turbulence quantities that can be reported for the DES model in combination with the Realizable k-ε model are as follows:

- all of the quantities available for the Realizable k-ε model
- **DES TKE Dissipation Multiplier**
- **Normalized Q Criterion** (listed under **Velocity...**)
- **Q Criterion** (listed under **Velocity...**)
- **Lambda 2 Criterion** (listed under **Velocity...**)

Note:

For a Detached Eddy Simulation with the Realizable k-ε model, the **DES TKE Dissipation Multiplier** represents the function f_d in Equation 4.279.

Turbulence quantities that can be reported for the k-ω models are as follows:

- **Turbulent Kinetic Energy (k)**
- **Turbulent Intensity**
- **Turbulent Dissipation Rate (Epsilon)**
- **Intermittency**
- **Specific Dissipation Rate (Omega)**
- **Production of k**
- **Turbulent Viscosity**
- **Effective Viscosity**
- **Turbulent Viscosity Ratio**
- **Effective Thermal Conductivity**
- **Effective Prandtl Number**
- **Wall Ystar**
- **Wall Yplus**
- **Turbulent Reynolds Number (Re_y)**
- **Curvature Correction Function fr** (only when the curvature correction is enabled)

Turbulence quantities that can be reported for the Standard and BSL k-ω model in combination with the SAS model are as follows:

- all of the quantities available for the k-ω model

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Normalized Q Criterion** (listed under **Velocity...**)
- **Q Criterion** (listed under **Velocity...**)
- **Lambda 2 Criterion** (listed under **Velocity...**)

Turbulence quantities that can be reported for the BSL or SST k-ω model in combination with the DES model are as follows:

- all of the quantities available for the k-ω models
- **DES TKE Dissipation Multiplier**
- **Normalized Q Criterion** (listed under **Velocity...**)
- **Q Criterion** (listed under **Velocity...**)
- **Lambda 2 Criterion** (listed under **Velocity...**)

Note:

For a Detached Eddy Simulation with BSL / SST, **DES TKE Dissipation Multiplier** represents the function F_{DES} in Equation 4.283 and F_{IDDES} in Equation 4.287, depending on the specified model. Within the DES concept, the **DES TKE Dissipation Multiplier** increases the destruction term in the transport equation for the turbulence kinetic energy in LES regions (F_{DES}>1). This increase in destruction terms reduces the eddy viscosity in LES regions.

Turbulence quantities that can be reported for the BSL or SST k-ω model in combination with the Shielded Detached Eddy Simulation (SDES) model or the Stress-Blended Eddy Simulation (SBES) model are as follows:

- all of the quantities available for the k-ω models
- **Shielding Function for SBES or SDES**
- **Normalized Q Criterion** (listed under **Velocity...**)
- **Q Criterion** (listed under **Velocity...**)
- **Lambda 2 Criterion** (listed under **Velocity...**)

Turbulence quantities that can be reported for the transition k-kl-ω model are as follows:

- **Turbulent Kinetic Energy (k)**
- **Laminar Kinetic Energy**
- **Total Fluctuation Energy**
- **Turbulent Intensity**
- **Turbulent Dissipation Rate (Epsilon)**
- **Specific Dissipation Rate (Omega)**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Production of k**
- **Production of laminar k**
- **Turbulent Viscosity**
- **Turbulent Viscosity (large-scale)**
- **Turbulent Viscosity (small-scale)**
- **Effective Viscosity**
- **Turbulent Viscosity Ratio**
- **Effective Thermal Conductivity**
- **Effective Prandtl Number**
- **Wall Ystar**
- **Wall Yplus**
- **Turbulent Reynolds Number (Re_y)**

Turbulence quantities that can be reported for the Transition SST model are as follows:

- **Turbulent Kinetic Energy (k)**
- **Turbulent Intensity**
- **Turbulent Dissipation Rate (Epsilon)**
- **Intermittency**
- **Intermittency Effective**
- **Momentum Thickness Re**
- **Geometric Roughness Height**
- **Specific Dissipation Rate (Omega)**
- **Production of k**
- **Turbulent Viscosity**
- **Effective Viscosity**
- **Turbulent Viscosity Ratio**
- **Effective Thermal Conductivity**
- **Effective Prandtl Number**
- **Wall Ystar**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Wall Yplus**
- **Turbulent Reynolds Number (Re_y)**
- **Curvature Correction Function fr** (only when the curvature correction is enabled)

Turbulence quantities that can be reported for the Transition SST model in combination with the SAS model are as follows:

- all of the quantities available for the Transition SST model
- **Normalized Q Criterion** (listed under **Velocity...**)
- **Q Criterion** (listed under **Velocity...**)
- **Lambda 2 Criterion** (listed under **Velocity...**)

Turbulence quantities that can be reported for the Transition SST model in combination with the DES model are as follows:

- all of the quantities available for the Transition SST model
- **DES TKE Dissipation Multiplier**
- **Normalized Q Criterion** (listed under **Velocity...**)
- **Q Criterion** (listed under **Velocity...**)
- **Lambda 2 Criterion** (listed under **Velocity...**)

Note:

For the Transition SST model in combination with the DES model, **DES TKE Dissipation Multiplier** represents the function F_{DES} in Equation 4.286 and F_{IDDES} in Equation 4.287 depending on the specified model. Within the DES concept, the **DES TKE Dissipation Multiplier** increases the destruction term in the transport equation for the turbulence kinetic energy in LES regions (F_{DES}>1). This increase in destruction terms reduces the eddy viscosity in LES regions.

Turbulence quantities that can be reported for the Transition SST model in combination with the Shielded Detached Eddy Simulation (SDES) model or the Stress-Blended Eddy Simulation (SBES) model are as follows:

- all of the quantities available for the Transition SST model
- **Shielding Function for SBES or SDES**
- **Normalized Q Criterion** (listed under **Velocity...**)
- **Q Criterion** (listed under **Velocity...**)
- **Lambda 2 Criterion** (listed under **Velocity...**)

Turbulence quantities that can be reported for the RSM are as follows:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Turbulent Kinetic Energy (k)**
- **Turbulent Intensity**
- **UU Reynolds Stress**
- **VV Reynolds Stress**
- **WW Reynolds Stress**
- **UV Reynolds Stress**
- **VW Reynolds Stress**
- **UW Reynolds Stress**
- **Turbulent Dissipation Rate (Epsilon)**
- **Specific Dissipation Rate (Omega)** (ω-based Reynolds stress models only)
- **Production of k**
- **Turbulent Viscosity**
- **Effective Viscosity**
- **Turbulent Viscosity Ratio**
- **Effective Thermal Conductivity**
- **Effective Prandtl Number**
- **Wall Ystar**
- **Wall Yplus**
- **Turbulent Reynolds Number (Re_y)** (ε-based Reynolds stress models only)

Turbulence quantities that can be reported for the ω-based Reynolds stress models in combination with the SAS model are as follows:

- all of the quantities available for the RSM model
- **Normalized Q Criterion** (listed under **Velocity...**)
- **Q Criterion** (listed under **Velocity...**)
- **Lambda 2 Criterion** (listed under **Velocity...**)

Turbulence quantities that can be reported for the SAS model in combination with the SST k-ω model are as follows:

- **Turbulent Kinetic Energy (k)**
- **Turbulent Intensity**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Turbulent Dissipation Rate (Epsilon)**
- **Intermittency**
- **Specific Dissipation Rate (Omega)**
- **Production of k**
- **Turbulent Viscosity**
- **Effective Viscosity**
- **Turbulent Viscosity Ratio**
- **Effective Thermal Conductivity**
- **Effective Prandtl Number**
- **Wall Yplus**
- **Wall Ystar**
- **Curvature Correction Function fr** (only when the curvature correction is enabled)
- **Normalized Q Criterion** (listed under **Velocity...**)
- **Q Criterion** (listed under **Velocity...**)
- **Lambda 2 Criterion** (listed under **Velocity...**)

Turbulence quantities that can be reported for the LES model are as follows:

- **Turbulence Kinetic Energy**
- **Turbulence Intensity**
- **Subgrid Kinetic Energy**
- **Production of k**
- **Subgrid Turbulent Viscosity**
- **Subgrid Effective Viscosity**
- **Subgrid Turbulent Viscosity Ratio**
- **Subgrid Filter Length**
- **Subgrid Test-Filter Length**
- **Subgrid Dissipation Rate**
- **Subgrid Dynamic Viscosity Const**
- **Subgrid Dynamic Prandtl Number**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Subgrid Dynamic Sc of Species**
- **Subtest Kinetic Energy**
- **Effective Thermal Conductivity**
- **Effective Prandtl Number**
- **Wall Ystar**
- **Wall Yplus**
- **Normalized Q Criterion** (listed under **Velocity...**)
- **Q Criterion** (listed under **Velocity...**)
- **Lambda 2 Criterion** (listed under **Velocity...**)

Additional turbulence quantities can be reported for the Embedded LES (ELES) model.

Note:

Within the Embedded LES zone, the modeled eddy viscosity is determined by an algebraic subgrid-scale eddy viscosity model. The global turbulence model is either frozen or, if it is either SAS, DES, SDES, or SBES with an underlying two-equation RANS model, runs in a "passive mode", that is without affecting the momentum equations.

Hence, most turbulence postprocessing in the Embedded LES zone refers to the frozen or "passive" global turbulence model. Special care is necessary with the modeled eddy viscosity, because there are two to consider: one from the frozen / passive global turbulence model; and one from the "local" algebraic subgrid-scale model that is running within the Embedded LES zone and actually affects the momentum equations.

- Some postprocessing quantities refer to a "passive" global turbulence model or are zero if the global model cannot run in "passive" mode and therefore is frozen in the Embedded LES zone.
  - **Turbulent Viscosity**
  - **Turbulent Viscosity Ratio**
- In the Embedded LES zone, the subgrid-scale eddy viscosity from the local algebraic model, which actually affects the momentum equations, is displayed as:
  - **LES Subgrid Turbulent Viscosity**
- The following quantities are specific to the Dynamic LES subgrid-scale eddy viscosity models. They are available in Embedded LES zones if the **Dynamic Smagorinsky** model is used in any Embedded LES zone; they are also available for global LES when the **Smagorinsky-Lilly** subgrid-scale model is selected with the **Dynamic Stress** option enabled or when the **Kinetic-Energy Transport** subgrid-scale model is selected:
  - **Subgrid Test-Filter Length**
  - **Subgrid Dynamic Viscosity Const**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– **Subtest Kinetic Energy**

All of these variables can be found in the **Turbulence...** category of the variable selection drop-down list that appears in postprocessing dialog boxes. See Field Function Definitions (p. 4167) for their definitions.

For additional information, see the following sections:

15.20.1. Custom Field Functions for Turbulence

15.20.2. Postprocessing Turbulent Flow Statistics

15.20.3. Troubleshooting

15.20.1. Custom Field Functions for Turbulence

In addition to the quantities listed in Postprocessing for Turbulent Flows (p. 2151), above, you can define your own turbulence quantities using the Custom Field Function Calculator Dialog Box (p. 5428).

User-Defineed → **Field Functions** → **Custom...**

The following functions may be useful:

- the ratio of production of k to its dissipation ($G_k / \rho\varepsilon$)
- the ratio of the mean flow to turbulent time scale, η ($\equiv Sk / \varepsilon$)
- the Reynolds stresses derived from the Boussinesq formula (for example, $-\overline{uv}=\nu_t \frac{\partial u}{\partial y}$)

15.20.2. Postprocessing Turbulent Flow Statistics

As described in Large Eddy Simulation (LES) Model (in the Theory Guide), LES involves the solution of a transient flow field, but it is the mean flow quantities that are of interest from an engineering standpoint.

For all other turbulent flow, if **Data Sampling for Time Statistics** is enabled in the Run Calculation Task Page (p. 5207), Ansys Fluent gathers data for time statistics while performing the simulation. The statistics that Ansys Fluent collects at each sampling interval (which consists of the mean and the root-mean-square-error (RMSE) values) can be postprocessed by selecting **Unsteady Statistics...** in any of the postprocessing dialog boxes. You can view several variables that include, but are not limited to, shear stresses (**Resolved UV/UW/VW Reynolds Stress**), flow heat fluxes (**Resolved UT/VT/WT Heat Flux**), and species statistics (**RMSE Mass Fraction of** species and **Mean Mass Fraction** of species). If you select **Unsteady Wall Statistics...** in any of the postprocessing dialog boxes, you can view wall statistics such as **Mean Pressure Coefficient**, **Mean Wall Shear Stress**, **Mean X-Wall Shear Stress**, **Mean Y-Wall Shear Stress**, **Mean Z-Wall Shear Stress**, **Mean Skin Friction Coefficient**, **Mean Surface Heat Flux**, **Mean Surface Heat Transfer Coef.**, **Mean Surface Nusselt Number**, **Mean Surface Stanton Number**. Note that these quantities are only the statistical evaluation of sampled solution data, and do not contain any kind of modeled turbulent fluctuations. See Postprocessing for Time-Dependent Problems (p. 3669) for details.

The **Sampled Time** field displays the time period over which data has been sampled for the postprocessing of the mean and RMSE values. As long as the time step size has been constant, dividing this by the time step size yields the number of data sets that have been collected. If the time step size is

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

varied, every contribution of data sets sampled is automatically weighted by the current time step size.

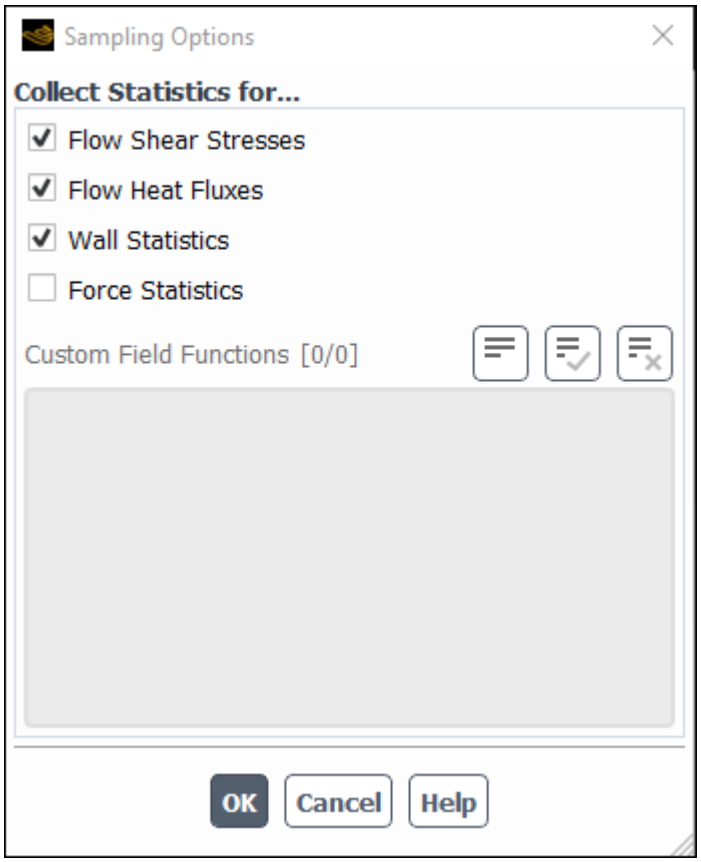
Important:

Note that *mean* statistics are collected only in interior cells and not on wall surfaces. Therefore, when node or cell values of mean quantities are plotted on the wall surface, you are actually plotting values in nearby cells attached to the wall.

If you want to control what sets of variables are available for postprocessing, there are two dialog boxes that you can use to do so. The **Sampling Options** dialog box allows you to and enable or disable the statistics shown in Figure 15.35: The Sampling Options Dialog Box (p. 2161). The alternative is to use the **Zone-Specific Sampling Options** dialog box (Figure 15.36: The Zone-Specific Sampling Options Dialog Box (p. 2162)), which provides more granular control over which zones and quantities are sampled.

Solution → Run Calculation → Sampling Options...

Figure 15.35: The Sampling Options Dialog Box



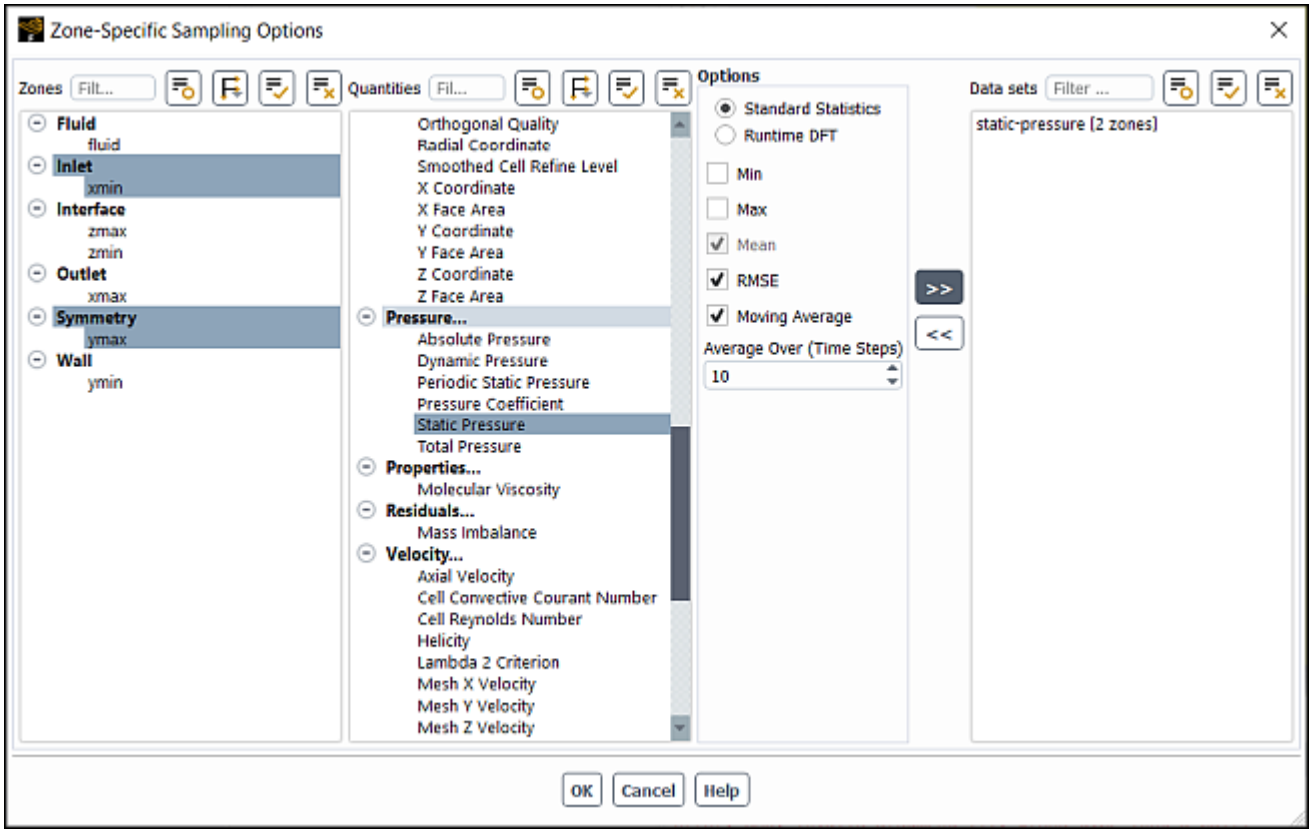
Important:

When including or excluding statistics on variables, it is recommended that you re-initialize the flow statistics.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution → Run Calculation → Sampling Options (Zone Selection)...

Figure 15.36: The Zone-Specific Sampling Options Dialog Box



Important:

When including or excluding statistics on variables, it is recommended that you re-initialize the flow statistics.

15.20.3. Troubleshooting

You can use the postprocessing options not only for the purpose of interpreting your results but also for investigating any anomalies that may appear in the solution. For instance, you may want to plot contours of the k field to check if there are any regions where k is erroneously large or small. You should see a high k region in the region where the production of k is large. You may want to display the turbulent viscosity ratio field in order to see whether or not the turbulence takes full effect. Usually the turbulent viscosity is at least two orders of magnitude larger than molecular viscosity for fully-developed turbulent flows modeled using the RANS approach (that is, not using LES). You may also want to check whether you are using an adequate near-wall mesh when using a y^+-insensitive wall treatment (see Grid Resolution for RANS Models (p. 2072) for guidelines). For EWT-ε, you can also display filled contours of Re_y (turbulent Reynolds number) overlaid on the mesh (see Enhanced Wall Treatment ε-Equation (EWT-ε) in the Theory Guide for more details).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 16: Modeling Thermal Energy

This chapter provides details about the heat transfer models available in Ansys Fluent. Information is presented in the following sections:

16.1. Introduction

16.2. Modeling Conductive and Convective Heat Transfer

16.3. Modeling Radiation

16.4. Modeling Periodic Heat Transfer

16.5. Modeling Heat Exchangers

16.6. Thermal Analysis of Printed Circuit Boards

16.1. Introduction

The flow of thermal energy from matter occupying one region in space to matter occupying a different region in space is known as heat transfer. Heat transfer can occur by three main mechanisms: conduction, convection, and radiation. Physical models involving conduction and/or convection only are the simplest (Modeling Conductive and Convective Heat Transfer (p. 2163)), while buoyancy-driven flow or natural convection (Buoyancy-Driven Flows and Natural Convection (p. 1387)), and flow involving radiation (Modeling Radiation (p. 2192)) are more complex. Depending on your problem, Ansys Fluent will solve a variation of the energy equation that takes into account the heat transfer methods you have specified. Ansys Fluent is also able to predict heat transfer in periodically repeating geometries (Modeling Periodic Heat Transfer (p. 2274)), therefore greatly reducing the required computational effort in certain cases.

16.2. Modeling Conductive and Convective Heat Transfer

Ansys Fluent allows you to include heat transfer within the fluid and/or solid regions in your model. Problems ranging from thermal mixing within a fluid to conduction in composite solids can therefore be handled by Ansys Fluent.

When your Ansys Fluent model includes heat transfer you will need to enable the relevant physical models, supply thermal boundary conditions, and enter material properties (which may vary with temperature) that govern heat transfer. For information about heat transfer theory, see Heat Transfer Theory in the Theory Guide. Information about heat transfer theory and how to set up and use heat transfer in your Ansys Fluent model is presented in the following subsections:

16.2.1. Solving Heat Transfer Problems

16.2.2. Solution Strategies for Heat Transfer Modeling

16.2.3. Postprocessing Heat Transfer Quantities

16.2.4. Natural Convection

16.2.5. Shell Conduction

16.2.6. Anisotropic Thermal Conductivity with Curvilinear Coordinate System (CCS)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

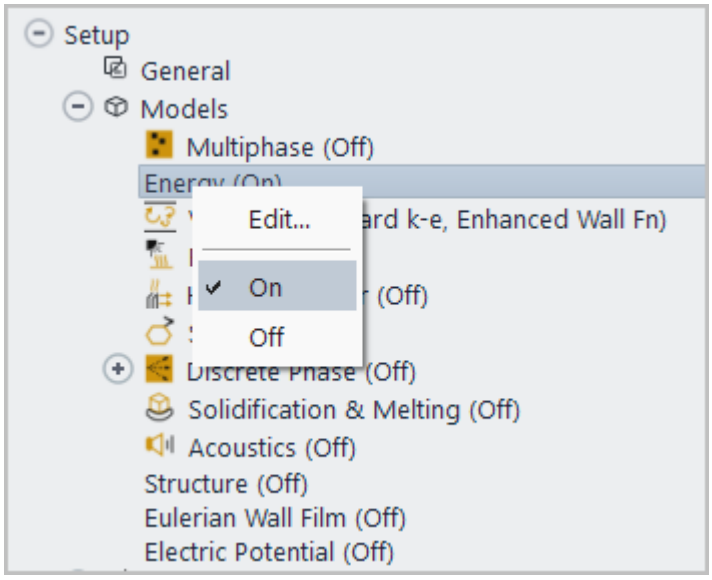
16.2.1. Solving Heat Transfer Problems

The procedure for setting up a heat transfer problem is described below. Note that this procedure includes only those steps necessary for the heat transfer model itself; you will need to define the other settings (for example, other models, boundary conditions) as usual.

1. To activate the calculation of heat transfer, enable the energy equation by right-clicking **Energy** in the tree (under **Setup/Models**) and clicking **On** in the menu that opens.

   Setup → **Models** → **Energy** → **On**

   Figure 16.1: Enabling the Energy Equation

2. (Optional, pressure-based solver only.) If you are modeling viscous flow and you want to include the viscous heating terms in the energy equation, enable the **Viscous Heating** option in the **Viscous Model** dialog box.

   Setup → **Models** → **Viscous** → **Edit...**

   As noted in Inclusion of the Viscous Dissipation Terms in the Theory Guide, the viscous heating terms in the energy equation are (by default) ignored by Ansys Fluent when the pressure-based solver is used. They are always included for the density-based solver. Viscous dissipation should be enabled when the shear stress in the fluid is large (for example, in lubrication problems) and/or in high-velocity, compressible flows. See Equation 5.10 in the Theory Guide.

3. Define thermal boundary conditions at flow inlets, flow outlets, and walls. The boundary condition dialog boxes can be opened by right-clicking the boundary name in the tree (under **Setup/Boundary Conditions**) and clicking **Edit...** in the menu that opens; alternatively, you can open them from the **Boundary Conditions** task page:

   Setup → **Boundary Conditions**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

At flow inlets and exits you will set the temperature; at walls you may use any of the following thermal conditions:

- specified heat flux
- specified temperature
- convective heat transfer
- external radiation
- combined external radiation and external convective heat transfer

Thermal Boundary Conditions at Walls (p. 1467) provides details on the model inputs that govern these thermal boundary conditions. The default thermal boundary condition at inlets is a specified temperature of 300 K; at walls the default condition is zero heat flux (adiabatic). See Cell Zone and Boundary Conditions (p. 1299) for details about boundary condition inputs.

Important:

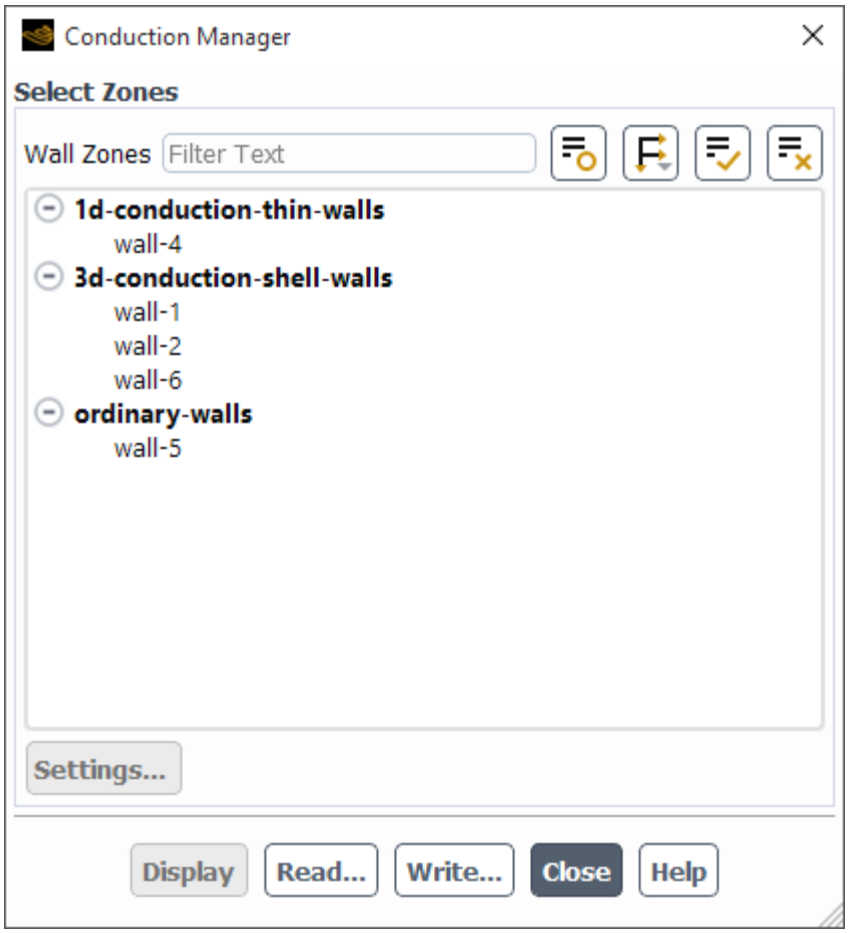
If your heat transfer application involves two separated fluid regions, see the information provided below.

---

4. If your case involves conduction through walls (either 1d conduction through "thin" walls or 3d conduction through shell walls), you can specify the required settings for any wall using Figure 16.2: The Conduction Manager Dialog Box (p. 2166).

   Physics → **Model Specific** → **Conduction Manager...**

Figure 16.2: The Conduction Manager Dialog Box



a. Select any number of walls and click **Settings...**.

b. Specify the conduction settings for the selected walls. For details, see Managing Conduction Walls (p. 2184).

5. Define material properties for heat transfer.

   Setup → **Materials**

   Heat capacity and thermal conductivity must be defined, and you can specify many properties as functions of temperature as described in Physical Properties (p. 1597).

16.2.1.1. Limiting the Predicted Temperature Range

For stability reasons, Ansys Fluent includes a limit on the predicted temperature range. The purpose of the temperature ceiling and floor is to improve the stability of calculations in which the temperature should physically lie within known limits. Sometimes intermediate solutions of the equations give rise to temperatures beyond these limits for which property definitions, and so on, are not well defined.

The temperature limits keep the temperatures within the expected range for your problem. If the Ansys Fluent calculation predicts a temperature above the maximum limit, the stored temperature

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

values are "pegged" at this maximum value. The default for the temperature ceiling is 5000 K. If the Ansys Fluent calculation predicts a temperature below the minimum limit, the stored temperature values are "pegged" at this minimum value. The default for the temperature minimum is 1 K.

If you expect the temperature in your domain to exceed 5000 K, use the **Solution Limits** dialog box to increase the **Maximum Static Temperature**.

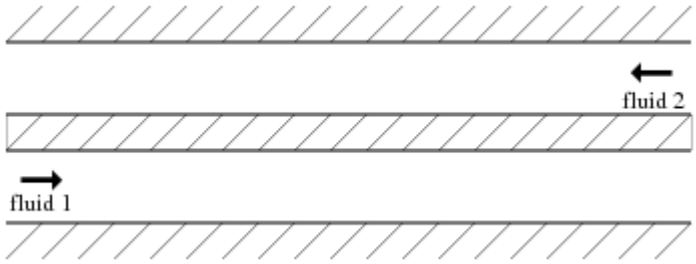
Solution → **Controls** **Limits...**

16.2.1.2. Modeling Heat Transfer in Two Separated Fluid Regions

If your heat transfer application involves two fluid regions separated by a solid zone or a wall, as illustrated in Figure 16.3: Typical Counterflow Heat Exchanger Involving Heat Transfer Between Two Separated Fluid Streams (p. 2167), you will need to define the problem with some care. Specifically:

- You should not use outflow boundary conditions in either fluid.
- You can establish separate fluid properties by selecting a different fluid material for each zone. For species calculations, however, you can only select a single mixture material for the entire domain.

Figure 16.3: Typical Counterflow Heat Exchanger Involving Heat Transfer Between Two Separated Fluid Streams



16.2.2. Solution Strategies for Heat Transfer Modeling

Although many simple heat transfer problems can be successfully solved using the default solution parameters assumed by Ansys Fluent, you may accelerate the convergence of your problem and/or improve the stability of the solution process using some of the guidelines provided in this section.

16.2.2.1. Under-Relaxation of the Energy Equation

When you use the pressure-based solver, Ansys Fluent under-relaxes the energy equation using the under-relaxation parameter defined by you in the **Solution Controls** task page, as described in Setting Under-Relaxation Factors (p. 3589).

Solution → **Controls**

If you are using the non-adiabatic non-premixed combustion model, you will set the energy under-relaxation factor as usual but you will also set an under-relaxation factor for temperature, as described below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Fluent uses a default under-relaxation factor of 1.0 for the energy equation, regardless of the form in which it is solved (temperature or enthalpy). In problems where the energy field impacts the fluid flow (via temperature-dependent properties or buoyancy) you should use a lower value for the under-relaxation factor, in the range of 0.8–1.0. In problems where the flow field is decoupled from the temperature field (no temperature-dependent properties or buoyancy forces), you can usually retain the default value of 1.0.

16.2.2.2. Under-Relaxation of Temperature When the Enthalpy Equation is Solved

When the enthalpy form of the energy equation is solved (that is, when you are using the non-adiabatic non-premixed combustion model), Ansys Fluent also under-relaxes the temperature, updating the temperature by only a fraction of the change that would result from the change in the (under-relaxed) enthalpy values. This second level of under-relaxation can be used to good advantage when you would like to let the enthalpy field change rapidly, but the temperature response (and its effect on fluid properties) to lag. Ansys Fluent uses a default setting of 1.0 for the under-relaxation on temperature and you can modify this setting using the **Solution Controls** task page.

Solution → **Controls**

16.2.2.3. Disabling the Species Diffusion Term

If you are solving for species transport using the pressure-based solver and you encounter convergence difficulties, you may want to consider disabling the **Diffusion Energy Source** option in the **Species Model** dialog box.

Setup → **Models** → **Species** **Edit...**

When this option is disabled, Ansys Fluent will neglect the effects of species diffusion on the energy equation.

Note that species diffusion effects are always included when the density-based solver is used.

16.2.2.4. Step-by-Step Solutions

Often the most efficient strategy for predicting heat transfer is to compute an isothermal flow first and then add the calculation of the energy equation. The procedure differs slightly, depending on whether or not the flow and heat transfer are coupled.

16.2.2.4.1. Decoupled Flow and Heat Transfer Calculations

If your flow and heat transfer are decoupled (no temperature-dependent properties or buoyancy forces), you can first solve the isothermal flow (energy equation turned off) to yield a converged flow-field solution and then solve the energy transport equation alone.

Important:

Since the density-based solver always solves the flow and energy equations together, the procedure for solving for energy alone applies to the pressure-based solver, only.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can temporarily disable the flow equations or the energy equation by deselecting them in the **Equations** dialog box:

Solution → **Controls** **Equations...**

You can also disable the energy equation:

Setup → **Models** → **Energy** **Off**

16.2.2.4.2. Coupled Flow and Heat Transfer Calculations

If the flow and heat transfer are coupled (that is, your model includes temperature-dependent properties or buoyancy forces), you can first solve the flow equations before enabling energy. Once you have a converged flow-field solution, you can enable energy and solve the flow and energy equations simultaneously to complete the heat transfer simulation.

16.2.2.5. Conjugate Heat Transfer

For conjugate heat transfer (CHT) problems, particularly those with combustion, the dominant time / iteration scales in the fluid and solid zones are often several orders of magnitude different. For transient simulations, solving the coupled fluid-solid interface with the same time step size for both fluid and solid domains is prohibitively expensive, and so Fluent allows for specifying a different time step size for solid domains. Such solid time stepping can be combined with a Loosely Coupled CHT options that provides instantaneous implicit and also time-averaged explicit options to couple fluid-solid interface, ensuring continuity of heat flow and temperature; this Loosely Coupled CHT option is also available for steady flows. You can also change the settings for solid zones that have an anisotropic thermal conductivity (other than biaxial).

Note:

For general CHT applications, the correct temporal evolution of temperature in the solid can only be obtained when the same time step size / number of iterations is used in fluid and solid zones.

For details, see the following sections:

16.2.2.5.1. Specifying the Solid Time Step Size

16.2.2.5.2. Loosely Coupled Conjugate Heat Transfer

16.2.2.5.3. Time Averaged Explicit Thermal Coupling

16.2.2.5.4. Settings for Anisotropic Solid Zones

16.2.2.5.1. Specifying the Solid Time Step Size

When solving both the fluid and solid zones with a time step size that is suitable for the fluid zone, the computation of the energy equation at the solid zones becomes expensive. Specifying a larger time-step size accounts for the high thermal inertia of the solid zones which allows them to reach their thermal equilibrium faster. Note that the thermal field for the solid zones will still be updated at the same rate as fluid zones and they will remain fully coupled, so specifying a larger time step size by itself does not reduce redundant calculations or increase the performance.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

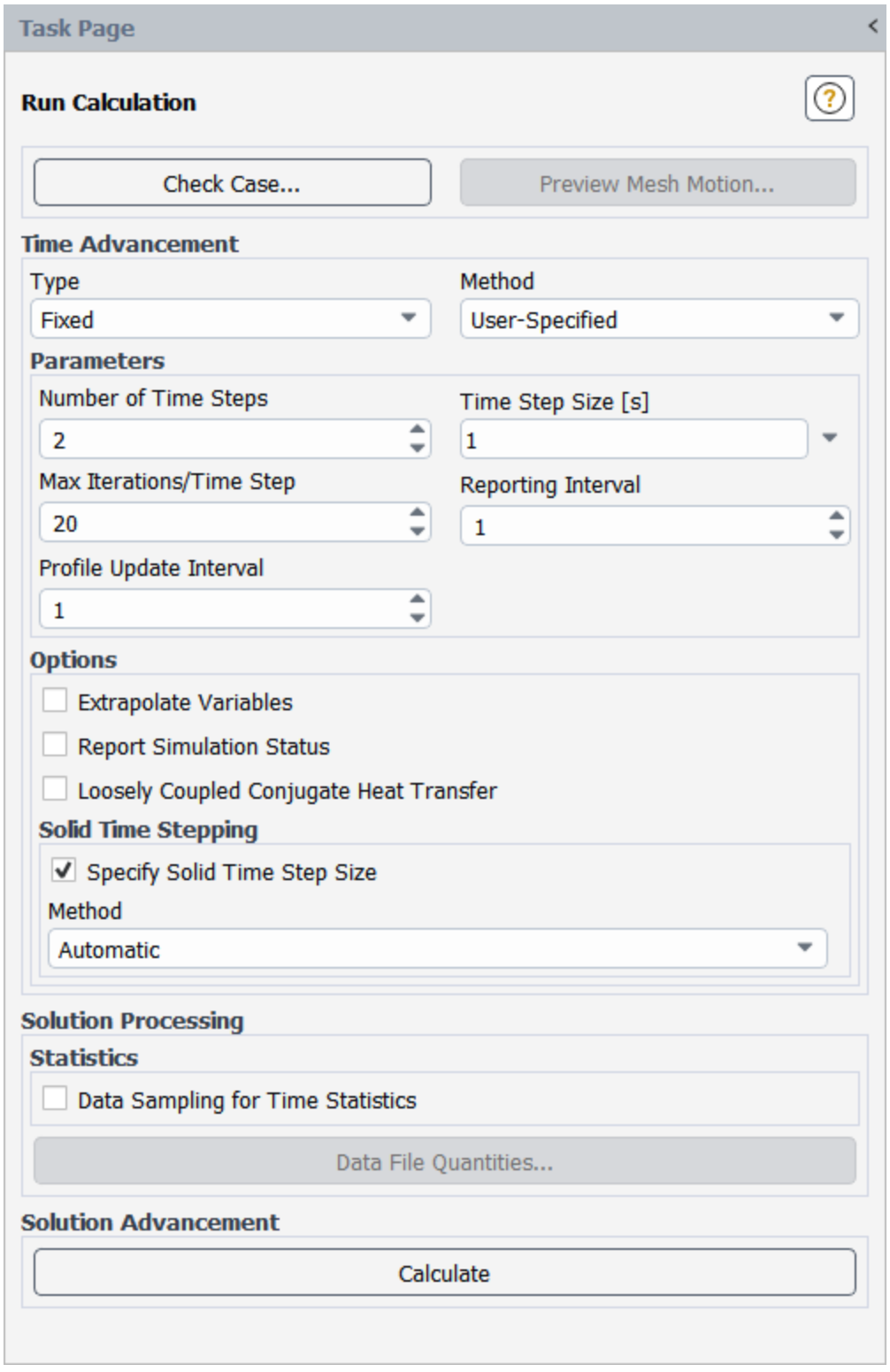
You can enhance the performance of the simulation for some conjugate heat transfer cases by using the **Loosely Coupled Conjugate Heat Transfer** option along with the **Specify Solid Time Step Size** option. For details and limitations, see Loosely Coupled Conjugate Heat Transfer (p. 2172).

For a transient case with solid and fluid zones and the energy equation enabled, you can specify a different solid time step size using the **Run Calculation** task page:

Solution → Run Calculation

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.4: The Run Calculation Task Page Showing Solid Time Stepping



1. Under **Options**, enable the **Specify Solid Time Step Size** option.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Use the default **Automatic** time step size from the **Method** drop-down list to have the time step size calculated as described in Automatic Time Step Size Calculation (p. 2172); otherwise, select **User-Specified** to enter your own **Time Step Size**.

Note:

You can enhance the performance of the simulation for some conjugate heat transfer cases by using the **Loosely Coupled Conjugate Heat Transfer** option along with the **Specify Solid Time Step Size** option. For details and limitations, see Loosely Coupled Conjugate Heat Transfer (p. 2172).

16.2.2.5.1.1. Automatic Time Step Size Calculation

The default **Automatic** time step size is calculated by ANSYS Fluent using the following formula:

$$\Delta t_{solid} = \frac{L_{scale_solid}^2}{\alpha} \tag{16.1}$$

where,

- L_{scale_solid} is the representative length scale and is calculated as $\sqrt[3]{Solid_Volume}$.
- α is $\frac{k}{\rho c_p}$, the representative velocity scale, where k is the conductivity, ρ is the density, and c_p is the specific heat capacity of the solid.

The calculation for Δt_{solid} results in an approximation for the solid time step size, which might aide in the solution running more efficiently. It should be noted, however, that this is only an engineering approximation as there is no general way to calculate a meaningful length scale for an arbitrary geometry.

16.2.2.5.2. Loosely Coupled Conjugate Heat Transfer

For conjugate heat transfer (CHT) simulations, by default a fully coupled approach is used, so that the whole domain (including fluid and solid zones) is solved together and the energy field in the solids and fluids is solved and updated at every iteration / fluid time step. You can instead loosely couple the CHT, so that multi-domain architecture is used (within a single Fluent session) to compute the energy field in solid zones at only a few instants during the simulation, and thereby increase the speed of the simulation. With this approach, the temperature boundary condition from the solid zones is assigned to the fluid domain when the solid zones are not solved, and then periodically (at coupling) both the fluid and solid zones are solved based on your selection for **Coupling Method**.

Note:

Loosely coupled CHT has following limitations and capabilities.

- The case must have both fluid and solid zones.
- The energy and flow equations must be enabled.
- It is not available for the density-based solver.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

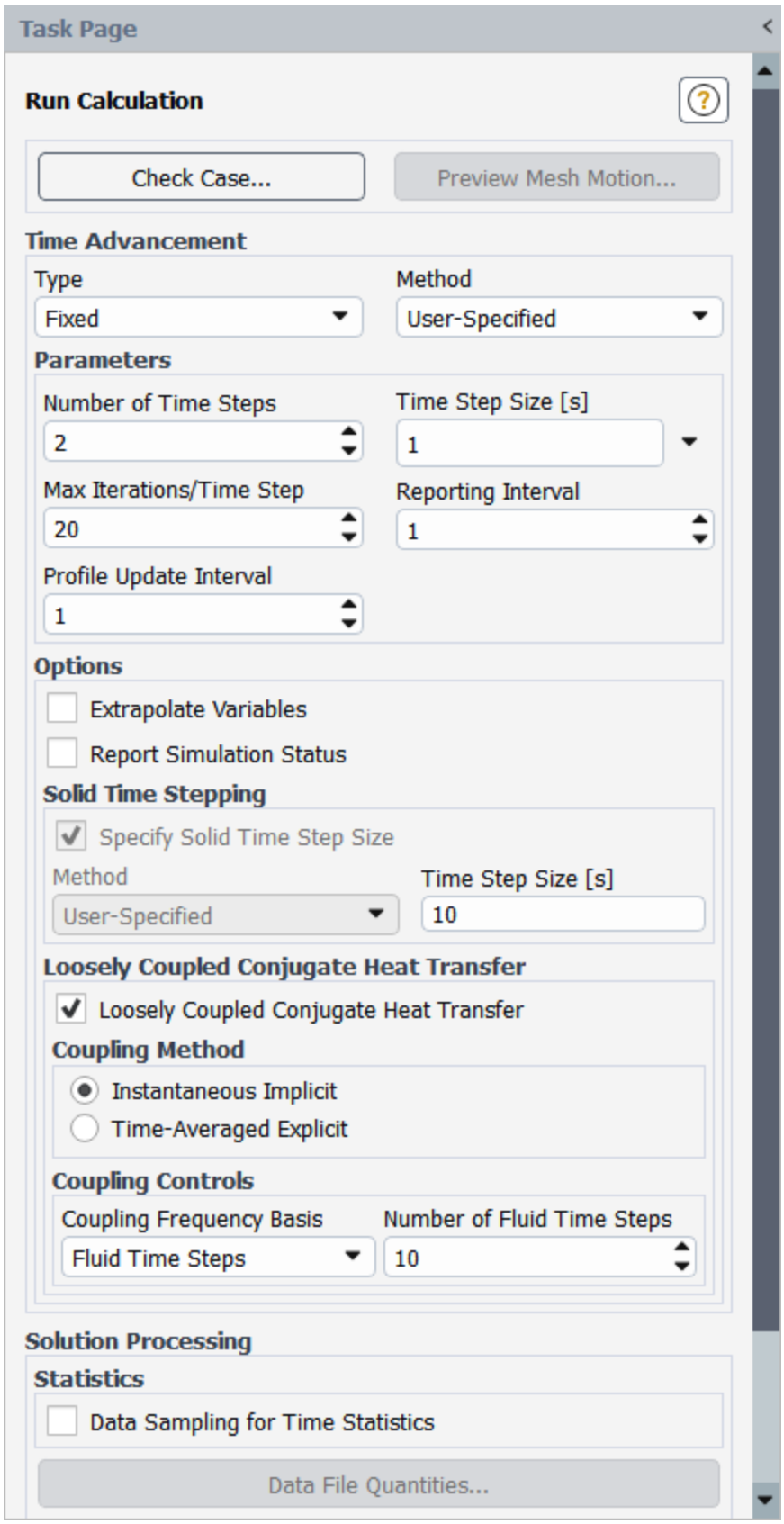
- It requires that the mesh does not change at the fluid-solid interface in order to be accurate. Therefore it is not supported when this interface undergoes remeshing and/or layering as part of a dynamic mesh or automatic adaption.
- It is not available with porous zones.
- It is not available if the case has a mesh interface that uses the **Mapped** option.
- It is disabled if time-averaged explicit thermal coupling is already selected for one or more coupled walls. For details, see Time Averaged Explicit Thermal Coupling (p. 2177).
- It is disabled if the case has shell conduction zones.
- Time-averaged explicit thermal coupling (Time Averaged Explicit Thermal Coupling (p. 2177)) and the **Time-Averaged Explicit** coupling method for Loosely Coupled CHT both employ the same explicit coupling method. The former solves the solid domain at each fluid time step, whereas the latter solves the solid domain at the specified coupling frequency.

You can loosely couple the CHT using the **Run Calculation** task page:

Solution → Run Calculation

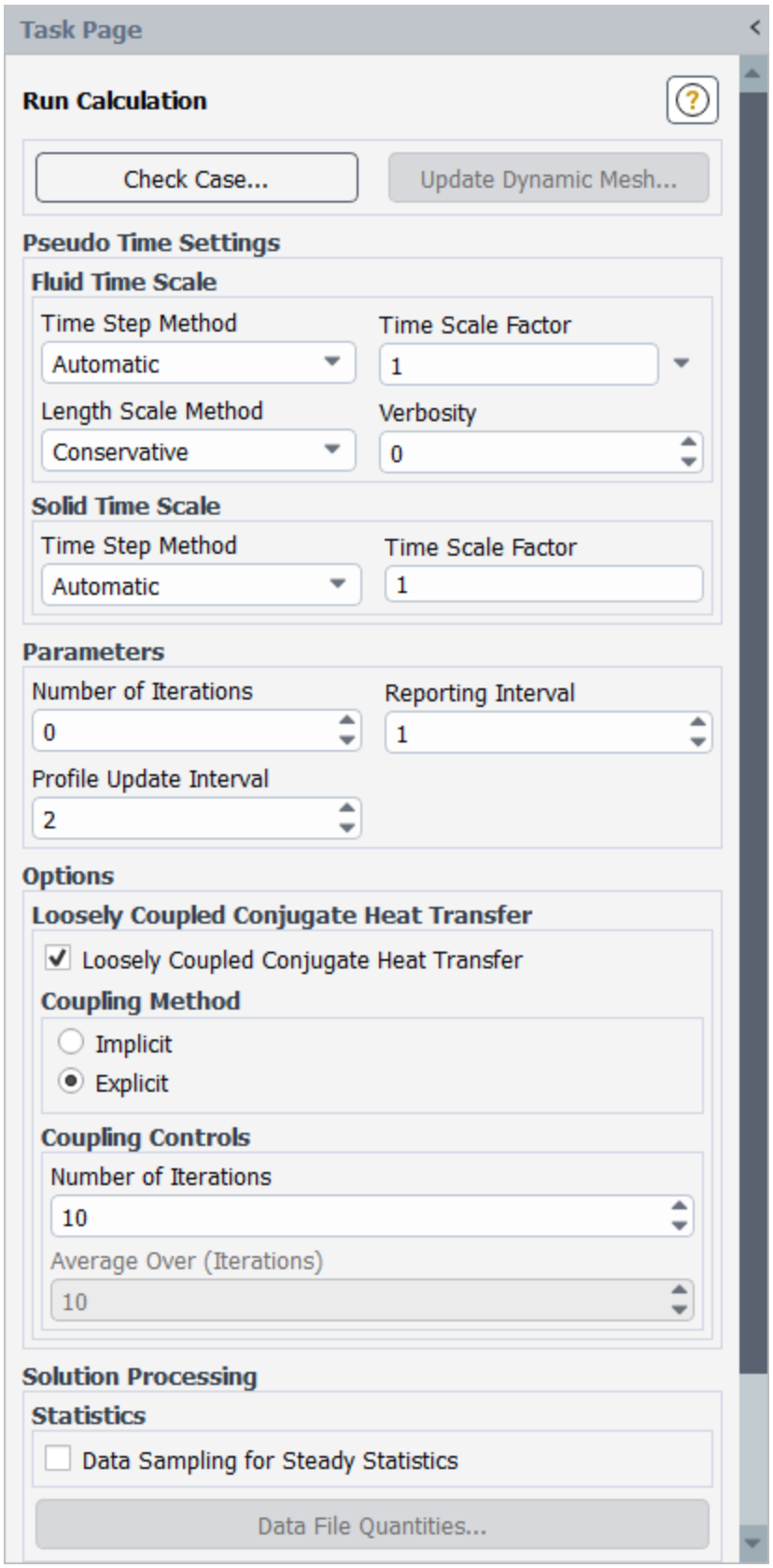
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.5: The Loosely Coupled Conjugate Heat Transfer Option for a Transient Flow



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.6: The Loosely Coupled Conjugate Heat Transfer Option for a Steady Flow



1. Under **Options**, enable the **Loosely Coupled Conjugate Heat Transfer** option.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. For transient simulations, perform the following steps:

   a. Make a selection from the **Coupling Method** list.

      - **Instantaneous Implicit**

        This method solves the fluid-solid interface in a fully coupled manner like conventional coupled thermal boundary condition at each coupling step. For details, see Thermal Conditions for Two-Sided Walls (p. 1472).

      - **Time-Averaged Explicit**

        In this method, Dirichlet – Neumann explicit thermal coupling is employed at each coupling step. A convective thermal boundary condition is applied at the wall adjacent to the solid and the instantaneous solid-side temperature is applied on the wall adjacent to the fluid. Time-averaged heat flow on the fluid side is used to compute the convective thermal boundary condition parameters (heat transfer coefficient and free-stream temperature) on the solid side.

   b. Under **Coupling Controls**, make a selection from the **Coupling Frequency Basis** drop-down list:

      - Select **Time** to have the coupling performed at a set **Time Period**. This option is only available for the **Instantaneous Implicit** coupling method.

      - Select **Fluid Time Steps** to have the coupling performed at a set **Number of Fluid Time Steps**. This is not available when using adaptive time stepping.

        If **Time-Averaged Explicit** is selected for the coupling method, **Average Over (Time Steps)** is shown for information only. This is the number of time steps used for computing time-averaged thermal variables. The input for this field is automatically copied from the specified **Number of Fluid Time Steps**.

   c. The **Specify Solid Time Step Size** option is automatically enabled with the **User-Specified** method selected (note that the **Automatic** method is not available for transient simulations with loosely coupled CHT). You should ensure that the **Time Step Size** is equal to or larger than the period for the loose coupling.

3. For steady simulations, perform the following steps:

   a. Make a selection from the **Coupling Method** list.

      - **Implicit**

        This method solves the fluid-solid interface in a fully coupled manner like conventional coupled thermal boundary condition at each iteration. For details, see Thermal Conditions for Two-Sided Walls (p. 1472).

      - **Explicit**

        In this method, Dirichlet – Neumann explicit thermal coupling is employed at each coupling iteration. A convective thermal boundary condition is applied at the wall adjacent to the solid and the instantaneous solid-side temperature is applied on the wall adjacent to the fluid. Iteration-averaged heat flow on the fluid side is used to compute

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the convective thermal boundary condition parameters (heat transfer coefficient and free-stream temperature) on the solid side.

b. Under **Coupling Controls**, enter the **Number of Iterations** at which you want the coupling performed.

If **Explicit** is selected for coupling method, **Average Over (Iterations)** is shown for information only. This is the number of iterations used for computing iteration-averaged thermal variables. The input for this field is automatically copied from the specified **Number of Iterations**.

Note:

The following practices are recommended:

- For transient CHT simulations, it is recommended that you use a high coupling frequency at the beginning and gradually decrease the coupling frequency when the temperature of the solid zones starts to reach its equilibrium state. This will help in the rapid and smooth convergence of solid temperature without compromising of the performance gain.
- Define the coupling frequency between fluid and solid to match the principal frequency of the fluid thermal field fluctuations, in order to improve accuracy.

16.2.2.5.3. Time Averaged Explicit Thermal Coupling

For transient simulations involving conjugate heat transfer, you have the option of specifying a time averaged explicit coupling. This method solves the energy equation in the fluid and solid concurrently, which is different from the Loosely Coupled Conjugate Heat Transfer (p. 2172) method, where solids are only solved with fluids when needed. The fluid and solid domains typically advance the solution at different time steps within each coupling cycle (see Specifying the Solid Time Step Size (p. 2169)). Each coupling cycle is specified for a number of time steps at which the exchange of thermal boundary condition quantities occurs.

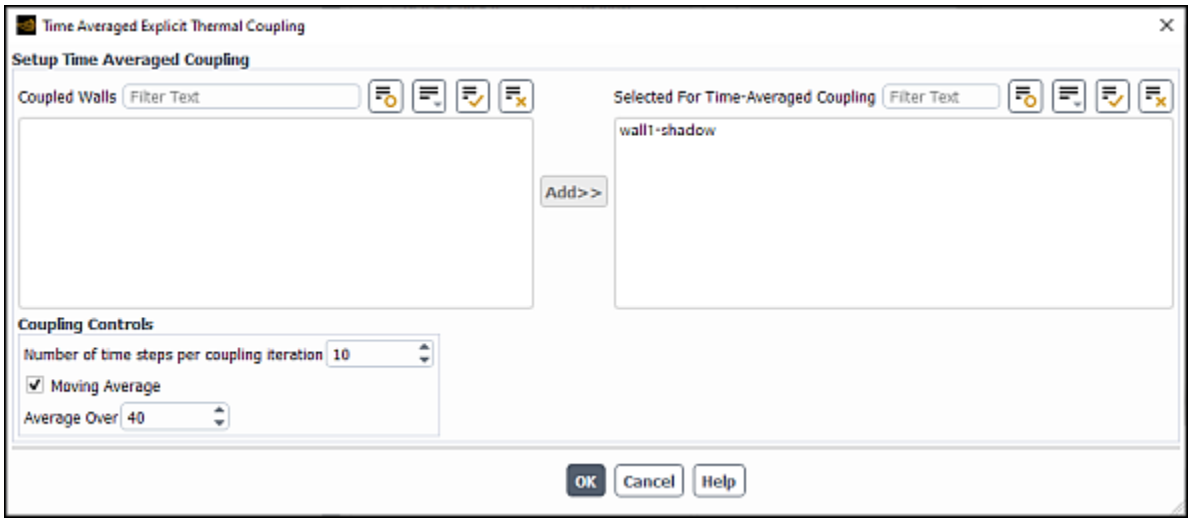
Well established Dirichlet – Neumann thermal coupling is employed at each cycle. At the end of each cycle, the convective thermal boundary condition is applied at the wall adjacent to the solid and the instantaneous solid side temperature is applied on the wall adjacent to the fluid. Time-averaged heat flow on the fluid side is used to compute the convective thermal boundary condition parameters (heat transfer coefficient and freestream temperature on the solid side).

You can enable this method for coupled walls from the **Time Averaged Explicit Thermal Coupling** dialog box:

Physics → **Model Specific** → **Explicit Thermal Coupling...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.7: The Time Averaged Explicit Thermal Coupling Dialog Box



1. Select the wall pairs under **Coupled Wall** that you would like to convert to explicit time-averaged coupling.

2. Click **Add>>**. A prompt will appear warning that this operation slits the selected wall pairs (therefore altering the mesh). If you would like to retain the data for the current case, you should save the `.cas` and `.dat` files before continuing.

   The walls selected for conversion will appear under **Selected For Time-Averaged Coupling**.

3. Under **Coupling Controls**, specify the **Number of time steps per coupling iteration** for the explicit coupling.

4. Optionally select **Moving Average** and specify **Average Over**. Fluent will still perform a coupling iteration with the frequency specified by **Number of time steps per coupling iteration** but will the use data averaged over the previous number of time steps as specified in **Average Over**.

   In the above example shown in Figure 16.7: The Time Averaged Explicit Thermal Coupling Dialog Box (p. 2178), the first coupling iteration will occur at the 40th time step with averaged data over the first 40 time steps. The next coupling iteration will occur at the 50th time step using data averaged from the 10th to 49th time steps.

   Upon enabling this option, Fluent automatically creates the required unsteady datasets for each coupled wall. For more information on unsteady statistics, see Inputs for Time-Dependent Problems (p. 3645).

5. Click **OK** to close the dialog box.

Note the following limitations:

- If you replace the mesh in your current case, you will have to redefine any transient explicit thermal coupled walls.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16.2.2.5.4. Settings for Anisotropic Solid Zones

When performing a steady or transient conjugate heat transfer (CHT) simulation, you can change the following settings for pressure-based simulations that have the energy equation enabled and an anisotropic option (other than biaxial) selected for the thermal conductivity of the solid.

Heat Flux Methods

You can select the method used to calculate the heat flux for the anisotropic solid zones. The determining factor when choosing the method is whether the solid has highly anisotropic thermal conductivity properties (that is, has a conductivity in one direction that is more than 100 times the value in another direction), as such properties can result in convergence problems or reports that you are exceeding temperature limits. The following heat flux methods are available:

- `boundary-stabilized-implicit`

  This method is selected by default. It is robust for many cases with highly anisotropic solids, while only moderately increasing the number of iterations / time steps needed to reach convergence.

- `stabilized-implicit`

  While this is the most robust method for cases with highly anisotropic solids, it also results in the greatest increase in the number of iterations / time steps needed to reach convergence of the available heat flux methods.

- `standard`

  This method results in the fastest convergence rate, but should only be used when the solid is not highly anisotropic.

To select the heat flux method for cases with anisotropic solids, use the following text command:

`solve` → `set` → `advanced` → `anisotropic-solid-heat-transfer` → `flux`

Temperature Gradient Methods

You can select an improved method (named `local-neighbors`) for calculating the temperature gradient for the anisotropic solid zones. Compared to the default `standard` method, the `local-neighbors` method increases the speed of the solid zone calculations to be 2–3 times faster.

To use the improved temperature gradient method, ensure that **Least Squares Cell Based** is selected from the **Gradient** drop-down list in the **Solution Methods** task page, and then select the `local-neighbors` method using the following text command:

`solve` → `set` → `advanced` → `anisotropic-solid-heat-transfer` → `gradient`

Relaxation for the Heat Flux Coefficient

You can reduce the relaxation for the heat flux coefficient to be less than the default value of 1 using the following text command. In certain anisotropic conductivity cases, such under-relaxation can help with convergence.

`solve` → `set` → `advanced` → `anisotropic-solid-heat-transfer` → `relaxation`

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For further details on anisotropic thermal conductivity properties, see Anisotropic Thermal Conductivity for Solids (p. 1648).

16.2.3. Postprocessing Heat Transfer Quantities

For information about postprocessing heat transfer quantities, see the following sections:

16.2.3.1. Available Variables for Postprocessing
16.2.3.2. Definition of Enthalpy and Energy in Reports and Displays
16.2.3.3. Reporting Heat Transfer Through Boundaries
16.2.3.4. Reporting Heat Transfer Through a Surface
16.2.3.5. Reporting Averaged Heat Transfer Coefficients
16.2.3.6. Exporting Heat Flux Data

16.2.3.1. Available Variables for Postprocessing

Ansys Fluent provides reporting options for simulations involving heat transfer. You can generate graphical plots or reports of the variables listed under the **Temperature...** or **Wall Fluxes...** category. See Field Function Definitions (p. 4167) for the list of available variables and their definitions.

16.2.3.2. Definition of Enthalpy and Energy in Reports and Displays

The reported enthalpy values may be different between the pressure-based and density-based solver depending on the properties of the working fluid, compressible, incompressible, ideal gas, etc. For details, see the discussion of enthalpy in The Energy Equation in chapter 5, Heat Transfer of the Theory Guide.

16.2.3.3. Reporting Heat Transfer Through Boundaries

You can use the **Flux Reports** dialog box to compute the heat transfer through each boundary of the domain, or to sum the heat transfer through all boundaries to check the heat balance.

Results → **Reports** → **Fluxes** → **Edit...**

It is recommended that you perform a heat balance check to ensure that your solution is truly converged.

Note:

- For the Composition PDF Transport model, the **Flux Reports** dialog box reports the **Total Heat Transfer Rate** and **Total Sensible Heat Transfer Rate** only for the Lagrangian phase solution, which does not always guarantee overall mass conservation due to the nature of the discrete phase model. To check the balance of heat transfer rates that are based on the continuous phase computation, you should use the **Surface Integrals** dialog box to create a report for the **Flow Rate** of **Enthalpy** (**Temperature** category). (See Reporting Heat Transfer Through a Surface (p. 2181) for details.)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Fluxes can only be computed on boundaries and not on interior or user defined surfaces such as planes, points, iso-surfaces, and so on.

16.2.3.4. Reporting Heat Transfer Through a Surface

You can use the **Surface Integrals** dialog box to compute the heat transfer through any boundary or any surface created using the methods described in Creating Surfaces and Cell Registers for Displaying and Reporting Data (p. 3813).

Results → **Reports** → **Surface Integrals** **Edit...**

To report the mass flow rate of enthalpy

$$Q = \int H \rho \vec{v} \cdot d\vec{A} \tag{16.2}$$

choose **Flow Rate** for the **Report Type** in the **Surface Integrals** dialog box, select **Enthalpy** (in the **Temperature...** category) as the **Field Variable**, and select the surface(s) on which to integrate.

16.2.3.5. Reporting Averaged Heat Transfer Coefficients

The **Surface Integrals** dialog box can also be used to generate a report of averaged heat transfer coefficient h on a surface ($\frac{1}{A}\int h dA$).

Results → **Reports** → **Surface Integrals** **Edit...**

In the **Surface Integrals** dialog box, choose **Area-Weighted Average** for **Report Type**, select **Surface Heat Transfer Coef.** (in the **Wall Fluxes...** category) as the **Field Variable**, and select the surface.

16.2.3.6. Exporting Heat Flux Data

It is possible to export heat flux data on wall zones (including radiation) to a generic file that you can examine or use in an external program. To save a heat flux file, you will use the `custom-heat-flux` text command.

`file` → `export` → `custom-heat-flux`

Heat transfer data will be exported in the following free format for each face zone that you select for export:

```
zone-name nfaces
x_f y_f z_f A Q T_w T_c HTC
.
.
.
```

Each block of data starts with the name of the face zone (`zone-name`) and the number of faces in the zone (`nfaces`). Next there is a line for each face (that is, `nfaces` lines), each containing the components of the face centroid (`x_f`, `y_f`, and, in 3D, `z_f`), the face area (A), the heat

transfer rate (Q), the face temperature (T_w), the adjacent cell temperature (T_c), and the heat transfer coefficient (HTC). If the heat transfer coefficient is calculated based on wall function (Equation 42.69 (p. 4281)), then Q is the convective heat transfer rate. Otherwise, Q will be the total heat transfer rate, including radiation heat transfer.

16.2.4. Natural Convection

For information on how to include the effects of natural convection in buoyancy-driven flow problems, see Buoyancy-Driven Flows and Natural Convection (p. 1387).

16.2.5. Shell Conduction

For information about shell conduction considerations, see the following sections:

16.2.5.1. Introduction

16.2.5.2. Physical Treatment

16.2.5.3. Limitations of Shell Conduction Walls

16.2.5.4. Managing Conduction Walls

16.2.5.5. Initializing Shells

16.2.5.6. Locking the Temperature for Shells

16.2.5.7. Postprocessing Shells

16.2.5.1. Introduction

By default, Ansys Fluent treats walls as having zero-thickness and presenting no thermal resistance to heat transfer across them. If a thickness is specified for a wall (thereby making it a thin wall, as described in Thin-Wall Thermal Resistance Parameters (p. 1470)) then the appropriate thermal resistance across the wall thickness is imposed, although conduction is considered in the wall in the normal direction only. There are applications, however, where conduction in the planar directions of the wall is also important. For these applications, you have two options: you can either mesh the thickness, or you can use the shell conduction approach. Shell conduction can be used to model one or more layers of wall cells without the need to mesh the wall thickness in a preprocessor. When the shell conduction approach is utilized, you have the ability to easily switch on and off conjugate heat transfer on any wall. If you create a shell—either by enabling **Shell Conduction** in an individual **Wall** dialog box (as described in Shell Conduction (p. 1475)) or by defining multiple walls as shell conduction zones using the **Conduction Manager** dialog box (as described in Managing Conduction Walls (p. 2184))—and define the settings for the shell layer(s) using the **Conduction Layers** dialog box, then Ansys Fluent will automatically grow the specified layers of cells. The cells that are grown will be either prism cells or hex cells, depending on the type of face mesh that is utilized; at no point are the cells visible in the displayed mesh.

Shell conduction can be used to account for thermal mass in transient thermal analysis problems (see Locking the Temperature for Solid and Shell Zones (p. 1382) for more information). It can also be used for multiple junctions and allows heat conduction through the junctions. Shell conduction can be applied on boundary walls as well as two-sided walls.

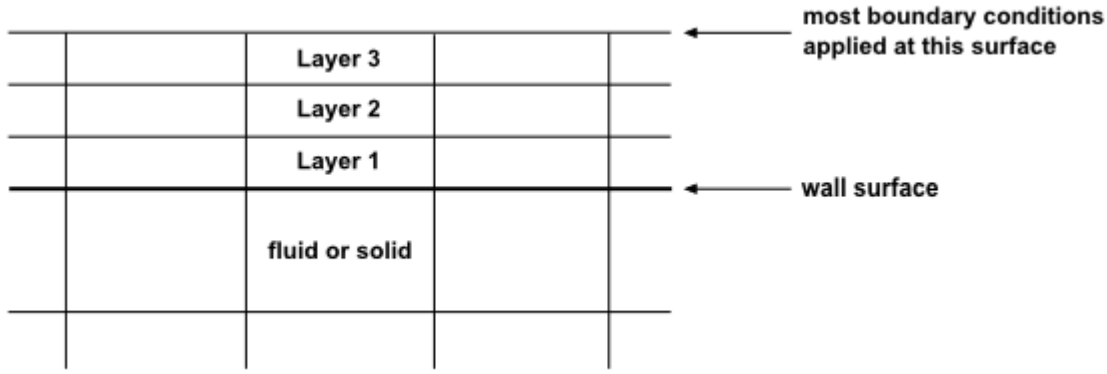
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16.2.5.2. Physical Treatment

In the case where shell conduction is applied on a boundary wall, the original surface is referred to as the "wall surface" and is always adjacent to the fluid / solid cells. The first time the shell model is enabled at any wall surface, Fluent grows the required shell zones. These layers are numbered sequentially, starting with the one closest to the wall surface. See Figure 16.8: A Boundary Wall with Shell Conduction (p. 2183) for a visualization of how the solver treats shells.

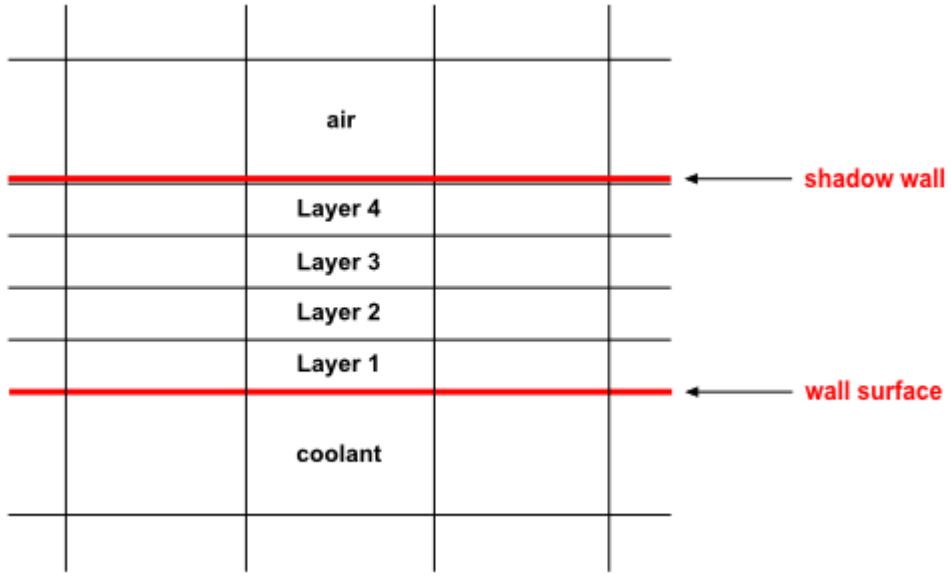
For the most part, the boundary conditions that you specify for the boundary wall are applied to the layer surface that is furthest from the wall surface; only the internal emissivity is applied at the wall surface. The sides of the shell zone require boundary conditions as well. If the shell is connected to another wall that does not have shell conduction enabled, the shell side will take the boundary condition of the attached wall. The sides will be adiabatic if they are connected to face zones that have a boundary condition type other than **wall**. If the attached wall has shell conduction enabled, then the common sides at the junction will be coupled.

Figure 16.8: A Boundary Wall with Shell Conduction



The layers of cells grown for two-sided walls can be visualized in a similar way to Figure 16.8: A Boundary Wall with Shell Conduction (p. 2183), except that the layer surface that is furthest from the wall surface is also adjacent to a shadow wall (which is then adjacent to the other fluid / solid zone).

Figure 16.9: A Two-Sided Wall with Shell Conduction



16.2.5.3. Limitations of Shell Conduction Walls

The following is a list of limitations for the shell conduction model:

- Shell conduction only supports constant wall thickness.
- Shell conduction cannot be used on moving wall zones.
- Shell conduction cannot be used with FMG initialization.
- Shell conduction is not available when the wall is set up to receive thermal data via system coupling.
- Shell conduction is not available for 2D.
- Shell conduction is available only when the pressure-based solver is used.
- Shell conducting walls cannot be split or merged. If you need to split or merge a shell conducting wall, you will need to turn off the **Shell Conduction** option for the wall (in the **Wall** dialog box), perform the split or merge operation, and then enable **Shell Conduction** for the new wall zones.
- The shell conduction model cannot be used on a wall zone that has undergone hanging node adaption. If you want to perform such adaption elsewhere in the computational domain, be sure to use a boolean operation in an adaption expression to exclude a register containing the shell conducting walls. For example, `NOT(region_0)`. *Refer to Adapting the Mesh (p. 3773) for additional information on adapting the mesh*.
- Fluxes at the ends of a shell conducting wall are not included in heat balance reports. These fluxes are accounted for correctly in the Ansys Fluent solution, but are not listed in the flux report.
- For a one-sided wall with shell conduction model enabled, the specified thermal BC is applied to the external surface (side of the wall away from the domain). Therefore, the reported flux on the wall would not correspond to the specified thermal BC.
- The junction of a wall with shell conduction enabled and a non-conformal coupled wall is not supported. Such a junction will not be thermally connected, that is, there will be no heat transfer between the shell and the mesh interface wall.
- When running the parallel solver with the shell conduction model, note that coupled walls are encapsulated. If you encounter problems with the partitioning of the mesh, you can try changing the encapsulation method to see if that resolves the problem (see Troubleshooting (p. 4342) for this and other troubleshooting options).
- Shells with a sharp bend may yield non-physical temperature fields when using Green-Gauss Node Based gradients. This applies to any angle (acute or obtuse) with the effect worsening as the angle approaches 90 degrees. To resolve the issue, you should consider breaking the sharp angled shells into multiple shells or, in the event that this cannot be done, enabling Warped Face Gradient Correction.

16.2.5.4. Managing Conduction Walls

The **Conduction Manager** dialog box provides a convenient way for you to manage, define, and display multiple conduction zones. You can

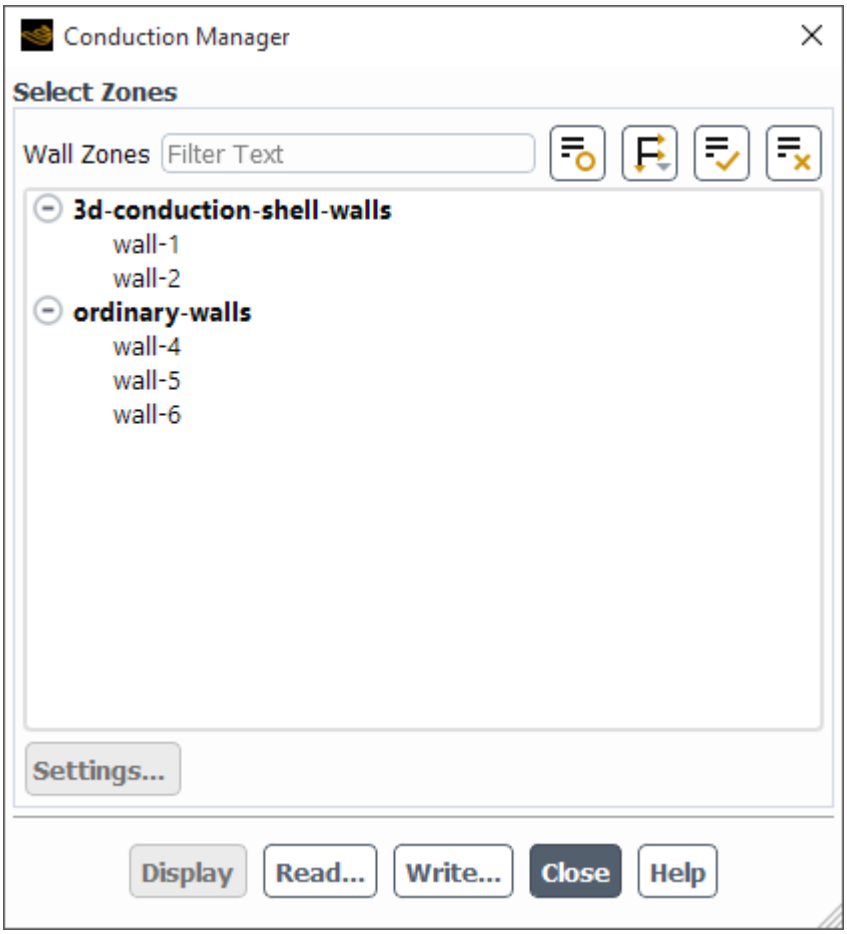
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- display any wall zone
- enable or disable shell conduction for a wall
- define the settings for conduction walls or shell layers, such as thickness, material, and heat generation rate

Note that you can still enable shell conduction for a wall and define the related settings using the **Wall** dialog box, as described in Shell Conduction (p. 1475). However, the **Conduction Manager** dialog box provides you with an alternative to do such activities from a single dialog box for all the walls, instead of visiting the individual **Wall** boundary condition dialog boxes.

Physics → Model Specific → Conduction Manager...

Figure 16.10: The Conduction Manager Dialog Box



To manage shell conduction using the **Conduction Manager** dialog box, perform the following steps:

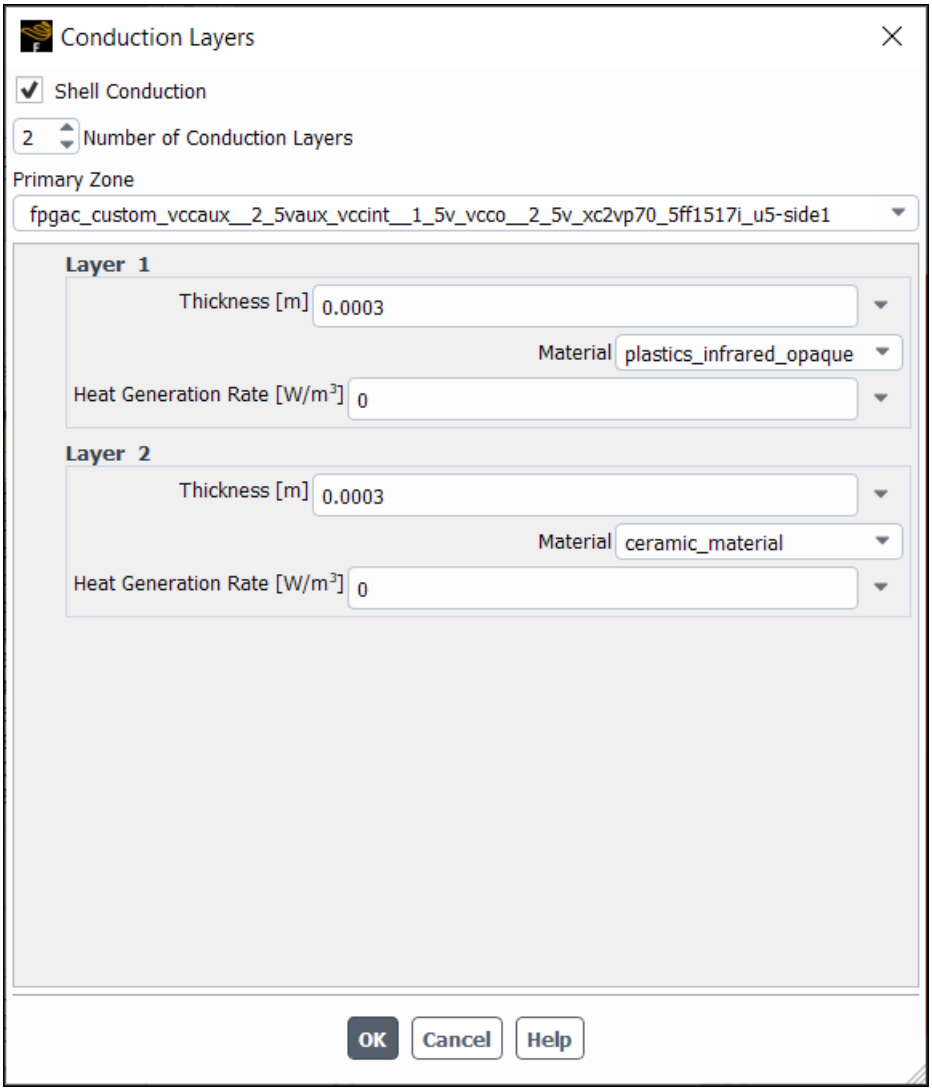
1. (optional) If you have previously generated a CSV file that contains shell conduction settings for the layers of the walls, you can read it by clicking **Read...** at the bottom of the **Conduction Manager** dialog box and using the dialog box that opens; otherwise, you should proceed to the steps that follow.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Using a `.csv` file can be helpful when you have a large number of layers and walls. It is not necessary to create the `.csv` file from nothing, as you can create a simplified `.csv` file with sample values using the **Write...** button, and then revise the settings using a spreadsheet program. See Conduction Settings File Format (p. 5694) for details about the format of the `.csv` file.

2. Select the walls you want to convert to shell conduction and click **Settings...**. The **Conduction Layer** dialog box opens.

Figure 16.11: Conduction Layers Dialog Box



3. Select **Shell Conduction** to enable shell conduction for the selected walls.

Note:

If the **Shell Conduction** option is disabled and you specify a thickness, the wall(s) will be treated as thin walls (1d conduction).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. If you have selected multiple walls, you can assign one to be the **Primary Zone**. The settings for the **Primary Zone** will be copied to the other walls that are currently selected.

5. Specify the number of layers, and then define the thickness, material, and heat generation rate for each layer. Note that you have the option of defining the heat generation rate using a user-defined function (UDF) that utilizes the `DEFINE_PROFILE` macro; for more information on creating and using user-defined functions, see the Fluent Customization Manual.

   Important:

   You must specify a nonzero-thickness for every layer.

6. Click **OK** to save your settings for the selected walls and close the **Conduction Layers** dialog box.

   The walls that have shell conduction enabled will appear listed under **3d-conduction-shell-walls** on Figure 16.10: The Conduction Manager Dialog Box (p. 2185).

7. If you want to save all of the settings you specified using the **Conduction Manager** dialog box as a `.csv` file, click **Write...** and specify a name in the dialog box that opens. You can edit the conduction settings in this `.csv` file using a spreadsheet program, and then read it in this or a separate case file, as described in step 1. Note that information will not be written for any wall that defines the heat generation rate using a UDF.

Note that you have the option of disabling shell conduction in every wall with a single action, by using the following text command. This capability is available in both serial and parallel mode.

`define` → `boundary-conditions` → `modify-zones` → `delete-all-shells`

16.2.5.5. Initializing Shells

Shell zones can be patched using the **Patch** dialog box, as described in Patching Values in Selected Cells (p. 3630).

Solution → **Initialization** **Patch...**

16.2.5.6. Locking the Temperature for Shells

You can lock (or "freeze") the temperature values for all the cells in shells and solid zones (including those to which you have a hooked an energy source through a UDF) by using the `solve/set/lock-solid-temperature?` text command, as described in Locking the Temperature for Solid and Shell Zones (p. 1382).

16.2.5.7. Postprocessing Shells

In order to facilitate the postprocessing of shells, surfaces (referred to as "shell surfaces") are created at the interface of adjacent layers, as well as at the layer surface that is furthest from the wall surface. Note that shell surfaces are not created on the sides of the shell zones or in the interior of the layers. The shell surfaces can then be selected from the **Surfaces** list available in a variety of dialog boxes

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

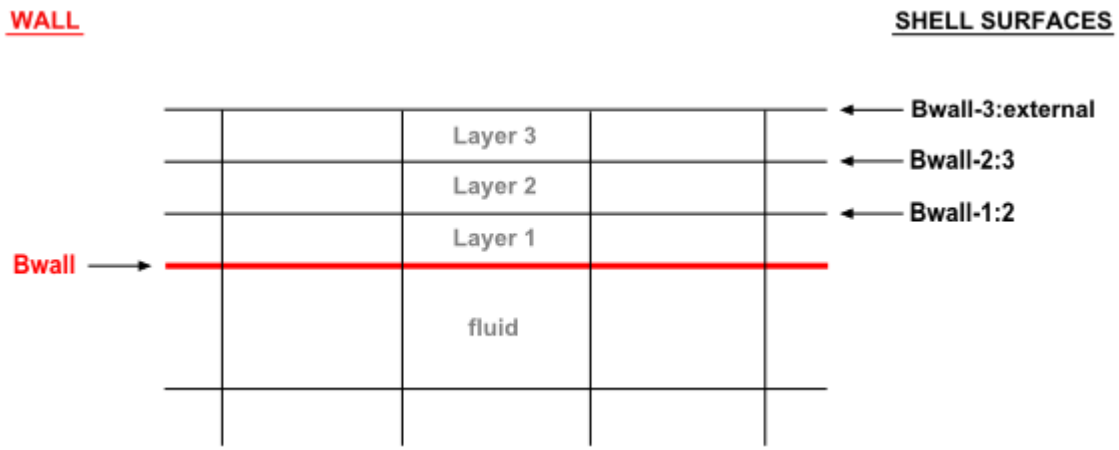
(for example, the **Contours** dialog box, the **Surface Report Definition** dialog box, and the **Solution XY Plot** dialog box) in the same way as the surfaces for the wall and (for two-sided walls) the shadow wall. When visualized in the graphics display window, these shell surfaces will be coincident with the wall they are associated with.

The default naming convention for these shell surfaces is <*wall_name*>**-**<*c0*>**:**<*c1*>, where:

- <*wall_name*> is the name of the wall on which the layers have been grown.
- <*c0*> is the rank of the layer on the c0 side of the interface (that is, the layer closest to the wall surface).
- <*c1*> is the rank of the layer on the c1 side of the interface (that is, the layer furthest from the wall surface). Note that the layer surface that is furthest from the wall surface does not have a layer on the c1 side, and so a different convention is used: for boundary walls, <*c1*> is set to **external**; for two-sided walls, <*c1*> is set to the name of the fluid / solid adjacent to the shadow wall.

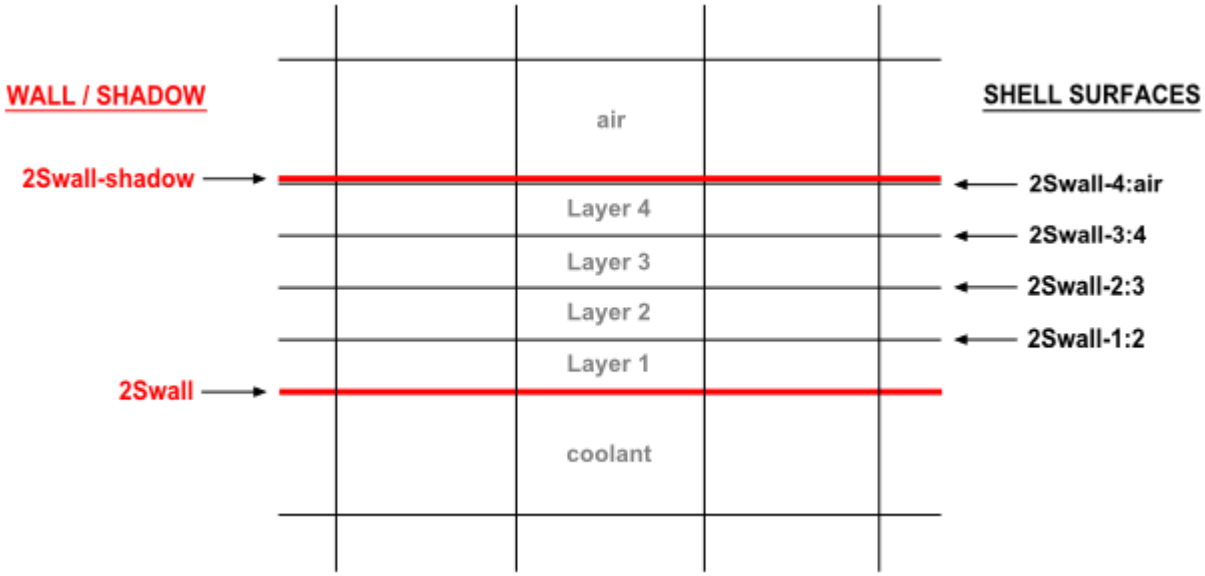
You have the ability to rename the shell surfaces, if you want. The following figures illustrate the default naming convention, where the wall/shadow surfaces are displayed in red and the shell surfaces are displayed in black.

Figure 16.12: Shell Surface Names for a Boundary Wall



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.13: Shell Surface Names for a Two-Sided Wall



You can use the walls, shadow walls, and shell surfaces to display the temperature of the interfaces and adjacent cells. The **Temperature...** category provides two options: the temperature of the cell on the c0 side of the surface is stored as **Static Temperature**; and the temperature of the surface itself is stored as **Wall Temperature**. If a more detailed analysis of the solid zone and surfaces is required, then you should consider creating layers of solid cells in your meshing application.

The following examples illustrate how to postprocess the surfaces shown in Figure 16.12: Shell Surface Names for a Boundary Wall (p. 2188):

- The contours of **Static Temperature** and **Wall Temperature** of **Bwall** will provide the temperature of the fluid cells adjacent to **Bwall** and the temperature of the **Bwall** surface itself, respectively.
- The contours of **Static Temperature** and **Wall Temperature** of **Bwall-1:2** will provide the temperature of the cells of **Layer 1** and the temperature of the interface between **Layer 1** and **Layer 2** (that is, the **Bwall-1:2** shell surface), respectively.
- The contours of **Static Temperature** and **Wall Temperature** of **Bwall-3:external** will provide the temperature of the cells of **Layer 3** and the temperature of the layer surface that is furthest from the wall surface (that is, the **Bwall-3:external** shell surface), respectively.

The following examples illustrate how to postprocess the surfaces shown in Figure 16.13: Shell Surface Names for a Two-Sided Wall (p. 2189):

- The contours of **Static Temperature** and **Wall Temperature** of **2Swall-4:air** will provide the temperature of the cells of **Layer 4** and the temperature of the interface between **Layer 4** and the shadow wall (that is, the **2Swall-4:air** shell surface), respectively.
- The contours of **Static Temperature** and **Wall Temperature** of **2Swall-shadow** will provide the temperature of the air cells adjacent to **2Swall-shadow** and the temperature of the shadow wall, respectively.

When postprocessing shells, note the following limitations/considerations:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- You should not enable **Node Values** when postprocessing any surfaces associated with shells surfaces. The layers share nodes at the edges, and thus the averaging at such nodes is erroneous.
- You cannot postprocess shells in Ansys CFD-Post, as shell surfaces are not supported.
- The **Boundary Values** option can be used for displaying contours on shell surfaces. The values with this option for **Wall Temperature** and **Static Temperature** are identical for shell surfaces.
- When post-processing the following options on shell surfaces, like any other surface, boundary values are considered:
  - Surface report definitions
  - XY Plot

16.2.6. Anisotropic Thermal Conductivity with Curvilinear Coordinate System (CCS)

Thermal conductivity can be defined using direction vectors but these directions are constant and therefore only relevant for geometries that are aligned with the global coordinate system. This feature allows the definition of anisotropic thermal conductivity where the anisotropy follows the shape of the geometry. This can be useful in defining anisotropic thermal conductivity for twisted geometries such as electric motor coils/windings.

To provide flexibility in handling such scenarios, a Curvilinear Coordinate System (CCS) that follows the shape of the geometry can be defined.

For more details on anisotropic conductivity, see Anisotropic Conductivity in Solids in the *Fluent Theory Guide*.

16.2.6.1. Workflow for Anisotropic Thermal Conductivity with CCS

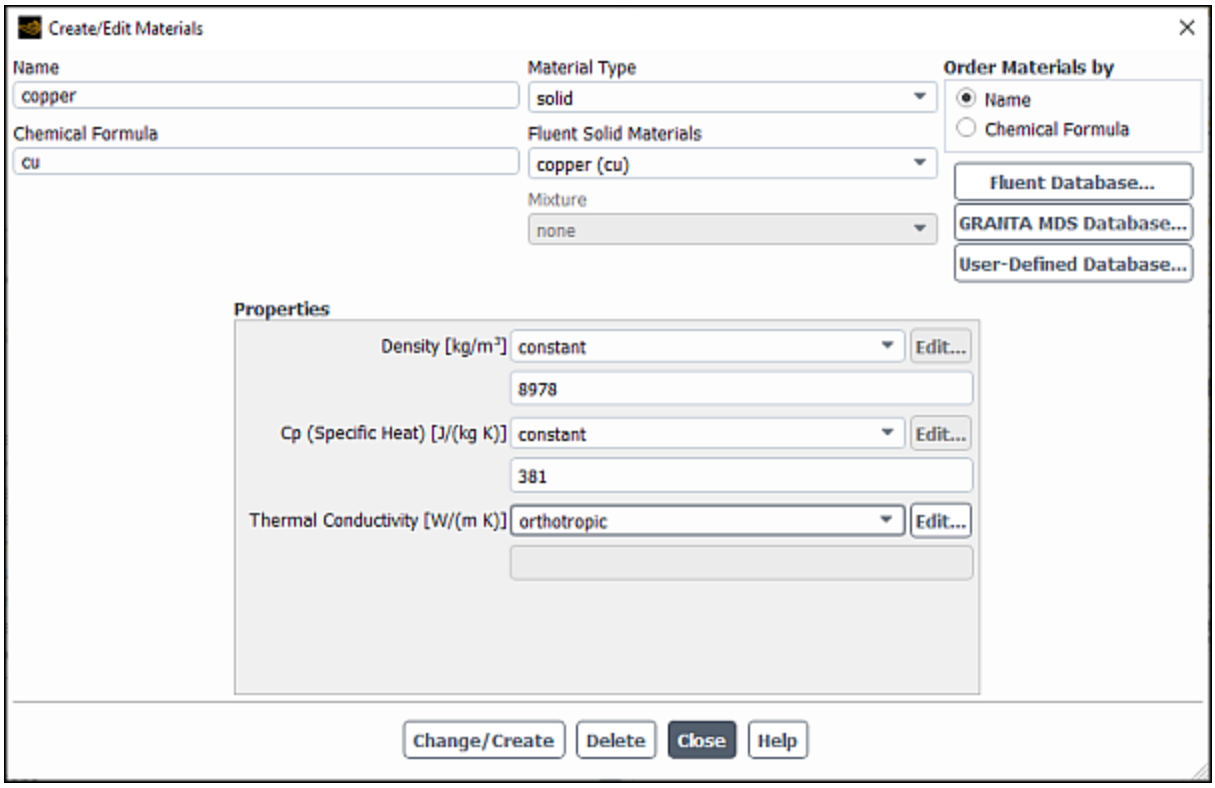
The general workflow for setting up anisotropic thermal conductivity is as follows:

1. For the solid material, set the **Thermal Conductivity** to **orthotropic**.

   Physics → **Materials** → **Create/Edit...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.14: Create/Edit Materials Dialog Box with Orthotropic Conductivity



2. Define a Curvilinear Coordinate System that follows the shape of the geometry. For details, see Curvilinear Coordinate Systems (p. 1164).

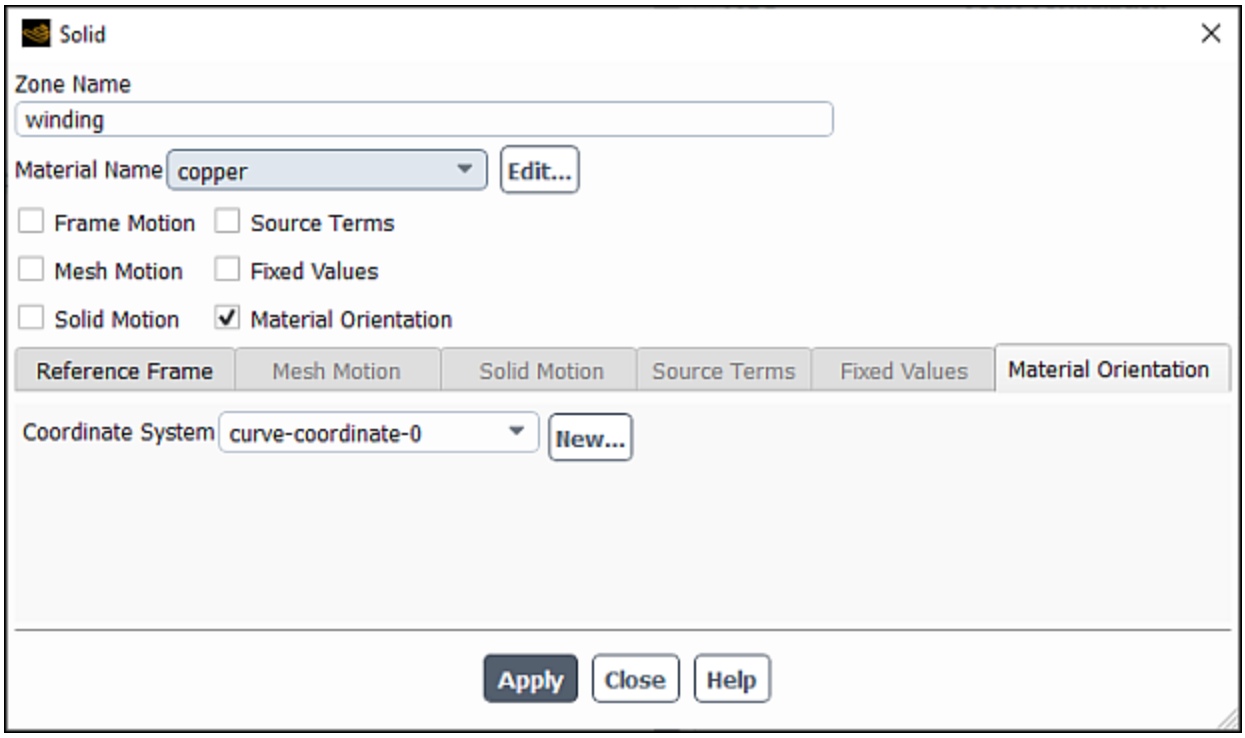
   Setup → Curvilinear Coordinate System → New...

3. Assign the CCS to the appropriate cell zone.

   Setup → Cell Zone Conditions → Solid → <cell zone name> → Edit...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.15: Solid Cell Zone Dialog Box with Defined CCS



1. Select **Material Orientation**.
2. On the **Material Orientation** sub-panel, for **Coordinate System**, select the CCS you created in step 2.

   If for **Coordinate System** you specify the default **From Material**, Fluent uses the directions specified in the Orthotropic Conductivity dialog box.

16.3. Modeling Radiation

Information about radiation modeling is presented in the following sections:

16.3.1. Using the Radiation Models
16.3.2. Setting Up the P-1 Model with Non-Gray Radiation
16.3.3. Setting Up the DTRM
16.3.4. Setting Up the S2S Model
16.3.5. Setting Up the DO Model
16.3.6. Setting Up the MC Model
16.3.7. Defining Material Properties for Radiation
16.3.8. Defining Boundary Conditions for Radiation
16.3.9. Solution Strategies for Radiation Modeling
16.3.10. Postprocessing Radiation Quantities
16.3.11. Solar Load Model

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For theoretical information about the radiation models in Ansys Fluent, refer to Modeling Radiation in the Theory Guide.

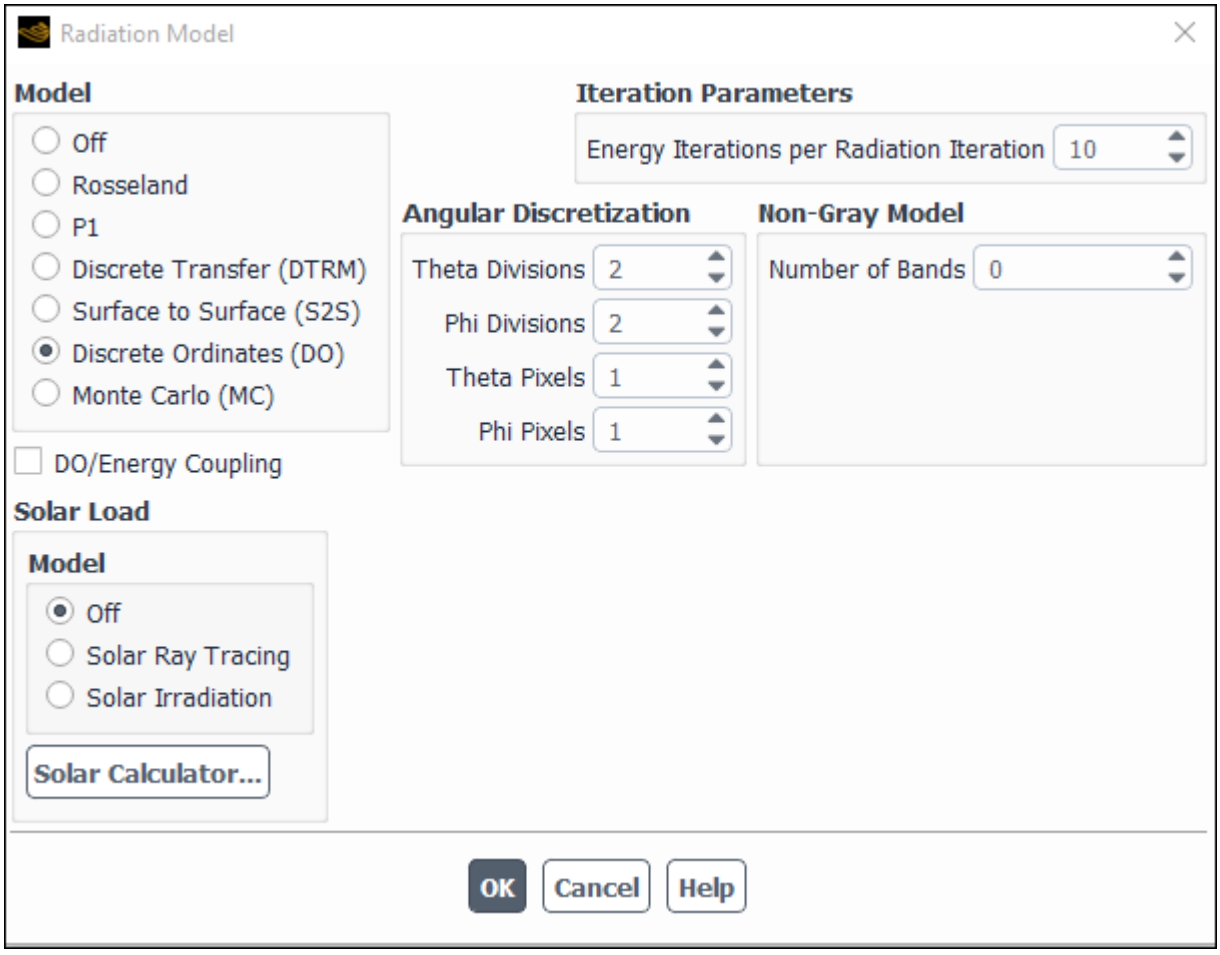
16.3.1. Using the Radiation Models

The procedure for setting up and solving a radiation problem is outlined below, and described in detail in referenced sections. Steps that are relevant only for a particular radiation model are noted as such. The steps that are pertinent to radiation modeling, are shown here. For information about inputs related to other models that you are using in conjunction with radiation, see the appropriate sections for those models.

1. Enable radiative heat transfer by selecting a radiation model (**Rosseland**, **P1**, **Discrete Transfer (DTRM)**, **Surface to Surface (S2S)**, **Discrete Ordinates**, or **Monte Carlo (MC)**) under **Model** in the **Radiation Model** dialog box (Figure 16.16: The Radiation Model Dialog Box (DO Model) (p. 2193)).

   Setup → Models → Radiation → Edit...

   Figure 16.16: The Radiation Model Dialog Box (DO Model)



Important:

The **Rosseland** model can be used only with the pressure-based solver.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that when the P-1, the DTRM, the S2S, DO, or MC model is enabled, the **Radiation Model** dialog box expands to show additional parameters. These parameters will not appear if you select the Rosseland model. If you are running a 3D case, you will have the added option of using the solar load model. The solar load options will be displayed in the dialog box, below the radiation model settings.

When the radiation model is active, the radiation fluxes will be included in the solution of the energy equation at each iteration. If you set up a problem with the radiation model turned on, and you then decide to turn it off completely, you must select the **Off** button in the **Radiation Model** dialog box.

Note that when you enable a radiation model, Ansys Fluent will automatically enable the energy equation so that step is not needed.

2. Set the appropriate radiation parameters.

   a. If you are modeling non-gray radiation using the P-1 model, define the non-gray radiation parameters as described in Setting Up the P-1 Model with Non-Gray Radiation (p. 2194).

   b. If you are using the DTRM, define the ray tracing as described in Setting Up the DTRM (p. 2195).

   c. If you are using the S2S model, define the surface clusters and view factors settings and compute or read the view factors as described in Setting Up the S2S Model (p. 2199).

   d. If you are using the DO model, choose **DO/Energy Coupling** if desired, define the angular discretization as described in Setting Up the DO Model (p. 2211) and, if relevant, define the non-gray radiation parameters as described in Defining Non-Gray Radiation for the DO Model (p. 2212).

   e. If you are using the MC model, specify the **Target Number of Histories** as described in Setting Up the MC Model (p. 2214) and, if relevant, similar to when using the DO model, define the non-gray radiation parameters as described in Defining Non-Gray Radiation for the DO Model (p. 2212).

3. Define the material properties as described in Defining Material Properties for Radiation (p. 2216).

4. Define the boundary conditions as described in Defining Boundary Conditions for Radiation (p. 2217). If your model contains a semi-transparent medium, see the information below on setting up semi-transparent media.

5. Set the parameters that control the solution (DTRM, DO, MC, S2S, and P-1 only) as described in Solution Strategies for Radiation Modeling (p. 2236).

6. Run the solution as described in Running the Calculation (p. 2241).

7. Postprocess the results as described in Postprocessing Radiation Quantities (p. 2243).

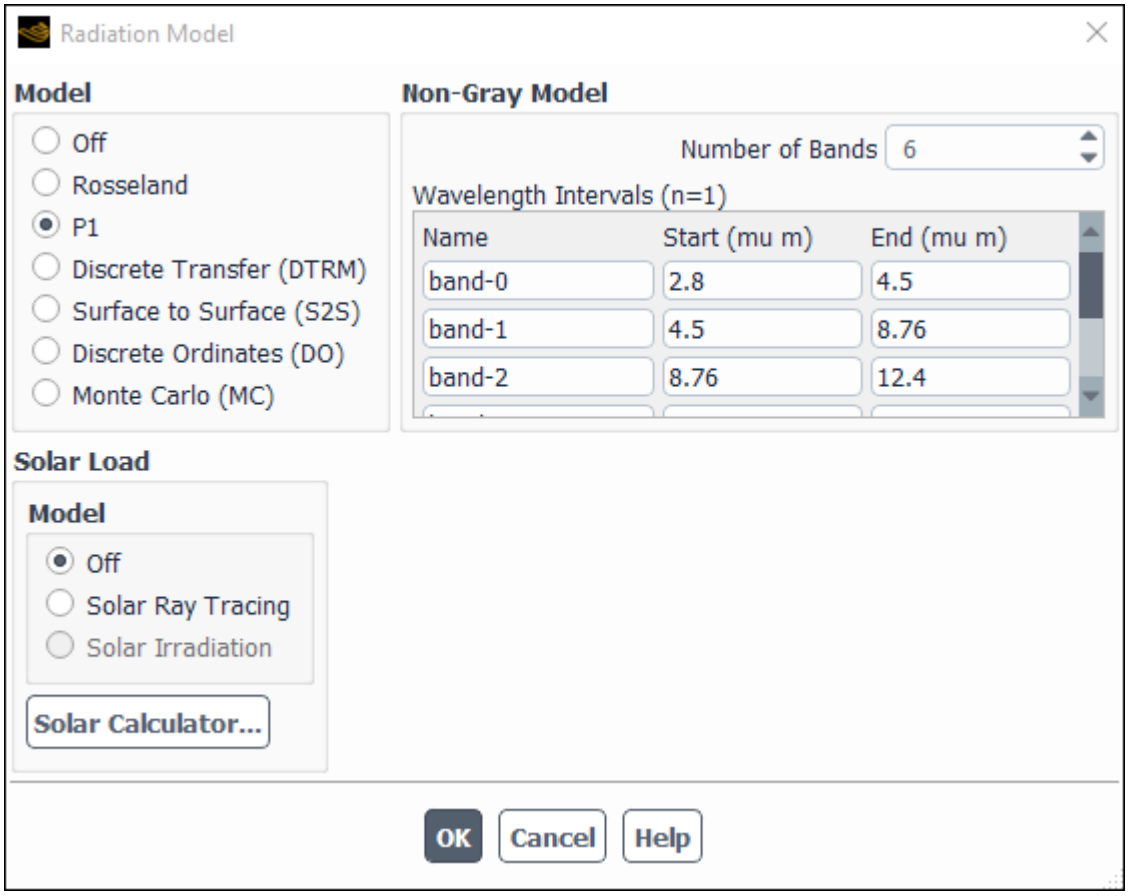
16.3.2. Setting Up the P-1 Model with Non-Gray Radiation

If you want to model non-gray radiation using the P-1 model, you can specify the **Number of Bands** under **Non-Gray Model** in the expanded **Radiation Model** dialog box (Figure 16.17: The Radiation Model Dialog Box (Non-Gray P-1 Model) (p. 2195)). By default, the **Number of Bands** is set to zero, indicating that only gray radiation will be modeled. Because the cost of computation increases directly with the number of bands, you should try to minimize the number of bands used. When a nonzero **Number of Bands** is specified, the **Radiation Model** dialog box will expand once again to show the

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Wavelength Intervals. For each wavelength band, you can specify a **Name**, as well as the **Start** and **End** wavelength of the band in μm. Note that the wavelength bands are specified for vacuum (n=1). For more information about non-gray radiation calculations, see Defining Non-Gray Radiation for the DO Model (p. 2212).

Figure 16.17: The Radiation Model Dialog Box (Non-Gray P-1 Model)



Ansys Fluent allows you to use a user-defined function (UDF) to modify the emissivity weighting factor $F\left(0\rightarrow n\lambda_2 T\right)-F\left(0\rightarrow n\lambda_1 T\right)$ (which otherwise defaults to the black body emission factor obtained from a standard Planck distribution). The emissivity weighting factor appears in the emission term of the radiative transfer equation for the non-gray model, as shown in Equation 5.64 in the Theory Guide. For more information, see the Fluent Customization Manual.

16.3.3. Setting Up the DTRM

For information about setting up the DTRM, see the following sections:

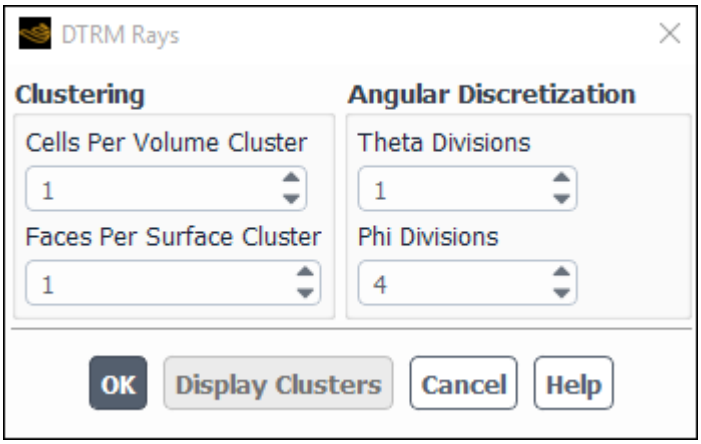
16.3.3.1. Defining the Rays
16.3.3.2. Controlling the Clusters
16.3.3.3. Controlling the Rays
16.3.3.4. Writing and Reading the DTRM Ray File
16.3.3.5. Displaying the Clusters

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16.3.3.1. Defining the Rays

When you select the **Discrete Transfer** model and click **OK** in the **Radiation Model** dialog box, the **DTRM Rays** dialog box (Figure 16.18: The DTRM Rays Dialog Box (p. 2196)) will open automatically. If you need to modify the current settings later in the problem setup or solution procedure, you can open this dialog box by clicking **DTRM Rays...** in the **Physics** ribbon tab (**Model Specific** group box).

Figure 16.18: The DTRM Rays Dialog Box



In this dialog box you will set parameters for and create the rays and clusters discussed in The DTRM Equations in the Theory Guide.

The procedure is as follows:

1. To control the number of radiating surfaces and absorbing cells, set the **Cells Per Volume Cluster** and **Faces Per Surface Cluster**. (See the explanation below.)

2. To control the number of rays being traced, set the number of **Theta Divisions** and **Phi Divisions**. (Guidelines are provided below.)

3. When you click **OK** in the **DTRM Rays** dialog box, The Select File Dialog Box (p. 931) will open prompting you for the name of the "ray file". After you have specified the filename and chosen whether to write a binary ray file, Ansys Fluent will write the ray file and then read it afterward. During the write process the status of the DTRM ray tracing will be reported in the Ansys Fluent console. For example:

```
Completed 25% tracing of DTRM rays

Completed 50% tracing of DTRM rays

Completed 75% tracing of DTRM rays

Completed 100 % tracing of DTRM rays
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See following sections for details on **DTRM Rays** dialog box inputs.

Important:

If you cancel the **DTRM Rays** dialog box without writing and reading the ray file, the DTRM will be disabled.

16.3.3.2. Controlling the Clusters

Your inputs for **Cells Per Volume Cluster** and **Faces Per Surface Cluster** will control the number of radiating surfaces and absorbing cells. By default, each is set to 1, so the number of surface clusters (radiating surfaces) will be the number of boundary faces, and the number of volume clusters (absorbing cells) will be the number of cells in the domain. For small 2D problems, these are acceptable numbers, but for larger problems you will want to reduce the number of surface and/or volume clusters in order to reduce the ray-tracing expense. See Clustering in the Theory Guide for details about clustering.

Note:

- Only zones attached to fluid cell zones are used for creating surface clusters.
- **Faces Per Surface Cluster** (FPSC) is indicative of the actual reduction factor except when the surfaces are planar. The surface clustering algorithm uses other criteria (angle between faces, and so on) to ensure the clusters are nearly planar, so it is possible that there will be no further reduction in the total number of clusters beyond a certain value of FPSC.

16.3.3.3. Controlling the Rays

Your inputs for **Theta Divisions** and **Phi Divisions** will control the number of rays being traced from each surface cluster (radiating surface).

Theta Divisions defines the number of discrete divisions in the angle θ used to define the solid angle about a point P on a surface. The solid angle is defined as θ varies from 0 to 90 degrees (in the Theory Guide), and the default setting of 1 for the number of discrete settings implies that there will be one ray traced from the surface.

Phi Divisions defines the number of discrete divisions in the angle ϕ used to define the solid angle about a point P on a surface. The solid angle is defined as ϕ varies from 0 to 360 degrees (Figure 5.2: Angles θ and ϕ Defining the Hemispherical Solid Angle About a Point P in the Theory Guide). The default setting of 4 implies that each ray traced from the surface will be located at a 90° angle, and in combination with the default setting for **Theta Divisions**, above, implies that 4 rays will be traced from each surface control volume. In many cases, it is recommended that you at least double the number of divisions in θ and ϕ.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16.3.3.4. Writing and Reading the DTRM Ray File

After you have activated the DTRM and defined all of the parameters controlling the ray tracing, you must create a ray file, which will be read back in and used during the radiation calculation. The ray file contains a description of the ray traces (for example, path lengths, cells traversed by each ray). This information is stored in the ray file, instead of being recomputed, in order to speed up the calculation process.

By default, a binary ray file will be written. You can also create text (formatted) ray files by turning off the **Write Binary Files** option in The Select File Dialog Box (p. 931).

Important:

Do not write or read a compressed ray file, because Ansys Fluent will not be able to access the ray tracing information properly from a compressed ray file.

The ray filename must be specified to Ansys Fluent only once. Thereafter, the filename is stored in your case file and the ray file will be automatically read into Ansys Fluent whenever the case file is read. Ansys Fluent will remind you that it is reading the ray file after it finishes reading the rest of the case file by reporting its progress in the console.

Note that the ray filename stored in your case file may not contain the full name of the directory in which the ray file exists. The full directory name will be stored in the case file only if you initially read the ray file through the GUI (or if you typed in the directory name along with the filename when using the text interface). In the event that the full directory name is absent, the automatic reading of the ray file may fail (since Ansys Fluent does not know in which directory to look for the file), and you will need to manually specify the ray file, using the **File/Read/DTRM Rays...** ribbon tab item. The safest approaches are to use the GUI when you first read the ray file or to supply the full directory name when using the text interface.

Important:

You must recreate the ray file whenever you do anything that changes the mesh, such as:

- change the type of a boundary zone
- adapt the mesh
- scale the mesh

You can open the **DTRM Rays** dialog box directly by clicking **DTRM Rays...** in the **Physics** ribbon tab (**Model Specific** group box).

16.3.3.5. Displaying the Clusters

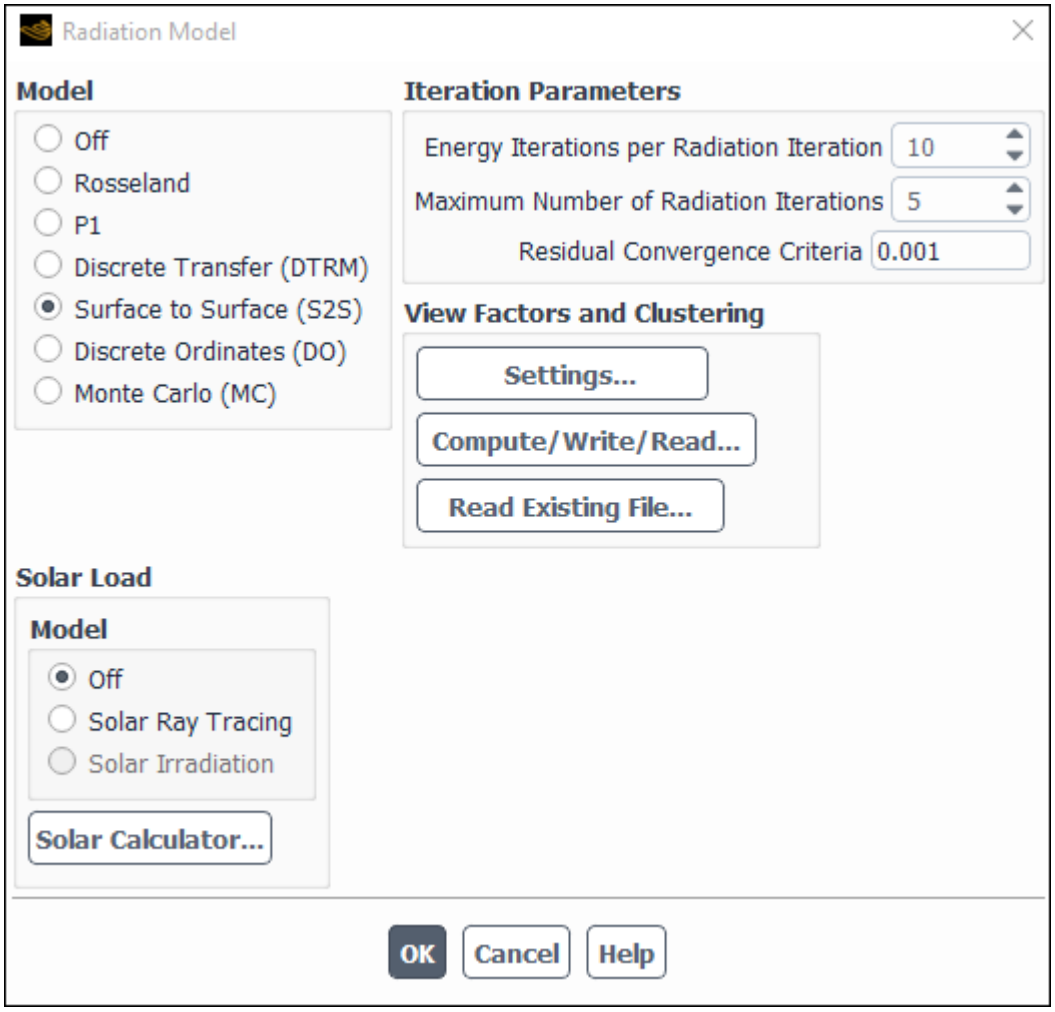
Once a ray file has been created or read in manually, you can click the **Display Clusters** button in the **DTRM Rays** dialog box to graphically display the clusters in the domain. See Displaying Rays and Clusters for the DTRM (p. 2245) for additional information about displaying rays and clusters.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16.3.4. Setting Up the S2S Model

When you select the **Surface to Surface (S2S)** model, the **Radiation Model** dialog box will expand to show additional parameters (see Figure 16.19: The Radiation Model Dialog Box (S2S Model) (p. 2199)). In these additional group boxes, you will set the solution parameters (see S2S Solution Parameters (p. 2238) for further details), access the view factors and clustering settings, and compute the view factors for your problem or read previously computed view factors into Ansys Fluent.

Figure 16.19: The Radiation Model Dialog Box (S2S Model)



The S2S radiation model is very expensive in terms of computation effort and memory requirements when there are a large number of radiating surfaces. You can reduce the number of radiating surfaces by grouping faces together to form surface clusters. The surface cluster information (coordinates and connectivity of the nodes, surface cluster IDs) is used by Ansys Fluent in the radiosity calculations and to compute the view factors.

Important:

- You must recreate the surface cluster information whenever you do anything that changes the mesh, such as:
  - change the type of a boundary zone

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– scale the mesh

Note that you do *not* need to recalculate view factors after shell conduction at any wall has been enabled or disabled (see Thermal Boundary Conditions at Walls (p. 1467) for more information about shell conduction). However, enabling or disabling shell conduction in the parallel version of Fluent does cause a migration of cells and hence a change in the partition. Whenever there is cell migration, the previously read view factor file is no longer valid, and the view factor file must be read again using either the **Read Existing File** button in the **Radiation Model** dialog box or the **File/Read/View Factors...** ribbon tab item.

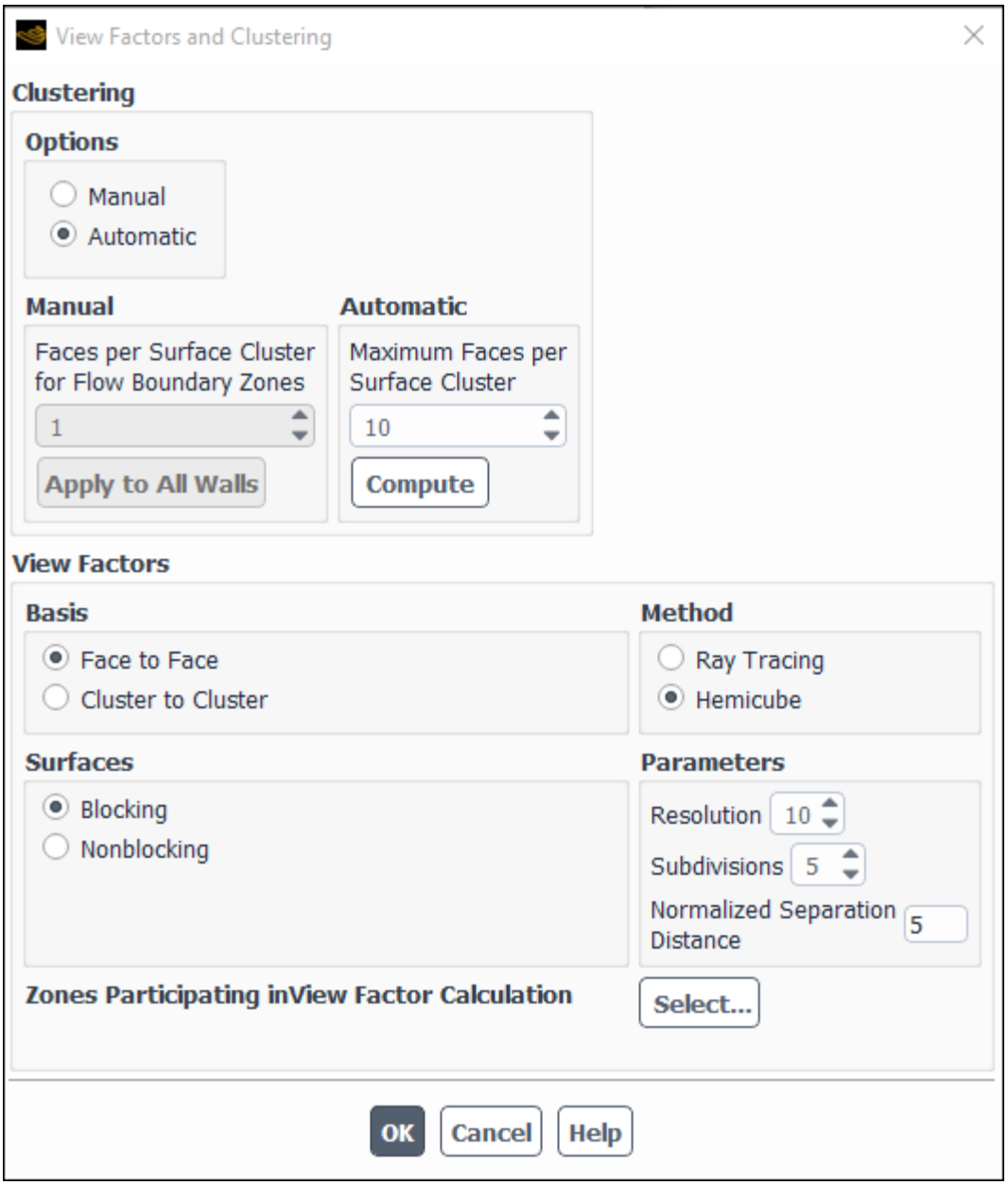
- Ansys Fluent will warn you to recreate the cluster/view factor file if a boundary zone has been changed from a wall to an internal wall (or vice versa), or if a boundary zone has been merged, separated, or fused.

16.3.4.1. View Factors and Clustering Settings

You can use the **View Factors and Clustering** dialog box (Figure 16.20: The View Factors and Clustering Dialog Box (p. 2201)) to define how the surface clusters are formed and how the view factors are calculated for the S2S model. To open this dialog box, click **Settings...** in the **View Factors and Clustering** group box in the Radiation Model Dialog Box (p. 4736) (Figure 16.19: The Radiation Model Dialog Box (S2S Model) (p. 2199)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.20: The View Factors and Clustering Dialog Box



16.3.4.1.1. Forming Surface Clusters

You can set the number of faces per surface cluster (FPSC) for each flow boundary (that is, exhaust fan, inlet vent, intake fan, outlet vent, mass-flow inlet, mass-flow outlet, pressure far-field, pressure inlet, pressure outlet, outflow, and velocity inlet boundary) and wall that is adjacent to a fluid zone; in this way, you can control the number of radiating surfaces and (if you select **Cluster to Cluster** for **Basis**) view factor surfaces. By default, the FPSC value is set to 1 everywhere, so the number of surface clusters (radiating/view factor surfaces) will be equal to the number of boundary faces. For small 2D problems, this is an acceptable number. For larger problems, you may want to reduce the number of surface clusters (that is, increase the FPSC) to reduce both the size of the view factor file and the memory requirement. Such a reduction in the number of clusters, however, comes at the cost of some accuracy. Note that for a thermally coupled mesh interface (that is, a fluid-fluid interface with **Coupled Wall** enabled, or an interface involving a

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

solid zone), the FPSC is always set to 1. See Clustering in the Theory Guide for details about clustering.

Important:

If you plan to use the cluster to cluster basis for the view factor calculation, be sure that the FPSC values are appropriate for both the radiosity calculation and the view factors.

Note that during the creation of a cluster on a non-conformal interface, its parent thread is taken into the consideration; it may happen that the cluster is somewhat larger than its child face, which may lead some inaccuracy in the flux calculation.

The **Clustering** group box in the **View Factors and Clustering** dialog box (Figure 16.20: The View Factors and Clustering Dialog Box (p. 2201)) provides two methods for revising the default FPSC values:

- manual
- automatic

If you select **Manual** in the **Options** group box, you can specify an FPSC value for all flow boundaries in the **Faces per Surface Cluster for Flow Boundary Zones** number-entry box in the **Manual** group box. If you then click **Apply to All Walls**, you will apply this value to all wall zones that are adjacent to fluid zones as well. For a wall that requires a different FPSC value (that is, you may want a lower value for walls in critical areas, and higher values in non-critical areas), you will need to open the boundary condition dialog box (Figure 16.21: The Wall Dialog Box (p. 2203)) for that particular wall and modify the **Faces Per Surface Cluster** in the **S2S Parameters** group box of the **Radiation** tab. Note that the **Radiation** tab also allows you to specify whether the wall participates in the view factor calculation, as described in Specifying Boundary Zone Participation (p. 2206).

Note:

- Only zones attached to fluid cell zones are used for creating surface clusters.
- **Faces Per Surface Cluster** (FPSC) is indicative of the actual reduction factor except when the surfaces are planar. The surface clustering algorithm uses other criteria (angle between faces, and so on) to ensure the clusters are nearly planar, so it is possible that there will be no further reduction in the total number of clusters beyond a certain value of FPSC.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.21: The Wall Dialog Box

Important:

The **Faces Per Surface Cluster** number-entry box will not be visible in the GUI on wall boundary zones that are attached to a solid.

Using the manual method to specify the FPSC values for walls can become very cumbersome if the model involves a large number of radiating faces, which is typically the case in underhood models. In such circumstances, it is recommended that you use the automatic clustering method instead. In this method, different FPSC values are assigned to the walls automatically, based on the distance of the walls from and the FPSC values of the walls that are defined as critical. The steps you will need to take are as follows:

1. Select **Automatic** from the **Options** list in the **View Factors and Clustering** dialog box.
2. For each wall that you deem critical, perform the following actions in the **Wall** dialog box (Figure 16.21: The Wall Dialog Box (p. 2203)):
   a. Click the **Radiation** tab.
   b. Enter an appropriate value for **Faces Per Surface Cluster** in the **S2S Parameters** group box.
   c. Enable the **Critical Zone** option.
   d. Click **OK** to close the **Wall** dialog box.
3. Enter the **Maximum Faces per Surface Cluster** value in the **View Factors and Clustering** dialog box and click the **Compute** button. Ansys Fluent will automatically calculate and update the **Faces Per Surface Cluster** values for all **Wall** dialog boxes adjacent to fluid zones that do not have **Critical Zone** enabled, without computing the clusters.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. You can check the automatically assigned FPSC values by opening the boundary condition dialog box of any non-critical wall of interest and examining the value for **Faces Per Surface Cluster** in the **S2S Parameters** group box of **Radiation** tab. You can manually modify the value for **Faces Per Surface Cluster** as necessary.

16.3.4.1.1.1. Setting the Split Angle for Clusters

Whether you set the FPSC value manually or automatically, you have the option of modifying the cutoff or "split" angle between adjacent face normals for the purpose of controlling surface clustering. The split angle sets the limit for which adjacent faces are clustered. A smaller split angle allows for a better representation of the view factor. By default, no surface cluster will contain any face that has a face normal greater than 20°. To modify the value of this parameter, you can use the `split-angle` text command:

`define` → `models` → `radiation` → `s2s-parameters` → `split-angle`

16.3.4.1.2. Setting Up the View Factor Calculation

You can control many aspects of how the view factors are calculated for your S2S model: how surfaces are defined; the computational method and related parameters; whether surface blocking will be accounted for; and which boundary zones will participate in the calculation. All of these controls are available in the **View Factors** group box in the **View Factors and Clustering** dialog box (Figure 16.20: The View Factors and Clustering Dialog Box (p. 2201)), and are described in the sections that follow.

16.3.4.1.2.1. Selecting the Basis for Computing View Factors

Ansys Fluent allows two ways to define the surfaces used for the view factor calculation, as described in Clustering and View Factors in the Theory Guide. If you want the surfaces to be the boundary faces of the mesh, select **Face to Face** for **Basis** in the **View Factor** group box of the **View Factors and Clustering** dialog box. This is the default selection.

Alternatively, you can select **Cluster to Cluster** for **Basis**, in order to reduce the computational expense and storage requirements. In this case, the surfaces used to calculate the view factors are the clusters defined by the settings in the **Cluster** group box of the **View Factors and Clustering** dialog box. The reduction in computational time will be proportional to the number of surface clusters used in the view factor calculation. The trade-offs of using the cluster to cluster basis for the calculation are that the accuracy may decrease, and the following limitations apply:

- The mesh must be 3D.
- You cannot subdivide the faces as part of the hemicube method parameters, which can cause the view factors to be overestimated.
- Polyhedral meshes are restricted to 1 face per surface cluster.
- When using **Cluster to Cluster**, surface clusters will form on non-conformal interfaces by default only with the **Ray Tracing** method enabled. If you do not want clusters to form on non-conformal interfaces, you can use the following command and answer **no**.

```
define/models/radiation/s2s-parameters/enable-mesh-interface-
clustering?
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16.3.4.1.2.2. Selecting the Method for Computing View Factors

Ansys Fluent provides two methods for computing view factors: the ray tracing method (which is selected by default) and the hemicube method. The following limitations apply:

- The hemicube method is available only for 3D and axisymmetric cases.
- The hemicube method should not be used when any of the zones are defined as periodic or symmetry boundaries, as these types are not currently supported.
- For the ray tracing method, when running in distributed parallel mode (including with Microsoft Job Scheduler), the working directory must be shared across all machines.

The hemicube method uses a differential area-to-area method and calculates the view factors on a row-by-row basis. The view factors calculated from the differential areas are summed to provide the view factor for the whole surface. This method originated from the use of the radiosity approach in the field of computer graphics [33] (p. 5714).

To use the hemicube method to compute the view factors, select **Hemicube** from the **Method** list in the **View Factors and Clustering** dialog box. It is recommended that you use the hemicube method for large, complex models with few obstructing surfaces between the radiating surfaces.

The hemicube method is based upon three assumptions about the geometry of the surfaces: aliasing, visibility, and proximity. To validate these assumptions, you can specify three different hemicube parameters, which can help you obtain better accuracy in calculating view factors. In most cases, however, the default settings will be sufficient.

- Aliasing—The true projection of each visible face onto the hemicube can be accurately accounted for by using a finite-resolution hemicube. As described previously, the faces are projected onto a hemicube. Because of the finite resolution of the hemicube, the projected areas and resulting view factors may be overestimated or underestimated. Aliasing effects can be reduced by increasing the value of the **Resolution** of the hemicube in the **Parameters** group box.
- Visibility—The visibility between any two faces does not change. In some cases, face i has a complete view of face k from its centroid, but some other face j occludes much of face k from face i. In such a case, the hemicube method will overestimate the view factor between face i and face k calculated from the centroid of face i. This error can be reduced by subdividing face i into smaller subfaces. You can specify the number of subfaces by entering a value for **Subdivisions** in the **Parameters** group box. Note that you cannot subdivide the faces when **Cluster to Cluster** is selected for **Basis**.
- Proximity—The distance between faces is great compared to the effective diameter of the faces. The proximity assumption is violated whenever faces are close together in comparison to their effective diameter or are adjacent to one another. In such cases, the distances between the centroid of one face and all points on the other face vary greatly. Since the view factor dependence on distance is nonlinear, the result is a poor estimate of the view factor.

  In the **Parameters** group box, you can set a limit for the **Normalized Separation Distance**, which is the ratio of the minimum face separation to the effective diameter of the face. If the computed normalized separation distance is less than the specified value, the face will then be divided into a number of subfaces until the normalized distances of the subfaces

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

are greater than the specified value. Alternatively, you can specify the number of subfaces to create for such faces by entering a value for **Subdivisions**.

While the hemicube method projects radiating surfaces onto a hemicube, the ray tracing method instead traces rays through the centers of every hemicube face to determine which surfaces are visible through that face. Note that the ray tracing method does not subdivide the faces (as can be done when using the hemicube method by setting the **Subdivisions** or **Normalized Separation Distance** parameters), and so the view factors may be less accurate than those calculated using the hemicube method for surfaces that have a normalized separation distance less than 5.

To use the ray tracing method to compute the view factors, select **Ray Tracing** from the **Method** list in the **View Factors and Clustering** dialog box. You can adjust the value of the **Resolution** in the **Parameters** group box in order to reduce the impact of aliasing effects, as described previously.

16.3.4.1.2.3. Accounting for Blocking Surfaces

View factor calculations depend on the geometric orientations of surface pairs with respect to each other. Two situations may be encountered when examining surface pairs:

- If there is no obstruction between the surface pairs under consideration, then they are referred to as "nonblocking" surfaces.
- If there is another surface blocking the views between the surfaces under consideration, then they are referred to as "blocking" surfaces. Blocking will change the view factors between the surface pairs and require additional checks to compute the correct value of the view factors.

For cases with blocking surfaces, select **Blocking** from the **Surfaces** list in the **View Factors and Clustering** dialog box. For cases with nonblocking surfaces, you can choose either **Blocking** or **Nonblocking** without affecting the accuracy. However, it is better to choose **Nonblocking** for such cases, as it takes less time to compute.

16.3.4.1.2.4. Specifying Boundary Zone Participation

You can choose to exclude walls and inlet and exit boundaries from participating in the view factor calculation. If you are unsure whether it is necessary to calculate view factors for a particular boundary zone ahead of time, it is recommended that you allow it to participate; you can always reverse this decision as part of a future calculation run.

There are two ways in which you can enable/disable the participation of walls and inlet and exit boundaries in the view factor calculation. One of those ways is to use the **Participates in View Factor Calculation** option in the **Radiation** tab of the boundary condition dialog box. The other method is to use the **Participating Boundary Zones** dialog box (Figure 16.22: The Participating Boundary Zones Dialog Box (p. 2207)), which is accessed by clicking the **Select...** button next to the **Zones Participating in View Factor Calculation** label in the **View Factors**

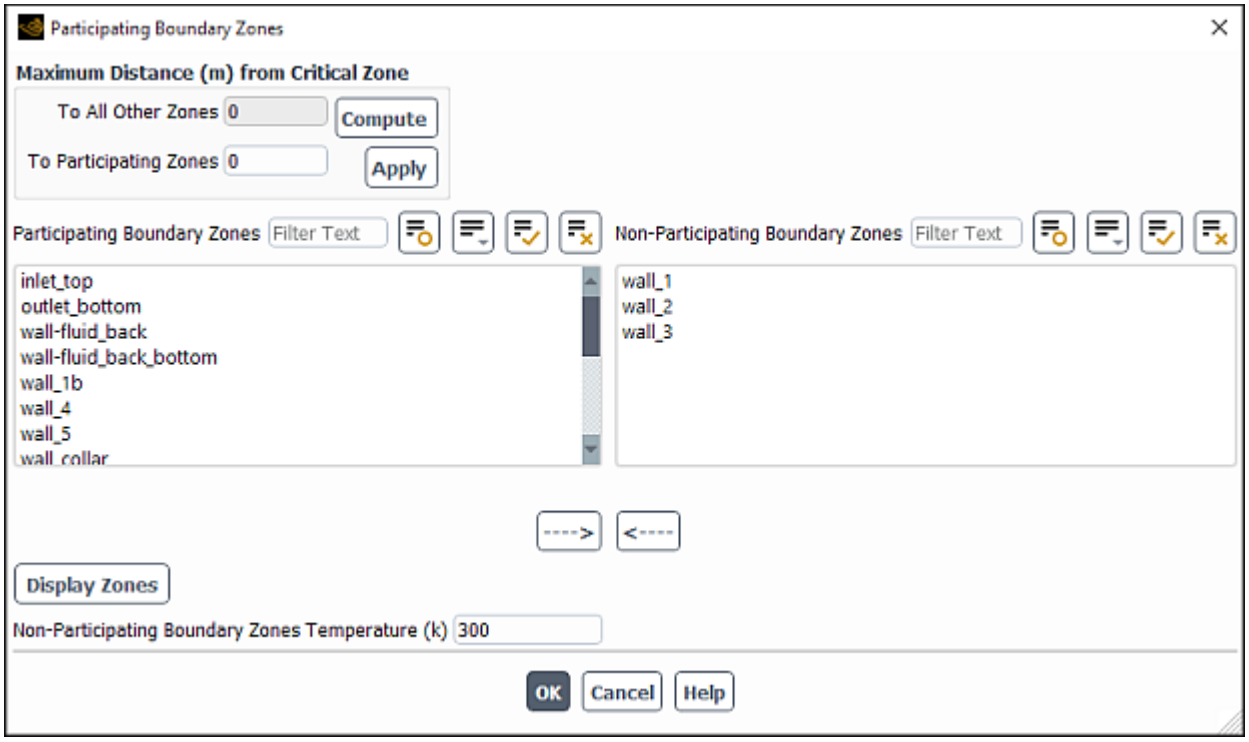
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and Clustering dialog box. For cases that are made up of a very large number of zones, such as underhood applications, the latter method is recommended.

Important:

If you compute the view factors and then later alter which boundary zones participate in the view factor calculation, you must recompute the view factors so that the data is up to date.

Figure 16.22: The Participating Boundary Zones Dialog Box



The **Participating Boundary Zones** dialog box allows you to easily specify those zones that are participating or non-participating without having to visit the boundary conditions dialog box of each zone. For cases that have a small number of boundary zones, you can simply select the zones that you do not want to participate in the view factor calculation from the **Participating Boundary Zones** list and click the arrow button that points to the right, so that the zones are moved to the **Non-Participating Boundary Zones** list; if you make an error, you can always reverse this process (that is, select a zone in the **Non-Participating Boundary Zones** list and click the arrow button that points to the left). This can be cumbersome for cases that have a large number of boundary zones, and so the following procedure is recommended instead:

1. Make sure that your clustering options (see Forming Surface Clusters (p. 2201)) are appropriate for your view factor settings:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. Select **Automatic** from the **Options** list in the **Clustering** group box of the **View Factors and Clustering** dialog box, as this enables some GUI items in the **Participating Boundary Zones** dialog box.

> **Important:**
>
> Note that if you do not want to use the **Automatic** option for the clustering, you can revert to the **Manual** option after you are done using the **Participating Boundary Zones** dialog box.

b. Verify that the walls you specified as critical (by enabling the **Critical Zone** option in the **Radiation** tab of the boundary condition dialog box) also correspond to those that are critical for the view factor calculation. You must have specified at least one critical zone for the steps that follow.

2. Click the **Compute** button in the **Maximum Distance from Critical Zone** dialog box, to update the value displayed for **To All Other Zones**. This value represents the maximum distance between the centroids of a critical zone and a non-critical zone in the mesh; it is for information purposes only, and cannot be edited in this dialog box.

3. Based on the value displayed for **To All Other Zones**, enter a threshold value for **To Participating Zones**. The value you enter will specify the maximum distance allowed between the centroids of a critical zone and a zone that participates in the view factor calculation. Then click the **Apply** button to move all zones beyond this distance into the **Non-Participating Boundary Zones** list.

4. Review the zones displayed in the **Participating Boundary Zones** and **Non-Participating Boundary Zones** lists. If necessary, select zones in these lists and use the arrow buttons to move them to the appropriate list, as described previously.

If at any point you want to visually identify zones displayed in the **Participating Boundary Zones** and **Non-Participating Boundary Zones** lists, select the zones and click the **Display Zones** button. Only the selected zones will be displayed in the graphics window.

If any zones are displayed in the **Non-Participating Boundary Zones** list, ensure to enter an appropriate temperature for **Non-Participating Boundary Zones Temperature**. In most cases the appropriate value is the ambient temperature, which by default is assumed to be 300 K.

After you have specified which zones do not participate in view factor calculation and set their temperature, click **OK** to store the settings and close the **Participating Boundary Zones** dialog box. You can then proceed to computing the view factors, as described in the section that follows.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16.3.4.2. Computing View Factors

Ansys Fluent can compute the view factors for your problem in the current session and save them to a file for use in the current session and future sessions.

Note:

You can accelerate view factor calculations by using Ansys Fluent in parallel. For more information see Accelerating View Factor Calculations Using General Purpose Graphics Processing Units (GPGPUs) (p. 4351).

To compute view factors in your current Ansys Fluent session, you must first set the parameters for the view factor calculation in the **View Factors and Clustering** dialog box (see View Factors and Clustering Settings (p. 2200) for details) and click **OK** to save them. When you have set the view factor and surface cluster parameters, click the **Compute/Write/Read...** button in the **View Factors and Clustering** group box of the **Radiation Model** dialog box. A **Select File** dialog box will open, prompting you for the name of the file to which you would like to save the surface cluster information and view factors.

View factor files can be created in any of the following formats:

- CFF S2S format (`filename.s2s.h5`)

  Selecting **CFF S2S Files** for **Files of type** automatically adds the `s2s.h5` extension.

- S2S compressed format (`filename.s2s.gz`)

  Selecting **S2S Files** for **Files of type** allows you to add the `s2s.gz` extension.

- S2S format (`filename.s2s`)

  Selecting **S2S Files** for **Files of type** allows you to add the `s2s` extension.

Important:

Writing S2S files in `.h5` format has the following limitations.

- Not supported when using GPU acceleration.
- Not supported for mixed Windows/Linux mode.

Note that when using the TUI, you must directly specify the extension in the file name you provide.

The option to **Write Binary Files** is enabled by default, which reduces the time needed to write and read view factor files. When you click **OK** in the **Select File** dialog box, Ansys Fluent writes the surface cluster information to the file. Ansys Fluent uses the surface cluster information to compute the view factors, save the view factors to the same file, and then automatically read the view factors. The Ansys Fluent console will report the status of the view factor calculation. For example:

```
Completed 25% calculation of view factors
```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

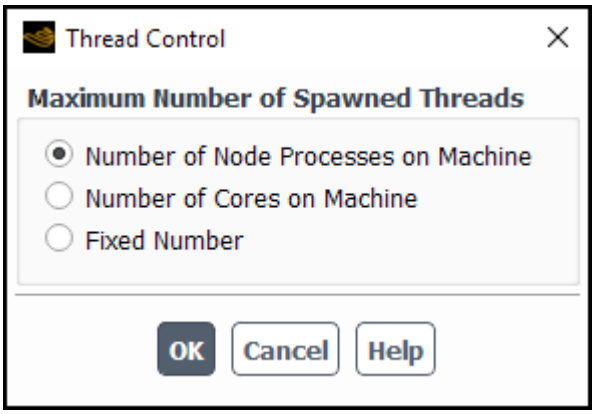
```
Completed 50% calculation of view factors

Completed 75% calculation of view factors

Completed 100 % calculation of view factors
```

Important:

- You must recompute the view factors if you take any of following actions after the initial computation:
  - if you alter which boundary zones participate in the view factor calculation (either by using the **Participating Boundary Zones** dialog box, or by enabling / disabling the **Participates in View Factor Calculation** option in the **Radiation** tab of the boundary condition dialog box)
  - if you scale or unscale the mesh by using the **Scale Mesh** dialog box
- The view factor file format for this version of Ansys Fluent is known as the compressed row format (CRF) and is a more efficient way of writing view factors than in versions that are prior to Fluent 6.4. In the CRF format, only nonzero view factors with their associated cluster IDs are stored to the file. This reduces the size of the `.s2s` file, and reduces the time it takes to read the file into Ansys Fluent. While the CRF file format is the default, you can still use the older file format if necessary. Contact your support engineer for more information.
- View factors using ray tracing are computed with the MPI/OpenMP hybrid model for parallel runs. By default, one OpenMP thread is used per MPI process, as specified in the Thread Control Dialog Box (p. 5631) (see Controlling the Threads (p. 4345) for details). To use all the available cores on your machine as OpenMP threads, select **Number of Cores on Machine**. A **Fixed Number** of cores can also be used based on your needs.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.23: The Thread Control Dialog Box



Note:

For cases that use the S2S model and contain non-conformal interfaces, if the mesh interfaces are defined after the case has been read, then the intersection of the interface zones will occur during the initialization process, which will cause relabeling of faces throughout the domain. If you attempt to compute view factors before solution initialization in such a case, then the following message will be displayed in the console and the view factors will be computed automatically during solution initialization to avoid a mismatch with the case file:

```
Intersection zones for non-conformal interfaces are not created.
View factors will be computed during solution initialization.
```

16.3.4.3. Reading View Factors into Ansys Fluent

If the view factors for your problem have already been computed and saved to a file, you can read them into Ansys Fluent. To read in the view factors, click **Read Existing File...** button in the **View Factors and Clustering** group box of the **Radiation Model** dialog box. A **Select File** dialog box will open where you can specify the name of the file containing the view factors. For **Files of type**, select **CFF S2S Files** to read .h5 files or select **S2S Files** to read other formats. Alternatively, you can read the view factors file using the **File/Read/View Factors...** ribbon tab item.

16.3.5. Setting Up the DO Model

For information about setting up the DO model, see the following sections:

16.3.5.1. Angular Discretization

16.3.5.2. Defining Non-Gray Radiation for the DO Model

16.3.5.3. Enabling DO/Energy Coupling

For information about accelerating the DO solver, see Accelerating Discrete Ordinates (DO) Radiation Calculations (p. 4352).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16.3.5.1. Angular Discretization

When you select the **Discrete Ordinates** model, the **Radiation Model** dialog box will expand to show inputs for **Angular Discretization** (see Figure 16.16: The Radiation Model Dialog Box (DO Model) (p. 2193)). In this section, you will set parameters for the angular discretization and pixelation described in Angular Discretization and Pixelation in the Theory Guide.

Theta Divisions (N_θ) and **Phi Divisions** (N_ϕ) will define the number of control angles used to discretize each octant of the angular space (see Figure 5.3: Angular Coordinate System in the Theory Guide). Note that higher levels of discretization are recommended for problems where specular exchange of radiation is important to increase the likelihood of the correct beam direction being captured. For a 2D model, Ansys Fluent will solve only 4 octants (due to symmetry); therefore, a total of $4N_\theta N_\phi$ directions $\vec{s}$ will be solved. For a 3D model, 8 octants are solved, resulting in $8N_\theta N_\phi$ directions $\vec{s}$. By default, the number of **Theta Divisions** and the number of **Phi Divisions** are both set to 2. For most practical problems, these settings are acceptable, however, a setting of 2 is considered to be a coarse estimate. Increasing the discretization of **Theta Divisions** and **Phi Divisions** to a minimum of 3, or up to 5, will achieve more reliable results. A finer angular discretization can be specified to better resolve the influence of small geometric features or strong spatial variations in temperature, but larger numbers of **Theta Divisions** and **Phi Divisions** will add to the cost of the computation.

Theta Pixels and **Phi Pixels** are used to control the pixelation that accounts for any control volume overhang (see Figure 5.7: Pixelation of Control Angle in the Theory Guide and the figures and discussion preceding it). For problems involving gray-diffuse radiation, the default pixelation of 1×1 is usually sufficient. For problems involving symmetry, periodic, specular, or semi-transparent boundaries, a pixelation of 3×3 is recommended and will achieve acceptable results. The computational effort, as a result of increasing the pixelation, is less than the computational effort caused by increasing the divisions. You should be aware, however, that increasing the pixelation adds to the cost of computation.

Important:

Note that pixelations are applied to boundary faces by default.

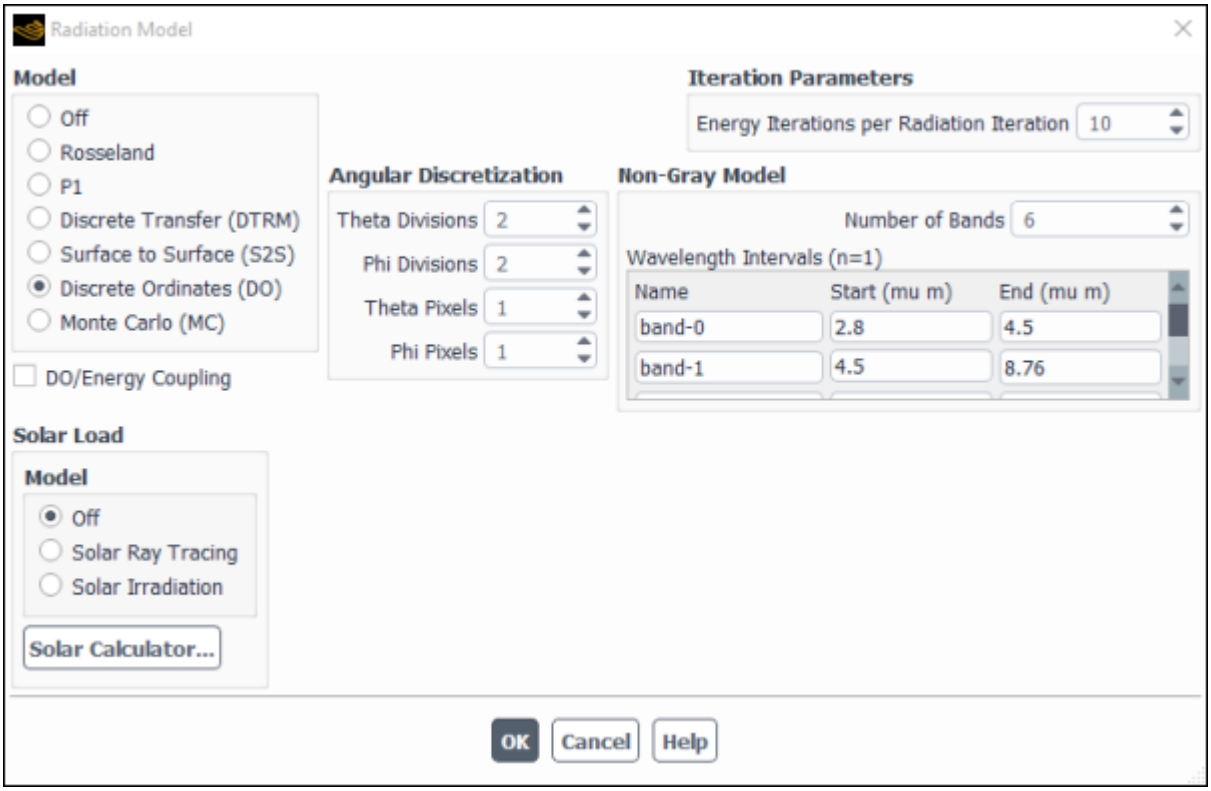
16.3.5.2. Defining Non-Gray Radiation for the DO Model

If you want to model non-gray radiation using the DO model, you can specify the **Number of Bands** (N) under **Non-Gray Model** in the expanded **Radiation Model** dialog box (Figure 16.24: The Radiation Model Dialog Box (Non-Gray DO Model) (p. 2213)). For a 2D model, Ansys Fluent will solve $4N_\theta N_\phi N$ directions. For a 3D model, $8N_\theta N_\phi N$ directions will be solved. By default, the **Number of Bands** is set to zero, indicating that only gray radiation will be modeled. Because the cost of computation increases directly with the number of bands, you should try to minimize the number of bands used. In many cases, the absorption coefficient or the wall emissivity is effectively constant for the wavelengths of importance in the temperature range of the problem. For such cases, the gray DO model can be used with little loss of accuracy. For other cases, non-gray behavior is important, but relatively few bands are necessary. For typical glasses, for example, two or three bands will frequently suffice.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When a nonzero **Number of Bands** is specified, the **Radiation Model** dialog box will expand once again to show the **Wavelength Intervals** (Figure 16.24: The Radiation Model Dialog Box (Non-Gray DO Model) (p. 2213)). You can specify a **Name** for each wavelength band, as well as the **Start** and **End** wavelength of the band in μm. Note that the wavelength bands are specified for vacuum (n=1). Ansys Fluent will automatically account for the refractive index in setting band limits for media with n different from unity.

Figure 16.24: The Radiation Model Dialog Box (Non-Gray DO Model)



The frequency of radiation remains constant as radiation travels across a semi-transparent interface. The wavelength, however, changes such that $n\lambda$ is constant. Therefore, when radiation passes from a medium with refractive index n_1 to one with refractive index n_2, the following relationship holds:

$$n_1\lambda_1 = n_2\lambda_2 \tag{16.3}$$

Here λ_1 and λ_2 are the wavelengths associated with the two media. It is conventional to specify the wavelength rather than frequency. Ansys Fluent requires you to specify wavelength bands (in μm) for an equivalent medium with n=1.

For example, consider a typical glass with a step jump in the absorption coefficient at a cut-off wavelength of λ_c. The absorption coefficient is a_1 for $\lambda \leq \lambda_c$ and a_2 for $\lambda > \lambda_c$. The refractive index of the glass is n_g. Since $n\lambda$ is constant across a semi-transparent interface, the equivalent cut-off wavelength for a medium with n=1 is $n_g\lambda_c$ using Equation 16.3 (p. 2213). You should choose two bands in this case, with the limits 0 to $n_g\lambda_c$ and $n_g\lambda_c$ to 100. Here, the upper wavelength limit has been chosen to be a large number, 100, in order to ensure that the entire spectrum is covered by the bands. When multiple materials exist, you should convert all the cut-off wavelengths to equivalent cut-off wavelengths for an n=1 medium, and choose the band boundaries accordingly.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

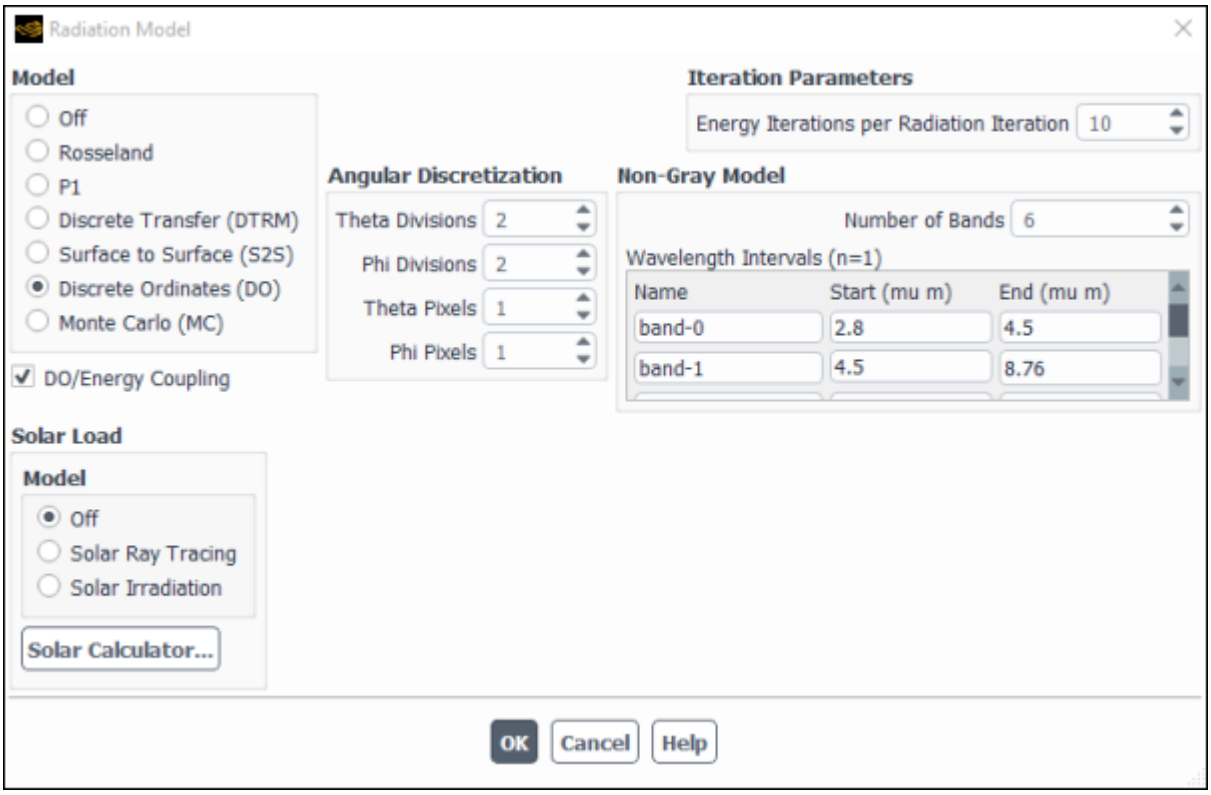
The bands can have different widths and need not be contiguous. You can ensure that the entire spectrum is covered by your bands by choosing $\lambda_{min}=0$ and $n\lambda_{max}T_{min}\geq 50,000$. Here λ_{min} and λ_{max} are the minimum and maximum wavelength bounds of your wavelength bands, and T_{min} is the minimum expected temperature in the domain.

Ansys Fluent allows you to use a user-defined function (UDF) to modify the emissivity weighting factor $F\left(0\rightarrow n\lambda_2 T\right)-F\left(0\rightarrow n\lambda_1 T\right)$ (which otherwise defaults to the black body emission factor obtained from a standard Planck distribution). The emissivity weighting factor appears in the emission term of the radiative transfer equation for the non-gray model, as shown in Equation 5.100 in the Theory Guide. For more information, see `DEFINE_EMISSIVITY_WEIGHTING_FACTOR` in the Fluent Customization Manual.

16.3.5.3. Enabling DO/Energy Coupling

For applications involving optical thicknesses greater than 10, you can enable the **DO/Energy Coupling** option in the **Radiation Model** (Figure 16.25: The Radiation Model Dialog Box with DO/Energy Coupling Enabled (p. 2214)) in order to couple the energy and intensity equations at each cell, solving them simultaneously. This approach accelerates the convergence of the finite volume scheme for radiative heat transfer and can be used with the gray or non-gray radiation model.

Figure 16.25: The Radiation Model Dialog Box with DO/Energy Coupling Enabled



16.3.6. Setting Up the MC Model

In the **Iteration Parameters** group box of the **Radiation Model** dialog box, you have the option of revising the **Target Number of Histories** to be tracked for the Monte Carlo (MC) simulation. The

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

default value is 100,000. This can have a large effect on the accuracy of simulation results. For details, see Monte Carlo Solution Accuracy in the Theory Guide.

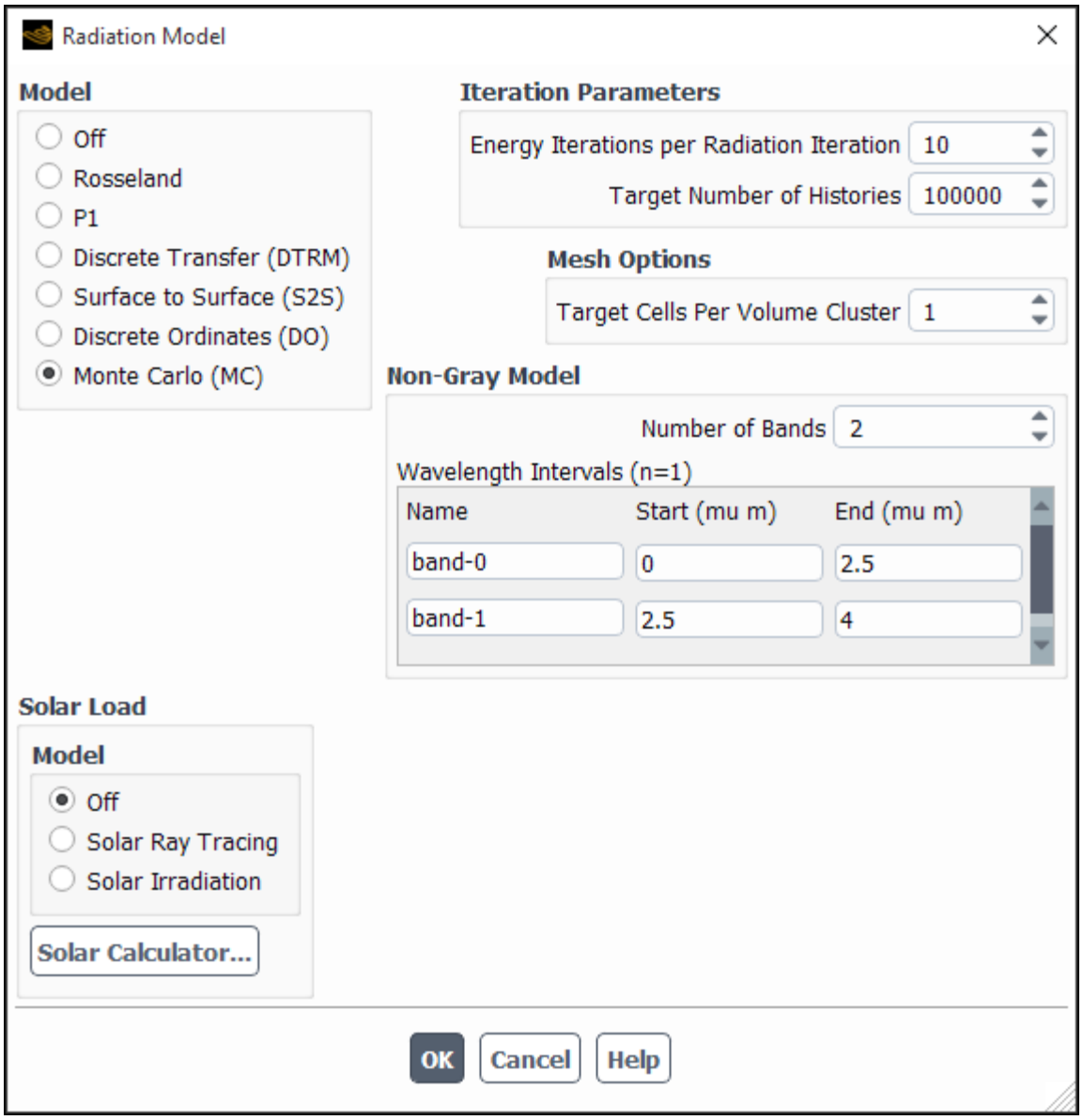
In some cases, Fluent might calculate substantially more histories than the specified **Target Number of Histories**, particularly if a fine surface mesh is used. The solver will ensure that an appropriate minimum number of histories are started at each surface face, and that can increase the total number of histories that are used during the calculation. Thus the CPU usage may not scale directly with changes to the specified target.

In the **Mesh Options** group box, you can coarsen the radiation mesh by specifying a **Target Cells Per Volume Cluster** greater than one. By default, it is set to one which implies no coarsening. Any value greater than one reduces the number of effective cells in the domain used by the MC radiation model. The coarsening is achieved by merging cells together and, therefore, does not affect the surface mesh resolution or alter the faces of any merged cells. The reduction in cells may speed up the radiation calculation and reduce peak memory usage, however, it may also have an impact on solution accuracy.

You have the option specifying multiple radiation bands using the **Non-Gray Model** group box. For each desired band, you must specify a start and end wavelength similar to Defining Non-Gray Radiation for the DO Model (p. 2212).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.26: The Radiation Model Dialog Box (MC)



16.3.7. Defining Material Properties for Radiation

When you are using the P-1, DO, MC, or Rosseland radiation model in Ansys Fluent, you should be sure to define both the absorption and scattering coefficients of the fluid in the **Create/Edit Materials** dialog box. Note that you can either enter a constant value for these parameters, or you can specify them using a user-defined function (UDF). For more information, see `DEFINE_PROPERTY` UDFs in the Fluent Customization Manual.

Setup → Materials

If you are modeling semi-transparent media using the DO or MC model, you should also define the refractive index for the semi-transparent fluid or solid material. When using the Rosseland model, you can specify the refractive index only for the fluid material. When using the P-1 model, you should

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

define the refractive index for the fluid material only. For the DTRM, you need to define only the absorption coefficient.

If your model includes gas phase species such as combustion products, absorption and/or scattering in the gas may be significant. The scattering coefficient should be increased from the default of zero if the fluid contains dispersed particles or droplets that contribute to scattering. Note that when using the non-gray model, you cannot specify a different scattering coefficient for each band. Alternatively, you can specify the scattering coefficient as a user-defined function (UDF). For more information, see DEFINE_PROPERTY UDFs in the Fluent Customization Manual.

Ansys Fluent allows you to enter a composition-dependent absorption coefficient for CO_2 and H_2O mixtures, using the WSGGM. The method for computing a variable absorption coefficient is described in Radiation in Combusting Flows in the Theory Guide. Radiation Properties (p. 1665) provides a detailed description of the procedures used for specification of radiation properties.

16.3.7.1. Absorption Coefficient for a Non-Gray Model

If you are using the non-gray P-1, DO, or MC model, you can specify a different constant absorption coefficient for each of the bands used by the gray-band model, as described in Radiation Properties (p. 1665). You cannot, however, compute a composition-dependent absorption coefficient in each band. If you use the WSGGM to compute a variable absorption coefficient, the value will be the same for all bands. Alternatively, you can specify a user-defined function (UDF) for the absorption coefficient. For more information, see DEFINE_PROPERTY UDFs in the Fluent Customization Manual.

16.3.7.2. Refractive Index for a Non-Gray Model

If you are using the non-gray P-1, DO, or MC model, you can specify a different constant refractive index for each of the bands used by the gray-band model, as described in Radiation Properties (p. 1665). You cannot, however, compute a composition-dependent refractive index in each band.

16.3.8. Defining Boundary Conditions for Radiation

When you set up a problem that includes radiation, you will set additional boundary conditions at inlets, outlets, and walls. These inputs are described in the sections that follow. The boundary condition dialog boxes can be opened by right-clicking the boundary name in the tree (under **Setup/Boundary Conditions**) and clicking **Edit...** in the menu that opens; alternatively, you can open them from the **Boundary Conditions** task page:

Setup → **Boundary Conditions**

The following topics are discussed:

16.3.8.1. Inlet and Outlet Boundary Conditions
16.3.8.2. Wall Boundary Conditions for the DTRM, P-1, S2S, and Rosseland Models
16.3.8.3. Wall Boundary Conditions for the DO Model
16.3.8.4. Wall Boundary Conditions for the MC Model
16.3.8.5. Solid Cell Zones Conditions for the DO or MC Models
16.3.8.6. Thermal Boundary Conditions

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16.3.8.1. Inlet and Outlet Boundary Conditions

16.3.8.1.1. Emissivity

When radiation is active, you can define the emissivity at each inlet and exit boundary when you are defining boundary conditions in the associated inlet or exit boundary dialog box (for example, **Pressure Inlet** dialog box, **Velocity Inlet** dialog box, **Pressure Outlet** dialog box). Enter the appropriate value for **Internal Emissivity**. The default value for all boundary types is 1. Alternatively, you can specify a user-defined function for emissivity. For more information, see `DEFINE_PROFILE` in the Fluent Customization Manual.

For the non-gray P-1, DO, and MC models, the specified constant emissivity will be used for all wavelength bands.

Important:

The **Internal Emissivity** boundary condition is not available with the Rosseland model.

16.3.8.1.2. Black Body Temperature

Ansys Fluent includes an option that allows you to take into account the influence of the temperature of the gas and the walls beyond the inlet/exit boundaries, and specify different temperatures for radiation and convection at inlets and exits. This is useful when the temperature outside the inlet or exit differs considerably from the temperature in the enclosure. For example, if the temperature of the walls beyond the inlet is 2000 K and the temperature at the inlet is 1000 K, you can specify the outside wall temperature to be used for computing radiative heat flux, while the actual temperature at the inlet is used for calculating convective heat transfer. To do this, you would specify a radiation temperature of 2000 K as the black body temperature.

Although this option allows you to account for both cooler and hotter outside walls, you must use caution in the case of cooler walls, since the radiation from the immediate vicinity of the hotter inlet or outlet almost always dominates over the radiation from cooler outside walls. If, for example, the temperature of the outside walls is 250 K and the inlet temperature is 1500 K, it might be misleading to use 250 K for the radiation boundary temperature. This temperature might be expected to be somewhere between 250 K and 1500 K; in most cases it will be close to 1500 K. Its value depends on the geometry of the outside walls and the optical thickness of the gas in the vicinity of the inlet.

In the flow inlet or exit dialog box (for example, **Pressure Inlet** dialog box, **Velocity Inlet** dialog box), select **Specified External Temperature** in the **External Black Body Temperature Method** drop-down list, and then enter the value of the radiation boundary temperature as the **Black Body Temperature**.

Important:

- If you want to use the same temperature for radiation and convection, retain the default selection of **Boundary Temperature** as the **External Black Body Temperature Method**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The **Black Body Temperature** boundary condition is not available with the Rosseland model.

16.3.8.1.3. Inlet and Outlet Boundary Conditions for the DO and MC Models

For the DO and MC models, you also have the option of specifying the **BC Type** with the following options:

- `opaque`

  The boundary behaves as described in Emissivity (p. 2218) and Black Body Temperature (p. 2218) above.

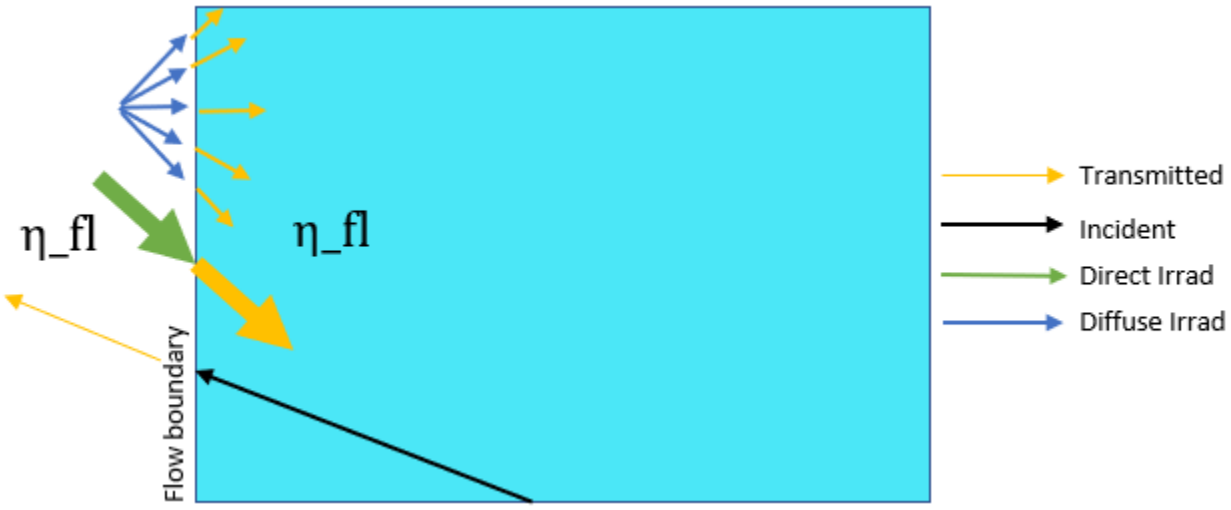
- `transparent`

  The boundary behaves similarly to an external semi-transparent wall, however, the irradiation flux passes through the transparent flow boundary from outside the computational domain into the adjacent fluid zone without getting reflected, absorbed or refracted (as shown in Figure 16.27: Treatment of Radiation at Transparent Boundary (p. 2219)). For further details see, Semi-Transparent Exterior Walls in the *Fluent Theory Guide*.

  The irradiation beam is defined in the same way as for an extremal semi-transparent wall. Specifically, for defining the beam width for the DO model, see Beam Width and Direction for External Irradiation Beam in the *Fluent Theory Guide*.

  Incident radiation on a flow boundary from within the domain is also transmitted specularly.

Figure 16.27: Treatment of Radiation at Transparent Boundary



Details for specifying a transparent boundary are similarly described for external semi-transparent wall boundaries for the DO and MC models, respectively.

- Semi-Transparent Walls for the DO Model (p. 2222)
- Semi-Transparent Walls for the MC Model (p. 2233)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16.3.8.2. Wall Boundary Conditions for the DTRM, P-1, S2S, and Rosseland Models

The DTRM, S2S, Rosseland, and gray P-1 models assume all walls to be gray and diffuse. The only radiation boundary condition required in the **Wall** dialog box is the **Internal Emissivity**. For the Rosseland model, the internal emissivity is automatically set to 1. For the DTRM, S2S, and gray P-1 models, you can enter the appropriate value for **Internal Emissivity** in the **Radiation** tab of the **Wall** dialog box. The default value is 1. Alternatively, you can specify a user-defined function for **Internal Emissivity**. For more information, see `DEFINE_PROFILE` in the Fluent Customization Manual.

For the non-gray P-1 model, specify a constant **Internal Emissivity** for each wavelength band in the **Radiation** tab of the **Wall** dialog box (the default value in each band is 1). Alternatively, you can specify the internal emissivity using a boundary condition parameter (see Creating a New Parameter (p. 1311)). See Defining Boundary Conditions for Radiation (p. 2217) for details.

16.3.8.2.1. Boundary Conditions for the S2S Model

When the S2S model is used, you can specify that some of the walls and inlet and exit boundaries are not participating in the view factor calculation. This capability allows you to save time computing the view factors and also reduce the memory required to store the view factor file during the Ansys Fluent calculation. See Specifying Boundary Zone Participation (p. 2206) for details.

Important:

- Whenever you revise which boundary zones participate in the calculation, you will need to recompute the view factors.
- The **Flux Reports** dialog box will not show the exact balance of the **Radiation Heat Transfer Rate** because the radiative heat transfer to those boundaries that do not participate in the view factor calculation is not included.

16.3.8.3. Wall Boundary Conditions for the DO Model

When the DO model is used, you can model opaque walls, as discussed in Boundary and Cell Zone Condition Treatment at Opaque Walls in the Theory Guide, as well as semi-transparent walls (Cell Zone and Boundary Condition Treatment at Semi-Transparent Walls in the Theory Guide).

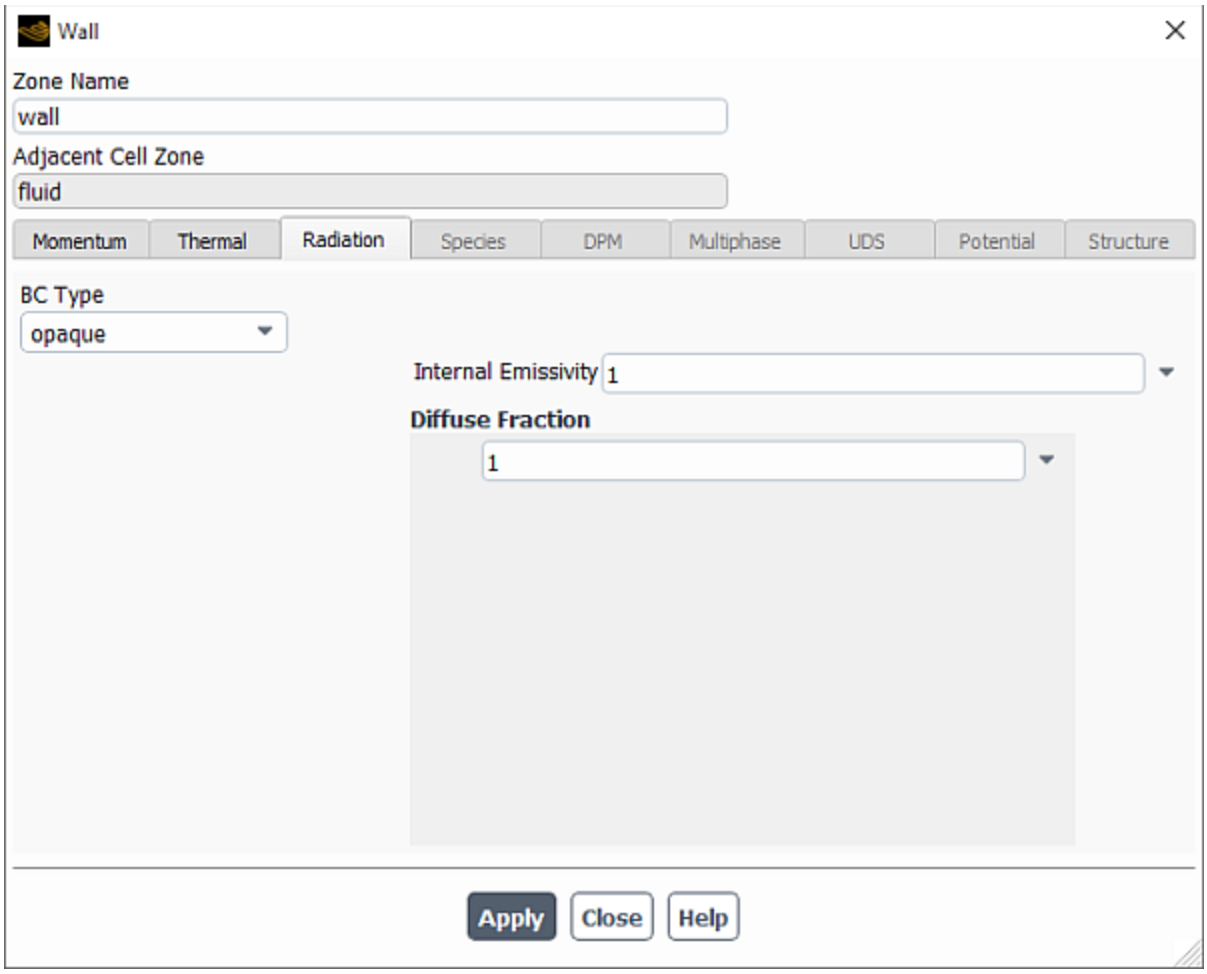
You can use a diffuse wall to model wall boundaries in many industrial applications since, for the most part, surface roughness makes the reflection of incident radiation diffuse. For highly polished surfaces, such as reflectors or mirrors, the specular boundary condition is appropriate. The semi-transparent boundary condition can be appropriate, for example, when modeling for glass panes in air.

16.3.8.3.1. Opaque Wall for the DO Model

In the **Radiation** tab of the **Wall** dialog box (Figure 16.28: The Wall Dialog Box Showing Radiation Conditions for an Opaque Wall (p. 2221)), select **opaque** in the **BC Type** drop-down list to specify an opaque wall.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.28: The Wall Dialog Box Showing Radiation Conditions for an Opaque Wall



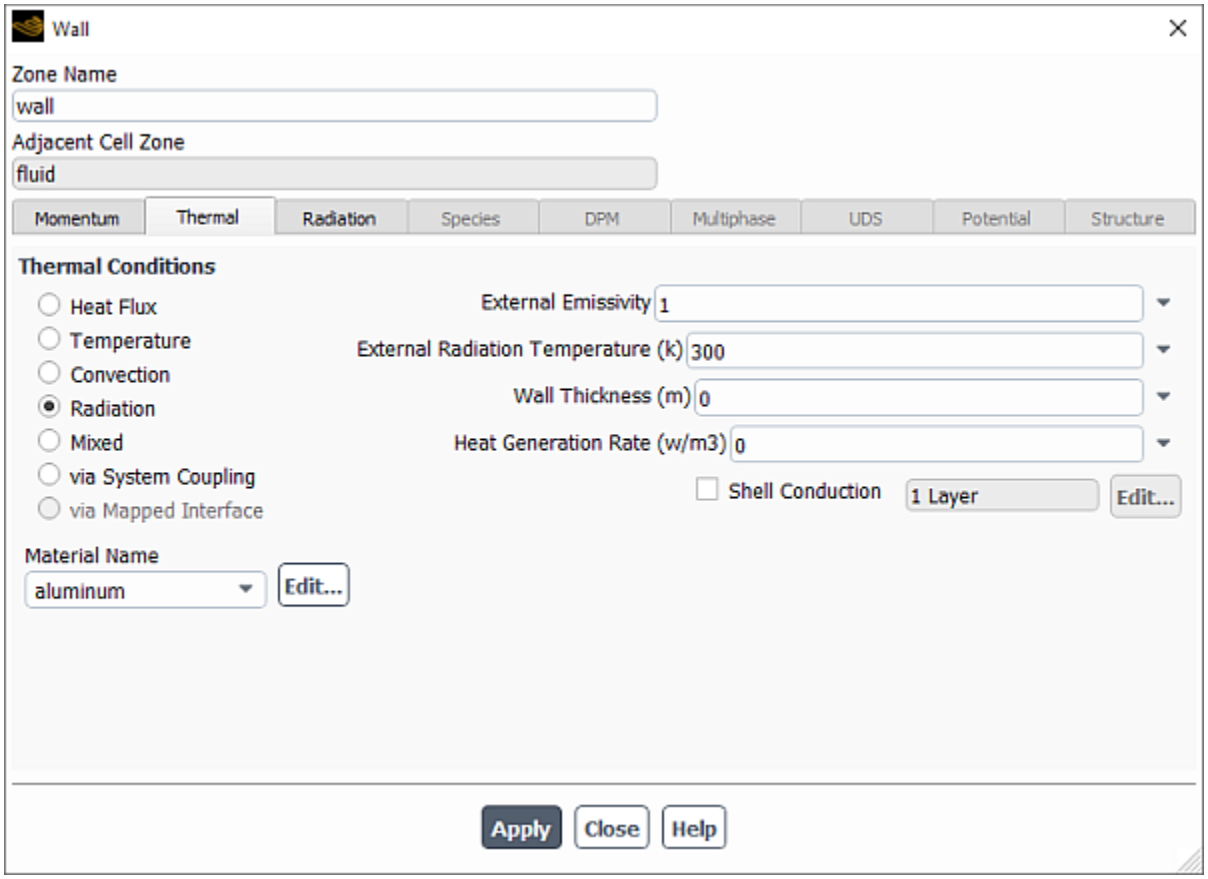
Specify the fraction of reflected radiation flux that is to be treated as diffuse. By default, the **Diffuse Fraction** is set to 1, indicating that all of the radiation is diffuse. A diffuse fraction equal to 0 indicates purely specular reflected radiation. A diffuse fraction between 0 and 1 will result in partially diffuse and partially specular reflected energy. If the non-gray DO model is being used, the **Diffuse Fraction** can be specified for each band. See Boundary and Cell Zone Condition Treatment at Opaque Walls in the Theory Guide for more details.

For gray-radiation DO models, enter the appropriate value for **Internal Emissivity** (default value is 1). For non-gray DO models, specify a constant **Internal Emissivity** for each wavelength band in the **Radiation** tab of the **Wall** dialog box. (The default value in each band is 1.) Alternatively, you can specify the internal emissivity using a boundary condition parameter (see Creating a New Parameter (p. 1311)).

You can also specify the external emissivity and external radiation temperature for an opaque wall when the thermal conditions are set to **Radiation** or **Mixed** in the **Wall** dialog box (Figure 16.29: The Wall Dialog Box Showing External Emissivity and External Radiation Temperature Thermal Conditions (p. 2222)). Alternatively, you can specify a UDF for these parameters; for more information, see `DEFINE_PROFILE` in the Fluent Customization Manual.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.29: The Wall Dialog Box Showing External Emissivity and External Radiation Temperature Thermal Conditions



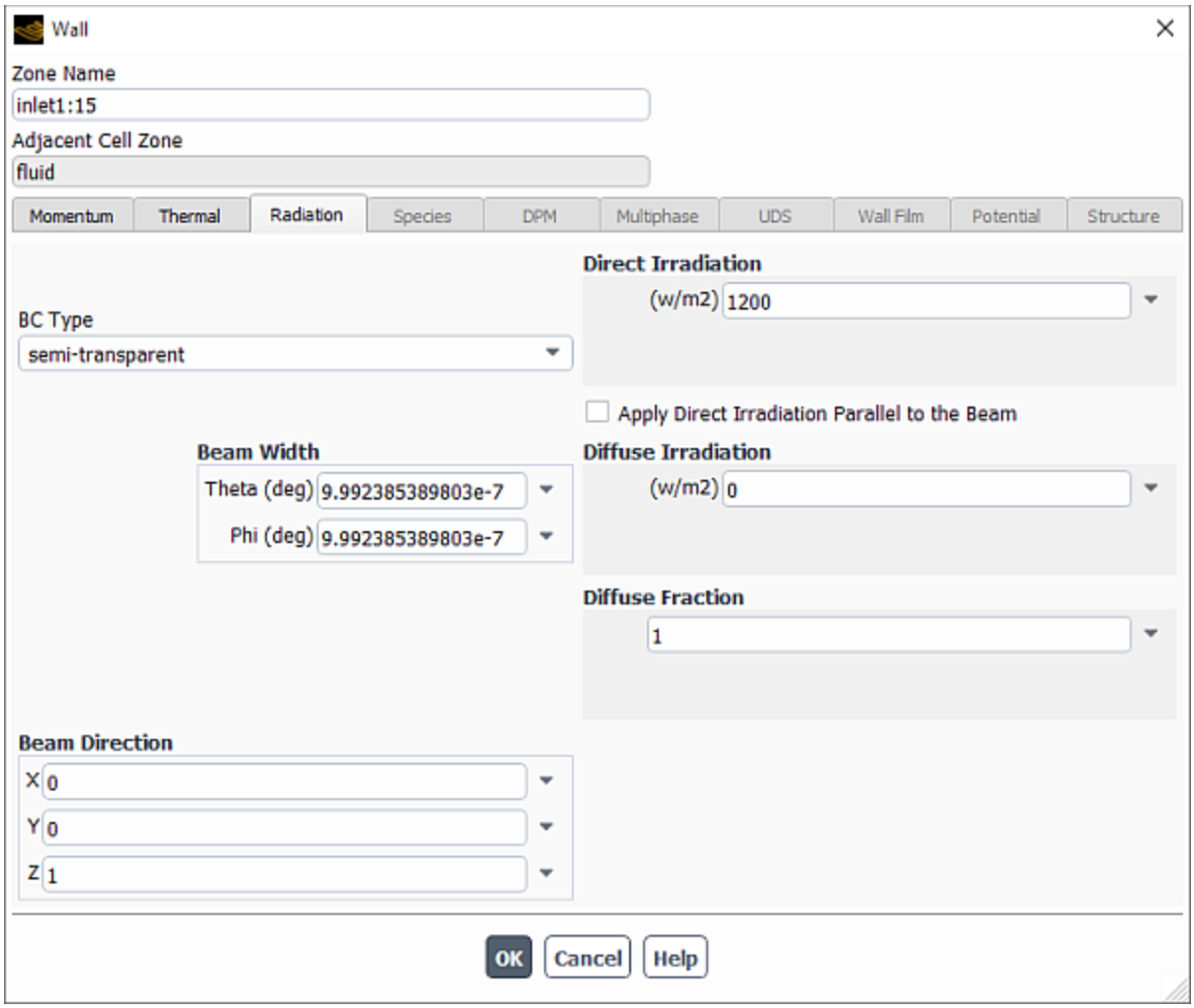
For more information on boundary condition treatment at opaque walls, see Boundary and Cell Zone Condition Treatment at Opaque Walls in the Theory Guide.

16.3.8.3.2. Semi-Transparent Walls for the DO Model

To define radiation for an exterior semi-transparent wall, click the **Radiation** tab in the **Wall** dialog box and then select **semi-transparent** in the **BC Type** drop-down list (Figure 16.30: The Wall Dialog Box for a Semi-Transparent Wall Boundary (p. 2223)). The dialog box will expand to display the semi-transparent wall inputs needed to define an external irradiation flux (Figure 16.30: The Wall Dialog Box for a Semi-Transparent Wall Boundary (p. 2223)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.30: The Wall Dialog Box for a Semi-Transparent Wall Boundary



Then perform the following steps:

1. Specify the value of the irradiation flux (in W/m^2) under **Direct** or **Diffuse Irradiation**. If the non-gray DO model is being used, a constant **Direct** or **Diffuse Irradiation** can be specified for each band.

   Important:

   Note that the external diffuse irradiation specified when using **Radiation** or **Mixed** thermal conditions (selected in the **Thermal** tab), or **Diffuse Irradiation** (in the **Radiation** tab) is always distributed hemispherically after transmission through semi-transparent walls (that is independent of whether the external semi-transparent boundary wall is defined as a diffusely or specularly reflecting type).

2. By default, **Apply Direct Irradiation Parallel to the Beam** is selected, which means Fluent assumes that the value you specify for **Direct Irradiation** is the irradiation flux parallel to the **Beam Direction**. When deselected, Ansys Fluent assumes that the value you specify for **Direct Irradiation** is the irradiation flux normal to the boundary. See Figure 5.12: DO Irradiation on

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

External Semi-Transparent Wall in Semi-Transparent Exterior Walls in the Theory Guide for details.

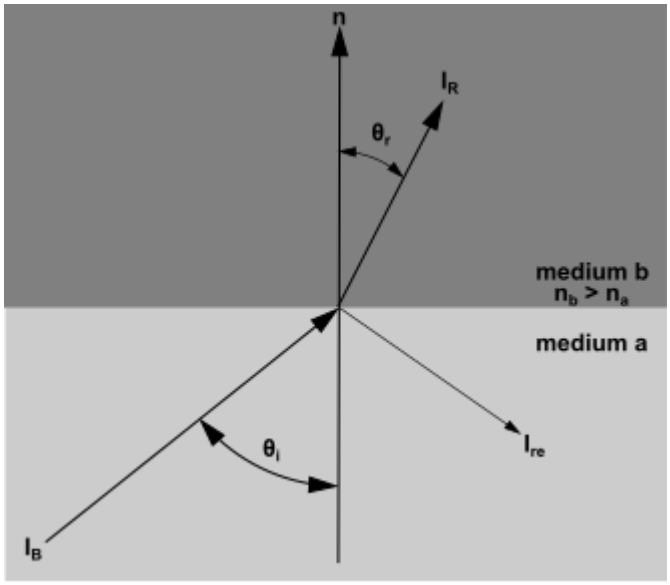
3. Define the **Beam Width** by specifying the beam **Theta** and **Phi** extents. Beam width is specified as the solid angle over which the irradiation is distributed. The default value for beam width is $1e^{-6}$, which is suitable for collimated beam radiation. A beam width less than this is likely to result in zero irradiation flux.

4. Specify the (**X,Y,Z**) vector that defines the **Beam Direction**. The beam direction is defined as the vector of the centroid of the solid angle (beam width). You can specify the **Beam Direction** as a constant, a profile or a UDF. This is especially useful in applications where the shape of the radiative source is circular or cylindrical (or nonlinear). For information about boundary profiles, see Reading and Writing Profile Files (p. 964).

   Note that the actual direction of the beam of radiation that enters the domain will be further influenced by the solid angles available from the number of divisions set up; the effective direction will be the direction vector of the solid angle that the incoming beam falls into. Finally, any nonzero diffuse fraction will act to spread out (hemispherically, proportional to the diffuse fraction) the irradiation that enters the domain.

   Note that the refractive index of the external medium is assumed to be 1. Therefore, if the refractive index of the fluid or solid material adjacent to the semi-transparent boundary is greater than 1, the direction of the prescribed direct irradiation after entering the computational domain will change due to refraction, and the magnitude of the incident radiation can be determined according to Figure 16.31: Refraction of Irradiation Entering Computational Domain (p. 2224).

   Figure 16.31: Refraction of Irradiation Entering Computational Domain



   For a UDF example that specifies the beam direction, see Example 5 - Beam Direction Profile at Semi-Transparent Walls in the Fluent Customization Manual.

5. Specify the **Diffuse Fraction**, the fraction of the reflected radiation from the wall that is treated as diffuse between 0 and 1. By default, the **Diffuse Fraction** is set to 1, indicating that all of

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the radiation is diffuse. A **Diffuse Fraction** of 0 treats the reflected radiation as purely specular. If you specify a value between 0 and 1, the radiation is treated as partially diffuse and partially specular. If the non-gray DO model is being used, the **Diffuse Fraction** can be specified for each band. See Diffuse Semi-Transparent Walls in the Theory Guide for details.

Important:

- If **Heat Flux** conditions are specified in the **Thermal** tab of the **Wall** dialog box, the specified heat flux is considered to be only the conduction and convection portion of the boundary flux. The given irradiation specifies the incoming exterior radiative flux; the radiative flux transmitted from the domain interior to the outside is computed as a part of the calculation by Ansys Fluent. Internal emissivity is ignored for semi-transparent surfaces.
- Note that when a boundary wall is made semi-transparent Ansys Fluent calculates the amount of radiation leaving as well as entering the domain. If you do not provide a source of irradiation or a radiating thermal condition (for example **Mixed** or **Radiation**) then you are effectively radiating to a temperature of 0 K and it is highly likely you may observe temperatures in your model that are lower than expected. Ensure that the external (incoming) radiant conditions give good account of the surroundings.

You can also specify the external emissivity and external radiation temperature for a semi-transparent wall when the thermal conditions are set to **Radiation** or **Mixed** in the **Wall** dialog box (Figure 16.29: The Wall Dialog Box Showing External Emissivity and External Radiation Temperature Thermal Conditions (p. 2222)). Alternatively, you can specify a user-defined function (UDF) for these parameters; for more information, see `DEFINE_PROFILE` in the Fluent Customization Manual.

Important:

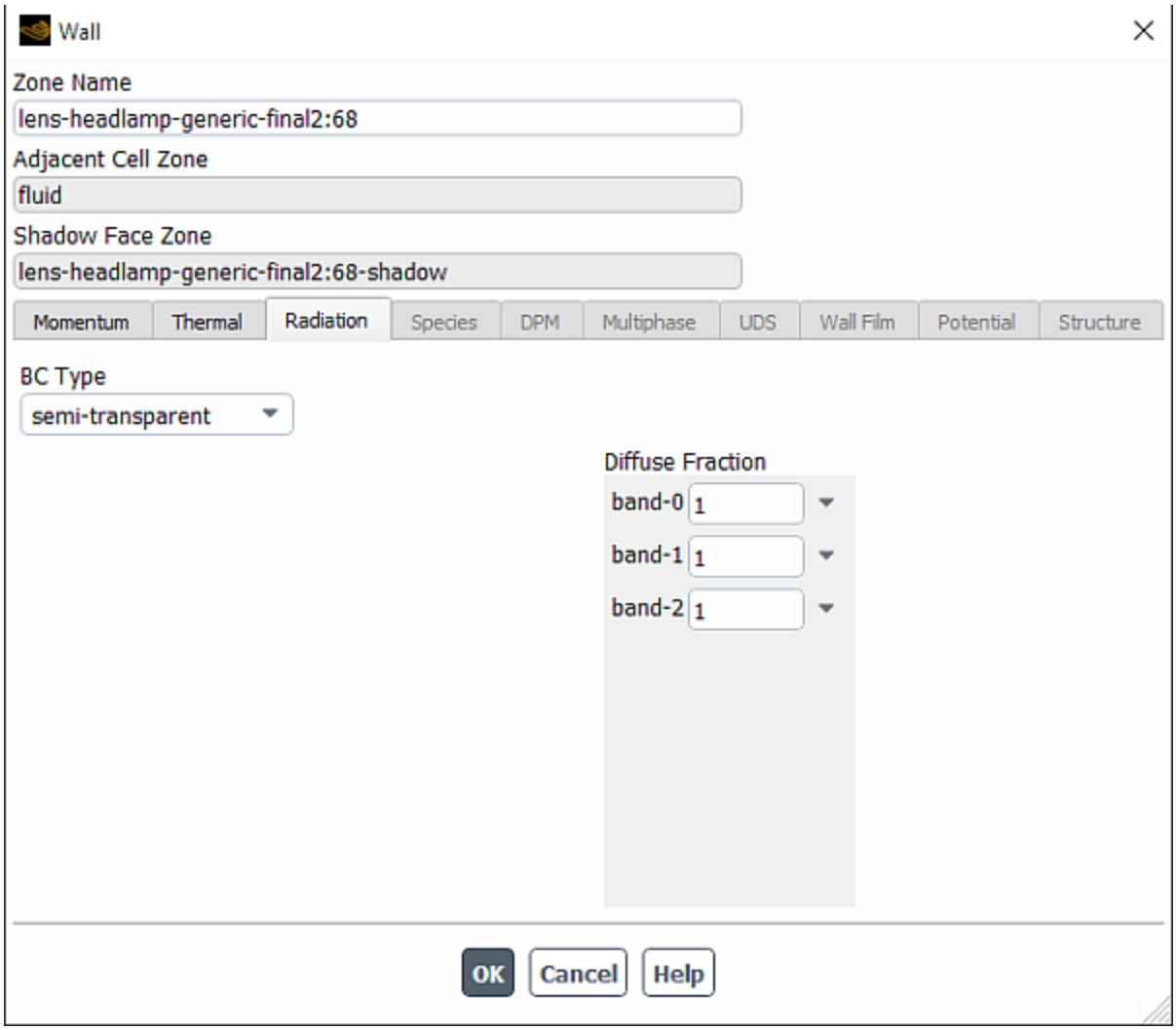
Note that for semi-transparent walls, **Internal Emissivity** is not available. If you want to include the effect of internal emissivity at semi-transparent walls, you must create a solid zone and specify that it participates in the radiation calculation; see Solid Semi-Transparent Media in the Theory Guide for details.

For a detailed description of boundary condition treatment at semi-transparent walls, see Cell Zone and Boundary Condition Treatment at Semi-Transparent Walls in the Theory Guide.

To define radiation for an interior (two-sided) semi-transparent wall, in the **Wall** dialog box, click the **Radiation** tab and then select **semi-transparent** in the **BC Type** drop-down list (Figure 16.32: The Wall Dialog Box for an Interior Semi-Transparent Wall (p. 2226)). Then specify the **Diffuse Fraction** as described for the previous case.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.32: The Wall Dialog Box for an Interior Semi-Transparent Wall



16.3.8.4. Wall Boundary Conditions for the MC Model

When the MC model is used, you can model walls as **opaque** or **semi-transparent**. Semi-transparent walls allows for the transmission of radiation through the wall (such as glass) whereas an opaque wall does not. Except in the case of an internal semi-transparent wall, you can also specify a **Boundary Source** where the wall will act as a radiation source with the specified characteristics (such as a light source).

16.3.8.4.1. Opaque Walls for the MC Model

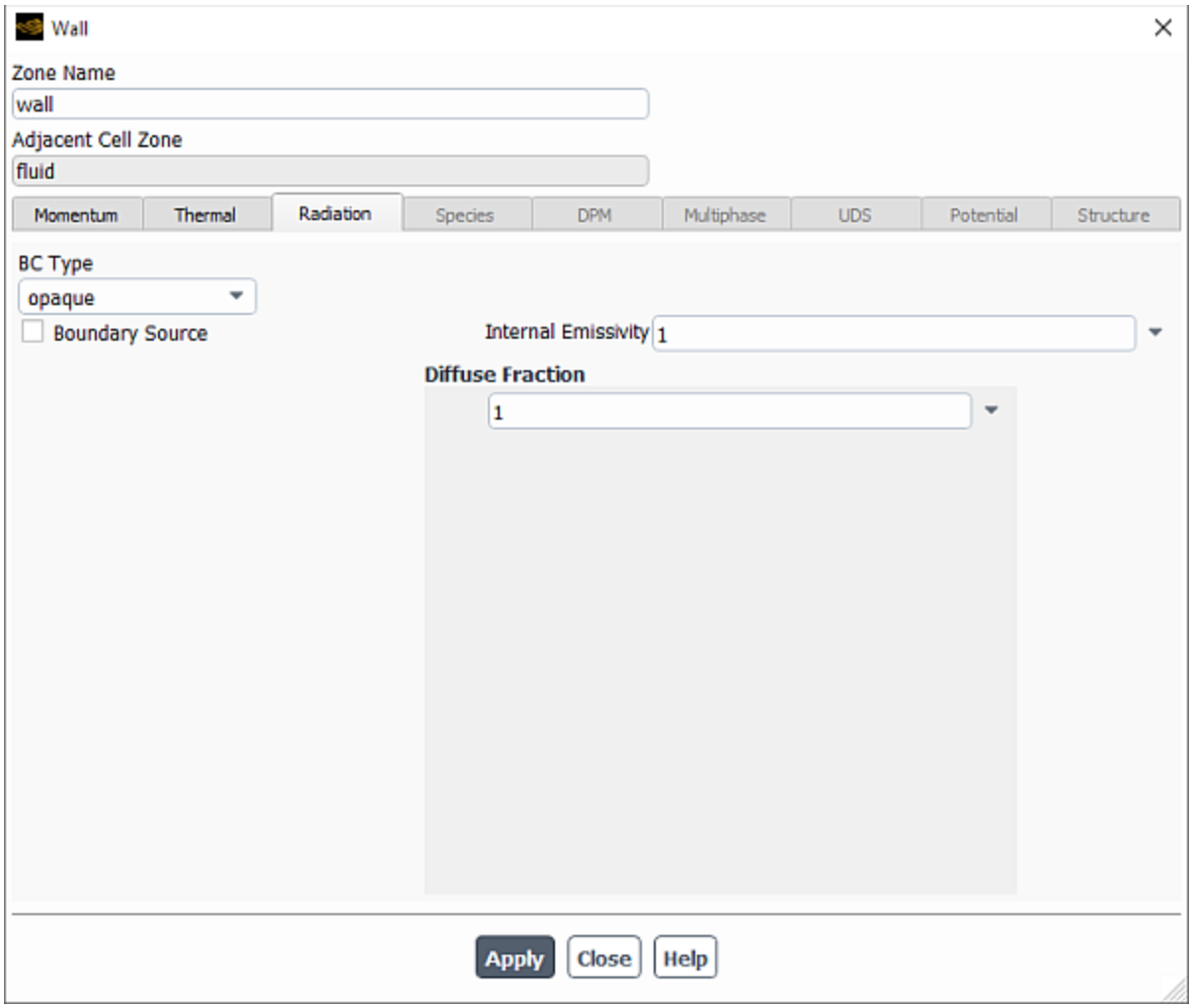
16.3.8.4.2. Semi-Transparent Walls for the MC Model

16.3.8.4.1. Opaque Walls for the MC Model

In the **Radiation** tab of the **Wall** dialog box (Figure 16.33: The Wall Dialog Box for an Opaque Wall with MC Model (Gray) (p. 2227)), select **opaque** in the **BC Type**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.33: The Wall Dialog Box for an Opaque Wall with MC Model (Gray)



You have the option of specifying an irradiation flux by selecting **Boundary Source**. (Figure 16.34: The Wall Dialog Box for an Opaque Wall with MC Model (Boundary Source) (p. 2228))

1. Specify the value of the irradiation flux (in W/m^2) under **Direct Irradiation**. If the non-gray MC model is being used, a constant **Direct Irradiation** can be specified for each band.

2. Specify the (**X,Y,Z**) vector that defines the **Beam Direction**. To model an isotropic source, you must set the **Beam Direction** to `0,0,0`.

   Note:

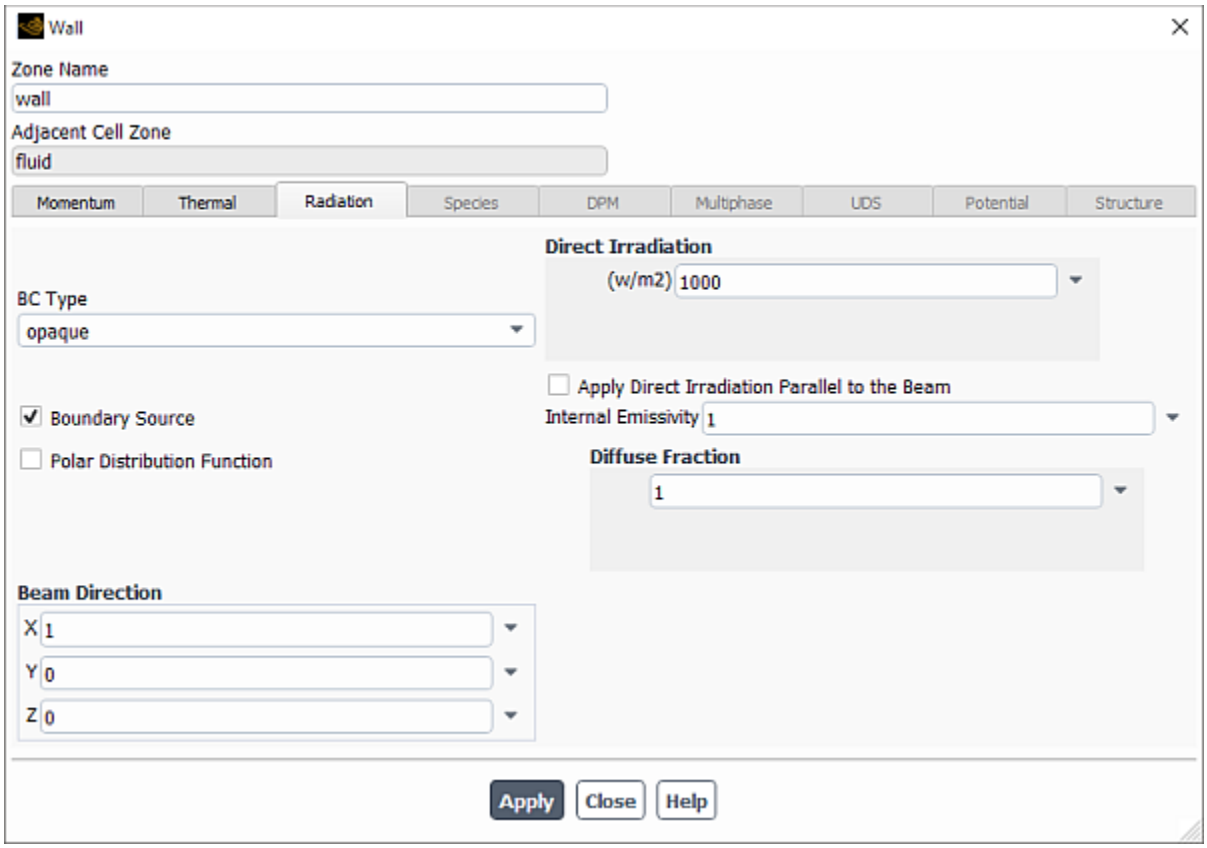
   For multiband cases, if Direct Sources are zero for all bands, and you want to model a diffuse source, copy the band-wise diffuse sources to direct sources (band0 to band0, and so on). When doing this, specify the Directional Vector as zero (0,0,0). This will model the diffuse source per band as expected.

3. By default, **Apply Direct Irradiation Parallel to the Beam** is selected, which means Fluent assumes that the value you specify for **Direct Irradiation** is the irradiation flux parallel to the

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Beam Direction. When deselected, Ansys Fluent assumes that the value you specify for **Direct Irradiation** is the irradiation flux normal to the boundary.

Figure 16.34: The Wall Dialog Box for an Opaque Wall with MC Model (Boundary Source)



4. Optionally select **Polar Distribution Function**, which can be used if the irradiation intensity varies with direction, but is rotationally symmetric about a given direction. For example, LED light sources.

   You have the option of specifying a polar distribution via an **Expression** or **Table**.

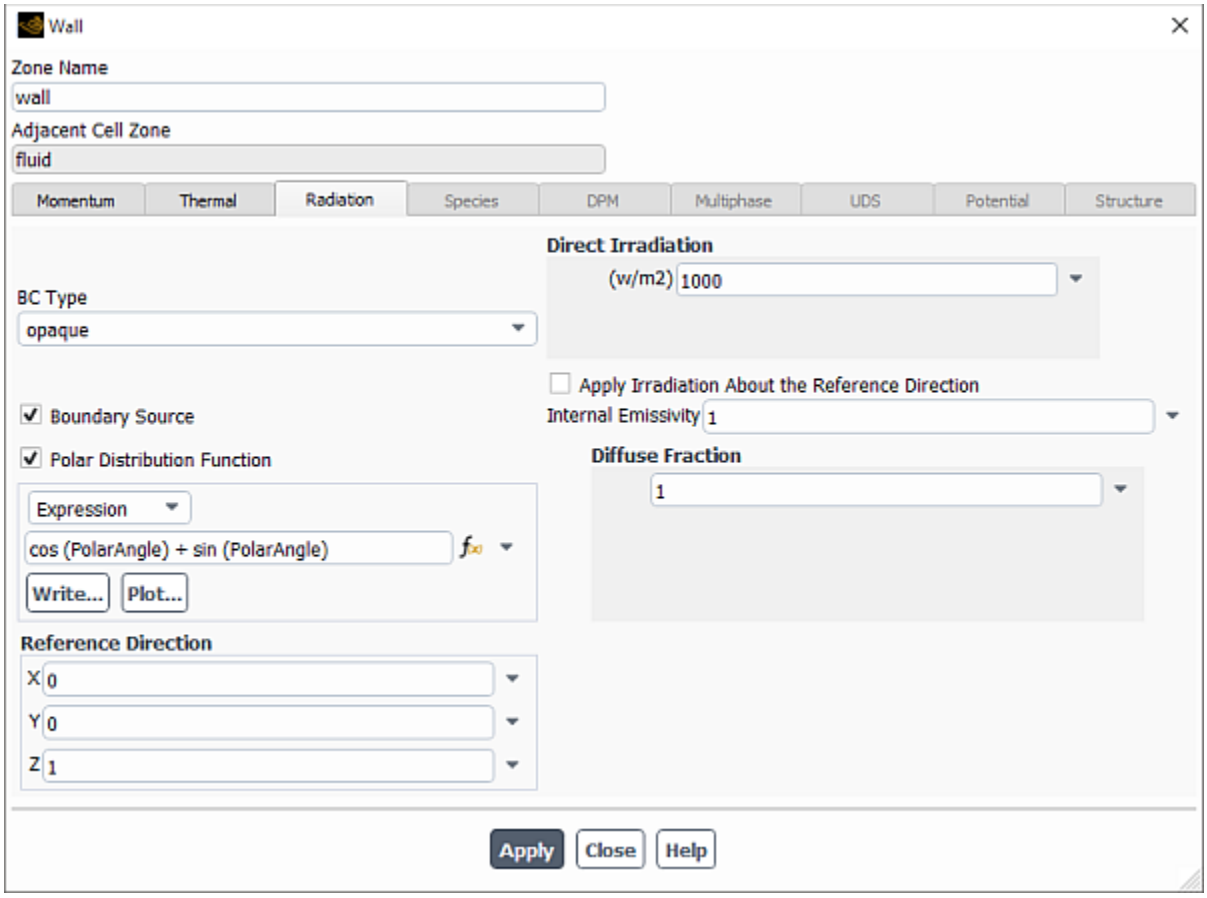
   If you select **Expression** (Figure 16.35: The Wall Dialog Box for an Opaque Wall with MC Model (Polar Distribution with Expression) (p. 2229)), enter an expression to define the polar distribution as a function of "PolarAngle". For example:

   ```
   cos(PolarAngle) + sin(PolarAngle)
   ```

   The expression is evaluated at 100 angles between 0 and 90 degrees to form data pairs of **Polar Angle** and **Relative Intensity**. You can write this polar distribution data via a `.csv` file using the **Write...** button or create a polar coordinates plot in the graphics window using the **Plot...** button (as shown in Figure 16.36: Polar Coordinate Plot of Radiation Intensity for Polar Distribution Function (p. 2230)).

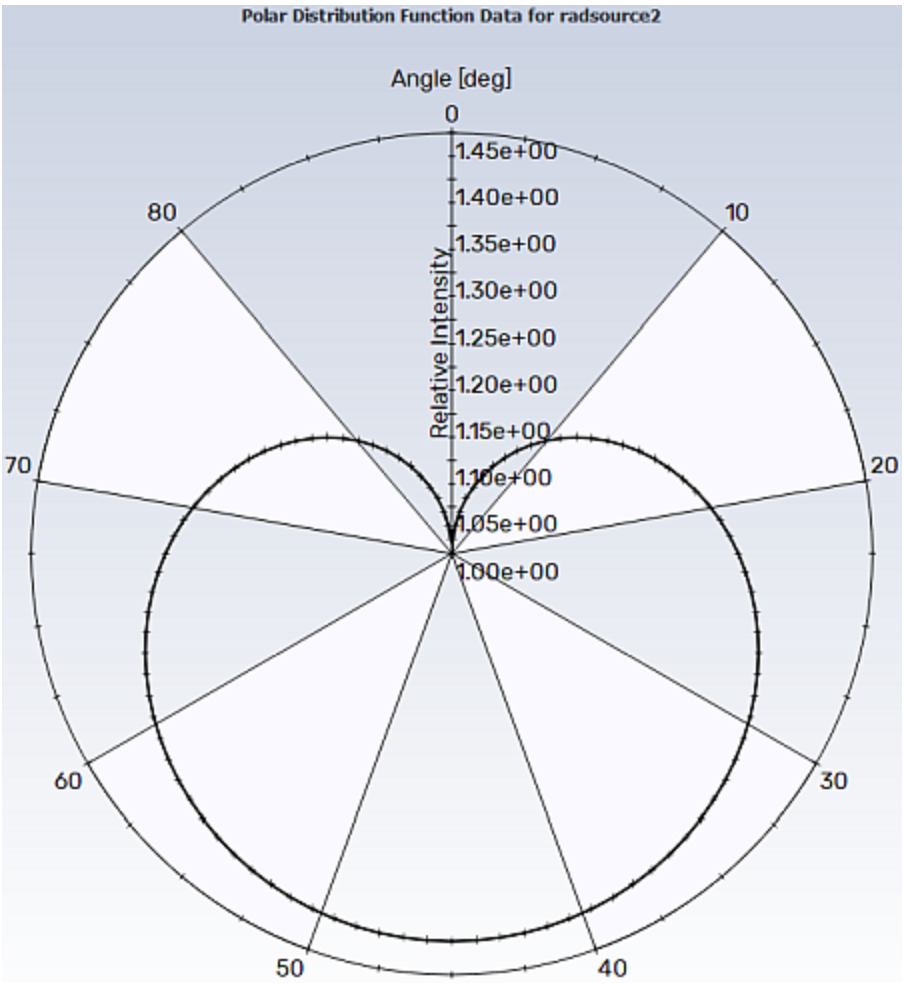
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.35: The Wall Dialog Box for an Opaque Wall with MC Model (Polar Distribution with Expression)



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.36: Polar Coordinate Plot of Radiation Intensity for Polar Distribution Function



If you select **Table** (Figure 16.37: The Wall Dialog Box for an Opaque Wall with MC Model (Polar Distribution with Table) (p. 2231)), you must manually enter the data pairs or read a previously created `.csv` file.

a. Select the **Number of Data Pairs** to define the polar coordinate system for your irradiation source.

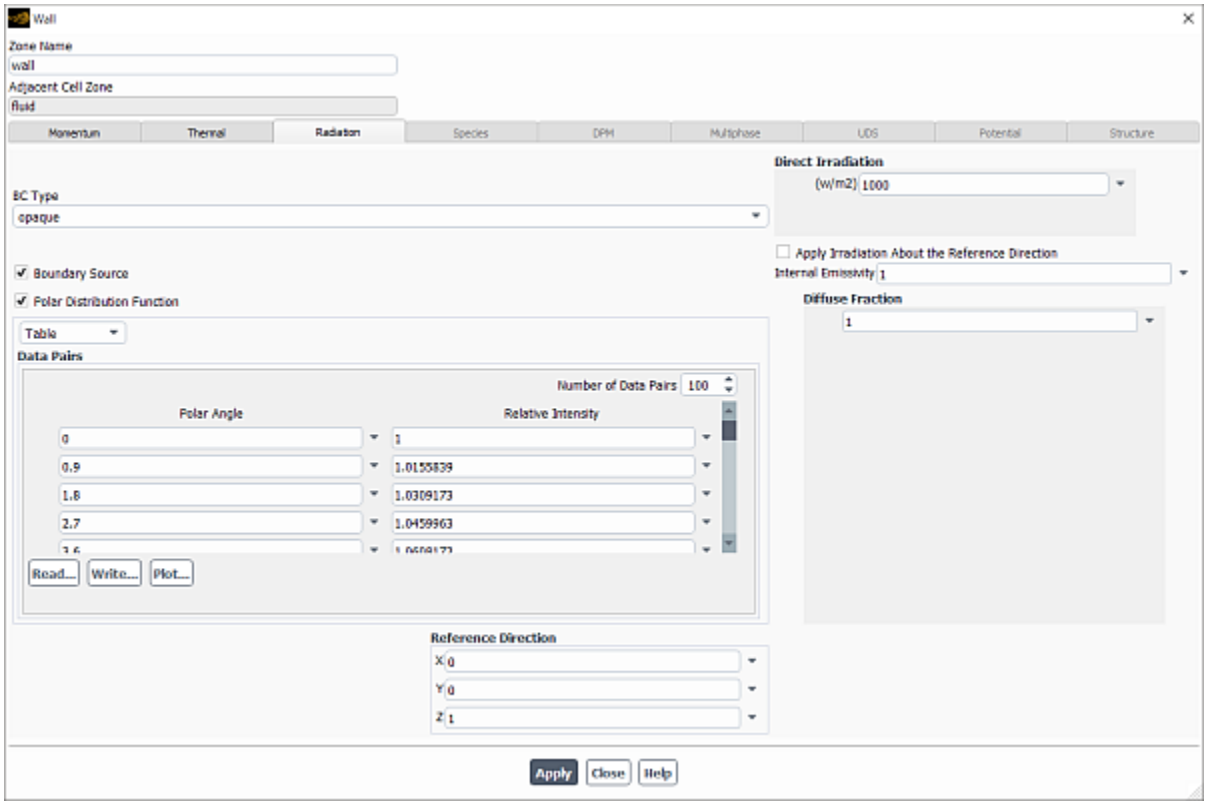
b. Enter the data pairs in the form of **Angle** and **Relative Intensity**.

   Note that the minimum number of data pairs is 2 and must include relative intensity data for 0 degrees. All angles must be between 0 and 90 degrees. **Relative Intensity** is typically between 0 and 1 although it does not have to be. A **Relative Intensity** of zero at a given angle means there is no radiation in that direction.

c. You have the option of reading and writing the polar distribution data (via `.csv` files) using the **Read...** and **Write...** buttons. You can create a polar coordinate plot in the graphics window by selecting **Plot...** (as shown in Figure 16.36: Polar Coordinate Plot of Radiation Intensity for Polar Distribution Function (p. 2230)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.37: The Wall Dialog Box for an Opaque Wall with MC Model (Polar Distribution with Table)



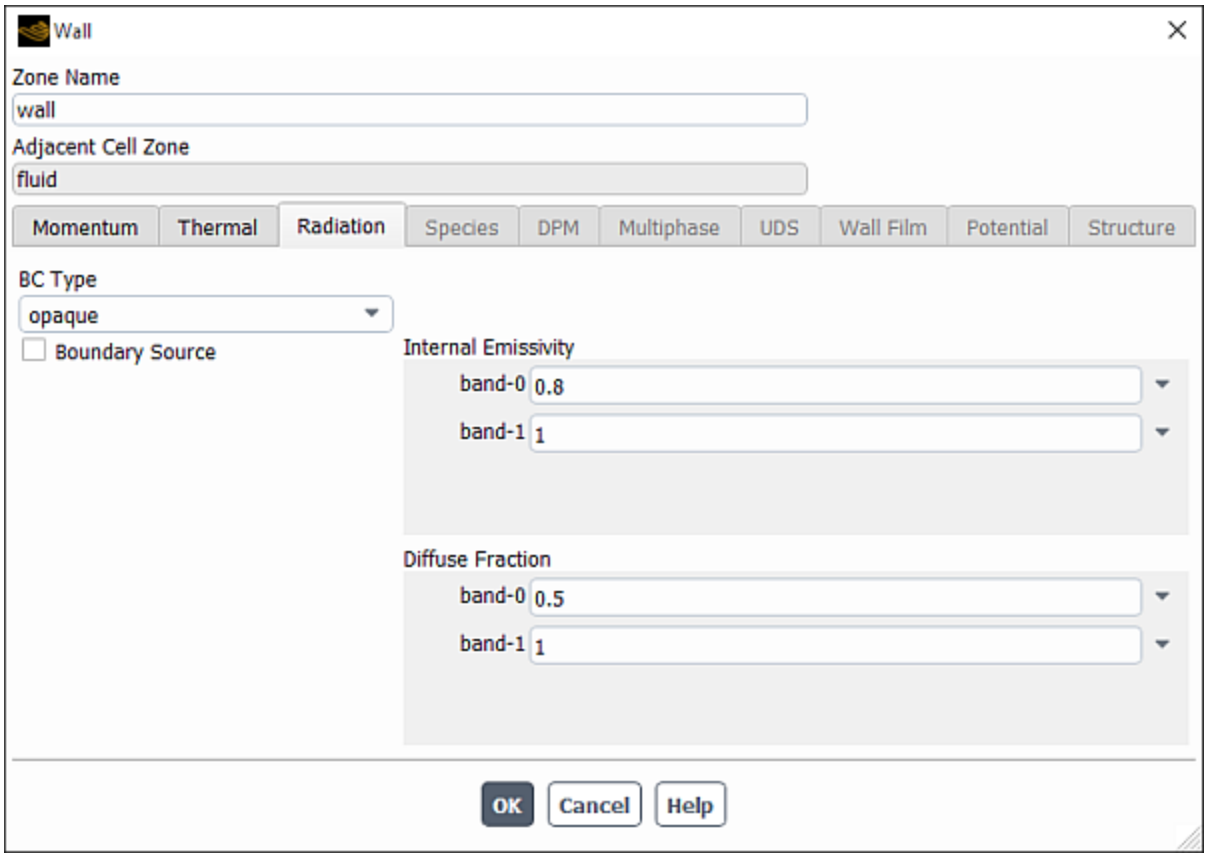
Specify the (**X,Y,Z**) vector that defines the **Reference Direction**. This determines the center of the polar distribution function.

By default, **Apply Irradiation About the Reference Direction** is selected, which means Fluent assumes that the value you specify for **Direct Irradiation** is the irradiation flux parallel to the **Reference Direction**. When deselected, Ansys Fluent assumes that the value you specify for **Direct Irradiation** is the irradiation flux normal to the boundary.

5. You will also be required to specify the **Internal Emissivity** or, in the case of the non-gray MC model, specify an **Internal Emissivity** for each wavelength band.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

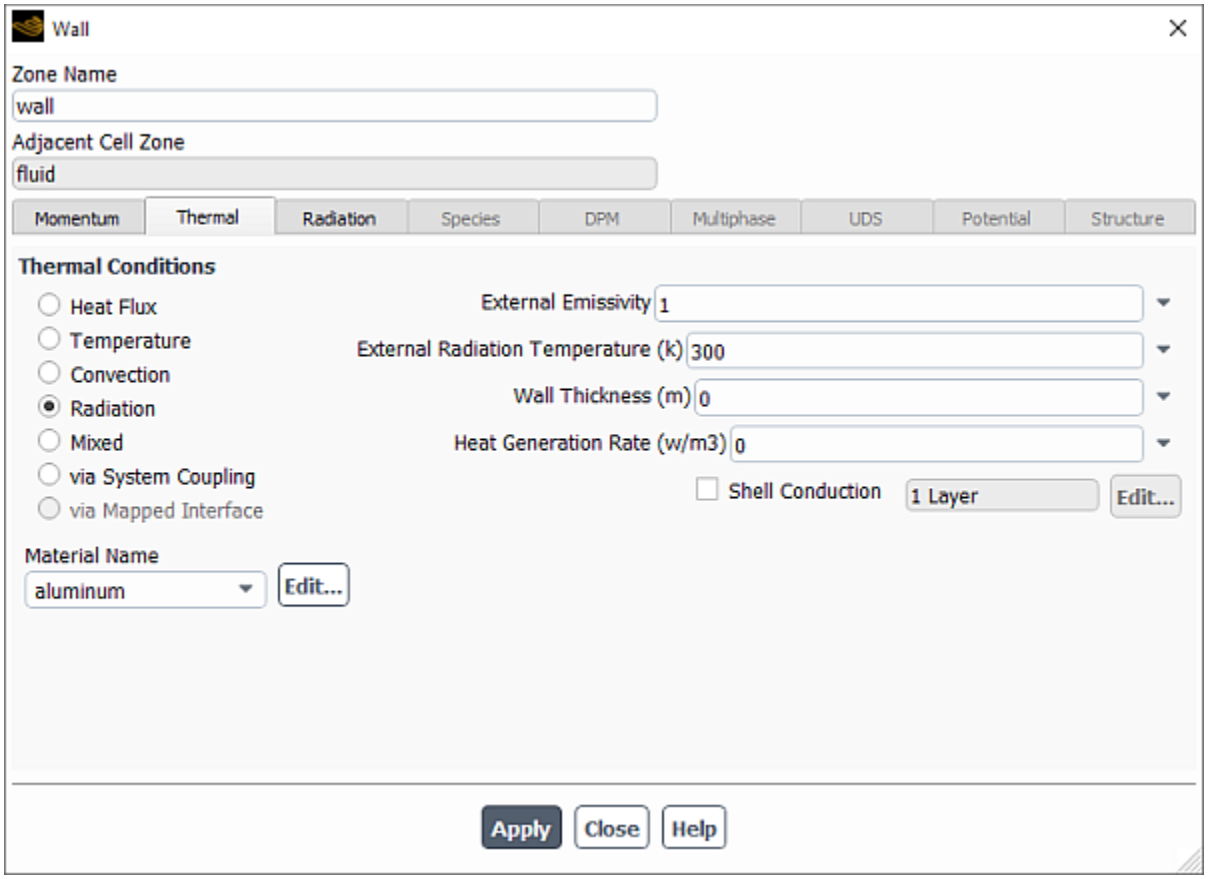
Figure 16.38: The Wall Dialog Box for an Opaque Wall with MC Model (Non-gray)



6. You can also specify the external emissivity and external radiation temperature for an external wall when the thermal conditions are set to **Radiation** or **Mixed** in the **Wall** dialog box (Figure 16.39: The Wall Dialog Box Showing External Emissivity and External Radiation Temperature Thermal Conditions (p. 2233)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.39: The Wall Dialog Box Showing External Emissivity and External Radiation Temperature Thermal Conditions



7. Specify the **Diffuse Fraction**, the fraction of the reflected radiation from the wall that is treated as diffuse between 0 and 1. By default, the **Diffuse Fraction** is set to 1, indicating that all of the radiation is diffuse. A diffuse fraction of 0 treats the radiation as purely specular. If you specify a value between 0 and 1, the radiation is treated as partially diffuse and partially specular. If the non-gray MC model is being used, the **Diffuse Fraction** can be specified for each band.

Note:

For two-sided walls, while thermal boundary conditions are copied between a wall and it's shadow, any specified irradiation is not copied and each side of the wall is treated separately.

16.3.8.4.2. Semi-Transparent Walls for the MC Model

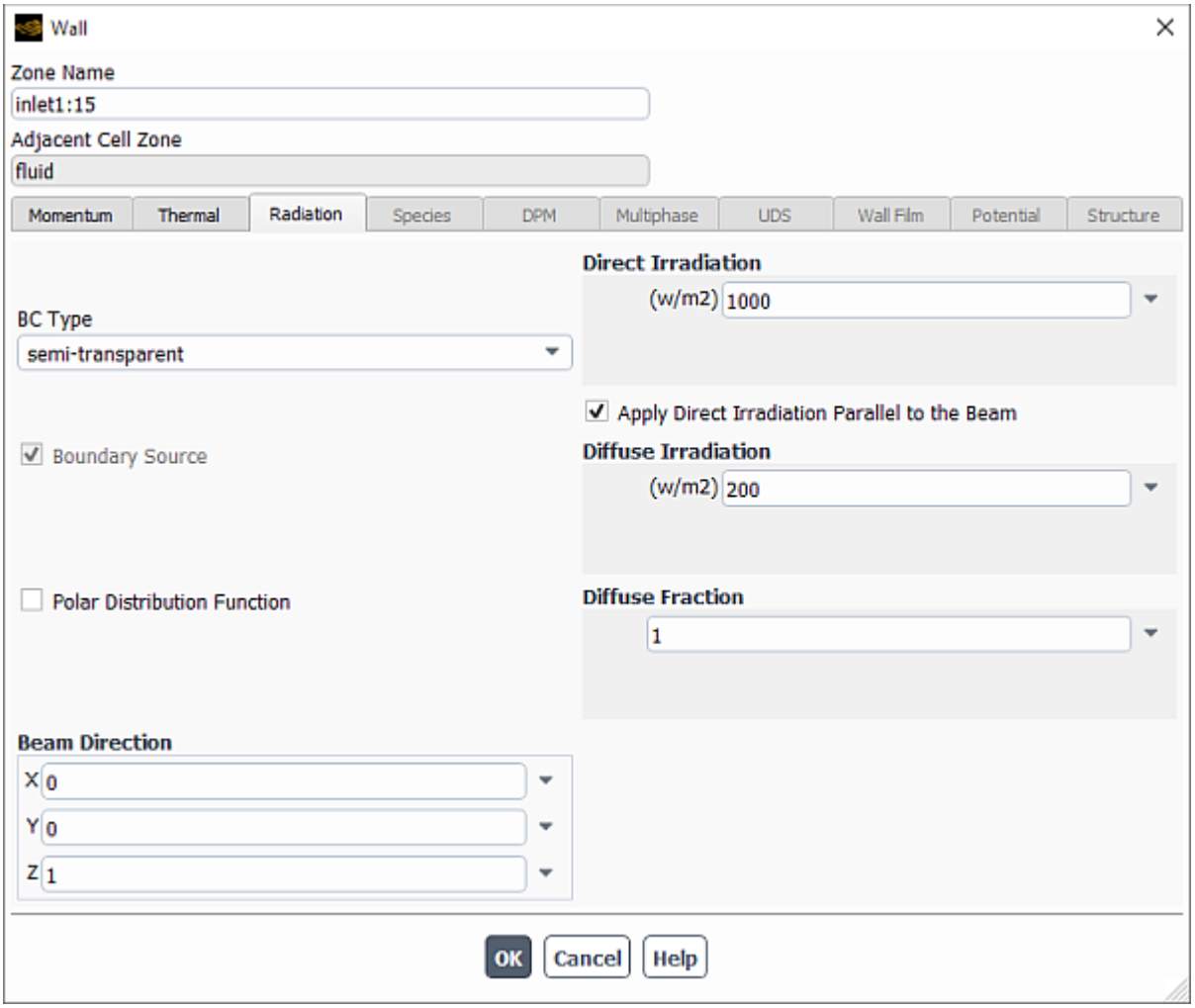
To define radiation for a semi-transparent wall, click the **Radiation** tab in the **Wall** dialog box and then select **semi-transparent** in the **BC Type**.

- For internal semi-transparent walls, the boundary source option is not available. Specify the **Internal Emissivity** and **Diffuse Fraction** as described in Opaque Walls for the MC Model (p. 2226).
- For external semi-transparent walls, **Boundary Source** is automatically enabled.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– Provide the external irradiation as described in Opaque Walls for the MC Model (p. 2226), although you can specify **Direct** or **Diffuse Irradiation** for semi-transparent external walls when using the MC model as shown in Figure 16.40: The Wall Dialog Box for a Semi-transparent Wall with MC Model (p. 2234).

Figure 16.40: The Wall Dialog Box for a Semi-transparent Wall with MC Model



– If you specify the external emissivity and external radiation temperature for a semi-transparent wall when the thermal conditions are set to **Radiation** or **Mixed** in the **Wall** dialog box, the external irradiation is treated as diffuse.

Important:

- Note that the refractive index of the external medium is assumed to be 1.
- If **Heat Flux** conditions are specified in the **Thermal** tab of the **Wall** dialog box, the specified heat flux is considered to be only the conduction and convection portion of the boundary flux. The given irradiation specifies the incoming exterior radiative flux; the radiative flux transmitted from the domain interior to the outside is computed as a part of the calculation by Ansys Fluent. Internal emissivity is ignored for semi-transparent surfaces.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

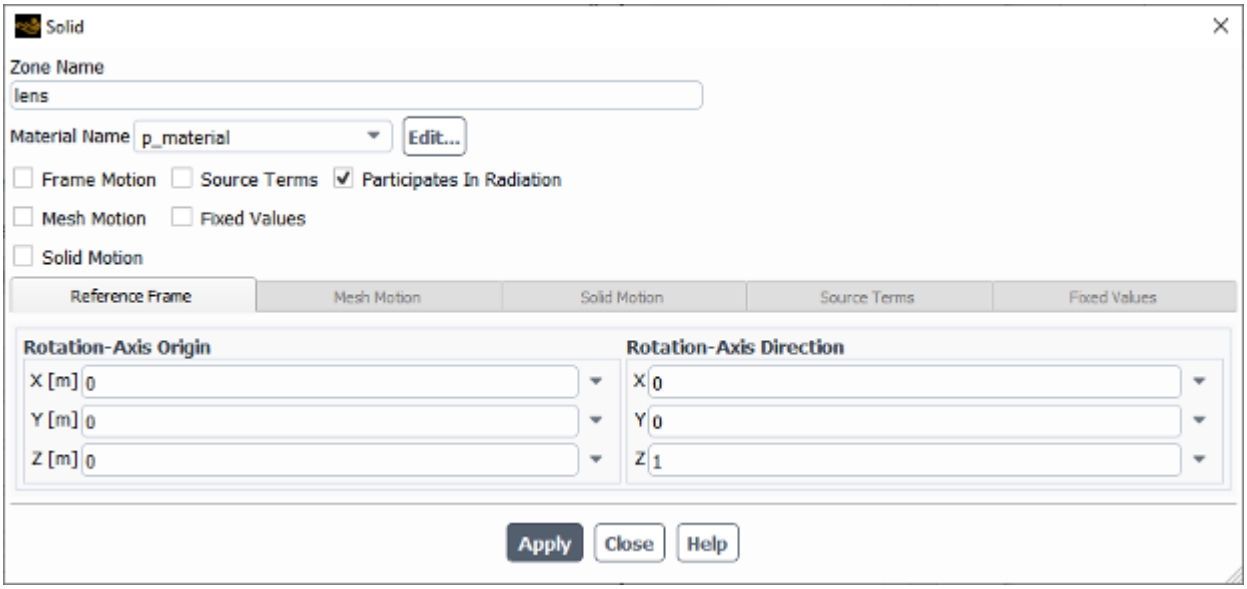
- Note that when a boundary wall is made semi-transparent Ansys Fluent calculates the amount of radiation leaving as well as entering the domain. If you do not provide a source of irradiation or a radiating thermal condition (for example **Mixed** or **Radiation**) then you are effectively radiating to a temperature of 0 K and it is highly likely you may observe temperatures in your model that are lower than expected. Ensure that the external (incoming) radiant conditions give good account of the surroundings.
- **Internal Emissivity** is assumed to be zero for external semi-transparent walls. For Internal semi-transparent walls, internal emissivity is only required on walls between fluid and solid zones, or between two solid zones.

For a detailed description of boundary condition treatment at semi-transparent walls (for the DO model), see Cell Zone and Boundary Condition Treatment at Semi-Transparent Walls in the Theory Guide.

16.3.8.5. Solid Cell Zones Conditions for the DO or MC Models

With the DO or MC model, you can specify whether or not you want to solve for radiation in each cell zone in the domain. By default, the DO or MC equations are solved in all fluid zones, but not in any solid zones. If you want to model semi-transparent media, for example, you can enable radiation in the solid zone(s). To do so, enable the **Participates In Radiation** option in the **Solid** dialog box (Figure 16.41: The Solid Dialog Box (p. 2235)).

Figure 16.41: The Solid Dialog Box



Important:

In general, you should *not* disable the **Participates In Radiation** option for any fluid zones.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See Solid Semi-Transparent Media in the Theory Guide for more information on solid semi-transparent media.

16.3.8.6. Thermal Boundary Conditions

In general, any well-posed combination of thermal boundary conditions can be used when any of the radiation models is active. The radiation model will be well-posed in combination with fixed temperature walls, conducting walls, and/or walls with set external heat transfer boundary conditions (Thermal Boundary Conditions at Walls (p. 1467)). You can also use any of the radiation models with heat flux boundary conditions defined at walls, in which case the heat flux you define will be treated as the sum of the convective and radiative heat fluxes. The exception to this is the case of semi-transparent walls for the DO model. Here, Ansys Fluent allows you to specify the convective and radiative portions of the heat flux separately.

16.3.9. Solution Strategies for Radiation Modeling

For the P-1, DTRM, S2S, DO, MC radiation models, there are several parameters that control the radiation calculation. You can use the default solution parameters for most problems, or you can modify these parameters to control the convergence and accuracy of the solution. Iteration parameters that are unique for a particular radiation model are specified in the **Radiation Model** dialog box. For the DTRM, S2S, DO, and MC models there are a common set of **Iteration Parameters** that control the frequency that the radiation model field is solved (as described in Iteration Parameters For Radiation Solve Frequency (p. 2240)). **Spatial Discretization** (Discretization in the Theory Guide) and **Under-Relaxation Factors** (Under-Relaxation of Variables in the Theory Guide) are specified in the **Solution Methods** and **Solution Controls** task pages, respectively. The **Convergence Criterion** (Modifying Convergence Criteria (p. 3693)) is set in the **Residual Monitors** dialog box.

There are no solution parameters to be set for the Rosseland model, since it impacts the solution only through the energy equation.

Important:

If radiation is the only model being solved in Ansys Fluent and all other equations are disabled, then the radiation transfer equation will automatically be solved for every iteration.

16.3.9.1. P-1 Model Solution Parameters

For the P-1 radiation model, you can control the convergence criterion and under-relaxation factor. You should also pay attention to the optical thickness, as described below.

The default convergence criterion for the P-1 model is 10^{-6}, the same as that for the energy equation, since the two are closely linked. See Monitoring Residuals (p. 3684) for details about convergence criteria. You can set the **Convergence Criterion** for **p1** in the **Residual Monitors** dialog box.

Solution → **Monitors** → **Residuals** **Edit...**

The under-relaxation factor for the P-1 model is set with those for other variables, as described in Setting Under-Relaxation Factors (p. 3589). Note that since the equation for the radiation temperature

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(Equation 5.60 in the Theory Guide) is a relatively stable scalar transport equation, in most cases you can safely use large values of under-relaxation (0.9–1.0).

For optimal convergence with the P-1 model, the optical thickness $(a+\sigma_s)L$ must be between 0.01 and 10 (preferably not larger than 5). Smaller optical thicknesses are typical for very small enclosures (characteristic size of the order of 1 cm), but for such problems you can safely increase the absorption coefficient to a value for which $(a+\sigma_s)L=0.01$. Increasing the absorption coefficient will not change the physics of the problem because the difference in the level of transparency of a medium with optical thickness = 0.01 and one with optical thickness <0.01 is indistinguishable within the accuracy level of the computation.

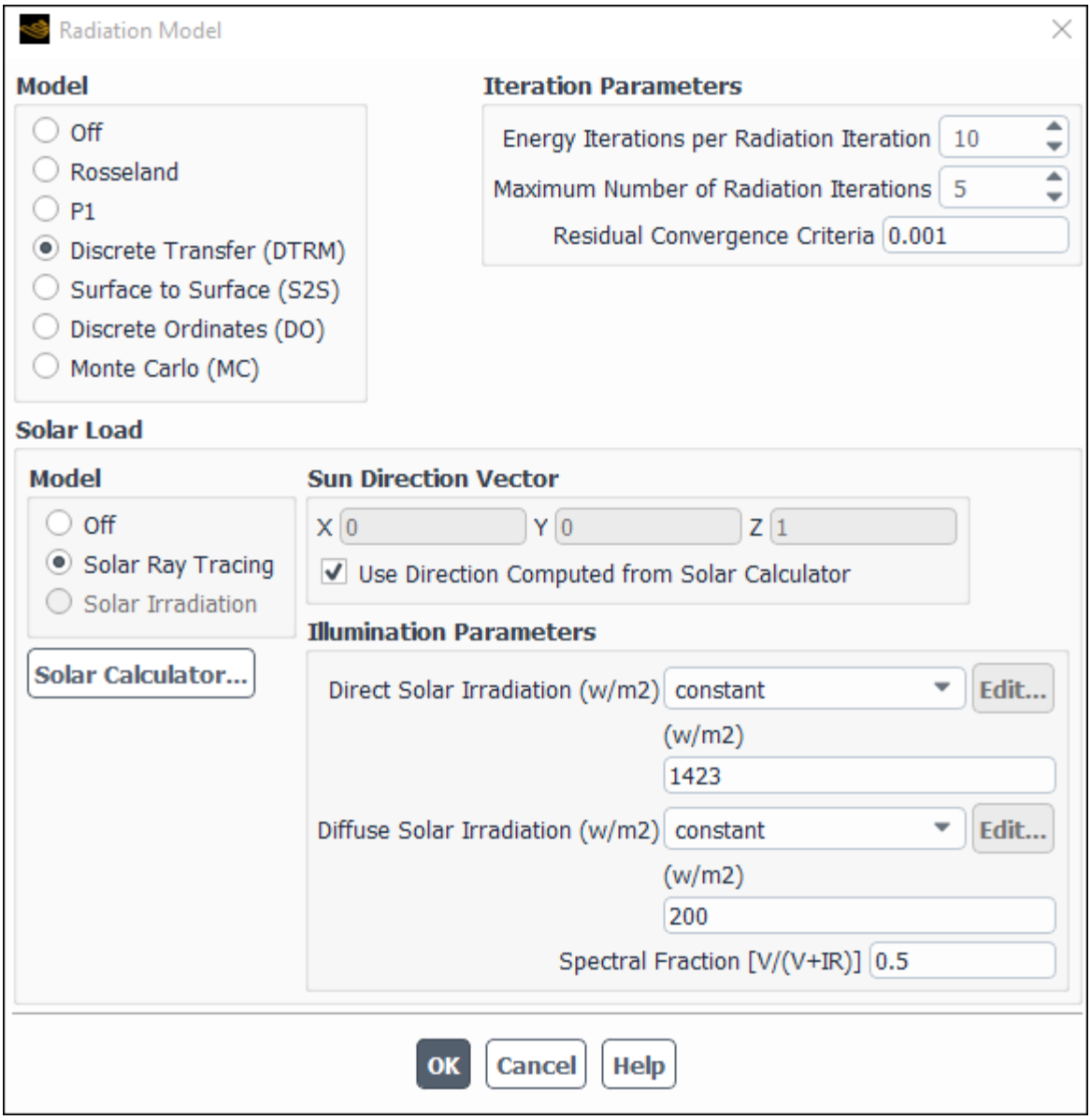
16.3.9.2. DTRM Solution Parameters

You can control the frequency that the radiation field is updated as the continuous phase solution proceeds for steady and transient simulations. For details, see Iteration Parameters For Radiation Solve Frequency (p. 2240).

When the DTRM is active, Ansys Fluent updates the radiation field during the calculation and computes the resulting energy sources and heat fluxes via the ray-tracing technique described in Ray Tracing in the Theory Guide. Ansys Fluent provides several solution parameters in the expanded portion of the **Radiation Model** dialog box (Figure 16.42: The Radiation Model Dialog Box (DTRM) (p. 2238)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.42: The Radiation Model Dialog Box (DTRM)



You can control the maximum number of iterations of the radiation calculation during each global iteration by changing the **Maximum Number of Radiation Iterations**. The default setting of 5 means that the radiosity will be updated up to 5 times. The actual number of iterations will be less if the residual convergence criterion is exceeded at any point during these iterations.

The **Residual Convergence Criteria** parameter (0.001 by default) determines when the radiation intensity update is converged. It is defined as the maximum normalized change in the surface intensity from one DTRM radiation iteration to the next (see Equation 16.4 (p. 2242)).

16.3.9.3. S2S Solution Parameters

You can control the frequency that the radiation field is updated as the continuous phase solution proceeds for steady and transient simulations. For details, see Iteration Parameters For Radiation Solve Frequency (p. 2240).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can control the maximum number of iterations of the radiation calculation during each global iteration by changing the **Maximum Number of Radiation Iterations**. The default setting of 5 means that the radiosity will be updated up to 5 times. The actual number of iterations will be less if the residual convergence criterion is exceeded at any point during these iterations.

The **Residual Convergence Criteria** (0.001 by default) determines when the radiosity update is converged. It is defined as the maximum normalized change in the radiosity from one S2S radiation iteration to the next (see Equation 16.5 (p. 2242)).

16.3.9.4. DO Solution Parameters

You can control the frequency that the radiation field is updated as the continuous phase solution proceeds for steady and transient simulations. For details, see Iteration Parameters For Radiation Solve Frequency (p. 2240).

For most problems, the default under-relaxation of 1.0 for the DO equations is adequate. For problems with large optical thicknesses ($aL>10$), you may experience slow convergence or solution oscillation. For such cases, under-relaxing the energy and DO equations is useful. Under-relaxation factors between 0.9 and 1.0 are recommended for both equations.

16.3.9.5. MC Solution Parameters

You can control the frequency that the radiation field is updated as the continuous phase solution proceeds for steady and transient simulations. For details, see Iteration Parameters For Radiation Solve Frequency (p. 2240).

`Wall Irradiation Flux.Normalized Std Deviation` and `Radiation Intensity.Normalized Std Deviation` (both available during postprocessing) can give an indication of the quality of the solution. They arise from the stochastic nature of the Monte Carlo model.

The Monte Carlo radiation solver computes the standard deviation error based on Poisson statistics. The user-specified number of histories is divided into several groups. Histories are selected from each group and their physical interactions (emission, absorption, reflection, and so on) are tracked through the domain. At the end of the calculation, each group provides values for the quantities of interest, such as irradiation heat flux or absorbed radiation. The mean value, and standard deviation, of each quantity of interest are computed from the groups. A normalized standard deviation is computed by dividing the standard deviation by the mean value.

- `Wall Irradiation Flux.Normalized Std Deviation` is the maximum normalized standard deviation of the irradiation flux at an element face on a boundary. The presence of small isolated boundary regions with values of `Wall Irradiation Flux.Normalized Std Deviation` larger than 30% is an indication that the element faces in those regions were insufficiently sampled. Increasing the **Target Number of Histories** may help address this, at the expense of computational effort.

- `Radiation Intensity.Normalized Std Deviation` is the maximum normalized standard deviation of the radiation intensity within an element. The value of `Radiation Intensity.Normalized Std Deviation` is expected to be less than 30% within a reliable radiation field solution.

Because of the stochastic behavior, it can be difficult to judge the convergence of MC radiation cases based on residuals alone, particularly if the standard deviations are large. It is recommended

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

that you create a temperature probe (for example, a **Surface Report Definition**) and monitor its stability during the solution. For more information, see Creating Report Definitions (p. 4098).

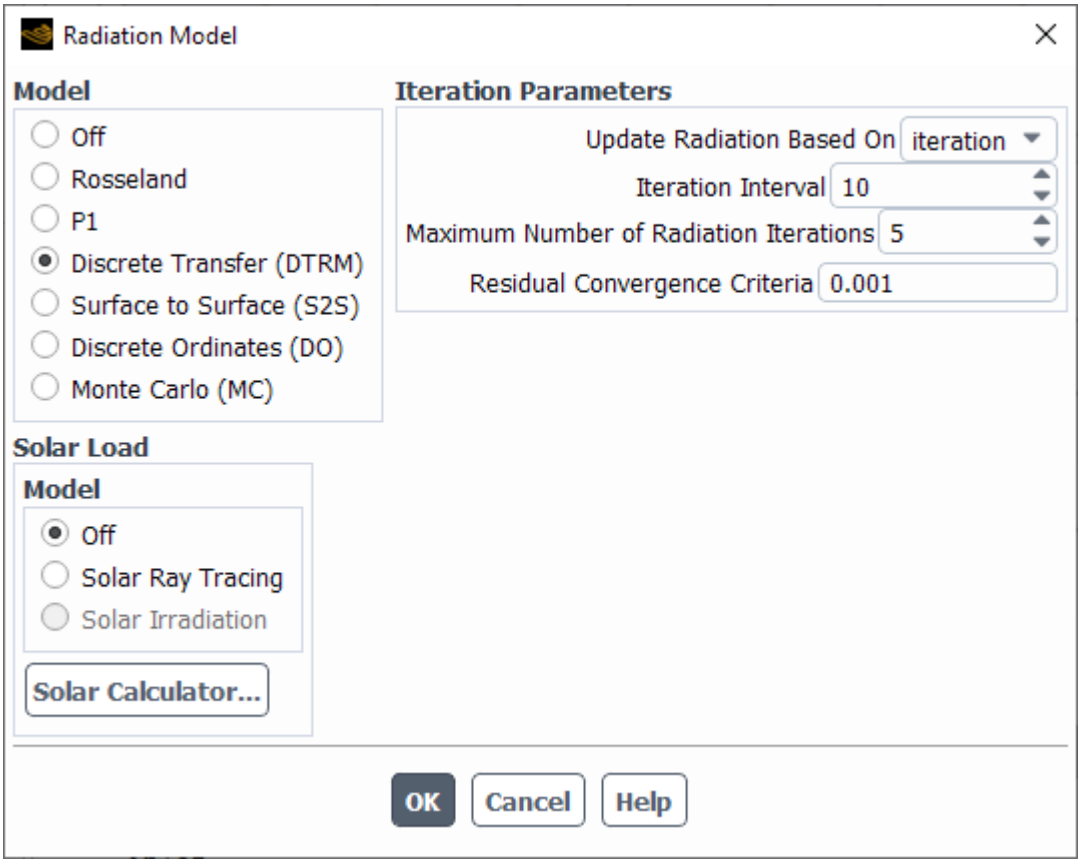
16.3.9.6. Iteration Parameters For Radiation Solve Frequency

For the DTRM, S2S, DO, and MC models, you can control the frequency that the radiation field is updated as the continuous phase solution proceeds. The options available to control the radiation solve frequency depends on whether the simulation is steady or transient.

Steady Simulations

Under **Iteration Parameters**, the option **Update Radiation Based On** can only be set to `iteration` with the default **Iteration Interval** set to 10. This means that the radiation calculation is performed once every 10 iterations of the solution process. Increasing the number can speed the calculation process, but may slow overall convergence.

Figure 16.43: Iteration Parameters for Steady Simulations



Note:

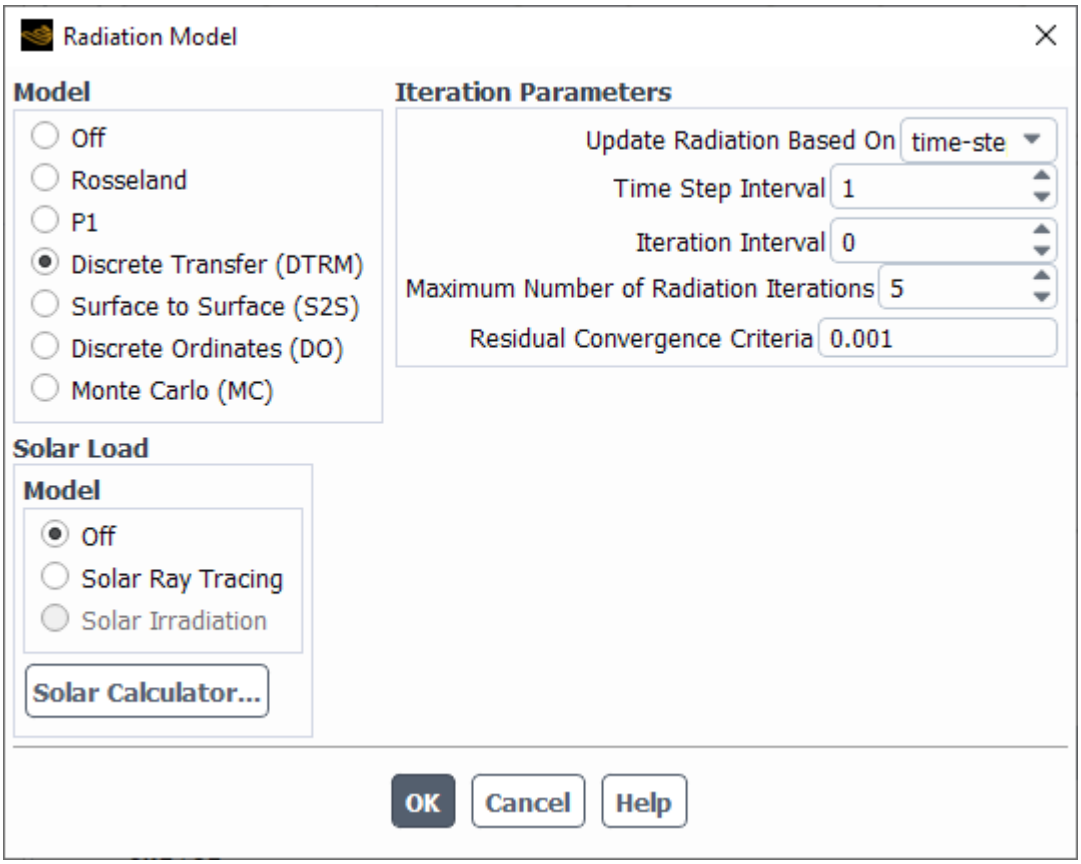
If you are using the pressure-based solver with the S2S model and you first solve the flow equations with the energy equation turned off, you should reduce the **Iteration Interval** from 10 to 1 or 2. This will ensure the convergence of the radiosity. If the default value of 10 is kept in this case, it is possible that the flow and energy residuals may converge and the solution will terminate before the radiosity is converged.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Transient Simulations

Under **Iteration Parameters**, the option **Update Radiation Based On** can be set to `time-step` or `flow-time`.

Figure 16.44: Iteration Parameters for Transient Simulations



For updating radiation based on `time-step`, you must specify the **Time Step Interval** and **Iteration Interval**. The default value of 1 for **Time Step Interval** means the radiation field will be updated every time step. The default **Iteration Interval** of 0 means the radiation field is updated after the first iteration in a time step. For example, a **Time Step Interval** of 2 means the radiation field is updated every other time step. An **Iteration Interval** of 5 means the radiation field is updated every 5 iterations within a timestep.

For updating radiation based on `flow-time`, you must specify the **Time Interval** in seconds and **Iteration Interval**. The default value of 1 second for **Time Interval** means the radiation field will be updated once for every 1 second of flow time. The default **Iteration Interval** of 0 means the radiation field is only updated to satisfy the **Time Interval** condition. For example, a **Time Interval** of 2 seconds means the radiation field is updated after every 2 seconds of flow time. An **Iteration Interval** of 5 means the radiation field is updated every 5 iterations within a timestep.

16.3.9.7. Running the Calculation

Once the radiation problem has been set up, you can proceed as usual with the calculation. Note that while the P-1 and DO models will solve additional transport equations and report residuals, the DTRM and the Rosseland and S2S models will not (since they impact the solution only through

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the energy equation). Residuals for the DTRM and S2S model radiation iterations are reported by Ansys Fluent every time such an iteration is performed, as described below.

When running the S2S model with the parallel solver, note that coupled walls are encapsulated, which may cause problems with the partitioning of the mesh. See Troubleshooting (p. 4342) for recommendations on correcting such problems.

16.3.9.7.1. Residual Reporting for the P-1 Model

The residual for radiation as calculated by the P-1 model is updated after each iteration and reported with the residuals for all other variables. Ansys Fluent reports the normalized P-1 radiation residual as defined in Monitoring Residuals (p. 3684) for the other transport equations.

16.3.9.7.2. Residual Reporting for the DO Model

After each DO iteration, the DO model reports a composite normalized residual for all the DO transport equations. The definition of the residuals is similar to that for the other transport equations (see Monitoring Residuals (p. 3684)).

16.3.9.7.3. Residual Reporting for the DTRM

Ansys Fluent does not include a DTRM residual in its usual residual report that is issued after each iteration. The effect of radiation on the solution can be gathered, instead, via its impact on the energy field and the energy residual. However, each time a DTRM iteration is performed, Ansys Fluent will print out the normalized radiation error for each DTRM radiation iteration. The normalized radiation error is defined as

$$E = \frac{\sum_{all\ radiating\ surfaces} \left(I_{new} - I_{old}\right)}{N\left(\sigma T^4 / \pi\right)} \tag{16.4}$$

where the error E is the maximum change in the intensity (I) at the current radiation iteration, normalized by the maximum surface emissive power, and N is the total number of radiating surfaces. Note that the default radiation convergence criterion, as noted in DTRM Solution Parameters (p. 2237), defines the radiation calculation to be converged when E decreases to 10^{-3} or less.

16.3.9.7.4. Residual Reporting for the S2S Model

Ansys Fluent does not include an S2S residual in its usual residual report that is issued after each iteration. The effect of radiation on the solution can be gathered, instead, via its impact on the energy field and the energy residual. However, each time an S2S iteration is performed, Ansys Fluent will print out the normalized radiation error for each S2S radiation iteration. The normalized radiation error is defined as

$$E = \frac{\sum_{all\ radiating\ surface\ clusters} \left(J_{new} - J_{old}\right)}{N\sigma T^4} \tag{16.5}$$

where the error E is the maximum change in the radiosity (J) at the current radiation iteration, normalized by the maximum surface emissive power, and N is the total number of radiating surface clusters. Note that the default radiation convergence criterion, as noted in DTRM Solution

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parameters (p. 2237), defines the radiation calculation to be converged when E decreases to 10^{-3} or less.

16.3.9.7.5. Disabling the Update of the Radiation Fluxes

Sometimes, you may want to set up your Ansys Fluent model with the radiation model active and then disable the radiation calculation during the initial calculation phase. For the P-1 and DO models, you can turn off the radiation calculation temporarily by deselecting **P1** or **Discrete Ordinates** in the **Equations** dialog box, which is accessed by right-clicking **Solution Controls** in the tree (under **Solution**) and clicking **Equations...** in the menu that opens. For the DTRM and the S2S model, there is no radiation calculation to disable in the **Equations** dialog box. You can instead set a very large number for **Energy Iterations per Radiation Iteration** in the **Iteration Parameters** group box of the **Radiation Model** dialog box.

If you turn off the radiation calculation, Ansys Fluent will skip the update of the radiation field during subsequent iterations, but will leave in place the influence of the current radiation field on energy sources due to absorption, wall heat fluxes, and so on. Turning the radiation calculation off in this way can therefore be used to initiate your modeling work with the radiation model inactive and/or to focus the computational effort on the other equations if the radiation model is relatively well converged.

16.3.10. Postprocessing Radiation Quantities

Information regarding postprocessing radiation quantities can be found in the following sections:

16.3.10.1. Available Variables for Postprocessing

16.3.10.2. Reporting Radiative Heat Transfer Through Boundaries

16.3.10.3. Overall Heat Balances When Using the DTRM

16.3.10.4. Displaying Rays and Clusters for the DTRM

16.3.10.5. Reporting Radiation in the S2S Model

16.3.10.1. Available Variables for Postprocessing

Ansys Fluent provides radiation quantities that you can use in postprocessing when your model includes the solution of radiative heat transfer. You can generate graphical plots or alphanumeric reports of the following variables/functions:

In the **Radiation...** category:

- **Incident Radiation** (P-1, DO, MC)
- **Incident Radiation (Band n)** (non-gray P-1, non-gray DO, non-gray MC)
- **Absorption Coefficient** (DTRM, P-1, DO, Rosseland)
- **Scattering Coefficient** (P-1, DO, Rosseland)
- **Refractive Index** (P1, DO, Rosseland)
- **Radiation Temperature** (P-1, DO, MC)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Surface Cluster ID** (S2S)
- **Volumetric Absorbed Radiation** (P-1, DO, MC)
- **Volumetric Emitted Radiation** (P-1, DO, MC)
- **Radiation Intensity.Normalized Std Deviation** (MC)
- **Wall Irradiation Flux.Normalized Std Deviation** (MC)

In the **Wall Fluxes...** category:

- **Radiation Heat Flux** (All)
- **Surface Incident Radiation** (S2S, DTRM, DO, MC)
- **Absorbed Radiation Flux** (DO, MC)
- **Reflected Radiation Flux** (DO, MC)
- **Transmitted Radiation Flux** (DO, MC)
- **Beam Irradiation Flux** (DO, MC)

See Field Function Definitions (p. 4167) for definitions of these postprocessing variables. Note that in addition, incident radiation, transmitted, reflected and absorbed radiation flux are also available on a per-band basis for the non-gray DO model.

Important:

- The sign convention on the radiative heat flux is such that the heat flux from the wall surface is a positive quantity.
- It is possible to export heat flux data on wall zones (including radiation) to a generic file that you can examine or use in an external program. See Exporting Heat Flux Data (p. 2181) for details.
- Take care not to confuse **Incident Radiation** and **Surface Incident Radiation**. **Incident Radiation** is a volumetric quantity giving the total radiant load passing through the cell (in all directions), whereas **Surface Incident Radiation** is the total radiant load hitting the surface (which will subsequently be absorbed, transmitted and reflected). There is no direct means to report how much radiation has been absorbed/emitted/scattered in cells.

16.3.10.2. Reporting Radiative Heat Transfer Through Boundaries

You can use the **Flux Reports** dialog box to compute the radiative heat transfer through each boundary of the domain, or to sum the radiative heat transfer through all boundaries.

Results → Reports → Fluxes → Edit...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See Fluxes Through Boundaries (p. 4135) for details about generating flux reports.

16.3.10.3. Overall Heat Balances When Using the DTRM

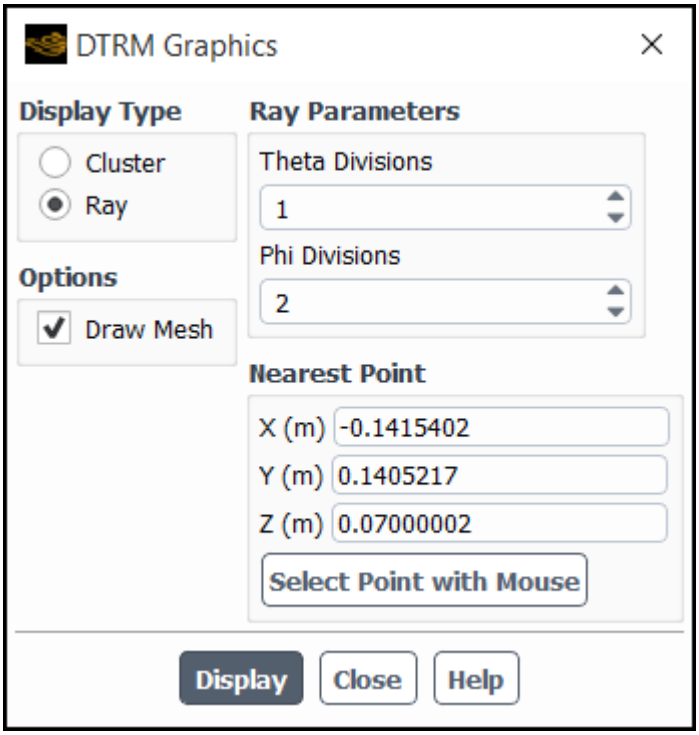
The DTRM yields a global heat balance and a balance of radiant heat fluxes only in the limit of a sufficient number of rays. In any given calculation, therefore, if the number of rays is insufficient you may find that the radiant fluxes do not obey a strict balance. Such imbalances are the inevitable consequence of the discrete ray tracing procedure and can be minimized by selecting a larger number of rays from each wall boundary.

16.3.10.4. Displaying Rays and Clusters for the DTRM

When you use the DTRM, Ansys Fluent allows you to display surface or volume clusters, as well as the rays that emanate from a particular surface cluster. You can use the **DTRM Graphics** dialog box (Figure 16.45: The DTRM Graphics Dialog Box (p. 2245)) for all of these displays.

Results → Model Specific → DTRM Graphics...

Figure 16.45: The DTRM Graphics Dialog Box



16.3.10.4.1. Displaying Clusters

To view clusters, select **Cluster** under **Display Type** and then select either **Surface** or **Volume** under **Cluster Type**.

To display all of the surface or volume clusters, select the **Display All Clusters** option under **Cluster Selection** and click the **Display** button.

To display only the cluster (surface or volume) nearest to a specified point, deselect the **Display All Clusters** option and specify the coordinates under **Nearest Point**. You may also use the

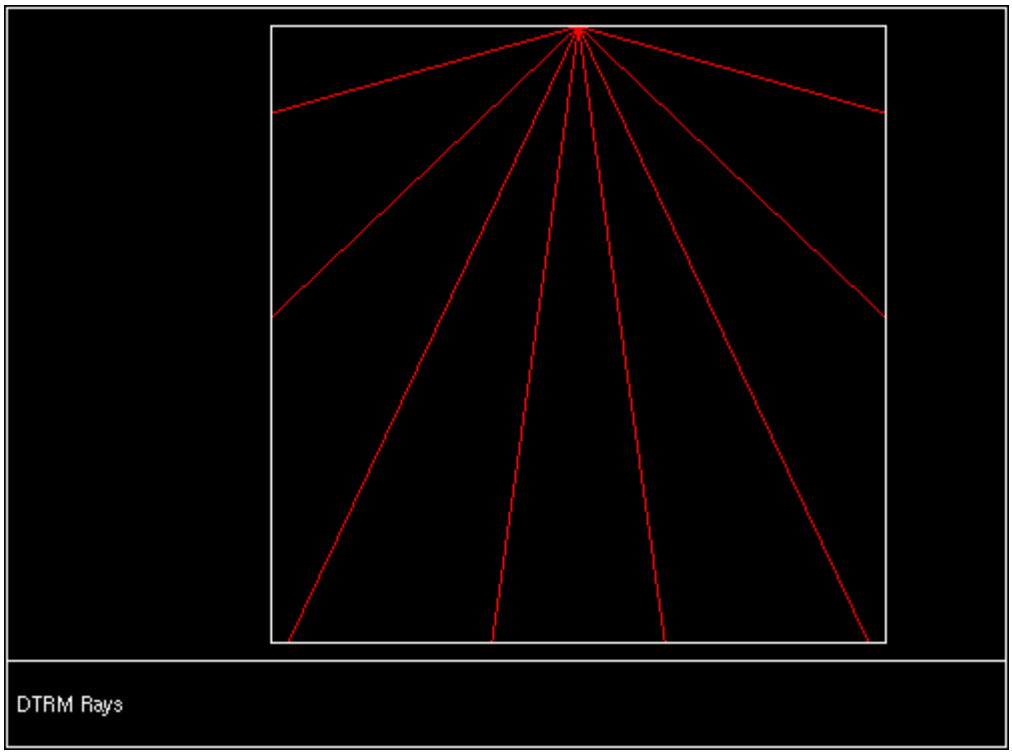
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

mouse to choose the nearest point. Click the **Select Point With Mouse** button and then right-click a point in the graphics window.

16.3.10.4.2. Displaying Rays

To display the rays emanating from the surface cluster nearest to the specified point, select **Ray** under **Display Type**. Set the appropriate values for **Theta** and **Phi Divisions** under **Ray Parameters** (see Setting Up the DTRM (p. 2195) for details), and then click the **Display** button. Figure 16.46: Ray Display (p. 2246) shows a ray plot for a simple 2D geometry.

Figure 16.46: Ray Display



16.3.10.4.3. Including the Mesh in the Display

For some problems, especially complex 3D geometries, you may want to include portions of the mesh in your ray or cluster display as spatial reference points. For example, you may want to show the location of an inlet and an outlet along with displaying the rays. This is accomplished by enabling the **Draw Mesh** option in the **DTRM Graphics** dialog box. The **Mesh Display** dialog box will appear automatically when you enable the **Draw Mesh** option, and you can set the mesh display parameters there. When you click **Display** in the **DTRM Graphics** dialog box, the mesh display, as defined in the **Mesh Display** dialog box, will be included in the ray or cluster display.

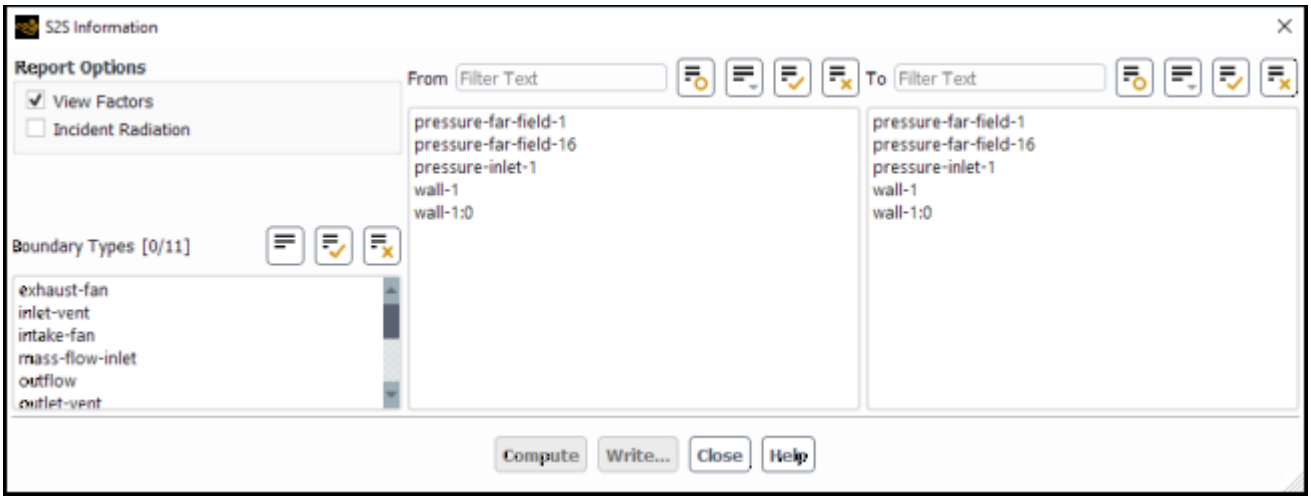
16.3.10.5. Reporting Radiation in the S2S Model

When you use the S2S model, Ansys Fluent allows you to view the values of the view factor and radiation emitted from one zone to any other zone. You can use the **S2S Information** dialog box (Figure 16.47: The S2S Information Dialog Box (p. 2247)) to generate a report of these values in the console or as a separate file.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Results → Model Specific → S2S Information...

Figure 16.47: The S2S Information Dialog Box



The steps for generating the report are as follows:

1. Specify the values in which you are interested by selecting **View Factors** and/or **Incident Radiation**.

2. Choose the zones for which you would like data by selecting them in the lists under **From** and **To** (at least one zone must be selected under each list). To select all of the zones of a particular type, click that category in the list under **Boundary Types**.

3. Specify how you would like to present the data. To report the values in the console, click the **Compute** button. To write the data as an S2S Info File (`.sif` format), click the **Write...** button and enter a file name in The Select File Dialog Box (p. 931).

The following is an example of how the data is presented:

```
S2S Information
From wall1 to:
                    View factor       Incident Radiation
          wall1          0.0000                  0.0000
          wall2          0.2929             171387.7813
          wall3          0.2929             155305.7969
          wall4          0.4142              29055.9023

From wall2 to:
                    View factor       Incident Radiation
          wall1          0.2929             306451.9688
          wall2          0.0000                  0.0000
          wall3          0.4142             214195.0938
          wall4          0.2929              19153.2715
```

Note that the header listed above (`S2S Information`) is not displayed in the console.

16.3.11. Solar Load Model

Ansys Fluent provides a solar load model that can be used to calculate radiation effects from the sun's rays that enter a computational domain. Two options are available for the model: **Solar Ray Tracing**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and **Solar Irradiation**. The Solar Ray Tracing approach is a highly efficient and practical means of applying solar loads as heat sources in the energy equations. In cases where you want to use the DO or MC model to calculate radiation effects within the domain, the Solar Irradiation option is available to supply outside beam direction and intensity parameters directly to the radiation model. The solar load model includes a solar calculator utility that can be used to construct the sun's location in the sky for a given time-of-day, date, and position. Solar load is available in the 3D solver only, and can be used to model steady and unsteady flows.

16.3.11.1. Introduction

Typical applications that are well-suited for solar load simulations include the following:

- automotive climate control (ACC) applications
- human comfort modeling applications in buildings

The effects of solar loading are needed in many ACC applications, where the temperature, humidity, and velocity fields around passengers (and drivers) are desired. ACC systems are tested for their capacity to cool down passenger compartments after they have been "soaked" in intense solar radiation. Ansys Fluent's solar load model will enable you to simulate solar loading effects and predict the time it will take to reasonably cool down the cabin of a car that has been exposed to solar radiation, as well as predict the time interval needed to lower the temperature in specified points and areas within the domain.

In the analysis of buildings, solar loading provides a significant burden on the cooling requirement in warm climates, particularly where architects want to use the aesthetics of glazed facades. Even in cooler climates, solar loading can provide a burden during warmer seasons where modern buildings are well insulated against thermal loss during winter months. As well as providing an engineer with a practical tool for determining the solar heating effect inside a building, Ansys Fluent's solar load model will allow the solar transmission through all glazed surfaces to be determined over the course of a day, allowing important decisions to be made before undertaking any flow studies. Ansys Fluent's solar load model also allows you to simulate porous blinds, which can transmit a portion of the solar radiation while also allowing fluid flow.

16.3.11.2. Solar Ray Tracing

The solar load model's ray tracing algorithm can be used to predict the direct illumination energy source that results from incident solar radiation. It takes a beam that is modeled using the sun position vector and illumination parameters, applies it to any or all wall, porous jump, and inlet/outlet boundary zones that you specify, performs a face-by-face shading analysis to determine well-defined shadows on all boundary faces and interior walls, and computes the heat flux on the boundary faces that results from the incident radiation.

Important:

The solar ray tracing model includes only boundary zones that are adjacent to fluid zones in the ray tracing calculation. Therefore, boundary zones that are attached to solid zones are treated as transparent and are not included in the ray tracing algorithm.

The resulting heat flux that is computed by the solar ray tracing algorithm is coupled to the Ansys Fluent calculation via a source term in the energy equation. The heat sources are added directly to

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

computational cells bordering each face and are assigned to adjacent cells in the following order: shell conduction cells, solid cells, and fluid cells. Heat sources are assigned to one of these types of adjacent cells, only. You can choose to override this order and include adjacent fluid cells in the solar load calculation by issuing a command in the text user interface (see Text Interface-Only Commands (p. 2268) for details). Note that the sun position vector and solar intensity can be entered either directly by you or computed from the solar calculator. Direct and diffuse irradiation parameters can also be specified using a user-defined function (UDF) and hooked to Ansys Fluent in the **Radiation Model** dialog box.

The solar ray tracing option allows you to include the effects of direct solar illumination as well as diffuse solar radiation in your Ansys Fluent model. A two-band spectral model is used for direct solar illumination and accounts for separate material properties in the visible and infrared bands. A single-band hemispherical-averaged spectral model is used for diffuse radiation. Opaque materials are characterized in terms of two-band absorptivities. A semi-transparent material requires specification of absorptivity and transmissivity. Values that you specify for transmissivity and absorptivity are defined for normal incident rays. Ansys Fluent recomputes/interpolates these values for the given angle of incidence.

The solar ray tracing algorithm also accounts for internal scattered and diffusive loading. The reflected component of direct solar irradiation is tracked. A fraction of this radiative heat flux, called internally scattered energy is applied to all the surfaces participating in the solar load calculation, weighted by area. The internally scattered energy depends on the scattering fraction that is specified in the TUI, whose default value is 1. Depending on the reflectivity of the primary surface, the scattering fraction can be responsible for the inclusion (or exclusion) of a large amount of radiation within the rest of the domain.

Also included as internally scattered energy is the contribution of the transmitted component of diffuse solar irradiation (which enters a domain through semi-transparent walls depending upon the hemispherical transmissivity). The total value of internally scattered energy is reported to the Ansys Fluent console. The ambient flux is obtained by dividing the internally scattered energy by the total surface area of the faces participating in the solar load calculation.

Note that **Solar Ray Tracing** is *not* a participating radiation model. It does not deal with emission from surfaces, and the reflecting component of the primary incident load is distributed uniformly across all surfaces rather than being local to the surfaces reflected to. If surface emission is an important factor in your case then you can consider implementing a radiation model (for example, P-1) in conjunction with **Solar Ray Tracing**.

Note the following limitation when using the solar ray tracing model:

- Non-conformal interfaces are not supported with **Solar Ray Tracing**.
- If solving in parallel, Fluent computes **Solar Ray Tracing** on only the host process although the calculation does employ multiple threads. Therefore, it is important to ensure the host process has sufficient available RAM for the computation.

16.3.11.2.1. Shading Algorithm

The shading calculation that is used for solar ray tracing is a straightforward application of vector geometry. A ray is traced from the centroid of a test face in the direction of the sun. Every other face is checked to determine if the ray intersects the candidate face and if the candidate face is in front of the test face. If both conditions are met, then an opaque face completely shades the test face. A semi-transparent face attenuates the incident energy.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A Barycentric coordinate formulation is used to construct triangle-ray intersections. A quadrilateral ray intersection method is used to handle the case when model surfaces contain quadrilaterals. A quad-tree preprocessing step is applied to reduce the ray tracing algorithm complexity that can lead to long run times for 10^4 faces and greater. The quad-tree refinement factor can be modified in the text interface. The default value of this parameter is 7, which is sufficient to cover the entire spectrum of mesh sizes between one cell and five million cells. If the mesh is greater than five million cells, an increase in this parameter would reduce the CPU time needed to compute the solar loads.

16.3.11.2.2. Glazing Materials

Incident solar radiation can be applied to glass and plastic glazing materials of various types at wall boundaries, and the effects of coated glazings modeled using the solar ray tracing algorithm. To model solar optical properties, you will need to specify the transmissivity and reflectivity of the material in the **Wall** boundary conditions dialog box. You can obtain these values from the glass (or plastic) manufacturer or use data from another source (for example, ASHRAE Handbook).

Glazing optical properties are dependent on incident angle, and the variation is significant for an incident angle greater than 40 degrees. As the incident angle increases from zero, transmissivity decreases, reflectivity increases, and absorptivity increases initially due to lengthened optical path, and then decreases as more incident radiation is reflected. The shape of the property curve varies with glass type and thickness. This difference is more pronounced for coated glass or for a multiple-pane glazing system. It cannot be assumed that all glazing systems have a universal angular dependence.

For coated glazings, the spectral transmissivity and reflectivity at any incident angle are approximated in the solar load model from the normal angle of incidence [45] (p. 5715).

Transmissivity is given by

$$T(\theta,\lambda)=T(0,\lambda)Tref(\theta) \tag{16.6}$$

where

$$Tref(\theta)=a0+a1cos(\theta)+a2(cos(\theta))^2+a3(cos(\theta))^3+a4(cos(\theta))^4 \tag{16.7}$$

Reflectivity is given by

$$R(\theta,\lambda)=R(0,\lambda)[1-Rref(\theta)]+Rref(\theta) \tag{16.8}$$

where

$$Rref(\theta)=b0+b1cos(\theta)+b2(cos(\theta))^2+b3(cos(\theta))^3+b4(cos(\theta))^4-Tref(\theta) \tag{16.9}$$

The constants used in Equation 16.6 (p. 2250) and Equation 16.8 (p. 2250) are for coated glazings and are taken from Finlayson and Arasteh. [45] (p. 5715). The normal transmissivity and reflectivity, $T(0,\lambda)$ and $R(0,\lambda)$ are specified in the **Wall** boundary conditions dialog box.

16.3.11.2.3. Inputs

The following inputs are required for the solar ray tracing algorithm:

- sun direction vector

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- direct solar irradiation
- diffuse solar irradiation
- spectral fraction
- direct and IR absorptivity (opaque wall)
- direct and IR absorptivity and transmissivity (semi-transparent wall and porous jump)
- diffuse hemispherical absorptivity and transmissivity (semi-transparent wall)
- solar transmissivity factor
- quad tree refinement factor
- scattering fraction
- ground reflectivity

The sun direction vector is the direction vector looking to the sun, from which the direct irradiation will be incident. You can enter the vector components (**X**, **Y**, **Z**) and the direct and diffuse solar irradiation fluxes in the **Radiation Model** dialog box, or you can have these parameters derived from the solar calculator. These irradiation fluxes can also be specified using a user-defined function (User-Defined Functions (UDFs) for Solar Load (p. 2256)). The spectral fraction is the final input in the **Radiation Model** dialog box. This defines the split of visible and infra-red (shortwave and longwave respectively) radiation, specifically the fraction of the direct irradiation flux that is in the visible band. These quantities can also be defined through the text interface.

The scattering fraction defines the amount of non-absorbed radiation that will be distributed (uniformly) across all participating surfaces. This is required as the solar load model does not track the rays beyond the first opaque surface. Therefore, a highly glazed space where incident radiation is likely to be reflected back out will have a low value. Conversely, a predominantly opaque (wall-bounded) space where reflected radiation is likely to be incident upon (and ultimately absorbed by) other opaque surfaces will have a high value. This parameter is defined through the text interface only, taking a default value of 1.0:

`define` → `models` → `radiation` → `solar-parameters` → `scattering-fraction`

The ground reflectivity is used by the solar calculator to compute the background diffuse radiation intensity component contributed to by radiation reflected off the ground. This should be based on typical figures for the surface reflectivity of the outside ground surfaces. By default this is set to 0.2, but can be adjusted through the text-interface:

`define` → `models` → `radiation` → `solar-parameters` → `ground-reflectivity`

The quad-tree-refinement parameter determines the level of detail used by the shading algorithm. By default this is set to 7 which will generally work well, but can lie between 0 and 10. This is defined only through the text interface:

`define` → `models` → `radiation` → `solar-parameters` → `quad-tree-refinement`

Further details on the text interface-only entries is provided later in this section (see Text Interface-Only Commands).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The absorptivity and transmissivity parameters related to a wall or porous jump are entered in the **Wall** (under the **Radiation** tab) or **Porous Jump** boundary condition dialog box, respectively, for the particular boundary zones you want to participate in solar ray tracing. On flow boundaries you have a solar transmissivity factor to allow you to attenuate the incoming solar flux, for example, set to 1 for a fully open inlet or set to 0 for a light obscuring louvered inlet.

16.3.11.3. Solar Irradiation

The solar load model's Solar Irradiation option provides you with an easy means of applying a solar load directly to the DO or MC model. Unlike the ray tracing solar load option, the Solar Irradiation method does not compute heat fluxes and apply them as heat sources to the energy equation. Instead, the irradiation flux is applied directly to semi-transparent walls (which you specify) as a boundary condition, and the radiative heat transfer is derived from the solution of the radiation model itself.

The following inputs are required for Solar Irradiation at semi-transparent walls:

- direct irradiation
- diffuse irradiation
- beam direction
- beam width (only required for DO model)
- diffuse fraction

In the **Wall** boundary condition dialog box for each semi-transparent wall you want to participate in Solar Irradiation, you can specify that the beam direction, direct irradiation, and diffuse irradiation be derived from the solar parameters (for example, solar calculator) that you set (or compute) in the **Radiation Model** dialog box. This is done by checking the **Use Beam Direction from Solar Load Model Settings** and **Use Irradiation from Solar Load Model Settings** boxes. When selected, Ansys Fluent sets the beam width (the angle subtended by the sun) to the default value of 0.53 degrees when using the DO model.

Important:

Note that the sign of the beam direction that is needed for the DO or MC model is opposite the sun direction vector that is entered or derived from the solar parameters. The beam direction in the DO or MC model is the direction of external radiation (for example, radiation coming from the sun), while the sun direction vector in the solar load model points to the sun.

16.3.11.4. Solar Calculator

Ansys Fluent provides a solar calculator that can be used to compute solar beam direction and irradiation for a given time, date, and position. These values can be used as inputs to the solar ray tracing algorithm or as semi-transparent wall boundary conditions for solar irradiation.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16.3.11.4.1. Inputs/Outputs

Inputs needed for the solar calculator are:

- global position (latitude, longitude, time zone)
- starting date and time
- mesh orientation
- solar irradiation method
- sunshine factor

Global position consists of latitude, longitude, and time zone (relative to GMT). The time of day for a transient simulation is the starting time plus the flow-time. For mesh orientation, you will need to specify the North and East direction vector in the CFD mesh. The default solar irradiation method is Fair Weather Conditions. Alternatively, you can choose the Theoretical Maximum method. The sunshine factor is simply a linear reduction factor for the computed incident load that allows for cloud cover to be accounted for, if appropriate.

You can specify these inputs in the **Solar Calculator** dialog box that is accessible from the **Radiation Model** dialog box (Figure 16.51: The Solar Calculator Dialog Box (p. 2260)). Alternatively, you can enter the parameters using text interface commands (Additional Text Interface Commands (p. 2270)).

The following values are computed by the solar calculator and are displayed in the console whenever the solar calculator is used:

- sun direction vector
- sunshine fraction
- direct normal solar irradiation at earth's surface
- diffuse solar irradiation - vertical and horizontal surface
- ground reflected (diffuse) solar irradiation - vertical surface

Direct normal solar irradiation is computed using the ASHRAE Fair Weather Conditions method, when this option is selected in the solar calculator. Note: Equation 20 and Table 7 from Chapter 30 of the 2001 ASHRAE Handbook of Fundamentals. The theoretical maximum values for direct normal solar irradiation and diffuse solar irradiation are computed using NREL's Theoretical Maximum method, when this option is selected. In practice, these values are unlikely to be experienced due to atmospheric conditions.

Ansys Fluent computes the diffuse solar irradiation components (vertical and horizontal) internally for each face in the domain. When the Theoretical Maximum method is chosen, these diffuse irradiation values provide estimates for the maximum vertical and horizontal surface effects.

16.3.11.4.2. Theory

Ansys Fluent provides two options for computing the solar load: Fair Weather Conditions method and Theoretical Maximum method. Although these methods are similar, there is a key difference.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Fair Weather Conditions method imposes greater attenuation on the solar load, which is representative of atmospheric conditions that are fair–but not completely clear.

The equation for normal direct irradiation applying the Fair Weather Conditions Method is taken from the ASHRAE Handbook:

$$Edn=\frac{A}{e^{\frac{B}{\sin(\beta)}}} \tag{16.10}$$

where A and B are apparent solar irradiation at air mass $m=0$ and atmospheric extinction coefficient, respectively. These values are based on the earth's surface on a clear day. β is the solar altitude (in degrees) above the horizontal.

The equation for direct normal irradiation that is used for the Theoretical Maximum Method is taken from NREL's Solar Position and Intensity Code (Solpos):

$$Edn=S_{etrn}S_{unprime} \tag{16.11}$$

where S_{etrn} is the top of the atmosphere direct normal solar irradiance and $S_{unprime}$ is the correction factor used to account for reduction in solar load through the atmosphere.

The calculation for the diffuse load in the solar model is based on the approach suggested in the 2001 ASHRAE Fundamental Handbook (Chapter 20, Fenestration). The equation for diffuse solar irradiation on a vertical surface is given by:

$$Ed=CYEdn \tag{16.12}$$

where C is a constant whose values are given in Table 7 from Chapter 30 of the 2001 ASHRAE Handbook of Fundamentals, Y is the ratio of sky diffuse radiation on a vertical surface to that on a horizontal surface (calculated as a function of incident angle), and Edn is the direct normal irradiation at the earth's surface on a clear day.

The equation for diffuse solar irradiation for surfaces other than vertical surfaces is given by:

$$Ed=CEdn\frac{\left(1+\cos\varepsilon\right)}{2} \tag{16.13}$$

where ε is the tilt angle of the surface (in degrees) from the horizontal plane.

The equation for ground reflected solar irradiation on a surface is given by:

$$Er=Edn\left(C+\sin\beta\right)\rho_g\frac{\left(1-\cos\varepsilon\right)}{2} \tag{16.14}$$

where ρ_g is the ground reflectivity. The total diffuse irradiation on a given surface will be the sum of Ed and Er when the input for diffuse solar radiation is taken from the solar calculator. Otherwise, if the `constant` option is selected in the **Radiation** dialog box, then the total diffuse irradiation will be the same as specified in the dialog box.

16.3.11.4.3. Computation of Load Distribution

In calculating the solar load that will be incident on each surface, it is necessary to distinguish between the calculation of diffuse and direct solar loads. A direct load will be tracked from participating transmissive boundary surfaces and non-participating boundary surfaces, the former provides some opportunity to attenuate the incoming flux by absorption and reflection, while the non-participating surfaces allow the flux to enter without any drop in intensity. The direct

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

load is then tracked through the model space until it is incident on an opaque surface, or it exists through a transmissive or non-participating boundary zone. During its passage, its intensity will be attenuated as it passes through participating semi-transparent internal walls, where some radiation may be absorbed and some may be reflected. The total amount of direct radiation that is reflected at internally facing surfaces will be added to the scattered radiation budget for further use later.

The diffuse load originates at participating transmissive boundary surfaces. It is these surfaces that permit diffuse radiation to enter, irrespective of their orientation relative to the direction vector. For each transmissive surface, some of the incoming diffuse load may be immediately absorbed and/or reflected to the outside. The rest is assumed to be transmitted inside and summed from all of these surfaces to give an initial diffuse budget. Onto this budget is added a fraction of the previously computed scattered radiation from the direct load, the fraction used is defined as an input to the model. This provides the total diffuse load. This is then uniformly distributed across all surfaces that are participating in the solar calculation, irrespective of whether they are opaque or semi-transparent. There is no scope to define local absorptivity for this distribution and no biasing with regards proximity to transmissive surfaces. Note that a non-participating boundary zone will allow direct load to enter the model space but will not provide an incoming quantity of diffuse load.

Note that the solar flux that is externally incident on an opaque surface will be completely disregarded, for example solar load on an opaque roof of a model whose internals only are modeled will not be included as a heat gain. Instead, this heat gain should be manually calculated and applied as a thermal condition, typically using a fixed heat flux or a radiation/mixed condition.

16.3.11.5. Using the Solar Load Model

When you want to run a steady-state solution with solar load enabled, you simply set up the solar load model (Setting Up the Solar Load Model (p. 2256)) and boundary conditions (Setting Boundary Conditions for Solar Loading (p. 2261)) for your case, and then run the simulation. The solution data file will contain the solar fluxes that you can use for postprocessing. For a steady-state solution, the solar loads are computed on initialization. If you want to initially solve a case without solar loading (say, for stability) and then add the effects of solar loading afterward, you will need to enable the solar load model through the text user interface (TUI).

Important:

Note that you can compute the solar load at any time once you have set up the model by using the `sol-on-demand` text interface command (see Additional Text Interface Commands (p. 2270) for details).

When you want to run a transient solar load simulation, the process is the same as for the steady-state case but you will need to specify the additional **Time Steps per Solar Load Update** parameter in the **Radiation Model** dialog box. Ansys Fluent will re-compute the sun position and irradiation and update solar loads with this specified frequency.

Important:

Note that parallel simulations involving the solar load model are set up and computed using the same steps as in serial.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16.3.11.5.1. User-Defined Functions (UDFs) for Solar Load

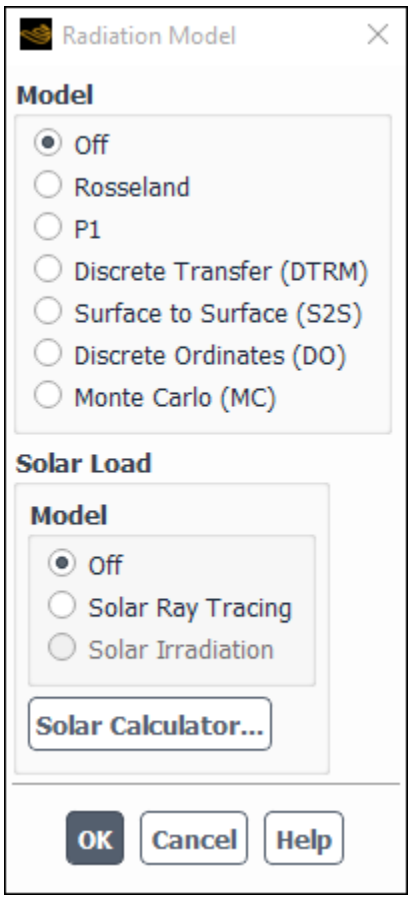
You can write a user-defined function (UDF) to specify direct and diffuse solar intensity using the `DEFINE_SOLAR_INTENSITY` macro. See `DEFINE_SOLAR_INTENSITY` for more information. After it is interpreted or compiled, you can hook your intensity UDF for direct or diffuse solar irradiation by selecting `user-defined` in the drop-down lists for these parameters in the **Radiation Model** dialog box. See Step 2 in Setting Up the Solar Load Model (p. 2256) for details.

16.3.11.5.2. Setting Up the Solar Load Model

The solar load model is enabled in the **Radiation Model** dialog box (Figure 16.48: The Radiation Model Dialog Box (p. 2256)).

Setup → Models → Radiation Edit...

Figure 16.48: The Radiation Model Dialog Box



Important:

Solar load is available in the 3D solver only, and can be used to model steady and unsteady flows.

The solar load model has two options: **Solar Ray Tracing** and **Solar Irradiation**. **Solar Ray Tracing** can be applied as a stand-alone solar loading model, or it can be used in conjunction with one

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

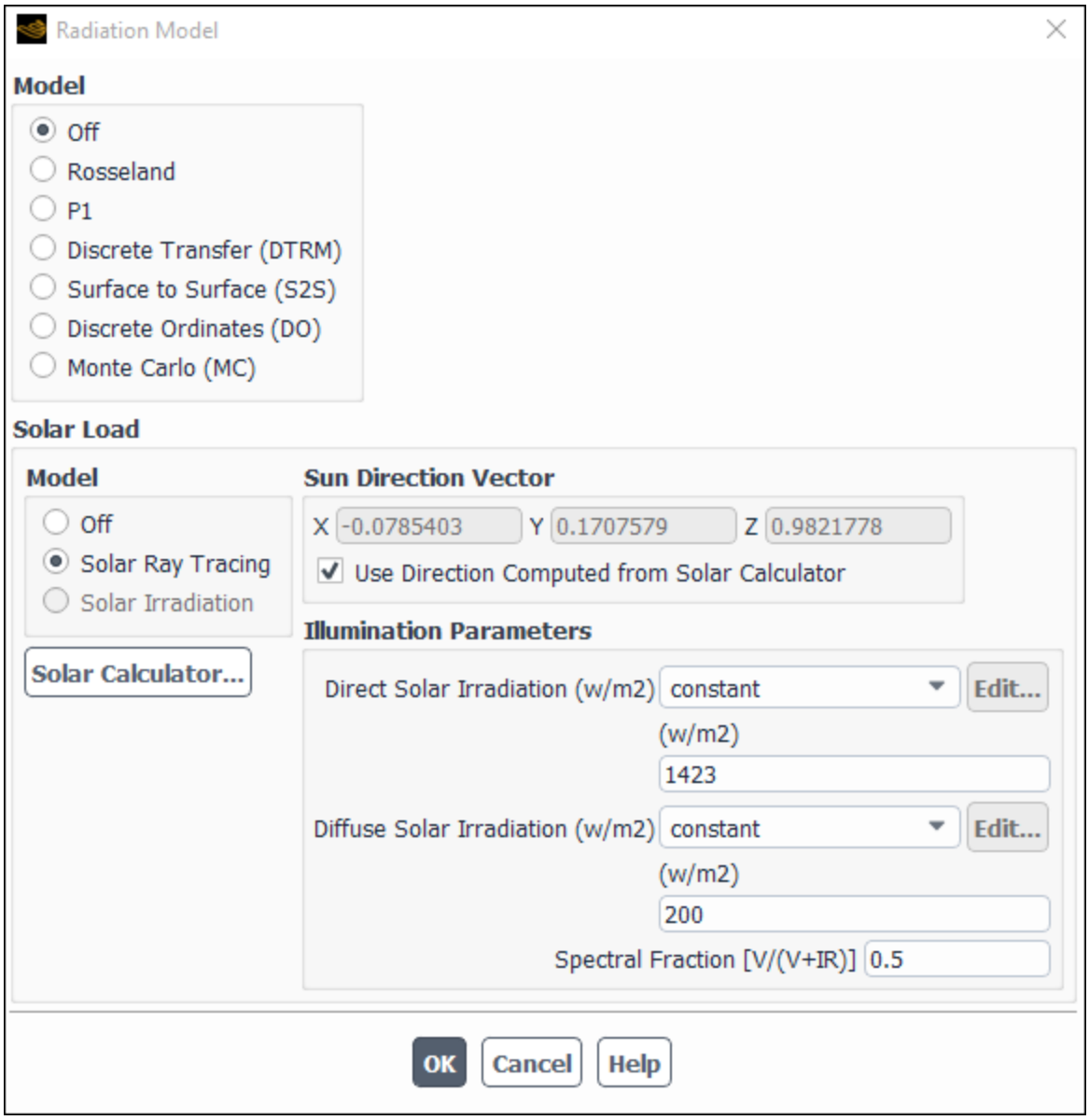
of the Ansys Fluent radiation models. **Solar Irradiation** is available only when the **Discrete Ordinates (DO)** or **Monte Carlo (MC)**radiation model is enabled.

To set up the solar load model, perform the following steps:

1. Enable the solar load model in the **Radiation Model** Dialog Box.

   a. To enable the solar ray tracing algorithm, select **Solar Ray Tracing** under **Solar Load** (Figure 16.49: The Radiation Model Dialog Box (With Solar Load Model Solar Ray Tracing Option) (p. 2257)).

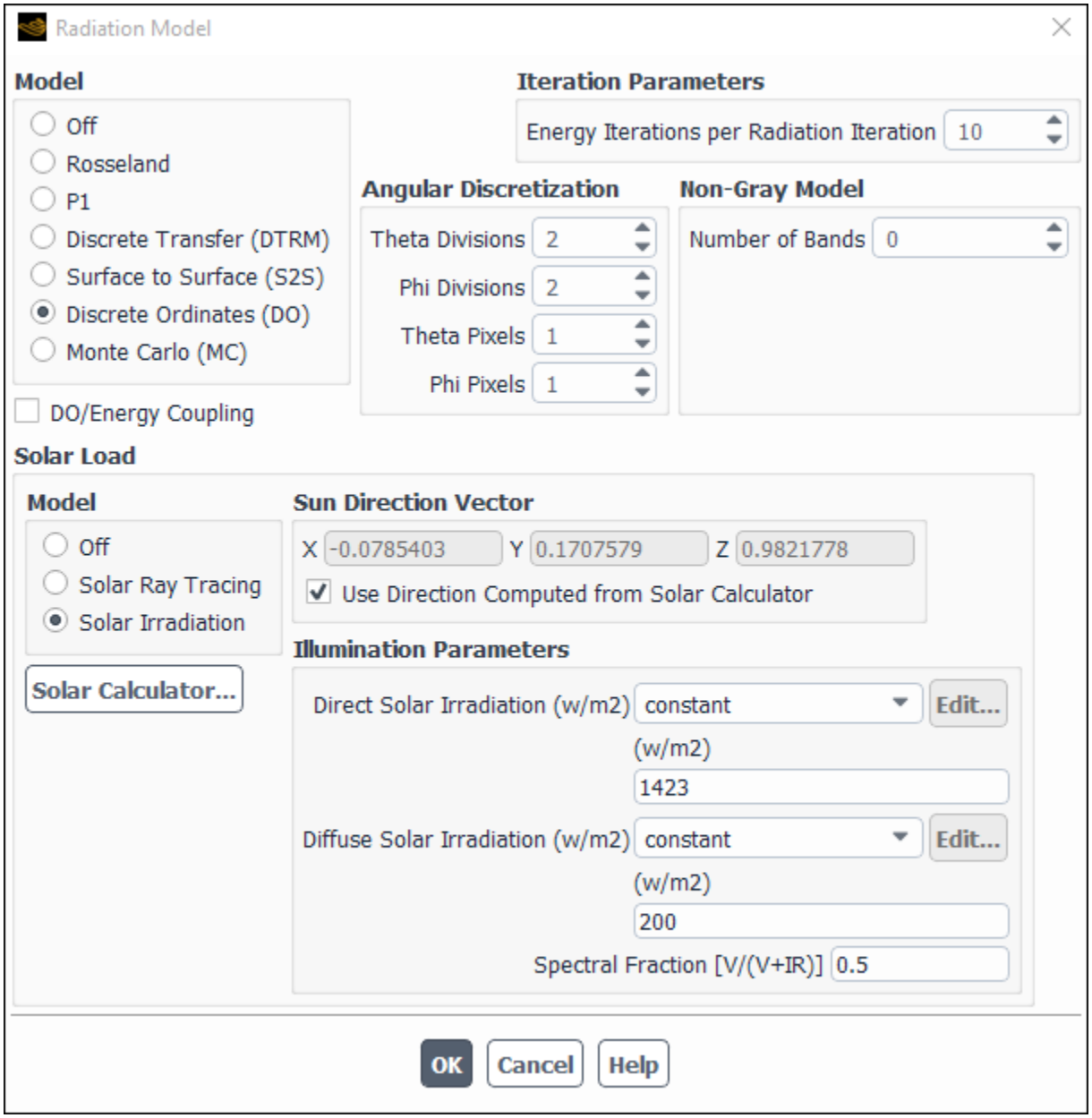
   Figure 16.49: The Radiation Model Dialog Box (With Solar Load Model Solar Ray Tracing Option)



   b. To enable the Solar Irradiation option, first select the **DO** or **MC** model, and then select **Solar Irradiation** under **Solar Load** (Figure 16.50: The Radiation Model Dialog Box (with Solar Load Model Solar Irradiation Option) (p. 2258)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.50: The Radiation Model Dialog Box (with Solar Load Model Solar Irradiation Option)



2. When using the non-gray DO or MC model, if the wavelengths of the specified bands do not cover the entire solar spectrum, a message is written to the console displaying what percentage of the full solar model has been covered. It is recommended that the full solar spectrum is considered.

   When using the non-gray MC model, if you only specify one band then Fluent will automatically cover the entire solar spectrum

3. Define the solar parameters.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. Enter values for the **X**, **Y**, and **Z** components of the **Sun Direction Vector**. Alternatively, you can choose to have this vector computed from the solar calculator by enabling the **Use Direction Computed from Solar Calculator** option.

b. Specify the illumination parameters.

i. Enter a value for **Direct Solar Irradiation** under **Illumination Parameters**. This parameter is the amount of energy per unit area in W / m^2 due to direct solar irradiation. This value may depend on the time of year and the clearness of the sky. Make your selection in the drop-down list next to **Direct Solar Irradiation** and either enter a **constant** value, have the value computed from the **solar calculator**, or specify it using a **user-defined function**. Note that only compiled (not interpreted) UDFs can be used to specify **Direct Solar Irradiation**. (For more information on writing solar intensity UDFs, see `DEFINE_SOLAR_INTENSITY` in the Fluent Customization Manual.) For transient simulations, you have the additional option of specifying a time-dependent **piecewise-linear** and **polynomial** profile for direct solar irradiation.

ii. Enter a value for **Diffuse Solar Irradiation** for the DO model only, which is the amount of energy per unit area in W / m^2 due to diffuse solar irradiation. This value may depend on the time of year, the clearness of the sky, and also on ground reflectivity. Make your selection in the drop-down list next to **Diffuse Solar Irradiation** and either enter a **constant** value, have the value computed from the **solar calculator**, or specify it using a **user-defined function**. Note that only compiled (not interpreted) UDFs can be used to specify **Diffuse Solar Irradiation**. (For more information on writing solar intensity UDFs, see `DEFINE_SOLAR_INTENSITY` in the Fluent Customization Manual.) For transient simulations, you have the additional option of specifying a time-dependent **piecewise-linear** and **polynomial** profile for diffuse solar irradiation.

iii. If you are using the **Solar Ray Tracing** solar load model (Figure 16.49: The Radiation Model Dialog Box (With Solar Load Model Solar Ray Tracing Option) (p. 2257)), then you will need to enter a value for **Spectral Fraction**. The spectral fraction is the fraction of incident solar radiation in the visible part of the solar radiation spectrum.

$$\text{Spectral Fraction} = \frac{V}{V+IR} \tag{16.15}$$

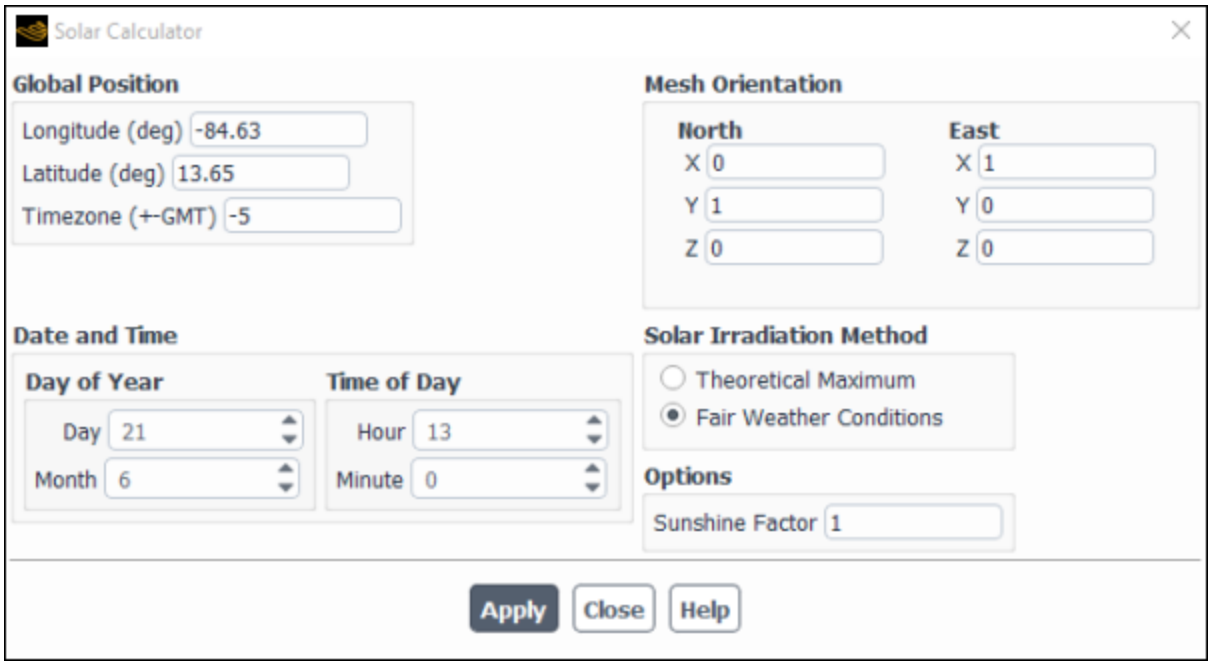
where V is the visible incident solar radiation, and $V+IR$ is the total incident solar radiation (visible plus infrared).

4. Use the solar calculator to compute solar beam direction and irradiation.

a. Click **Solar Calculator...** in the **Radiation Model** dialog box to open the **Solar Calculator** dialog box (Figure 16.51: The Solar Calculator Dialog Box (p. 2260)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.51: The Solar Calculator Dialog Box



b. In the **Solar Calculator** dialog box, define the **Global Position** by the following parameters:

   i. Enter a real number in degrees for **Longitude**. Values may range from -180 to 180 where negative values indicate the Western hemisphere and positive values indicate the Eastern hemisphere.

   ii. Enter a real number for **Latitude** in degrees. Values can range from -90° (the South Pole) to 90° (the North Pole), with 0° defined as the equator.

   iii. Enter an integer for **Timezone** that is the local time zone in hours relative to Greenwich Mean Time (+-GMT). This value can range from $+12$ to -12.

Important:

Note that you must specify all three **Global Position** parameters for the solar calculator.

---

c. Define the local **Date and Time** by the following parameters:

   i. Enter an integer for **Day** and **Month** under **Day of Year**.

   ii. Enter an integer for **Hour** that ranges from 0 to 24 under **Time of Day**. Enter an integer or floating point number for **Minute**.

   The time of day is based on a 24-hour clock: 0 hours and 0 minutes corresponds to 12:00 a.m. and 23 hours 59.99 min corresponds to 11:59.99 p.m. For example, if the local time was 12:01:30 a.m., you would enter 0 for **Hour** and 1.5 for **Minute**. If the local time was 4:17 p.m., you would enter 16 for **Hour** and 17 for **Minute**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

d. Define the **Mesh Orientation** as the vectors for North and East in the CFD mesh system of coordinates.

e. Select the appropriate **Solar Irradiation Method**. The **Fair Weather Conditions** is the default method.

f. Enter an integer for **Sunshine Fraction** (default = 1).

g. Click **Apply**.

The solar calculator output parameters are computed and the results are reported in the console. The default values are shown below:

```
Fair Weather Conditions:
 Sun Direction Vector: X: -0.0785396, Y: 0.170758, X: 0.982178
 Sunshine Fraction: 1
 Direct Normal Solar Irradiation (at Earth's surface) [W/m^2]:
 881.635
 Diffuse Solar Irradiation - vertical surface: [W/m^2]:
 152.107
 Diffuse Solar Irradiation - horizontal surface: [W/m^2]:
 118.727
 Ground Reflected Solar Irradiation - vertical surface: [W/m^2]:
 96.4649
```

5. For transient simulations, enter the **Time Steps Per Solar Load Update** under **Update Parameters**. The number of time steps that you specify will direct the Ansys Fluent solver to update the solar load data for the specified flow-time intervals in the unsteady solution process.

16.3.11.5.3. Setting Boundary Conditions for Solar Loading

Once you have defined the solar parameters for the solar load model (Setting Up the Solar Load Model (p. 2256)), you will need to set up boundary conditions for boundary zones that will participate in solar loading. The boundary condition dialog boxes can be opened by right-clicking the boundary name in the tree (under **Setup/Boundary Conditions**) and clicking **Edit...** in the menu that opens; alternatively, you can open them from the **Boundary Conditions** task page:

Setup → **Boundary Conditions**

16.3.11.5.4. Solar Ray Tracing

Note:

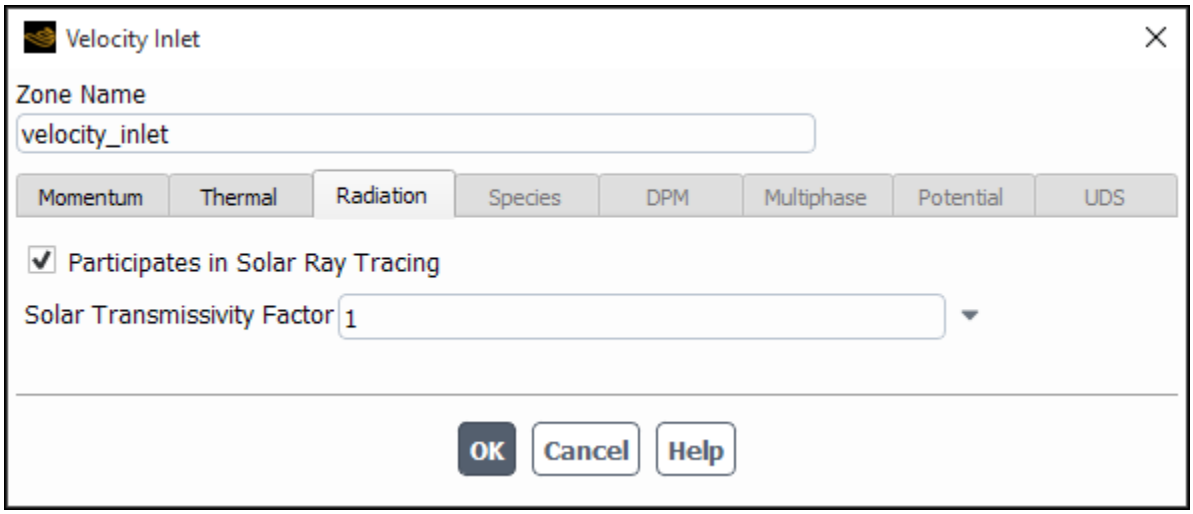
Solar ray tracing is designed to be applied to your geometrical information, therefore, you must include every side of the external, or outer, enclosure/boundary for ray-tracing purposes, unless you expect that solar irradiation cannot enter through such boundaries. If you exclude any internal zone from the solar load computation, then this would mean that the resulting obstruction or blockage has been removed which would also be equivalent to a transparent surface. Excluded walls will also not participate in the computation of scattered energy by the solar load model and, therefore, do not gain any heat from scattered energy.

1. Set the boundary condition for each inlet and exit boundary zone that you want to include in solar loading.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. Open the inlet or exit boundary condition dialog box (for example, **Velocity Inlet**) and click the **Radiation** tab (Figure 16.52: The Velocity Inlet Dialog Box (p. 2262)).

Figure 16.52: The Velocity Inlet Dialog Box



b. Enable the **Participates in Solar Ray Tracing** option (this option is enabled for all boundary conditions by default). If you deactivate solar ray tracing by disabling this option the surface will be ignored and the solar ray will pass through it with no interaction, regardless of the boundary condition type.

c. Enter a value between `0` and `1` for the **Solar Transmissivity Factor**. This will allow you to control the amount of solar irradiation entering the domain. By reducing the solar transmissivity factor from 1 to 0.5, you can effectively cut the total internal energy source entering the domain by half.

Important:

Note that the solar transmissivity factor is applied to both direct and diffuse solar irradiation components.

---

d. Click **OK**.

2. Set the boundary condition for each wall boundary zone that you want to include in solar loading.

a. Open a **Wall** boundary condition dialog box and click the **Radiation** tab.

b. Define the wall as **opaque** or **semi-transparent**. (An opaque wall will not allow any solar radiation to pass through it, while a semi-transparent surface will allow a portion of the solar radiation to pass through it.)

i. For an opaque wall, select **opaque** from the drop-down list for **BC Type** (Figure 16.53: The Wall Dialog Box (p. 2263)). Then enable the **Participates in Solar Ray Tracing** option (this option is enabled for all boundary conditions by default) in the **Solar Boundary Conditions** group box and enter constant values for **Direct Visible**

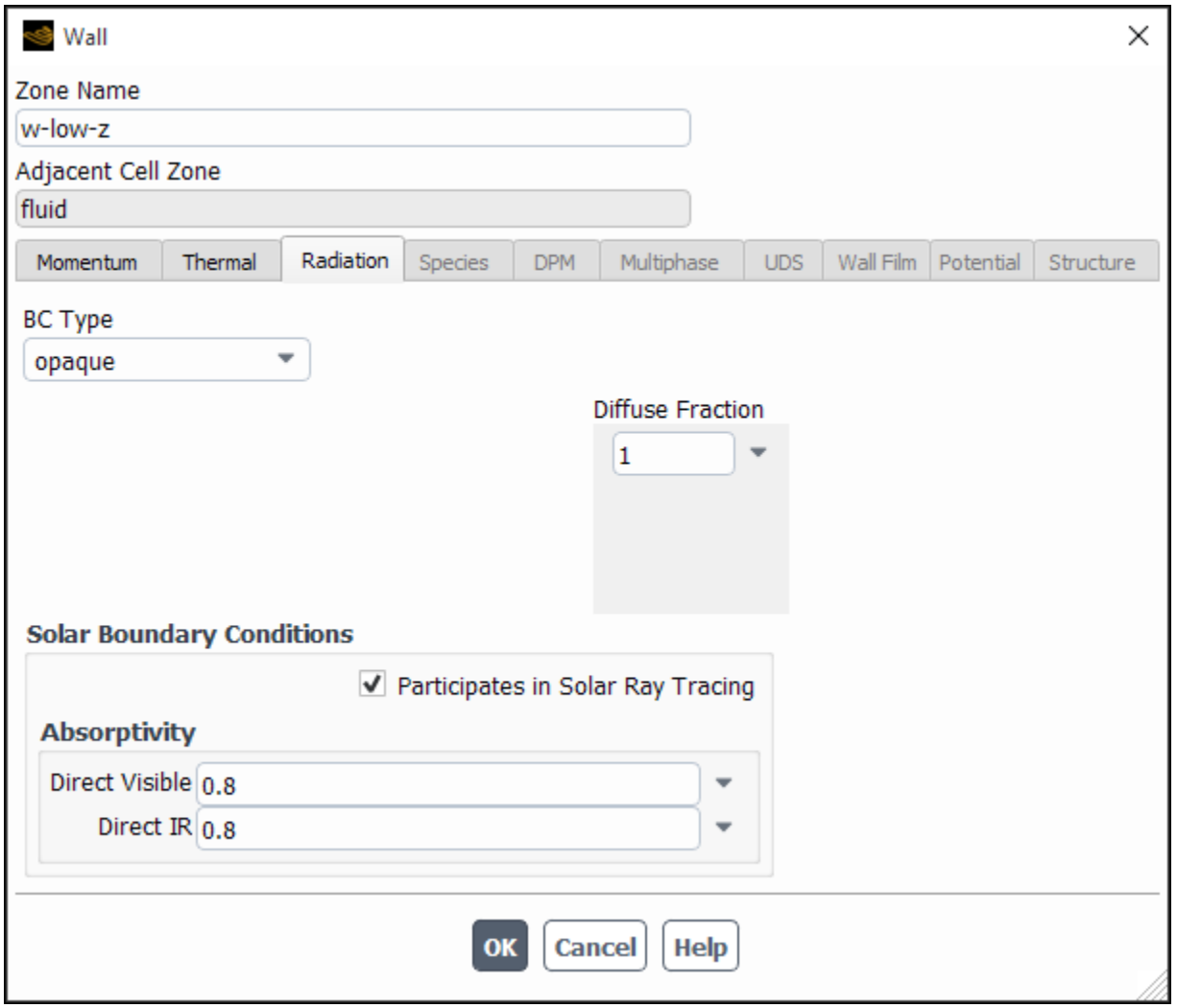
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and **Direct IR** absorptivity. Note that if you deactivate solar ray tracing by disabling the **Participates in Solar Ray Tracing** option, the surface will be ignored and the solar ray will pass through it with no interaction, regardless of the boundary condition type.

Important:

Absorption in the visible and infrared portions of the spectrum define the surface material for the opaque wall.

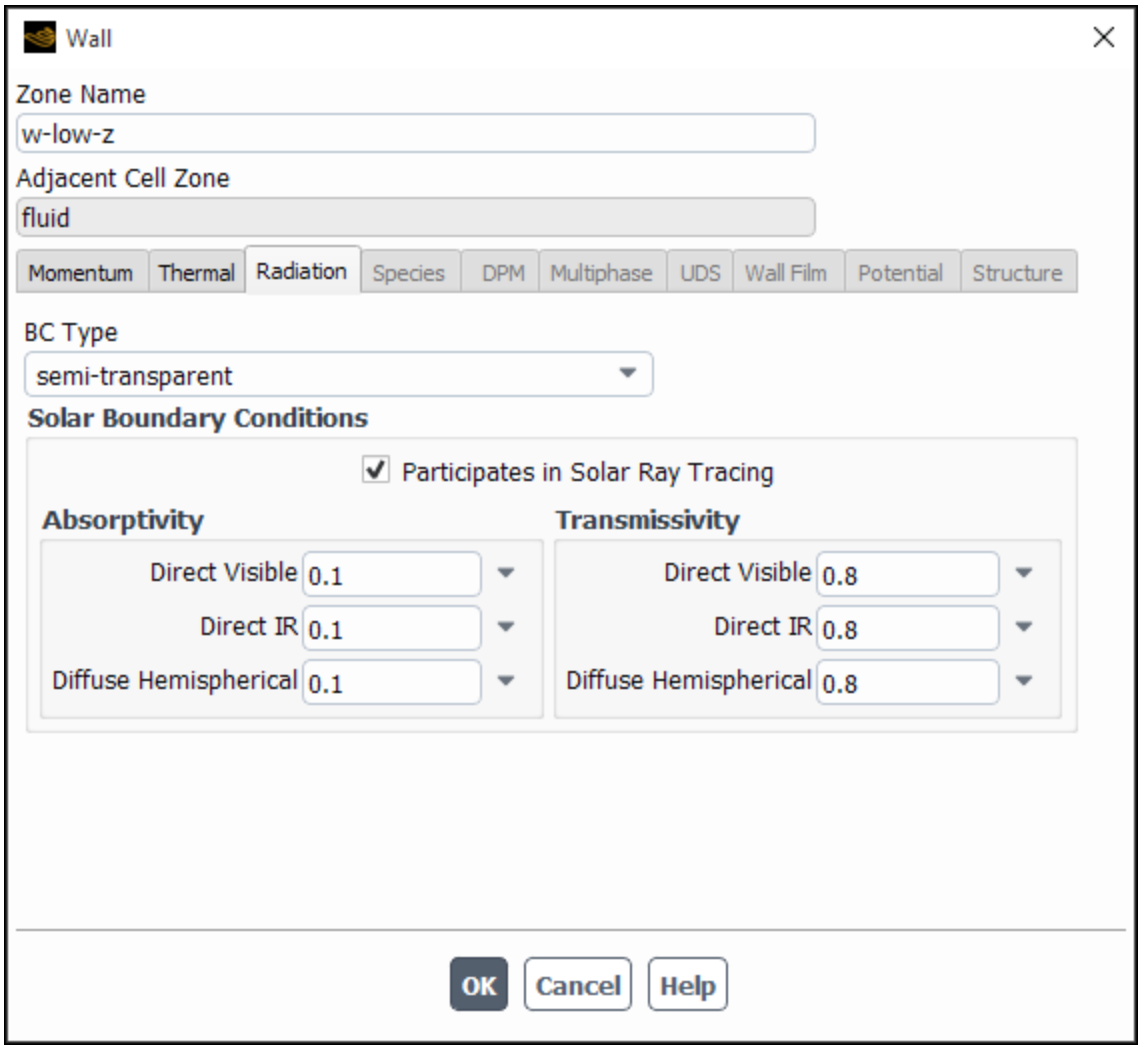
Figure 16.53: The Wall Dialog Box



ii. For a semi-transparent wall, select **semi-transparent** from the drop-down list for **BC Type** (Figure 16.54: The Wall Dialog Box (p. 2264)). Then, enable the **Participates in Solar Ray Tracing** option (this option is enabled for all boundary conditions by default) in the **Solar Boundary Conditions** group box and enter constant values for **Direct Visible**, **Direct IR**, and **Diffuse Hemispherical** absorptivity and transmissivity. Note that if you deactivate solar ray tracing by disabling the **Participates in Solar Ray Tracing** option, the surface will be ignored and the solar ray will pass through it with no interaction, regardless of the boundary condition type.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.54: The Wall Dialog Box



Important:

For semitransparent coupled walls, the solar ray tracing settings are defined on the original wall and not on the shadow wall.

Absorption and transmittance in the visible and infrared portions of the spectrum, as well as the "shading" formulation (**Diffuse Hemispherical**), define the surface material for a semi-transparent wall. These parameters are properties of the glazed unit and should be provided by the glazing manufacturer. The direct components are based on normal incident radiation (Ansys Fluent adjusts this for the actual angle of incidence). Most manufacturers present this information in a slightly different way so it may be necessary to seek guidance from the supplier. Another useful source of data can be found in the ASHRAE Fundamentals Handbook, chapter on Fenestration.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

iii. Click **OK**.

Important:

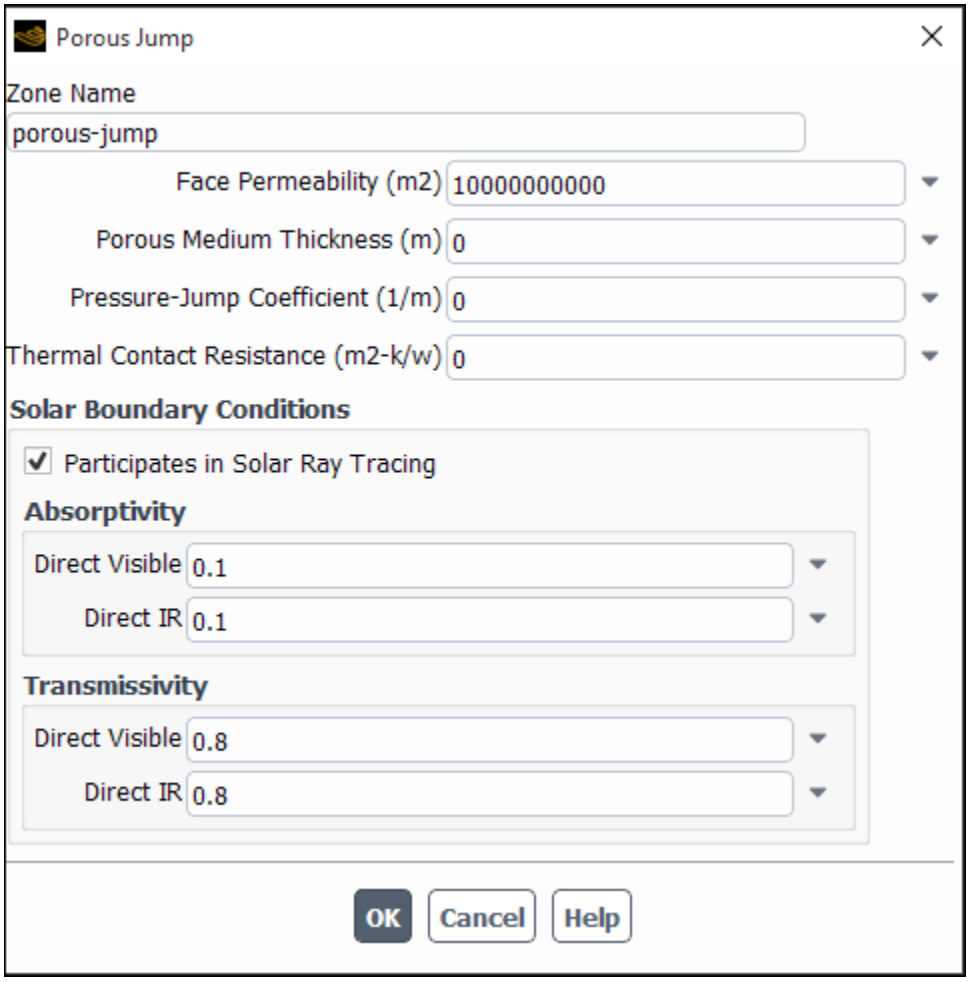
Ansys Fluent will calculate the reflectivity as the difference between one and the sum of absorptivity and transmissivity:

$$reflectivity = 1 - \left(absorptivity + transmissivity\right) \tag{16.16}$$

3. Set the boundary condition for each porous jump boundary zone that you want to include in solar loading. You can define the boundary such that it acts as a semi-transparent surface that will allow a portion of the solar radiation to pass through it, along with fluid flow. In this way you can represent a partial opening (for example, a grate, a grill, or a louver) that is a combination of gaps and (typically) opaque surfaces.

   a. Open the **Porous Jump** boundary condition dialog box (Figure 16.55: The Porous Jump Dialog Box (p. 2265)).

Figure 16.55: The Porous Jump Dialog Box



b. Define the **Face Permeability**, **Porous Medium Thickness**, **Pressure-Jump Coefficient**, and **Thermal Contact Resistance** as described in User Inputs for the Porous Jump Model (p. 1519).

c. Define the settings in the **Solar Boundary Conditions** group box.

i. Enable the **Participates in Solar Ray Tracing** option (this option is enabled for all boundary conditions by default). Note that if you deactivate solar ray tracing by disabling the **Participates in Solar Ray Tracing** option, the surface will be ignored and the solar ray will pass through it with no interaction, regardless of the boundary condition type.

ii. Enter constant values (between 0 and 1) for **Direct Visible** and **Direct IR** in the **Absorptivity** group box. These values act as multipliers for the visible and infrared portions of the direct solar radiation spectrum, respectively, to account for the absorption of the porous jump.

Important:

One reasonable way to estimate the **Direct Visible** and **Direct IR** absorptivity values for a grill, and so on, is to use the product of the obstructed area fraction and the surface absorptivity. For example, if 40% of the grill facial area is obstructed with grill slats, and the slats have a surface absorptivity of 0.7, you could estimate the absorptivity as being 0.28.

iii. Enter constant values (between 0 and 1) for **Direct Visible** and **Direct IR** in the **Transmissivity** group box. These values act as multipliers for the visible and infrared portions of the direct solar radiation spectrum, respectively, to account for the transmissivity of the porous jump.

Important:

One reasonable way to estimate the **Direct Visible** and **Direct IR** transmissivity values for a grill, and so on, is to use the open area fraction. For example, if a grill has openings between the slats that amount to 60% of the area, you could estimate the transmissivity as being 0.6.

d. Click **OK**.

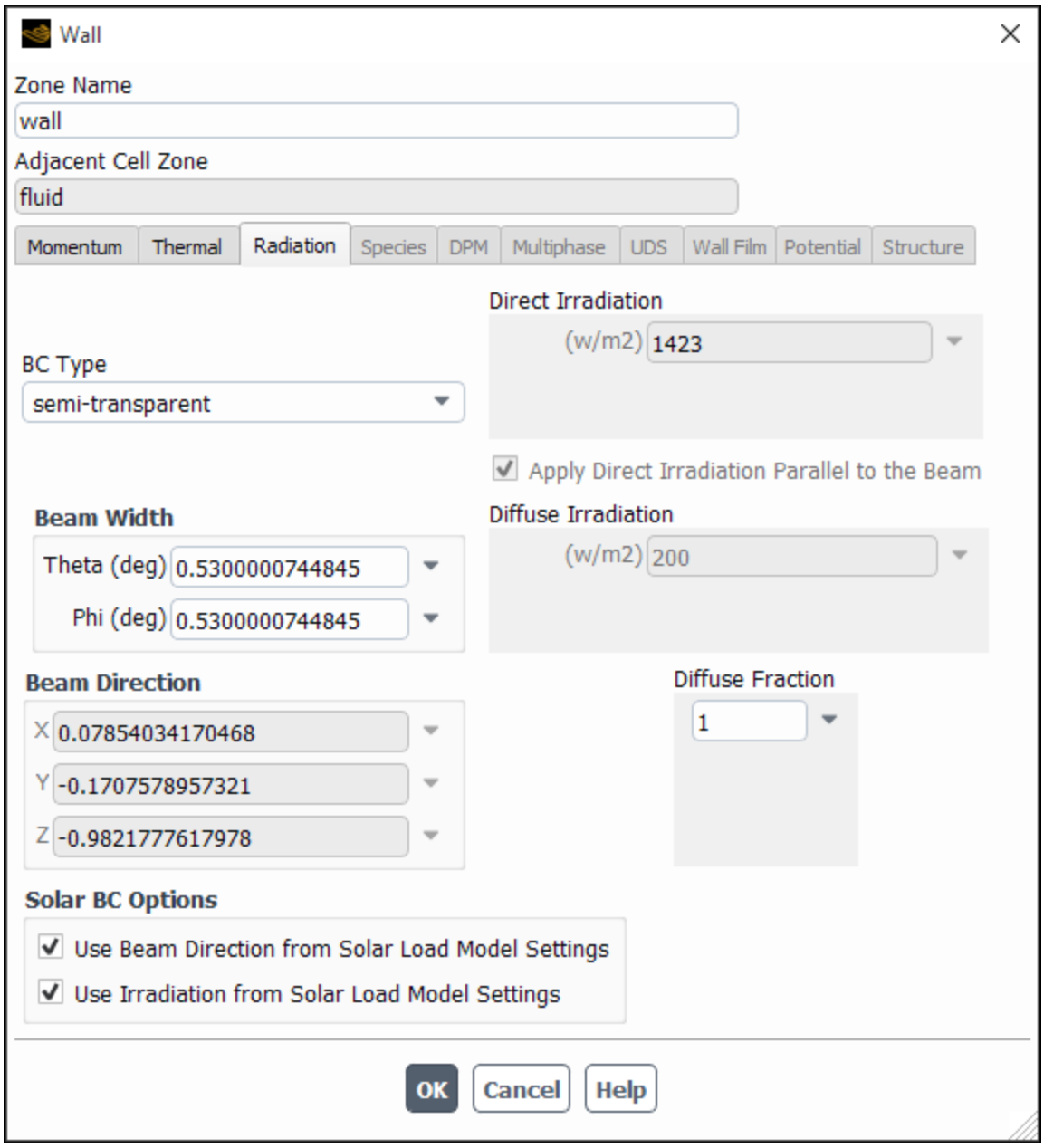
16.3.11.5.5. Solar Irradiation

1. For Solar Irradiation, all boundary conditions are set up as normal for the DO or MC model, except that now you can select semi-transparent boundary surfaces that will provide a source of solar irradiation.

a. Open a **Wall** boundary condition dialog box and click the **Radiation** tab (Figure 16.56: The Wall Dialog Box Radiation tab with Solar Irradiation (p. 2267)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.56: The Wall Dialog Box Radiation tab with Solar Irradiation



b. Select **semi-transparent** from the drop-down list for **BC Type**.

c. Enable the **Use Beam Direction from Solar Load Model Settings** option, under **Solar BC Options**, to have the values for beam direction applied from the **Solar Load Model** settings in the **Radiation** dialog box.

Important:

Note that the sign of the beam direction that is needed for the DO or MC model is opposite the sun direction vector that is entered or derived from the solar parameters. The beam direction in the DO or MC model is the direction

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

of external radiation (for example, radiation coming from the sun), while the sun direction vector in the solar load model points to the sun.

d. Enable the **Use Irradiation from Solar Load Model Settings** option to have the irradiation specified on the Radiation Model dialog box for the Solar Load model applied for direct and diffuse irradiation.

   If selected while using the multiband approach for the DO or MC models, the direct and diffuse irradiation is calculated and applied in each band, according to the black body emission weighting factor.

   When **Use Irradiation from Solar Load Model Settings** is enabled, the beam width will automatically be set to 0.53 degrees - the angle subtended by the sun.

e. Click **OK**.

16.3.11.5.6. Text Interface-Only Commands

Ansys Fluent has provided some additional commands for solar load setup that are only available in the text interface. These commands are present in the following sections.

16.3.11.5.6.1. Automatically Saving Solar Ray Tracing Data

It is possible to direct Ansys Fluent to automatically save solar load data to a generic file that you can examine or use in an external program. This is done by executing the text command `autosave-solar-data` from the text interface. Note, this command is only available on Linux in serial mode -t0.

`define` → `models` → `radiation` → `solar-parameters` → `autosave-solar-data`

1. Enter a value for the autosave solar data file frequency when prompted, in order to specify the frequency (in time steps) at which you want the solar load data written to a file. The default value is zero, which means that no automatic saving is performed.

2. Enter the filename, in quotations.

3. Choose to write file in binary format.

The text interface command for `autosave-solar-data` for a file named `solar` and a frequency of 1 is shown below:

```
/define/models/radiation/solar-parameters> autosave-solar-data
 Autosave Solar Data File Frequency [0] 1
 Enter Filename [""] "solar"
```

16.3.11.5.6.2. Automatically Reading Solar Data

When you are executing a transient simulation in serial or parallel Ansys Fluent version 18.2 or later and you want to read the solar load data file from a serial run performed in an earlier version, you can use the following text command:

`define` → `models` → `radiation` → `solar-parameters` → `autoread-solar-data`

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Enter a value for the autoread solar data file frequency when prompted, in order to specify the frequency (in time steps) at which you want the solar load data read from the file generated during the serial run. The default value is zero, which means that no automatic reading is performed.

2. Enter the filename, in quotations.

The text interface command for `autoread-solar-data` for a file named `solar` and a frequency of 1 is shown below:

```
/define/models/radiation/solar-parameters> autoread-solar-data
 Autoread Solar Data File Frequency [0] 1
 Enter Filename [""] "solar"
 Use Binary Format for Reading Data Files [yes]
```

16.3.11.5.6.3. Aligning the Camera Direction With the Position of the Sun

When the solar load model is enabled, you can direct Ansys Fluent to align the camera direction with the sun position using the text interface command:

`define` → `models` → `radiation` → `solar-parameters` → `sol-camera-pos`

This command is useful when you are executing a transient simulation and you want to capture an image of your model with solar load parameters displayed (such as solar heat flux) as the sun position changes with time in order to create an animation. See Postprocessing Solar Load Quantities (p. 2271) for details.

16.3.11.5.6.4. Specifying the Scattering Fraction

You can modify the default scattering fraction (1) using the text interface command:

`define` → `models` → `radiation` → `solar-parameters` → `scattering-fraction`

The scattering fraction is the amount of direct radiation that has been reflected from opaque surfaces (after entering through the transparent surfaces) that will be considered to remain within the space and be evenly distributed among all surfaces. The value is between 0 and 1.

The text interface command for specifying a `scattering-fraction` of 0.5 is shown below:

```
/define/models/radiation/solar-parameters> scattering-fraction
 Scattering Fraction [1] .5
```

16.3.11.5.6.5. Applying the Solar Load on Adjacent Fluid Cells

You can direct Ansys Fluent to apply the solar load that is computed from the solar ray tracing algorithm to adjacent fluid cells by issuing the following command at the text interface:

`define` → `models` → `radiation` → `solar-parameters` → `sol-adjacent-fluidcells`

The text interface command is shown below:

```
/define/models/radiation/solar-parameters> sol-adjacent-fluidcells
 Apply Solar Load on adjacent Fluid Cells? [no] y
```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This command allows you to apply solar loads to adjacent fluid cells only, even if solid or shell conduction zones are present. By applying the solar load on adjacent fluid cells, you are overruling the default order of the adjacent cell assignment in Ansys Fluent which is shell, solid, fluid.

16.3.11.5.6.6. Specifying Quad Tree Refinement Factor

You can modify the default value (7) for the maximum quad tree refinement factor in the solar ray tracing algorithm using the text command:

`define` → `models` → `radiation` → `solar-parameters` → `quad-tree-parameters`

The text interface command is shown below, when a new maximum refinement value of 10 is specified:

```
/define/models/radiation/solar-parameters> quad-tree-parameters
 Maximum Quad-Tree Refinement [7] 10
```

16.3.11.5.6.7. Specifying Ground Reflectivity

You can modify the default value (0.2) for the ground reflectivity using the text command:

`define` → `models` → `radiation` → `solar-parameters` → `ground-reflectivity`

Ground reflectivity ρ_g (Equation 16.14 (p. 2254)) includes the contribution of reflected solar radiation from ground surfaces. It is treated as part of the total diffuse solar irradiation when the solar calculator is used in conjunction with the `Diffuse Solar Irradiation` illumination parameter. The default value is 0.2.

```
/define/models/radiation/solar-parameters> ground-reflectivity
 Ground Reflectivity [0.2] 0.5
```

16.3.11.5.6.8. Reverting to Single Band Implementation of DO Model

For consistency with cases set up previous to Fluent 2019 r1, you can use the following command so that the complete solar load is applied to the first band only.

`define` → `models` → `radiation` → `apply-full-solar-irradiation?`

```
  /define/models/radiation/apply-full-solar-irradiation?
Apply full solar irradiation to first band? [no] yes
```

16.3.11.5.6.9. Additional Text Interface Commands

Some solar load commands that are available in the graphical user interface are also made available in the text interface. For example, you can turn the solar load model on using the text command:

`define` → `models` → `radiation` → `solar?`

You can also enter the solar calculator parameters in the text interface by executing the command:

`define` → `models` → `radiation` → `solar-calculator`

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Once invoked, you will be prompted to enter the solar calculator input parameters.

To set the illumination parameters, select this option from the `solar-parameters` menu:

`define` → `models` → `radiation` → `solar-parameters` → `illumination-parameters`

And finally, you can direct Ansys Fluent to compute the solar load on demand, by issuing the text command:

`define` → `models` → `radiation` → `solar-parameters` → `sol-on-demand`

When the command is initiated, the solar data are written to the console.

16.3.11.6. Postprocessing Solar Load Quantities

The following solar load quantities can be used to visualize the illuminated areas and shadows created by solar radiation.

- solar heat flux (that is, sum of visible and IR absorbed solar flux on opaque walls)
- absorbed visible and IR solar flux (semi-transparent walls and porous jump boundaries only)
- reflected visible and IR solar flux (semi-transparent walls and porous jump boundaries only)
- transmitted visible and IR solar flux (semi-transparent walls and porous jump boundaries only)

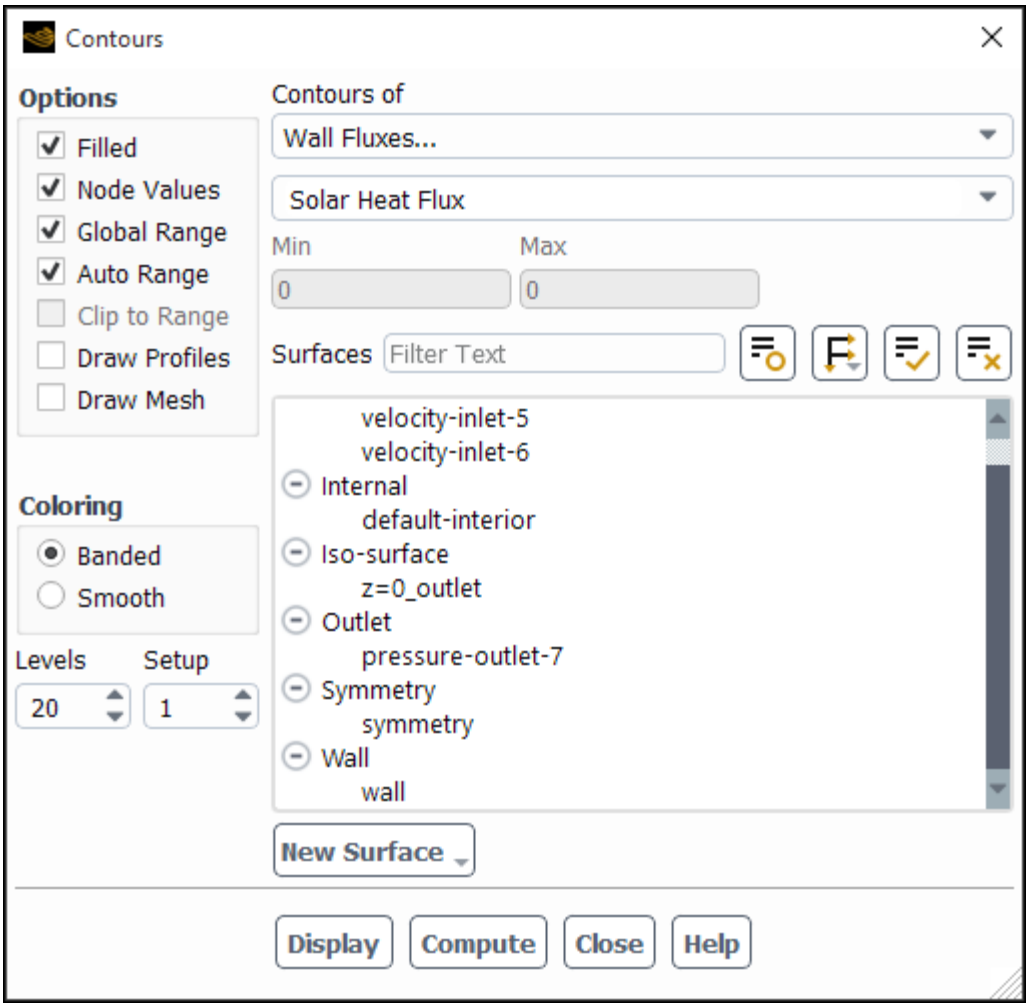
These quantities are available for postprocessing of solar loading at wall boundaries and can be displayed as contours of **Wall Fluxes** in the **Contours** dialog box. For steady-state simulations, the solar flux data is computed at solution initialization and is available for postprocessing. You can also compute the solar load at any time during your Ansys Fluent session, after you have set up the model and applied boundary conditions. To compute the solar load on demand, you can issue the `sol-on-demand` command in the text interface (see Additional Text Interface Commands (p. 2270) for details).

Solar heat flux, for example, can be displayed for surfaces using the **Contours** dialog box. A sample dialog box is shown below (Figure 16.57: The Contours Dialog Box (p. 2272)).

Results → **Graphics** → **Contours** **New...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.57: The Contours Dialog Box



16.3.11.6.1. Solar Load Animation at Different Sun Positions

The solar camera alignment command is useful when you want to take timed pictures of solar loading effects of your model during transient simulations, and later create animations of the image files using an external program. Follow the procedure below.

1. Read (or set up) your transient case file in Ansys Fluent.

2. Set up the automatic execution of solution commands in the **Execute Commands** dialog box that will: 1) display solar load parameter graphics, 2) re-position the solar camera such that the view is aligned with the instantaneous sun direction, and 3) generate a picture image file (`.tiff`) during the solution process in the **Execute Commands** dialog box.

   Solution → **Calculation Activities** → **Execute Commands** **Edit...**

3. Initialize and run the solution.

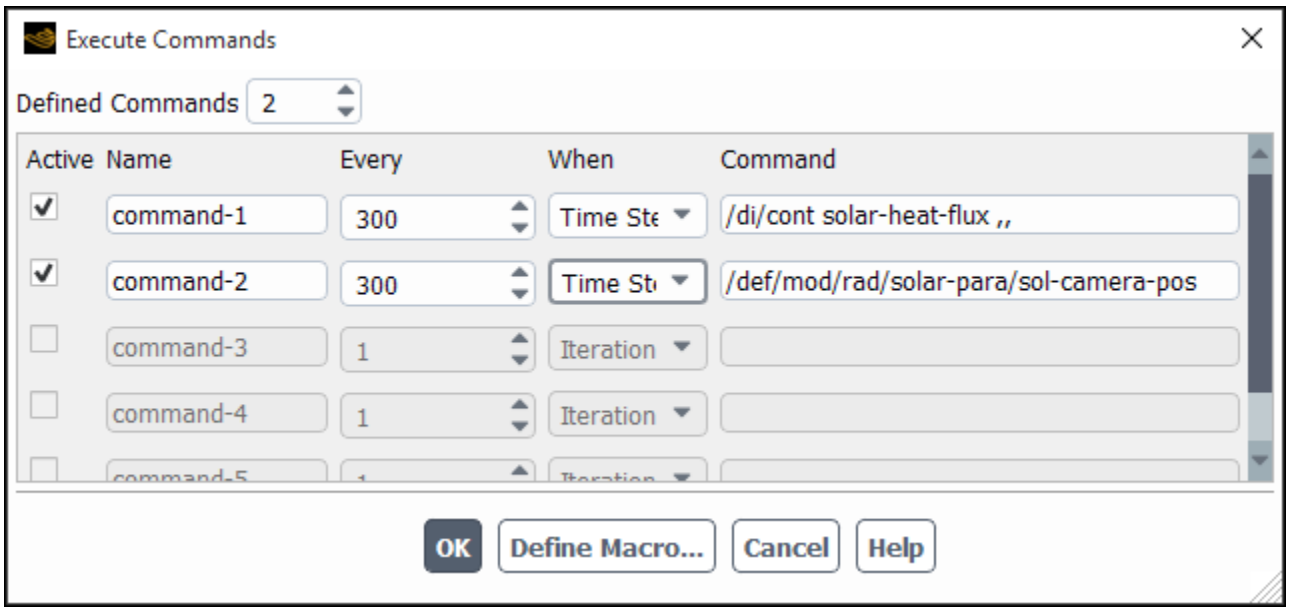
4. Animate the `.tiff` files using an external animation tool.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following commands entered in the **Execute Commands** dialog box will direct Ansys Fluent to display contours of solar heat flux, align the camera with the current direction of the sun, and then generate a picture image file (`.tiff`) of the solar heat flux contour every 300 time steps during the unsteady simulation. See Figure 16.58: The Execute Commands Dialog Box (p. 2273).

```
/di/cont solar-heat-flux ,,
/def/mod/rad/solar-para/sol-camera-pos
/di/hc "flux-%t.tiff"
```

Figure 16.58: The Execute Commands Dialog Box



16.3.11.6.2. Reporting and Displaying Solar Load Quantities

Ansys Fluent provides some additional solar load variables that you can use for postprocessing when your model includes solar ray tracing. You can generate graphical plots or alphanumeric reports of the following variables:

In the **Wall Fluxes...** category:

- **Solar Heat Flux**
- **Transmitted Visible Solar Flux** (semi-transparent walls and porous jump boundaries)
- **Transmitted IR Solar Flux** (semi-transparent walls and porous jump boundaries)
- **Reflected Visible Solar Flux** (semi-transparent walls and porous jump boundaries)
- **Reflected IR Solar Flux** (semi-transparent walls and porous jump boundaries)
- **Absorbed Visible Solar Flux** (semi-transparent walls and porous jump boundaries)
- **Absorbed IR Solar Flux** (semi-transparent walls and porous jump boundaries)

See Field Function Definitions (p. 4167) for their definitions.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16.4. Modeling Periodic Heat Transfer

Ansys Fluent is able to predict heat transfer in periodically repeating geometries, such as compact heat exchangers, by including only a single periodic module for analysis.

This section discusses streamwise-periodic heat transfer. The treatment of streamwise-periodic flows is discussed in Periodic Flows (p. 1747), and a description of no-pressure-drop periodic flow is provided in Periodic Boundary Conditions (p. 1501).

Information about streamwise-periodic heat transfer is presented in the following sections:

16.4.1. Overview and Limitations
16.4.2. Theory
16.4.3. Using Periodic Heat Transfer
16.4.4. Solution Strategies for Periodic Heat Transfer
16.4.5. Monitoring Convergence
16.4.6. Postprocessing for Periodic Heat Transfer

16.4.1. Overview and Limitations

The following sections contain information about periodic heat transfer:

16.4.1.1. Overview
16.4.1.2. Constraints for Periodic Heat Transfer Predictions

16.4.1.1. Overview

As discussed in Overview and Limitations (p. 1747), streamwise-periodic flow conditions exist when the flow pattern repeats over some length L, with a constant pressure drop across each repeating module along the streamwise direction.

Periodic thermal conditions may be established when the thermal boundary conditions are of the constant wall temperature or wall heat flux type. In such problems, the temperature field (when scaled in an appropriate manner) is periodically fully developed [116] (p. 5719). As for periodic flows, such problems can be analyzed by restricting the numerical model to a single module or periodic length.

16.4.1.2. Constraints for Periodic Heat Transfer Predictions

In addition to the constraints for streamwise-periodic flow discussed in Limitations for Modeling Streamwise-Periodic Flow (p. 1748), the following constraints must be met when periodic heat transfer is to be considered:

- The pressure-based solver must be used.
- The thermal boundary conditions must be of the specified heat flux or constant wall temperature type. Furthermore, in a given problem, these thermal boundary types cannot be combined: all boundaries must be either constant temperature or specified heat flux. You can, however, include constant-temperature walls and zero-heat-flux walls in the same problem. For the constant-temperature case, all walls must be at the same temperature (profiles are not allowed) or zero

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

heat flux. For the heat flux case, profiles and/or different values of heat flux may be specified at different walls.

- When constant-temperature wall boundaries are used, you cannot include viscous heating effects or any volumetric heat sources.
- In cases that involve solid regions, the regions cannot straddle the periodic plane.
- The thermodynamic and transport properties of the fluid (heat capacity, thermal conductivity, viscosity, and density) cannot be functions of temperature. (You cannot, therefore, model reacting flows.) Transport properties may, however, vary spatially in a periodic manner, and this allows you to model periodic turbulent flows in which the effective turbulent transport properties (effective conductivity, effective viscosity) vary with the (periodic) turbulence field.
- The periodic boundary condition with specified non-zero mass flow rate and/or pressure gradient cannot be used for variable density flows.

Theory (p. 2275) and Using Periodic Heat Transfer (p. 2276) provide more detailed descriptions of the input requirements for periodic heat transfer.

16.4.2. Theory

Streamwise-periodic flow with heat transfer from constant-temperature walls is one of two classes of periodic heat transfer that can be modeled by Ansys Fluent. A periodic fully developed temperature field can also be obtained when heat flux conditions are specified. In such cases, the temperature change between periodic boundaries becomes constant and can be related to the net heat addition from the boundaries as described in this section.

Important:

Periodic heat transfer can be modeled only if you are using the pressure-based solver.

16.4.2.1. Definition of the Periodic Temperature for Constant-Temperature Wall Conditions

For the case of constant wall temperature, as the fluid flows through the periodic domain, its temperature approaches that of the wall boundaries. However, the temperature can be scaled in such a way that it behaves in a periodic manner. A suitable scaling of the temperature for periodic flows with constant-temperature walls is [116] (p. 5719)

$$\theta = \frac{T(\vec{r}) - T_{wall}}{T_{bulk,inlet} - T_{wall}} \tag{16.17}$$

The bulk temperature, $T_{bulk,inlet}$, is defined by

$$T_{bulk,inlet} = \frac{\int_A T \left| \rho \vec{v} \cdot d\vec{A} \right|}{\int_A \left| \rho \vec{v} \cdot d\vec{A} \right|} \tag{16.18}$$

where the integral is taken over the inlet periodic boundary (A). It is the scaled temperature, θ, which obeys a periodic condition across the domain of length L.

16.4.2.2. Definition of the Periodic Temperature Change σ for Specified Heat Flux Conditions

When periodic heat transfer with heat flux conditions is considered, the form of the unscaled temperature field becomes analogous to that of the pressure field in a periodic flow:

$$\frac{T\left(\vec{r}+\vec{L}\right)-T\left(\vec{r}\right)}{L}=\frac{T\left(\vec{r}+2\vec{L}\right)-T\left(\vec{r}+\vec{L}\right)}{L}=\sigma. \tag{16.19}$$

where $\vec{L}$ is the periodic length vector of the domain. This temperature gradient, σ, can be written in terms of the total heat addition within the domain, Q, as

$$\sigma=\frac{Q}{\dot{m}c_pL}=\frac{T_{bulk,exit}-T_{bulk,inlet}}{L} \tag{16.20}$$

where $\dot{m}$ is the specified or calculated mass flow rate.

16.4.3. Using Periodic Heat Transfer

A typical calculation involving both streamwise-periodic flow and periodic heat transfer is performed in two parts. First, the periodic velocity field is calculated (to convergence) without consideration of the temperature field. Next, the velocity field is frozen and the resulting temperature field is calculated. These periodic flow calculations are accomplished using the following procedure:

1. Set up a mesh with translationally periodic boundary conditions.
2. Specify constant thermodynamic and molecular transport properties.
3. Specify either the periodic pressure gradient or the net mass flow rate through the periodic boundaries.
4. Compute the periodic flow field, solving momentum, continuity, and (optionally) turbulence equations.
5. Specify the thermal boundary conditions at walls as either heat flux or constant temperature.
6. Define an inlet bulk temperature.
7. Solve the energy equation (only) to predict the periodic temperature field.

In order to model the periodic heat transfer, you will need to set up your periodic model in the manner described in User Inputs for the Pressure-Based Solver (p. 1749) for periodic flow models with the pressure-based solver, noting the restrictions discussed in Limitations for Modeling Streamwise-Periodic Flow (p. 1748) and Constraints for Periodic Heat Transfer Predictions (p. 2274). In addition, you will need to provide the following inputs related to the heat transfer model:

1. Enable the solution of the energy equation by right-clicking **Energy** in the tree (under **Setup/Models**) and clicking **On** in the menu that opens (Figure 16.1: Enabling the Energy Equation (p. 2164)).

   Setup → **Models** → **Energy** → **On**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Define the thermal boundary conditions according to one of the following procedures. The boundary condition dialog boxes can be opened by right-clicking the boundary name in the tree (under **Setup/Boundary Conditions**) and clicking **Edit...** in the menu that opens; alternatively, you can open them from the **Boundary Conditions** task page:

   Setup → Boundary Conditions

   - If you are modeling periodic heat transfer with specified-temperature boundary conditions, set the wall temperature T_{wall} for all wall boundaries in their respective **Wall** dialog box. Note that all wall boundaries must be assigned the same temperature and that the entire domain (except the periodic boundaries) must be "enclosed" by this fixed-temperature condition, or by symmetry or adiabatic (q=0) boundaries.

   - If you are modeling periodic heat transfer with specified-heat-flux boundary conditions, set the wall heat flux in the **Wall** dialog box for each wall boundary. You can define different values of heat flux on different wall boundaries, but you should have no other types of thermal boundary conditions active in the domain.

3. Define solid regions, if appropriate, according to one of the following procedures. The cell zones can be defined by right-clicking the zone name in the tree (under **Setup/Cell Zone Conditions**), or using the **Cell Zone Conditions** task page:

   Setup → Cell Zone Conditions

   - If you are modeling periodic heat transfer with specified-temperature conditions, conducting solid regions can be used within the domain, provided that on the perimeter of the domain they are enclosed by the fixed-temperature condition. Heat generation within the solid regions is not allowed when you are solving periodic heat transfer with fixed-temperature conditions.

   - If you are modeling periodic heat transfer with specified-heat-flux conditions, you can define conducting solid regions at any location within the domain, including volumetric heat addition within the solid, if desired.

4. Set constant material properties (density, heat capacity, viscosity, thermal conductivity), *not* temperature-dependent properties, using the **Create/Edit Materials** dialog box. This dialog box can be opened by right-clicking the material name in the tree (under **Setup/Materials**) and clicking **Edit...** in the menu that opens; alternatively, you can open them from the **Materials** task page:

   Setup → Materials

5. Specify the **Upstream Bulk Temperature** in the **Periodic Conditions** dialog box, which can be opened from **Boundary Conditions** task page:

   Setup → Boundary Conditions → Periodic Conditions...

   Important:

   If you are modeling periodic heat transfer with specified-temperature conditions, the bulk temperature should not be equal to the wall temperature, since this will give you the trivial solution of constant temperature everywhere.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Set the solution parameters as described in Solution Strategies for Periodic Heat Transfer (p. 2278).
7. Run the solution and monitor the convergence as described in Monitoring Convergence (p. 2278).
8. Postprocess the results as described in Postprocessing for Periodic Heat Transfer (p. 2279).

16.4.4. Solution Strategies for Periodic Heat Transfer

After completing the inputs described in Using Periodic Heat Transfer (p. 2276), you can solve the flow and heat transfer problem to convergence. The most efficient approach to the solution, however, is a sequential one in which the periodic flow is first solved without heat transfer and then the heat transfer is solved leaving the flow field unaltered. This sequential approach is accomplished as follows:

1. Disable solution of the energy equation in the **Equations** dialog box; this dialog box is accessed by right-clicking **Solution Controls** in the tree (under **Solution**) and clicking **Equations...** in the menu that opens.

   Solution → **Controls** **Equations...**

2. Solve the remaining equations (continuity, momentum, and, optionally, turbulence parameters) to convergence to obtain the periodic flow field.

   Important:

   When you initialize the flow field before beginning the calculation, use the mean value between the inlet bulk temperature and the wall temperature for the initialization of the temperature field.

3. Return to the **Equations** dialog box and turn off solution of the flow equations and turn on the energy solution.
4. Solve the energy equation to convergence to obtain the periodic temperature field of interest.

While you can solve your periodic flow and heat transfer problems by considering both the flow and heat transfer simultaneously, you will find that the procedure outlined above is more efficient.

16.4.5. Monitoring Convergence

If you are modeling periodic heat transfer with specified-temperature conditions, you can monitor the value of the bulk temperature ratio

$$\theta = \frac{T_{wall} - T_{bulk,inlet}}{T_{wall} - T_{bulk,exit}} \tag{16.21}$$

during the calculation by creating a report plot that includes bulk temperature ratio (**periodic-bulk-temperature-ratio**), to ensure that you reach a converged solution. See Monitoring Statistics (p. 3695) for additional information.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

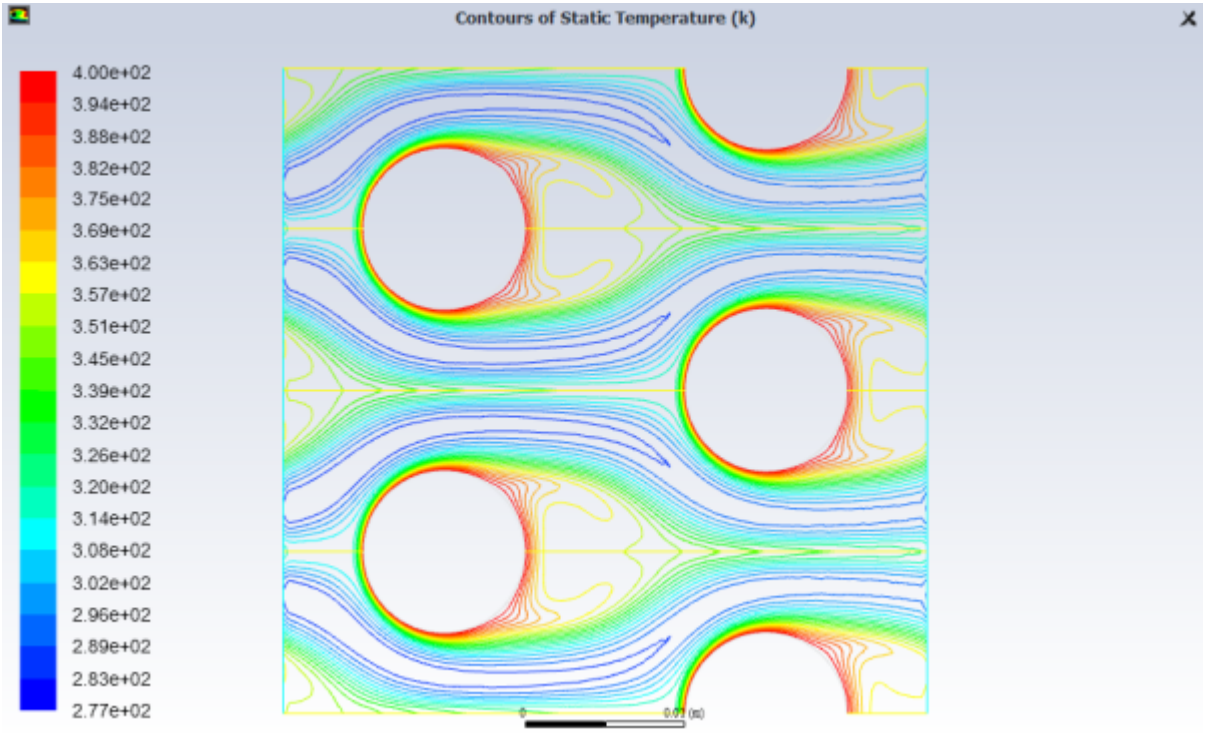
16.4.6. Postprocessing for Periodic Heat Transfer

The actual temperature field predicted by Ansys Fluent in periodic models will not be periodic, and viewing the temperature results during postprocessing will display this actual temperature field ($T(\vec{r})$ of Equation 16.17 (p. 2275)). The displayed temperature may exhibit values outside the range defined by the inlet bulk temperature and the wall temperature. This is permissible since the actual temperature profile at the inlet periodic face will have temperatures that are higher or lower than the inlet bulk temperature.

Static Temperature is found in the **Temperature...** category of the variable selection drop-down list that appears in postprocessing dialog boxes.

Figure 16.59: Temperature Field in a 2D Heat Exchanger Geometry With Fixed Temperature Boundary Conditions (p. 2279) shows the temperature field in a periodic heat exchanger geometry.

Figure 16.59: Temperature Field in a 2D Heat Exchanger Geometry With Fixed Temperature Boundary Conditions



16.5. Modeling Heat Exchangers

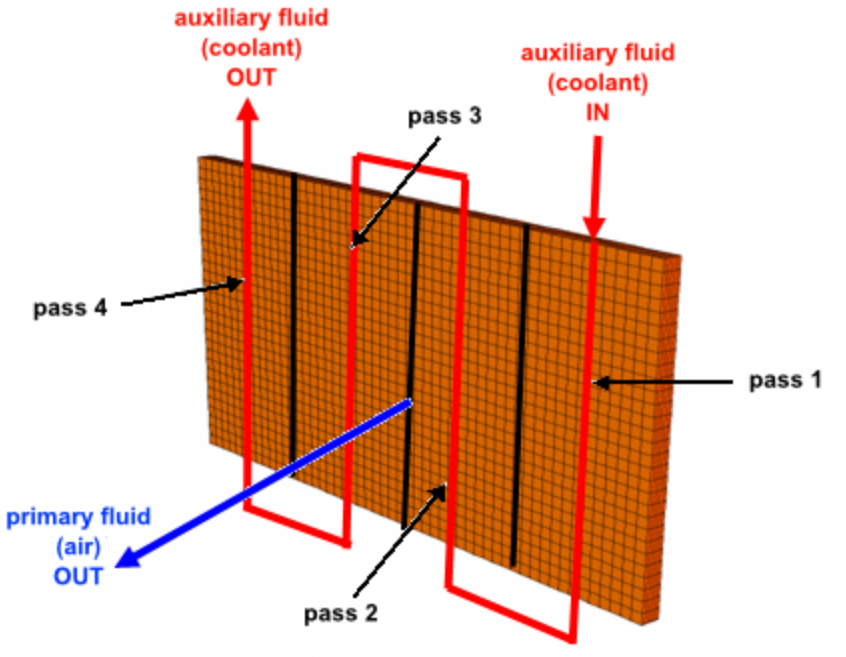
Many engineering systems, including power plants, climate control, and engine cooling systems typically contain tubular heat exchangers. However, for most engineering problems, it is impractical to model individual fins and tubes of a heat exchanger core. In principle, heat exchanger cores introduce a pressure drop to the primary fluid stream and transfer heat from or to a second fluid (for example, a coolant), referred to here as the auxiliary fluid.

Ansys Fluent provides two distinct methods of modeling a heat exchanger: the dual cell model and the macro model. These models can be used to compute the auxiliary fluid inlet temperature for a fixed heat rejection or the total heat rejection for a fixed auxiliary fluid inlet temperature. The dual cell

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

model allows the solution of the passes of the auxiliary flow on a separate mesh, that is, other than the primary fluid mesh (see Figure 16.60: An Example of a Four-Pass Heat Exchanger (p. 2280)), unlike the macro model, where the auxiliary flow passes are modeled as 1D flow.

Figure 16.60: An Example of a Four-Pass Heat Exchanger



Important:

Note that the heat exchanger models are not appropriate for modeling a cold-only flow; in such a case, you should instead use the porous media model, as described in Porous Media Conditions (p. 1333).

For theoretical information about the various heat exchanger models, refer to Heat Exchangers in the Theory Guide.

The following sections contain information about the heat exchanger models:

16.5.1. Choosing a Heat Exchanger Model
16.5.2. The Dual Cell Model
16.5.3. The Macro Heat Exchanger Models
16.5.4. Postprocessing for the Heat Exchanger Model
16.5.5. Useful Reporting TUI Commands

16.5.1. Choosing a Heat Exchanger Model

Ansys Fluent provides various options for modeling heat exchangers, each with their own features and limitations. The following instructions can help you determine which option/combination of options is the most appropriate for your problem.

1. Decide whether you want to use the dual cell model or the macro model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. The dual cell model provides the greatest flexibility with regard to the shape of the heat exchanger core and the nature of the mesh, and allows the auxiliary fluid to be highly non-uniform as it enters the core (for example, due to passing through arbitrary shaped inlet tanks). However, the dual cell model may need to be discounted as an option because of the following limitations:

   - If the heat exchanger performance data that you have is in the form of a velocity vs. effectiveness curve, the dual cell model cannot be used.
   - The dual cell model does not allow you to model phase change in the auxiliary fluid.

   If the previous limitations are not relevant for your problem, you can proceed directly to using the dual cell model, as described in Using the Dual Cell Heat Exchanger Model (p. 2283).

b. While the macro model has more restrictions than the dual cell model, it is quite suitable for a thin 3D heat exchanger core with a rectangular cross-section, where the pass-to-pass plane is perpendicular to the primary flow direction, the auxiliary flow is uniform (so it can be treated as a 1D flow), and the mesh is uniform and structured. To verify that your problem can tolerate the limitations of the macro model, see Restrictions (p. 2293).

2. If you chose to use the macro model in the previous step, you must decide whether you want to use the grouped or ungrouped version of this model. If you want to define a single heat exchanger using multiple fluid zones, or if you would like to connect the fluid flow path among multiple heat exchangers, you should use the grouped version, as described in Using the Grouped Macro Heat Exchanger Model (p. 2305). Otherwise, you can use the ungrouped version, as described in Using the Ungrouped Macro Heat Exchanger Model (p. 2294).

3. If you chose to use the macro model in the step 1., you must decide whether you want to model the heat transfer using the number-of-transfer-units (NTU) method or the simple effectiveness method. This decision largely rests on the kind of experimental data that you have: the NTU method requires that you provide the various primary fluid flow rates and auxiliary fluid flow rates and the corresponding heat transfer values; the simple effectiveness method requires that you provide the data points for a curve that defines how the effectiveness (that is, the ratio of actual rate of heat transfer between the primary and auxiliary fluids to the maximum possible rate of heat transfer) varies with the fluid velocity. Additionally, you should consider the differences between the two heat transfer models outlined in Table 16.1: NTU Model Vs. Simple Effectiveness Model (p. 2281).

Table 16.1: NTU Model Vs. Simple Effectiveness Model

	NTU Model	Simple Effectiveness Model
How many phases are allowed in the auxiliary flow?	single phase only	single phase and two phase
Can you model primary fluid-side reverse flow?	Yes	No
Can the primary fluid have a variable density?	Yes	No
Must the primary fluid heat capacity be less than the auxiliary fluid heat capacity?	No	Yes

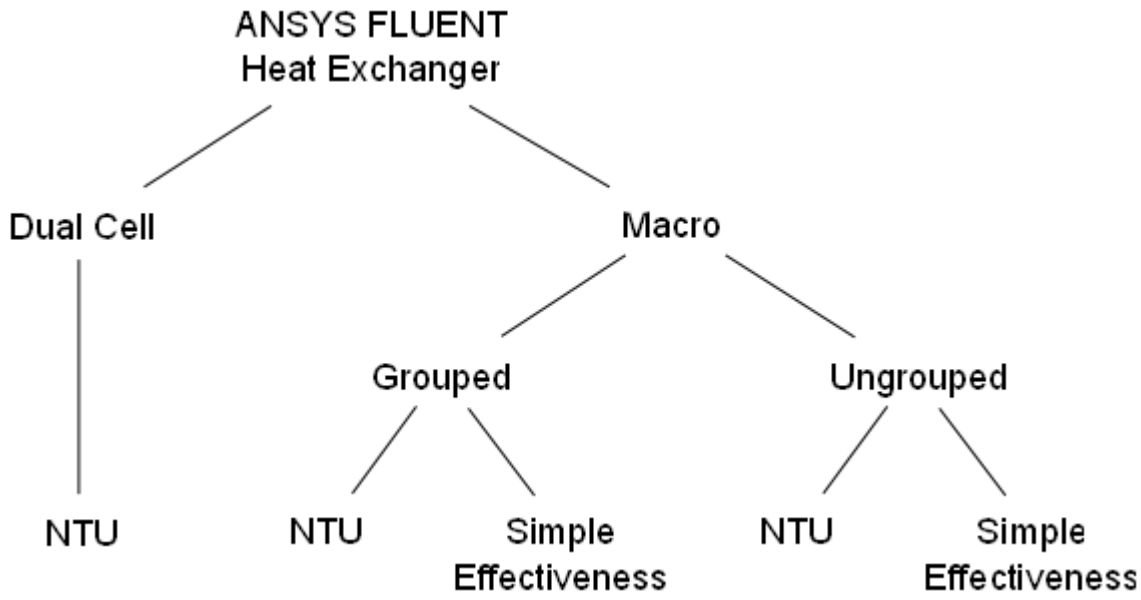
You will specify whether you want the NTU or simple effectiveness model in the **Model Data** tab of either the **Ungrouped Macro Heat Exchanger** dialog box or the **Heat Exchanger Group** dialog

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

box, depending on whether you opted for the ungrouped or grouped version of the macro model, respectively.

An overview of the options available to you when modeling heat exchangers is shown in Figure 16.61: Heat Exchanger Modeling Options (p. 2282).

Figure 16.61: Heat Exchanger Modeling Options



16.5.2. The Dual Cell Model

The dual cell heat exchanger model allows the solution of both the primary and auxiliary flow on separate co-located meshes and couples the two flows only through heat transfer at the heat exchanger core.

For theoretical information about this model, refer to The Dual Cell Model in the Theory Guide.

16.5.2.1. Restrictions

The following restrictions exist for the dual cell heat exchanger models:

- The simple effectiveness model is not available.
- The heat exchanger performance data must be in the form of heat transfer rates or number of transfer units (NTU) for auxiliary / primary fluid flow rate combinations, rather than a velocity vs. effectiveness curve.
- In the case of a heat exchanger in which the primary and auxiliary meshes are not identical, heat transfer may be non-conservative (that is, the heat lost by the hot fluid may not equal the heat gained by the cold fluid). To minimize the difference in heat transfer, the topology and size of the primary and auxiliary cells should be as similar as possible, with the ideal being one-to-one cell conductivity. Note that you can make the meshes identical by copying a zone via the `mesh/modify-zones/copy-move-cell-zone` text command.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16.5.2.2. Using the Dual Cell Heat Exchanger Model

The steps for setting up the dual cell heat exchanger model is as follows:

1. Read the mesh file.

   a. If the mesh file does not already contain overlapping heat exchanger cores for primary and auxiliary fluids, then you must create a duplicate of the zone that represents the core using the following text command:

      `mesh` → `modify-zones` → `copy-move-cell-zone`

      See Copying Cell Zones (p. 1279) for details on copying meshes.

   b. Make sure that the auxiliary fluid mesh is divided into separate zones, one for each pass.

      Domain → **Zones** → **Separate** → **Cells...**

      See Separating Cell Zones (p. 1265) for details on separating meshes.

   c. Make sure that the inlet and outlet for the heat exchanger are two separate zones.

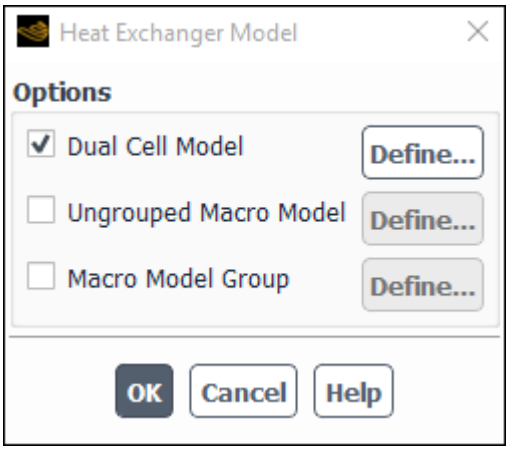
2. Enable the calculation of energy in the **Energy** dialog box.

   Setup → **Models** → **Energy** **ON**

3. Enable the **Dual Cell Model** in the **Heat Exchanger Model** dialog box and click **Define...** (Figure 16.62: The Heat Exchanger Model Dialog Box (p. 2283)).

   Setup → **Models** → **Heat Exchanger** **Edit...**

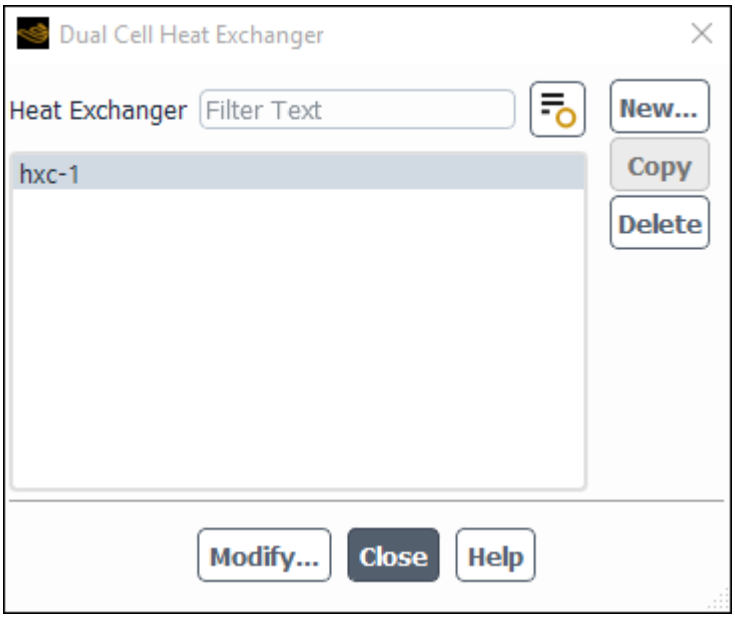
   Figure 16.62: The Heat Exchanger Model Dialog Box



4. Specify the inputs to the dual cell heat exchanger model, using the **Dual Cell Heat Exchanger** dialog box (Figure 16.63: The Dual Cell Heat Exchanger Dialog Box (p. 2284)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

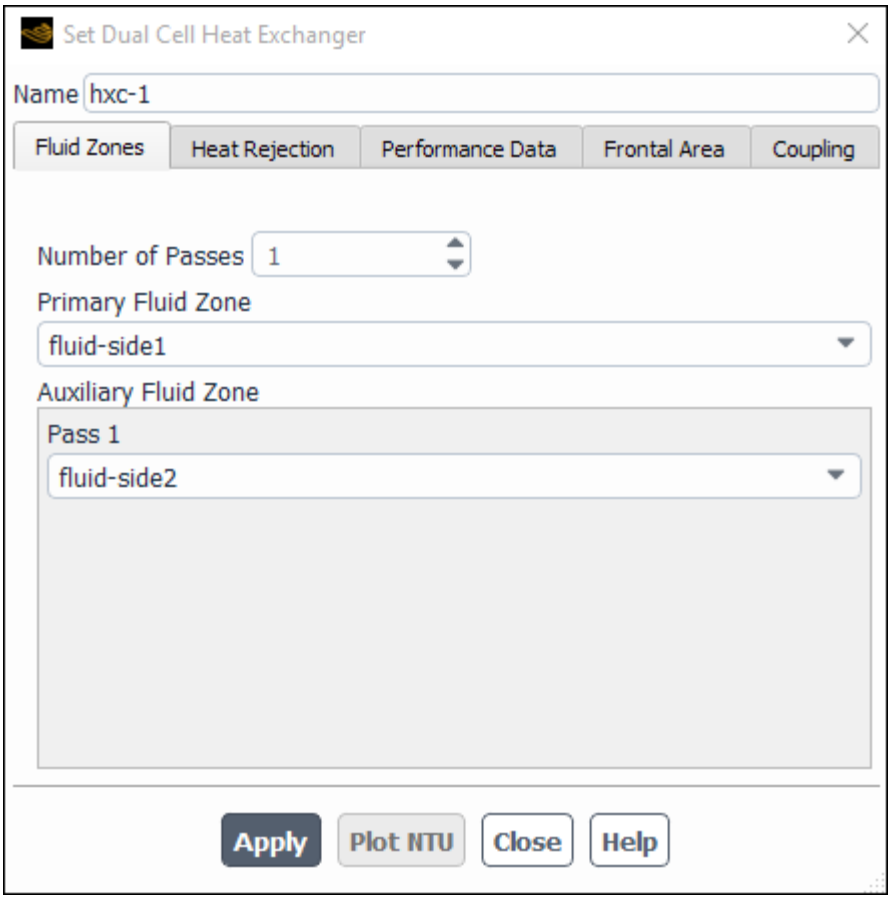
Figure 16.63: The Dual Cell Heat Exchanger Dialog Box



5. Click **New...** to define the heat exchanger. The **Set Dual Cell Heat Exchanger** dialog box will appear (Figure 16.64: The Set Dual Cell Heat Exchanger Dialog Box (p. 2285)), where you will define the heat exchanger parameters.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.64: The Set Dual Cell Heat Exchanger Dialog Box



a. Enter the heat exchanger **Name** or keep the default name. The suffix `-1` is incremented automatically on defining more than one heat exchanger.

b. In the **Fluid Zones** tab (Figure 16.64: The Set Dual Cell Heat Exchanger Dialog Box (p. 2285))

   i. Specify the **Number of Passes** of your heat exchanger.

   ii. Select the appropriate **Primary** and **Auxiliary Fluid Zone**, representing the heat exchanger core.

Important:

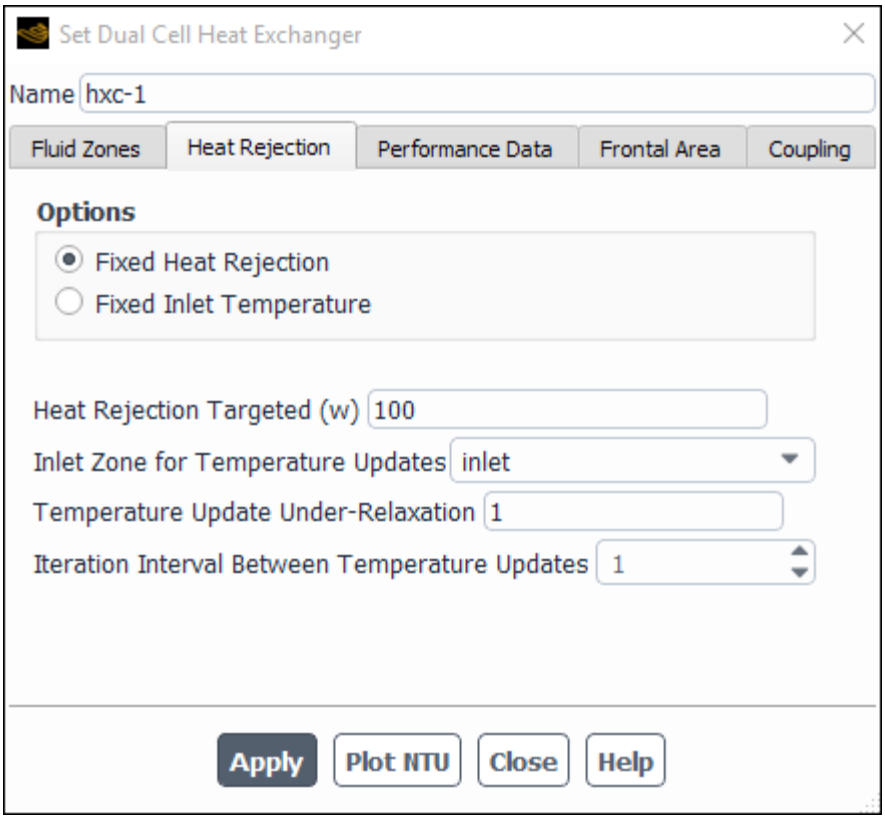
The selected zones must be overlapping in physical space.

---

c. Click the **Heat Rejection** tab (Figure 16.65: The Heat Rejection Tab (p. 2286)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.65: The Heat Rejection Tab



i. If you select **Fixed Heat Rejection**, set the inputs for the following:

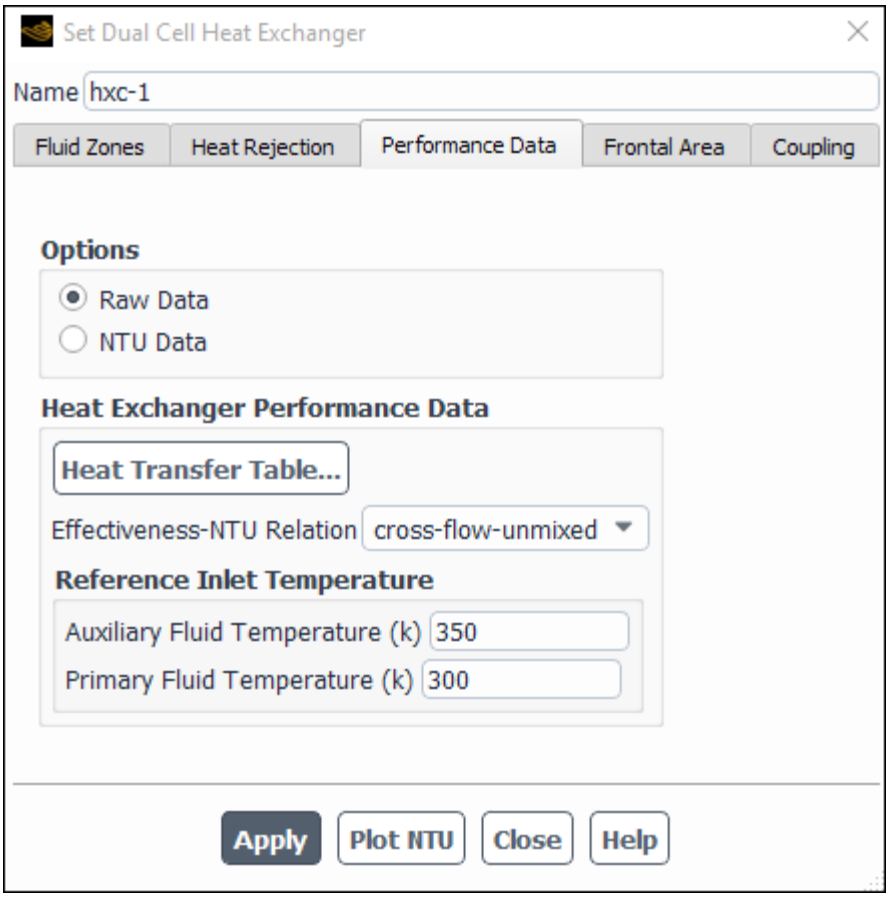
- **Heat Rejection Targeted** which is the heat rejection desired from the heat exchanger.
- **Inlet Zone for Temperature Updates** allows Ansys Fluent to change the temperature of the specified inlet zone in order to match the targeted heat rejection.
- **Temperature Update Under-Relaxation** is a factor that controls convergence.
- **Iteration Interval Between Temperature Updates** is used to control divergence.

ii. Select **Fixed Inlet Temperature** if the output desired is total heat rejection.

d. Click the **Performance Data** tab (Figure 16.66: The Performance Data Tab (p. 2287)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.66: The Performance Data Tab



You must decide whether you will select **Raw Data** or **NTU Data** from the **Options** list. Using the **Raw Data** option specifies that the effectiveness (ε) is evaluated as the ratio of the rate of heat transfer between the primary and auxiliary fluids (which you define in a table format) to the maximum possible rate of heat transfer (q_{max}) for the current flow regime (see Equation 6.7 in the Theory Guide). The calculated efficiency is then used to calculate the NTU value using the crossflow formula for each and every set of primary and auxiliary flow rates.

If the heat exchanger manufacturer used a similar approach in evaluating the NTU values, then there is no difference between the **Raw Data** and **NTU Data** options. However, if a different methodology was used by the manufacturer that accounts for some possible non-uniformity of the profile or for the partial mixing of the flow stream, then the **NTU Data** option is likely to be more accurate. Similarly, you may be able to assess more accurate NTU values for your case using tables from the relevant literature. This can even include some mixing of the oncoming flow stream, for situations where the primary flow can deviate from the direction normal to the heat exchanger tubes; this can happen in heat exchangers where the fins are either very spaced or absent. Therefore, the **NTU Data** option can be more accurate if the NTU values are gathered at non-ideal conditions that bear a close resemblance to your simulated scenario.

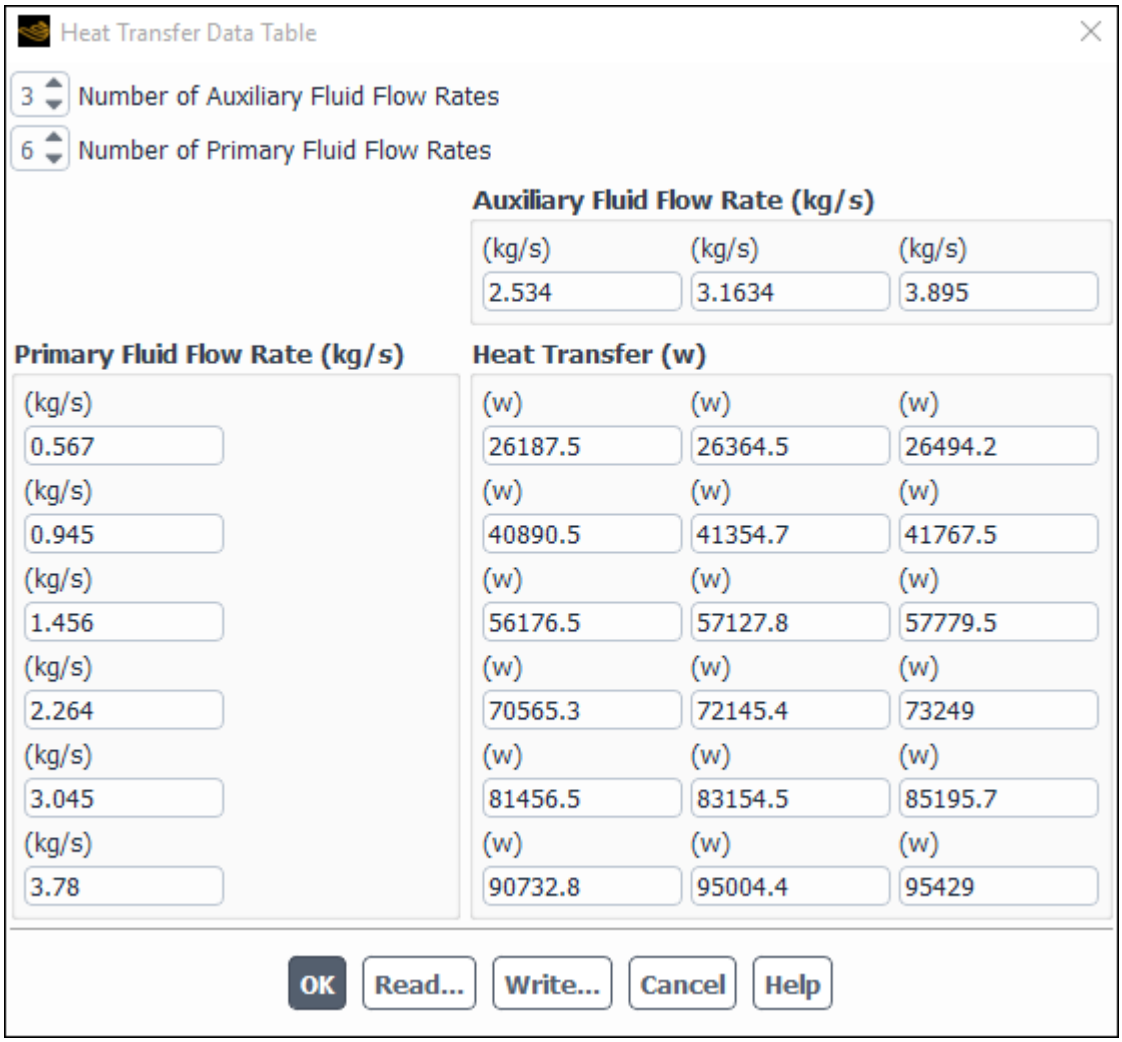
i. If you select the **Raw Data** option, then specify the following:

   - **Heat Transfer Data...** opens the Heat Transfer Data Table Dialog Box (p. 4753), where you will enter experimental data that defines how heat transfer values relate to the

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

fluid flow rates (Figure 16.67: The Heat Transfer Data Table Dialog Box (p. 2288)). See Specifying Heat Exchanger Performance Data (p. 2299) more details.

Figure 16.67: The Heat Transfer Data Table Dialog Box



- **Effectiveness-NTU Relation** computes the NTU values from the heat transfer data. Choose **cross-flow-unmixed**, **parallel-flow**, or **counter-flow**, all of which are described in NTU Relations.
- **Auxiliary Fluid Temperature** is the inlet reference temperature for the auxiliary fluid.
- **Primary Fluid Temperature** is the inlet reference temperature for the primary fluid.

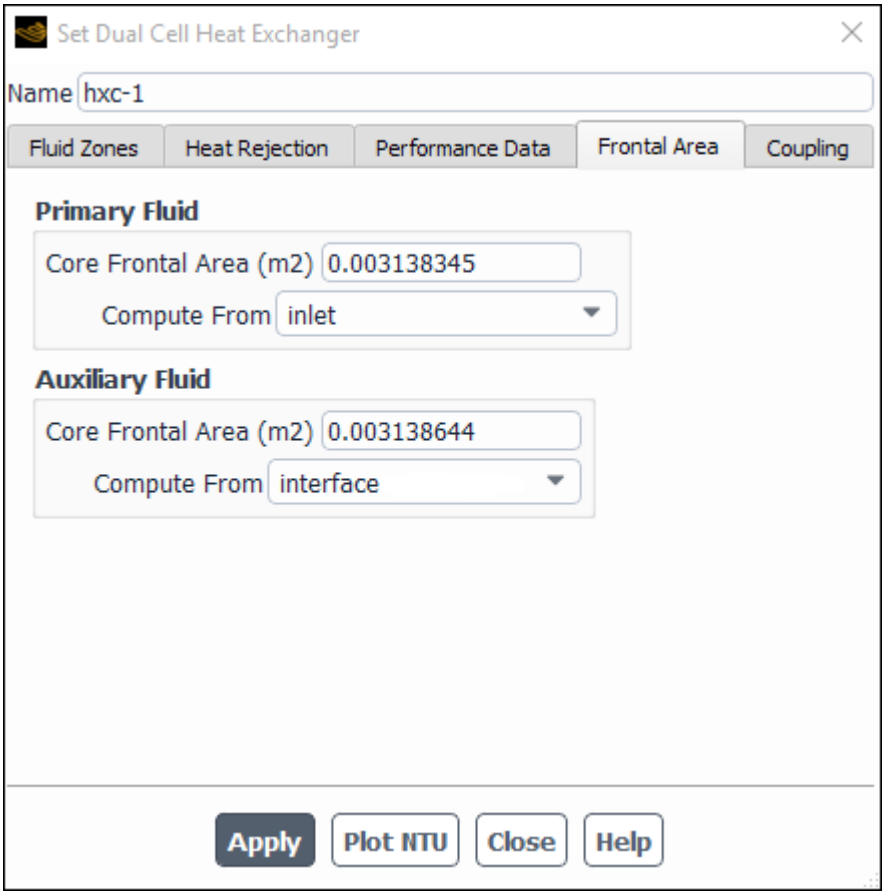
ii. If you select the **NTU Data** option, click **NTU Table...** to access the **NTU Table** dialog box. Populate this table in the same manner as described previously for the Heat Transfer Data Table Dialog Box (p. 4753). More information is available in Specifying Heat Exchanger Performance Data (p. 2299).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

e. Click the **Frontal Area** tab. You have the option to specify the **Primary** and **Auxiliary Fluid Core Frontal Area** directly, or compute the area from a surface zone, as shown in Figure 16.68: The Frontal Area Tab (p. 2289).

> **Note:**
>
> For a multipass configuration, you only need to specify the frontal area of a single pass, for a fluid which is being routed through such passes. This can be applicable either to an auxiliary fluid, primary fluid, or to both of them, depending on the flow arrangement through the core of a heat exchanger.

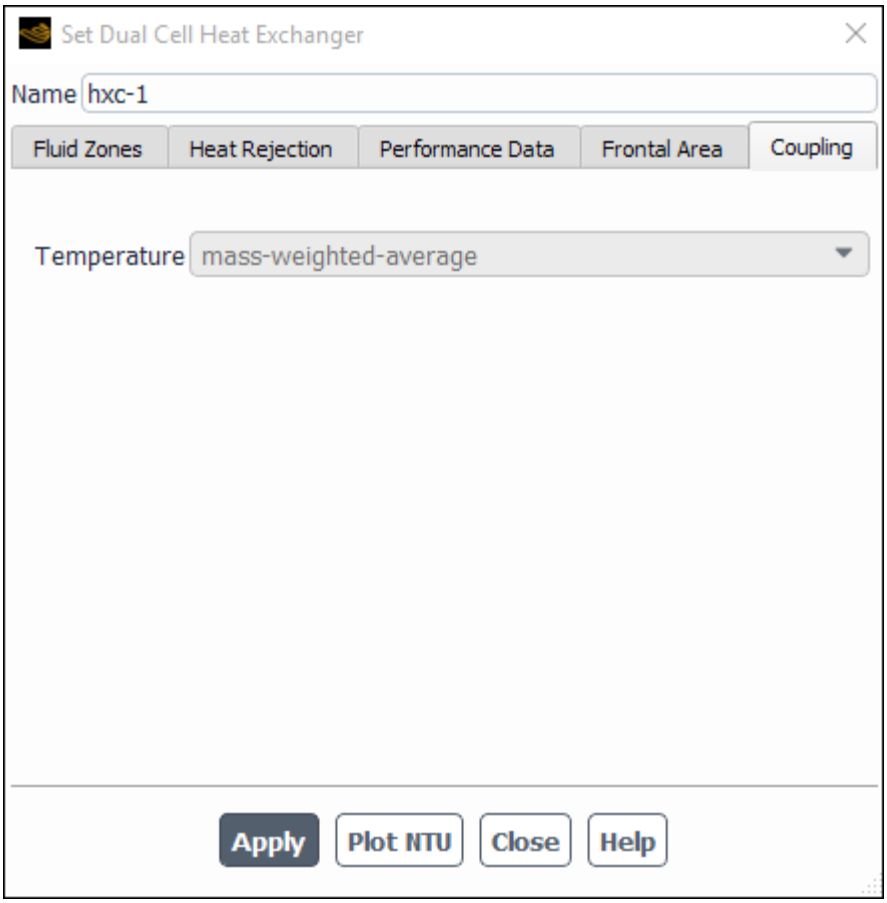
Figure 16.68: The Frontal Area Tab



f. Click the **Coupling** tab if you want to couple the heat exchanger passes (Figure 16.69: The Coupling Tab (p. 2290)).

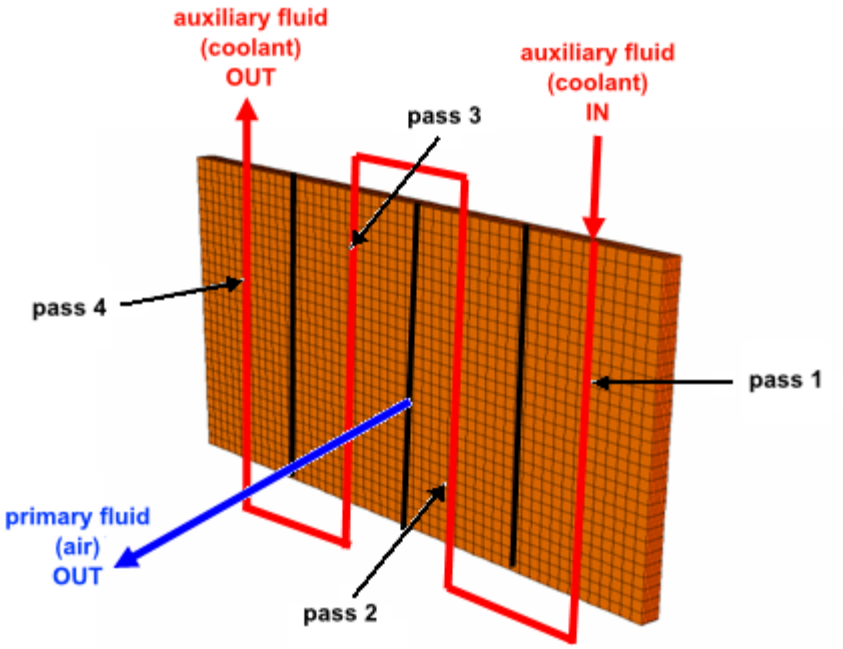
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.69: The Coupling Tab



Consider the following example illustrating the coupling of a four-pass heat exchanger.

Figure 16.70: An Example of a Four-Pass Heat Exchanger



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.70: An Example of a Four-Pass Heat Exchanger (p. 2290) shows a four-pass heat exchanger with air as the primary fluid and the coolant as the auxiliary fluid. The coolant flows through the tubes in a serpentine manner and air flows normal to the tubes, forming a cross flow pattern. To model this type of flow using the dual cell heat exchanger model, you must first generate the mesh. The mesh should contain the following:

i. A single primary cell zone.

ii. Four adjacent auxiliary cell zones, one for each pass. Each auxiliary zone should be separated from the other by a coupled or uncoupled wall. Each pass will have its own inlet and outlet zones.

iii. The primary and four auxiliary zones should overlap in physical space.

In the **Coupling** tab, **mass-weighted-average** is selected by default for the **Temperature** of the outlet of Pass 1 to the inlet of Pass 2. Similarly, the mass-weighted-average temperature of the outlet of Pass 2 will be applied at the inlet zone of Pass 3, and so on. Alternatively, you can couple the passes by using **Profiles...** in the **Boundary Conditions** task page or by using user-defined functions to define the **Temperature** in the **Thermal** tab of the relevant boundary condition dialog boxes. If you do so, make sure you select **none** from the **Temperature** drop-down list in the **Coupling** tab of the **Set Dual Heat Exchanger** dialog box.

Important:

Make sure to specify the auxiliary zones in the correct order (that is the zone for Pass 1 should be selected first, then Pass 2, and so on) in the **Fluid Zones** tab of the **Set Dual Cell Heat Exchanger** dialog box.

g. Click **Apply** to save the heat exchanger inputs.

6. To view the plot of NTU vs. primary mass flow rate for each auxiliary mass flow rate, click **Plot NTU**.

The **Plot NTU** button will plot the performance data curve for the selected heat exchanger. The performance data is supplied through the **Performance Data** tab.

When you close the **Set Dual Cell Heat Exchanger** dialog box, you will return to the **Dual Cell Heat Exchanger** dialog box, where you should now see the heat exchanger name in the **Heat Exchanger** list.

You can

- Modify the settings of heat exchanger by selecting it from the list and clicking **Modify...**.
- Copy the data of one heat exchanger to another using the **Copy** button, assuming you have more than one heat exchanger.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Delete any unwanted heat exchangers by selecting the heat exchanger from the list and clicking **Delete**.

Important:

All the inputs are copied except for the name, primary fluid zone and auxiliary fluid zone.

16.5.3. The Macro Heat Exchanger Models

To use the macro heat exchanger model, you must define one or more fluid zone(s) to represent the heat exchanger core. Typically, the fluid zone is sized to the dimension of the core itself. As part of the setup procedure, you will define the auxiliary fluid path, the number of macros, and the physical properties and operating conditions of the core (pressure drop parameters, heat exchanger effectiveness, auxiliary fluid flow rate, and so on).

Additional information about macro heat exchangers can be found in Overview of the Macro Heat Exchanger Models in the Theory Guide. For the theory behind these models, refer to Macro Heat Exchanger Model Theory in the Theory Guide.

Ansys Fluent provides two heat transfer models: the default NTU model and the simple effectiveness model. The simple effectiveness model interpolates the effectiveness from the velocity vs. effectiveness curve that you provide. For the NTU model, Ansys Fluent calculates the effectiveness, ε, from the NTU value that is calculated by Ansys Fluent from the heat transfer data provided by you in tabular format. Ansys Fluent will automatically convert this heat transfer data to a primary fluid mass flow rate vs. NTU curve (this curve will be piecewise linear). This curve will be used by Ansys Fluent to calculate the NTU for macros based on their size and primary fluid flow rate.

The NTU model provides the following features:

- The model can be used to check heat capacity for both the primary and the auxiliary fluid and takes the lesser of the two for the calculation of heat transfer.
- The model can be used to model heat transfer to the primary fluid from the auxiliary fluid, and vice versa.
- The model can be used to model primary fluid-side reverse flow.
- The model can be used with variable density of the primary fluid.
- The model can be used in either the serial or parallel Ansys Fluent solvers.
- The model can be used to make a network of heat exchangers using a heat exchanger group (Using the Grouped Macro Heat Exchanger Model (p. 2305)).
- Transient profiles can be used for the auxiliary fluid inlet temperature and for total heat rejection.
- Transient profiles can be used for auxiliary mass flow rates.

The simple effectiveness model provides the following features:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The model can be used to model heat transfer from the auxiliary fluid to the primary fluid, and vice versa.
- The auxiliary fluid properties can be a function of pressure and temperature, therefore allowing phase change of the auxiliary fluid.
- The model can be used by serial as well as parallel solvers.
- The model can be used to make a network of heat exchangers using a heat exchanger group (Using the Grouped Macro Heat Exchanger Model (p. 2305)).
- Transient profiles can be used for the auxiliary fluid inlet temperature and for total heat rejection.
- Transient profiles can be used for auxiliary mass flow rates.

For additional information, see the following sections:

16.5.3.1. Restrictions
16.5.3.2. Using the Ungrouped Macro Heat Exchanger Model
16.5.3.3. Using the Grouped Macro Heat Exchanger Model

16.5.3.1. Restrictions

The following restrictions exist for the macro heat exchanger models:

- The core must be a 3D mesh with a cross-section that is approximately rectangular in shape.
- The primary fluid streamwise direction (see Equation 6.1 in the Theory Guide) must be aligned with one of the three orthogonal axes defined by the rectangular core.
- The pass-to-pass plane must be perpendicular to the primary fluid streamwise direction.
- The two dimensions of the pass-to-pass plane can each be discretized into multiple macroscopic cells (macros), but in the direction perpendicular to this plane the macros cannot be subdivided.
- It is highly recommended that the free-form **Tet** mesh is not used in the macro heat exchanger model. Instead, evenly distributed **Hex/Wedge** cells should be used for improved accuracy and a more robust solution process.
- Flow acceleration effects are neglected in calculating the pressure loss coefficient.
- For the simple effectiveness model, the primary fluid must have a capacity rate that is less than that of the auxiliary fluid.
- Auxiliary fluid phase change cannot be modeled using the NTU model.
- The macro-based method requires that an equal number of cells reside in each macro of equal size and shape.
- The auxiliary fluid flow is assumed to be 1D.
- The pass width has to be uniform.
- Accuracy is not guaranteed when the mesh is not structured or layered.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Accuracy is not guaranteed when there is upstream diffusion of temperature at the inlet/outlet of the core.
- Non-conformal meshes cannot be attached to the inlet/outlet of the core. An extra layer has to be created to avoid it.

16.5.3.2. Using the Ungrouped Macro Heat Exchanger Model

The heat exchanger model settings may be written into and read from the boundary conditions file (Reading and Writing Boundary Conditions (p. 971)) using the text commands, `file/write-settings` and `file/read-settings`, respectively. Otherwise, the steps for setting up the ungrouped macro heat exchanger model is as follows:

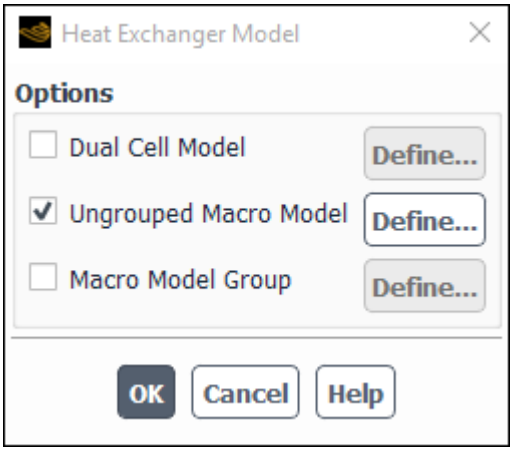
1. Read the mesh and make sure that the inlet and outlet for the heat exchanger are two separate zones.
2. Enable the calculation of energy in the Energy Dialog Box (p. 4714).

   Setup → Models → Energy → ON

3. Enable the **Ungrouped Macro Model** option and click the **Define...** button in the Heat Exchanger Model Dialog Box (p. 4748) (Figure 16.71: The Heat Exchanger Model Dialog Box (p. 2294)) to access the **Ungrouped Macro Heat Exchanger** dialog box.

   Setup → Models → Heat Exchanger → Edit...

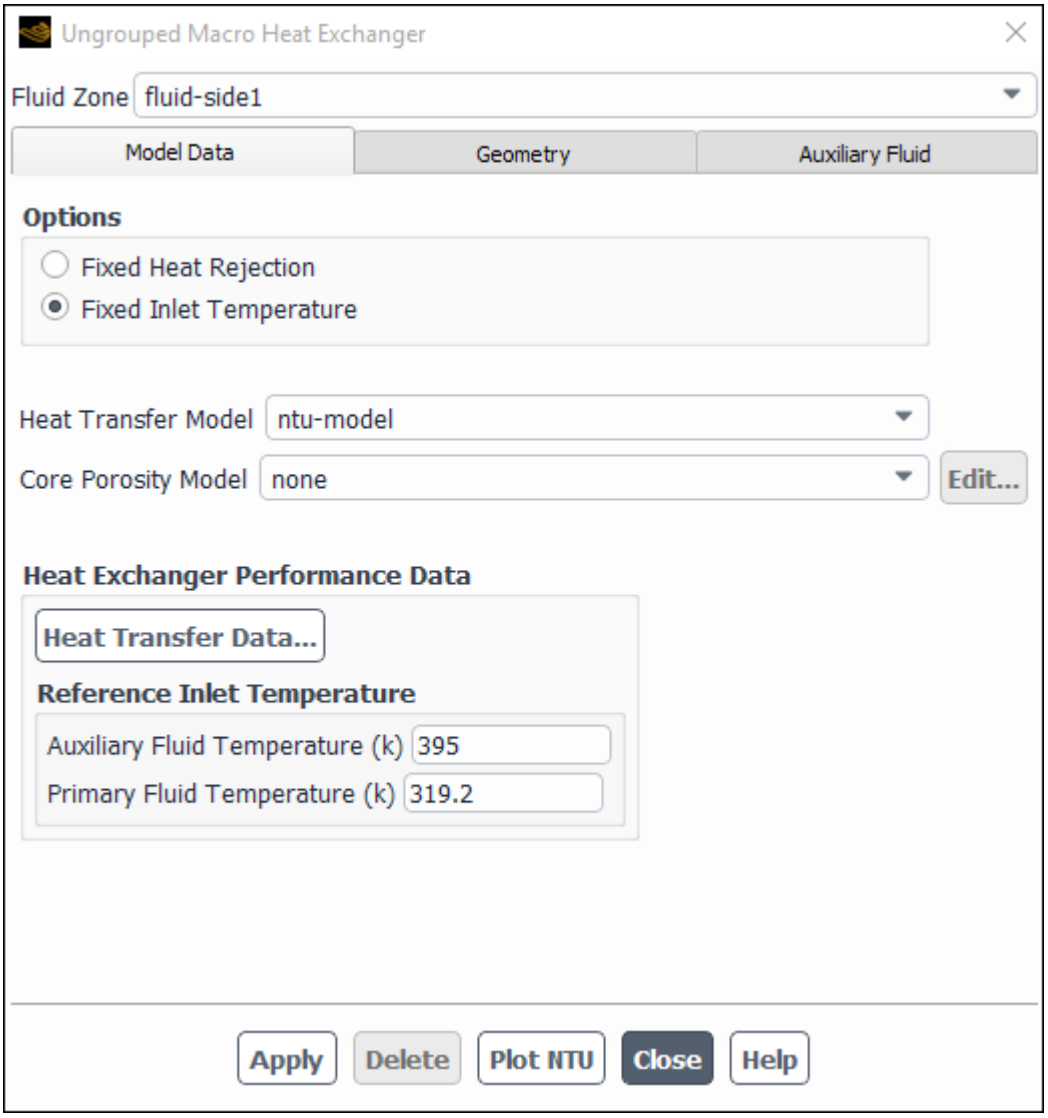
   Figure 16.71: The Heat Exchanger Model Dialog Box

4. Specify the heat exchanger inputs in the **Ungrouped Macro Heat Exchanger** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.72: The Ungrouped Macro Heat Exchanger Dialog Box Displaying the Model Data Tab



a. In the **Fluid Zone** drop-down list, select the fluid zone representing the heat exchanger core.

b. Under the **Model Data** tab, choose **Fixed Heat Rejection** or **Fixed Inlet Temperature**, as required (Figure 16.72: The Ungrouped Macro Heat Exchanger Dialog Box Displaying the Model Data Tab (p. 2295)).

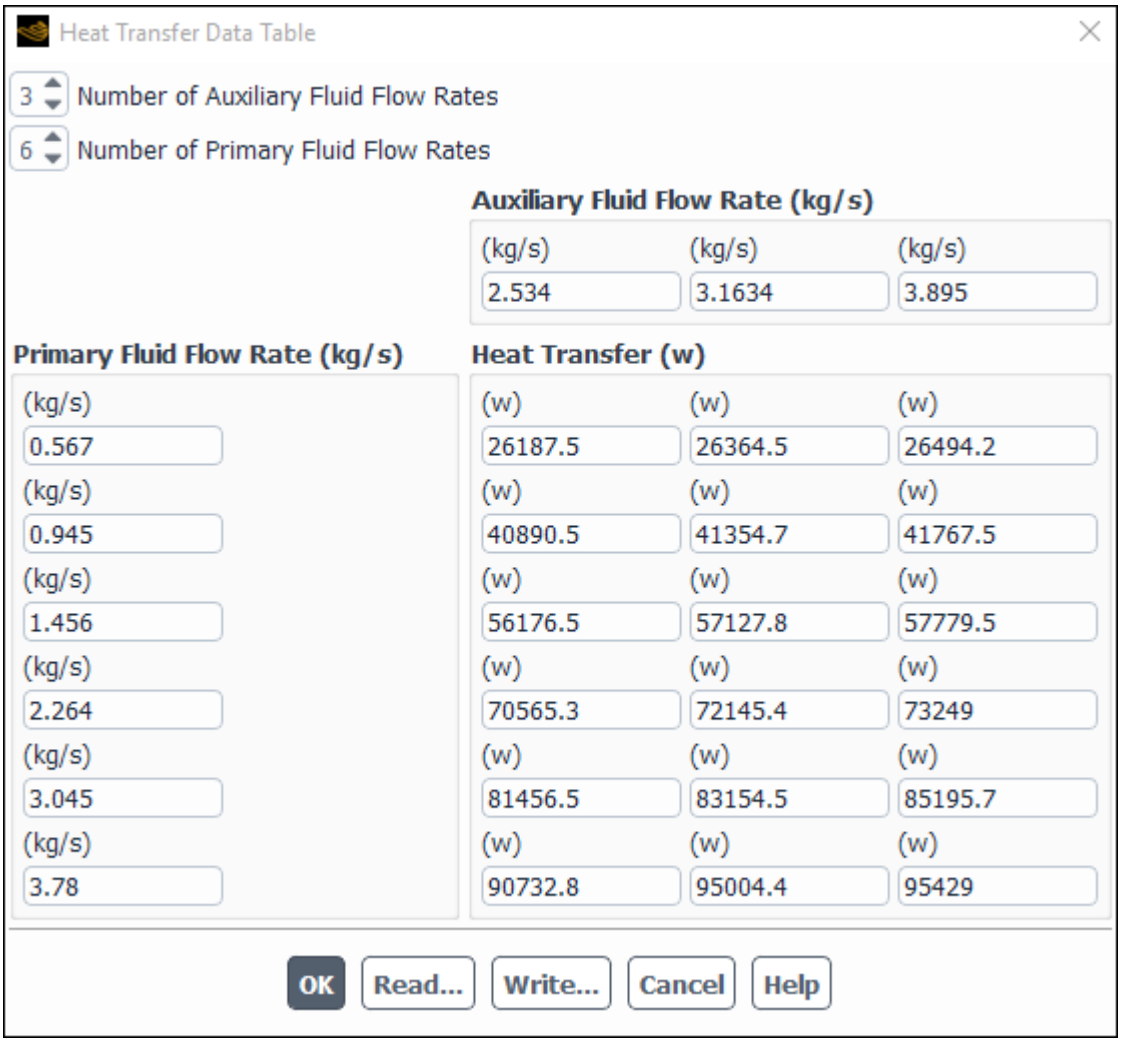
c. Specify the **Heat Transfer Model** as either the default **ntu-model** or the **simple-effectiveness-model**. See step 3 in Choosing a Heat Exchanger Model (p. 2280) for details about the differences between these two models.

d. Specify the **Core Porosity Model** if you want Ansys Fluent to use the pressure loss coefficient function to automatically compute (and update) the porous media coefficients in the cell zones condition dialog box, as described in Streamwise Pressure Drop in the Theory Guide. More information is available in Setting the Pressure-Drop Parameters and Effectiveness (p. 2303).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

e. If the **ntu-model** is chosen, a **Heat Transfer Data...** button will appear under **Heat Exchanger Performance Data**. Clicking the **Heat Transfer Data...** button will open the Heat Transfer Data Table Dialog Box (p. 4753), where you will enter experimental data that defines how heat transfer values relate to the fluid flow rates (Figure 16.67: The Heat Transfer Data Table Dialog Box (p. 2288)). See Specifying Heat Exchanger Performance Data (p. 2299) more details.

Figure 16.73: The Heat Transfer Data Table Dialog Box for the NTU Model



f. Enter the **Auxiliary Fluid Temperature** and the **Primary Fluid Temperature** for the **ntu-model**. These are the fixed inlet temperatures at which the test was performed to obtain the heat transfer data.

g. If the **simple-effectiveness-model** is chosen, then clicking the **Velocity Effectiveness Curve...** button, under the **Heat Exchanger Performance Data**, allows you to set the velocity and corresponding effectiveness for each point. More information is available in Specifying Heat Exchanger Performance Data (p. 2299).

h. In the **Geometry** tab, define the heat exchanger macros using the **Number of Passes**, the **Number of Rows/Pass**, and the **Number of Columns/Pass** fields. The **Number of Rows/Pass** is along the auxiliary flow direction (height) and the **Number of Columns/Pass** is defined in the pass-to-pass (width) direction. Also, enter the **Auxiliary Fluid Inlet Direction** and

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Pass-to-Pass Direction. You may want to snap the plane tool to either the inlet or outlet of the heat exchanger using the **Update from Plane Tool**. Note that the plane tool must be attached exactly and oriented so that its green arrow points in the auxiliary flow direction, and its blue arrow points in the pass-to-pass direction.

Important:

To attach the plane tool exactly, exact coordinates (printed in the console by probing) of the three corner nodes must be entered in the plane tool (x0, x1, x2) in a specific order. Also, note that x0 to x1 is the auxiliary flow direction and x1 to x2 is the pass-to-pass direction.

More information is available in Specifying the Auxiliary Fluid Inlet and Pass-to-Pass Directions (p. 2300) and Defining the Macros (p. 2300).

Figure 16.74: The Ungrouped Macro Heat Exchanger Dialog Box Displaying the Geometry Tab

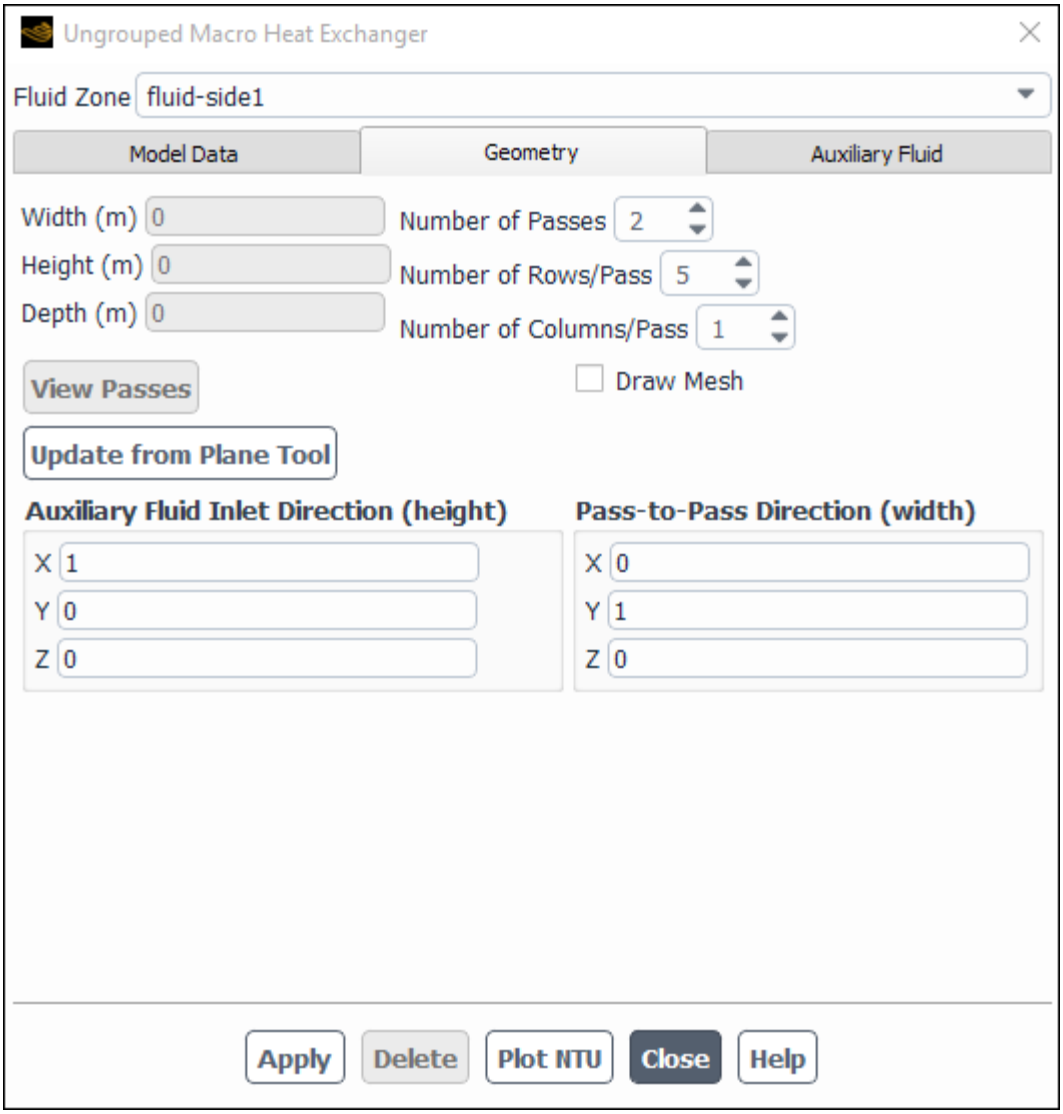
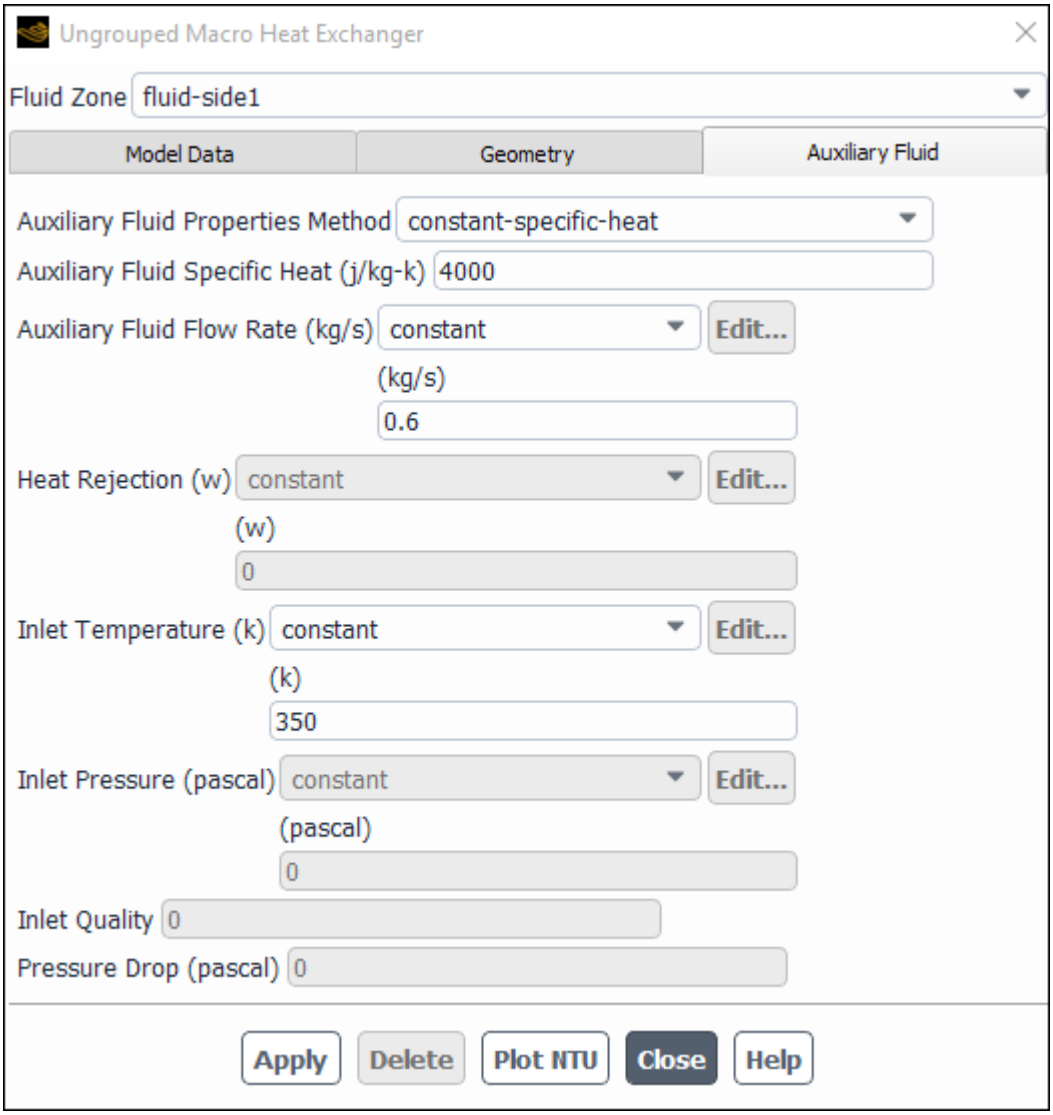


Figure 16.75: The Ungrouped Macro Heat Exchanger Dialog Box Displaying the Auxiliary Fluid Tab



i. In the **Auxiliary Fluid** tab, specify the **Auxiliary Fluid Properties Method**, either as a **constant-specific-heat** or as a **user-defined-enthalpy**.

j. **Auxiliary Fluid Flow Rate**, **Heat Rejection**, **Inlet Temperature**, and **Inlet Pressure** can be provided as a **constant**, **polynomial** or **piecewise-linear** profile that is a function of time. If **user-defined-enthalpy** is selected as the **Auxiliary Fluid Properties Method**, you will need to specify the **Inlet Quality** and the **Pressure Drop**. More information is available in Specifying the Auxiliary Fluid Properties and Conditions (p. 2302).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

k. Click **Apply** in the **Ungrouped Macro Heat Exchanger** dialog box to save all the settings. Once you click the **Apply** button, the NTU matrix will be computed from the raw data. Therefore, make sure you click **Apply** at the very end of your setup.

Important:

When you click **Apply**, look for any error or warning message in the Ansys Fluent console. Some of the common errors you may see displayed are due to the NTU computations not converging. In such cases, check that

- you have entered the data correctly
- the values of the data are reasonable
- the operating condition for the auxiliary fluid flow rate is not too far from the range of the heat transfer data

Other error messages you may encounter may be due to macros not getting any cells assigned to it. In such cases, make sure that

- the heat exchanger core is rectangular
- the directions are correct
- you are using uniformly spaced cells in both directions
- the mesh is either a hexahedra or wedge and is structured

---

l. Repeat steps (a)–(k) for any other heat exchanger fluid zones.

To use multiple fluid zones to define a single heat exchanger, or to connect the auxiliary fluid flow path among multiple heat exchangers, see Using the Grouped Macro Heat Exchanger Model (p. 2305).

For additional information, see the following sections:

16.5.3.2.1. Selecting the Zone for the Heat Exchanger
16.5.3.2.2. Specifying Heat Exchanger Performance Data
16.5.3.2.3. Specifying the Auxiliary Fluid Inlet and Pass-to-Pass Directions
16.5.3.2.4. Defining the Macros
16.5.3.2.5. Specifying the Auxiliary Fluid Properties and Conditions
16.5.3.2.6. Setting the Pressure-Drop Parameters and Effectiveness

16.5.3.2.1. Selecting the Zone for the Heat Exchanger

Choose the fluid zone for which you want to define a heat exchanger in the **Fluid Zone** drop-down list.

16.5.3.2.2. Specifying Heat Exchanger Performance Data

Based on the heat transfer model you choose in the **Model Data** tab, some performance data must be entered for the heat exchanger.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **ntu-model**: For the **ntu-model** you will provide the heat transfer for different primary and auxiliary fluid flow rates. Click the **Heat Transfer Data...** button to open up a tabular dialog box. Set the number of auxiliary flow rates and primary fluid flow rates. The dialog box will resize itself accordingly. You will need to provide various primary fluid flow rates and auxiliary fluid flow rates and the corresponding heat transfer values. You may write this data to a file that can be read later.

- **simple-effectiveness-model**: For this model, you will need to provide velocity versus effectiveness data. To provide this you can click the **Velocity Effectiveness Curve...** button. This will open up a tabular dialog box. In this dialog box, you can set the number of points in the curve, then you can provide velocities and corresponding effectiveness values (note that the effectiveness values must be within the range of 0–1). This data can be written to a file and read back.

16.5.3.2.3. Specifying the Auxiliary Fluid Inlet and Pass-to-Pass Directions

To define the auxiliary fluid direction and flow path, you will specify direction vectors for the **Auxiliary Fluid Inlet Direction** and the **Pass-to-Pass Direction** in the **Geometry** tab. Figure 16.76: 1x4x3 Macros (p. 2301) shows these directions relative to the macros.

For some problems in which the principal axes of the heat exchanger core are not aligned with the coordinate axes of the domain, you may not know the auxiliary fluid inlet and pass-to-pass direction vectors a priori. In such cases, you can use the plane tool as follows to help you to determine these direction vectors.

1. Drag the plane tool onto the boundary of the heat exchanger core. Refer to Plane Surfaces (p. 3830) for additional information on plane surfaces and using the plane tool.

2. Translate and rotate the axes of the tool appropriately until they are aligned with the principal directions of the heat exchanger core. The depth direction is determined by the red axis, the height direction by the green axis, and the width direction by the blue axis.

3. Once the axes are aligned, click the **Update from Plane Tool** button in the **Ungrouped Macro Heat Exchanger** dialog box. The directional vectors will be set automatically. (Note that the **Update from Plane Tool** button will also set the height, width, and depth of the heat exchanger core.)

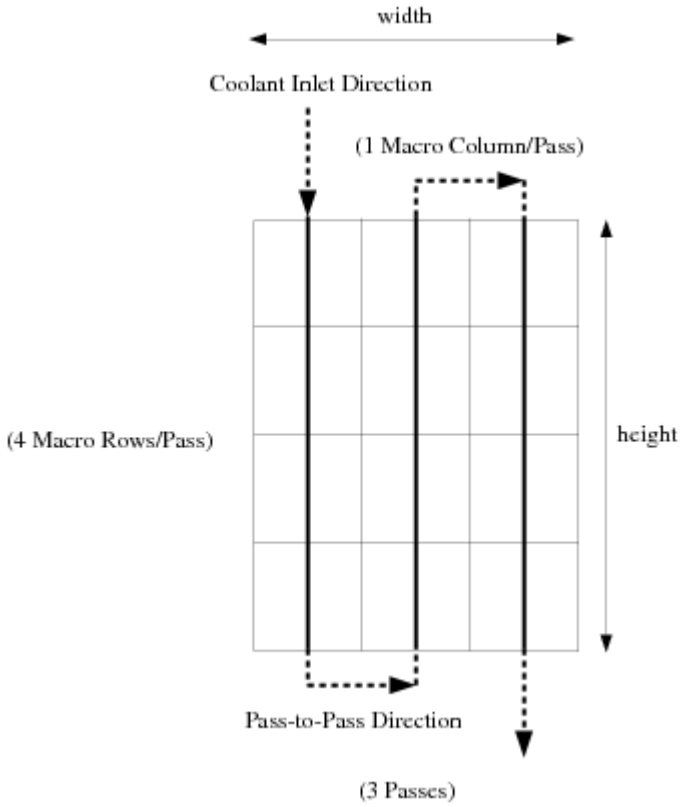
   Open the Plane Surface Dialog Box (p. 5580) to make the plane tool appear.

16.5.3.2.4. Defining the Macros

As discussed in The Macro Heat Exchanger Models (p. 2292), the fluid zone representing the heat exchanger core is split into macros. Macros are constructed based on the specified number of passes, the number of macro rows per pass, the number of macro columns per pass, and the corresponding auxiliary fluid inlet and pass-to-pass directions (see Figure 16.76: 1x4x3 Macros (p. 2301)). Macros are numbered from 0 to $(n-1)$ in the direction of auxiliary fluid flow, where n is the number of macros.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.76: 1x4x3 Macros



In the **Ungrouped Macro Heat Exchanger** dialog box, in the **Geometry** tab, specify the **Number of Passes**, the **Number of Rows/Pass**, and the **Number of Columns/Pass**. The model will automatically extrude the macros to the depth of the heat exchanger core. For each pass, the **Number of Rows/Pass** are defined in the direction of the auxiliary flow inlet direction and the **Number of Columns/Pass** are defined in the direction of the pass-to-pass direction.

Important:

The **Number of Rows/Pass**, as well as the **Number of Columns/Pass** must be divisible by the number of cells in their respective directions. For example, if you have 50 cells in the auxiliary flow direction, you can use 25 for the **Number of Rows/Pass**, but you should not use 26 or 24. If you have 51 cells in that direction, you can only use 51 for the **Number of Rows/Pass**. The same holds true for the other direction.

16.5.3.2.4.1. Viewing the Macros

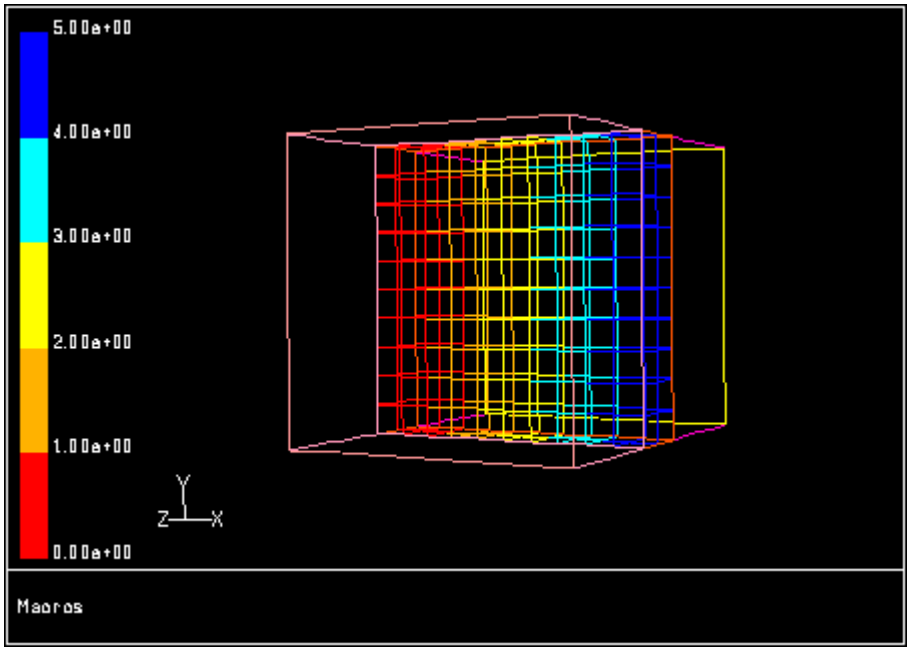
You can view the auxiliary fluid path by displaying the macros. To view the macros for your specified **Number of Passes**, **Number of Rows/Pass**, and **Number of Columns/Pass**, click the **Apply** button at the bottom of the dialog box, then click the **View Passes** button to display it. To appropriately visualize the passes in the graphics window, you should ensure **Edges** are visible from the Mesh Display Dialog Box (p. 4688), by selecting **Draw Mesh**. The path of the auxiliary fluid is color-coded in the display: macro 0 is red and macro $n-1$ is blue.

For some problems, especially complex geometries, you may want to include portions of the computational-domain mesh in your macros plot as spatial reference points. For example, you

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

may want to show the location of an inlet and an outlet along with the macros. This is accomplished by enabling the **Draw Mesh** option. The Mesh Display Dialog Box (p. 4688) will appear automatically when you enable the **Draw Mesh** option, where you can set the mesh display parameters. When you click the **View Passes** button in the **Ungrouped Macro Heat Exchanger** dialog box, the mesh display, as defined in the **Mesh Display** dialog box, will be included in the macros plot (see Figure 16.77: Mesh Display With Macros (p. 2302)).

Figure 16.77: Mesh Display With Macros



16.5.3.2.5. Specifying the Auxiliary Fluid Properties and Conditions

To define the auxiliary fluid properties and conditions, you will specify the **Auxiliary Fluid Flow Rate** ($\dot{m}$) in the **Auxiliary Fluid** tab. The properties of the auxiliary fluid can be specified using the **Auxiliary Fluid Properties Method** drop-down list. You can choose a **constant-specific-heat** (c_p) and set the value in the **Auxiliary Fluid Specific Heat** field below, or as a user-defined function for the enthalpy using the **user-defined-enthalpy** option and selecting the corresponding UDF from the **Auxiliary Fluid Enthalpy UDF** drop-down list.

The function should return a single value depending on the index:

- Enthalpy for given values of temperature, pressure, and quality.
- Temperature for given values of enthalpy and pressure
- Specific heat for given values of temperature and pressure

The user-defined function should be of type

```
DEFINE_SOURCE(udf_name, cell_t c, Thread *t, real d[n], int index).
```

where `n` in the expression `d[n]` would be `0` for temperature, `1` for pressure, or `2` for quality. The variable `index` is `0` for enthalpy, `1` for temperature, or `2` for specific heat. This user-defined function should return

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
real value; /* (temperature or enthalpy or Cp depending on index). */
```

- If you want Ansys Fluent to compute the auxiliary fluid inlet temperature for a specified heat rejection, follow the steps below:

  1. Enable the **Fixed Heat Rejection** option in the **Model Data** tab.
  2. Specify the **Heat Rejection** (q_{total} in Equation 6.15 in the Theory Guide) in the **Auxiliary Fluid** tab.
  3. Specify the **Initial Temperature**, which will be used by Ansys Fluent as an initial guess for the inlet temperature ($T_{in,auxiliary}$ in Equation 6.11 and Equation 6.13 in the Theory Guide).

- If you want Ansys Fluent to compute the total heat rejection of the core for a given inlet auxiliary fluid temperature, follow the steps below:

  1. Enable the **Fixed Inlet Temperature** option in the **Model Data** tab.
  2. Specify the **Inlet Temperature** ($T_{in,auxiliary}$ in Equation 6.11 and Equation 6.13 in the Theory Guide) in the **Auxiliary Fluid** tab.

- If you enable the **User Defined Enthalpy** option under the **Auxiliary Fluid Properties Method**, you must also specify the **Inlet Pressure** (p_{in} in Equation 6.21 in the Theory Guide) and **Inlet Quality** (x in Equation 6.20 in the Theory Guide).

16.5.3.2.6. Setting the Pressure-Drop Parameters and Effectiveness

The pressure drop parameters and effectiveness define the **Core Porosity Model**. If you would like Ansys Fluent to set the porosity of this a heat exchanger zone using a particular core model, you can select the appropriate model. This will automatically set the porous media inputs. There are three ways to specify the **Core Porosity Model** parameters:

- Use the values in Ansys Fluent's default model.
- Define a new core porosity model with your own values.
- Read a core porosity model from an external file.

If you do not choose a core porosity model, you will need to set the porosity parameters in the cell zone conditions dialog box for the heat exchanger zone(s). To do this, follow the procedures described in User Inputs for Porous Media (p. 1343).

The models you define will be saved in the case file.

16.5.3.2.6.1. Using the Default Core Porosity Model

Ansys Fluent provides a default model for a typical heat exchanger core. The **default-model** core porosity model is a list of constant values from the **Ungrouped Macro Heat Exchanger** dialog box. These constants are used for setting the porous media parameters. To use these values, simply retain the selection of **default-model** in the **Core Porosity Model** drop-down list in the **Ungrouped Macro Heat Exchanger** dialog box. (You can view the default parameters as described below.)

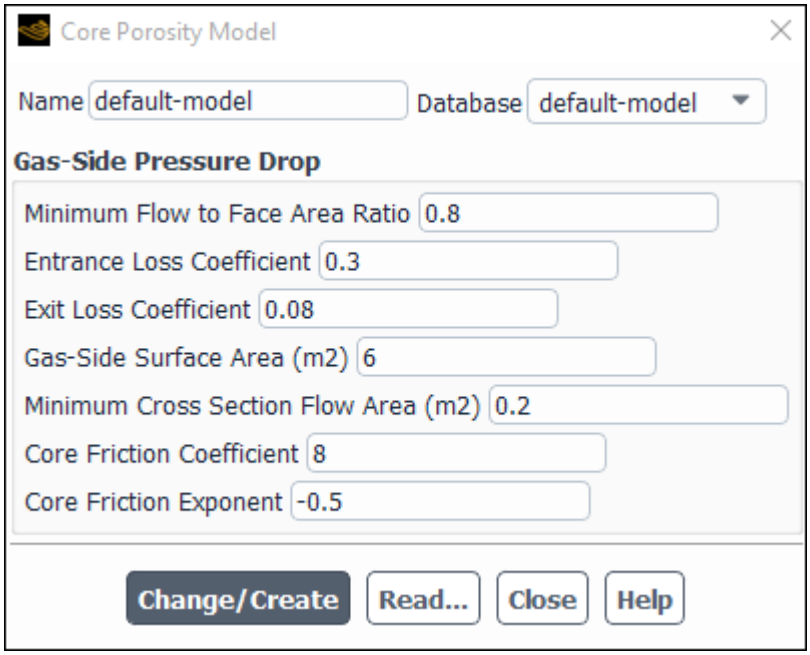
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16.5.3.2.6.2. Defining a New Core Porosity Model

If you want to define pressure-drop and effectiveness parameters that are different from those in the default core porosity model, you can create a new model. The steps for creating a new model are as follows:

1. Click the **Edit...** button to the right of the **Core Porosity Model** drop-down list, for which **default-model** should have been selected. This will open the Core Porosity Model Dialog Box (p. 4761) (Figure 16.78: The Core Porosity Model Dialog Box (p. 2304)).

   Figure 16.78: The Core Porosity Model Dialog Box

2. Enter the name of your new model in the **Name** box at the top of the dialog box.

3. Under **Gas-Side Pressure Drop**, specify the following parameters used in Equation 6.2 in the Theory Guide:

   - **Minimum Flow to Face Area Ratio** (σ)
   - **Entrance Loss Coefficient** (K_c)
   - **Exit Loss Coefficient** (K_e)
   - **Gas-Side Surface Area** (A)
   - **Minimum Cross Section Flow Area** (A_c)

   and the **Core Friction Coefficient** and **Core Friction Exponent** (a and b, respectively, in Equation 6.3 in the Theory Guide).

4. Click the **Change/Create** button. This will add your new model to the database.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16.5.3.2.6.3. Reading Heat Exchanger Parameters from an External File

You can read parameters for your **Core Porosity Model** from an external file. A sample file is shown below:

```
("modelname"
 (0.73 0.43 0.053 5.2 0.33 9.1 0.66))
```

The first entry in the file is the name of the model (for example, `modelname`). The second set of numbers contains the gas-side (primary-side) pressure drop parameters:

(σ K_c K_e A A_c a b)

To read an external heat exchanger file, you will follow these steps:

1. In the **Core Porosity Model** dialog box, click the **Read...** button.
2. In the resulting **Select File** dialog box, specify the **HXC Parameters File** name and click **OK**. Ansys Fluent will read the core porosity model parameters, and add the new model to the database.

16.5.3.2.6.4. Viewing the Parameters for an Existing Core Model

To view the parameters associated with a core porosity model that you have already defined, select the model name in the **Database** drop-down list (in the **Core Porosity Model** dialog box). The values for that model from the database will be displayed in the **Core Porosity Model** dialog box.

16.5.3.3. Using the Grouped Macro Heat Exchanger Model

To define a single heat exchanger that uses multiple fluid zones, or to connect the auxiliary fluid flow path among multiple heat exchangers, you can use heat exchanger groups. To use heat exchanger groups, perform the following steps:

1. Read the mesh and make sure that the inlet and outlet for the heat exchanger are two separate zones.
2. Enable the calculation of energy in the Energy Dialog Box (p. 4714).

   Setup → Models → Energy → ON

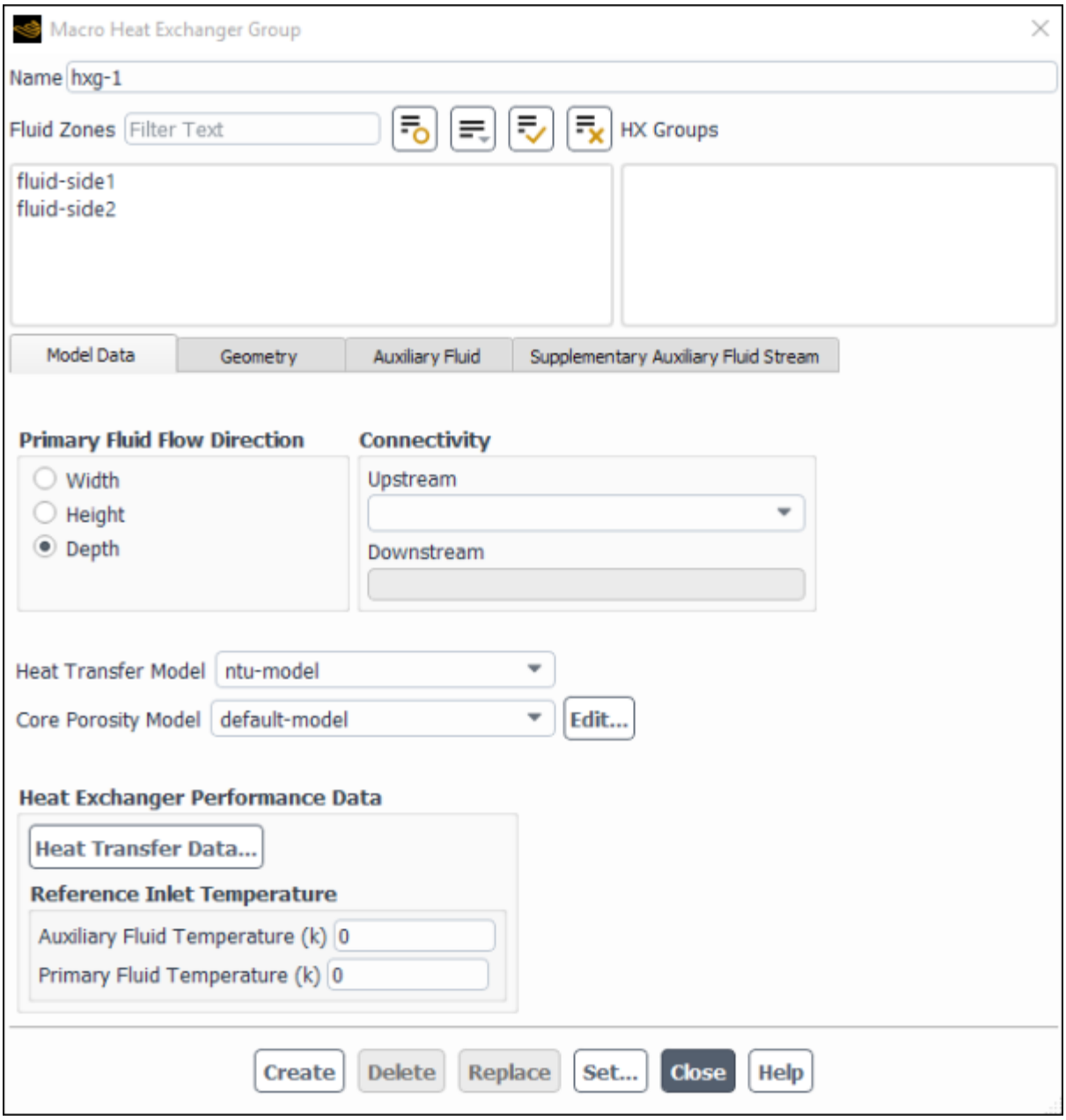
3. Enable the **Macro Model Group** option and click the **Define...** button in the Heat Exchanger Model Dialog Box (p. 4748) to access the **Macro Heat Exchanger Group** dialog box.

   Setup → Models → Heat Exchanger → Edit...

4. Specify the inputs to the heat exchanger group in the **Macro Heat Exchanger Group** dialog box (Figure 16.79: The Macro Heat Exchanger Group Dialog Box (p. 2306)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.79: The Macro Heat Exchanger Group Dialog Box



a. Enter the **Name** of the heat exchanger group.

b. Under **Fluid Zones**, select the fluid zones that you want to define in the heat exchanger group (Selecting the Fluid Zones for the Heat Exchanger Group (p. 2311)).

c. Click the **Model Data** tab.

   i. Under **Primary Fluid Flow Direction**, specify the primary fluid flow direction as either **Width**, **Height**, or **Depth**.

   ii. Under **Connectivity**, select the **Upstream** heat exchanger group if such a connection exists (see Selecting the Upstream Heat Exchanger Group (p. 2311)).

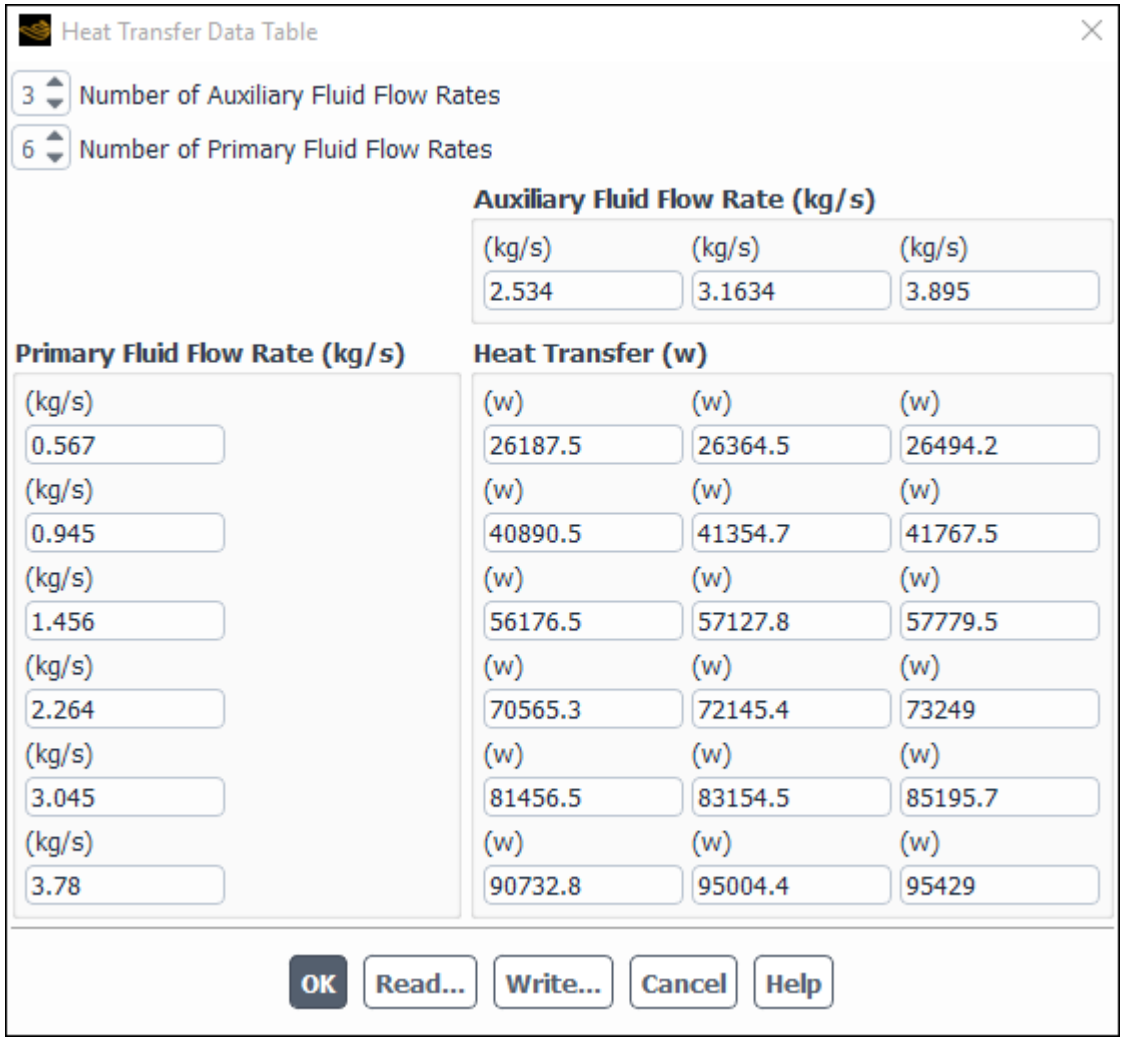
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

iii. In the **Heat Transfer Model** drop-down list, choose either the **ntu-model** or the **simple-effectiveness-model**. See step 3 in Choosing a Heat Exchanger Model (p. 2280) for details about the differences between these two models.

iv. From the **Core Porosity Model** drop-down list, specify the core model that should be used to calculate the porous media parameters for the zones in the group. More information is available in Setting the Pressure-Drop Parameters and Effectiveness (p. 2303).

v. If the **ntu-model** is chosen, a **Heat Transfer Data...** button will appear under **Heat Exchanger Performance Data**. Clicking the **Heat Transfer Data...** button will open the Heat Transfer Data Table Dialog Box (p. 4753), where you will enter experimental data that defines how heat transfer values relate to the fluid flow rates (Figure 16.67: The Heat Transfer Data Table Dialog Box (p. 2288)). See Specifying Heat Exchanger Performance Data (p. 2299) more details.

Figure 16.80: The Heat Transfer Data Table Dialog Box for the NTU Model



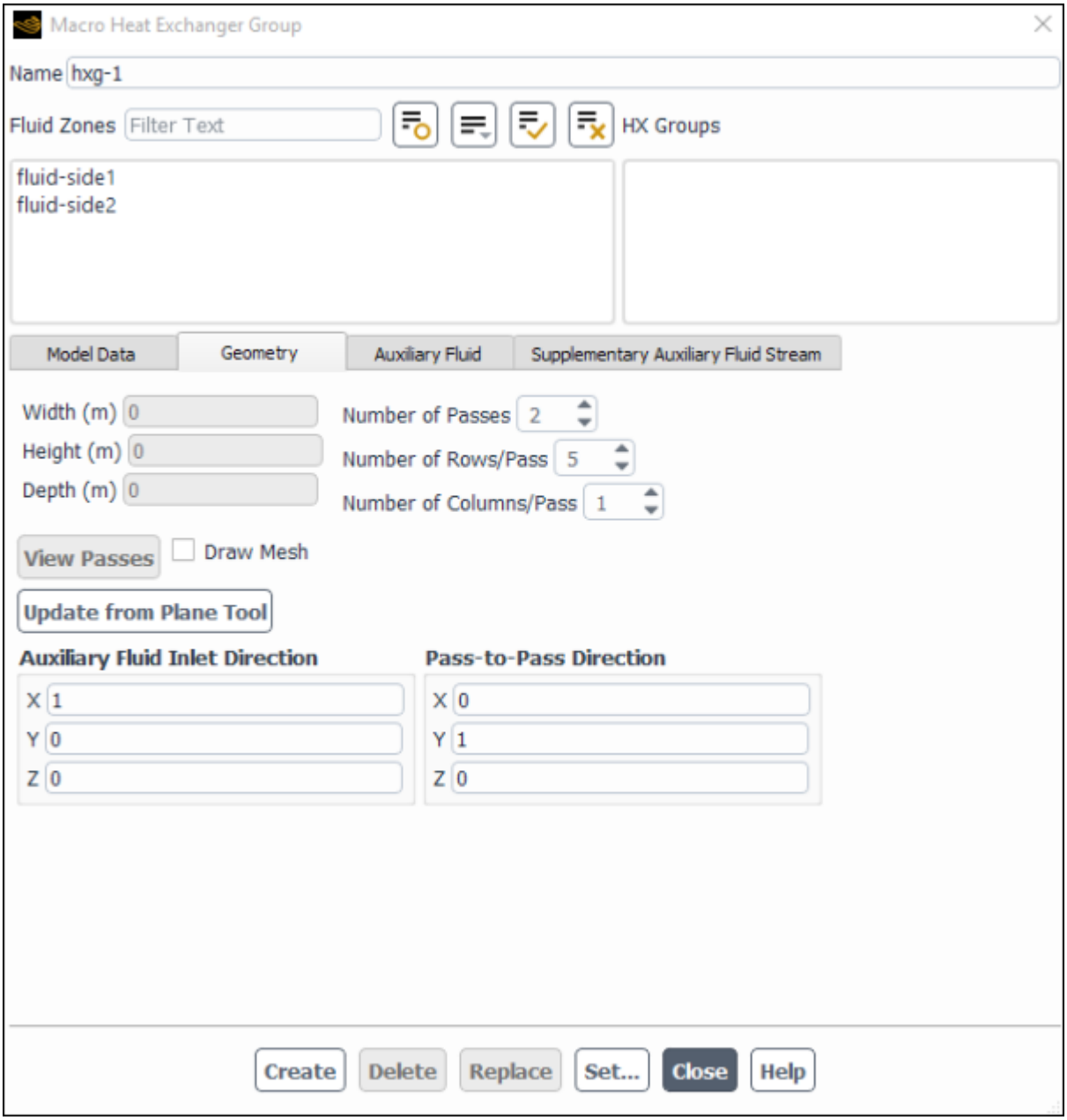
vi. Enter the **Auxiliary Fluid Temperature** and the **Primary Fluid Temperature** for the **ntu-model**. These are the fixed inlet temperatures at which the test was performed to obtain the heat transfer data.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

vii. If the **simple-effectiveness-model** is chosen, then clicking the **Velocity Effectiveness Curve...** button, under the **Heat Exchanger Performance Data**, allows you to set the velocity and corresponding effectiveness for each point. More information is available in Specifying Heat Exchanger Performance Data (p. 2299).

d. Click the **Geometry** tab (Figure 16.81: The Macro Heat Exchanger Group Dialog Box - Geometry Tab (p. 2308)).

Figure 16.81: The Macro Heat Exchanger Group Dialog Box - Geometry Tab



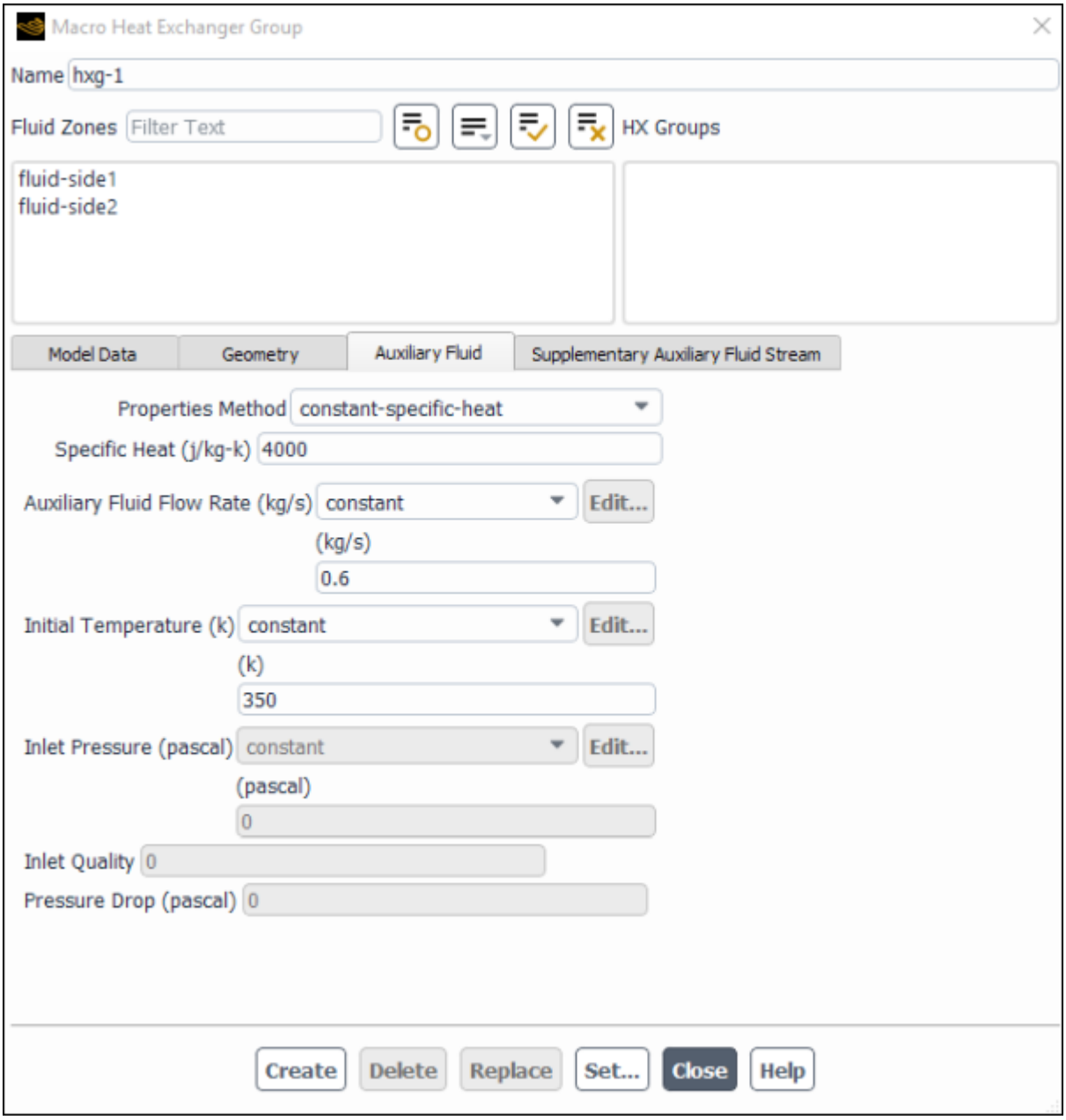
i. Define the heat exchanger macros by specifying the **Number of Passes**, the **Number of Rows/Pass**, and the **Number of Columns/Pass**. More information is available in Specifying the Auxiliary Fluid Inlet and Pass-to-Pass Directions (p. 2300) and Defining the Macros (p. 2300).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ii. Specify the **Auxiliary Fluid Inlet Direction** and **Pass-to-Pass Direction** (see Specifying the Auxiliary Fluid Inlet and Pass-to-Pass Directions (p. 2312)).

e. Click the **Auxiliary Fluid** tab (Figure 16.82: The Macro Heat Exchanger Group Dialog Box - Auxiliary Fluid Tab (p. 2309)) to specify the auxiliary fluid operating conditions.

Figure 16.82: The Macro Heat Exchanger Group Dialog Box - Auxiliary Fluid Tab



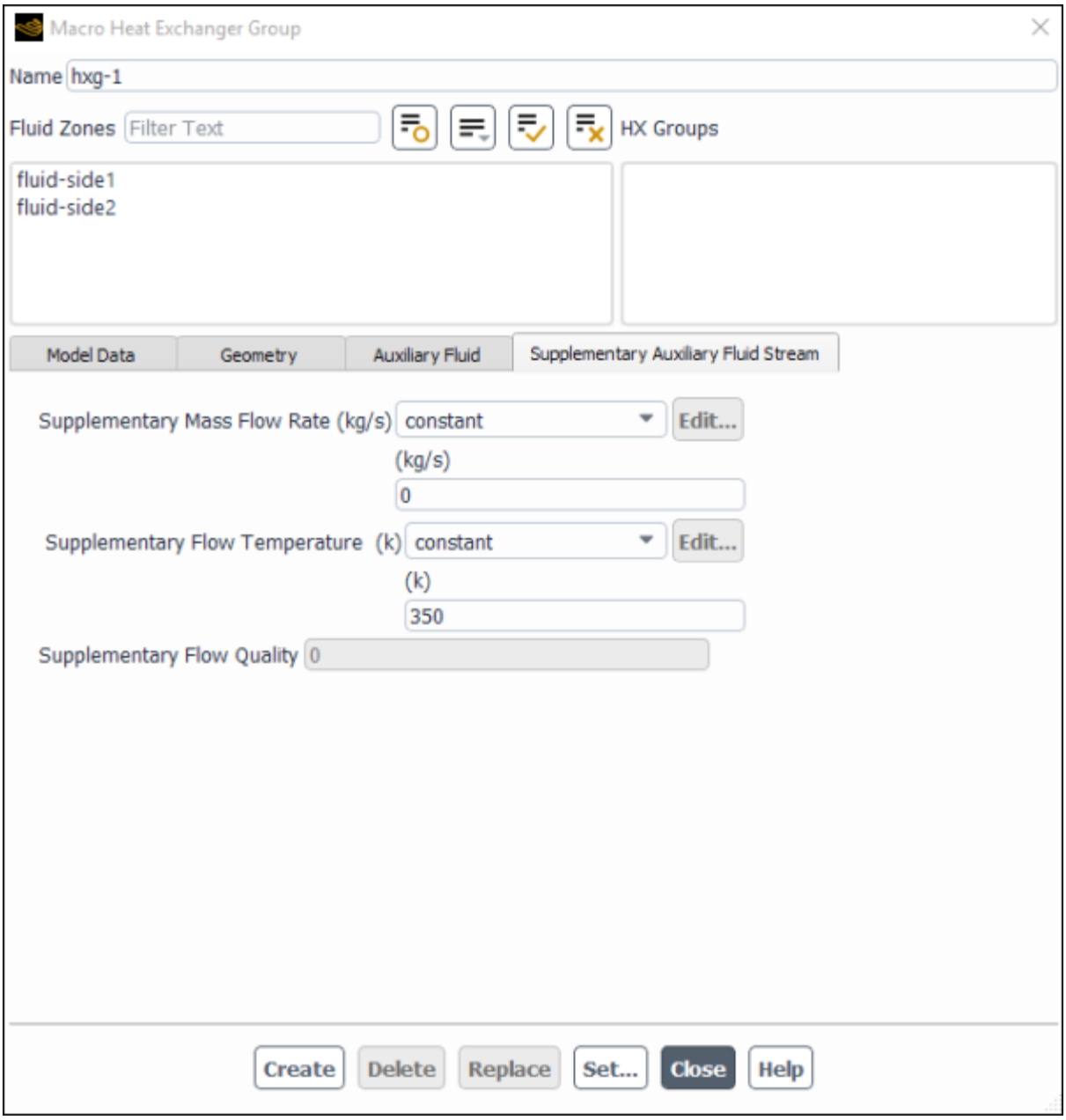
i. Specify the **Specific Heat** as either a **constant-specific-heat** or as a **user-defined-enthalpy**.

ii. **Auxiliary Fluid Flow Rate**, **Initial Temperature**, and **Inlet Pressure** can be provided as a **constant**, **polynomial** or **piecewise-linear** profile that is a function of time (see Specifying the Auxiliary Fluid Properties and Conditions (p. 2302)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

f. If a supplementary auxiliary stream is to be modeled, click the **Supplementary Auxiliary Fluid Stream** tab.

Figure 16.83: The Macro Heat Exchanger Group Dialog Box - Supplementary Auxiliary Fluid Stream Tab



i. You can specify the **Supplementary Mass Flow Rate** as a **constant**, **polynomial** or **piecewise-linear** profile that is a function of time.

ii. You can specify the **Supplementary Flow Temperature** as a **constant**, **polynomial** or **piecewise-linear** profile that is a function of time.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

g. Click **Create** or **Replace** in the **Macro Heat Exchanger Group** dialog box to save all the settings. **Replace** changes the parameters of the already existing group that is selected in the **HX Groups** list.

Important:

Creating or replacing any heat exchanger group initializes any previously calculated values for temperature and enthalpy for all macros.

5. If a heat exchanger group is composed of multiple fluid zones and you want to override any of the inputs defined in the previous steps, click the **Set...** button to open the **Ungrouped Macro Heat Exchanger** dialog box (Figure 16.72: The Ungrouped Macro Heat Exchanger Dialog Box Displaying the Model Data Tab (p. 2295)). Select the particular fluid zone as usual. Notice that the individual heat exchanger inherits the properties of the group by default. You may override any of the following:

   - **Number of Passes**, **Number of Rows/Pass**, and **Number of Columns/Pass**
   - **Auxiliary Fluid Inlet Direction** and the **Pass-to-Pass Direction**
   - **Core Porosity Model**

For additional information, see the following sections:

16.5.3.3.1. Selecting the Fluid Zones for the Heat Exchanger Group
16.5.3.3.2. Selecting the Upstream Heat Exchanger Group
16.5.3.3.3. Specifying the Auxiliary Fluid Inlet and Pass-to-Pass Directions
16.5.3.3.4. Specifying the Auxiliary Fluid Properties
16.5.3.3.5. Specifying Supplementary Auxiliary Fluid Streams
16.5.3.3.6. Initializing the Auxiliary Fluid Temperature

16.5.3.3.1. Selecting the Fluid Zones for the Heat Exchanger Group

Select the fluid zones that you want to define in the heat exchanger group in the **Fluid Zones** drop-down list. The auxiliary fluid flow in all these zones will be in parallel. Note that one zone cannot be included in more than one heat exchanger group.

16.5.3.3.2. Selecting the Upstream Heat Exchanger Group

If you want to connect the current group in series with another group, choose the upstream heat exchanger group. Note that any group can have at most one upstream and one downstream group. Also, a group cannot be connected to itself. Select a heat exchanger group from the **Upstream** drop-down list under **Connectivity** in the **Model Data** tab of the **Macro Heat Exchanger Group** dialog box.

Important:

Connecting to an upstream heat exchanger group can be done only while creating a heat exchanger group. The connection will persist even if the connection is later

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

changed and the **Replace** button is clicked. To change a connection to an upstream heat exchanger group, you need to delete the connecting group and create a new heat exchanger group with the proper connection.

16.5.3.3.3. Specifying the Auxiliary Fluid Inlet and Pass-to-Pass Directions

The **Auxiliary Fluid Inlet Direction** and **Pass-to-Pass Direction**, in the **Geometry** tab can be specified as directed in Specifying the Auxiliary Fluid Inlet and Pass-to-Pass Directions (p. 2300) in the **Ungrouped Macro Heat Exchanger** dialog box. Note that the **Update from Plane Tool** will set the height, width, and depth as the average of the fluid zones selected in the **Fluid Zones**.

16.5.3.3.4. Specifying the Auxiliary Fluid Properties

The auxiliary fluid can be specified as having a **constant-specific-heat**, or a user-defined function can be written to calculate the enthalpy, as described in Specifying the Auxiliary Fluid Properties and Conditions (p. 2302).

16.5.3.3.5. Specifying Supplementary Auxiliary Fluid Streams

The addition or removal of a supplementary auxiliary fluid is allowed in any of the heat exchanger groups. Note that auxiliary streams are not allowed for individual zones. You will specify the mass flow rate, temperature, and quality of the supplementary auxiliary fluid. You will also need to specify the heat transfer for various flow rates of primary and auxiliary flows. The auxiliary stream has the following assumptions:

- The magnitude of a negative auxiliary stream must be less than the primary auxiliary fluid inlet flow rate of the heat exchanger group.
- Added streams will be assumed to have the same fluid properties as the primary inlet auxiliary fluid.

16.5.3.3.6. Initializing the Auxiliary Fluid Temperature

When the heat exchanger group is connected to an upstream heat exchanger group, Ansys Fluent will automatically set the initial guess for the auxiliary fluid inlet temperature, $T_{in,auxiliary}$, to be equal to the $T_{in,auxiliary}$ of the upstream heat exchanger group. Thus the boundary condition $T_{in,auxiliary}$ for the first heat exchanger group in a connected series will automatically propagate as an initial guess for every other heat exchanger group in the series. However, when it is necessary to further improve convergence properties, you will be allowed to override $T_{in,auxiliary}$ for any connected heat exchanger group by providing a value in the **Initial Temperature** field. Whenever such an override is supplied, Ansys Fluent will automatically propagate the new $T_{in,auxiliary}$ to any heat exchanger groups further downstream in the series. Similarly, every time the $T_{in,auxiliary}$ boundary condition for the first heat exchanger group is modified, Ansys Fluent will correspondingly update every downstream heat exchanger group.

If you want to impose a non-uniform initialization on the auxiliary fluid temperature field, first connect the heat exchanger groups and then set $T_{in,auxiliary}$ for each heat exchanger group in streamwise order.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

All heat exchangers included in a group must use the fixed $T_{in,auxiliary}$ option. All heat exchangers within a heat exchanger group must have the same $T_{in,auxiliary}$. In other words, no local override of this setting is possible through the **Ungrouped Macro Heat Exchanger** dialog box.

16.5.4. Postprocessing for the Heat Exchanger Model

Postprocessing for the heat exchanger models involves computing the total heat rejection rate by setting up volume monitors and reporting of variables such as computed heat rejection, inlet or outlet temperature, specific heat, and mass flow rate.

Note that when postprocessing dual cell heat exchanger models using dialog boxes other than the **Heat Exchanger Report** dialog box (which requires you to explicitly specify whether you want reports on either the **Primary** or the **Auxiliary** fluid), attention must be paid to the fact that there are overlapping cell zones in the heat exchanger region. For example:

- When creating an isosurface and making selections from the **From Zones** selection list of the **Iso-Surface** dialog box, you should never select both the primary and the auxiliary fluid zones at the same time. Note that if no selections are made in this list, then all the cell zones are selected.
- When making selections from the **Surfaces** list of the **Contours** dialog box, you should not select a surface that is associated with the primary fluid zone and a surface associated with the auxiliary fluid zone at the same time, if those surfaces are spatially coincident.

For additional information, see the following sections:

16.5.4.1. Heat Exchanger Reporting

16.5.4.2. Total Heat Rejection Rate

16.5.4.1. Heat Exchanger Reporting

Reporting the results for the heat exchanger models is done using the **Heat Exchanger Report** dialog box.

Results → Reports → Heat Exchanger **Edit...**

The following variable options are available for reporting:

- **Computed Heat Rejection**
- **Inlet Temperature**
- **Outlet Temperature**
- **Mass Flow Rate**
- **Specific Heat**

16.5.4.1.1. Computed Heat Rejection

To display the **Computed Heat Rejection**

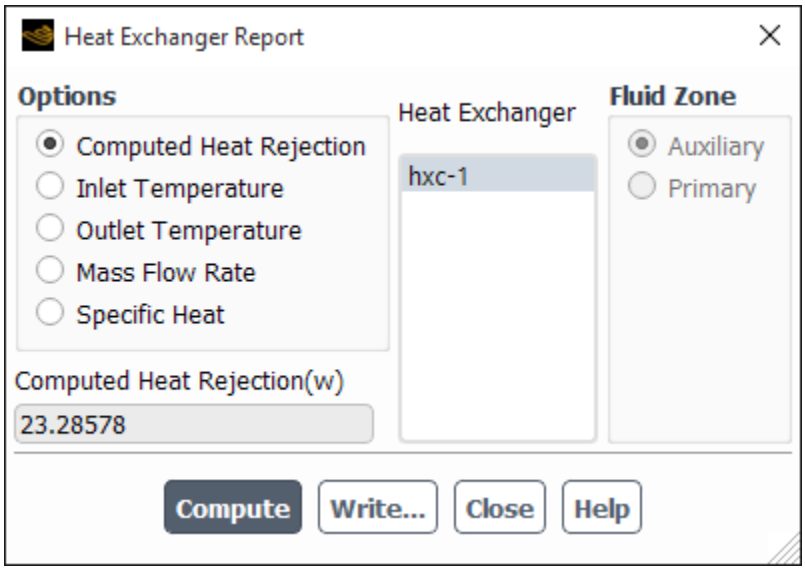
1. Select **Computed Heat Rejection** from the **Options** list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Select the **Heat Exchanger** from the selection list.

3. Click **Compute**.

Figure 16.84: The Heat Exchanger Report Dialog Box for Reporting Computed Heat Rejection



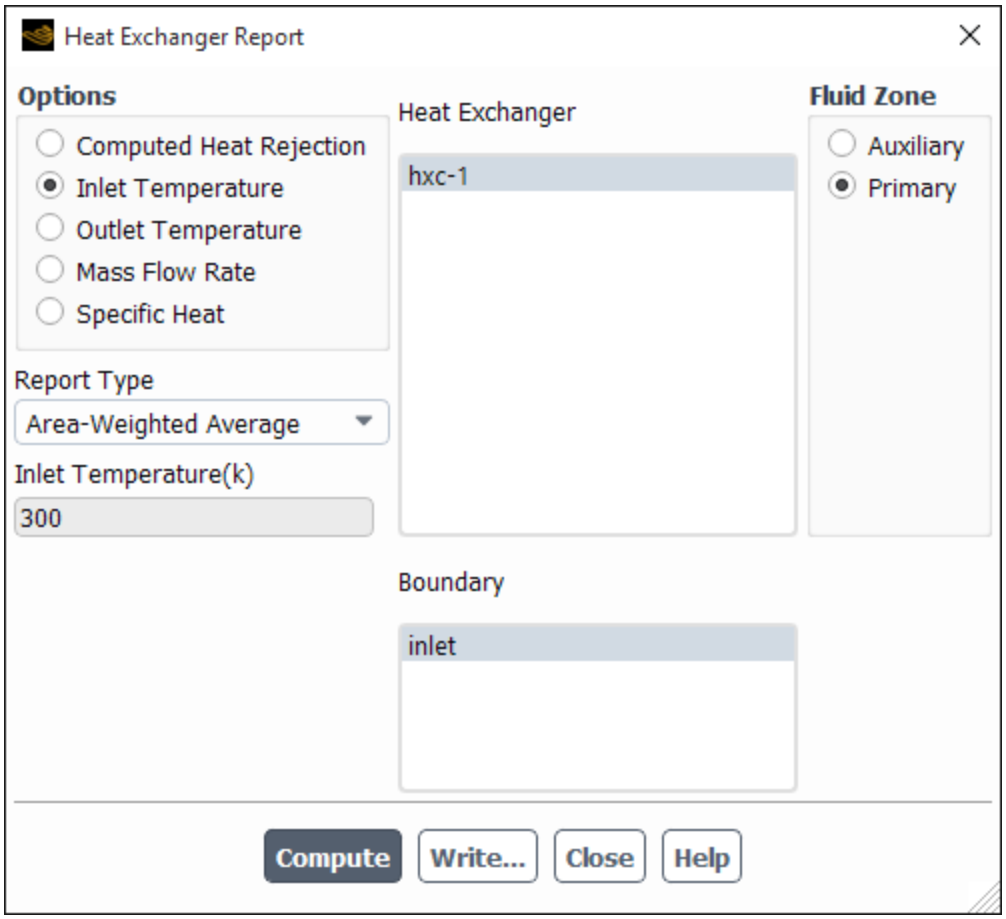
You can write the computed data to a file by clicking the **Write...** button and entering the name of the heat exchanger report file in the **Select File** dialog box.

16.5.4.1.2. Inlet/Outlet Temperature

Inlet/Outlet Temperature can be reported for both primary and auxiliary fluid in the **Heat Exchanger Report** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.85: The Heat Exchanger Report Dialog Box for Reporting the Inlet Temperature



1. Select **Inlet Temperature** or **Outlet Temperature** from the **Options** list.
2. Select the heat exchanger from the **Heat Exchanger** selection list.
3. Select either **Auxiliary** or **Primary** as the **Fluid Zone**.
4. Select the appropriate boundary zone and report type from the **Boundary** and **Report Type** lists, respectively.

   Important:

   Note that the macro heat exchangers (unlike the dual cell heat exchanger) do not contain an auxiliary cell zone. Hence, the **Boundary** and **Report Type** fields will not appear in the dialog box if you are reporting an auxiliary inlet/outlet temperature.

5. Click **Compute**.

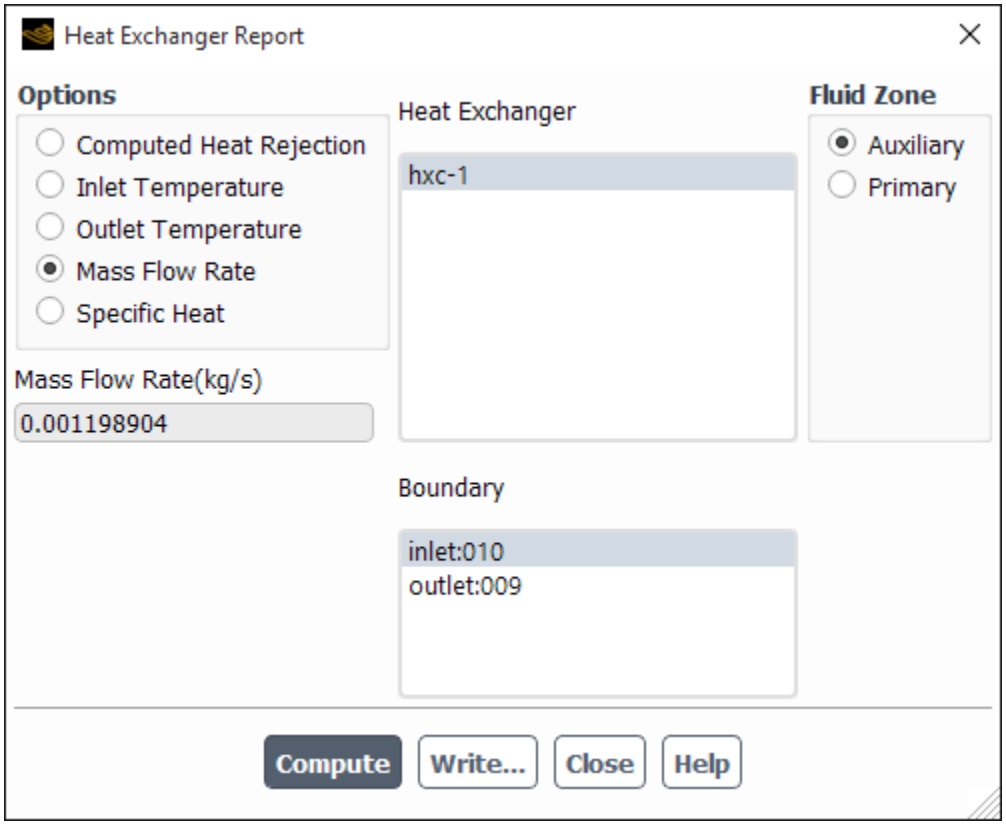
You can write the computed data to a file by clicking the **Write...** button and entering the name of the heat exchanger report file in the **Select File** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16.5.4.1.3. Mass Flow Rate

Mass Flow Rate can be reported for both the primary and auxiliary fluid in the **Heat Exchanger Report** dialog box.

Figure 16.86: The Heat Exchanger Report Dialog Box for Reporting Mass Flow Rate



1. Select **Mass Flow Rate** from the **Options** group box.
2. Select the heat exchanger from the **Heat Exchanger** selection list.
3. Select either **Auxiliary** or **Primary** as the **Fluid Zone**.
4. Select the appropriate boundary zone and report type from the **Boundary** and **Report Type** lists, respectively.
5. Click **Compute**.

Important:

Note that the macro heat exchangers (unlike the dual cell heat exchanger) do not contain an auxiliary cell zone. Hence, the **Boundary** field will not appear in the dialog box if you are reporting an auxiliary fluid mass flow rate.

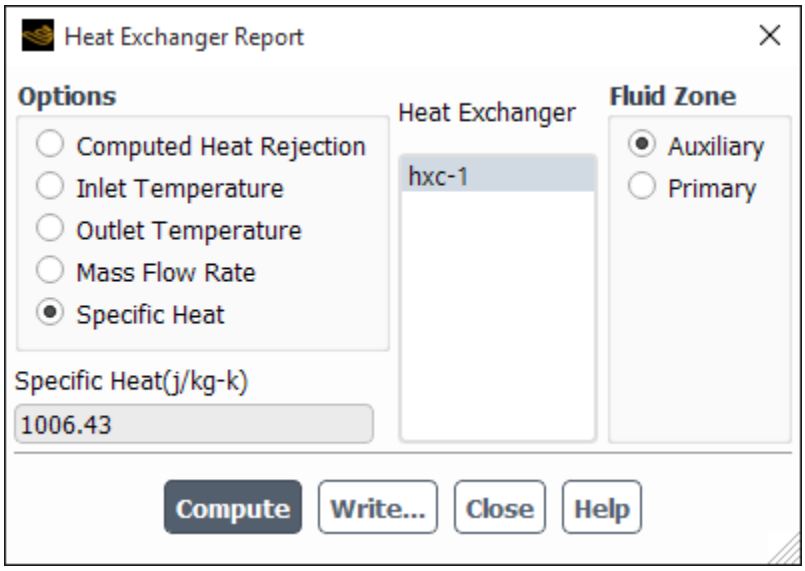
You can write the computed data to a file by clicking the **Write...** button and entering the name of the heat exchanger report file in the **Select File** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16.5.4.1.4. Specific Heat

Specific Heat for the primary or auxiliary fluid can be reported through the **Heat Exchanger Report** dialog box. If specific heat is defined as a function of temperature, the specific heat reported will be a cell volume averaged value.

Figure 16.87: The Heat Exchanger Report Dialog Box for Reporting Specific Heat



1. Select **Specific Heat** from the **Options** group box.
2. Select the heat exchanger from the **Heat Exchanger** selection list.
3. Select either **Auxiliary** or **Primary** as the **Fluid Zone**.
4. Click **Compute**.

16.5.4.2. Total Heat Rejection Rate

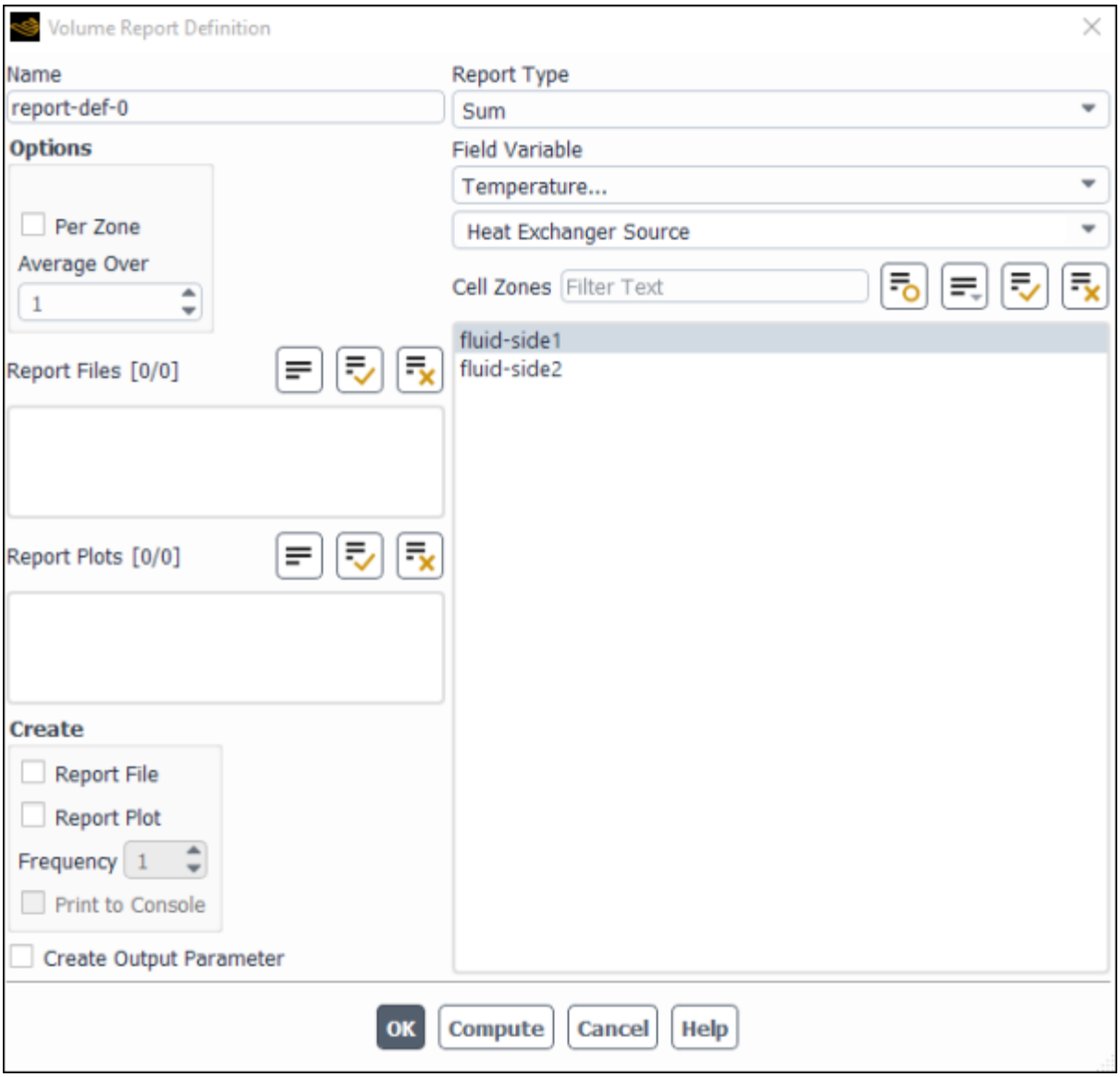
To postprocess the total heat rejection rate, you can set up a volume report definition to monitor convergence and view the computed values.

Solution → **Reports** → **Definitions** → **New** → **Volume Report** → **Sum...**

1. Select **Temperature** and **Heat Exchanger Source** from the **Field Variables** drop-down lists.
2. Select the appropriate **Cell Zones** and click **OK** to close the **Volume Monitor** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.88: The Volume Report Definition Dialog Box



Important:

Note that the macro heat exchangers contain only the primary fluid as the cell zone, but in case of the dual cell model, you can select either the primary or auxiliary fluid.

16.5.5. Useful Reporting TUI Commands

To report the results for the macro heat exchangers, you can use the following text command:

`define` → `models` → `heat-exchanger` → `macro-model` → `heat-exchanger-macro-report`

Specify the fluid zone *id / name* for which you want to obtain information.

To view the connectivity of the heat exchanger groups, use the text command:

```
(report-connectivity)
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16.6. Thermal Analysis of Printed Circuit Boards

The thermal conductivity of a printed circuit board (PCB) varies considerably due to the layered construction and highly variable metal fraction at any particular location. Therefore, thermal analysis of a PCB requires knowledge of the orthotropic components of thermal conductivity (that is, X, Y, and Z components of thermal conductivity).

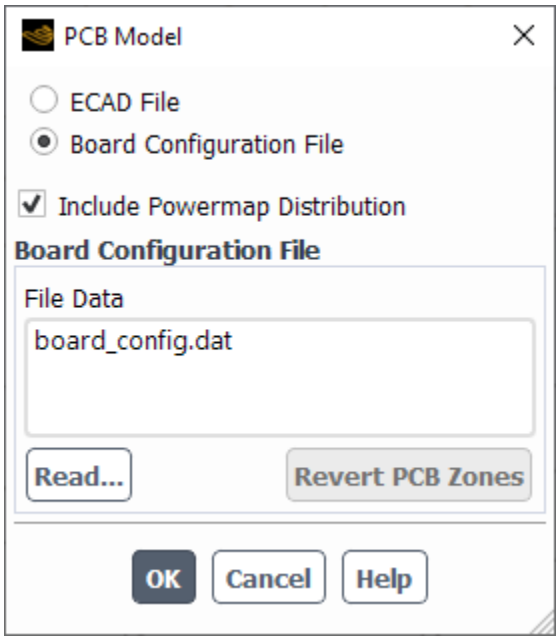
For the powermap distribution, Fluent uses SIWave DC IR drop analysis. DC IR Drop analysis computes the joule heating losses and exports it into power map file (`.outb`) for each conducting layer.

Ansys Fluent provides the **PCB Model** to automate the workflow, enabling the necessary case settings to prepare for thermal analysis of the PCB zones. This includes computing the orthotropic components of thermal conductivity and, by extension, the power distribution map of a PCB. This requires certain files from dependant on whether you are starting from an **ECAD File** or **Board Configuration File**.

The PCB Model is initiated from Figure 16.89: The PCB Model Dialog Box (p. 2319):

Physics → **Model Specific** → **PCB Model...**

Figure 16.89: The PCB Model Dialog Box



Note:

Switching between the ECAD File-based and the Board Configuration File-based workflow results in reverting all PCB zones and settings. An informational dialog box appears to confirm this revert.

These two workflows and additional information are discussed below:

16.6.1. PCB Model Workflow with ECAD File

16.6.2. PCB Model Workflow with Board Configuration File

16.6.3. Postprocessing for the PCB Model

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

16.6.4. Limitations for the PCB Model

16.6.1. PCB Model Workflow with ECAD File

To compute the orthotropic thermal conductivity using an ECAD File, you must have the following files:

- Electronic CAD mesh file `.aedb/edb.def` (ECAD for Metal and di-electric layers, generated in ANSYS SIWave or ANSYS Electronic Desktop.)

Note:

Currently supported materials for conducting layers in the `.aedb` file are `Cu-Pure` and `copper` and for dielectric layers in the `.aedb` file, the supported materials are `FR-4`, `FR4_epoxy`, and `air`.

For the power distribution map, you also require:

- SIWave Export Files (`.outb`)

Important:

The zone that is to be designated as PCB must be meshed using the Multizone meshing architecture of Fluent Meshing. Multizone meshing creates only hex cells while the surrounding fluid domain can be a poly or tet mesh. After multizone meshing is complete, the hex cells of the PCB zone will have non-conformal interfaces with the surrounding poly or tet cells. For details, see Adding Multizone Controls (p. 344). After meshing, you can switch to Solution Mode to complete the rest of the PCB Model workflow described in this section.

Note:

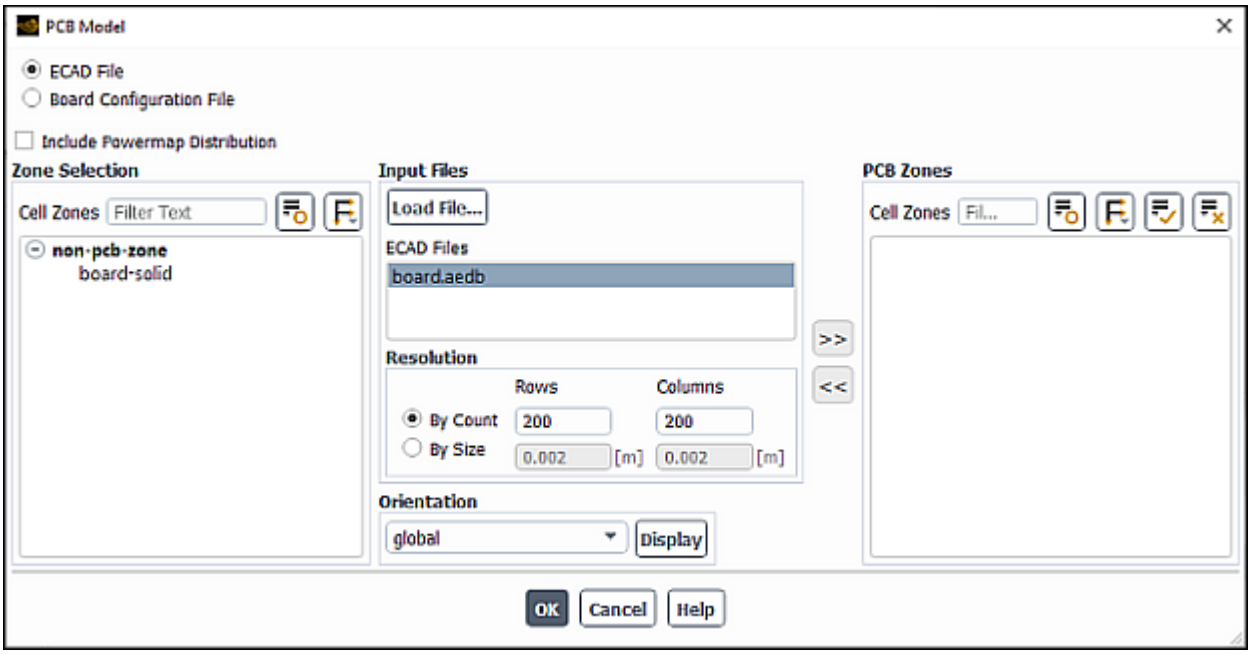
If you read both ECAD mesh and MCAD (mechanical) mesh, then the appropriate interfaces will need to be created between the two separate mesh zones.

16.6.1.1. Using the PCB Model with ECAD File

After selecting **ECAD File** on the **PCB Model** dialog box, as shown in Figure 16.90: The PCB Model Dialog Box for ECAD File (p. 2321), the following steps are required to complete the PCB Model setup

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.90: The PCB Model Dialog Box for ECAD File



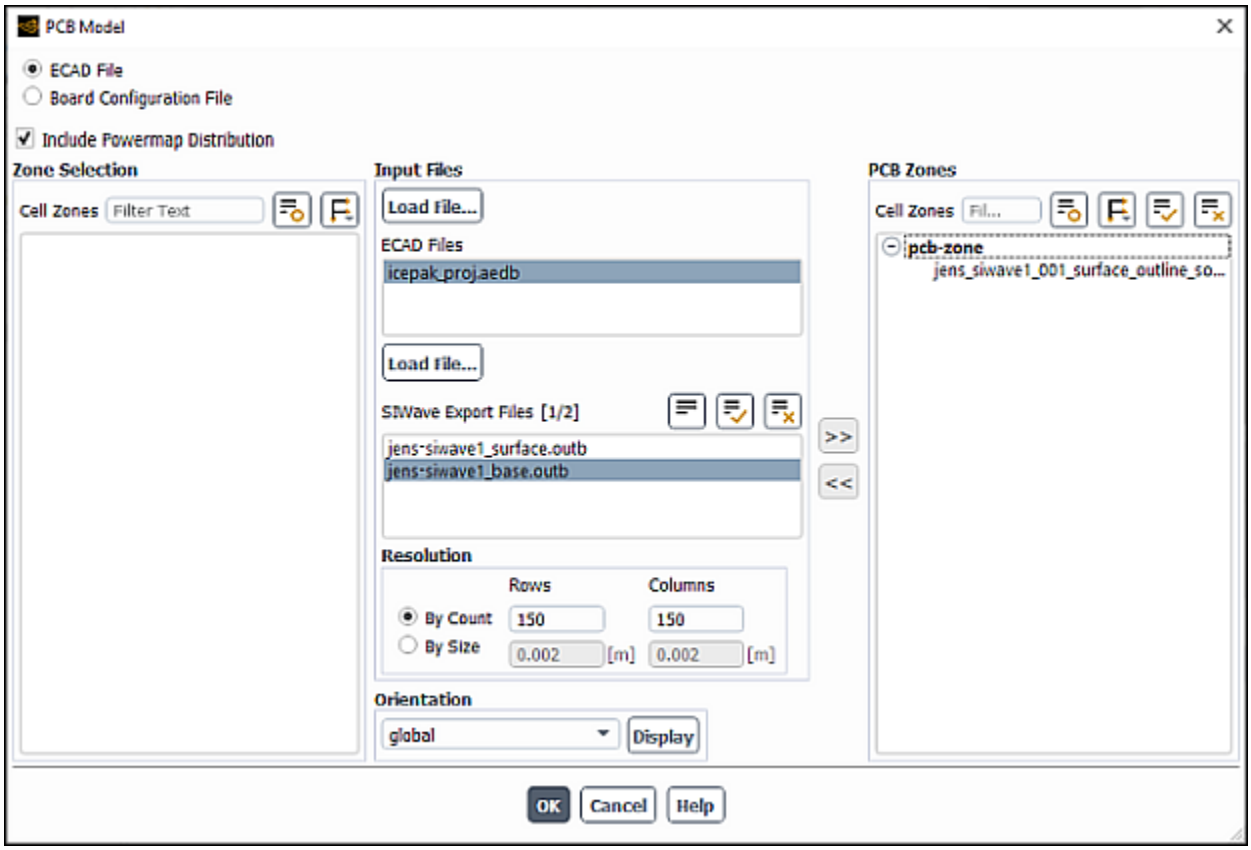
1. Under **Zone Selection**, all solid cell zones are listed. Any zone in this list are non-pcb zones. Select the zone(s) for which you would like to assign PCB settings.

2. Under **Input Files**, load an ECAD file (`.aedb/edb.def`). Click **Load File...** to open a dialog box to read the `edb.def` file. The `.aedb` file will then appear in the **ECAD Files** field. Different ECAD files can be read for different cell zones. If you want to re-use an already loaded ECAD file, then you do not need to load it again, instead you can select it from the **ECAD Files** field for any other cell zone.

3. Optionally you can **Include Powermap Distribution** that requires SIWave Export Files (`.outb`) in the current working directory. These are binary files output by SIWave using the **Export Icepak Power Map** option from **DC IR Drop Results** in SIwave. A DC IR drop simulation in ANSYS SIWave generates power dissipation data for the copper layers.

   An extra field to specify the **SIWave Export Files** is exposed when **Include Powermap Distribution** is selected. Note that multiple SIWave export files can be selected.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Under **Resolution**, you can specify **By Count** or **By Size**. The ECAD file can be thought of as an image and to generate the metal fraction, you need to obtain "pixelated" data from the file. You will need to provide details on the level of pixelation.

   - **By Count** - You provide the number of divisions along the length (**Rows**) and width (**Columns**) to divide the ECAD file.
   - **By Size** - You provide the size of each "pixel".

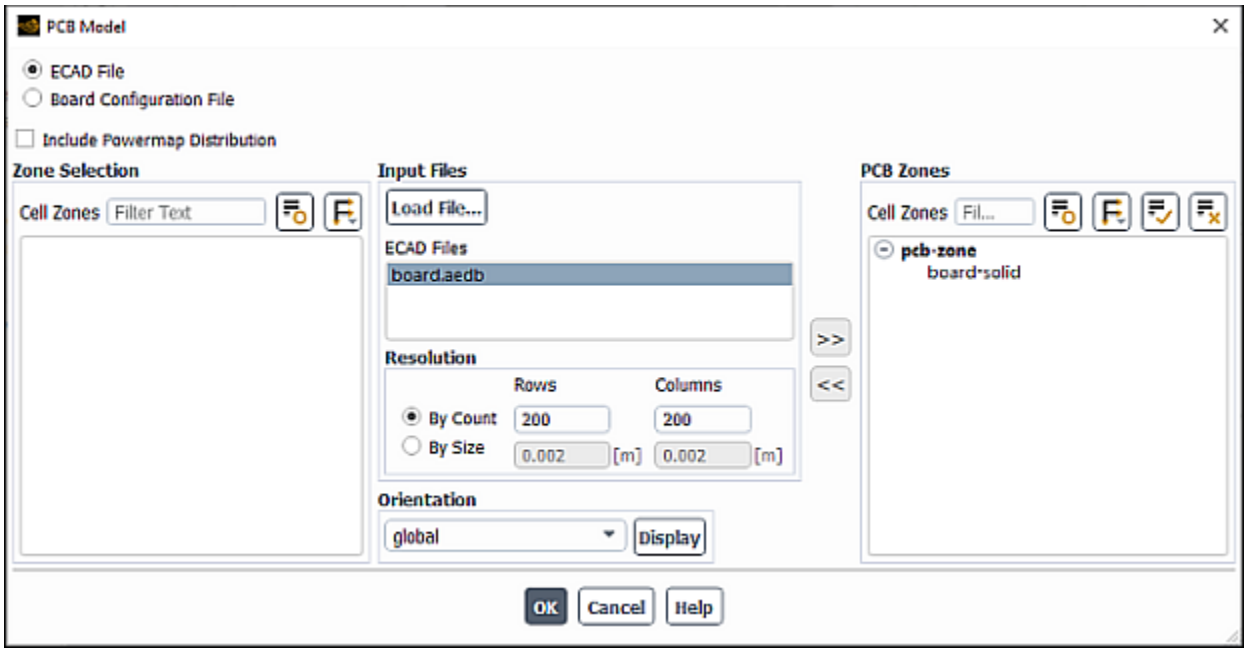
5. Under **Orientation**, you provide the local orientation of the PCB board. There is a drop-down menu where you can either select **global** as the local coordinate system if the origin of the PCB lies at (0,0,0) and is aligned along the coordinate axes, or you can select **Create New** to create a new reference frame. For details, see Reference Frames (p. 1160).

   When any cell zone is selected and you click **Create New**, the **Origin** is pre-filled with the minimum node coordinates of the selected cell zone. Therefore, if you intend to create a local reference frame on the cell zone which should be centered at the minimum node coordinates and aligned along the X – Y plane, no further inputs are requires and you can click **OK** to create the reference frame.

   You can also click **Display** to display the selected zone (pcb or non-pcb) along with the chosen reference frame. This can help to verify that the chosen reference frame is correctly aligned with the selected zone.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Click **>>** to designate the selected cell zone(s) a PCB Zone. The zone will now appear under **PCB Zones**. Note at any time, you can select a PCB zone from the list to see all the PCB settings associated with that zone.



You can also select a PCB zone(s) and click **<<** to revert all PCB settings for that zone. A dialog box will appear to confirm the revert. If accepted, the zone will move to the **Cell Zones** selection list.

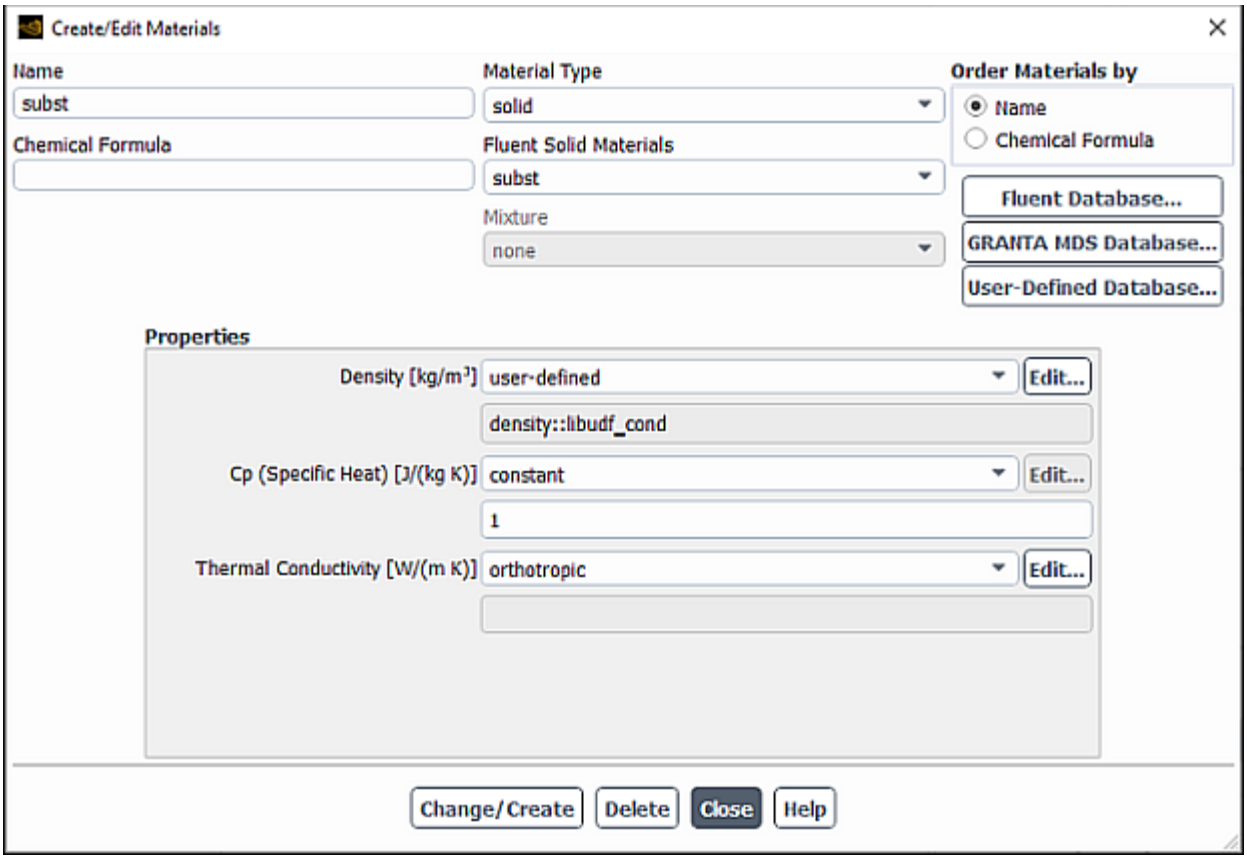
7. Click **OK** and Fluent starts an automatic workflow from generating the metal fraction and board configuration files to determining the thermal conductivity map of the PCB and then interpolation of the conductivity map on to the CFD Mesh.

Fluent automatically creates a new solid material called `subst` which can be viewed from the Materials list in the outline view. By default, `subst` is assigned to PCB cell zones.

Setup → Materials → Solid → subst

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.91: Material Properties for PCB Model Material

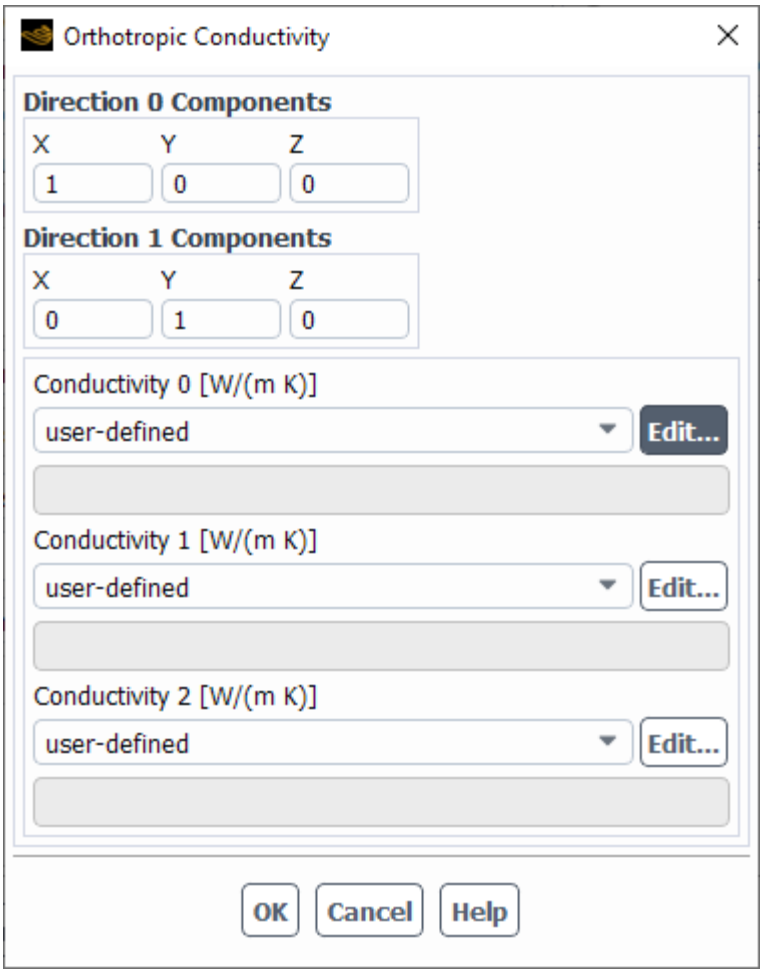


The material properties of `subst` are automatically defined:

- Density is defined via UDF that is automatically set by the PCB model.
- Specific heat is assigned a constant value of 1 because the density defined by the UDF is itself multiplied by the specific heat.
- Thermal Conductivity is defined as `orthotropic`. Clicking **Edit...** will open the Orthotropic Conductivity dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.92: Orthotropic Conductivity for PCB Model Material



Conductivity 0, **1**, and **2** are all automatically defined via UDF from the PCB model.

For more details on orthotropic conductivity, see Orthotropic Thermal Conductivity (p. 1650).

At this point, the case can be initialized and run.

Note:

The PCB settings are preserved in the Fluent `.cas` file and applied at run initialization. Therefore, a case can be restarted without any accompanying ECAD or Board Configuration files.

PCB Model Workflow with ECAD File Text Command

Use the command `define/models/addon-module` and enter **12** for the PCB Model.

You will be prompted to select from the menu as shown below. Enter **1** for the ECAD File workflow and follow the prompts.

```
Read ECAD File.              (1)
Read Board Configuration File.(2)
```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
Delete PCB Settings.          (3)

Choose an option from above. [1]
```

16.6.2. PCB Model Workflow with Board Configuration File

To compute the orthotropic thermal conductivity using a Board Configuration File, you must have the following files:

- Case file `.cas` (case file from Ansys Icepak project). Case file should be appended to current Fluent case/mesh before executing PCB model workflow with a board configuration file.
- Board configuration file from Ansys Icepak ("`board_config.dat`")
- Metal fraction information from Ansys Icepak (`.cond` file)

For the power distribution map, you also require:

- Power profiles from Ansys Icepak (.prof files)

Note:

Each `thread_id` specified in the `board_config.dat` file must match the PCB cell zone thread in Fluent.

Note:

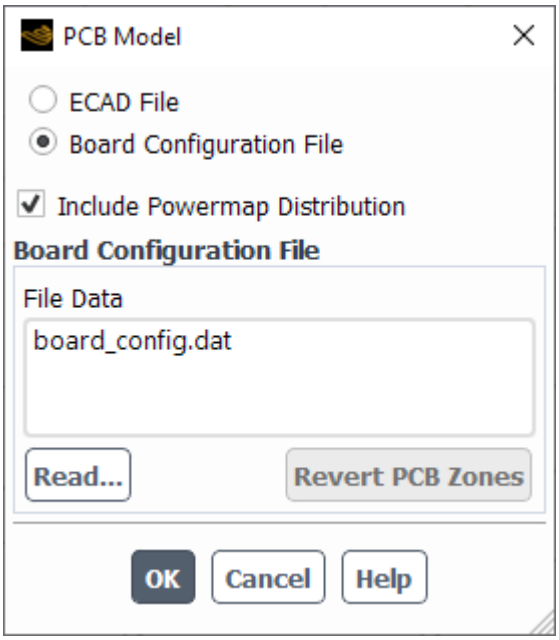
If you read both ECAD mesh and MCAD (mechanical) mesh, then the appropriate interfaces will need to be created between the two separate mesh zones.

16.6.2.1. Using the PCB Model with Board Configuration File

After selecting **Board Configuration File** on the **PCB Model** dialog box, as shown in Figure 16.93: The PCB Model Dialog Box for Board Configuration File (p. 2327), the following steps are required to complete the PCB Model setup

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.93: The PCB Model Dialog Box for Board Configuration File



1. Click **Read...** to select the board configuration file.
2. Optionally select **Include Powermap Distribution** if desired and ensure the appropriate `.prof` are available.

   Important:

   You must ensure that the appropriate `.prof` files are available in your working directory and are specified at the end of the `board_config.dat` file. For example, if you have three powermap files (`power-01.prof`, `power-02.prof`, `power-03.prof`), the end of the `board_config.dat` should read as follows:

   ```
   .
   .
   .
   power_prof=power-01.prof power-02.prof power-03.prof
   end Board 1
   ```

3. Click **OK**.

   Fluent automatically loads and configures the appropriate files based on the board configuration file.

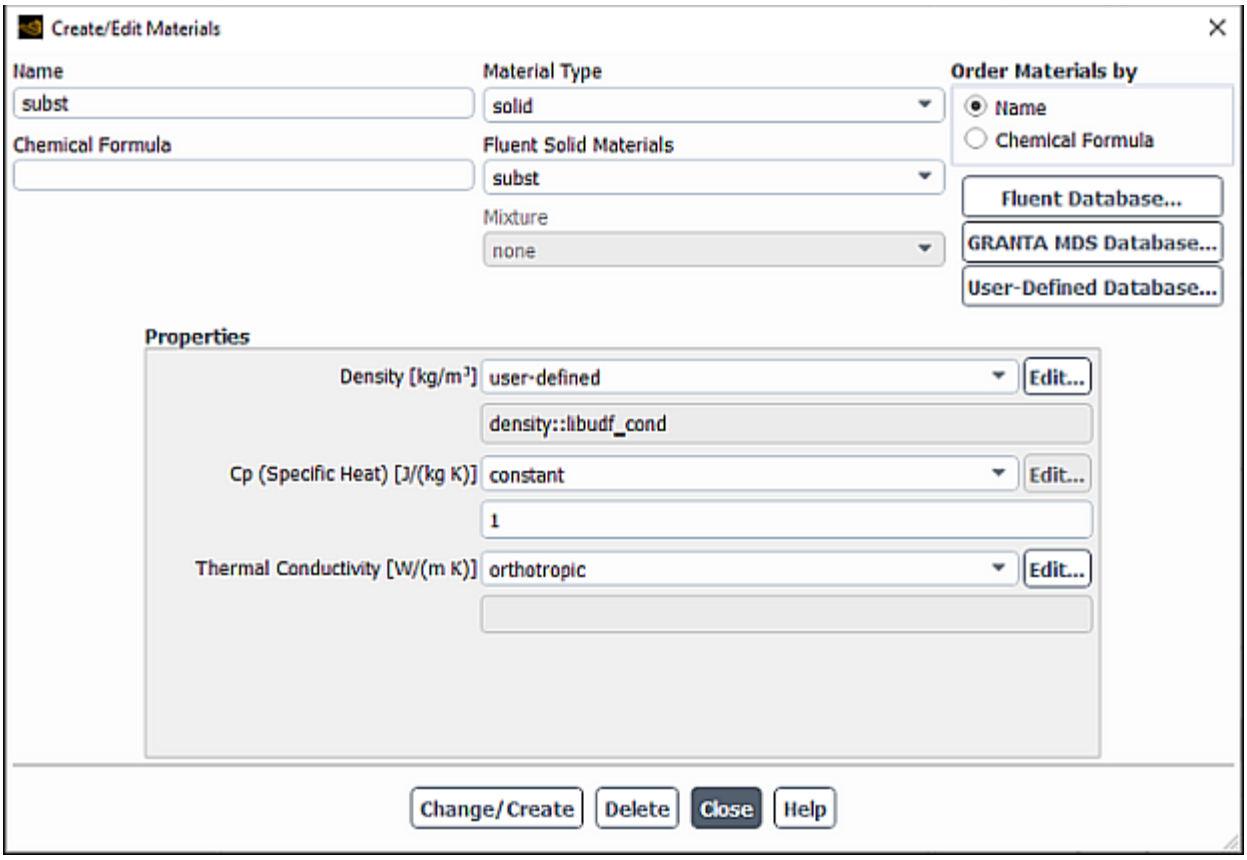
4. If needed, you can click **Revert PCB Zones** to revert all PCB settings for the affected zones. A dialog box will appear to confirm the revert.

Fluent automatically creates a new solid material called `subst` which can be viewed from the Materials list in the outline view. By default, `subst` is assigned to PCB cell zones.

Setup → Materials → Solid → subst

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.94: Material Properties for PCB Model Material

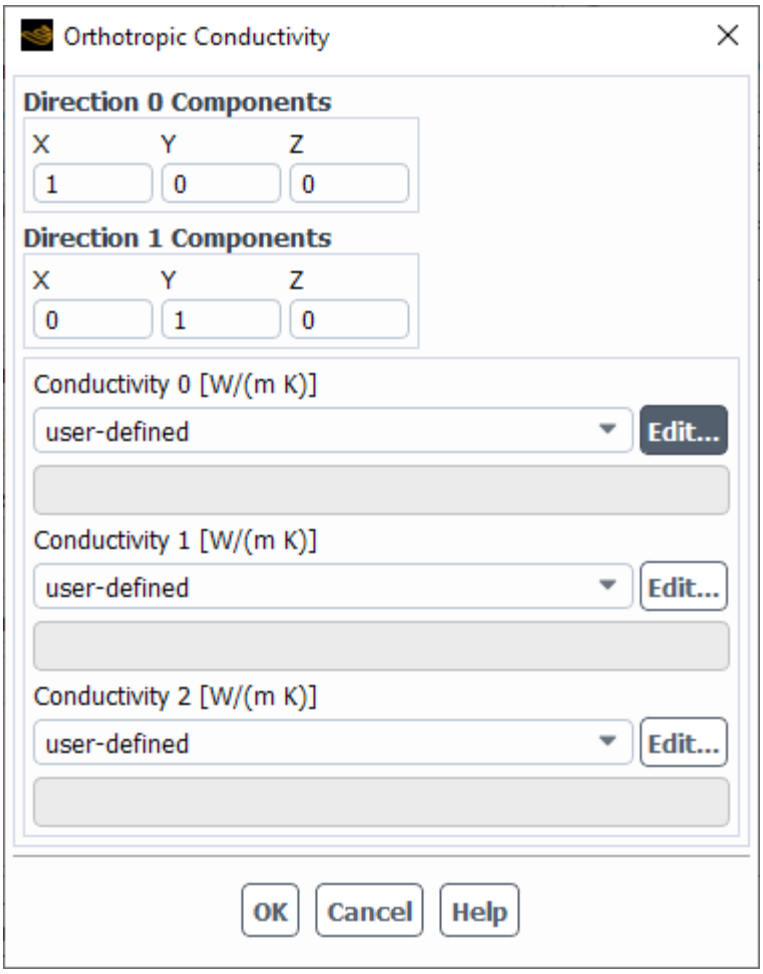


The material properties of `subst` are automatically defined:

- Density is defined via UDF that is automatically set by the PCB model.
- Specific heat is assigned a constant value of 1 because the density defined by the UDF is itself multiplied by the specific heat.
- Thermal Conductivity is defined as `orthotropic`. Clicking **Edit...** will open the Orthotropic Conductivity dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 16.95: Orthotropic Conductivity for PCB Model Material



Conductivity 0, **1**, and **2** are all automatically defined via UDF from the PCB model.

For more details on orthotropic conductivity, see Orthotropic Thermal Conductivity (p. 1650).

At this point, the case can be initialized and run.

Note:

The PCB settings are preserved in the Fluent `.cas` file and applied at run initialization. Therefore, a case can be restarted without any accompanying ECAD or Board Configuration files.

PCB Model Workflow with Board Configuration File Text Command

Use the command `define/models/addon-module` and enter **12** for the PCB Model.

You will be prompted to select from the menu as shown below. Enter **2** for the Board Configuration File workflow and follow the prompts.

```
Read ECAD File.              (1)
Read Board Configuration File.(2)
```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
Delete PCB Settings.          (3)

Choose an option from above. [2]
```

16.6.3. Postprocessing for the PCB Model

The following additional variables are available for postprocessing with the PCB model:

- Thermal Conductivity X
- Thermal Conductivity Y
- Thermal Conductivity Z

Note that these orthotropic thermal conductivities are computed in the global coordinate system.

Postprocessing Powermap Distribution

The powermap distribution can be postprocessed as any volumetric user energy source shown on the **Flux Reports** dialog box:

Results → **Reports** → **Fluxes...**

On the **Total Heat Transfer Rate** flux report, the total powermap sources appear under **User Source**.

For more details, see Generating a Flux Report (p. 4136).

16.6.4. Limitations for the PCB Model

The following are limitations associated with the PCB model:

- PCB model is not supported with mixed Windows/Linux platforms.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 18: Fluent's Virtual Blade Model

This chapter describes Fluent's Virtual Blade Model (VBM) which is used for analyzing the aerodynamic interaction between multiple rotors and airframes. With this technique, the 3D rotors are replaced with actuator disks that introduce the effect of the rotors via implicit momentum source terms in the governing equations. Since the actual blades are not physically modeled, the technique is hence referred to as the Virtual Blade Model. The model supports hybrid unstructured meshes for easy handling of multiple rotor geometries in close proximity, with convenient local mesh clustering. The non-linear aerodynamic interaction of rotor wakes with each other and with other structural components is obtained by coupling the VBM with Fluent's Navier-Stokes solver. In addition, when using the VBM, rotor trimming can be performed at desired thrust and moments in an automatic and robust fashion using an iterative method to account for the non-linear relation between blade pitch and rotor performance. A propeller and a well-studied simplified single rotor helicopter model in forward flight are presented in as tutorials to show the capabilities of Fluent VBM. See Fluent's Virtual Blade Model Tutorials for more details.

18.1. Introduction

The flow around a rotorcraft is strongly unsteady and three-dimensional. This complexity is attributable to the unique aerodynamic characteristics of rotary wings and to the reciprocal aerodynamic interaction of the stationary and rotating components that make up a rotorcraft. For a simple helicopter, for example, strong aerodynamic interactions can be observed between the main rotor and the airframe, between the main rotor and the tail rotor, and between the tail rotor and airframe. These interactions dominate the overall flow field and therefore the performance of the helicopter.

Due to the complexity of the flow around rotorcraft, engineers require powerful tools to optimize its performance and stability. Wind tunnel time is difficult to obtain and very expensive, and often of limited effectiveness. Hence, the ability to accurately simulate these complicated flow fields with computational tools becomes increasingly important. Numerical methods can handle many levels of complexity, therefore the computational approach is dictated by the objective of the simulation, which can range from obtaining detailed blade characteristics (stall behavior, noise, FSI, etc.), to cases where the time-averaged cumulative effects of the rotors on each other and the airframe are more than sufficient. For the latter, typical applications would be the prediction of fuselage drag and tail forces for a range of operating conditions, as well the effect of engine exhaust plume distribution and impingement on skin heating and IR-signature prediction. These time-averaged analyses are carried out using reduced-order models such as the Virtual Blade Model.

Historically, in time-averaged simulations, the rotors and their blades are replaced by rotor disk surrogates of equal diameter, or actuator disks, that are coupled with three-dimensional Navier-Stokes or Euler solvers. Since the rotors and their blades are not modelled directly, the computational grids are much smaller, less complex and require significantly less mesh generation effort. With the time-averaged approach, the computational time is also drastically reduced.

Two distinct actuator disk approaches exist. The pressure-disk rotor model simulates helicopter rotors or propellers in a time-averaged manner using a disk composed of two surfaces, one side representing an outflow boundary, the other an inflow boundary. The disk does not need to have a finite thickness,

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and the grid nodes of the two surfaces need not be paired and coincident. A pressure jump, function of radius and azimuth, is imposed across the disk, subject to the constraint that mass be explicitly conserved through the disk surfaces.

Zori et al.6 (References (p. 2352)) developed an alternate technique that replaces the rotor system with momentum sources acting on a one-cell-thick actuator disk that is an integral part of the mesh and does not have inflow and outflow boundaries. The advantage of this approach is that mass is automatically conserved across the disk. Both methods compute the rotor forces according to the Blade Element Theory, using lift and drag coefficient look-up tables for the stacks of airfoils that represent the blades.

In general, for rotorcraft, a rotor operates at the desired thrust and zero moments about the hub. These targets can be met by varying the collective and the cyclic blade pitch angles through user-input or by a trimming algorithm included in the rotor disk model. Typically, the relationship between the thrust coefficient and the collective pitch angle, and between the hub moments and the cyclic pitch angles is assumed to be linear, making the trim routine easy to implement but numerically unstable. The lack of a robust trim routine historically limited the simulation to single rotor configurations, therefore ignoring the tail rotor. Only recently, Yang et al.6, suggested an automatic trim routine based on a Newton-Raphson iterative method to account for the non-linear relation between blade pitch and rotor performance. The current implementation of Fluent VBM is based on the models developed by Zori et al. and Yang et al. (References (p. 2352)), with additional enhancements.

Fluent VBM allows for the specification of rotor blades represented by stacks of 2D airfoil sections varying in twist, chord and airfoil type. The airfoil look-up tables, containing lift and drag coefficients versus angle of attack, can also be functions of Mach and Reynolds number, allowing the accurate treatment of both low- and high-speed regimes. Furthermore, the code can handle up to twenty-five individual rotors simultaneously, permitting simulations of complete rotorcraft with both main and tail rotors, or other configurations, such as quadcopters, quad-tiltrotors, or other exotic multi-rotor configurations. Simulations of multiple rotorcraft operating in close proximity are also possible. Furthermore, in the current implementation, the rotor disks can also be meshed with hybrid unstructured grids, simplifying mesh construction for multiple rotors in close proximity and with convenient individual local mesh clustering.

18.2. The Virtual Blade Model (VBM)

This technique models the effect of the rotor on the flow field in a time-averaged manner through momentum equation source terms placed in a disk volume swept by the spinning rotor. Two tasks must be addressed simultaneously in this approach:

1. Time-averaged solution of the flow field through and around the rotor.

2. Reduced-order solution of the rotor blade aerodynamics to compute the required momentum source terms.

In our approach for the first task, the flow field is solved using Ansys Fluent, inviscid or viscous laminar or turbulent simulations can be obtained, assuming that the fluid is either compressible or incompressible. The following is, therefore, an outline of our approach to the formulation and implementation of task 2.

The momentum source terms, unknown at the start of the simulations, are linked to the flow solution through the actuator disks and therefore evolve as part of the iterative solution. The momentum source terms are evaluated using the Blade Element Theory by replacing the rotor blades with a stack of airfoil sections in the spanwise direction. Each section is treated as if it were a 2D airfoil. The geometric

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

properties of each section, such as chord length, airfoil type and twist can vary along the span of the blade.

In typical CFD simulations, rotorcraft are assumed to be stationary and immersed in a moving airstream, hence airflow solutions around rotorcraft are computed in the absolute frame of reference. Since the blades are rotating around an axis with velocity Ω, they are best described in a relative frame of reference defined by their rotation axis and origin, the radial direction along the blade and the azimuthal angle that defines the position of the blade in its circle of rotation. In VBM simulations, a third local 2D frame of reference is needed, defined by the tangential velocity vector at the blade section and the normal vector of the rotor disk. In order to compute the forces acting on a blade section, it is necessary to transform the velocity field $\vec{U}$ computed in the absolute frame of reference to the local blade section frame of reference to obtain the relative velocity field, from which the local angle of attack (α), Mach number (Ma), and Reynolds number (Re), can be determined. Look-up tables corresponding to each blade section are then used to extract the local C_L and C_D values, from which the instantaneous rotor forces per unit rotor span can be computed in the form:

$$f_{L,D}=C_{L,D}\left(\alpha,Ma,\mathrm{Re}\right)\cdot c\left(r/R\right)\cdot\tfrac{1}{2}\rho u_{rel}^{2}$$

where C is the chord length as a function of the non-dimensional radius r/R, R is the rotor tip radius and u_{rel} is the instantaneous relative airflow velocity experienced by each blade cross-section in the frame of reference of the rotor. However, for the time-averaged solution in the absolute frame of reference, these forces must be time-averaged over one rotation. Assuming that the rotational speed is constant, time averaging over one period is identical to geometric averaging over an angle 2π. Therefore, the time-averaged forces on a cell with face area ΔA on the rotor disk becomes:

$$F_{L,D}\big|_{\mathrm{cell}}=N_b\frac{\Delta A}{2\pi r}f_{L,D}$$

where N_b is the number of blades, and r is the radial coordinate of cell center.

The above force vector F is transformed back into the absolute flow field reference frame vector F so that it can be included in the airflow calculation. Let V_{cell} be the cell volume, then:

$$S'_{cell}=\frac{F'_{cell}}{V_{cell}}$$

is the time-averaged source term per unit volume that is added to the momentum equations in every cell attached to the rotor disk. Once these source terms have been computed, the solution of the flow field can be obtained, and the iterative procedure is repeated until the convergence criteria are met. This generalized implementation supports both structured and hybrid unstructured mesh topologies with hexahedral and prismatic elements.

The rotor performance is controlled by the thrust and moment coefficients, which are defined as:

$$C_T=\frac{T}{C_{NA-EU}\rho {V_{tip}}^2\pi R^2}$$

$$C_{Mx}=\frac{M_x}{C_{NA-EU}\rho {V_{tip}}^2\pi R^3}$$

$$C_{My}=\frac{M_y}{C_{NA-EU}\rho {V_{tip}}^2\pi R^3}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where C_{NA-EU} = 1 in North American (NA) practice and C_{NA-EU} = ½ in European practice. In NA practice, the coefficient ½ is absorbed into the thrust and moment coefficients, whereas in European practice the formulae explicitly contains the coefficient ½, therefore

$$\left(c_T, c_{Mx}, c_{My}\right)\Big|_{EU} = 2\left(c_T, c_{Mx}, c_{My}\right)\Big|_{NA}$$

In Fluent VBM, C_{NA-EU} is set to 1.

Finally, a comment should be made regarding turbulence modeling. CFD solvers such as Ansys Fluent employ a wide range of turbulence models to account for the effects of turbulence on the mean flow for steady-state fluid flow simulations. For applications where the VBM is used, it is suggested that standard two equation models such as $k-\varepsilon$ or $k-\omega$ be employed. It should be noted that the VBM itself does not affect the turbulence model directly by increasing the production of turbulence kinetic energy within the modeled rotor zone. However, turbulent mixing is modeled downstream of the VBM rotor zones, and therefore turbulence is indirectly influenced by the momentum generated by the rotors.

18.2.1. VBM Mode

Fluent VBM replaces the rotor system with momentum sources acting on a layer of cells for each rotor zone. These cells can be created in the mesh as a one-cell-thick actuator disk, called **Embedded Disk** method (EDM), or can be searched in the mesh when cut by a disk surface geometry, called **Floating Disk** method (FDM).

If using EDM, an internal surface in the shape of a flat disk should be created in the flow domain at the location of the rotor. In the CAD, this surface should be marked as an internal wall or internal surface that will be discretized by a single layer of nodes, such that the mesh is continuous across the surface. A single uniform layer of prisms or hexahedral cells should be extruded from the cell faces attached to the disk in the direction normal to the disk. The prismatic layer of cells in the downstream of the disk should be marked as a separate cell zone such that each rotor has its own separate cell zone. When calculating the airflow, Fluent VBM reads cells of each zone in a loop and calculate the source terms. Creating a mesh with these criteria would be difficult specially for a case with many rotors. However, it will provide more accurate and stable results. In addition, for unsteady simulation in this mode, any movement of disk requires mesh displacement or replacement which is not practical.

When using FDM, there is no need to create rotor disks and corresponding prismatic layers in the mesh. Instead, you only create a mesh with any type of element around the fuselage or any bodies in the domain without considering the rotor disk. The floating disk can be created using the post-processing tools available in Fluent to create a plane surface and clip it by iso-clip, or it can be imprinted as a `.stl` file. In VBM's Graphical User Interface (GUI), you can simply set up the disk center, orientation, and tip and root radius, and press a button to create the floating disk. Meshing Guidelines and Creating the VBM Disk (p. 2341) explains the mesh guidelines and how to define and create the floating disk in the mesh.

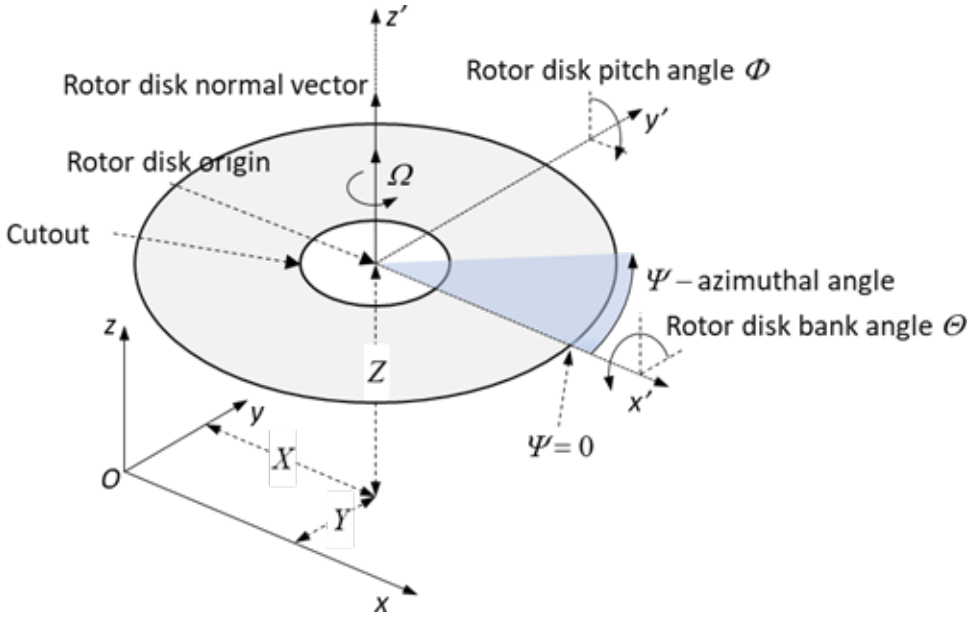
18.2.2. Rotor Disks

The current implementation of the VBM allows a maximum of twenty-five rotor disks. This enables the simulation of complex rotorcraft with multiple rotors, from simple helicopters with main and tail rotor, to multiple tilt rotors, to quadcopters and even to multiple rotorcraft, and single- or multi-engine propeller aircraft. It also can be used to simulate airflow around wind turbines in a large wind farm.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The rotor disk geometry is characterized by the location of its center {*X*,*Y*,*Z*}, its tip and root radius and its pitch and bank angles (or the rotor disk normal vector) as shown in Figure 18.1: Rotor Disks Schematic (p. 2335).

Figure 18.1: Rotor Disks Schematic



In the current implementation of VBM, the natural orientation of the disk's normal vector points in the positive *Z*-direction. Should the orientation of the normal vector be different, the disk must be rotated from its natural orientation first by using the disk pitch angle ϕ (rotation around the Y-axis) and then bank angle Θ (rotation around the *x′*-axis $\{cos\phi, 0, sin\phi\}$). Therefore, when a pitch rotation is applied, the *x′*-axis of the rotor will no longer be parallel to the X-axis.

Note:

- Rotations are not additive.
- Rotations are non-commutative and must be applied consistently in this order.
- Rotations follow the right-hand rule.

This also applies to the inclination of the rotor during CAD construction – the rotations must be applied in the same order. For example, in the case of an aircraft equipped with a propeller whose normal axis is aligned in the negative *X*-direction, the pitch and bank angles must be set to ϕ = –90˚, Θ = 0˚. Similarly, if the rotor axis is aligned in the positive *Y*-direction, the pitch and bank angles should be set to ϕ = 0˚, Θ = -90˚.

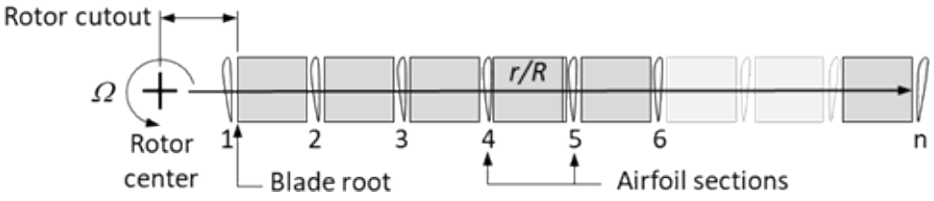
Rotor disk position can also be set using disk normal vector components instead of disk pitch and bank angles. In that case, Fluent's VBM computes disk pitch and bank angles according to the above mentioned natural orientation and disk normal vector.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

18.2.3. Blade Geometry

The blades are represented as stacks of airfoils which can vary in chord, twist and airfoil type along the normalized blade span r/R, as shown in Figure 18.2: 3D Blade Geometry Represented by a Stack of 2D Airfoils (p. 2336). The VBM assumes a linear distribution of chord, twist and C_L and C_D between the two ends of each section. Simulation accuracy increases if the airfoil look-up tables contain data for a range of Reynolds (Re) and Mach numbers (Ma) covering all possible operating conditions, since the VBM can determine the local Re and Ma on the blade and interpolate C_L and C_D from the tables accordingly.

Figure 18.2: 3D Blade Geometry Represented by a Stack of 2D Airfoils



18.2.4. Blade Pitch

The blade pitch angle $q(y)$, not to be confused with the rotor pitch angle, is the angle of the blade with respect to the rotor disk plane. To a first-order approximation (neglecting blade torsional flexing and other harmonics), it consists of the baseline collective angle θ_o and a cyclic pitch component expressed as a function of the sine and cosine of the azimuthal angle ψ:

$$\theta(\psi)=\theta_0-\theta_{lc}\cos\psi-\theta_{ls}\sin\psi$$

The blade pitch angles θ_0, θ_{1c}, and θ_{1s} can either be prescribed by you or calculated by the automatic rotor trim routine. The collective angle θ_0 is the angle of the blade at some point on its span with respect to the disk surface. The angles θ_{1c} and θ_{1s}, the amplitudes of the cosine and sine components of the cyclic pitch, are directly related to the angles of the swash plate found on most helicopter rotors and therefore control the pitch and roll attitude of the rotorcraft. Most rotor/propeller blades also have a spanwise twist. The blade twist adds to the collective pitch, therefore, reference point for the collective pitch and twist angles can be defined arbitrarily at any position along the blade as long as their sum provides correct rotor pitch angle. Traditionally, for helicopters, the reference point for the twist is located at 3-quarters of the blade span, therefore, helicopter main rotors usually have negative twist towards the tip. Negative twist progressively reduces the angle of attack towards the tip, increases the stall and buffeting margins, unloads the outer blade region and increases flapping stability.

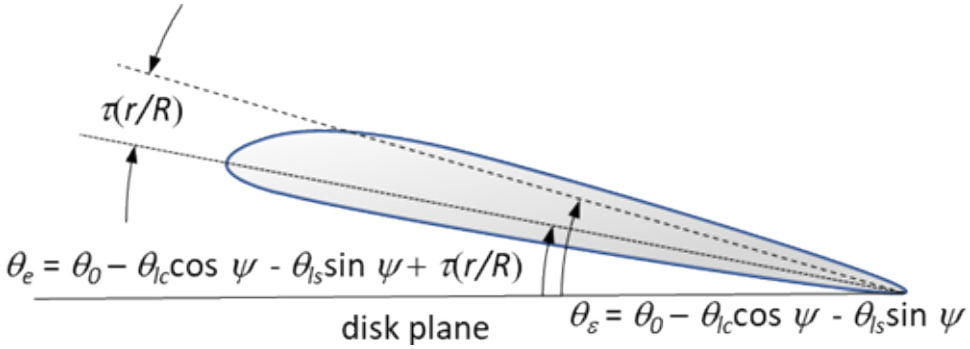
The VBM saves the effective pitch angle over the entire rotor as one of several VBM data fields in the solution file. The effective pitch angle is defined as:

$$\theta_e(\psi,\tau,r)=\theta_0-\theta_{lc}\cos\psi-\theta_{ls}\sin\psi+\tau(r/R)$$

where $\tau(r/R)$ is the twist angle as a function of the normalized radius, as shown in Figure 18.3: Effective Pitch Angle as a Function of r/R and ψ (p. 2337).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 18.3: Effective Pitch Angle as a Function of r/R and ψ



18.2.5. Blade Flapping

Articulated helicopter rotor blades can flap in response to the centrifugal and aerodynamic forces they endure. The flapping motion is not computed by the model, however, if the characteristics of the flapping motion is known, the model can account for the rotor coning angle and the first flapping harmonics by transforming the velocity components from the rotor disk plane to the actual tip path plane. Therefore, the baseline coning angle β_0 and the longitudinal and lateral flapping coefficients β_{1c} and β_{1s} determine the resultant flapping angle: $\beta(\psi)$:

$$\beta(\psi)=\beta_0-\beta_{lc}\cos\psi-\beta_{ls}\sin\psi$$

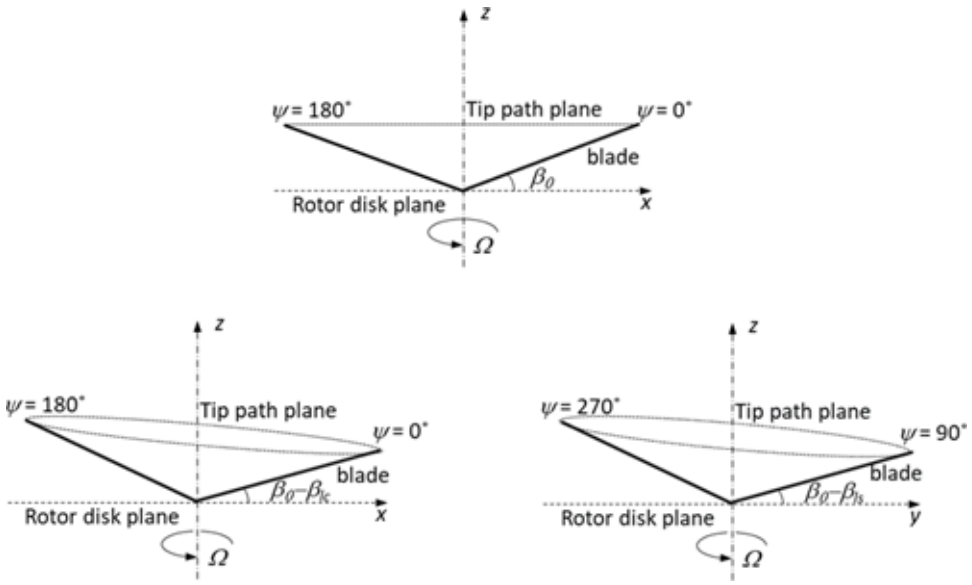
Figure 18.4: Blade Flapping – Baseline β_0 (Top), Longitudinal and Lateral Components β_{1c}, β_{1s} (Bottom) (p. 2338) shows a schematic of the flapping blades in the *x-z* and *y-z* planes.

Note:

The disk of the rotor remains flat and the flapping of the blades is simulated.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 18.4: Blade Flapping – Baseline β_0 (Top), Longitudinal and Lateral Components β_{1c}, β_{1s} (Bottom)



The change in the angle of attack due to flapping is computed, however the relatively small flapping velocity component $\partial\beta/\partial t$ that should be added to the velocity component normal to the blade path is neglected. Furthermore, due to its relatively small effect, the lead-lag motion of the blade is neglected.

18.2.6. Rotor Trimming

The rotors of helicopters operating in forward flight generate asymmetrical load distributions with respect to the plane defined by the forward velocity vector and the rotor normal vector. In addition to the collective pitch control required to maintain level flight at the desired altitude, cyclic pitch is necessary to balance the load across the rotor and control the pitching and rolling moments. Accurate aerodynamic simulations are only possible when the rotors are operating at the desired thrust and pitching/rolling moment settings. These settings, however, must be transformed into rotor blade pitch angles - a non-trivial operation. An automatic trim routine is available to compute the correct rotor settings without costly empirical guesswork. Thrust and moments with respect to the center of the rotor can be formulated in terms of non-dimensional thrust and moment coefficients.

Rotorcraft

$$T = C_T \rho_\infty V_{tip}^2 A_{\text{rotor}}$$

$$M_x = C_{M_x} \rho_\infty V_{tip}^2 A_{\text{rotor}} R_{rotor}$$

$$M_y = C_{M_y} \rho_\infty V_{tip}^2 A_{\text{rotor}} R_{rotor}$$

$$V_{tip} = \Omega R_{rotor}$$

$$_{\text{rotor}} = \pi R_{rotor}^2$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Propeller

$$T = C_T \rho_\infty {V_a}^2 A_{prop}$$

$$M_x = C_{M_x} \rho_\infty {V_a}^2 A_{\text{prop}} R_{\text{prop}}$$

$$M_y = C_{M_y} \rho_\infty {V_a}^2 A_{\text{prop}} R_{\text{prop}}$$

$$A_{\text{prop}} = \pi {R_{prop}}^2$$

where V_{tip} is the rotor tip speed and V_a is the aircraft forward speed.

Since rotorcrafts in hover have no forward velocity, the dynamic pressure must be formulated in terms of the rotor tip speed, whereas for propellers the aircraft forward speed is normally used.

Important:

In Fluent VBM, the thrust and moment coefficients are defined as functions of the rotor tip speed for both propellers and rotors.

The reference area of the rotor A_{prop} is defined solely by its tip radius R and includes the disc cutout, or the spinner cutout in the case of a propeller.

The VBM trimming routine calculates the collective and cyclic pitch angles to achieve the desired thrust and pitching (y-axis) and rolling (x-axis) moments around the hub. Let the three coefficients be expressed as functions of the collective and cyclic angles:

$$c_T = c_T\left(\theta_0, \theta_{lc}, \theta_{ls}\right)$$
$$c_{Mx} = c_{Mx}\left(\theta_0, \theta_{lc}, \theta_{ls}\right)$$
$$c_{My} = c_{My}\left(\theta_0, \theta_{lc}, \theta_{ls}\right)$$

Note:

The effect of the flapping hinge offset is neglected in the calculation of the moments.

Since the relationship between the rotor aerodynamic parameters and the blade pitch is non-linear, an iterative technique is used to drive the trim procedure to convergence. Following Yang et al. (References (p. 2352)), a Newton-Raphson iterative method is employed to automatically trim any number of rotors in the computational domain to achieve the user-set target values $\left\{c_T, c_{Mx}, c_{My}\right\}_i^{\text{target}}$.

The simulation begins with a user-supplied initial guess for the angles $\left\{\theta_0, \theta_{lc}, \theta_{ls}\right\}_i^0$ of each rotor i. After every N airflow iterations, called update frequency, the flow solver pauses and the VBM computes the rotor parameters $\left\{c_T, c_{Mx}, c_{My}\right\}_i^n$. A new set of angles is computed for each rotor *i* with the Newton-Raphson iterative approach. Let

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\begin{Bmatrix} c_T \\ c_{Mx} \\ c_{My} \end{Bmatrix}_i^n + \begin{bmatrix} \frac{\partial c_T}{\partial \theta_0} & \frac{\partial c_T}{\partial \theta_{lc}} & \frac{\partial c_T}{\partial \theta_{ls}} \\ \frac{\partial c_{Mx}}{\partial \theta_0} & \frac{\partial c_{Mx}}{\partial \theta_{lc}} & \frac{\partial c_{Mx}}{\partial \theta_{ls}} \\ \frac{\partial c_{My}}{\partial \theta_0} & \frac{\partial c_{My}}{\partial \theta_{lc}} & v\frac{\partial c_{My}}{\partial \theta_{ls}} \end{bmatrix}_i^n \begin{Bmatrix} \Delta\theta_0 \\ \Delta\theta_{lc} \\ \Delta\theta_{ls} \end{Bmatrix}_i^n = \begin{Bmatrix} c_T \\ c_{Mx} \\ c_{My} \end{Bmatrix}_i^{\text{target}}$$

The system of equations can be solved easily by recasting it in the form

$$\begin{Bmatrix} \Delta\theta_0 \\ \Delta\theta_{lc} \\ \Delta\theta_{ls} \end{Bmatrix}_i^n = \left[J_i^n \right]^{-1} \begin{Bmatrix} \Delta c_T \\ \Delta c_{Mx} \\ \Delta c_{My} \end{Bmatrix}_i^n$$

With initial angles $\{\theta_0, \theta_{lc}, \theta_{ls}\}^0$ The angles are updated as follows

$$\begin{Bmatrix} \theta_0 \\ \theta_{lc} \\ \theta_{ls} \end{Bmatrix}^{n+1} = \begin{Bmatrix} \theta_0 \\ \theta_{lc} \\ \theta_{ls} \end{Bmatrix}^n + \sigma_u \begin{Bmatrix} \Delta\theta_0 \\ \Delta\theta_{lc} \\ \Delta\theta_{ls} \end{Bmatrix}^n$$

where σ_u is a tunable under-relaxation factor, such that $0 \leq \sigma_u \leq 1$. The recommended value is 0.8.

The VBM then recomputes the source terms and the flow solution resumes. Since there are no analytical equations for computing the coefficients of the Jacobian matrix, the derivatives are obtained in the frozen flow field at time level *n* by computing the thrust and moment coefficients after perturbing each angle independently by the amount $\pm a$. For example, the first coefficient, representing the derivative of the thrust coefficient with respect to the collective angle, would be approximated by

$$\frac{\partial c_T}{\partial \theta_0} \approx \frac{c_T(\theta_0+\alpha) - c_T(\theta_0-\alpha)}{(\theta_0+\alpha)-(\theta_0-\alpha)}$$
$$\approx \frac{c_T(\theta_0+\alpha) - c_T(\theta_0-\alpha)}{2\alpha}$$

Since the derivatives are computed numerically, the algorithm is really the Secant Method. This procedure is repeated at every N iterations, until the rotor performance parameters reach their target values and the flow solution is converged. The L2 norms of ΔC_T, ΔC_{Mx} and ΔC_{My} are computed and printed on the Fluent **Console** to show the convergence of the trimming routine. The trimming routine is robust and yields converged results for cases simulating multiple rotors.

18.2.7. Tip Losses

Rotor blades usually have very high aspect ratios, therefore the lift and drag forces can be computed to a reasonable approximation at each spanwise location by assuming two-dimensional flow. However, near the tip, the circulation around the blade induced by the lift force is turned by 90˚ and shed downstream. The turning of the circulation creates the tip vortex, which introduces lift and drag losses that the VBM is unable to model directly.

To obtain a more realistic solution, the tip loss effects are incorporated in VBM using a tip loss factor F that corrects lift and drag coefficients near the tip of the disk, such that:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$C_{l_{\text{cor.}}} = F * C_l(Re, Ma, \alpha)$$

$$C_{d_{\text{cor.}}} = F * C_d(Re, Ma, \alpha)$$

where the lift/drag coefficient C_l/C_d is interpolated in airfoil data tables based on the Re, Ma, α at the cell centroid of each element on the disk. Since drag coefficient cannot be lower than the minimum drag coefficient for any airfoil, corrected drag coefficient is enforced to be above this number.

$$C_{d_{\text{cor.}}} = C_{d_{\min}} \text{ if } C_{d_{\text{cor.}}} < C_{d_{\min}}$$

Two options are available in VBM to calculate the tip loss factor as follows:

- **Quadratic Tip Loss Function**: Using this option, tip losses are approximated by a damping function inversely proportional to the square of the distance to the blade tip, starting from a user-specified radial threshold, named tip loss limit r_{tl} (default set at 96% of the rotor span R).

$$F = 1 - \left(\left(\frac{r}{R} - \frac{r_{tl}}{100} \right) \Big/ \left(1 - \frac{r_{tl}}{100} \right) \right)^2 \quad \text{if } \left(\frac{r}{R} > \frac{r_{tl}}{100} \right)$$

- **Modified Prandtl's Tip Loss Function**: Prandtl's tip loss function is generally used in Blade Element Momentum Theory (BEMT) that equates two set of equations of the momentum balance on a rotating annular stream tube passing on a disk (momentum theory, or MT) and the forces acting on the blade element at the corresponding sections along the blade span (blade element theory or BET). In VBM, however, Reynolds-averaged Navier–Stokes equations (RANS) and BET are applied such that RANS can capture the tip loss effects near the tip of rotor disk. However, that does not represent the tip loss near the tip of the actual rotor. Therefore, a modified version of this function is used in VBM such that you can tune it for each application. In VBM, the tip loss factor F is calculated using:

$$F = \frac{2}{\pi} \cos^{-1}\left(e^{-gf} \right) \quad \& \quad f = \frac{N}{2} \frac{R - r}{r \sin(\varphi)}$$

  where g is the tuning coefficient and f is given in terms of the number of the blades N, the radial position of blade element r, the rotor radius R, and induced inflow angle φ, defined as the angle between the normal and tangential components of relative incident velocity at each blade section where an element exists.

$$\varphi = \tan^{-1}\left(\frac{V_n}{V_t} \right)$$

  The tuning coefficient g should be larger than 1, set by you in the VBM user interface.

18.3. Meshing Guidelines and Creating the VBM Disk

Fluent VBM replaces the rotor system with momentum sources acting on a layer of cells for each rotor zone. These cells can be marked in the mesh as a separate cell zone, called **Embedded Disk**, or can be searched in the mesh when cut by a disk surface geometry, called **Floating Disk**. In this section, the meshing guideline for each mode and general considerations to create a proper mesh for VBM simulations is described.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

18.3.1. Embedded Disk

For **Embedded Disk**, an internal surface in the shape of a flat disk should be created in the flow domain at the location of the rotor. In the CAD, this surface should be marked as an internal wall or internal surface that will be discretized by a single layer of nodes, such that the mesh is continuous across the surface (for example, no duplicate nodes). The dead part of the rotor near the hub - the cutout - can be neglected. Figure 18.5: Schematic of a Cut Through a Structured VBM Grid. The Green Cells Are Assigned to the VBM (p. 2343) shows the disk analogue of a simple rotor and its cutout.

A single uniform layer of prisms or hexahedral cells should be extruded from the cell faces attached to the disk in the direction normal to the disk. Although there is no definitive rule regarding the thickness of these cells, it is recommended that the layer remains between 1/40th to 1/100th of the diameter of the disk. The single prismatic cell layer in the direction downstream of the disk should be marked as a separate zone (rotor zone).

Note:

Pyramids and tetra cells attached to the disk cells to provide the blending with the rest of the grid are part of the flow domain, not the rotor zone.

18.3.2. Floating Disk

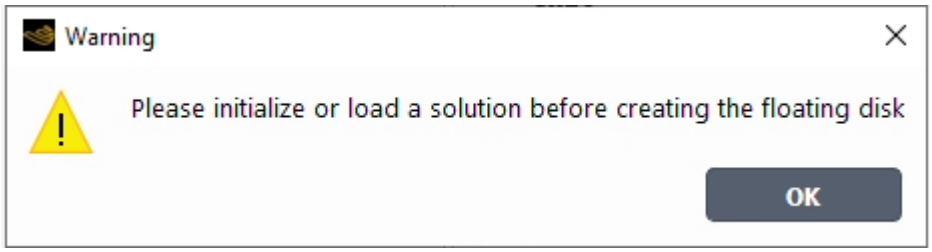
For the **Floating Disk** mode, there is no need to create any rotor disk and corresponding prismatic layers in the mesh. Instead, the floating disk can be created through the VBM's graphical user interface based on the rotor geometry definition and orientation (**Rotor Radius [m]**, **Rotor Root Cutout [m]**, **Rotor Disk Origin**, and **Rotor Disk Normal Vector**), see Meshing Guidelines and Creating the VBM Disk (p. 2341) for more details. In case the disk orientation is set based on **Rotor Disk Angles**, Fluent will calculate the disk normal vector components automatically. After pressing **Create Floating Disk** for each **Active Rotor Zone**, the following steps will take place to create and select the floating disk:

- Creates a **Custom Field Function** named **radius-fdisk-RotorName** to calculate the radius from the **Rotor Disk Origin**. (**RotorName** refers to the **Active Rotor Name**)
- Creates a **Plane Surface** named **plane-fdisk-RotorName** based on the **Rotor Disk Origin** and **Rotor Disk Normal Vector**.
- Creates the floating disk, named **srf-fdisk-RotorName** by cutting the **plane-fdisk-RotorName** using an iso-clip to include the area with **radius-fdisk-RotorName** between **Rotor Root Cutout [m]** and **Rotor Radius [m]**.
- Selects **srf-fdisk-RotorName** in the graphical user interface under **Rotor Surface**.

Note:

The iso-clip will only be activated if a solution is present. Therefore, an initial solution should be loaded or initialized before pressing **Create Floating Disk**. Otherwise, Fluent will give a warning message.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



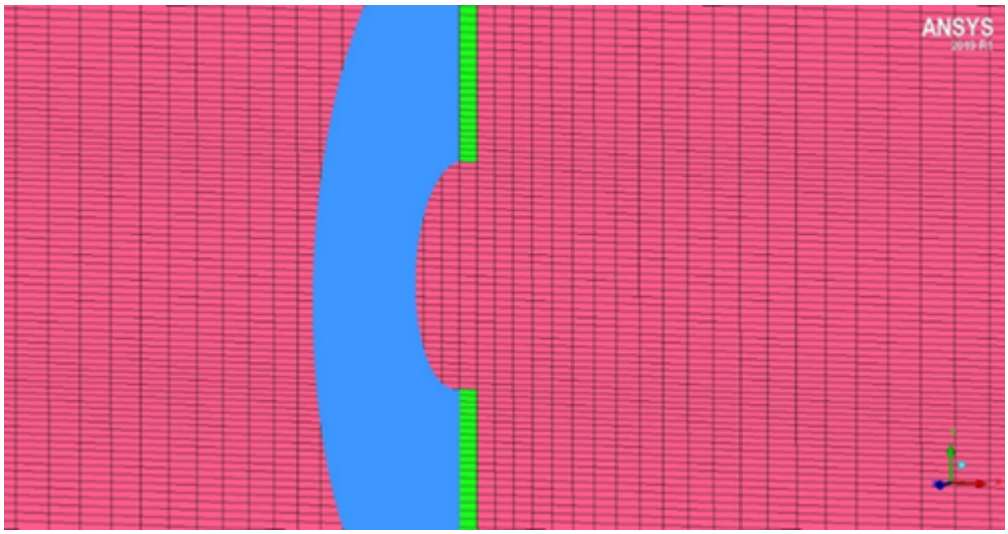
The floating disk can be any surface with any shape and not necessarily planar like the disk created automatically through the graphical user interface. There are several ways to create a surface in Fluent. For non-planar surface (for example, a conical surface), you can imprint the disk from a `.stl` file.

18.3.3. General Considerations

The VBM supports both structured and unstructured hybrid grids. Mesh clustering is recommended near the disk and on the inner and outer circumference of the disk since these are regions where interference effects and loss models are usually acting. The current default quadratic tip loss model, for example, will not produce smooth solutions if the grid in the range $0.96 \leq r/R \leq 1.0$ is very coarse.

Attention must be paid to the size of the inlet/exit boundaries of the computational domain, particularly for heavy helicopters in hover, to ensure that the mass flow rate of air passing through the domain is many times larger than the mass flow through all the rotors. If this detail is overlooked the overall simulation might diverge, converge very poorly, or the rotors may not be able to produce the desired amount of thrust. The reason is that VBM simulations (and also higher-fidelity methods, such as the MRF) are over-specified: while rotorcraft or propeller-driven aircraft will fly at a speed resulting from the balance of forces acting on them, in the CFD simulation this speed is specified by you. It is then possible to specify speeds that are incompatible with the settings of the rotor. For this reason, for example, a suitable non-zero mass flow rate should be specified on the farfield boundaries of a helicopter in hover.

Figure 18.5: Schematic of a Cut Through a Structured VBM Grid. The Green Cells Are Assigned to the VBM



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Finally, the grid in the downstream of the rotor should have sufficient clustering to capture the complex effects of the momentum change imparted by the VBM and the interaction of the wake of the rotor with other rotors (for example, main and tail rotors of a helicopter) or with other components of the aircraft/rotorcraft. Since the load distribution of a rotor is generally non-uniform, secondary vortices are likely to appear and should be captured in as much detail as possible.

18.4. Airfoil File Format

VBM results will only be as accurate as the lift and drag data supplied for each blade section. The blade is represented by a stack of airfoils and therefore multiple airfoil files can be used. Rotating components can have high tip velocities, therefore, if possible, the airfoil data file should contain lift and drag data for a range of Reynolds and Mach numbers sufficient to cover the entire spectrum of aerodynamic conditions that will be encountered. Since the receding blades of helicopters in forward flight can experience reversed flow, the data must be provided for the full $-180^{\circ} \leq \text{AoA} \leq +180^{\circ}$ spectrum of angles of attack. The names of the airfoil data files present in the working directory should always have the `.dat` suffix, as in `naca0012.dat`. The airfoil file follows the format listed in Table 18.1: Format of the Airfoil Data File (p. 2344).

Table 18.1: Format of the Airfoil Data File

`Airfoilxyz`	Name of the airfoil (same as the airfoil file name and 30 characters maximum)
`Itot`	Total number of C_L and C_D tables in the file (25 airfoil tables maximum (cl plus cd))
`Cl`	Label at the beginning of the C_L table. (10 characters maximum)
`Re`	Reynolds number of the table
`Ma`	Mach number of the table
`Jtot`	Number of lines in the subsequent table
`-180.0 0.00`	Angle of attack and C_L values (two blanks-separated columns and 250 data points maximum)
...	
`-45.0 -1.50`	
...	
`0.0 0.0`	
...	
`45.0 1.50`	
...	
`180.0 0.00`	
`Cd`	Label at the beginning of the C_D table. (10 characters maximum)
`Re`	Reynolds number of the table
`Ma`	Mach number of the table
`Jtot`	Number of lines in the subsequent table

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

`-180.0 0.006`	Angle of attack and C_D values (two blanks-separated columns and 250 data points maximum)
…	
`180.0 0.006`	
…	Repeat, if more C_L and C_D tables are available.

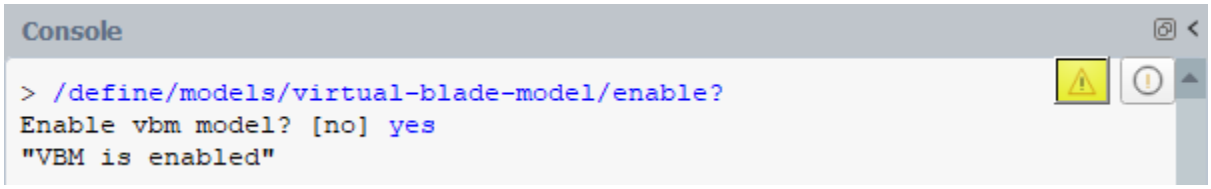
The following shows an example of an airfoil data file.

```
naca0015
2
cl
100000.0
0.1
 41
     -180.00  0.0000
     -172.00  0.7800
     :
        0.00  0.0000
     :
      172.50 -0.7800
      180.00  0.0000
cd
100000.0
0.1
71
     -180.00 0.0220
     -175.00 0.0620
     :
        0.00 0.0088
      :
      175.00 0.0620
      180.00 0.0220
```

18.5. Enabling the VBM

The Fluent VBM can be enabled through the through **VBM Rotor Inputs** dialog box or text user interface (TUI). Use the following TUI command in the Fluent **Console** to enable the VBM:

`define/models/virtual-blade-model/enable?`



Note:

The VBM is only available in 3D and **Parallel** mode. The **Serial** option is deprecated.

The VBM can only be enabled when a valid Ansys Fluent case or mesh file has been set or read.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

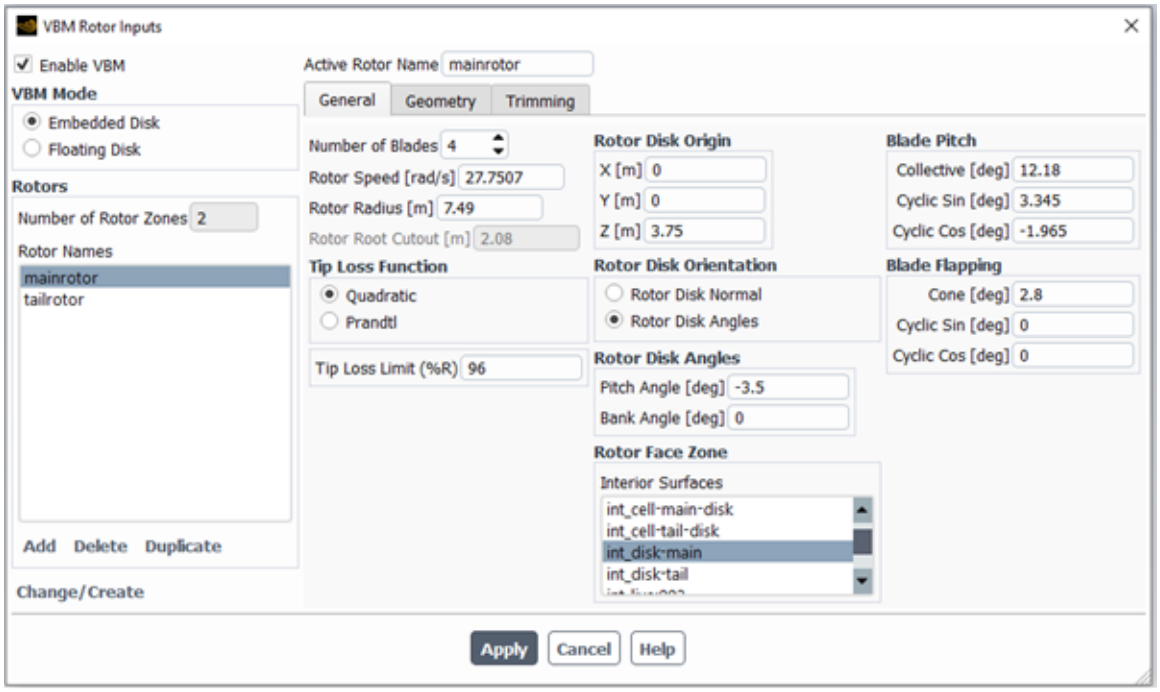
VBM's current compatibility is limited as it does not work with multiphase or inviscid flow. If any of these are active, Fluent will prevent VBM from being enabled and will display a warning message in the **Console** or a pop-up message.

18.6. VBM Configuration

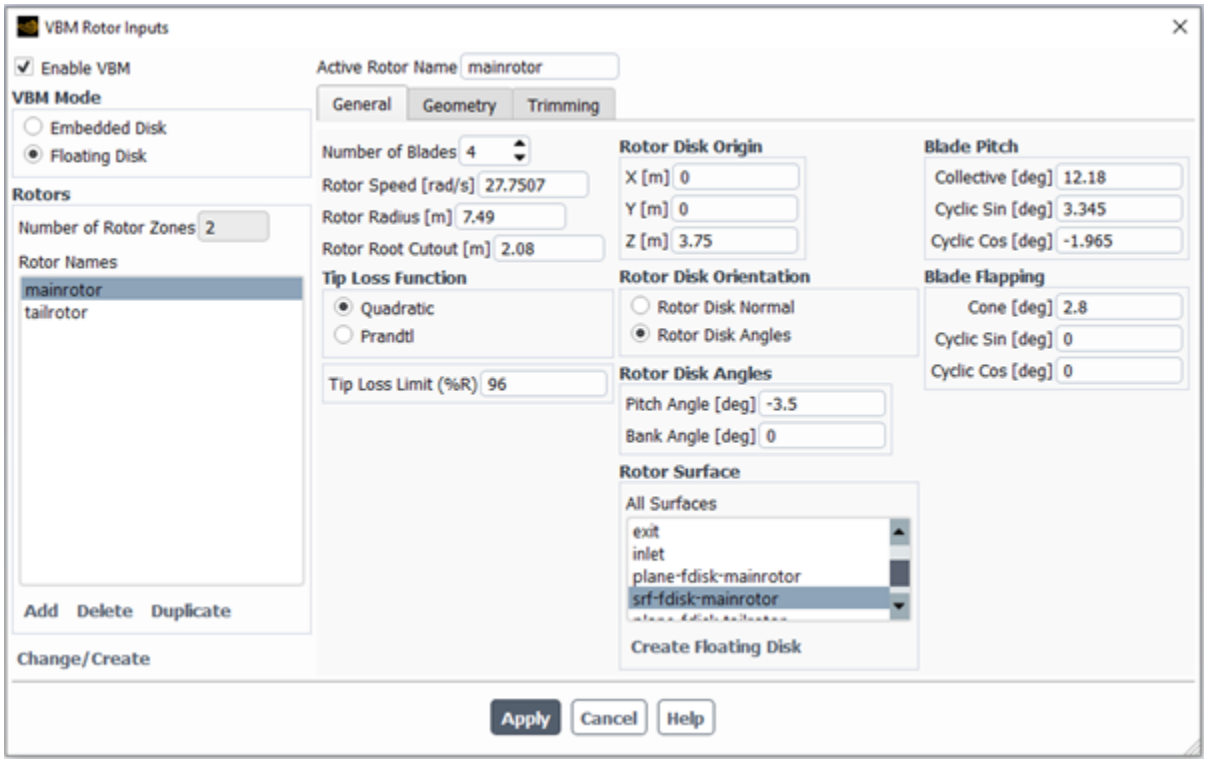
VBM can be configured through its graphical user interface (GUI), **VBM Rotor Inputs** dialog box or text user interface (TUI). This section discusses the VBM graphical user interface. VBM text commands are listed in the Fluent Text Command List.

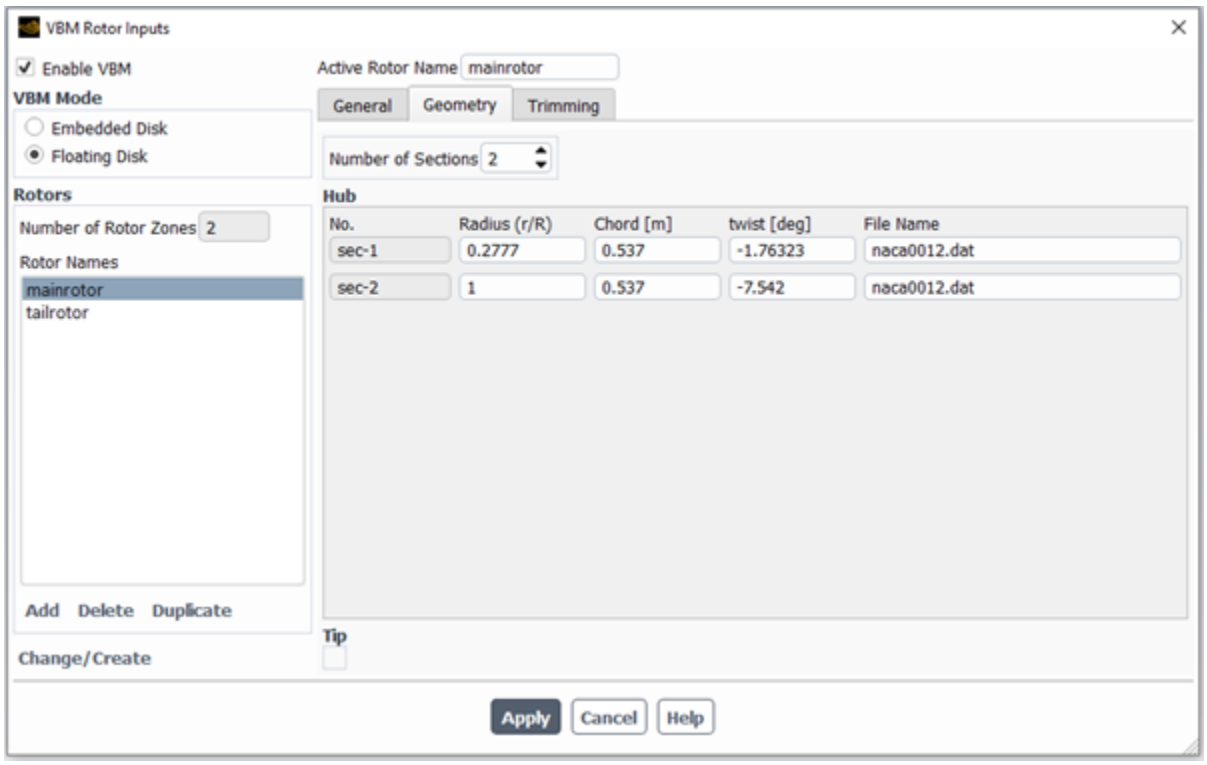
Depending on the VBM mode (**Embedded Disk** / **Floating Disk**) and running conditions for each rotor, the graphical user interface will display up to three configuration panels, accessible through the following tabs, **General**, **Geometry** and **Trimming**, as shown in Figure 18.6: VBM Graphical User Interface Configuration Panels (p. 2346). Parameters are the same for **Embedded Disk** and **Floating Disk** modes, unless otherwise stated for each mode separately. The **General** tab for these modes is different, however, **Geometry** and **Trimming** tabs are identical.

Figure 18.6: VBM Graphical User Interface Configuration Panels

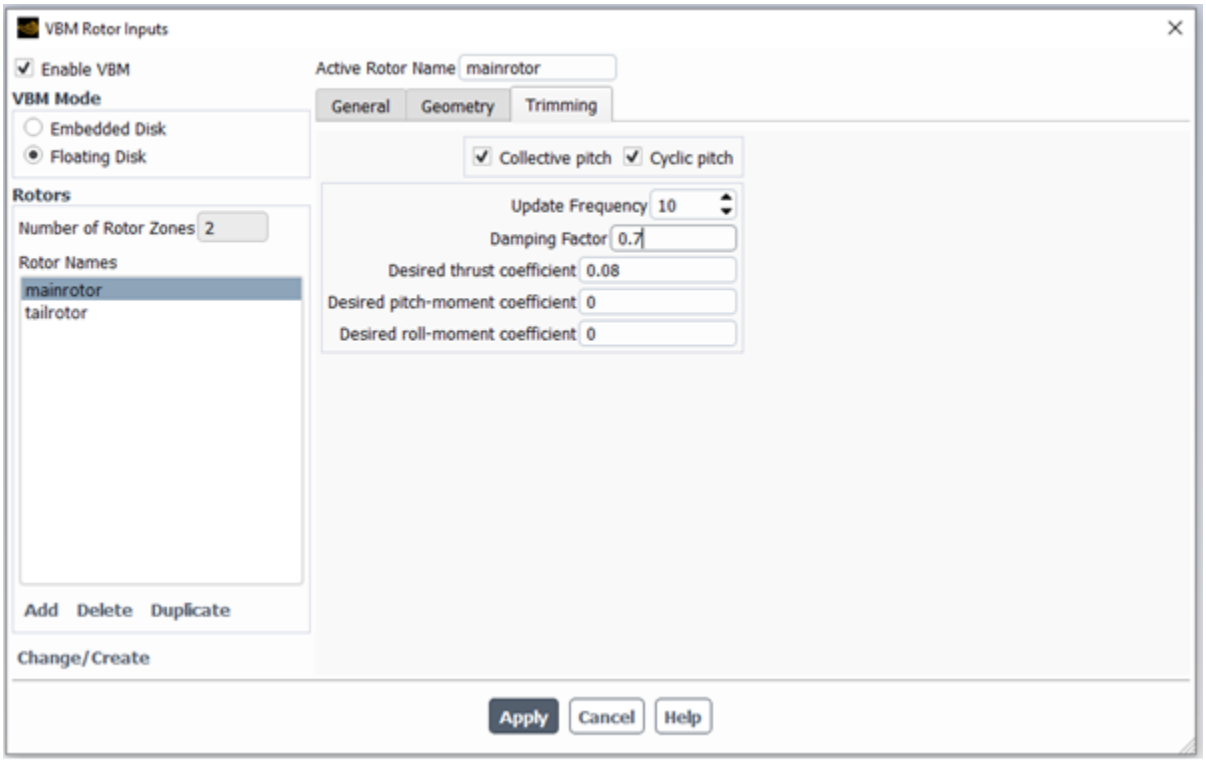


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Follow these steps to configure the rotor inputs:

1. Select one of the following **VBM Mode**: **Embedded Disk** or **Floating Disk**. Since the **VBM Mode** for all **Rotor Zones** must be the same, you can only select it for the first **Rotor Zone**. This option is not active for other **Rotor Zones**.

2. Press **Add** under **Rotor Names** list to create a new VBM rotor or select an existing rotor name from the list to edit. The total **Number of Rotors Zones** are shown above the list. The VBM supports up to 25 rotor disks.

3. Enter the **Active Rotor Name** or retain the default name.

4. Enter the **Number of Blades**.

5. Enter the **Rotor Speed [rad/s]**. The rotation direction follows the right-hand rule with respect to rotor normal vector.

6. Enter the **Rotor Radius [m]** which is corresponding to the rotor tip.

7. Enter the **Rotor Root Cutout [m]**. This entry is enabled when using **Floating Disk** mode only.

8. Select **Tip Loss Function**:

   - Enter the **Tip Loss Limit** if choosing **Quadratic** function. (% of normalized rotor radius, default is 96%. A value of 100% means that tip losses will not be introduced)

   - Enter **Tuning Coefficient** if using **Prandtl**'s function. (Set a value >= **1**; default is **1**)

9. To set the **X**, **Y**, **Z** coordinates of **Rotor Disk Origin**:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Enter **X**, **Y**, **Z** coordinates of the center of the rotor disk cell zone (not disk center) if using **Embedded Disk Mode**.
- Enter **X**, **Y**, **Z** coordinates of the center of the rotor disk if using **Floating Disk Mode**.

10. To set **Rotor Disk Orientation**:
    - Enter the **Pitch and Bank Angle [deg]** if choosing the **Rotor Disk Angles** option (see Figure 18.1: Rotor Disks Schematic (p. 2335))
    - Enter **Disk Normal X, Y, Z** if choosing the **Rotor Disk Normal** option.
11. To represent the rotor:
    - Select the **Rotor Face Zone** from the **Interior Surfaces** list when using **Embedded Disk Mode**.
    - Select **Rotor Surface** from the **All Surfaces** list if using **Floating Disk Mode**.
12. Follow these steps to create the **Floating Disk** when using **Floating Disk Mode**:
    - Set **Rotor Radius**, **Rotor Root Cutout**, **Rotor Disk Origin** and **Rotor Disk Orientation** in advance.
    - Before creating the disk, an initial solution should be loaded or initialized. Otherwise, Fluent will give a warning message.
    - Press **Create Floating Disk**.

    Fluent will create and select the **Rotor Surface** named **srf- fdisk-RotorName** (**RotorName** refers to the **Active Rotor Name**). See Floating Disk (p. 2342) for more information.
13. Enter the initial blade **Collective [deg]** pitch angle θ_0.
14. Enter the **Cyclic Sin [deg]** and **Cyclic Cos [deg]** coefficients (amplitude) of the **Blade Pitch** angle. (θ_{1s}, θ_{1c})
15. Enter the **Cone [deg]** flapping angle (β_0).
16. Enter the **Cyclic Sin [deg]** and **Cyclic Cos [deg]** coefficients of the **Blade Flapping** angle. (β_{1s}, β_{1c})
17. Click the **Geometry** button on the ribbon.
18. Enter the **Number of Sections**. Up to 25 are supported by default.
19. For each section, starting at the innermost location (cutout), enter the **Radius (r/R)**, **Chord [m]** length, **twist [deg]** angle and **File Name** of the airfoil file containing the CL and CD data for the section (with or without the `.dat` suffix).

Note:

All files containing the airfoil tables must have `.dat` suffix. If omitted from the airfoil file name entered in the graphical user interface, it will be added automatically. Whether the

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

name **naca0012** or **naca0012.dat** is entered into the graphical user interface, a file named naca0012.dat must exist in the project directory.

20. Click the **Trimming** tab on the ribbon, if trimming is required for the active rotor.

21. Enable **Collective pitch** and/or **Cyclic pitch**. Depending on the trimming operation selected, additional input boxes will be activated.

22. Enter the **Damping Factor** (or under-relaxation factor). Default value is 0.1, but faster convergence can be achieved by increasing this value. Max value is 1, recommended value is 0.7.

23. Enter values for the **Desired thrust coefficient**.

24. The **Desired pitch-moment** and **Desired roll-moment** coefficients are normally left at 0, however non-zero values can be entered if a pitch or roll moment is required.

25. Click the **Change/Create** button to save the data for this rotor.

26. Repeat steps 2 – 25 to configure the other rotor zones.

27. Click the **Apply** button to apply VBM and perform pre-processing input data. This will also terminate and close the **VBM Rotor Inputs** dialog box.

After the initial configuration, every time a change is made to one or more of the parameters, the **Change/Create** button must be clicked before switching to another rotor or clicking on the **Apply** button. Clicking on the **Change/Create** button saves the data of each rotor to a pointer, whereas clicking on the **Apply** button stores the data in a Fluent memory pointer, calculate some pre-processing values, and closes the VBM graphical user interface. When the .cas file is saved, the information of all the rotors will be saved as a pointer in the file.

18.7. VBM Field Variables

VBM shows 15 VBM data fields for each computational cell in the entire domain. The VBM variables are cell-based and will not display correctly as node variables. They are calculated only in the cells attached to the actuator disks. The labels of the VBM parameters are listed in Table 18.2: VBM Field Variables (p. 2350).

Table 18.2: VBM Field Variables

Variable Name	Description
VBM Radius	Radius
VBM Incident Velocity	Airflow relative incident velocity
VBM Pitch Angle	Effective Pitch angle (as defined in Blade Pitch)
VBM Chord	Blade chord
VBM Reynolds Number	Reynolds number
VBM Mach Number	Mach number
VBM AoA	Angle of attack
VBM cl	Lift coefficient

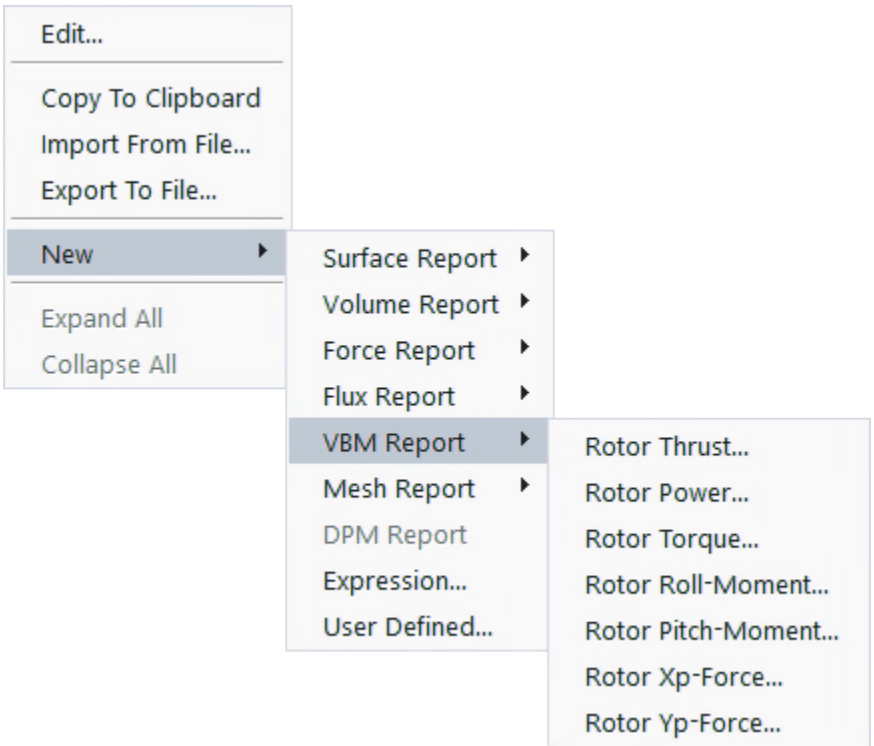
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Variable Name	Description
VBM cd	Drag coefficient
VBM Normal Force	Normal component of instantaneous rotor force
VBM Tangential Force	Tangential component of instantaneous rotor force
VBM Radial Force	Radial component of instantaneous rotor force
VBM X-Force	x-component of time-averaged force ($F'_{L,D}$)
VBM Y-Force	y-component of time-averaged force ($F'_{L,D}$)
VBM Z-Force	z-component of time-averaged force ($F'_{L,D}$)

18.8. VBM Monitoring and Report Definition

By default, VBM writes the `Rotor_xx_Loads.csv` files for each rotor `xx` to report the VBM global outputs. These outputs include the force, moment and power on the rotor disk, as defined in Table 18.3: VBM Global Outputs for Each Rotor (p. 2352), considering the local coordinates $x'y'z'$ shown in Figure 18.1: Rotor Disks Schematic (p. 2335). In this table, x' , y' and z' repesents the local coordinates and r stands for the radius of a facet cell. The components of the time-averaged force on the facet cell are denoted as $F_{x'}$, $F_{y'}$, $F_{z'}$ and F_t represents the tangential force on the facet cell. Within this table, V_t=$R\Omega$ is the rotor speed, A_d=πR^2 is the rotor disk area, R is the rotor radius and Ω is the rotational speed of the rotor. Throughout the simulation, you have the option of monitoring, printing, writing and/or plotting these global solutions for any rotor using VBM report definitions. To set up VBM reports using the **VBM Report Definition** dialog box, go to:

Solution → **Reports** → **Definitions** → **New...** → **VBM Report** → **Rotor...**



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

From there, you can choose one of the available report type options. Additionally, you can set the **Output Type** to **Coefficient** if required.

It's important to define VBM reports individually if you intend to have multiple report definitions printed to the **Console**, written to a file or plotted in the **Graphics** window. This is useful for monitoring convergence and recording time histories.

Table 18.3: VBM Global Outputs for Each Rotor

Output Type	Definition - Quantity	Definition - Coefficient
Xp-force (N)	$\mathcal{F}_{x'}=\sum F_{x'}$	$C_{F_{y'}}=\frac{\mathcal{F}_{x'}}{\rho_{ref}V_t^2A_d}$
Yp-force (N)	$\mathcal{F}_{y'}=\sum F_{y'}$	$C_{F_{y'}}=\frac{\mathcal{F}_{y'}}{\rho_{ref}V_t^2A_d}$
`Thrust(N)`	$T=\sum F_{z'}$	$C_T=\frac{T}{\rho_{ref}V_t^2A_d}$
`Roll-moment(N·m)`	$M_{x'}=\sum F_{z'}*y'$	$C_{M_{x'}}=\frac{M_{x'}}{\rho_{ref}V_t^2A_dR}$
`Pitch-moment(N·m)`	$M_{y'}=\sum F_{z'}*x'$	$C_{M_{y'}}=\frac{M_{y'}}{\rho_{ref}V_t^2A_dR}$
`Torque(N·m)`	$Q=\sum F_t*r$	$C_Q=\frac{Q}{\rho_{ref}V_t^2A_dR}$
`Power(W)`	$P=Q*\Omega$	$C_P=\frac{P}{\rho_{ref}V_t^3A_d}$

Note:

When reading a Fluent data file, the new collective and cyclic pitches (calculated during rotor trimming) will be printed to the **Console**. However, they will not be updated within the VBM setup. If you want to continue the simulation using new collective and cyclic pitches, you should update your VBM setup manually.

18.9. References

1. Chaffin, M.S., "A guide to the use of the pressure disk rotor model as implemented in INS3D-UP", NASA CR-4692, September 1995.

2. Zori, L.A.J., Rajagopalan, R.G., "Navier-Stokes Calculation of Rotor-Airframe Interaction in Forward Flight", Journal of the American Helicopter Society, Vol.40, April 1995.

3. Stepniewski, W.Z. and Keys, C.N., "Rotary-Wing Aerodynamics," Dover Publications Inc., New York, USA, 1984.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Leishman, J.G., "Principles of Helicopter Aerodynamics", Cambridge Aerospace Series, Cambridge University Press, 2000.

5. "Ansys Fluent Customization Manual 2019R3," ANSYS, Inc., Canonsburg, PA, USA.

6. Yang, Z., Sankar L.N., Smith, M., Bauchau, O., "Recent Improvements to a Hybrid Method for Rotors in Forward Flight", AIAA Conference Reno, 2000.

7. "Fluent User's Guide 2019R3," ANSYS, Inc., Canonsburg, PA, USA.

8. Capitao Patrao, A., "Description and Validation of the rotorDiskSource Class for Propeller Performance Estimation," Proceedings of CFD with Open Source Software, edited by Nilsson, H., 2017.

9. Liou, S.G., Komerath, N.M., McMahon, H.M, "Velocity Measurements of Airframe Effects on a Rotor in Low-Speed Forward Flight" J. Aircraft Vol.26, No.4, 1989.

10. Brand, A., Komerath, N., McMahon, H., "Results from Laser Sheet Visualization of a Periodic Rotor Wake", J. Aircraft Vol.26, No.5, 1989.

11. Liou, S.G., Komerath, N.M., McMahon, H.M., "Velocity Field of a Cylinder in the Wake of a Rotor in Forward Flight", J. Aircraft Vol.27, No.9, 1990.

12. Liou, S.G., Komerath, N.M., McMahon, H.M, "Measurements of the Interaction Between a Rotor Tip Vortex and a Cylinder", AIAA J. Vol.28, No.6, 1990.

13. Marvis, D.N., Komerath, N.M., McMahon, H.M., "Prediction of Aerodynamic Rotor-Airframe Interaction in Forward Flight", Journal of the American Helicopter Society, Vol.34, No.4, 1989.

14. Brand, A.G., Komerath, N.M., McMahon, H.M., "Windtunnel data from a rotor wake/airframe interaction study", Georgia Institute of Technology, US Army research contract No. DAAG 29-82-K-0094, 1986.

15. Prouty, R., "Helicopter Performance, Stability and Control", Notes by The University of Kansas Continuing Education, September 13-17, 2004.

16. Lorber, P.F., Egolf, T.A., "An Unsteady Helicopter Rotor-Fuselage Aerodynamic Interaction Analysis", Journal of the American Helicopter Society, Vol.35, No.3, 1990.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 19: Modelling with Finite-Rate Chemistry

This chapter contains information about the following finite-rate chemistry modeling capabilities in Ansys Fluent:

19.1. Modeling Species Transport and Finite-Rate Chemistry
19.2. Modeling a Composition PDF Transport Problem
19.3. Using Chemistry Acceleration

19.1. Modeling Species Transport and Finite-Rate Chemistry

Species transport modeling capabilities, both with and without reactions, and the inputs you provide when using the model are described in this section. For theoretical information about species transport, see Species Transport and Finite-Rate Chemistry in the Theory Guide.

Note that you may also want to consider modeling your turbulent reacting flame using one of the following approaches:

- Mixture fraction approach for non-premixed systems (Modeling Non-Premixed Combustion (p. 2471))
- Reaction progress variable approach for premixed systems (Modeling Premixed Combustion (p. 2536))
- Partially premixed approach (Modeling Partially Premixed Combustion (p. 2543))
- Composition PDF Transport approach (Modeling a Composition PDF Transport Problem (p. 2442))
- Multiphase species transport and finite-rate chemistry (Modeling Multiphase Flows (p. 2915))

For simulations using Ansys Fluent reacting flow models, you can specify mixtures consisting of up to 700 chemical species.

The following models and features may not be used with more than 50 species:

- Density-based solver
- Eulerian PDF Transport model
- Melting/solidification model
- Crevice model
- Thermal diffusivity model
- Surface species

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Site species

Note:

If the species model is used in combination with a multiphase model, the mass fraction of a species may be displayed incorrectly in Ansys CFD-Post if the volume fraction of a cell is zero. This applies for all Ansys Fluent files written prior to release 19.2.

For all multiphase, multi-species simulations written in release 19.2 and beyond in the legacy format (`.dat` files), the mass fractions of species will not be available in Ansys CFD-Post, unless you add the mass fraction of a species as an additional quantity to be saved in the data file (through the **Data File Quantities** dialog box).

Information is divided into the following sections:

19.1.1. Volumetric Reactions
19.1.2. Wall Surface Reactions and Chemical Vapor Deposition
19.1.3. Particle Reactions
19.1.4. Electrochemical Reactions
19.1.5. Species Transport Without Reactions
19.1.6. Reacting Channel Model
19.1.7. Reactor Network Model

19.1.1. Volumetric Reactions

Information about using species transport and finite-rate chemistry as related to volumetric reactions is presented in the following subsections. For more information about the theoretical background of volumetric reactions, see Volumetric Reactions in the Theory Guide.

19.1.1.1. Overview of User Inputs for Modeling Species Transport and Reactions
19.1.1.2. Enabling Species Transport and Reactions and Choosing the Mixture Material
19.1.1.3. Importing a Volumetric Kinetic Mechanism in CHEMKIN Format
19.1.1.4. Defining Properties for the Mixture and Its Constituent Species
19.1.1.5. Setting up Coal Simulations with the Coal Calculator Dialog Box
19.1.1.6. Defining Cell Zone and Boundary Conditions for Species
19.1.1.7. Defining Other Sources of Chemical Species
19.1.1.8. Solution Procedures for Chemical Mixing and Finite-Rate Chemistry
19.1.1.9. Postprocessing for Species Calculations

19.1.1.1. Overview of User Inputs for Modeling Species Transport and Reactions

The basic steps for setting up a problem involving species transport and reactions are listed below.

1. Select species transport and volumetric reactions, and specify the mixture material. See Enabling Species Transport and Reactions and Choosing the Mixture Material (p. 2359). The mixture material concept is explained in Mixture Materials (p. 2357).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. If you are also modeling wall or particle surface reactions, select wall surface and/or particle surface reactions. See Wall Surface Reactions and Chemical Vapor Deposition (p. 2400) Combusting Particle Surface Reactions (p. 2406) for details.

3. Check and/or define the properties of the mixture. See Defining Properties for the Mixture and Its Constituent Species (p. 2374). Mixture properties include the following:

   - species in the mixture
   - reactions
   - other physical properties (for example, viscosity, specific heat)

4. Check and/or set the properties of the individual species in the mixture. See Defining Properties for the Mixture and Its Constituent Species (p. 2374).

5. Set species cell zone and boundary conditions. See Defining Cell Zone and Boundary Conditions for Species (p. 2394).

In many cases, you will not need to modify any physical properties of mixture material because the solver gets species properties, reactions, and so on, from the materials database when you choose the mixture material. Some properties, however, may not be defined in the database. You will be warned when you choose your material if any required properties need to be set, and you can then assign appropriate values for these properties. You may also want to check the database values of other properties to be sure that they are correct for your particular application. For details about modifying an existing mixture material or creating a new one from nothing, see Defining Properties for the Mixture and Its Constituent Species (p. 2374). Modifications to the mixture material can include the following:

- Addition or removal of species
- Changing the chemical reactions
- Modifying other material properties for the mixture
- Modifying material properties for the mixture's constituent species

If you are solving a reacting flow, you will usually want to define the mixture's specific heat as a function of composition, and the specific heat of each species as a function of temperature. You may want to do the same for other properties as well. By default, most species specific heats in the database are piecewise-polynomial functions of temperature, but you may choose to specify a different temperature-dependent function if you know of one that is more suitable for your problem.

19.1.1.1.1. Mixture Materials

The concept of mixture materials has been implemented in Ansys Fluent to facilitate the setup of species transport and reacting flow. A mixture material may be thought of as a set of species and a list of rules governing their interaction. The mixture material carries with it the following information:

- A list of the constituent species, referred to as "fluid" materials

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- A list of mixing laws dictating how mixture properties (density, viscosity, specific heat, and so on) are to be derived from the properties of individual species if composition-dependent properties are desired
- A direct specification of mixture properties if composition-independent properties are desired
- Diffusion coefficients for individual species in the mixture
- Other material properties (for example, absorption and scattering coefficients) that are not associated with individual species
- A set of reactions, including a reaction type (finite-rate, eddy-dissipation, and so on) and stoichiometry and rate constants

The mixture materials are stored in the Ansys Fluent materials database. Many common mixture materials are included (for example, methane-air, propane-air). Generally, one or two-step reaction mechanisms and many physical properties of the mixture and its constituent species are defined in the database.

The Ansys Fluent materials database is accessed in one of two ways, depending on the sequence of your workflow:

- through the **Species Model** dialog box before any one of the species models is enabled
- through the **Create/Edit Materials** dialog box after one of the available species models is enabled

When you enable one of the species models, the mixture material that you have selected for your application, its constituent fluid materials, and properties will be loaded into your simulation. If any necessary information about the selected material (or the constituent fluid materials) is missing, the solver will inform you that you need to specify it. In addition, you may choose to modify any of the predefined properties. See Using the **Create/Edit Materials** Dialog Box (p. 1599) for information about the sources of Ansys Fluent database property data.

For example, if you plan to model combustion of a methane-air mixture using global kinetics, you do not need to explicitly specify the species involved in the reaction or the reaction itself. You will simply select **methane-air** as the mixture material to be used, and the relevant species (CH_4, O_2, CO_2, H_2O, and N_2) and reaction data will be loaded into the solver from the database. You can then check the species, reactions, and other properties and define any properties that are missing and/or modify any properties for which you want to use different values or functions. You will generally want to define a composition- and temperature-dependent specific heat, and you may want to define additional properties as functions of temperature and/or composition.

The use of mixture materials gives you the flexibility to select and load from the Fluent database one of the many predefined mixtures that is appropriate for your case or, if you want to create your own custom mixture material, a mixture-template (the default) consisting of H_2O, O_2, and N_2. Once the mixture material has been copied to the solver, you can modify it in the Create/Edit

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Materials Dialog Box (p. 4885), as described in Defining Properties for the Mixture and Its Constituent Species (p. 2374).

> **Warning:**
>
> Material mixtures use a copy of the original constituent fluid's properties, rather than pointing to those constituent materials. This means that if you modify any of the constituent material's properties, they will not be propagated to the mixture, unless the modification is made prior to creating the mixture.
>
> If you intend on creating a mixture where the constituent fluid's properties will change, rename the modified material to help you ensure you are clear on which material you are modifying and what it's properties are.

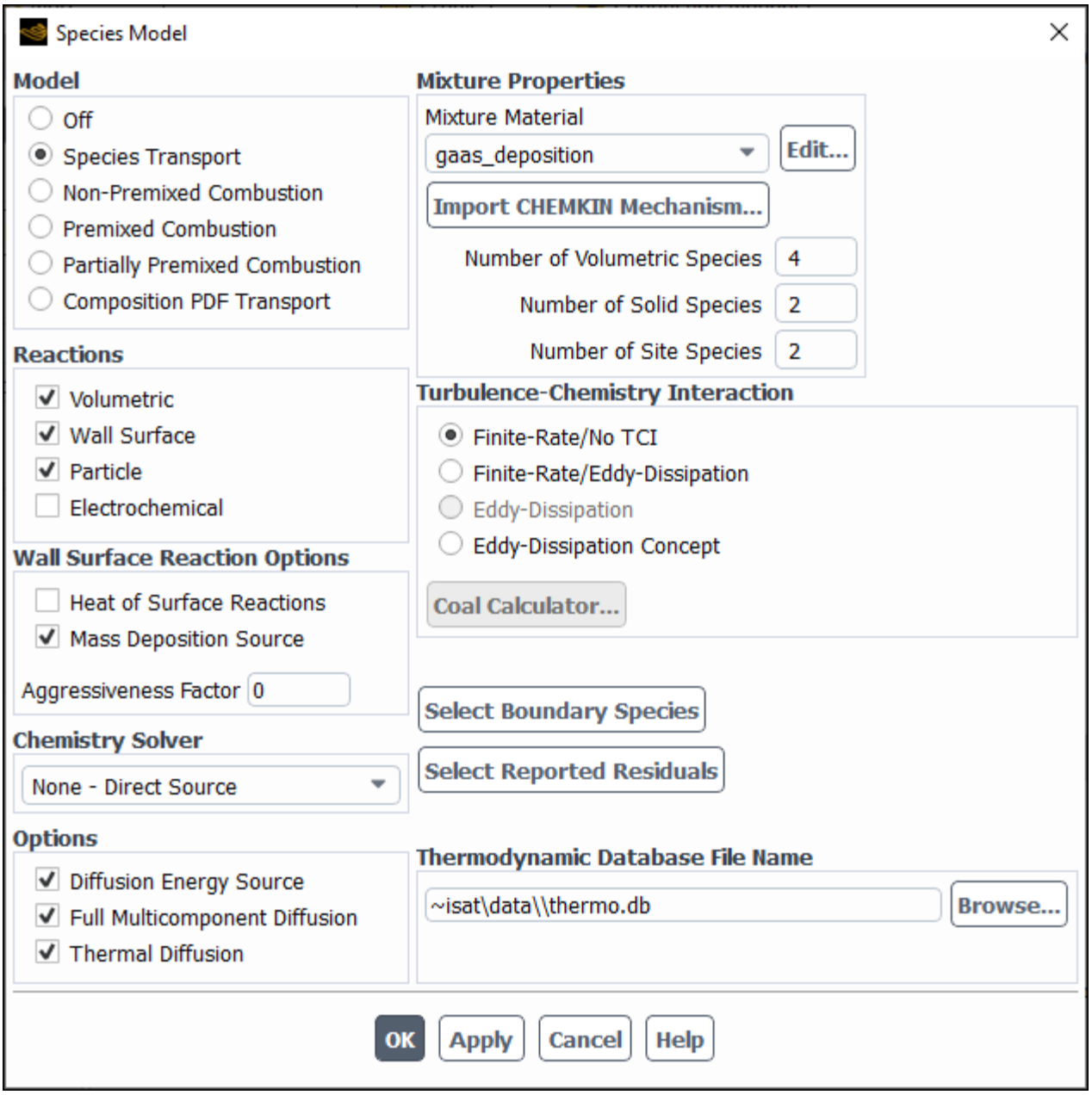
19.1.1.2. Enabling Species Transport and Reactions and Choosing the Mixture Material

The problem setup for species transport and volumetric reactions begins in the **Species Model** dialog box (Figure 19.1: The Species Model Dialog Box (p. 2360)). For cases that involve multiphase species transport and reactions, refer to Modeling Species Transport in Multiphase Flows in the Theory Guide.

Setup → **Models** → **Species** **Edit...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 19.1: The Species Model Dialog Box



1. Under **Model**, select **Species Transport**.

2. Under **Reactions**, select **Volumetric**.

3. In the **Mixture Material** drop-down list under **Mixture Properties**, select which mixture material you want to use in your problem or, if you want to create your own mixture material, retain the default selection of **mixture-template** consisting of H_2O, O_2, and N_2. The drop-down list includes all of the mixtures that are currently defined in the database. If there is a mixture material listed that is similar to your desired mixture, you may choose that material and modify it following the steps provided in Defining Properties for the Mixture and Its Constituent Species (p. 2374).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The number of species in the selected mixture material is displayed in the **Number of Volumetric Species** field and, if applicable, in the **Number of Solid Species** and **Number of Site Species** fields. To review the mixture composition and reactions of the selected Fluent database mixture material, click the **View...** button to the right of **Mixture Material**. This opens the **Fluent Database Materials** dialog box. Clicking, in turn, the **View...** buttons next to **Mixture Species** and **Reaction** opens the corresponding dialog boxes allowing you to examine the constituent species and reactions data. Note, that the mixture properties cannot be edited before the mixture material is copied to your Fluent problem. For this reason, all related controls are dimmed while viewing the mixture properties.

When you enable the species transport model by clicking either **OK** or **Apply**, Ansys Fluent performs the following:

- The selected Fluent database mixture material including all constituent fluid materials (species) and properties are automatically copied to your case and appear under **Mixture** in the tree and the **Materials** task page.
- If any properties for the selected mixture material (or the constituent fluid materials) are missing, the solver prints to the console a notice of the required data.
- When the mixture material is copied, the **Mixture Material** drop-down list in the **Species Model** dialog box changes to contain only the mixtures defined for your Fluent problem.
- The **View...** button is replaced by the **Edit...** button giving you an option to access the **Edit Material** dialog box (Edit Material Dialog Box (p. 4957)) from the **Species Model** dialog box.
- The **Create/Edit Materials** dialog box lists **mixture** in the **Material Type** drop-down list allowing you to access the Fluent mixture material database. If you want to check or modify any properties of the mixture material, use the Create/Edit Materials Dialog Box (p. 4885), as described in Defining Properties for the Mixture and Its Constituent Species (p. 2374).

You can add more mixture materials to your case by copying them from the database, as described in Copying Materials from the Ansys Fluent Database (p. 1602), or by creating a new mixture, as described in Creating a New Material (p. 1606) and Defining Properties for the Mixture and Its Constituent Species (p. 2374).

4. Select the **Turbulence-Chemistry Interaction** (TCI) modeling option. The following options are available:

   Finite-Rate/No TCI

   computes only the Arrhenius rate (see Equation 7.21 in the Theory Guide) and neglects turbulence-chemistry interaction. This model is recommended for laminar or turbulent flows using complex chemistry where either the turbulence time-scales are expected to be fast relative to the chemistry time scales or where the chemistry is sufficiently complex that the chemistry timescales of importance are highly disparate throughout the simulation. You can specify the following inputs:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Flow Iterations per Chemistry Update (steady-state only)

Increasing the number reduces the computational expense of the chemistry calculations. By default, Ansys Fluent will update the chemistry once per flow iteration. This option is not available when the **None - Direct Source** option is selected for **Chemistry Solver**.

Aggressiveness Factor (steady-state only)

This is a numerical factor that controls the robustness and the convergence speed. This value ranges between 0 and 1, where 0 is the most robust, but results in the slowest convergence. The default value for the **Aggressiveness Factor** is 0.5. This option is available only when **Stiff Chemistry Solver** or **CHEMKIN CFD Solver** is selected for **Chemistry Solver**.

Temperature Threshold

The reaction rate is highly temperature-dependent. When temperature is low, the reaction rate is fairly small and can be ignored. This may reduce the computational cost without sacrificing accuracy. Ansys Fluent sets the chemistry reaction rate to zero in cells where temperature is below the **Temperature Threshold** you specified. The default value is 200 K. This option is available only when **Stiff Chemistry Solver** or **CHEMKIN CFD Solver** is selected for **Chemistry Solver**.

Finite-Rate/Eddy-Dissipation (turbulent flows only)

computes both the Arrhenius rate and the mixing rate and uses the smaller of the two.

Eddy-Dissipation (turbulent flows only)

computes only the mixing rate (see Equation 7.39 and Equation 7.40 in the Theory Guide).

Eddy-Dissipation Concept (turbulent flows only)

models turbulence-chemistry interaction with detailed chemical mechanisms (see Equation 7.21 and Equation 7.42 in the Theory Guide). When using this model, you can select the following submodels in the **EDC Model** group box:

- **Constant Coefficients** (default): Is the standard EDC model described in The Standard EDC Model in the *Fluent Theory Guide*
- **Partially Stirred Reactor**: Is the partially stirred reactor EDC model described in The Partially Stirred Reactor EDC Model in the *Fluent Theory Guide*

For both standard and partially stirred reactor EDC models, you can adjust the following parameters:

Flow Iterations per Chemistry Update (steady-state only)

Increasing the number reduces the computational expense of the chemistry calculations. By default, Ansys Fluent will update the chemistry once per flow iteration.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Aggressiveness Factor (steady-state only)

This is a numerical factor that controls the robustness and the convergence speed. This value ranges between 0 and 1, where 0 is the most robust, but results in the slowest convergence. The default value for the **Aggressiveness Factor** is 0.5.

Temperature Threshold

The reaction rate is highly temperature-dependent. When temperature is low, the reaction rate is fairly small and can be ignored. This may reduce the computational cost without sacrificing accuracy. Ansys Fluent sets the chemistry reaction rate to zero in cells where temperature is below the **Temperature Threshold** you specified. The default value is 200 K.

For the **Constant Coefficients** model, you can specify the following parameters:

- **Volume Fraction Constant**: Is C_{ξ} in Equation 7.44 in the Theory Guide. The default value is recommended.
- **Time Scale Constant**: Is C_{τ} in Equation 7.45 in the Theory Guide. The default value is recommended.

For the **Partially Stirred Reactor** model, you can select the **Mixing Model** to compute C_{mix} in Equation 7.50 in the *Fluent Theory Guide*:

- **Constant Cmix**: Sets C_{mix} to a constant value specified in **Mixing Constant**
- **Dynamic Cmix** (default): Computes C_{mix} by Equation 7.51 in the *Fluent Theory Guide* with the fractal dimension D in Equation 7.52 in the *Fluent Theory Guide* specified in **Fractal Dimension**

In addition, you can use your own compiled user-defined functions to calculate EDC scales and to modify the reaction rate as described in `DEFINE_EDC_SCALES` and `DEFINE_EDC_MDOT` in the *Fluent Customization Manual*.

5. From the **Chemistry Solver** drop-down list, select the solver for your simulation (note that the solvers available in the drop-down list are dependent on the selected **Turbulence-Chemistry Interaction** model):
   - If you want the stiff chemistry to relax to chemical equilibrium without solving reactions, select **Relax to Chemical Equilibrium**. This option is not available for **Eddy Dissipation Concept**. For information about the Relaxation to Chemical Equilibrium model, refer to The Relaxation to Chemical Equilibrium Model in the Theory Guide

---

Important:

The **Relax to Chemical Equilibrium** chemistry solver requires thermodynamic data of all species to calculate chemical equilibrium. You can supply your own database using the **Import CHEMKIN Format Mechanism** dialog box (opened by clicking **Import CHEMKIN mechanism...**) (see Importing a Volumetric Kinetic Mechanism in CHEMKIN Format (p. 2368) and Importing a Surface Kinetic Mechanism in CHEMKIN Format (p. 2400) for details). A sample database is also contained in the file

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

`.../fluentx.x/cpropep/data/thermo.db`. If you define your own species in Ansys Fluent, as determined by the **Chemical Formula** in the **Create/Edit Materials** dialog box, and this species name is not in the thermo.db file, Ansys Fluent will report an error. To overcome this error, you must manually enter all the unknown species in the thermo file.

- If you want to solve finite-rate chemistry, select either **Stiff Chemistry Solver** or **CHEMKIN-CFD Solver**. These two solvers are available when using the **Finite-Rate/No TCI** or **Eddy-Dissipation Concept** TCI option.

  Selecting **CHEMKIN-CFD Solver** will enable you to use the Ansys CHEMKIN-CFD solution algorithms, which are based on and compatible with the Ansys Chemkin technology. Once you click **OK** or **Apply**, you will be prompted to import a CHEMKIN mechanism, if you have not already done so. Refer to the Ansys Chemkin Input and Theory manuals for details on the chemistry formulation options. See Using Ansys Encrypted Mechanisms (p. 2369) for instructions on establishing a CHEMKIN-mechanism based material. In addition, Appendix C: Controlling CHEMKIN-CFD Solver Parameters Using Text Commands (p. 5697) includes information about advanced solver settings and diagnostic information for the CHEMKIN-CFD solver. With the **Chemkin-CFD Solver** selected, the methods for mixture material properties will be defaulted to **chemkin**. See Procedure for Importing Volumetric CHEMKIN Mechanisms (p. 2369) for details.

- If you want to make explicit use of chemistry source terms in the species transport equations, without a stiff-chemistry solver, select **None - Direct Source**. This option is desirable for use with the density-based solver, where the flow and species transport equations are solved monolithically and the explicit use of the chemistry source terms allows the strongest equation coupling. It is however not recommended with the pressure-based solver, where one of the relaxation or stiff-chemistry solvers should be used instead.

6. (optional) For TCI model options that consider finite-rate chemistry, set the integration parameters using the **Integration Parameters** dialog box (opened by clicking the **Integration Parameters...** button). In the **Integration Parameters** dialog box, you can the following select acceleration methods (see Using Chemistry Acceleration (p. 2456) for more details):

   - For **Stiff Chemistry Solver** (single-phase flow): **Dynamic Mechanism Reduction** (DMR)

     See Using Dynamic Mechanism Reduction (p. 2461) for details.

   - For **Stiff Chemistry Solver** (single-phase flow) or **Relax to Chemical Equilibrium**: **Chemistry Agglomeration** (CA)

     See Using Chemistry Agglomeration (p. 2465) for details.

   - For any chemistry solver: **ISAT**

     See Using ISAT (p. 2457) for details.

   - For the **CHEMKIN-CFD Solver**: **Dynamic Cell Clustering** (DCC))

     See see Using Dynamic Cell Clustering (p. 2467) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. (optional) Select any **Options** desired. If you want to model full multicomponent (Stefan-Maxwell) diffusion or thermal (Soret) diffusion, select the **Full Multicomponent Diffusion** or **Thermal Diffusion** option, respectively.

   See Full Multicomponent Diffusion in the *Fluent Theory Guide* and Multicomponent Method Inputs (p. 1672) for details.

8. Enable the **Thickened Flame Model** when modeling unsteady laminar flames, or when using an LES model for turbulent premixed and partially-premixed flames. In the **Thickened Flame Model Options** group box, specify:

   - **Efficiency Function**: Allows you to select one of the following options:
     - **none**
     - **Colin** (default)
     - **Charlette**

     When **none** is selected, then the alteration of the thickening of the flame on its interaction with the turbulence is not considered.

     See The Thickened Flame Model in the *Fluent Theory Guide* for more information about these options.

   - **Number of Grid Points in Flame**: Default is 8 grid points.

   - (simulations with LES only) **Integral Length Scale**: Is the representative of the largest eddy sizes, l_t in Equation 7.57 in the *Fluent Theory Guide*. You should provide a value based on the geometry and size of the characteristic bodies (such as inlet, burners, or bluff bodies). Typically, a value ranging from 1/4 to 1/2 of the characteristic dimension is used. This item is available only with the **Colin** efficiency function.

   Consequently, in the **Create/Edit Materials** dialog box, you will specify the **Laminar Flame Speed**, for which you have a choice of **constant**, **user-defined**, or **metghalchi-keck-law**, and the **Laminar Flame Thickness**, for which you have a choice of **constant**, **user-defined**, or **diffusivity-over-flame-speed**.

   Note:

   Since the thickened flame model must capture the correct flame speed, it is imperative that the combination of your chemical mechanism and your molecular transport properties (diffusion coefficients and thermal conductivity) reproduce the correct laminar flame speed. It is recommended that you verify this with a 1D premixed flame simulation using Ansys Chemkin, for example.

   When the **Thickened Flame Model** is enabled (Figure 19.2: The Species Model Dialog Box Displaying the Thickened Flame Model (p. 2366)), you have the option to hook a UDF in the **User-**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

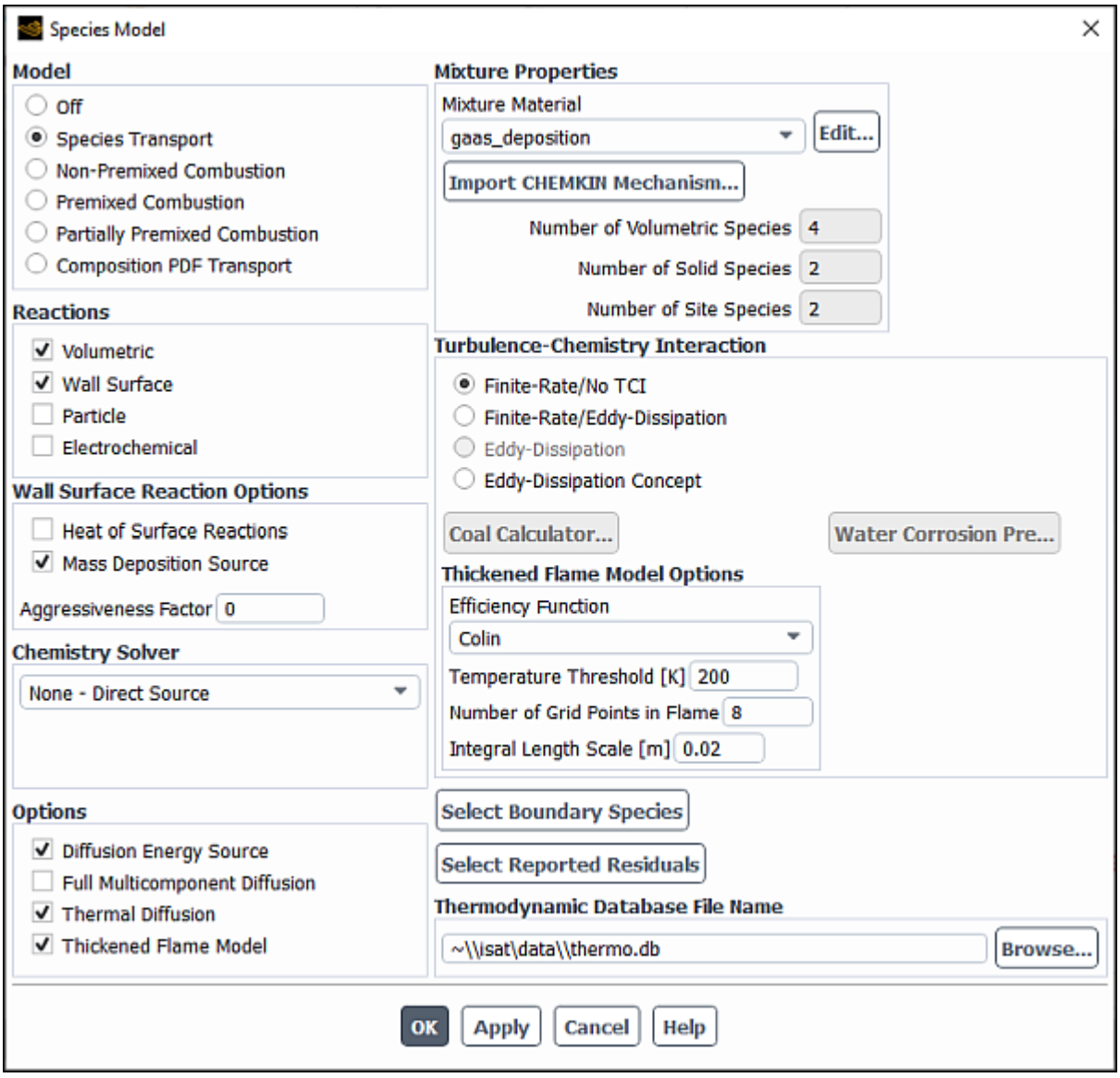
Defined Function Hooks dialog box to customize the **Thickened Flame Model** parameters. Refer to `DEFINE_THICKENED_FLAME_MODEL` in the *Fluent Customization Manual* for details.

Important:

The **Thickened Flame Model** is available only for unsteady laminar or turbulent (LES / DES / SAS / SBES / SDES) flows, with **Species Transport** and **Volumetric Reactions** enabled.

For information about the theory behind the **Thickened Flame Model**, refer to The Thickened Flame Model in the Theory Guide.

Figure 19.2: The Species Model Dialog Box Displaying the Thickened Flame Model



The Species Model Dialog Box

9. (optional) Select the species for which you want to specify boundary conditions. Only the species that you have selected will be listed in the relevant boundary condition dialog boxes. This will

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

facilitate the problem setup, especially when a large number of species are used in a gas mixture. Note that the **Patch** and **Hybrid Initialization** dialog boxes, as well as the **Solution Initialization** task page will also display only the selected species.

a. Click **Select Boundary Species**.

Figure 19.3: The Select Boundary Species Dialog Box

By default, all the gas species for which the transport equations are to be applied are set as boundary species and appear in the **Selected** multiple-selection list of the **Select Boundary Species** dialog box (see Figure 19.3: The Select Boundary Species Dialog Box (p. 2367)).

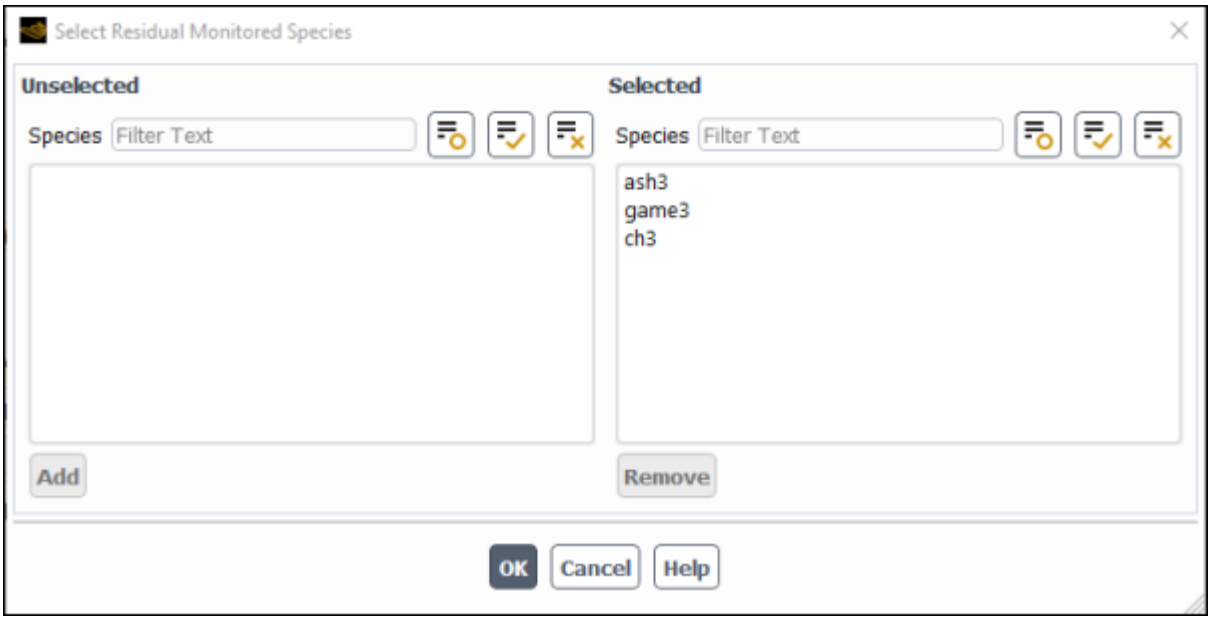
b. Modify the boundary species list as follows:

- To remove the species from the **Selected Species** multiple-selection list, select it and click **Remove**. The species will be moved from the **Selected Species** list to the **Unselected Species** list.
- To add the species back to the **Selected Species** list, simply select it in the **Unselected Species** multiple-selection list and click **Add**. The species will be moved from the **Unselected Species** list to the **Selected Species** list.

10. (optional) In a similar manner, click the **Select Reported Residuals** button and select the species that you want to monitor. Only the residuals for the selected species will be monitored and plotted in the graphics window during the simulation.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 19.4: The Select Residual Monitored Species



By default, all the species for which the species transport equations are to be solved are set as monitored species, and their residual plots will be drawn in the graphics window during the run. For a large number of monitored species, it may be difficult to distinguish the overlapping plot lines. By decreasing the number of monitored species, you can obtain clearer residual plots.

Important:

- The Fluent solver performs the convergence checks only for the monitored species. To prevent false convergence, make sure to include the species that may have difficulties converging in the monitored species list.
- You can modify the list of the monitored species dynamically during a run.

19.1.1.3. Importing a Volumetric Kinetic Mechanism in CHEMKIN Format

You can import gas kinetics mechanism in CHEMKIN format, which can be used with any solver option. For the **CHEMKIN-CFD Solver**, native CHEMKIN computations will determine reaction rates, so full compatibility with Ansys Chemkin is provided.

Note:

If a mechanism includes at least one reaction type that is not supported in the Fluent material-definition interface (that is, it is not included in the options for manual setup of reactions), the reactions of the CHEMKIN mechanism will not be displayed in the Fluent interface, even though they will be correctly used in Fluent computations.

For information about the format of CHEMKIN volumetric kinetics mechanisms, see Gas-phase Kinetics Input in the *Chemkin Input Manual*.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

19.1.1.3.1. Using Ansys Encrypted Mechanisms

For certain types of simulations, Fluent supports the use of encrypted gas kinetics mechanisms, such as those derived from the Ansys Model Fuels Library (encrypted) or through Ansys services. In order to use an encrypted mechanism, you must first import both the gas-phase kinetics and thermodynamic files as described in Procedure for Importing Volumetric CHEMKIN Mechanisms (p. 2369).

When using encrypted CHEMKIN gas mechanisms, note the following points:

- Encrypted mechanisms can only be used with the CHEMKIN-CFD solver or the **Relax to Chemical Equilibrium** option
- Prior to importing a flamelet or reading a PDF table generated with an encrypted mechanism, you must first import the mechanism
- Chemical equilibrium PDF table generation is not supported
- Multiphase and Decoupled Detailed Chemistry models are not supported
- You cannot combine encrypted and plain-text gas kinetics, surface kinetics and thermodynamics files in a simulation
- The chemistry acceleration options Dynamic Mechanism Reduction (DMR) and Dimension Reduction (DR) are not supported.

Note:

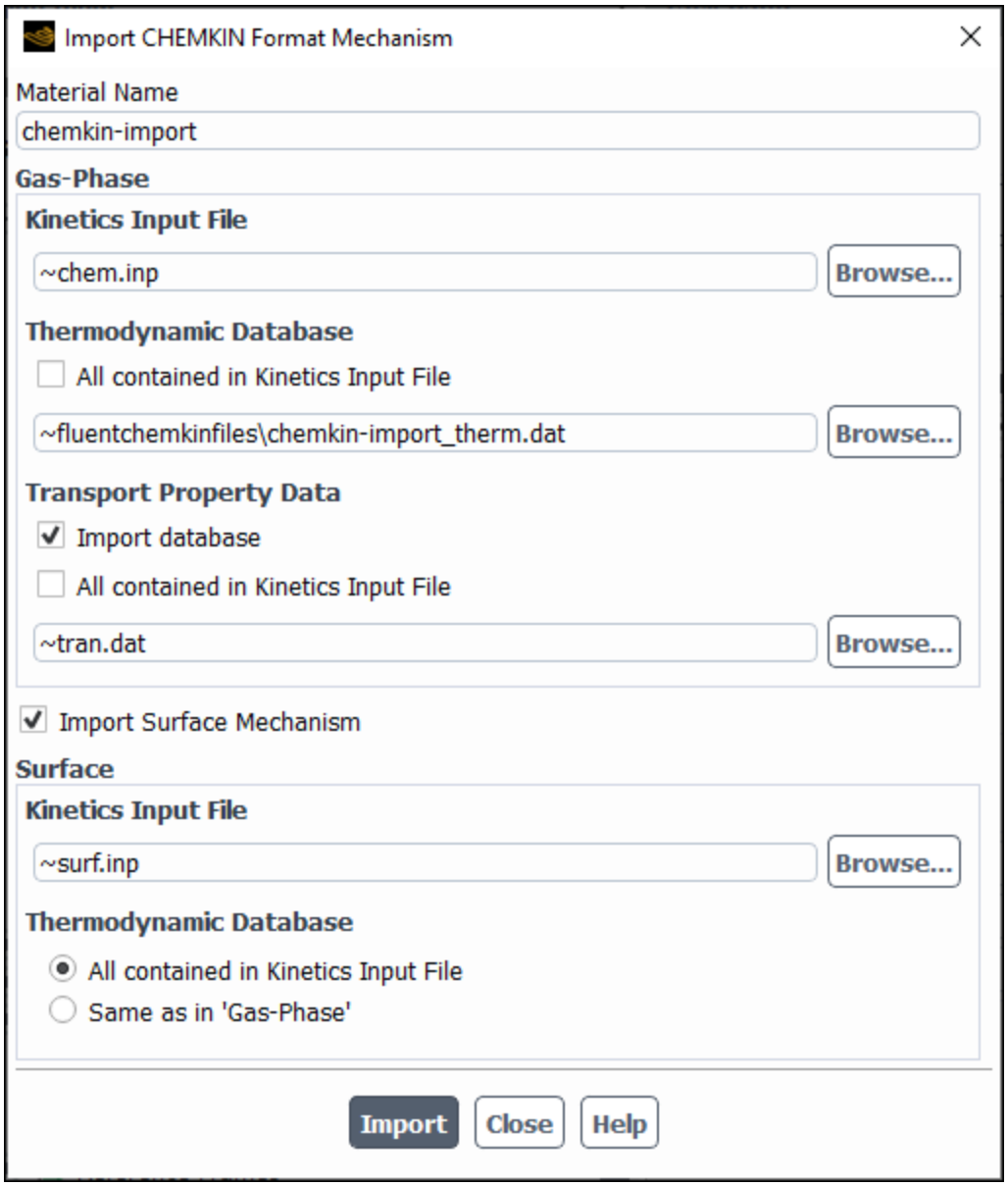
The proprietary reaction data will not be displayed in the Fluent interface.

19.1.1.3.2. Procedure for Importing Volumetric CHEMKIN Mechanisms

You can import the CHEMKIN mechanism file into Ansys Fluent using the **Import CHEMKIN Format Mechanism** dialog box (Figure 19.5: The Import CHEMKIN Format Mechanism Dialog Box for Volumetric Kinetics (p. 2370)) (opened by clicking **Import CHEMKIN Mechanism...** in the **Species Model** dialog box).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 19.5: The Import CHEMKIN Format Mechanism Dialog Box for Volumetric Kinetics



In the **Import CHEMKIN Format Mechanism** dialog box

1. Enter a name for the chemical mechanism under **Material Name**.
2. Enter the path to the CHEMKIN file (for example, *path/file*.inp) in the **Kinetics Input File** text field in the **Gas-Phase** group box.

Important:

- Since Ansys Fluent does not solve for the last species, you must ensure that the last species in the CHEMKIN mechanism species list is an abundant species, such as nitrogen. If not, edit the CHEMKIN mechanism file before importing it into Ansys Fluent, and move the most abundant species (that is the species in your system with the largest total mass fraction) to the end of the species list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The format for kinetics files is detailed in Chemkin Input Manual. Note that the keyword END must be added at the end of each group of elements in the input file.

  The following is an example of a valid format:

```
ELEMENTS
C H N
END
SPECIES
CH O2 CO2 H2O N2
END
```

  An example of invalid format is shown below:

```
ELEMENTS
C H N

SPECIES
CH O2 CO2 H2O N2
```

3. Specify species thermodynamic property data. You can include the data in the kinetics input file and/or in a separate file as follows:

   - If you want to specify thermodynamic data for all species in the kinetics input file, enable **All contained in Kinetics Input File** to avoid the need to specify a separate file.
   - If you provide some or all species thermodynamic data in a separate file, disable **All contained in Kinetics Input File** and specify the thermodynamic data file location.

   You can either use the default thermodynamic database file, `thermo.db`, provided in the `cpropep\data` directory in your Ansys Fluent installation area or specify the path of your thermodynamic database file if `thermo.db` does not contain all the gas-phase species in the CHEMKIN mechanism. The format for `thermo.db` is detailed in the CHEMKIN manual [79] (p. 5717).

   Note:

   Model Fuel Library (MFL) encrypted mechanisms for various fuels have thermodynamic data included in the kinetics input file. See the separate Ansys Model Fuel Library Getting Started Guide for more information about MFL mechanisms.

4. (optional) To import transport property data, enable **Import database** under **Transport Property Data (Optional)** and specify the location of the species transport property data. You can include the data in the kinetics input file and/or in a separate file as follows:

   - If you want to specify transport property data for all species in the kinetics input file, enable **All contained in Kinetics Input File**, to avoid the need to specify a separate file.
   - If you want to provide some or all species transport data in a separate file, disable **All contained in Kinetics Input File** and specify the transport property data file location.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Fluent will enable mixture-averaged multicomponent diffusion and set the Lennard-Jones kinetic theory parameters for all the species in the imported CHEMKIN mechanism. If you would like to use Stefan-Maxwell diffusion, enable the **Full Multicomponent Diffusion** in the **Species** dialog box.

Once the CHEMKIN files are imported, Ansys Fluent creates a material with the specified name, which will contain the data for the species and reactions, and add it to the list of available **Mixture Materials** in the Create/Edit Materials Dialog Box (p. 4885). The methods to compute the mixture's material properties are set automatically depending on which chemistry solver you selected.

For cases with imported transport property data, the methods used to calculate properties are shown in the table below.

Material Property	Stiff Chemistry Solver	CHEMKIN-CFD Solver
Specific Heat	mixing law	chemkin [a]
Viscosity	ideal-gas-mixing-law	chemkin [a]
Thermal Conductivity	ideal-gas-mixing-law	chemkin [a]
Mass Diffusivity	kinetics-theory	chemkin [a]
Thermal Diffusion	kinetics-theory	chemkin [a]

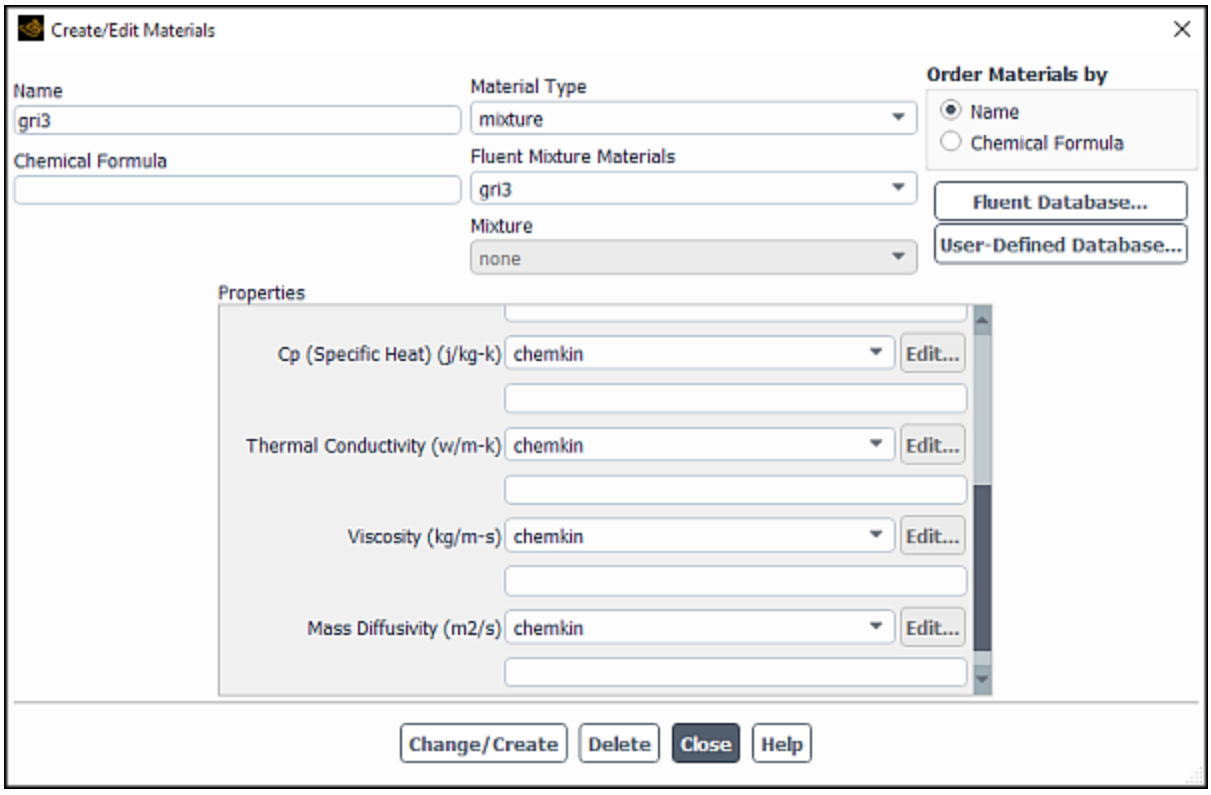
a. The material property value will be computed by Ansys Chemkin. You may choose to use a different property method if it is more suitable for your problem.

For cases with the **CHEMKIN-CFD Solver** that do not use transport property data, only the **Specific Heat** mixture property will be automatically set to the **chemkin** method.

Note that when **Full Multicomponent Diffusion** is enabled in the **Species** dialog box, Ansys Chemkin returns full multicomponent diffusivities. If **Full Multicomponent Diffusion** is disabled, Ansys Chemkin returns mixture averaged mass diffusivities for each species.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 19.6: The Material Dialog Box When Importing CHEMKIN Transport Properties



5. Click the **Import** button.

   Once you click **Import**, Ansys Fluent automatically saves the reaction mechanism and thermal files in the data structure. The original input files are not required to be retained along with the Fluent case file. Ansys Fluent will automatically regenerate these files as needed (for example, when using the CHEMKIN-CFD solver).

For information on how to import a surface kinetic mechanism in CHEMKIN format, see Importing a Surface Kinetic Mechanism in CHEMKIN Format (p. 2400).

19.1.1.3.3. CHEMKIN Mechanisms Included with Ansys Fluent

A CHEMKIN mechanism is a chemical reaction mechanism (also called a chemistry set) that has the following mechanism components:

- A file containing thermochemical data for the chemical species of interest
- A file containing a description of the reactions occurring in the gas phase
- (Optional) A file containing a description of the reactions occurring at the gas-surface interface or on the surface
- (Optional) A file containing gas-phase transport data

A set of gas-phase and/or surface reactions are generally developed using specific thermochemical data. Thus, the gas-phase kinetics, surface kinetics, and thermodynamic data are treated as a chemistry "set" in reacting-flow simulations. Ansys Fluent provides the following fuel-combustion

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

mechanisms (including reactions, thermodynamic data, and transport data) that are appropriate for use in common combustion simulations:

- Methane / Ethane: 50-species version of GRI-Mech 3.0 (`grimech30_50spec`)

  This is the original `GRI-mech` 3.0 (53 species) [148] (p. 5721) with three species (argon AR, propane C3H8, and propyl radical C3H7) and their reactions removed.

  In fuel/air or fuel/oxygen combustion, the removal of these species is expected to have negligible effect on the mechanism predictive capabilities for methane and ethane as fuels. The applicability of the GRI mechanism and validation cases are provided on the GRI mechanism web site [148] (p. 5721).

- Propane: 37-species high-temperature mechanism (`Propane-NOx_highT`)

  The mechanism has 211 elementary reaction steps and was obtained by removing species and reactions that are unimportant for propane combustion from the comprehensive master mechanism [104] (p. 5719). Ansys Chemkin Reaction Workbench software was used to reduce the master mechanism under the conditions of slightly lean and high pressure laminar flame of a propane/air mixture using the Directed Relation Graph (DRG) method.

- Two hydrogen mechanisms:

  – 9-species version for H2 combustion (`H2_mech`)

    The mechanism has 21 elementary reaction steps and has been extracted from Lighthill [88] (p. 5718) mechanism for CO and other hydrocarbons.

  – 21-species version for H2 combustion with NOx (`H2_NOx_mech`)

    The mechanism has 105 elementary reaction steps. The hydrogen combustion sub-mechanism is the same as the `H2_mech` mechanism, above. The NO_x sub-mechanism is based on the study by the Glarborg group [127] (p. 5720).

19.1.1.4. Defining Properties for the Mixture and Its Constituent Species

As discussed in Overview of User Inputs for Modeling Species Transport and Reactions (p. 2356), if you use a mixture material from the Ansys Fluent database, most mixture and species properties will already be defined. You may follow the procedures in this section to check the current properties, modify some of the properties, or set all properties for a brand-new mixture material that you are defining from nothing.

Remember that you will need to define properties for the mixture material and also for its constituent species. It is important that you define the mixture properties before setting any properties for the constituent species, since the species property inputs may depend on the methods you use to define the properties of the mixture.

The recommended sequence for property inputs is as follows:

1. Define the mixture species, and reaction(s), and define physical properties for the mixture. Remember to click the **Change/Create** button when you are done setting properties for the mixture material.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Define physical properties for the species in the mixture. Remember to click the **Change/Create** button after defining the properties for each species.

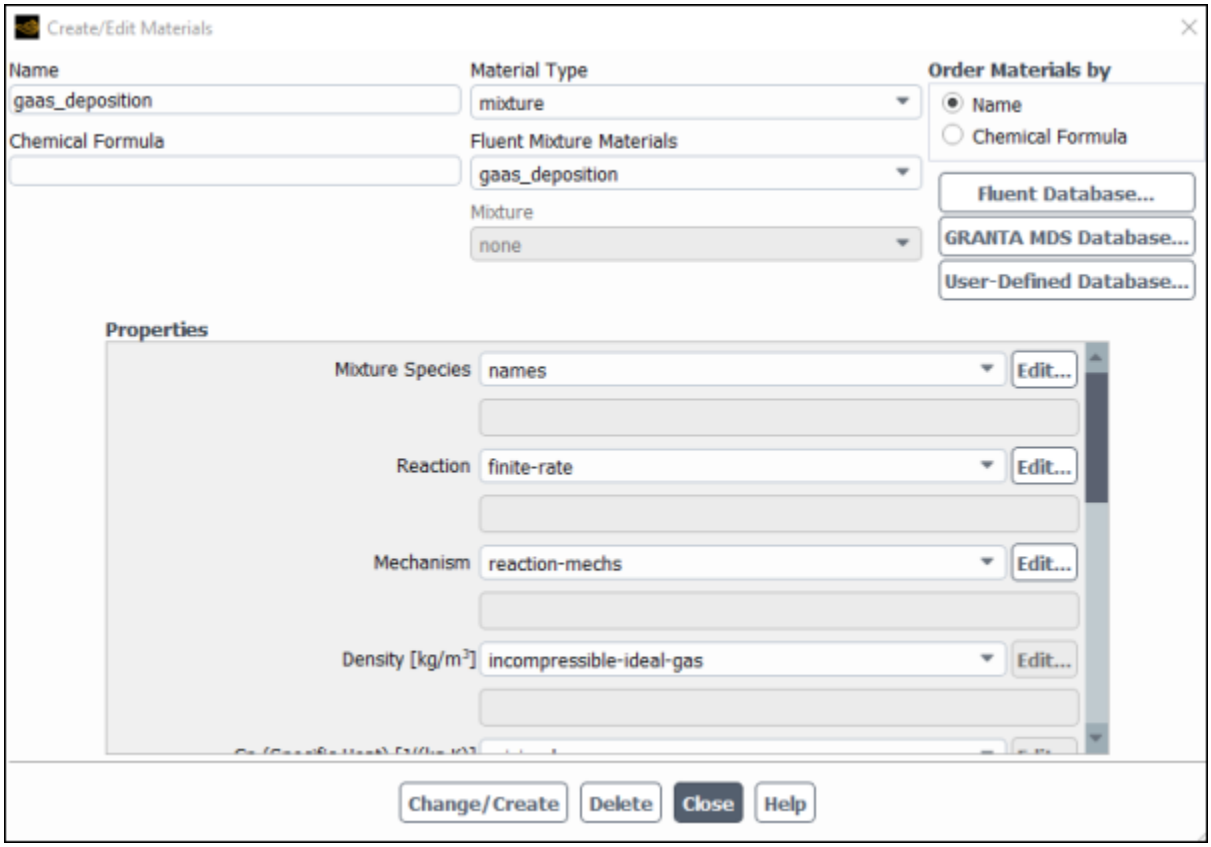
These steps, all of which are performed in the Create/Edit Materials Dialog Box (p. 4885), are described in detail in this section.

Setup → Materials

19.1.1.4.1. Defining the Species in the Mixture

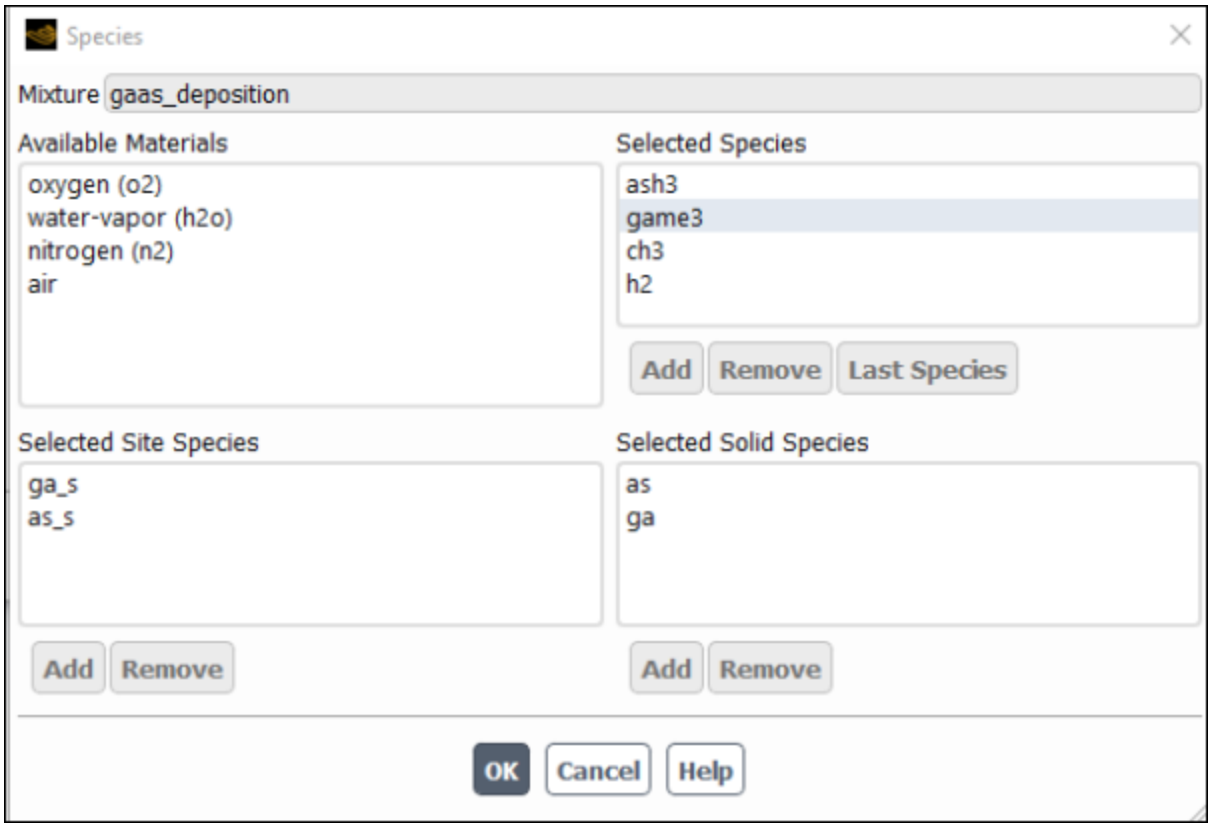
In the **Create/Edit Materials** dialog box (Figure 19.7: The Create/Edit Materials Dialog Box (Showing a Mixture Material) (p. 2375)), check that the **Material Type** is set to **mixture** and your mixture is selected in the **Fluent Mixture Materials** list.

Figure 19.7: The Create/Edit Materials Dialog Box (Showing a Mixture Material)



Click the **Edit...** button to the right of **Mixture Species** under the **Properties** group box and in the Species Dialog Box (p. 4929) (Figure 19.8: The Species Dialog Box (p. 2376)) that opens define the species in the mixture material.

Figure 19.8: The Species Dialog Box



Important:

For Chemkin mechanisms, the recommended approach for modifying mixture materials is to modify the Chemkin mechanism file and then re-import it into your Fluent case.

19.1.1.4.1.1. Overview of the Species Dialog Box

In the **Species** dialog box, the **Selected Species** list shows all of the fluid-phase species in the mixture. If you are modeling wall or particle surface reactions, the **Selected Solid Species** list will show all of the bulk solid species in the mixture. Solid species are species that are deposit to, or etch from, wall boundaries or discrete-phase particles (for example, Si(s)) and do not exist as fluid-phase species. If you are modeling wall surface reactions with site balancing, where species adsorb onto the wall surface, react, and then desorb off the surface, the **Selected Site Species** list will show all of the site species in the mixture.

The use of solid and site species with wall surface reactions is described in Wall Surface Reactions and Chemical Vapor Deposition (p. 2400). See Combusting Particle Surface Reactions (p. 2406) for information about particle surface reactions.

Important:

The order of the species in the **Selected Species** list is very important. Ansys Fluent considers the last species in the list to be the bulk species. You should therefore

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

be careful to retain the most abundant species (by mass) as the last species when you add species to or delete species from a mixture material.

The **Available Materials** list shows materials that are available but not in the mixture. Generally, you will see air in this list, since air is always available by default.

19.1.1.4.1.2. Adding Species to the Mixture

If you are creating a mixture from nothing or starting from an existing mixture and adding some missing species, you will first need to load the desired species from the database (or create them, if they are not present in the database) so that they will be available to the solver. You will need to close the **Species** dialog box before you begin, since it is a "modal" dialog box that will not allow you to do anything else when it is open. The procedure for adding species is as follows:

1. In the **Create/Edit Materials** dialog box, click the **Fluent Database...** button to open the **Fluent Database Materials** dialog box and copy the desired species, as described in Copying Materials from the Ansys Fluent Database (p. 1602). Remember that the constituent species of the mixture are fluid materials, so you should select **fluid** as the **Material Type** in the **Fluent Database Materials** dialog box to see the correct list of choices. Note that available solid and site species (for surface reactions) are also contained in the **fluid** list.

   Important:

   If you do not see the species you are looking for in the database, you can create a new fluid material for that species, following the instructions in Creating a New Material (p. 1606), and then continue with step 2, below.

2. Re-open the **Species** dialog box. You will see that the fluid materials you copied from the database (or created) are listed in the **Available Materials** list.

3. To add a species to the mixture, select it in the **Available Materials** list and click the **Add** button below the **Selected Species** list (or below the **Selected Site Species** or **Selected Solid Species** list, to define a site or solid species). The species will be added to the end of the relevant list and removed from the **Available Materials** list.

4. Repeat the previous step for all the desired species. When you are finished, click the **OK** button.

   Important:

   Adding a species to the list will alter the order of the species. You should be sure that the last species in the list is the bulk species, and you should check all cell and boundary zone conditions, under-relaxation factors, and other solution parameters that you have set, as described in detail in the following sections.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

19.1.1.4.1.3. Removing Species from the Mixture

To remove a species from the mixture, simply select it in the **Selected Species** list (or the **Selected Site Species** or **Selected Solid Species** list) and click the **Remove** button below the list. The species will be removed from the list and added to the **Available Materials** list.

Important:

- Removing a species from the list will alter the order of the species. You should be sure that the last species in the list is the bulk species, and you should check any cell zone or boundary conditions, under-relaxation factors, or other solution parameters that you have set, as described in detail in the following sections.
- Any species that is a reactant, product, or enhanced third-body in an active reaction cannot be deleted from the mixture material.

19.1.1.4.1.4. Assigning the Last Species

If you find that the last species in the **Selected Species** list is not the most abundant species (as it should be), you will need to rearrange the species to obtain the proper order.

1. Select the bulk species in the **Selected Species** list.
2. Click **Last Species**. The selected species will be placed at the end of the **Selected Species** list. The transport equation of the last species will not be solved.

19.1.1.4.1.5. The Naming and Ordering of Species

As discussed previously, you should retain the most abundant species as the last one in the **Selected Species** list when you add or remove species. Additional considerations you should be aware of when adding and deleting species are presented here.

There are three characteristics of a species that identify it to the solver: name, chemical formula, and position in the list of species in the **Species** dialog box. Changing these characteristics will have the following effects:

- You can change the **Name** of a species (using the Create/Edit Materials Dialog Box (p. 4885), as described in Renaming an Existing Material (p. 1602)) without any consequences.
- You should *never* change the given **Chemical Formula** of a species.
- You will change the order of the species list if you add or remove any species. When this occurs, all cell zone or boundary conditions, solver parameters, and solution data for species will be reset to the default values. (Solution data, cell zone or boundary conditions, and solver parameters for other flow variables will not be affected.) Therefore, if you add or remove species you should take care to redefine species cell zone and boundary conditions and solution parameters for the newly defined problem. In addition, you should recognize that patched species concentrations or concentrations stored in any data file that was based on the original species ordering will be incompatible with the newly defined problem. You can use the data file as a starting guess, but you should be aware that the species concentrations in the data file may provide a poor initial guess for the newly defined model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

19.1.1.4.2. Defining Reactions

If your Ansys Fluent model involves chemical reactions, you can define the reactions in which the defined species participate. This will be necessary only if you are creating a mixture material from nothing, you have modified the species, or you want to redefine the reactions for some other reason.

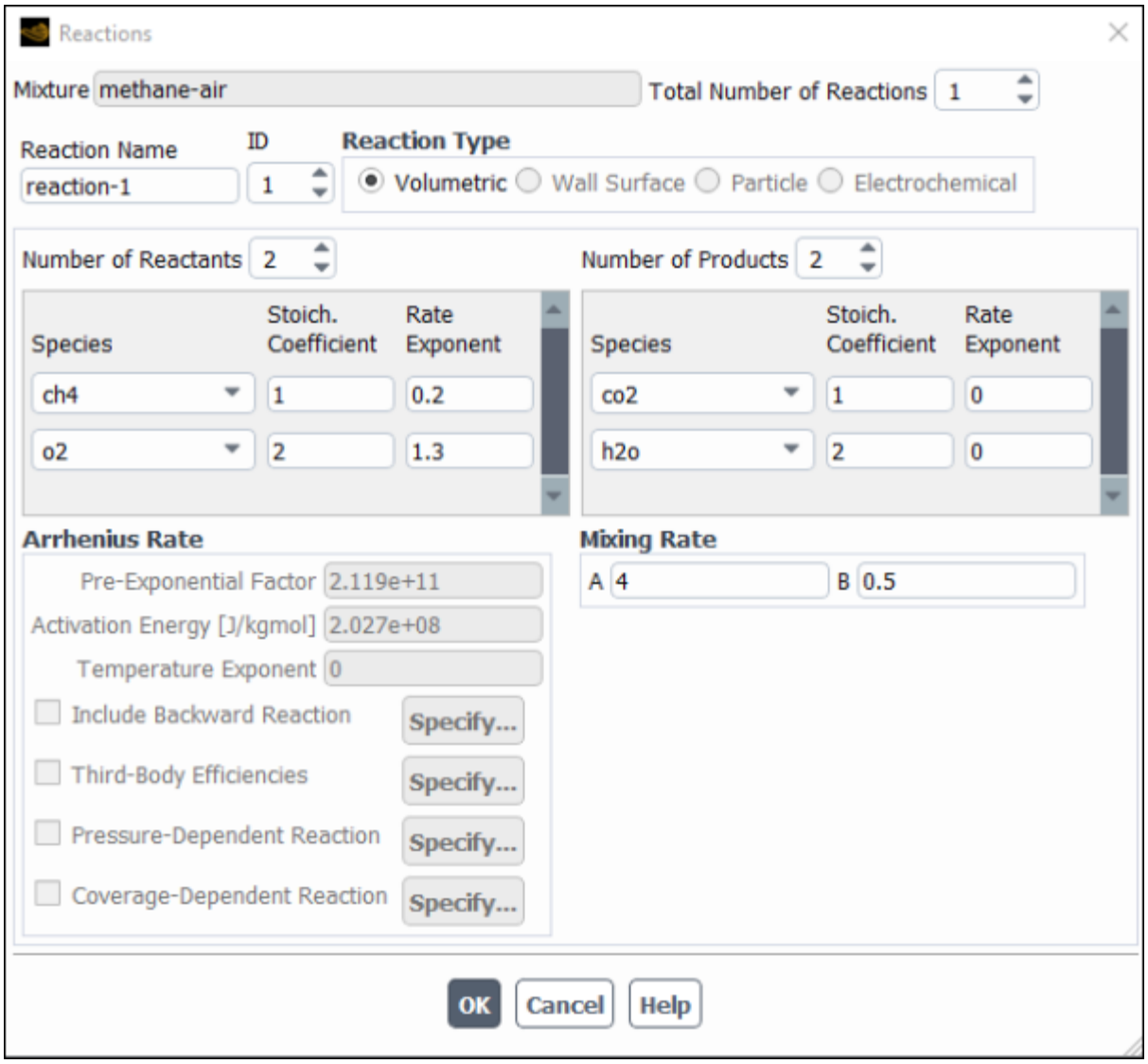
Depending on which turbulence-chemistry interaction model you selected in the **Species Model** dialog box (see Enabling Species Transport and Reactions and Choosing the Mixture Material (p. 2359)), the appropriate reaction model will be displayed in the **Reaction** drop-down list in the **Edit Material** dialog box. If you are using the **Finite-Rate/No TCI** or **Eddy-Dissipation Concept** model, the reaction model will be **finite-rate**; if you are using the **Eddy-Dissipation** model, the reaction model will be **eddy-dissipation**; if you are using the **Finite-Rate/Eddy-Dissipation** model, the reaction model will be **finite-rate/eddy-dissipation**.

19.1.1.4.2.1. Inputs for Reaction Definition

To define the reactions, click the **Edit...** button to the right of **Reaction**. The **Reactions** dialog box (Figure 19.9: The Reactions Dialog Box (p. 2380)) will open.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 19.9: The Reactions Dialog Box



The steps for defining reactions are as follows:

1. Set the total number of reactions (volumetric reactions, wall surface reactions, and particle surface reactions) in the **Total Number of Reactions** field. Use the arrows to change the value, or type in the value and press **Enter**.

   Note that if your model includes discrete-phase combusting particles, you should include the particulate surface reaction(s) (for example, char burnout, multiple char oxidation) in the number of reactions *only* if you plan to use the multiple surface reactions model for surface combustion.

2. Specify the **Reaction Name** of the reaction you want to define.

3. Set the **ID** of the reaction you want to define. Again, if you type in the value be sure to press **Enter**.

4. If this is a fluid-phase reaction, keep the default selection of **Volumetric** as the **Reaction Type**. If this is a wall surface reaction (described in Wall Surface Reactions and Chemical

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Vapor Deposition (p. 2400)) or a particle reaction (described in Particle Reactions (p. 2406)), select **Wall Surface** or **Particle** as the **Reaction Type**.

5. Specify how many reactants and products are involved in the reaction by increasing the value of the **Number of Reactants** and the **Number of Products**. Select each reactant or product in the **Species** drop-down list and then set its stoichiometric coefficient and rate exponent in the appropriate **Stoich. Coefficient** and **Rate Exponent** fields. The stoichiometric coefficient is the constant $\nu'_{i,r}$ or $\nu''_{i,r}$ in Equation 7.20 in the Theory Guide and the rate exponent is the exponent on the reactant or product concentration, $\eta'_{j,r}$ or $\eta''_{j,r}$ in Equation 7.21 in the Theory Guide.

   There are two general classes of reactions that can be handled by the **Reactions** dialog box, so it is important that the parameters for each reaction are entered correctly. The classes of reactions are as follows:

   - Global forward reaction (no reverse reaction): Product species generally do not affect the forward rate, so the rate exponent for all products ($\eta''_{j,r}$) should be 0. For reactant species, set the rate exponent ($\eta'_{j,r}$) to the desired value. If such a reaction is not an elementary reaction, the rate exponent will generally not be equal to the stoichiometric coefficient ($\nu'_{i,r}$) for that species. An example of a global forward reaction is the combustion of methane:

     $$CH_4+2O_2\rightarrow CO_2+2H_2O \tag{19.1}$$

     where $\nu'_{CH_4}=1$, $\eta'_{CH_4}=0.2$, $\nu'_{O_2}=2$, $\eta'_{O_2}=1.3$, $\nu''_{CO_2}=1$, $\eta''_{CO_2}=0$, $\nu''_{H_2O}=2$, and $\eta''_{H_2O}=0$.

     Figure 19.9: The Reactions Dialog Box (p. 2380) shows the coefficient inputs for the combustion of methane. See also the **methane-air** mixture material in the Fluent Database Materials Dialog Box (p. 4897).

     Note that, in certain cases, you may want to model a reaction where product species affect the forward rate. For such cases, set the product rate exponent ($\eta''_{j,r}$) to the desired value. An example of such a reaction is the oxidation of carbon-monoxide (see the **carbon-monoxide-air** mixture material in the Fluent Database Materials Dialog Box (p. 4897)), in which the presence of water has an effect on the reaction rate:

     $$CO+\tfrac{1}{2}O_2+H_2O\rightarrow CO_2+H_2O$$

     In the gas-shift reaction, the rate expression may be defined as:

     $$k[CO][O_2]^{1/4}[H_2O]^{1/2} \tag{19.2}$$

     where $\nu'_{CO}=1$, $\eta'_{CO}=1$, $\nu'_{O_2}=0.5$, $\eta'_{O_2}=0.25$, $\nu''_{CO_2}=1$, $\eta''_{CO_2}=0$, $\nu''_{H_2O}=0$, and $\eta''_{H_2O}=0.5$.

   - Reversible reaction: An elementary chemical reaction that assumes the rate exponent for each species is equivalent to the stoichiometric coefficient for that species. An example of an elementary reaction is the oxidation of SO_2 to SO_3:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$SO_2+\frac{1}{2}O_2\rightleftharpoons SO_3$$

where $\nu'_{SO_2}=1$, $\eta'_{SO_2}=1$, $\nu'_{O_2}=0.5$, $\eta'_{O_2}=0.5$, $\nu''_{SO_3}=1$, and $\eta''_{SO_3}=1$.

See step 7 below for information about how to enable reversible reactions.

6. If you are using the laminar finite-rate, finite-rate/eddy-dissipation, eddy-dissipation concept or PDF transport model for the turbulence-chemistry interaction, enter the following parameters for the Arrhenius rate in the **Arrhenius Rate** group box:

Note:

All quantities for the Arrhenius rate parameters are in units of kgmol, m^3, and seconds.

Pre-Exponential Factor

(the constant A_r in Equation 7.23 in the Theory Guide). The units of A_r must be specified such that the units of the molar reaction rate, $\hat{R}_{i,r}$ in Equation 7.19 in the Theory Guide, are moles/volume-time (for example, kmol/m^3-s) and the units of the volumetric reaction rate, R_i in Equation 7.19 in the Theory Guide, are mass/volume-time (for example, kg/m^3-s).

Important:

It is important to note that if you have selected the British units system, the Arrhenius factor should still be entered in SI units. This is because Ansys Fluent applies no conversion factor to your input of A_r (the conversion factor is 1.0) when you work in British units, as the correct conversion factor depends on your inputs for $\nu'_{i,r}$, β_r, and so on.

Activation Energy

(the constant E_r in the forward rate constant expression, Equation 7.23 in the Theory Guide).

Temperature Exponent

(the value for the constant β_r in Equation 7.23 in the Theory Guide).

Third-Body Efficiencies

(the values for $\gamma_{j,r}$ in Equation 7.22 in the Theory Guide). If you have accurate data for the efficiencies and want to include this effect on the reaction rate (that is, include Γ in Equation 7.21 in the Theory Guide), select the **Third Body Efficiencies** option and click the **Specify...** button to open the Third-Body Efficiency Dialog Box (p. 4936) (Figure 19.10: The Third-Body Efficiency Dialog Box (p. 2383)).

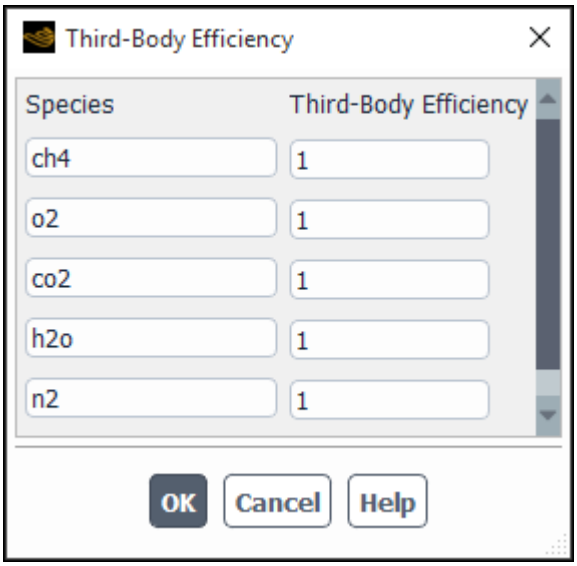
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For each **Species** in the dialog box, specify the **Third-Body Efficiency**.

Important:

It is not necessary to include the third-body efficiencies. You should not enable the **Third-Body Efficiencies** option unless you have accurate data for these parameters.

Figure 19.10: The Third-Body Efficiency Dialog Box

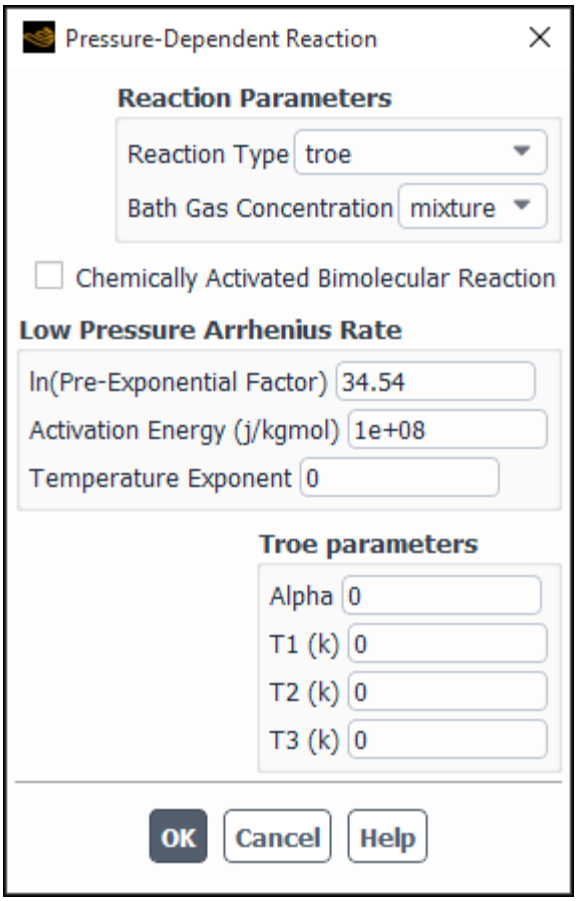


Pressure-Dependent Reaction

(if relevant) If you are using the laminar finite-rate or eddy-dissipation concept model for turbulence-chemistry interaction, or have selected the composition PDF transport model (see Modeling a Composition PDF Transport Problem (p. 2442)), and the reaction is a pressure fall-off reaction (see Pressure-Dependent Reactions in the Theory Guide), enable the **Pressure-Dependent Reaction** option for the **Arrhenius Rate** and click the **Specify...** button to open the **Pressure-Dependent Reaction** dialog box (Figure 19.11: The Pressure-Dependent Reaction Dialog Box (p. 2384)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 19.11: The Pressure-Dependent Reaction Dialog Box



Under **Reaction Parameters**, select the appropriate **Reaction Type** (**lindemann**, **troe**, or **sri**). See Pressure-Dependent Reactions in the Theory Guide for details about the three methods. Next, specify if the **Bath Gas Concentration** ($[M]$ in Equation 7.32 in the Theory Guide) is to be defined as the concentration of the **mixture**, or as the concentration of one of the mixture's constituent species, by selecting the appropriate item in the drop-down list.

Enabling the **Chemically Activated Bimolecular Reaction** option results in a net rate constant at any pressure being defined as Equation 7.38 in the Theory Guide.

The parameters you specified under **Arrhenius Rate** in the **Reactions** dialog box represent the high-pressure Arrhenius parameters. You can, however, specify values for the following parameters under **Low Pressure Arrhenius Rate**:

ln(Pre-Exponential Factor)

(A_{low} in Equation 7.30 in the Theory Guide) The pre-exponential factor A_{low} is often an extremely large number, so you will input the natural logarithm of this term.

Activation Energy

(E_{low} in Equation 7.30 in the Theory Guide)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Temperature Exponent

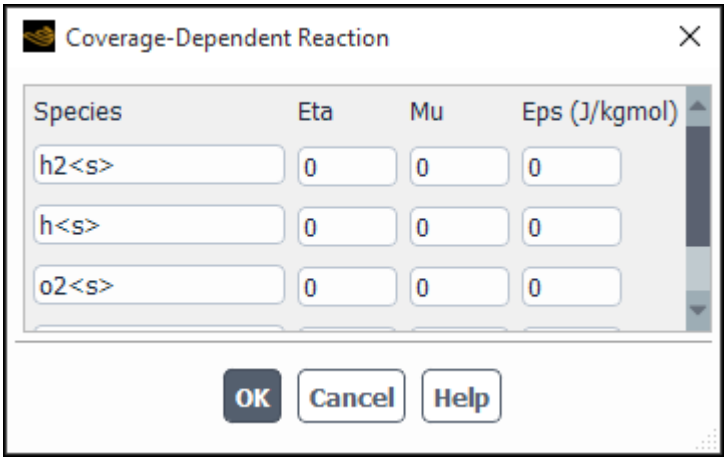
(β_{low} in Equation 7.30 in the Theory Guide)

If you selected **troe** for the **Reaction Type**, you can specify values for **Alpha**, **T1**, **T2**, and **T3** (α, T_1, T_2, and T_3 in Equation 7.35 in the Theory Guide) under **Troe parameters**. If you selected **sri** for the **Reaction Type**, you can specify values for **a**, **b**, **c**, **d**, and **e** (a, b, c, d, and e in Equation 7.36 in the Theory Guide) under **SRI parameters**.

Coverage Dependent Reaction

If you are modeling **Wall Surface** reactions with site-balancing and you have reaction rates that depend on site coverages, you can enable the **Coverage Dependent Reaction** option. Click **Specify...** to open the **Coverage Dependent Reaction** dialog box (Figure 19.12: The Coverage Dependent Reaction Dialog Box (p. 2385)) and input the coverage parameters.

Figure 19.12: The Coverage Dependent Reaction Dialog Box



In the **Coverage Dependent Reaction** dialog box, all the site species of the reaction will be present with a default value of 0 for all the parameters, corresponding to no surface coverage modification. Enter the relevant values of the parameters μ, ε, and η (as defined in Equation 7.81 in the Theory Guide) for all the species for which the reaction has coverage dependence. μ and ε are dimensionless, and η is in units of J/kgmol.

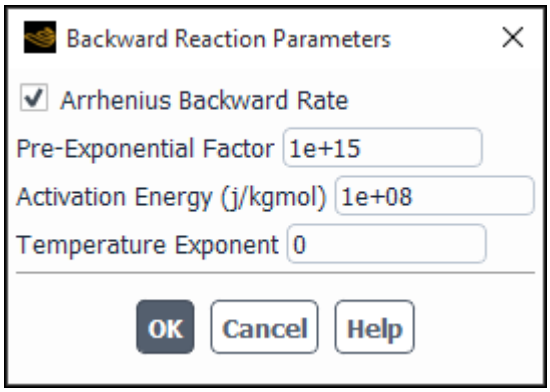
7. If you are using the laminar finite-rate, eddy-dissipation concept or PDF transport model for turbulence-chemistry interaction, and the reaction is reversible, enable the **Include Backward Reaction** option for the **Arrhenius Rate**. When this option is enabled, you will not be able to edit the **Rate Exponent** for the product species, which instead will be set to be equivalent to the corresponding product **Stoich. Coefficient**. By default, Ansys Fluent calculates the reverse reaction rate constants using Equation 7.24 in the *Fluent Theory Guide*. You can overwrite the Ansys Fluent's default parameters for the reactions or define your own reactions. In both scenarios, you also need to specify the standard-state enthalpy and standard-state entropy for mixture material. Ansys Fluent will use these values in the calculation of the backward reaction rate constant (Equation 7.24 in the *Fluent Theory Guide*).

   In addition, you have the option to provide your own backward reaction parameters. To do this, click **Specify...** next to **Include Backward Reaction** and enable the **Arrhenius**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Backward Rate option in the **Backward Reaction Parameters** dialog box that opens (see Figure 19.13: Backward Reaction Parameters Dialog Box (p. 2386)).

Figure 19.13: Backward Reaction Parameters Dialog Box



You can specify the following backward rate parameters:

Pre-Exponential Factor

(the constant $A_{b,r}$ in Equation 7.28 in the *Fluent Theory Guide*).

Activation Energy

(the constant $E_{b,r}$ in the backward rate constant expression in Equation 7.28 in the *Fluent Theory Guide*).

Temperature Exponent

(the constant $\beta_{b,r}$ in Equation 7.28 in the *Fluent Theory Guide*).

Ansys Fluent will use your custom values to calculate the backward rate constant of the reversible reaction according to Equation 7.28 in the *Fluent Theory Guide*.

Note that the reversible reaction option is not available for either the eddy-dissipation or the finite-rate/eddy-dissipation turbulence-chemistry interaction model.

8. If you are using the eddy-dissipation or finite-rate/eddy-dissipation model for turbulence-chemistry interaction, you can enter values for **A** and **B** under the **Mixing Rate** heading. These values should not be changed unless you have reliable data. In most cases you will use the default values.

   A is the constant A in the turbulent mixing rate (Equation 7.39 and Equation 7.40 in the Theory Guide) when it is applied to a species that appears as a reactant in this reaction. The default setting of 4.0 is based on the empirically derived values given by Magnussen et al. [94] (p. 5718).

   B is the constant B in the turbulent mixing rate (Equation 7.40 in the Theory Guide) when it is applied to a species that appears as a product in this reaction. The default setting of 0.5 is based on the empirically derived values given by Magnussen et al. [94] (p. 5718).

9. Repeat steps 2–8 for each reaction you need to define. After defining all reactions, click **OK**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

19.1.1.4.2.2. Defining Species and Reactions for Fuel Mixtures

Quite often, combustion systems will include fuel that is not easily described as a pure species (such as CH_4 or C_2H_6). Complex hydrocarbons, including fuel oil or even wood chips, may be difficult to define in terms of such pure species. However, if you have available the heating value and the ultimate analysis (elemental composition) of the fuel, you can define an equivalent fuel species and an equivalent heat of formation for this fuel. Consider, for example, a fuel known to contain 50% C, 6% H, and 44% O by weight. Dividing by atomic weights, you can arrive at a "fuel" species with the molecular formula $C_{4.17}H_6O_{2.75}$. You can start from a similar, existing species or create a species from nothing, and assign it a molecular weight of 100.04 kg/kmol (4.17 × 12 + 6 × 1 + 2.75 × 16). The chemical reaction would be considered to be

$$C_{4.17}H_6O_{2.75}+4.295O_2 \rightarrow 4.17CO_2+3H_2O \tag{19.3}$$

You will need to set the appropriate stoichiometric coefficients for this reaction.

The heat of formation (or standard-state enthalpy) for the fuel species can be calculated from the known heating value ΔH since

$$\Delta H = \sum_{i=1}^{N} h_i^0 \left(\nu''_{i,r} - \nu'_{i,r} \right) \tag{19.4}$$

where h_i^0 is the standard-state enthalpy on a molar basis. Note the sign convention in Equation 19.4 (p. 2387) : ΔH is negative when the reaction is exothermic.

19.1.1.4.3. Defining Zone-Based Reaction Mechanisms

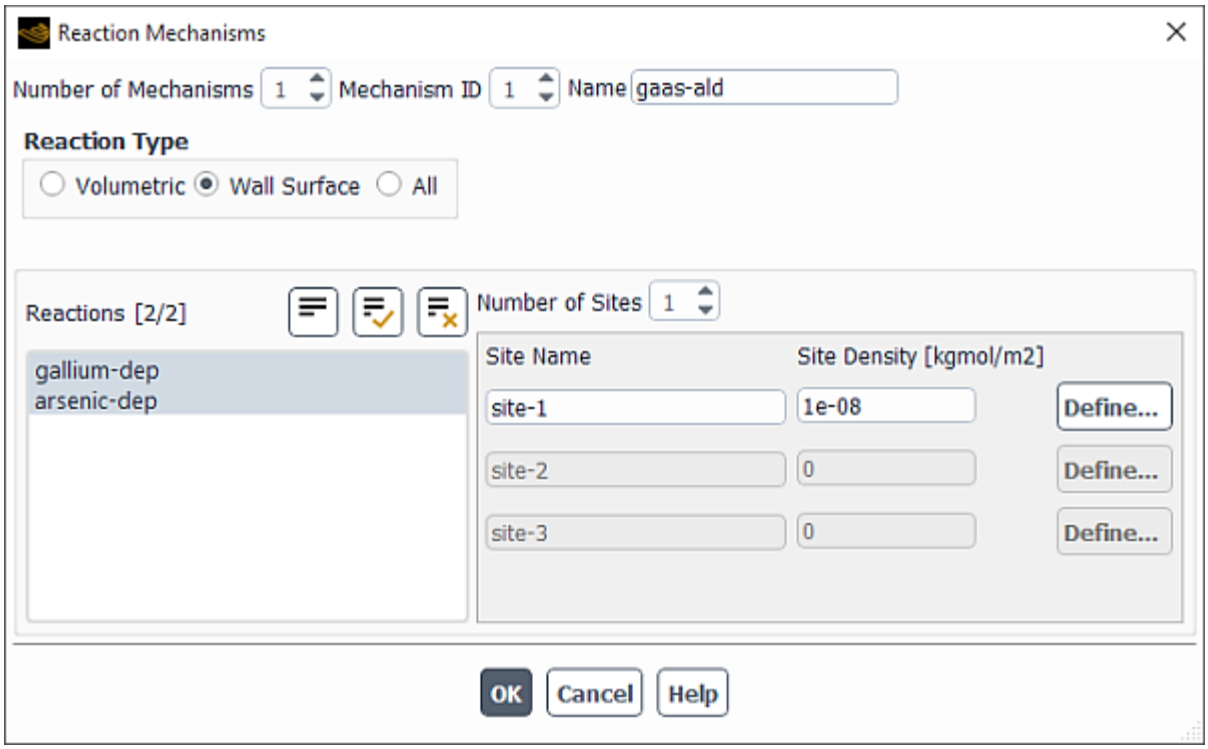
If your Ansys Fluent model involves reactions that are confined to a specific area of the domain, you can define "reaction mechanisms" to enable different reactions selectively in different geometrical zones. You can create reaction mechanisms by selecting reactions from those defined in the **Reactions** dialog box and grouping them. You can then assign a particular mechanism to a particular zone.

19.1.1.4.3.1. Inputs for Reaction Mechanism Definition

To define a reaction mechanism, click the **Edit...** button to the right of **Mechanism**. The Reaction Mechanisms Dialog Box (p. 4941) (Figure 19.14: The Reaction Mechanisms Dialog Box (p. 2388)) will open.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 19.14: The Reaction Mechanisms Dialog Box



The steps for defining a reaction mechanism are as follows:

1. Set the total number of mechanisms in the **Number of Mechanisms** field. Use the arrows to change the value, or type the value and press **Enter**.

2. Set the **Mechanism ID** of the mechanism you want to define. Again, if you type in the value, be sure to press **Enter**.

3. Specify the **Name** of the mechanism.

4. Select the type of reaction to add to the mechanism under **Reaction Type**. If you select **Volumetric**, the **Reactions** list will display all available fluid-phase reactions. If you select **Wall Surface** or **Particle**, the **Reactions** list will display all available wall surface reactions (described in Wall Surface Reactions and Chemical Vapor Deposition (p. 2400)) or particle reactions (described in Particle Reactions (p. 2406)). If you select **All**, the **Reactions** list will display all available reactions. This option is meant for backward compatibility with Ansys Fluent 6.0 or earlier cases.

5. Select the reactions to be included in the mechanism.

   - For **Volumetric** or **Particle** reactions, select available reactions for the mechanism in the **Reactions** list.

   - For **Wall Surface** reactions, use the following procedure:

     a. Select available wall surface reactions for the mechanism in the **Reactions** list.

     b. If any site species appear in the selected reaction(s), set the number of sites in the **Number of Sites** field. Use the arrows to change the value, or type the value and

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

press **Enter**. See Reaction-Diffusion Balance for Surface Chemistry in the Theory Guide for details about site species in wall surface reactions.

c. If you specify a **Number of Sites** that is greater than zero, specify the properties of the site.

Site Name

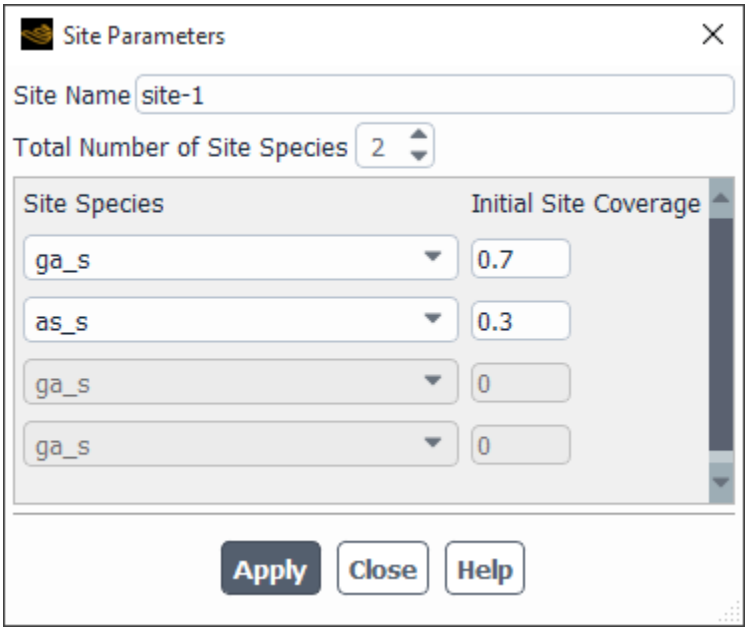
(optional)

Site Density

(in kmol/m^2) This value is typically in the range of 10^{-8} to 10^{-6}.

Click the **Define...** button. This will open the Site Parameters Dialog Box (p. 4943) (Figure 19.15: The Site Parameters Dialog Box (p. 2389)), where you will define the parameters of the site species.

Figure 19.15: The Site Parameters Dialog Box



Site Name

is the optional name of the site that was specified in the **Reaction Mechanisms** dialog box.

Total Number of Site Species

is the number of adsorbed species that are to be modeled at the site. (Use the arrows to change the value, or type the value and press **Enter**.)

Under **Site Species**, select the appropriate species from the drop-down list(s) and specify the fractional **Initial Site Coverage** for each species. For steady-state calculations, it is recommended (though not strictly required) that the initial values

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

of **Initial Site Coverage** sum to unity. For transient calculations, it is required that these values sum to unity.

Click **Apply** in the **Site Parameters** dialog box to store the new values.

6. Repeat steps 2–5 for each reaction mechanism you need to define. When you are finished defining all reaction mechanisms, click **OK**.

19.1.1.4.4. Defining Physical Properties for the Mixture

When your Ansys Fluent model includes chemical species, the following physical properties must be defined, either by you or by the database, for the mixture material:

- density, which you can define using the gas law or as a volume-weighted function of composition
- viscosity, which you can define as a function of composition
- thermal conductivity and specific heat (in problems involving solution of the energy equation), which you can define as functions of composition
- mass diffusion coefficients and Schmidt number, which govern the mass diffusion fluxes (Equation 7.2 and Equation 7.4 in the Theory Guide)

Detailed descriptions of these property inputs are provided in Physical Properties (p. 1597).

Important:

Remember to click the **Change/Create** button when you are done setting the properties of the mixture material. The properties that appear for each of the constituent species will depend on your settings for the properties of the mixture material. If, for example, you specify a composition-dependent viscosity for the mixture, you will need to define viscosity for each species.

19.1.1.4.5. Defining Physical Properties for the Species in the Mixture

For each of the fluid materials in the mixture, you (or the database) must define the following physical properties:

- molecular weight, which is used in the gas law and/or in the calculation of reaction rates and mole-fraction inputs or outputs
- standard-state (formation) enthalpy and reference temperature (in problems involving solution of the energy equation)
- viscosity, if you defined the viscosity of the mixture material as a function of composition
- thermal conductivity and specific heat (in problems involving solution of the energy equation), if you defined these properties of the mixture material as functions of composition
- standard-state entropy, if you are modeling reversible reactions

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- thermal and momentum accommodation coefficients, if you have enabled the low-pressure boundary slip model.

Detailed descriptions of these property inputs are provided in Physical Properties (p. 1597).

Important:

Global reaction mechanisms with one or two steps inevitably neglect the intermediate species. In high-temperature flames, neglecting these dissociated species may cause the temperature to be overpredicted. A more realistic temperature field can be obtained by increasing the specific heat capacity for each species. Rose and Cooper [130] (p. 5720) have created a set of specific heat polynomials as a function of temperature.

The specific heat capacity for each species is calculated as

$$c_p(T) = \sum_{k=0}^{m} a_k T^k \tag{19.5}$$

The modified c_p polynomial coefficients (J/kg-K) are provided in Table 19.1: Modified Specific Heat Capacity (Cp) Polynomial Coefficients (p. 2391).

Table 19.1: Modified Specific Heat Capacity (Cp) Polynomial Coefficients

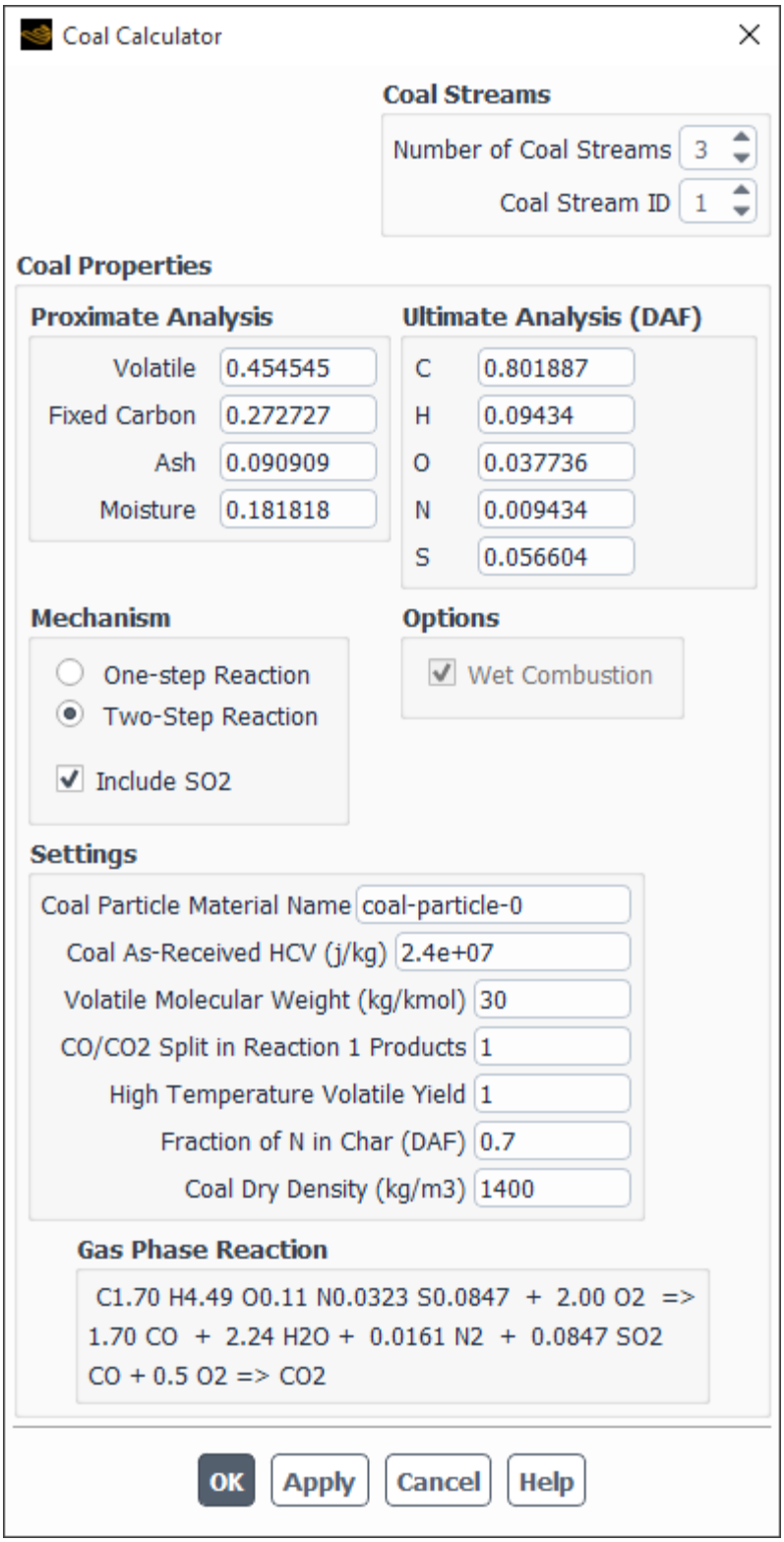
Coefficient	CO_2	H_2O	O_2
a_0	5.35446e+02	1.93780e+03	8.76317e+02
a_1	1.27867e+00	-1.18077e+00	1.22828e-01
a_2	-5.46776e-04	3.64357e-03	5.58304e-04
a_3	-2.38224e-07	-2.86327e-06	-1.20247e-06
a_4	1.89204e-10	7.59578e-10	1.14741e-09
a_5	—	—	-5.12377e-13
a_6	—	—	8.56597e-17

19.1.1.5. Setting up Coal Simulations with the Coal Calculator Dialog Box

The **Coal Calculator** dialog box automates calculation and setting of the relevant input parameters for the Species, Discrete-Phase (DPM) and Pollutant models associated with coal combustion. It is available in the **Species** dialog box for the **Species Transport** model when the **Eddy-Dissipation** or **Finite-Rate/Eddy-Dissipation** turbulence-chemistry option is selected. You can define up to three coal streams in your simulation using the **Number of Coal Streams** entry box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 19.16: The Coal Calculator Dialog Box



For each coal stream selected in the **Coal Stream ID**, you need to define the following properties:

1. Coal **Proximate Analysis**, which is the mass fraction of **Volatile, Fixed Carbon, Ash** and **Moisture** in the coal. Ansys Fluent will normalize the mass fractions so that they sum to unity.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Coal **Ultimate Analysis**, which is the mass fraction of atomic **C, H, O, N** and optionally **S**, in the Dry-Ash-Free (DAF) coal. Ansys Fluent will normalize the mass fractions so that they sum to unity.

3. A choice of **One-step** or **Two-step** chemical mechanism. The one-step mechanism is,

$$vol + \nu_{O_2} O_2 \rightarrow \nu_{CO_2} CO_2 + \nu_{H_2O} H_2O + \nu_{SO_2} SO_2 + \nu_{N_2} N_2 \tag{19.6}$$

   The two-step mechanism involves oxidation of volatiles to CO in the first reaction and oxidation of CO to CO_2 in the second reaction:

$$\begin{aligned} vol + \nu_{O_2} O_2 &\rightarrow \nu_{CO} CO + \nu_{CO_2} CO_2 + \nu_{H_2O} H_2O + \nu_{SO_2} SO_2 + \nu_{N_2} N_2 \\ CO + 0.5 O_2 &\rightarrow CO_2 \end{aligned} \tag{19.7}$$

   The stoichiometric coefficients in Equation 19.6 (p. 2393) and Equation 19.7 (p. 2393) are calculated from the ultimate and proximate analyses.

4. An option to **Include SO2**. When this is enabled, an input for the atomic mass fraction of sulphur, **S**, appears in the ultimate analysis frame.

5. The **Coal Particle Material Name**. A DPM combusting-particle material will be created with this name. The default name is *coal-particle*.

6. The **Coal As-Received HCV**, where HCV denotes the Higher Calorific Value.

7. **Volatile Molecular Weight** is the molecular weight of pure volatiles.

8. The **CO/CO2 Split in Reaction 1 Products** can be used to specify the molar fraction of CO to CO_2 in the first reaction of Equation 19.7 (p. 2393). The default value of 1 implies that all carbon is reacted to CO, with no CO_2 produced.

9. The **High Temperature Volatile Yield**. Enhanced devolatization at higher temperatures can cause the volatile yield to exceed the proximate analysis fraction. To model this, the actual volatile fraction used is calculated as that specified in the **Proximate Analysis** input multiplied by the **High Temperature Volatile Yield**. The actual fixed carbon fraction is then calculated as one minus the sum of the actual volatile, ash and moisture fractions.

10. **Fraction of N in Char (DAF)**. This input is used in calculating the split of atomic nitrogen for the Fuel NOx model.

11. **Coal Dry Density** is used to calculate the volume fraction of liquid-water for the **Wet Combustion** option in the **Injections** dialog box.

Clicking **Apply** will save the **Coal Property** data for the selected **Coal Stream ID**. When you click **OK**, Ansys Fluent makes the following changes:

1. A Mixture material is created, named *coal-volatiles-air*, with a one or two step reaction mechanism as specified in the **Mechanism** option. If the **Fluid** material species (O_2, CO, CO_2, and so on) do not exist, they are created. A **Fluid** material called *coal-volatiles*, is also created with a standard state enthalpy calculated from the ultimate and proximate analyses, as-received HCV and volatile molecular weight. For the second and third coal streams, additional **Fluid** materials and reactions with the properties corresponding to the second and third coal streams are also created in the mixture.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. A combusting-particle material is created for each coal stream with **Volatile Component Fraction** and **Combustible Fraction** calculated from the ultimate and proximate analyses. The discrete phase model (DPM) is enabled.

3. For the fuel NOx model, the default fuel species is set to *vol*, the char N conversion is set to *NO*, and the fuel NOx **Volatile** and **Char** mass fractions are set according to the ultimate and proximate compositions of the first coal stream. Note that even though some of the default fuel NOx parameters are changed, the fuel NOx model itself is not enabled.

4. If the **Moisture** mass fraction in the coal proximate analysis is nonzero, then the **Wet Combustion** option is enabled in the **Coal Calculator** dialog box, and all subsequent injections that are created will have wet combustion enabled. The liquid material will be set to `water-liquid`, and the volume fraction of water will be calculated from the **Moisture** mass fraction specified in the proximate analysis and the **Coal Dry Density**. The wet combustion settings are not modified for existing injections. The **Density** for the combusting-particle in the **Create/Edit Materials** dialog box will also be set to **Coal Dry Density**.

Note:

Irrespective of the **Wet Combustion** option in the **Coal Calculator** dialog box you can modify the **Wet Combustion** model setting for the injections in the **Set Injection Properties** dialog box. In addition, you will need to take care to enter the flow-rate of each injection on a wet or dry basis, consistent with the **Wet Combustion** model setting.

19.1.1.6. Defining Cell Zone and Boundary Conditions for Species

You will need to specify the inlet mass fraction for all species in your simulation. In addition, for pressure outlets you will set species mass fractions to be used in case of backflow. At walls, Ansys Fluent will apply a zero-gradient (zero-flux) boundary condition for all species by default, although you can change each species boundary condition to a specified value. If you have surface reactions defined (see Wall Surface Reactions and Chemical Vapor Deposition (p. 2400)), you can choose to enable wall-surface reactions and select the chemical mechanism. For fluid zones, you also have the option of specifying a reaction mechanism. Input of cell zone and boundary conditions is described in Cell Zone and Boundary Conditions (p. 1299).

Note:

In a single-phase case with all fluid zones, only one mixture material can be assigned to the zones. However, if the fluid zones are separated by walls, then one mixture and any number of non-mixture (pure fluid) materials can be assigned to different zones.

Important:

- Non-reflecting boundary conditions (NRBCs) are not compatible with species transport models. They are mainly used to solve ideal-gas single species flow. For information about NRBCs, see Boundary Acoustic Wave Models (p. 1527).

- Note that you will explicitly set mass fractions only for the first $N-1$ species. The solver will compute the mass fraction of the last species by subtracting the total of

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the specified mass fractions from 1. If you want to explicitly specify the mass fraction of the last species, you must reorder the species in the list (in the Create/Edit Materials Dialog Box (p. 4885)), as described in Defining Properties for the Mixture and Its Constituent Species (p. 2374).

19.1.1.6.1. Diffusion at Inlets with the Pressure-Based Solver

For the pressure-based solver in Ansys Fluent, the net transport of species at inlets consists of both convection and diffusion components. The convection component is fixed by the specified inlet species mass or mole fraction, whereas the diffusion component depends on the gradient of the computed species concentration field (which is not known a priori). At very small convective inlet velocities, for example when modeling perforated combustion liners with an inlet, substantial mass can be gained or lost through the inlet due to diffusion. For this reason, inlet diffusion is disabled by default, but can be enabled with the `define/models/species/inlet-diffusion` text command.

19.1.1.7. Defining Other Sources of Chemical Species

You can define a source or sink of a chemical species within the computational domain by defining a source term in the **Fluid** dialog box. You may choose this approach when species sources exist in your problem but you do not want to model them through the mechanism of chemical reactions. Defining Mass, Momentum, Energy, and Other Sources (p. 1382) describes the procedures you would follow to define species sources in your Ansys Fluent model. If the source is not a constant, you can use a user-defined function. See the Fluent Customization Manual for details about user-defined functions.

19.1.1.8. Solution Procedures for Chemical Mixing and Finite-Rate Chemistry

While many simulations involving chemical species may require no special procedures during the solution process, you may find that one or more of the solution techniques noted in this section helps to accelerate the convergence or improve the stability of more complex simulations. The techniques outlined below may be of particular importance if your problem involves many species and/or chemical reactions, especially when modeling combusting flows.

19.1.1.8.1. Stability and Convergence in Reacting Flows

Obtaining a converged solution in a reacting flow can be difficult for a number of reasons. First, the impact of the chemical reaction on the basic flow pattern may be strong, leading to a model in which there is strong coupling between the mass/momentum balances and the species transport equations. This is especially true in combustion , where the reactions lead to a large heat release and subsequent density changes and large accelerations in the flow. All reacting systems have some degree of coupling, however, when the flow properties depend on the species concentrations. These coupling issues are best addressed by the use of a two-step solution process, as described below, and by the use of under-relaxation as described in Setting Under-Relaxation Factors (p. 3589).

A second convergence issue in reacting flows involves the magnitude of the reaction source term. When the Ansys Fluent model involves very rapid reaction rates (reaction time scales are much faster than convection and diffusion time scales), the solution of the species transport equations

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

becomes numerically difficult. Such systems are termed "stiff" systems. Stiff systems with finite-rate chemistry should be solved using either the pressure-based or density-based solver together with either **Stiff Chemistry Solver** or the **CHEMKIN-CFD Solver** (see Solution of Stiff Chemistry Systems (p. 2397)). Finite-rate chemistry may also be used for turbulent combustion. For turbulent combustion, turbulence-chemistry interactions can either be neglected (**Finite-rate/No TCI** model) or accounted for using **Eddy-Dissipation Concept** or **PDF Transport** models.

19.1.1.8.2. Two-Step Solution Procedure (Steady-state Only)

Solving a reacting flow as a two-step process is a practical method for reaching a stable converged solution to your Ansys Fluent problem. In this process, you begin by solving the flow, energy, and species equations with reactions disabled (the "cold-flow", or unreacting flow). When the basic flow pattern has thereby been established, you can re-enable the reactions and continue the calculation. The cold-flow solution provides a good starting solution for the calculation of the combusting system. This two-step approach to combustion modeling can be accomplished using the following procedure:

1. Set up the problem including all species and reactions of interest.

2. Temporarily disable reaction calculations by turning off **Volumetric** in the Species Model Dialog Box (p. 4768).

   Setup → **Models** → **Species** **Edit...**

3. Disable calculation of the product species in the Equations Dialog Box (p. 5166).

   Solution → **Controls** → **Equations...**

4. Calculate an initial (cold-flow) solution. (Note that it is generally not productive to obtain a fully converged cold-flow solution unless the non-reacting solution is also of interest to you.)

5. Enable the reaction calculations by turning on **Volumetric** again in the Species Model Dialog Box (p. 4768).

6. Enable all equations in the **Equations** dialog box. If you are using the **Finite-Rate/No TCI**, **Finite-Rate/Eddy-Dissipation**, **Eddy-Dissipation Concept** or **PDF Transport** model for the turbulence-chemistry interaction, you may need to patch an ignition source (as described in Ignition in Steady-State Combustion Simulations (p. 2397)).

19.1.1.8.3. Density Under-Relaxation

One of the main reasons a combustion calculation can have difficulty converging is that large changes in temperature cause large changes in density, which can, in turn, cause instabilities in the flow solution. When you use the pressure-based solver, Ansys Fluent allows you to under-relax the change in density to alleviate this difficulty. The default value for density under-relaxation is 1, but if you encounter convergence trouble you may want to reduce this to a value between 0.5 and 1 (in the **Solution Controls** task page).

Note that if you are using a real-gas model the solution might converge at a slower rate than when running ideal-gas flow. To improve the convergence performance of your analysis, you need to reduce the underrelaxation for density to values as low as 0.1.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

19.1.1.8.4. Ignition in Steady-State Combustion Simulations

If you introduce fuel to an oxidant or have a premixed fuel/oxidant mixture, but you are not simulating the transient ignition process, then you must numerically force the solution to the burned state (rather than the unburned state). Generally, spontaneous ignition does not occur unless the temperature of the mixture exceeds the activation energy threshold required to maintain combustion. For steady-state Ansys Fluent combustion simulations, whether you are using the species transport or PDF transport model and regardless of turbulence-chemistry interaction option, you have to supply an ignition source to initiate combustion. This ignition source may be a heated surface or inlet mass flow that heats the gas mixture above the required ignition temperature. Often, however, it is the equivalent of a spark: an initial solution state that causes combustion to proceed. You can supply this initial "spark" by patching a hot temperature into a region of the Ansys Fluent model that contains an ignitable fuel/air mixture.

Solution → **Initialization** → **Patch...**

Depending on the model, you may need to patch both the temperature and the fuel/ oxidant/product concentrations to produce ignition in your model. The initial patch has no impact on the final steady-state solution—no more than the location of a match determines the final flow pattern of the torch that it lights. See Patching Values in Selected Cells (p. 3630) for details about patching initial values.

19.1.1.8.5. Solution of Stiff Chemistry Systems

When modeling stiff chemistry systems, such as flames with finite-rate kinetics, you can either use the pressure-based solver with the **Stiff Chemistry Solver** option enabled or **CHEMKIN-CFD Solver** selected as seen in the Species Model Dialog Box (p. 4768) (Figure 19.1: The Species Model Dialog Box (p. 2360)), or the density-based solver.

For steady simulations using the pressure-based solver, the **Stiff Chemistry Solver** option approximates the reaction rate R_i in the species transport equation (see Equation 7.19 in the Theory Guide) as,

$$R_i^* = \frac{1}{\tau}\int_0^{\tau} R_i dt \tag{19.8}$$

where τ is an appropriate time step size. Note that as τ tends to zero the approximation becomes exact but the stiff numerics will cause the pressure-based solver to diverge. On the other hand, as τ tends to infinity, the approximated reaction rate R_i^* tends to zero and, while the numerical stiffness is alleviated, there is no reaction. In Ansys Fluent, the default value for τ is set to one-tenth of the minimum convective or diffusive time-scale in the cell. This value was found to be sufficiently accurate and robust, although it can be modified using the `solve/set/stiff-chemistry` text command. ISAT is employed to integrate the stiff chemistry in Equation 19.8 (p. 2397). Equation 19.8 (p. 2397) is also applied for unsteady simulations, with τ = flow time step size.

Details about the ISAT algorithm may be found in Particle Reaction in the Theory Guide and Using ISAT Efficiently (p. 2459). For efficient and accurate use of ISAT, a review of this section is highly recommended.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The density-based implicit solver may also be used with either chemistry solver option. This option allows a larger stable Courant (CFL) number specification, although additional calculations are required to calculate the eigenvalues of the chemical system Jacobian [170] (p. 5722). When enabling the stiff-chemistry solver with the density-based solver, the following must be specified:

- **Temperature Positivity Rate Limit**: limits new temperature changes by this factor multiplied by the old temperature. Its default value is 0.2.
- **Temperature Time Step Reduction**: limits the local CFL number when the temperature is changing too rapidly. Its default value is 0.25.

19.1.1.8.6. Eddy-Dissipation Concept Model Solution Procedure

The Eddy Dissipation Concept (EDC) model is used to consider detailed chemistry and its interaction with turbulence. Due to the use of an ODE solver to obtain the fine scale species mass fractions and temperature, the computational expense of the EDC model can be high. One efficient solution procedure is to first use the relaxation to chemical equilibrium model (see The Relaxation to Chemical Equilibrium Model in the *Fluent Theory Guide*) to quickly obtain an initial solution and then switch to the EDC model to complete the runs for the desired accuracy.

Note that the default numerical parameters for the solution of the EDC equations are set to provide maximum robustness with slowest convergence. The convergence rate can be increased by setting the **Aggressiveness Factor** in the **Species** dialog box or with the text command:

`define` → `models` → `species` → `set-turb-chem-interaction`

The **Aggressiveness Factor** can be set from 0 (slow but stable) to 1 (fast but least stable). The default is 0.5.

19.1.1.9. Postprocessing for Species Calculations

Ansys Fluent can report chemical species as mass fractions, mole fractions, and molar concentrations. You can also display laminar and effective mass diffusion coefficients. The following variables are available for postprocessing of species transport and reaction simulations:

- **Mass fraction of species-n**
- **Mole fraction of species-n**
- **Molar Concentration of species-n**
- **Lam Diff Coef of species-n**
- **Eff Diff Coef of species-n**
- **Thermal Diff Coef of species-n**
- **Enthalpy of species-n** (pressure-based solver calculations only)
- **species-n Source Term** (density-based solver calculations only)
- **Relative Humidity**
- **TFM Thickening Factor** (Thickened Flame Model only)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **TFM Omega** (Thickened Flame Model only)
- **TFM Thickening Factor** (turbulent cases (LES / DES / SAS / SBES / SDES) with Thickened Flame Model only)
- **Laminar Flame Speed** (Thickened Flame Model only)
- **Laminar Flame Thickness** (Thickened Flame Model only)
- **Cell Time Scale** (Eddy-Dissipation Concept and laminar finite-rate with stiff-chemistry only)
- **Fine Scale Mass fraction of species-n** (Eddy-Dissipation Concept model only)
- **EDC Cell Volume Fraction** (Eddy-Dissipation Concept model only)
- **Fine Scale Temperature** (Eddy-Dissipation Concept model only)
- **Net Rate of species-n** (Eddy-Dissipation Concept and laminar finite-rate with stiff-chemistry only)
- **Kinetic Rate of Reaction-n**
- **Turbulent Rate of Reaction-n**
- **Liquid species mass fraction of species-n** (solidification and melting model only)
- **Heat of Reaction**

These variables are contained in the **Species...**, **Temperature...**, and **Reactions...** categories of the variable selection drop-down list that appears in postprocessing dialog boxes. See Field Function Definitions (p. 4167) for a complete list of flow variables, field functions, and their definitions. Displaying Graphics (p. 3869) and Reporting Alphanumeric Data (p. 4097) explain how to generate graphics displays and reports of data.

Note:

The net reaction rates of species are computed during the solution of transport equations, and, therefore, you need to run at least one iteration to obtain correct values for postprocessing.

19.1.1.9.1. Averaged Species Concentrations

Averaged species concentrations at inlets and exits, and across selected planes (that is, surfaces that you have created using **Create** in the **Domain** ribbon tab (**Surface**group box)) within your model can be obtained using the Surface Integrals Dialog Box (p. 5325), as described in Surface Integration (p. 4150).

Results → Reports → Surface Integrals **Edit...**

Select the **Molar Concentration of species-n** for the appropriate species in the **Field Variable** drop-down list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

19.1.2. Wall Surface Reactions and Chemical Vapor Deposition

For gas-phase reactions, the reaction rate is defined on a volumetric basis and the rate of creation and destruction of chemical species becomes a source term in the species conservation equations. Heterogeneous surface reactions create sources (and sinks) of chemical species in the gas-phase as well, but also alter surface coverages for surface site reactions, and may deposit or etch species for bulk (solid) reactions.

For more information about the theoretical background of wall surface reactions and chemical vapor depositions, see Wall Surface Reactions and Chemical Vapor Deposition in the Theory Guide. Information about using wall surface reactions is presented in the following subsections:

19.1.2.1. Overview of Surface Species and Wall Surface Reactions
19.1.2.2. Importing a Surface Kinetic Mechanism in CHEMKIN Format
19.1.2.3. Manual Inputs for Wall Surface Reactions
19.1.2.4. Including Mass Transfer To Surfaces in the Continuity Equation
19.1.2.5. Wall Surface Mass Transfer Effects in the Energy Equation
19.1.2.6. Modeling the Heat Release Due to Wall Surface Reactions
19.1.2.7. Solution Procedures for Wall Surface Reactions
19.1.2.8. Postprocessing for Surface Reactions

19.1.2.1. Overview of Surface Species and Wall Surface Reactions

Ansys Fluent treats chemical species adsorbed and desorbed, as well as those deposited into or etched from the bulk solid surfaces as distinct from the same chemical species in the gas. Similarly, surface reactions are defined distinctly and treated differently than gas-phase reactions involving the same chemical species.

Surface reactions can be limited so that they occur on only some of the wall boundaries (while the other wall boundaries remain free of surface reaction). The surface reaction rate is defined and computed per unit surface area, in contrast to the fluid-phase reactions, which are based on unit volume.

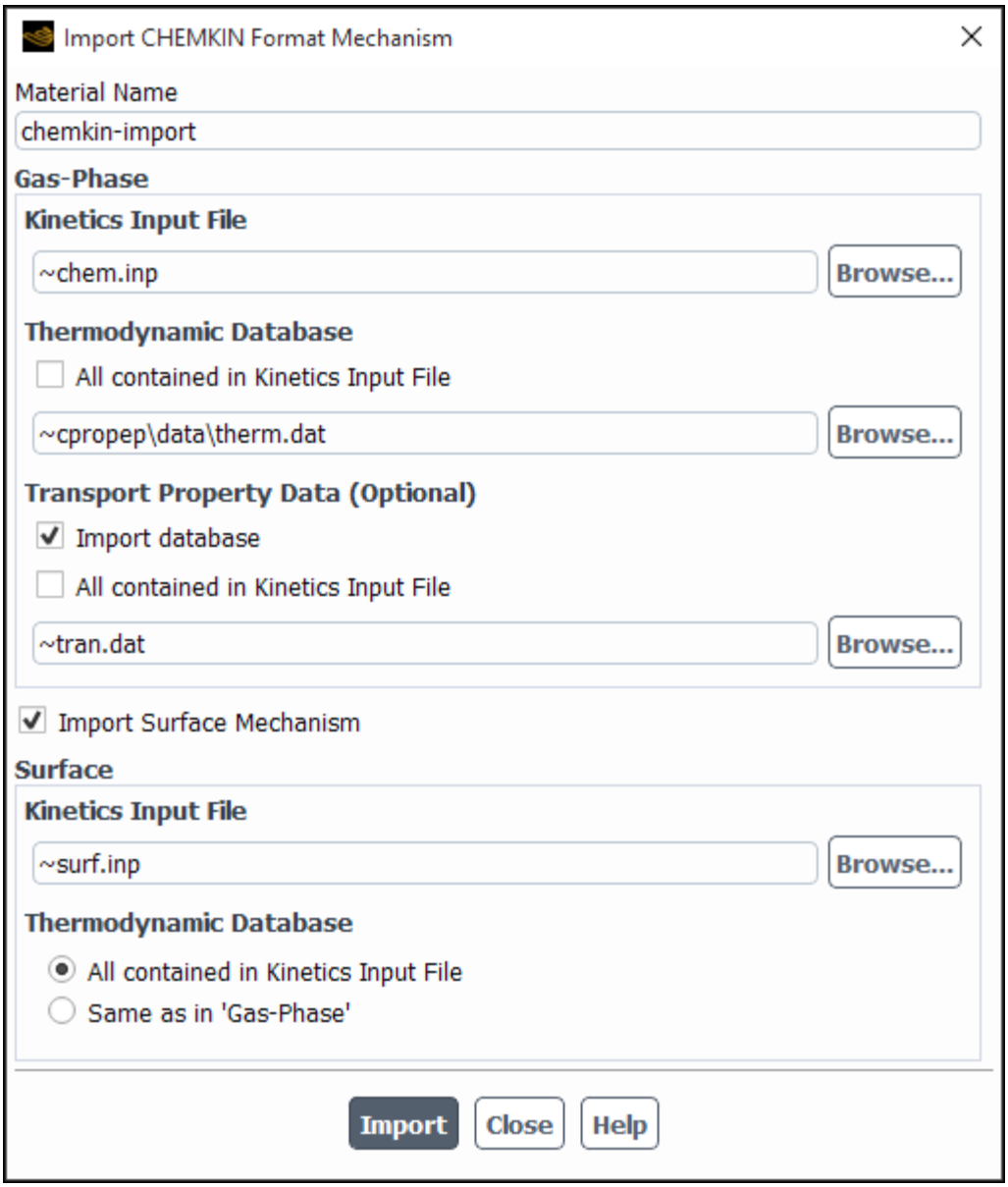
19.1.2.2. Importing a Surface Kinetic Mechanism in CHEMKIN Format

Importing surface kinetic mechanisms in CHEMKIN format requires that the gas-phase mechanism file accompanies the surface mechanism file for full compatibility with CHEMKIN (for more information on how to import the gas-phase mechanism, see Importing a Volumetric Kinetic Mechanism in CHEMKIN Format (p. 2368)). If the gas-phase mechanism file is not available, then you will need to create one that you will import along with the surface mechanism file. For information about the format of CHEMKIN surface kinetics mechanisms, see Surface Kinetics Input in the *Chemkin Input Manual*.

The mechanism files are imported into Ansys Fluent using the **Import CHEMKIN Format Mechanism** dialog box (Figure 19.17: The Import CHEMKIN Format Mechanism Dialog Box for Surface Kinetics (p. 2401)) which is opened by clicking **Import CHEMKIN Mechanism...** in the **Species Model** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 19.17: The Import CHEMKIN Format Mechanism Dialog Box for Surface Kinetics



In the **Import CHEMKIN Format Mechanism** dialog box:

1. Enter a name for the chemical mechanism under **Material Name**.
2. Enable **Import Surface Mechanism**.
3. Enter the paths in the **Kinetic Input File** text fields in the **Gas-Phase** and **Surface** group boxes (for example, *path*/`chem.inp` and *path*/ *path*/`surf.inp`, respectively).
4. Specify species thermodynamic property data. You can include the data in the kinetics input file and/or in a separate file as follows:
   - If you want to specify thermodynamic data for all species in the kinetics input file, enable **All contained in Kinetics Input File** to avoid the need to specify a separate file.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you provide some or all species thermodynamic data in a separate file, disable **All contained in Kinetics Input File** and specify the thermodynamic data file location.

You can either use the default thermodynamic database file, `thermo.db`, provided in the `cpropep\data` directory in your Ansys Fluent installation area or specify the path of your thermodynamic database file if `thermo.db` does not contain all the gas-phase species in the CHEMKIN mechanism. The format for `thermo.db` is detailed in the CHEMKIN manual [79] (p. 5717).

5. To specify the surface species thermodynamic database, you can:

   - Select **All contained in Kinetics Input File** under the **Thermodynamic Database** and include the surface species thermodynamics data in the surface kinetics input file.
   - Select **Same as 'Gas-Phase'** and include the surface species thermodynamics data in the thermodynamics data file used in the gas-phase mechanism. Note that the default `thermo.db` file contains only gas-phase species. If you want to use this file, you will need to supply a thermodynamic database that includes surface species defined in your simulation.

   Note:

   The surface species thermodynamic data must be included either in the surface kinetics input file or in the gas-phase thermodynamics data file. If you do not import thermodynamics data as a separate file in the gas-phase mechanism, then all surface thermodynamics data must be contained in the surface kinetics input file.

6. Click the **Import** button.

Ansys Fluent will create a material with the specified name, which will contain the CHEMKIN data for the species and reactions, and add it to the list of **Fluent Mixture Materials**. You can view all of the reactions by clicking the **Edit...** button to the right of **Mechanism**, under **Properties** in the Create/Edit Materials Dialog Box (p. 4885).

Ansys Fluent will also automatically save the imported data within the Fluent case file. Note that carrying the original input files along with the Fluent case file is not required. Ansys Fluent will automatically retrieve these files when needed (for example, when using the CHEMKIN-CFD solver).

For details on importing gas species transport properties, see Procedure for Importing Volumetric CHEMKIN Mechanisms (p. 2369).

Important:

A surface reaction rate constant can be expressed in terms of a sticking coefficient in CHEMKIN mechanisms. When **Stiff Chemistry Solver** is selected from the **Chemistry Solver** drop-down list, Ansys Fluent will convert this sticking coefficient form to an equivalent Arrhenius rate expression [78] (p. 5717). For full compatibility with Ansys Chemkin, select **CHEMKIN-CFD Solver** instead.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

19.1.2.2.1. Compatibility and Limitations for Gas Phase Reactions

Generally, Ansys Fluent handles all reaction types that are supported in Ansys Chemkin. However, some limitations still exist for the following models and features:

- the **None – Explicit Source** chemistry solver (non-stiff chemistry)
- the **Finite-Rate/Eddy-Dissipation** turbulence-chemistry interaction

In these cases, only the following volume reaction types can be used:

- Arrhenius, including those reactions with arbitrary reaction order, third-body efficiencies and non-integer stoichiometric coefficients
- Pressure-dependent reactions using the Lindemann, Troe and SRI forms
- CHEMKIN mechanism files with CHEMKIN-default rate parameter units
- Duplicate reactions (keyword DUP)
- Arbitrary reverse reaction (keyword REV)

19.1.2.2.2. Compatibility and Limitations for Surface Reactions

As with volume reactions, Ansys Fluent generally handles all reaction types that are supported in Ansys Chemkin. However, with the **None – Explicit Source** chemistry solver or **Finite-Rate/Eddy-Dissipation** turbulence-chemistry interaction, only the following surface reaction types can be used:

- Arrhenius reactions, including those with arbitrary reaction order, third-body efficiencies and non-integer stoichiometric coefficients
- Sticking coefficients (keyword STICK). Ansys Fluent converts these to an equivalent Arrhenius expression.
- Arbitrary reaction units
- Duplicate reactions (keyword DUP)
- Surface coverage modification (keyword COV)

For a detailed description of the keywords, see Ansys Chemkin Input Manual.

19.1.2.3. Manual Inputs for Wall Surface Reactions

The basic steps for setting up a problem involving wall surface reactions are the same as those presented in Overview of User Inputs for Modeling Species Transport and Reactions (p. 2356) for setting up a problem with only fluid-phase reactions, with a few additions:

1. In the Species Model Dialog Box (p. 4768):

   Setup → **Models** → **Species** **Edit...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. Enable **Species Transport**, select **Volumetric** and **Wall Surface** under **Reactions**, and specify the **Mixture Material**. See Enabling Species Transport and Reactions and Choosing the Mixture Material (p. 2359) for details about this procedure, and Mixture Materials (p. 2357) for an explanation of the mixture material concept.

b. (optional) If you want to model the heat release due to wall surface reactions, enable the **Heat of Surface Reactions** option.

c. (optional) If you want to include the effect of surface mass transfer in the continuity equation, enable the **Mass Deposition Source** option. When the density-based solver is used, this option is always enabled.

d. To control the robustness and the convergence speed, enter a value between 0 and 1 for the **Aggressiveness Factor**. A value of 0 is the most robust, but results in the slowest convergence. The default value for the **Aggressiveness Factor** is 0.5. This parameter appears only when the pressure-based solver is used.

e. (optional) If you are using the pressure-based solver and you want to include species diffusion effects in the energy equation, enable the **Diffusion Energy Source** option. See Wall Surface Mass Transfer Effects in the Energy Equation (p. 2405) for details.

f. (optional, but recommended for CVD) If you want to model full multicomponent (Stefan-Maxwell) diffusion or thermal (Soret) diffusion, enable the **Full Multicomponent Diffusion** or **Thermal Diffusion** option. See Full Multicomponent Diffusion in the *Fluent Theory Guide* for details.

2. Check and/or define the properties of the mixture. See Defining Properties for the Mixture and Its Constituent Species (p. 2374).

   Setup → Materials

   Mixture properties include the following:

   - species in the mixture
   - reactions
   - other physical properties (for example, viscosity, specific heat)

   Important:

   - You will find all species (including the solid/bulk and site species) in the list of **Fluent Fluid Materials**. For a deposited species such as Si, you will need both Si(g) and Si(s) in the materials list for the **fluid** material type.
   - Note that the *final* gas-phase species named in the **Selected Species** list should be the carrier gas. This is because Ansys Fluent will not solve the transport equation for the final species. Note also that any reordering, adding or deleting of species should be handled with caution, as described in Assigning the Last Species (p. 2378).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Check and/or set the properties of the individual species in the mixture. (See Defining Properties for the Mixture and Its Constituent Species (p. 2374).) Note that if you are modeling the heat of surface reactions, you should be sure to check (or define) the formation enthalpy for each species.

4. Set species boundary conditions.

   Setup → Boundary Conditions

   In addition to the boundary conditions described in Defining Cell Zone and Boundary Conditions for Species (p. 2394), you will first need to indicate whether or not surface reactions are in effect on each wall. If so, you will then need to assign a reaction mechanism to the wall. To enable the effect of surface reaction on a wall, enable the **Reaction** option in the **Species** section of the **Wall** dialog box.

   Important:

   If you have enabled the global **Low-Pressure Boundary Slip** option in the Viscous Model Dialog Box (p. 4715), the **Shear Condition** for each wall will be reset to **No Slip** even though the slip model will be in effect. Note that the **Low-Pressure Boundary Slip** option is available only when the **Laminar** model is selected in the **Viscous Model** dialog box.

   See Inputs at Wall Boundaries (p. 1458) for details about boundary condition inputs for walls. See User Inputs for Porous Media (p. 1343) for details about boundary condition inputs for porous media.

19.1.2.4. Including Mass Transfer To Surfaces in the Continuity Equation

In the surface reaction boundary condition described above, the effects of the wall normal velocity or bulk mass transfer to the wall are not included in the computation of species transport. The momentum of the net surface mass flux from the surface is also ignored because the momentum flux through the surface is usually small in comparison with the momentum of the flow in the cells adjacent to the surface. However, you can include the effect of surface mass transfer in the continuity equation by activating the **Mass Deposition Source** option in the Species Model Dialog Box (p. 4768). With the density-based solver, the effect of surface mass transfer in the continuity equation is always included.

19.1.2.5. Wall Surface Mass Transfer Effects in the Energy Equation

Species diffusion effects in the energy equation due to wall surface reactions are included in the normal species diffusion term described in Treatment of Species Transport in the Energy Equation in the Theory Guide.

If you are using the pressure-based solver, you can neglect this term by disabling the **Diffusion Energy Source** option in the Species Model Dialog Box (p. 4768). For the density-based solvers, this term is always included; you cannot disable it. Neglecting the species diffusion term implies that errors may be introduced to the prediction of temperature in problems involving mixing of species with significantly different heat capacities, especially for components with a Lewis number far from unity. While the effect of species diffusion should go to zero at Le = 1, you may see subtle effects due to differences in the numerical integration in the species and energy equations.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

19.1.2.6. Modeling the Heat Release Due to Wall Surface Reactions

The heat release due to a wall surface reaction is, by default, ignored by Ansys Fluent. You can, however, choose to include the heat of surface reaction by activating the **Heat of Surface Reactions** option in the Species Model Dialog Box (p. 4768) and setting appropriate formation enthalpies in the Edit Material Dialog Box (p. 4957).

19.1.2.7. Solution Procedures for Wall Surface Reactions

As in all CFD simulations, your surface reaction modeling effort may be more successful if you start with a simple problem description, adding complexity as the solution proceeds. For wall surface reactions, you can follow the same guidelines presented for fluid-phase reactions in Solution Procedures for Chemical Mixing and Finite-Rate Chemistry (p. 2395).

In addition, if you are modeling the heat release due to surface reactions and you are having convergence trouble, you should try temporarily turning off the **Heat of Surface Reactions** and **Mass Deposition Source** options in the Species Model Dialog Box (p. 4768).

If you are modeling surface site species, good estimates of the **Initial Site Coverage** will aid convergence.

19.1.2.8. Postprocessing for Surface Reactions

In addition to the gas-phase variables listed in Postprocessing for Species Calculations (p. 2398), for surface reactions you can display/report the surface coverage as well as the deposition rate of the solid species deposited on a surface. Select **Surface Coverage of species-n**, **Surface Deposition Rate of species-n**, or **Porous Deposition Rate of species-n** in the **Species...** category of the variable selection drop-down list.

Important:

For surface reactions involving porous media, you can display/report the surface reaction rates using the **Kinetic Rate of Reaction-n(Porous)** in the **Reactions...** category of the variable selection drop-down list.

19.1.3. Particle Reactions

Information about using particle reactions is divided into the following sections:

19.1.3.1. Combusting Particle Surface Reactions

19.1.3.2. Multicomponent Particles with Chemical Reactions

19.1.3.1. Combusting Particle Surface Reactions

As described in The Multiple Surface Reactions Model in the Theory Guide, it is possible to define multiple particle surface reactions to model the surface combustion of a combusting discrete-phase particle. For more information about the theoretical background of particle surface reactions, see Combusting Particle Surface Reactions in the Theory Guide. Information about using particle surface reactions is provided in the following subsections:

19.1.3.1.1. Limitations

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

19.1.3.1.2. User Inputs for Particle Surface Reactions

19.1.3.1.3. Modeling Gaseous Solid Catalyzed Reactions

19.1.3.1.4. Using the Multiple Surface Reactions Model for Discrete-Phase Particle Combustion

19.1.3.1.1. Limitations

Note the following limitations of the multiple surface reactions model:

- The model is available only with the species transport model for volumetric reactions, and not with the nonpremixed, premixed, or partially premixed combustion models.
- The multiple surface reactions model for combusting particles cannot be used with the breakup, collision and coalescence discrete phase models.

19.1.3.1.2. User Inputs for Particle Surface Reactions

The setup procedure for particle surface reactions requires only a few inputs in addition to the procedure for volumetric reactions described in Overview of User Inputs for Modeling Species Transport and Reactions (p. 2356) – Defining Other Sources of Chemical Species (p. 2395). These additional inputs are as follows:

1. In the **Species Model** dialog box, enable the **Particle** option under **Reactions**.

   Setup → **Models** → **Species** **Edit...**

2. When you specify the species involved in the particle surface reaction, be sure to identify the surface species, as described in Defining Properties for the Mixture and Its Constituent Species (p. 2374).

   Important:

   You will find all species (including the surface species) in the list of **Fluent Fluid Materials**. If, for example, you are modeling coal gasification, you will find solid carbon, C(s), in the materials list for the **fluid** material type.

3. For each particle surface reaction, select **Particle** as the **Reaction Type** in the **Reactions** dialog box, and specify the following parameters (in addition to those described in Defining Reactions (p. 2379)):

   Diffusion-Limited Species

   When there is more than one gaseous reactant taking part in the particle surface reaction, the diffusion-limited species is the species for which the concentration gradient between the bulk and the particle surface is the largest. See Figure 7.1: A Reacting Particle in the Multiple Surface Reactions Model in the Theory Guide for an illustration of this concept. In most cases, there is a single gas-phase reactant and the diffusion-limited species does not need to be defined.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Catalyst Species

This option is available only when there are no solid species defined in the stoichiometry of the particle surface reaction. In such a case, you will need to specify the solid species that acts as a catalyst for the reaction. The reaction will proceed only on the particles that contain this solid species. See Using the Multiple Surface Reactions Model for Discrete-Phase Particle Combustion (p. 2408) for details on defining the particle surface species mass fractions.

Diffusion Rate Constant

($C_{1,r}$ in Equation 7.104 in the Theory Guide)

Effectiveness Factor

(η_r in Equation 7.102 in the Theory Guide)

19.1.3.1.3. Modeling Gaseous Solid Catalyzed Reactions

The catalytic particle surface reaction option is enabled in Ansys Fluent when **Particle** is selected as the **Reaction Type** in the Reactions Dialog Box (p. 4930) and there are no solid species in the reaction stoichiometry. The solid species acting as a catalyst for the reaction is defined in the **Reactions** dialog box. The catalytic particle surface reaction will proceed only on those particles containing the catalyst species.

19.1.3.1.4. Using the Multiple Surface Reactions Model for Discrete-Phase Particle Combustion

When you use the multiple surface reactions model, the procedure for setting up a problem involving a discrete phase is slightly different from that outlined in Steps for Using the Discrete Phase Models (p. 2677). The revised procedure is as follows:

1. Enable any of the discrete phase modeling options, if relevant, as described in Physical Models for the Discrete Phase Model (p. 2695).
2. Specify the initial conditions, as described in Setting Initial Conditions for the Discrete Phase (p. 2718).
3. Define the boundary conditions, as described in Setting Boundary Conditions for the Discrete Phase (p. 2779).
4. Define the material properties, as described in Setting Material Properties for the Discrete Phase (p. 2811).

Important:

You must select **multiple-surface-reactions** in the **Combustion Model** drop-down list in the **Create/Edit Materials** dialog box before you can proceed to the next step.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. If you have defined more than one particle surface species, for example, carbon (**C<s>**) and sulfur (**S<s>**), you will need to return to the **Set Injection Properties** dialog box (or **Set Multiple Injection Properties** dialog box) to specify the mass fraction of each particle surface species in the combusting particle. Click the **Multiple Reactions** tab, and enter the **Species Mass Fractions**. These mass fractions refer to the combustible fraction of the combusting particle, and should sum to 1. If there is only one surface species in the mixture material, the mass fraction of that species will be set to 1, and you will not specify anything under **Multiple Surface Reactions**.

6. Set the solution parameters and solve the problem, as described in Solution Strategies for the Discrete Phase (p. 2827).

7. Examine the results, as described in Postprocessing for the Discrete Phase (p. 2833).

Important:

Solid deposition reactions on the particle are not allowed together with custom laws.

19.1.3.2. Multicomponent Particles with Chemical Reactions

Note:

The following limitation currently exists for reacting multicomponent particles:

- Composition-dependent transport properties are not compatible with reacting multicomponent particles. For the particle-mixture material, the **Viscosity**, **Thermal Conductivity**, and **Droplet Surface Tension** properties should not use the default **mass-weighted-mixing-law** option and can only be set to constant or temperature-dependent methods.

To model multicomponent particles with chemical reactions, you should follow the steps described in Steps for Using the Discrete Phase Models (p. 2677) to setup multicomponent particle injections, with the modifications to the setup procedure outlined below. For more information about the theoretical background of the multicomponent particle reactions, see Multicomponent Particles with Chemical Reactions in the *Fluent Theory Guide*.

1. In the **Species Model** dialog box, enable both the **Volumetric** and **Particle** options (**Reactions** group box). This will enable multicomponent particle modeling.

2. In the **Set Injection Properties** dialog box, create an injection of the **Multicomponent** particle type.

3. In the **Create/Edit Materials dialog** box, define the following materials:
   - mixture

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- particle-mixture

Note:

- You can either compose your own mixture and particle-mixture materials or use one of the following predefined mixture and particle-mixture pairs from the Fluent database materials:
  - **Calcium-carbonate-decomposition** and **calcium-carbonate-calcium-oxide**
  - **Urea-water-deposits-air-brack** and **urea-water-deposits**
- The multicomponent particle may contain reacting, volatile, or inert components that do not participate in any reactions.

a. For the particle-mixture material, select **multicomponent-reactions** from the **Reaction Model** drop-down list (**Properties** group box).

   The **Information** dialog box will notify you that the multicomponent particle reactions must be specified in the **Reactions** dialog box for the mixture material.

b. Specify the components of the mixture and particle-mixture materials in the **Species** dialog box (accessed by clicking **Edit...** next to **Mixture Species** (**Properties** group box)).

   When the **multicomponent-reactions** is used as the reaction model, then the **Available Materials** list is populated with both the fluid and droplet-particle materials for the mixture and the particle-mixture materials. In particular, for the particle-mixture material, the **Available Materials** list contains the fluid materials listed in the **Selected Solid/Condensed Species** list for the mixture material (see Figure 19.18: The Species Dialog Box for the Mixture Material (p. 2411) and Figure 19.19: The Species Dialog Box for the Particle-Mixture Material with Reacting Components (p. 2411)).

   Reacting components must be included in the **Selected Solid/Condensed Species** list of the **Species** dialog box for the mixture material, and the particle species participating in reactions must be added to the **Selected Species** list for the particle-mixture material as shown below.

   Note:

   The reacting components included in the **Selected Solid/Condensed Species** list of the **Species** dialog box for the mixture material are assumed to be in solid state, and consequently the viscosity and surface tension properties are not available in the **Create/Edit Materials** dialog box.

   As an exception, generic values of viscosity and surface tension of the materials **urea-liquid**, **biuret-liquid**, and **isocyanic-acid-liquid** of the predefined **urea-water-deposits** and **urea-water-deposits-air-brack** mixture materials can be specified using the text command:

   ```
   define/models/dpm/options/scr-urea-deposition/set-liquid-properties
   ```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 19.18: The Species Dialog Box for the Mixture Material

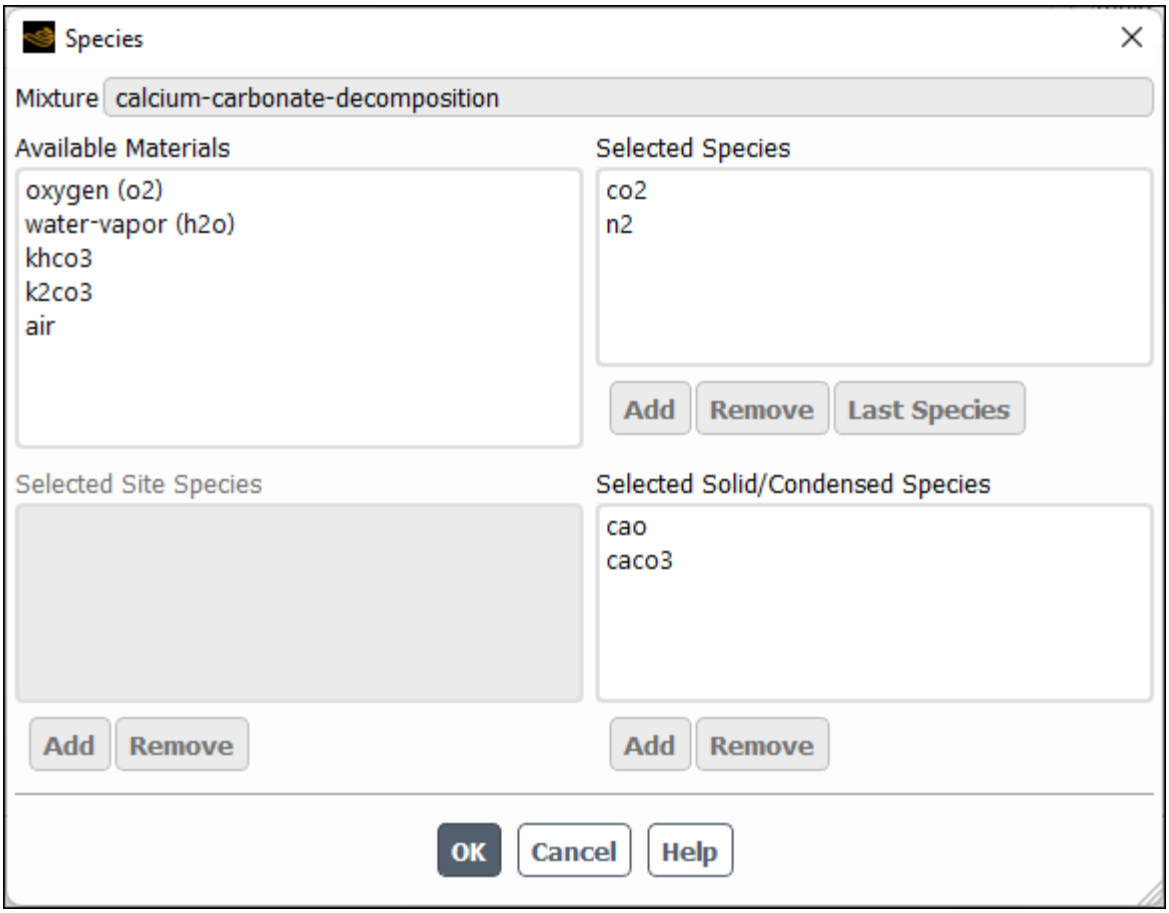
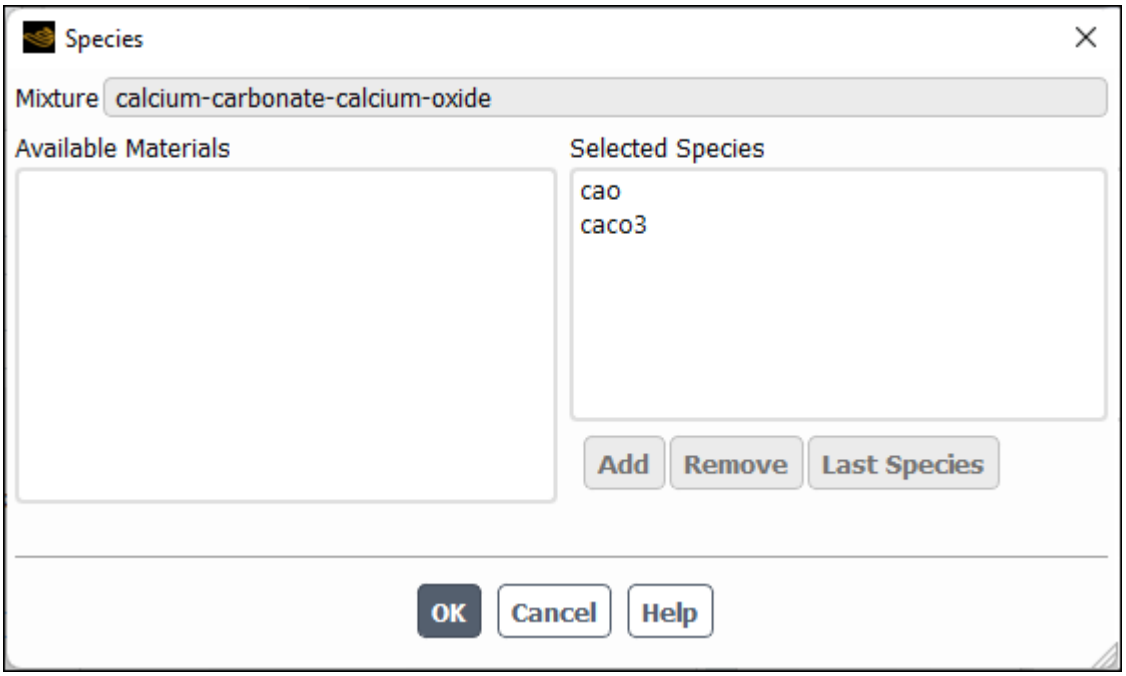


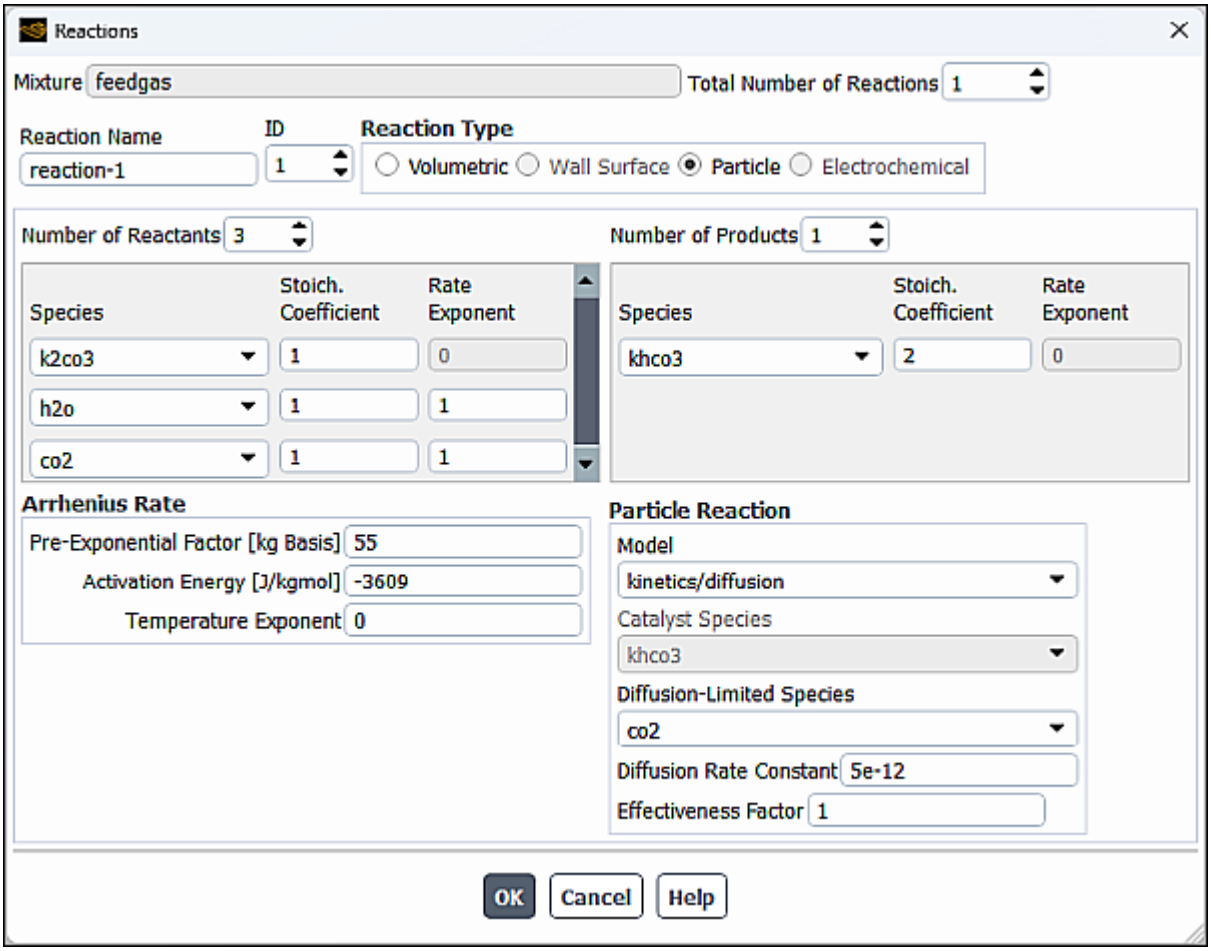
Figure 19.19: The Species Dialog Box for the Particle-Mixture Material with Reacting Components



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. For the mixture material, define particle reactions in the **Reactions** dialog box (accessed by clicking **Edit...** next to the **Reaction** drop-down list). See Inputs for Reaction Definition (p. 2379) for more information.

Figure 19.20: The Reaction Dialog Box of a Mixture Material with Particle Reactions Enabled



i. In the **Reaction Type** group box, enable **Particle**.

ii. Specify parameters for each reaction. Note that reactions can include particle species or gas phase species as either reactants or products.

iii. In the **Particle Reaction** group box, select the particle reaction **Model** from one of the following options:

- **kinetics/diffusion**

  Calculates the reaction rate according to Ansys Fluent Model Formulation in the *Fluent Theory Guide*. This is the same model used for the combusting particle material when the **multiple-surface-reactions** option is selected as the **Combustion Model** in the **Create/Edit Materials** dialog box. For information about the kinetics/diffusion model parameters, see User Inputs for Particle Surface Reactions (p. 2407).

- **surface-kinetics**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Calculates the reaction rate by Equation 7.112 in the *Fluent Theory Guide*.

- **volume-kinetics**

  Calculates the reaction rate according to Equation 7.114 in the *Fluent Theory Guide*

- **phase-change**

  Calculates the reaction rate according to Equation 7.115 in the *Fluent Theory Guide*

- **thermolysis**

  Calculates the thermolysis reaction rate according to Equation 7.116 in the *Fluent Theory Guide*. Note that the thermolysis reaction only occurs in freestream particles, and not in the Lagrangian wall-film particles.

- **user-defined**

- Allows you to specify a custom particle reaction using the `DEFINE_PR_RATE` user-defined function (UDF). See `DEFINE_PR_RATE` in the *Fluent Customization Manual* for details.

4. In the **Species Model** dialog box, make sure that the mixture material with the particle reactions is selected as the **Mixture Material** (**Mixture Properties** group box).

5. In the **Set Injection Properties** dialog box, configure the following settings:

   a. Select the particle-mixture from the **Material** drop-down list.

   b. In the **Components** tab, specify the multicomponent particle composition.

      Note the following:

      - If a gas phase species is selected from the **Evaporating Species** dropdown list, this component is considered volatile and will evaporate according to Multicomponent Particle Definition (Law 7) in the *Fluent Theory Guide*.

      - If **none** is selected for the **Evaporating Species**, the component is available to participate in particle reactions or is inert/non-reacting.

      - The chemical reactions begin after all components with a corresponding gas phase species selected in the **Evaporating Species** drop-down list have evaporated.

Note:

It is recommended that for complex reaction mechanisms, you enable **Multicomponent** in the **Coupled Heat-Mass Solution** group box of the **Discrete Phase Model** dialog box (**Numerics** tab).

Once the solution is completed, the mass fractions of reacting and non-reacting components will be available for postprocessing similar to other particle variables.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

19.1.4. Electrochemical Reactions

Similar to wall surface reactions, electrochemical reactions generate sources (and sinks) in chemical species transport equations.

For more information about the theoretical background of electrochemical reactions, see Electrochemical Reactions in the *Fluent Theory Guide*. Information about using electrochemical reactions is presented in the following sections:

19.1.4.1. Overview and Limitation of Electrochemical Reactions
19.1.4.2. User Inputs for Electrochemical Reactions
19.1.4.3. Electrochemical Reaction Effects in the Energy Equation
19.1.4.4. Electrochemical Reaction Effects in the Species Transport Equation
19.1.4.5. Including Mass Transfer in Continuity
19.1.4.6. Solution Procedures for Electrochemical Reactions
19.1.4.7. Modeling Corrosion with the Water Corrosion Pre Tool

19.1.4.1. Overview and Limitation of Electrochemical Reactions

An electrochemical reaction occurs only on wall surfaces where the electrolyte and the electrode meet. Both aqueous and solid species can participate in electrochemical reactions. Solid species that can be involved in electrochemical reactions are corroded or deposited species.

Electrochemical reactions can be limited, so that they can only occur on some of the wall boundaries (while the other wall boundaries remain free of reaction). The reaction rate is defined and computed per unit surface area, in contrast to the fluid-phase reactions, which are based on unit volume.

You can specify different reaction mechanisms for different walls. Each reaction mechanism can have multiple electrochemical reactions.

Electrochemical reactions are not available with the density-based solver.

19.1.4.2. User Inputs for Electrochemical Reactions

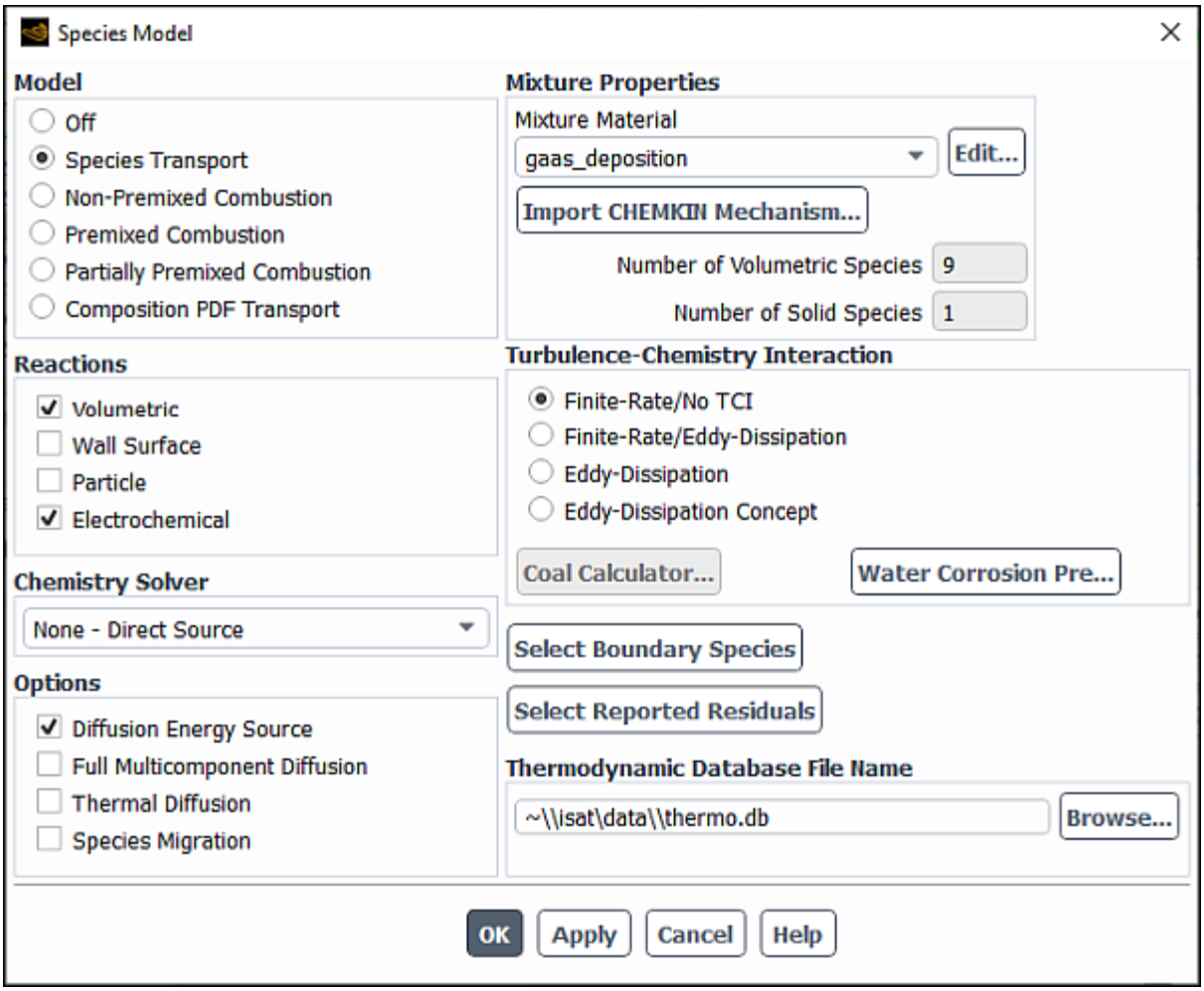
The steps needed to incorporate electrochemical reactions in your problem are as follows:

1. For unsteady simulations, select **Transient** from the **General** task page (**Solver** group).
2. Enable chemical species transport model.

   Setup → **Models** → **Species** **Edit...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 19.21: The Species Model Dialog Box with Electrochemical Reactions Enabled



a. In the **Species Model** dialog box, under **Model**, enable **Species Transport**.

b. Under **Reactions**, select **Volumetric** and **Electrochemical**.

> **Note:**
>
> Once you enable the **Electrochemical** reactions, the electric potential solver will be automatically enabled (under the **Models** tree branch).

3. In the **Create/Edit Materials** dialog box, define all the species materials that are present in the electrolyte and electrodes.

Setup → Materials

a. Copy the new species from the Fluent database to your case as described in Copying Materials from the Ansys Fluent Database (p. 1602). If your species material is not present in the

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

database, copy a similar species material and edit its properties, including **Name** and **Chemical Formula**.

Note:

- When copying species (including corroded and deposited solid species), make sure that **fluid** is selected as **Material Type** in the **Fluent Database Materials** dialog box.
- It is not necessary to represent the electron in the reaction, and e^- can be safely ignored.

b. Under the **Properties** group box, set **Charge Number** for the new species.

Important:

By default, **Charge Number** is set to 0, which means that the species is neutral. Make sure to specify the **Charge Number** for all relevant species.

Note:

Do not specify **Electrical Conductivity** for individual species. You will specify **Electrical Conductivity** for the mixture.

4. Edit the mixture composition using the **Species** dialog box (opened by clicking **Edit** next to **Mixture Species** under **Properties** of the **Create/Edit Materials** dialog box).

   a. Remove unused species and add the electrochemical species to the **Selected Species** list as described in Adding Species to the Mixture (p. 2377).

   b. If solid species are involved in the electrochemical reactions (for example, in corrosion or electroplating simulations), add them to the **Selected Solid Species** list.

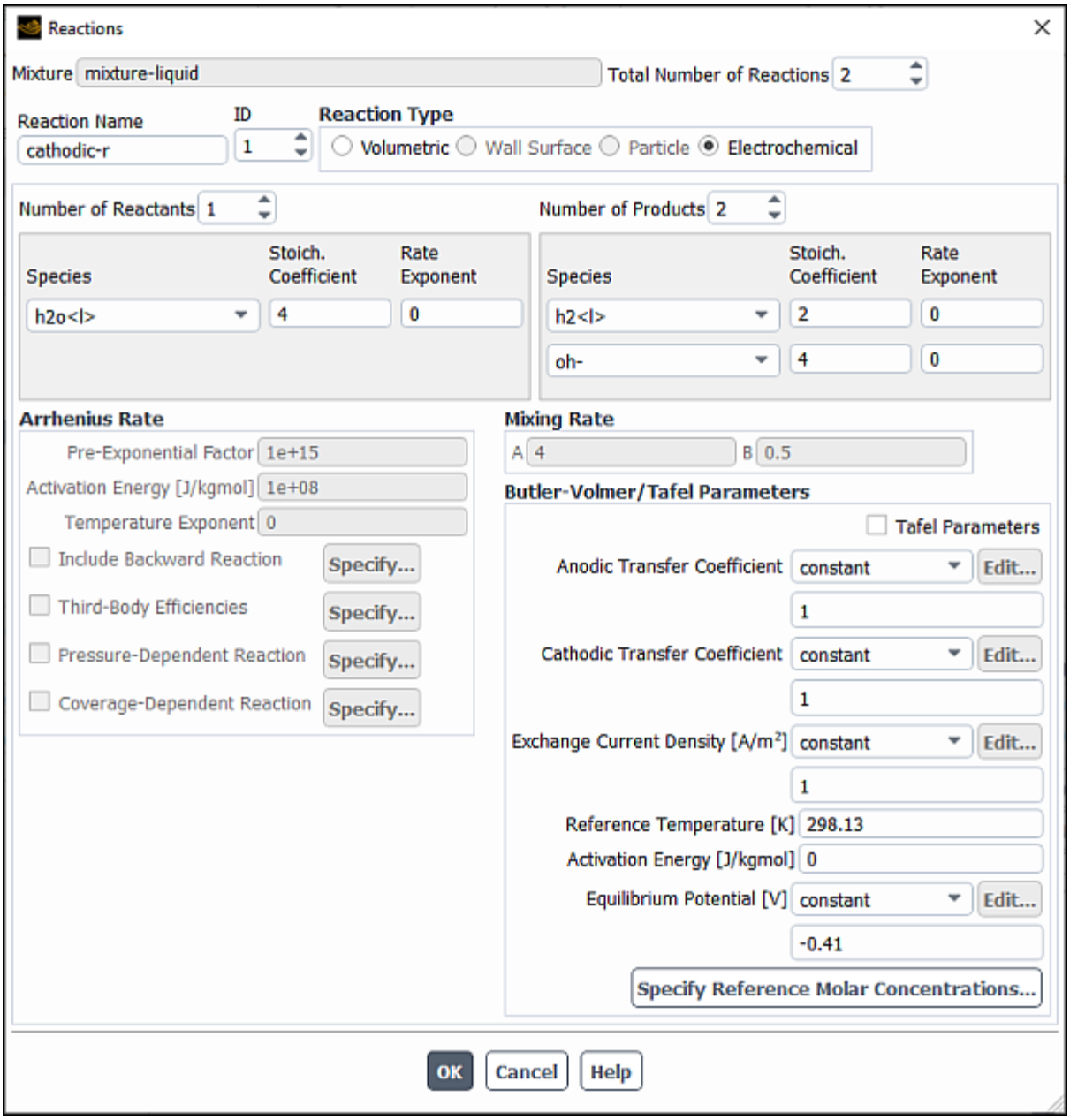
   Note:

   Make sure that the bulk species is the last species in the **Selected Species** list. Ansys Fluent solves transport equations for all but the last species in the **Selected Species** list. The order of the solution is the same as the order of the species in **Selected Species**.

5. Specify the reactions in the **Reactions** dialog box (opened by clicking **Edit** next to **Reactions** under **Properties** of the **Create/Edit Materials** dialog box).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 19.22: The Reactions Dialog Box



For each reaction, configure the following settings:

a. Select **Electrochemical** as the reaction type.

b. If you want the Tafel Equation 7.119 in the *Fluent Theory Guide*, rather than the Bulter-Volmer Equation 7.118, to be used in the kinetics calculation, select **Tafel Parameters**.

c. Specify the following **Butler-Volmer/Tafel Parameters**:

- Butler-Volmer parameters (the **Tafel Parameters** check box is cleared):
  - **Anodic Transfer Coefficient** (α_a in Equation 7.118 in the *Fluent Theory Guide*)
  - **Cathodic Transfer Coefficient** (α_c in Equation 7.118 in the *Fluent Theory Guide*)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Tafel parameters (the **Tafel Parameters** check box is selected):
  - **Anodic Tafel Slope** ($\beta_{a,r}$ in Equation 7.119 in the *Fluent Theory Guide*)
  - **Cathodic Tafel Slope** ($\beta_{c,r}$ in Equation 7.119 in the *Fluent Theory Guide*)
- **Exchange Current Density** ($i_{o,r}$ in Equation 7.118 and Equation 7.119 in the *Fluent Theory Guide*)
- **Reference Temperature** (T_{ref} in Equation 7.118 and Equation 7.119 in the *Fluent Theory Guide*)
- **Activation Energy** (E in Equation 7.118 and Equation 7.119 in the *Fluent Theory Guide*)
- **Equilibrium Potential** ($E_{eq,r}$ in Equation 7.120 in the *Fluent Theory Guide*)

You can define a **constant**, **polynomial**, **piecewise-linear**, or **user-defined** function for any of the aforementioned Butler-Volmer/Tafel parameters.

d. Enter the k^{th} species dimensionless power (γ_k in Equation 7.118 in the *Fluent Theory Guide*) as **Rate Exponent**.

e. Under the **Butler-Volmer/Tafel Parameters** group box, click the **Specify Reference Molar Concentrations...** button and in the **Reference Molar Concentrations** dialog box that opens (see Reference Molar Concentrations Dialog Box (p. 4941)), enter the reference species molar concentrations ($C_{k,r}$ in Equation 7.118 in the *Fluent Theory Guide*).

The reference molar concentrations essentially act as scaling factors in the Butler-Volmer equation. In the literature, when a reaction mechanism is discussed, the authors usually provide values for the reference molar concentration as well as the current density and other kinetics parameters. In Ansys Fluent, the default value is 1 kmol/m^3 for all the reference molar concentrations.

Note:

You can customize the electrochemical reaction rate by using the following UDFs:

- `DEFINE_EC_KINETICS_PARAMETER`: to customize an individual kinetics parameter in the Butler-Volmer reaction rate
- `DEFINE_EC_RATE`: to define a totally different function form for the reaction rate.

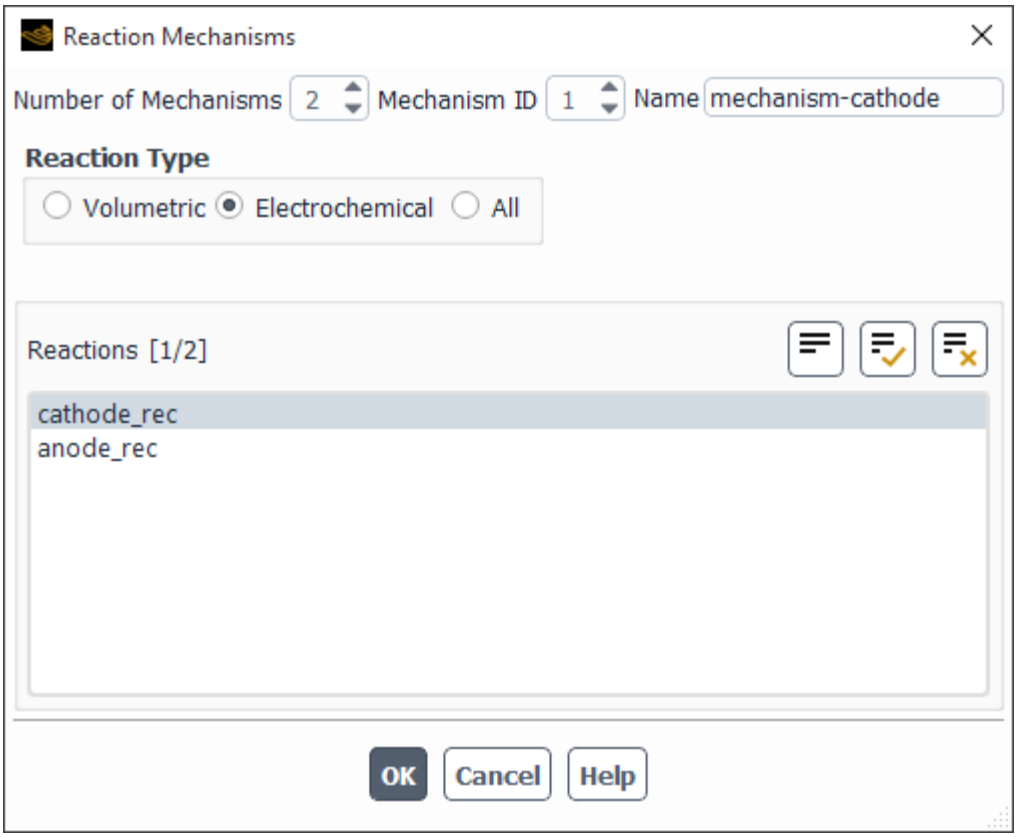
Refer to `DEFINE_EC_KINETICS_PARAMETER` and `DEFINE_EC_RATE` in the *Fluent Customization Manual* for details about those functions.

---

6. Define reaction mechanisms using the **Reaction Mechanisms** dialog box (opened by clicking **Edit** next to **Mechanism** under **Properties** of the **Create/Edit Materials** dialog box).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 19.23: The Reaction Mechanisms Dialog Box



a. Specify **Number of Mechanisms**. If you type in the value, be sure to press **Enter**.

b. For each electrochemical reaction mechanism, configure the following settings:

   i. Set the **Mechanism ID** of the mechanism you want to define. Again, if you type in the value, be sure to press **Enter**.

   ii. Select **Electrochemical** as a reaction type.

   iii. Under **Reactions**, select the reaction and in the **Name** text entry field, specify a meaningful name for the mechanism (for example, **`mechanism-anode`** or **`mechanism-cathode`**).

7. In the **Create/Edit Materials** dialog box, set the remaining mixture properties, such as **Density** (typically volume-weighted-mixing-law), **Mass Diffusivity**, and **Electrical Conductivity**.

   Note that if the anode, cathode, and electrolyte have different material properties, such as species mass diffusivities, it is best to use a UDF. Alternatively, you can create new materials for the anode and cathode by copying existing materials from the Fluent database, renaming them, and then assigning these materials to the anode and cathode fluid zones in the **Cell Zone Conditions** dialog box.

8. For electrolyte and electrode zones, set cell zone conditions using Cell Zone Conditions Task Page (p. 4958).

   Setup → Cell Zone Conditions

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

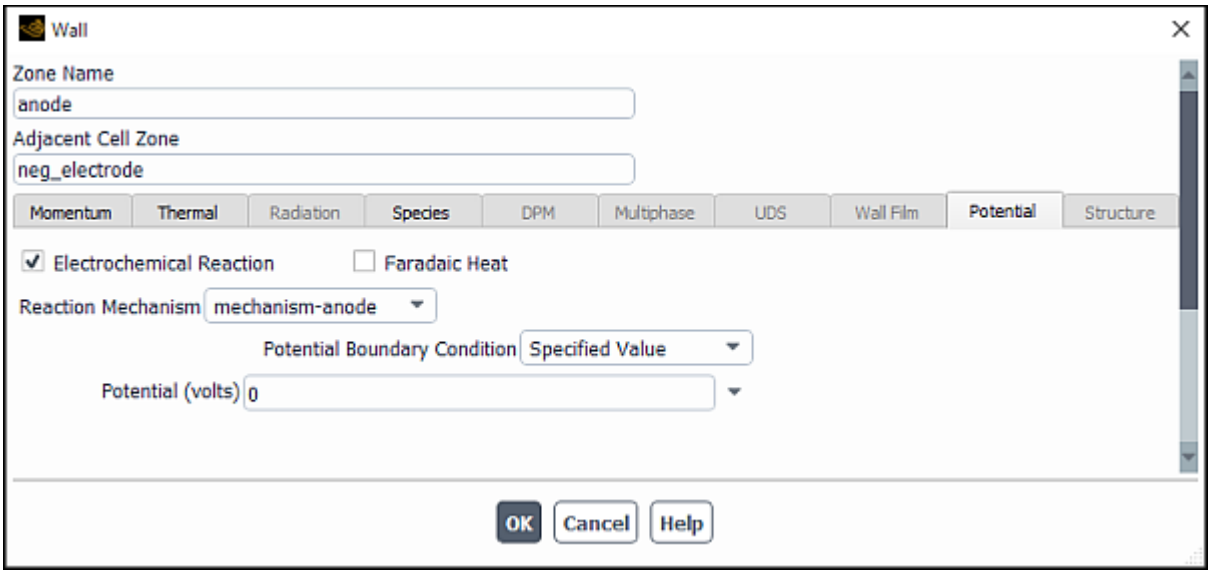
a. In the **Fluid** dialog box, specify whether or not the selected zone is an electrolyte zone by selecting or clearing the **Electrolyte** check box. By default, all fluid zones are electrolyte zones.

b. If there is no volumetric reactions, clear the **Reaction** check box.

9. Set the electrochemical boundary conditions for all relevant walls using Boundary Conditions Task Page (p. 4991).

Setup → Boundary Conditions

Figure 19.24: Wall Potential Boundary Condition



Electrochemical reaction can be defined at either exterior (1-sided) walls or interior (2-sided) walls with only one side exposed to electrolyte.

In the **Wall** dialog box, configure the following settings:

a. For exterior (1-sided) walls, set the potential boundary conditions as described in Setting Up the Electric Potential Model (p. 3266).

b. Specify whether or not the wall is Faradaic by selecting or clearing the **Electrochemical Reaction** check box.

c. For Faradaic walls, from the **Reaction Mechanism** drop-down list, select the reaction mechanism that you have previously defined.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

d. If the electrochemical reaction is defined on the wall (**Electrochemical Reaction** is selected), you can include the reaction heat in the energy equation by selecting the **Faradaic Heat** check box.

Note:

For the 2-sided wall, you need to specify the potential boundary conditions only on one side of the wall. The settings you have specified will be automatically copied to the other side of the wall.

---

10. (optional) You can include additional sources in the energy, continuity, and species transport equations as described in the following sections:

    - Electrochemical Reaction Effects in the Energy Equation (p. 2422)
    - Electrochemical Reaction Effects in the Species Transport Equation (p. 2422)
    - Including Mass Transfer in Continuity (p. 2422).

11. Set the solution parameters in the Solution Methods Task Page (p. 5157) and Solution Controls Task Page (p. 5163).

    In many electrochemical applications, fluid convection can be neglected, in which case the **Flow** equation should be disabled in the **Equations** control box (accessible from the Solution Controls Task Page (p. 5163)).

    Similarly, for isothermal simulations, the **Energy** equation should be disabled.

    Important:

    If the potential equation has difficulty converging, you can lower the **Faradaic Interface Current** under-relaxation factor in the **Solution Controls** task page to improve stability.

    ---

12. Once the solution is converged, you can display or report the following variables that are available under the **Potential** category:

    - **Electric Potential**
    - **Electric Conductivity**
    - **Faradaic Current Density**
    - **Total Surface Corrosion Rate**
    - **Electrode Surface Potential**
    - **Current Density**
    - **Joule Heat Source**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Faradaic Heat Source**
- **Total Echem Heat Source**

In addition, you can display

- **Echem Reaction Rate** of each reactions (under the **Reactions** category)
- **Surface Corrosion Rate** of each solid species on a wall (under **Species** category)

19.1.4.3. Electrochemical Reaction Effects in the Energy Equation

When the electrochemical reaction model is enabled, you can specify two extra sources in the energy equation:

- Joule heating in the fluid and solid zones

  You can include the Joule heating in the energy equation calculation by selecting the **Include Joule Heating in Energy Equation** option in the **Electric Potential** dialog box (Setting Up the Electric Potential Model (p. 3266)).

- Electrochemical reaction heat at Faradaic interfaces

  You can include the electrochemical reaction heat by selecting the **Faradaic Heat** option in the boundary condition dialog boxes for each reacting wall.

19.1.4.4. Electrochemical Reaction Effects in the Species Transport Equation

When the electrochemical reaction model is enabled, you can specify an extra source in the species transport equation for the ion-species:

- Species migration in the electric field

You can include this effect by selecting the **Species Migration** option in the Species Model Dialog Box (p. 4768).

19.1.4.5. Including Mass Transfer in Continuity

When a solid species (corroded or deposited) is involved in an electrochemical reaction, the solid material can enter or leave the flow domain. Usually the mass transfer of the solid species is small and can be ignored in the continuity equation. However, if you want, you can include the mass source in the continuity equation by using the following Scheme command:

```
(rpsetvar 'potential/corrosion-source-in-continuity? #t)
```

19.1.4.6. Solution Procedures for Electrochemical Reactions

The strongly nonlinear coupling between species and electric field can cause convergence problems. To improve the stability of the solution, you can lower the **Faradaic Interface Current** under-relaxation factor in the Solution Controls Task Page (p. 5163).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In addition, you can try temporarily disabling the **Include Joule Heating in Energy Equation** and **Faradaic Heat** options (see Electrochemical Reaction Effects in the Energy Equation (p. 2422)).

You can enable these options once the solution becomes more stable.

19.1.4.7. Modeling Corrosion with the Water Corrosion Pre Tool

19.1.4.7.1. Background

One important electrochemical reaction application is corrosion. In the oil and gas industry, pipelines and processing equipment, which are made of mild steel, may be subject to the CO_2 aqueous environment. As a result, the so called "sweet" CO_2 corrosion may occur in such systems. Depending on the aqueous environment (strong acid or CO_2), different homogenous volumetric reactions and heterogeneous electrochemical reactions can be present in the system.

The heterogeneous electrochemical reactions are the same for all the environments. The anodic reaction is the dissolution of iron in a water solution:

$$Fe \rightarrow Fe^{2+} + 2e^{-} \tag{19.9}$$

The reaction rate is computed from the following Tafel equation:

$$i_{Fe} = i_{0,Fe} \times \exp\left[-\frac{E}{R}\left(\frac{1}{T} - \frac{1}{T_{ref}}\right)\right] \times 10^{\left(\phi_s - \phi_e - E_{eq}\right)/\beta_a} \tag{19.10}$$

with the following parameter values taken from [106] (p. 5719):

$i_{0,Fe}$ = 1 A/m^2

E_{eq} = -0.488 V

β_a = 0.04 V

E = 37.5 kJ/mol

T_{ref} = 298.15 K

Refer to Equation 7.119 in the *Fluent Theory Guide* for the notation used in Equation 19.10 (p. 2423).

The cathodic reaction is a hydrogen evolution reaction via the reduction of H^+ ions:

$$2H^{+}\langle l\rangle + 2e^{-} \rightarrow H_2\langle l\rangle \tag{19.11}$$

The charge transfer current density can be calculated using the Tafel equation as:

$$i_{H^+} = i_{0,H^+} \times \exp\left[-\frac{E}{R}\left(\frac{1}{T} - \frac{1}{T_{ref}}\right)\right] \times \left(\frac{c_{H^+}}{c_{H^+,ref}}\right)^{0.5} \times 10^{-\left(\phi_s - \phi_e - E_{eq}\right)/\beta_c} \tag{19.12}$$

with the following parameter values taken from [106] (p. 5719):

i_{0,H^+} = 0.03 A/m^2

E_{eq} = -0.24 V

β_c = 0.12 V

E = 30 kJ/mol

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

T_{ref} = 298.15 K

$c_{H^+,ref}$ = 1.0e^{-4} M

The homogeneous reactions are different depending on the environment:

- Strong acid environment

  In such environments, the acid is practically fully dissociated in water, and there is no need to solve any homogeneous reactions. The species involved in such systems include H+<l>, OH-<l>, Fe2+<l>, H2<l>, H2O<l>, and Fe<b>.

- CO2 environment

  When CO2 gas is dissolved in water, a series of chemical reactions occurs, and that may make the resultant solution corrosive to mild steel. The reactions in this environment are:

  – CO2 hydration:

  $$CO_2\langle l\rangle + H_2O\langle l\rangle \overset{k_f,k_b}{\Longleftrightarrow} H_2CO_3\langle l\rangle$$

  – Carbonic acid dissociation:

  $$H_2CO_3\langle l\rangle \overset{k_f,k_b}{\Longleftrightarrow} H^+\langle l\rangle + HCO_3^-\langle l\rangle$$

  – Bicarbonate ion dissociation:

  $$HCO_3^-\langle l\rangle \overset{k_f,k_b}{\Longleftrightarrow} H^+\langle l\rangle + CO_3^{2-}\langle l\rangle$$

  – Water dissociation:

  $$H_2O\langle l\rangle \overset{k_f,k_b}{\Longleftrightarrow} H^+\langle l\rangle + OH^-\langle l\rangle$$

  Each reaction is reversible. The reaction kinetics parameters for the forward and backward reactions are taken from [106] (p. 5719)

  The species involved in such systems include H+<l>, OH-<l>, CO2<l>, H2CO3<l>, HCO3-<l>, CO32-<l>, Fe2+<l>, H2<l>, H2O<l>, and Fe<b>.

19.1.4.7.2. Procedure for Setting Corrosion Simulations using the Water Corrosion Pre Tool

The Water Corrosion Pre tool is designed to assist you in setting a corrosion case quickly in Ansys Fluent. Using this tool, you can conduct water chemistry calculation and create corresponding material and reactions for the environment.

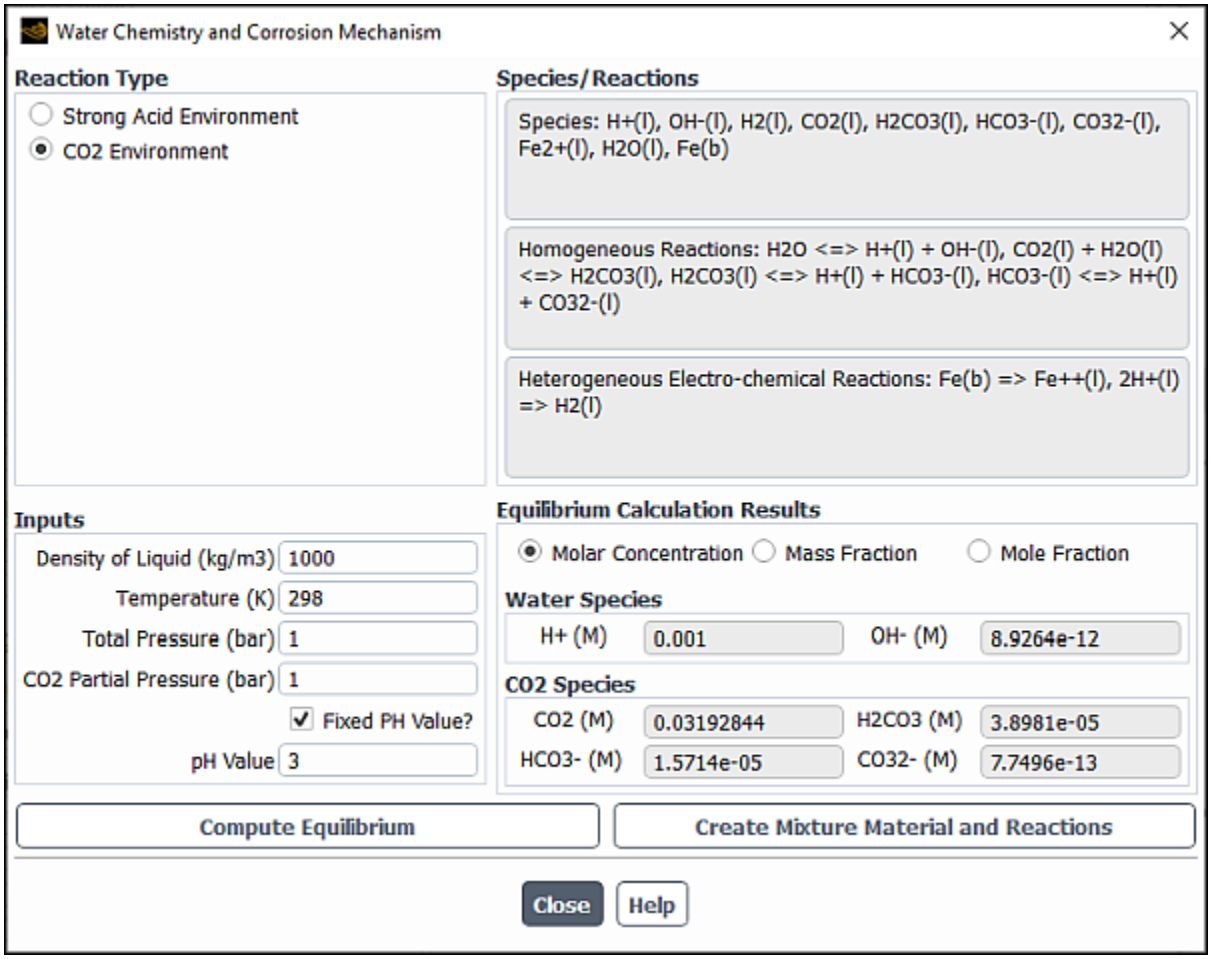
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following procedure provides an overview of the steps required for setting up a corrosion case. Only the steps that are pertinent to corrosion simulations are shown here. For information about inputs related to other models that you want to use in your case, see the appropriate sections for those models in the Ansys Fluent User's Guide (p. 1)

1. In the **Species Model** dialog box, enable the **Species Transport** model, and in the **Reactions** group box, enable the following reactions:
   - **Volumetric**
   - **Wall Surface**
   - **Electrochemical**

2. Click the **Water Corrosion Pre...** button.

   Figure 19.25: The Water Chemistry and Corrosion Mechanism Dialog Box



3. In the **Water Chemistry and Corrosion Mechanism** dialog box that opens (see Figure 19.25: The Water Chemistry and Corrosion Mechanism Dialog Box (p. 2425)), select the **Reaction Type** from the following options:

   - **Strong Acid Environment**

   - **CO2 Environment**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The species and reactions involved in the water chemistry for the selected reaction type are shown in the **Species/Reactions** group box. For additional information, see Background (p. 2423).

4. Specify **Inputs** for the selected reaction type.

   For the **CO2 Environment** reaction type, you need to specify the following parameters:

   Density of Liquid

   is the water solution density.

   Temperature

   is the solution temperature at which the equilibrium calculation is carried out.

   Total Pressure

   is the pressure used to calculate some of the water equilibrium constants.

   CO2 Partial Pressure

   is the partial pressure of CO_2 in the gas phase, which is used to calculate the aqueous phase (CO_2) concentration in the CO_2 dissolution reaction (CO_2<g> <=> CO_2<l>).

   Fixed pH Value?

   when enabled, allows you to specify the **pH Value** of the solution.

   pH Value

   is the pH value of the solution. This item is available only when **Fixed pH Value?** is enabled.

   For the **Strong Acid Environment** reaction type, you need to specify only the following parameters:

   - **Density of Liquid**
   - **pH Value**.

5. Click the **Compute Equilibrium** button to initiate the water chemistry calculation.

   The species concentration from the equilibrium calculation are printed to the Ansys Fluent console. They are also displayed in the **Equilibrium Calculation Results** group box of the **Water Chemistry and Corrosion Mechanism** dialog box. You can select to display the results in the following units:

   - **Molar Concentration**
   - **Mass Fraction**
   - **Mole Fraction**

   The equilibrium calculation results are automatically passed to the corresponding boundary condition dialog boxes and **Solution Initialization** task page where the relevant fields are updated.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Click the **Create Mixture Material and Reactions** button to create a mixture material, reactions, and chemistry mechanisms for the selected reaction type.

   Depending on the environment, one of the following materials is created:

   - mixture-strong-acid for **Strong Acid Environment**
   - mixture-co2-corrosion for **CO2 Environment**

   For each mixture, all the species involved, reactions, and mechanisms are pre-defined. For the both environments, the following three electrochemical reaction mechanisms are created:

   - **mechanism-anodic**: Includes only the anodic reaction
   - **mechanism-cathodic**: Includes nly the cathodic reaction
   - **mechanism-combined-echem**: Includes both the anodic and cathodic reactions

   For the CO2 environment, where some homogeneous reactions are present, a volumetric reaction mechanism, **mechanism-volumetric**, is also created. This mechanism can be assigned to a fluid zone.

   In a galvanic corrosion, where anodic and cathodic reactions occur at different surfaces, the **mechanism-anodic** and **mechanism-cathodic** mechanisms can be assigned to the corresponding surfaces. In a uniform corrosion case, where the anodic and cathodic reactions occur at the same surface, the **mechanism-combined-echem** mechanism can be used.

   In addition, Ansys Fluent automatically applies the following changes:

   - Sets the newly created material as the mixture material in the **Species Model** dialog box.
   - Assigns the newly created material to the fluid zone(s).
   - Updates the species mass fractions in the **Species** tab of the relevant boundary conditions dialog boxes.

7. Specify boundary conditions using the equilibrium calculation results.

   a. Set boundary conditions for all species at all inlets

   b. Hook the electrochemical reaction mechanism to all walls where corrosion occurs.

8. In the **Fluid** dialog box, specify cell zone conditions for the fluid zones. For the **CO2 Environment** reaction type, hook the **mechanism-volumetric** mechanism to all fluid zones where CO2 environment corrosion occurs.

9. Initialize the species field.

10. Run the simulation.

19.1.5. Species Transport Without Reactions

In addition to the volumetric and surface reactions described in the previous sections, you can also use Ansys Fluent to solve a species mixing problem without reactions. The species transport equations

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

that Ansys Fluent will solve are described in Volumetric Reactions in the Theory Guide, and the procedure you will follow to set up the non-reacting species transport problem is the same as that described in Overview of User Inputs for Modeling Species Transport and Reactions (p. 2356) – Defining Other Sources of Chemical Species (p. 2395), with some simplifications.

The basic steps are listed below:

1. Enable **Species Transport** in the **Species Model** dialog box and select the appropriate **Mixture Material**.

   Setup → **Models** → **Species** **Edit...**

   See Overview of User Inputs for Modeling Species Transport and Reactions (p. 2356) for information about the mixture material concept, and Enabling Species Transport and Reactions and Choosing the Mixture Material (p. 2359) for more details about using the **Species Model** dialog box.

2. (optional) If you want to model full multicomponent (Stefan-Maxwell) diffusion or thermal (Soret) diffusion, enable the **Full Multicomponent Diffusion** or **Thermal Diffusion** option.

3. Check and/or define the properties of the mixture and its constituent species.

   Setup → **Materials**

   Mixture properties include the following:

   - species in the mixture
   - other physical properties (for example, viscosity, specific heat)

   See Defining Properties for the Mixture and Its Constituent Species (p. 2374) for details.

4. Set species boundary conditions, as described in Defining Cell Zone and Boundary Conditions for Species (p. 2394).

No special solution procedures are usually required for a non-reacting species transport calculation. Upon completion of the calculation, you can display or report the following quantities:

- **Mass fraction of species-n**
- **Mole fraction of species-n**
- **Concentration of species-n**
- **Lam Diff Coef of species-n**
- **Eff Diff Coef of species-n**
- **Enthalpy of species-n** (pressure-based solver calculations only)
- **Relative Humidity** (when the water vapors (H_2O) is present in the species mixture material)
- **Mean Molecular Weight**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Liquid species mass fraction of species-n** (solidification and melting model only)

These variables are contained in the **Species...** and **Properties...** categories of the variable selection drop-down list that appears in postprocessing dialog boxes. See Field Function Definitions (p. 4167) for a complete list of flow variables, field functions, and their definitions. Displaying Graphics (p. 3869) and Reporting Alphanumeric Data (p. 4097) explain how to generate graphics displays and reports of data.

19.1.6. Reacting Channel Model

Information about using the reacting channel model is presented in the following subsections. For more information about the theoretical background, see Reacting Channel Model in the Theory Guide.

19.1.6.1. Overview and Limitations of the Reacting Channel Model

19.1.6.2. Enabling the Reacting Channel Model

19.1.6.3. Boundary Conditions for Channel Walls

19.1.6.4. Postprocessing for Reacting Channel Model Calculations

19.1.6.1. Overview and Limitations of the Reacting Channel Model

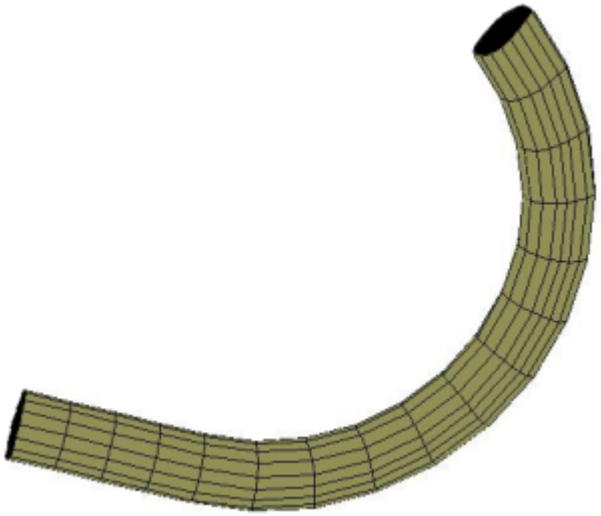
The reacting channel model in Ansys Fluent offers an efficient solution methodology for solving reacting flow in shell and tube heat exchangers with long and thin channels. The volume inside the reacting channels is not meshed and solved with the outer flow. Instead, a plug flow approximation is used for the reacting channels, which is coupled to the outer flow through heat transfer across the channel walls.

The reacting channel model has the following limitations:

- The reacting channel model is available only for steady-state 3D problems.
- Shell conduction cannot be used on reacting channel walls.
- The reacting channel model solves the one-dimensional species equations at the centerline of the reacting channel. The accuracy of the model depends on how well Ansys Fluent computes the channel centerline from the channel wall surface mesh. It is highly recommended to use a swept structured surface mesh on the channel wall (see Figure 19.26: Optimal Surface Mesh on the Reacting Channel Wall (p. 2430)). Unstructured channel wall surface meshes may result in loss of accuracy due to inconsistent channel centerlines.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 19.26: Optimal Surface Mesh on the Reacting Channel Wall



19.1.6.2. Enabling the Reacting Channel Model

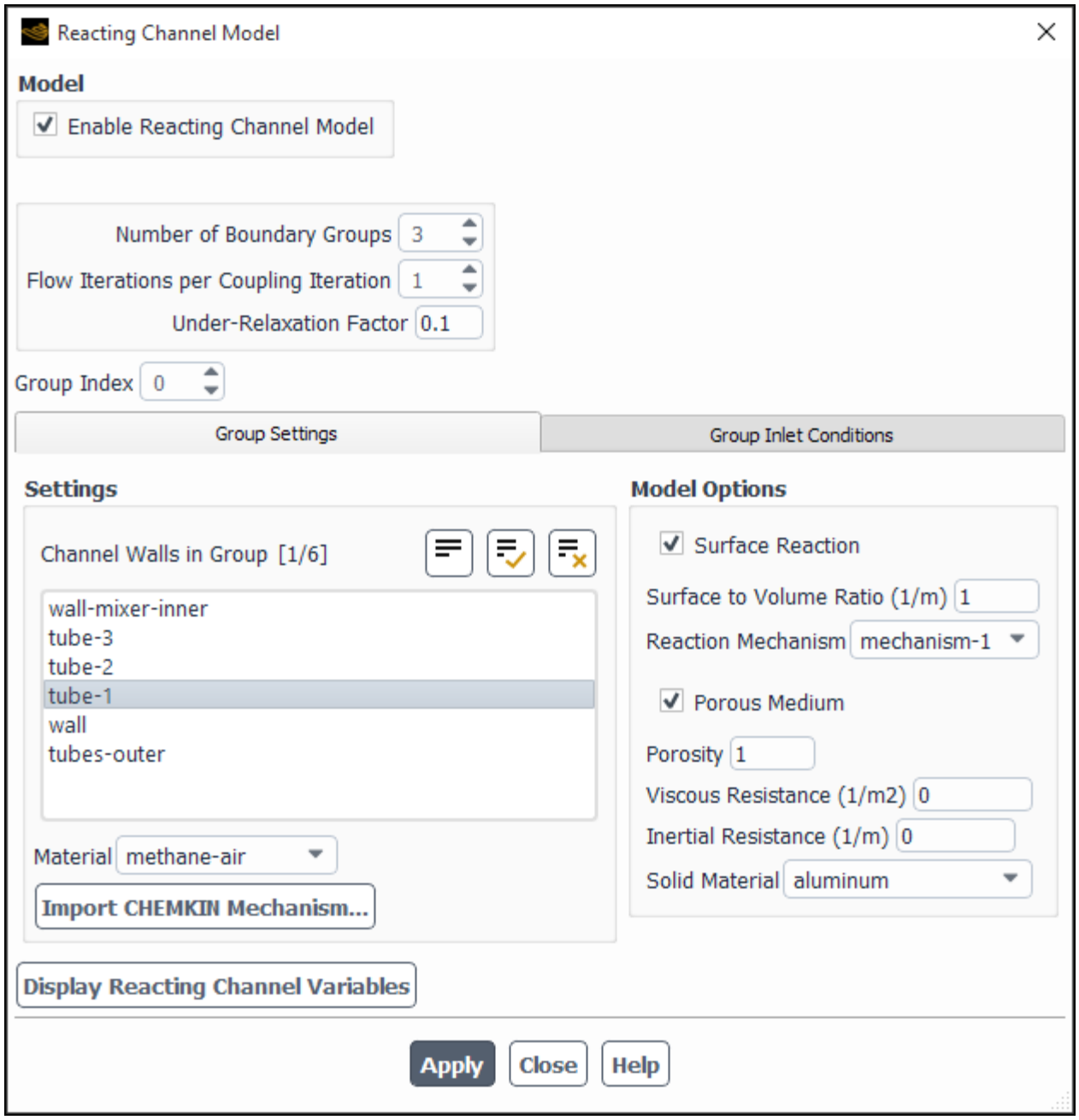
The steps and procedure to set up a reacting channel model are outlined below. In this section, the steps pertinent to the reacting channel model only are explained. The setting up of other models used in conjunction with reacting channel models are explained in other sections of the User's Guide.

1. Enable the reacting channel model.

   Setup → Models → Species → Reacting Channel Model → Edit...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 19.27: The Reacting Channel Model Dialog Box



When you enable the **Enable Reacting Channel Model** option, the dialog box will expand to show the relevant inputs (see Figure 19.27: The Reacting Channel Model Dialog Box (p. 2431)).

2. Specify the **Number of Boundary Groups**. If you have multiple channels, you can group together channels with common flow direction, mixture materials, inlet compositions, temperature,

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

pressure, and mass flow rate. You will be required to provide the inputs for each boundary group.

Note:

If you have multiple tubes, you will need to define each tube as a different wall and set up the model by grouping the walls together if they have the same boundary conditions.

3. Enter the number of **Flow Iterations Per Coupling Iteration**. This is the number of outer flow iterations for each channel flow iteration.

4. Optionally change an **Under-Relaxation Factor**. The value of the under-relaxation factor is used to update the heat flux from the reacting channel. See Equation 7.145 in the Theory Guide.

5. Set the **Group Index**. Note that the **Group Index** for the first boundary group is set to `0`.

   The inputs for each boundary group are entered in the **Group Settings** and **Group Inlet Conditions** tabs as further described.

6. Under the **Group Settings** tab, specify the model settings and options. (see Figure 19.28: The Reacting Channel Model Dialog Box (Group Inlet Conditions Tab) (p. 2433)).

   a. In the **Channel Walls in Group** selection list, select the wall boundary zones that correspond to the reacting channels.

   b. Select the group material from the **Material** drop-down list. You can choose any mixture material available in the case as a group material.

   c. Optionally, import a CHEMKIN mechanism using **Import CHEMKIN Mechanism...** button. For details see Importing a Volumetric Kinetic Mechanism in CHEMKIN Format (p. 2368).

   d. To model surface reactions within the channel, enable **Surface Reactions** under **Model Options** group box. The dialog box will expand to show the related inputs as shown in Figure 19.27: The Reacting Channel Model Dialog Box (p. 2431). These inputs include:

      - **Surface to Volume Ratio** of the channel group
      - Surface reaction mechanism of the group (selected from the **Reaction Mechanism** drop-down list)

   e. To model porous medium, enable **Porous Medium** under **Model Options** group box. The dialog box will expand to show porous medium inputs as shown in Figure 19.27: The Reacting Channel Model Dialog Box (p. 2431). These inputs include:

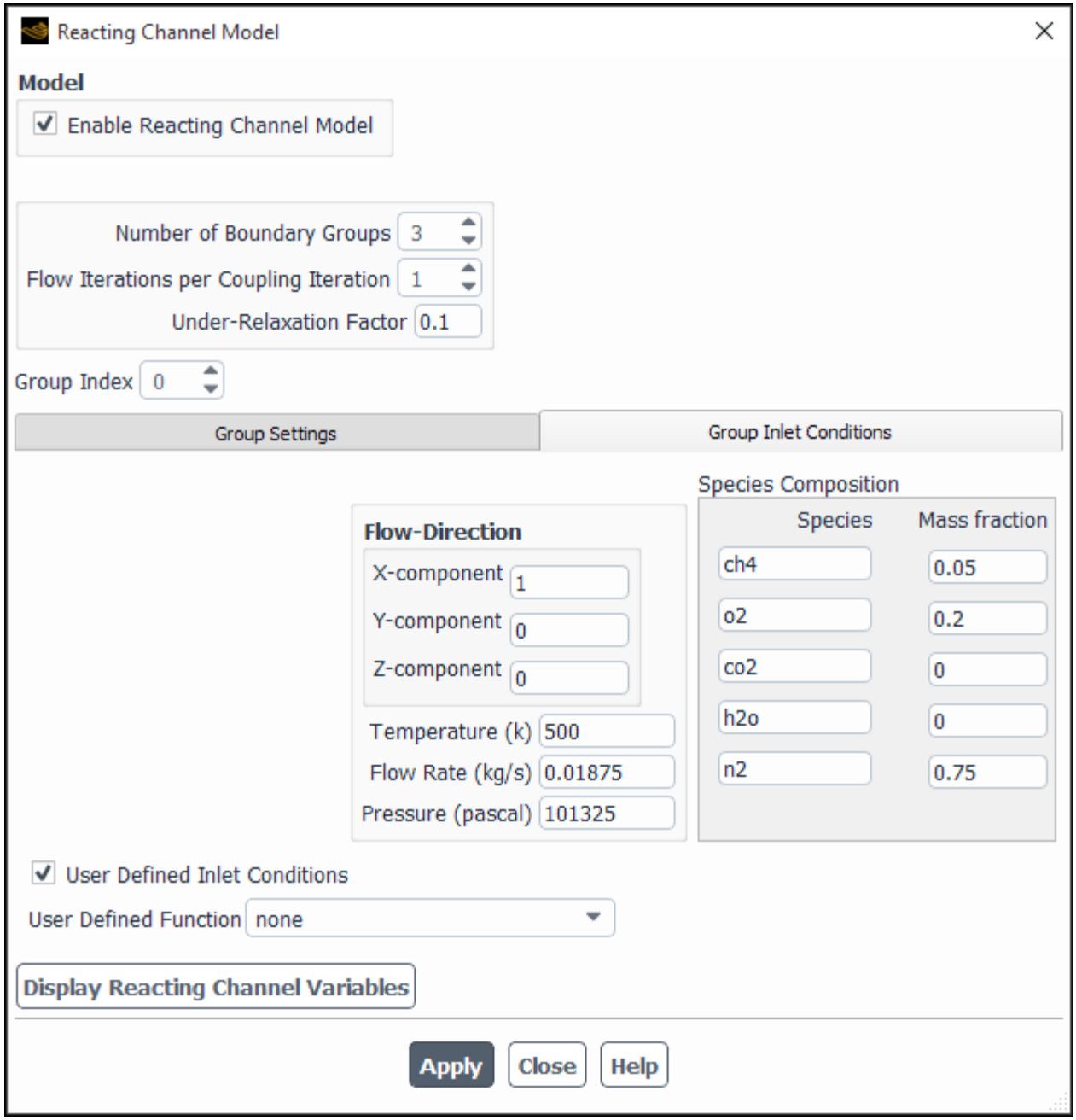
      - **Porosity** of the channel
      - **Viscous Resistance** in axial direction of the channel
      - **Inertial Resistance** in axial direction of the channel

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Solid material of the porous medium inside the channel (selected from the **Solid Material** drop-down list)

7. Under the **Group Inlet Conditions** tab, specify the inlet conditions for the boundary group (see Figure 19.28: The Reacting Channel Model Dialog Box (Group Inlet Conditions Tab) (p. 2433)). You can specify the inlet condition parameters as constant values. Ansys Fluent provides you also with the option of redefining any or all of the inlet condition parameters using user-defined functions (UDFs).

Figure 19.28: The Reacting Channel Model Dialog Box (Group Inlet Conditions Tab)



a. Specify the **X-**, **Y-**, and **Z-component** of the flow at the inlets of the channels in the current group in the **Flow Direction** group box.

> **Note:**
>
> Since there is no surface or zone to identify the channel inlet or outlet, the inlet flow direction of the group is used to determine the channel inlet. Therefore, It is very important to specify the flow direction correctly.

b. Enter the inlet channel **Temperature**, **Flow Rate** and **Pressure** in the group.

c. Specify the inlet mass fractions for each species in the group in the **Species Composition** group box.

d. To specify an inlet condition using a user-defined function (UDF), select the **User Defined Inlet Conditions** check box and then select the appropriate user-defined function from the drop-down list. For information about specifying and hooking DEFINE_REACTING_CHANNEL_BC UDFs, see `DEFINE_REACTING_CHANNEL_BC` in the *Fluent Customization Manual*.

8. Click the **Apply** button to save the settings.

9. If you specified more than one boundary group in step 1, change the group ID value in **Group Index** integer number entry box and repeat steps 5–7 for each remaining boundary group.

Once the reacting channel model is defined, you can generate reports or plots of reacting channel variables using the **Reacting channel 2D Curves** dialog box (Figure 19.30: Reacting Channel 2D Curves Dialog Box (Plot) (p. 2436)). You can open the **Reacting channel 2D Curves** dialog box by clicking the **Display Reacting Channel Variables** button in the **Reacting Channel Model** dialog box. For information about the plotting of reacting channel model variables, see Postprocessing for Reacting Channel Model Calculations (p. 2435).

Modeling Curvilinear Reacting Channels

In the reacting channel model, the two ends of a channel are identified as the inlet and the outlet based on the inputs provided for the inlet flow direction of the group. However, for some special channel configurations, such as U-tubes, it is not possible to identify the inlet/outlet of the channel based on the information of the inlet flow direction. For such cases, an additional input is required to differentiate between the end points of the channel as inlet or outlet. The additional input is to provide the centroid coordinates of the inlet of the U shaped channel. Therefore, for U-tube configurations, after setting up the reacting channel model following the earlier steps in this section, it is required to use the following text user interface commands to set the inlet and outlets correctly:

```
/define/models/species> reacting-channel-model-options
Are any of the channels a U-tube configuration? [no] yes
Is wall tube1 a U-tube configuration? [no] yes
Enter the coordinates of the center of the inlet for wall tube1
X coordinate [0] 0.01
Y coordinate [0] 0
Z coordinate [0] 0
```

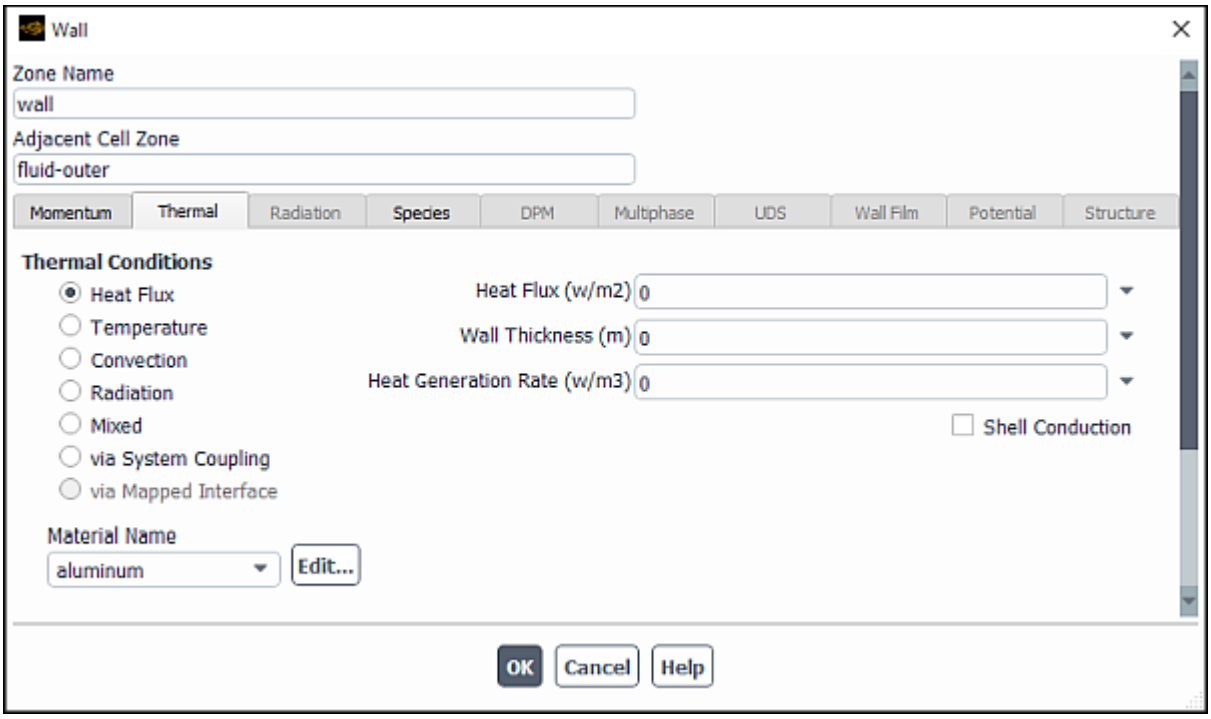
Where `tube1` is a reacting channel wall having a U shape. You will be asked the same questions for each of the existing reacting channel walls.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

19.1.6.3. Boundary Conditions for Channel Walls

In the Ansys Fluent wall boundary conditions dialog box, the wall surfaces that you selected as channel walls in the **Reacting Channel Model** dialog box will have zero heat flux boundary conditions, which are disabled as shown in Wall Surface Reactions and Chemical Vapor Deposition (p. 2400)).

Figure 19.29: The Wall Boundary Condition Dialog Box for the Reacting Channel Model



The heat flux at the channel walls is calculated internally through the coupling of the reacting channel model with the outer flow as described in Reacting Channel Model in the Theory Guide. For all other wall surfaces, which are not channel walls, all boundary conditions as described in Wall Boundary Conditions (p. 1457) are applicable.

19.1.6.4. Postprocessing for Reacting Channel Model Calculations

Ansys Fluent provides postprocessing options for plotting and reporting channel variables using the **Reacting Channel 2D Curves** dialog box that opens by clicking the **Display Reacting Channel Variables** button in the **Reacting Channel Model** dialog box. You can generate X-Y plots or reports of the following reacting channel variables:

- Bulk mean temperature of the channel
- Wall temperature of the channel
- Nusselt Number (for the channel wall on the channel side)
- Wall heat flux through the channel walls
- Velocity inside the channel
- Density of the channel

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

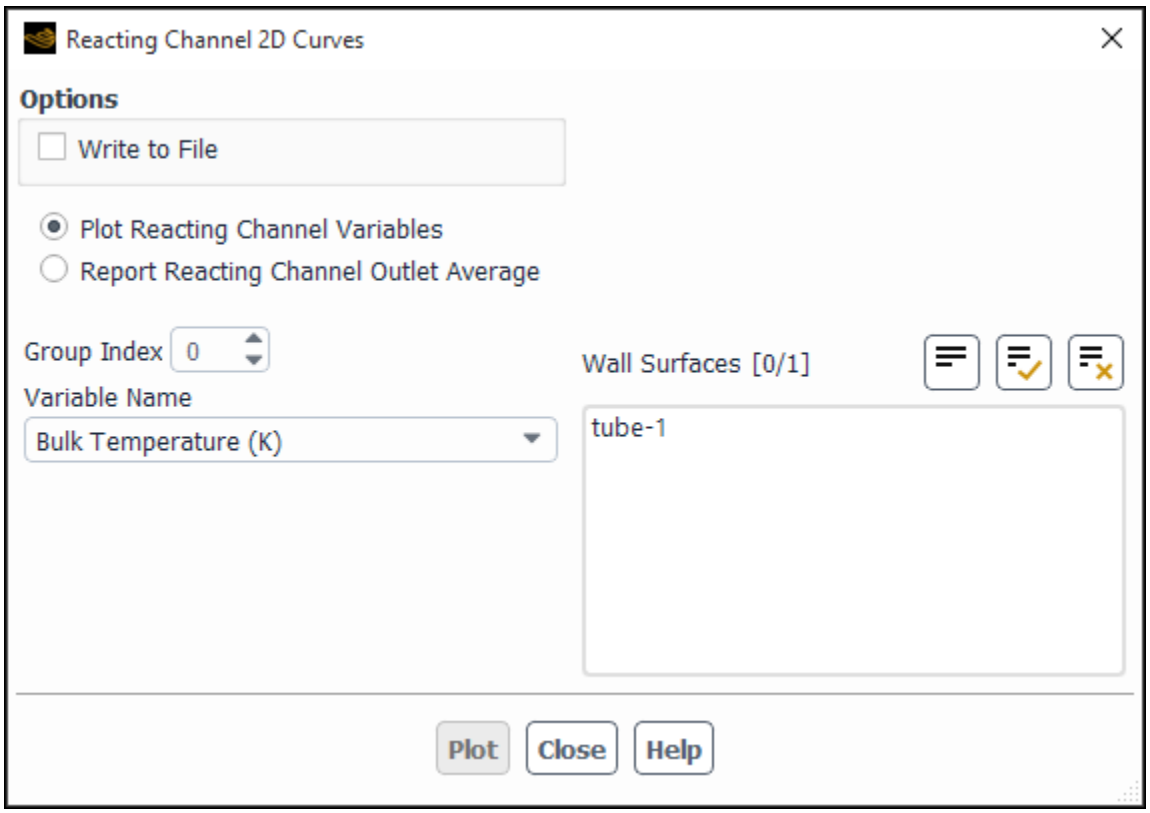
- Mass fraction of the species inside the channel
- The surface coverage and the deposition rate of the surface species.

The steps for generating the plots/reports are as follows:

1. Select either **Plot Reacting Channel Variables** or **Report Reacting Channel Outlet Average**.

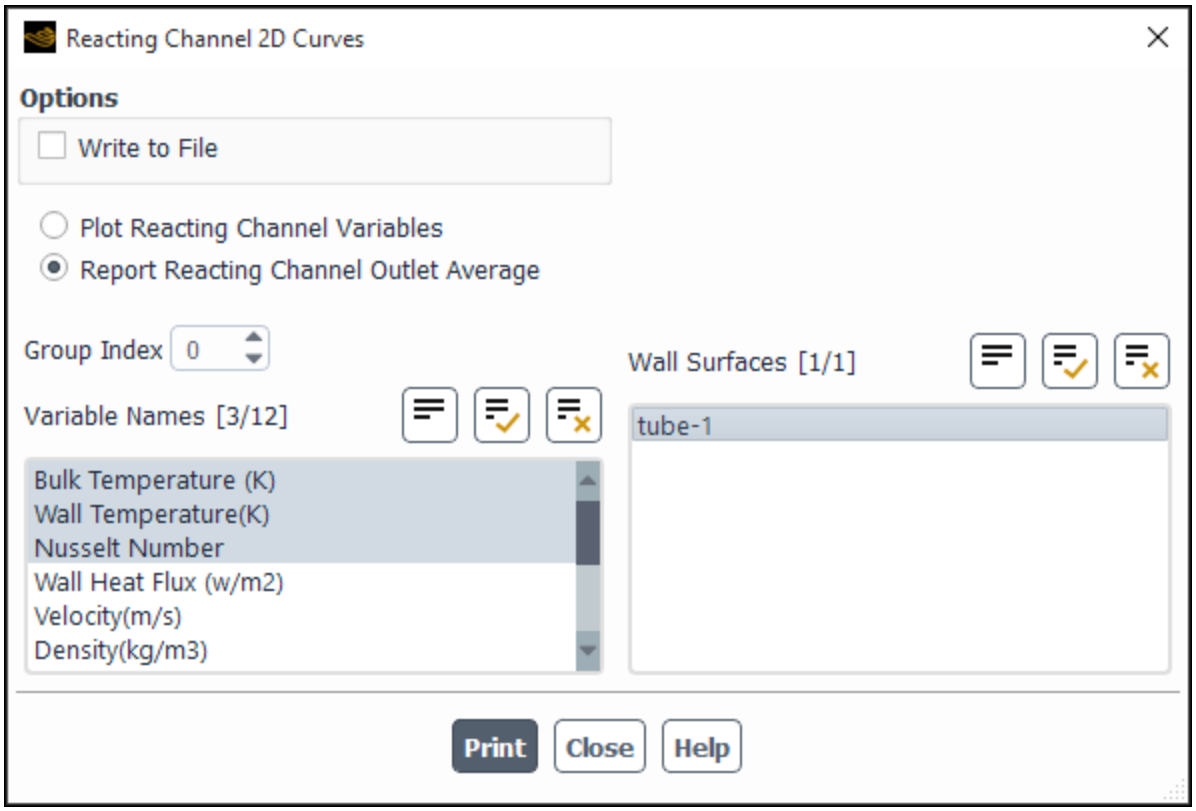
   The **Reacting Channel 2D Curves** dialog box displays either the **Variable Name** drop-down list for selecting a single species for X-Y plot generation, or the **Variable Names** selection list for selecting multiple species for report generation as shown in Figure 19.30: Reacting Channel 2D Curves Dialog Box (Plot) (p. 2436) and Figure 19.31: Reacting Channel 2D Curves Dialog Box (Report) (p. 2437).

   Figure 19.30: Reacting Channel 2D Curves Dialog Box (Plot)



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 19.31: Reacting Channel 2D Curves Dialog Box (Report)



2. Specify whether you want to write the plot data to a file by using the **Write To File** check box.
3. If more than one boundary group is defined for reacting channel model, specify the group ID in the **Group Index** integer number entry box.
4. The **Variable Name** drop-down/**Variable Names** selection list contains reacting channel variables of the specified boundary group. Select the reacting channel variable(s) from the list.
5. Select the wall surface(s) from the **Wall Surfaces** selection list on which the plot/report will be generated (at least one wall surface must be selected).
6. Click one of the following as applicable:
   - **Plot** to view X-Y plot in the graphics window
   - **Print** to view report in the console window
   - **Write...** to save to file

19.1.7. Reactor Network Model

Information about using the Reactor Network model is presented in the following subsections.

19.1.7.1. Overview and Limitations of the Reactor Network Model
19.1.7.2. Solving Reactor Networks

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

19.1.7.3. Postprocessing Reactor Network Calculations

For more information about the theoretical background, see Reactor Network Model in the *Fluent Theory Guide*.

19.1.7.1. Overview and Limitations of the Reactor Network Model

The Reactor Network model allows rapid solution of combustors using detailed chemical kinetic mechanisms. This speed is achieved by first performing a Fluent CFD simulation with a fast chemistry model, such as a Non-Premixed, Partially-Premixed, or Eddy-Dissipation model, and then using the Reactor Network model to solve with a detailed kinetic mechanism. The Reactor Network model agglomerates cells from the CFD solution into a user-specified small number of reactors and computes detailed chemistry on this reactor network.

Since the chemical kinetics is decoupled from the flow, reactor network simulations are applicable to steady simulations where the chemical kinetics does not affect the flow significantly, such as pollutant formation. Inherently unsteady simulations, such as ignition and global extinction, cannot be modeled with reactor networks. Note that the Reactor Network model can be used for unsteady simulations of statistically steady flows, such as LES, where the time-averaged flow and species fields are used to construct the reactor network.

19.1.7.2. Solving Reactor Networks

The steps and procedure for setting up and solving a Reactor Network model are outlined below.

1. Enable the Reactor Network model.

   Setup → **Models** → **Species** → **Reactor Network** **Edit...**

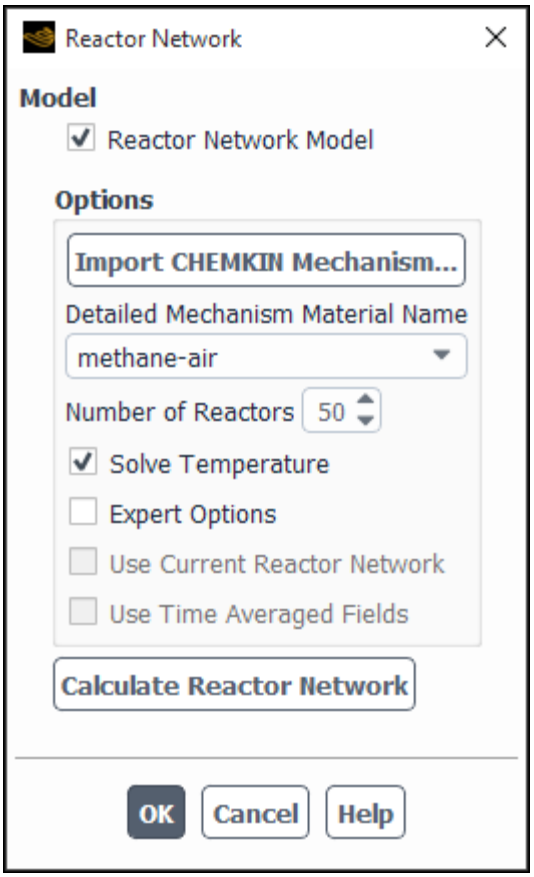
   Important:

   Note that this model is only available when one of the reacting species models is enabled.

   When you select the **Reactor Network Model** option, the dialog box expands to show the relevant modeling controls (see Figure 19.32: Reactor Network Dialog Box (Steady-State Flow) (p. 2439)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 19.32: Reactor Network Dialog Box (Steady-State Flow)



2. If a detailed chemical mechanism has already been imported, select the mixture material from the **Detailed Mechanism Material Name** drop-down list. Otherwise import a mechanism in CHEMKIN format by clicking **Import CHEMKIN Mechanism...**. The mechanism files are imported into your case using the **Import CHEMKIN Format Mechanism** dialog box as described in Specifying a Chemical Mechanism File for Flamelet Generation (p. 2482).

3. Set the **Number of Reactors** in the reactor network. The default value of 50 is a good starting value, but it can be increased for greater accuracy.

4. Specify whether or not the temperature is calculated from the equation of state by selecting or clearing the **Solve Temperature** check box. By default, this option is selected, and Ansys Fluent will determine the temperatures in the reactor network.

5. Once cells are agglomerated and a reactor-network solution is calculated, you can select the **Use Current Reactor Network** option to perform further reactor network simulations with the existing network of connected reactors. This option can be used to continue iterations from the previous reactor- network solution if additional convergence is desired. The option also allows you to change the chemical mechanism or species boundary conditions to generate a new solution on the current reactor network.

6. (for transient analysis with the **Data Sampling For Time Statistics** option selected in the **Run Calculation** task page only) Specify whether or not you want to use the time-averaged composition fields to agglomerate the reactors and time-averaged velocity fields to calculate the reactor

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

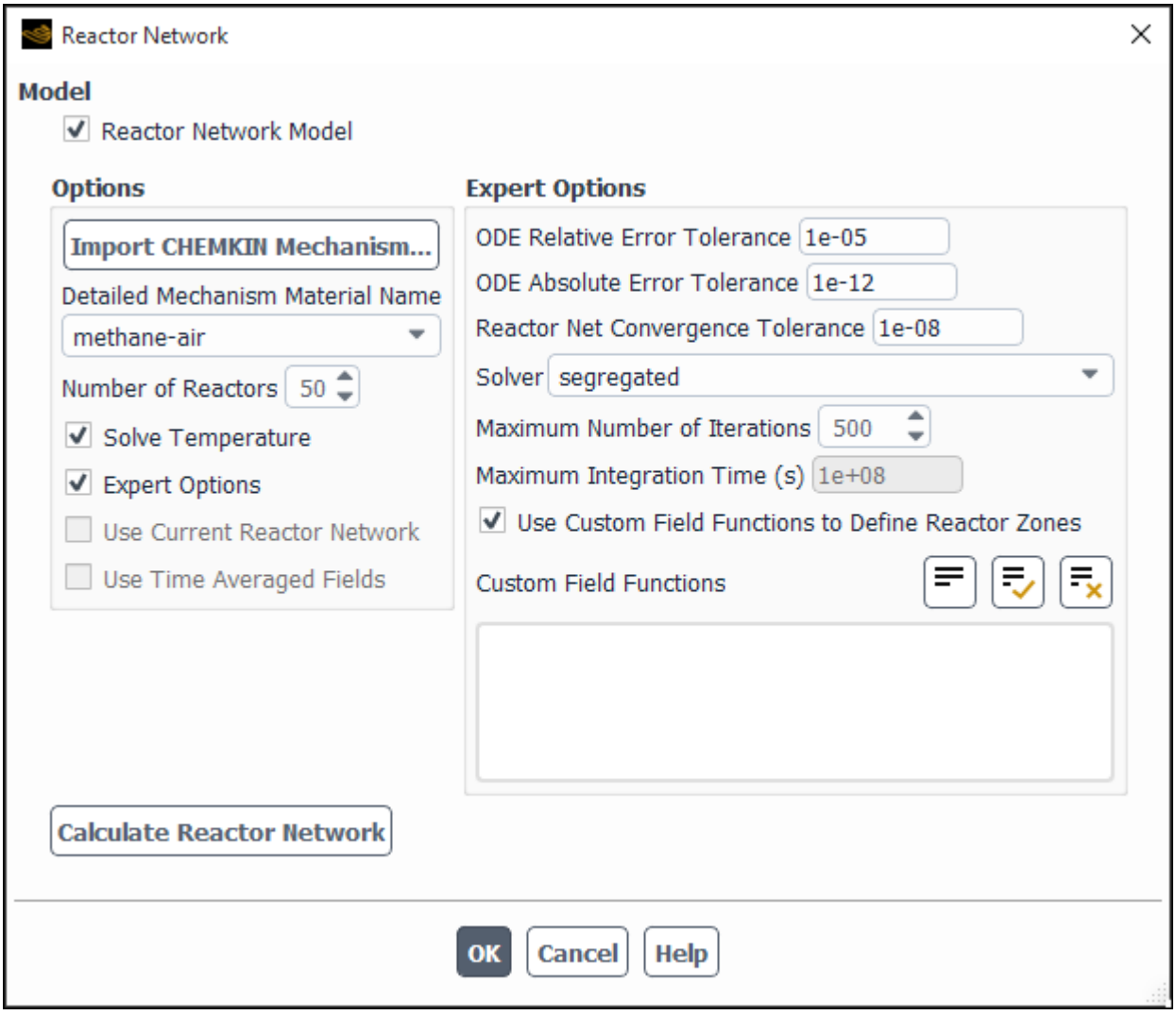
mass flux matrix (instead of the instantaneous fields) by selecting or clearing the **Use Time Averaged Fields** option.

Note:

The **Use Time Averaged Fields** option should be used for unsteady simulations (for example, LES) of statistically stationary combustors.

7. If necessary, specify custom-field functions and adjust the default solver settings using the **Expert Options** control. When **Expert Options** is selected, the **Reactor Network** dialog box expands to reveal the following advanced options:

Figure 19.33: Reactor Network Dialog Box - Expert Options



ODE Relative Error Tolerance, ODE Absolute Error Tolerance

The default values for the ODE relative and absolute error tolerances are 1e-05 and 1e-12, respectively. If the ODE solver fails to converge, it is often helpful to lower the **ODE Relative Error Tolerance** and the **ODE Absolute Error Tolerance**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reactor Network Convergence Tolerance

The default value for the reactor network convergence tolerance is 1e-08. Residuals should be well converged at the default tolerance. Higher tolerances may be acceptable for engineering accuracy.

Solver

The default segregated solver typically converges faster than the coupled solver. However, when using the segregated solver, residuals may stall above an acceptable tolerance. In this case, the coupled solver should be used.

Maximum Number of Iterations (segregated solver)

If residuals are still decreasing and better convergence is desired, increase the maximum number of iterations. If residuals are stalled and a quicker exit from the solver is desired, decrease the maximum number of iterations.

Maximum Integration Time (coupled solver)

The ODE solver terminates the calculation at this time if residuals fail to converge.

Use Custom Field Functions to Define Reactor Zones

When selected, this option allows you to use custom field functions (Custom Field Functions (p. 4285)) as composition variables to create the reactor volumes. Ansys Fluent clusters CFD cells that have similar compositions using the following default composition variables:

- temperature and mixture fraction for Non-Premixed and Partially-Premixed cases
- temperature and mass fractions of N_2 and H_2O for Species Transport cases

You can override these default composition variables by selecting **Use Custom Field Functions to Define Reactor Zones** and then selecting up to four custom field functions from the **Custom Field Functions** selection list. Ansys Fluent will group CFD cells that are close in these custom field function composition variables. In the example shown in Figure 19.33: Reactor Network Dialog Box - Expert Options (p. 2440), cells with similar turbulent kinetic energy, x coordinate, and y coordinate will be agglomerated to form the reactor network.

8. Click **Calculate Reactor Network** to obtain a flow solution.

As the calculation is progressing, Ansys Fluent prints the reactor network residuals for each iteration in the console.

Note:

The reactor network solution should converge with the default settings. However, if the model fails to converge with sufficient accuracy, you can adjust the default reactor network solver settings by using the **Expert Options** control.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

19.1.7.3. Postprocessing Reactor Network Calculations

Ansys Fluent provides additional postprocessing options for reactor network calculations. The following reactor network variables are available for generating graphical plots:

- **Reactor Net Zone ID**
- **Reactor Net Temperature**
- **Reactor Net Mass fraction of species-n**

These variables will appear under the **Reactor Network...** postprocessing category of the variable selection drop-down lists.

19.2. Modeling a Composition PDF Transport Problem

This section discusses how to use the composition PDF transport model for modeling finite-rate chemistry in turbulent flames in Ansys Fluent. Ansys Fluent has two different discretizations of the composition PDF transport equation, namely Lagrangian and Eulerian. The Lagrangian method is more accurate than the Eulerian method, but requires significantly longer run time to converge. For information about the theory behind the composition PDF transport model, see Composition PDF Transport in the Theory Guide.

Information about using this model is presented in the following sections:

19.2.1. Limitation
19.2.2. Steps for Using the Composition PDF Transport Model
19.2.3. Enabling the Lagrangian Composition PDF Transport Model
19.2.4. Enabling the Eulerian Composition PDF Transport Model
19.2.5. Initializing the Solution
19.2.6. Monitoring the Solution
19.2.7. Postprocessing for Lagrangian PDF Transport Calculations
19.2.8. Postprocessing for Eulerian PDF Transport Calculations

19.2.1. Limitation

A limitation that applies to the composition PDF transport model is that you must use the pressure-based solver as the model is not available with the density-based solver.

19.2.2. Steps for Using the Composition PDF Transport Model

The procedure for setting up and solving a composition PDF transport problem is outlined below, and then described in detail. Remember that only steps that are pertinent to composition PDF transport modeling are shown here. For information about inputs related to other models that you are using in conjunction with the composition PDF transport model, see the appropriate sections for those models.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Read a CHEMKIN-formatted gas-phase mechanism file and the associated thermodynamic data file in the **CHEMKIN Mechanism** dialog box (see Importing a Volumetric Kinetic Mechanism in CHEMKIN Format (p. 2368)).

   File → **Import** → **CHEMKIN Mechanism...**

   Important:

   If your chemical mechanism is not in CHEMKIN format, you will have to enter the mechanism into Ansys Fluent as described in Overview of User Inputs for Modeling Species Transport and Reactions (p. 2356).

2. Enable a turbulence model.

   Setup → **Models** → **Viscous** **Edit...**

3. Enable the **Composition PDF Transport** model and set the related parameters. Refer to Enabling the Lagrangian Composition PDF Transport Model (p. 2444) and Enabling the Eulerian Composition PDF Transport Model (p. 2447) for further details.

   Setup → **Models** → **Species** **Edit...**

4. Check the material properties in the **Create/Edit Materials** dialog box and the reaction parameters in the **Reactions** dialog box. The default settings should be sufficient.

   Setup → **Materials**

5. Set the operating conditions, cell zone conditions, and boundary conditions.

   Setup → **Cell Zone Conditions** → **Operating Conditions...**

   Setup → **Boundary Conditions**

6. Check the solver settings.

   Solution → **Methods**

   Solution → **Controls**

   The default settings should be sufficient, although you should change the discretization to second-order once the solution has converged.

7. Initialize the solution. You may need to patch a high-temperature region to ignite the flame.

   Solution → **Initialization** → **Initialize**

   Solution → **Initialization** → **Patch...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. Run the solution.

   Solution → **Run Calculation**

9. Solve the problem and perform postprocessing.

Important:

A good initial condition can reduce the solution time substantially. You should start from an existing solution calculated using the Laminar Finite-Rate, EDC model, non-premixed combustion model, or partially premixed combustion model. See Modeling Species Transport and Finite-Rate Chemistry (p. 2355), Modeling Non-Premixed Combustion (p. 2471), and Modeling Partially Premixed Combustion (p. 2543) for further information on such simulations.

19.2.3. Enabling the Lagrangian Composition PDF Transport Model

To enable the composition PDF transport model, select **Composition PDF Transport** in the **Species Model** dialog box (Figure 19.34: The Species Model Dialog Box for Lagrangian Composition PDF Transport (p. 2445)).

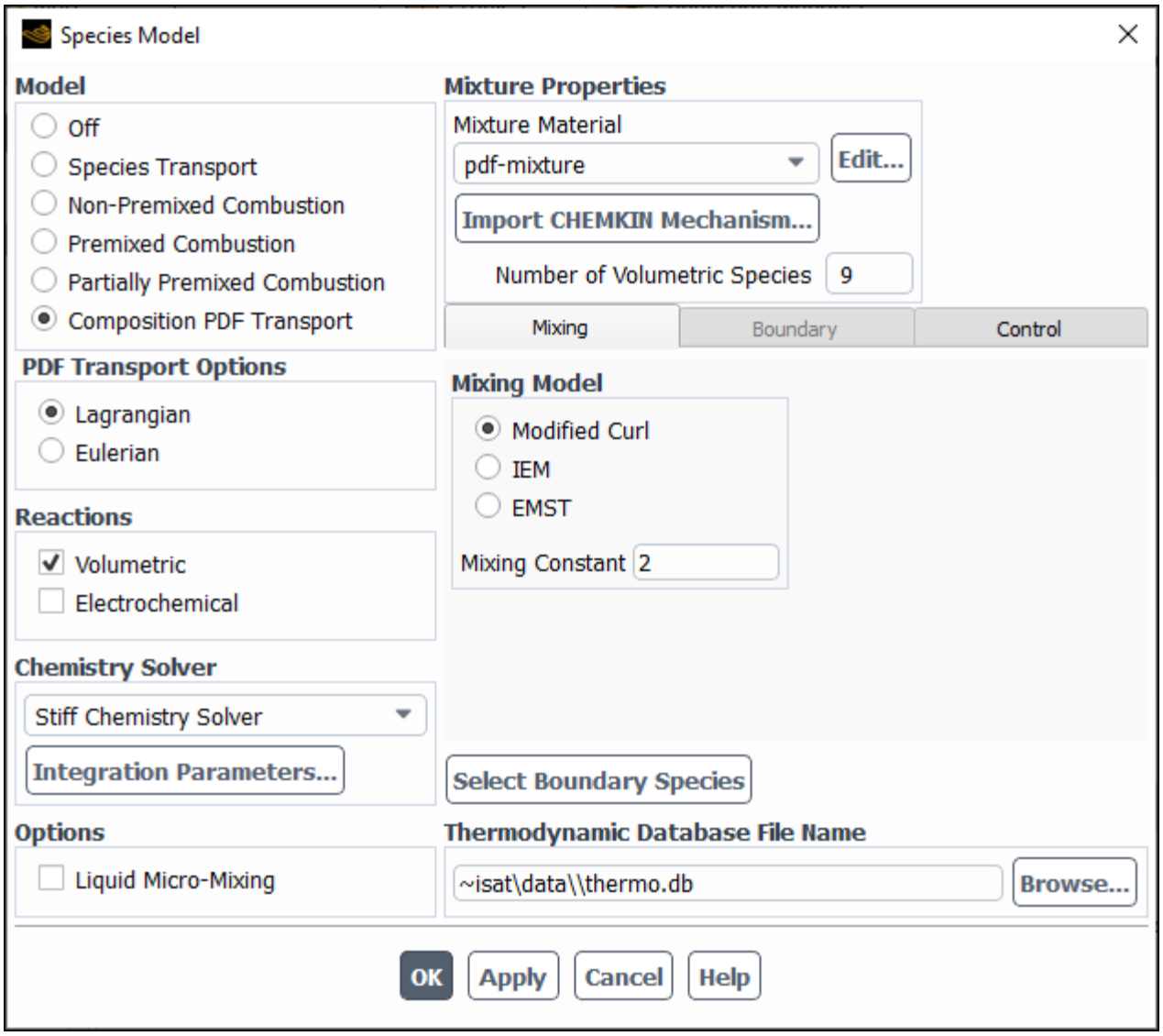
Setup → **Models** → **Species** **Edit...**

When you enable **Composition PDF Transport**, the dialog box will expand to show the relevant inputs.

1. Select **Lagrangian** under **PDF Transport Options**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 19.34: The Species Model Dialog Box for Lagrangian Composition PDF Transport



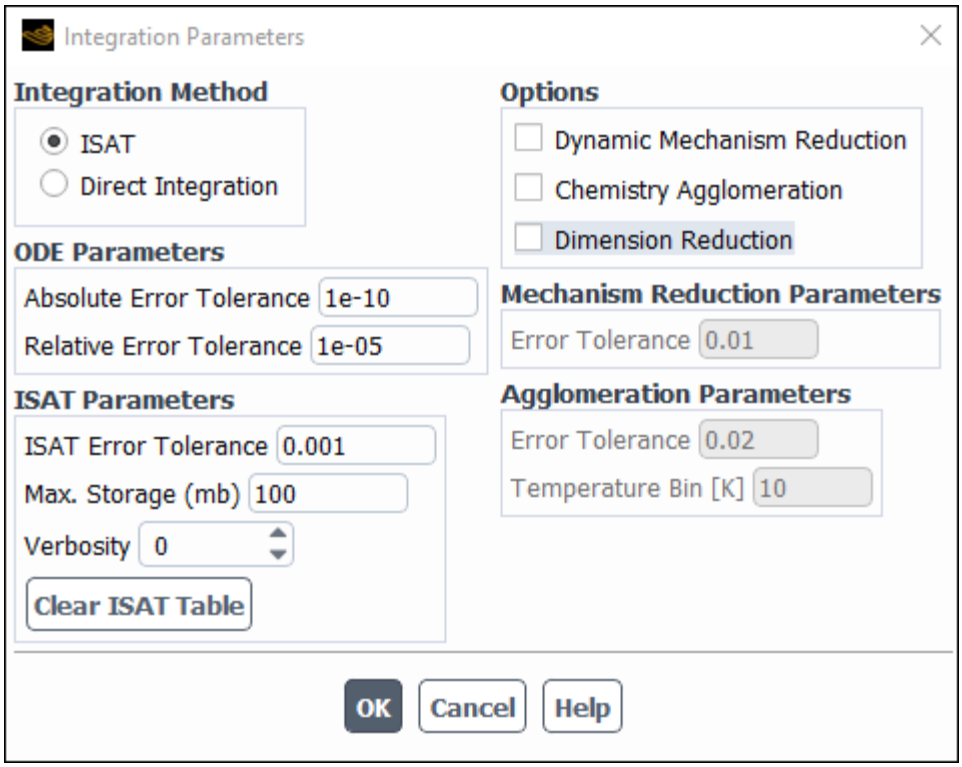
2. Enable **Volumetric** under **Reactions**.

   The **Composition Pdf Transport** model uses Ansys Fluent stiff chemistry solver with ISAT enabled by default. See Using ISAT (p. 2457) for more details.

3. Click the **Integration Parameters...** button to open the Integration Parameters Dialog Box (p. 4795) (Figure 19.35: The Integration Parameters Dialog Box (p. 2446)). For additional information, see Using ISAT (p. 2457).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 19.35: The Integration Parameters Dialog Box



4. Enable **Liquid Micro-Mixing** to interpolate C_ϕ from turbulence models and scalar spectra, as noted in Liquid Reactions in the Theory Guide. This is applied to cases where reactions in liquids occur at low turbulence levels, among reactants with low diffusivities. Therefore, a default value of C_ϕ=2 may not be desirable, as it over-estimates the mixing rate.

5. In the **Mixing** tab, select **Modified Curl**, **IEM**, or **EMST** under **Mixing Model** and specify the value of the **Mixing Constant** (C_ϕ in Equation 7.153 in the Theory Guide). For more information about particle diffusion, see Particle Mixing in the Theory Guide.

   Important:

   If the **Liquid Micro-Mixing** option is enabled, you cannot set the **Mixing Constant**.

6. You will not be specifying species boundary conditions in the **Boundary** tab. This is only applicable to the **Eulerian PDF Transport Option**.

7. In the **Control** tab, you will specify the Lagrangian PDF transport control parameters.

   Particles Per Cell

   sets the number of PDF particles per cell. Higher values of this parameter will reduce statistical error, but increase computational time.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Local Time Stepping

is available for steady-state simulations and can increase the convergence rate by taking maximum allowable time-steps on a cell-by-cell basis. (see Equation 7.151 of the Theory Guide). If **Local Time Stepping** is enabled, then you can specify the following parameters:

Convection #

specifies the particle convection number (see Δt_{conv} in Equation 7.151 in the Theory Guide).

Diffusion #

specifies the particle diffusion number (see Δt_{diff} in Equation 7.151).

Mixing #

specifies the particle mixing number (see Δt_{mix} in Equation 7.151).

19.2.4. Enabling the Eulerian Composition PDF Transport Model

To enable the composition PDF transport model, select **Composition PDF Transport** in the **Species Model** dialog box (Figure 19.34: The Species Model Dialog Box for Lagrangian Composition PDF Transport (p. 2445)).

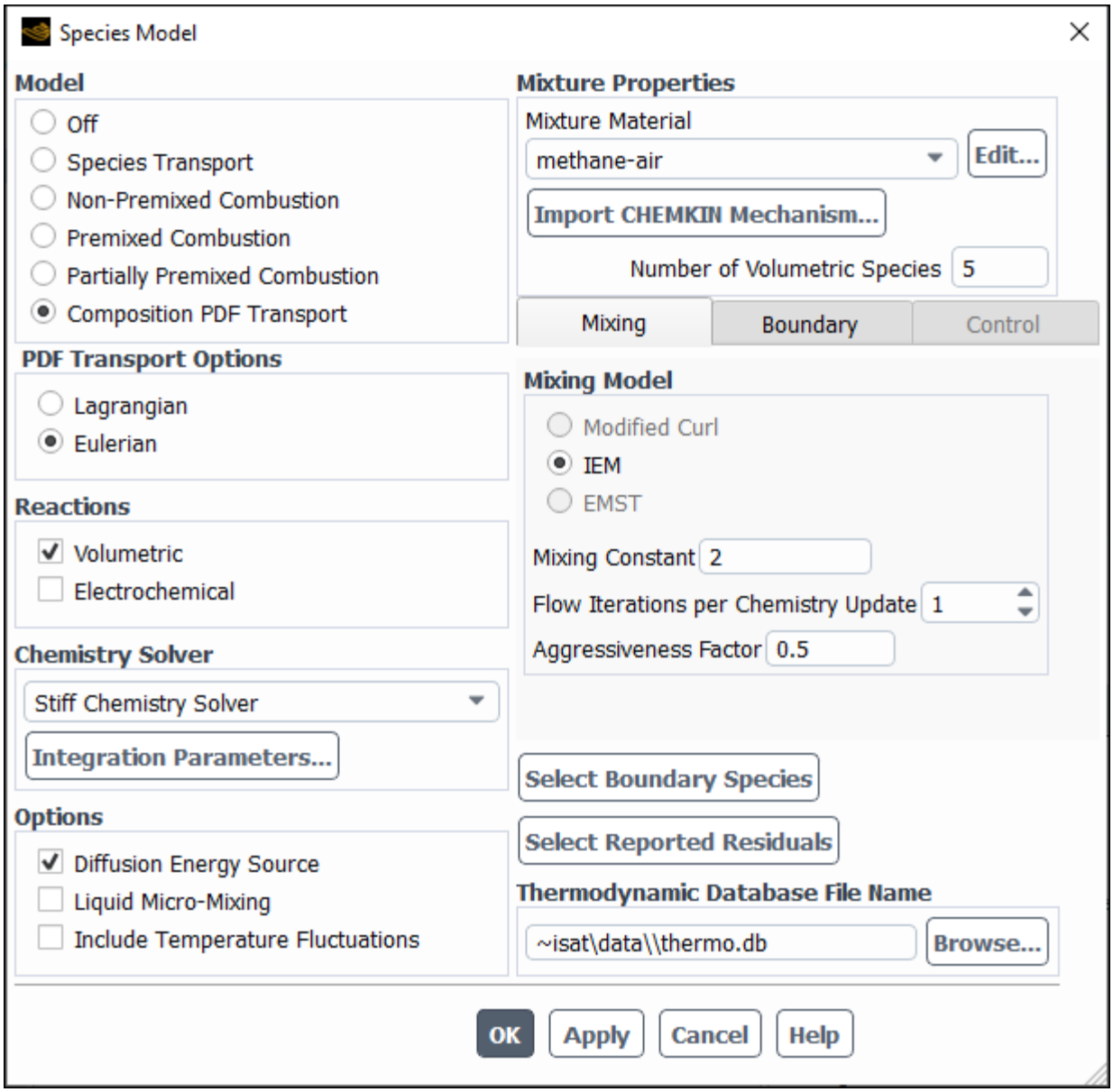
Setup → Models → Species Edit...

When you enable **Composition PDF Transport**, the dialog box will expand to show the relevant inputs.

1. Select **Eulerian** under **PDF Transport Options**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 19.36: The Species Model Dialog Box for Eulerian Composition PDF Transport



2. Enable **Volumetric** under **Reactions**. The **Stiff Chemistry Solver** is disabled by default and should be enabled if the kinetic mechanism is numerically stiff.

3. Click the **Integration Parameters...** button to open the Integration Parameters Dialog Box (p. 4795) (Figure 19.35: The Integration Parameters Dialog Box (p. 2446)). See Using ISAT (p. 2457) for detailed information about this dialog box.

4. Make sure that **Diffusion Energy Source** is enabled if you want to include species diffusion effects in the energy equation.

5. Enable **Liquid Micro-Mixing** if the fuel and oxidizer are liquids with low diffusivities (high Schmidt numbers). In this case, the **Mixing Constant** will be calculated according to Equation 7.160 in the Theory Guide.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. By default, the Laminar (one mode) energy equation is solved, where temperature fluctuations are ignored. By enabling **Include Temperature Fluctuations**, the multi-mode energy equation will be solved as done for species.

7. In the **Mixing** tab, only the **IEM** mixing model is available for Eulerian PDF transport.

   a. Specify the value of the **Mixing Constant**. The default value is 2 for gas phase species.

   Important:

   If the **Liquid Micro-Mixing** option is enabled, you cannot set the **Mixing Constant**.

   b. Enter the number of **Flow Iterations per Chemistry Update**. This is the frequency at which Ansys Fluent will update the chemistry during the calculation. Increasing the number can reduce the computational expense of the chemistry calculations.

   c. Enter the **Aggressiveness Factor**. This is a numerical factor that controls the robustness and the convergence speed. This value ranges between 0 and 1, where 0 is the most robust, but results in the slowest convergence. The default value for the **Aggressiveness Factor** is 0.5.

8. In the **Boundary** tab, define the compositions of the fuel and oxidizer. You can select the boundary species to be displayed as described in Overview of the Problem Setup Procedure (p. 2472).

9. (**Eulerian** PDF Transport only) Optionally, you can select the monitored species as described in Overview of the Problem Setup Procedure (p. 2472).

For additional information, see the following sections:

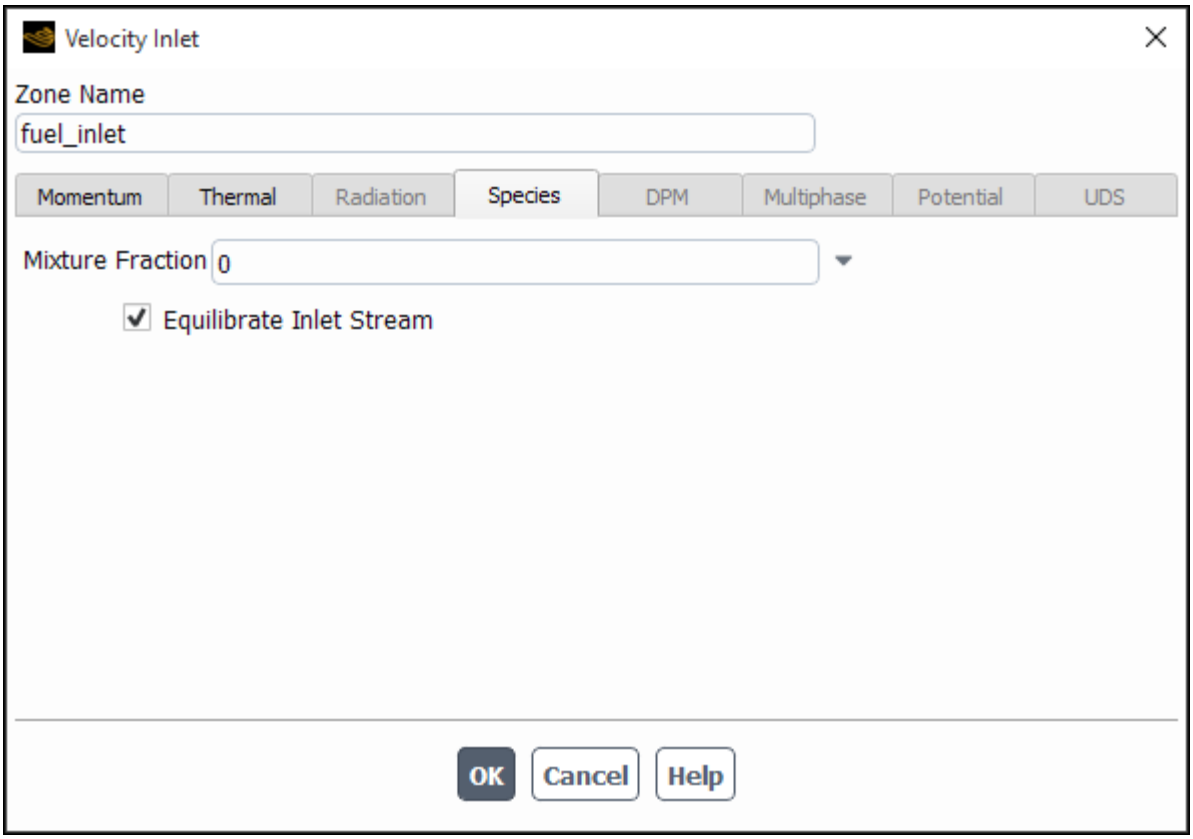
19.2.4.1. Defining Species Boundary Conditions

19.2.4.1. Defining Species Boundary Conditions

At flow inlets, specify the **Mixture Fraction** in the **Species** tab of the boundary condition inlet dialog boxes, as shown in Figure 19.37: The Velocity Inlet Dialog Box for Eulerian Composition PDF Transport (p. 2450). At outlet boundaries, similarly specify the **Backflow Mixture Fraction**. Ansys Fluent always applies zero flux boundary conditions at walls for all species.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 19.37: The Velocity Inlet Dialog Box for Eulerian Composition PDF Transport



19.2.4.1.1. Equilibrating Inlet Streams

The **Equilibrate Inlet Stream** option in the **Species** tab of the boundary conditions dialog boxes will set the inlet compositions to their chemical equilibrium values. This option can be used to model pilot flames and exhaust gas recirculation.

Important:

This option should not be enabled for pure fuel or oxidizer inlets.

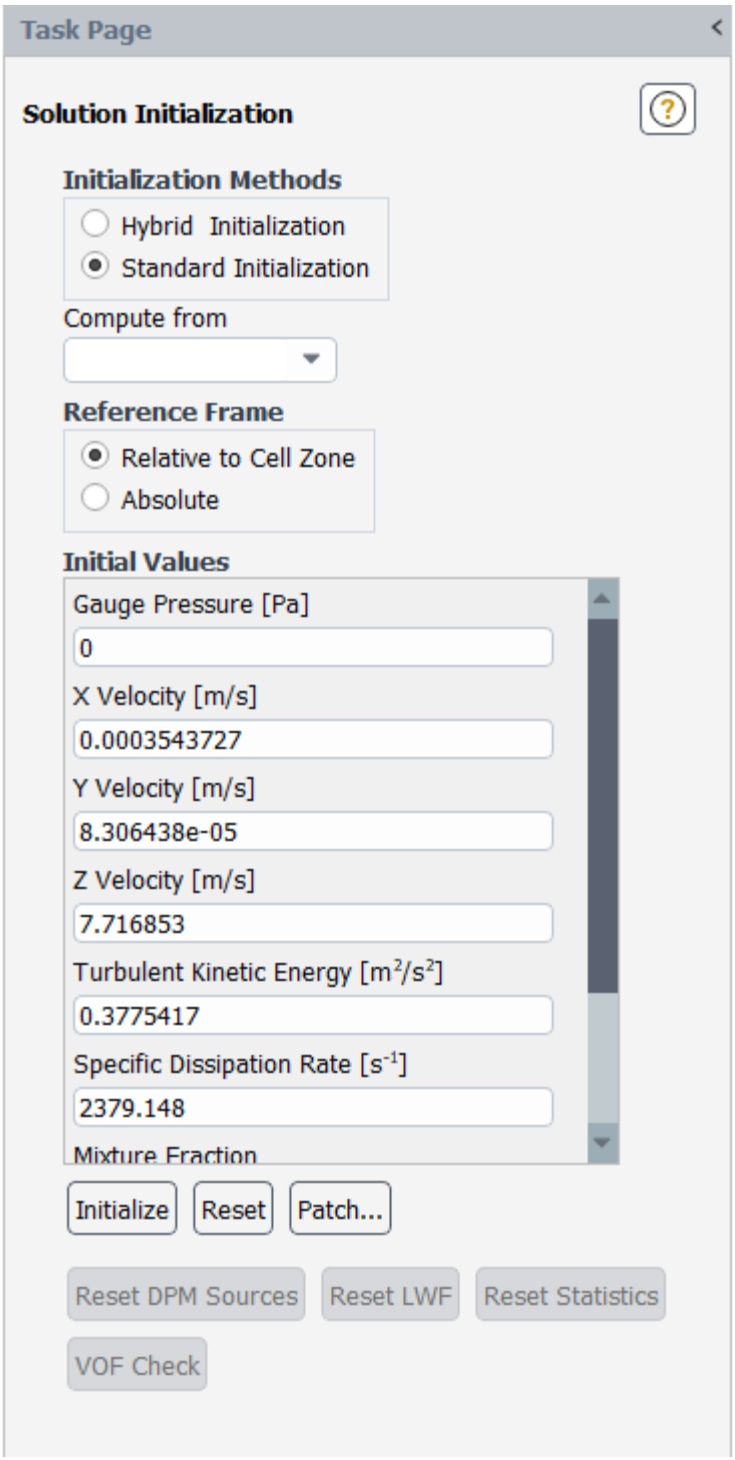
If you are using the Eulerian PDF transport model, specify the discretization and under-relaxation for the **Eulerian PDF** composition transport in the **Solution Controls** and **Solution Methods** task page.

19.2.5. Initializing the Solution

For the Eulerian PDF Transport model, the initialization variables are the mixture fraction and the temperature. The species mass fractions and enthalpy are calculated based on the fuel and oxidizer composition (specified in the **Species** dialog box) and the initialized mixture fraction and temperature.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 19.38: The Solution Initialization Task Page for Eulerian Composition PDF Transport

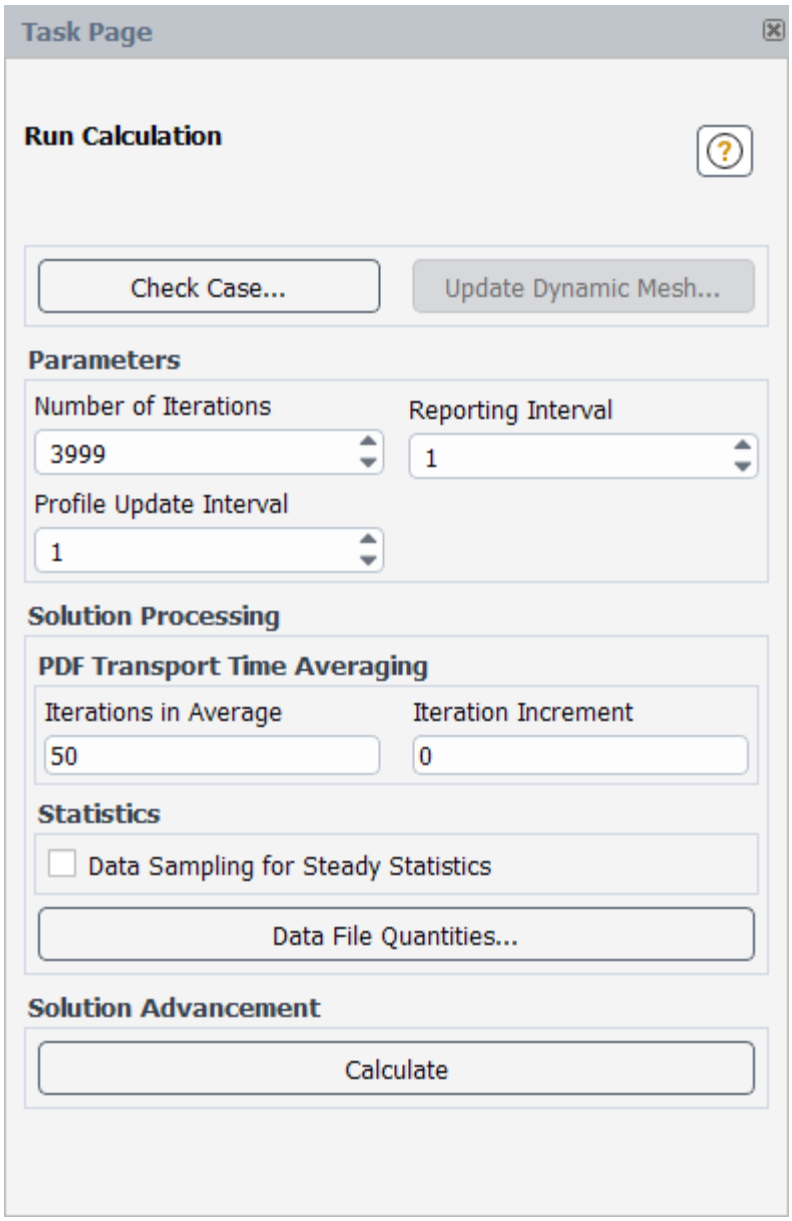


19.2.6. Monitoring the Solution

At low speeds, combustion couples to the fluid flow through density. The Lagrangian PDF transport algorithm has random fluctuations in the density field, which in turn causes fluctuations in the flow field. For steady-state flows, statistical fluctuations are decreased by averaging over a number of previous iterations in the **Run Calculation** task page (Figure 19.39: The Run Calculation Task Page for Composition PDF Transport (p. 2452)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 19.39: The Run Calculation Task Page for Composition PDF Transport



Averaging reduces statistical fluctuations and stabilizes the solution. However, Ansys Fluent often indicates convergence of the flow field before the composition fields (temperatures and species) are converged. You should lower the default convergence criteria in the Residual Monitors Dialog Box (p. 5599), and always check that the **Total Heat Transfer Rate** in the Flux Reports Dialog Box (p. 5320) is balanced. It is also recommended that you monitor temperature/species on outlet boundaries and ensure that these are steady.

The Lagrangian PDF method has the following additional solution controls: **Iterations in Average** and **Iteration Increment**. By increasing the **Iterations in Average**, fluctuations are smoothed out and residuals level off at smaller values. However, the composition PDF method requires a larger number of iterations to reach steady-state. It is recommended that you use the default of 50 **Iterations in Average** until the steady-state solution is obtained. Then, to gradually decrease the residuals, increase the **Iterations in Average** by setting a **Iteration Increment** to a value from 0 to 1 (the value

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0.2 is recommended). Subsequent iterations will increase the **Iterations in Average** by the **Iteration Increment**.

For additional information, see the following sections:

19.2.6.1. Running Unsteady Composition PDF Transport Simulations

19.2.6.2. Running Compressible Lagrangian PDF Transport Simulations

19.2.6.3. Running Lagrangian PDF Transport Simulations with Conjugate Heat Transfer

19.2.6.1. Running Unsteady Composition PDF Transport Simulations

For unsteady Lagrangian composition PDF transport simulations, a fractional step scheme is employed where the PDF particles are advanced over the time step, and then the flow is advanced over the time step. Unlike steady-state simulations, composition statistics are not averaged over iterations, and to reduce statistical error you should increase the number of particles per cell in the **Solution Monitors** dialog box.

For low speed flows, the thermo-chemistry couples to the flow through density. Statistical errors in the calculation of density may cause convergence difficulties between time step iterations. If you experience this, increase the number of PDF particles per cell, or decrease the density under-relaxation.

19.2.6.2. Running Compressible Lagrangian PDF Transport Simulations

Compressibility is included when **ideal-gas** is selected as the density method in the Create/Edit Materials Dialog Box (p. 4885). For such flows, particle internal energy is increased by $p\Delta v$ over the time step Δt, where p is the cell pressure and Δv is the change in the particle specific volume over the time step.

19.2.6.3. Running Lagrangian PDF Transport Simulations with Conjugate Heat Transfer

When solid zones are present in the simulation, Ansys Fluent solves the energy equation in the turbulent flow zones by the Lagrangian Monte Carlo particle method, and the energy equation in the solid zones by the finite-volume method.

19.2.7. Postprocessing for Lagrangian PDF Transport Calculations

For additional information, see the following sections:

19.2.7.1. Reporting Options

19.2.7.2. Particle Tracking Options

19.2.7.1. Reporting Options

Ansys Fluent provides several reporting options for the Lagrangian composition PDF transport calculations. You can generate graphical plots or alphanumeric reports of the following items:

- **Static Temperature**
- **Mean Static Temperature**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **RMSE Static Temperature**
- **Mass fraction of species-n**
- **Mean species-n Mass Fraction**
- **RMS species-n Mass Fraction**

The instantaneous composition (**Static Temperature** and **Mass fraction of species-n**) in a cell are calculated as,

$$\phi_{instant} = \frac{\sum_{i=1}^{N_c} \phi_p m_p}{\sum_{i=1}^{N_c} m_p} \tag{19.13}$$

where

$\phi_{instant}$ = instantaneous cell species mass fraction or temperature at the present iteration

N_c = number of particles in the cell

ϕ_p = particle mass fraction or temperature

m_p = particle mass

Mean and root-mean-square-error (RMSE) temperatures are calculated in Ansys Fluent by averaging instantaneous temperatures over a user-specified number of previous iterations (see Monitoring the Solution (p. 2451)).

Note that for steady-state simulations, instantaneous temperatures and species represent a Monte Carlo realization and are as such unphysical. Mean and RMSE quantities are much more useful.

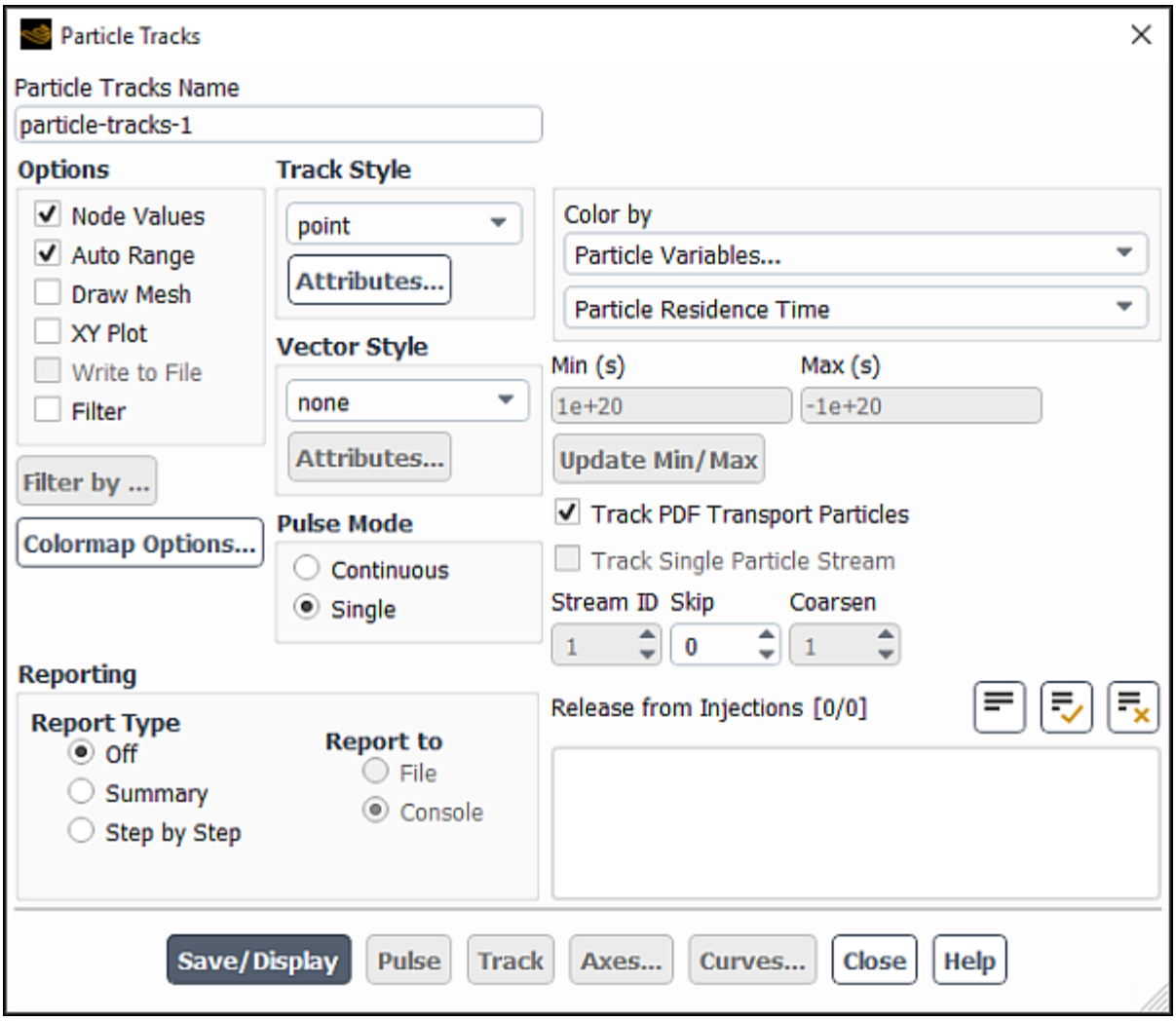
19.2.7.2. Particle Tracking Options

When you have enabled the Lagrangian composition PDF transport model, you can display the trajectories of the PDF particles using the Particle Tracks Dialog Box (p. 5558) (Figure 19.40: The Particle Tracks Dialog Box for Tracking PDF Particles (p. 2455)).

Results → **Graphics** → **Particle Tracks** **New...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 19.40: The Particle Tracks Dialog Box for Tracking PDF Particles



Select the **Track PDF Transport Particles** option to enable PDF particle tracking. To speed up the plotting process, you can specify a value n for **Skip**, which will plot only every n^{th} particle. For details about setting other parameters in the **Particle Tracks** dialog box, see Displaying of Trajectories (p. 2834).

When you have finished setting parameters, click **Display** to display the particle trajectories in the graphics window.

19.2.8. Postprocessing for Eulerian PDF Transport Calculations

For additional information, see the following sections:

19.2.8.1. Reporting Options

19.2.8.1. Reporting Options

To postprocess the Eulerian composition PDF transport model, the following variables are available for each mode:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Mixture fraction**
- **Mass fraction of species in mode n**
- **Temperature of mode n**
- **Sensible Enthalpy of mode n**

19.3. Using Chemistry Acceleration

Ansys Fluent can model detailed chemical kinetics in laminar and turbulent flames. Laminar flames are modeled with the **Finite-Rate/No TCI** option, while four turbulence-chemistry interaction models are available for turbulent flames (**Finite-Rate/No TCI**, **Eddy-Dissipation Concept**, **Lagrangian PDF Transport**, and **Eulerian PDF Transport**). Detailed chemical mechanisms are invariably numerically stiff and compute-intensive. Ansys Fluent provides following methods to accelerate these computations:

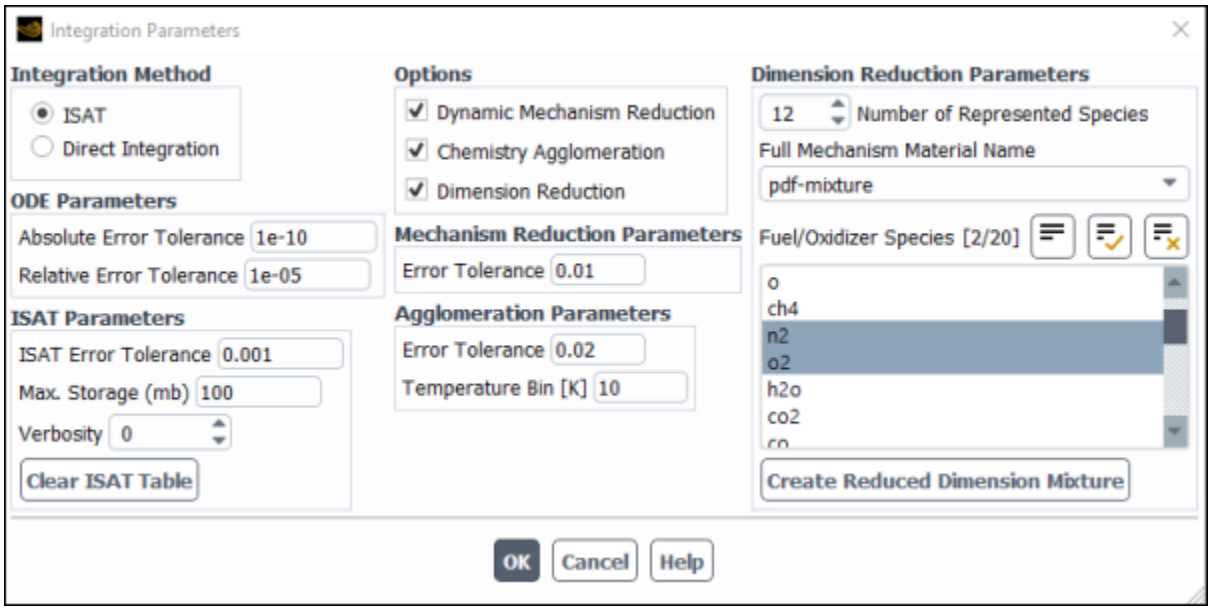
- ISAT
- Dynamic Mechanism Reduction (DMR)
- Chemistry Agglomeration
- Dimension Reduction
- Dynamic Cell Clustering (DCC) (available with the Ansys CHEMKIN-CFD solver only)
- Dynamic Adaptive Chemistry (DAC) (available with the Ansys CHEMKIN-CFD solver only)

All of the methods are enabled or disabled in the **Integration Parameters** dialog box, accessed from the **Species Model** dialog box by clicking the **Integration Parameters...** button in the **Reactions** group box:

Setup → **Models** → **Species** **Edit...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 19.41: The Integration Parameters Dialog Box



All of the acceleration methods induce some accuracy loss, and the controlling parameters should be carefully adjusted to ensure that this inaccuracy is acceptable, see Chemistry Acceleration in the *Fluent Theory Guide*. In most cases, applying ISAT and Dynamic Mechanism Reduction (DMR) together should give the best performance. Further information about using the chemistry acceleration models is presented in the following sections:

19.3.1. Using ISAT
19.3.2. Using Dynamic Mechanism Reduction
19.3.3. Using Chemistry Agglomeration
19.3.4. Dimension Reduction
19.3.5. Using Dynamic Cell Clustering
19.3.6. Using Dynamic Adaptive Chemistry with Ansys Fluent CHEMKIN-CFD Solver

19.3.1. Using ISAT

In-Situ Adaptive Tabulation (ISAT) is a storage-retrieval method that constructs a chemistry table at run time (in-situ) with a user-specified interpolation accuracy (adaptive tabulation). ISAT can be used with all Chemistry solvers except the **None - Direct Source** solver option. ISAT is not available for simulations with surface reactions.

For ODE solvers (Stiff Chemistry and CHEMKIN-CFD), Ansys Fluent uses two error tolerances (under **ODE Parameters**):

- **Absolute Error Tolerance**: by default, set to 10^{-8}.
- **Relative Error Tolerance**: by default, set to 10^{-9} for the Stiff Chemistry solver and 10^{-4} for the CHEMKIN-CFD solver.

The default values should be sufficient for most applications, although these tolerances may need to be decreased for some cases such as ignition. For problems in which the accuracy of the chemistry

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

integrations is crucial, it may be useful to test the accuracy of the error tolerances in simple zero-dimensional and one-dimensional test simulations with parameters comparable to those in the full simulation.

If you want to use chemistry agglomeration together with ISAT, enable **Chemistry Agglomeration** in the **Integration Parameters** dialog box and specify the chemistry agglomeration **Error Tolerance** (ε_{tol}). More information can be found in Using Chemistry Agglomeration (p. 2465).

For additional information, see the following sections:

19.3.1.1. ISAT Parameters

19.3.1.2. Monitoring ISAT

19.3.1.3. Using ISAT Efficiently

19.3.1.4. Reading and Writing ISAT Tables

19.3.1.1. ISAT Parameters

If you have selected **ISAT** under **Integration Method**, you will then be able to set additional ISAT parameters. The numerical error in the ISAT table is controlled by the **ISAT Error Tolerance** under **ISAT Parameters**. The default **ISAT Error Tolerance** of 0.001 may be sufficiently accurate for temperature and major species, but will most likely need to be decreased to get accurate minor species and pollutant predictions. For steady-state simulations, it may help to start with a high error tolerance during the initial iterations towards a converged solution. A larger error tolerance results in smaller tables and quicker run times, but greater error.

Important:

After your steady simulation is converged, you should always decrease the **ISAT Error Tolerance** and perform further iterations until the species that you are interested in are unchanged.

The **Max. Storage** is the maximum RAM used by the ISAT table, and has a default value of 100 MB. Generally, you do not need to set it to a value larger than 500 MB. The value of **Verbosity** allows you to monitor ISAT performance in different levels of detail. See Monitoring ISAT (p. 2458) for details about this parameter.

To purge the ISAT table, click the **Clear ISAT Table** button. See Using ISAT Efficiently (p. 2459) for more details.

19.3.1.2. Monitoring ISAT

You can monitor ISAT performance by setting the **Verbosity** in the **Integration Parameters** dialog box. For a **Verbosity** of 1 or 2, Ansys Fluent writes the following information periodically to a file named *case_file_name*_`stats.out`:

- total number of *queries*
- total number of queries resulting in *retrieves*
- total number of queries resulting in *grows*

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- total number of queries resulting in *adds*
- total number of queries resulting in *direct integrations*
- cumulative CPU seconds in ISAT
- cumulative CPU seconds outside ISAT
- cumulative wall-clock time in seconds (that is, total CPU time in ISAT plus total CPU time out of ISAT plus CPU idle time)

The ISAT **Verbosity** option of 2 is for expert users who are familiar with ISAT v5.0 [123] (p. 5720). Ansys Fluent writes out the following files for **Verbosity** = 2:

- *table_name*_`stats.out`, as described above
- *table_name*_`ODE_accuracy.out` reports the accuracy of the ODE integrations. For every new ISAT table entry, if the maximum absolute error in temperature or species is greater than any previous error, a line is written to this file. This line consists of the total number of ODE integrations performed up to this time, the maximum absolute species error, the absolute temperature error, the initial temperature and the time step.
- *table_name*_`ODE_diagnostic.out` prints diagnostics from the ODE solver
- *table_name*_`ODE_warning.out` prints warnings from the ODE solver

Initially, the table name is equal by default to the current case name, and is changed as the table is written or read.

Each processor builds its own ISAT table. If **Verbosity** is enabled, each compute node writes out the **Verbosity** file(s) with the node ID number appended to the file name.

19.3.1.3. Using ISAT Efficiently

Efficient use of ISAT requires thoughtful control. What follows are some detailed recommendations concerning the achievement of this goal.

Important:

- The numerical error in the ISAT table is controlled by the **ISAT Error Tolerance**, which has a default value of 0.001. This value is relatively large, which allows faster convergence times for steady-state simulations. However, once the solution has converged, it is important to reduce this **ISAT Error Tolerance** and re-converge. This process should be repeated until the species that you are interested in modeling are unchanged. Unsteady simulations should be run with a sufficiently small **ISAT Error Tolerance** so that the species of interest are unaffected by this parameter. Note that as the error tolerance is decreased, the memory and time requirements to build the ISAT table will increase substantially. There is a large performance penalty in specifying an error tolerance smaller than is needed to achieve acceptable accuracy, and the error tolerance should be decreased gradually and judiciously.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- As described in Mesh Partitioning and Load Balancing (p. 4315), you can select portions of the simulation to consider when performing dynamic load balancing in multiprocessor simulations. If the **ISAT** option is selected in the **Weighting** tab of the Partitioning and Load Balancing Dialog Box (p. 5567), the time required for solving chemistry will be factored in when assigning the computational cells to each available processor. It is recommended that you select this option when solving stiff chemistry, particularly when the Dynamic Mechanism Reduction method is enabled; thus, more resources can be allocated for the cells with the larger mechanisms.

During the initial iterations, before a steady-state solution is attained, transient composition states occur that are not present in the steady-state solution. For example, you might patch a high temperature region in a cold fuel-air mixing zone to ignite the flame, whereas the converged solution never has hot reactants without products. Since all states that are realized in the simulation are tabulated in ISAT, these initial mappings are wasteful of memory, and can degrade ISAT performance. If the table fills the allocated memory and contains entries from an initial transient that are no longer accessed, it may be beneficial to purge the ISAT table. This is achieved by either clearing it in the **Integration Parameters** dialog box, or saving your case and data files, exiting Ansys Fluent, then restarting Ansys Fluent and reading in the case and data. This can also be done with the TUI command `define/models/species/clear-isat-table`.

From experience, ISAT performs very well on premixed turbulent flames, where the range of composition states are smaller than in non-premixed flames. ISAT performance degrades in flames with large residence times, where more work is required in the ODE integrator.

19.3.1.4. Reading and Writing ISAT Tables

Ansys Fluent can write and subsequently read ISAT tables. However, it is in general not recommended to write and read ISAT tables for the following reason: ISAT tabulates chemical states that are specific to a single simulation. The ISAT tabulated composition states are determined by the geometry, boundary conditions, physics models such as turbulence and radiation, thermodynamic and transport properties, as well the chemical mechanism. If any of these parameters change, the realized composition space changes, and significant parts of the existing ISAT table are no longer accessed. These un-accessed ISAT table entries slow interpolation and decrease the number of useful accessed table entries that can be added. Since the time consumed building the ISAT table is typically much smaller than simulation run times, it is advised to rebuild the table when restarting.

When Ansys Fluent is run in parallel, each partition builds its own ISAT table and does not exchange information with ISAT tables on other compute nodes. You can save the ISAT tables on all compute nodes:

File → **Write** → **ISAT Table...**

Each compute node writes out its ISAT table to the specified file name, with the node ID number appended to the file name. For example, a specified file name of `my_name` on a two compute node run will write two files called `my_name-0.isat` and `my_name-1.isat`.

Subsequent runs can start from existing ISAT tables by reading them into memory.

File → **Read** → **ISAT Table...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Files can be read in two ways:

- Parallel nodes can read in corresponding ISAT tables saved from a previous parallel simulation. The appended node ID should not be removed from the input file name.

  Note:

  The ability to use ISAT tables generated from a parallel simulation with a different number of parallel nodes is not supported.

- All nodes can read one unique ISAT table. You might use this approach if you have a large table from a serial simulation. Ansys Fluent first checks to see if the exact file name that you specified exists, and if it does, all nodes will read this one file.

19.3.2. Using Dynamic Mechanism Reduction

Computational time increases with the size of the chemical kinetics mechanism used in the simulation. Dynamic Mechanism Reduction (DMR) decreases the mechanism size at each cell (or particle) to include only those species and reactions necessary for accurate modeling (within defined tolerances) of the chemical kinetics at the local conditions. This is performed at every flow iteration (for steady simulations) or time step (for transient simulations). Because the mechanism is only required to be accurate at the local conditions, smaller reduced mechanisms can be used with less accuracy loss than when using skeletal mechanism reduction, where a single reduced mechanism is used throughout the simulation. The speed-up from this approach is problem-dependent, but on average is about a factor of two or more compared to a case with no chemistry acceleration.

For an even greater increase in performance, you can use Dynamic Mechanism Reduction with any combination of the other chemistry acceleration options available in Ansys Fluent. In most cases, applying ISAT and DMR together should give the best performance. Note, however, that ISAT performance degrades when accuracy requirements are very strict or when conditions are not revisited often in a simulation. An example is unsteady ignition simulations, where ISAT table entries may not be reused. However, the important species and reactions can vary widely during the simulation; for example, many of the low temperature ignition reactions are only required for the small spark zone and short spark duration. In such situations, you may achieve better performance using Dynamic Mechanism Reduction along with direct integration method rather than ISAT.

Combining multiple chemistry acceleration methods, however, compounds accuracy loss, since each of these methods accelerates the simulation by sacrificing some accuracy. In general, greater speed-up is achievable when the full mechanism is larger, because there is potential for a greater degree of reduction.

Important:

When using Dynamic Mechanism Reduction along with ISAT, setting the error tolerance for either approach above its default value degrades accuracy faster than when either method is used alone.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Dynamic Mechanism Reduction is performed using the Directed Relation Graph (DRG) algorithm. For details of the DMR algorithm, refer to the discussion in Dynamic Mechanism Reduction in the *Fluent Theory Guide*.

You can enable Dynamic Mechanism Reduction through either the graphical user interface (GUI) or the text user interface (TUI) as follows.

- In the GUI, enable the **Dynamic Mechanism Reduction** option in the **Integration Parameters** dialog box.
- In the TUI, enter the following text command in the console:

  ```
  define/models/species/integration-parameters
  ```

  When prompted with `Enable Dynamic Mechanism Reduction?`, answer with `yes`.

  Note that it is not necessary to enable Chemistry Acceleration Expert mode in order to use Dynamic Mechanism Reduction.

For additional information, see the following sections:

19.3.2.1. Mechanism Reduction Parameters

19.3.2.2. Monitoring and Postprocessing Dynamic Mechanism Reduction

19.3.2.3. Using Dynamic Mechanism Reduction Effectively

19.3.2.1. Mechanism Reduction Parameters

In Ansys Fluent, Dynamic Mechanism Reduction is controlled by the error tolerance and the target species list, as described below.

Error Tolerance

(ε in Equation 7.172 in the *Fluent Theory Guide*).The default value for error tolerance ε is `0.01`, which should work well for most simulations, balancing speed and accuracy. However, a larger tolerance may be sufficient for some simulations (for example, steady-state simulations with moderate accuracy requirements), while a smaller tolerance may be required in other cases with stricter accuracy requirements (for example, modeling auto-ignition delay times for a highly complex fuel). In general, a larger tolerance yields faster, but less accurate simulations.

The error tolerance can be adjusted through the GUI or TUI as follows:

- In the GUI, set **Error Tolerance** to the desired value in the **Mechanism Reduction Parameters** group box in the **Integration Parameters** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- In the TUI, enter the following text command: `define/models/species/integration-parameters`. When prompted with `Mechanism Reduction Error Tolerance`, enter the desired tolerance (a positive, nonzero value) or press **Enter** to retain the default value.

Note:

- In the TUI, you can also enable Chemistry Acceleration Expert mode, other chemistry acceleration methods, and specify ODE integrator parameters (see Using ISAT (p. 2457) for details about these parameters).
- Chemistry Acceleration Expert mode should only be enabled if you want to modify the default target species list.

Target Species List

As described in the Dynamic Mechanism Reduction in the *Fluent Theory Guide*, the target species are those species that you want to predict most accurately. The default number of target species N_{target} is 3. The first default target species is hydrogen radical. DRG will add two other species with the largest mass fractions to complete the target species list at each flow time step or iteration. DRG will then identify all other species that must also be included in the mechanism in order to accurately model these targets.

Although you should rarely need to alter the default target species list, Ansys Fluent provides an option to specify target species of your choice. The designated target species can be specified only in the TUI when Chemistry Acceleration Expert mode is enabled. Enter the text command `define/models/species/integration-parameters` and then answer `yes` when prompted with `Enable Chemistry Acceleration expert?`.

Ansys Fluent provides the ability to explicitly specify target species. From your mixture material list, you can select any number N_{user} of target species to be included in the kinetics mechanism. In case no targets were selected (N_{user}=0) or the number of user-selected targets N_{user} is less than the minimum number of target species N_{target} (N_{user}<N_{targets}), the DRG algorithm will add the (N_{targets}−N_{user}) species with the largest mass fractions to the target species list.

As discussed in the Dynamic Mechanism Reduction in the *Fluent Theory Guide*, there is an option to remove a species from the target list whenever its mass fraction is below a specified threshold. This option is disabled by default in Ansys Fluent (that is, the default value for minimum mass fraction is 0).

When specifying Target Species List, you will receive the following prompts in the console:

1. `Minimum number of target species [3]`

   Enter the desired number of targets.

2. `Minimum mass fraction of target species [0]`

   Enter minimum allowable target mass fraction.

   The current target species list will be displayed in the console window (for example, the console output for the default target species consisting of hydrogen radical will be `Current target species list = (h)`).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. `Enter target species list...`

   `Species name [""]`

   Enter target species name, for example, **"ch4"**.

   Important:

   You *must* enter the complete name of the species as it appears in the **Chemical Formula** field in the **Create/Edit Materials** dialog box within quotes (" ").

   After you enter the first target species, you will be prompted to specify the second one, and so on, until you press **Enter** to complete the setup.

   Note, that if you have entered **0** for the minimum number of target species N_{target} and have not provided any target species (that is, the number of user-selected targets, N_{user}, is zero), Ansys Fluent will automatically select the three species with the largest mass fraction and use them as the target species in the DRG algorithm.

19.3.2.2. Monitoring and Postprocessing Dynamic Mechanism Reduction

When Chemistry Acceleration expert is enabled in the TUI, two additional field variables are available to be monitored or examined in postprocessing:

- **DRG Reduced Number of Species** in the **Species...** category
- **DRG Reduced Number of Reactions** in the **Reactions...** category

These field variables quantify the size of the reduced mechanism at each cell or particle in the domain (the number of retained species and reactions, respectively).

When you use DMR in combination with ISAT, Ansys Fluent will report values of zero for **DRG Reduced Number of Species** and **DRG Reduced Number of Reactions** for those cells where the Ansys solver computed the solution using table lookup instead of direct integration in the ISAT algorithm. A cell value of zero **DRG Reduced Number of Species** does not imply that the DMR algorithm eliminated all species from the mechanism; rather, it indicates that the Ansys solver performed ISAT table lookup to obtain the chemistry solution at the cell.

If you want to view the cells for which ISAT table lookup was performed in the last iteration or time step, display the **DRG Reduced Number of Species** plot clipped to a range of 0 to 0. (Make sure that the **Node Value** option is deselected in the postprocessing dialog boxes.)

If you want to view the cells for which DMR was performed in the last iteration or time step, display the **DRG Reduced Number of Species** plot clipped to a range of 1 to a global maximum value (as it appears in the **Max** field when **Global Range** is selected in the postprocessing dialog boxes).

Note:

The DMR postprocessing field variables are not stored in the data file and, hence, will not be available for postprocessing in the next session. If you want to postprocess these variables in your next session, read the data file and perform a single iteration or time

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

step (for steady-state or transient simulations, respectively) in order for the DMR data to be calculated and available for viewing.

19.3.2.3. Using Dynamic Mechanism Reduction Effectively

As described in Dynamic Mechanism Reduction in the *Fluent Theory Guide*, mechanism reduction is performed at each cell or particle, once per time step (for transient simulations) or per flow iteration (for steady-state simulations). It is assumed that the reduced mechanism created for the starting conditions will remain valid for the entire transport time step over which the chemistry ODE is integrated. If the chemistry integration time interval is very large, the chemical state can change significantly, degrading the accuracy of the reduced mechanism. In some cases (particularly transient simulations such as ignition) it may be necessary to enforce smaller flow time steps in order to effectively use Dynamic Mechanism Reduction. In this way the reduced mechanisms are updated more frequently to match the changing chemical states.

Note that Dynamic Mechanism Reduction works best when there are significant regions in a computational domain and/or times during a simulation with relatively low chemical activity (for example, low temperature, low mixing of fuel/oxidizer, low concentrations of reactive species, and so on). Consider a simple opposed flow diffusion flame problem with pure fuel as one stream and pure oxygen as the other stream. Large mechanisms (which, depending on the error tolerance, may be close to the full mechanism size) will likely be used in the mixing region where the flame is located, while smaller mechanisms would be used elsewhere in the domain. You should not expect much speed-up if a very large fraction of your grid cells (for example, 90%) are in the flame region.

In steady-state simulations, you can improve the solution time by first running to convergence with a larger error tolerance ε, and then restarting the simulation repeatedly, gradually decreasing the error tolerance ε until you reach the desired accuracy level. For example, if you first converge the solution with the error tolerance ε=0.1, then iterate the solution further to convergence with the lower error tolerance, ε=0.05, and then with the desired error tolerance ε=0.01, you will generally be able to achieve faster convergence than iterating from initial conditions with the error tolerance ε=0.01.

Important:

As described in Mesh Partitioning and Load Balancing (p. 4315), you can select portions of the simulation to consider when performing dynamic load balancing in multiprocessor simulations. When using the Dynamic Mechanism Reduction along with ISAT, it is recommended that you select the **ISAT** option in the **Weighting** tab of the Partitioning and Load Balancing Dialog Box (p. 5567) when solving stiff chemistry. If **ISAT** is selected, the time required for solving chemistry will be factored in when assigning the computational cells to each available processor; thus, more resources can be allocated for the cells with the larger mechanisms.

19.3.3. Using Chemistry Agglomeration

The Chemistry Agglomeration method reduces the number of calls to the computationally expensive ODE integrator by clustering cells with similar compositions. The size of these clusters is determined by the agglomeration parameters **Error Tolerance** (ε_{tol}) and **Temperature Bin** (Δ_T) in the **Integration**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parameters dialog box. Larger values of ε_{tol} and Δ_T result in a larger number of agglomerated cells, fewer calls to the reaction integrator, increased run-time speed, but greater error.

You can also set the chemistry agglomeration parameters using the text user interface (TUI) command `define/models/species/integration-parameters`. Once you enable agglomeration chemistry, Ansys Fluent prompts you for `Agglomerate Chemistry Error Tolerance` and `Agglomerate Chemistry Temperature Bin`. Enter the desired values or press **Enter** to retain the default values of `0.02` for `Agglomerate Chemistry Error Tolerance` and `10` for `Agglomerate Chemistry Temperature Bin`.

More information can be found in Chemistry Agglomeration in the *Fluent Theory Guide*.

Important:

The solution accuracy can be improved by decreasing the tolerances. Generally, it should not be necessary to reduce the tolerances below 0.01 and 10 K.

19.3.4. Dimension Reduction

Detailed kinetic mechanisms typically contain a multitude of intermediate species that far exceed the number of major fuel, oxidizer, and product species. Chemical mechanism **Dimension Reduction** reduces the number of intermediate species transport equations (called representative species) that are solved, and reconstructs the 'unrepresented' species using chemical equilibrium assumptions.

Important:

Since Ansys Fluent is limited to a maximum of 700 transported species, the main use of **Dimension Reduction** is to enable simulation with chemical mechanisms containing more than 700 species.

Follow these steps to use **Dimension Reduction**:

- Import your CHEMKIN mechanism. Note that you can import CHEMKIN mechanisms that contain more than 700 species.
- Click the **Integration Parameters** button in the **Species Model** dialog box, and enable **Dimension Reduction**.
- Set the **Number of Represented Species**. This must be greater than 10 and less than the number of species in the full mechanism. The **Number of Represented Species** must also be less than 700 minus the number of unrepresented elements (the number of chemical elements in the unrepresented species). A larger **Number of Represented Species** will increase accuracy, but also increase computational expense. The default of 12 has been chosen to provide a good compromise between accuracy and speed.
- Select the **Full Mechanism Material Name**, which is typically the name of the CHEMKIN mechanism that you imported. Ansys Fluent can store several imported CHEMKIN mechanisms in different mixture materials whose mechanisms you are not using. However, since mechanisms are typically large, you should delete unused mixtures to reduce memory requirements.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Set the boundary and initial fuel and oxidizer, as well as product species, as represented species. These are set in the **Fuel/Oxidizer Species** list. Note that you can force other species to be represented by selecting them here. Species of interest, especially species that are not near chemical equilibrium, such as pollutants, and their associated intermediate species in the mechanism should also be included. Intermediate species that occur in large mass fractions relative to the fuel and oxidizer species should be included, as well as species important in the chemical pathway. For example, for methane combustion in air, CH_3 should be included as a represented species since CH_4 pyrolizes to CH_3 first.

- Click **Calculate Reduced Dimension Mixture**. This will create a new mixture material called `reduced-dimension-mixture`, which contains the represented species as well as proxy 'species' for the unrepresented elements. These unrepresented elements have "u" prepended to the element name. You should never rename the reduced-dimension-mixture mixture material, or select another mixture as the active material, while **Dimension Reduction** is enabled.

- Continue with the setup, solution, and postprocessing as for other detailed chemistry cases. Note that the boundary and initial mass fraction of unrepresented elements should always be zero. The entries for the unrepresented element mass fractions are disabled in the Ansys Fluent GUI.

For postprocessing, the unrepresented elements are available in the **Species** list as the element name with "u" prepended. All species in the full mechanism, consisting of both represented and unrepresented species, are available in the **Full Mechanism Species...** category of the postprocessing dialog boxes.

After you obtain a preliminary solution with **Dimension Reduction**, it is recommended that you check the magnitude of all unrepresented species, which are available in the **Full Mechanism Species...** option in the **Contours** dialog box. If the mass fraction of any unrepresented species is larger than other represented species, you should repeat the simulation with this species included in the represented species list. In turn, the mass fraction of all unrepresented elements should decrease.

Note:

Note that **Dimension Reduction** is only available with ISAT. **Dimension Reduction** is initially comparable in speed to a simulation with the full mechanism, but iterations become significantly faster at later times when the ISAT table is populated.

For information about the theory of this option, see Chemical Mechanism Dimension Reduction in the *Fluent Theory Guide*.

19.3.5. Using Dynamic Cell Clustering

The Dynamic Cell Clustering (DCC) method is available with the Ansys CHEMKIN-CFD solver. DCC groups computational cells with similar temperatures, pressures, and initial species mass fractions into clusters.

To use the DCC method, select **Dynamic Cell Clustering** in the **Integration Parameters** dialog box.

The following parameters are available for controlling dynamic cell clustering:

- **Max. Temperature Dispersion** (default =10 K)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Max. Equiv. Ratio Dispersion** (default =0.05)
- **Max. Clusterization** (default = 10) and **Min. Clusterization** (default = 0)

  These parameters limit the number of generated clusters and allocate the required amount of memory in your analysis.

- **Reactants Threshold (mass fraction)** (default = 1e-09)

  If the computed value of the species mass fraction is above the specified threshold, then the solver calculates the clusterization parameters; otherwise, it is considered to be zero.

The default values for these parameters have been found to provide accurate results for a wide range of combustion cases. Varying these parameters may affect the computational cost. For example, reducing maximum temperature and equivalence ratio of dispersions increases the number of clusters, which leads to a higher CPU cost.

For background information about DCC, refer to Dynamic Cell Clustering with Ansys Fluent CHEMKIN-CFD Solver in the *Fluent Theory Guide*.

19.3.6. Using Dynamic Adaptive Chemistry with Ansys Fluent CHEMKIN-CFD Solver

---

Note:

Dynamic Adaptive Chemistry is not compatible with ISAT.

The **Dynamic Adaptive Chemistry** method is controlled by the following parameters:

- **DAC Error Tolerance**: Controls the level of accuracy.

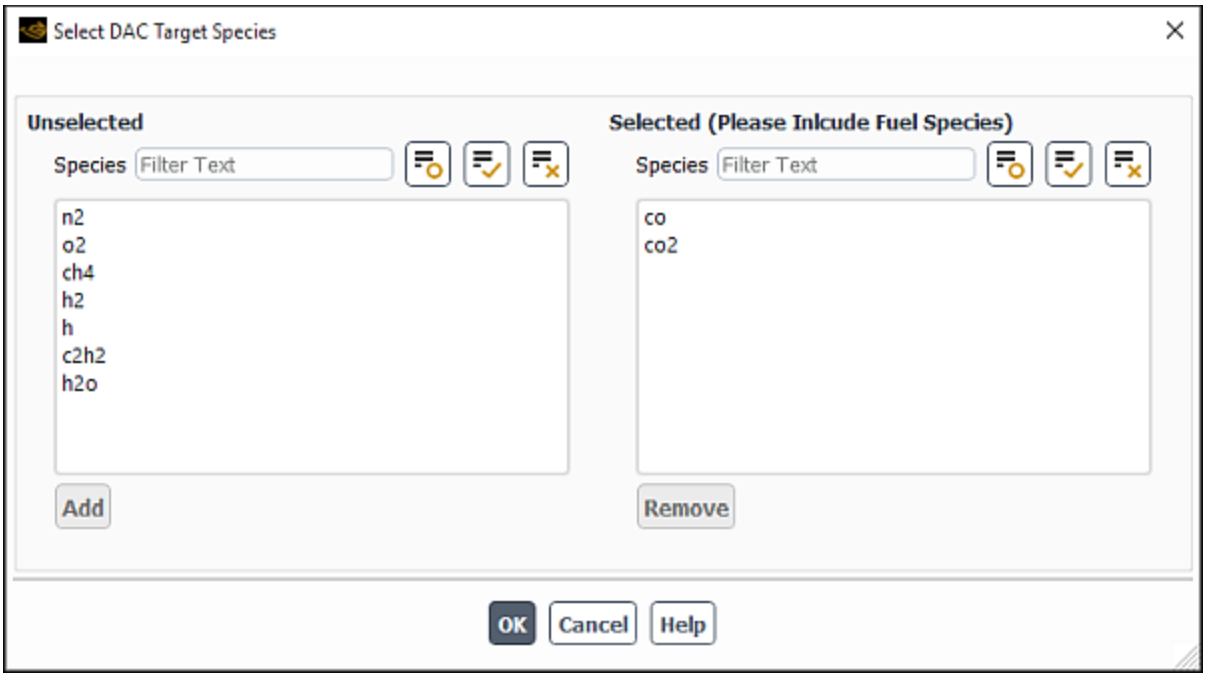
  A larger tolerance would tend to lower the accuracy but might speed up the computation. The default value for DAC error tolerance is 0.001, which should be suitable for most cases.

- DAC target species: Are the initial species to be tracked by the DAC algorithm.

  You can select the target species for your analysis in the **Select DAC Target Species** dialog box that opens when you click **Select Target Species** in the **Integration Parameters** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 19.42: The Select DAC Target Species Dialog Box



For both diesel and gasoline surrogate fuels, CO, HO_2, and fuel species are an effective choice for the target species (367 and 368). By default, only **co** and **ho2** are selected as the DAC target species. You must also select fuel species for your analysis.

To modify the DAC target species list:

– To remove the species from the **Selected Species** multiple-selection list, select it and click **Remove**. The species will be moved from the **Selected Species** list to the **Unselected Species** list.

– To add the species back to the **Selected Species** list, select it in the **Unselected Species** multiple-selection list and click **Add**. The species will be moved from the **Unselected Species** list to the **Selected Species** list.

For information about the theory of this method, see Dynamic Adaptive Chemistry with Ansys Fluent CHEMKIN-CFD Solver in the *Fluent Theory Guide*.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 20: Modelling of Turbulent Combustion With Reduced Order

This chapter contains information about the following turbulent combustion modeling capabilities in Ansys Fluent"

20.1. Modeling Non-Premixed Combustion

20.2. Modeling Premixed Combustion

20.3. Modeling Partially Premixed Combustion

20.1. Modeling Non-Premixed Combustion

This section provides details about the non-premixed combustion modeling capabilities in Ansys Fluent.

The non-premixed combustion model is presented in the following sections:

20.1.1. Steps in Using the Non-Premixed Model

20.1.2. Setting Up the Equilibrium Chemistry Model

20.1.3. Setting Up the Steady and Unsteady Diffusion Flamelet Models

20.1.4. Defining the Stream Compositions

20.1.5. Setting Up Control Parameters

20.1.6. Calculating the Flamelets

20.1.7. Calculating the Look-Up Tables

20.1.8. Standard Files for Diffusion Flamelet Modeling

20.1.9. Setting Up the Inert Model

20.1.10. Defining Non-Premixed Boundary Conditions

20.1.11. Defining Non-Premixed Physical Properties

20.1.12. Solution Strategies for Non-Premixed Modeling

20.1.13. Enabling Robust Numerics for Combustion with a PDF Table

20.1.14. Postprocessing the Non-Premixed Model Results

For theoretical background on the non-premixed combustion model, see Non-Premixed Combustion in the Theory Guide.

20.1.1. Steps in Using the Non-Premixed Model

A description of the inputs for the non-premixed model is provided in the sections that follow.

20.1.1.1. Preliminaries

20.1.1.2. Defining the Problem Type

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

20.1.1.3. Overview of the Problem Setup Procedure

20.1.1.1. Preliminaries

Before turning on the non-premixed combustion model, you must enable turbulence calculations in the Viscous Model Dialog Box (p. 4715).

Setup → Models → Viscous Edit...

If your model is non-adiabatic, you should also enable heat transfer (and radiation, if required).

Setup → Models → Energy ON

Setup → Models → Radiation Edit...

Figure 8.7: Reacting Systems Requiring Non-Adiabatic Non-Premixed Model Approach in the Theory Guide illustrates the types of problems that must be treated as non-adiabatic.

20.1.1.2. Defining the Problem Type

Your first task is to define the type of reaction system and reaction model that you intend to use. This includes selection of the following options:

- Non-premixed or partially premixed model option (see Modeling Partially Premixed Combustion (p. 2543)).
- Equilibrium chemistry model, steady diffusion flamelet model, unsteady diffusion flamelet model, or diesel unsteady flamelet.
- Adiabatic or non-adiabatic modeling options (see Non-Adiabatic Extensions of the Non-Premixed Model in the Theory Guide).
- Addition of a secondary stream (equilibrium model only).
- Empirically defined fuel and/or secondary stream composition (equilibrium model only).

You can make these model selections using the **Species Model** dialog box (Figure 20.6: The Species Model Dialog Box (Chemistry Tab) (p. 2477)).

Setup → Models → Species Edit...

20.1.1.3. Overview of the Problem Setup Procedure

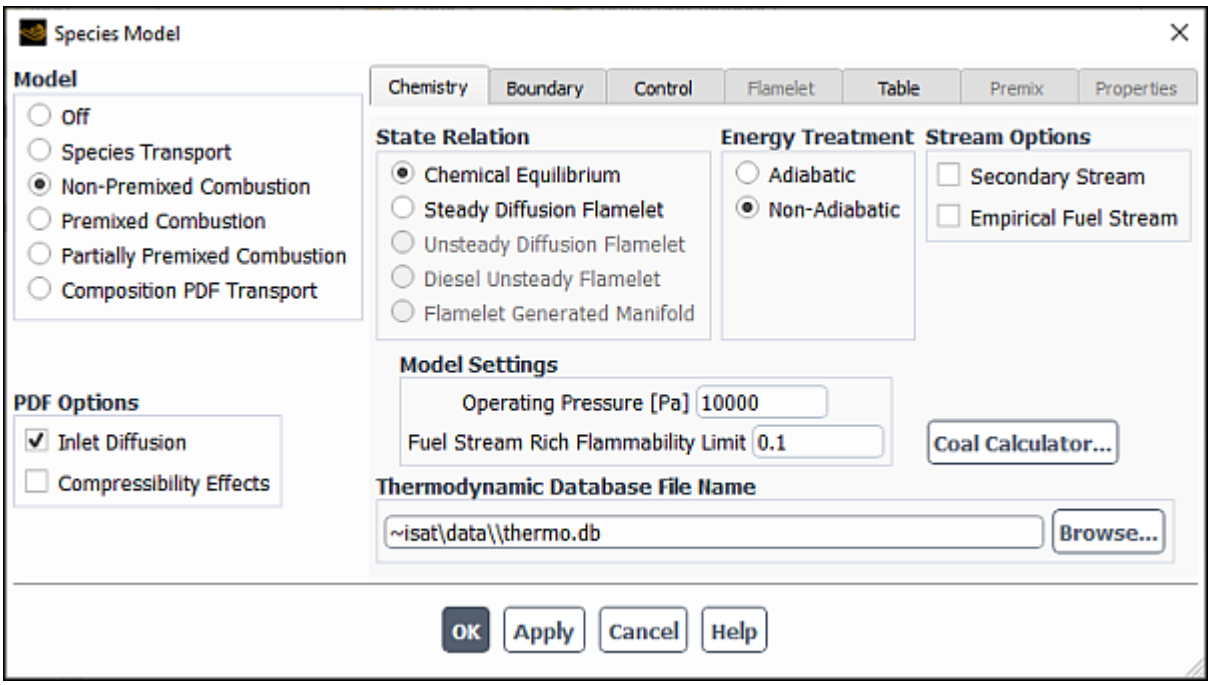
For a single-mixture-fraction problem, you will perform the following steps:

1. Choose the chemical description of the system: chemical equilibrium, steady diffusion flamelet, unsteady diffusion flamelet, or diesel unsteady flamelet (Figure 20.1: Defining Equilibrium Chemistry (p. 2473)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

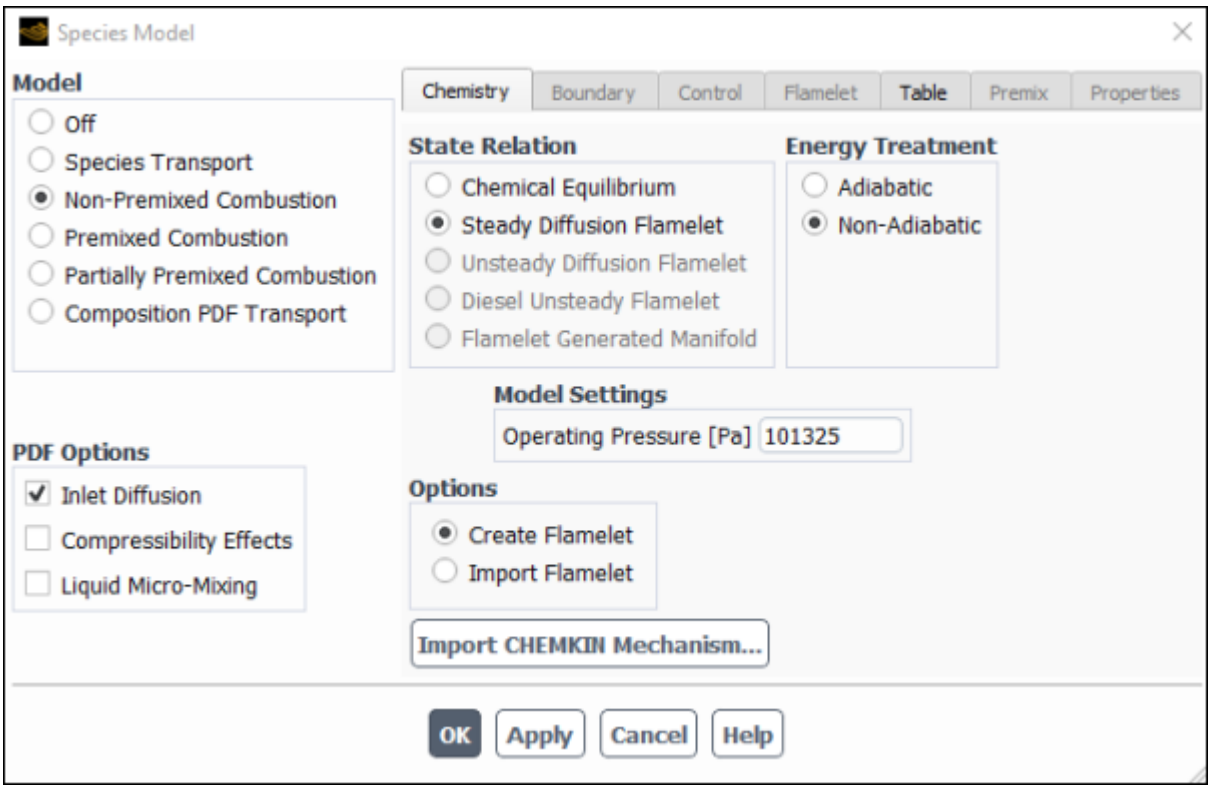
2. Indicate whether the problem is adiabatic or non-adiabatic.

Figure 20.1: Defining Equilibrium Chemistry



3. (steady diffusion flamelet model only) Import a flamelet file or appropriate CHEMKIN mechanism file if generating flamelets (Figure 20.2: Defining Steady Diffusion Flamelet Chemistry (p. 2473)).

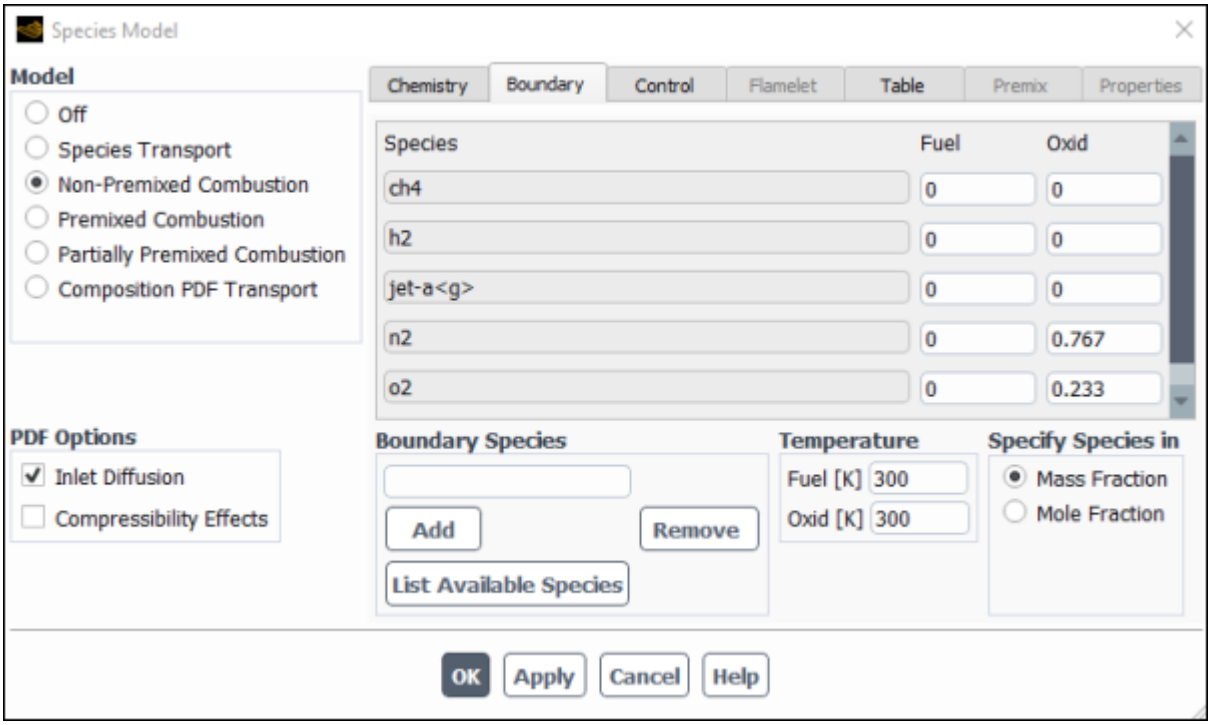
Figure 20.2: Defining Steady Diffusion Flamelet Chemistry



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Define the chemical boundary species to be considered for the streams in the reacting system model. Note that this step is not relevant in the case of flamelet import (Figure 20.3: Defining Chemical Boundary Species (p. 2474)). For more information, see Defining the Stream Compositions (p. 2487).

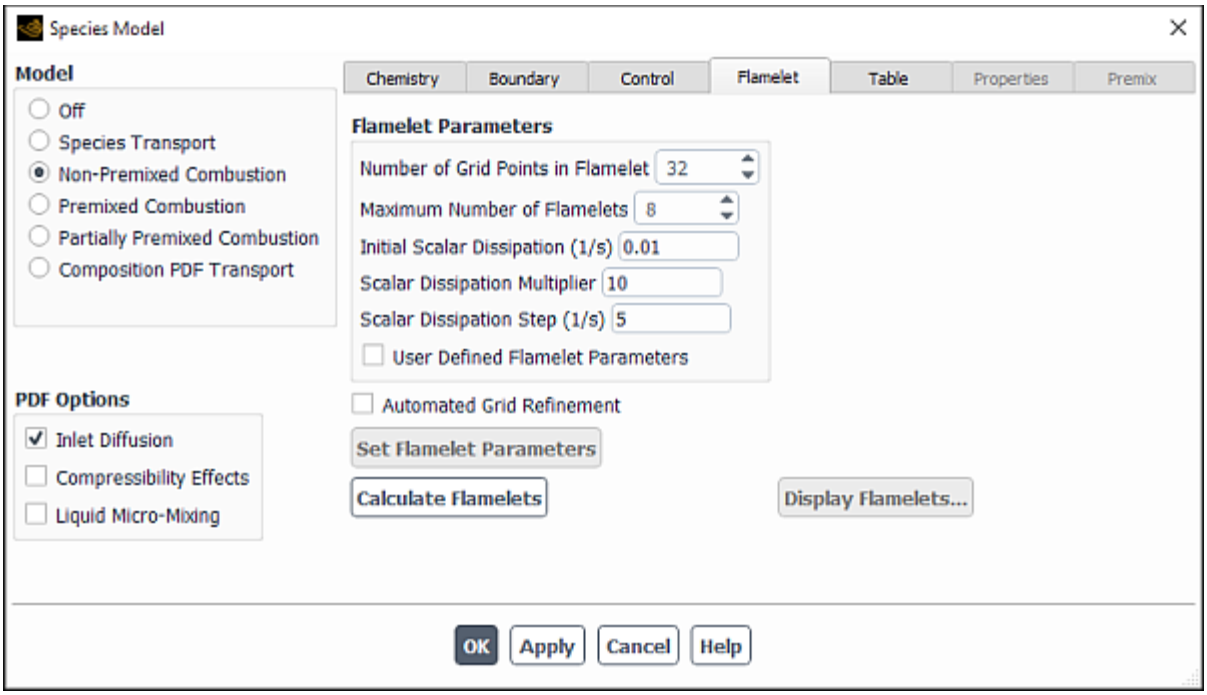
Figure 20.3: Defining Chemical Boundary Species



5. (steady diffusion flamelet model only) If you are generating flamelets, compute the flamelet state relationships of species mass fractions, density, and temperature as a function of mixture fraction and scalar dissipation (Figure 20.4: Calculating Steady Diffusion Flamelets (p. 2475)).

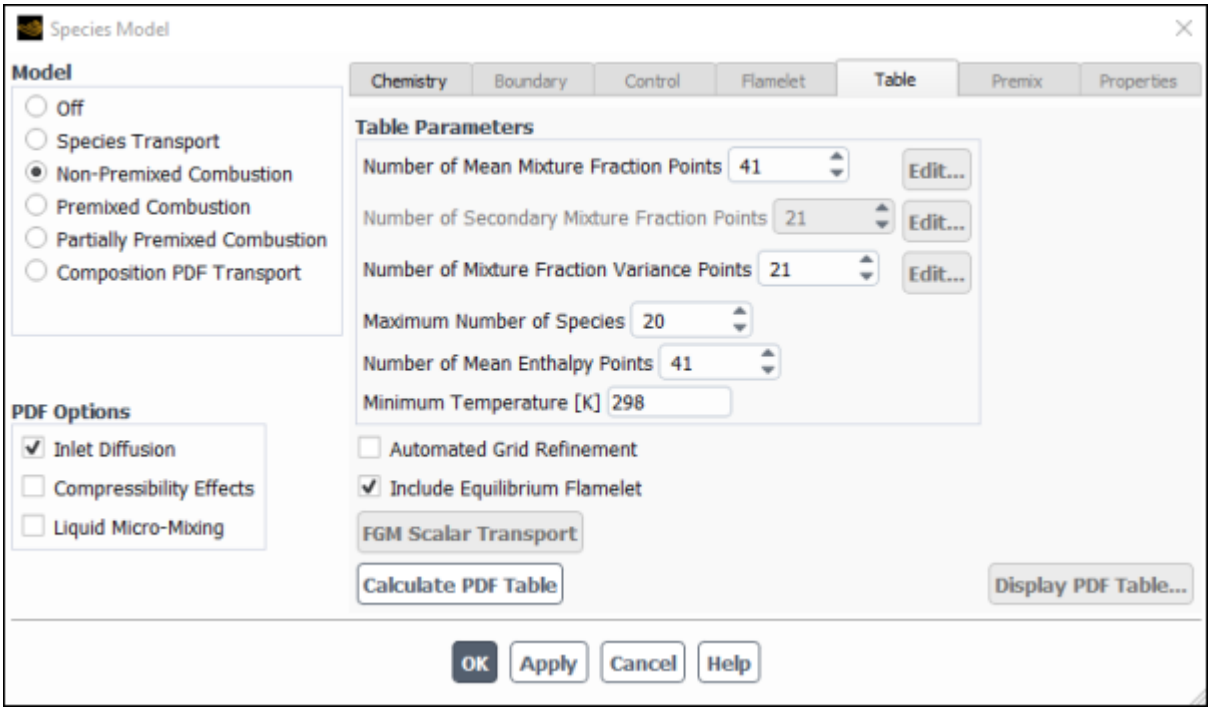
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 20.4: Calculating Steady Diffusion Flamelets



6. Compute the final chemistry look-up table, containing mean values of species fractions, density, and temperature as a function of mean mixture fraction, mixture fraction variance, and possibly enthalpy and scalar dissipation. The contents of this look-up table will reflect your preceding inputs describing the turbulent reacting system (Figure 20.5: Calculating the Chemistry Look-Up Table (p. 2475)).

Figure 20.5: Calculating the Chemistry Look-Up Table



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The look-up table is the stored result of the integration of Equation 8.17 (or Equation 8.25) and Equation 8.19 (in the Theory Guide). The look-up table will be used in Ansys Fluent to determine mean species mass fractions, density, and temperature from the values of mean mixture fraction ($\overline{f}$), mixture fraction variance ($\overline{f'^2}$), and possibly mean enthalpy ($\overline{H}$) and mean scalar dissipation ($\overline{\chi_{st}}$) as they are computed during the Ansys Fluent calculation of the reacting flow. See Look-Up Tables for Adiabatic Systems and Figure 8.8: Visual Representation of a Look-Up Table for the Scalar (Mean Value of Mass Fractions, Density, or Temperature) as a Function of Mean Mixture Fraction and Mixture Fraction Variance in Adiabatic Single-Mixture-Fraction Systems and Figure 8.10: Visual Representation of a Look-Up Table for the Scalar as a Function of Mean Mixture Fraction and Mixture Fraction Variance and Normalized Heat Loss/Gain in Non-Adiabatic Single-Mixture-Fraction Systems in the Theory Guide.

For a problem that includes a secondary stream (and, therefore, a second mixture fraction), you will perform the first two steps listed above for the single-mixture-fraction approach and then prepare a look-up table of instantaneous properties using Equation 8.13 or Equation 8.15 in the Theory Guide.

20.1.2. Setting Up the Equilibrium Chemistry Model

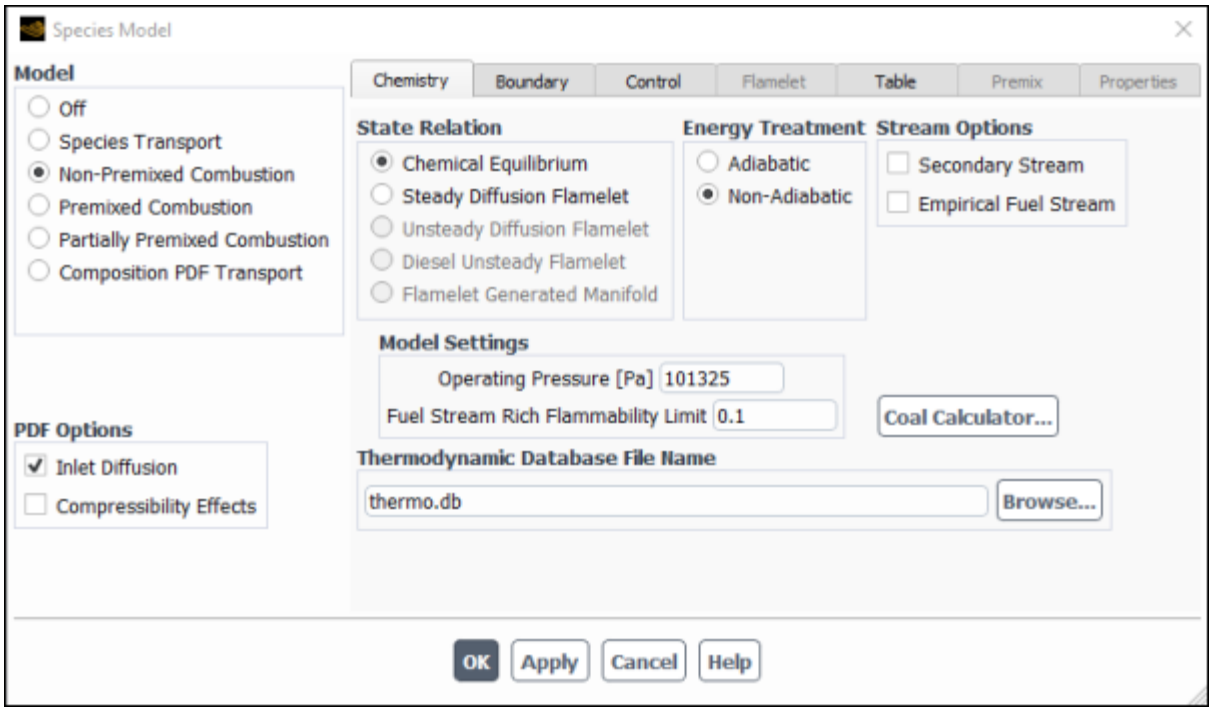
In the equilibrium chemistry model, the concentrations of species of interest are determined from the mixture fraction using the assumption of chemical equilibrium (see Non-Premixed Combustion and Mixture Fraction Theory in the Theory Guide). With this model, you can include the effects of intermediate species and dissociation reactions, producing more realistic predictions of flame temperatures than the Eddy-Dissipation model. When you choose the equilibrium chemistry option, you will have the opportunity to use the rich flammability limit (RFL) option.

To enable the equilibrium chemistry model

1. Select **Non-Premixed Combustion** in the **Species Model** dialog box.
2. Select **Chemical Equilibrium** in the **Chemistry** tab of the **Species Model** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 20.6: The Species Model Dialog Box (Chemistry Tab)



For additional information, see the following sections:

20.1.2.1. Choosing Adiabatic or Non-Adiabatic Options

20.1.2.2. Specifying the Operating Pressure for the System

20.1.2.3. Enabling a Secondary Inlet Stream

20.1.2.4. Choosing to Define the Fuel Stream(s) Empirically

20.1.2.5. Enabling the Rich Flammability Limit (RFL) Option

20.1.2.1. Choosing Adiabatic or Non-Adiabatic Options

You should use the non-adiabatic modeling option if your problem definition in Ansys Fluent will include one or more of the following:

- radiation or wall heat transfer
- multiple fuel inlets at different temperatures
- multiple oxidant inlets at different temperatures
- liquid fuel, coal particles, and/or heat transfer to inert particles

Note that the adiabatic model is a simpler model involving a two-dimensional look-up table in which scalars depend only on $\overline{f}$ and $\overline{f'^2}$ (or on f_{fuel} and p_{sec}). If your model is defined as adiabatic, you will not need to solve the energy equation in Ansys Fluent and the system temperature will be determined directly from the mixture fraction and the fuel and oxidant inlet temperatures. The non-adiabatic case will be more complex and more time-consuming to compute, requiring the generation of three-dimensional look-up tables. However, the non-adiabatic model option allows you to include the types of reacting systems described above.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select **Adiabatic** or **Non-Adiabatic** in the **Chemistry** tab of the **Species Model** dialog box.

20.1.2.2. Specifying the Operating Pressure for the System

The system **Operating Pressure** is used to calculate density using the ideal gas law. For non-adiabatic simulations, the **Compressibility Effects** under **PDF Options** can be enabled to account for cases where substantial pressure changes occur in time and/or space. In such cases it is assumed that the species mass fractions do not change with pressure, and the density is calculated as

$$\overline{\rho}=\rho_{op}\frac{\overline{p}}{p_{op}} \tag{20.1}$$

where ρ_{op} is the density at the specified **Operating Pressure** (p_{op}), and $\overline{p}$ is the local mean pressure in an Ansys Fluent cell.

When the **Compressibility Effects** option is enabled, the flow operating pressure (set in the **Operating Conditions** dialog box) can differ from the Non-Premixed model operating pressure. To distinguish this difference, the **Operating Pressure** name tag in the **Species Model** dialog box changes to **Equilibrium Operating Pressure** when the compressibility effects option is enabled.

Note that for the **Operating Pressure** or **Equilibrium Operating Pressure**, you should specify a value close to the absolute pressure in the system.

See Solution Strategies for Non-Premixed Modeling (p. 2528) for details about enabling compressibility effects.

20.1.2.3. Enabling a Secondary Inlet Stream

If you are modeling a system consisting of a single fuel and a single oxidizer stream, you do not need to enable a secondary stream in your PDF calculation. As discussed in Definition of the Mixture Fraction in the Theory Guide, a secondary stream should be enabled if your PDF reaction model will include any of the following:

- two dissimilar gaseous fuel streams
  - In these simulations, the fuel stream defines one of the fuels and the secondary stream defines the second fuel.
- mixed fuel systems of dissimilar gaseous and liquid fuel
  - In these simulations, the fuel stream defines the gaseous fuel and the secondary stream defines the liquid fuel (or vice versa).
- mixed fuel systems of dissimilar gaseous and coal fuels
  - In these simulations, you can use the fuel stream or the secondary stream to define either the coal or the gaseous fuel. See Modeling Coal Combustion Using the Non-Premixed Model (p. 2491) regarding coal combustion simulations with the non-premixed combustion model.
- mixed fuel systems of coal and liquid fuel

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - In these simulations, you can use the fuel stream or the secondary stream to define either the coal or the liquid fuel. See Modeling Coal Combustion Using the Non-Premixed Model (p. 2491) regarding coal combustion simulations with the non-premixed combustion model.

- coal combustion

  - Coal combustion can be more accurately modeled by using a secondary stream to track the distinct volatile and char off-gases. The fuel stream must define the char and the secondary stream must define the volatile components of the coal. See Modeling Coal Combustion Using the Non-Premixed Model (p. 2491) regarding coal combustion simulations with the non-premixed combustion model.

- a single fuel with two dissimilar oxidizer streams

  - In these simulations, the fuel stream defines the fuel, the oxidizer stream defines one of the oxidizers, and the secondary stream defines the second oxidizer.

To include a secondary stream in your model, turn on the **Secondary Stream** option under **Stream Options** in the **Chemistry** tab.

Important:

Using a secondary stream can substantially increase the calculation time for your simulation since the multi-dimensional PDF integrations are performed at run time. Alternatively, Ansys Fluent can perform a full tabulation of the PDF integrations, as detailed in Full Tabulation of the Two-Mixture-Fraction Model (p. 2514).

When the secondary stream is present, only instantaneous species mass fraction, temperature and mixture properties are stored inside the PDF table. The probability density function (PDF) calculates averaged species mass fraction and mixture properties at the run time. After a PDF table has been generated or read into Ansys Fluent, you can select the shape of the assumed PDF from the **Probability Density Function** drop-down list (**PDF Options** group box):

- **double delta**: as given by Equation 8.21 in the *Fluent Theory Guide*.
- **beta**: as given by Equation 8.22 in the *Fluent Theory Guide*.

20.1.2.4. Choosing to Define the Fuel Stream(s) Empirically

The empirical fuel option provides an alternative method for defining the composition of the fuel or secondary stream when the individual species components of the fuel are unknown. That is, you will define the elemental fraction *not* the individual species. When this option is disabled, you will define the chemical species that are present in each stream and the mass or mole fraction of each species, as described in Defining the Stream Compositions (p. 2487). The option for defining an empirical fuel stream is particularly useful for coal combustion simulations (see Modeling Coal Combustion Using the Non-Premixed Model (p. 2491)) or for simulations involving other complex hydrocarbon mixtures.

To define a fuel or secondary stream empirically

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Turn on the **Empirical Fuel Stream** option under **Stream Options** in the **Chemistry** tab of the **Species Model** dialog box. If you have a secondary stream, enable the **Empirical Secondary Stream** option, or both as appropriate.

2. Specify the appropriate lower heating value (for example **Empirical Fuel Lower Caloric Value**, **Empirical Secondary Lower Caloric Value**), specific heat (**Empirical Fuel Specific Heat**, **Empirical Secondary Specific Heat**), and molecular weight (**Empirical Fuel Molecular Weight**, **Empirical Secondary Molecular Weight**) for each empirically defined stream.

Important:

The empirical definition option is available only with the full equilibrium chemistry model. It cannot be used with the rich flammability limit (RFL) option or the steady and unsteady diffusion flamelet models, since equilibrium calculations are required for the determination of the fuel composition.

Important:

The empirical fuel and secondary molecular weights are only required if your empirical streams are entering the domain via an inlet boundary, or if you are using the partially premixed model. If you are using the non-premixed model and the empirically defined streams originate from the dispersed phase (for example, if you are modeling coal or liquid fuel combustion) the molecular weights are not required for the computation.

20.1.2.5. Enabling the Rich Flammability Limit (RFL) Option

You can define a rich limit on the mixture fraction when the equilibrium chemistry option is used. Input of the rich limit is accomplished by specifying a value of the **Rich Flammability Limit** for the appropriate **Fuel Stream**, **Secondary Stream**, or both. You will not be allowed to specify the **Rich Flammability Limit** if you have used the empirical definition option for fuel composition.

Ansys Fluent will compute the composition at the rich limit using equilibrium. For mixture fraction values above this limit, Ansys Fluent will suspend the equilibrium chemistry calculation and will compute the composition based on mixing, but not burning, of the fuel with the composition at the rich limit. A value of 1.0 for the rich limit implies that equilibrium calculations will be performed over the full range of mixture fraction. When you use a rich limit that is less than 1.0, equilibrium calculations are suspended whenever f, f_{fuel}, or f_{sec} exceeds the limit. This RFL model is often more accurate than the assumption of chemical equilibrium for rich mixtures, and also avoids complex equilibrium calculations, speeding up the preparation of the look-up tables. An RFL value of approximately twice the stoichiometric mixture fraction is appropriate.

For the **Secondary Stream**, the rich flammability limit controls the equilibrium calculation for the secondary mixture fraction. If your secondary stream is not a fuel, you should use an RFL value of 1. A value of 1.0 for the rich limit implies that equilibrium calculations will be performed over the full range of mixture fraction. When you input a rich limit that is less than 1.0, equilibrium calculations

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

are suspended whenever f_{sec} exceeds the limit. (Note that it is the secondary mixture fraction f_{sec} and not the partial fraction p_{sec} that is used here.)

Important:

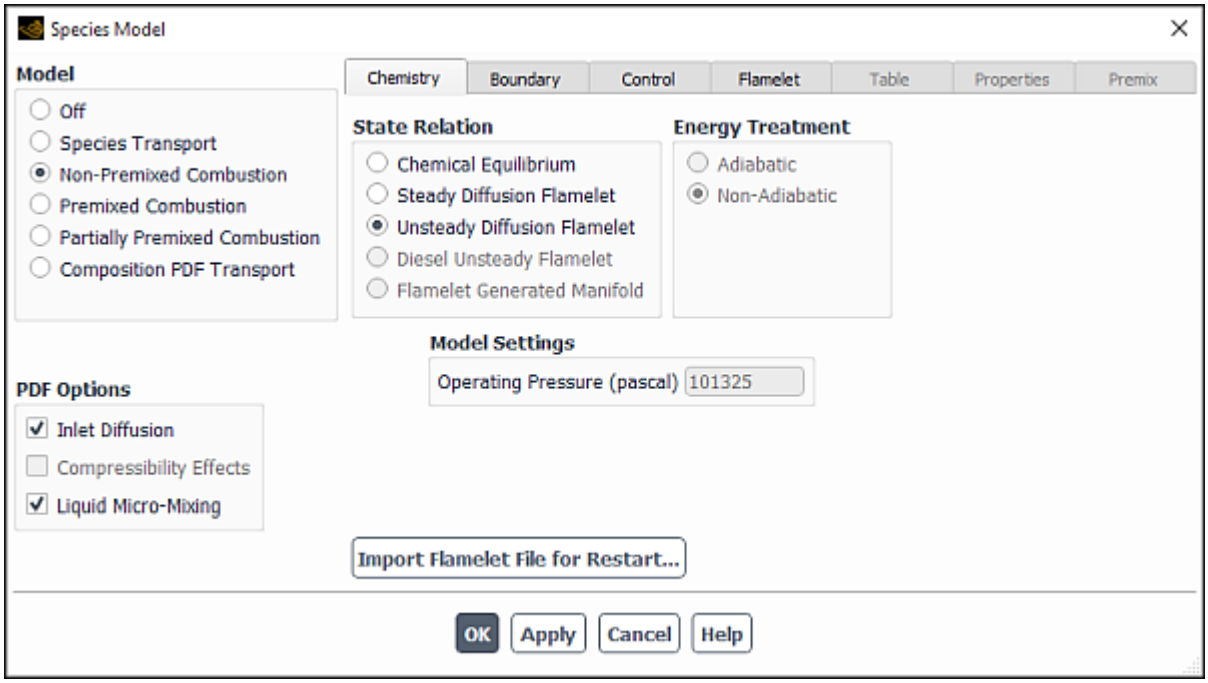
Experimental studies and reviews [20] (p. 5714), [144] (p. 5721) have shown that although the fuel-lean flame region approximates thermodynamic equilibrium, non-equilibrium kinetics will prevail under fuel-rich conditions. Therefore, for non-empirically defined fuels, the RFL model is strongly recommended.

20.1.3. Setting Up the Steady and Unsteady Diffusion Flamelet Models

To enable the diffusion flamelet models

1. Select **Non-Premixed Combustion** in the **Species Model** dialog box.
2. Select **Steady Diffusion Flamelet** or **Unsteady Diffusion Flamelet** in the **Chemistry** tab of the **Species Model** dialog box. See Using the Unsteady Diffusion Flamelet Model (p. 2482).

Figure 20.7: The Chemistry Tab for the Unsteady Diffusion Flamelet Model



For additional information, see the following sections:

20.1.3.1. Choosing Adiabatic or Non-Adiabatic Options

20.1.3.2. Specifying the Operating Pressure for the System

20.1.3.3. Specifying a Chemical Mechanism File for Flamelet Generation

20.1.3.4. Importing a Flamelet

20.1.3.5. Using the Unsteady Diffusion Flamelet Model

20.1.3.6. Using the Diesel Unsteady Laminar Flamelet Model

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

20.1.3.7. Resetting Diesel Unsteady Flamelets

20.1.3.1. Choosing Adiabatic or Non-Adiabatic Options

Select **Adiabatic** or **Non-Adiabatic** in the **Chemistry** tab of the **Species Model** dialog box. See the discussion in Choosing Adiabatic or Non-Adiabatic Options (p. 2477) about the two options.

20.1.3.2. Specifying the Operating Pressure for the System

The system **Operating Pressure** is used to calculate density using the ideal gas law. When the **Compressibility Effects** option is enabled, the name **Operating Pressure** is changed to **Equilibrium Operating Pressure** since the non-premixed combustion model operating pressure can differ from the flow operating pressure. Specifying the Operating Pressure for the System (p. 2478) provides more information about this value.

You can use the steady or unsteady diffusion flamelet model for reactions in liquid systems. To do so, enable **Liquid Micro-Mixing** under **PDF Options**. The **Liquid Micro-Mixing** option is discussed in detail in Liquid Reactions in the Theory Guide.

20.1.3.3. Specifying a Chemical Mechanism File for Flamelet Generation

If you are generating a flamelet file yourself, you will need to read in the chemical kinetic mechanism and thermodynamic data. The mechanism and thermodynamic data must be in CHEMKIN format [79] (p. 5717).

To read in a CHEMKIN mechanism, select the **Create Flamelet** option in the **Chemistry** tab of the **Species Model** dialog box and click the **Import CHEMKIN Mechanism...** button. Using the **Import CHEMKIN Format Mechanism** dialog box, import the gas-phase files into Ansys Fluent as described in Importing a Volumetric Kinetic Mechanism in CHEMKIN Format (p. 2368). Note that for flamelet generation, only a gas-phase kinetic mechanism and a thermodynamic database are required.

Note that the import is limited to mechanisms with 700 or fewer species.

20.1.3.4. Importing a Flamelet

To import an existing flamelet file

1. Select the **Import Flamelet** option in the **Chemistry** tab of the **Species Model** dialog box.
2. (steady diffusion flamelet only) Select either **Standard** or **CFX-RIF** format under **File Type**.
3. Click the **Import Flamelet File...** button. In The Select File Dialog Box (p. 931), select the file (for a single flamelet) or files (for multiple flamelets) to be read in to Ansys Fluent.

After you have completed this step, you can skip ahead to the **Table** tab of the **Species Model** dialog box (see Calculating the Look-Up Tables (p. 2509)).

20.1.3.5. Using the Unsteady Diffusion Flamelet Model

The unsteady diffusion flamelet model can only be enabled from a valid steady-state, steady diffusion flamelet solution. When enabled, the unsteady diffusion flamelet model will change this case to

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

unsteady and postprocess marker probability equations on the frozen flow field. You should hence ensure that the starting steady-state, steady diffusion flamelet solution is fully converged.

When the **Unsteady Diffusion Flamelet** is enabled in the **Chemistry** tab, the **Import Flamelet File for Restart...** button appears in the dialog box, allowing you to run the simulation from a previously saved case, data and unsteady flamelet file.

The Unsteady Diffusion Flamelet Model requires four user inputs in the **Flamelet** tab:

- The **Number of Grid Points in Flamelet**.
- The **Mixture Fraction Lower Limit for Initial Probability**. The initial condition of the marker probability field is unity for all mean mixture fractions above the **Mixture Fraction Lower Limit for Initial Probability**, and zero for mean mixture fractions below it. Note that this should be specified to be greater than the stoichiometric mixture fraction.
- **Maximum Scalar Dissipation**. Flamelets may extinguish at high scalar dissipations because diffusion in the flamelet overwhelms reaction. It is possible to have unrealistically high modeled scalar dissipation in the 2D or 3D Ansys Fluent simulations, which gets transferred to the 1D unsteady flamelet. In order to avoid excessive diffusion in the 1D unsteady flamelet, the instantaneous scalar dissipation in the 1D flamelet is limited to the specified **Maximum Scalar Dissipation**.
- **Courant Number**. The time step size for the unsteady probability marker equation is calculated automatically by Ansys Fluent based on the **Courant Number**. Larger values imply a smaller number of time steps to convect/diffuse the marker probability out of the domain, but also results in a larger numerical error. The **Courant Number** should be small enough so that the unsteady flamelet mean mass fractions are unchanged with any smaller **Courant Number**. The default value of 1 should be sufficient for most applications.
- **Number of Flamelets**. The number of unsteady laminar flamelets that Ansys Fluent will generate during the run. The marker probability equation Equation 8.57 in the *Fluent Theory Guide* will be solved for each flamelet.

When these inputs have been set, clicking the **Initialize Unsteady Flamelet Probability** button initializes the marker probability equation for each flamelet, automatically enabling the **Unsteady** solver, while disabling all equations except the **Unsteady Flamelet Probability** equation in the **Solution Controls** task page. This initialization in the **Flamelet** tab also sets the **Time Step Size** in the **Run Calculation** task page.

Important:

- Do not initialize your solution from the tree or the **Solution Initialization** task page. Note that you are postprocessing a probability field on the frozen steady-state flow field, and by clicking the **Initialize Unsteady Flamelet Probability** button, you have already initialized the probability marker field.
- If you disable the **Unsteady Diffusion Flamelet** model and you want to revert to solving a steady diffusion flamelet simulation, make sure you enable **Steady** (either in the **General** task page or in the tree from the **Setup**/**General**/ **Analysis Type** tree item) and enable all the equations (in the **Solution Controls** task page or in the tree from the **Solution**/**Solution Controls** tree item).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

20.1.3.6. Using the Diesel Unsteady Laminar Flamelet Model

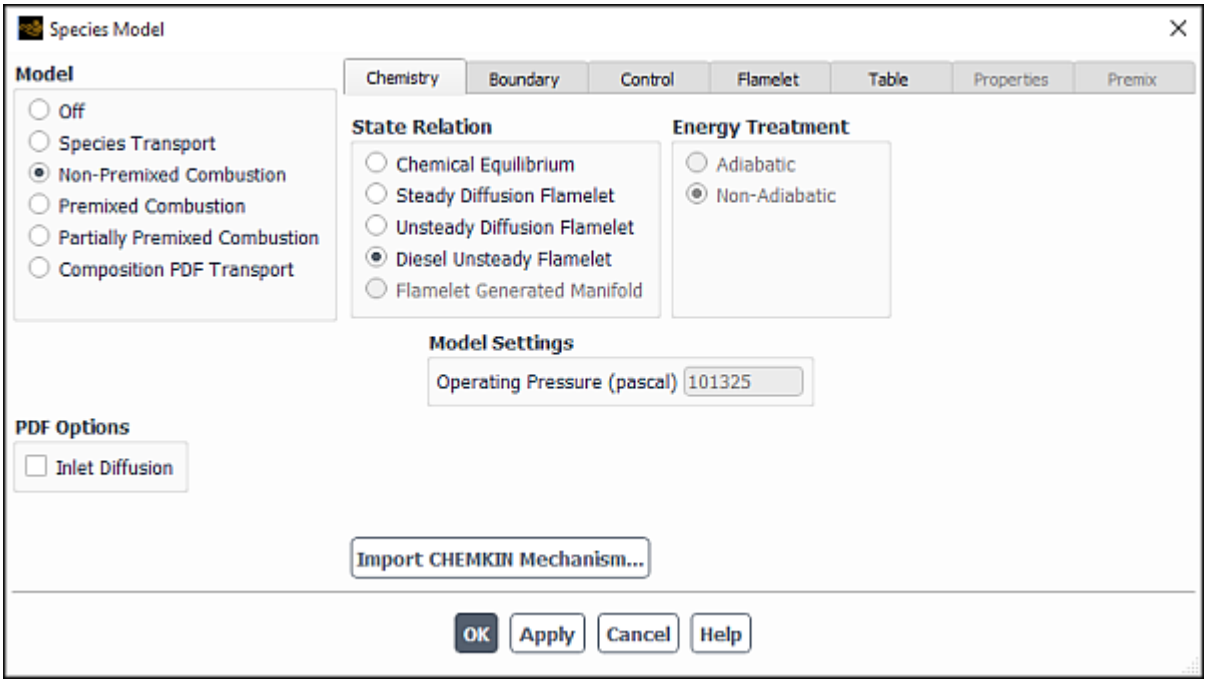
The diesel unsteady laminar flamelet model can only be enabled when conditions for compression-ignition are met:

- The **Transient** solver is selected in the **General** task page (or in the tree from the **Setup**/**General**/**Analysis Type**).
- The **In-Cylinder** dynamic mesh is enabled.
- The Discrete Phase model option **Interaction with Continuous Phase** is selected.

The basic steps for setting the diesel unsteady laminar flamelet models are as follows.

1. In the **Chemistry** tab, select **Diesel Unsteady Flamelet**.

   Figure 20.8: The Enabled Diesel Unsteady Flamelet Model



2. If a detailed chemical mechanism containing kinetic reactions appropriate for compression ignition has not yet been defined in your case, you can import a mechanism in CHEMKIN format as described in Importing a Volumetric Kinetic Mechanism in CHEMKIN Format (p. 2368).

   The mechanism can include pollutant formation reactions as well if you are interested in modeling emissions.

3. In the **Boundary** tab, define the stream compositions as described in Defining the Stream Compositions (p. 2487).

4. In the **Flamelet** tab, set the following flamelet parameters.

   a. Set the **Number of Grid Points in Flamelet**.

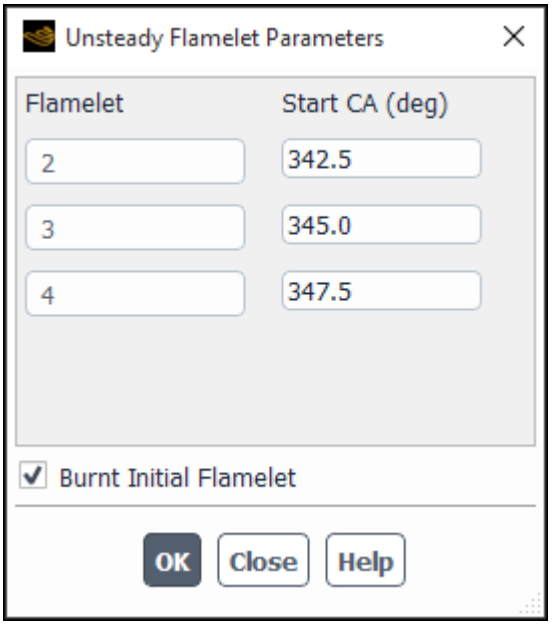
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. Set the **Number of Unsteady Flamelets** that Ansys Fluent will generate during simulation.

c. (for multiple unsteady flamelets) Set the flamelet start times.

Ansys Fluent automatically sets the start time for the first flamelet, but you must set the start time for each consecutive flamelet using the **Unsteady Flamelet Parameters** dialog box. Open it by clicking **Set Flamelet Parameters** and enter the start time for each flamelet either in seconds or in crank angles if the dynamic mesh is enabled.

Figure 20.9: The Unsteady Flamelet Parameters Dialog Box



Ansys Fluent starts simulation with only one flamelet, and then it automatically introduces new flamelets into the reacting domain at the times you have specified.

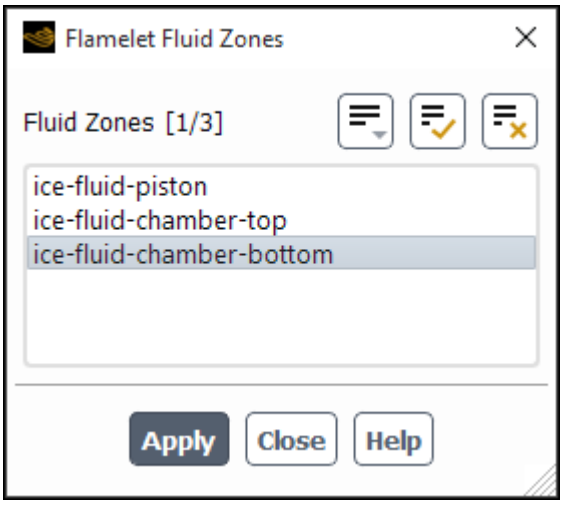
d. The default initial condition for an unsteady flamelet is unburnt. Ansys Fluent provides the **Burnt Initial Flamelet** option that allows you to set the initial flamelet condition to a chemical equilibrium burnt state. This option is useful if you are modeling internal combustion engines where residual gases may be present in the cylinder before the spray is injected, which would be incorrectly modeled by the unburnt state. Note that the **Burnt Initial Flamelet** is only used at case initialization.

e. (optional) Specify the flamelet fluid zones.

Ansys Fluent calculates diesel unsteady flamelets using the zone-averaged pressure and scalar dissipation at every time step. By default, the averaging is performed over all fluid zones in the domain, but you can also select and/or deselect the fluid zones using the **Flamelet Fluid Zones** dialog box. Open this dialog box by clicking **Set Flamelet Fluid Zones** and select the fluid zones to be used for calculating average pressure and scalar dissipation. If no fluid zone is selected, Ansys Fluent will compute domain average pressure and scalar dissipation using all fluid zones.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 20.10: The Flamelet Fluid Zones Dialog Box



Note:

For internal combustion cases, it is recommended that you select the cylinder fluid zones and deselect the intake and exhaust fluid zones.

Note that Ansys Fluent calculates flamelets at every time step of the run. For this reason, the option to calculate the flamelets as a preprocessing step before running your simulation is unavailable, and **Calculate Flamelets** appears dimmed.

5. In the **Table** tab, set the PDF table parameters as described in Calculating the Look-Up Tables (p. 2509).

   Note that Ansys Fluent calculates the PDF table at every time step of the run. For this reason, the option to calculate the PDF table as a preprocessing step before running your simulation is unavailable, and **Calculate PDF Table** appears dimmed.

20.1.3.6.1. Recommended Settings for Internal Combustion Engines

When setting up and using the Diesel Unsteady Flamelet model for internal combustion engine simulations, the following recommendations apply:

1. Number of Flamelets

   a. You must specify at least two diesel unsteady flamelets. Ansys Fluent will use the first flamelet to model trapped burnt gases from the previous cycle. The second flamelet will start at the crank angle (CA) of fuel injection, specified in the **Unsteady Flamelet Parameters** dialog box (see Figure 20.10: The Flamelet Fluid Zones Dialog Box (p. 2486)).

   b. To model split injections where an initial fuel mass is injected and burns before the main fuel injection, three or more unsteady flamelets are required.

2. Flamelet Initialization

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

By default, the flamelet is initialized as mixed-but-unburnt. However, in all practical scenarios there is always some trapped gas remaining inside the cylinder. Therefore, it is recommended that you use the **Burnt Initial Flamelet** option in the **Unsteady Flamelet Parameters** dialog box (see Figure 20.10: The Flamelet Fluid Zones Dialog Box (p. 2486)). When this option is selected, Ansys Fluent performs a constant temperature equilibrium calculation and sets the initial flamelet condition to a chemical equilibrium burnt state.

3. Multi-cycle simulations

   a. To accurately model multiple cycles of internal combustion engines, the flamelets must be reset at the end of each cycle. This is performed by defining the **Diesel Unsteady Flamelet Reset** event, typically at the specified crank angle, just before the inlet valve opens. Refer to the Resetting Diesel Unsteady Flamelets (p. 2487) for details. This approach is recommended for modeling the EGR trapped gases with the first burnt unsteady flamelet.

   b. If you are using the Inert (EGR) model in order to track the trapped inert mixture, you need to define the **Inert EGR Reset** event at the specified crank angle just before the inlet valve opens. See Resetting Inert EGR (p. 2525) for details.

20.1.3.7. Resetting Diesel Unsteady Flamelets

In order to simulate multiple cycles in internal combustion engines, flamelets should be reset at the end of every cycle. In addition, the burned trapped gases must be modeled, which can be done in one of the two ways. The first and recommended approach is to use the **Diesel Unsteady Flamelet Reset** option. The second approach is to use the inert (EGR) model and the **Inert EGR Reset** option. You can access the **Diesel Unsteady Flamelet Reset** and **Inert EGR Reset** options via the **Dynamic Mesh Events** dialog box. There, you need to set the crank angle at which this event occurs (usually shortly before the inlet valves open) and the participating fluid zones (usually only the combustion chamber and not the intake and exhaust port zones).

When the **Diesel Unsteady Flamelet Reset** event is executed, all flamelets are deleted and a new flamelet is introduced with a state set to the probability-weighted average condition of all flamelets present before reset. The other new flamelets are introduced during a new cycle in a similar fashion to that described in Using the Diesel Unsteady Laminar Flamelet Model (p. 2484).

When the inert EGR reset event is executed with the diesel unsteady flamelet model, the burnt gas in the selected Inert EGR Reset zones is converted to inert, all flamelets are deleted, and a new unburnt flamelet is introduced into the domain.

Note:

The **Diesel Unsteady Flamelet Reset** option is available only when the selected number of flamelets is greater than one.

20.1.4. Defining the Stream Compositions

In modeling a non-premixed combustion problem, you will only specify the boundary species (that is, the fuel, oxidizer, and if necessary, secondary stream species). The intermediate and product species will be determined automatically.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

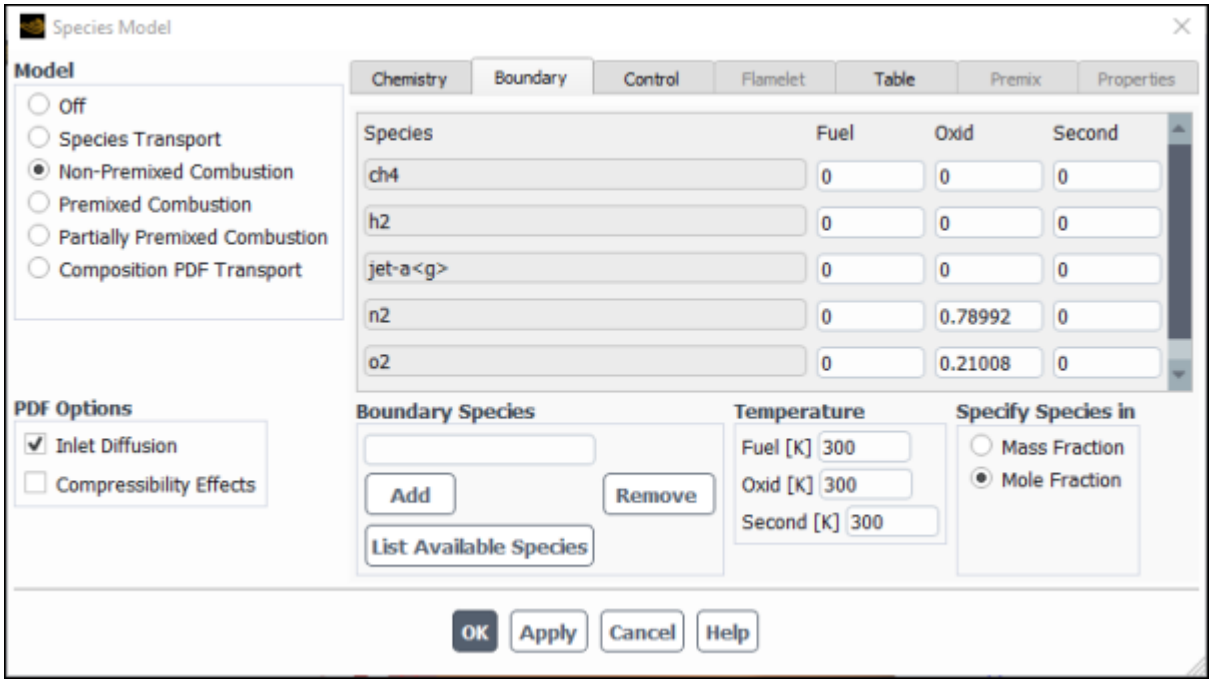
Ansys Fluent provides you with an initial list of common boundary species (ch4, h2, jet-a<g>, n2 and o2). If your fuel and/or oxidizer is composed of different species, you can add them to the boundary **Species** list. All boundary species must exist in the chemical database and you must enter their names in the same format used in the database, otherwise an error message will be issued.

After defining the boundary species that will be considered in the reaction system, you must define their mole or mass fractions at the fuel and oxidizer inlets and at the secondary inlet, if one exists. (If you choose to define the fuel or secondary stream composition empirically, you will instead enter the parameters described at the end of this section.) For the example shown in Figure 8.12: Chemical Systems That Can Be Modeled Using a Single Mixture Fraction in the Theory Guide, for example, the fuel inlet consists of 60% CO_4, 20% CO, 10% CO_2, and 10% C_3H_8.

Finally, the inlet stream temperatures of your reacting system are required for construction of the look-up table and computation of the equilibrium chemistry model.

For the equilibrium chemistry model, the species names are entered using the **Boundary** tab in the **Species Model** dialog box (Figure 20.11: The Species Model Dialog Box (Boundary Tab) (p. 2488)). If you are generating a steady or unsteady diffusion flamelet, the list of boundary species will be automatically filled as all the species in the CHEMKIN mechanism, and you will be unable to change these.

Figure 20.11: The Species Model Dialog Box (Boundary Tab)



The steps for adding new species and defining their compositions are as follows:

1. (chemistry equilibrium chemistry model only) If your fuel, oxidizer, or secondary streams are composed of species other than the default species list, type the chemical formula (for example, so2 or S02 for SO_2) under **Boundary Species** and click **Add**. The species will be added to the **Species** list. Continue in this manner until all of the boundary species you want to include are shown in the **Species** list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To remove a species from the list, type the chemical formula under **Boundary Species** and click **Remove**. To print a list of all species in the thermodynamic database file (`thermo.db`) in the console window, click **List Available Species**.

2. (non-equilibrium chemistry models) Optionally, select the boundary species as described in Overview of the Problem Setup Procedure (p. 2472).

3. Under **Specify Species In**, specify whether you want to enter the **Mass Fraction** or **Mole Fraction**. **Mass Fraction** is the default.

4. For each relevant species in the **Species** list, specify its mass or mole fraction for each stream (**Fuel**, **Oxid**, or **Second** as appropriate) by entering values in the table. Note that if you change from **Mass Fraction** to **Mole Fraction** (or vice versa), all values will be automatically converted if they sum to 0 or 1, so be sure that you are entering either all mass fractions or all mole fractions as appropriate. If the values do not sum to 0 or 1, an error will be issued.

5. Under **Temperature**, specify the following inputs:

   Fuel

   is the temperature of the fuel inlet in your model. In adiabatic simulations, this input (together with the oxidizer inlet temperature) determines the inlet stream temperatures that will be used by Ansys Fluent. In non-adiabatic systems, this input should match the inlet thermal boundary condition that you will use in Ansys Fluent (although you will enter this boundary condition again in the Ansys Fluent session). If your Ansys Fluent model will use liquid fuel or coal combustion, define the inlet fuel temperature as the temperature at which vaporization/de-volatilization begins (that is, the **Vaporization Temperature** specified for the discrete-phase material—see Setting Material Properties for the Discrete Phase (p. 2811)). For such non-adiabatic systems, the inlet temperature will be used only to adjust the look-up table grid (for example, the discrete enthalpy values for which the look-up table is computed). Note that if you have more than one fuel inlet, and these inlets are not at the same temperature, you must define your system as non-adiabatic. In this case, you should enter the fuel inlet temperature as the value at the dominant fuel inlet.

   Oxid

   is the temperature of the oxidizer inlet in your model. The issues raised in the discussion of the input of the fuel inlet temperature (directly above) pertain to this input as well.

   Second

   is the temperature of the secondary stream inlet in your model. (This item will appear only when you have defined a secondary inlet.) The issues raised in the discussion of the input of the fuel inlet temperature (directly above) pertain to this input as well.

For additional information, see the following sections:

20.1.4.1. Setting Boundary Stream Species

20.1.4.2. Modifying the Database

20.1.4.3. Composition Inputs for Empirically-Defined Fuel Streams

20.1.4.4. Modeling Liquid Fuel Combustion Using the Non-Premixed Model

20.1.4.5. Modeling Coal Combustion Using the Non-Premixed Model

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

20.1.4.1. Setting Boundary Stream Species

In combustion, a large number of intermediate and product species may be produced from a small number of initial boundary species. In Ansys Fluent you must specify only the species composition of your boundary species in the fuel, oxidizer, and (if appropriate) secondary streams. Ansys Fluent will calculate all intermediate and product species automatically. The following suggestions may be helpful in the definition of the system chemistry:

- For coal combustion, char in the coal should be represented by C(s).

  Important:

  Care should be taken to distinguish atomic carbon, C, from solid carbon, C(s). Atomic carbon should be selected only if you are using the empirically-defined input method.

- If your fuel composition is known empirically (for example, $C_{0.9}$ H_3 $O_{0.2}$), use the option for an empirically-defined stream (see below).
- If you want to include the sulfur that may be present in a hydrocarbon fuel, note that this may hinder the convergence of the equilibrium solver, especially if the concentration of sulfur is small. It is therefore recommended that you include sulfur in the calculation only if it is present in considerable quantities.

20.1.4.1.1. Including Condensed Species

In addition to gaseous species, liquid and solid species can be included in the chemistry calculations. They are often indicated by an "l" or an "s" in parentheses after the species name. If you add a condensed species to the equilibrium chemical system, its density will be retrieved from Ansys Fluent's chemical property database file `propdb.scm` if you are using the thermodynamic database file `thermo.db` that is also supplied with Ansys Fluent. If you are using a custom thermodynamic database file and want to include a condensed species in the equilibrium system that does not exist in `propdb.scm`, a density of 1000 kg/m^3 will be assumed. The condensed species density can be changed in the Create/Edit Materials Dialog Box (p. 4885) after the PDF table has been calculated. If you modify the condensed species density in this manner, you will then need to recalculate the PDF table.

20.1.4.2. Modifying the Database

If you want to include a new species in your reacting system that is not available in the chemical database, you can add it to the database file, `thermo.db`. The format for `thermo.db` is detailed in [79] (p. 5717). If you choose to modify the standard database file, you should create copies of the original file.

20.1.4.3. Composition Inputs for Empirically-Defined Fuel Streams

As mentioned in Defining the Problem Type (p. 2472), you can define the composition of a fuel stream (that is, the standard fuel or a secondary fuel) empirically. For an empirically-defined stream, you will need to enter the atomic mass or mole fractions in addition to the inputs for lower caloric (heating) value of the fuel and the mean specific heat of the fuel that were described previously.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The heat of formation of an empirically defined stream is calculated from the heating value and the atomic composition. The fuel inlet temperature and fuel specific heat are used to calculate the sensible enthalpy. The molecular weight is used for the computation of the unburnt stream density. Note that the unburnt density is only required if the stream enters via an inlet boundary, or if you are using the partially-premixed model.

When an empirically-defined fuel or secondary stream is specified in the **Chemistry** tab (equilibrium chemistry model only) of the **Species Model** dialog box, you must specify the atom fractions of C, H, O, N, and S in that stream instead of the species mass or mole fractions. To avoid confusion, the species fraction inputs for an empirically-defined stream will be grayed out in the table within the **Boundary** tab, leaving only the fields for atom fractions (that is, **c**, **h**, **o**, **n**, and **s**).

20.1.4.4. Modeling Liquid Fuel Combustion Using the Non-Premixed Model

Liquid fuel combustion can be modeled with the discrete phase and non-premixed models. In Ansys Fluent, the fuel vapor, which is produced by evaporation of the liquid fuel, is defined as the fuel stream. (See Defining the Stream Compositions (p. 2487).) The liquid fuel that evaporates within the domain appears as a source of the mean fuel mixture fraction.

Within Ansys Fluent, you define the liquid fuel discrete-phase model in the usual way. The gas phase (oxidizer) flow inlet is modeled using an inlet mixture fraction of zero and the fuel droplets are introduced as discrete phase injections (see Setting Initial Conditions for the Discrete Phase (p. 2718)). The property inputs for the liquid fuel droplets are unaltered by the non-premixed model (see Setting Material Properties for the Discrete Phase (p. 2811)). Note that when you are requested to input the gas phase species destination for the evaporating liquid, you should specify the species that make up the evaporating stream.

If the fuel stream was defined as a mixture of components, you should select the largest of these components as the "evaporating species". Ansys Fluent will ensure that the mass evaporated from the liquid droplet enters the gas phase as a source of the fuel mixture that you defined. The evaporating species you select here is used only to compute the diffusion controlled driving force in the evaporation rate.

20.1.4.5. Modeling Coal Combustion Using the Non-Premixed Model

If your model involves coal combustion, the fuel and secondary stream compositions can be input in one of several ways. You can use a single mixture fraction (fuel stream) to represent the coal, defining the fuel composition as a mixture of volatiles and char (solid carbon). Alternatively, you can use two mixture fractions (fuel and secondary streams), defining the volatiles and char separately. In two-mixture-fraction models for coal combustion, the fuel stream represents the char and the secondary stream represents volatiles. This section describes the modeling options and special input procedures for coal combustion models using the non-premixed approach.

There are three options for coal combustion:

- When coal is the only fuel in the system, you can model the coal using two mixture fractions, where the primary stream represents the char and the secondary stream represents the volatiles. Generally, the char stream composition is defined as 100% C(s). The volatile stream composition

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

is defined by selecting appropriate species and setting their mole or mass fractions. Alternatively, you can use the empirical method (input of atom fractions) for defining these compositions.

Important:

Using two mixture fractions to model coal combustion is more accurate than using one mixture fraction as the volatile and char streams are modeled separately. However, the two-mixture-fraction model incurs significant additional computational expense since the multi-dimensional PDF integrations are performed at run time.

- When coal is the only fuel in the system, you can choose to model the coal using a single mixture fraction (the fuel stream). When this approach is adopted, the fuel composition you define includes both volatile species and char. Char is typically represented by including C(s) in the species list. You can define the fuel stream composition by selecting appropriate species and setting their mole fractions, or by using the empirical method (input of atom fractions). Definition of the composition is described in detail below.

  Important:

  Using a single mixture fraction for coal combustion is less accurate than using two mixture fractions. However, convergence in Ansys Fluent should be substantially faster than the two-mixture-fraction model.

- When coal is used with another (gaseous or liquid) fuel of different composition, you must model the coal with one mixture fraction and use a second mixture fraction to represent the second (gaseous or liquid) fuel. The stream associated with the coal composition is defined as detailed below for single-mixture-fraction models.

20.1.4.5.1. Defining the Coal Composition: Single-Mixture-Fraction Models

When coal is modeled using a single mixture fraction (the fuel stream), the fuel stream composition can be input using the conventional approach or the empirical fuel approach.

- Conventional approach:

  To use the conventional approach, you will need to define the mixture of species in the coal and their mole or mass fractions in the fuel stream. Use the **Boundary** tab in the **Species Model** dialog box to input the list of species (for example, C_3H_8, CH_4, CO, CO_2, C(s)) that approximate the coal composition, and their mole or mass fractions.

  Note that C(s) is used to represent the char content of the coal. For example, consider a coal that has a molar composition of 40% volatiles and 60% char on a dry ash free (DAF) basis. Assuming the volatiles can be represented by an equimolar mixture of C_3H_8 and CO, the fuel stream composition defined in the **Boundary** tab would be C_3H_8=0.2, CO = 0.2, and C(s)=0.60. Note that the coal composition should always be defined on an ash-free basis, even if ash will be considered in the Ansys Fluent calculation.

  To define ash properties, go to the **Create/Edit Materials** dialog box and select **combusting-particle** as the **Material Type**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following table illustrates the conversion from a typical mass-based proximate analysis to the species fraction inputs required by Ansys Fluent. Note that the conversion requires that you make an assumption regarding the species representing the volatiles. Here, the volatiles are assumed to exist as an equimolar mix of propane and carbon monoxide.

Proximate Analysis	Weight %	Mass Fraction (DAF)	Moles (DAF)	Mole Fraction (DAF)
Volatiles	30			
$-C_3H_8$		0.2035	0.004625	0.07134
– CO		0.1295	0.004625	0.07134
Fixed Carbon (C(s))	60	0.667	0.05558	0.85732
Ash	10	-	-	-
(Total)			0.06483	1.0

Moisture in the coal can be considered by adding it in the fuel composition as liquid water, $H_2O(l)$. The moisture can also be defined as water vapor, H_2O, provided that the corresponding latent heat is included in the discrete phase material inputs in Ansys Fluent. If the liquid water is used as a boundary species, it should be removed from the list of excluded species (see Forcing the Exclusion and Inclusion of Equilibrium Species (p. 2499)).

Important:

Note that if water is included in the coal, the water release is not modeled as evaporation, which is typically the case in the wet combustion model, described in Particle Types (p. 2722).

- Empirical fuel approach:

  To use the empirical approach, enable the **Empirical Fuel Stream** option in the **Chemistry** tab. This method is ideal if you have an elemental analysis of the coal.

  In the **Chemistry** tab, enter the lower heating value and mean specific heat of the coal. Ansys Fluent will use these inputs to determine the mole fractions of the chemical species you have included in the system. Then, in the **Boundary** tab, define the atom fractions of C, H, N, S, and O in the fuel stream.

Note that for both of these composition input methods, you should take care to distinguish atomic carbon, C, from solid carbon, C(s). Atomic carbon should only be selected if you are using the empirical fuel input method.

See Additional Coal Modeling Inputs in Ansys Fluent (p. 2495) for details about further inputs for modeling coal combustion.

20.1.4.5.2. Defining the Coal Composition: Two-Mixture-Fraction Models

You can model coal using the two mixture fractions model, where the primary stream represents the char and the secondary stream represents the volatiles.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

As in single-mixture-fraction cases, the fuel stream and secondary stream compositions in a two-mixture-fraction case can be entered using either the conventional approach or the empirical fuel approach.

- Conventional approach:

  To use the conventional approach, you will need to define the mixture of species in the coal and their mole or mass fractions in the fuel and secondary streams.

  Use the **Boundary** tab of **Species Model** dialog box to define the mole or mass fractions of volatile species in the secondary stream (for example, C_3H_8, CH_4, CO, CO_2, C(s)). Next, define the mole or mass fractions of species used to represent the char. Generally, you will specify 100% C(s) for the fuel stream.

- Empirical fuel approach:

  To use the empirical fuel approach, enable the **Empirical Secondary Stream** option in the **Chemistry** tab for the volatile (secondary) stream. This method is ideal if you have an elemental analysis of the coal.

  In the **Chemistry** tab, input the lower heating value and mean specific heat of the coal volatiles. Then, in the **Boundary** tab, define the mole or mass fractions of species used to represent the char. Generally, you will specify 100% C(s) for the fuel stream. Finally, define the atom fractions of C, H, N, S, and O in the volatiles. Ansys Fluent will use these inputs to determine the mole fractions of the chemical species you have included in the system. For example, consider coal with the following DAF (dry ash free) data and elemental analysis:

Proximate Analysis	Wt % (dry)	Wt % (DAF)
Volatiles	28	30.4
Char (C(s))	64	69.6
Ash	8	-

Element	Wt % (DAF)	Wt % (DAF)
C	89.3	89.3
H	5.0	5.0
O	3.4	3.4
N	1.5	2.3
S	0.8	-

(Note that in the final column, for modeling simplicity, the sulfur content of the coal has been combined into the nitrogen mass fraction.)

You can combine the proximate and ultimate analysis data to yield the following elemental composition of the volatile stream:

Element	Mass	Mass Fraction	Moles	Mole Fraction
C	(89.3 - 69.6)	0.65	5.4	0.24
H	5.0	0.16	16	0.70

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Element	Mass	Mass Fraction	Moles	Mole Fraction
O	3.4	0.11	0.7	0.03
N	2.3	0.08	0.6	0.03
Total	30.4		22.7	

This adjusted composition is used to define the secondary stream (volatile) composition.

The lower heating value of the volatiles can be computed from the known heating value of the coal and the char (DAF):

$$LCV_{coal,DAF}=35.3\,\mathrm{MJ/kg}$$

$$LCV_{char,DAF}=32.9\,\mathrm{MJ/kg}$$

You can compute the heating value of the volatiles as

$$LCV_{vol}=\frac{35.3\,\mathrm{MJ/kg}-0.696\times 32.9\,\mathrm{MJ/kg}}{0.304} \tag{20.2}$$

or

$$LCV_{vol}=40.795\,\mathrm{MJ/kg} \tag{20.3}$$

Note that for both of these composition input methods, you should take care to distinguish atomic carbon, C, from solid carbon, C(s). Atomic carbon should only be selected if you are using the empirical fuel input method.

20.1.4.5.3. Additional Coal Modeling Inputs in Ansys Fluent

Within Ansys Fluent, the DPM coal combustion simulation is defined as usual when the non-premixed combustion model is selected. The air (oxidizer) inlets are defined as having a mixture fraction value of zero. No gas phase fuel inlets will be included and the sole source of fuel will come from the coal devolatilization and char burnout. The coal particles are defined as injections using the **Set Injection Properties** dialog box in the usual way, and physical properties for the coal material are specified as described in Setting Material Properties for the Discrete Phase (p. 2811). Remember the following issues when you are defining injections and discrete-phase material properties for coal materials:

- In the **Set Injection Properties** dialog box, you will specify for the **Oxidizing Species** one of the components of the oxidizer stream. This species concentration field will be used to calculate the diffusion-controlled driving force in the char burnout law (if applicable), and is O_2 by default.

  The specification of the char and volatile streams differs depending on the type of model you are defining:

  – If coal is modeled using a single mixture fraction, the gas phase species representing the volatiles and the char combustion are represented by the mixture fraction used by the non-premixed combustion model.

  – If coal is modeled using two mixture fractions, rather than specifying a destination species for the volatiles and char, you will instead specify the **Devolatilizing Stream** (as secondary) and the **Char Stream** (as primary).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - If coal is modeled using one mixture fraction, and another fuel is modeled using a second mixture fraction, you should specify the stream representing the coal as both the **Devolatilizing Stream** and the **Char Stream**.

- In the **Create/Edit Materials** dialog box, **Vaporization Temperature** should be set equal to the fuel inlet temperature. This temperature controls the onset of the devolatilization process. The fuel inlet temperature that you define in the **Boundary** tab of the **Species Model** dialog box should be set to the temperature at which you want to initiate devolatilization. This way, the look-up tables will include the appropriate temperature range for your process.

- In the **Create/Edit Materials** dialog box, **Volatile Component Fraction** and **Combustible Fraction** should be set to values that are consistent with the coal composition used to define the fuel (and secondary) stream composition.

- Also in the **Create/Edit Materials** dialog box, you will be prompted for the **Burnout Stoichiometric Ratio** and for the **Latent Heat**. The **Burnout Stoichiometric Ratio** is used in the calculation of the diffusion controlled burnout rate but has no other impact on the system chemistry when the non-premixed combustion model is used. The **Burnout Stoichiometric Ratio** is the mass of oxidant required per mass of char. The default value of 2.67 assumes that C(s) is oxidized by O_2 to yield CO_2. The **Latent Heat** input determines the heat required to generate the vapor phase volatiles defined in the non-premixed system chemistry. You can usually set this value to zero when the non-premixed model is used, since your definition of volatile species will have been based on the overall heating value of the coal. However, if the coal composition includes the water content, the latent heat should be set as follows:

  - Set the latent heat to zero if the water content of the coal has been defined as **H2O(L)**. In this case, the system chemistry will include the latent heat required to vaporize the liquid water.

  - Set the latent heat to the value for water (2.25×10^6 J/kg), adjusted by the mass loading of water in the volatiles, if the water content of the coal has been defined using water vapor, **H2O**. In this case, the water content you defined will be evolved along with the other species in the coal but the system chemistry does not include the latent heat effect.

- The **Density** you define for the coal in the **Create/Edit Materials** dialog box should be the apparent density, including ash content.

- You will not be asked to define the **Heat of Reaction for Burnout** for the char combustion.

20.1.4.5.4. Postprocessing Non-Premixed Models of Coal Combustion

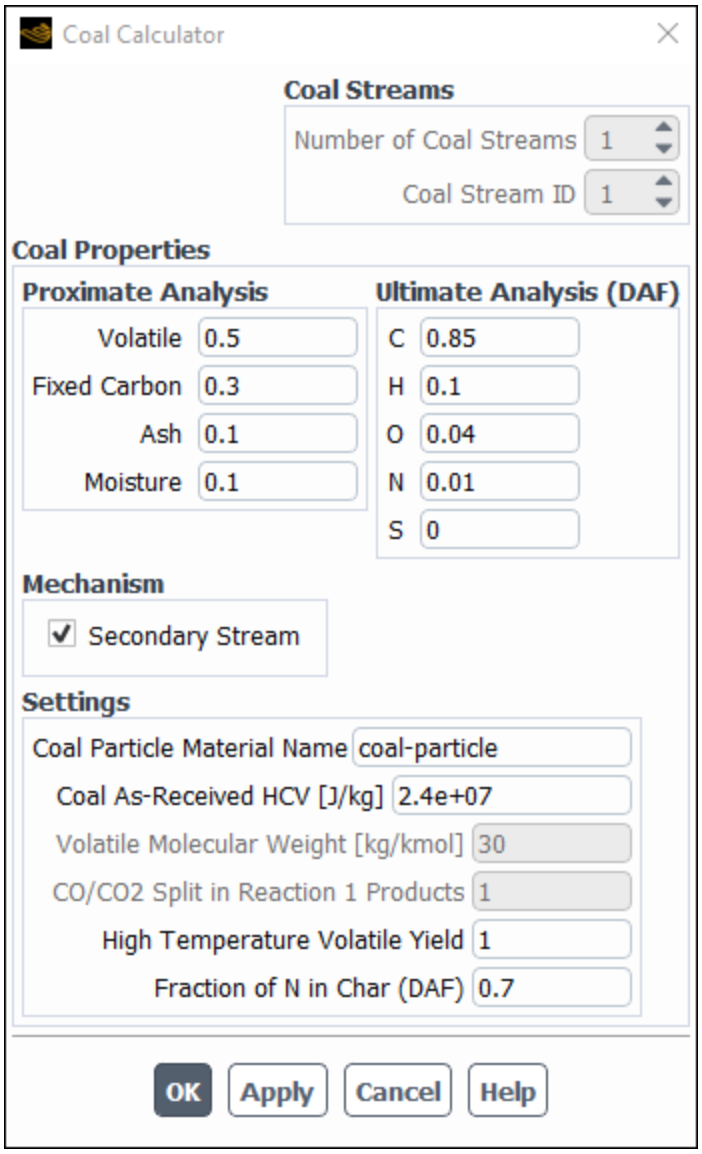
Ansys Fluent reports the rate of volatile release from the coal using the **DPM Evaporation/Devolatilization** postprocessing variable. The rate of char burnout is reported in the **DPM Burnout** variable.

20.1.4.5.5. The Coal Calculator

The **Coal Calculator** dialog box automates the calculations described above for setting up a coal case from the proximate and ultimate analyses.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 20.12: The Coal Calculator Dialog Box



The inputs to the **Coal Calculator** dialog box are:

1. Coal **Proximate Analysis**, which is the mass fraction of **Volatile, Fixed Carbon, Ash** and **Moisture** in the coal.

2. Coal **Ultimate Analysis**, which is the mass fraction of atomic **C, H, O, N** and optionally **S**, in the Dry-Ash-Free (DAF) coal.

3. The option to use a **Secondary Stream**. If enabled, the two mixture fraction model will be set with the primary stream representing char as $C<s>$, and an empirical secondary stream representing the volatiles.

4. The **Coal Particle Material Name**. A DPM Combusting Particle Material will be created with this name. The default name is *coal-particle*.

5. The **Coal As-Received HCV** (Higher Calorific Value).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. The **High Temperature Volatile Yield**. Proximate analyses are generally done with slower heating rates and lower temperatures than would occur in a real flame. Therefore, enhanced devolatization at higher temperatures can cause the volatile yield to exceed the proximate analysis fraction. To model this, the actual volatile fraction used is calculated as that specified in the **Proximate Analysis** input multiplied by the **High Temperature Volatile Yield**. The actual **Fixed Carbon** fraction is then calculated as one minus the sum of the actual **Volatile**, **Ash**, and **Moisture** fractions.

7. **Fraction of N in Char (DAF)**. This input is used in calculating the split of atomic nitrogen for the Fuel NOx model.

   When the **OK** button is clicked, Ansys Fluent makes the following changes:

   a. The empirical fuel atomic compositions in the **Boundary** tab are set, and the **Non-Adiabatic** model is enabled as required for DPM. The Empirical Fuel Lower Calorific Value is calculated as follows. First the DAF LCV of the coal is computed as,

   $$LCV_{coal}^{DAF} = \frac{HCV_{coal}^{ar} - h_{H_2O}^{latent} Moisture}{1 - Moisture - Ash} - \frac{H_{ar} W_{H_2O}}{2 W_H} h_{H_2O}^{latent} \tag{20.4}$$

   where $Moisture$ and Ash are the proximate moisture and ash fractions, H_{ar} is the ultimate H fraction, W_{H_2O} and W_H are the molecular weight of water and atomic hydrogen, respectively, and $h_{H_2O}^{latent}$ is the latent heat of water.

   If you are using the **Secondary Stream** option, LCV_{vol} is calculated from LCV_{coal}^{DAF} using,

   $$LCV_{vol} = \frac{LCV_{coal}^{DAF}\left(1 - Moisture - Ash\right) - LCV_{char} FixedCarbon}{Volatile} \tag{20.5}$$

   where $FixedCarbon$ and $Volatile$ are the proximate fixed carbon and volatile fractions, respectively.

   b. A combusting particle material is created with **Volatile Component Fraction** and **Combustible Fraction** calculated from the ultimate and proximate analyses. The Discrete Phase Model (DPM) is enabled.

   c. For the Fuel NOx model, the char N conversion is set to *NO*, and the Fuel NOx **Volatile** and **Char** mass fractions are set according to the ultimate and proximate compositions. Note that even though some of the Fuel NOx parameters are changed, the Fuel NOx model itself is not enabled.

After the **Coal Calculator** has set up the relevant models, you must build the PDF Table by clicking **Calculate PDF Table** in the **Table** tab. You will also need to create injections if you have not done this yet. After converging your coal combustion case, you may want to enable the NOx model for postprocessing nitrogen-oxide pollutants.

20.1.5. Setting Up Control Parameters

For information about setting up control parameters, see the following sections:

20.1.5.1. Forcing the Exclusion and Inclusion of Equilibrium Species

20.1.5.2. Defining the Flamelet Controls

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

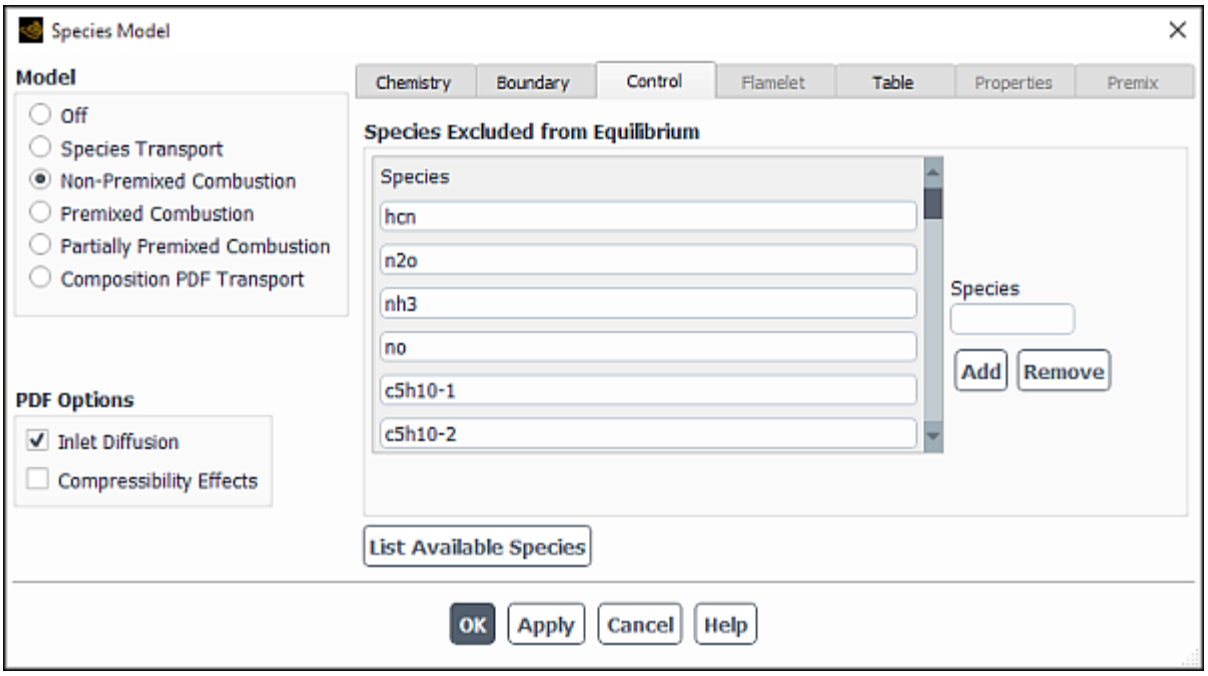
20.1.5.3. Zeroing Species in the Initial Unsteady Flamelet

20.1.5.1. Forcing the Exclusion and Inclusion of Equilibrium Species

Because Ansys Fluent calculates all intermediate and product species automatically during the equilibrium calculation, certain species will be included that are generally not in chemical equilibrium. Principal among these are the NOx species. Specifically, the NOx reaction rates are slow and should not be treated using an equilibrium assumption. Instead, the NOx concentration is predicted most accurately using the Ansys Fluent NOx postprocessor, where finite-rate kinetics are included (see NOx Formation (p. 2585)). The NOx species can be safely excluded from the equilibrium calculation since they are present at low concentrations and have little impact on the density, temperature, and other species.

To force the exclusion of a species from the equilibrium calculation, click the **Control** tab in the **Species Model** dialog box (Figure 20.13: The Species Model Dialog Box (Control Tab) (p. 2499)).

Figure 20.13: The Species Model Dialog Box (Control Tab)



Under **Species Excluded From Equilibrium**, enter the chemical formula for the desired species in the **Add/Remove Species** field. Next, click **Add** to add the species to the **Species** list or **Remove** to remove an existing species from the **Species** list.

If there are species that you want to include in your PDF table that would be ignored by Ansys Fluent due to their low concentration (for example, CH, CH_2, CH_3 for the NO_x calculation), you can force Ansys Fluent to include them using the text interface:

`define` → `models` → `species` → `non-premixed-combustion`

When the console window prompts you with `Force Equilibrium Species to Include...`, specify the appropriate species by entering the chemical formula(s) in double quotes (for example, `"ch"`, `"ch2"`).

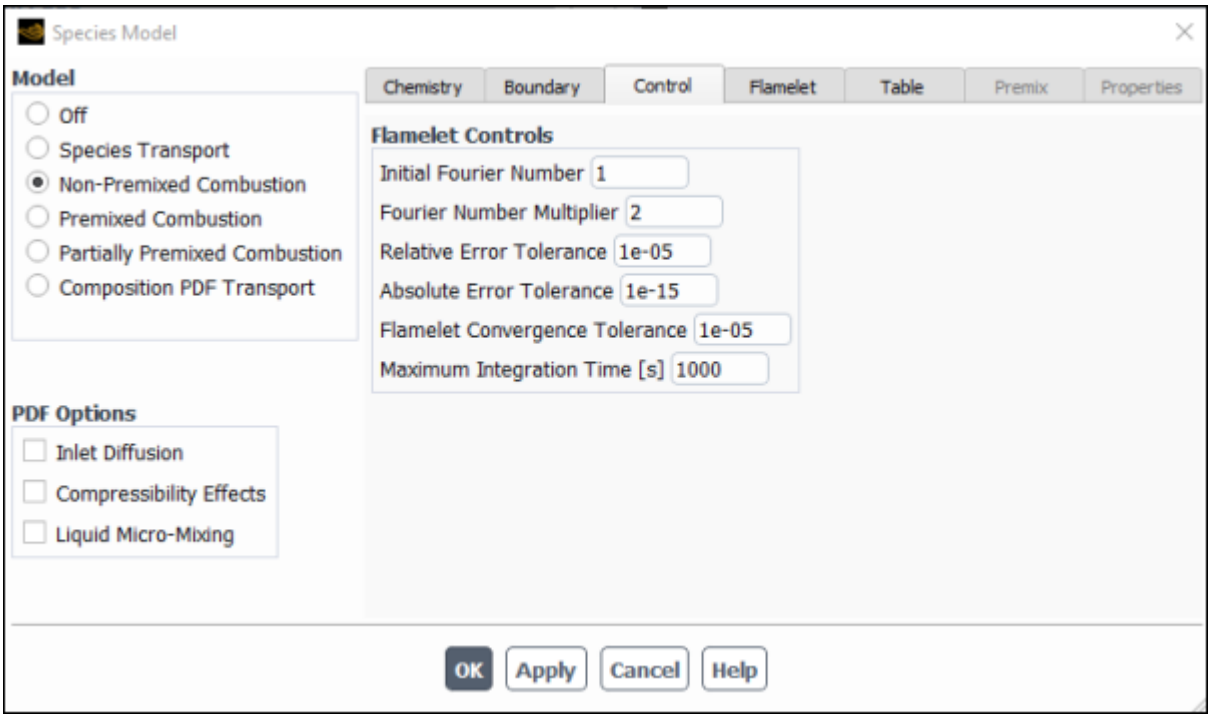
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that you will have to first set up the inputs for the fuel and oxidizer before you are given the option to include the species.

20.1.5.2. Defining the Flamelet Controls

When the steady diffusion flamelet model is selected, and you have created or imported a flamelet, you can adjust the controls for the flamelet solution in the **Control** tab of the **Species Model** dialog box (Figure 20.14: The Species Model Dialog Box (Control Tab) for the Steady Diffusion Flamelet Model (p. 2500)).

Figure 20.14: The Species Model Dialog Box (Control Tab) for the Steady Diffusion Flamelet Model



The **Initial Fourier Number** sets the first time step size for the solution of the flamelet equations (Equation 8.47 and Equation 8.48 in the Theory Guide). This first time step size is calculated as the explicit stability-limited diffusion time step size multiplied by this value. If the solution diverges before the first time step is complete, the value should be lowered.

The **Fourier Number Multiplier** increases the time step size at subsequent times. Every time step after the first is multiplied by this value. If the solution diverges after the first time step, this value should be reduced.

During the numerical integration of the flamelet equations, the local error is controlled to be less than

$$error_{loc,i} = \varepsilon_{rel}\phi_i + \varepsilon_{abs} \tag{20.6}$$

where ϕ_i represents the species mass fractions and temperature at point i in the 1D flamelet. ε_{rel} is the value of the **Relative Error Tolerance** and ε_{abs} is the value of the **Absolute Error Tolerance**, both of which you can specify.

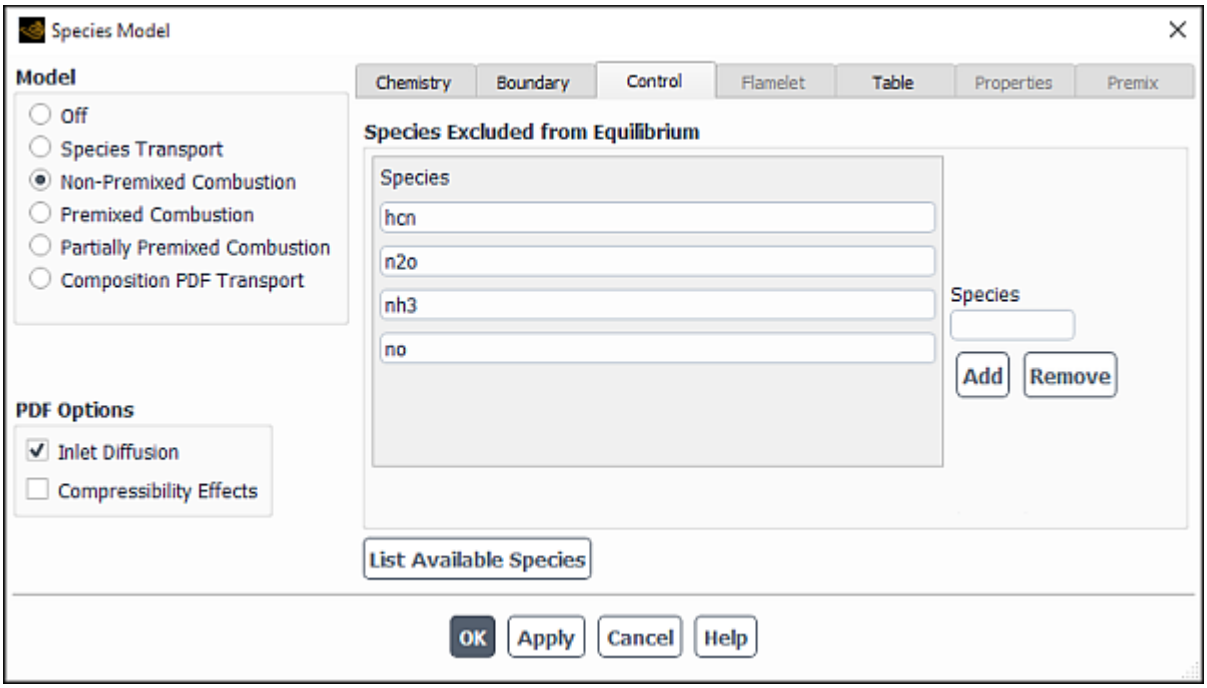
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Because steady flamelets are obtained by time-stepping, they are considered converged only when the maximum absolute change in species fraction or temperature at any discrete mixture-fraction point is less than the specified **Flamelet Convergence Tolerance**. Between time steps, the flamelet species fractions and temperature will sometimes oscillate, which causes absolute changes that are always greater than the flamelet convergence tolerance. In such cases, Ansys Fluent will stop the flamelet calculation after the total elapsed time has exceeded the **Maximum Integration Time**.

20.1.5.3. Zeroing Species in the Initial Unsteady Flamelet

When modeling gas-phase combustion using the Eulerian unsteady laminar flamelet model, the flamelet fields are initialized to a burning, steady-flamelet solution in order to model ignition. However, assuming steady-flamelet profiles for slow-forming species is inaccurate. A better approximation is to identify the slow species and to set them to zero, which is done in the **Control** tab. By default, Ansys Fluent selects some NO_x species (NO, NO_2, N_2O, N, NH, NH_2, NH_3, NNH, HCN, HNO, CN, H2CN, HCNN, HCNO, HOCN, HNCO, HCO), as well as liquid water H_2<l> and solid carbon C<s> to be zeroed. See Figure 20.15: Method to Zero Out the Slow Chemistry Species (p. 2501).

Figure 20.15: Method to Zero Out the Slow Chemistry Species



20.1.6. Calculating the Flamelets

For information about calculating flamelets, see the following sections:

20.1.6.1. Steady Diffusion Flamelet

20.1.6.2. Unsteady Diffusion Flamelet

20.1.6.3. Saving the Flamelet Data

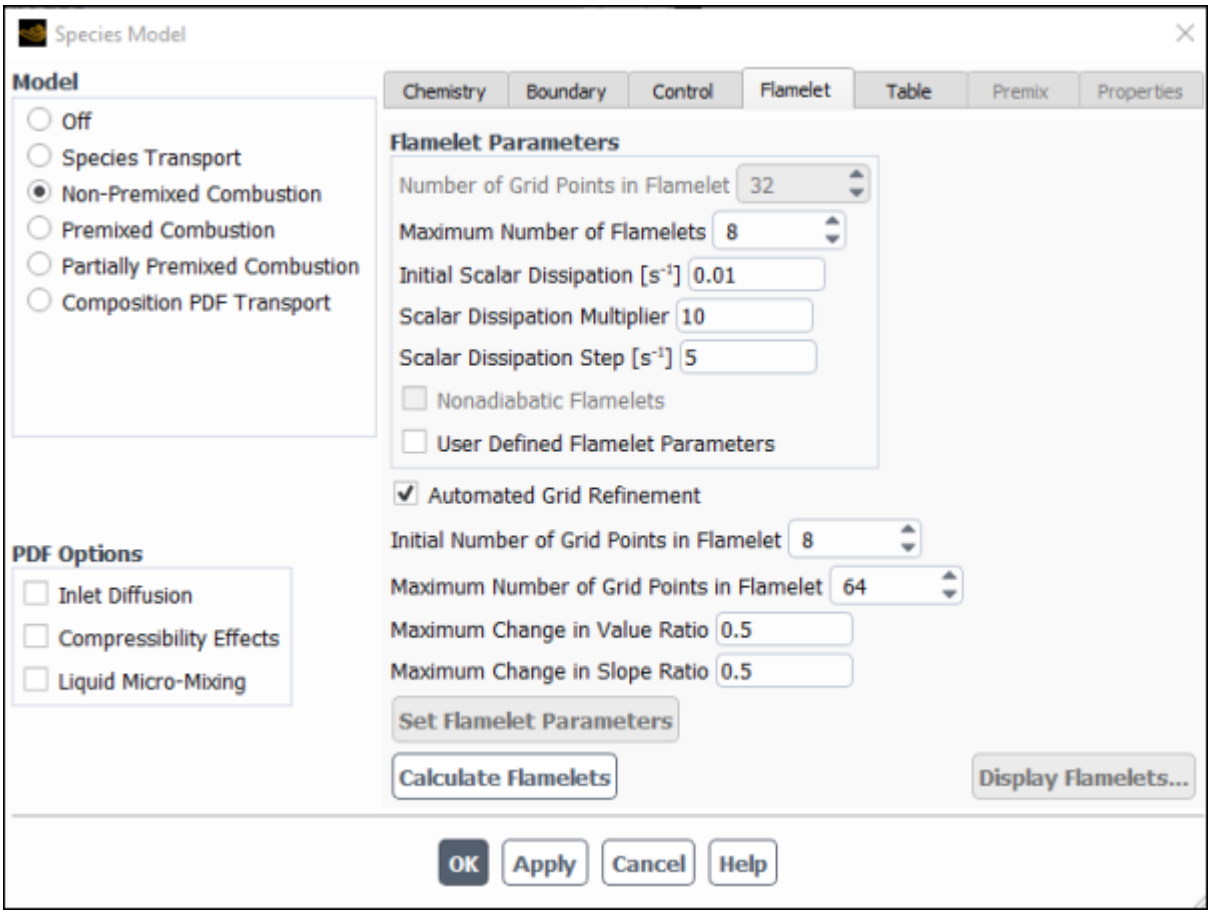
20.1.6.4. Postprocessing the Flamelet Data

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

20.1.6.1. Steady Diffusion Flamelet

In the **Flamelet** tab of the **Species Model** dialog box (Figure 20.16: The Species Model Dialog Box (Flamelet Tab) (p. 2502)), you will enter values for parameters of the flamelet(s).

Figure 20.16: The Species Model Dialog Box (Flamelet Tab)



The **Flamelet Parameters** are as follows:

Number of Grid Points in Flamelet

specifies the number of mixture fraction grid points distributed between the oxidizer ($f=0$) and the fuel ($f=1$). Increased resolution will provide greater accuracy, but since the flamelet species and temperature are solved coupled and implicit in f space, the solution time and memory requirements increase greatly with the number of f grid points.

Maximum Number of Flamelets

specifies the maximum number of flamelet profiles to be calculated. If the flamelet extinguishes before this number is reached, flamelet generation is halted and the actual number of flamelets in the flamelet library will be less than this value.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Initial Scalar Dissipation

is the scalar dissipation of the first flamelet in the library. This corresponds to χ_0 in Equation 8.53 in the Theory Guide.

Scalar Dissipation Multiplier

specifies the ratio of the scalar dissipation step in which successive flamelets are generated when the scalar dissipation is less than 1 s^{-1}. This corresponds to $\Delta\chi$ for $\chi < 1$ in Equation 8.53 in the Theory Guide.

Scalar Dissipation Step

specifies the interval between scalar dissipation values (in s^{-1}) for which multiple flamelets will be calculated. This corresponds to $\Delta\chi$ for $\chi \geq 1$ in Equation 8.53 in the Theory Guide.

Note:

Scalar Dissipation Multiplier and **Scalar Dissipation Step** are used to specify the interval of scalar dissipation for $\chi < 1$ and $\chi \geq 1$, respectively. For example, for initial scalar dissipation of 1e-3 s^{-1} with a **Scalar Dissipation Multiplier** of 10, and a **Scalar Dissipation Step** of 5, the flamelets will be generated with scalar dissipations of 1e-3, 1e-2, 0.1, 1.0, 6, 11, 16, and so on.

User Defined Flamelet Parameters

enables you to hook a user-defined function for scalar dissipation and mean mixture fraction (or progress variable) grid discretization

Automated Grid Refinement

employs an adaptive algorithm, which inserts grid points so that the change of values, as well as the change of slopes, between successive grid points is less than user-specified tolerances. For information about this option, refer to Steady Diffusion Flamelet Automated Grid Refinement in the Theory Guide. Once this option is enabled, you can specify the following parameters:

Initial Number of Grid Points in Flamelet

calculates a steady solution on a coarse grid, with a default of 8. See Equation 8.54 in the Theory Guide.

Maximum Number of Grid Points in Flamelet

has a default of 64.

Maximum Change in Value Ratio

has a default of 0.5 and is ε_v in Equation 8.28 in the Theory Guide.

Maximum Change in Slope Ratio

has a default of 0.5 and is ε_s in Equation 8.29 in the Theory Guide.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Click **Calculate Flamelets** to begin the diffusion flamelet calculation. Sample output for a flamelet calculation is shown below.

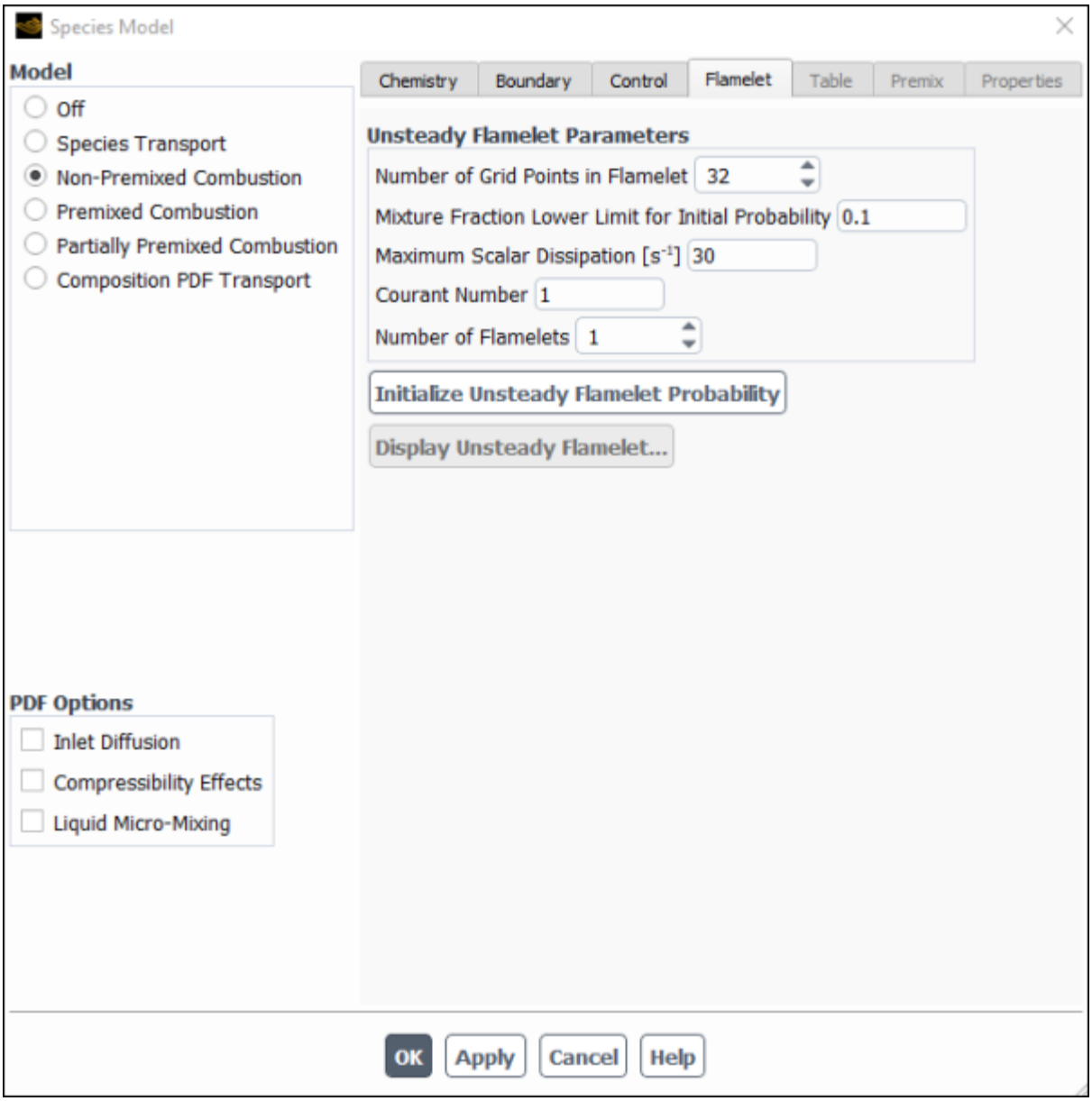
```
 Generating flamelet 1 at scalar dissipation  0.01 /s

Time (s)   Temp (K)   Residual
1.679e-05   2233.7   3.779e+00
5.038e-05   2233.0   7.734e-02
1.175e-04   2231.5   1.648e-01
2.519e-04   2228.6   3.652e-01
5.206e-04   2223.6   8.295e-01
1.058e-03   2215.7   2.100e+00
2.133e-03   2205.5   3.540e+00
4.282e-03   2197.0   4.607e+00
8.581e-03   2193.6   6.639e+00
1.718e-02   2193.1   4.905e+00
3.437e-02   2193.4   5.792e+00
6.877e-02   2194.3   4.659e+00
1.375e-01   2195.3   3.922e+00
2.751e-01   2192.2   3.181e+00
5.502e-01   2188.6   2.549e+00
1.100e+00   2184.8   1.639e+00
2.201e+00   2182.9   4.604e+00
4.402e+00   2186.8   1.307e+00
8.804e+00   2189.6   4.420e-01
1.761e+01   2190.0   8.581e-02
3.522e+01   2190.0   1.199e-02
7.043e+01   2190.0   1.735e-03
1.409e+02   2190.0   4.217e-04
2.817e+02   190.0    6.892e-05
5.635e+02   2190.0   6.777e-06

Flamelet successfully generated
```

20.1.6.2. Unsteady Diffusion Flamelet

In the **Flamelet** tab of the **Species Model** dialog box (Figure 20.17: The Flamelet Tab for the Unsteady Diffusion Flamelet Model (p. 2505)), you will enter values for parameters of the flamelet.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 20.17: The Flamelet Tab for the Unsteady Diffusion Flamelet Model



The **Unsteady Flamelet Parameters** are as follows:

Number of Grid Points in Flamelet

specifies the number of mixture fraction grid points distributed between the oxidizer ($f=0$) and the fuel ($f=1$). Increased resolution will provide greater accuracy, but since the flamelet species and temperature are solved coupled and implicit in f space, the solution time and memory requirements increase with the number of f grid points.

Mixture Fraction Lower Limit for Initial Probability

is the mixture fraction above which the marker probability will be initialized to 1, and below which the marker probability will be initialized to 0. In general, it should be set greater than the stoichiometric mixture fraction.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Maximum Scalar Dissipation

is where flamelets extinguish at large scalar dissipation (mixing) rates. To prevent excessive mixing in the flamelet, Ansys Fluent allows you to specify a **Maximum Scalar Dissipation** rate for the 1D flamelet equations. A reasonable value for this is the steady flamelet extinction scalar dissipation. The default value of 30/s is near the steady extinction scalar dissipation of a methane-air flame at standard temperature and pressure.

Courant Number

is the number at which Ansys Fluent automatically selects the time step for the probability equation based on this convective Courant number.

Number of Flamelets

is the number of unsteady laminar flamelets to be initiated in the simulation. The probability marker equation will be solved for each flamelet.

Click **Initialize Unsteady Flamelet Probability** to initialize the unsteady flamelet and its probability marker equation. Ansys Fluent is now ready for postprocessing the 1D unsteady flamelet and the 2D/3D unsteady marker probability equation.

20.1.6.3. Saving the Flamelet Data

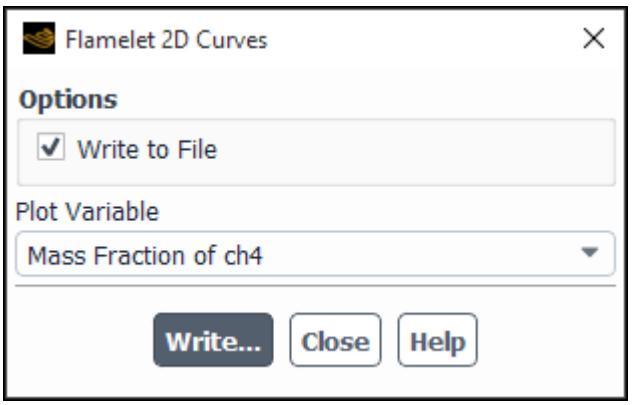
The flamelet tables may be written to a file for import into later sessions of Ansys Fluent. You may want to do this, for example, to change the number of discretization points in the PDF table, or to plot the flamelet profiles in Ansys Fluent. The flamelet tables should be saved before you create the PDF table:

File → **Write** → **Flamelet...**

20.1.6.4. Postprocessing the Flamelet Data

For the flamelet model, you can display or write flamelet curves. Click the **Display Flamelets...** or **Display Unsteady Flamelet...** button. If you have a single flamelet, as for the unsteady diffusion flamelet model, you can access the **Flamelet 2D Curves** dialog box (Figure 20.18: The Flamelet 2D curves Dialog Box (p. 2506)).

Figure 20.18: The Flamelet 2D curves Dialog Box

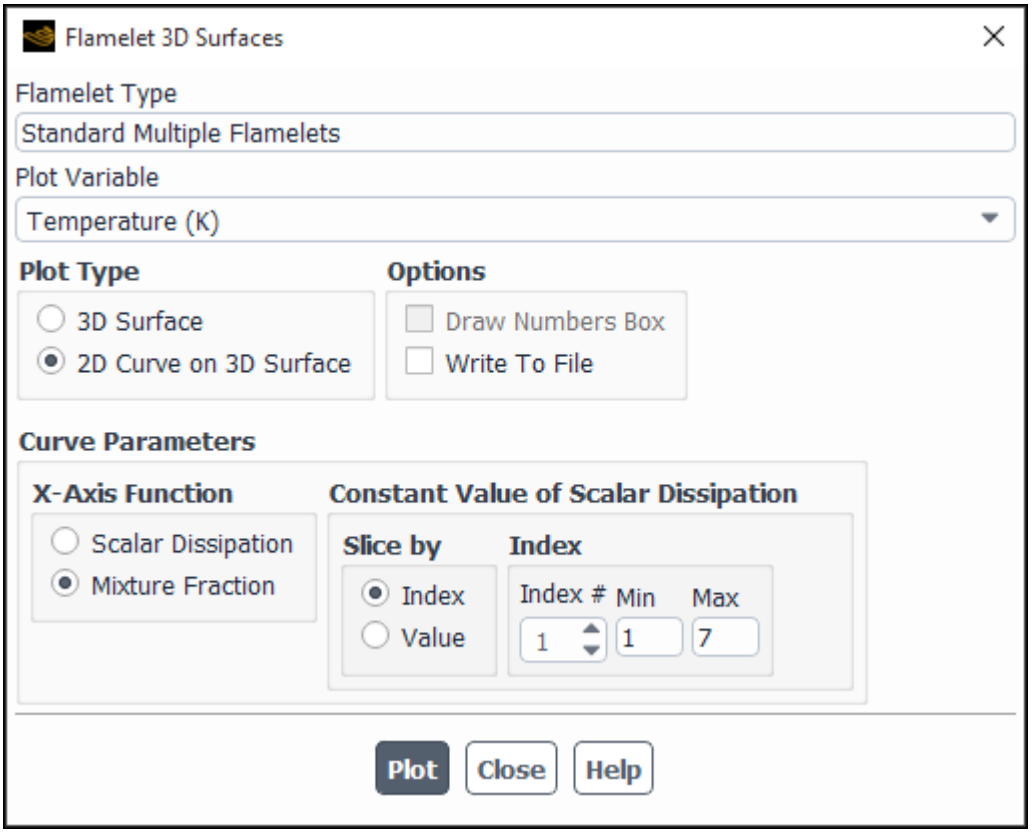


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For the steady diffusion flamelet model with more than one flamelet, you can display 2D plots and 3D surfaces showing the variation of species fraction or temperature with the mean mixture fraction or scalar dissipation using the **Flamelet 3D Surfaces** dialog box (for example, Figure 20.19: The Flamelet 3D Surfaces Dialog Box (p. 2507)).

To access this dialog box, click the **Display Flamelets...** button in the **Flamelet** tab of the **Species Model** dialog box, as shown in Figure 20.16: The Species Model Dialog Box (Flamelet Tab) (p. 2502).

Figure 20.19: The Flamelet 3D Surfaces Dialog Box



To display the flamelet tables graphically, use the following procedure:

1. In the **Flamelet 3D Surfaces** dialog box, from the **Plot Variable** drop-down list, select the variable you want to display graphically.
2. Specify the **Plot Type** as either **3D Surface** or **2D Curve on 3D Surface**.
   - For a 3D surface, enable or disable **Draw Numbers Box** under **Options**. When this option is turned on, the display will include a wireframe box with the numerical limits in each coordinate direction.
   - For a 2D curve on a 3D surface:
     a. Specify whether you want to write the plot data to a file by toggling **Write To File** under **Options**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. Specify the **X-Axis Function** against which the plot variable will be displayed by selecting **Scalar Dissipation** (χ), or **Mixture Fraction** (f). The variable that is not selected will be held constant.

c. Specify the type of discretization (that is, how the flamelet data will be sliced) for the variable that is being held constant (under **Constant Value of Mixture Fraction** or **Constant Value of Scalar Dissipation**).

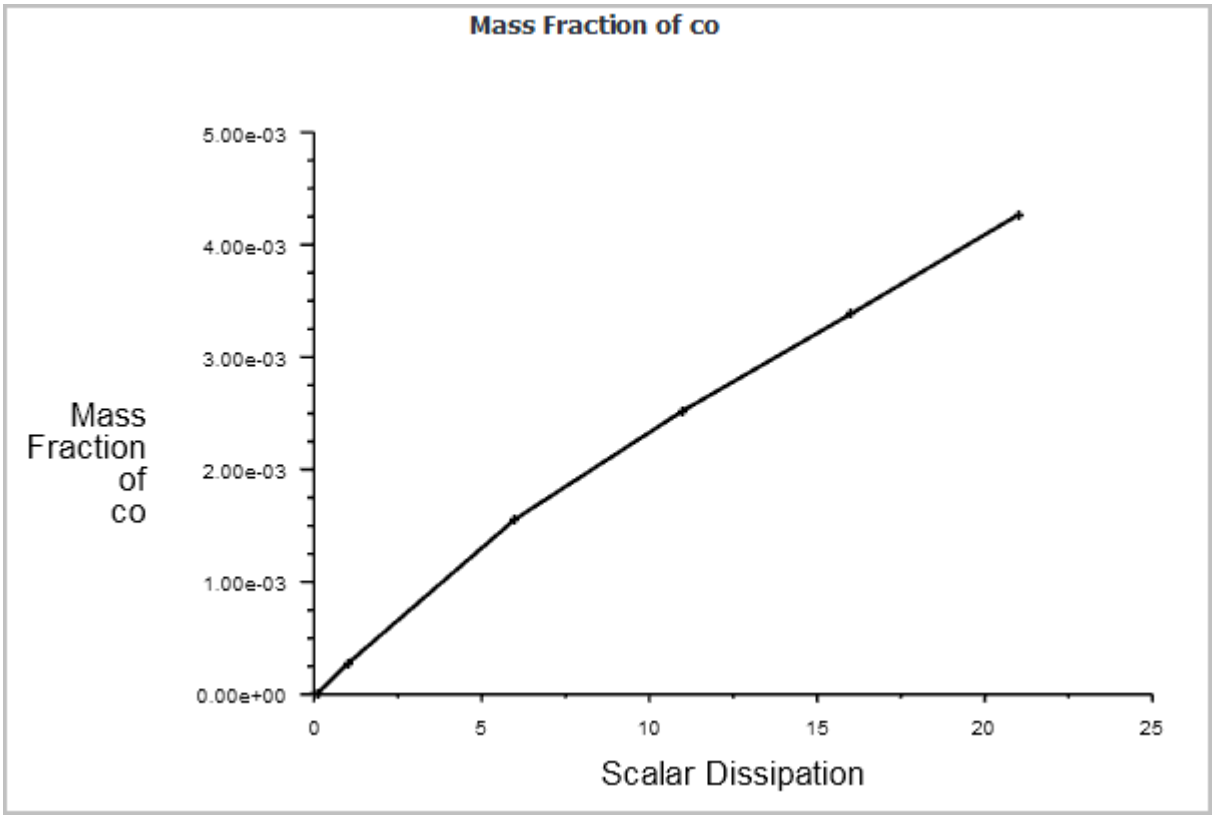
   – If you selected **Index** under **Slice by**, specify the discretization **Index** of the variable that is being held constant. The range of integer values that you are allowed to choose from is displayed under **Min** and **Max**, and is equivalent to the number of points specified for that variable in the **Flamelet** tab of the **Species Model** dialog box (see Calculating the Flamelets (p. 2501)).

   – If you selected **Value** under **Slice by**, specify the numerical **Value** of the variable that is being held constant. The range of values that you can specify is displayed under **Min** and **Max**.

3. Write or display the flamelet table results. If you have turned on the **Write To File** option for a 2D plot, click **Write** and specify a name for the file in The Select File Dialog Box (p. 931). Otherwise, click **Plot** or **Display** as appropriate to display a 2D plot or 3D surface in the graphics window.

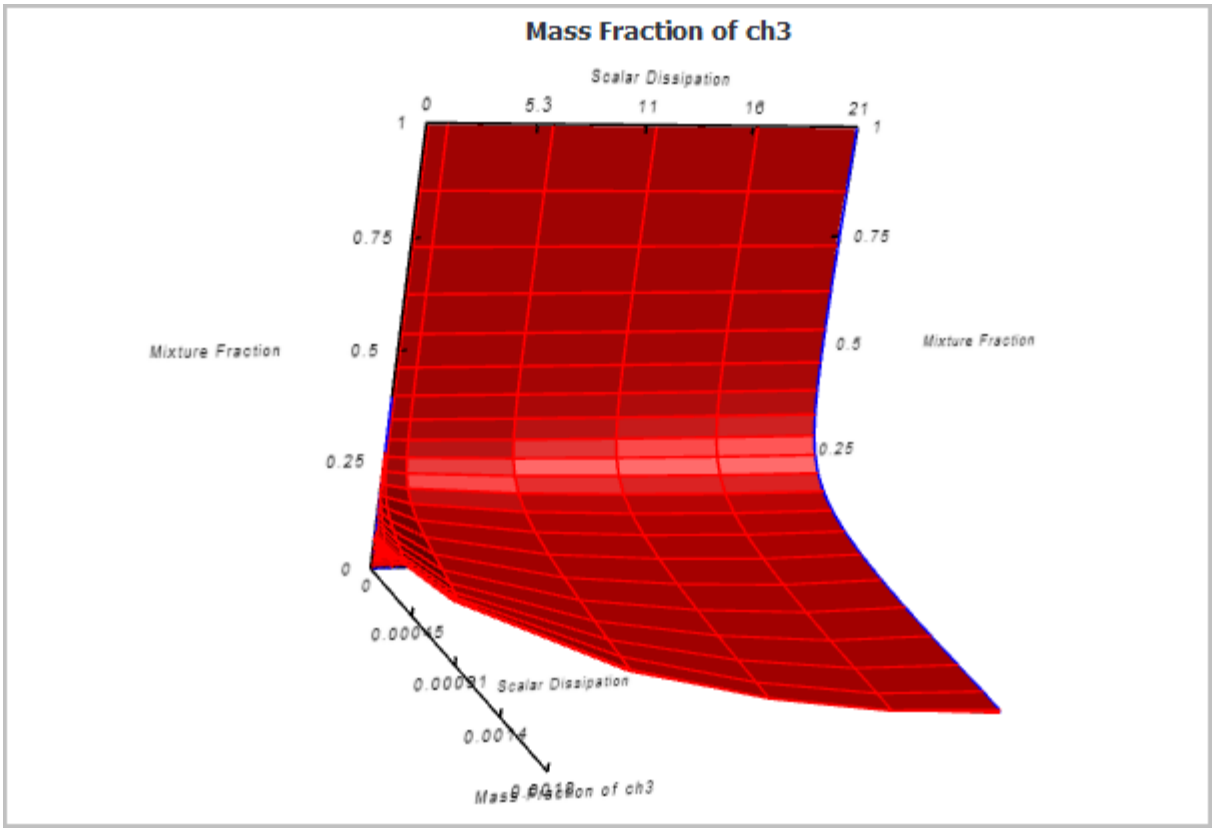
Figure 20.20: Example 2D Plot of Flamelet Data (p. 2508) and Figure 20.21: Example 3D Plot of Flamelet Data (p. 2509) show examples of a 2D curve plot and 3D surface plot of a flamelet table.

Figure 20.20: Example 2D Plot of Flamelet Data



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 20.21: Example 3D Plot of Flamelet Data



20.1.7. Calculating the Look-Up Tables

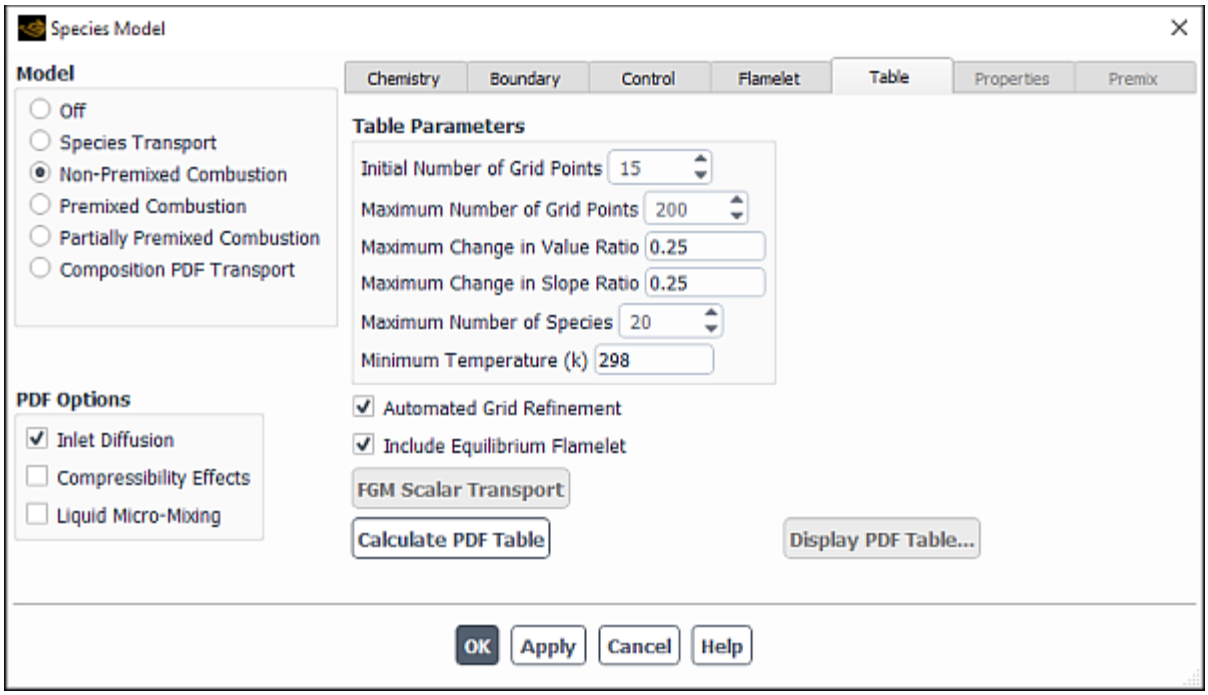
Ansys Fluent requires additional inputs that are used in the creation of the look-up tables. Several of these inputs control the number of discrete values for which the look-up tables will be computed. These parameters are input in the **Table** tab of the **Species Model** dialog box. When **Automated Grid Refinement** is enabled, you will specify the table parameters displayed in Figure 20.22: The Species Model Dialog Box (Table) Tab Displaying Automated Grid Refinement (p. 2510). If **Automated Grid Refinement** is disabled, you will specify the table parameters displayed in Figure 20.23: The Species Model Dialog Box (Table) Tab Excluding Automated Grid Refinement (p. 2511).

Note:

Automated Grid Refinement is not available with two mixture fractions.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 20.22: The Species Model Dialog Box (Table) Tab Displaying Automated Grid Refinement



The look-up table parameters when **Automated Grid Refinement** is enabled are as follows:

Initial Number of Grid Points

specifies the number of grid points for the resolution of the mean mixture fraction, mixture fraction variance, and mean enthalpy (for non-adiabatic systems).

Maximum Number of Grid Points

specifies the maximum number of grid points used for tabulation. The grid refinement procedure will stop inserting the points when either the change in value and slope between successive points is within tolerance or the maximum number of grid points are generated.

Maximum Change in Value Ratio

specifies the maximum allowable change in value of table variables between successive grid points as specified by Equation 8.28 in the Theory Guide.

Maximum Change in Slope Ratio

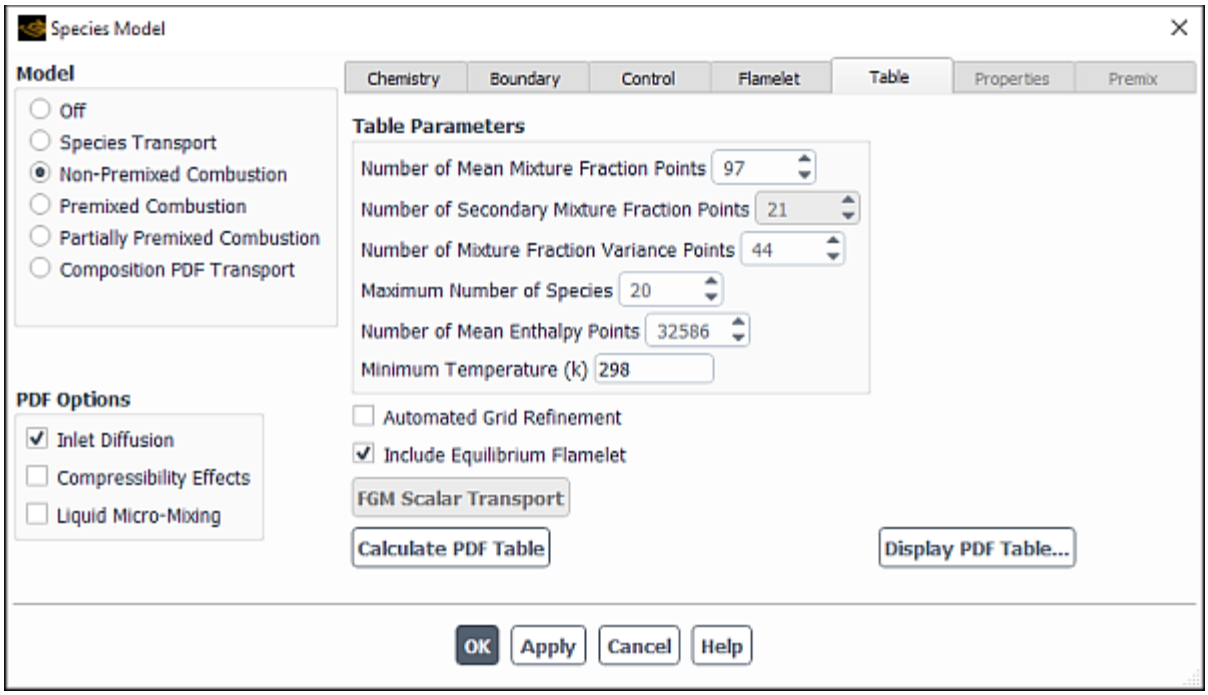
specifies the maximum change in the slope of table variables between successive grid points as specified by Equation 8.29 in the Theory Guide.

Maximum Number of Species

is the maximum number of species stored in the lookup tables.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 20.23: The Species Model Dialog Box (Table) Tab Excluding Automated Grid Refinement



The look-up table parameters when **Automated Grid Refinement** is disabled are as follows:

Number of Mean Mixture Fraction Points

is the number of discrete values of $\overline{f}$ at which the look-up tables will be computed. For a two-mixture-fraction model, this value is the number of points in the instantaneous state profile used to compute the PDF if you choose the β PDF model (see Tuning the PDF Parameters for Two-Mixture-Fraction Calculations (p. 2531)). Increasing the number of points will yield a more accurate PDF shape, but the calculation will take longer. The mean mixture fraction points will be automatically clustered around the stoichiometric mixture fraction value.

Number of Mixture Fraction Variance Points

is the number of discrete values of $\overline{f'^2}$ at which the look-up tables will be computed. Lower resolution is acceptable because the variation along the $\overline{f'^2}$ axis is, in general, slower than the variation along the $\overline{f}$ axis of the look-up tables. This option is available only when no secondary stream has been defined.

Number of Secondary Mixture Fraction Points

is the number of discrete values of p_{sec} at which the look-up tables will be computed. Like the **Number of Mean Mixture Fraction Points**, Ansys Fluent will use the **Number of Secondary Mixture Fraction Points** to compute the equilibrium state-relation if you choose the β PDF option (see Tuning the PDF Parameters for Two-Mixture-Fraction Calculations (p. 2531)) for a two-mixture-fraction model. A larger number of points will give a more accurate shape for the PDF, but with a longer calculation time. This option is available only when a secondary stream has been defined.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Maximum Number of Species

is the maximum number of species that will be included in the look-up tables. The maximum number of species that can be included is 200. Ansys Fluent will automatically select the species with the largest mole fractions to include in the PDF table. Note that the PDF table values of density and specific heat are pre-calculated with all the species, and hence the convergence behavior of Ansys Fluent will not be affected by the input for the **Maximum Number of Species**. Hence, to keep table sizes small, you should set the **Maximum Number of Species** to only include the species that you are interested in postprocessing.

Number of Mean Enthalpy Points

is the number of discrete values of enthalpy at which the look-up tables will be computed. This input is required only if you are modeling a non-adiabatic system. The number of points required will depend on the chemical system that you are considering, with more points required in high heat release systems (for example, hydrogen/oxygen flames). This option is not available with the unsteady flamelet model.

Minimum Temperature

is used to determine the lowest temperature for which the look-up tables are generated. Your input should correspond to the minimum temperature expected in the domain (for example, an inlet or wall temperature). The minimum temperature should be set 10–20 K below the minimum system temperature. This option is available only if you are modeling a non-adiabatic system. This option is not available with the unsteady flamelet model.

Include Equilibrium Flamelet

specifies that an equilibrium flamelet (that is, $\chi=0$) will be generated in Ansys Fluent and appended to the flamelet library before the PDF table is calculated. This option is available when generating or importing multiple flamelets, as well as when a single flamelet is considered. In the latter case, the PDF table will consist of two scalar dissipation slices, namely the equilibrium slice at $\chi=0$, and the flamelet slice. This option is available only with the steady diffusion flamelet model.

When you are satisfied with your inputs, click **Calculate PDF Table** to generate the look-up tables.

The computations performed for a single-mixture-fraction calculation culminate in the discrete integration of Equation 8.17 (or Equation 8.25 in the Theory Guide) as represented in Figure 8.5: Logical Dependence of Averaged Scalars on Mean Mixture Fraction, the Mixture Fraction Variance, and the Chemistry Model (Adiabatic, Single-Mixture-Fraction Systems) (or Figure 8.6: Logical Dependence of Averaged Scalars on Mean Mixture Fraction, the Mixture Fraction Variance, Mean Enthalpy, and the Chemistry Model (Non-Adiabatic, Single-Mixture-Fraction Systems) in the Theory Guide). For a two-mixture-fraction calculation, Ansys Fluent will calculate the physical properties using Equation 8.15 or its adiabatic equivalent. The computation time will be shortest for adiabatic single-mixture-fraction equilibrium calculations and longest for non-adiabatic calculations involving multiple flamelet generation. Below, sample outputs are shown for an adiabatic single-mixture-fraction equilibrium calculation and a non-adiabatic calculation with flamelets:

```
Generating PDF lookup table
 Type of the PDF Table: Adiabatic Table (Two Streams)
Calculating table .....

1271 points calculated
22 species added
PDF Table successfully generated!
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
 Generating PDF lookup table
  Type of the PDF Table: Nonadiabatic Table with Strained Flamelet Model (Two Streams)
  Calculating table .....
  calculating temperature limits .....
  calculating temperature limits .....
  calculating scalar dissipation slices .....
   - scalar dissipation slice  9
  calculating equilibrium slice .....
 Performing PDF integrations.....

 16810 points calculated
 17 species added
 PDF Table successfully generated!
Initializing PDF table arrays and structures.
```

Important:

Note that there is a significant difference in run time between the one-mixture fraction model and the two-mixture fraction model. In the one-mixture fraction model, the PDF table contains the mean data of density, temperature, and specific heats, and is three-dimensional for an equilibrium nonadiabatic case (mean mixture fraction, mixture fraction variance, and mean heat loss). For this case, Ansys Fluent updates properties every flow iteration. In the case of the two-mixture fraction model, only the instantaneous state relationships are stored and mean properties are calculated from these by performing PDF integrations in every cell of the Ansys Fluent simulation. Since this is computationally expensive, Ansys Fluent provides the option of only updating properties after a specified number of iterations.

For a two-mixture fraction model, you can also refer to Full Tabulation of the Two-Mixture-Fraction Model (p. 2514) for more information.

After completing the calculation at the specified number of mixture fraction points, Ansys Fluent reports that the calculation succeeded. In a single-mixture-fraction case, the resulting look-up tables take the form illustrated in Figure 8.8: Visual Representation of a Look-Up Table for the Scalar (Mean Value of Mass Fractions, Density, or Temperature) as a Function of Mean Mixture Fraction and Mixture Fraction Variance in Adiabatic Single-Mixture-Fraction Systems in the Theory Guide (or Figure 8.10: Visual Representation of a Look-Up Table for the Scalar as a Function of Mean Mixture Fraction and Mixture Fraction Variance and Normalized Heat Loss/Gain in Non-Adiabatic Single-Mixture-Fraction Systems, for non-adiabatic systems). These look-up tables can be plotted using the available graphics tools, as described in Postprocessing the Look-Up Table Data (p. 2515).

Note that in non-adiabatic calculations, the console window will report that the temperature limits and enthalpy slices have been calculated.

For a two-mixture-fraction case, the resulting look-up tables take the form illustrated in Figure 8.9: Visual Representation of a Look-Up Table for the Scalar ϕ_I as a Function of Fuel Mixture Fraction and Secondary Partial Fraction in Adiabatic Two-Mixture-Fraction Systems in the Theory Guide (or Figure 8.11: Visual Representation of a Look-Up Table for the Scalar ϕ_I as a Function of Fuel Mixture Fraction and Secondary Partial Fraction, and Normalized Heat Loss/Gain in Non-Adiabatic Two-Mixture-Fraction Systems, for non-adiabatic systems).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

20.1.7.1. Full Tabulation of the Two-Mixture-Fraction Model

The default algorithm for the two-mixture-fraction model is to perform PDF integrations of the equilibrium state relationships at run time. Since these are multi-dimensional integrals, the default two-mixture-fraction model can be computationally demanding.

Alternatively, you may want to pre-compute these integrations and create 4D look-up tables for adiabatic simulations, or 5D tables for non-adiabatic simulations. Such high-dimensional tables are computationally expensive to build, and may require large memory and disk storage, but can offer substantial improvement in run time speed.

The option to create a fully-tabulated two-mixture-fraction table is available in cases with the two-mixture-fraction model enabled, via the text command:

```
define/models/species/full-tabulation?
```

If you are modeling pollutant formation, note that the full tabulation option is not compatible with the **mixture-fraction** option for **PDF Mode** in the **Turbulence Interaction Mode** tab settings. See Setting Turbulence Parameters (p. 2598) for details on Turbulence Interaction options for pollutant modeling.

20.1.7.2. Stability Issues in Calculating Chemical Equilibrium Look-Up Tables

Complex chemistry and non-adiabatic effects may make the equilibrium calculation more time-consuming and difficult. In some instances the equilibrium calculation may even fail. You may be able to eliminate any difficulties that you encounter by trying the calculation as an adiabatic system. Adiabatic system calculations are generally quite straightforward and can provide valuable insight into the optimal inputs to the non-adiabatic calculation.

Additional stability issues may arise for solid or heavy liquid fuels that have been defined using the empirical fuel approach. You may find that, for rich fuel mixtures, the equilibrium calculation produces very low temperatures and eventually fails. This indicates that strong endothermic reactions are taking place and the mixture is not able to sustain them. In this situation, you may need to raise the heating value of the fuel until Ansys Fluent produces acceptable results. Provided that your fuel will be treated as a liquid or solid (coal) fuel, you can maintain the desired heating value in your Ansys Fluent simulation. This is accomplished by defining the difference between the desired and the adjusted heating values as latent heat (in the case of combusting solid fuel) or heat of pyrolysis (in the case of liquid fuel).

20.1.7.3. Saving the Look-Up Tables

The look-up tables may be stored in a file that you can read back into later sessions of Ansys Fluent. The look-up tables should be saved before you exit from the current Ansys Fluent session.

File → **Write** → **PDF...**

By default, the file will be saved as formatted (ASCII, or text). To save a binary (unformatted) file, turn on the **Write Binary Files** option in the **Select File** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

20.1.7.4. Postprocessing the Look-Up Table Data

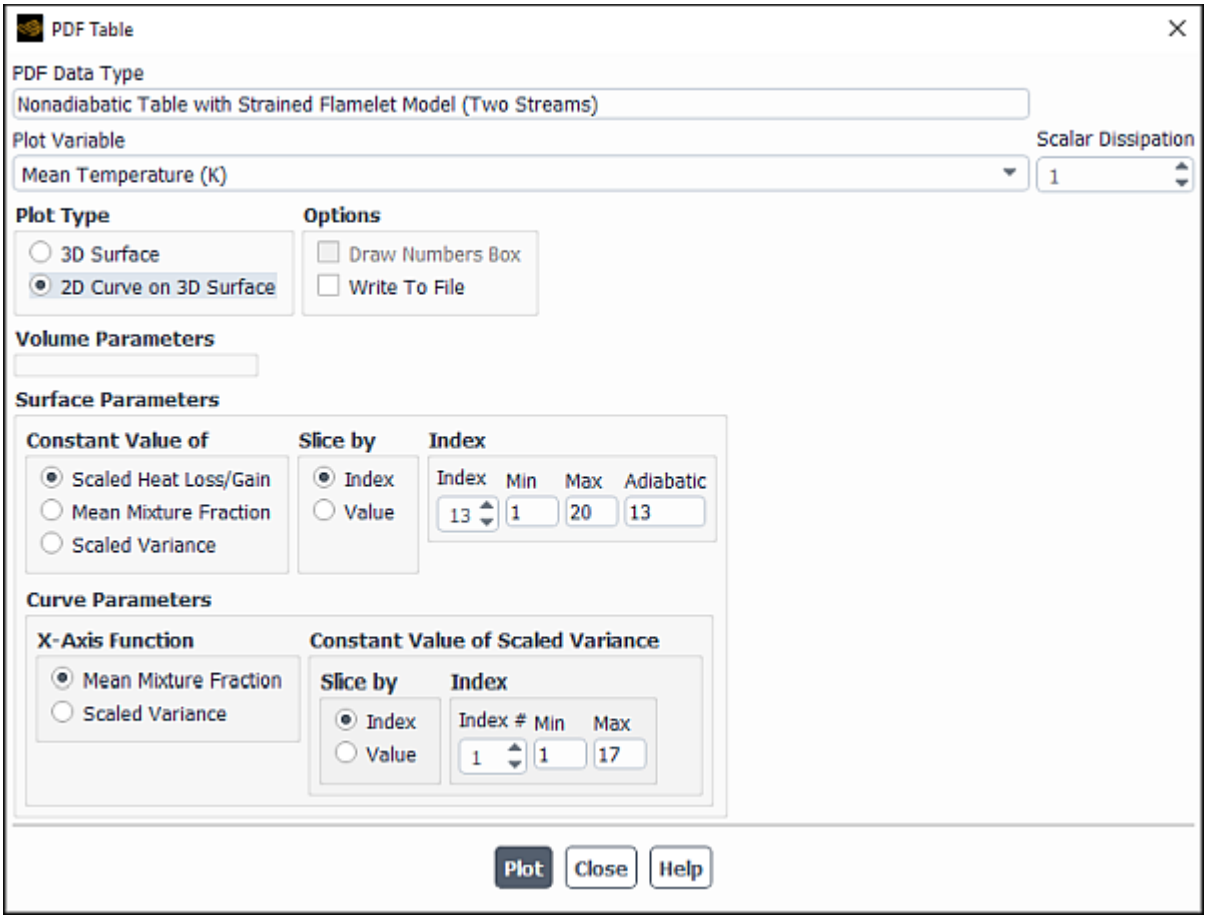
It is important for you to view your temperature and species tables to ensure that they are adequately but not excessively resolved. Inadequate resolution will lead to inaccuracies, and excessive resolution will lead to unnecessarily slow calculation times.

After a PDF table has been generated or read into Ansys Fluent, you can display 2D plots and 3D surfaces showing the variation of species mole fraction, density, or temperature with the mean mixture fraction, mixture fraction variance, or enthalpy using the **PDF Table** dialog box (for example, Figure 20.24: The PDF Table Dialog Box (Non-Adiabatic Case With Flamelets) (p. 2515)). The **PDF Table** dialog box can be accessed in one of two ways:

- by clicking the **Display PDF Table...** button in the **Table** tab of the **Species Model** dialog box (as shown in Figure 20.23: The Species Model Dialog Box (Table) Tab Excluding Automated Grid Refinement (p. 2511))
- by using the following path:

  Postprocessing → **Model Specific** → **PDF Table...**

Figure 20.24: The PDF Table Dialog Box (Non-Adiabatic Case With Flamelets)



To display the look-up tables graphically, use the following procedure:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the **PDF Table** dialog box, in the **Plot Variable** drop-down list you can select temperature, density, or species mole fraction as the variable to be plotted.

2. (multiple flamelets only) Specify the value of the **Scalar Dissipation**. In the case of non-adiabatic flamelets, there is the additional parameter of mean enthalpy. In addition to varying the mean enthalpy and mean mixture fraction, you can vary the display of the PDF table by changing the value of the scalar dissipation, which gives the table a fourth "dimension".

3. Specify the **Plot Type** as either **3D Surface** or **2D Curve on 3D Surface**. In the equilibrium model, a 2D curve is a slice of a 3D surface, and therefore some options selected for a 3D surface may impact the display of a 2D curve.

   - For a 3D surface:

     a. Enable or disable **Draw Numbers Box** under **Options**. When this option is turned on, the display will include a wireframe box with the numerical limits in each coordinate direction.

     b. (non-adiabatic cases only) Under **Surface Parameters**, specify the discrete independent variable to be held constant in the look-up table (**Constant Value of**).

        – For a single-mixture-fraction case, select **Scaled Heat Loss/Gain** ($\overline{H}$), **Mean Mixture Fraction** ($\overline{f}$), or **Scaled Variance** ($\overline{f'^2}$). For any mean mixture fraction $\overline{f}$, the variance varies between a minimum of 0 and a maximum of $\overline{f}\left(1-\overline{f}\right)$. In order to view the mixture fraction variance, it is normalized by Equation 20.7 (p. 2516) so that for any mean mixture fraction the scaled variance ranges from 0 to 0.25.

$$\overline{f_s'^2}=0.25\frac{\overline{f'^2}}{\overline{f}\left(1-\overline{f}\right)} \tag{20.7}$$

        – For a two-mixture-fraction case, the **Scaled Heat Loss/Gain** is the only available option.

     c. (non-adiabatic cases only) Specify whether the 3D array of data points available in the look-up table will be sliced by **Index** or **Value** under **Slice by**.

        – If you selected **Index**, specify the discretization **Index** of the variable that is being held constant. The range of integer values that you are allowed to choose from is displayed under **Min** and **Max**, and is equivalent to the number of points specified for that variable in the **Table** tab of the **Species Model** dialog box (see Calculating the Look-Up Tables (p. 2509)). If you specified to hold the enthalpy (**Scaled Heat Loss/Gain**) constant, the enthalpy slice index corresponding to the adiabatic case will be displayed in the **Adiabatic** field.

        – If you selected **Value**, specify the numerical **Value** of the variable that is being held constant. The range of values that you can specify is displayed under **Min** and **Max**.

   - For a 2D curve on a 3D surface:

     a. Specify whether you want to write the plot data to a file by toggling **Write To File** under **Options**.

     b. Under **Curve Parameters**, specify the **X-Axis Function** against which the plot variable will be displayed.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– For an adiabatic single-mixture-fraction case, select **Mean Mixture Fraction** ($\overline{f}$), or **Scaled Variance** ($\overline{f'^2}$).

– For a non-adiabatic single-mixture-fraction case, the options will depend on what was selected under **Constant Value of** under **Surface Parameters**, but will include two of the following: **Scaled Heat Loss/Gain** ($\overline{H}$), **Mean Mixture Fraction**, and **Scaled Variance**.

– For a two-mixture-fraction case, select **Fuel Mixture Fraction** (f_{fuel}) or **Secondary Partial Fraction** (p_{sec}).

c. Specify the type of discretization (that is, how the look-up table data will be sliced) for the variable that is being held constant (under **Constant Value of Mean Mixture Fraction**, **Constant Value of Scaled Variance**, and so on). Note that for non-adiabatic cases, each 3D surface slice contains a full set of 2D slices.

– If you selected **Index** under **Slice by**, specify the discretization **Index** of the variable that is being held constant. The range of integer values that you are allowed to choose from is displayed under **Min** and **Max**, and is equivalent to the number of points specified for that variable in the **Table** tab of the **Species Model** dialog box (see Calculating the Look-Up Tables (p. 2509)).

– If you selected **Value** under **Slice by**, specify the numerical **Value** of the variable that is being held constant. The range of values that you can specify is displayed under **Min** and **Max**.

4. Write or display the look-up table results. If you have turned on the **Write To File** option for a 2D plot, click **Write** and specify a name for the file in The Select File Dialog Box (p. 931). Otherwise, click **Plot** or **Display** as appropriate to display a 2D plot or 3D surface in the graphics window.

Figure 20.25: Mean Species Mole Fraction Derived From an Equilibrium Chemistry Calculation (p. 2518) and Figure 20.26: Mean Temperature Derived From an Equilibrium Chemistry Calculation (p. 2518) show examples of 2D plots derived for a very simple hydrocarbon system.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 20.25: Mean Species Mole Fraction Derived From an Equilibrium Chemistry Calculation

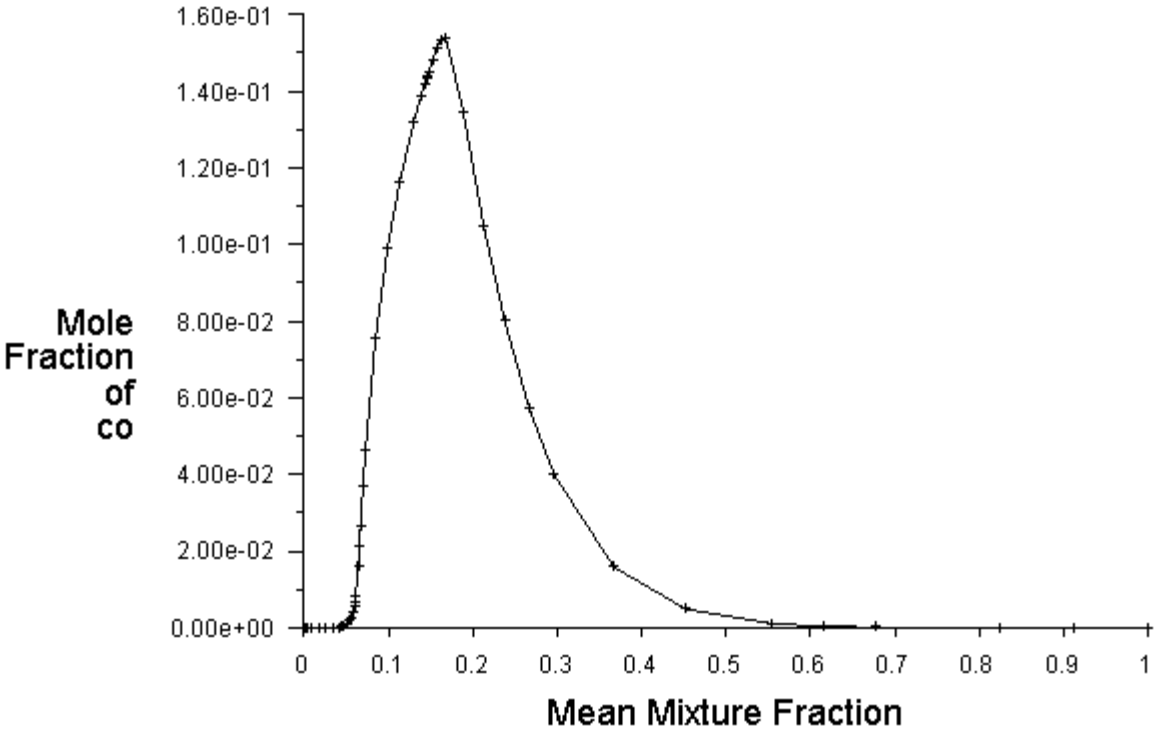
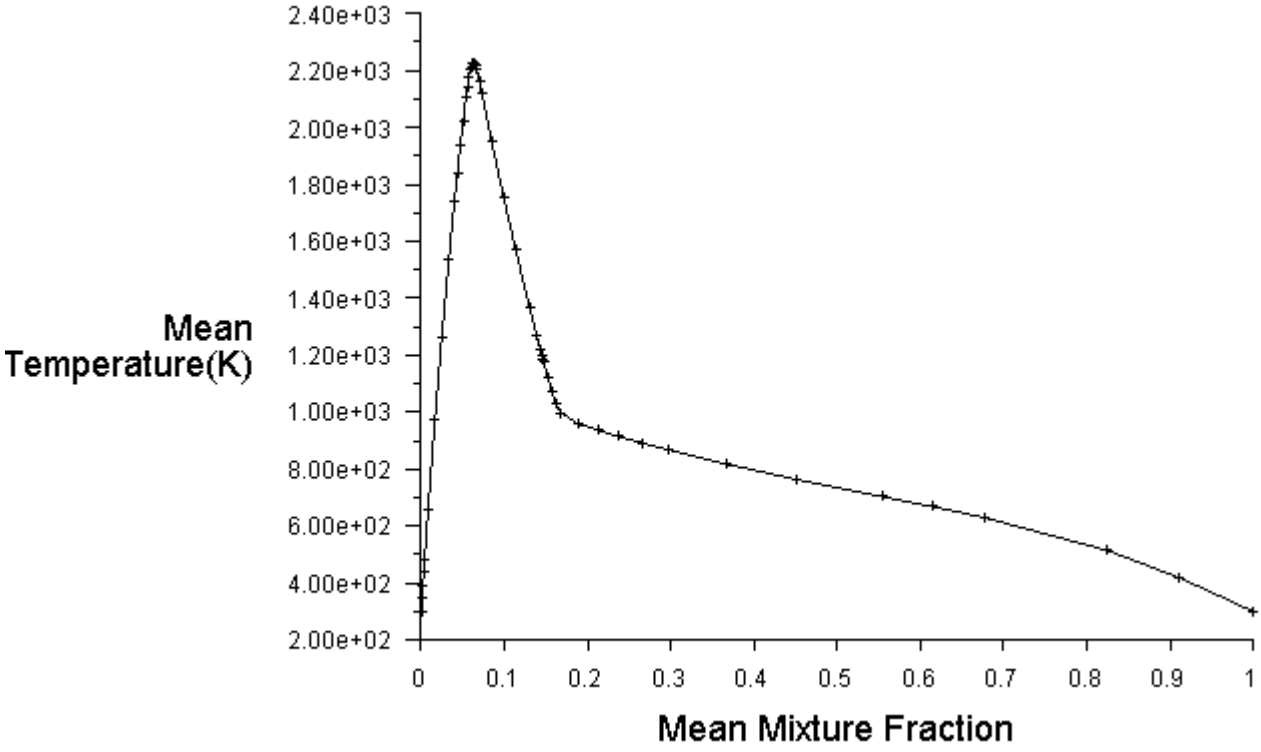


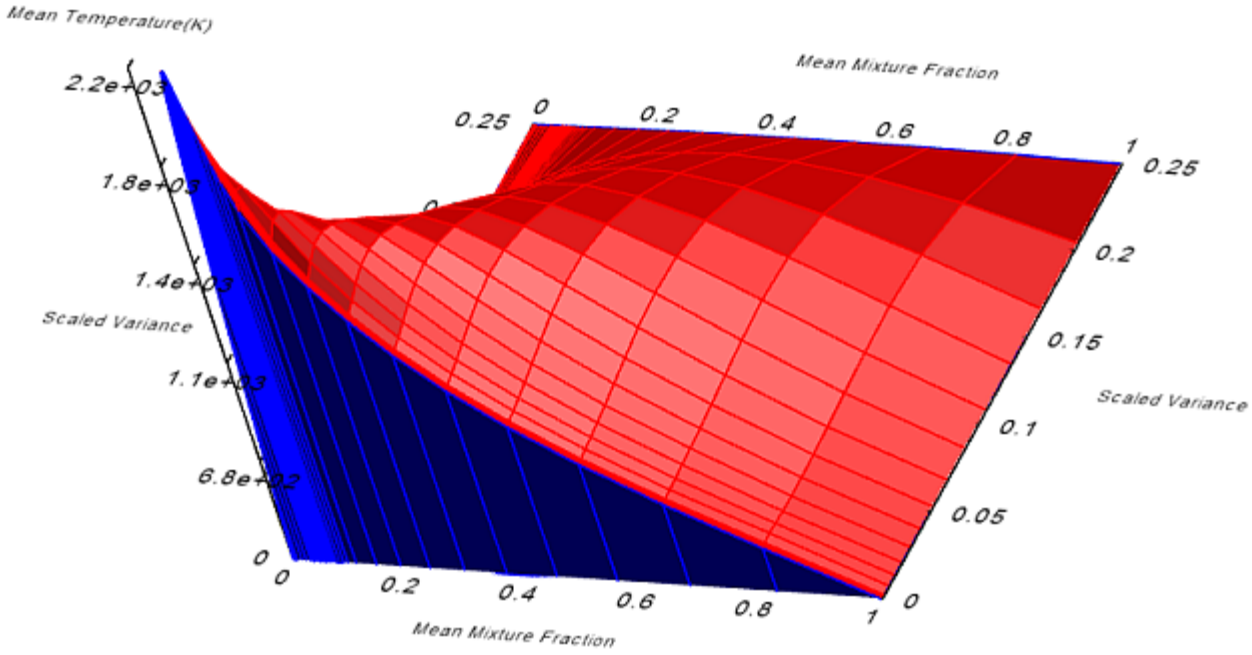
Figure 20.26: Mean Temperature Derived From an Equilibrium Chemistry Calculation



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 20.27: 3D Plot of Look-Up Table for Temperature Generated for a Simple Hydrocarbon System (p. 2519) shows an example of a 3D surface derived for the same system.

Figure 20.27: 3D Plot of Look-Up Table for Temperature Generated for a Simple Hydrocarbon System



20.1.8. Standard Files for Diffusion Flamelet Modeling

The standard flamelet file format can be used to read and write non-premixed (diffusion) flamelets. The data structure for the standard flamelet file format is based on keywords that precede each data section. If any of the keywords in your flamelet data file do not match the supported keywords, you will have to manually edit the file and change the keywords to one of the supported types. (The Ansys Fluent flamelet filter is case-insensitive, so you need not worry about capitalization within the keywords.)

The following keywords are supported by the Ansys Fluent filter:

Description	Keyword
Header section	`HEADER`
Main body section	`BODY`
Number of species	`NUMOFSPECIES`
Number of grid points	`GRIDPOINTS`
Pressure	`PRESSURE`
Strain rate for diffusion flamelet generated in physical space	`STRAINRATE`
Stoichiometric scalar dissipation for diffusion flamelet	`STOICH_SCADIS` or `CHI`
Stoichiometric mixture fraction	`STOICH_Z`
Temperature	`TEMPERATURE` and `TEMP`

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description	Keyword
Mass fraction	MASSFRACTION-
Mixture fraction	Z
Mole fraction	MOLEFRACTION-

Note:

The strain rate output is written if the flamelets are generated in physical space, for example, using Ansys CHEMKIN.

20.1.8.1. Sample Standard Diffusion Flamelet File

A sample diffusion flamelet file in the standard format is provided below. Note that not all species are listed in this file.

```
HEADER
STOICH_SCADIS   1.000000E-02
NUMOFSPECIES    60
GRIDPOINTS      50
STOICH_Z        6.618652E-02
PRESSURE        1.013250E+05
BODY
Z
 0.000000000E+00 8.273315430E-03 1.240997314E-02 1.447830200E-02 1.551246643E-02
 1.602954865E-02 1.654663086E-02 2.068328857E-02 2.481994629E-02 3.309326172E-02
 4.136657715E-02 4.963989258E-02 5.377655029E-02 5.791320801E-02 6.204986572E-02
 6.618652344E-02 6.912528540E-02 7.206404735E-02 7.500280931E-02 7.794157127E-02
 7.867626176E-02 7.941095225E-02 8.088033323E-02 8.381909519E-02 8.675785714E-02
 8.969661910E-02 9.263538106E-02 9.557414302E-02 1.014516669E-01 1.073291909E-01
 1.132067148E-01 1.190842387E-01 1.249617626E-01 1.367168104E-01 1.425943343E-01
 1.484718583E-01 1.543493822E-01 1.572881441E-01 1.602269061E-01 1.672799348E-01
 1.743329635E-01 1.884390209E-01 2.166511357E-01 2.448632505E-01 2.730753653E-01
 3.407844408E-01 4.084935163E-01 5.709952975E-01 7.659974350E-01 1.000000000E+00
 TEMPERATURE
 3.000000000E+02 6.268546798E+02 7.802882604E+02 8.544333704E+02 8.908936814E+02
 9.089771589E+02 9.269654154E+02 1.067550740E+03 1.203418117E+03 1.462664100E+03
 1.707202552E+03 1.934669997E+03 2.037622483E+03 2.128475386E+03 2.200397351E+03
 2.244572249E+03 2.254177637E+03 2.245072250E+03 2.220638695E+03 2.186735136E+03
 2.177535664E+03 2.168109526E+03 2.148737393E+03 2.108856496E+03 2.068758176E+03
 2.029154219E+03 1.990673114E+03 1.953782923E+03 1.885112633E+03 1.821857119E+03
 1.761797286E+03 1.703540208E+03 1.647197627E+03 1.543457773E+03 1.496624533E+03
 1.452642813E+03 1.410968313E+03 1.390849359E+03 1.371237774E+03 1.326602075E+03
 1.286669965E+03 1.222176660E+03 1.134221753E+03 1.064763074E+03 1.004219101E+03
 8.845385948E+02 7.871510772E+02 6.152847269E+02 4.605312740E+02 3.000000000E+02
 massfraction-h
 0.000000000E+00 8.470246609E-15 1.689876963E-13 7.823505733E-13 1.604836416E-12
 2.259465253E-12 3.137901769E-12 2.297692554E-11 1.299314819E-10 2.009221085E-09
 3.364446760E-08 4.504321512E-07 1.488600091E-06 4.041919731E-06 9.259044043E-06
 1.764854561E-05 2.446982906E-05 2.948355957E-05 3.067005444E-05 2.858096141E-05
 2.763204904E-05 2.660609671E-05 2.447858861E-05 2.024662524E-05 1.571691024E-05
 1.207106883E-05 9.769042791E-06 8.144795604E-06 5.683704776E-06 3.862438419E-06
 2.529575950E-06 1.577642294E-06 9.493198194E-07 3.597224020E-07 2.397461608E-07
 1.672683885E-07 1.109095223E-07 8.352078253E-08 5.847760877E-08 2.024498078E-08
 6.804126487E-09 1.241975019E-09 1.664800458E-10 3.824650210E-11 1.058872219E-11
 1.103611172E-12 2.070333150E-13 5.521048377E-15 6.116361056E-17 0.000000000E+00
 massfraction-o2
 2.330000000E-01 2.037872969E-01 1.891809523E-01 1.818777426E-01 1.782261116E-01
 1.764002861E-01 1.745744526E-01 1.599674402E-01 1.453586904E-01 1.161277458E-01
 8.685446531E-02 5.764270922E-02 4.337649028E-02 2.986903537E-02 1.792882492E-02
 8.656537748E-03 4.260825775E-03 1.718417261E-03 5.787962652E-04 1.809733781E-04
 1.319197735E-04 9.626119601E-05 5.195145104E-05 1.624530279E-05 5.000207346E-06
 1.625791857E-06 5.935574928E-07 2.314963476E-07 5.242565474E-08 1.388575920E-08
 4.396692299E-09 1.701489188E-09 7.617025651E-10 3.172983452E-10 2.146818725E-10
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
 1.500353242E-10 1.052693603E-10 8.680804793E-11 7.032480088E-11 3.785587351E-11
 1.627110024E-11 1.631110236E-12 2.316226342E-14 1.904385354E-16 8.159316969E-19
 2.851864505E-21 7.569901388E-23 1.016058744E-25 1.304158252E-26 0.000000000E+00
 massfraction-oh
 .  .   .    .    .
 .  .   .    .    .
 .  .   .    .    .
```

20.1.8.2. Missing Species

Ansys Fluent will check whether all species in the flamelet data file exist in the thermodynamic properties databases `thermo.db`. If any of the species in the flamelet file do not exist, Ansys Fluent will issue an error message and halt the flamelet import. If this occurs, you can either add the missing species to the database, or remove the species from the flamelet file.

You should not remove a species from the flamelet data file unless its species concentration is very small (10^{-3} or less) throughout the flamelet profile. If you remove a low-concentration species, you will not have the species concentrations available for viewing in the Ansys Fluent calculation, but the accuracy of the Ansys Fluent calculation will otherwise be unaffected.

Important:

If you choose to remove any species, be sure to also update the number of species (keyword `NUMOFSPECIES`) in the flamelet data file, to reflect the loss of any species you have removed from the file.

If a species with relatively large concentration is missing from the Ansys Fluent thermodynamic databases, you will have to add it. Removing a high-concentration species from the flamelet file is not recommended.

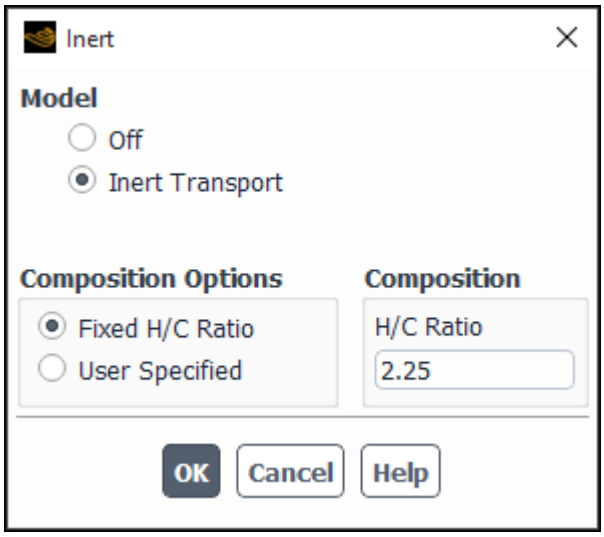
20.1.9. Setting Up the Inert Model

This section describes how to set up and apply the inert model. For a discussion about the theory, refer to Using the Non-Premixed Model with the Inert Model in the Theory Guide.

To enable the inert model, make sure that the non-premixed or partially premixed model is selected in the **Species Model** dialog box, or that a PDF file is read. Refer to Steps in Using the Non-Premixed Model (p. 2471) and Modeling Partially Premixed Combustion (p. 2543) to learn more about these models.

Setup → Models → Inert → Edit...

Figure 20.28: The Inert Model Dialog Box



The **Inert** dialog box will be displayed (Figure 20.28: The Inert Model Dialog Box (p. 2522)). To enable this model, select **Inert Transport**. The following steps will walk you through setting up the inert model:

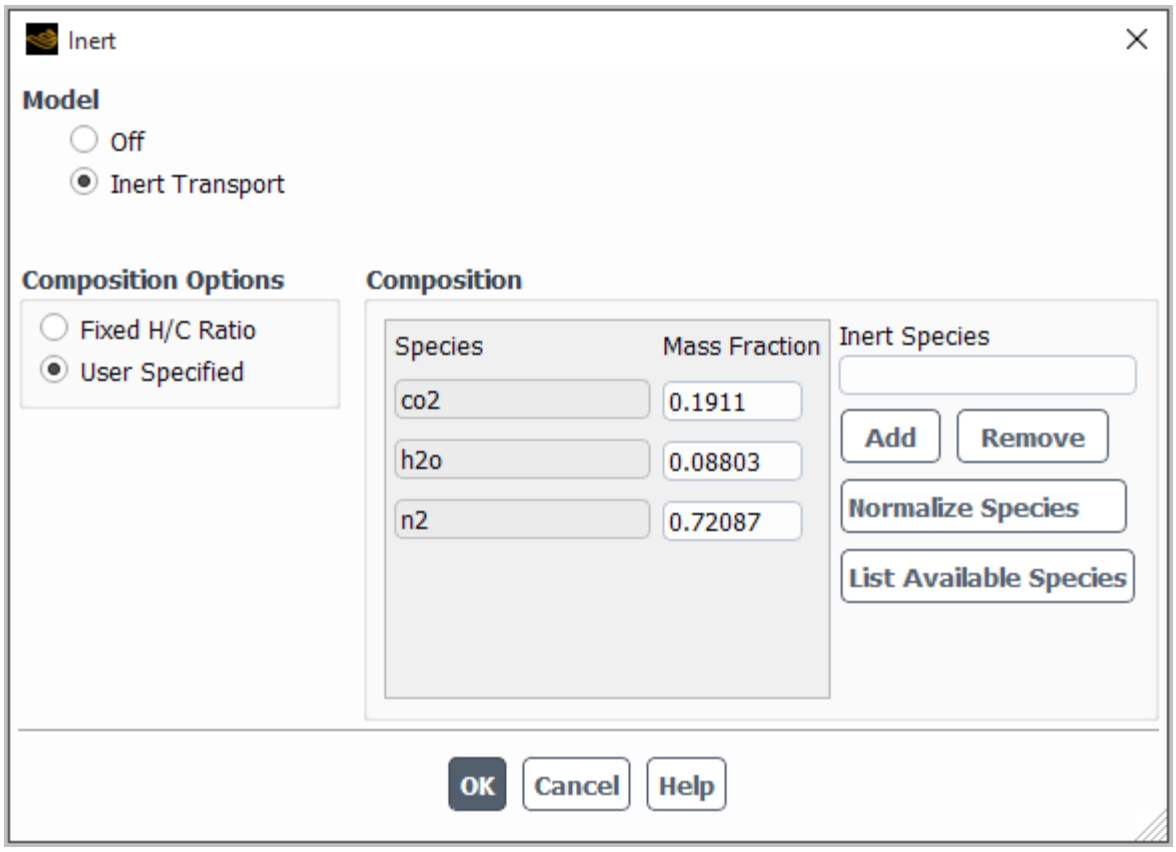
1. Select **Fixed H/C Ratio** as the **Composition Option** if the hydrogen to carbon ratio is known. For example, for methane (CH_4) enter 4 for **H/C Ratio**.

   Setting the H/C ratio assumes that the burned gas resulted from the complete, stoichiometric combustion of that hydrocarbon fuel with air, and the only products of the combustion are CO_2, H_2O and N_2.

2. Select **User Specified** as the **Composition Option** if you want to specify an arbitrary composition for the inert stream, as shown in Figure 20.29: The Inert Model Dialog Box (p. 2523).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 20.29: The Inert Model Dialog Box



You can specify your composition stream by adding or removing species if your stream is composed of species other than the default species list.

a. To add species to the **Species** list, type the chemical formula under **Inert Species** and click **Add**.

b. Enter the **Mass Fraction** of the newly added species. Continue in this manner until all of the inert species you want to include are shown in the **Species** list.

c. Make sure the sum of the mass fractions add up to 1. Ansys Fluent will normalize the species mass fractions for you when you click **Normalize Species**.

d. To remove a species from the list, type the chemical formula under **Inert Species** and click **Remove**.

e. To print a list of all species in the thermodynamic database file (`thermo.db`) in the console window, click **List Available Species**.

20.1.9.1. Setting Boundary Conditions for Inert Transport

You will need to set appropriate boundary conditions at flow inlets and exits for the inert tracer mass fraction, Y_I. The tracer species mass fraction must be between zero and one, with the value of one meaning that all of the material entering the domain comes from the inert stream. The values for flow boundaries are set in the **Inert Stream** field of the inlet boundary condition dialog boxes, under the **Species** tab.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

20.1.9.2. Initializing the Inert Stream

The main assumption of the inert model is that the composition of the inert stream does not change with combustion. For some dilutants, this is a very reasonable assumption, however, it is not valid for rich combustion where there is fuel in the exhaust stream. For cases where there is fuel or oxidizer left in the exhaust gas, the accuracy of your results will depend upon taking the fuel or oxidizer species into account when setting initial conditions.

20.1.9.2.1. Inert Fraction

Initialization of the inert mass fraction Y_I is done in the same way as other variables: by entering in the appropriate value in the **Solution Initialization** task page. Another option for initialization is to patch the value of Y_I in a region of the domain. When the value of Y_I is patched in this way, Ansys Fluent automatically recalculates the enthalpy field for the current temperature field in order to account for the change in composition.

See Patching Values in Selected Cells (p. 3630) for details about patching values of solution variables.

20.1.9.2.2. Inert Composition

For combustion calculations that burn a hydrocarbon fuel, Ansys Fluent provides a straightforward way of setting the initial composition of the inert stream. The inert composition can be set by assuming a ratio of hydrogen to carbon in the following overall oxidation reaction (from Heywood [63] (p. 5716)):

$$C_aH_b+\left(a+\frac{b}{4}\right)\left(O_2+3.773N_2\right)=aCO_2+\frac{b}{2}H_2O+3.773\left(a+\frac{b}{4}\right)N_2 \tag{20.8}$$

which can be rewritten in terms of the ratio of hydrogen to carbon atoms ($y=b/a$) in the fuel as

$$CH_y+\left(1+\frac{y}{4}\right)\left(O_2+3.773N_2\right)=CO_2+\frac{y}{2}H_2O+3.773\left(1+\frac{y}{4}\right)N_2 \tag{20.9}$$

If Equation 20.9 (p. 2524) is solved for the mass fractions of CO_2, H_2O and N_2, the following relations are obtained:

$$\begin{aligned} Y_{CO_2} &= \frac{44.01}{y_{tot}} \\ Y_{H_2O} &= \frac{9.01y}{y_{tot}} \\ Y_{N_2} &= \frac{26.56192y+106.24768}{y_{tot}} \end{aligned} \tag{20.10}$$

where

$$y_{tot}=35.57192y+150.25768$$

Setting the H/C ratio assumes that the burned gas resulted from the complete, stoichiometric combustion of that hydrocarbon fuel, and the only products of the combustion are CO_2, H_2O and N_2. An arbitrary composition for the inert stream can also be specified in the interface.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

20.1.9.3. Resetting Inert EGR

Ansys Fluent allows burned gases to be converted to an inert gas. This has been designed with in-cylinder combustion in mind to aid the simulation of multiple cycles of such engines using the partially-premixed combustion model with exhaust gas recirculation (EGR). At the end of a combustion stroke, the premixed progress variable of the burnt gases is unity. When fresh charge, with progress variable of zero, is mixed with the burnt trapped gases, combustion will be initiated unless these trapped gases are converted to inert.

This facility is accessed via the **Dynamic Mesh Events** dialog box, as described in Resetting Inert EGR (p. 1903). There, you will need to set the crank angle at which the event occurs (usually shortly before the inlet valves open) and at the fluid zones (usually the combustion chamber) where the burnt gases are converted to inert.

When Ansys Fluent performs an inert reset, the inert composition is calculated as the stoichiometric composition. For lean mixtures, the stoichiometric inert is mixed with oxidizer, and for rich mixtures the stoichiometric inert is mixed with fuel, so that overall stoichiometry is conserved. This entails:

- Calculating the stoichiometric mixture fraction, which is defined as the mixture where both fuel and oxidizer are completely consumed.
- Calculating the inert species mass fractions as the stoichiometric species mass fractions. Species with mole fractions less than 0.01 are discarded.
- Setting the inert mass fraction and the mixture fraction as follows:
  - Lean mixtures

$$Y_i = Y_{i,old} + \left(1.0 - Y_{i,old}\right) \cdot \tilde{c} \cdot \frac{\tilde{f}}{\tilde{f}_{sto}}$$

$$\tilde{f} = \tilde{f}_{old} \cdot \left(1.0 - \tilde{c}\right)$$

  - Rich mixtures

$$Y_i = Y_{i,old} + \left(1.0 - Y_{i,old}\right) \cdot \tilde{c} \cdot \frac{1.0 - \tilde{f}_{old}}{1.0 - \tilde{f}_{sto}}$$

$$\tilde{f} = \tilde{f}_{old} \cdot \left(1.0 - \tilde{c}\right) + \tilde{c} \cdot \frac{\tilde{f}_{old} - \tilde{f}_{sto}}{1.0 - \tilde{f}_{sto}}$$

    where Y_i is the inert mass fraction, f is the mixture fraction, c is the reaction progress, subscript *sto* denotes stoichiometric, subscript *old* denotes values before inert reset, tilde superscript denotes Favre averaging.

- Setting the reaction progress to zero.

  For the G-Equation model, G (the mean distance from the flame front) is set to a negative value.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Since there will be some small errors in the species mass fractions from before the EGR reset, the temperature will change due to the change in the specific heat. By default, Ansys Fluent adjusts the enthalpy so that temperature is unchanged after EGR reset. This may be limited by the PDF table temperature limits for the new mixture fraction, in which case the temperature after inert reset will not equal the temperature before the inert reset event.

20.1.10. Defining Non-Premixed Boundary Conditions

For information about defining non-premixed boundary conditions, see the following sections:

20.1.10.1. Input of Mixture Fraction Boundary Conditions

20.1.10.2. Diffusion at Inlets

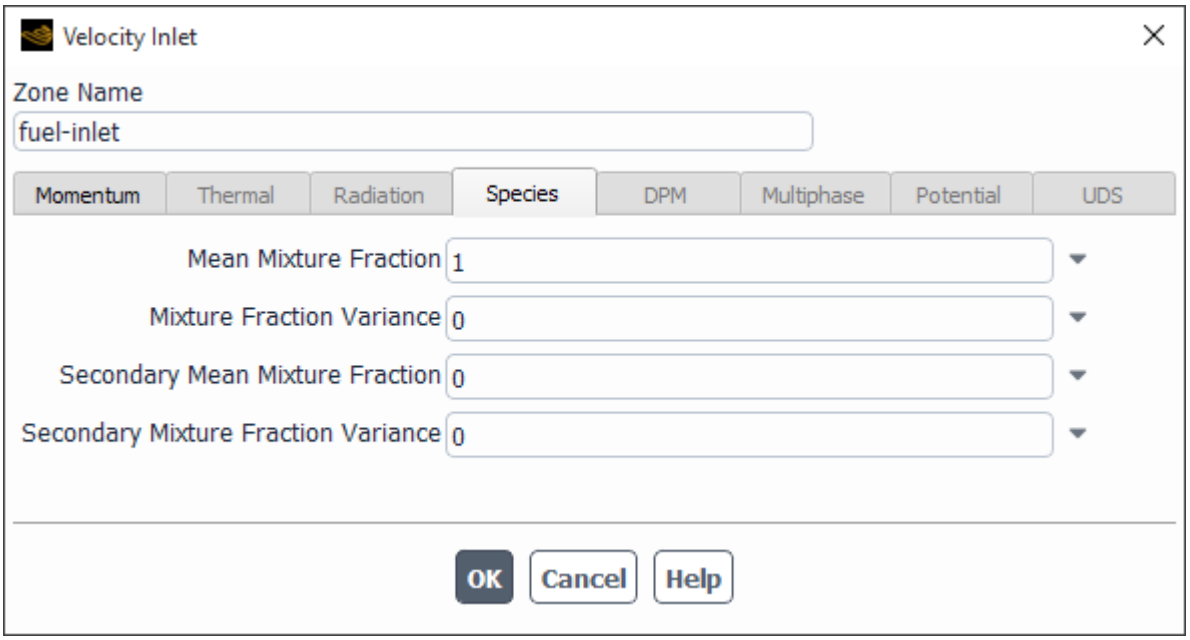
20.1.10.3. Input of Thermal Boundary Conditions and Fuel Inlet Velocities

20.1.10.1. Input of Mixture Fraction Boundary Conditions

When the non-premixed combustion model is used, flow boundary conditions at inlets and exits (that is, velocity or pressure, turbulence intensity) are defined in the usual way. Species mass fractions at inlets are not required. Instead, you define values for the mean mixture fraction, $\overline{f}$, and the mixture fraction variance, $\overline{f'^2}$, at inlet boundaries. (For problems that include a secondary stream, you will define boundary conditions for the mean secondary partial fraction and its variance as well as the mean fuel mixture fraction and its variance.) These inputs provide boundary conditions for the conservation equations you will solve for these quantities. The inlet values are supplied in the boundary conditions task page, under the available tabs, for the selected inlet boundary (for example, Figure 20.30: The Velocity Inlet Dialog Box Showing Mixture Fraction Boundary Conditions (p. 2526)).

Setup → Boundary Conditions

Figure 20.30: The Velocity Inlet Dialog Box Showing Mixture Fraction Boundary Conditions



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Click the **Species** tab and specify the **Mean Mixture Fraction** and **Mixture Fraction Variance** (and the **Secondary Mean Mixture Fraction** and **Secondary Mixture Fraction Variance**, if you are using two mixture fractions). In general, the inlet value of the mean fractions will be 1.0 or 0.0 at flow inlets: the mean fuel mixture fraction will be 1.0 at fuel stream inlets and 0.0 at oxidizer or secondary stream inlets; the mean secondary mixture fraction will be 1.0 at secondary stream inlets and 0.0 at fuel or oxidizer inlets. The fuel or secondary mixture fraction will lie between 0.0 and 1.0 only if you are modeling flue gas recycle, as illustrated in Figure 8.15: Using the Non-Premixed Model with Flue Gas Recycle and discussed in Definition of the Mixture Fraction in the Theory Guide. The fuel or secondary mixture fraction variance can usually be taken as zero at inlet boundaries.

20.1.10.2. Diffusion at Inlets

In some cases, you may want to include the diffusive transport of mixture fraction through the inlets of your domain. You can do this by enabling inlet mixture-fraction diffusion. By default, Ansys Fluent excludes the diffusion flux of mixture fraction at inlets. To enable inlet diffusion, use either the

```
define/models/ species/inlet-diffusion?
```

text command, or the **Species Model** dialog box (Figure 20.1: Defining Equilibrium Chemistry (p. 2473))

Setup → **Models** → **Species** **Edit...**

and enable the **Inlet Diffusion** option.

20.1.10.3. Input of Thermal Boundary Conditions and Fuel Inlet Velocities

If your model is non-adiabatic, you should specify the **Temperature** at the flow inlets. Recall that the inlet temperatures were requested during the table construction in the **Chemistry** tab of the **Species Model** dialog box, and were used in the construction of the look-up tables. The inlet temperatures for each fuel, oxidizer, and secondary inlet in your non-adiabatic model should be defined, in addition, as boundary conditions in Ansys Fluent. It is acceptable for the inlet temperature boundary conditions to differ slightly from those you input for the look-up table calculations. If the inlet temperatures differ significantly from those in the **Chemistry** tab, however, your look-up tables may provide inaccurate interpolation. This is because the discrete points in the look-up tables were clustered around a finite range of the temperatures defined in the **Chemistry** tab, and your input in the Ansys Fluent inlet boundary condition may fall outside this range.

Wall thermal boundary conditions should also be defined for non-adiabatic non-premixed combustion calculations. You can use any of the standard conditions available in Ansys Fluent, including specified wall temperature, heat flux, external heat transfer coefficient, or external radiation. If radiation is to be included within the domain, the wall emissivity should be defined as well. See Thermal Boundary Conditions at Walls (p. 1467) for details about thermal boundary conditions at walls.

20.1.11. Defining Non-Premixed Physical Properties

When you use the non-premixed combustion model, the material used for all fluid zones is automatically set to **pdf-mixture**. This material is a special case of the mixture material concept discussed in Mixture Materials (p. 2357). The constituent species of this mixture are the species that were calculated in the PDF look-up table creation; you cannot change them directly. When the non-premixed model is used, heat capacities, molecular weights, and enthalpies of formation for each species considered

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

are extracted from the chemical database, so you will not modify any properties for the constituent species in the PDF mixture. For the PDF mixture itself, the mean density and mean specific heat are determined from the look-up tables.

The physical property inputs for a non-premixed combustion problem are therefore only the transport properties (viscosity, thermal conductivity, and so on) for the PDF mixture. To set these in the Create/Edit Materials Dialog Box (p. 4885), choose **mixture** as the **Material Type**, **pdf-mixture** (the default, and only choice) in the **Mixture Materials** list, and set the desired values for the transport properties.

Setup → Materials

See Physical Properties (p. 1597) for details about setting physical properties. The transport properties in a non-premixed combustion problem can be defined as functions of temperature, if desired, but not as functions of composition. In practice, since turbulence effects will dominate, it will be of little benefit to include even the temperature dependence of the laminar transport properties.

If you are modeling radiation heat transfer, you will also enter radiation properties, as described in Radiation Properties (p. 1665). Composition-dependent absorption coefficients (using the WSGGM) are allowed.

The non-premixed model can also be used with real gas models in Ansys Fluent, if the **Compressibility Effects** option is enabled in the **Species Model** dialog box. In this case, the density method is based on one of the four real gas models, discussed in Equation of State (p. 1686) and Using the Cubic Equation of State Real Gas Models (p. 1692).

20.1.12. Solution Strategies for Non-Premixed Modeling

The non-premixed model setup and solution procedure in Ansys Fluent differs slightly for single- and two-mixture-fraction problems. Below, an overview of each approach is provided. Note that your Ansys Fluent case file must always meet the restrictions listed for the non-premixed modeling approach in Restrictions on the Mixture Fraction Approach in the Theory Guide. In this section, details are provided regarding the problem definition and calculation procedures you follow in Ansys Fluent.

For additional information, see the following sections:

20.1.12.1. Single-Mixture-Fraction Approach

20.1.12.2. Two-Mixture-Fraction Approach

20.1.12.3. Starting a Non-Premixed Calculation From a Previous Case File

20.1.12.4. Solving the Flow Problem

20.1.12.1. Single-Mixture-Fraction Approach

For a single-mixture-fraction system, when you have completed the calculation of the PDF look-up tables, you are ready to begin your reacting flow simulation. In Ansys Fluent, you will solve the flow field and predict the spatial distribution of $\overline{f}$ and $\overline{f'^2}$ (and $\overline{H}$ if the system is non-adiabatic or $\overline{\chi}$ if the system is based on laminar flamelets). Ansys Fluent will obtain the corresponding values of temperature and individual chemical species mass fractions from the look-up tables.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

20.1.12.2. Two-Mixture-Fraction Approach

When a secondary stream is included, Ansys Fluent will solve transport equations for the mean secondary partial fraction (p_{sec}) and its variance in addition to the mean fuel mixture fraction and its variance. Ansys Fluent will then look up the instantaneous values for temperature, density, and individual chemical species in the look-up tables, compute the PDFs for the fuel and secondary streams, and calculate the mean values for temperature, density, and species.

Note that in order to avoid both inaccuracies and unnecessarily slow calculation times, it is important for you to view your temperature and species tables in Ansys Fluent to ensure that they are adequately but not excessively resolved.

20.1.12.3. Starting a Non-Premixed Calculation From a Previous Case File

You can read a previously defined Ansys Fluent case file as a starting point for your non-premixed combustion modeling. If this case file contains inputs that are incompatible with the current non-premixed combustion model, Ansys Fluent will alert you when the non-premixed model is turned on and it will turn off those incompatible models. For example, if the case file includes species that differ from those included in the PDF file created by Ansys Fluent, these species will be disabled. If the case file contains property descriptions that conflict with the property data in the chemical database, these property inputs will be ignored.

Important:

PDF files created by prePDF 2 or older are not supported by this version of Ansys Fluent. The files generated by PrePDF version 3 or newer, are fully compatible.

In the **Species Model** dialog box, select **Non-Premixed Combustion** under the **Model** heading. When you click **OK** in the **Species Model** dialog box, The Select File Dialog Box (p. 931) will immediately appear, prompting you for the name of the PDF file containing the look-up tables created in a previous Ansys Fluent session. (The PDF file is the file you saved using the **File/Write/PDF...** ribbon tab item after computing the look-up tables.) Ansys Fluent will indicate that it has successfully read the specified PDF file:

```
Reading "/home/mydirectory/adiabatic.pdf"...
read 5 species (binary c, adiabatic fluent)
pdf file successfully read.
Done.
```

After you read in the PDF file, Ansys Fluent will inform you that some material properties have changed. You can accept this information; you will be updating properties later on.

You can read in an altered PDF file at any time by using the **File/Read/PDF...** ribbon tab item.

Important:

Recall that the non-premixed combustion model is available only when you used the pressure-based solver; it cannot be used with the density-based solvers. Also, the non-premixed combustion model is available only when turbulence modeling is active.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you are modeling a non-adiabatic system and you want to include the effects of compressibility, re-open the **Species Model** dialog box and turn on **Compressibility Effects** under **PDF Options**. This option tells Ansys Fluent to update the density according to Equation 20.1 (p. 2478). When the non-premixed combustion model is active, you can enable compressibility effects only in the **Species Model** dialog box. For other models, you will specify compressible flow (**ideal-gas**, **boussinesq**, and so on) in the **Create/Edit Materials** dialog box. See Specifying the Operating Pressure for the System (p. 2478) and Tuning the PDF Parameters for Two-Mixture-Fraction Calculations (p. 2531) for more information about compressibility effects.

20.1.12.3.1. Retrieving the PDF File During Case File Reads

The PDF filename is specified to Ansys Fluent only once. Thereafter, the filename is stored in your Ansys Fluent case file and the PDF file will be automatically read into Ansys Fluent whenever the case file is read. Ansys Fluent will remind you that it is reading the PDF file after it finishes reading the rest of the case file by reporting its progress in the text (console) window.

Note that the PDF filename stored in your case file may not contain the full name of the directory in which the PDF file exists. The full directory name will be stored in the case file only if you initially read the PDF file through the GUI (or if you typed in the directory name along with the filename when using the text interface). In the event that the full directory name is absent, the automatic reading of the PDF file may fail (since Ansys Fluent does not know which directory to look in for the file), and you will need to manually specify the PDF file. The safest approaches are to use the GUI when you first read the PDF file or to supply the full directory name when using the text interface.

20.1.12.4. Solving the Flow Problem

The next step in the non-premixed combustion modeling process in Ansys Fluent is the solution of the mixture fraction and flow equations. First, initialize the flow. By default, the mixture fraction and its variance have initial values of zero, which is the recommended value; you should generally not set nonzero initial values for these variables. See Initializing the Solution (p. 3627) for details about solution initialization.

Solution → **Initialization**

Next, begin calculations in the usual manner.

Solution → **Run Calculation**

During the calculation process, Ansys Fluent reports residuals for the mixture fraction and its variance in the `fmean` and `fvar` columns of the residual report:

```
iter       cont       x-vel       y-vel          k     epsilon       fmean       fvar
  28    1.57e-3     4.92e-4     4.80e-4    2.68e-2     2.59e-3     9.09e-1    1.17e+0
  29    1.42e-3     4.43e-4     4.23e-4    2.48e-2     2.30e-3     8.89e-1    1.15e+0
  30    1.28e-3     3.98e-4     3.75e-4    2.29e-2     2.04e-3     8.88e-1    1.14e+0
```

(For two-mixture-fraction calculations, columns for `psec` and `pvar` will also appear.)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

20.1.12.4.1. Under-Relaxation Factors for PDF Equations

The transport equations for the mean mixture fraction and mixture fraction variance are quite stable and high, under-relaxation can be used when solving them. By default, an under-relaxation factor of 1 is used for the mean mixture fraction (and secondary partial fraction) and 0.9 for the mixture fraction variance (and secondary partial fraction variance). If the residuals for these equations are increasing, you should consider decreasing these under-relaxation factors, as discussed in Setting Under-Relaxation Factors (p. 3589).

20.1.12.4.2. Density Under-Relaxation

One of the main reasons a combustion calculation can have difficulty converging is that large changes in temperature cause large changes in density, which can, in turn, cause instabilities in the flow solution. Ansys Fluent allows you to under-relax the change in density to alleviate this difficulty. The default value for density under-relaxation is 1, but if you encounter convergence trouble you may want to reduce this to a value between 0.5 and 1 (in the **Solution Controls** task page).

Note that if you are using a real-gas model the solution might converge at a slower rate than when running ideal-gas flow. To improve the convergence performance of your analysis, you need to reduce the underrelaxation for density to values as low as 0.1.

20.1.12.4.3. Tuning the PDF Parameters for Two-Mixture-Fraction Calculations

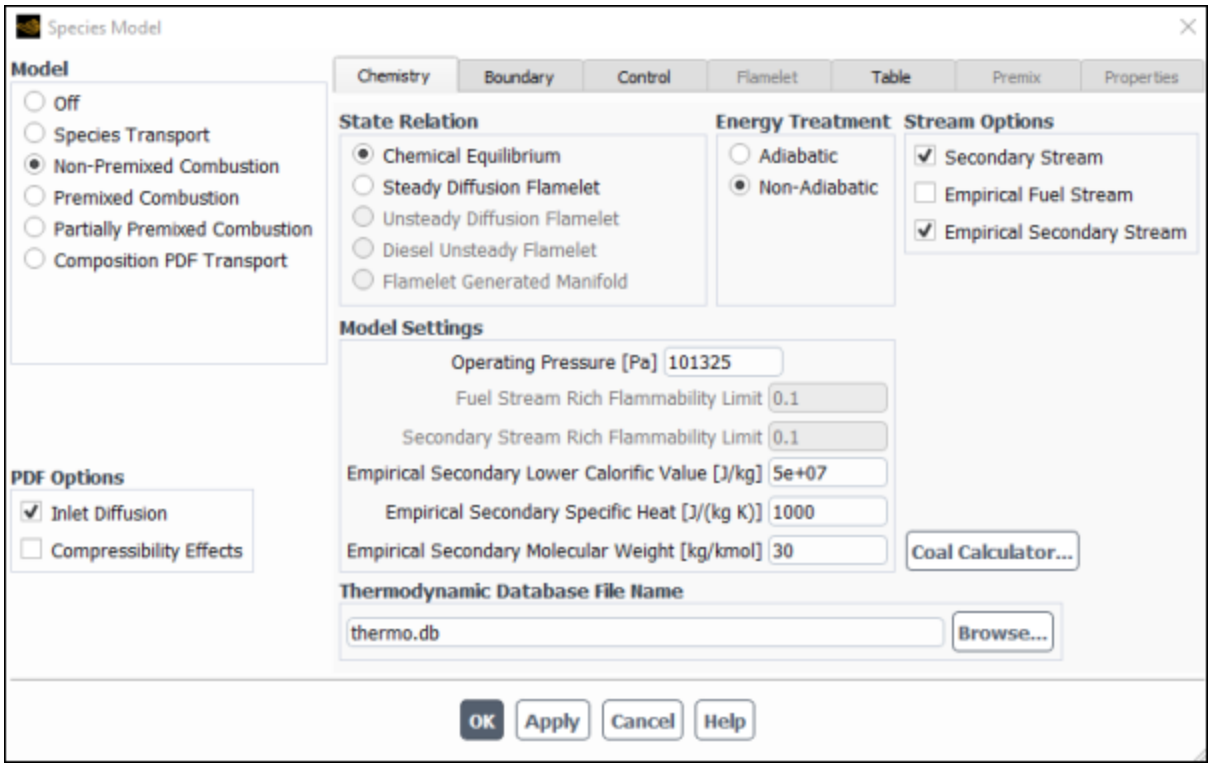
For cases that include a secondary stream, the PDF integrations are performed inside Ansys Fluent.

The parameters for these integrations are defined in the Species Model Dialog Box (p. 4768) (Figure 20.31: The Species Model Dialog Box for a Two-Mixture-Fraction Calculation (p. 2532)).

Setup → **Models** → **Species** **Edit...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 20.31: The Species Model Dialog Box for a Two-Mixture-Fraction Calculation



The parameters are as follows:

Compressibility Effects

(non-adiabatic systems only) tells Ansys Fluent to update the density, temperature, species mass fraction, and enthalpy from the PDF tables to account for the varying pressure of the system.

20.1.13. Enabling Robust Numerics for Combustion with a PDF Table

You may be able to increase the robustness of combustion simulations that use a PDF table by applying poor mesh numerics during the calculation. When you initialize or read data for a case with non-premixed or partially premixed combustion, a cell register named `pdf_stability_marker` is created (which marks cells where the enthalpy—as defined by the **PDF Table Heat Loss/Gain** field variable described in Alphabetical Listing of Field Variables and Their Definitions (p. 4205)—is below the minimum PDF table value), and this cell register is used as part of the settings for a register-based definition named `pmn_pdf_stability_marker` that applies poor mesh numerics treatment every 5 iterations (for steady cases) or every 2 time steps (for unsteady cases). Note that this definition is automatically created, but is deactivated by default; in order to use it in the current session of Fluent, you must enter the following text command prior to running the calculation:

```
> solve/set/poor-mesh-numerics/register-based/edit pmn_pdf_stability_marker active? yes
```

If you would like to use such definitions in all future sessions of Fluent without having to enter the previous text command, you can enable the **Robust Numerics for Combustion with PDF Tables** option in the **Simulation** branch of the **Preferences** dialog box (accessed through **File/Preferences...**).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that if the default settings for this treatment are not appropriate for your case, you can edit the cell register and/or register-based definition for poor mesh numerics, as described in Manage Cell Registers (p. 3864) and Robustness with Meshes of Poor Quality (p. 3741), respectively.

20.1.14. Postprocessing the Non-Premixed Model Results

The final step in the non-premixed combustion modeling process is the postprocessing of species concentrations and temperature data from the mixture fraction and flow-field solution data. The following variables are of particular interest:

- **Mean Mixture Fraction** (in the **Pdf...** category)
- **Secondary Mean Mixture Fraction** (in the **Pdf...** category)
- **Mixture Fraction Variance** (in the **Pdf...** category)
- **Secondary Mixture Fraction Variance** (in the **Pdf...** category)
- **Fvar Prod** (in the **Pdf...** category, which is the production term in the mixture fraction variance transport equation)
- **Fvar2 Prod** (in the **Pdf...** category)
- **Scalar Dissipation** (in the **Pdf...** category)
- **PDF Table Adiabatic Enthalpy** (in the **Pdf...** category)
- **PDF Table Heat Loss/Gain** (in the **Pdf...** category)
- **Mass fraction of *species-n*** (in the **Species...** category)
- **Mole fraction of *species-n*** (in the **Species...** category)
- **Molar Concentration of *species-n*** (in the **Species...** category)
- **RMS *species-n* Mass Fraction** (in the **Species...** category)
- **Static Temperature** (in the **Temperature...** category)
- **RMS Temperature** (in the **Temperature...** category)
- **Enthalpy** (in the **Temperature...** category)
- **Probability** (in the **Unsteady Flamelet...** category)
- **Mean Temperature** (in the **Unsteady Flamelet...** category)
- **Mean Mass Fraction of *species-n*** (in the **Unsteady Flamelet...** category)

Important:

For the unsteady diffusion flamelet model, mean species mass fractions are displayed for the first fifty species in the flamelet kinetic mechanism.

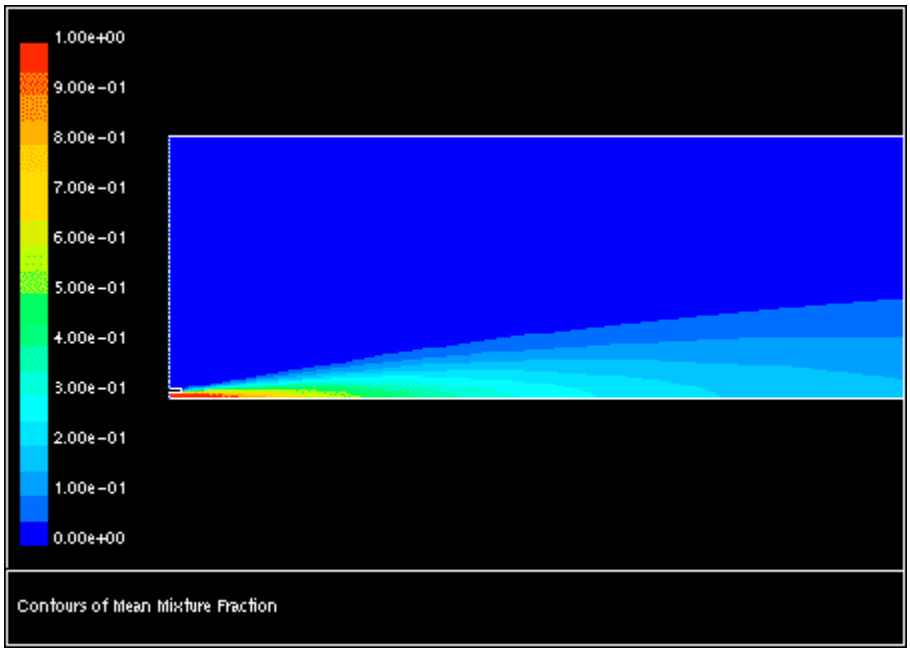
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These quantities can be selected for display in the indicated category of the variable-selection drop-down list that appears in postprocessing dialog boxes. See Field Function Definitions (p. 4167) for their definitions.

In all cases, the species concentrations are derived from the mixture fraction/variance field using the look-up tables. Note that temperature and enthalpy can be postprocessed even when your Ansys Fluent model is an adiabatic non-premixed combustion simulation in which you have not solved the energy equation. In both the adiabatic and non-adiabatic cases, the temperature is derived from the look-up table.

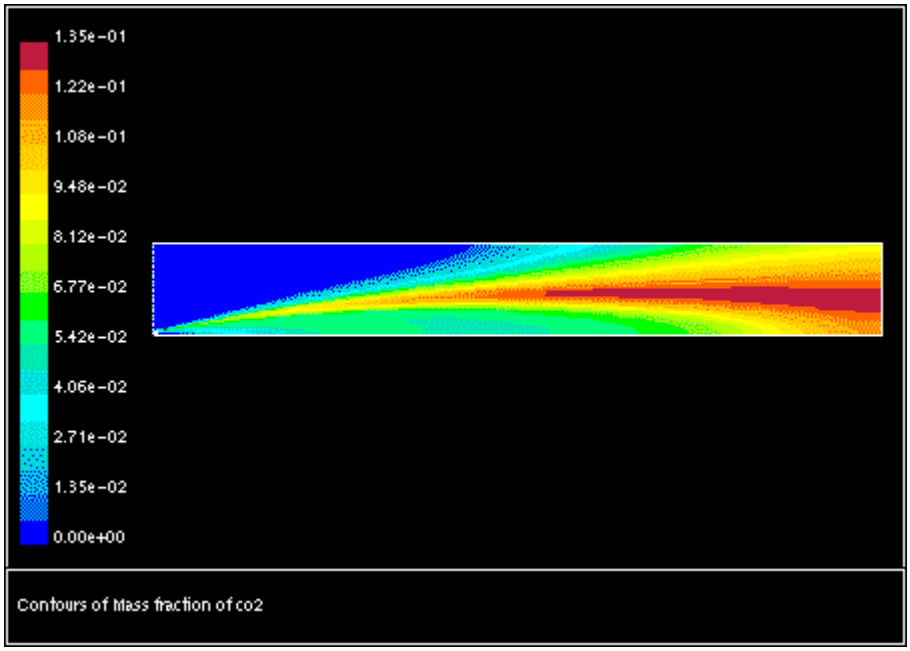
Figure 20.32: Predicted Contours of Mixture Fraction in a Methane Diffusion Flame (p. 2534) and Figure 20.33: Predicted Contours of CO2 Mass Fraction Using the Non-Premixed Combustion Model (p. 2535) illustrate typical results for a methane diffusion flame modeled using the non-premixed approach.

Figure 20.32: Predicted Contours of Mixture Fraction in a Methane Diffusion Flame



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 20.33: Predicted Contours of CO2 Mass Fraction Using the Non-Premixed Combustion Model



For additional information, see the following section:

20.1.14.1. Postprocessing for Inert Calculations

20.1.14.1. Postprocessing for Inert Calculations

Ansys Fluent provides several additional reporting options for post-processing calculations with the inert model. You can generate graphical plots or alphanumeric reports of the same items that are available with the non-premixed or partially premixed models, and in addition the following variables are available:

- **Inert Mass Fraction**
- **Inert Specific Heat**
- **Inert Density**
- **Inert Enthalpy**
- **Pdf Enthalpy**
- **Pdf Fmean**
- **Pdf Fvar**
- **Mass Fraction of Inert** Y_i

where the mass fraction of the i^{th} inert species, Y_i, is calculated as

$$Y_i = Y_I Y_i^I$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where Y_I is the inert tracer and Y_i^I is the mass fraction of the i^{th} inert species defined in the **Inert** dialog box.

Note these variables appear in the **Inert...** category.

20.2. Modeling Premixed Combustion

This sections discusses how to simulate premixed turbulent combustion in Ansys Fluent. For theoretical background on this model, see Premixed Combustion in the Theory Guide.

Information about using this model is provided in the following sections:

20.2.1. Limitations of the Premixed Combustion Model
20.2.2. Using the Premixed Combustion Model
20.2.3. Setting Up the C-Equation and G-Equation Models
20.2.4. Postprocessing for Premixed Combustion Calculations

20.2.1. Limitations of the Premixed Combustion Model

The following limitations apply to the premixed combustion model:

- You must use the pressure-based solver. The premixed combustion model is not available with the density-based solver.
- The premixed combustion model is valid only for turbulent, subsonic flows. These types of flames are called deflagrations. Explosions, also called detonations, where the combustible mixture is ignited by the heat behind a shock wave, can be modeled with the finite-rate model using the density-based solver. See Modeling Species Transport and Finite-Rate Chemistry (p. 2355) for information about the finite-rate model.
- The premixed combustion model cannot be used in conjunction with the pollutant (that is, soot and NOx) models. However, a perfectly premixed system can be modeled with the partially premixed model (see Modeling Partially Premixed Combustion (p. 2543)), which can be used with the pollutant models.
- You cannot use the premixed combustion model to simulate reacting discrete-phase particles, because these would result in a partially premixed system. Only inert particles can be used with the premixed combustion model.
- The G-Equation model can be used only with the unsteady solver because it tracks the flame front in time. However, RANS solutions, which tend to a steady-state, can be modeled by evolving them in time until the solution is stationary.

20.2.2. Using the Premixed Combustion Model

The procedure for setting up and solving a premixed combustion model is outlined below, and then described in detail. Remember that only the steps that are pertinent to premixed combustion modeling are shown here. For information about inputs related to other models that you are using in conjunction with the premixed combustion model, see the appropriate sections for those models.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Enable the premixed turbulent combustion model and set the related parameters.

   Note:

   Make sure a turbulence model (other than the Spalart-Allmaras model) is selected before selecting a combustion model.

   Setup → **Models** → **Species** **Edit...**

2. Define the physical properties for the unburnt and burnt material in the domain.

   Setup → **Materials** → **Create/Edit...**

3. Set the value of the progress variable c at flow inlets and exits.

   Setup → **Boundary Conditions**

4. Initialize and patch the value of the progress variable.

   Solution → **Initialization** → **Patch...**

5. Solve the problem and perform postprocessing.

   Important:

   If you are interested in computing the concentrations of individual species in the domain, you can use the partially premixed model described in Modeling Partially Premixed Combustion (p. 2543). Alternatively, compositions of the unburnt and burnt mixtures can be obtained from external analyses using equilibrium or kinetic calculations.

For additional information, see the following sections:

20.2.2.1. Enabling the Premixed Combustion Model

20.2.2.2. Choosing an Adiabatic or Non-Adiabatic Model

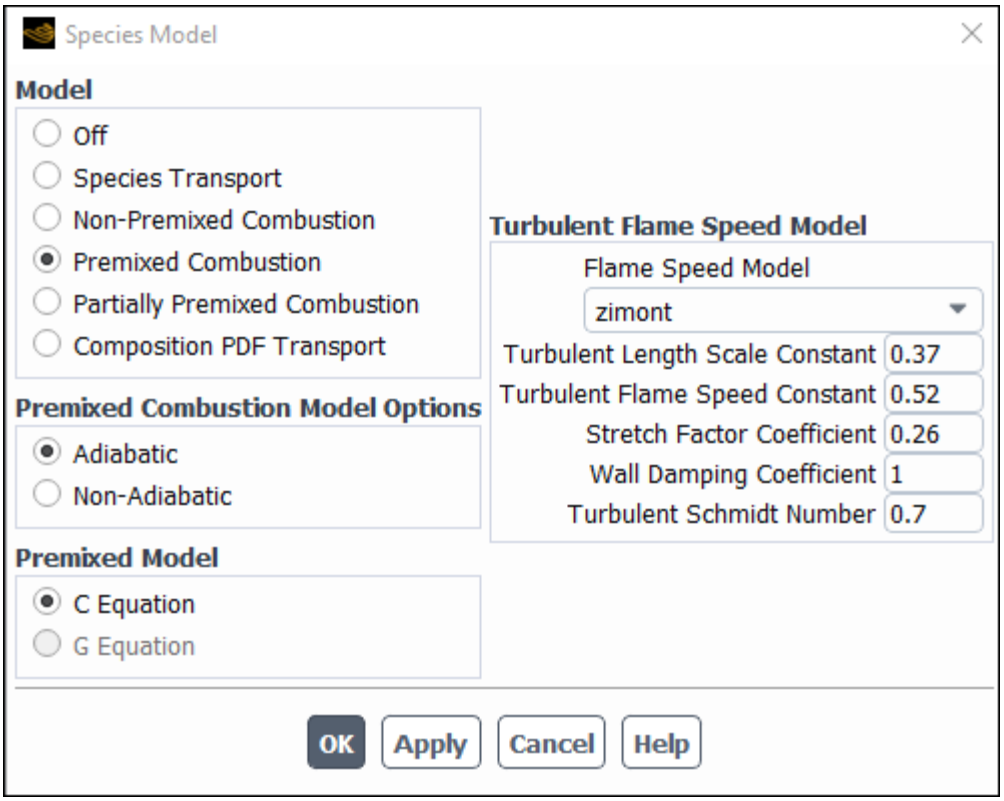
20.2.2.1. Enabling the Premixed Combustion Model

To enable the premixed combustion model, select **Premixed Combustion** under **Model** in the Species Model Dialog Box (p. 4768) (Figure 20.34: The Species Model Dialog Box for Premixed Combustion (p. 2538)).

Setup → **Models** → **Species** **Edit...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 20.34: The Species Model Dialog Box for Premixed Combustion



When you enable **Premixed Combustion**, the dialog box expands to show the relevant inputs.

20.2.2.2. Choosing an Adiabatic or Non-Adiabatic Model

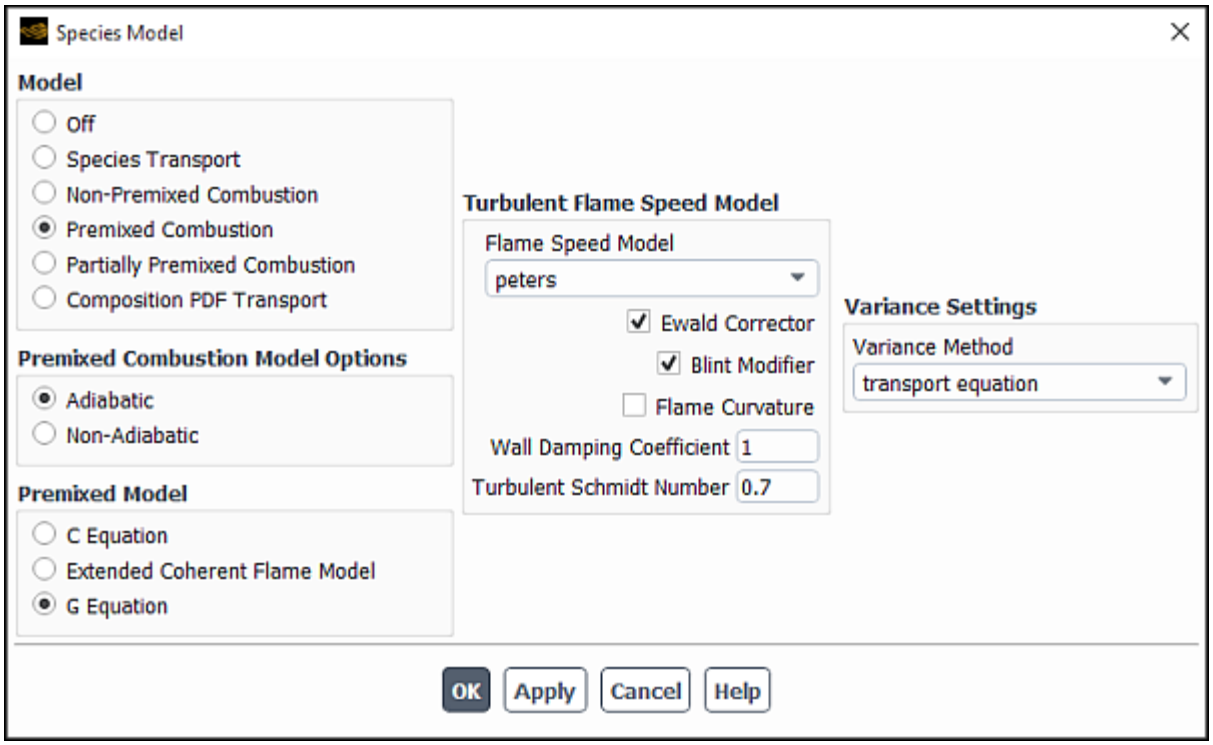
Under **Premixed Combustion Model Options** in the Species Model Dialog Box (p. 4768), choose either **Adiabatic** (the default) or **Non-Adiabatic**. This choice affects only the calculation method used to determine the temperature (either Equation 8.97 or Equation 8.98 in the Theory Guide).

20.2.3. Setting Up the C-Equation and G-Equation Models

When **C Equation** or **G Equation** is selected as the **Premixed Model**, you can then choose to use the **zimont** or **peters Flame Speed Model** (described in Turbulent Flame Speed Models). You can also modify a number of model constants, as described in Modifying the Constants for the Zimont Flame Speed Model (p. 2539) and Modifying the Constants for the Peters Flame Speed Model (p. 2540).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 20.35: The Species Model Dialog Box for the G-Equation Model



For a non-adiabatic premixed combustion model, note that the value you specify for the **Turbulent Schmidt Number** will also be used as the Prandtl number for energy. (The **Energy Prandtl Number** will therefore not appear in the **Viscous Model** dialog box for non-adiabatic premixed combustion models.) These parameters control the level of diffusion for the progress variable and for energy. The progress variable is closely related to energy (because the flame progress results in heat release), so it is important that the transport equations use the same level of diffusion.

For additional information, see the following sections:

20.2.3.1. Modifying the Constants for the Zimont Flame Speed Model

20.2.3.2. Modifying the Constants for the Peters Flame Speed Model

20.2.3.3. Additional Options for the G-Equation Model

20.2.3.4. Defining Physical Properties for the Unburnt Mixture

20.2.3.5. Setting Boundary Conditions for the Progress Variable

20.2.3.6. Initializing the Progress Variable

20.2.3.1. Modifying the Constants for the Zimont Flame Speed Model

In general, you will not need to modify the constants used in the equations presented in C-Equation Model Theory in the Theory Guide. The default values are suitable for a wide range of premixed flames.

You can set the **Turbulent Length Scale Constant** (C_D in Equation 8.79), **Turbulent Flame Speed Constant** (A in Equation 8.77), the **Stretch Factor Coefficient** (μ_{str} in Equation 8.85), the **Turbulent Schmidt Number** (Sc_t in Equation 8.70), and the **Wall Damping Coefficient** (α_w in Equation 8.88).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

20.2.3.2. Modifying the Constants for the Peters Flame Speed Model

The Peters turbulent flame speed model constants, as described in Peters Flame Speed Model in the Theory Guide, are not available in the GUI because they are generally suitable for a wide range of premixed flames. They can, however, be accessed from the TUI.

The **Ewald Corrector** is enabled by default and described in Peters Flame Speed Model. The **Turbulent Schmidt Number** (Sc_t in Equation 8.70), and the **Wall Damping Coefficient** (α_w in Equation 8.88) are the same as those described for the Zimont model.

20.2.3.3. Additional Options for the G-Equation Model

When the G-Equation model is enabled, the **G Equation Settings** group box appears. You can select either the **transport equation** or **algebraic** option for the calculation of the flame distance variance. Consult Peters Flame Speed Model in the Theory Guide for the variance transport and algebraic equation expressions (Equation 8.74 and Equation 8.75). It is recommended that you use the **transport equation** option for RANS and the **algebraic** option for LES.

When **Flame Curvature Source** is enabled, the curvature source term in the G-Equation, which is the last term in Equation 8.72, is included. By default, this term is excluded.

20.2.3.4. Defining Physical Properties for the Unburnt Mixture

The fluid material in your domain should be assigned the properties of the unburnt mixture, including the thermal diffusivity (α in Equation 8.77 in the Theory Guide). α is defined as $k/\rho c_p$, and values at standard conditions can be found in combustion handbooks (for example, [83] (p. 5718)).

For both adiabatic and non-adiabatic combustion models, you will need to specify the **Laminar Flame Speed** (U_l in Equation 8.77 in the Theory Guide) as a material property, in the **Create/Edit Materials** dialog box. You may choose to enter a **constant** value, use a **user-defined** function, or apply the **metghalchi-keck-law**. See Laminar Flame Speed in the Theory Guide and Defining Physical Properties for the Unburnt Mixture (p. 2540) for information about setting the other properties for the unburnt material.

When using the Zimont turbulent flame speed model, if you want to include the flame stretch effect in your model, you will also need to specify a lower value than the default **Critical Rate of Strain** (g_{cr} in Equation 8.86 in the Theory Guide). As discussed in Flame Stretch Effect in the Theory Guide, g_{cr} is set to a very high value ($1\times10^8 s^{-1}$) by default, so no flame stretching occurs. To include flame stretching effects, you will need to adjust the **Critical Rate of Strain** based on experimental data for the burner. Because the flame stretching and flame extinction can influence the turbulent flame speed (as discussed in Flame Stretch Effect in the Theory Guide), a realistic value for the **Critical Rate of Strain** is required for accurate predictions. Typical values for CH_4 lean premixed combustion range from 3000 to 8000 s^{-1} [177] (p. 5723). Note that you can specify constant values or user-defined functions to define the **Laminar Flame Speed** and **Critical Rate of Strain**. See the Fluent Customization Manual for details about user-defined functions.

For adiabatic models, you will also specify the **Adiabatic Burnt Temperature** (T_{ad} in Equation 8.97 in the Theory Guide), which is the temperature of the burnt products under adiabatic conditions. This temperature will be used to determine the linear variation of temperature in an adiabatic premixed combustion calculation. You can specify a constant value or use a user-defined function.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For non-adiabatic models, you will instead specify the **Heat of Combustion** per unit mass of fuel and the **Unburnt Fuel Mass Fraction** (H_{comb} and Y_{fuel} in Equation 8.99 in the Theory Guide). Ansys Fluent will use these values to compute the heat losses or gains due to combustion, and include these losses/gains in the energy equation that it uses to calculate temperature. The **Heat of Combustion** can be specified only as a constant value, but you can specify a constant value or use a user-defined function for the **Unburnt Fuel Mass Fraction**.

To specify the density for a premixed combustion model, choose **premixed-combustion** in the **Density** drop-down list and set the **Adiabatic Unburnt Density** and **Adiabatic Unburnt Temperature** (T_u and ρ_u in Equation 8.100 in the Theory Guide). For adiabatic premixed models, your input for **Adiabatic Unburnt Temperature** (T_u) will also be used in Equation 8.97 in the Theory Guide to calculate the temperature.

The other properties specified for the unburnt mixture are viscosity, specific heat, thermal conductivity, and any other properties related to other models that are being used in conjunction with the premixed combustion model.

20.2.3.5. Setting Boundary Conditions for the Progress Variable

For premixed combustion models, you will need to set an additional boundary condition at flow inlets and exits: the progress variable, c. Valid inputs for the **Progress Variable** are as follows:

- $\overline{c}=0$: unburnt mixture
- $\overline{c}=1$: burnt mixture

20.2.3.6. Initializing the Progress Variable

Often, it is sufficient to initialize the progress variable c to 1 (burnt) everywhere and allow the unburnt ($c=0$) mixture entering the domain from the inlets to blow the flame back to the stabilizer. A better initialization is to patch an initial value of 0 (unburnt) upstream of the flame holder and a value of 1 (burnt) in the downstream region (after initializing the flow field in the **Solution Initialization** task page).

Solution → **Initialization** → **Patch...**

See Patching Values in Selected Cells (p. 3630) for details about patching values of solution variables.

20.2.4. Postprocessing for Premixed Combustion Calculations

Ansys Fluent provides several additional reporting options for premixed combustion calculations. You can generate graphical plots or alphanumeric reports of the following items:

- **Progress Variable**
- **Damkohler Number**
- **Stretch Factor**
- **Turbulent Flame Speed**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Static Temperature**
- **Product Formation Rate**
- **Laminar Flame Speed**
- **Critical Strain Rate**
- **Unburnt Fuel Mass Fraction**
- **Adiabatic Flame Temperature**

These variables are contained in the **Premixed Combustion...** category of the variable selection drop-down list that appears in postprocessing dialog boxes. See Field Function Definitions (p. 4167) for a complete list of flow variables, field functions, and their definitions. Displaying Graphics (p. 3869) and Reporting Alphanumeric Data (p. 4097) explain how to generate graphics displays and reports of data.

Note that **Static Temperature** and **Adiabatic Flame Temperature** will appear in the **Premixed Combustion...** category only for adiabatic premixed combustion calculations; for non-adiabatic calculations, **Static Temperature** will appear in the **Temperature...** category. **Unburnt Fuel Mass Fraction** will appear only for non-adiabatic models.

Ansys Fluent also provides two additional reporting options for premixed combustion calculations with the G-Equation model:

- **Mean Distance from Flame**
- **Variance of Distance from Flame**

The **Variance of Distance from Flame** is the variance of the distance of the instantaneous flame from the mean flame position. These variables appear in the **Premixed Combustion...** category.

For additional information, see the following section:

20.2.4.1. Computing Species Concentrations

20.2.4.1. Computing Species Concentrations

If you know the composition of the unburnt and burnt mixtures in your model (that is, if you have performed separate Ansys Fluent or external analyses of chemical equilibrium calculations or 1D premixed flames), you can compute the species concentrations in the domain using custom field functions:

- To determine the concentration of a species in the unburnt mixture, define the custom function $Y_u\left(1-c\right)$, where Y_u is the mass fraction for the species in the unburnt mixture (specified by you) and c is the value of the progress variable (computed by Ansys Fluent).
- To determine the concentration of a species in the burnt mixture, define the custom function $Y_b c$, where Y_b is the mass fraction for the species in the burnt mixture (specified by you) and c is the value of the progress variable (computed by Ansys Fluent).

See Custom Field Functions (p. 4285) for details about defining and using custom field functions.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

20.3. Modeling Partially Premixed Combustion

This section describes how to use the Ansys Fluent's partially-premixed combustion model that is based on the non-premixed combustion model (Modeling Non-Premixed Combustion (p. 2471)) and the premixed combustion model (Modeling Premixed Combustion (p. 2536)). For information about the theory behind the partially premixed combustion model, see Partially Premixed Combustion in the *Fluent Theory Guide*.

Information about using the partially premixed combustion model is presented in the following sections:

20.3.1. Limitations

20.3.2. Using the Partially Premixed Combustion Model

20.3.1. Limitations

- The underlying theory, assumptions, and limitations of the non-premixed and premixed models apply directly to the partially premixed model. In particular, the single-mixture-fraction approach is limited to two inlet streams, which may be pure fuel, pure oxidizer, or a mixture of fuel and oxidizer. The two-mixture-fraction model extends the number of inlet streams to three, but incurs a major computational overhead. See Limitations of the Premixed Combustion Model (p. 2536) for additional limitations.
- The Flamelet Generated Manifold (FGM) model is limited to the c-Equation partially-premixed model.

20.3.2. Using the Partially Premixed Combustion Model

The procedure for setting up and solving a partially premixed combustion problem combines parts of the non-premixed combustion setup and the premixed combustion setup. An outline of the procedure is provided in Setup and Solution Procedure (p. 2543), along with information about where to look in the non-premixed and premixed combustion sections for details. Inputs that are specific to the partially premixed combustion model are provided in this section.

Information is provided in the following sections:

20.3.2.1. Setup and Solution Procedure

20.3.2.2. Importing a Flamelet

20.3.2.3. Flamelet Generated Manifold

20.3.2.4. Calculating the Look-Up Tables

20.3.2.5. Standard Files for Flamelet Generated Manifold Modeling

20.3.2.6. Setting Premix Flame Propagation Parameters

20.3.2.7. Modifying the Unburnt Mixture Property Polynomials

20.3.2.8. Modeling Strained Laminar Flame Speed

20.3.2.9. Modeling In Cylinder Combustion

20.3.2.10. Postprocessing for FGM Scalar Transport Calculations

20.3.2.1. Setup and Solution Procedure

1. Read your mesh file into Ansys Fluent and set up any other models you plan to use in conjunction with the partially premixed combustion model (turbulence, radiation, and so on).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Enable the partially premixed combustion model.

   a. Turn on the **Partially Premixed Combustion** model in the **Species Model** dialog box.

   Note:

   Make sure a turbulence model (other than the Spalart-Allmaras model) is selected before selecting a combustion model.

   Setup → Models → Species Edit...

   b. If you select the **C Equation** as the **Premixed Model**, then you have the choice of selecting one of the following **State Relation** models in the **Chemistry** tab: **Chemical Equilibrium**, **Steady Diffusion Flamelet**, **Unsteady Diffusion Flamelet** (for steady-state simulations starting from a converged Steady Diffusion Flamelet solution), or **Flamelet Generated Manifold**.

   c. If you select the **G Equation** as the **Premixed Model**, then you have the choice of selecting one of the following **State Relation** models in the **Chemistry** tab: **Chemical Equilibrim** or **Steady Diffusion Flamelet**.

3. In the **Chemistry** tab of the **Species Model** dialog box, select the appropriate **State Relation** based on the type of partially premixed model you want to simulate. In Ansys Fluent, there are two types of partially premixed models, namely a thin-flamelet model (equilibrium and diffusion flamelet) and a finite-thickness flamelet model, which is a Flamelet Generated Manifold. You can refer to Non-Premixed Combustion in the *Fluent Theory Guide* to learn about diffusion flamelets or Partially Premixed Combustion Theory in the *Fluent Theory Guide* for FGMs. Also, you can refer to Overview in the *Fluent Theory Guide* for general information about the thin-flamelet and finite-thickness flamelet models.

4. If you select **Flamelet Generated Manifold**, you will also need to define whether the FGM is calculated from a premixed or a diffusion flamelet, as well as the **Turbulence-Chemistry Interaction** in the **Premix** tab. The available options (**Finite-Rate** and **Turbulent Flame Speed**) are discussed in FGM Turbulent Closure in the *Fluent Theory Guide*.

5. In the **Flamelet** tab, you can modify the **Flamelet Parameters** as described in Flamelet Generated Manifold (p. 2546).

6. Generate a PDF look-up table. The table parameters are defined in Calculating the Look-Up Tables (p. 2554).

   Important:

   If Ansys Fluent warns you during the partially premixed properties calculation that any parameters are out of the range for the laminar flame speed function, you will need to modify the piecewise-linear points manually before saving the PDF file. See Modifying the Unburnt Mixture Property Polynomials (p. 2564) for details. Also, the calculation of the thermal diffusivity uses the thermal conductivity in the **Create/Edit Materials** dialog box. More accurate thermal diffusivity polynomials can be obtained

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

by editing the thermal conductivity in the **Create/Edit Materials** dialog box and then clicking **Recalculate Properties** in the **Properties** tab.

7. Define the physical properties for the unburnt material in the domain.

   Setup → Materials

   Ansys Fluent will automatically select the **prepdf-polynomial** function for **Laminar Flame Speed**, indicating that the piecewise-linear polynomial function from the PDF look-up table will be used to compute the laminar flame speed. You may also choose to enter a **constant** value, use a **user-defined** function, **metghalchi-keck-law**, **laminar-flame-speed-computed**, or apply **laminar-flame-speed-library** instead of a piecewise-linear polynomial function. See Laminar Flame Speed in the *Fluent Theory Guide* and Defining Physical Properties for the Unburnt Mixture (p. 2540) for information about setting the other properties for the unburnt material.

8. Set the values for the mean progress variable ($\overline{c}$) and the mean mixture fraction ($\overline{f}$) and its variance ($\overline{f'^2}$) at flow inlets and exits. (For problems that include a secondary stream, you will define boundary conditions for the mean secondary partial fraction and its variance as well.)

   Setup → Boundary Conditions

   See Defining Non-Premixed Boundary Conditions (p. 2526) for guidelines on setting mixture fraction and variance conditions, as well as thermal and velocity conditions at inlets.

   Important:

   There are two ways to specify a premixed inlet boundary condition:

   a. If you defined the fuel composition in the **Boundary** tab to be the premixed inlet species, then you should set $\overline{f}=1$ and $\overline{c}=0$ in the boundary condition dialog boxes.

   b. If you set the fuel composition to pure fuel in the **Boundary** tab, you will need to set the correct equivalence ratio ($0<\overline{f}<1$) and $\overline{c}=0$ at your premixed inlet boundary condition.

   For example, if the premixed inlet of methane and air is at an equivalence ratio of 0.3, you can

   a. specify the mass fraction of the fuel composition of Y_{CH_4} = 0.017, Y_{O_2} = 0.236, and Y_{N_2} = 0.747 in the **Boundary** tab and $\overline{f}$ = 1 and $\overline{c}$ = 0 in the boundary condition dialog box.

   b. specify the mass fraction of the fuel composition of Y_{CH_4} = 1.0 in the **Boundary** tab and $\overline{f}$ = 0.017 and $\overline{c}$ = 0 in the boundary condition dialog box.

      Method (a) is preferred since it will have more points in the flame zone than method (b).

9. Initialize the value of the progress variable.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution → **Initialization** → **Patch...**

See Initializing the Progress Variable (p. 2541) for details.

10. You may be able to increase the robustness of the calculations by applying poor mesh numerics treatment to the cells that have an enthalpy value that is less than the minimum enthalpy defined in the PDF table. For details on how to easily apply such treatment, see Enabling Robust Numerics for Combustion with a PDF Table (p. 2532).

11. Solve the problem and perform postprocessing.

    See Solving the Flow Problem (p. 2530) for guidelines about setting solution parameters. (These guidelines are for non-premixed combustion calculations, but they are relevant for partially premixed as well.)

20.3.2.2. Importing a Flamelet

To import an existing flamelet file

1. Select the **Import Flamelet** option in the **Chemistry** tab of the **Species Model** dialog box.

2. Click the **Import Flamelet File...** button. In The Select File Dialog Box (p. 931), select the file to be read in to Ansys Fluent.

After you have completed this step, you can skip ahead to the **Table** tab of the **Species Model** dialog box (see Calculating the Look-Up Tables (p. 2554)).

20.3.2.3. Flamelet Generated Manifold

The Ansys Fluent Flamelet Generated Manifold (FGM) model has several advantages over the thin-flame equilibrium or diffusion steady flamelet models, such as the ability to model flame quenching due to dilution. The manifold can be modeled with either a **Premixed Flamelet** or a **Diffusion Flamelet**.

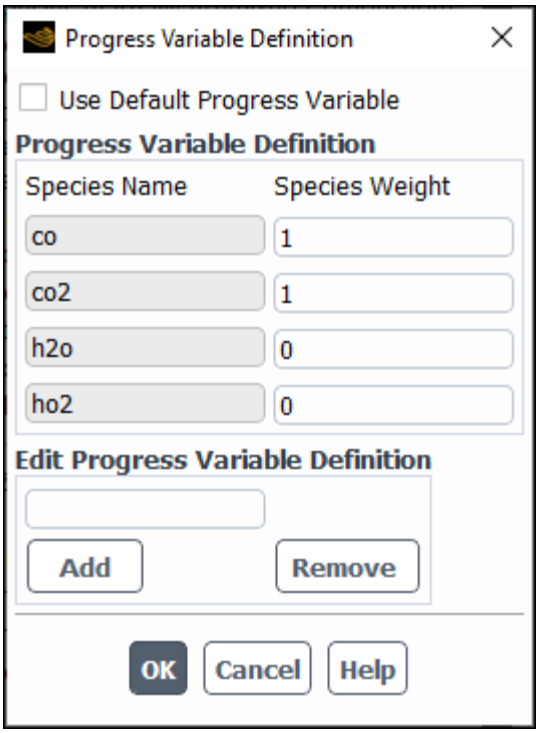
When **Premixed Flamelet** is selected as the **Flamelet Type**, you can select to solve the flamelets either in the **Progress Variable Space** or in the **CHEMKIN Physical Space** (**Flamelet Solution Method** group box). For more information about these methods, see Premixed FGMs in Reaction Progress Variable Space and Premixed FGMs in Physical Space in the *Fluent Theory Guide*.

When the **Flamelet Generated Manifold** state relation is selected in the **Species Model** dialog box (**Chemistry** tab), Ansys Fluent solves the reaction progress variable equation (Equation 8.113 in the *Fluent Theory Guide*). By default, Ansys Fluent uses the definition of the reaction progress variable as described in Premixed FGMs in Reaction Progress Variable Space in the *Fluent Theory Guide*. The default progress variable is a good and recommended choice for a wide range of hydrocarbon flames. However, if you want, you can adjust the definition of the progress variable as follows:

1. In the **Species Model** dialog box, click the **Set Progress Variable...** button (**Boundary** tab) to open the **Progress Variable Definition** dialog box. (See Figure 20.36: The Progress Variable Definition Dialog Box (p. 2547).)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 20.36: The Progress Variable Definition Dialog Box



The species used in the progress variable definition and their weights are displayed under **Species Name** and **Species Weight**, respectively.

2. Disable **Use Default Progress Variable**.

3. Edit the weight of the species as desired.

4. If you want to include a species in the progress variable composition, type its chemical formula (for example, h2o) under **Edit Progress Variable Definition** and click **Add**. The species will be added to the **Species Name** list.

   Similarly, to remove a species from the **Species Name** list, type its chemical formula under **Edit Progress Variable Definition** and click **Remove**.

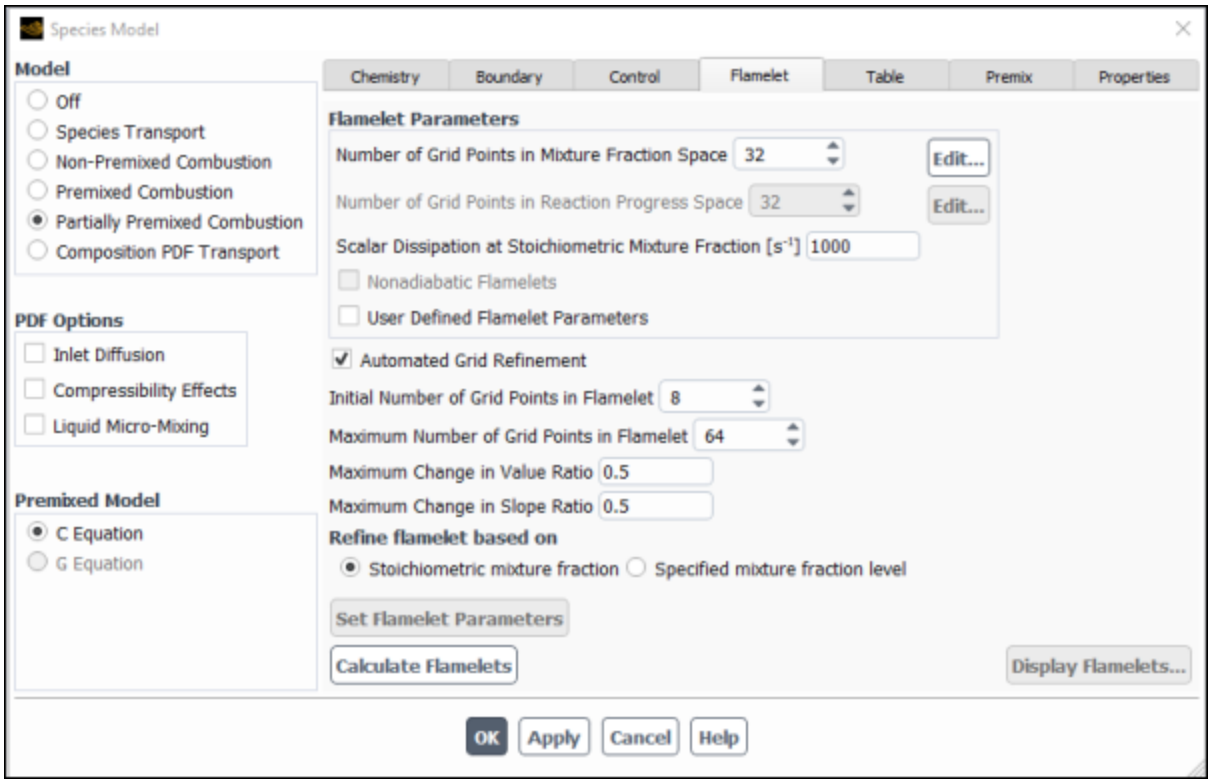
To generate a flamelet file, go to the **Flamelet** tab of the **Species Model** dialog box (Figure 20.16: The Species Model Dialog Box (Flamelet Tab) (p. 2502)), where you will enter values for parameters of the flamelet.

20.3.2.3.1. Premixed Flamelet Generated Manifolds

For premixed Flamelet Generated Manifolds, details about specifying the Scalar Dissipation at Stoichiometric Mixture Fraction can be found in Premixed FGMs in Reaction Progress Variable Space in the *Fluent Theory Guide*.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 20.37: Premixed Flamelet Generated Manifolds (Flamelet Tab)



The **Flamelet Parameters** for premixed FGMs are as follows:

Number of Grid Points in Mixture Fraction Space

specifies the number of mixture fraction grid points distributed between the oxidizer ($f=0$) and the fuel ($f=1$). Increased resolution will provide greater accuracy, but since premixed flamelets are solved at all mixture fraction points, the flamelet computation time will increase linearly with this input.

Number of Grid Points in Reaction Progress Space

specifies the number of reaction progress grid points distributed between unburnt ($c=0$) and burnt ($c=1$) states. Increased resolution will provide greater accuracy. However, when the flamelets are generated in the **Progress Variable Space**, the computation time and the required memory for the flamelets calculation will increase with increasing resolution.

Scalar Dissipation at Stoichiometric Mixture Fraction

specifies the premixed flamelet strain rate. The default of 1000/s is an approximate value of the scalar dissipation of a freely propagating (unstrained) premixed flame at stoichiometric mixture fraction (unity equivalence ratio). This option is available only when the premixed FGM is generated in the **Progress Variable Space**.

User Defined Flamelet Parameters

enables you to hook a user-defined function for scalar dissipation and mean mixture fraction (or progress variable) grid discretization

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Automated Grid Refinement

employs an adaptive algorithm, which inserts grid points so that the change of values, as well as the change of slopes, between successive grid points is less than user-specified tolerances. For information about this option, refer to Steady Diffusion Flamelet Automated Grid Refinement in the Theory Guide. Once this option is enabled, you can specify the following parameters:

Initial Number of Grid Points in Flamelet

calculates a steady solution on a coarse grid, with a default of 8. See Equation 8.54 in the Theory Guide.

Maximum Number of Grid Points in Flamelet

has a default of 64.

Maximum Change in Value Ratio

has a default of 0.5 and is ε_v in Equation 8.28 in the Theory Guide.

Maximum Change in Slope Ratio

has a default of 0.5 and is ε_s in Equation 8.29 in the Theory Guide.

Refine flamelet based on

allows you to select either the **Stoichiometric mixture fraction** or **Specified mixture fraction**. For **Specified mixture fraction**, you must also specify a mixture fraction level at which the flamelet is solved during the grid refinement.

Note:

When **CHEMKIN Physical Space** is selected as the **Flamelet Solution Method** for the **Premixed Flamelet**, the following default values are used for the **Automated Grid Refinement** parameters:

- **Initial Number of Grid Points in Flamelet** = 12
- **Maximum Number of Grid Points in Flamelet** =250
- **Maximum Change in Value Ratio** = 0.5
- **Maximum Change in Slope Ratio** = 0.1

Click **Calculate Flamelets** to begin the premixed flamelet calculation.

20.3.2.3.1.1. Editing the Flamelet Grid Distribution

The default grid distribution for flamelet parameters is sufficient for most of the cases and should not be modified. However, in very specific cases, changing the grid distribution may improve accuracy. You can edit the grid distribution for the mixture fraction, and for with no automated grid refinement (AGR), for the reaction progress variable.

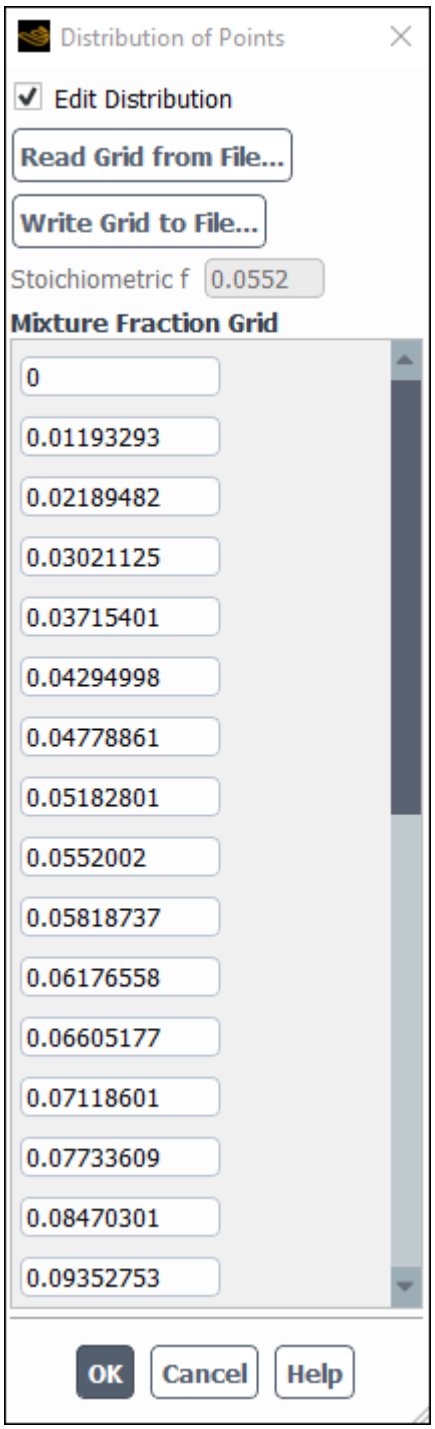
To edit the grid points, follow these steps:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click the **Edit...** button next to the flamelet parameter for which you want to change the distribution grid points.

   The **Distribution of Points** dialog box opens, showing the stoichiometric mixture fraction (in the **Stoichiometric f** field) and the default distribution of the grid points for the chosen flamelet parameter. In the example shown in Figure 20.38: The Distribution of Points Dialog Box (p. 2550), the grid point values are displayed in the **Mixture Fraction Grid** group box.

Figure 20.38: The Distribution of Points Dialog Box



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Enable **Edit Distribution** and modify the values of the grid points for the chosen flamelet parameter.

3. If you want to export the grid point data, use the **Write Grid to File...** button. The data will be saved in an ASCII file in the XY plot file format (see XY Plot File Format (p. 4071) for details). Grid point pairs contain the point numbers and their values, separated by a space.

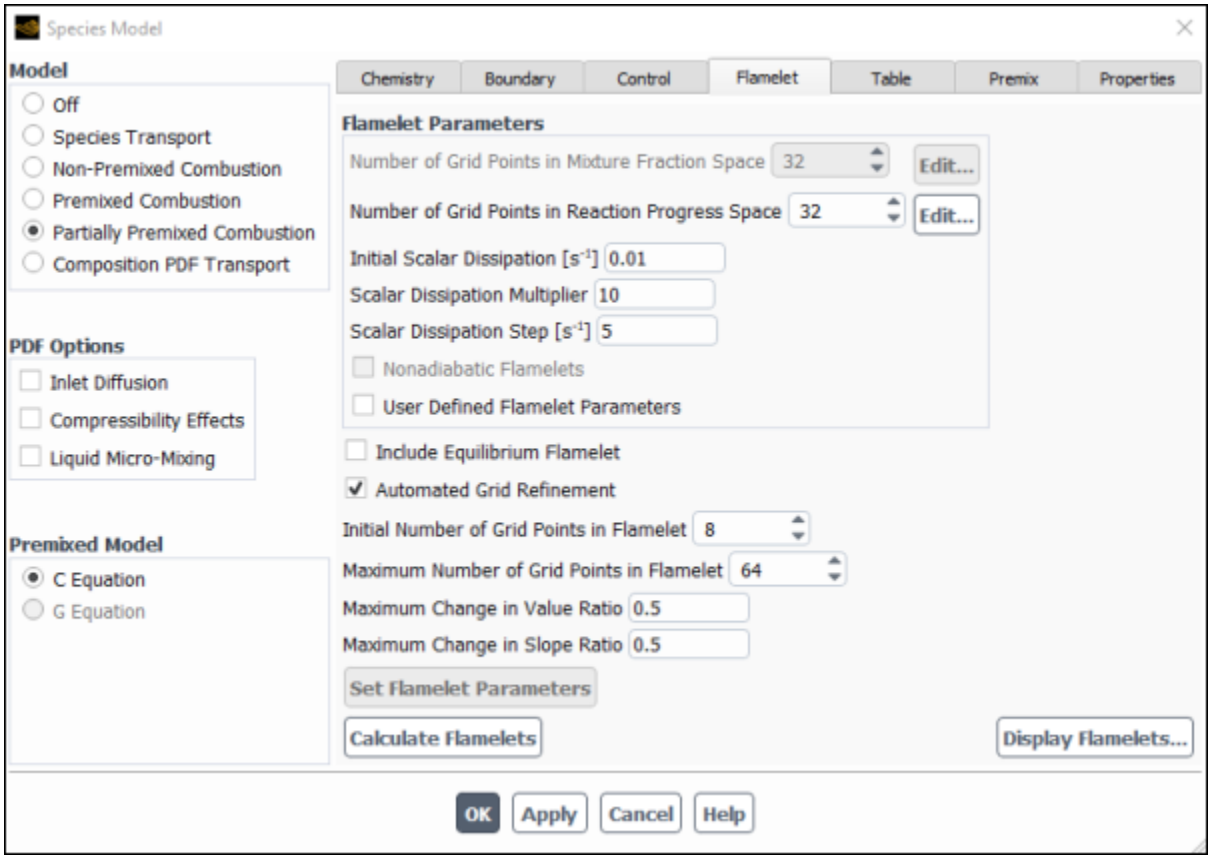
   You can use the **Read Grid from File...** button to import a previously saved grid point file into your case.

4. Click **OK**.

20.3.2.3.2. Diffusion Flamelet Generated Manifolds

Generating diffusion Flamelet Generated Manifolds is similar to generating Steady Diffusion Flamelets, as described in Calculating the Flamelets (p. 2501).

Figure 20.39: Diffusion Flamelet Generated Manifolds (Flamelet Tab)



The parameters for the diffusion FGM flamelet are slightly different than those for the premixed FGM flamelet (see Premixed Flamelet Generated Manifolds (p. 2547) for the definitions of the parameters). The additional diffusion FGM flamelet parameters are as follows:

Initial Scalar Dissipation

is the scalar dissipation of the first flamelet in the library. This corresponds to χ_0 in Equation 8.53 in the *Fluent Theory Guide*.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Scalar Dissipation Multiplier

specifies the ratio of the scalar dissipation step in which successive flamelets are generated when the scalar dissipation is less than 1 s^{-1}. This corresponds to $\Delta\chi$ for $\chi < 1$ in Equation 8.53 in the Theory Guide.

Scalar Dissipation Step

specifies the interval between scalar dissipation values (in s^{-1}) for which multiple flamelets will be calculated. This corresponds to $\Delta\chi$ for $\chi \geq 1$ in Equation 8.53 in the Theory Guide.

Note:

- The FGM model does not require the **Maximum Number of Flamelets** parameter. Instead, Ansys Fluent increases the scalar dissipation until the flamelet extinguishes. Hence, the **Scalar Dissipation Step** should be set at approximately one twentieth of the counterflow diffusion flamelet extinction strain rate, so that about twenty flamelets are calculated. The final, extinguishing unsteady diffusion flamelet is used to model the manifold between the unburnt state (c=0) and the last steady diffusion flamelet at the highest scalar dissipation rate just before extinction.
- **Scalar Dissipation Multiplier** and **Scalar Dissipation Step** are used to specify the interval of scalar dissipation for $\chi < 1$ and $\chi \geq 1$, respectively. For example, for initial scalar dissipation of 1e-3 s^{-1} with a **Scalar Dissipation Multiplier** of 10, and a **Scalar Dissipation Step** of 5, the flamelets will be generated with scalar dissipations of 1e-3, 1e-2, 0.1, 1.0, 6, 11, 16, and so on.

Include Equilibrium Flamelet

determines how the progress variable (Equation 8.107 in the *Fluent Theory Guide*) is calculated. If this option is selected, Ansys Fluent will compute the progress variable using equilibrium. Otherwise, the progress variable will be computed using the solution of flamelet generated with initial scalar dissipation as a denominator in Equation 8.107 in the *Fluent Theory Guide*.

Note:

The default grid distribution for flamelet parameters is sufficient for most of the cases and should not be modified. However, similar to the premixed FGM flamelet, in very specific cases, changing the grid distribution may improve accuracy. You can edit the grid distribution for the mixture fraction, and for cases with no AGR, for the reaction progress variable. The procedure for changing the grid points is described in Editing the Flamelet Grid Distribution (p. 2549).

20.3.2.3.3. Using the Heat Loss Modeling Capability for Nonadiabatic FGM

When the following options are selected in the **Species Model** dialog box (**Chemistry** tab):

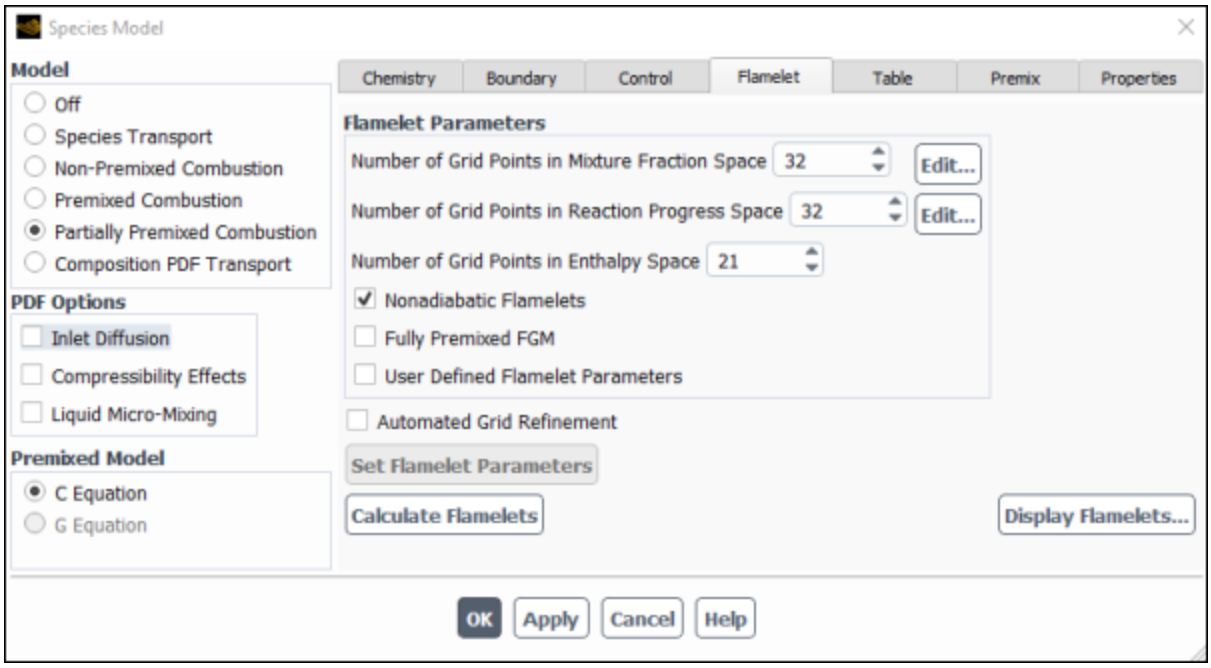
- **Premixed Flamelet** as the **Flamelet Type**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Non-Adiabatic** as the **Energy Treatment**
- **CHEMKIN Physical Space** as the **Flamelet Solution Method**

you can select **Nonadiabatic Flamelets** in the **Flamelet** tab to generate the nonadiabatic flamelets as described in Nonadiabatic Flamelet Generated Manifold (FGM) in the *Fluent Theory Guide*.

Figure 20.40: Non-adiabatic Premixed Flamelet Generated Manifolds (Flamelet Tab)



For nonadiablatic flamelet generation, you need to provide inputs in the **Flamelet Parameters** group box (**Flamelet** tab). Some parameters are the same as for premixed flamelet generated manifolds described in section Premixed Flamelet Generated Manifolds in the Ansys Fluent User's Guide. The additional parameters specific to nonadiabatic flamelets are as follows:

Number of Grid Points in Enthalpy Space

specifies the number of enthalpy levels for which the flamelets will be generated. Ansys Fluent automatically distributes the enthalpy levels based on the reference fuel and oxidizer temperatures. A higher number of points provides the increased resolution in enthalpy space, but requires more computation time because each additional point in enthalpy space will generate a set of flamelets for entire mixture fraction space.

Fully Premixed FGM

is an option to generate nonadiabatic flamelets only for one mixture fraction specified in the **Mixture Fraction Value for Flamelet Generation** entry field. This is useful for fully premixed CFD problems when the mixture fraction is constant, and flamelets are required only for a single mixture fraction. When this option is enabled, the **Number of Grid Points in Mixture Fraction Space** is automatically set to **1**. Since the number of mixture fraction space is only **1**, this option is not only fast to generate the manifold, but it also produces flamelet and PDF files that are greatly reduced size.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

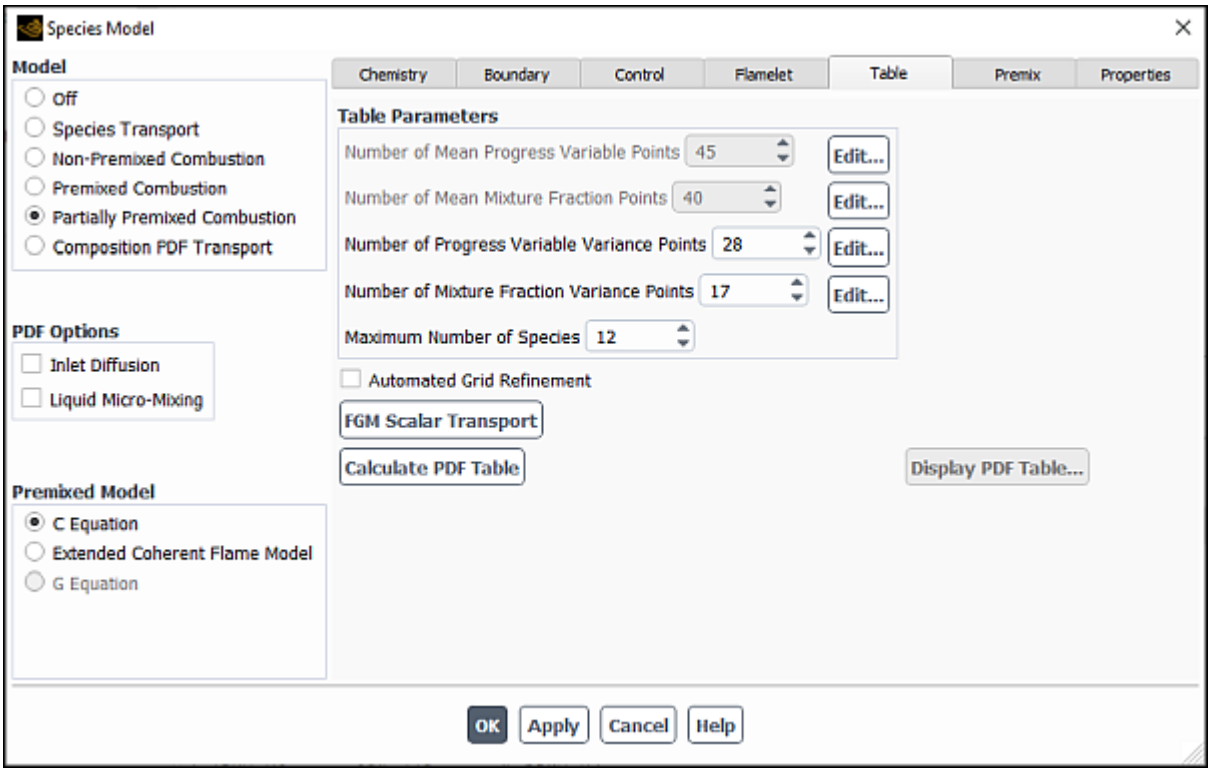
Once you click **Calculate Flamelets**, Ansys Fluent will generate premixed nonadiabatic flamelets as described in Nonadiabatic Flamelet Generated Manifold (FGM) in the *Fluent Theory Guide*.

20.3.2.4. Calculating the Look-Up Tables

Ansys Fluent requires additional inputs that are used in the creation of the look-up tables. Several of these inputs control the number of discrete values for which the look-up tables will be computed. These parameters are input in the **Table** tab of the **Species Model** dialog box.

For the example of modeling a partially-premixed flame with either the premixed or the diffusion FGM model, the **Table** tab of the **Species Model** dialog box is shown in Figure 20.41: The Species Model Dialog Box: Table Tab with no Automated Grid Refinement (p. 2554) when the Automated Grid Refinement (AGR) is disabled and Figure 20.42: The Species Model Dialog Box: Table Tab Displaying Automated Grid Refinement (p. 2556) when the AGR is enabled.

Figure 20.41: The Species Model Dialog Box: Table Tab with no Automated Grid Refinement



When the automated grid refinement is not selected, the look-up table parameters specific to the partial-premixed model are as follows:

Number of Mean Progress Variable Points

is the number of discrete values of $\overline{c}$ at which the look-up tables will be computed. This number is determined from the number of grid points in the flamelet.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Number of Progress Variable Variance Points

is the number of discrete values of $\overline{c'^2}$ at which the look-up tables will be computed. Lower resolution is acceptable because the variation along the $\overline{c'^2}$ axis is, in general, not as steep as the variation along the $\overline{c}$ axis of the look-up tables.

Maximum Number of Species

is the maximum number of species that will be included in the look-up tables. The maximum number of species that can be included is 200. Ansys Fluent will automatically select the species with the largest mole fractions to include in the PDF table. Note that the PDF table values of density and specific heat are pre-calculated with all the species, and hence the convergence behavior of Ansys Fluent will not be affected by the input for the **Maximum Number of Species**. Hence, to keep table sizes small, you should set the **Maximum Number of Species** to only include the species that you are interested in postprocessing.

Number of Mean Enthalpy Points

is the number of discrete values of enthalpy at which the look-up tables will be computed. This input is required only if you are modeling a non-adiabatic system. The number of points required will depend on the chemical system that you are considering, with more points required in high heat release systems (for example, hydrogen/oxygen flames).

Minimum Temperature

is used to determine the lowest temperature for which the look-up tables are generated. Your input should correspond to the minimum temperature expected in the domain (for example, an inlet or wall temperature). The minimum temperature should be set 10–20 K below the minimum system temperature. This option is available only if you are modeling a non-adiabatic system.

Note:

The default grid distribution for table parameters is sufficient for most of the cases and should not be modified. However, similar to flamelet parameters, the grid distribution can be modified when the AGR is disabled. Changing the grid distribution should be done only in very specific cases in order to improve accuracy. The procedure for changing the grid points is described in Editing the Flamelet Grid Distribution (p. 2549).

You can edit the grid point for the following table parameters:

- Mean mixture fraction
- Progress variable variance

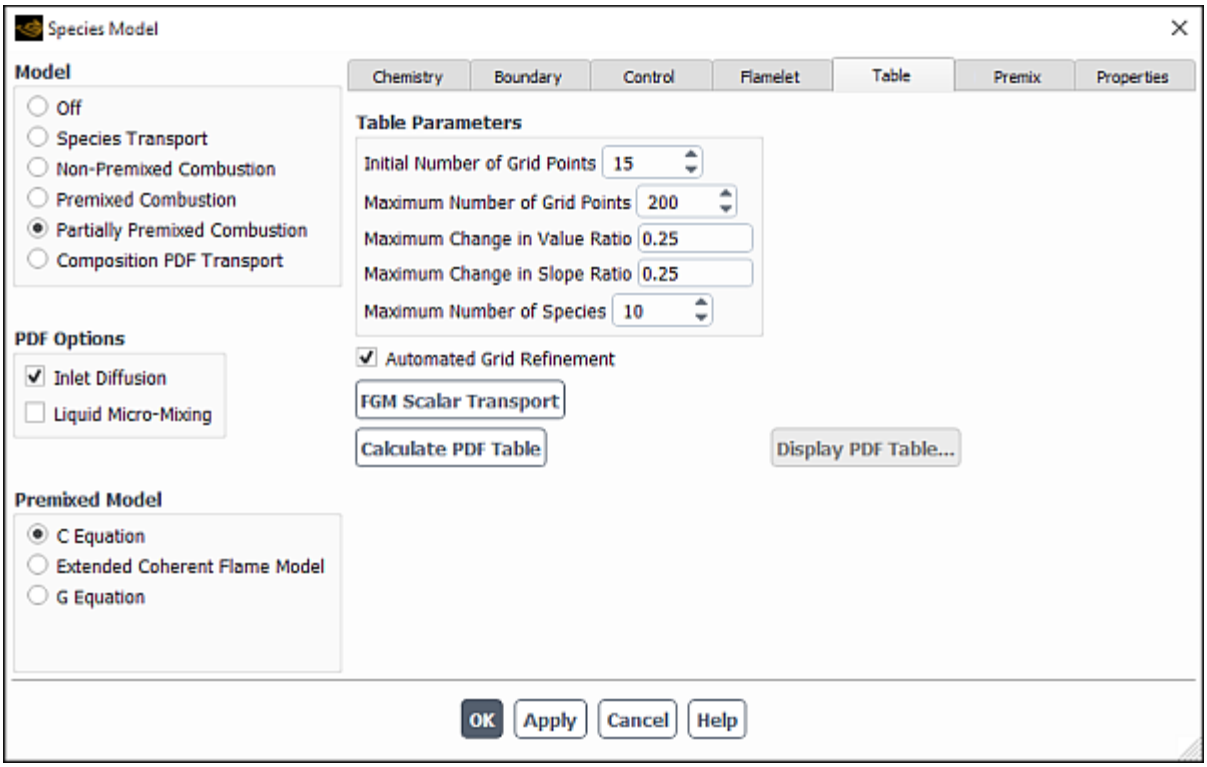
  Note that for the mean mixture fraction, you can enable the **Use Flamelet Distribution** option if you want to use the points of the grid distribution of the FGM flamelet.

- Mean fraction variance

For the mean progress variable, you can view the grid distribution points, but you cannot edit them.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 20.42: The Species Model Dialog Box: Table Tab Displaying Automated Grid Refinement



The look-up table parameters when **Automated Grid Refinement** is selected are as follows:

Initial Number of Grid Points

specifies the number of grid points for the resolution of the mean mixture fraction, mixture fraction variance, and mean enthalpy (for non-adiabatic systems); and mean progress variable and progress variable variance (for FGM).

Maximum Number of Grid Points

specifies the maximum number of grid points used for tabulation. The grid refinement procedure will stop inserting the points when either the change in value and slope between successive points is within tolerance or the maximum number of grid points are generated.

Maximum Change in Value Ratio

specifies the maximum allowable change in value of table variables between successive grid points as specified by Equation 8.28 in the Theory Guide.

Maximum Change in Slope Ratio

specifies the maximum change in the slope of table variables between successive grid points as specified by Equation 8.29 in the Theory Guide.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Maximum Number of Species

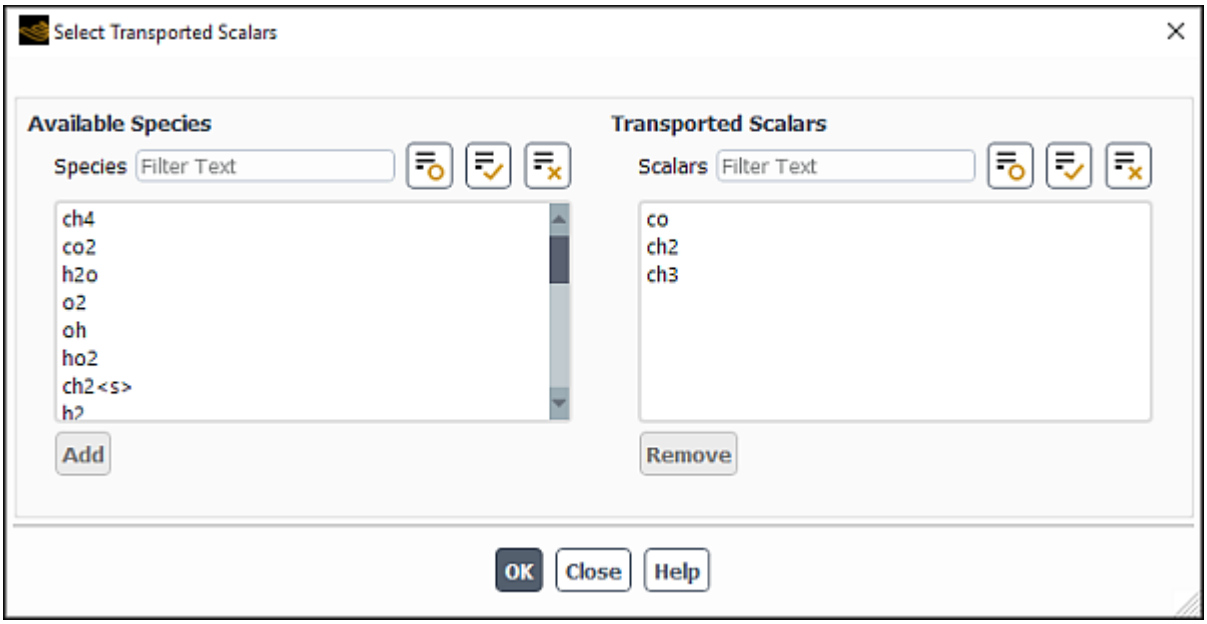
is the maximum number of species stored in the lookup tables.

Note:

Automated Grid Refinement is not available with two mixture fractions.

For the FGM model, you can solve additional transport equations for selected species mass fractions, called FGM scalars, as described in Scalar Transport with FGM Closure in the *Fluent Theory Guide*. To do so, click **FGM Scalar Transport**.

Figure 20.43: The Select Transported Scalars Dialog Box



In the **Select Transported Scalars** dialog box that opens, select the FGM scalars as follows:

- To add a species to the **Transported Scalars** list, select it in the **Available Species** multiple-selection list and click **Add**.
- To remove a species from the **Transported Scalars** list back to the **Available Species** multiple-selection list, select it and click **Remove**.

For flow boundaries, you can specify the boundary conditions for the **Scalar Mass Fractions** for the selected transported scalars (in the **Species** tab of the corresponding boundary conditions dialog boxes).

During the solution run, Ansys Fluent will solve transport equations for the selected scalars and take their pre-computed averaged reaction rates from the PDF look-up table.

Note:

- You must select the FGM transported scalars before calculating the PDF table. Once the table is calculated, you will no longer be able to modify the definition of the

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

transported FGM scalars. In order to modify the FGM scalar definition after the PDF table has been calculated, you will need to re-import or regenerate the flamelets.

- Since the scalar equations do not couple back into the flow equations or mixture properties, they could be solved in a postprocessing mode for a steady-state simulation, such as pollutant formation. In such cases, you could deselect **FGM Scalar Transport** from the **Equations** list solved by Ansys Fluent (in the **Equations** dialog box), and then select them again once the steady flow field has converged.

When you are satisfied with your inputs, click **Calculate PDF Table** to generate the look-up tables.

20.3.2.4.1. Postprocessing the Look-Up Tables with Flamelet Generated Manifolds

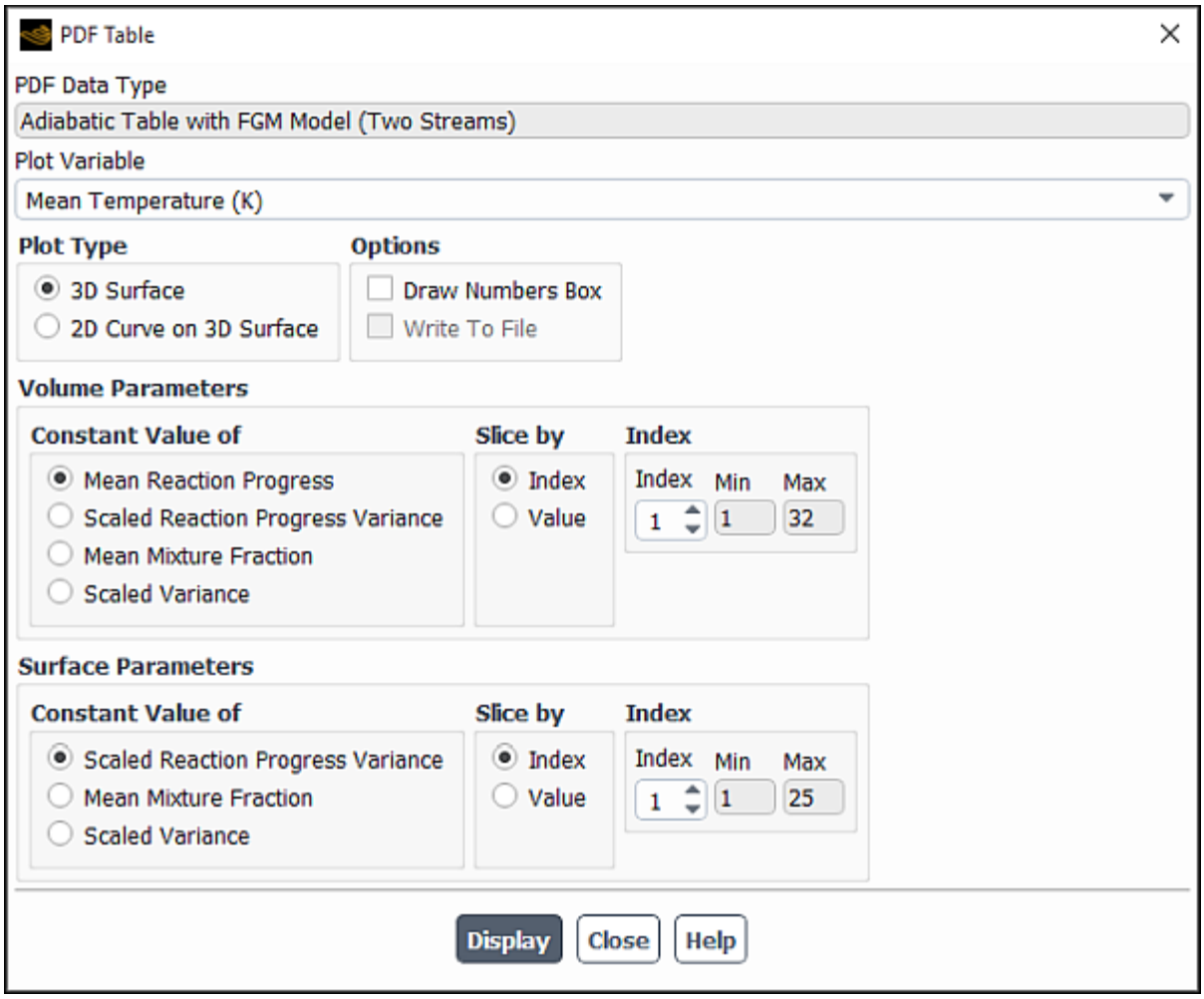
For the FGM model, the PDF table is four-dimensional for adiabatic cases and five-dimensional for non-adiabatic cases. For adiabatic PDF tables, the four independent variables are Mean mixture fraction, Mean reaction progress variable, Mixture fraction variance, and Progress variable variance. For non-adiabatic tables, the additional fifth independent variable is enthalpy. The FGM PDF tables can be post-processed using 2D curves and 3D surfaces by fixing two or more independent variables.

The procedure to display the lookup tables graphically is similar to that outlined in Postprocessing the Look-Up Table Data (p. 2515) with additional steps as follows.

- Adiabatic PDF Tables

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 20.44: The PDF Table Dialog Box (Adiabatic Case With FGM)



– 3D surfaces:

1. Under **Volume Parameters**, select the discrete independent parameter to be held constant and fix it by specifying either a slice **Index** number or a **Value**. (For more information, see Postprocessing the Look-Up Table Data (p. 2515).)

2. In a similar manner, under **Surface Parameters**, select and fix the second discrete independent parameter and click **Plot**.

   The resulting plot is a 3D surface of the selected **Plot Variable** as a function of the remaining two independent variables that have not been fixed.

– 2D curves on 3D Surface:

1. Under **Volume Parameters** and **Surface Parameters**, fix discrete independent parameters as described above.

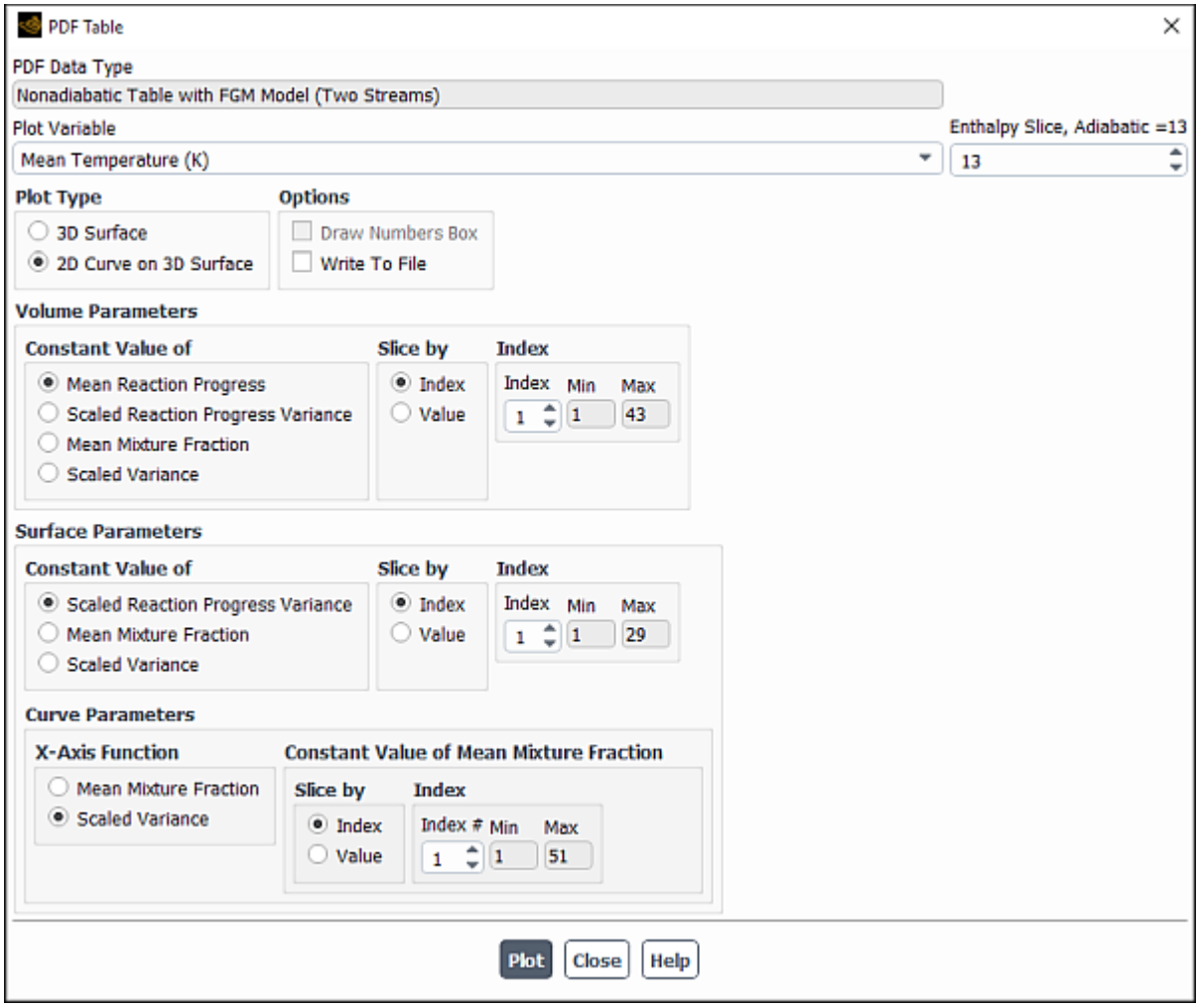
2. Under the **Curve Parameters** group box, select the **X-Axis Function** against which the plot variable will be displayed, fix the remaining discrete independent parameter using controls in the **Constant Value of *`Independent Parameter`*** group box, and click **Plot**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The resulting plot is a 2D curve of the selected **Plot Variable** as a function of the remaining independent variable that has not been fixed.

- Non-Adiabatic PDF Tables

Figure 20.45: The PDF Table Dialog Box (Non-Adiabatic Case With FGM)



1. Fix the enthalpy slice level by specifying **Enthalpy Slice**.

   For non-adiabatic PDF tables, heat loss/gain is the fifth independent variable, which is expressed as enthalpy levels. By default, the enthalpy level is fixed at the adiabatic level, which is displayed in the **Enthalpy Slice, Adiabatic =** ***slice number*** label, as shown in Figure 20.45: The PDF Table Dialog Box (Non-Adiabatic Case With FGM) (p. 2560). Enthalpy slice numbers below the adiabatic slice number are associated with heat loss, while enthalpy slice numbers above the adiabatic slice number are associated with enthalpy gain.

2. Depending on whether you want to display a 3D surface or a 2D Curve on 3D Surface, follow appropriate steps for Adiabatic PDF Tables described above.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

20.3.2.5. Standard Files for Flamelet Generated Manifold Modeling

A standard flamelet file format can be used to read and write FGM flamelets. The data structure for the standard flamelet file format is based on keywords that precede each data section. If any of the keywords in your flamelet data file do not match the supported keywords, you will have to manually edit the file and change the keywords to one of the supported types. (The Ansys Fluent flamelet filter is case-insensitive, so you need not worry about capitalization within the keywords.)

The following keywords are supported by the Ansys Fluent filter:

- Header section: `HEADER`
- Main body section: `BODY`
- Number of species: `NUMOFSPECIES`
- Number of grid points: `GRIDPOINTS`
- Pressure: `PRESSURE`
- Temperature: `TEMPERATURE` and `TEMP`
- Mass fraction: `MASSFRACTION-`
- Mixture fraction: `Z`
- Mole fraction: `MOLEFRACTION-`
- Scalar dissipation for premixed flamelet: `PREMIX_CHI`
- Reaction progress: `REACTION_PROGRESS`
- Reaction progress source term: `PREMIX_YCDOT`

Note:

Note that the FGM standard flamelet file format is identical for both premixed and diffusion flamelets. However, this format is different from the standard format for steady diffusion Laminar Flamelets, which is described in Calculating the Look-Up Tables (p. 2509).

20.3.2.5.1. Sample Standard FGM File

A sample FGM file in the standard FGM format is provided below. Note that not all species are listed in this file.

```
HEADER
PREMIX_STOICH_SCADIS   1.000000E+01
Z         0.000000E+00
NUMOFSPECIES  53
GRIDPOINTS  32
STOICH_Z    5.520020e-02
PRESSURE  1.013250E+05
BODY
REACTION_PROGRESS
0.000000000E+00   5.625000000E-02   1.125000000E-01   1.687500000E-01   2.250000000E-01
2.812500000E-01   3.375000000E-01   3.937500000E-01   4.500000000E-01   5.062500000E-01
```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

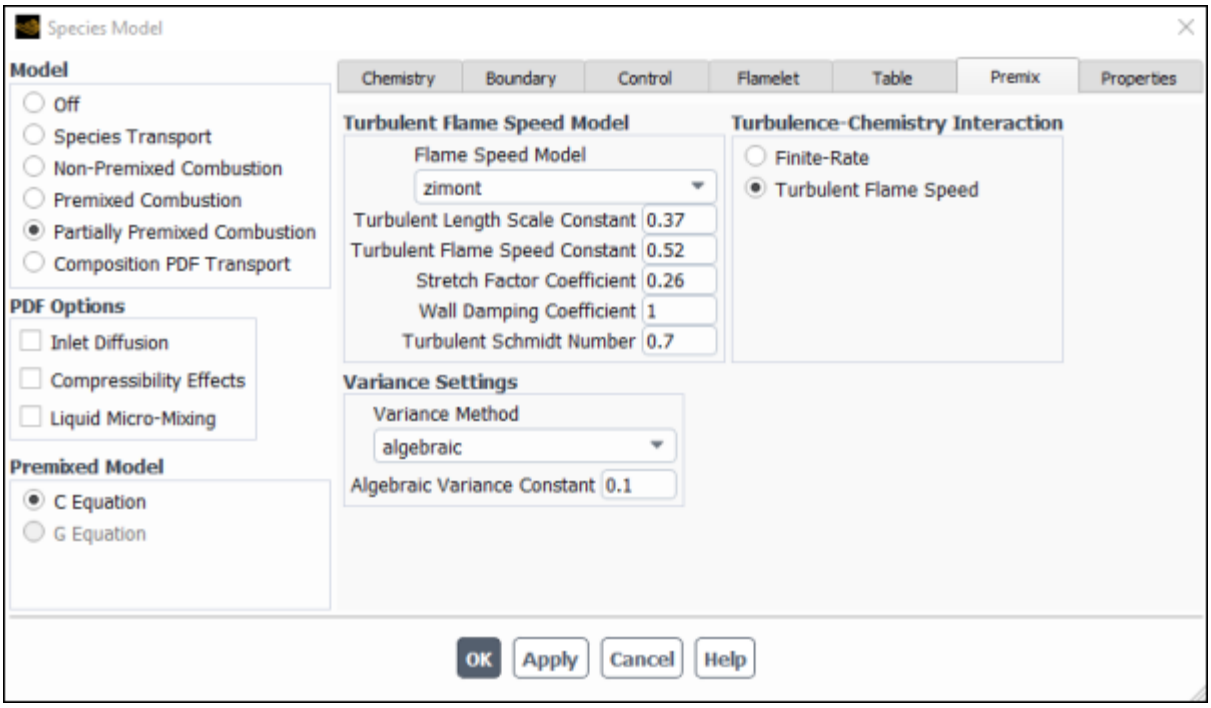
```
5.625000000E-01   6.187500000E-01   6.750000000E-01   7.312500000E-01   7.875000000E-01
8.437500000E-01   9.000000000E-01   9.508132748E-01   9.604275079E-01   9.668740305E-01
9.718608224E-01   9.759835686E-01   9.795270729E-01   9.826516818E-01   9.854574013E-01
9.880111737E-01   9.903602004E-01   9.925391181E-01   9.945741609E-01   9.964857158E-01
9.982899681E-01   1.000000000E+00
TEMPERATURE
3.000000000E+02   3.000000063E+02   3.000000126E+02   3.000000189E+02   3.000000251E+02
3.000000314E+02   3.000000377E+02   3.000000440E+02   3.000000503E+02   3.000000566E+02
3.000000628E+02   3.000000691E+02   3.000000754E+02   3.000000817E+02   3.000000880E+02
3.000000943E+02   3.000001006E+02   3.000001062E+02   3.000001073E+02   3.000001080E+02
3.000001086E+02   3.000001090E+02   3.000001094E+02   3.000001098E+02   3.000001101E+02
3.000001104E+02   3.000001107E+02   3.000001109E+02   3.000001111E+02   3.000001113E+02
3.000001115E+02   3.000001117E+02
massfraction-h2
0.000000000E+00   1.095241207E-53   2.190482414E-53   3.285723620E-53   4.380964827E-53
5.476206034E-53   6.571447241E-53   7.666688448E-53   8.761929655E-53   9.857170861E-53
1.095241207E-52   1.204765328E-52   1.314289448E-52   1.423813569E-52   1.533337690E-52
1.642861810E-52   1.752385931E-52   1.851324229E-52   1.870044058E-52   1.882596053E-52
1.892305813E-52   1.900333194E-52   1.907232735E-52   1.913316647E-52   1.918779651E-52
1.923752089E-52   1.928325868E-52   1.932568429E-52   1.936530852E-52   1.940252832E-52
1.943765883E-52   1.947095479E-52
 .  .   .    .    .
 .  .   .    .    .
PREMIX_YCDOT
0.000000000E+00   0.000000000E+00   0.000000000E+00   0.000000000E+00   0.000000000E+00
0.000000000E+00   0.000000000E+00   0.000000000E+00   0.000000000E+00   0.000000000E+00
0.000000000E+00   0.000000000E+00   0.000000000E+00   0.000000000E+00   0.000000000E+00
0.000000000E+00   0.000000000E+00   0.000000000E+00   0.000000000E+00   0.000000000E+00
0.000000000E+00   0.000000000E+00   0.000000000E+00   0.000000000E+00   0.000000000E+00
0.000000000E+00   0.000000000E+00   0.000000000E+00   0.000000000E+00   0.000000000E+00
0.000000000E+00   0.000000000E+00
```

20.3.2.6. Setting Premix Flame Propagation Parameters

Once the PDF look-up table is generated, you can specify the turbulent flame speed model options in the **Premixed** tab of the **Species Model** dialog box. For the FGM model, you can also specify the variance settings and select the turbulence-chemistry interaction model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 20.46: The Species Model Dialog Box(Premix Tab)



The following controls are available in the **Turbulent Flamelet Speed Model** group box.

Flame speed Model

allows you to select the method to calculate turbulent flame speed as described in Turbulent Flame Speed Models in the *Fluent Theory Guide*.

If you selected **zimont**, you can adjust the Zimont model parameters as described in Modifying the Constants for the Zimont Flame Speed Model (p. 2539).

When the SBES turbulence model is used with the C Equation premixed model, you can also specify the following additional inputs:

Turbulent Length Scale Constant (RANS)

is the modeling constant in the RANS region (C_D in Equation 8.79 in the Theory Guide).

Turbulent Flame Speed Constant (RANS)

is the modeling constant in the RANS region (A in Equation 8.77 in the Theory Guide).

If you selected **peters**, you can adjust the Peters model parameters as described in Modifying the Constants for the Peters Flame Speed Model (p. 2540).

For the FGM model, you can adjust the following additional parameters:

Turbulence-Chemistry Interaction

allows you to choose the method for modeling the source term in the progress variable equation (Equation 8.113 in the *Fluent Theory Guide*). The following methods are available:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Finite-Rate**: Uses Equation 8.114 in the *Fluent Theory Guide* to calculate progress variable source terms from the PDF table.
- **Turbulent Flame Speed**: Uses Equation 8.115 in the *Fluent Theory Guide* to calculate progress variable reaction source terms.

For more information about these models, see FGM Turbulent Closure in the Theory Guide.

Variance Settings

contains the **Variance Method** drop-down list that allows you to choose between the following methods for calculating the progress variable variance:

- **transport equation** (Equation 8.116 in the *Fluent Theory Guide*)
- **algebraic** (Equation 8.117 in the *Fluent Theory Guide*)
- (SBES only) **hybrid** (Equation 8.118 in the *Fluent Theory Guide*)

For the **hybrid** option, when applying Equation 8.118 in the *Fluent Theory Guide*, the algebraic model is used for $\widetilde{Y}''_{c,LES}$ and the transport equation model is used for $\widetilde{Y}''_{c,RANS}$.

For the **algebraic** and **hybrid** options, you need to specify **Algebraic Variance Constant** (c_{var} in Equation 8.117 in the *Fluent Theory Guide*).

It is recommended that you use the **transport equation** option for RANS, the **algebraic** option for LES, and the **hybrid** option for SBES.

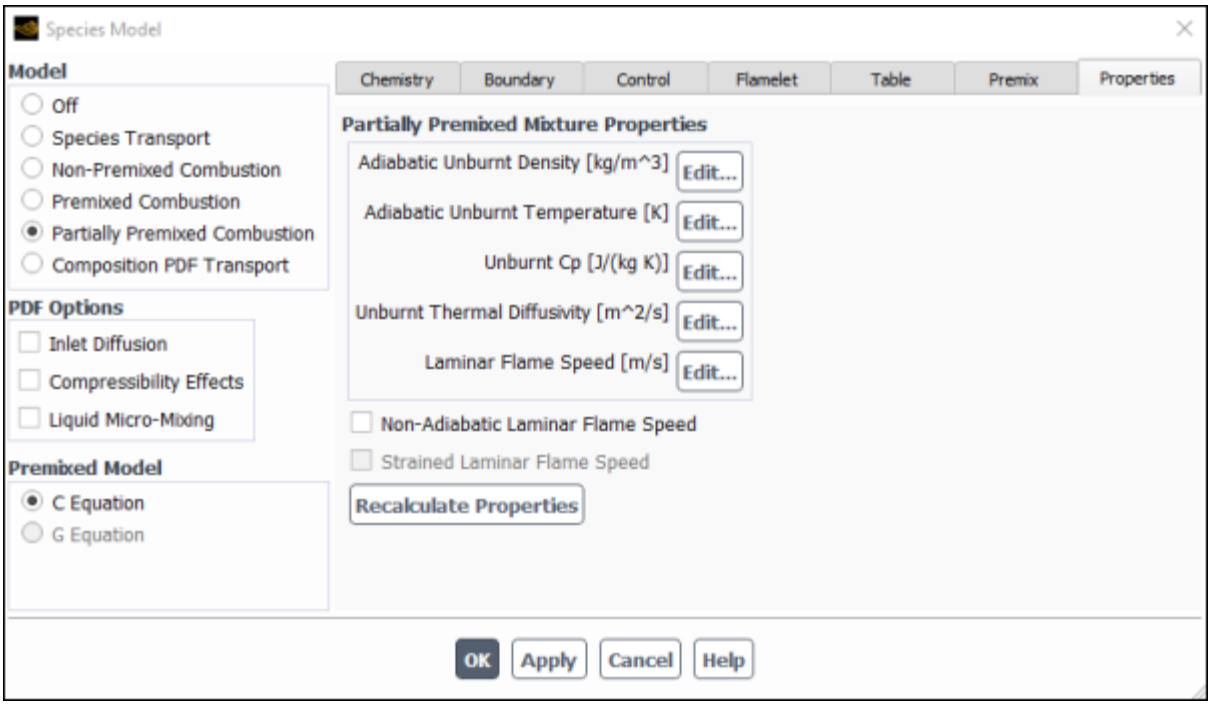
20.3.2.7. Modifying the Unburnt Mixture Property Polynomials

After building the PDF table, Ansys Fluent automatically calculates the temperature, density, heat capacity, and thermal diffusivity of the unburnt mixture as polynomial functions of the mean mixture fraction, $\overline{f}$ (see Equation 8.127 in the *Fluent Theory Guide*). The laminar flame speed is automatically calculated as a piecewise-linear polynomial function of $\overline{f}$.

However, as outlined in Partially Premixed Combustion Theory in the *Fluent Theory Guide*, the laminar flame speed depends on details of the chemical kinetics and molecular transport properties, and in the prepdf-polynomial method, it is not calculated directly. Instead, curve fits are made to flame speeds determined from detailed simulations [55] (p. 5716). These fits are limited to a range of fuels (H_2, CH_4, C_2H_2, C_2H_4, C_2H_6, and C_3H_8), air as the oxidizer, equivalence ratios of the lean limit through unity, unburnt temperatures from 298 K to 800 K, and pressures from 1 bar to 40 bars. If your parameters fall outside this range, Ansys Fluent will warn you when it computes the look-up table. In this case, you will need to modify the piecewise-linear points in the **Properties** tab of the **Species Model** dialog box (Figure 20.47: The Species Model Dialog Box (Properties Tab) (p. 2565)) before you save the PDF file. If you selected the **laminar-flame-speed-computed** or **laminar-flame-speed-library** method, Ansys Fluent will automatically calculate the polynomial coefficient from computed flame speed or library. In both methods, the polynomial coefficient values of the laminar flame speed cannot be modified.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 20.47: The Species Model Dialog Box (Properties Tab)

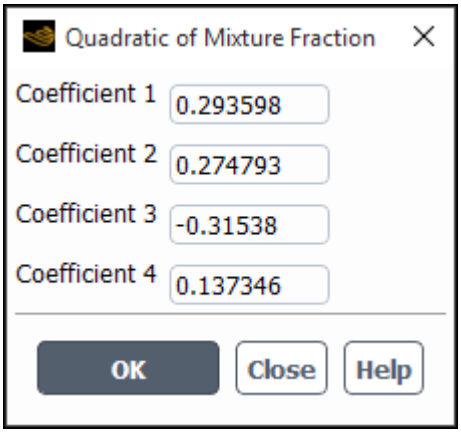


For each of the following polynomial functions of the mean mixture fraction $\overline{f}$ in the **Partially Premixed Mixture Properties** group box:

- **Adiabatic Unburnt Density**
- **Adiabatic Unburnt Temperature**
- **Unburnt Cp**
- **Unburnt Thermal Diffusivity**

you can specify values for **Coefficient 1**, **Coefficient 2**, **Coefficient 3**, and **Coefficient 4** (the polynomial coefficients in Equation 8.127 in the *Fluent Theory Guide*) in the **Quadratic of Mixture Fraction** dialog box, which opens when you click the **Edit...** button to the right of each property name. See Figure 20.48: The Quadratic of Mixture Fraction Dialog Box (p. 2565).

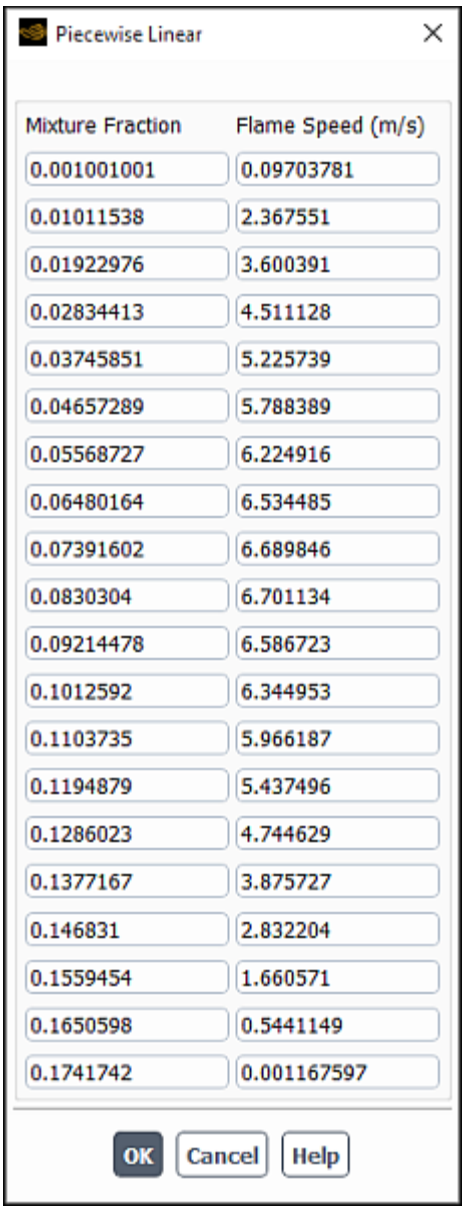
Figure 20.48: The Quadratic of Mixture Fraction Dialog Box



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you selected the **prepdf-polynomial** method for **Laminar Flame Speed**, you can also specify the piecewise-linear **Mixture Fraction** and its corresponding **Flame Speed** for 20 different points in the **Piecewise Linear** dialog box (Figure 20.49: The Piecewise Linear Dialog Box (p. 2566)). The first set of values is the lower limit and the last set of values is the upper limit. Outside of either limit, the laminar flame speed is constant and equal to that limit. If you selected the laminar-flame-speed-library method, you can display the table, but you cannot modify the table values. To open the **Piecewise Linear** dialog box, click the **Edit...** button next to **Laminar Flame Speed** in the **Properties** tab.

Figure 20.49: The Piecewise Linear Dialog Box



Important:

Note also that if you choose to use a user-defined function for the laminar flame speed in the Create/Edit Materials Dialog Box (p. 4885), the piecewise-linear fit becomes irrelevant.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the secondary mixture fraction model is enabled, the unburnt properties are a function of both the mean and secondary mixture fractions. An additional column is added in the **Piecewise Linear** dialog box for the secondary mixture fraction unburnt polynomial coefficients.

Note that the **Non-Adiabatic Laminar Flame Speed** option when enabled includes the non-adiabatic effects on the laminar flame speed by tabulating the laminar speeds in the PDF table. See Laminar Flame Speed in the *Fluent Theory Guide*.

20.3.2.8. Modeling Strained Laminar Flame Speed

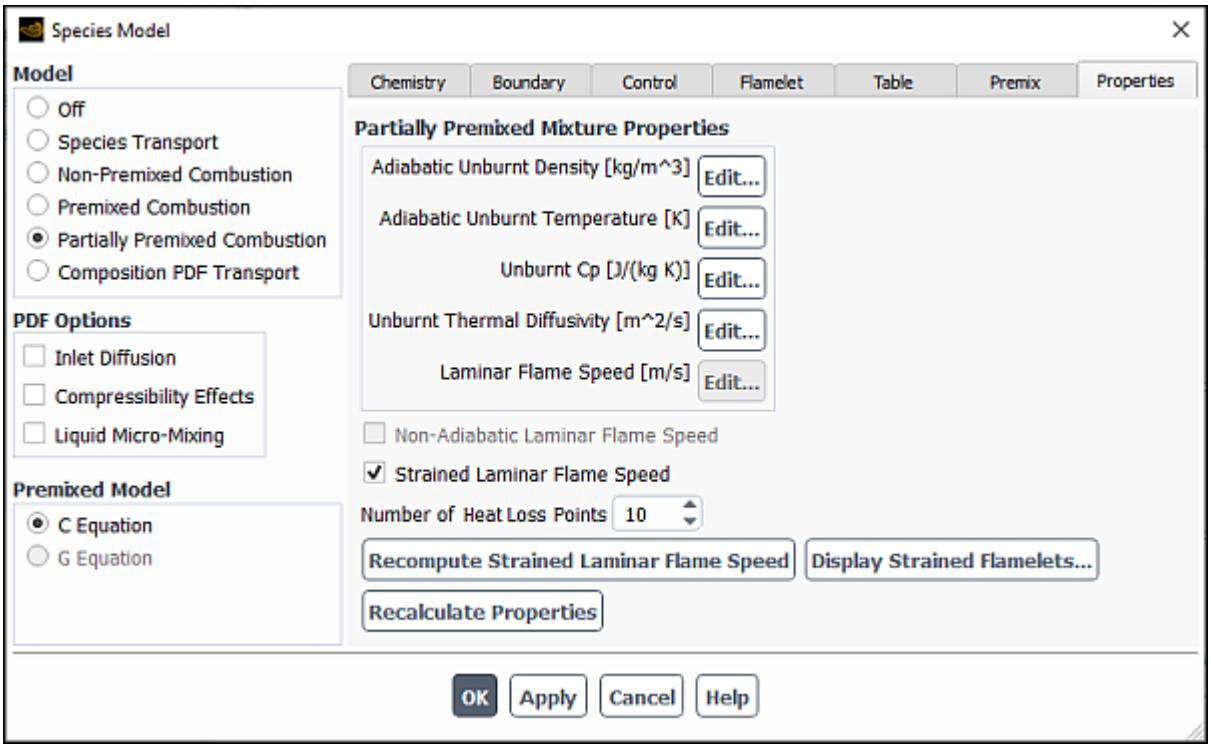
For the Flamelet Generated Manifold (FGM) model with the turbulent flame speed model, you can model the strained laminar flame speed as follows:

1. Make sure **Flamelet Generated Manifold** is selected as the **State Relation** (**Chemistry** tab).

2. After calculating the flamelets and the PDF table (**Flamelet** and **Table** tabs), select **Turbulent Flame Speed** in the **Turbulence-Chemistry Interaction** group box and specify all relevant parameters (**Premix** tab).

3. In the **Properties** tab, enable **Strained Laminar Flame Speed**. This option appears only if you have imported a transport data file along with CHEMKIN mechanism and thermodynamic data files.

Figure 20.50: The Species Model Dialog Box: Strained Laminar Flame Speed



4. Click the **Compute Strained Laminar Flame Speed Table** button to generate strained flamelets for different mixture fraction points and strain rates.

5. If you are modeling non-adiabatic strained flame speed, then set the **Number of Heat Loss Points** to a value other than 1.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

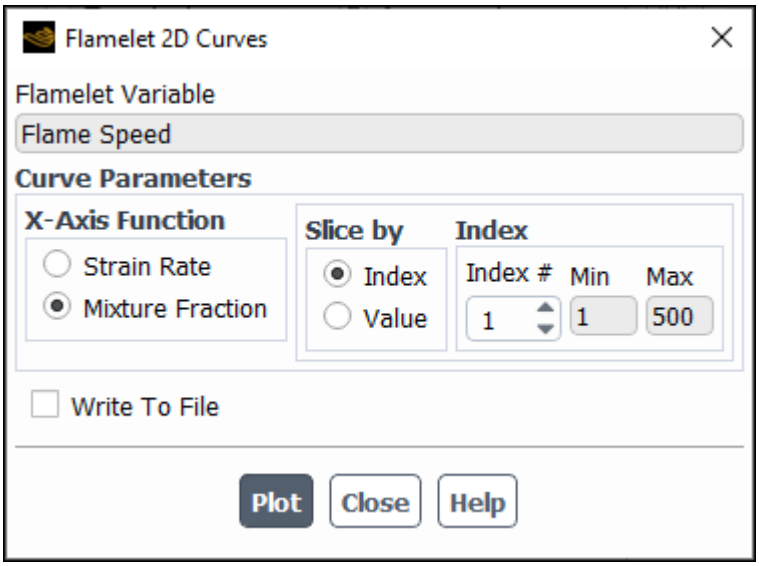
The heat loss points represent the different values of the heat loss parameter (Equation 8.139 in the *Fluent Theory Guide*). The default value of 1 corresponds to adiabatic strained flame speed. If you use more than one heat loss point in your simulation, the strained flame speed will be computed in increments of 0.05 of the heat loss parameter. For example, for three heat loss points, the strained non-adiabatic flame speed is computed for the heat loss parameters of 0, 0.05, and 0.1. For most applications, less than 10 heat loss points is sufficient.

See Strained Non-Adiabatic Flame Speed in the *Fluent Theory Guide* for more information.

6. Once the flamelets are generated, you can display the flame speed as a function of the mixture fraction or strain rate using the **Display Strained Flamelets...** button.

   In the **Flamelet 2D Curves** dialog box that opens, **Flame Speed** appears as **Flamelet Variable**.

   Figure 20.51: The Flamelet 2D Curves Dialog Box



   a. Specify the **X-Axis Function** against which the plot variable will be displayed by selecting **Strain Rate** or **Mixture Fraction**. The variable that is not selected will be held constant.

   b. Specify the type of discretization (that is, how the flamelet data will be sliced) for the variable that is being held constant.

      - If you selected **Index** under **Slice by**, specify the discretization **Index** of the variable that is being held constant. The range of integer values that you are allowed to choose from is displayed under **Min** and **Max**, and is equivalent to the number of points used in postprocessing the flamelet data.

      - If you selected **Value** under **Slice by**, specify the numerical **Value** of the variable that is being held constant. The range of values that you can specify is displayed under **Min** and **Max**.

   c. Click **Plot**.

   Note that you can write flamelet table results to a file by using the **Write to File** option.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For more information about this model, refer to Strained Laminar Flame Speed in the *Fluent Theory Guide*.

20.3.2.9. Modeling In Cylinder Combustion

Each of the partially premixed combustion models may be used for modeling in cylinder combustion. When modeling more than one cycle of such engines, you may choose to model trapped combustion products and exhaust gas recirculation (EGR) using the inert species model (see Setting Up the Inert Model (p. 2521)). A **Dynamic Mesh Event** has also been provided for calculating the inert composition, converting burnt gases to inert and resetting the combustion process ready for the next cycle (see Resetting Inert EGR (p. 1903) and Resetting Inert EGR (p. 2525)).

In the case of the G-Equation model, this is the only way to automatically model multiple in cylinder combustion cycles and take into account trapped burnt gases remaining in the cylinder from one cycle to the next.

20.3.2.10. Postprocessing for FGM Scalar Transport Calculations

Ansys Fluent provides additional reporting options when you use the **FGM Scalar Transport** option for solving transport equations for the FGM scalars selected in the **Select Transported Scalars** dialog box. Quantities for reporting scalar mass fractions of the FGM scalars are contained in the **Premixed Combustion…** category the postprocessing dialog boxes.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 21: Modeling Engine Ignition

This chapter discusses how to use the engine ignition models available in Ansys Fluent in the following sections. For information about the theory behind these ignition models, see Engine Ignition in the Theory Guide.

21.1. Using the Spark Model

21.2. Using the Autoignition Models

21.3. Using the Crevice Model

21.1. Using the Spark Model

The spark model in Ansys Fluent will be described in the context of the premixed and partially premixed combustion models, including the species transport model. For information regarding the theory of this model, see Spark Model in the Theory Guide.

You can model a single spark or multiple sparks. To use the spark model, perform the following steps:

1. Select **Transient** from the **Time** list in the **General** task page (or from the **General** → **Analysis Type** in the tree).

2. In the **Species Model** dialog box, select one of the following reaction models:

   Setup → **Models** → **Species** **Edit...**

   - Under **Model**, select **Species Transport**, and under **Reactions**, enable **Volumetric**.

   - Under **Model**, select **Premixed Model** with the **C Equation** or **G Equation** model enabled. See Setting Up the C-Equation and G-Equation Models (p. 2538) for more information.

   The **Spark Ignition** model will now appear in the **Models** task page and under the **Models** tree item.

   Important:

   When you read in a R14.5 case file, Ansys Fluent will revert to the R14.5 spark model by default. To switch to the current spark model, you can either

   - open the **Spark Ignition** dialog box, or

   - use the TUI `define/models/species/spark-model` command.

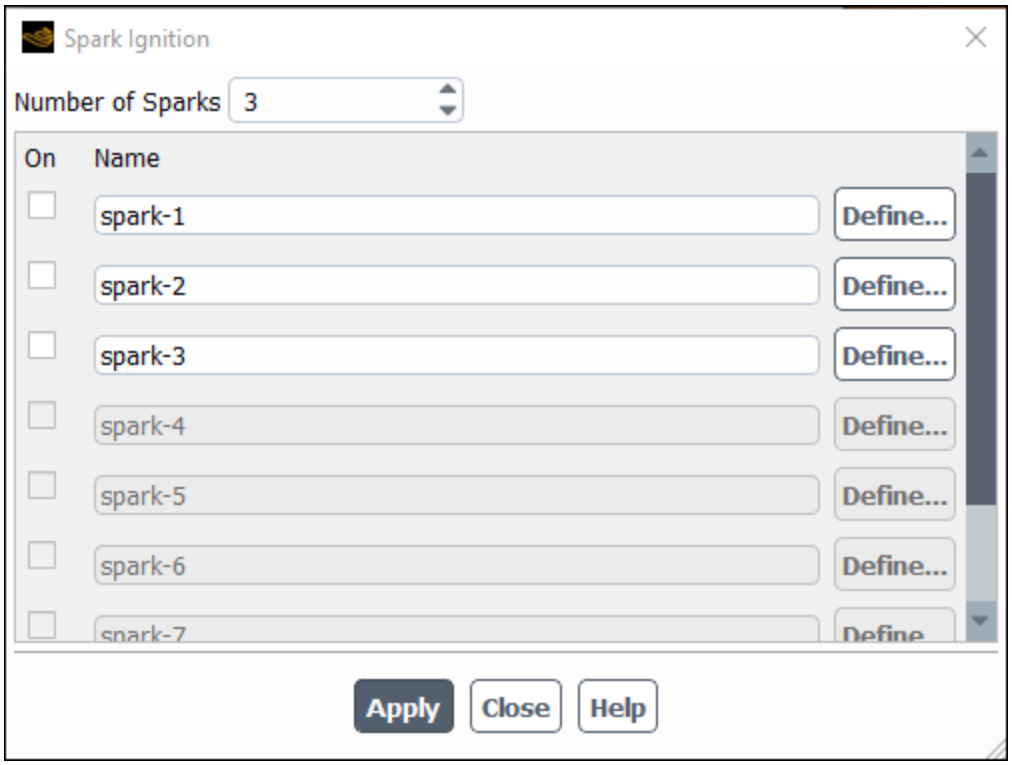
   You can also use the TUI `define/models/species/spark-model` command to revert back to the R14.5 spark model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Define sparks for your case using the **Spark Ignition** dialog box (see Figure 21.1: The Spark Ignition Dialog Box (p. 2572)).

   Setup → Models → Species → Spark Ignition Edit...

   Figure 21.1: The Spark Ignition Dialog Box



4. Specify the **Number of Sparks** you would like to include in your simulation. You can define up to 16 sparks.

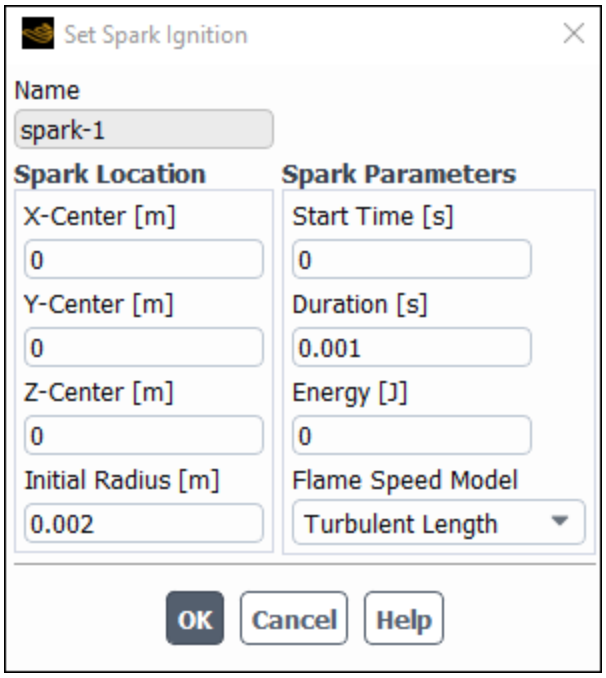
5. While you can define several sparks, you can choose which ones to turn on using the **On** option.

6. Enter the **Name** of the spark, or simply keep the default name.

7. Click the **Define...** button to open the **Set Spark Ignition** dialog box (Figure 21.2: The Set Spark Ignition Dialog Box (p. 2573)), where you will set the parameters of the selected spark ignition models.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 21.2: The Set Spark Ignition Dialog Box



8. Set the spark model parameters.

   - Set X, Y, and Z coordinates of the spark center and the **Initial Radius** of the spark kernel in the **Spark Location** group box.
   - Enter **Start Time** and **Duration** in the **Spark Parameters** group box.

     Important:

     When the in-cylinder model is turned on, the **Start Time** is entered in crank angle degrees instead of seconds, while the spark **Duration** is still in seconds. The value you enter for **Start Time** is for one complete engine cycle. In subsequent cycles, the spark is enabled according to Equation 13.30 (p. 1897), where θ_{event} corresponds to the spark start crank angle.

   - (optional) Set **Energy** to a positive value if you want to model the higher temperature levels within the spark kernel. The energy input will be evenly distributed across the specified spark duration. This energy is not required to initiate combustion. The spark model will control the spark kernel growth and combustion progress. For this reason the default energy input is zero. If you specify extra energy here, it will raise the kernel temperature beyond that given by the combustion process.
   - Select the **Flame Speed Model** from the drop-down list. For details of these models see Spark Model Theory in the *Fluent Theory Guide*.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

21.2. Using the Autoignition Models

Autoignition phenomena in engines are due to the effects of chemical kinetics of the reacting flow inside the cylinder. There are two types of autoignition models considered in Ansys Fluent:

- knock model in spark-ignited (SI) engines
- ignition delay model in diesel engines

For information regarding the theory behind autoignition models, see Autoignition Models in the Theory Guide.

To use the autoignition model, perform the following steps:

1. Select **Transient** from the **Time** list in the **General** task page (or from the **General** → **Analysis Type** in the tree).
2. Select an appropriate reaction model in the **Species Model** dialog box.

   Setup → **Models** → **Species** **Edit...**
3. The models in the **Species Model** dialog box that are compatible with the autoignition model are **Species Transport**, **Premixed Combustion**, and **Partially Premixed Combustion**.

   The **Autoignition** model will now appear in the **Models** task page and under the **Models** tree item.

   Important:

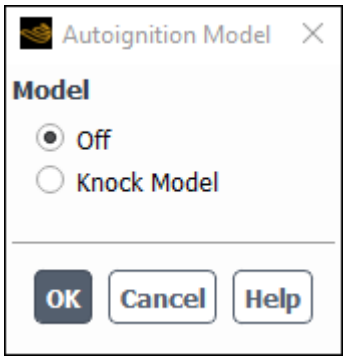
   - If you select **Species Transport**, you must also enable the **Volumetric** option in the **Reactions** group box.
   - The **Premixed Combustion** and **Partially Premixed Combustion** models are only available for turbulent flows using the pressure-based solver.
4. Select the Autoignition model.

   Setup → **Models** → **Species** → **Autoignition** **Edit...**

   - If **Species Transport** is selected in the **Species Model** dialog box, you can only select the **Ignition Delay Model**.

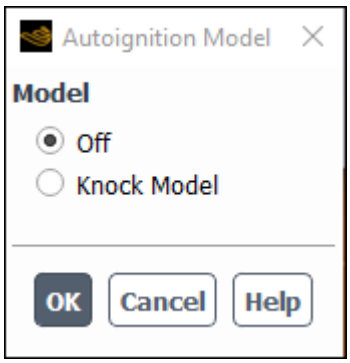
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 21.3: The Ignition Delay Model in the Autoignition Model Dialog Box



- If **Premixed Combustion** is selected in the **Species Model** dialog box, you can only select the **Knock Model**.

Figure 21.4: The Knock Model in the Autoignition Model Dialog Box

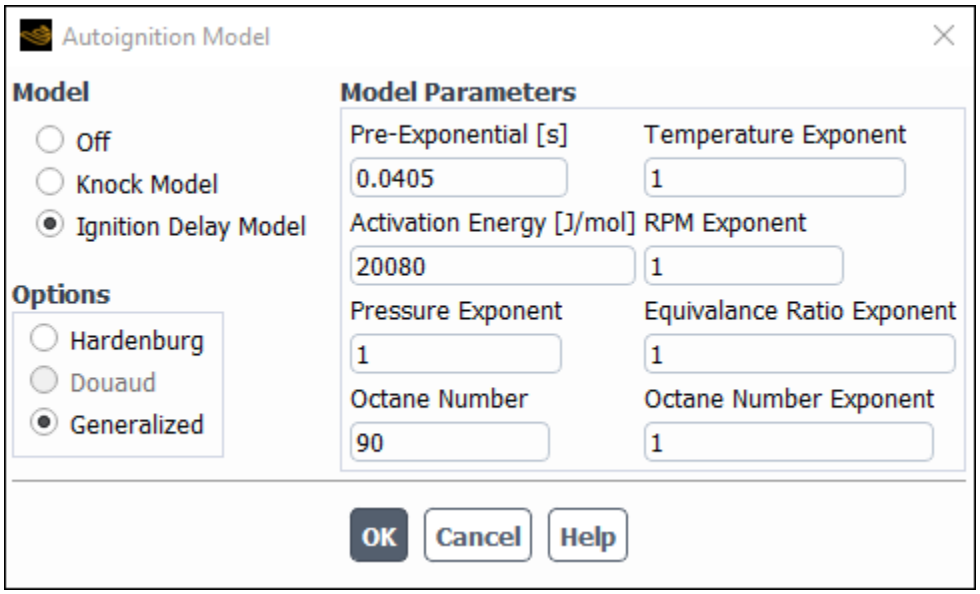


- If **Partially Premixed Combustion** is selected in the **Species Model** dialog box, you can select either the **Knock Model** or the **Ignition Delay Model**.

5. When the **Ignition Delay Model** is enabled, the dialog box expands to include the modeling parameters for this model (Figure 21.5: The Ignition Delay Model for the Partially Premixed Combustion Model (p. 2576)). The two correlation options that exist with this model are **Hardenburg** and **Generalized**. Depending on which correlation option is selected, the appropriate modeling parameters will appear in the dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 21.5: The Ignition Delay Model for the Partially Premixed Combustion Model



- The **Hardenburg** option is typically used for heavy duty diesel engines. With this option, the following parameters are available:
  - **Pre-Exponential**
  - **Pressure Exponent**
  - **Activation Energy**
  - **Cetane Number**

  For the **Species Transport** model, **Fuel Species** is selected from the drop-down list.
- The **Generalized** option is described by Equation 10.9 in the Theory Guide. With this option, the following parameters are available:
  - **Pre-Exponential**
  - **Temperature Exponent**
  - **Activation Energy**
  - **RPM Exponent**
  - **Pressure Exponent**
  - **Equivalence Ratio Exponent**
  - **Octane Number**
  - **Octane Number Exponent**

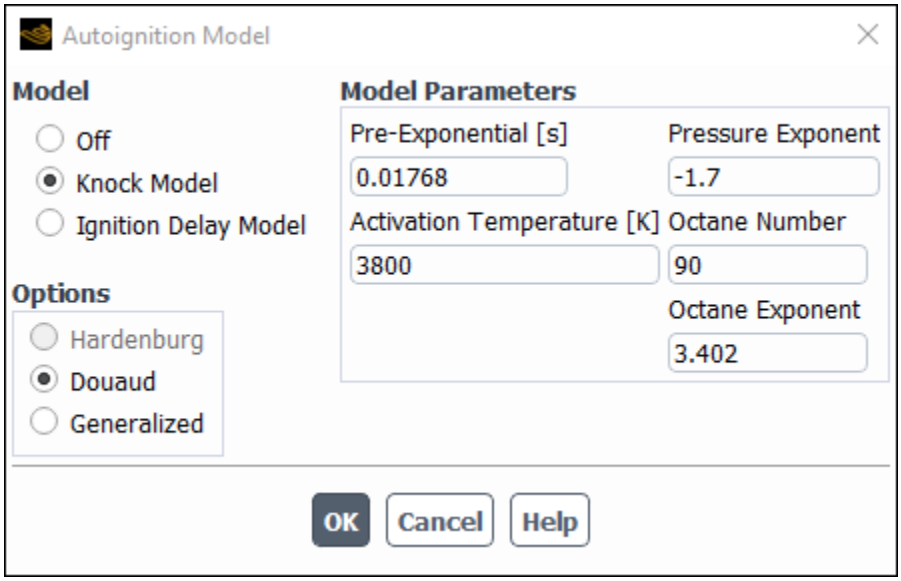
  For the **Species Transport** model, **Fuel Species** is selected from the drop-down list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Default values of these parameters can be found in Default Values of the Variables in the Hardenburg Correlation in the *Fluent Theory Guide*.

6. When the **Knock Model** is enabled, the dialog box expands to include modeling parameters for this model (Figure 21.6: The Knock Model with the Partially Premixed Combustion Model Enabled (p. 2577)). The two correlation options that exist with this model are **Douaud** and **Generalized**. Depending on which correlation option is selected, the appropriate modeling parameters will appear in the dialog box.

Figure 21.6: The Knock Model with the Partially Premixed Combustion Model Enabled



- The **Douaud** option is used for knock in SI engines. The modeling parameters that are specified in the **Autoignition Model** dialog box for this option are the **Pre-Exponential**, **Pressure Exponent**, **Activation Temperature**, **Octane Number**, and **Octane Exponent** (Equation 10.8 in the Theory Guide).

- The **Generalized** option (Equation 10.9 in the Theory Guide) in the knock model requires the same parameters as in the ignition delay model.

21.3. Using the Crevice Model

For information regarding the theory behind the crevice model, see Crevice Model in the Theory Guide. Using the crevice models in Ansys Fluent are described in the following sections:

21.3.1. Setup Procedure

21.3.2. Crevice Model Solution Details

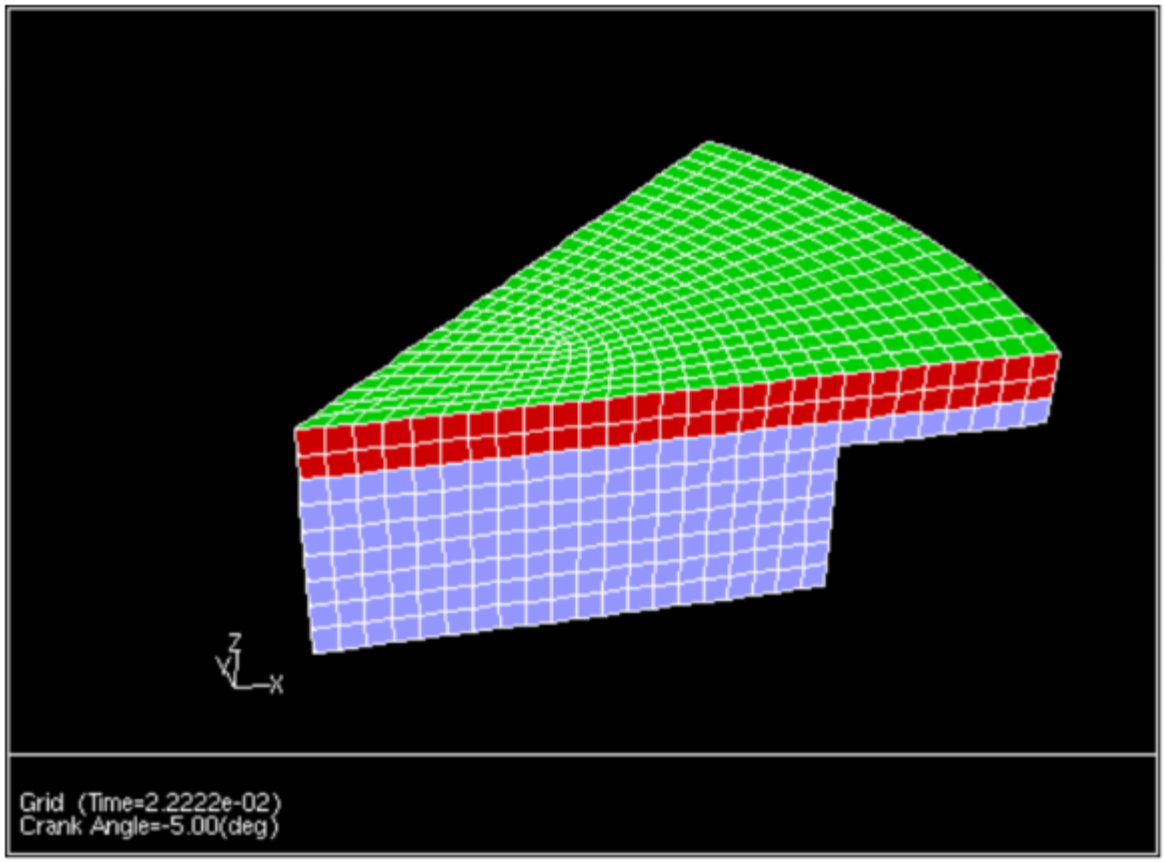
21.3.3. Postprocessing for the Crevice Model

21.3.1. Setup Procedure

An optical experimental engine [37] (p. 5715) is used below to show a working example of how to use the crevice model as it is implemented in Ansys Fluent. The mesh at ten crank angle degrees before top center is shown in Figure 21.7: Experimental Engine Mesh (p. 2578).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 21.7: Experimental Engine Mesh



The following example shows the necessary steps to enable the crevice model for a typical in-cylinder flow.

1. From the > prompt, enter the `define/models` menu by using the following text command:

   `define` → `models`

2. Enable the crevice model, as follows:

```
  /define/models   crevice-model?

  Enable crevice model? [no] yes
```

3. Enter the ring pack geometry:

```
  /define/models   crevice-model-controls

  Cylinder bore (m) [0.1] 0.1397
  Piston to bore clearance (m) [3.0e-5] 5.08e-05
  Piston crevice temperature (K) [400] 433
  Piston sector angle (deg) [360] 45
  Ring discharge coefficient [0.8] 0.7
  Pressure in crankcase (exit pressure) (Pa) [101325]
  Write  out  crevice  data  to  a  file?  [no]   yes

  output file name ["crev.out"]

 Available  wall  threads  are:  (wall.1  wall  wall-8)
 Leaking  wall []   wall.1
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
Shared boundary [] wall-8
   Selected  boundary  threads  :  (wall.1  wall-8)
  Use  these  zones?  [yes]   yes

Solve crevice model ? [no] yes
Number of rings [3]
          Width  of  ring  number  0  is:  [0.00375]
      Thickness  of  ring  number  0  is:  [0.0015]
        Spacing  of  ring  number  0  is:  [0.008]
   Land  Length  for  ring  number  0  is:  [0.00391]
       Top  Gap  of  ring  number  0  is:  [6e-05]
     Middle  Gap  of  ring  number  0  is:  [4e-05]
     Bottom  Gap  of  ring  number  0  is:  [6e-05]

          Width  of  ring  number  1  is:  [0.00375]
      Thickness  of  ring  number  1  is:  [0.0015]
        Spacing  of  ring  number  1  is:  [0.008]
   Land  Length  for  ring  number  1  is:  [0.00391]
       Top  Gap  of  ring  number  1  is:  [6e-05]
     Middle  Gap  of  ring  number  1  is:  [4e-05]
     Bottom  Gap  of  ring  number  1  is:  [6e-05]

          Width  of  ring  number  2  is:  [0.00375]
      Thickness  of  ring  number  2  is:  [0.0015]
        Spacing  of  ring  number  2  is:  [0.00391]
   Land  Length  for  ring  number  2  is:  [0.00391]
       Top  Gap  of  ring  number  2  is:  [6e-05]
     Middle  Gap  of  ring  number  2  is:  [4e-05]
     Bottom  Gap  of  ring  number  2  is:  [6e-05]


  Initial  conditions  in  ring  pack
  Pressure  1  is:  [4600623.5]
  Pressure  2  is:  [4173522.5]
  Pressure  3  is:  [3689110.5]
  Pressure  4  is:  [3130620]
  Pressure  5  is:  [2214841.8]
```

A fast way to set up multiple rings in the ring pack is to specify only one ring and enter the geometry. Once the ring geometry is entered, invoke the `crevice-model- controls` menu a second time and specify the number of rings desired. When the number of rings changes, the geometry from the first ring is copied to all subsequent rings. Default values can be taken for the rest of the way through the menu structure.

A summary of the crevice model is printed out by entering the `(crevice-summary)` command at the command prompt:

```
  >(crevice-summary)
                      crevice/n-rings  :  3
                   crevice/ring-width  :  (0.00375  0.00375  0.00375)
               crevice/ring-thickness  :  (0.0015  0.0015  0.0015)
                    crevice/ring-mass  :  (0.00375  0.00375  0.00375)
                 crevice/ring-spacing  :  (0.008  0.008  0.00391)
                  crevice/land-length  :  (0.00391  0.00391  0.00391)
                 crevice/top-ring-gap  :  (6e-05  6e-05  6e-05)
                 crevice/mid-ring-gap  :  (4e-05  4e-05  4e-05)
                 crevice/bot-ring-gap  :  (6e-05  6e-05  6e-05)
           crevice/piston-temperature  :  433
                 crevice/sector-angle  :  45
                   crevice/mid-gap-cd  :  0.7
                crevice/exit-pressure  :  101325
                      crevice/threads  :  (5  6)
           names  of  crevice/threads  :  (wall.1  wall-8)
                crevice/unit-roundoff  :  5.9604645e-08
        crevice/piston-bore-clearance  :  5.08e-05
                       crevice/write?  :  #t
                  crevice/output-file  :  crev.out
```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
                        crevice/solve?  :  #t
                      crevice/enabled?  :  #t
                    crevice/pressures   :  (4600623.5  4173522.5  3689110.5  3130620  2214841)
```

21.3.2. Crevice Model Solution Details

The under-relaxation factor for the crevice model source terms can be found in the **Solution Controls** task page. The default value for **Crevice Model Sources** is 0.8, which has been found to work well for motored engine simulations. Once the crevice model is enabled, the solution proceeds normally.

Solution → **Controls**

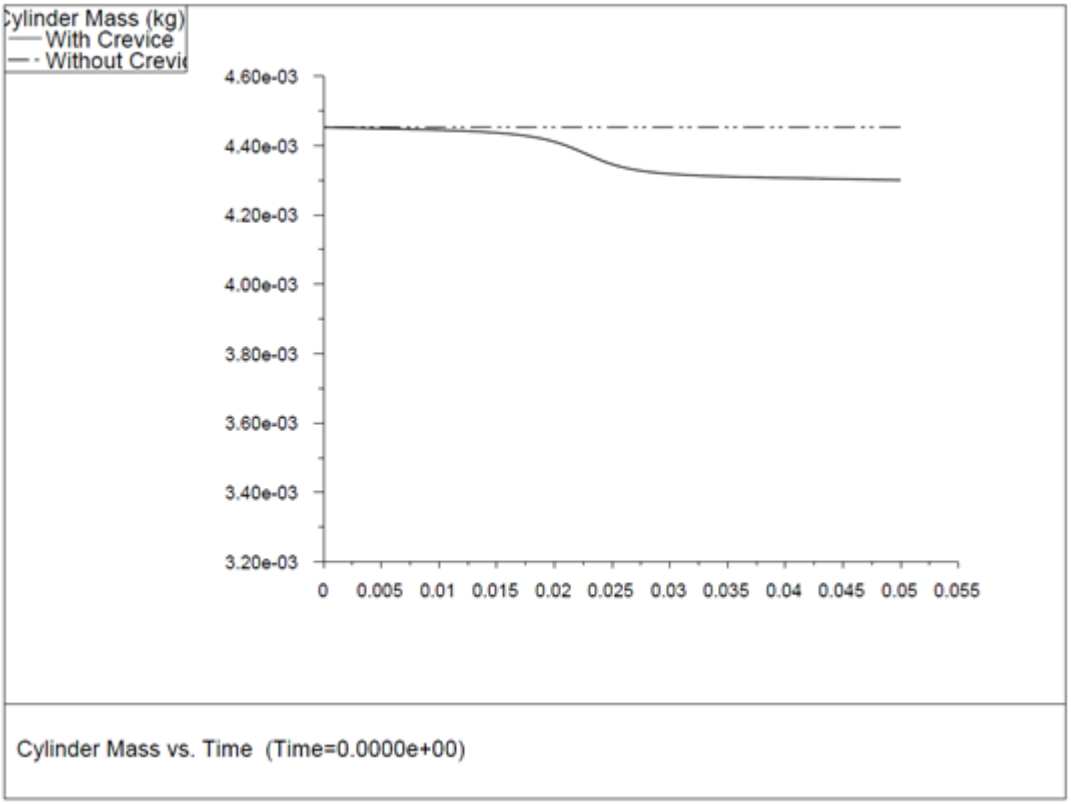
Solution → **Initialization**

Solution → **Run Calculation**

21.3.3. Postprocessing for the Crevice Model

A plot of cylinder mass with and without the crevice model during the motored engine simulation is shown in Figure 21.8: Cylinder Mass vs. Crank Angle (p. 2580). The rate of mass loss from the crevice is proportional to the pressure difference between the cylinder and the crankcase pressure defined in the text interface.

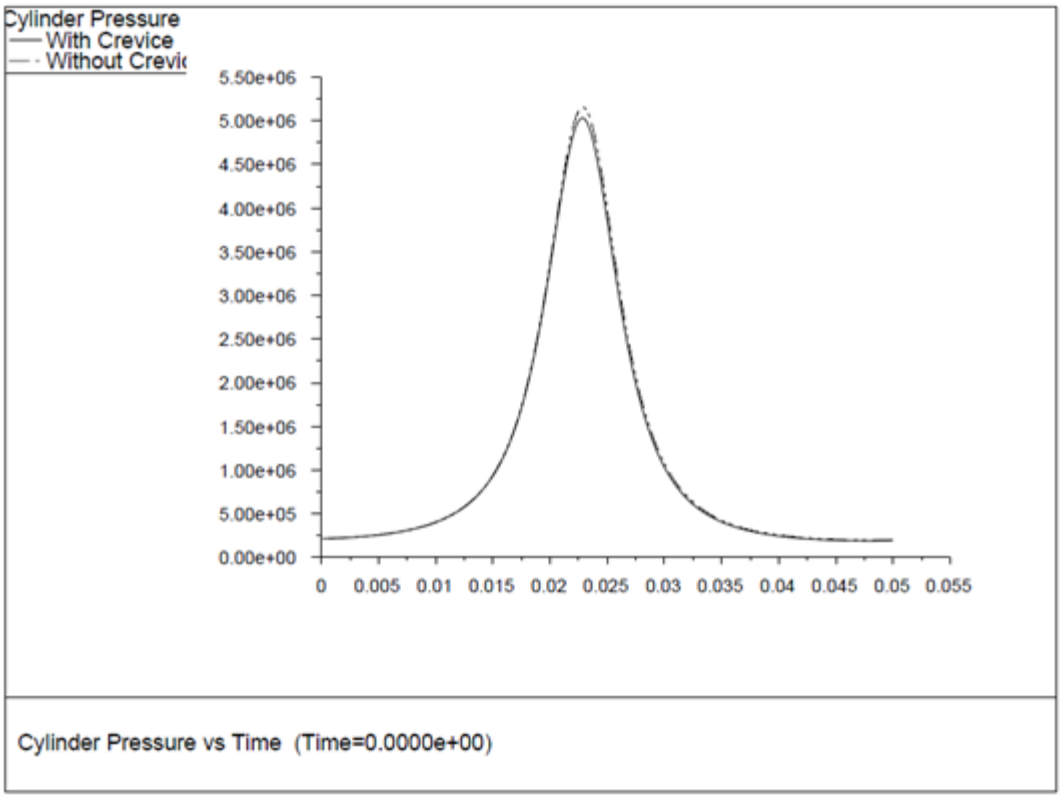
Figure 21.8: Cylinder Mass vs. Crank Angle



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A plot of cylinder pressure with and without the crevice model for the same engine simulation is shown in Figure 21.9: Cylinder Pressure vs. Crank Angle (p. 2581). The effect of the mass loss from the crevice is to lower the peak pressure in proportion to the total mass loss from the cylinder.

Figure 21.9: Cylinder Pressure vs. Crank Angle



21.3.3.1. Using the Crevice Output File

The pressure in the top ring land is defined as the cylinder pressure (that is, the pressure in the cells defining the ring landing). Intermediate pressures are available at any point during the Ansys Fluent session through the `(crevice-summary)` command as previously shown. If the optional data file output is chosen in the `crevice-model-controls`, the intermediate pressures in the defined crevices are printed to the file `crev.out` at the start of each new time step. The format of the file is as follows:

```
 # crank (deg) data-press[0...1...2...3...4...5...6] total_mdot
1.95500e+02 2.16650e+05 1.01325e+05 1.01325e+05 1.01325e+05 1.01325e+05 1.01325e+05 1.01325e+05 0.0
1.96000e+02 2.09945e+05 1.06794e+05 1.81553e+05 1.04111e+05 1.48582e+05 1.02202e+05 1.01325e+05 -1.6
1.96500e+02 2.17787e+05 1.13070e+05 1.88242e+05 1.07960e+05 1.53544e+05 1.03526e+05 1.01325e+05 -1.6
1.97000e+02 2.17434e+05 1.19065e+05 1.88060e+05 1.11705e+05 1.53475e+05 1.04830e+05 1.01325e+05 -1.6
1.97500e+02 2.17652e+05 1.24777e+05 1.88299e+05 1.15286e+05 1.53668e+05 1.06081e+05 1.01325e+05 -1.6
1.98000e+02 2.17937e+05 1.30215e+05 1.88594e+05 1.18711e+05 1.53900e+05 1.07283e+05 1.01325e+05 -1.6
```

where the first column is the current flow time (or crank angle), and the next $n_{cv}+2$ columns are the ring pressures (where n_{cv} is the number of crevice volumes, or $2n_r-1$), including the face pressure on the crevice cell, and the defined pressure at the crevice exit. The final column is the mass flow past the top ring. This file is currently formatted so that it can be read into the free **Gnuplot** plotting package, which is available at `www.gnuplot.info`.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To read the crevice output file into Ansys Fluent as a data file, you will need to put each column of the crevice output file in its own individual file. The first three lines of each column of the data file should be of the following form:

```
"Title"
"X-Label" "Y-Label"
0 0 0 0
```

where the title, x-label, and y-label strings are enclosed by double quotes and the third line of the file contains four zeros. The lines following the first three lines of the file are the columns you want to plot. For example, to plot column 1 versus column 3 of the crevice model output file in Ansys Fluent, you would enter the following commands in a Linux terminal:

```
cat > crev_col_1_3.dat.h5
"Column 1 vs Column 3"
"Crank Angle (deg)" "Pressure behind ring 1 (Pa)"
0 0 0 0
ctrl-d
```

where `ctrl-d` is the end-of-file character made holding down the **Ctrl** key and pressing **d** (**Ctrl**+**d**). To append columns 1 and 3 to this file, enter the following:

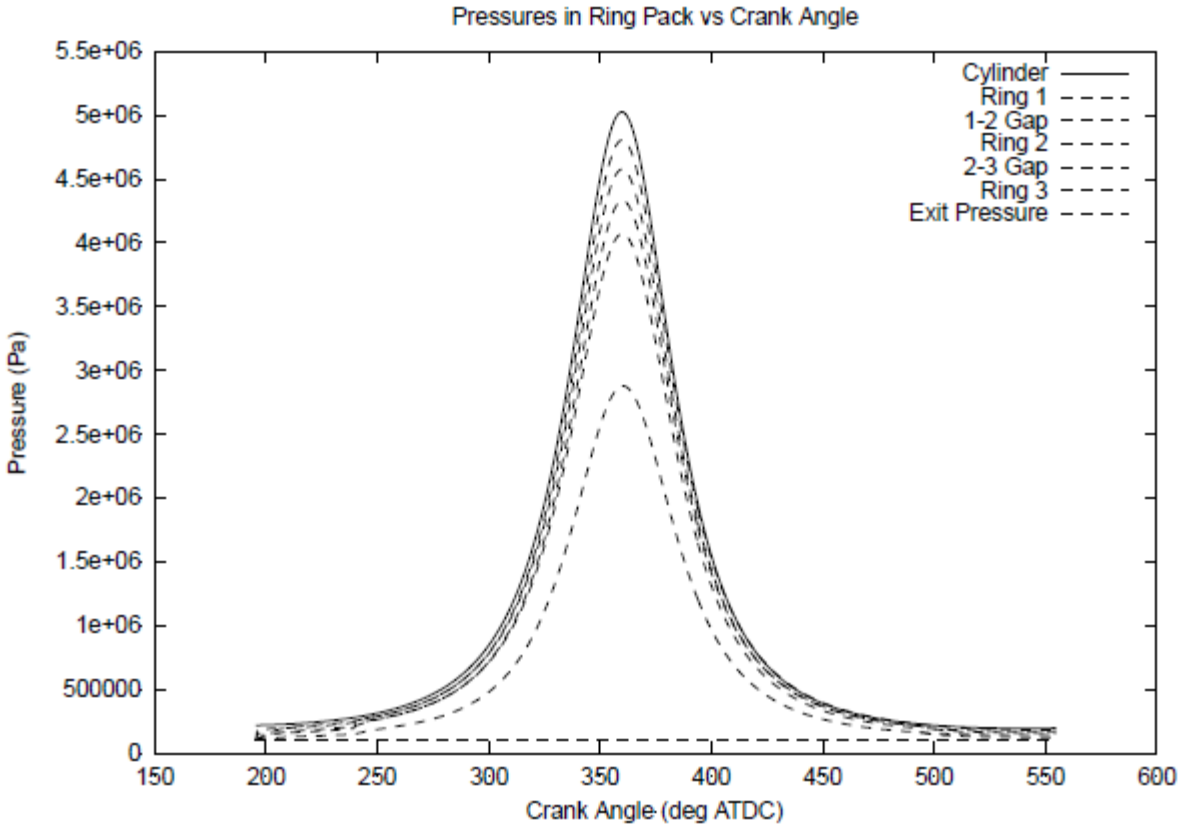
```
tail +2 crev.out | awk '{print $1, $3}' >> crev_col_1_3.dat.h5
```

The file `crev_col_1_3.dat.h5` can now be read into Ansys Fluent using the Plot Data Sources Dialog Box (p. 5297). See Creating an XY Plot From Multiple Data Sources (Including Files) (p. 4064) for details about creating x- y plots. For Windows users, the file `crev.out` can be imported into Excel for plotting purposes without any modification.

A **Gnuplot** plot of the pressure in the ring pack crevices for the above engine simulation is shown in Figure 21.10: Crevice Pressures (p. 2583). After an initial transient period where the flows in the network settle down, Figure 21.10: Crevice Pressures (p. 2583) shows that the pressure in the ring crevices follows the cylinder pressure in form, though with pressure magnitudes that are controlled by the ring pack geometry.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 21.10: Crevice Pressures



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 22: Modeling Pollutant Formation

This chapter discusses how to use the models available in Ansys Fluent for modeling pollutant formation. For information about the theory behind the models in Ansys Fluent, see Pollutant Formation in the Theory Guide.

Information is presented in the following sections:

22.1. NOx Formation

22.2. Soot Formation

22.3. Using the Decoupled Detailed Chemistry Model

22.1. NOx Formation

The following sections describe how to use the NO_x models in Ansys Fluent. For information about the theory behind the NO_x models in Ansys Fluent, see NOx Formation in the Theory Guide.

22.1.1. Using the NOx Model

22.1.2. Solution Strategies

22.1.3. Postprocessing

22.1.1. Using the NOx Model

22.1.1.1. Decoupled Analysis: Overview

NO_x concentrations generated in combustion systems are generally low. As a result, NO_x chemistry has negligible influence on the predicted flow field, temperature, and major combustion product concentrations. It follows that the most efficient way to use the NO_x model is as a postprocessor to the main combustion calculation.

The recommended procedure is as follows:

1. Calculate your combustion problem using Ansys Fluent as usual.

Important:

The premixed combustion model is not compatible with the NO_x model.

Important:

If you plan to use the Ansys Fluent selective non-catalytic reduction (SNCR) model for NO_x reduction, you must first include ammonia NH_3 or urea $CO(NH_2)_2$ (depending

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

upon which reagent is employed) as a fluid species in the main combustion calculation and define appropriate species injections, as described later in this section. See Defining the Species in the Mixture (p. 2375) for details about adding species to your model and Setting Initial Conditions for the Discrete Phase (p. 2718) for details about creating injections.

2. Enable the desired NO_x models (thermal, prompt, fuel, and/or N_2O intermediate NO_x, with or without reburn), define the fuel streams (for prompt NO_x and fuel NO_x only), and set the appropriate parameters, as described in this section.

   Setup → Models → Species → NOx Edit...

3. Define the boundary conditions for NO (and HCN, NH_3, or N_2O if necessary) at flow inlets.

   Setup → Boundary Conditions

4. (steady state only) In the Equations Dialog Box (p. 5166), turn off the solution of all variables except species NO (and HCN, NH_3, or N_2O, based on the model selected).

   Solution → Controls → Equations...

5. (transient cases only) In the **Run Calculation** task page, select **Postprocess Pollutants** and adjust **Max Post Iterations/Time Step** as needed.

   Solution → Run Calculation

6. Perform calculations until convergence (that is, until the NO—and HCN, NH_3, or N_2O, if solved—species residuals are below 10^{-6}).

   Solution → Run Calculation

7. Review the mass fractions of NO (and HCN, NH_3, or N_2O) by generating graphical plots or alphanumeric reports in the usual way.

8. Save a new set of case and data files, if desired.

   File → Write → Case & Data...

Inputs specific to the calculation of NO_x formation are explained in the remainder of this section.

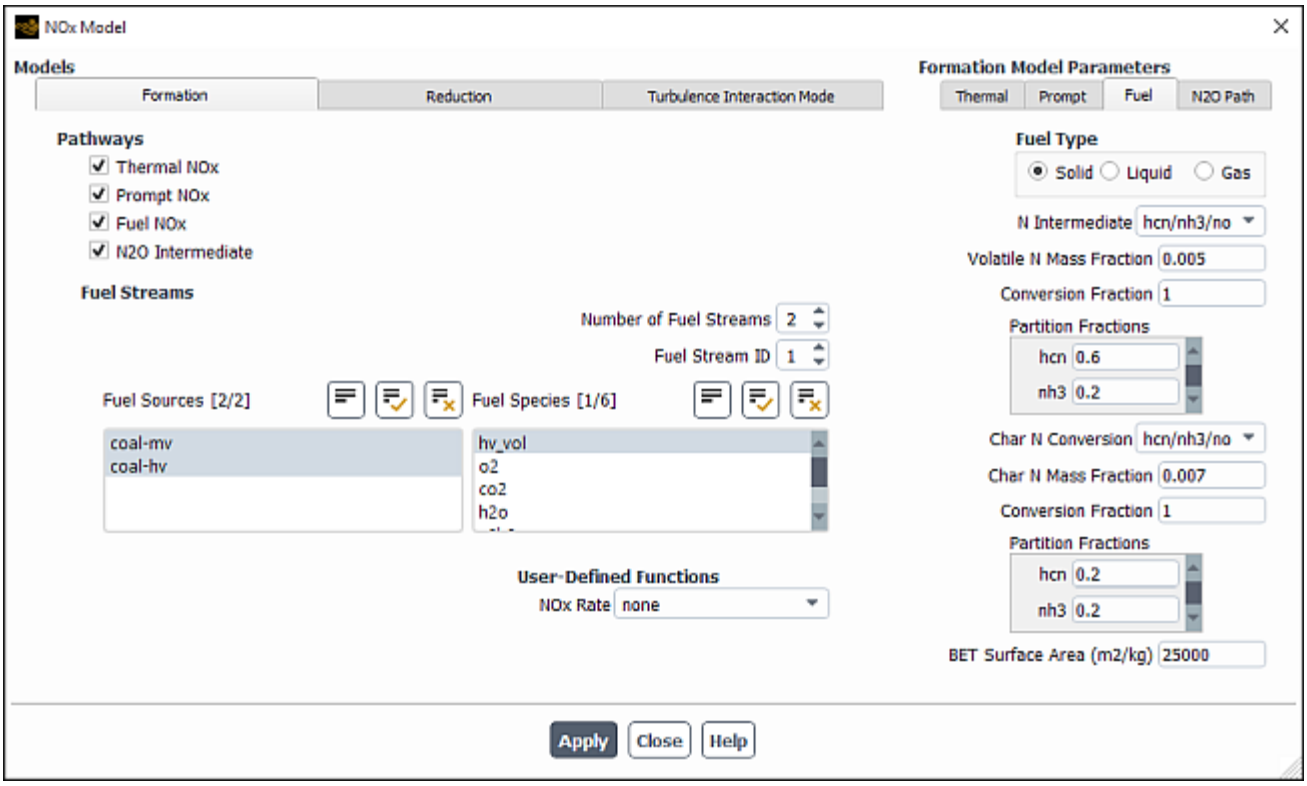
22.1.1.2. Enabling the NOx Models

To enable the NO_x models and set related parameters, you will use the NOx Model Dialog Box (p. 4816) (for example, Figure 22.1: The NOx Model Dialog Box (p. 2587)).

Setup → Models → Species → NOx Edit...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 22.1: The NOx Model Dialog Box



In the **Formation** tab, select the NO_x models under **Pathways** to be used in the calculation of the NO and HCN, NH_3, or N_2O concentrations:

- To enable thermal NO_x, turn on the **Thermal NOx** option.
- To enable prompt NO_x, turn on the **Prompt NOx** option.
- To enable fuel NO_x, turn on the **Fuel NOx** option.

> **Important:**
>
> When using the non-premixed combustion model, the following limitation exists for the **Fuel NOx** option:
>
> – The **Fuel NOx** option is only available if the DPM model is also enabled.

- To enable the formation of NO_x through N_2O intermediate, turn on the **N2O Intermediate** option. (Note that the N_2O option will not appear until you have enabled one of the other NO models listed above.)

Your selection(s) under **Pathways** will enable the calculation of thermal, prompt, fuel, and/or N_2O-intermediate NO_x in accordance with the chemical kinetic models described in Thermal NOx Formation through NOx Formation from Intermediate N2O in the Theory Guide. Mean NO formation rates will be computed directly from mean concentrations and temperature in the flow field.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

22.1.1.3. Defining the Fuel Streams

When modeling fuel NO_x formation, Ansys Fluent allows you to define up to three separate fuel streams, and to select N fuel sources for each fuel stream.

You can define multiple fuel streams to include in your model with the following configurations:

- Solid and liquid fuels (combusting particle and droplet) both contributing to fuel NO_x.
- Two or three solid fuels with different N-content and NO_x model parameters (two or three combusting particles), for example coal blends, coal-biomass cofiring, and so on.
- Two or three liquid fuels with different N-content and NO_x model parameters (two or three droplet or multicomponent particles).
- Gas and solid (or droplet) fuel both contributing to fuel NO_x.

  In addition, you can model one solid fuel contributing to NO_x in the presence of another reacting solid particle not containing any N (for example, sorbent injection in a coal furnace or calcination reaction in cement kiln) by specifying the active N fuel source for NO_x formation. For this configuration, you will not need to define multiple fuel streams in your model.

If **Fuel NOx** is enabled in the **Pathways** group box in the **Formation** tab, perform the following steps to define multiple fuel streams:

1. Specify the **Number of Fuel Streams** in the **Fuel Streams** group box.

   Note:

   You are allowed up to three separate fuel streams.

2. Define the first fuel stream.

   a. Select the fuel stream to be defined by using the arrow keys of the **Fuel Stream ID** field.

   b. Select the **Fuel Type** in the **Fuel** tab for the **Formation Model Parameters**.

   c. If your **Fuel Type** is **Liquid** or **Solid**, select the N sources from the **Fuel Sources** list.

      - If the **Fuel Type** is **Solid** and you have defined multiple injections with different combusting particle materials in your reacting flow calculation, the available combusting particle materials will be listed in the **Fuel Sources** list. Select one or more materials from the list to be included as fuel sources in the **Fuel NOx** calculation. Your selection will be used to determine the char burnout rate S_c and volatile release rate S_{vol} in the coal fuel NO_x formation rate models described in Fuel NOx from Coal in the Theory Guide. Make sure to deselect all combusting particle materials that do not contribute to NO_x.
      - If the **Fuel Type** is **Liquid** and you have defined multiple injections with different droplet or multicomponent particle materials in your reacting flow calculation, the available materials will be listed in the **Fuel Sources** list. Select one or more materials from the list to

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

be included as fuel sources in the **Fuel NOx** calculation. Your selection will be used to determine the fuel release rate S_{fuel} in the liquid fuel NO_x formation rate models described in Fuel NOx from Intermediate Hydrogen Cyanide (HCN) and Fuel NOx from Intermediate Ammonia (NH3) in the Theory Guide. Make sure to deselect all droplet and multicomponent materials that do not contribute to NO_x.

d. If you are also modeling **Prompt NOx**, select the fuel species from the **Fuel Species** list. You cannot select more than 5 fuel species for each fuel stream, and the total number of fuel species selected for all the fuel streams combined cannot exceed 10. You can define the same, or different **Fuel Species**, **Fuel Carbon Number** and **Equivalence Ratio** for each stream. The **Fuel Carbon Number** and **Equivalence Ratio** can be modified in the **Prompt** tab under **Formation Model Parameters**.

e. If the **Fuel Type** in your model is **Gas**, select one or more materials from the **Fuel Species** list. Your selection will be used to determine the mean limiting reaction rate $\mathfrak{R}_{cf}$ in the gaseous fuel NO_x formation rate models described in Fuel NOx from Intermediate Hydrogen Cyanide (HCN) and Fuel NOx from Intermediate Ammonia (NH3) in the Theory Guide. If you are also modeling **Prompt NOx** formation, the same fuel selection will apply also for the prompt NO_x model.

f. Set the other parameters associated with your selected pathway(s) in the **Prompt** and/or **Fuel** tabs under **Formation Model Parameters**. See Setting Prompt NOx Parameters (p. 2591) and Setting Fuel NOx Parameters (p. 2592) for details.

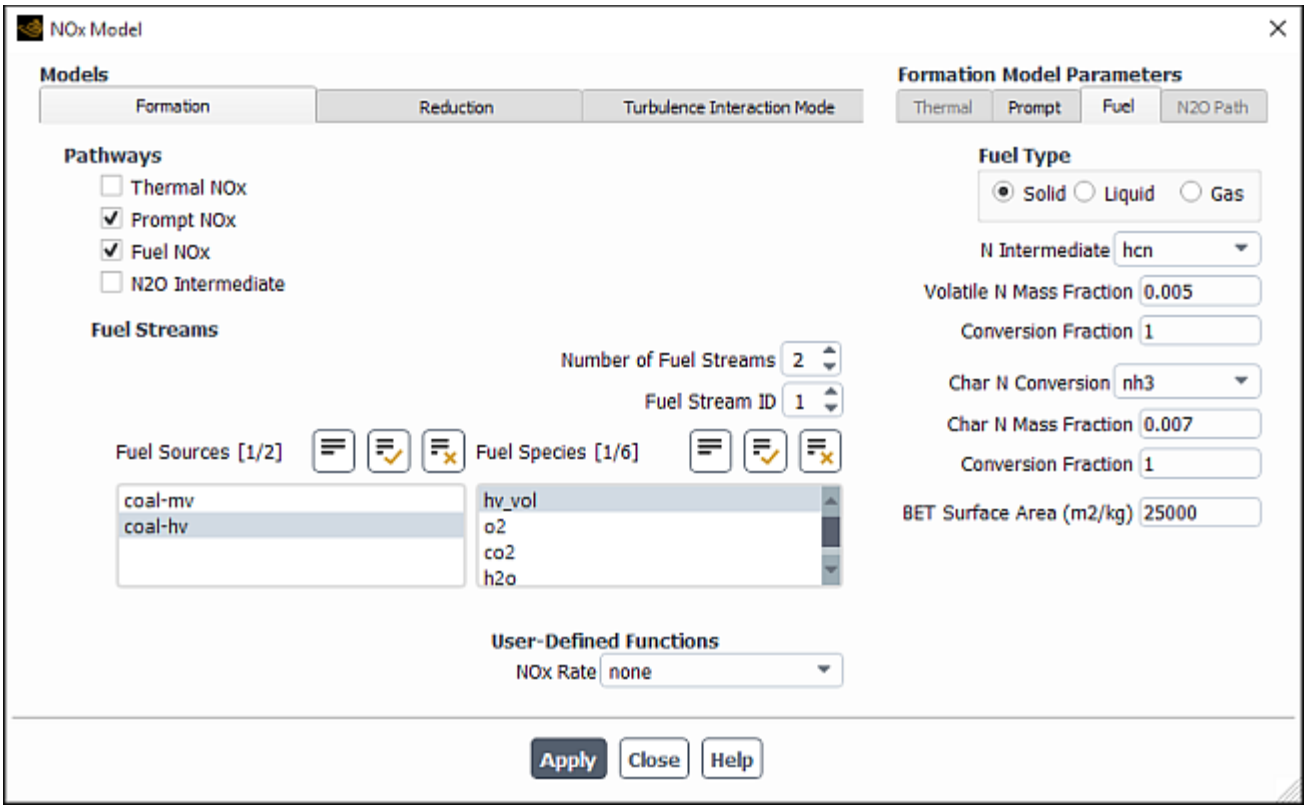
3. Repeat steps 2.(a)–2.(f) for each additional fuel stream.

Important:

- The gaseous fuel option is available only when the species model is enabled.
- Important considerations should be made when reading case and data files set up in a version of Ansys Fluent 14.5 or earlier:
  - When reading a case and data file with multiple injection materials that was set up in Ansys Fluent version 14.5 or earlier, Ansys Fluent will initialize the injection material specific fuel N sources for the fuel NO_x model. Ansys Fluent will perform a DPM iteration when the flow iterations are initiated.
  - When reading a case file that was set up in a version older than Ansys Fluent 14.5 with multiple fuel streams defined for the **Fuel NOx** model, you must review the setup and select the N sources from the **Fuel Sources** list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 22.2: The NOx Model Dialog Box Displaying the Fuel Streams



22.1.1.4. Specifying a User-Defined Function for the NOx Rate

You can choose to specify a user-defined function for the rate of NO_x production. By default, the rate returned from the UDF is added to the rate returned from the standard NO_x production options, if any are selected. You also have the option of replacing any or all of Ansys Fluent's NO_x rate calculations with your own user-defined NO_x rate.

In addition to or instead of using the UDF to specify the NO_x rate, you can use it to specify custom values for the maximum limit (T_{max}) that is used for the integration of the temperature PDF (when temperature is accounted for in the turbulence interaction modeling).

To use a UDF to add a rate to Ansys Fluent's NO_x rate calculations, you must compile and load the desired function, and then select it from the **NOx Rate** drop-down list in the **User-Defined Functions** group box in the **Formation** tab. After you have selected the UDF, you have the following options:

- You can specify that your custom rate is added to the Ansys Fluent NO_x rate calculations, by retaining the default selection of **Add to Fluent Rate** in the **UDF Rate** group box for the appropriate NO_x formation pathway(s) (for example, in the **Fuel** tab).
- You can replace the Ansys Fluent NO_x rate calculations with your custom rate, by selecting **Replace Fluent Rate** in the **UDF Rate** group box for the appropriate NO_x formation pathway(s) (for example, in the **Fuel** tab).
- You can specify custom values for T_{max}, by selecting **user-defined** from the **Tmax Option** drop-down list in the **Turbulence Interaction Mode** tab.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See the Fluent Customization Manual for details about user-defined functions.

22.1.1.5. Setting Thermal NOx Parameters

The NO_x routines employ three methods for calculation of thermal NO_x (as described in Method 1: Equilibrium Approach in the Theory Guide). You will specify the method to be used in the **Thermal** tab, under **Formation Model Parameters** in the NOx Model Dialog Box (p. 4816):

- To choose the equilibrium method, select **equilibrium** in the **[O] Model** drop-down list.
- To choose the partial equilibrium method, select **partial-equilibrium** in the **[O] Model** or **[OH] Model** drop-down list.
- To use the predicted O and/or OH concentration, select **instantaneous** in the **[O] Model** or **[OH] Model** drop-down list.

Important:

Note that the urea model uses the [OH] model.

If you hooked a user-defined function in the **Formation** tab, you can make a selection in the **UDF Rate** group box to specify the treatment of the user-defined NO_x rate:

- Select **Replace Fluent Rate** to replace Ansys Fluent's thermal NO_x rate calculations with the custom NO_x rate produced by your UDF.
- Select **Add to Fluent Rate** to add the custom NO_x rate produced by your UDF to Ansys Fluent's thermal NO_x rate calculations.

22.1.1.6. Setting Prompt NOx Parameters

Prompt NO_x formation is predicted using Equation 9.26 and Equation 9.28 in the Theory Guide. For each fuel stream specified in the **Fuel Stream ID** field in the **Formation** tab, set the parameters in the **Prompt** tab under **Formation Model Parameters** in the NOx Model Dialog Box (p. 4816) in the following manner:

- Set the **Fuel Carbon Number** to specify the number of carbon atoms per fuel molecule.
- Set the **Equivalence Ratio** as follows:

$$\text{Equivalence Ratio} = \frac{\text{actual fuel-to-air ratio}}{\text{stoichiometric fuel-to-air ratio}} \tag{22.1}$$

If you hooked a user-defined function in the **Formation** tab, you can make a selection in the **UDF Rate** group box to specify the treatment of the user-defined NO_x rate:

- Select **Replace Fluent Rate** to replace Ansys Fluent's prompt NO_x rate calculations with the custom NO_x rate produced by your UDF.
- Select **Add to Fluent Rate** to add the custom NO_x rate produced by your UDF to Ansys Fluent's prompt NO_x rate calculations.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

22.1.1.7. Setting Fuel NOx Parameters

When using the fuel NO_x model, you must set the parameters in the **Fuel** tab under **Formation Model Parameters** for each fuel stream specified in the **Fuel Stream ID** field in the **Formation** tab.

If you hooked a user-defined function in the **Formation** tab, you can make a selection in the **UDF Rate** group box to specify the treatment of the user-defined NO_x rate:

- Select **Replace Fluent Rate** to replace Ansys Fluent's fuel NO_x rate calculations with the custom NO_x rate produced by your UDF.
- Select **Add to Fluent Rate** to add the custom NO_x rate produced by your UDF to Ansys Fluent's fuel NO_x rate calculations.

If there is no NO_x rate UDF or if you selected **Add to Fluent Rate**, you must define fuel parameters. To begin, specify the fuel type in the following manner:

- For solid fuel NO_x, select **Solid** under **Fuel Type**.
- For liquid fuel NO_x, select **Liquid** under **Fuel Type**.
- For gaseous fuel NO_x, select **Gas** under **Fuel Type**.

Note that you can use only one of the fuel types for a given fuel stream. The **Gas** option is available only when the **Species Transport** model is enabled (see Enabling Species Transport and Reactions and Choosing the Mixture Material (p. 2359)).

22.1.1.7.1. Setting Gaseous and Liquid Fuel NOx Parameters

If you have selected **Gas** or **Liquid** as the **Fuel Type**, you must also specify the following:

- Select the intermediate species (**hcn**, **nh3**, or **hcn/nh3/no**) in the **N Intermediate** drop-down list.
- Set the correct mass fraction of nitrogen in the fuel (kg nitrogen per kg fuel) in the **Fuel N Mass Fraction** field.
- Specify the overall fraction of the fuel N, by mass, that will be converted to the intermediate species and/or product NO in the **Conversion Fraction** field. The **Conversion Fraction** for the **N Intermediate** has a default value of 1. Therefore, any remaining N will not contribute to NO_x formation. This is based on the assumption that the remaining volatile N will convert to gas phase nitrogen. However, this has very little effect on the overall mass fraction of gas phase nitrogen. Therefore, you do not have to solve for nitrogen species when solving pollutant transport equations.
- If you selected **hcn/nh3/no** as the intermediate, specify the fraction of the converted fuel N, by mass, that will become **hcn** and **nh3** under **Partition Fractions**. The fraction of fuel N that will become NO will be calculated by the remainder.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that setting a partition fraction of 0 for both HCN and NH_3 is equivalent to assuming that all fuel N is converted to the final product NO, whereas a partition fraction of 0 for HCN and 1 for NH_3 is the same as selecting **nh3** as the intermediate.

Ansys Fluent will use Equation 9.31 and Equation 9.32 in the Theory Guide (for HCN) or Equation 9.42 and Equation 9.43 in the Theory Guide (for NH_3) to predict NO formation for a gaseous or liquid fuel.

Important:

Note that there is a limitation that must be considered when defining more than one liquid fuel stream. See Defining the Fuel Streams (p. 2588) for details.

22.1.1.7.2. Setting Solid (Coal) Fuel NOx Parameters

For solid (coal) fuel, Ansys Fluent will use Equation 9.55 and Equation 9.56 in the Theory Guide (for HCN) or Equation 9.62 and Equation 9.63 in the Theory Guide (for NH_3) to predict NO formation. Several inputs are required for the coal fuel NO_x model as follows:

- Select the intermediate species (**hcn**, **nh3**, or **hcn/nh3/no**) in the **N Intermediate** drop-down list.
- Specify the mass fraction of nitrogen in the volatiles in the **Volatile N Mass Fraction** field.
- Specify the overall fraction of the volatile N, by mass, that will be converted to the intermediate species and/or product NO in the **Conversion Fraction** field.
- If you selected **hcn/nh3/no** as the volatile N intermediate, specify the fraction of the converted volatile N, by mass, that will become **hcn** and **nh3** under **Partition Fractions**. The fraction of volatile N that will become NO will be calculated by the remainder.
- Select the char N conversion path from the **Char N Conversion** drop-down list as **no**, **hcn**, **nh3**, or **hcn/nh3/no**. Note that **hcn** or **nh3** can be selected only if the same species has been selected as the intermediate species in the **N Intermediate** drop-down list.
- Specify the mass fraction of nitrogen in the char in the **Char N Mass Fraction** field.
- Specify the overall fraction of the char N, by mass, that will be converted to the intermediate species and/or product NO in the **Conversion Fraction** field.
- If you selected **hcn/nh3/no** as the char N conversion, specify the fraction of the converted char N, by mass, that will become **hcn** and **nh3** under **Partition Fractions**. The fraction of char N that will become NO will be calculated by the remainder.
- Define the BET internal pore surface area (see BET Surface Area in the Theory Guide for details) of the particles in the **BET Surface Area** field.

Important:

Note that there are limitations that must be considered when defining more than one solid fuel stream. See Defining the Fuel Streams (p. 2588) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following equations are used to determine the mass fraction of nitrogen in the volatiles and char:

$$\dot{m}_{N_{v/c}} = \dot{m}_{v/c} * mf_{N_{v/c}} \tag{22.2}$$

where

$\dot{m}_{N_{v/c}}$ = release rate of fuel nitrogen in kg/s

$\dot{m}_{v/c}$ = release rate of volatiles (v) or char (c) in kg/s

$mf_{N_{v/c}}$ = mass fraction of nitrogen in volatiles or char

Let

TN_{fuel} = total nitrogen mass fraction in daf coal (that is, from daf ultimate analysis)

N_{split} = char nitrogen as a fraction of total nitrogen

F_{vol} = mass fraction of volatiles in daf coal

F_{char} = mass fraction of char in daf coal

Then the following should hold:

$$F_{vol} + F_{char} = 1 \tag{22.3}$$

$$\frac{F_{char} * mf_{N_c}}{TN_{fuel}} = N_{split} \tag{22.4}$$

$$F_{vol} * mf_{N_v} + F_{char} * mf_{N_c} = TN_{fuel} \tag{22.5}$$

$$mf_{N_v} = \left(1 - N_{split}\right) * \frac{TN_{fuel}}{F_{vol}} \tag{22.6}$$

$$mf_{N_c} = N_{split} * \frac{TN_{fuel}}{F_{char}} \tag{22.7}$$

22.1.1.8. Setting N2O Pathway Parameters

The formation of NO through an N_2O intermediate can be predicted by any of quasi-steady and transported-simple methods, which can be specified in the **N2O Path** tab.

- To choose the quasi-steady-state method, select **quasi-steady** in the **N2O Model** drop-down list.

Important:

The transport equation for the species N_2O will not be solved for N_2O. However, N_2O will be updated at every iteration. Therefore, the residual values that appear for N_2O are always zero. Do not be alarmed if the solver keeps printing zero at each iteration.

- To choose the simplified form of the N_2O-intermediate mechanism, select **transported-simple** in the **N2O Model** drop-down list. Here, the species N_2O is added to the list of pollutant species, and its mass fraction is solved via a transport equation.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The atomic O concentration will be calculated according to the thermal NO_x **[O] Model** that you have specified previously. If you have not selected the **Thermal NOx** pathway, then you will be given the option to specify an **[O] Model** for the N_2O pathway calculation. The same three options for the thermal NO_x **[O] Model** will be the available options.

If you hooked a user-defined function in the **Formation** tab, you can make a selection in the **UDF Rate** group box to specify the treatment of the user-defined NO_x rate:

- Select **Replace Fluent Rate** to replace the NO_x rate calculated by Ansys Fluent using N_2O intermediates with the custom NO_x rate produced by your UDF.
- Select **Add to Fluent Rate** to add the custom NO_x rate produced by your UDF to the NO_x rate calculated by Ansys Fluent using N_2O intermediates.

22.1.1.9. Setting Parameters for NOx Reburn

To enable NO_x reduction by reburning, click the **Reduction** tab in the NOx Model Dialog Box (p. 4816) and enable the **Reburn** option under **Methods**. In the expanded portion of the dialog box, as shown in Figure 22.3: The NOx Dialog Box Displaying the Reburn Reduction Method (p. 2596), click the **Reburn** tab under **Reduction Method Parameters**, where you can choose from the following options:

- To choose the instantaneous method, select **instantaneous [CH]** in the **Reburn Model** drop-down list.

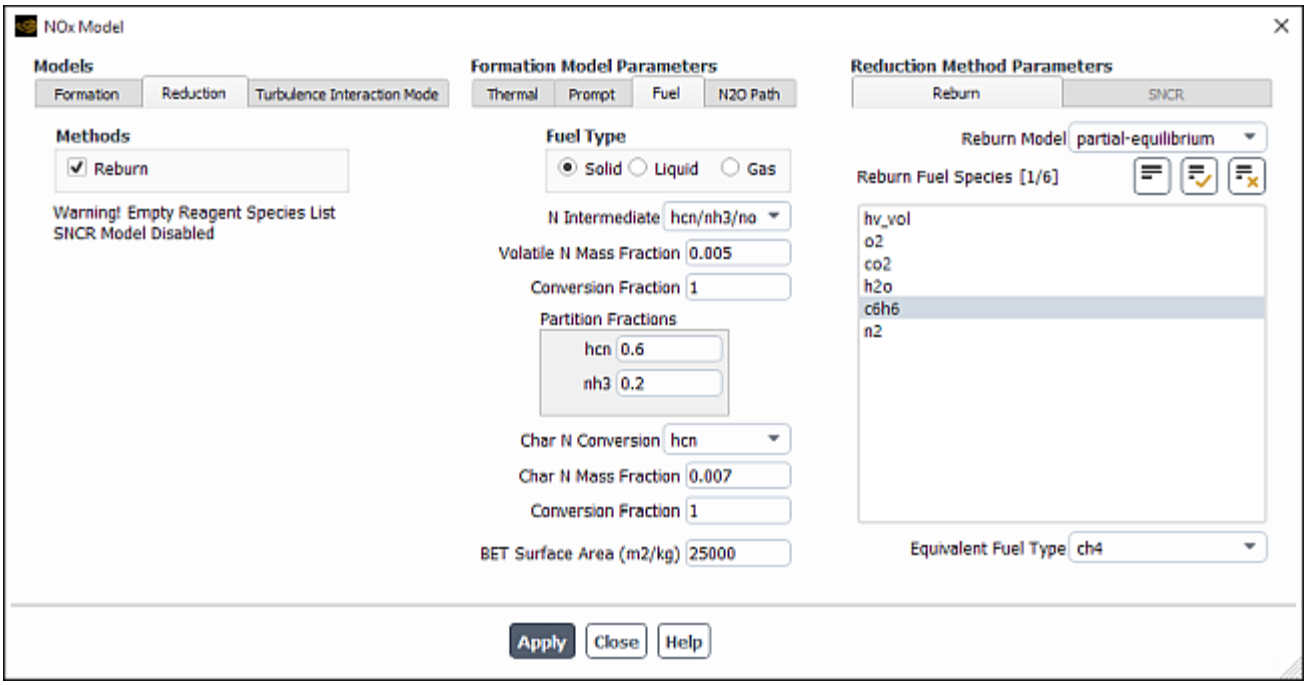
  Important:

  When you use this method, you must be sure to include the species CH, CH_2, and CH_3 in your problem definition. See NOx Reduction by Reburning in the Theory Guide for details.

- To choose the partial equilibrium method, select **partial-equilibrium** in the **Reburn Model** drop-down list. You then must select the **Reburn Fuel Species** from the list of available species. Ansys Fluent will allow you select up to six reburn fuel species. Specify the **Equivalent Fuel Type** (**ch4**, **ch3**, **ch2**, or **ch**). For example, if you choose methane as the reburn fuel, then the **Equivalent Fuel Type** would be **ch4**. If you choose a reburn fuel such as **hv_vol** (a volatile component of coal), then you must specify the most appropriate equivalent hydrocarbon fuel type so that the partial equilibrium model will be set up correctly.
- Due to coal volatiles behaving very differently, it is important to select the correct equivalent fuel type. You must first consider the volatile fuel composition, then check the C/H ratio to find the fuel that most closely matches CH, CH_2, CH_3, or CH_4 [90] (p. 5718). While the method for best determining the equivalent fuel is debatable, considering the C/H ratio of the fuel itself is a reasonable indicator.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 22.3: The NOx Dialog Box Displaying the Reburn Reduction Method

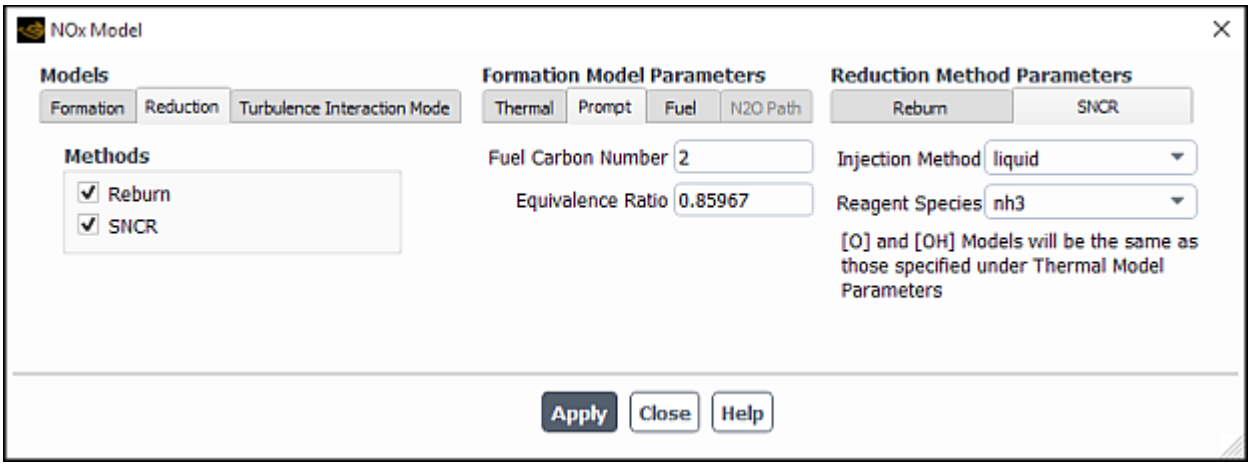


22.1.1.10. Setting SNCR Parameters

Prior to enabling reduction by SNCR, make sure that you have included in the species list **nh3** (for reduction by ammonia injection) and **co<nh2>2** (for reduction by urea injection). See NOx Reduction by SNCR in the Theory Guide for detailed information about SNCR theory.

1. To enable NO_x reduction by SNCR, click the **Reduction** tab in the NOx Model Dialog Box (p. 4816) and enable the **SNCR** option under **Methods**, as shown in Figure 22.4: The NOx Dialog Box Displaying the SNCR Reduction Method (p. 2596).

   Figure 22.4: The NOx Dialog Box Displaying the SNCR Reduction Method



2. In the **SNCR** tab under **Reduction Method Parameters**,set from the following options:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To have ammonia or urea included as a gas-phase pollutant species from the injection locations, select **gaseous** in the **Injection Method** drop-down list.

  If you plan to select this option for NO_x postprocessing, then you must also include ammonia or urea as a gas-phase species. Additionally, you must specify the mass fraction of ammonia or urea at the respective inlet for the SNCR injection. You must include this set of inputs prior to the main Ansys Fluent combustion calculation.

- To have ammonia included as a liquid-phase pollutant species from the injection locations, select **liquid** in the **Injection Method** drop-down list. If urea is injected as a liquid solution, then select **liquid** in the **Injection Method** drop-down list. Note that you must enable the DPM model with urea or ammonia included as a material.

  Note:

  All particle types such as droplets, combusting, and multicomponent particles are compatible with the SNCR model.

  If you plan to select this option for NO_x postprocessing, then you must include NH_3 as both a gas-phase and liquid-phase species. Additionally, you must specify injection locations for liquid droplet ammonia particles and set gaseous ammonia as the evaporation species. You must include this set of inputs in conjunction with the main Ansys Fluent combustion calculation.

  Since urea is a subliming solid, and usually is injected as a solution, mixed in water, you have to define solid properties for urea under the Create/Edit Materials Dialog Box (p. 4885). It is assumed that the water evaporates before urea begins its subliming process. The sublimation process is modeled similar to the single rate devolatilization model of coal. You will supply the value for the sublimation rate (s^{-1}). You must specify the water content while defining the injection properties.

- Specify the SNCR **Reagent Species** as **nh3** (ammonia) or **co<nh2>2** (urea) in the drop-down list.

- When using the non-premixed combustion model with a liquid-phase reagent injection, enter a value in the **Reagent Fraction in Stream** to specify the mass fraction of the reagent in the reagent stream. The remaining mass fraction is assumed to be water. If you enabled a secondary stream in your PDF calculation, by default the secondary stream will act as the reagent stream. You can assign the primary stream as the reagent stream by using the text command that follows (enter `0` in response to the `PDF Stream ID` prompt that follows the `Injection Method` prompt):

  ```
  define/models/nox-parameters/nox-chemistry
  ```

- If the **Reagent Species** selected is **co <nh2> 2**, then you will either accept the **rate-limiting** option for **Urea Decomposition**, or specify the **NH3 Conversion** value when selecting a **user-specified Urea Decomposition**.

  You will use the urea decomposition under the **SNCR** tab to define which of the two decomposition models is to be used. The first model (which is the default) is the rate-limiting decomposition model, as given in Table 9.3: Two-Step Urea Breakdown Process in the Theory Guide.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

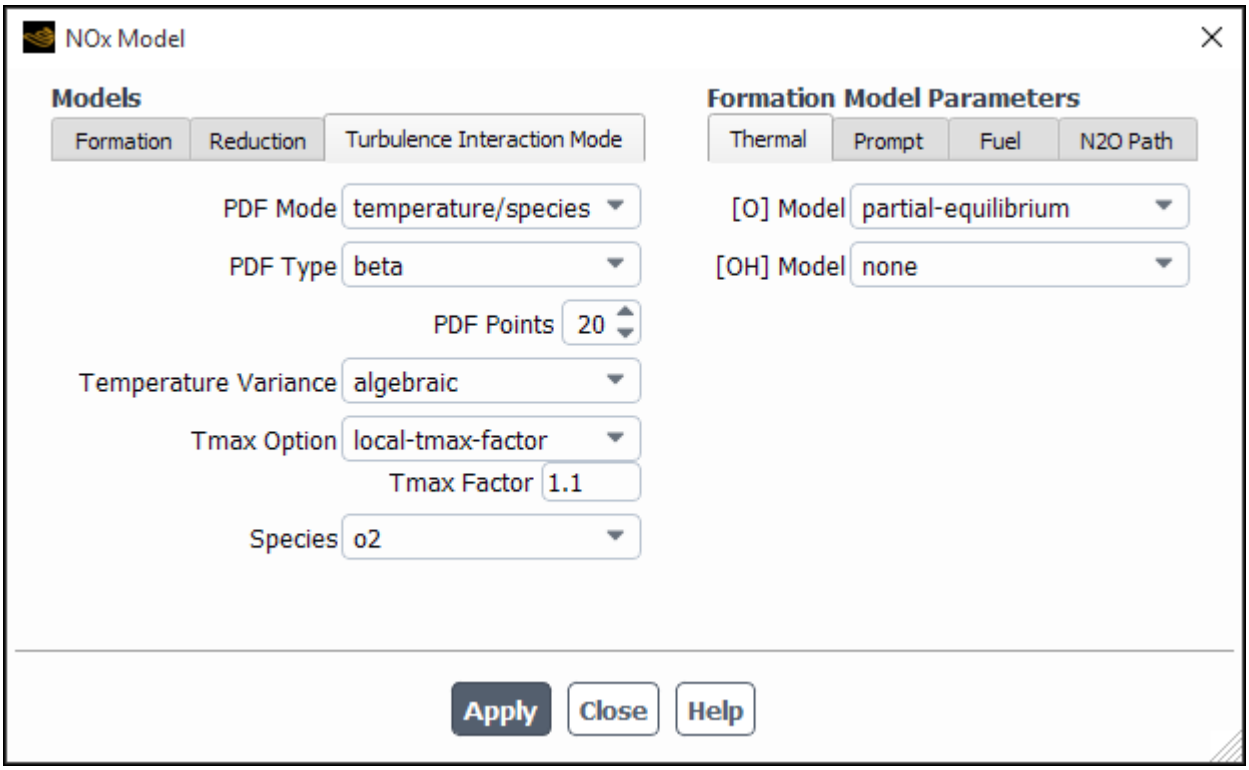
Ansys Fluent will then calculate the source terms according to the rates given in Table 9.3: Two-Step Urea Breakdown Process in the Theory Guide. The second model is for when you assume that the urea decomposes instantly into ammonia and HNCO at a given proportion. In this case, you will specify the molar conversion fraction for ammonia, assuming that the rest of the urea is converted to HNCO. An example value is given above.

The value for **user-specified** NH_3 conversion is the mole fraction of NH_3 in the mixture of NH_3 and HNCO that is instantly created from the reagent injection. In this case, there is no urea source because all of reagent is assumed to convert to both NH_3 and HNCO, instantly.

22.1.1.11. Setting Turbulence Parameters

If you want to take into account turbulent fluctuations (as described in NOx Formation in Turbulent Flows in the Theory Guide) when you compute the specified NO_x formation (thermal, prompt, and/or fuel, with or without reburn), define the turbulence parameters in the **Turbulence Interaction Mode** tab.

Figure 22.5: The NOx Model Dialog Box and the Turbulence Interaction Mode Tab



Select one of the options in the **PDF Mode** drop-down list in the **Turbulence Interaction Mode** tab:

- Select **temperature** to take into account fluctuations of temperature.
- Select **temperature/species** to take into account fluctuations of temperature and mass fraction of the species selected in the **Species** drop-down list (which appears when you select this option).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- (non-premixed and partially premixed combustion calculations only) Select **mixture fraction** to take into account fluctuation in the mixture fraction(s).

Important:

When modeling the formation of other pollutants along with NO_x, you should compare the selections made in the **PDF Mode** drop-down lists in the **Turbulence Interaction Mode** group boxes of the NOx Model Dialog Box (p. 4816) and the **Turbulence Interaction Mode** group box of the **Soot Model** dialog box. If **mixture fraction** is selected in any of these dialog boxes, then it must be selected in all of the others as well.

The **mixture fraction** option is available only if you are using either the non-premixed or partially premixed combustion model to model the reacting system. If you use the **mixture fraction** option, the instantaneous temperatures and species concentrations are taken from the PDF look-up table as a function of mixture fraction and enthalpy and the instantaneous NO_x rates are calculated at each cell. The PDF used for convoluting the instantaneous NO_x rates is the same as the one used to compute the mean flow-field properties. For example, for single-mixture fraction models the beta PDF is used, and for two-mixture fraction models, the beta or the double delta PDF can be used. The PDF for mixture fraction is calculated from the values of mean mixture fraction and variance at each cell, and the instantaneous NO_x rates are convoluted with the mixture fraction PDF to yield the mean rates in turbulent flow.

Note:

When `mixture fraction` is selected as **PDF mode**, Ansys Fluent calculates and tabulates the NO_x rates apriori for faster computations of NO_x rates. The NO_x rates are tabulated at the first NO_x iteration and a message

```
Performing Pollutant PDF integrations...
```

is displayed in the Fluent console.

If you selected **temperature** or **temperature/species** for the **PDF Mode**, you should define the following parameters in the **Turbulence Interaction Mode** tab:

PDF Type

allows you to specify the shape of the PDF, which is then integrated to obtain mean rates for the temperature and (if you selected **temperature/species** for the **PDF Mode**) the species. If you select **beta**, the PDF will be modeled using Equation 9.108 in the Theory Guide. If you select **gaussian**, the PDF will be modeled using Equation 9.111 in the Theory Guide.

PDF Points

allows you to specify the number of points used to integrate the beta or Gaussian function in Equation 9.105 or Equation 9.106 in the Theory Guide on a histogram basis. The default value of 10 is a minimum value. It is recommended that you run the calculation with this minimum value to convergence, and then keep increasing the value (for example, 20–25) until the pollutants of interest stop changing. Increasing this value may improve accuracy, but will also increase the computation time.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Temperature Variance

allows you to specify the form of transport equation that is solved to calculate the temperature variance. The default selection is **algebraic**, which is an approximate form of the transport equation (see Equation 9.114 of the Theory Guide). You also have the option of selecting **transported** to solve Equation 9.113 in the Theory Guide. Though the **transported** form is more exact, it is more expensive computationally.

Tmax Option

provides various options for determining the maximum limit of the temperature used for the integration of the PDF (see Equation 8.44 in the *Fluent Theory Guide*) to calculate mean rates for turbulent fluctuations of NO_x:

- The default selection is **global-tmax**, which sets the limit as the maximum temperature in the flow field.
- You can select **local-tmax** if you would rather obtain cell-based maximum temperature limits by multiplying the local cell mean temperature by the value entered in **Tmax Factor**.
- You can select **specified-tmax** to set the limit for each cell to be the value entered in **Tmax**.
- If you have selected a user-defined function in the **NOx Rate** drop-down list in the **Formation** tab, then you can select **user-defined** so that the temperature limit is specified by a user-defined function. See the Fluent Customization Manual for details about user-defined functions.

Species

only appears if you have selected **temperature/species** for the **PDF Mode**. Your selection in this drop-down menu determines which species' mass fraction is included in the NO_x formation calculations.

Important:

Note that the species variance will always be calculated using the algebraic form of the transport equation (Equation 9.114 in the Theory Guide).

22.1.1.12. Defining Boundary Conditions for the NOx Model

At flow inlet boundaries, you must specify the **Pollutant NO Mass Fraction**, and if necessary, the **Pollutant HCN Mass Fraction**, **Pollutant NH3 Mass Fraction**, and **Pollutant N2O Mass Fraction**.

Setup → Boundary Conditions

You can retain the default inlet values of zero for these quantities or you can enter nonzero numbers as appropriate for your combustion system.

22.1.2. Solution Strategies

To solve for NO_x products, perform the following steps:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. (optional) If the discrete phase model (DPM) is turned on (by enabling the **Interaction with Continuous Phase**) to run with the NO_x model, then set the **DPM Iteration Interval** to `0` such that no DPM iterations are performed as the NO_x case is being solved.

2. In the Equations Dialog Box (p. 5166) of the Solution Controls Task Page (p. 5163), turn off the solution of all variables except species NO (and HCN, NH_3, or N_2O, based on the model selected).

3. **Solution** → **Controls** → **Equations...**

4. In the **Solution Controls** task page, set a suitable value for the NO (and HCN, NH_3, or N_2O, if appropriate) under-relaxation. A value of 1.0 is suggested, although lower values may be required for certain problems (that is, if convergence cannot be obtained, try a lower under-relaxation value).

5. In the Residual Monitors Dialog Box (p. 5599), decrease the convergence criterion for NO (and HCN, NH_3, or N_2O, if appropriate) to 10^{-6}.

   Solution → **Monitors** → **Residuals** **Edit...**

6. Perform calculations until convergence (that is, until the NO—and HCN, NH_3, or N_2O, if solved—species residuals are below 10^{-6}) to ensure that the NO and HCN or NH_3 concentration fields are no longer evolving.

   Solution → **Run Calculation**

22.1.3. Postprocessing

When you compute NO_x formation, the following additional variables will be available for postprocessing:

- **Mass fraction of Pollutant no**
- **Mass fraction of Pollutant hcn** (appropriate fuel NO_x model only)
- **Mass fraction of Pollutant nh3** (appropriate fuel NO_x model only)
- **Mass fraction of Pollutant n2o** (N_2O-intermediate model only)
- **Mass fraction of Pollutant urea** (SNCR urea injection)
- **Mass fraction of Pollutant hnco** (SNCR urea injection)
- **Mass fraction of Pollutant nco** (SNCR urea injection)
- **Mole fraction of Pollutant no**
- **Mole fraction of Pollutant hcn** (appropriate fuel NO_x model only)
- **Mole fraction of Pollutant nh3** (appropriate fuel NO_x model only)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Mole fraction of Pollutant n2o** (N_2O-intermediate model only)
- **Mole fraction of Pollutant urea** (SNCR urea injection)
- **Mole fraction of Pollutant hnco** (SNCR urea injection)
- **Mole fraction of Pollutant nco** (SNCR urea injection)
- **no Density**
- **hcn Density** (appropriate fuel NO_x model only)
- **nh3 Density** (appropriate fuel NO_x model only)
- **n2o Density** (N_2O-intermediate model only)
- **urea Density** (SNCR urea injection)
- **hnco Density** (SNCR urea injection)
- **nco Density** (SNCR urea injection)
- **Rate of no** (from the individual pathways)
- **Rate of hcn** (appropriate fuel NO_x model only)
- **Rate of nh3** (appropriate fuel NO_x model only)
- **Rate of n2o** (N_2O-intermediate model only)
- **Rate of urea** (SNCR urea injection)
- **Rate of hnco** (SNCR urea injection)
- **Rate of nco** (SNCR urea injection)
- **RMS of Temperature** (appropriate when either **temperature** or **temperature/species** is selected from the **PDF Mode** drop-down list in the **NOx Model** dialog box (**Turbulence Interaction Mode** tab)

These variables are contained in the **NOx...** category of the variable selection drop-down list that appears in postprocessing dialog boxes. Additional NO rates from individual pathways, **Thermal**, **Prompt**, **Fuel**, **N2O Path**, and **SNCR** can be plotted.

22.2. Soot Formation

This section contains information about using the soot formation models in Ansys Fluent. For information about the theory behind the soot models in Ansys Fluent, see Soot Formation in the Theory Guide.

22.2.1. Using the Soot Models

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

22.2.1. Using the Soot Models

When the mass fraction of soot is relatively large (for example, 10%) or if your problem involves the effect of radiation, the soot formation should be computed as part of the main combustion solution and not through postprocessing (as is done for the NO_x model). The procedure for setting up and solving a soot formation model is outlined below. Only the steps that are pertinent to soot modeling are shown here. For information about inputs related to other models that you are using in conjunction with the soot formation model, see the appropriate sections for those models.

1. Set up your combustion problem using Ansys Fluent as usual. Note the following limitations:

   - None of the soot models are compatible with premixed combustion.
   - The one-step and two-step soot formation models are only available for turbulent flows.

2. Enable the desired soot formation model and set the related parameters, as described in this section.

   Setup → Models → Species → Soot → Edit...

3. Define the boundary conditions for soot (and nuclei, if you are not using the one-step model) at flow inlets.

   Setup → Boundary Conditions

4. In the Solution Controls Task Page (p. 5163), set a suitable value for the soot (and nuclei, if you are not using the one-step model) under-relaxation factor(s). The default value is 0.9. If convergence cannot be obtained with this value, try a lower under-relaxation value.

   Solution → Controls

5. Perform calculations until convergence (that is, until the soot / nuclei residual is below 10^{-6}) to ensure that the soot (and nuclei) field is no longer evolving.

   Solution → Run Calculation

   Note that when **Soot-Radiation Interaction** is enabled in the **Soot Model** dialog box, soot equations (soot, nuclei, and tar species, if available) must be solved together with the combustion solution to obtain correct radiation coupling. Therefore, when **Soot-Radiation Interaction** is enabled, the soot equations are not available for pollutant postprocessing, but will be solved together with the combustion solution. However, if other pollutants are to be solved (while soot/radiation interaction is enabled), those pollutant equations can be postprocessed.

6. Review the mass fraction of soot (and nuclei) by generating graphical plots or alphanumeric reports in the usual way.

7. Save a new set of case and data files, if desired.

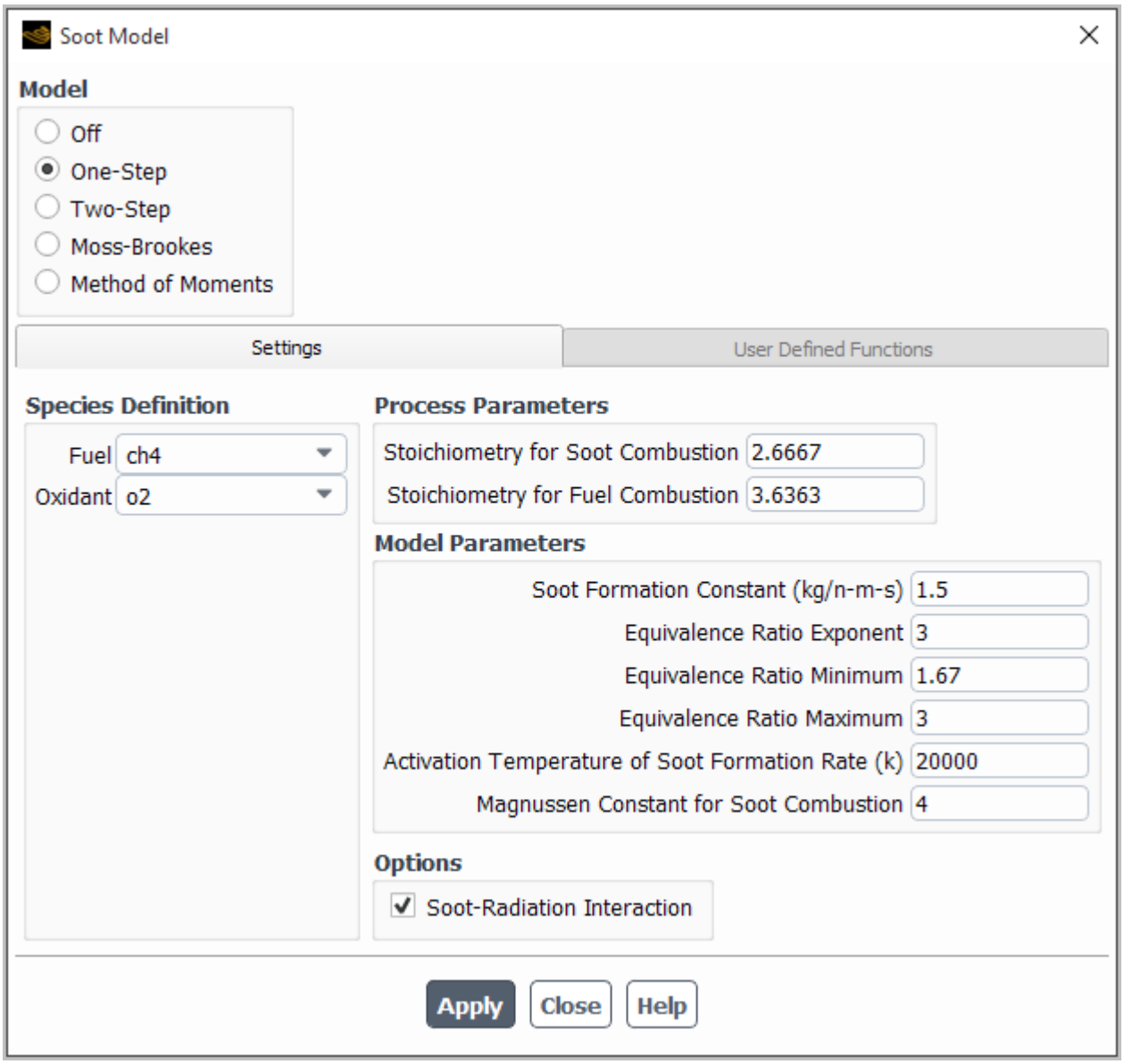
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

22.2.1.1. Setting Up the One-Step Model

You can enable and set up the one-step soot formation model by using the Soot Model Dialog Box (p. 4825) (Figure 22.6: The Soot Model Dialog Box for the One-Step Model (p. 2604)).

Setup → Models → Species → Soot Edit...

Figure 22.6: The Soot Model Dialog Box for the One-Step Model



1. Under **Model**, select **One-Step**. The dialog box will expand to show the appropriate inputs.

2. Define the fuel and oxidizer species. Under **Species Definition**, select the fuel in the **Fuel** drop-down list and the oxidizer in the **Oxidant** drop-down list. If you are using the non-premixed model for the combustion calculation and your fuel stream consists of a mixture of components, you should choose the most appropriate species as the **Fuel** species for the soot formation

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

model. Similarly, the most significant oxidizing component (for example, O_2) should be selected as the **Oxidant**.

3. If you want to include the effects of soot formation on the radiation absorption coefficient, enable **Soot-Radiation Interaction** in the **Options** group box. For more details, see The Effect of Soot on the Absorption Coefficient in the Theory Guide.

4. Define the following **Process Parameters**:

   Stoichiometry for Soot Combustion

   is the mass stoichiometry, ν_{soot}, in Equation 9.120 in the Theory Guide, which computes the soot combustion rate. The default value of 2.6667 assumes that the soot is pure carbon and the oxidizer is O_2.

   Stoichiometry for Fuel Combustion

   is the mass stoichiometry, ν_{fuel}, in Equation 9.120 in the Theory Guide, which computes the soot combustion rate. The default value supplied by Ansys Fluent (3.6363) is for combustion of propane (C_3H_8) by oxygen (O_2).

5. Set the **Modeling Parameters** that are used in Equation 9.117, Equation 9.119, and Equation 9.120 in the Theory Guide:

   Soot Formation Constant

   is the parameter C_s (kg/n-m-s) in Equation 9.117 in the Theory Guide.

   Equivalence Ratio Exponent

   is the exponent r in Equation 9.117 in the Theory Guide.

   Equivalence Ratio Minimum

   and **Equivalence Ratio Maximum** are the minimum and maximum values of the fuel equivalence ratio ϕ in Equation 9.117 in the Theory Guide. This equation will be solved only if **Equivalence Ratio Minimum** $<\phi<$ **Equivalence Ratio Maximum**; if ϕ falls outside of this range, it is assumed that soot does not form.

   Activation Temperature of Soot Formation Rate

   is the term E / R in Equation 9.117 in the Theory Guide.

   Magnussen Constant for Soot Combustion

   is the constant A used in the rate expressions governing the soot combustion rate (Equation 9.119 and Equation 9.120 in the Theory Guide).

Note that the default values for these parameters are for propane fuel [32] (p. 5714), [167] (p. 5722), which are valid for a wide range of hydrocarbon fuels.

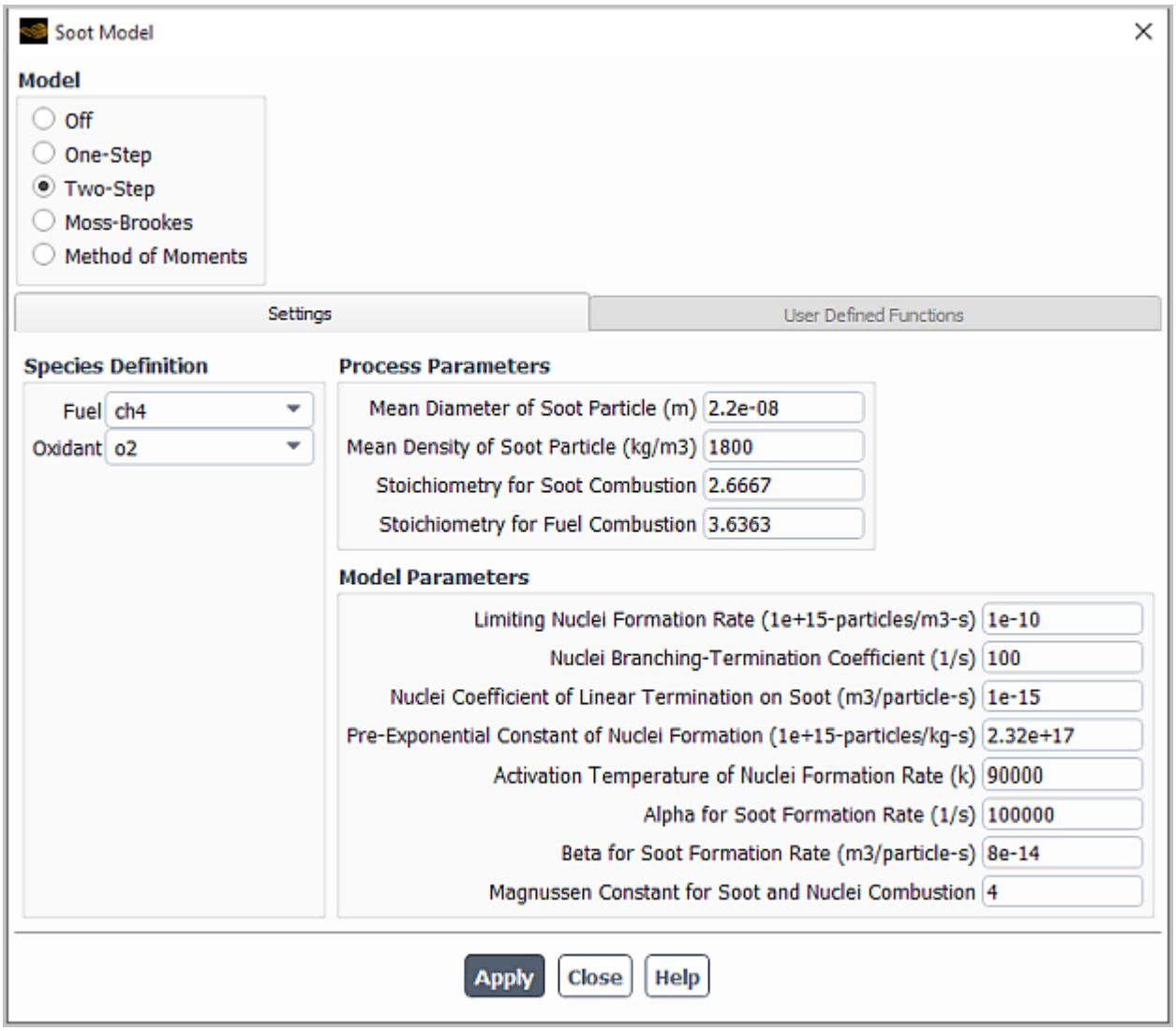
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

22.2.1.2. Setting Up the Two-Step Model

You can enable and set up the two-step soot formation model by using the Soot Model Dialog Box (p. 4825) (Figure 22.7: The Soot Model Dialog Box for the Two-Step Model (p. 2606)).

Setup → Models → Species → Soot → Edit...

Figure 22.7: The Soot Model Dialog Box for the Two-Step Model



1. Under **Model**, select **Two-Step**. The dialog box will expand to show the appropriate inputs.

   Important:

   Note that the two-step Tesner model should only be used when the eddy-dissipation model is used to define the turbulence-chemistry interaction.

2. Under **Species Definition**, select the fuel in the **Fuel** drop-down list and the oxidizer in the **Oxidant** drop-down list. If you are using the non-premixed model for the combustion calculation

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and your fuel stream consists of a mixture of components, you should choose the most appropriate species as the **Fuel** species for the soot formation model. Similarly, the most significant oxidizing component (for example, O_2) should be selected as the **Oxidant**.

3. If you want to include the effects of soot formation on the radiation absorption coefficient, enable **Soot-Radiation Interaction** in the **Options** group box. For more details, see The Effect of Soot on the Absorption Coefficient in the Theory Guide.

4. Define the following **Process Parameters**:

   Mean Diameter of Soot Particle, Mean Density of Soot Particle

   are the assumed average diameter and average density of the soot particles in the combustion system, used to compute the soot particle mass m_p in Equation 9.123 in the Theory Guide. Note that the default values for soot density and diameter are taken from [94] (p. 5718).

   Stoichiometry for Soot Combustion

   is the mass stoichiometry ν_{soot} in Equation 9.120 in the Theory Guide, which computes the soot combustion rate. The default value of 2.6667 assumes that the soot is pure carbon and the oxidizer is O_2.

   Stoichiometry for Fuel Combustion

   is the mass stoichiometry, ν_{fuel}, in Equation 9.120 in the Theory Guide, which computes the soot combustion rate. The default value supplied by Ansys Fluent (3.6363) is for combustion of propane ($C_3 H_8$) by oxygen (O_2).

5. Set the **Modeling Parameters** that are used in Equation 9.119, Equation 9.120, Equation 9.123, Equation 9.125, and Equation 9.126 in the Theory Guide:

   Limiting Nuclei Formation Rate

   is the limiting value of the kinetic nuclei formation rate, η_0 in Equation 9.126 in the Theory Guide. Below this limiting value, the branching and termination term, $(f-g)$ in Equation 9.125 in the Theory Guide, is not included.

   Nuclei Branching-Termination Coefficient

   is the term $(f-g)$ in Equation 9.125 in the Theory Guide.

   Nuclei Coefficient of Linear Termination on Soot

   is the term g_0 in Equation 9.125 in the Theory Guide.

   Pre-Exponential Constant of Nuclei Formation

   is the pre-exponential term a_0 in the kinetic nuclei formation term, Equation 9.126 in the Theory Guide.

   Activation Temperature of Nuclei Formation Rate

   is the term E/R in the kinetic nuclei formation term, Equation 9.126 in the Theory Guide.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Alpha for Soot Formation Rate

is α, the constant in the soot formation rate equation, Equation 9.123 in the Theory Guide.

Beta for Soot Formation Rate

is β, the constant in the soot formation rate equation, Equation 9.123 in the Theory Guide.

Magnussen Constant for Soot and Nuclei Combustion

is the constant A used in the rate expressions governing the soot combustion rate (Equation 9.119 and Equation 9.120 in the Theory Guide).

The default values for the two-step model are the same as in Magnussen and Hjertager [94] (p. 5718) (for acetylene flame), except for α_0, which is assumed to have the original value from Tesner et al. [162] (p. 5722). If your model involves propane fuel rather than acetylene, $\alpha = 3.5\text{x}10^8$ [5] (p. 5713) is recommended. For best results, you should modify both of these parameters using empirically determined inputs for your specific combustion system.

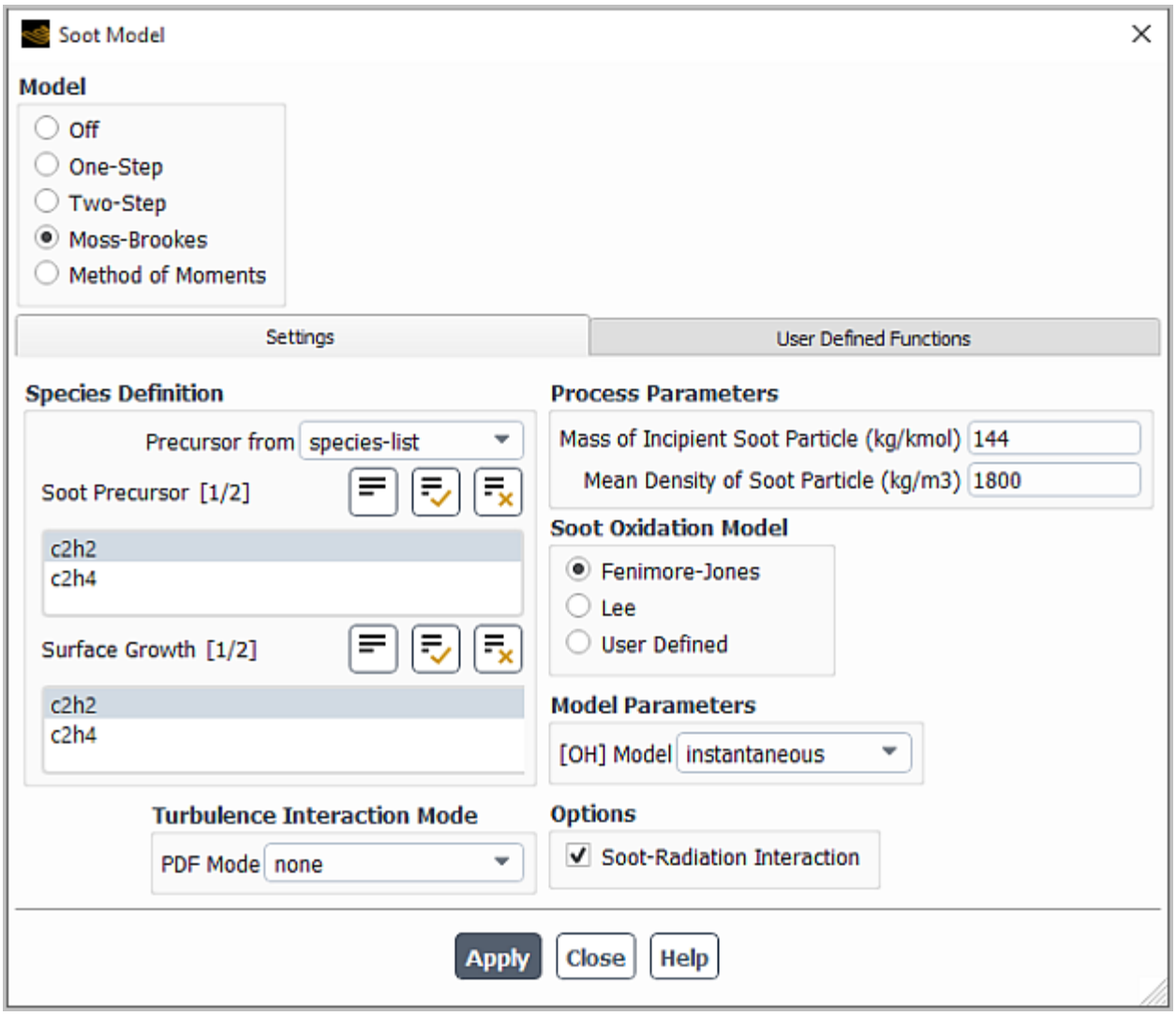
22.2.1.3. Setting Up the Moss-Brookes Model and the Hall Extension

You can enable and set up the Moss-Brookes and Moss-Brookes-Hall soot formation models by using the Soot Model Dialog Box (p. 4825) (Figure 22.8: The Soot Model Dialog Box for the Moss-Brookes Model (p. 2609)).

Setup → Models → Species → Soot → Edit...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 22.8: The Soot Model Dialog Box for the Moss-Brookes Model



1. Under **Model**, select **Moss-Brookes** or **Moss-Brookes-Hall**. The dialog box will expand to show the appropriate inputs. Note the following about these models:

   - The **Moss-Brookes** model was originally proposed for soot prediction in methane flames. However, it can be equally applicable to higher hydrocarbon species by appropriately modifying the soot precursor and participating surface growth species.

   - The **Moss-Brookes-Hall** model is applicable for higher hydrocarbon fuels (for example, kerosene) and will only be available when C_2H_2, C_6H_6, C_6H_5, and H_2 are present in the gas phase species list.

2. Define the precursor species in the **Species Definition** group box. When suitable precursor species are present in the species list, you can select **species-list** from the **Precursor from** drop-down list, and then select the **Soot Precursor** species and the **Surface Growth** species from the selection lists. Note that for the **Moss-Brookes** model, you can select acetylene (**c2h2**), ethylene (**c6h4**), and/or benzene (**c6h6**) for the **Soot Precursor**; if neither are present or if you would like to specify a different precursor correlation, then curve fitting will be used to determine the precursor and surface growth species mass fractions (see Species Definition for the Moss-

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Brookes Model with a User-Defined Precursor Correlation (p. 2613) for further details regarding curve fitting).

3. Specify how turbulent fluctuations will be accounted for in the soot formation calculations by defining the turbulence parameters in the **Turbulence Interaction Mode** group box. Select one of the options in the **PDF Mode** drop-down list:

   - Select **none** to ignore turbulence and use laminar soot rate calculations.
   - Select **temperature** to take into account fluctuations of temperature.
   - Select **temperature/species** to take into account fluctuations of temperature and mass fraction of the species selected in the **Species** drop-down list (which appears when you select this option).
   - (non-premixed and partially premixed combustion calculations only) Select **mixture fraction** to take into account fluctuation in the mixture fraction(s). This is the recommended approach, as it generally yields the best results and accuracy.

   Important:

   When modeling the formation of other pollutants along with soot, you should compare the selections made in the **PDF Mode** drop-down lists in the **Turbulence Interaction Mode** tab of the NOx Model Dialog Box (p. 4816) and the **Turbulence Interaction Mode** group box of the **Soot Model** dialog box. If **mixture fraction** is selected in any of these dialog boxes, then it must be selected in all of the others as well.

   The **mixture fraction** option is available only if you are using either the non-premixed or partially premixed combustion model to model the reacting system. If you use the **mixture fraction** option, the instantaneous temperatures and species concentrations are taken from the PDF look-up table as a function of mixture fraction and enthalpy and the instantaneous soot production rates are calculated at each cell. The PDF used for convoluting the instantaneous soot rates is the same as the one used to compute the mean flow-field properties. For example, for single-mixture fraction models the beta PDF is used, and for two-mixture fraction models, the beta or the double delta PDF can be used. The PDF in terms of mixture fraction is calculated from the values of mean mixture fraction and variance at each cell, and the instantaneous soot rates are convoluted with the mixture fraction PDF to yield the mean rates in turbulent flow.

   Note:

   When **mixture fraction** is selected as **PDF mode**, Ansys Fluent calculates and tabulates the soot rates apriori for faster computations of soot rates. The soot rates are tabulated at the first soot iteration and a message

   ```
   Performing Pollutant PDF integrations...
   ```

   is displayed in the Fluent console.

4. If you selected **temperature** or **temperature/species** for the **PDF Mode**, you should define the following parameters in the **Turbulence Interaction Mode** group box:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

PDF Type

allows you to specify the shape of the PDF, which is then integrated to obtain mean rates for the temperature and (if you selected **temperature/species** for the **PDF Mode**) the species. If you select **beta**, the PDF will be modeled using Equation 9.108 in the Theory Guide. If you select **gaussian**, the PDF will be modeled using Equation 9.111 in the Theory Guide.

PDF Points

allows you to specify the number of points used to integrate the beta or Gaussian function in Equation 9.105 or Equation 9.106 in the Theory Guide on a histogram basis. The default value of 10 is a minimum value. It is recommended that you run the calculation with this minimum value to convergence, and then keep increasing the value (for example, 20–25) until the pollutants of interest stop changing. Increasing this value may improve accuracy, but will also increase the computation time.

Temperature Variance

allows you to specify the form of transport equation that is solved to calculate the temperature variance. The default selection is **algebraic**, which is an approximate form of the transport equation (see Equation 9.114 of the Theory Guide). You have the option of selecting **transported** to instead solve Equation 9.113 in the Theory Guide. Though the **transported** form is more exact, it is also more expensive computationally.

Tmax Option

provides various options for determining the maximum limit of the temperature used for the integration of the PDF (see Equation 8.44 in the *Fluent Theory Guide*) to calculate mean rates for turbulent fluctuations of soot. The default selection is **global-tmax**, which sets the limit as the maximum temperature in the flow field. You can select **local-tmax** if you would rather obtain cell-based maximum temperature limits by multiplying the local cell mean temperature by the value entered in **Tmax Factor**. You can select **specified-tmax** to set the limit for each cell to be the value entered in **Tmax**. Finally, if you have compiled a user-defined function for the soot rate and loaded the library into Ansys Fluent, then you can select **user-defined** so that the limit is specified by a user-defined function.

Species

only appears if you have selected **temperature/species** for the **PDF Mode**. Your selection in this drop-down menu determines which species' mass fraction is included in the soot formation calculations.

Important:

Note that the species variance will always be calculated using the algebraic form of the transport equation (Equation 9.114 in the Theory Guide).

---

5. Under **Process Parameters**, enter information about the mass and mean density of the soot particles:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mass of Incipient Soot Particles

is M_P in Equation 9.131 and Equation 9.133 in the Theory Guide. Note that this value was assumed to be 144 kg/kmol (12 carbon atoms) in the work of Brookes and Moss, whereas the Hall extension model assumed it to be 1200 kg/kmol (100 carbon atoms).

Mean Density of Soot Particle

is ρ_{soot} in Equation 9.130 and Equation 9.133 in the Theory Guide. Note that this value was assumed to be 1800 kg/m^3 in the work of Brookes and Moss [28] (p. 5714), whereas Hall et al. [60] (p. 5716) assumed it to be 2000 kg/m^3.

6. Select the **Soot Oxidation Model**. Your choices include the **Fenimore-Jones** model, as originally used in Brookes and Moss' work, or the **Lee** extended model. The **Lee** model will model soot oxidation due to hydroxyl radicals as in the **Fenimore-Jones** model, as well as the oxidation due to molecular oxygen.

7. Set the **Modeling Parameters**:

   [OH] Model

   allows you to specify the method by which the OH radical concentration is calculated. The recommended selection from the drop-down list is **instantaneous**, although this option is only available when OH is available in the species list and is calculated by the combustion model. The other option is the **partial-equilibrium** model, which necessitates the availability of O radical concentration within the field.

   [O] Model

   must be defined when you have selected **partial-equilibrium** for the **[OH] Model**, and specifies the method by which the O radical concentration is calculated. The options include **equilibrium**, **partial-equilibrium**, and **instantaneous**.

8. If you want to include the effects of soot formation on the radiation absorption coefficient, enable **Soot-Radiation Interaction** in the **Options** group box. For more details, see The Effect of Soot on the Absorption Coefficient in the Theory Guide.

Note that in Ansys Fluent, the oxidation rate scaling parameter (C_{oxid} in Equation 9.131 in the Theory Guide) is set to unity. If you would like to change the value of this parameter, you can use the `define/models/soot-parameters` text command. A lower value will reduce the amount of soot oxidation.

22.2.1.3.1. Specifying a User-Defined Function for the Soot Oxidation Rate

You can choose to specify a user-defined function for the soot oxidation rate. The compiled UDF hook is available from the **User Defined Oxidation Rate** drop-down list of the **Soot Model** dialog box. See `DEFINE_SOOT_OXIDATION_RATE` in the *Fluent Customization Manual* for details about user-defined functions.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

22.2.1.3.2. Specifying a User-Defined Function for the Soot Precursor Concentration

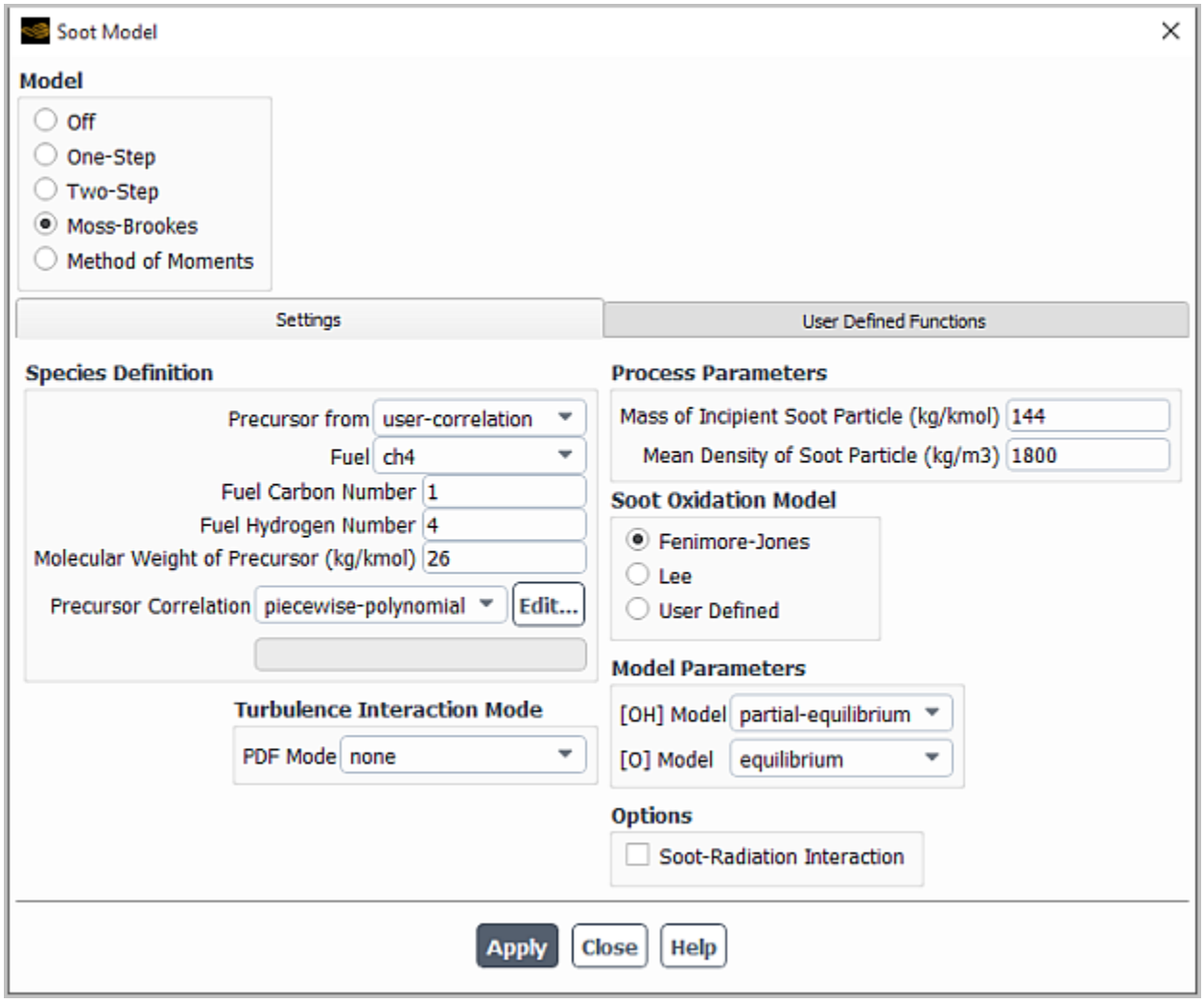
You can choose to specify a user-defined function for the soot oxidation rate. The compiled UDF hook is available from the **User Defined Precursor** drop-down list in the **Species Definition** group box of the **Soot Model** dialog box. See `DEFINE_SOOT_PRECURSOR` in the *Fluent Customization Manual* for details about user-defined functions.

22.2.1.3.3. Species Definition for the Moss-Brookes Model with a User-Defined Precursor Correlation

Ansys Fluent accepts the following as possible precursor species for the Moss-Brookes model: C_2H_2, C_6H_6, and C_2H_4. If none of these species are present in the species list (as is often the case when using the eddy-dissipation model) or if you would prefer to specify a different precursor correlation, your setup for the Moss-Brookes model will be different than noted previously. Under such circumstances, you should select **user-correlation** from the **Precursor from** drop-down list in the **Species Definition** group box (note that this is only option possible when the appropriate species are not present). The Soot Model Dialog Box (p. 4825) will then be as shown in Figure 22.9: The Soot Model Dialog Box for the Moss-Brookes Model with a User-Defined Precursor Correlation (p. 2614). The parameters you set in the **Species Definition** group box allow Ansys Fluent to calculate a mixture fraction based on the mass fractions of the oxidant and the carbon/hydrogen contributed by a designated fuel species. The precursor species mass fraction will then be computed as a function (which you will also define) of this mixture fraction.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 22.9: The Soot Model Dialog Box for the Moss-Brookes Model with a User-Defined Precursor Correlation



1. In the **Species Definition** group box, select a **Fuel** species and enter the related **Fuel Carbon Number** and **Fuel Hydrogen Number** for use in the mixture fraction calculation.

2. Enter the **Molecular Weight of Precursor** (the default value is for acetylene).

3. Make a selection in the **Precursor Correlation** drop-down list to indicate how the precursor mass fraction will be related to the mixture fraction. A **piecewise-polynomial** profile is defined by default.

   Important:

   Note that the default values for the **piecewise-polynomial** profile are only valid for a methane diffusion flame simulation, in which both the air and fuel initial temperatures are set to 290 K, and acetylene is assumed as the soot precursor.

   If you decide not to use the default values for **Precursor Correlation**, define the correlation between the precursor mass fraction and the mixture fraction. This correlation should be based on a laminar flamelet profile that you have generated. If possible, you should generate

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the profile using the **Species Model** dialog box (as described in Setting Up the Steady and Unsteady Diffusion Flamelet Models (p. 2481)) with **Steady Flamelet** selected in the **Chemistry** tab. If there is no mechanism, you can generate the profile using the equilibrium chemistry model (see Setting Up the Equilibrium Chemistry Model (p. 2476) for details). You may also use a third-party software package of your choice. You should then apply a curve-fitting technique to your generated profile, to obtain either a constant value or a piecewise-polynomial function.

In a piecewise-polynomial function, the laminar flamelet profile is divided into a number of mixture fraction ranges. In each range, the precursor species mass fraction Y_{prec} is defined using the following equation:

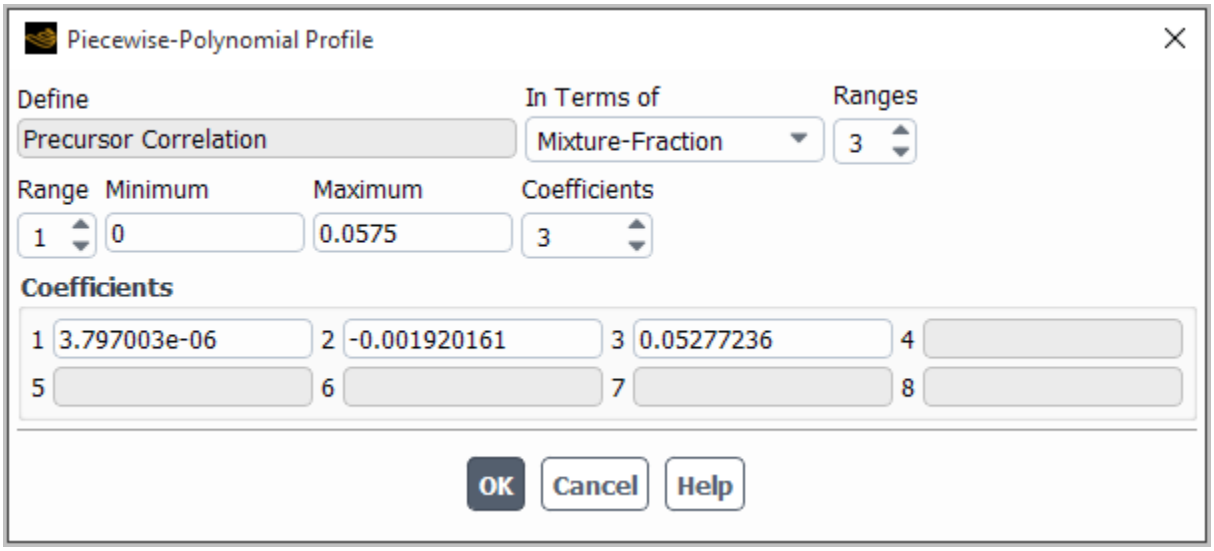
$$Y_{prec} = \sum_{i=1}^{i=NC} C_i f^{i-1} \tag{22.8}$$

where NC is the number of coefficients C, and f is the mixture fraction. The following piecewise-polynomial function corresponds to the default settings in Ansys Fluent:

$$Y_{prec} = \begin{cases} \text{for } 0 \le f < 0.0575: \\ 3.797003\times10^{-6} - 1.920161\times10^{-3} f \\ +5.277237\times10^{-2} f^2 \\ \text{for } 0.0575 \le f < 0.128: \\ 1.051312 - 71.34743 f + 1964.038 f^2 - 28182.59 f^3 \\ +223543.4 f^4 - 932192 f^5 + 1599627 f^6 \\ \text{for } 0.128 \le f < 1: \\ 7.988928\times10^{-3} - 8.440912\times10^{-3} f \\ +4.273195\times10^{-4} f^2 \end{cases} \tag{22.9}$$

4. To define a piecewise-polynomial profile to relate the precursor mass fraction to the mixture fraction, select **piecewise-polynomial** from the **Precursor Correlation** drop-down list and click the **Edit...** button.

Figure 22.10: The Piecewise-Polynomial Profile Dialog Box



In the **Piecewise-Polynomial Profile** Piecewise-Polynomial Profile Dialog Box (p. 4909), set the following controls:

a. Enter the number of **Ranges**. For the example shown in Equation 22.9 (p. 2615), three ranges of mixture fraction are defined, which is the maximum allowed.

b. For the first range (**Range** = 1), enter the **Minimum** and **Maximum** mixture fraction values, and the number of **Coefficients**. (Up to eight coefficients are available.) The number of coefficients defines the order of the polynomial. An input of 1 defines a polynomial of order 0, and the mass fraction will be constant and equal to the single coefficient. An input of 2 defines a polynomial of order 1, and the mass fraction will vary linearly with mixture fraction, and so on.

c. Define the values for the coefficients in the **Coefficients** group box. The dialog box in Figure 22.10: The Piecewise-Polynomial Profile Dialog Box (p. 2615) shows the inputs for the first range of Equation 22.9 (p. 2615).

d. Increase the value of **Range** and enter the **Minimum** and **Maximum** mixture fractions, number of **Coefficients**, and the values for the **Coefficients** for the next range. Repeat if there is a third range.

Important:

Note when defining the ranges, you must start with the lowest mixture fraction range, and then proceed in order to the highest range. The solver will *not* sort them for you.

---

5. To define a constant profile to relate the precursor mass fraction to the mixture fraction, select **constant** from the **Precursor Correlation** drop-down list and enter a value in the accompanying text entry box.

22.2.1.4. Setting Up the Method of Moments Soot Model

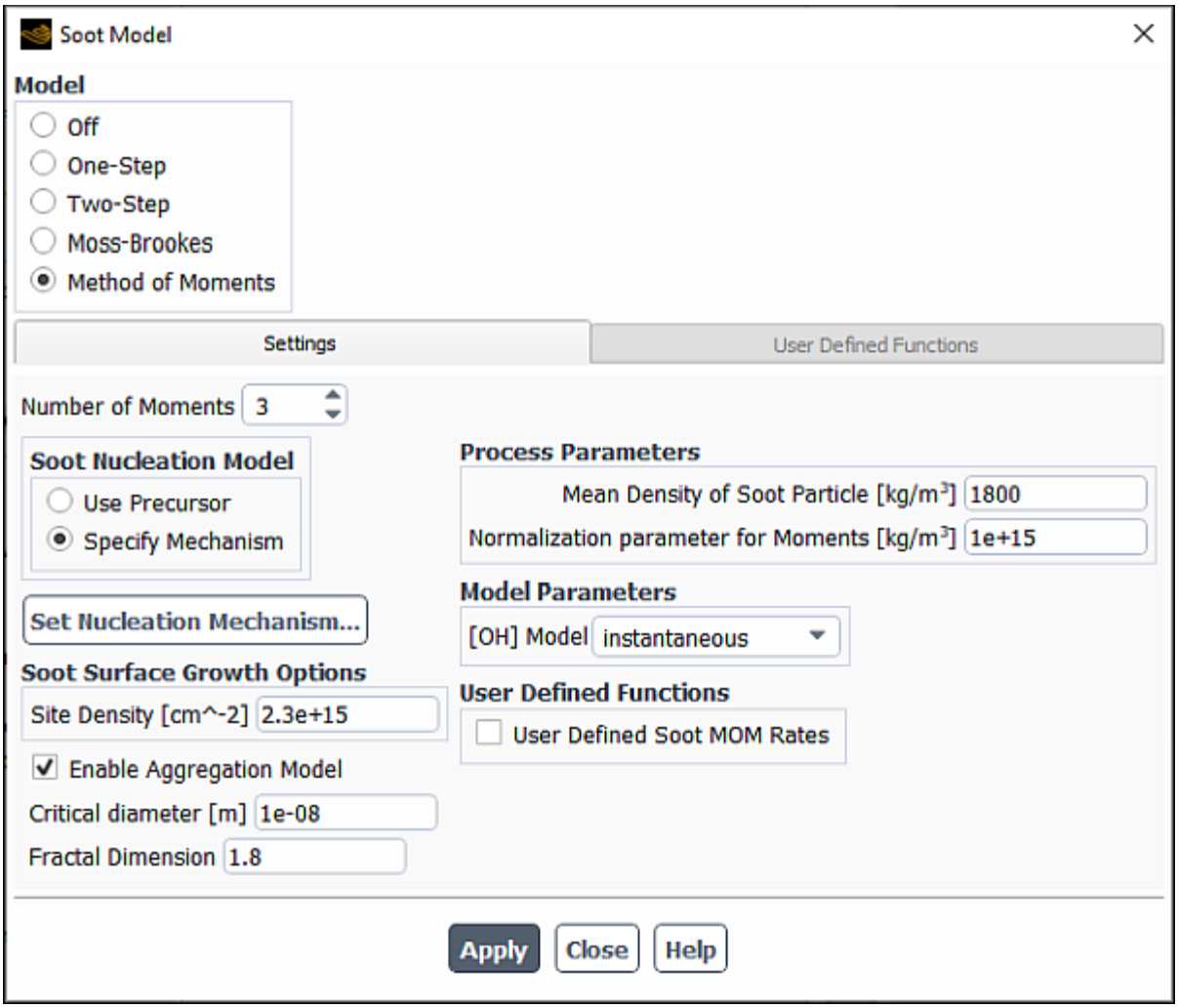
You can enable and set up the Method of Moments model by using the **Soot Model** dialog box (Figure 22.11: The Soot Model Dialog Box for the Method of Moments Model (p. 2617)).

Setup → Models → Species → Soot → Edit...

1. In the **Soot Model** dialog box under **Model**, select **Method of Moments**. The dialog box will expand to show the appropriate inputs.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 22.11: The Soot Model Dialog Box for the Method of Moments Model



2. Specify **Number Of Moments**.

   Ansys Fluent will solve an equal number of moment transport equations. The default value of 3 moments works reasonably well for a wide range of applications. However, you can specify up to 6 moments to achieve higher accuracy.

3. From the **Soot Nucleation Model** group box, select one of the following options:

   - **Use Precursor** (default): Enables you to define the precursor species for your soot prediction simulation.
   - **Specify Mechanism**: Enables you to provide the nucleation rate and specify nucleation as a reaction between soot precursors. Using this option, you can achieve more accurate results where the nucleation rate data is available.

4. (if **Use Precursor** is selected for the soot nucleation model) Under the **Species Definition** group box, define precursor species.

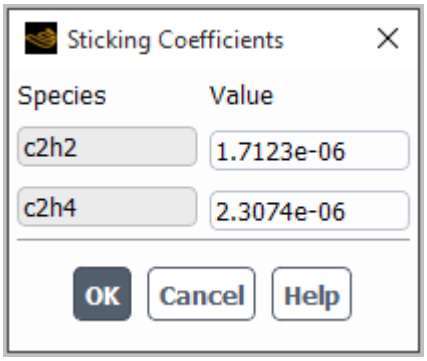
   a. In the **Soot Precursor** group list (which is populated with all precursors available in your analysis), select one or more precursors.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The soot precursor species must be a hydrocarbon species. Ansys Fluent uses kinetic theory to calculate the nucleation rates considering coagulation between precursor species.

b. Click **Sticking Coefficients...** to specify the sticking coefficient for each precursor in the **Sticking Coefficients** dialog box that will open.

Figure 22.12: Sticking Coefficients for Soot Precursors



You must specify the sticking coefficient for each precursor because the nucleation rates calculated based on kinetic theory of collision of two soot precursors are generally very large, causing numerical errors.

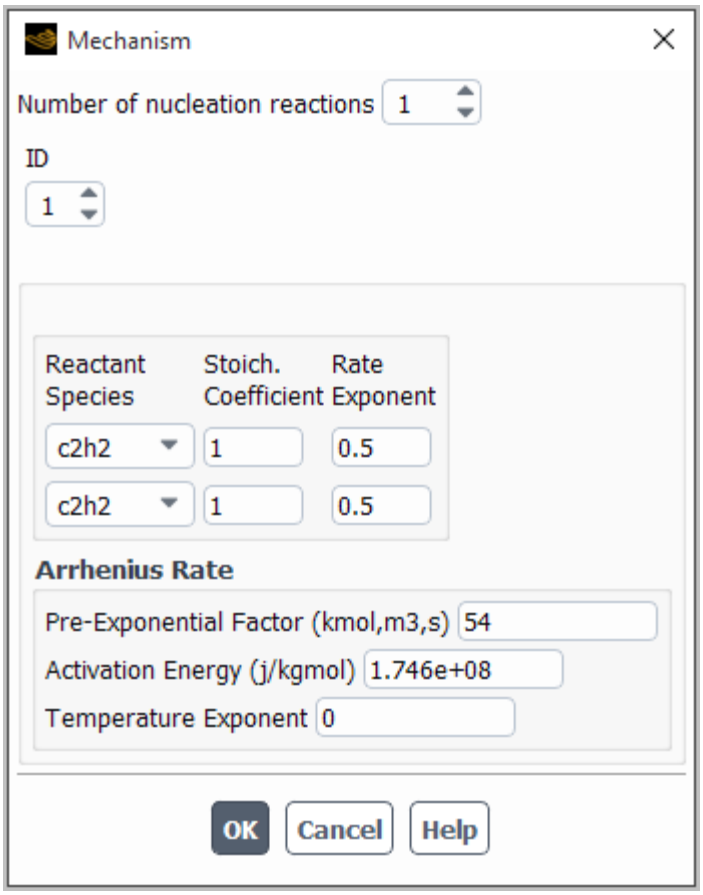
Important:

The default value for a sticking coefficient is calculated based on precursor molecular weight. In general, you can use smaller values for sticking coefficients for smaller precursors. For guidelines on specifying the sticking coefficients for precursor species, see Nucleation in the *Fluent Theory Guide*.

5. (if **Specify Mechanism** is selected for the soot nucleation model) Click **Set Nucleation Mechanism...** and enter the reaction rates inputs in the **Mechanism** dialog box that will open.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 22.13: Settings for the Nucleation Mechanism



a. Specify **Number of nucleation reactions**.

   The nucleation reactions can be single or multi-step.

b. For each reaction, specify the following:

   - Reactant species and corresponding stoichiometric coefficient and rate exponent
   - The **Arrhenius Rate** parameters (**Pre-Exponential Factor**, **Activation Energy**, and **Temperature Exponent**)

Note:

The inputs in the **Mechanism** dialog box are required only for reactant species as the main product of specified nucleation reaction is always assumed to be a soot nuclei, and all other products are not considered in calculations of soot nucleation rates.

---

6. If you want to account for soot aggregate formation, select **Enable Aggregation Model**

   In addition to the moment transport equations, Ansys Fluent will solve an equal number of the particle moment transport equations.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Once you have enabled the aggregation model, you can specify the following parameters:

Critical diameter: A value of the soot particle diameter above which the coagulation regime is switched from coalescent coagulation to aggregate coagulation.

Fractal Dimension: A parameter that describes the fractal structure of the aggregate. Values between 1.7 and 2.0 work reasonably good for the aggregate coagulation.

Note:

When the **Enable Aggregation Model** option is enabled, the Method of Moments model tends to over-predict the production of soot. This is due, in part, to the linear interpolation of the particle moments. This aspect of the over-prediction can be overcome by employing quadratic interpolation of the particle moments. Another reason for the over-prediction of soot is due to how Ansys Fluent currently ignores the depletion of acetylene during the soot calculations. Acetylene is consumed during the surface growth of soot, and since the gas field stays frozen and has no knowledge of the soot process, thereby producing a higher surface growth of soot, and eventually higher soot yields.

For information about the aggregation model, see Soot Aggregation in the *Fluent Theory Guide*.

7. Under **Soot Surface Growth Options**, specify **Soot Density**.

   The growth of soot due to surface reactions and depletion due to oxidation reactions are modeled using HACA mechanism (see Surface Growth and Oxidation in the separate Theory Guide). For reaction rates and reaction steps, Ansys Fluent uses default values and requires no user inputs except for the soot site density.

   Note:

   You can specify and/or edit the reaction rates of surface reactions using the `define/models/soot-parameters` text command.

   Important:

   The soot site density is always provided in units of number of sites per cm^2 irrespective of the unit system selected in Ansys Fluent.

   ---

8. Under **Process Parameters**, specify **Mean Density of Soot Particle** and **Normalization Parameter for Moments**.

   The normalization parameter is used to minimize numerical errors when solving the normalized soot moment transport equations. The default value for this parameter is sufficient, and you may not need to change it.

9. Specify the **Modeling Parameters**:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **[OH] Model**: Specifies the method by which the [OH] radical concentration is calculated. For **OH Model**, the following options are available:

  – `instantaneous` (recommended where possible)

    This option is only available when OH is present in the species list and participate in the combustion simulation.

  – `partial-equilibrium`

    This option necessitates the availability of [O] atom concentration within the field.

- **[O] Model** (available only if `instantaneous` is not selected for **OH Model**): Specifies the method by which the [O] radical concentration is calculated. The following options are available:

  – `equilibrium`

  – `partial-equilibrium`

  – `instantaneous`

22.2.1.5. Defining Boundary Conditions for the Soot Model

At flow inlet boundaries, you must specify the **Soot Mass Fraction** and (when not using the one-step model) the **Nuclei** mass concentration. These correspond to Y_{soot} in Equation 9.115 and Equation 9.129 and b^*_{nuc} in Equation 9.121 and Equation 9.129 in the Theory Guide, respectively.

Setup → Boundary Conditions

You can retain the default inlet values of zero for both quantities or you can enter nonzero numbers as appropriate for your combustion system.

22.2.1.6. Reporting Soot Quantities

Ansys Fluent provides additional reporting options when your model includes soot formation. You can generate graphical plots or alphanumeric reports of the following items:

- **Mass fraction of Soot**
- **Mole fraction of Soot**
- **Soot Density**
- **Soot Volume fraction**
- **Rate of Soot**
- **Normalized Concentration of Nuclei** (unavailable for one-step model)
- **Rate of Nuclei** (unavailable for one-step model)
- **Rate of Soot Mass Nucleation** (Moss-Brookes and Moss-Brookes-Hall models only)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Rate of Surface Growth** (Moss-Brookes and Moss-Brookes-Hall models only)
- **Rate of Oxidation** (Moss-Brookes and Moss-Brookes-Hall models only)
- **Rate of Nucleation** (Moss-Brookes and Moss-Brookes-Hall models only)
- **Rate of Coagulation** (Moss-Brookes and Moss-Brookes-Hall models only)
- **Soot Surface Area** (Method of Moments model only)
- **Soot Mean Diameter** (Method of Moments model only)
- **Normalized soot moments** (Method of Moments model only)
- **Normalized Soot Aggregation Moments** (Method of Moments model with Aggregation only)
- **Average Number of Particles in Soot Aggregate** (Method of Moments model with Aggregation only)
- **Primary Particle Diameter** (Method of Moments model with Aggregation only)

These parameters are contained in the **Soot...** category of the variable selection drop-down list that appears in postprocessing dialog boxes.

22.3. Using the Decoupled Detailed Chemistry Model

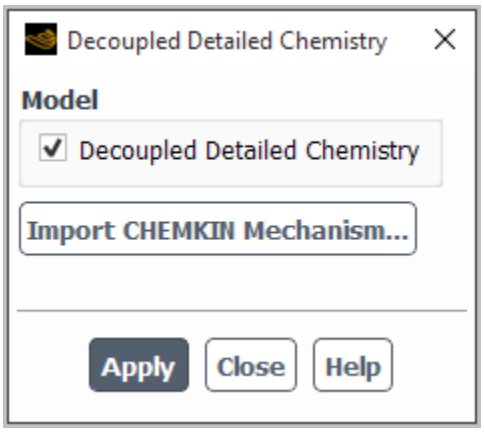
The decoupled detailed chemistry pollutant model is used to postprocess slowly-forming, trace pollutant species on a steady-state flow field using detailed chemical kinetic mechanisms. For theoretical information, refer to Decoupled Detailed Chemistry Model. The recommended procedure is as follows:

1. Calculate your steady combustion problem using the Species Transport, Non-Premixed, Partially-Premixed, or PDF Transport models in Ansys Fluent. It is recommended that you save your case and data file, as it may be difficult to revert to the original settings after you set up the decoupled detailed chemistry pollutant model.
2. Enable the **Decoupled Detailed Chemistry** option in the **Decoupled Detailed Chemistry** dialog box. To access this dialog box, make sure that either the non-premixed or partially premixed model is selected, or that reactions are enabled if the species transport or PDF transport models are used.

   Setup → **Models** → **Species** → **Decoupled Detailed Chemistry** **Edit...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 22.14: The Decoupled Detailed Chemistry Dialog Box



3. Click **Import CHEMKIN Mechanism...** to import your detailed chemistry mechanism in CHEMKIN format. The **Import CHEMKIN Format Mechanism** dialog box (described in Importing a Volumetric Kinetic Mechanism in CHEMKIN Format (p. 2368)) will open, where you will browse to the file. After the CHEMKIN mechanism is imported, the **Decoupled Detailed Chemistry** dialog box will expand to include the species.

   The species in the initial detailed chemical kinetic mechanism used to obtain the steady-state solution appear in the **Original Species** list. The species in the detailed chemistry mechanism you have imported appear in the **Pollutant Species** list. Note that if the species is present in both mechanisms, it will be listed only under **Original Species** and will not be available as a pollutant species.

4. Select the **Pollutant Species** and click **Apply**. The pollutant species are typically slowly forming (far from chemical equilibrium), and occur at miniscule mass fractions. Ansys Fluent will create a mixture material called **pollutant-mixture** and enable the **Species Transport** model with the **Stiff Chemistry Solver**. All species in the imported mechanism that were not in the original combustion model and are not pollutant species will be calculated by chemical equilibrium at the cell temperature. The solution of all transport equations, except the selected pollutant species, are disabled.

   Note:

   Species that are not identified as pollutants and do not participate in the reactions amongst the pollutant species are eliminated. Similarly, reactions in the imported CHEMKIN mechanism, which do not include any pollutant species are also eliminated.

5. Click **Integration Parameters...** to open the **Integration Parameters** dialog box. Set the **ISAT Parameters**, such as the **ISAT Error Tolerance** and **Max. Storage**. An **ISAT Error Tolerance** of 1e-5 is recommended, and **Max. Storage** should be set to a large fraction of the free RAM memory available. To learn more about setting integration parameters, refer to Using ISAT (p. 2457).

6. Define the boundary conditions for all pollutants at flow inlets (these are usually zero).

   Setup → Boundary Conditions

7. Iterate until convergence.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution → Run Calculation

8. Review the mass fractions of pollutants by generating graphical plots or alphanumeric reports in the usual way.

9. Save a new set of case and data files, if desired.

File → Write → Case & Data...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 23: Predicting Aerodynamically Generated Noise

This chapter provides an overview of Ansys Fluent's approaches to computing aerodynamically generated sound, the model setup, and the procedure for computing sound. Details are also provided about using sponge layers to eliminate non-physical pressure wave reflections from boundary zones with the wave equation model or the pressure-based solver.

23.1. Overview

23.2. Using the Ffowcs Williams and Hawkings Acoustics Model

23.3. Using the Acoustics Wave Equation Model

23.4. Using the Broadband Noise Source Models

23.5. Sponge Layers

For more information about the underlying theory behind aerodynamically generated sound, see Aerodynamically Generated Noise in the Theory Guide.

23.1. Overview

Considering the breadth of the discipline and the challenges encountered in aerodynamically generated noise, it is not surprising that a number of computational approaches have been proposed over the years whose sophistication, applicability, and cost widely vary.

Ansys Fluent offers three approaches to computing aerodynamically generated noise: a direct approach, a hybrid approach, and an approach that utilizes broadband noise source models. Within the hybrid approach, Fluent offers the two methods, which are the integral method by Ffowcs Williams and Hawkings, and a differential sound propagation method based on a finite volume solver for a wave equation.

For additional information, see the following sections:

23.1.1. Direct Method

23.1.2. Integral Method by Ffowcs Williams and Hawkings

23.1.3. Method Based on Wave Equation

23.1.4. Broadband Noise Source Models

23.1.1. Direct Method

In this method, both generation and propagation of sound waves are directly computed by solving the appropriate fluid dynamics equations. Prediction of sound waves always requires time-accurate solutions to the governing equations. Furthermore, in most practical applications of the direct method, one has to employ governing equations that are capable of modeling viscous and turbulence

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

effects, such as unsteady Navier-Stokes equations (that is, DNS), RANS equations, and filtered equations used in LES and hybrid RANS-LES models.

The direct method is therefore computationally difficult and expensive inasmuch as it requires highly accurate numerics, very fine computational meshes all the way to receivers, and acoustically nonreflecting boundary conditions. The computational cost becomes prohibitive when sound is to be predicted in the far field (for example, hundreds of chord-lengths in the case of an airfoil). The direct method becomes feasible when receivers are in the near field (for example, cabin noise). In many such situations involving near-field sound, sounds (or pseudo-sounds for that matter) are predominantly due to local hydrodynamic pressure, which can be predicted with a reasonable cost and accuracy.

Since sound propagation is directly resolved in this method, you would normally solve the compressible form of the governing equations (for example, compressible RANS equations, compressible form of filtered equations for LES). Only in situations where the flow is low subsonic and the receivers in the near field sense primarily local hydrodynamic pressure fluctuations (that is, pseudo sound) can incompressible flow formulations be used. But this incompressible treatment will also not allow you to simulate resonance and feedback phenomena.

23.1.2. Integral Method by Ffowcs Williams and Hawkings

For predictions of mid- to far-field noise, the methods based on Li's acoustic analogy [87] (p. 5718) offer viable alternatives to the direct method. In this approach, the near-field flow obtained from appropriate governing equations such as unsteady RANS equations, DES, SAS, SDES, SBES, or LES are used to predict the sound with the aid of analytically derived integral solutions to wave equations. The acoustic analogy essentially decouples the propagation of sound from its generation, allowing one to separate the flow solution process from the acoustics analysis.

Ansys Fluent offers a method based on the Ffowcs Williams and Hawkings (FW-H) formulation [44] (p. 5715). The FW-H formulation adopts the most general form of Lighthill's acoustic analogy, and is capable of predicting sound generated by equivalent acoustic sources such as monopoles, dipoles, and quadrupoles. Ansys Fluent adopts a time-domain integral formulation wherein time histories of sound pressure, or acoustic signals, at prescribed receiver locations are directly computed by evaluating a few surface integrals.

Time-accurate solutions of the flow-field variables, such as pressure, velocity components, and density on source (emission) surfaces, are required to evaluate the surface integrals. Time-accurate solutions can be obtained from unsteady Reynolds-averaged Navier-Stokes (URANS) equations, large eddy simulation (LES), or hybrid RANS-LES models as appropriate for the flow at hand and the features that you want to capture (for example, vortex shedding). The source surfaces can be placed not only on impermeable walls, but also on interior (permeable) surfaces, which enables you to account for the contributions from the quadrupoles enclosed by the source surfaces. Both broadband and tonal noise can be predicted depending on the nature of the flow (noise source) being considered, turbulence model employed, and the time scale of the flow resolved in the flow calculation.

The FW-H acoustics model in Ansys Fluent allows you to select multiple source surfaces and receivers. It also permits you either to save the source data for a future use, or to carry out an "on the fly" acoustic calculation simultaneously as the transient flow calculation proceeds, or both. Sound pressure signals therefore obtained can be processed using the fast Fourier transform (FFT) and associated postprocessing capabilities to compute and plot such acoustic quantities as the overall sound pressure level (SPL) and power spectra.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

One important limitation of Ansys Fluent's FW-H model is that it is applicable only to predicting the propagation of sound toward *free space*. Therefore, while the model can be legitimately used to predict far-field noise due to external aerodynamic flows, such as the flows around ground vehicles and aircraft, it cannot be used for predicting the noise propagation inside ducts or wall-enclosed space.

23.1.3. Method Based on Wave Equation

In this hybrid simulation method, which is intended to simulate aeroacoustics of low Mach number flows, the incompressible flow model is used for the calculation of sound sources, while the differential wave equation is used to calculate the propagation of sound generated by these sources. The acoustics wave equation implemented in Ansys Fluent is derived from the acoustics perturbation equations by Ewert and Schroeder [178] (p. 5723) under the assumption of constant density flow. The main advantages of this model are:

• Extended applicability compared to the Ffowcs Williams and Hawking integral solver, which can be used only to model the sound propagation in open space (see Integral Method by Ffowcs Williams and Hawkings (p. 2626)).

• Simple and convenient workflow of hybrid aeroacoustics simulations, without any data exchange between different software components via disk files, and without the interpolation of sound sources on a different mesh. A transient co-simulation of fluid flow and noise propagation using the same domain and mesh has been chosen for the current implementation.

23.1.4. Broadband Noise Source Models

In many practical applications involving turbulent flows, noise does not have any distinct tones, and the sound energy is continuously distributed over a broad range of frequencies. In those situations involving *broadband noise*, statistical turbulence quantities readily computable from RANS equations can be utilized, in conjunction with semi-empirical correlations and Lighthill's acoustic analogy, to shed some light on the source of broadband noise.

Ansys Fluent offers several such *source models* that enable you to quantify the local contribution (per unit surface area or volume) to the total acoustic power generated by the flow. They include the following:

- Proudman's formula
- jet noise source model
- boundary layer noise source model
- source terms in the linearized Euler equations
- source terms in Lilley's equation

Considering that one would ultimately want to come up with some measures to mitigate the noise generated by the flow in question, the source models can be employed to extract useful diagnostics on the noise source to determine which portion of the flow is primarily responsible for the noise generation. Note, however, that these source models do not predict the sound at receivers.

Unlike the direct method and the FW-H integral method, the broadband noise source models do not require transient solutions to any governing fluid dynamics equations. All the source models need is

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

what typical RANS models would provide, such as the mean velocity field, turbulent kinetic energy (k) and the dissipation rate (ε). Therefore, the use of broadband noise source models requires the least computational resources.

23.2. Using the Ffowcs Williams and Hawkings Acoustics Model

The procedure for computing sound using the FW-H acoustics model in Ansys Fluent consists largely of two steps. In the first step, a time-accurate flow solution is generated from which time histories of the relevant variables (for example, pressure, velocity, and density) on the selected source surfaces are obtained. In the second step, sound pressure signals at the user-specified receiver locations are computed using the source data collected during the first step.

Important:

Note that you can also use the FW-H model for a steady-state simulation in the case where your model has a single moving reference frame. Here, the thickness and loading noise due to the motion of the noise sources is computed using the FW-H integrals (see Equation 11.5 and Equation 11.6 in the Theory Guide), except that the term involving the time derivative of surface pressure (contribution to $\dot{L}_r$ in Equation 11.6 in the Theory Guide) is set to zero.

In computing sound pressure using the FW-H integral solution, Ansys Fluent uses a so-called "forward-time projection" to account for the time delay between the emission time (the time at which the sound is emitted from the source) and the reception time (the time at which the sound arrives at the receiver location). The forward-time projection approach enables you to compute sound at the same time "on the fly" as the transient flow solution progresses, without having to save the source data.

In this section, the procedure for setting up and using the FW-H acoustics model is outlined first, followed by detailed descriptions of each of the steps involved. Remember that only the steps that are pertinent to acoustics modeling are discussed here. For information about the inputs related to other models that you are using in conjunction with the FW-H acoustics model, see the appropriate sections for those models.

The general procedure for carrying out an FW-H acoustics calculation in Ansys Fluent is as follows:

1. Calculate a converged flow solution. For a transient case, run the transient solution until you obtain a "statistically steady-state" solution as described below.

2. Enable the FW-H acoustics model and set the associated model parameters.

   Setup → Models → Acoustics → Edit...

3. Specify the source surface(s) and choose the options associated with acquisition and saving of the source data. For a steady-state case, specify the rotating surface zone(s) as the source surface(s).

4. Specify the receiver location(s).

5. Continue the transient solution for a sufficiently long period of time and save the source data (transient cases only).

   Solution → Run Calculation

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Compute and save the sound pressure signals.

   Solution → **Run Calculation** → **Acoustic Signals...**

7. Postprocess the sound pressure signals.

   Results → **Plots** → **FFT** **Edit...**

Important:

Before you start the acoustics calculation for a transient case, an Ansys Fluent transient solution should have been run to a point where the transient flow field has become "statistically steady". In practice, this means that the unsteady flow field under consideration, including all the major flow variables, has become fully developed in such a way that its statistics do not change with time. Monitoring the major flow variables at selected points in the domain is helpful for determining if this condition has been met.

As discussed earlier, URANS, LES, and hybrid RANS-LES models are all legitimate candidates for transient flow calculations. For stationary source surfaces, the frequency of the aerodynamically generated sound heard at the receivers is largely determined by the time scale or frequency of the underlying flow. Therefore, one way to determine the time-step size for the transient computation is to make it small enough to resolve the smallest characteristic time scale of the flow at hand that can be reproduced by the mesh and turbulence adopted in your model.

Once you have obtained a statistically stationary flow-field solution, you are ready to acquire the source data.

For additional information, see the following sections:

23.2.1. Enabling the FW-H Acoustics Model
23.2.2. Specifying Source Surfaces
23.2.3. Specifying Acoustic Receivers
23.2.4. Specifying the Time Step Size
23.2.5. Postprocessing the FW-H Acoustics Model Data
23.2.6. FFT of Acoustic Sources: Band Analysis and Export of Surface Pressure Spectra

23.2.1. Enabling the FW-H Acoustics Model

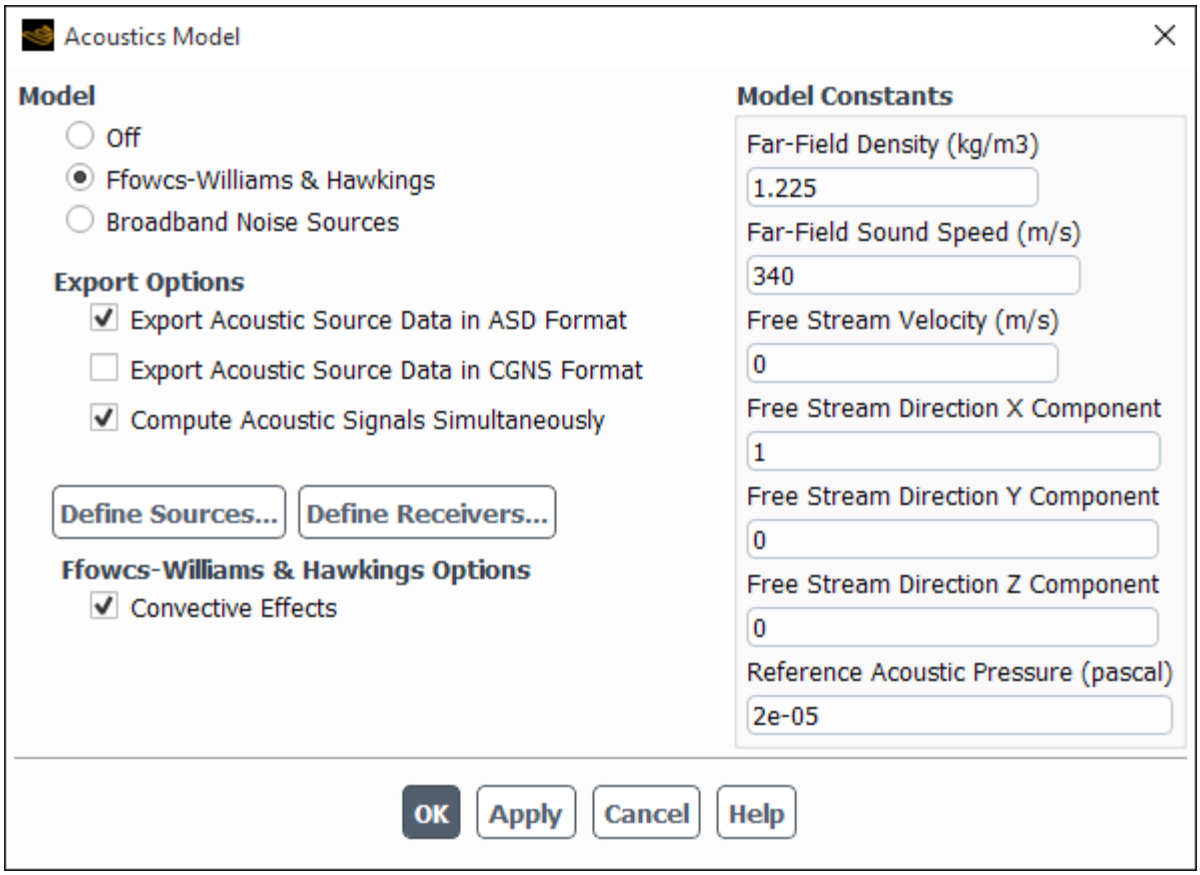
To enable the FW-H acoustics model, select **Ffowcs Williams & Hawkings** in the **Acoustics Model** dialog box (Figure 23.1: The Acoustics Model Dialog Box (p. 2630)).

Setup → **Models** → **Acoustics** **Edit...**

When you select **Ffowcs Williams & Hawkings**, the dialog box will expand to show the relevant fields for user inputs.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.1: The Acoustics Model Dialog Box



23.2.1.1. Setting Model Constants

Under **Model Constants** in the **Acoustics Model** dialog box, specify the relevant acoustic parameters and constants used by the model.

Far-Field Density

(for example, ρ_0 in Equation 11.1 in the Theory Guide) is the far-field fluid density.

Far-Field Sound Speed

(for example, a_0 in Equation 11.1) is the sound speed in the far field (= $\sqrt{\gamma R T_0}$).

Free Stream Velocity and Free Stream Direction

are required when the convective effects are taken into account. They become available in the interface when **Convective Effects** is enabled.

Important:

The use of **Convective Effects** with the proper **Free Stream Velocity** and **Free Stream Direction** is strongly recommended for all cases dealing with external flows around bodies.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reference Acoustic Pressure

(for example, p_{ref} in Equation 40.15 (p. 4086)) is used to calculate the sound pressure level in dB (see Using the FFT Utility (p. 4081)). The default reference acoustic pressure is 2×10^{-5} Pa.

Number of Time Steps Per Revolution

is available only for steady-state cases that have a single moving reference frame. Here you will specify the number of equivalent time steps that it will take for the rotating zone to complete one revolution.

Number of Revolutions

is available only for steady-state cases that have a single moving reference frame. Here you will specify the number of revolutions that will be simulated in the model.

Source Correlation Length

is required when sound is to be computed using a 2D flow result. The FW-H integrals will be evaluated over this length in the depth-wise direction using the identical source data (see Figure 23.2: The Acoustics Model Dialog Box for a 3D Steady-State Case with a Single Moving Reference Frame (p. 2632)).

The default values are appropriate for sound propagating in air at atmospheric pressure and temperature.

23.2.1.2. Computing Sound "on the Fly"

The FW-H acoustics model in Ansys Fluent allows you to perform simultaneous calculation of the sound pressure signals at the prescribed receivers without having to write the source data to files, which can save a significant amount of disk space on your machine. To enable this "on-the-fly" calculation of sound, enable the **Compute Acoustic Signals Simultaneously** option in the **Acoustics Model** dialog box.

Important:

Because the noise computation takes a negligible percentage of memory and computational time compared to a transient flow calculation, this option can be used by itself or along with the process of source data file export and sound calculation. For the latter, computing signals "on the fly" allows you to see when the signals have become statistically steady so you can know when to stop the simulation.

When the **Compute Acoustic Signals Simultaneously** option is enabled, the Ansys Fluent console window will print a message at the end of each time step indicating that the sound pressure signals have been computed (for example, `Computing sound signals at x receiver locations...`, where `x` is the number of receivers you specified). Enabling this option instructs Ansys

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fluent to compute sound pressure signals at the end of each time step, which will slightly increase the computation time.

Important:

Note that this option is available only when the FW-H acoustics model has been enabled. See below for details about exporting source data without enabling the FW-H model.

23.2.1.3. Writing Source Data Files

Although the "on-the-fly" capability is a convenient feature, you will want to save the source data as well, because the acquisition of source data during a transient flow-field calculation is the most time-consuming part of acoustics computations, and you most likely will not want to discard it. By saving the source data, you can always reuse it to compute the sound pressure signals at new or additional receiver locations.

To save the source data to files, enable either the **Export Acoustic Source Data in ASD Format** or the **Export Acoustic Source Data in CGNS Format** option, or both. After you have made your selection, the relevant source data at all face elements of the selected source surfaces will be written into the files you specify. The source data vary depending on the solver option you have chosen and whether the source surface is a wall or not. Table 23.1: Source Data Saved in Source Data Files (p. 2632) shows the flow variables saved as the source data.

Figure 23.2: The Acoustics Model Dialog Box for a 3D Steady-State Case with a Single Moving Reference Frame

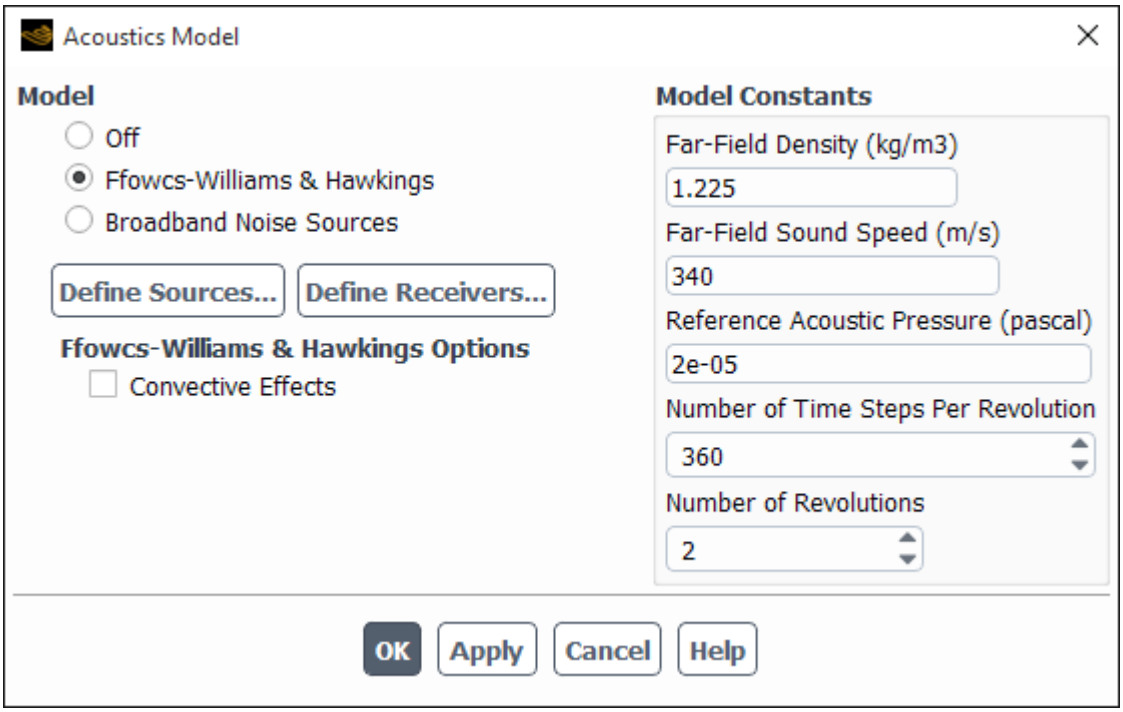


Table 23.1: Source Data Saved in Source Data Files

Solver Option	Source Surface	Source Data
incompressible	walls	p

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solver Option	Source Surface	Source Data
incompressible	permeable surfaces	p,u,v,w
compressible	walls	p
compressible	permeable surfaces	ρ,p,u,v,w

See Specifying Source Surfaces (p. 2635) for details on how to specify parameters for exporting source data.

23.2.1.3.1. Exporting Source Data Without Enabling the FW-H Model: Using the Ansys Fluent ASD Format

You can export sound source data for use with SYSNOISE without having to enable the Ffowcs Williams and Hawkings (FW-H) model. You still must specify source surfaces (see Specifying Source Surfaces (p. 2635)), as `.index` and `.asd` files are required by SYSNOISE. In addition, you can choose fluid zones as emission sources if you want to export quadrupole sources. To enable the selection of fluid zones as sources, use the

`define` → `models` → `acoustics` → `export-volumetric-sources?`

text command and change the selection to `yes`.

SYSNOISE also requires centroid data for source zones that are being exported.

For fan noise calculations, once you have specified the source zones in the **Acoustic Sources** dialog box and you have selected **Export Acoustic Source Data in ASD Format** from the **Acoustics Model** dialog box, you can export geometry in cylindrical coordinates by using the

`define` → `models` → `acoustics` → `cylindrical-export?`

text command and changing the selection to `yes`. By default, Ansys Fluent exports source zones for SYSNOISE in Cartesian coordinates.

You can then export the centroid data to a data file using the following text command:

`define` → `models` → `acoustics` → `write-centroid-info`

Since you will not be using the FW-H model to compute signals, you will not need to specify any acoustic model parameters or receiver locations. Also, you will not be able to enable the **Compute Acoustic Signals Simultaneously** option in the **Acoustics Model** dialog box, and **Acoustic Signals...** will not be available in the **Run Calculation** task page and in the menu for the **Run Calculation** Outline View item.

23.2.1.3.2. Exporting Source Data Without Enabling the FW-H Model: Using the CGNS Format

The sound source data for non-permeable surfaces can be exported in the CGNS file format (for Simcenter 3D Acoustics) without having to enable the Ffowcs Williams and Hawkings (FW-H) model. Enable the **Export Acoustic Source Data in CGNS Format** option in the **Acoustics Model** dialog box (Figure 23.3: The Acoustics Model Dialog Box for Exporting in CGNS

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

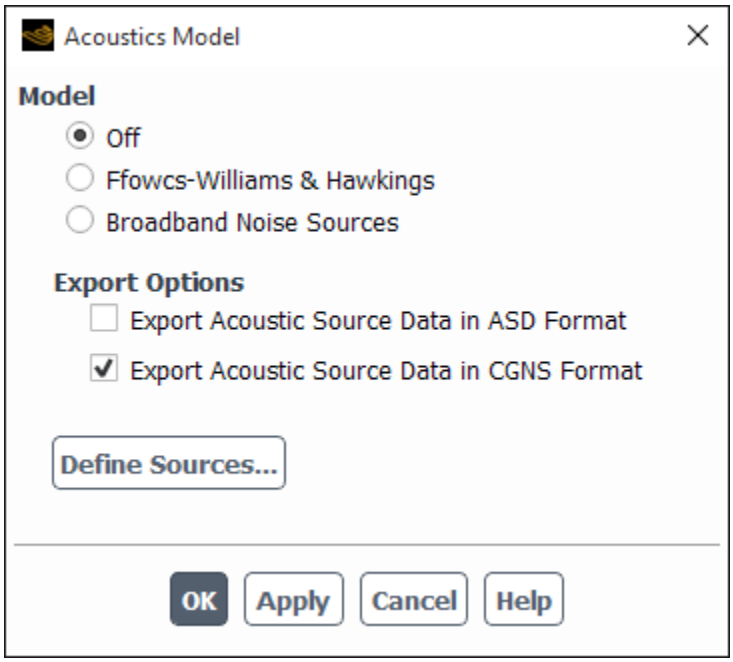
Format (p. 2634)). Specify the source surfaces in the **Acoustics Sources** dialog box (see Specifying Source Surfaces (p. 2635)) where, by default, the **Number of Time Steps per File** is set to 1.

Note:

The following limitation currently exists for exporting acoustic source data:

- Source data cannot be simultaneously exported for multiple differently rotating walls. All walls must be either non-rotating, or rotate with the same rate around the same axis. It is a responsibility of a user to select a consistent set of wall zones. Selecting differently rotating zones may cause incorrect acoustics result.

Figure 23.3: The Acoustics Model Dialog Box for Exporting in CGNS Format



Simcenter 3D Acoustics requires a mesh data file (named `<prefix>_mesh.cgns`) and a solution data file (named `<prefix>_<timestep>.cgns`). The string `<prefix>` is a generic name, which you will specify in the **File Name** in the **Acoustics Sources** dialog box. There is one single solution data file (`.cgns`) per time level exported, which contains the static pressure at the wall-face centroid location. The `.cgns` files will be stored in a directory, which you specify (named `<directory_name>/<prefix>`) in the **File Name**.

In addition, you can export quadrupole sources data by choosing fluid zones as emission sources. To enable the selection of fluid zones as sources, use the text command:

`define` → `models` → `acoustics` → `export-volumetric-sources-cgns?`

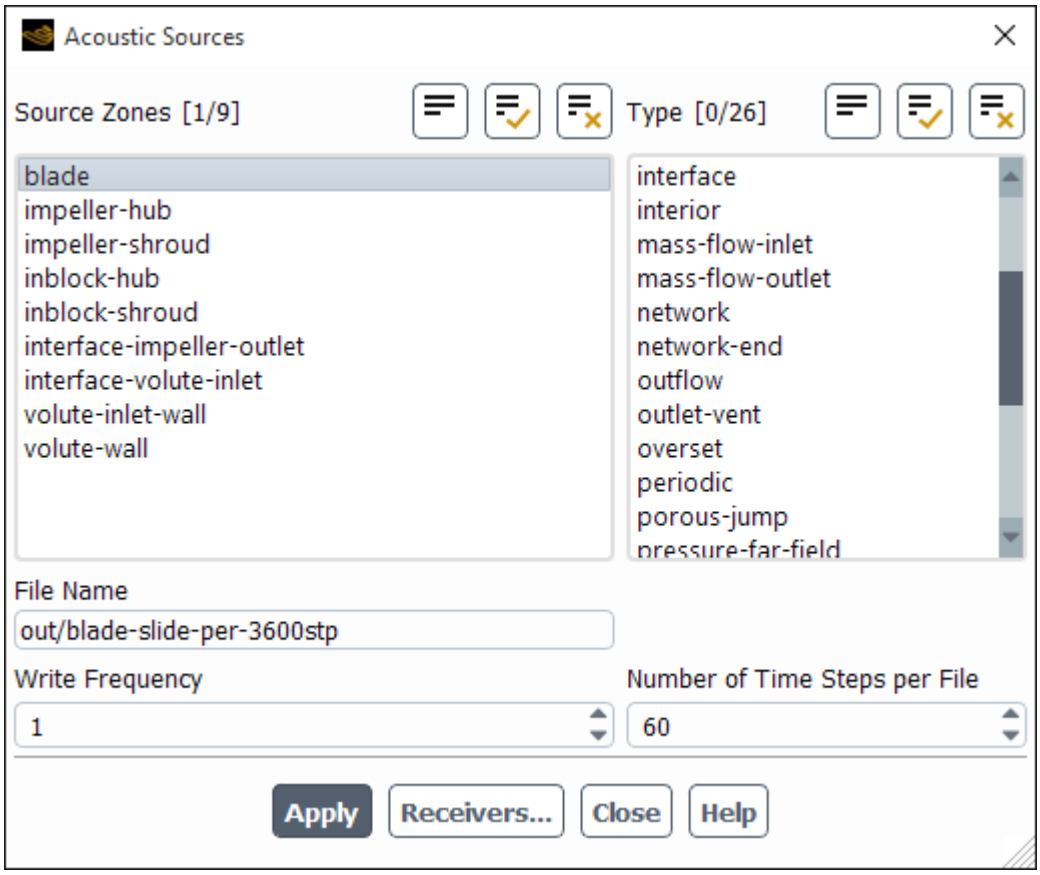
When asked if you would like to `Export volumetric sources?` enter `yes`. Note that Simcenter 3D Acoustics requires volumetric mesh data file (`<prefix>_Q_mesh.cgns`) and quadrupole solution data files (`<prefix>_Q_<timestep>.cgns`). The `.cgns` file will be stored in a similar way to that of dipole data export, in the directory you specified in the **File Name** text entry box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

23.2.2. Specifying Source Surfaces

In the **Acoustics Model** dialog box, click the **Define Sources...** button to open the **Acoustic Sources** dialog box (Figure 23.4: The Acoustics Model Dialog Box (p. 2635)). Here you will specify the source surface(s) to be used in the acoustics calculation and the inputs associated with saving source data to files.

Figure 23.4: The Acoustics Model Dialog Box



Under **Source Zones**, you can select multiple emission (source) surfaces and the surface **Type** that you can select is *not* limited to a **wall**. You can also choose **interior** surfaces and sliding interfaces (both stationary and rotating) as source surfaces.

Important:

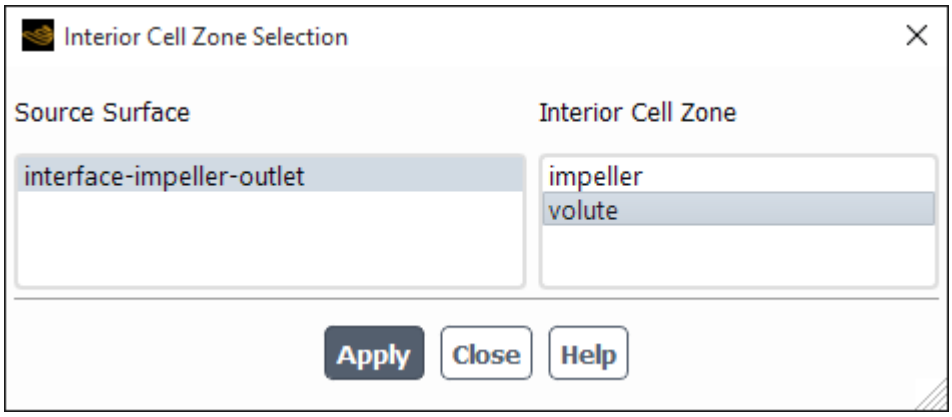
The ability to choose multiple source surfaces is useful for investigating the contributions from individual source surfaces. The results based on the use of multiple source surfaces are valid as long as there are negligible acoustic interactions among the surfaces. Therefore, some caution should be taken when selecting multiple source surfaces.

In cases where multiple source surfaces are selected, no source surface may enclose any of the other source surfaces. Otherwise, the sound pressure calculated based on the source surfaces will not be accurate, as the contribution from the enclosed (inner) source surfaces is over predicted, since the FW-H model is unable to account for the shading of the sound from the inner source surfaces by the enclosure surface.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you specify any interior surfaces as source surfaces, the interior surface must be generated in advance, in such a way that the two cell zones adjacent to the surface have different cell zone IDs. Furthermore, you must correctly specify which of the two zones is occupied by the quadrupole sources (interior cell zone). This will allow Ansys Fluent to determine the direction in which the sound will propagate. When you first attempt to select a legitimate interior surface (that is, an interior surface having two different cell zones on both sides) as a source surface, the Interior Cell Zone Selection Dialog Box (p. 4869) (Figure 23.5: The Interior Cell Zone Selection Dialog Box (p. 2636)) will appear. You must then select the interior cell zones from the two zones listed under the **Interior Cell Zone**. Figure 23.6: An Interior Source Surface (p. 2637) shows an example of an interior source surface.

Figure 23.5: The Interior Cell Zone Selection Dialog Box



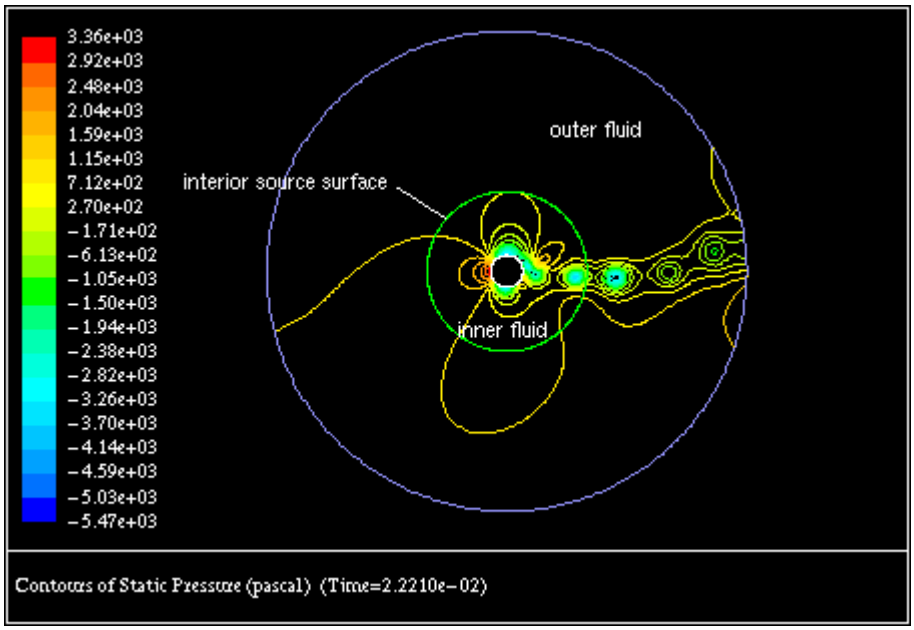
Like general interior surfaces, if the source surfaces selected are sliding interfaces, a dialog box similar to Figure 23.5: The Interior Cell Zone Selection Dialog Box (p. 2636) will appear that will show the two adjacent cell zones and you will be asked to specify the zone that has the sound sources.

Important:

When a permeable surface (either interior or sliding interface) is chosen as the source surface, other wall surfaces inside the volume enclosed by the permeable surface that generate sound should not be chosen for the acoustics calculation. For example, when running an "on-the-fly" calculation, if both these surfaces are selected, the sound pressure will be counted twice.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.6: An Interior Source Surface



23.2.2.1. Saving Source Data

To save the source data, you have to specify the **File Name**, **Write Frequency** (in number of time steps), and **Number of Time Steps per File** in the **Acoustic Sources** dialog box.

The **File Name** is used to give the names of the source data files (for example, `acoustic_example`xxxx`.asd`, where `xxxx` is the global time-step index of the transient solution) and an index file (for example, `acoustic_example.index`) that will store the information associated with the source data. The **Write Frequency** allows you to control how often the source data will be written. This will enable you to save disk space if the time-step size used in the transient flow simulation is smaller than necessary to resolve the sound frequency you are attempting to predict. In most situations, however, you will want to save the source data at every time step and use the default value of `1`.

Since acoustics calculations usually generate thousands of time steps of source data, you may want to split the data into several files. Specifying the **Number of Time Steps per File** allows you to write the source data into separate files for different simulation intervals, the duration of which (in terms of the number of transient flow time steps) is specified by you. For example, if you specify 100 for this parameter, each file will contain source data for an interval length of 100 time steps regardless of the write frequency.

You will find this feature useful if you want to use a selected number of source data files to compute the sound pressure rather than using all the data. For example, you may want to exclude an initial portion of the source data from your acoustics calculation because you may realize later that the flow field has not fully attained a statistically steady state.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

After you click **Apply**, Ansys Fluent will create the index file (for example, `acoustics_example.index`), which contains information about the source data.

Important:

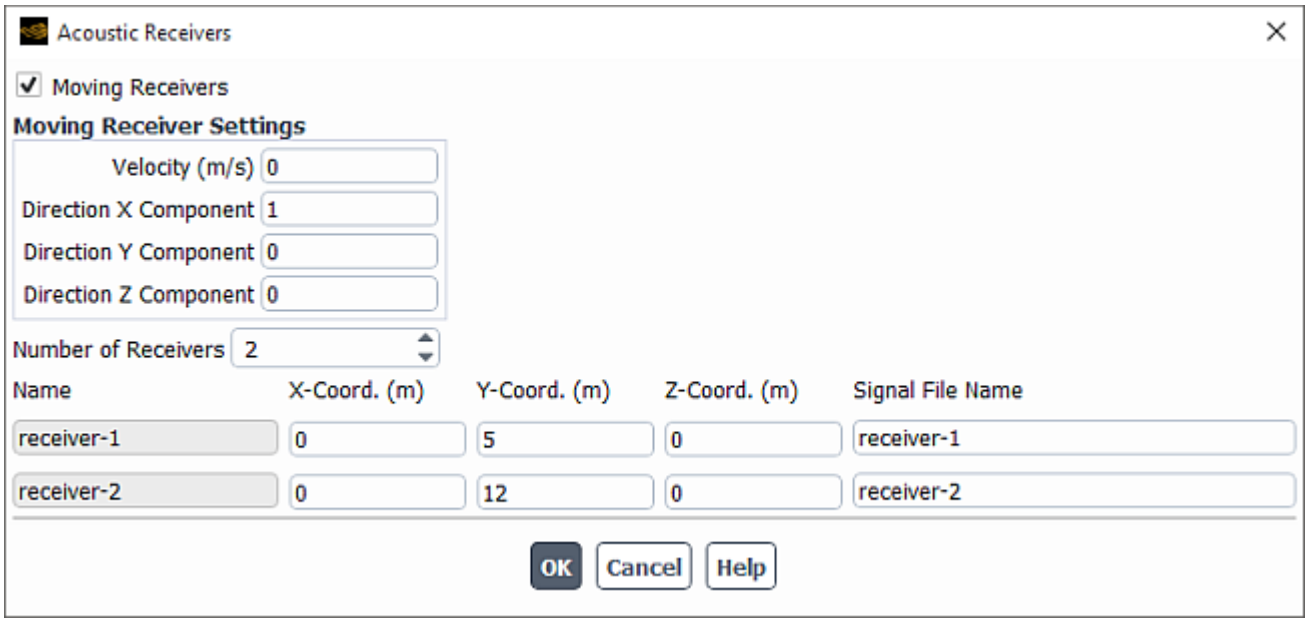
If you choose to save source data, keep in mind that the source data can use up a considerable amount of disk space, especially if the mesh being used has a large number of face elements on the source surfaces you selected. Ansys Fluent will print out the disk space requirement per time step at the time of source surface selection if the **Export Acoustic Source Data in ASD Format** option is enabled in the **Acoustics Model** dialog box.

At this point, if you have chosen to perform your acoustics calculation in two steps, (that is, saving the source data first, and computing the sound at a later time), you can go ahead and instruct Ansys Fluent to perform a suitable number of time steps, and the source data will be saved to the disk. If you chose to perform an "on-the-fly" acoustic calculation, then you must specify receiver locations (see Specifying Acoustic Receivers (p. 2638)) before you run the unsteady Ansys Fluent solution any further.

23.2.3. Specifying Acoustic Receivers

In the **Acoustics Model** dialog box, click the **Define Receivers...** button to open the **Acoustic Receivers** dialog box (Figure 23.7: The Acoustic Receivers Dialog Box (p. 2638)).

Figure 23.7: The Acoustic Receivers Dialog Box



Important:

Note that you can also open the **Acoustic Receivers** dialog box by clicking the **Receivers...** button in the **Acoustic Sources** or the **Acoustic Signals** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If required, you can enable **Moving Receivers** to specify the receiver motion. In this case, the defined **Velocity** magnitude and **Direction** apply to all receivers. The receiver locations, defined below, are then interpreted as the starting locations at the time that sound emission originates. The origination of sound emission is determined as follows:

- For "on the Fly" use of the FW-H model (see Computing Sound "on the Fly" (p. 2631)) the sound emission time is counted from the physical time of activation of the **Compute Acoustic Signals Simultaneously** option (see Figure 23.1: The Acoustics Model Dialog Box (p. 2630)).
- When the FW-H model reads the previously written acoustic source data (see Reading Unsteady Acoustic Source Data (p. 2641)), the sound emission time starts from the physical time associated with the first selected source data file.

Important:

When using moving receivers, note that if you start from a different source data file, this results in different receiver locations due to the different emission time origin associated with the different source data file.

Increase the **Number of Receivers** to the total number of receivers for which you want to compute sound, and enter the coordinates for each receiver in the **X-Coord.**, **Y-Coord.**, and **Z-Coord.** fields. Note that because Ansys Fluent's acoustics model is ideally suited for far-field noise prediction, the receiver locations you define should be at a reasonable distance from the sources of sound (that is, the selected source surfaces). The receiver locations can also fall outside of the computational domain.

For each receiver, you can specify a file name root in the **Signal File Name** field. The signal file name extension for the FW-H model is fixed to `.ard`. These files will be used to store the sound pressure signals at the corresponding receivers. By default, the files will be named `receiver-1.ard`, `receiver-2.ard`, and so on.

Once the receiver locations have been defined, the setup for your acoustic calculation is complete.

23.2.4. Specifying the Time Step Size

When using an implicit-in-time solution algorithm (dual-time stepping), and depending on the physical time step size and the most important time scales in the flow, you can write the acoustic source data at every time step. You can also coarsen it in time by a given frequency factor. The highest possible frequency the acoustic analysis can generate is based on the time step size of the collected acoustic source data.

When using the **Density-Based** explicit solver, with the **Explicit Transient Formulation** selected in the **Solution Methods** task page, the physical time step size must be computed by the solver, based on the CFL condition (Courant number). Due to the possibly large fluctuations of the physical time step size, an adapting time-stepping procedure can be used when the FW-H acoustics model is enabled. This allows you to use a user-specified time interval for sampling the acoustic data. In turn, the solver adapts its time step size, when necessary, without violating the CFL conditions to make sure that data is available at the desired time interval (hence, avoiding data interpolations).

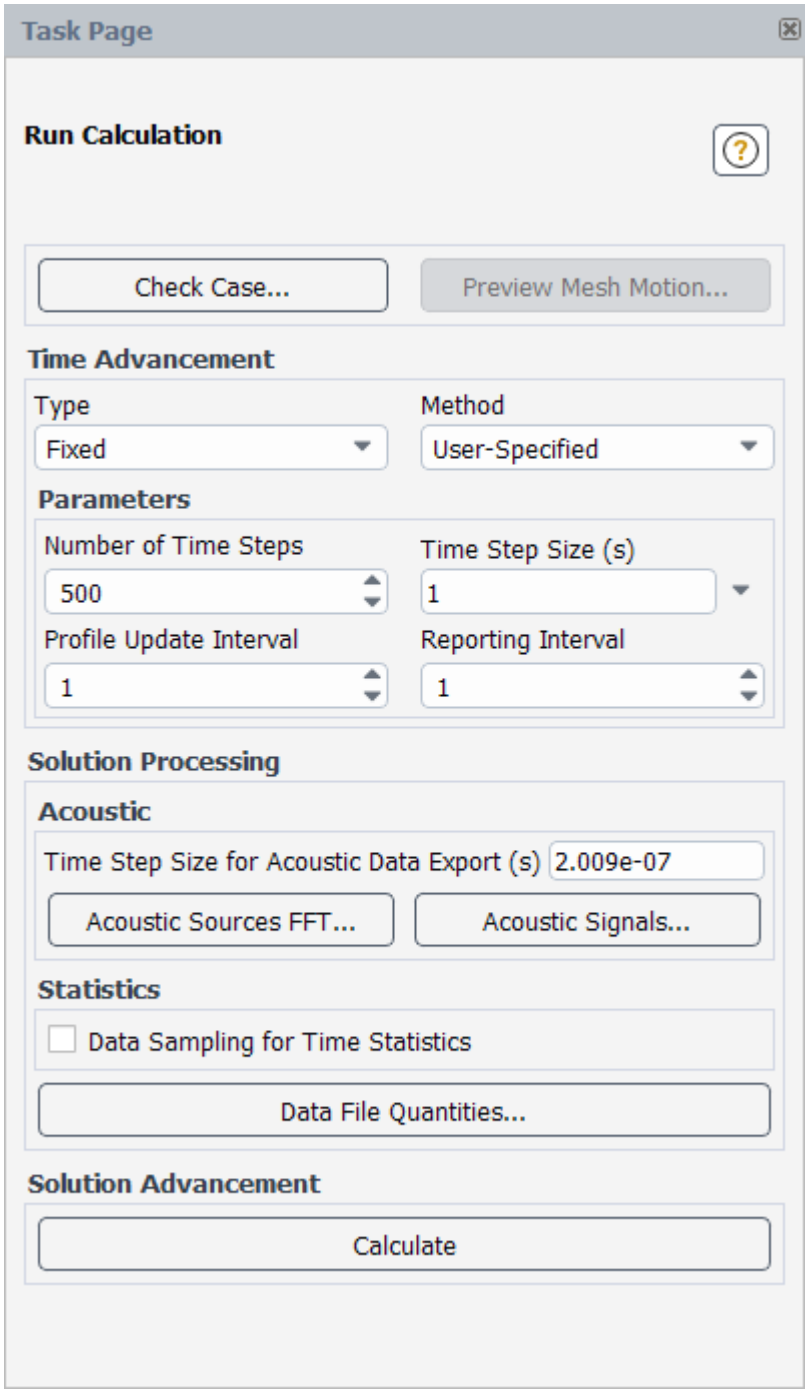
In the **Run Calculation** task page (Figure 23.8: The Run Calculation Task Page (p. 2640)), enter the **Time Step Size for Acoustic Data Export** to specify the time interval for acoustic data sampling. The value of this constant time step size determines the highest frequency that the acoustic analysis reproduces.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can refer to Performing Time-Dependent Calculations (p. 3643) for more information about the **Run Calculation** task page.

Solution → **Run Calculation**

Figure 23.8: The Run Calculation Task Page



You can now proceed to instruct Ansys Fluent to perform a transient calculation for a suitable number of time steps. When the calculation is finished, you will have either the source data saved on files (if you chose to save it to a file or files), or the sound pressure signals (if you chose to perform an

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

acoustic calculation "on the fly"), or both (if you chose to save the source data to files *and* if you chose to perform the acoustic calculation "on the fly").

If you chose to save the source data to files, the Ansys Fluent console will print a message at the end of each time step indicating that source data have been written (or appended to) a file (for example, `acoustic_example240.asd`).

23.2.5. Postprocessing the FW-H Acoustics Model Data

At this point, you will have either the source data saved to files or the sound pressure signals computed, or both. You can process these data to compute and plot various acoustic quantities using Ansys Fluent's FFT capabilities (Fast Fourier Transform (FFT) Postprocessing (p. 4078)). Additionally, you can perform Ansys Sound analysis on computed pressure signals. Refer to Performing an Ansys Sound Analysis (p. 4089) for additional information.

23.2.5.1. Writing Acoustic Signals

If you chose to perform the acoustic calculation "on the fly", you must write the sound pressure data to files. To do so, select **Write Acoustic Signals** under **Options** in the **Acoustic Signals** dialog box (Figure 23.9: The Acoustic Signals Dialog Box (p. 2642)) and then click **Write**. The computed acoustic pressure will be saved from internal buffer memory into a separate file for each receiver you defined in the **Acoustic Receivers** dialog box (for example, `receiver-1.ard`).

Solution → **Run Calculation** → **Acoustic Signals...**

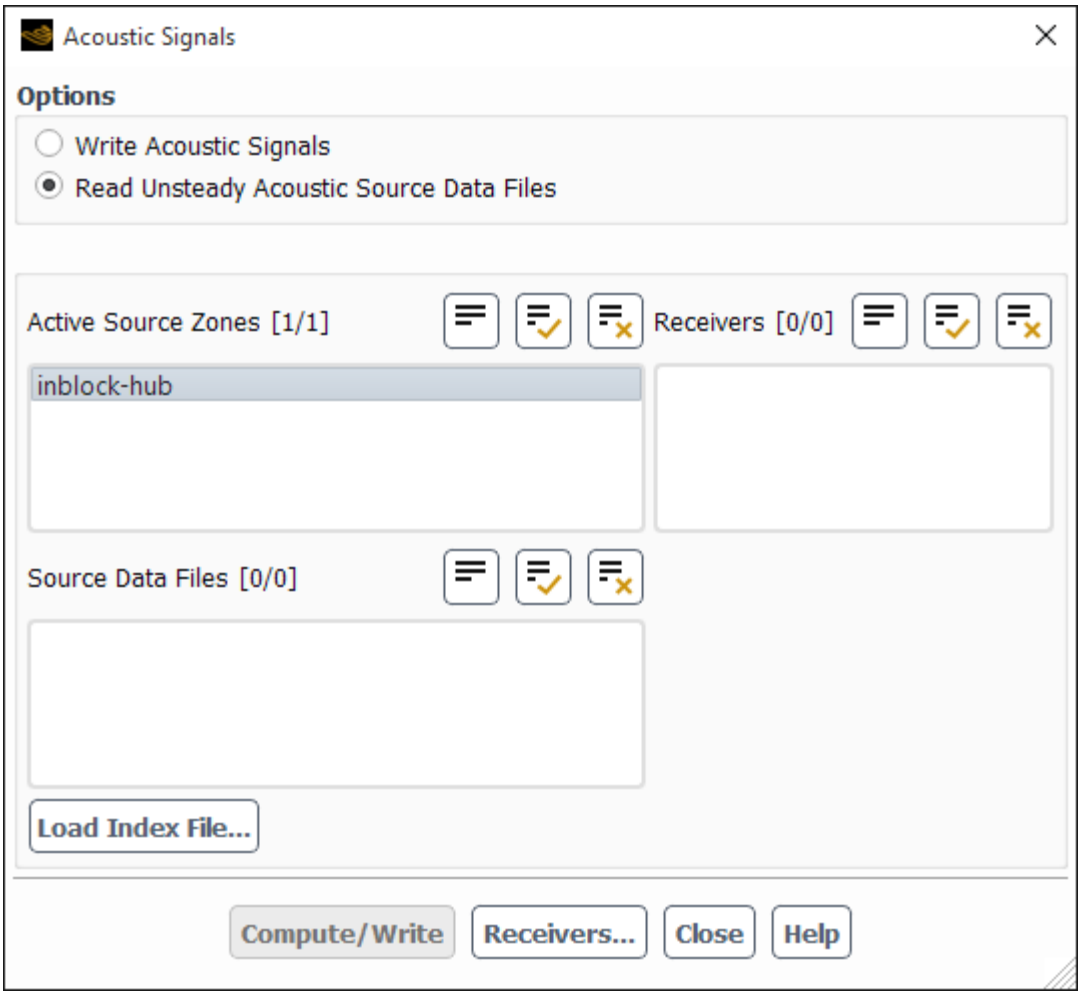
23.2.5.2. Reading Unsteady Acoustic Source Data

Computing the sound pressure signals using the source data saved to files is done in the **Acoustic Signals** dialog box (Figure 23.9: The Acoustic Signals Dialog Box (p. 2642))

Solution → **Run Calculation** → **Acoustic Signals...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.9: The Acoustic Signals Dialog Box



To compute the sound data, use the following procedure:

1. In the **Acoustic Signals** dialog box, select **Read Unsteady Acoustic Source Data Files** under **Options**.
2. Click **Load Index File...** and select the index file for your computation in the **Select File** dialog box. The file will have the name you entered in the **File Name** field in the **Acoustic Sources** dialog box, followed by the `.index` suffix (for example, `acoustic_example.index`).
3. In the **Source Data Files** list, select the source data files that you want to use to compute sound. Source data files will all contain the specified root file name followed by the suffix `.asd`.

   Important:

   You can use any number of source data files. However, note that you should select only consecutive files.

4. In the **Active Source Zones** list, select the source zones you want to include to compute sound. See Specifying Source Surfaces (p. 2635) for details about proper source surface selection.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. In the **Receivers** list, select the receivers for which you want to compute and save sound.

   Optionally, you can click the **Receivers...** button to open the **Acoustic Receivers** dialog box and define additional receivers.

6. Click the **Compute/Write** button to compute and save the sound pressure data. One file will be saved for each receiver you previously specified in the **Acoustic Receivers** dialog box (for example, `receiver-1.ard`).

Important:

If you enabled both the **Export Acoustic Source Data in ASD Format** and **Compute Acoustic Signals Simultaneously** options in the **Acoustics Model** dialog box, you must first select the **Write Acoustic Signals** option in the **Acoustic Signals** dialog box after the flow simulation has been completed. If you select the **Read Unsteady Acoustic Source Data Files** before writing out the "on-the-fly" data in such a case, the data will be flushed out of the internal buffer memory. To avoid such a loss of data, you should save the Ansys Fluent case and data files whenever you begin to do an acoustic computation in the **Acoustic Signals** dialog box. The sound pressure data calculated "on the fly" will then be saved into the `.dat` file. Finally, after the "on-the-fly" data is saved, make sure to change the file names of the receivers before doing a sound pressure calculation with the **Read Unsteady Acoustic Source Data Files** option enabled, to avoid overwriting the "on-the-fly" signal files.

Important:

Note that you can compute and write sound pressure signals only when the FW-H acoustics model has been enabled. See Exporting Source Data Without Enabling the FW-H Model: Using the Ansys Fluent ASD Format (p. 2633) for details about exporting source data (for example, for SYSNOISE) without enabling the FW-H model.

23.2.5.2.1. Pruning the Signal Data Automatically

Before the computed sound pressure data at each receiver is saved, it is by default automatically pruned. Pruning of the receiver data means clipping the tails of the signal where incomplete source information is available.

The acoustic source data is tabulated from time τ_0 to τ_n. Without auto-pruning, the receiver register begins receiving the earliest sound pressure signal at

$$t_0 = \tau_0 + \frac{r_{min}}{a_0} \tag{23.1}$$

where r_{min} is the shortest distance between the source surfaces and the receiver. However, the receiver will not receive the sound pressure signal from the farthest point on the source surfaces (r_{max}) until the receiver time becomes

$$t_1 = \tau_0 + \frac{r_{max}}{a_0} \tag{23.2}$$

From time t_0 to t_1, the sound accumulated on the receiver register does not include the contribution from the entire source surface area, and therefore the sound pressure data received during that time is not complete. The same thing occurs during the period from

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$t_m = \tau_m + \frac{r_{min}}{a_0} \tag{23.3}$$

to

$$t_n = \tau_n + \frac{r_{max}}{a_0} \tag{23.4}$$

Therefore, pruning means clipping the signal on the incomplete ends, from t_0 to t_1 and t_m to t_n. Auto-pruning can be disabled using the `define` → `models` → `acoustics` → `auto-prune` text command. Although auto-pruning can be disabled, it is expected that you will use only the complete sound pressure data.

23.2.5.3. Reporting the Static Pressure Time Derivative

The RMS value of the static pressure time derivative ($\partial p / \partial t$) is available for postprocessing only on wall surfaces, which are at the same time sources of sound, when the FW-H acoustics model is used.

You can select **Surface dpdt RMS** in the **Acoustics...** category only when you specify at least one wall surface, which is also marked as an acoustic source, in the relevant postprocessing dialog boxes.

23.2.5.4. Using the FFT Capabilities for Sound Pressure Signals

Once the sound pressure signals are computed and saved in files, the sound data is ready to be analyzed using Ansys Fluent's FFT tools. In the **Fourier Transform** dialog box (Figure 40.124: The Fourier Transform Dialog Box (p. 4082)), click **Load Input File...** and select the appropriate `.ard` file. If the receiver data is still in Ansys Fluent's memory, then it can directly be processed using the **Process Receiver** option. See Fast Fourier Transform (FFT) Postprocessing (p. 4078) for more information on Ansys Fluent's FFT capabilities.

Results → **Plots** → **FFT** **Edit...**

23.2.6. FFT of Acoustic Sources: Band Analysis and Export of Surface Pressure Spectra

During a transient flow simulation, the time histories of the instant surface pressure values can be written into the binary acoustic source data (ASD) files. This operation is performed for all mesh cell faces of those surface zones that are selected as acoustic sources. The primary purpose of the ASD files is to supply input data for the Ffowcs Williams and Hawkings acoustic solver (see Reading Unsteady Acoustic Source Data (p. 2641)). Additionally, these files can be used to compute and visualize the spectral properties of the stored flow pressure signals. The supported spectral properties for the individual Fourier modes are the complex amplitudes. For the frequency bands, the surface pressure level (SPL) values, in decibels, and the dimensional power spectral density (PSD) values of the pressure

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

time derivative are provided. Both the constant width and the proportional width bands (octaves and 1/3 octaves) can be specified.

Note:

Despite the use of the common acronym "SPL", the calculated values characterize the transient flow pressure on a surface, and not the acoustic pressure; that is, in this context, the acronym refers to the "Surface Pressure Level" (as mentioned previously) and not the "Sound Pressure Level".

In addition to the visualization capabilities, you can also use the computed spectral data to export surface fields of the complex amplitudes of flow pressure as CGNS files. The primary purpose of such files is to allow you to use the surface spectrum fields as distributed excitation forces when performing a harmonic response analysis with Ansys Mechanical. The ability to import CGNS files is available in Ansys Mechanical to match this export feature in Fluent; for details, see **FLUREAD** in the separate Mechanical APDL *Command Reference*. Using the acoustic wave equation solver of Ansys Mechanical, it is possible to perform frequency domain simulations for applications such as the vehicle cabin noise caused by external turbulent flow.

23.2.6.1. Using the FFT of Acoustic Sources

The **Acoustic Sources FFT...** button is available in the **Run Calculation** task page under the following conditions:

- The simulation is set up as a three-dimensional transient case.
- The **Export Acoustic Source Data in ASD Format** option is enabled in the **Acoustics Model** dialog box.

Clicking the **Acoustic Sources FFT...** button opens the **Acoustic Sources FFT** dialog box, which has four tabs (each of which is shown in the figures that follow):

- **Read ASD Files**
- **Compute FFT Fields**
- **FFT Surface Variables**
- **Write CGNS Files**

The first two tabs are obligatory, because the actions performed there prepare the Fourier spectra for the actions that follow, including creation of the surface variables to visualize (performed in the third tab) and/or CGNS export (performed in the fourth tab).

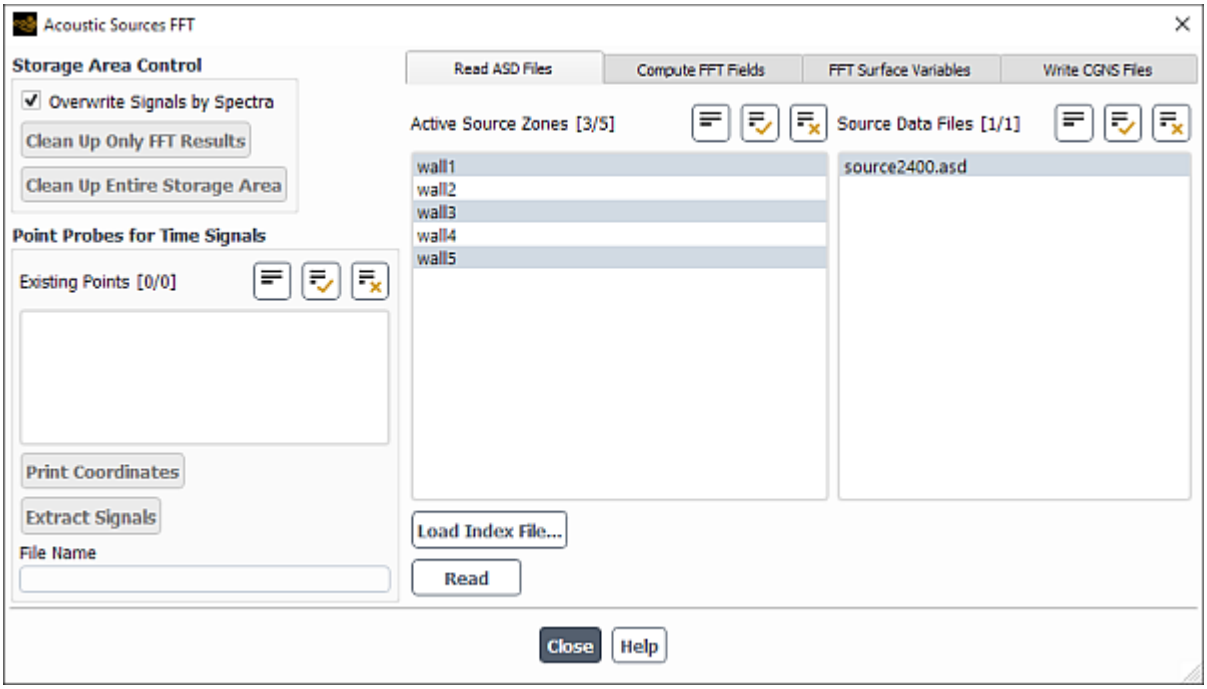
Postprocessing begins in the **Read ASD Files** tab. The **Active Source Zones** selection list shows all of the face zones for which transient export to ASD files has been performed (see Figure 23.10: The Read ASD Files Tab of the Acoustic Source FFT Dialog Box (p. 2646)). The **Source Data Files** selection list shows all of the available ASD files. Note that you can update this information by clicking the **Load Index File...** button and specifying the name of the index file to be read; the index file was created during the ASD files export, and is an ASCII file containing a list of the source zones and a list of the ASD file names. Select in the selection lists the desired face zones as well as the desired

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ASD files from which you want to read, and click the **Read** button. The console displays information about the data, such as the following example:

```
Overall 10000 timesteps have been read
from 3.0001 s to 4 s
Time step is 0.0001 s
```

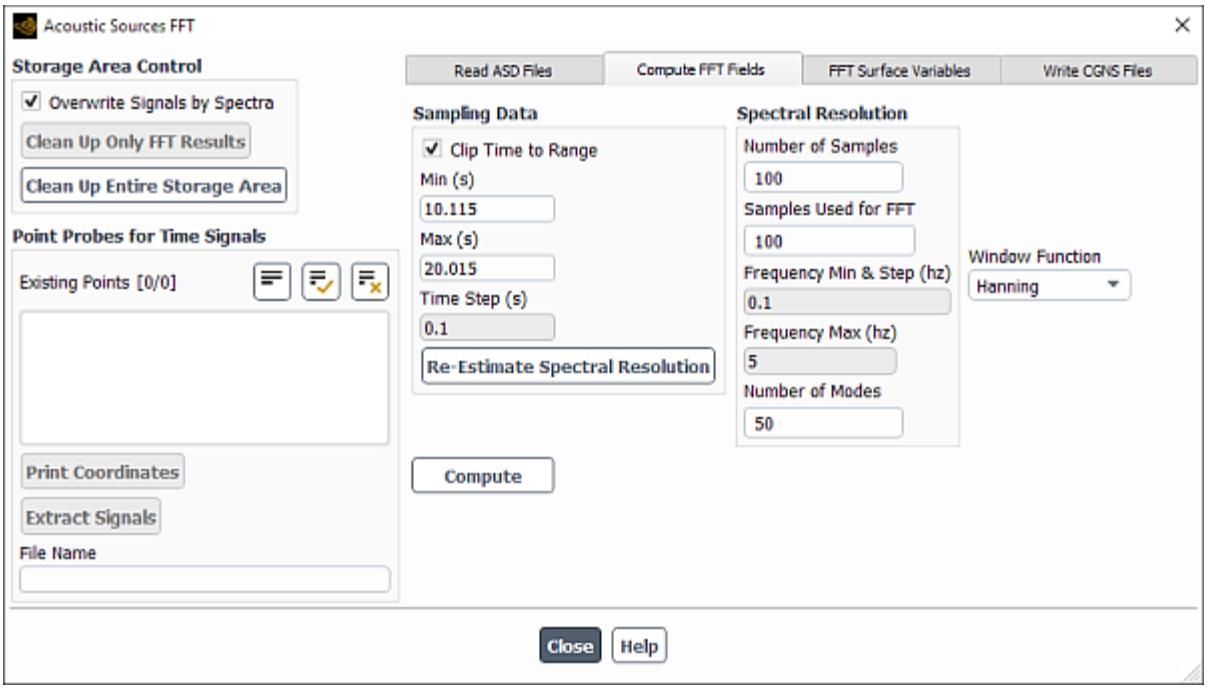
Figure 23.10: The Read ASD Files Tab of the Acoustic Source FFT Dialog Box



Having read the pressure histories, open the **Compute FFT Fields** tab (Figure 23.11: The Compute FFT Fields Tab of the Acoustic Source FFT Dialog Box (p. 2647)). The values shown in the **Spectral Resolution** group box provide information about the statistical properties, which depend on the sampling rate (that is, the simulation time step size) and the signal length (that is, the simulation time). These values will change if you enable the **Clip Time to Range** option in the **Sampling Data** group box, reduce the time range for the signals by changing the **Min** and/or **Max** fields, and click the **Re-Estimate Spectral Resolution** button. The **Window Function** selection list allows you to specify a window function using the same choices that are available for plotting the single signal FFT in the **Plot/Modify Input Signal** dialog box. The default settings use the full signals and the Hanning window function. The FFT calculation begins when you click the **Compute** button.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.11: The Compute FFT Fields Tab of the Acoustic Source FFT Dialog Box



The fields of the pressure histories and the computed fields of the Fourier spectra are stored in a large memory array, referred to here as the "storage area". This memory is allocated by Fluent dynamically when you read the pressure signals from the ASD files. If you read data for all face zones available in the ASD files, and if they are the wall zones, then the size of the allocated storage area is equal to the total size of the ASD files being read. Since the same or a slightly smaller amount of data is produced by the FFT algorithm, the same storage area can be re-used to store the Fourier spectra without allocating additional memory. Alternatively, the pressure histories can be left intact in memory, and for their spectra the storage area is correspondingly increased.

These two modes of memory allocation are controlled by the **Overwrite Signals by Spectra** option in the **Storage Area Control** group box. When the **Overwrite Signals by Spectra** option is enabled, the Fourier spectra displace in memory the original time signals, which are not kept after the FFT has been computed. Therefore, if you want to re-compute the FFT using the **Clip Time to Range** tool or a different **Window Function** (which are both in the **Compute FFT Fields** tab), you have to first de-allocate the storage area by clicking the **Clean Up Entire Storage Area** button, and then re-read the pressure histories from the ASD files. When the **Overwrite Signals by Spectra** option is disabled, you can re-compute the FFT without re-reading the pressure histories, as many times as you want. To do this, click the **Clean Up Only FFT Results** button, and proceed with computing spectra from the available pressure histories. De-allocation of the storage area (which can be very large) is recommended when you are done using the **Acoustic Sources FFT** dialog box and plan to either proceed to other postprocessing work or continue your transient simulation without re-starting Fluent.

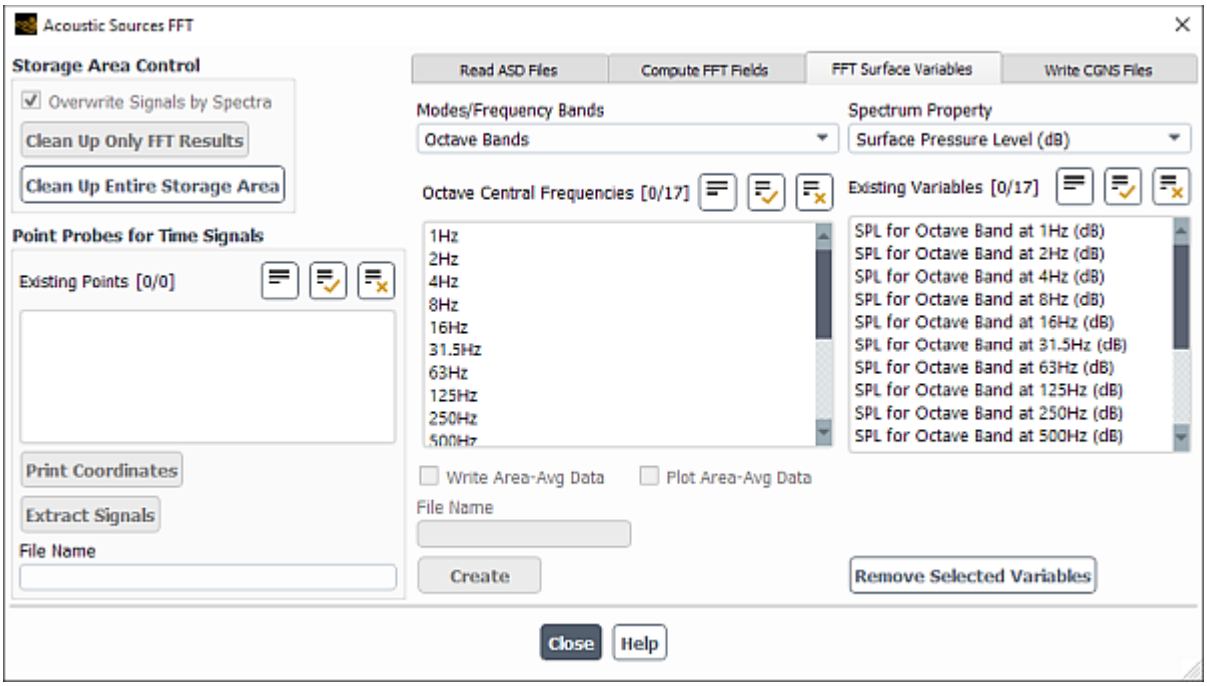
When the pressure histories are read and kept in the storage area, you can create point probes within the surface zones that have been read and extract the single pressure time signals at these probes. The extracted signals and their spectra may help to analyze the transient flow properties. For example, by creating an XY plot of a signal, you can figure out the need for clipping of the sampling time; by creating an XY plot of its spectrum, you can detect the presence of the tonal noise components.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To create a point probe, open the Point Surface Dialog Box (p. 5582) by clicking **Create** and selecting **Point...** in the **Domain** ribbon tab (**Surface** group box), and create a point surface by any available method (for example, by specifying the coordinates). The point probe will be created at the face center that is nearest to the point surface on the read surface zones. The created point probes are then listed under the point surface names in the **Existing Points** selection list, which is located in the **Point Probes for Time Signals** group box. You can print the coordinates of the point probes to the Fluent console by selecting the desired probes and clicking the **Print Coordinates** button. Clicking the **Extract Signals** button outputs the pressure histories for the selected probes in the ASCII files; these files can then be analyzed using the Fourier Transform Dialog Box (p. 5302) (available from the Plots Task Page (p. 5289)) by selecting **Process File Data** and then reading the exported ASCII file using the **Load Input File...** button. The exported files names are by default the point probe names with the extension `.dat`. The **File Name** field allows you to specify a prefix for these file names.

The **FFT Surface Variables** tab is shown in Figure 23.12: The FFT Surface Variables Tab of the Acoustic Source FFT Dialog Box for the Octave Bands (p. 2648) and Figure 23.14: The FFT Surface Variables Tab of the Acoustic Source FFT Dialog Box for a Set of Individual Modes (p. 2650). This tab allows you to **Create** new Fluent variables using the computed spectrum fields, which reside in the storage area and correspondingly increase its size. These variables will be defined only on those wall face zones for which the Fourier transformation has been computed; on the other face zones, they will show zero values. Created variables can be used for any kind of further postprocessing in Fluent, which typically includes contour plotting (see the variables marked with the restriction *asfft* in Table 42.21:**Acoustics** Category (p. 4203)) or exporting to CFD-Post.

Figure 23.12: The FFT Surface Variables Tab of the Acoustic Source FFT Dialog Box for the Octave Bands



Variables may characterize either the individual Fourier modes (if you select **Set of Modes** from the **Modes/Frequency Bands** drop-down list), or frequency bands that each represent a range of frequencies (if you select the other choices in the **Modes/Frequency Bands** drop-down list). The three types of frequency bands that are available include:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Octave Bands**

  These are proportional bands corresponding to the standard technical octaves.

- **1/3 Octave Bands**

  These are proportional bands corresponding to the standard technical thirds.

- **Constant Width Bands**

  These are user-defined equidistant bands.

The **Spectrum Property** drop-down list displays the type of variables created according to the selected choice in the **Modes/Frequency Bands** drop-down list. For a set of modes, the created variables are the real and the imaginary parts of the complex Fourier amplitudes, with a pair of variables per specified mode. For the frequency bands, the two types of variables can be created:

- The surface pressure level (SPL) fields, in decibels. The transformation to the decibel units is done by default using the standard acoustic reference pressure value of 2×10^{-5} Pa; you can change this value before creating the variables by opening the **Acoustic Models** dialog box, selecting the **Ffowcs Williams & Hawkings** model, and revising the **Reference Acoustic Pressure**.

- The fields of the dimensional power spectral density (PSD) of the pressure time derivative, in units of pressure squared over time squared.

For any type of frequency band (octave, 1/3, or constant width), the area-averaged band-specific pressure fluctuations are calculated and printed out to the Fluent console in decibels when you click **Create**. Conversion to decibels is done after the area-averaging of the band-filtered RMS pressure fluctuation fields. Similarly, the area-averaged band-specific values of PSD of the pressure time derivative are calculated and printed out, if this variable type is selected. The area-averaged data are printed only for those frequency bands that are covered by the resulting pressure spectrum. If you enable the **Write Area-Avg Data** option and specify a file name in the corresponding field, then the area-averaged data will be written to an ASCII file in the XY plot file format (see XY Plot File Format (p. 4071)). It is recommended that you use the file name extension `.xy` to more easily recognize this file when using the XY plotting tool.

If you enable the **Plot Area-Avg Data** option, then the data will be plotted in the form of a bar chart in the graphics window when you click **Create**, as shown in Figure 23.13: Bar Chart of Surface Pressure Level for Octave Bands (p. 2650).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.13: Bar Chart of Surface Pressure Level for Octave Bands

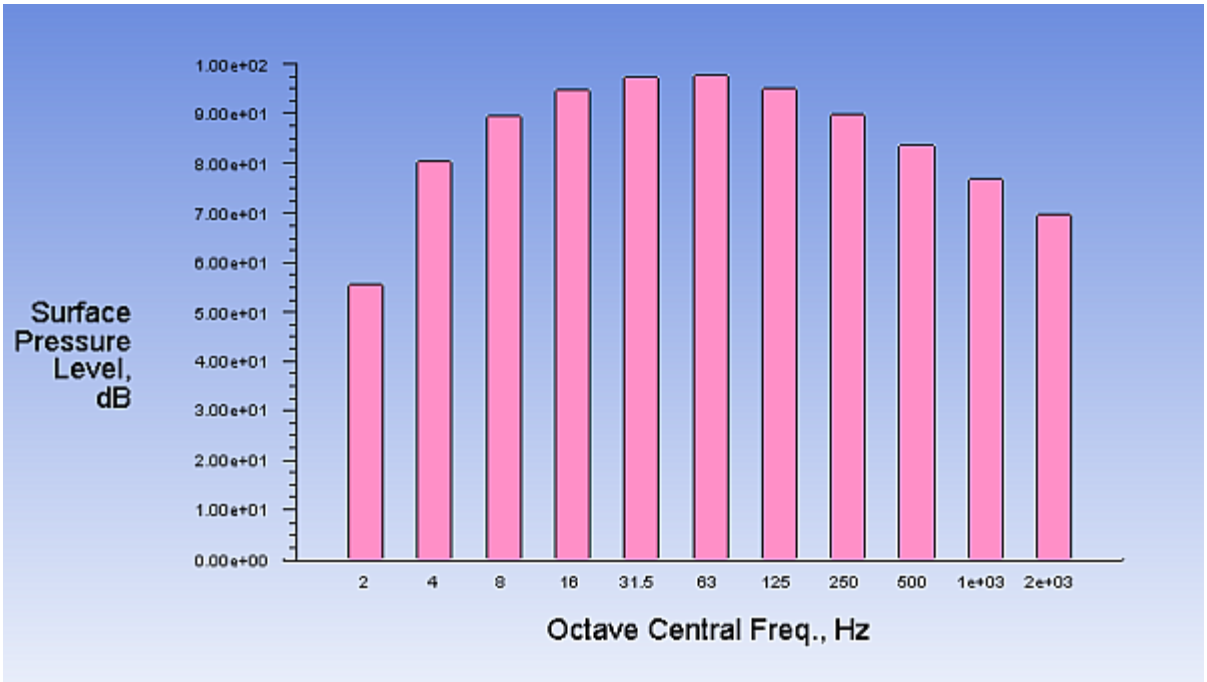
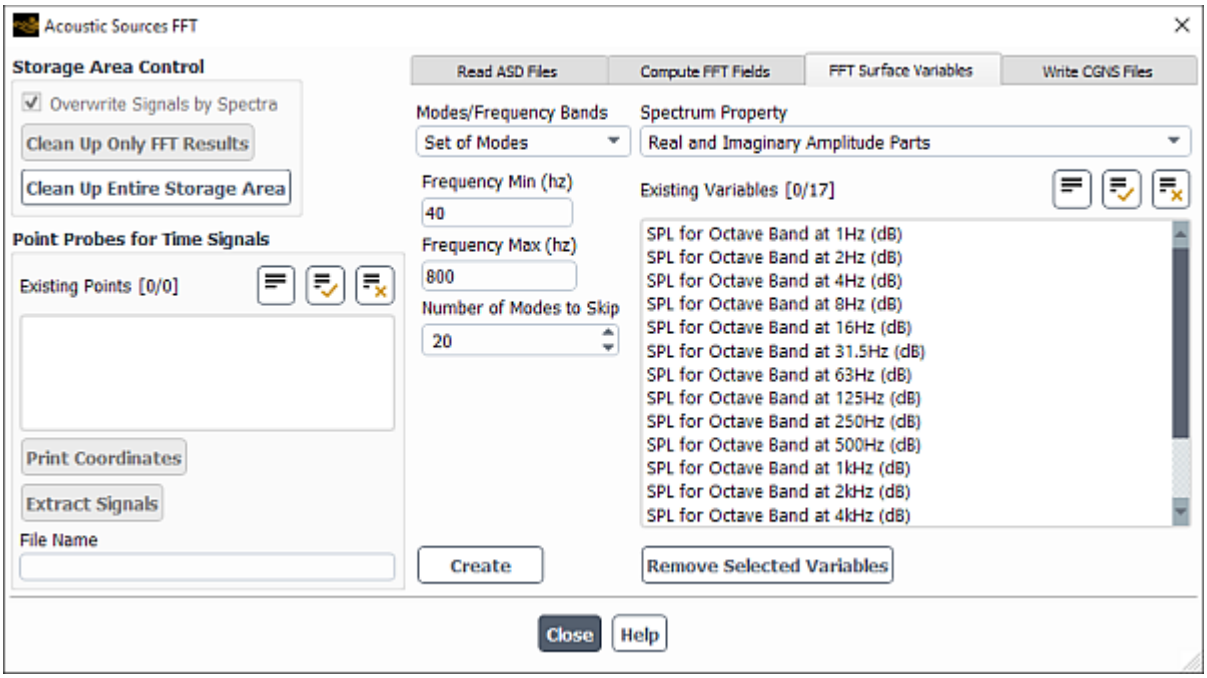


Figure 23.14: The FFT Surface Variables Tab of the Acoustic Source FFT Dialog Box for a Set of Individual Modes



The allowed number of variables makes it possible for you to simultaneously keep SPL and PSD of dp/dt for all octave and 1/3 octave bands, as well as SPL and PSD of dp/dt for up to 20 constant width bands and the real and imaginary amplitudes for up to 20 individual frequencies. The SPL and PSD of dp/dt variables for the octaves and the 1/3 octaves include the band central frequency in their names (for example, **SPL for Octave Band at 250Hz (dB)** or **PSD of dp/dt for 1/3-Octave Band at 1.25kHz**). As for the user-defined frequencies and constant width bands, their variables are simply numbered from 0 to 19. In order to provide you with the information about the meaning

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

of each such variable, Fluent prints a table in the console when you click **Create**. The following is an example of the table corresponding to the settings in the previous figure:

```
Creating variables for 20 modes from 40.5983 Hz to 466.88 Hz every 22.4359 Hz.
f( 0) = 40.5983 Hz     f( 1) = 63.0342 Hz
f( 2) = 85.4701 Hz     f( 3) = 107.906 Hz
f( 4) = 130.342 Hz     f( 5) = 152.778 Hz
f( 6) = 175.214 Hz     f( 7) = 197.65 Hz
f( 8) = 220.085 Hz     f( 9) = 242.521 Hz
f(10) = 264.957 Hz     f(11) = 287.393 Hz
f(12) = 309.829 Hz     f(13) = 332.265 Hz
f(14) = 354.701 Hz     f(15) = 377.137 Hz
f(16) = 399.573 Hz     f(17) = 422.009 Hz
f(18) = 444.444 Hz     f(19) = 466.88 Hz

Variables for frequencies above 466.88 Hz are not created,
because the limit of 20 modes has been reached.
```

You can analyze more than 20 individual modes or constant width bands, if you process them by portions of 20 items and delete the processed variables by selecting them in the **Existing Variables** list and clicking the **Remove Selected Variables** button.

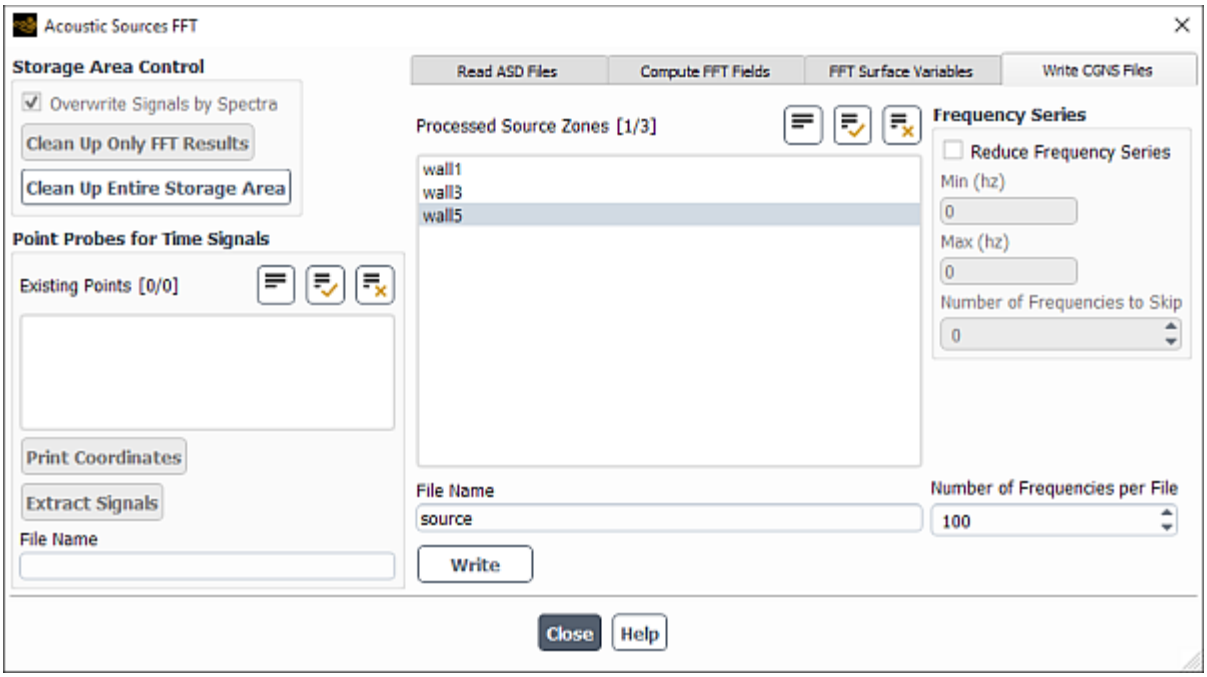
Clicking either the **Clean Up Entire Storage Area** or the **Clean Up Only FFT Results** button on the left side of the dialog box not only de-allocates the spectrum storage array, but also removes all created variables.

The **Write CGNS Files** tab is shown in Figure 23.15: The Write CGNS Files Tab of the Acoustic Source FFT Dialog Box (p. 2652). This tab allows you to specify the zones you want to export to CGNS, by selecting them in the **Processed Source Zones** list. You can reduce the exported spectrum data by enabling the **Reduce Frequency Series** option and specifying the frequency range (**Min** and **Max** frequencies), as well as the frequency step (the **Number of Frequencies to Skip** integer field). The latter must be used with care and only if really needed, because the Fourier amplitudes in the broadband noise spectrum depend on the frequency resolution. By skipping frequencies from the exported spectrum (for example, exporting every other Fourier mode), you will artificially coarsen the frequency resolution. However, the exported amplitudes will not be automatically re-scaled for the increased frequency step, but will retain their originally computed values.

Pressure spectra, which result from the scale-resolving flow simulations, may contain thousands of frequencies. Together with dense surface meshes, this can result in a very large amount of data to export. The total size of the disk space required to store fields of the complete spectra is approximately equal to the total size of the ASD files containing the wall pressure histories. Therefore, the wall pressure spectra are written to a series of CGNS files according to the **Number of Frequencies per File** field. It is recommended that you select an appropriate value for this parameter, to avoid extremely large output files or a large number of very small files. The total number of frequencies in the spectrum can be seen in the **Number of Modes** field of the **Compute FFT Fields** tab (see Figure 23.11: The Compute FFT Fields Tab of the Acoustic Source FFT Dialog Box (p. 2647)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.15: The Write CGNS Files Tab of the Acoustic Source FFT Dialog Box



Finally, you can specify a name for the exported CGNS files in the **File Name** field. For example, if you enter `wall_pressure_spectrum` and then click the **Write** button, Fluent will create the following set of files in your working folder:

wall_pressure_spectrum.cgns

wall_pressure_spectrum_1.cgns

wall_pressure_spectrum_2.cgns

...

wall_pressure_spectrum.flst

If you leave the **File Name** field empty, the files will be named **cgns**, **1.cgns**, **2.cgns**, **...**, **flst**. The first CGNS file (without a number in its name) contains the mesh data, as well as links to all of the other CGNS files, which in turn contain the spectrum fields with the specified number of modes per file. The CGNS links work like the file links in the UNIX file system: in order to import spectra to the Ansys Mechanical software, you only have to specify the name of the first file, and all other files will be imported automatically if they reside in the same folder. The file with the extension **flst** is an ASCII file, which contains the list of the exported frequencies.

Important:

Note the following:

- You should not specify a relative path as a part of the **File Name** input. If you do that, all files will be exported in the desired folder according to the specified path. However, the CGNS links, which are written in the first CGNS file, will also include the specified

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

path. Therefore, the linked CGNS files will become readable only after the correspondent file relocation—for example, moving the first CGNS file in your Fluent working folder.

- The public domain CGNS library, which is used to process CGNS files in Fluent and Ansys Mechanical, has a performance issue when run on Windows. Namely, for CGNS files that have many links (as is the case in this application), the input / output operations may be very slow if the CGNS files are located on a network disk. Therefore, for Windows it is strongly recommended that you select either a built-in local disk or a USB disk directly connected to your computer as the location of the CGNS files. This warning concerns both the export of CGNS files from Fluent and their import in Ansys Mechanical.

23.3. Using the Acoustics Wave Equation Model

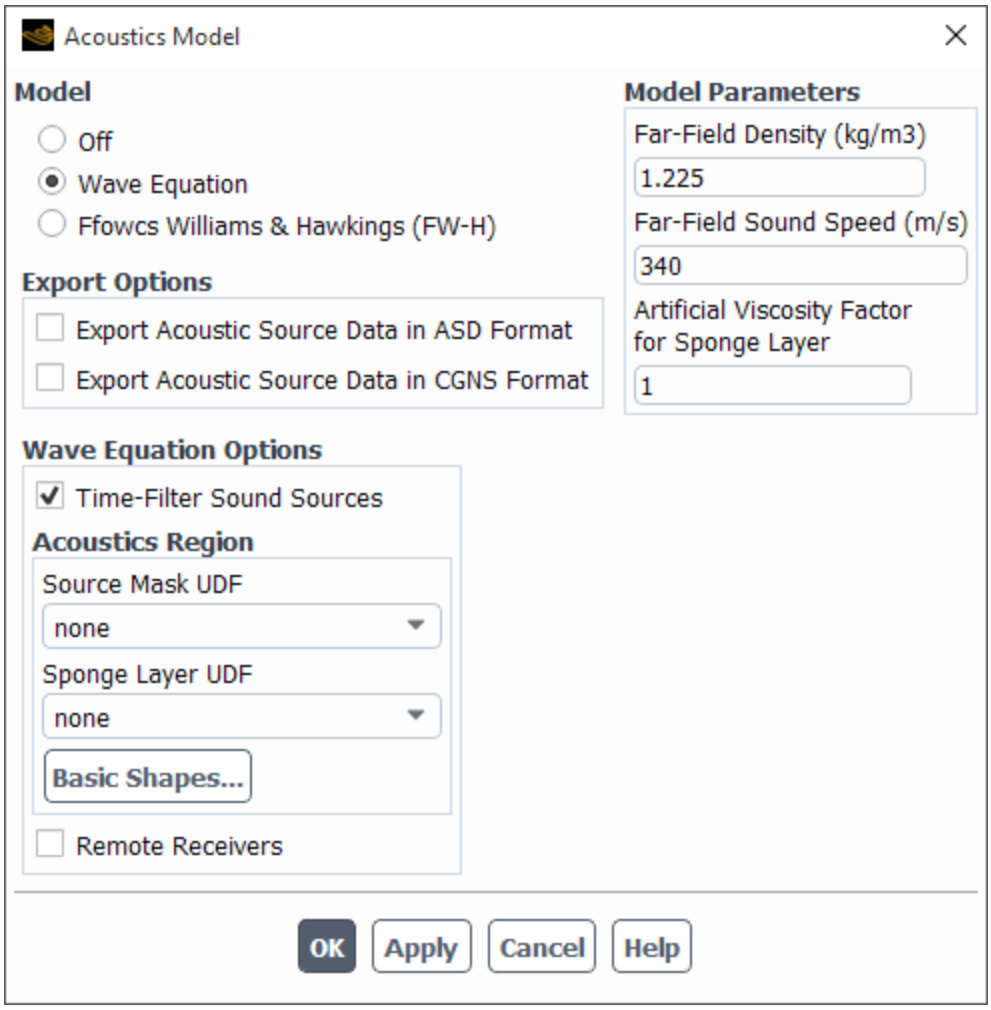
The acoustics wave equation model is recommended to be used with scale-resolving simulations of turbulent flows at low Mach numbers (LES, SBES, DES, SAS). Due to the current limitation of constant density and sound speed in the background fluid flow, no significant variation of these two parameters due to non-uniform temperature or mixture composition is allowed.

It is highly recommended that you read Preventing Non-Physical Reflections of Sound Waves in the *Fluent Theory Guide* for strategies to improve simulation quality.

The wave equation model is available for transient flow simulations with the second order or bounded second order implicit time scheme active. The model is not available for the 2-D axisymmetric version of Fluent.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.16: The Acoustics Model Dialog Box



You can enable the **Wave Equation** using the TUI command:

```
/define/models/acoustics/wave-equation?
```

The wave equation options are found in the menu:

```
/define/models/acoustics/wave-equation-options
```

The field **Artificial Viscosity Factor for Sponge Layer** is used to specify parameter C, which participates in Equation 11.13. The base level of artificial viscosity δ can be changed by the TUI command:

```
/define/models/acoustics/wave-equation-options/sponge-layer-base-level
```

The far-field parameters can be set using the TUI command:

```
/define/models/acoustics/far-field-parameters
```

23.3.1. Specifying Source Mask and Sponge Regions

The **Acoustics Region** group box is used to set up the geometry of the source mask and sponge regions.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

There are two methods to define geometry data:

- Preparing UDFs
- Use of basic shapes (rectangular hexahedron, cylinder, sphere)

The two methods may be combined together. The source mask and the sponge layer UDFs are created using the following macros:

```
DEFINE_SOURCE_MASK(src_mask_define, c, t)
```

```
DEFINE_SPONGE_LAYER(sponge_define, c, t)
```

where `src_mask_define` and `sponge_define` are example function names, `c` is a cell number, and `t` is a pointer to a cell thread. There can only be one UDF of each of these two types per simulation. Each UDF returns a single real value, which fills a corresponding storage variable (SV_SOUND_MODEL_SRC_MASK or SV_SOUND_SPONGE) in the cell `c` of the thread `t`.

Simple geometries can be built out of the basic shapes (rectangular hexahedron, cylinder, sphere), which must be preliminarily defined as cell registers of the type **Region**.

Solution → **Cell Registers** **New** → **Region...**

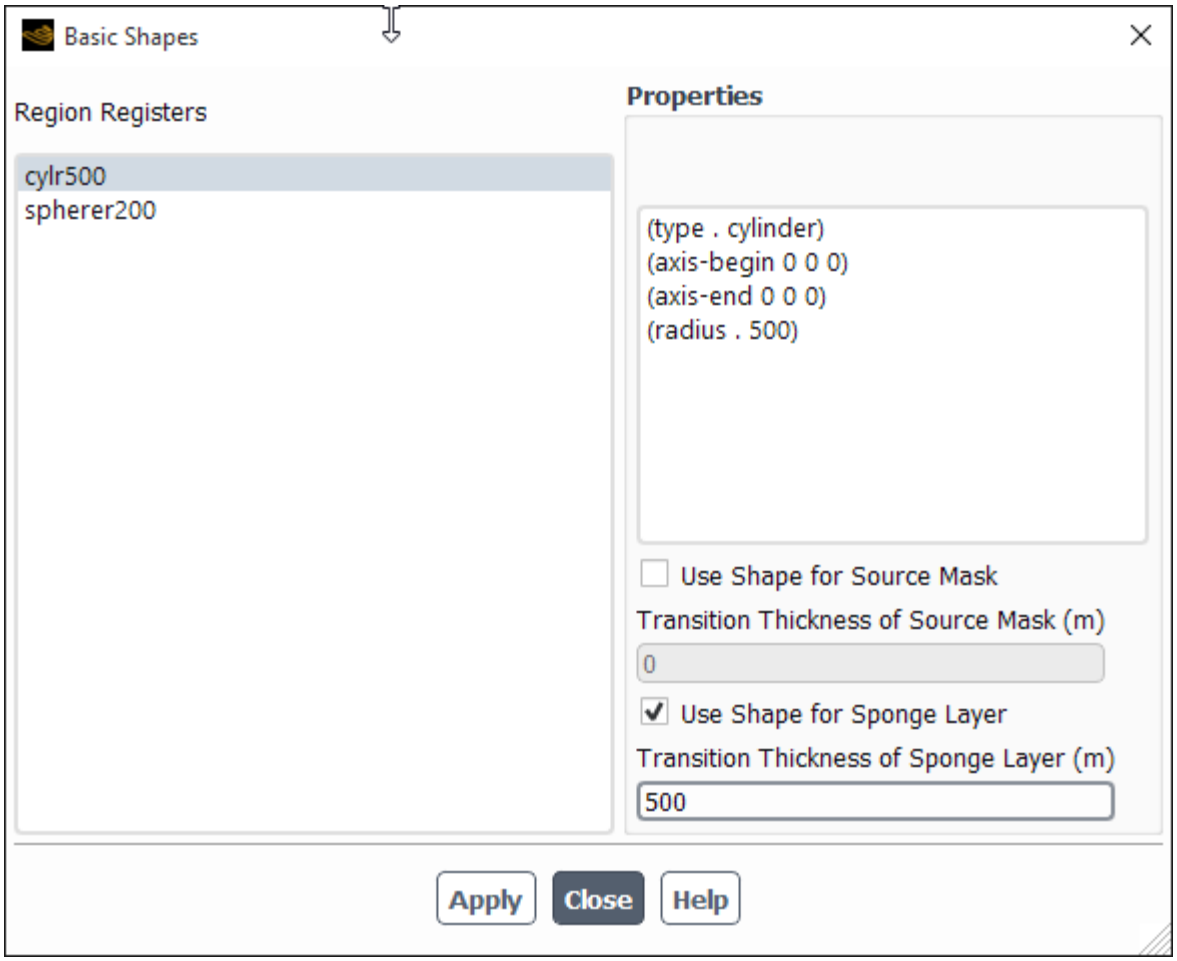
After you create one or more cell registers, you can use them to specify the desired geometry by clicking **Basic Shapes...** in the **Acoustics Model** dialog box (Figure 23.16: The Acoustics Model Dialog Box (p. 2654)), which opens the **Basic Shapes** dialog box (Figure 23.17: The Basic Shapes Dialog Box (p. 2656)). In this window you can add cell registers to either the source mask or sponge layer geometry definitions, and specify the individual transition thickness for each shape for each purpose.

Note:

After the cell registers have been used for the source mask and sponge layer geometry definitions, subsequent editing or deleting of these cell registers do not influence the already specified source mask and sponge layer geometry.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.17: The Basic Shapes Dialog Box



The TUI commands for creating and editing source mask and sponge layer geometry can be found in the menu:

```
/define/models/acoustics/wave-equation-options/basic-shapes/
```

To visualize the source mask and the sponge layer geometries, use the field variables **Sound WaveEq Model Source Mask** and **Sound Sponge Layer Marker** from the **Acoustics...** variable group. These output fields directly show the storage variables SV_SOUND_MODEL_SRC_MASK and SV_SOUND_SPONGE without any transformations.

Note:

The sponge layer described in this section is part of the wave equation model and is based on the viscosity. An alternative sponge layer model that is based on density is also available for compressible flows that use the pressure-based solver, as described in Sponge Layers (p. 2667).

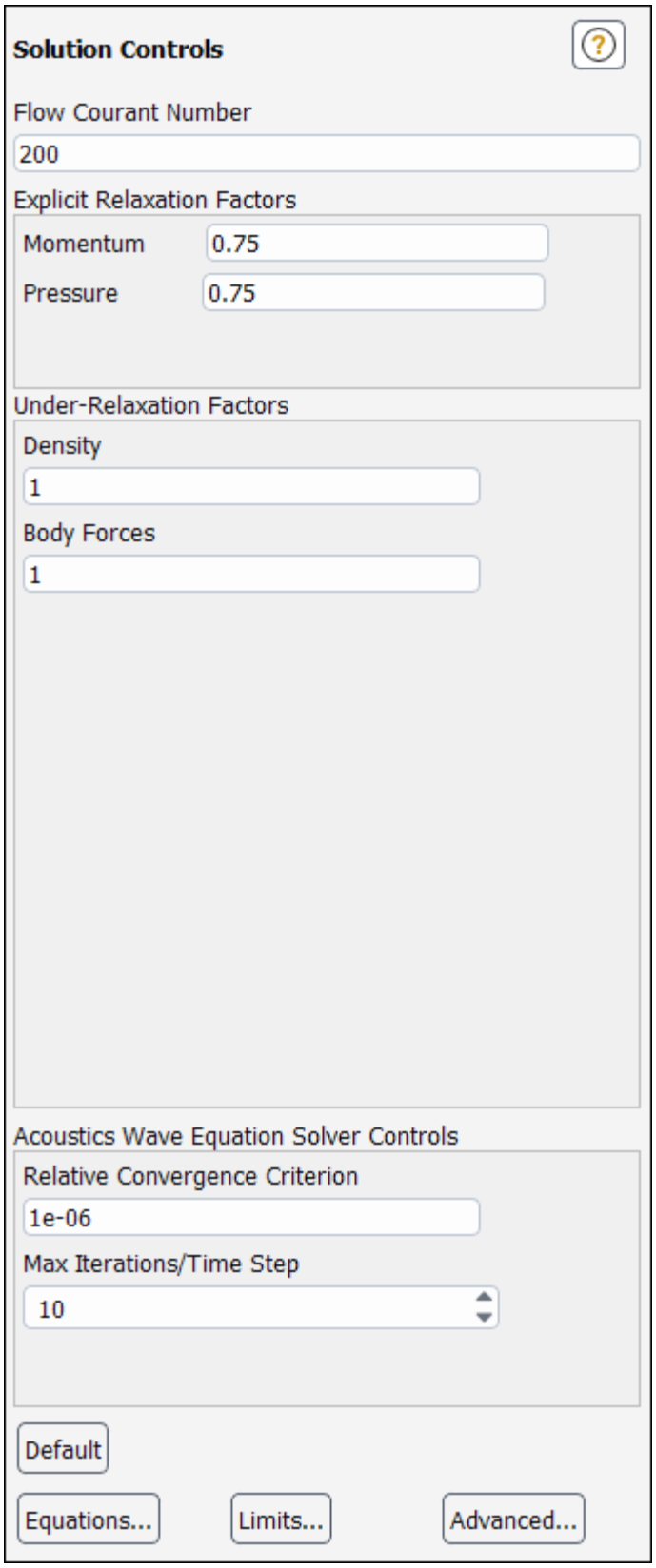
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

23.3.2. Solution Controls for the Acoustics Wave Equation

When the wave equation model is active, the **Solution Controls** task page includes an additional frame titled **Acoustics Wave Equation Solver Controls** (see Figure 23.18: The Acoustics Wave Equation Solver Controls Task Page (p. 2658)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.18: The Acoustics Wave Equation Solver Controls Task Page



The parameters **Relative Convergence Criterion** and **Max Iterations/Time Step** control the sub-iterations of the implicit wave equation solver. The need for sub-iterations comes from explicit skewness

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

corrections to the discretization of the basic Laplacian term in Equation 11.11, as well as the artificial viscosity term. At each sub-iteration, the system of linear equations is processed by the AMG linear solver to reduce the residual norm by a factor of 0.1 (this can be controlled by changing the rp-variable `acoustics-waveeq/amg-alpha`). The optimal values of these three algorithm parameters may depend on the mesh quality. Their default values may change in future releases after accumulating more experience with the wave equation model.

The TUI commands for the acoustics wave equation solution controls are found in the menu:

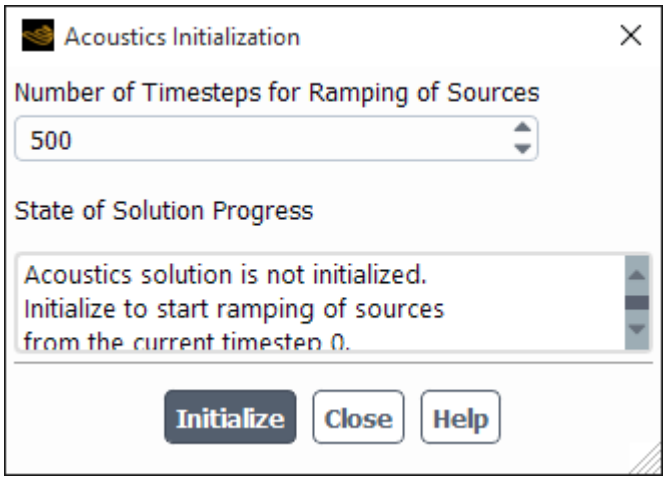
```
/solve/set/acoustics-wave-equation-controls/
```

The sub-menu `/expert` allows you to specify under-relaxation factors to enforce convergence on low quality meshes. Under normal circumstances, under-relaxation is not recommended.

23.3.3. Solution Initialization

The wave equation model is to be activated after the development of a time-dependent flow solution. Therefore the acoustics field requires a separate initialization process, that includes ramping in time the sound source term found in Equation 11.11. With the wave equation model enabled, a **Sound Potential** field and a new button **Initialize Acoustics...** appear on the **Solution Initialization** task page. The initial **Sound Potential** value should always be kept at zero. The **Initialize Acoustics...** button opens the **Acoustics Initialization** dialog box (Figure 23.19: The Acoustics Initialization Dialog Box (p. 2659)), where you can specify the number of timesteps for the ramping process.

Figure 23.19: The Acoustics Initialization Dialog Box



You can re-initialize the acoustics field separately at any time, since it does not influence the transient flow solution which is simply continued further.

You can specify the number of timesteps for the ramping process using the TUI command:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

/solve/initialize/init-acoustics-options

Note:

If you enable the wave equation model in a non-initialized setup, then the initialization of the entire simulation will also initialize acoustics. However the ramping duration will be set to zero. You can reset the ramping process by re-initializing acoustics separately.

The User Defined Function of the type `DEFINE_INIT` `DEFINE_INIT` cannot be used to initialize the wave equation model. Please use instead the `DEFINE_ON_DEMAND` function `DEFINE_ON_DEMAND`.

23.3.4. Postprocessing

The following field variables are available in the **Acoustics...** group for post-processing:

- **Sound Pressure**

  – main practical interest, connected to the sound potential by Equation 11.12. The sound pressure field is less smooth than the sound potential field because of numerical differentiation.

- **Sound Potential**

  – primary solution variable.

- **Sound DP/Dt**

  – time derivative of the sound pressure (a second time derivative of the primary solution variable). This field may display a lot of noise.

- **Sound WaveEq Model Source**

  – original model source term.

- **Sound WaveEq Model Source Smoothed**

  – source term smoothed by the time filter.

- **Sound WaveEq Model Source Mask**

  – source masking marker.

- **Sound Sponge Layer Marker**

  – sponge layer marker.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also postprocess results using the **Ansys Sound Analysis** dialog box. Refer to Postprocessing the FW-H Acoustics Model Data (p. 2641) for additional information.

Note:

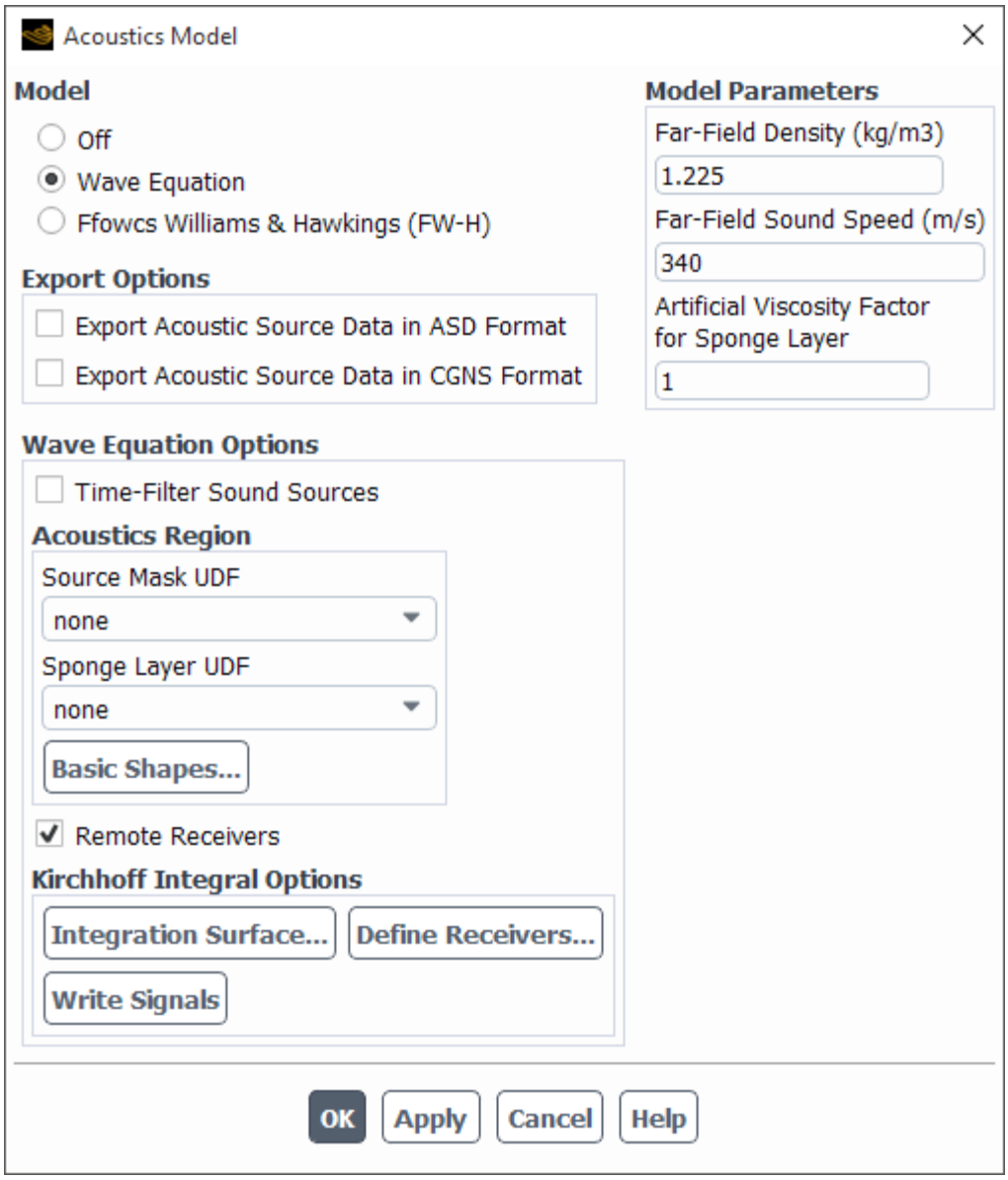
Both the **Sound WaveEq Model Source** and **Sound WaveEq Model Source Smoothed** do not include the effects of source masking and ramping. In order to see the source field after the application of masking and ramping, you can use the advanced output field **Sound WaveEq Source** in the **Acoustics...** group, which can be enabled by entering `yes` to the following text command: `/solve/set/advanced/retain-temporary-solver-mem`.

23.3.5. Using the Kirchhoff Integral Model

The Kirchhoff integral model is available for a three-dimensional acoustics simulation with the wave equation model. It is activated by selecting the checkbox, **Remote Receivers**, in the **Acoustics Model** dialog box. When this checkbox is selected, a subframe, **Kirchhoff Integral Options**, with additional buttons appear in the **Wave Equation Options** frame, as shown in Figure 23.20: Acoustics Model Dialog Box with Kirchhoff Integral Options (p. 2662).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

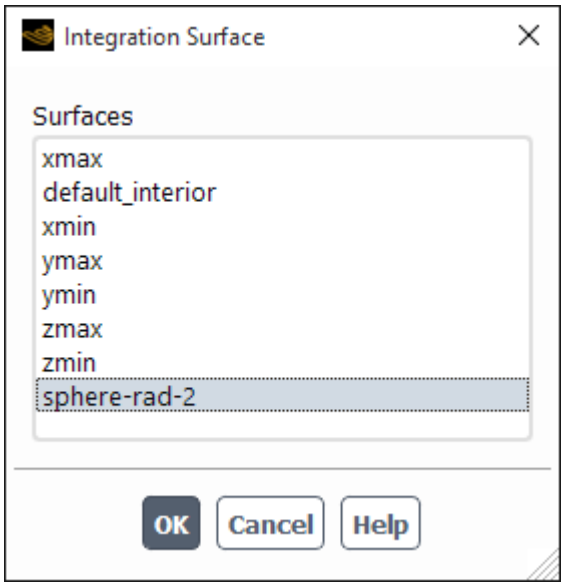
Figure 23.20: Acoustics Model Dialog Box with Kirchhoff Integral Options



Clicking **Integration Surface...** opens a list of all currently available surfaces, see an example in Figure 23.21: Integration Surface Dialog Box (p. 2663). The two most useful surface types for the Kirchhoff model are quadric surfaces (ribbon menu **Domain→Surface→Create→Quadric**) and iso-surfaces. With the current implementation only one surface can be selected in the list and used for the integration.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 23.21: Integration Surface Dialog Box



Clicking **Define Receivers...** opens the same **Acoustic Receivers** dialog box that is used to set up the Ffowcs Williams and Hawkings (FW-H) model, see Specifying Acoustic Receivers (p. 2638). For convenience, to simplify the simultaneous use of the FW-H model and the wave equation-based Kirchhoff integral model, receivers are shared between the two models and do not need to be set up twice. The signal file name extension for the Kirchhoff model is fixed to `.ark`.

Clicking **Write Signals** writes the computed sound signals at the remote receiver locations to ASCII files. The written Kirchhoff signals are always auto-pruned, see Pruning the Signal Data Automatically (p. 2643) for the details. If a sound signal is too short and does not yet contain the complete sound pressure data, then it is not written, and a warning message is printed in the Fluent console.

Important:

- The **Moving Receivers** option is supported only by the FW-H model. In the current implementation of the Kirchhoff model this option is ignored.
- The resulting signals are stored in memory and written in the data file separately for the FW-H and Kirchhoff model, which enables their simultaneous use and their comparison. When, however, the sound signals are being written in ASCII receiver files (see entry fields **Signal File Name** in Figure 23.7: The Acoustic Receivers Dialog Box (p. 2638)), both models share the same user-specified file name roots with the different model-specific extensions: `.ard` for the FW-H model signals, and `.ark` for the Kirchhoff model signals.

The Kirchhoff model is currently available only in "on-the-fly" mode, which means that the next set of signal samples is computed during a transient simulation after each time step. The "read-and-compute" mode for the FW-H model makes it possible to use the same transient flow simulation for both the FW-H and the wave equation model (with the Kirchhoff integral). To perform such a simulation:

1. Select the **Wave Equation** model and mark both the **Export Acoustic Source Data in ASD Format** and **Remote Receivers** checkboxes, as shown in Figure 23.20: Acoustics Model Dialog Box with Kirchhoff Integral Options (p. 2662).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Specify integration surfaces for the **FW-H** model by clicking **Define Sources...** under the **Export Options** frame.

3. Specify an integration surface for the Kirchhoff model by selecting **Integration Surface...** in the **Kirchhoff Integral Options** frame.

4. During a transient flow simulation, at every timestep, the acoustics wave equation will be solved and the sound signals at the remote receivers will be advanced in time using the Kirchhoff integral model. Source data for the FW-H model will also be exported in the ASD format. After the end of the simulation, write the Kirchhoff signals to ASCII files by clicking **Write Signals**.

5. Switch to the **FW-H** model, and then on the **Run Calculations** task page, start the FW-H model computations with the input from the ASD files using the **Acoustic Signals...** button. More details about the "read-and-compute" mode of the FW-H model are available in Writing Source Data Files (p. 2632).

The text commands for the Kirchhoff model, which can be used in the Fluent console and in journal files, are shown here in bold.

```
/define/models/acoustics/wave-equation-options>
basic-shapes/                                sponge-layer-base-level
remote-receivers-options/                    sponge-layer-factor
remote-receivers?                            sponge-layer-udf
source-mask-udf                              time-filter-source?


/define/models/acoustics/wave-equation-options/remote-receivers-options>
integration-surface         write-signals
```

23.4. Using the Broadband Noise Source Models

In this section, the procedure for setting up and using the broadband noise source models is outlined first, followed by descriptions of each of the steps involved.

The general procedure for carrying out a broadband noise source calculation in Ansys Fluent is as follows:

1. Calculate a steady or unsteady RANS solution.

2. Enable the broadband noise model and set the associated model parameters.

   Setup → **Models** → **Acoustics** **Edit...**

3. Postprocess the noise sources.

   Results → **Graphics** → **Contours** **Edit...**

For additional information, see the following sections:

23.4.1. Enabling the Broadband Noise Source Models

23.4.2. Postprocessing the Broadband Noise Source Model Data

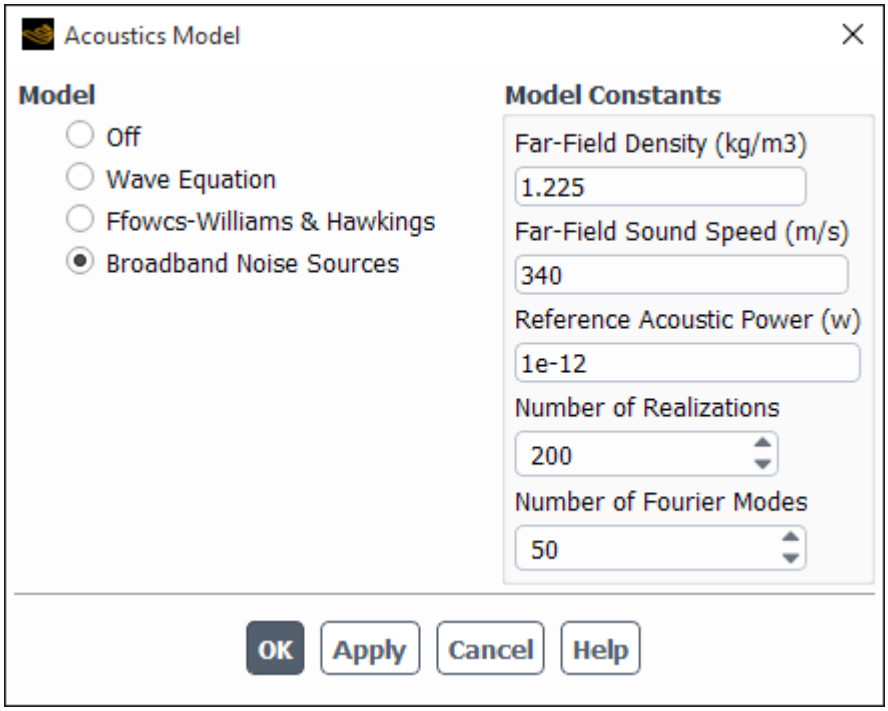
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

23.4.1. Enabling the Broadband Noise Source Models

To enable the broadband noise sources models, select **Broadband Noise Sources** in the **Acoustics Model** dialog box (Figure 23.22: The Acoustics Model Dialog Box for Broadband Noise (p. 2665)).

Setup → Models → Acoustics Edit...

Figure 23.22: The Acoustics Model Dialog Box for Broadband Noise



23.4.1.1. Setting Model Constants

Under **Model Constants** in the **Acoustics Model** dialog box, specify the relevant acoustic parameters and constants used by the model. See Enabling the FW-H Acoustics Model (p. 2629) for the definitions of **Far-Field Density** and **Far-Field Sound Speed**.

Reference Acoustic Power

(for example, P_{ref} in Equation 11.19 in the Theory Guide) is used to compute the acoustic power outputs in decibels (dB). The default value is 10^{-12}. Note that the units for the reference acoustic power will be different in 2D (W/m^2) and 3D (W/m^3) cases.

Number of Realizations

is the number of samples used in the SNGR to compute the averaged source terms of LEE and Lilley's equations. The default value is 200.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Number of Fourier Modes

(N in Equation 11.39 in the Theory Guide) is the number of the Fourier modes used to compute the turbulent velocity field and its derivatives. The turbulent velocity field is then used to compute the LEE and Lilley's source terms. The default value is 50.

23.4.2. Postprocessing the Broadband Noise Source Model Data

The final step in the broadband noise source modeling process is the postprocessing of acoustic power and noise source data. The following variables are available in the **Acoustics...** postprocessing category:

- **Acoustic Power Level (dB)**
- **Acoustic Power**
- **Jet Acoustic Power Level (dB)** (axisymmetric models only)
- **Jet Acoustic Power** (axisymmetric models only)
- **Surface Acoustic Power Level (dB)**
- **Surface Acoustic Power**
- **Lilley's Self-Noise Source**
- **Lilley's Shear-Noise Source**
- **Lilley's Total Noise Source**
- **LEE Self-Noise X-Source**
- **LEE Shear-Noise X-Source**
- **LEE Total Noise X-Source**
- **LEE Self-Noise Y-Source**
- **LEE Shear-Noise Y-Source**
- **LEE Total Noise Y-Source**
- **LEE Self-Noise Z-Source** (3D models only)
- **LEE Shear-Noise Z-Source** (3D models only)
- **LEE Total Noise Z-Source** (3D models only)

You can also postprocess results using the **Ansys Sound Analysis** dialog box. Refer to Postprocessing the FW-H Acoustics Model Data (p. 2641) for additional information.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

23.5. Sponge Layers

In order to eliminate non-physical pressure wave reflections from boundary zones in a transient simulation for a single-phase, compressible flow with the pressure-based solver, you can designate a layer of the domain adjacent to sets of boundary zones as sponge layers. Within each sponge layer, the density is blended from the value calculated by the solver to a user-specified far-field value. A quadratic blending function gives more weight to the far-field value over a defined ramping length that begins at the interior boundary of the sponge layer and extends toward the boundaries. Such density-based sponge layers are an alternative to the viscosity-based sponge layer (which is only available with the acoustics wave equation model, as described in Using the Acoustics Wave Equation Model (p. 2653)) and the boundary acoustic wave models such as NRBCs (as described in Boundary Acoustic Wave Models (p. 1527)).

Note:

Density-based sponge layers are not supported with a multiphase model.

To use density-based sponge layers, perform the following steps:

1. Select **Pressure-Based** for the solver **Type** in the **General** task page.

   Setup → **General**

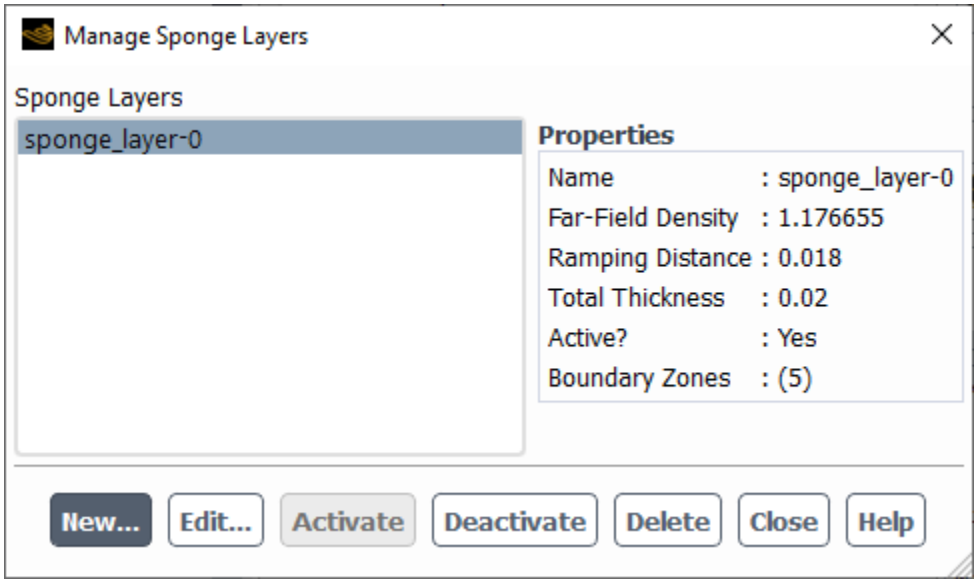
2. Select **Transient** from the **Time** list in the **General** task page.

3. Define a compressible fluid by making an appropriate selection from the drop-down list to the right of **Density** in the Create/Edit Materials Dialog Box (p. 4885), such as the ideal-gas law, real gas law, or compressible liquid.

4. Open the **Manage Sponge Layers** dialog box.

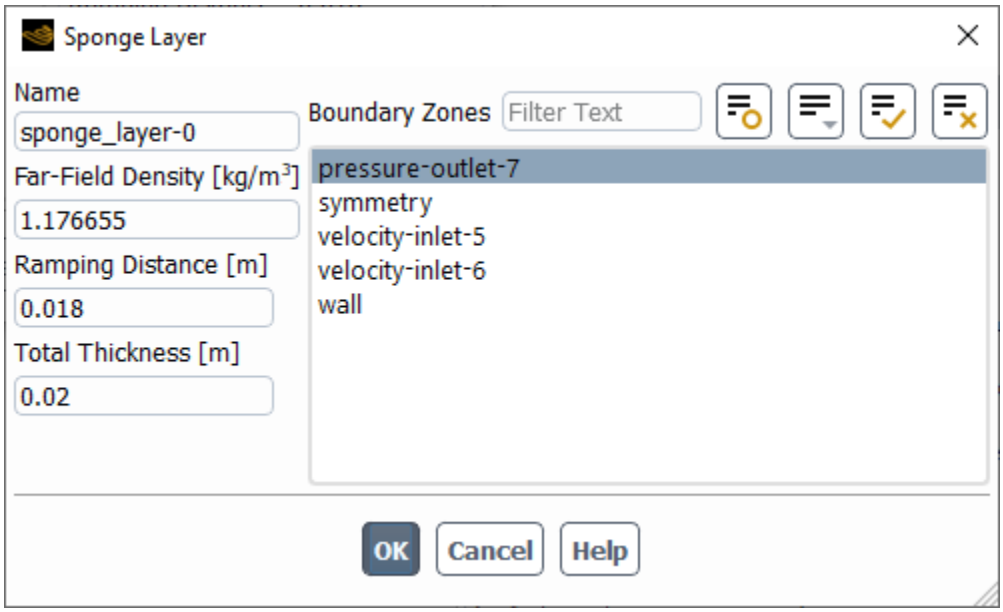
   Physics → **Models** → **Acoustics** → **Sponge Layers...**

Figure 23.23: The Manage Sponge Layers Dialog Box



a. Click the **New...** button to open the **Sponge Layer** dialog box, where you can define a sponge layer.

Figure 23.24: The Sponge Layer Dialog Box



i. Enter a **Name** for this sponge layer definition.

ii. Enter a value for the **Far-Field Density** that is used for blending with the solver value. It is recommended that you use the average density at the boundaries throughout the simulation.

iii. Enter a value for the **Ramping Distance**, a distance over which the density transitions from the solver value to the **Far-Field Density**. It is recommended that this distance is 80–100% of the **Total Thickness**.

iv. Enter a value for the **Total Thickness** of the sponge layer, a distance from the selected boundaries to the interior boundary of the sponge layer. It is recommended that this thickness is at least twice the wavelength of the expected pressure waves.

v. Make selections from the **Boundary Zones** list to specify where you want the sponge layer applied.

vi. Click **OK** to close the **Sponge Layer** dialog box.

b. You can repeat the previous step if you would like additional sponge layer definitions. Whenever you create a new sponge layer, it will automatically be set as active.

Fluent marks sponge layer cells based on the distance from a cell to a sponge layer boundary zone. If there are multiple active sponge layers, then they might overlap. For such cases, Fluent prints a warning message and assigns each cell in the overlapping regions to the sponge layer that has the most distant interior boundary.

c. After you have created one or more definitions, you can use the **Manage Sponge Layers** dialog box to take additional actions prior to running the calculation. If you make a selection from the **Sponge Layers** list, the **Properties** of the definition are displayed and you can use the buttons at the bottom of the dialog box:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Click **Edit...** to revise the definition of the current selection in the **Sponge Layer** dialog box.
- Click **Activate** to make the current selection active.
- Click **Deactivate** to make the current selection inactive.
- Click **Delete** to delete the selected definition.

5. It can be useful to set up a solution animation of an XY plot of the static pressure near the boundaries, so that you can evaluate if your sponge layers are indeed eliminating the wave reflections. For details, see Animating the Solution (p. 3714) and XY Plots of Solution Data (p. 4060).

6. After initializing the calculation, the following field variables are available for postprocessing (in the **Mesh...** category):

   - **Sponge Layer Distance**

     This is the distance from each cell inside a sponge layer to the interior boundary of a sponge layer. It changes linearly from 0 at the interior boundary to the thickness value of the sponge layer at the boundary zone.

   - **Sponge Blending Function**

     This shows how a blending function changes across a sponge layer, where a value of 0 means the solver density is used and a value of 1 means the far-field value is used.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 24: Modeling Discrete Phase

This chapter describes how to use the Lagrangian discrete phase capabilities available in Ansys Fluent. For information about the theory behind the discrete phase models, see Discrete Phase in the Theory Guide. Information is organized into the following sections:

24.1. Introduction

24.2. Steps for Using the Discrete Phase Models

24.3. Setting Initial Conditions for the Discrete Phase

24.4. Setting Boundary Conditions for the Discrete Phase

24.5. Particle Erosion Coupled with Dynamic Meshes

24.6. Modeling Lagrangian Wall Films

24.7. Setting Material Properties for the Discrete Phase

24.8. Solution Strategies for the Discrete Phase

24.9. Postprocessing for the Discrete Phase

24.10. Parallel Processing for the Discrete Phase Model

24.1. Introduction

In addition to solving transport equations for the continuous phase, Ansys Fluent allows you to simulate a discrete second phase in a Lagrangian frame of reference. This second phase consists of spherical particles (which may be taken to represent droplets or bubbles) dispersed in the continuous phase. Ansys Fluent computes the trajectories of these discrete phase entities, as well as heat and mass transfer to/from them. The coupling between the phases and its impact on both the discrete phase trajectories and the continuous phase flow can be included.

Ansys Fluent provides the following discrete phase modeling options:

- calculation of the discrete phase trajectory using a Lagrangian formulation that includes the discrete phase inertia, hydrodynamic drag, and the force of gravity, for both steady and unsteady flows
- prediction of the effects of turbulence on the dispersion of particles due to turbulent eddies present in the continuous phase
- heating/cooling of the discrete phase
- vaporization and boiling of liquid droplets
- combusting particles, including volatile evolution and char combustion to simulate coal combustion
- optional coupling of the continuous phase flow field prediction to the discrete phase calculations
- droplet breakup and coalescence

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- consideration of particle/particle collisions and voidage of discrete phase

These modeling capabilities allow Ansys Fluent to simulate a wide range of discrete phase problems including particle separation and classification, spray drying, aerosol dispersion, bubble stirring of liquids, liquid fuel combustion, and coal combustion. The physical equations used for these discrete phase calculations are described in Discrete Phase in the *Fluent Theory Guide*, and instructions for setup, solution, and postprocessing are provided in the remaining sections of this chapter.

Alternative models for multiphase systems use the Euler-Euler approach rather than the Euler-Lagrange approach used in the Discrete Phase Model. The Euler-Euler models are discussed in Modeling Multiphase Flows (p. 2915) along with the Dense Discrete Phase Model (DDPM) which is a hybrid Euler-Euler and Euler-Lagrange approach.

The Lagrangian discrete phase model can be connected to the Eulerian VOF model via VOF-to-DPM model transfer mechanisms as described in Using the VOF-to-DPM Model Transition for Dispersion of Liquid in Gas (p. 3020). For example, in the simulation of liquid sprays, approximately spherical liquid structures in the VOF solution can be converted automatically into Lagrangian particle parcels. This, in conjunction with dynamic solution-adaptive mesh refinement, allows for a detailed, yet time-efficient simulation of primary atomization, such as, for example, in gas turbines and internal combustion engines.

For additional information, see the following sections:

24.1.1. Concepts

24.1.2. Limitations

24.1.1. Concepts

This section introduces several concepts in the treatment of discrete phase particles in Fluent that are important to understand in order to get the most out of the remaining information in this chapter.

24.1.1.1. Uncoupled vs. Coupled DPM

24.1.1.2. Steady vs. Unsteady Tracking

24.1.1.3. Parcels

24.1.1.1. Uncoupled vs. Coupled DPM

When the fluid changes the particles, there will be corresponding effects on the fluid. For example, when drag force acts on a particle, the exchange of momentum can change the fluid flow. When simulating particles using DPM, you can choose whether or not to include these effects in the flow solution; these alternatives are called Coupled and Uncoupled DPM. See Options for Interaction with the Continuous Phase (p. 2678) for details about how to specify whether your simulation uses Coupled or Uncoupled DPM.

In Uncoupled DPM, the only purpose of the DPM particles is for postprocessing, and so particles are not tracked except when you request them, for example to calculate and display particle tracks. The particles can still change by heat and mass transfer, but the corresponding changes (such as vapor from an evaporating droplet) do not affect the flow solution.

In a Coupled DPM simulation, the effects of the particles are used to influence the flow solution. These effects are transmitted to the flow as DPM Sources. The DPM solution and the flow solution should reach a converged, self-consistent solution. Therefore, there are several options for running these solutions together—see Solution Strategies for the Discrete Phase (p. 2827).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.1.1.2. Steady vs. Unsteady Tracking

As described in Steps for Using the Discrete Phase Models (p. 2677), to set up a DPM simulation you specify the starting conditions of a set of particles by defining an injection. By specifying boundary conditions and physical sub-models, you also specify how these particles interact with other zones in the geometry and how they eventually leave the model—for instance, they might bounce off some walls but be trapped by others.

You must also specify how Fluent is to track the particles you have defined. If Steady Tracking is enabled, then, as soon as a particle is released, it is tracked until it reaches its final destination according to the specified boundary behavior (or until a fixed number of particle time steps have been used). Therefore, each particle typically travels through many cells of the model, interacting with the flow and (in a Coupled DPM simulation) changing the DPM Sources in each cell. These sources influence the flow solution for a defined number of iterations or time steps—the flow solution can be steady or unsteady. Then, if required, a new set of particle trajectories is tracked, the DPM Sources are updated, and the sequence is repeated. An example of using Steady DPM with unsteady flow is when the chosen flow models require a transient simulation, although the final goal is a steady solution.

If Unsteady Tracking is enabled, then each particle is advanced by a specified number of particle time steps, not necessarily reaching a final destination, before the flow solution is updated. When Unsteady DPM is coupled to unsteady flow solution, the particles and the flow develop in time together concurrently, although different time steps can be used for DPM and flow.

Unsteady DPM can also be coupled to steady flow solution; this makes sense if there is a continuous source of DPM particles that pass through the system. For steady or unsteady flow, there are several DPM models where Unsteady Tracking is required. For example, in spray coalescence and collision models, particles change with time on the basis of interactions with other particles, so they must be tracked simultaneously

24.1.1.3. Parcels

Especially when using Coupled DPM, the mass flow rate of a particle injection will often be a required and relevant input parameter since it determines the absolute value of the DPM Sources. This mass flow rate could be converted into the number of particles injected per unit time. However, it is often prohibitive to track that number of particles in a simulation. Strictly speaking, the model therefore tracks a number of 'parcels', and each parcel is representative of a fraction of the total continuous mass flow rate (in Steady tracking) or a fraction of the total mass flow released in a time step (in Unsteady tracking).

It is still sometimes helpful to refer to each parcel as a representative particle, because it has a specified particle diameter, and its trajectory in fluid flow uses the relaxation time appropriate for a single particle. (The relaxation time is a ratio of particle momentum to drag force). However, the parcel's mass (or mass flow rate) becomes important when calculating the DPM Sources: for example, if a representative droplet loses a small amount of vapor by evaporation, the overall effect from the whole parcel will typically be much larger. Other models also use the parcel mass (or mass flow rate) to calculate total concentrations of DPM material—in particular, the Dense Discrete Phase Model (DDPM) uses this concentration to feed into the volume fraction of the Eulerian phase that represents the same material. See Dense Discrete Phase Model in the *Fluent Theory Guide* for details of the Dense Discrete Phase Model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The concept of parcels is particularly important in the Discrete Element Method (DEM), where parcels occupy a finite volume and obstruct other DEM parcels. The volume occupied by a parcel is calculated directly from the mass that it represents (so that a realistic density is created when parcels pack together). The equivalent 'parcel diameter' is used for calculating parcel-parcel contacts and forces. However, for trajectories through fluid, it is still the 'particle diameter' that is used. See Discrete Element Method Collision Model in the *Fluent Theory Guide* for details of the Discrete Element Model.

The number of parcels in a DPM model is chosen in the model settings, and not defined by the true number of particles. There are several inputs that can be used to adjust the number of parcels when defining initial conditions such as the number of injection locations and (for Unsteady Tracking) the injection frequency (Setting Initial Conditions for the Discrete Phase (p. 2718)) Other inputs in sub-models include: the number of sizes in a size distribution (Using the Rosin-Rammler Diameter Distribution Method (p. 2745)); the number of stochastic tries in turbulent dispersion (Specifying Turbulent Dispersion of Particles (p. 2772)); and the breakup characteristics of some sprays (Breakup (p. 2769)). A high number of parcels can be computationally expensive, but it is often helpful for convergence, so that no single parcel has an overwhelming effect on the flow In general you should arrange for enough parcels to produce a statistical sample, representative of the full range of particle behavior.

24.1.2. Limitations

The discrete phase model limitations are provided in the following subsections.

24.1.2.1. Limitation on the Particle Volume Fraction

24.1.2.2. Limitation on the Particle Knudsen Number

24.1.2.3. Limitation on Modeling Continuous Suspensions of Particles

24.1.2.4. Limitations on Modeling Particle Rotation

24.1.2.5. Limitations on Using the Discrete Phase Model with Other Ansys Fluent Models

24.1.2.6. Limitations on Using the Hybrid Parallel Method

24.1.2.1. Limitation on the Particle Volume Fraction

The discrete phase formulation used by Ansys Fluent contains the assumption that the second phase is sufficiently dilute that particle-particle interactions and the effects of the particle volume fraction on the gas phase are negligible. In practice, these issues imply that the discrete phase must be present at a fairly low volume fraction, usually less than 10–12%. Note that the mass loading of the discrete phase may greatly exceed 10–12%: you may solve problems in which the mass flow of the discrete phase equals or exceeds that of the continuous phase. See Modeling Multiphase Flows (p. 2915) for information about when you might want to use one of the general multiphase models instead of the discrete phase model.

This limitation is relaxed for some variants of DPM. For example, the Dense Discrete Phase Model (DDPM) adds effects due to friction and volume fraction, so that the concentration can approach the packing limit. Where high local concentrations of spray droplets cause coalescence and collision, these phenomena can be included in some spray models. Parcel-parcel contacts between solid particles are modeled in detail in Discrete Element Models, so that parcels can pack together closely.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.1.2.2. Limitation on the Particle Knudsen Number

In general, the discrete phase model is limited to the continuum regime where the mean free path of the continuous phase gas molecules is much smaller than the particle diameter (Knudsen Number $Kn = \lambda / diam \ll 1$).

For situations where the mean free path of the gas phase is the same order of magnitude as the particle diameter, you can use the Stokes-Cunningham drag law. See Stokes-Cunningham Drag Law in the *Fluent Theory Guide* for additional information on this drag law. The discrete phase model should not be applied to the free-molecular flow regime ($Kn \gg 1$). Note that by default, Ansys Fluent limits the smallest particle diameter to 10 nm. If you want to model particles below this limit, you can set the minimum particle diameter using the `define/models/dpm/options/set-minimum-particle-diameter` text command.

24.1.2.3. Limitation on Modeling Continuous Suspensions of Particles

The steady-particle Lagrangian discrete phase model is suited for flows in which particle streams are injected into a continuous phase flow with a well-defined entrance and exit condition. The Lagrangian model does not effectively model flows in which particles are suspended indefinitely in the continuum, as occurs in solid suspensions within closed systems such as stirred tanks, mixing vessels, or fluidized beds. The unsteady-particle discrete phase model, however, is capable of modeling continuous suspensions of particles. See Modeling Multiphase Flows (p. 2915) for information about when you might want to use one of the general multiphase models instead of the discrete phase models.

24.1.2.4. Limitations on Modeling Particle Rotation

When using rotating particles, note the following limitations:

- In simulations with enabled stochastic particle collision models (Including Collision and Droplet Coalescence (p. 2700)), particle rotation is not affected by particle/particle collisions.
- Particle rotation is not available for massless particles.
- Particle rotation is not compatible with moving reference frame simulations.
- For atomizer injections, the initial angular velocity is set to zero.
- The effect of the Magnus lift force on the fluid is not taken into account in DPM coupled simulations.

24.1.2.5. Limitations on Using the Discrete Phase Model with Other Ansys Fluent Models

The following restrictions exist on the use of other models with the discrete phase model:

- When tracking particles with the DPM model in combination with any of the multiphase flow models (VOF, mixture, or Eulerian—see Modeling Multiphase Flows (p. 2915)) the **Shared Memory** method cannot be selected (Parallel Processing for the Discrete Phase Model (p. 2888)). (Note that using the **Message Passing** or **Hybrid** method enables the compatibility of all multiphase flow models with the DPM model.)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- When using the DPM model with the Eulerian multiphase model, the tracked particles rely only on the primary phase to compute drag, heat, and mass transfer. Also, any DPM related source terms are applied to the primary phase. Particle tracking relative to a secondary phase is not provided.
- Streamwise periodic flow (either specified mass flow rate or specified pressure drop) cannot be modeled with steady particle tracks in coupled simulation. It is possible using transient particle tracks.
- Only non-reacting particles can be included when the premixed combustion model is used.
- Surface injections will be moved with the mesh when a sliding mesh or a moving or deforming mesh is being used, however only those surfaces associated with a boundary will be recalculated. Injections from cut plane surfaces will not be moved with the mesh and will be deleted when remeshing occurs.
- The wall film model is only valid for liquid materials. If a nonliquid particle interacts with a wall film boundary, the boundary condition will default to the reflect boundary condition.
- By default, particles are tracked relative to the local reference frame motion. This implies that particle injection velocities (specified in the **Set Injection Properties** dialog box) must also be defined relative to the reference frame motion in which the injection is located.

  If necessary, particle tracking can be done in the absolute frame of reference. To do so, you can either use the `define/models/dpm/options/track-in-absolute-frame` text command or enable the **Track in Absolute Frame** option in the **Discrete Phase Model dialog** box option (**Numerics** tab). In this case, the injection velocities must also be specified in the absolute frame.
- Relative particle tracking cannot be used in combination with sliding and moving deforming meshes. If sliding and/or deforming meshes are used with the DPM model, the particles will always be tracked in the absolute frame. Switching to the relative frame is not permitted.
- Surface injections that use particle release surfaces that are not based on mesh zones are not compatible with the following models:
  - Mesh motion (sliding meshes)
  - Dynamic meshes
  - Automatic adaption
- When using the DPM model with an overset mesh, note the following:
  - The **High-Res Tracking** option must be enabled
  - Wall film particles cannot move between different overset meshes

24.1.2.6. Limitations on Using the Hybrid Parallel Method

- The hybrid parallel DPM tracking method is not available with the following models and features:
  - the PDF Transport model
  - the DDPM (Dense DPM) model with the **Volume Fraction Approaching Packing Limit** option selected in the **Multiphase Model** dialog box (**Phases** tab)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - Graphical display of particle tracks
  - Export of particle tracks into particle history files

  If any of these models or features are used, Fluent will silently fall back to the **Message Passing** parallel DPM tracking option.

- The **Use DPM Domain** option of the **Hybrid** parallel DPM tracking method will not be used in any of the following situations:
  - if the Eulerian Wall Film model is enabled
  - if the PDF Transport model is enabled
  - if the case has a dynamic zone
  - if the case has a sliding interface
  - if the case includes non-zonal surface injections (that is, injections that have positions defined through surfaces that are not zone surfaces)

  If the **Use DPM Domain** option is enabled in such cases, Ansys Fluent will issue a message warning that the DPM Domain is suppressed. You can avoid this message by disabling the **Use DPM Domain** option in the **Parallel** tab of the **Discrete Phase Model** dialog box.

  For additional limitations associated with the hybrid parallel method, see Limitations on Using the Discrete Phase Model with Other Ansys Fluent Models (p. 2675).

24.2. Steps for Using the Discrete Phase Models

You can include a discrete phase in your Ansys Fluent model by defining the initial position, velocity, size, and temperature of individual particles. These initial conditions, along with your inputs defining the physical properties of the discrete phase, are used to initiate trajectory and heat/mass transfer calculations. The trajectory and heat/mass transfer calculations are based on the force balance on the particle and on the convective/radiative heat and mass transfer from the particle, using the local continuous phase conditions as the particle moves through the flow. The predicted trajectories and the associated heat and mass transfer can be viewed graphically and/or alphanumerically.

The procedure for setting up and solving a problem involving a discrete phase is outlined below, and described in detail in Options for Interaction with the Continuous Phase (p. 2678) – Postprocessing for the Discrete Phase (p. 2833). Only the steps related specifically to discrete phase modeling are shown here. For information about inputs related to other models that you are using in conjunction with the discrete phase models, see the appropriate sections for those models.

1. Enable any of the discrete phase modeling options, if relevant, as described in Options for Interaction with the Continuous Phase (p. 2678).
2. Choose a transient or steady treatment of particles as described in Steady/Transient Treatment of Particles (p. 2679).
3. Specify tracking parameters as described in Tracking Settings for the Discrete Phase Model (p. 2683).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Enable the required physical submodels for the discrete phase model, as described in Physical Models for the Discrete Phase Model (p. 2695).

5. Set the numerics parameters and solve the problem, as described in Numerics of the Discrete Phase Model (p. 2710) and Solution Strategies for the Discrete Phase (p. 2827).

6. Specify the injection-specific models, initial conditions, and particle size distributions as described in Setting Initial Conditions for the Discrete Phase (p. 2718).

7. Define the boundary conditions, as described in Setting Boundary Conditions for the Discrete Phase (p. 2779).

8. Define the material properties, as described in Setting Material Properties for the Discrete Phase (p. 2811).

9. Initialize the flow field.

10. Solve the coupled or uncoupled flow (Solution Strategies for the Discrete Phase (p. 2827)).

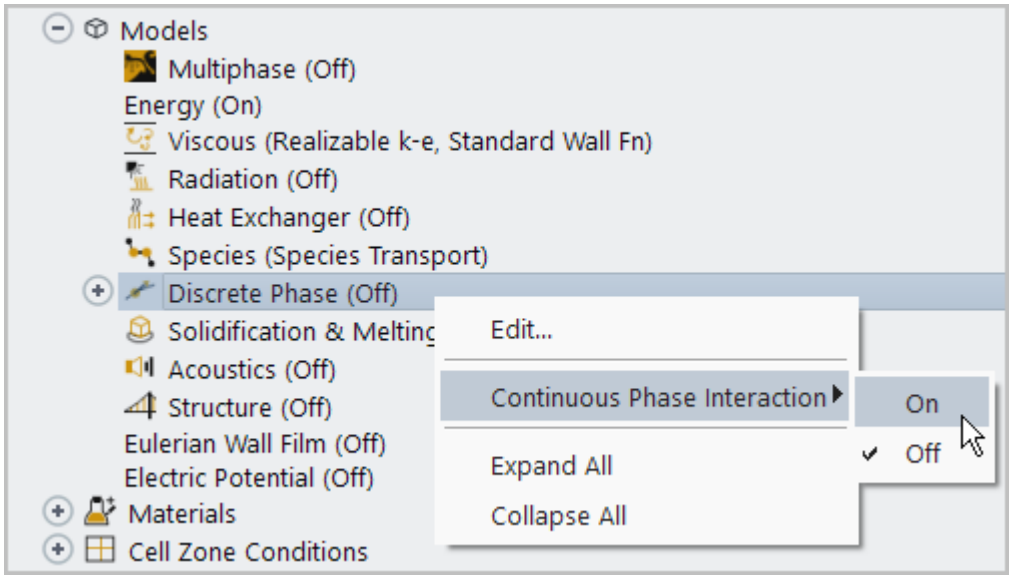
11. For transient cases, advance the solution in time by taking the desired number of time steps. Particle positions will be updated as the solution advances in time. If you are solving an uncoupled flow, the particle position will be updated at the end of each time step. For a coupled calculation, the positions are iterated on or within each time step.

12. Examine the results, as described in Postprocessing for the Discrete Phase (p. 2833).

24.2.1. Options for Interaction with the Continuous Phase

If the discrete phase interacts (that is, exchanges mass, momentum, and/or energy) with the continuous phase, you should enable **Interaction with Continuous Phase**.

Setup → Models → Discrete Phase → Continuous Phase Interaction → On



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

An input for the **DPM Iteration Interval** will appear, which allows you to control the frequency at which the particles are tracked. The DPM source terms are computed at every DPM Iterations. The DPM sources applied to the flow equations remain constant between two DPM Iterations. The amount of source considered by the solver is controlled by the under-relaxation factor specified in the **Solution Control** task page (see Under-Relaxation of the Interphase Exchange Terms (p. 2830)).

For steady-state simulations, increasing the **DPM Iteration Interval** will increase stability but might require more iterations to converge.

The additional option **Update DPM Sources Every Flow Iteration** allows you to control how DPM sources are applied to the continuous phase equations. When this option is enabled, the sources are successively updated in the flow equations at every flow solver iteration according to Equation 12.512 through Equation 12.514 in the *Fluent Theory Guide* without repeated tracking of the particles. For unsteady simulations, this option is default and recommended. Given that the flow solver uses sufficient iterations per time step (see Under-Relaxation of the Interphase Exchange Terms (p. 2830)), the option ensures that the complete DPM sources are consumed within a time step.

Note:

When the **Update DPM Sources Every Flow Iteration** option is enabled, the value reported for DPM source terms (**Discrete Phase Sources...** category) contains the full source term.

24.2.2. Steady/Transient Treatment of Particles

The Discrete Phase Model uses a Lagrangian approach to derive the equations for the underlying physics, which are solved transiently. Transient numerical procedures in the Discrete Phase Model can be applied to resolve steady flow simulations as well as transient flows.

In the Discrete Phase Model Dialog Box (p. 4845) you have the option of choosing whether you want to treat the particles in an unsteady or a steady fashion. This option can be chosen independent of the settings for the solver. Thus, you can perform steady-state trajectory simulations even when selecting a transient solver for numerical reasons. You can also specify unsteady particle tracking when solving the steady continuous phase equations. This can be used to improve numerical stability for very large particle source terms or simply for postprocessing purposes. Whenever you enable a breakup or collision model to simulate sprays, the **Unsteady Particle Tracking** will be switched on automatically.

When **Unsteady Particle Tracking** is enabled, several new options appear. If steady-state equations are solved for the continuous phase, you simply enter the **Particle Time Step Size** and the **Number of Time Steps**, thus tracking particles every time a DPM iteration is conducted. When you increase the **Number of Time Steps**, the droplets penetrate the domain faster.

When solving unsteady equations for the continuous phase, you must decide whether you want to use **Fluid Flow Time Step** to inject the particles, or you prefer a **Particle Time Step Size** independent of the **Fluid Flow Time Step**. With the latter option, you can use the Discrete Phase Model in combination with changes in the time step for the continuous equations, as it is done when using adaptive flow time stepping.

If you do not use **Fluid Flow Time Step**, you must decide when to inject the particles for a new time step. You can either **Inject Particles at Particle Time Step** or at the **Fluid Flow Time Step**. In any case, the particles will always be tracked in such a way that they coincide with the flow time of the

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

continuous flow solver, as long as the maximum number of time steps used to compute a single trajectory is sufficient (see Tracking Settings for the Discrete Phase Model (p. 2683) for details).

You can use a user-defined function (`DEFINE_DPM_TIMESTEP`) to change the time step size for DPM particle tracking. The time step size can be prescribed for special applications where a certain time step size is needed. For more information about changing the time step size for DPM particle tracking, see `DEFINE_DPM_TIMESTEP` in the Fluent Customization Manual.

Important:

When the density-based solver is used with the explicit unsteady formulation, the particles are advanced once per time step and are calculated at the start of the time step (before the flow is updated).

Additional inputs are required for each injection in the Set Injection Properties Dialog Box (p. 5608), as detailed in Defining Injection Properties (p. 2756). For **Unsteady Particle Tracking**, the injection **Start Time** and **Stop Time** must be specified under **Point Properties**. Injections with start and stop times set to zero will be injected only at the start of the calculation ($t=0$). If the In-Cylinder mesh motion is enabled, the start and stop times are replaced by **Start Crank Angle** and **Stop Crank Angle**, respectively. The injection specified in this way will be repeated at the starting and stopping crank angle if the simulation is run through more than one cycle. Changing injection settings during a transient simulation will not affect particles currently released in the domain. At any point during a simulation, you can clear particles that are currently in the domain by clicking the **Clear Particles** button in the Discrete Phase Model Dialog Box (p. 4845).

You can choose the **Parcel Release Method** to determine how Ansys Fluent creates parcels. For an overview of the concept of parcels, see Parcels (p. 2673). The methods available are:

standard

injects a single parcel per injection stream per time step. The number of particles in the parcel, NP, is determined as follows:

$$NP=\dot{m}_s\frac{\Delta t}{m_p} \tag{24.1}$$

where,

NP is the number of particles in a parcel

$\dot{m}_s$ is the mass flow rate of the particle stream

Δt is the time step size

m_p is the particle mass

This is the default method.

constant-number

injects parcels with a user-specified number of particles per parcel. The number of parcels is determined to satisfy the specified mass-flow rate and particle size distribution for the injection.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

constant-mass

injects parcels with a user-specified parcel mass. The number of parcels is determined to satisfy the specified mass-flow rate and particle size distribution for the injection.

constant-diameter

injects parcels with a user-specified parcel diameter. The number of parcels is determined to satisfy the specified mass-flow rate and particle size distribution for the injection.

Note that for atomizer injections, **file** injections, or injections that use the `DEFINE_DPM_INJECTION_INIT` user defined function, the Fluent solver automatically uses the default standard parcel release method. Other parcel release methods are not available.

For cases involving sprays and particle size distributions in general, the recommended setting for **Parcel Release Method** is **constant-number**. For DEM simulations, you can use **constant-diameter** or **constant-mass** to ensure that the parcel diameter does not exceed the size of the smallest cells in the computational mesh.

Note that a lower value specified for **constant-number**, **constant-mass**, or **constant-diameter** will result in a larger number of parcels injected and a finer discretization of the DPM phase. This may be beneficial for accuracy and stability of the calculation, at the expense of additional computational cost.

The **Parcel Release Method** is specified in the **Parcel** tab of the **Set Injection Properties** dialog box.

You can also choose one of several methods to control when the particles are tracked.

- In steady-state simulations, particles are tracked at the user-specified intervals (**DPM Iteration Interval**). Individual particles are tracked from their injection position until either they escape the domain or certain termination criteria are met. Particle tracking is performed only if the global **Number of Iterations** is equal to or exceeds **DPM Iteration Interval**. The particle sources are updated during tracking, and then applied to the gas phase in subsequent flow iterations.

- In transient flow simulations, particle tracking always occurs at the beginning of each flow time step, irrespective of **DPM Iteration Interval**. The particles are advanced in time based on the current continuous-phase solution, and DPM sources are updated. The DPM sources prevail up to the next time the particle tracker is run.

- (transient flow) If the **DPM Iteration Interval** is larger than or equal to **Max Iterations/Time Step**, then particles are tracked only once per time step. If the **DPM Iteration Interval** is less than **Max Iterations/Time Step**, then the particle tracker is run multiple times during the time step. Each time the particle tracker is used, particles are returned to their original state at the beginning of the time step. Then particles are advanced in time based on the current continuous-phase solution, and DPM sources are updated. If **Particle Time Step** is larger than **Fluid Flow Time Step** (to allow injection of particles at a different time scale than the fluid), then particles are advanced only for the fluid flow time step.

- (transient flow) If you specify a value of zero as the **DPM Iteration Interval**, the particles are advanced at the end of the time step. Note that if you have previously enabled the resetting of DPM source terms at the beginning of every time step using the TUI command `define/models/dpm/interaction/reset-sources-at-timestep?`, Ansys Fluent will disable this

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

option once you set the **DPM Iteration Interval** to zero, because this feature combination is not advisable.

In cases where **Interaction with Continuous Phase** is enabled, you must provide a sufficient number of particle source term updates to ensure that particle source terms reach their final values (see Figure 24.58: Effect of Number of Source Term Updates on Source Term Applied to Flow Equations (p. 2831)). This can be achieved either by setting the DPM under-relaxation factor to 1 or in other cases by enabling **Update DPM Sources Every Flow Iteration**. If the latter is disabled and the DPM under-relaxation factor is less than 1, then the value you specify for **DPM Iteration Interval** must be small compared to **Max Iterations/Time Step**.

Important:

- In steady-state discrete phase modeling, particles do not interact with each other and are tracked one at a time in the domain.
- If the collision model is used, you will not be able to set the **DPM Iteration Interval**. Refer to Collision and Droplet Coalescence Model Theory in the *Fluent Theory Guide* for details about this limitation.

Note:

As explained in Setting Initial Conditions for the Discrete Phase (p. 2718), the injection flow rate is defined per unit meter of depth in 2D problems. In such cases, each injected parcel is considered to be a solid cylinder of diameter d_{2D} and depth 1 meter. The parcel volume is then expressed as:

$$V_{2D}=\frac{\pi}{4}{d_{2D}}^2\cdot 1\,m$$

However, the particles that are dense-packed in the parcel are assumed to be spheres rather than pucks (ice-hockey). The volume of each particle is the same as in 3D cases:

$$V_p=\frac{\pi}{6}{d_p}^3$$

Consider next the theoretical atomic packing factor β of spherical particles in a cylinder of one meter depth, also expressed as the volume ratio of all particles to all injected parcels.

In 2D, the particles of diameter d_p are stacked on top of one another in a cylinder of diameter d_p and depth 1 meter. The total number of particles that can be stacked in the cylinder in such a way is $1\,\mathrm{m}/d_p$. The total volume occupied by these particles is

$$V_1=\frac{\pi}{6}{d_p}^3\cdot 1\,\mathrm{m}/d_p$$

The volume of the surrounding parcel cylinder of depth 1 m is

$$V_2=\frac{\pi}{4}{d_p}^2\cdot\ 1\,\mathrm{m}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The atomic packing factor β will have a constant value of 2/3 when expressed as:

$$\beta = V_1 / V_2 = \left(\frac{\pi}{6} {d_p}^3 \cdot 1\,\mathrm{m} / d_p\right) \Big/ \left(\frac{\pi}{4} {d_p}^2 \cdot\ 1\,\mathrm{m}\right) = \frac{2}{3}$$

In order to use the same parameters (spring constant k and coefficient of restitution η) for the DEM spring dashpot models in both 3D and 2D simulations, parcels of the same mass must be injected in 3D and 2D.

In 3D, the injected parcel is a sphere with diameter d_{3D} and mass $m_{3D} = \rho_p \cdot \frac{\pi}{6} {d_{3D}}^3$. The number of spherical particles in a spherical parcel is calculated by

$$NP = m_{3D} / m_p = \rho_p \cdot \frac{\pi}{6} {d_{3D}}^3 \Big/ \rho_p \cdot \frac{\pi}{6} {d_p}^3$$

In 2D, the parcel is a cylinder of 1 m depth. This cylinder has the same mass as the spherical parcel in 3D ($m_{2D} = m_{3D}$), but its volume differs and is calculated as:

$$V_{prcl,2D} = \frac{\pi}{4} {d_{2D}}^2 \cdot\ 1\,\mathrm{m}$$

The number of particles per cylindrical parcel NP and the volume they occupy is the same as in 3D, that is

$$V_{p,2D} = V_{p,3D} = NP\ \frac{\pi}{6} {d_p}^3$$

Since the ratio of these two volumes must be equal to the theoretical atomic packing factor β, we obtain

$$\left(NP \cdot\ \frac{\pi}{6} {d_p}^3\right) \Big/ \left(\frac{\pi}{4} {d_{2D}}^2 \cdot\ 1m\right) =\ \beta = \frac{2}{3}$$

The parcel diameter in 2D can then be calculated as:

$$d_{2D} = \sqrt{NP \cdot\ {d_p}^3 / 1\,\mathrm{m}} = \sqrt{{d_{3D}}^3 / 1\,\mathrm{m}}$$

Note that in 2D, the parcel diameter d_{2D} is used as a collision range and it is different than the parcel diameter d_{3D} in 3D.

For each parcel release method, the input values for the parcel mass and parcel diameter (m_{3D} and d_{3D}, respectively) are the same in both 3D and 2D, while the output values for parcel diameters are different: d_{2D} in 2D simulations and d_{3D} in 3D simulations.

24.2.3. Tracking Settings for the Discrete Phase Model

You will use two parameters to control the time integration of the particle trajectory equations:

- the maximum number of time steps

  This factor is used to abort trajectory calculations when the particle never exits the flow domain.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

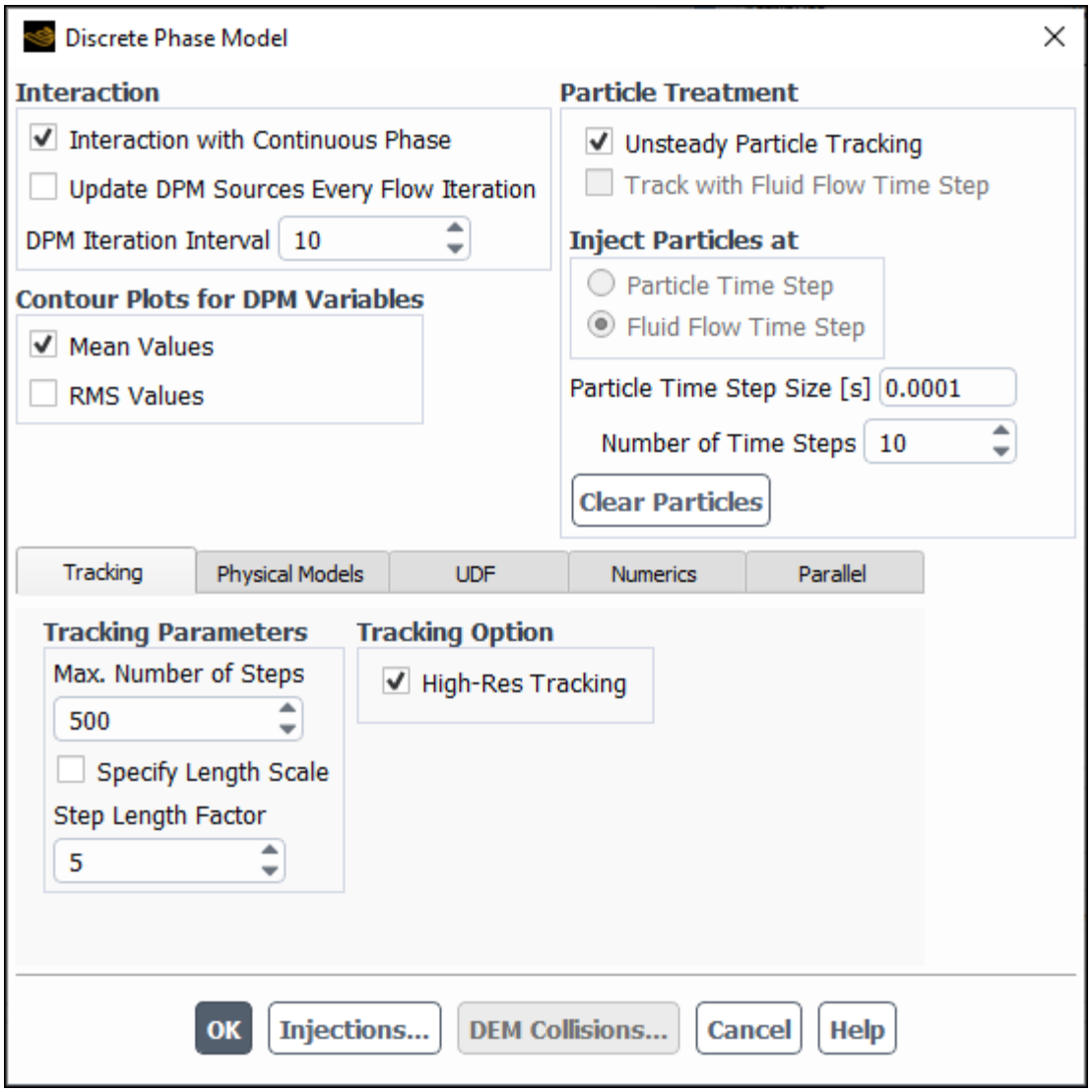
- the length scale/step length factor

  This factor is used to set the time step size for integration within each control volume.

Each of these parameters is set in the Discrete Phase Model Dialog Box (p. 4845) (Figure 24.1: The Discrete Phase Model Dialog Box - Tracking Tab (p. 2684)) under **Tracking Parameters** in the **Tracking** tab.

Setup → Models → Discrete Phase → Edit...

Figure 24.1: The Discrete Phase Model Dialog Box - Tracking Tab



For the **Tracking Parameters**, you can specify:

Max. Number of Steps

is the maximum number of time steps used to compute a single particle trajectory via integration of Equation 12.1 in the Theory Guide. When the maximum number of steps is exceeded, Ansys Fluent abandons the trajectory calculation for the current particle injection and reports the trajectory fate as "incomplete". The limit on the number of integration time steps eliminates the

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

possibility of a particle being caught in a recirculating region of the continuous phase flow field and being tracked infinitely. The default value for this parameter is 50,000 for steady-state particle tracking and 500 for unsteady particle tracking. Note that you may easily create problems in which the default value is insufficient for completion of the trajectory calculation. In this case, when trajectories are reported as incomplete within the domain and the particles are not recirculating indefinitely, you can increase the maximum number of steps (up to a limit of 10^9).

Note:

If, when opening a previous simulation with unsteady DPM particle tracking, you receive a warning reporting that a fraction of the discrete phase mass is "behind in time", you can use the reported percentage values to determine if it is a significant enough discrepancy to warrant revising the simulation to have a higher value for **Max. Number of Steps**.

Length Scale

controls the integration time step size used to integrate the equations of motion for the particle. The integration time step size is computed by Ansys Fluent based on a specified length scale L, and the velocity of the particle (u_p) and of the continuous phase (u_c):

$$\Delta t = \frac{L}{u_p + u_c} \tag{24.2}$$

where L is the **Length Scale** that you define. As defined by Equation 24.2 (p. 2685), L is proportional to the integration time step size and is equivalent to the distance that the particle will travel before its motion equations are solved again and its trajectory is updated. A smaller value for the **Length Scale** increases the accuracy of the trajectory and heat/mass transfer calculations for the discrete phase.

(Note that particle positions are always computed when particles enter/leave a cell; even if you specify a very large length scale, the time step size used for integration will be such that the cell is traversed in one step.)

Length Scale will appear in the **Discrete Phase Model** dialog box when the **Specify Length Scale** option is enabled.

Step Length Factor

also controls the time step size used to integrate the equations of motion for the particle. It differs from the **Length Scale** in that it allows Ansys Fluent to compute the time step size in terms of the number of time steps required for a particle to traverse a computational cell. To set this parameter instead of the **Length Scale**, turn off the **Specify Length Scale** option.

The integration time step size is computed by Ansys Fluent based on a characteristic time that is related to an estimate of the time required for the particle to traverse the current continuous phase control volume. If this estimated transit time is defined as Δt^*, Ansys Fluent chooses a time step size Δt as

$$\Delta t = \frac{\Delta t^*}{\lambda} \tag{24.3}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where λ is the **Step Length Factor**. As defined by Equation 24.3 (p. 2685), λ is inversely proportional to the integration time step size and is roughly equivalent to the number of time steps required to traverse the current continuous phase control volume. A larger value for the **Step Length Factor** decreases the discrete phase integration time step size. The default value for the **Step Length Factor** is 5. **Step Length Factor** will appear in the **Discrete Phase Model** dialog box when the **Specify Length Scale** option is off (the default setting).

For steady-state particle tracking, one simple general rule to follow when setting the parameters above is that if you want the particles to advance through a domain consisting of N mesh cells into the main flow direction, the **Step Length Factor** times N should be approximately equal to the **Max. Number of Steps**.

When **Accuracy Control** is enabled in the **Numerics** tab, the settings for **Step Length Factor** and **Length Scale** will be used only to estimate the time step size of the first integration step. In all subsequent integration steps, the particle integration time step size is adapted to achieve the tolerance specified in Numerics of the Discrete Phase Model (p. 2710).

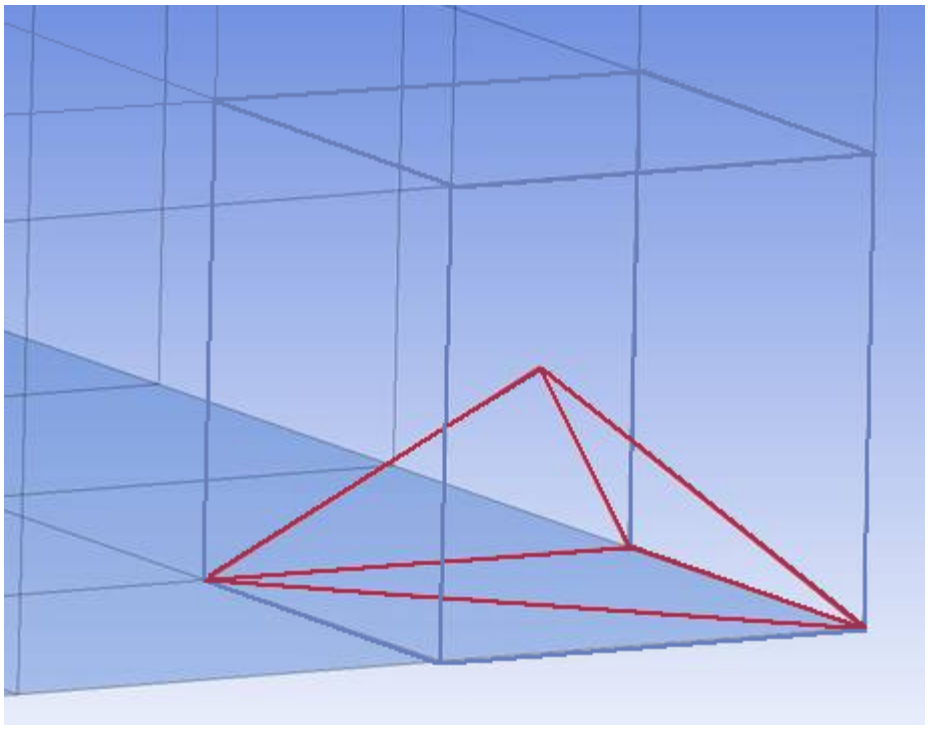
High-Resolution Tracking

When the **High-Res Tracking** option is enabled in the **Tracking Option** group box (default), the computational cells are decomposed into tetrahedrons (subtets) and the particles tracked through the subtets. This option provides a more robust tracking algorithm and improved variable interpolation. If **High-Res Tracking** is not enabled, particles are tracked through the computational cells directly. Flow variables that appear in particle equations either use cell-center values or a truncated Taylor series approximation for interpolation to the particle position.

Figure 24.2: A Subtet Formed From Decomposing a Hexagonal Cell (p. 2687) shows one of the twelve subtets produced by decomposing a hexagonal cell. In this example, the subtet is formed by the edges that connect the cell centroid and three of the eight cell nodes.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.2: A Subtet Formed From Decomposing a Hexagonal Cell



In conjunction with tracking particles through the subtets, the barycentric interpolation method is used to obtain the flow solution (such as, velocity and temperature) at the particle position. The barycentric interpolation involves linearly interpolating the flow variables from the subtet vertices to the particle position. This method provides a continuous interpolation field throughout the domain. The current interpolation method in Ansys Fluent uses the truncated Taylor series expansion, which can result in discontinuities at cell boundaries. When combined with high-resolution tracking, the barycentric interpolation provides improved accuracy and robustness and reduces the dependency on parallel partitioning. Barycentric interpolation is required for high-resolution tracking and will be enabled by Ansys Fluent automatically.

For cases with significant variations in viscosity or density (for example, cases that involve VOF or other multiphase models) the text command options `interpolate-flow-density?` and/or `interpolate-flow-viscosity?` described below must be enabled to ensure that these properties are continuous throughout the domain. Otherwise, discontinuous material properties may result in particles being trapped at interior cell faces. Ansys Fluent automatically enables these settings when multiphase models are enabled.

High-resolution tracking can be used with both free-stream particles and with Lagrangian wall film particles.

Note that when using high-resolution tracking, interpolation of flow gradients and temporal interpolation of the flow solution are not compatible. If both options are selected, Ansys Fluent disables the transient variable interpolation.

Once the high-resolution tracking is enabled through either the **Discrete Phase Model** dialog box or the text command

```
define/models/dpm/numerics/high-resolution-tracking/
```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
enable-high-resolution-tracking?
Activate high-resolution-tracking? [no] yes
```

the below text commands become available under the `high-resolution-tracking` text menu, depending on the mesh type and solver settings.

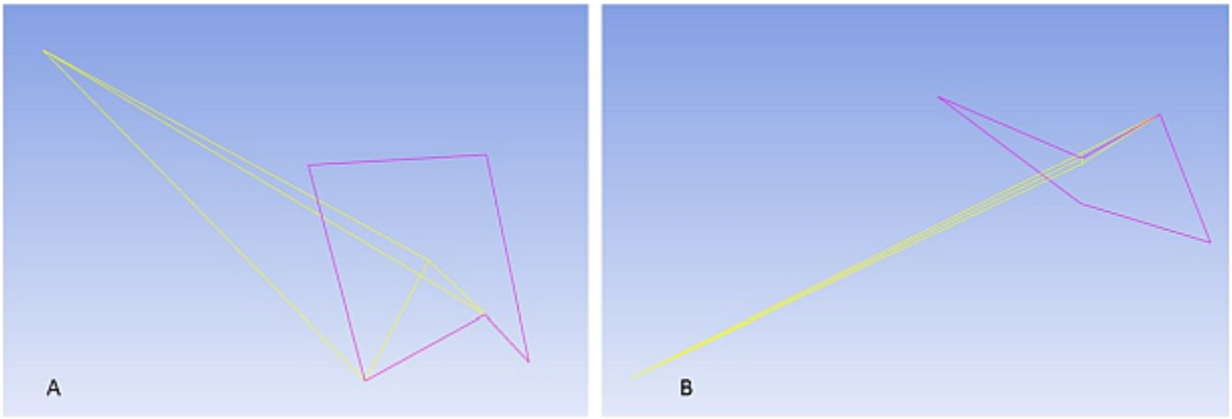
General High Resolution Tracking Text Commands

`define/models/dpm/numerics/high-resolution-tracking/check-subtet-validity?`

When enabled, checks the validity of a subtet when the particle first enters it. If the subtet is found to be degenerate, the tracking algorithm modifies to accommodate it. This option is useful for degeneracy in polyhedral cells shown in Figure 24.3: Degenerate Subtet in a Polyhedral Mesh (p. 2688). In such cases, an intersection can be missed due to a degenerate subtet. By default, this option is disabled to improve performance. It is recommended to always perform a mesh check (for example, by using the `mesh/check` text command) to determine if there are degenerate subtets in the domain.

An inverted subtet with negative volume can be produced when decomposing cells with highly warped faces. Figure 24.3: Degenerate Subtet in a Polyhedral Mesh (p. 2688) and Figure 24.4: Degenerate Subtet in a Hex Mesh (p. 2689) show examples of degenerate subtets.

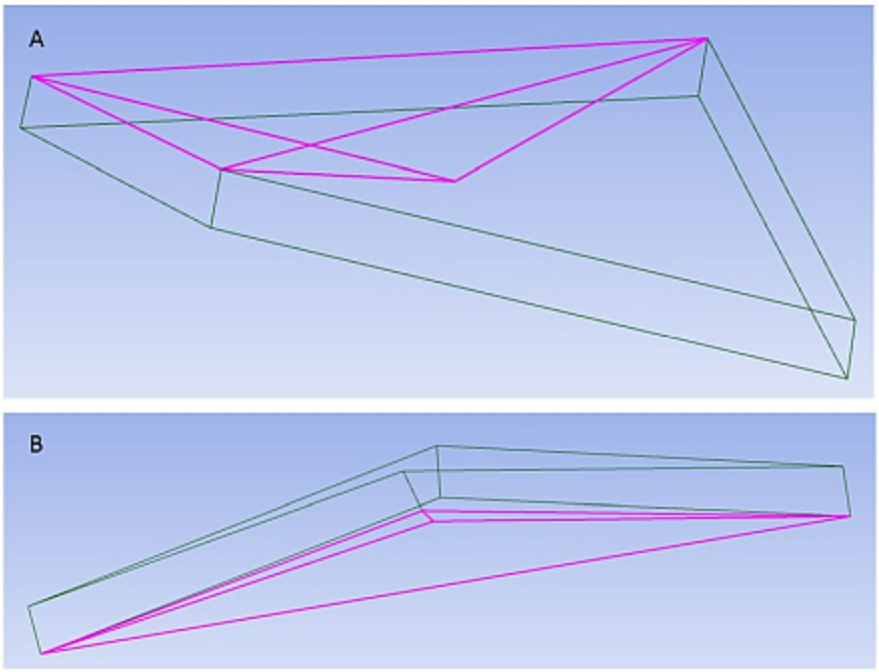
Figure 24.3: Degenerate Subtet in a Polyhedral Mesh



In Figure 24.3: Degenerate Subtet in a Polyhedral Mesh (p. 2688), the left-hand side (A) illustrates a degenerate subtet (in yellow) resulting from a warped polyhedral face (in magenta). The right-hand side (B) shows the same subtet from a different angle.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.4: Degenerate Subtet in a Hex Mesh



In Figure 24.4: Degenerate Subtet in a Hex Mesh (p. 2689), the top image (A) illustrates a degenerate subtet resulting from a warped quadrilateral face of a hexagonal cell while the bottom image (B) illustrates the same subtet from a different angle.

Tracking particles through degenerate subtets can result in missed face intersections and lost particles. The degeneracy may also result in the failure of the initial location of the particle when it is first injected.

`define/models/dpm/numerics/high-resolution-tracking/use-quad-face-centroid?`

Enables/disables using quad face centroids when creating subtets. This option changes the way hexahedral cells are decomposed to avoid creating degenerate subtets. It is useful in situations where the particle cannot be initially located in degenerate subtets formed by a warped hexahedral cell (see Figure 24.4: Degenerate Subtet in a Hex Mesh (p. 2689)). It is recommended to always perform a mesh check (for example, by using the `mesh/check` text command) to determine if there are degenerate subtets in the domain. If degenerate subtets are found in the domain during the mesh check, Ansys Fluent automatically enables the appropriate settings (`use-quad-face-centroid?` and/or `check-subtet-validity?`) and displays the following message:

```
Note: Settings to improve the robustness of pathline and
      particle tracking have been automatically enabled.
```

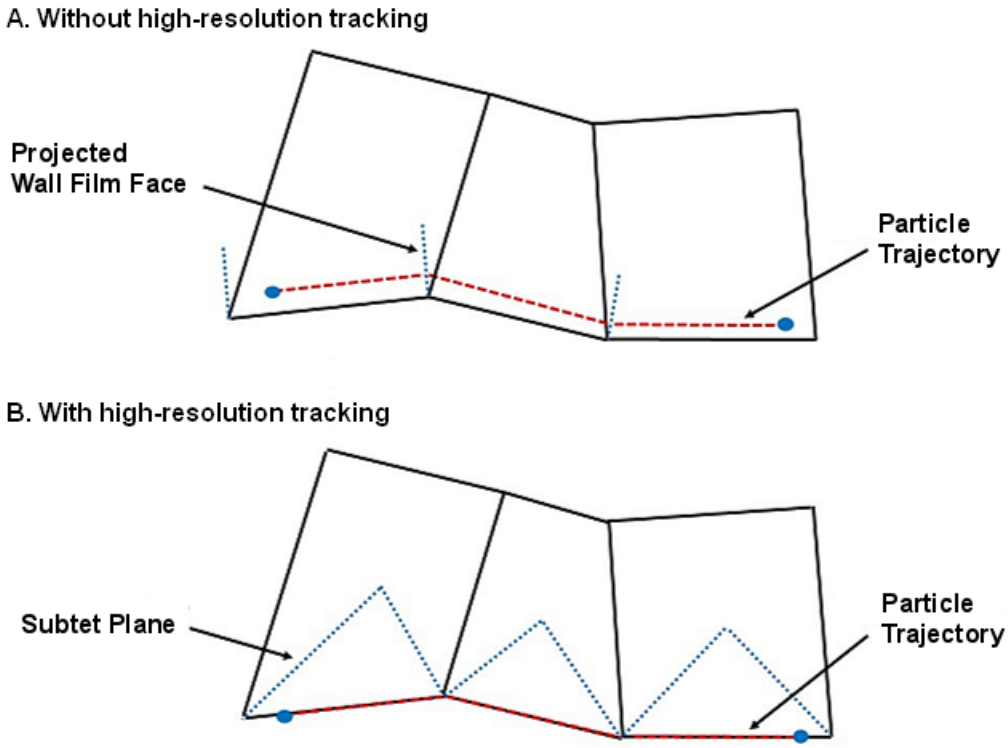
`define/models/dpm/numerics/high-resolution-tracking/project-wall-film-particles-to-film?`

Enables/disables projecting existing particles to Lagrangian wall film to track using high-resolution tracking.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When not using the high-resolution tracking method in Ansys Fluent, the particles are tracked slightly above the wall film surface to avoid incomplete or lost particles. When **High-Res Tracking** is enabled, particles are tracked directly on the surface, which improves accuracy and robustness, as shown in Figure 24.5: Lagrangian Wall Film Tracking (p. 2690).

Figure 24.5: Lagrangian Wall Film Tracking



When reading in a data file that contains wall film particles previously tracked without high-resolution tracking, you need to either clear the particles from the domain or project their positions to the wall film surface using the `project-wall-film-particles-to-film?` text command prior to using the high-resolution tracking method. After tracking the particles for one timestep, this option can be disabled to improve performance.

Note:

Projecting LWF particles that have been tracked without high-resolution tracking to the film zone and continuing with high-resolution tracking sacrifices both accuracy and robustness. Therefore, removing such particles from the domain is recommended.

`define/models/dpm/numerics/high-resolution-tracking/enable-automatic-intersection-tolerance?`

Enables/disables the automatic calculation of intersection tolerance. By default, the tolerance used in intersection calculations is scaled by the residence time of the particle in the cell to improve robustness. For most cases, the scaled tolerance is sufficient to identify all intersections of the particle trajectory and the subtet faces. You can set the intersection tolerance manually using the `set-subtet-intersection-tolerance` text command.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

define/models/dpm/numerics/high-resolution-tracking/set-subtet-intersection-tolerance

Specifies the tolerance used in intersection calculations. This tolerance will be scaled by the characteristic cell crossing time of the particle if the `enable-automatic-intersection-tolerance?` text command is enabled. If that option is disabled, the specified tolerance will be used without scaling. The default intersection tolerance is 10^{-5}.

define/models/dpm/numerics/high-resolution-tracking/boundary-layer-tracking?

Enables/disables the calculation of the particle time step size that considers both the cell aspect ratio and the particle trajectory. This method improves the accuracy of the predictions in boundary layer cells, particularly in layers where flow gradients are large.

define/models/dpm/numerics/high-resolution-tracking/enable-barycentric-intersections?

Enables/disables an alternative method of calculating intersections with cell boundaries. Barycentric intersections are linear calculations and are faster than the default intersection algorithm. The default intersection algorithm is second-order for stationary meshes; therefore, using the barycentric intersection may sacrifice accuracy. You must verify that the barycentric intersections provide comparable results to the default intersection method. This option is available only for 3D stationary meshes and the double precision solver.

define/models/dpm/numerics/high-resolution-tracking/use-barycentric-sampling?

When enabled, this option provides improved accuracy and parallel consistency when sampling particles at planes. This item is available only with the 3D solver. Using the double-precision solver and bounded planes is recommended.

define/models/dpm/numerics/high-resolution-tracking/use-particle-timestep-for-intersection-tolerance?

Enables/disables the use of the particle timestep for the subtet intersection tolerance with axisymmetric grids (default: enabled). If disabled, the tolerance will be calculated in the same manner as non-axisymmetric meshes (a scaled value of the tolerance which is set using the `define/models/dpm/numerics/high-resolution-tracking/set-subtet-intersection-tolerance` text command).

define/models/dpm/numerics/high-resolution-tracking/use-velocity-based-error-control?

Enables/disables an alternative method of timestep adaption. By default, Ansys Fluent uses the half-step method of timestep adaption with particle integration. This alternative method of controlling the integration timestep based upon velocity changes is faster; however, you need to ensure that the accuracy is comparable for your specific application.

define/models/dpm/numerics/high-resolution-tracking/ set-film-spreading-parameter

Provides the ability to add a random component to the Lagrangian wall-film particle acceleration to reduce streaking that may result from tracking particles on faceted geometry. As small variations in face normals can produce streaks in the wall-film appearance, this parameter makes

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the film look smoother. The value of this parameter should range between 0 and 1.0. The default value of 0 prevents any artificial smoothing.

define/models/dpm/numerics/high-resolution-tracking/always-use-face-centroid-with-periodics?

When enabled, Ansys Fluent uses quad face centroids when creating subtets in cases with periodic boundaries.

To avoid discontinuities in the particle forces at periodic boundaries, Ansys Fluent automatically enables the `use-quad-face-centroid?` option in simulations with periodic zones. You can disable this option for cases where the particles are not crossing the periodic boundaries. This may improve performance for meshes predominantly made up of hex cells. The recommended method to ensure that degenerate subtets are still handled appropriately is to execute the following commands in this order:

```
define/models/dpm/numerics/high-resolution-tracking/always-use-face-centroid-with-periodics? no
define/models/dpm/numerics/high-resolution-tracking/use-quad-face-centroid? no
mesh/check
```

define/models/dpm/numerics/high-resolution-tracking/sliding-interface-crossover-fraction

Specifies the fraction of the distance to the subtet center to move the particle.

At non-conformal interfaces, the nodes used for the barycentric interpolation are different on either side of the interface. This may result in incomplete particles due to discontinuities in the variable interpolation. The number of incomplete particles may be reduced by moving the particles slightly off of the sliding interface. Recommended values range between 0 and 0.5.

Barycentric Interpolation Method Text Commands

In addition, the following interpolation options are available under the `define/models/dpm/numerics/high-resolution-tracking/barycentric-interpolation/` text menu:

define/models/dpm/numerics/high-resolution-tracking/barycentric-interpolation/zero-nodal-velocity-on-walls?

When enabled, sets the velocity at wall nodes to zero. (By default, the nodal velocities on walls are first reconstructed from cell and face values and then corrected to ensure that there are no velocity components directed towards the walls). This may be useful if you want to consider particle impingement on the walls. Note that enabling this option will more likely produce incomplete particles as some particles may settle on the walls.

define/models/dpm/numerics/high-resolution-tracking/barycentric-interpolation/interpolate-flow-density?

Enables/disables the barycentric interpolation of the flow density. This option is recommended when the density varies with position to avoid discontinuities in the interpolated variable at cell boundaries. For constant density flows, this option is unnecessary.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

define/models/dpm/numerics/high-resolution-tracking/barycentric-interpolation/interpolate-flow-viscosity?

Enables/disables the barycentric interpolation of flow viscosity to the particle position. This option is recommended when the flow viscosity varies with position to avoid discontinuities in the interpolated variable at cell boundaries. For flows with constant viscosity, this option is unnecessary.

define/models/dpm/numerics/high-resolution-tracking/barycentric-interpolation/interpolate-flow-cp?

Enables/disables the barycentric interpolation of specific heat to the particle position. This option is recommended when the specific heat varies with position to avoid discontinuities in the interpolated variable at cell boundaries. For flows with constant specific heat, this option is unnecessary.

define/models/dpm/numerics/high-resolution-tracking/barycentric-interpolation/interpolate-flow-solution-gradients?

When enabled, flow solution gradients are interpolated to the particle position. This can be useful when using physical models that depend on these gradients (for example, the thermophoretic force, pressure-gradient force, or virtual mass force). Interpolating the gradients is highly recommended when using the trapezoidal numerics scheme, which is the default method.

define/models/dpm/numerics/high-resolution-tracking/barycentric-interpolation/interpolate-temperature?

Enables/disables the barycentric interpolation of temperature to the particle position. The cell temperature is used by default in calculations of heat transfer to/from the particle.

define/models/dpm/numerics/high-resolution-tracking/barycentric-interpolation/interpolate-wallfilm-properties?

When enabled, the wallfilm properties (film height, film mass, and wall shear) are interpolated to the particle position.

define/models/dpm/numerics/high-resolution-tracking/barycentric-interpolation/precompute-pdf-species?

When this option is enabled for premixed or non-premixed combustion simulations, the species composition in each cell is precomputed prior to tracking particles. This approach may improve performance for cases with many particles and relatively few cells. By default, this option is set to no, and Ansys Fluent calculates the species composition during particle tracking. The solution results will be identical for both methods.

define/models/dpm/numerics/high-resolution-tracking/barycentric-interpolation/user-interpolation-function

Provides a hook for the DEFINE_DPM_INTERPOLATION user-defined function (UDF) used to interpolate cell UDM to the particle position when using high-resolution particle tracking. See DEFINE_DPM_BODY_FORCE in the *Fluent Customization Manual* for more information about this UDF.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Particle Relocation Text Commands

The following location options are available under the `define/models/dpm/numerics/high-resolution-tracking/particle-relocation/` text menu:

`define/models/dpm/numerics/high-resolution-tracking/particle-relocation/enhanced-cell-relocation-method?`

When enabled, Ansys Fluent uses a more rigorous method of locating the cell in which the particle is currently contained. This approach is computationally more expensive than the default method; however, it is recommended for cases when Fluent reports that particles are lost after the mesh has been changed due to MDM, dynamic adaption, or repartitioning.

`define/models/dpm/numerics/high-resolution-tracking/particle-relocation/enhanced-wallfilm-location-method?`

When enabled, Ansys Fluent uses a more robust method of locating the wall film particles on the film zone after the domain has been remeshed either due to dynamic mesh movement or dynamic adaption. The improved robustness involves a performance penalty. This method is recommended for cases when Fluent reports that wall film particles are lost during relocation.

`define/models/dpm/numerics/high-resolution-tracking/particle-relocation/overset-relocation-robustness-level`

Sets the performance/robustness trade-off when relocating particles in overset meshes. The default value is 1, which corresponds to a more robust algorithm. Performance may be improved by setting the value to 0; however, this may result in particle loss, particularly if the mesh is moving. You need to make sure that no particle loss occurs.

`define/models/dpm/numerics/high-resolution-tracking/particle-relocation/use-legacy-particle-location-method?`

Provides the ability to use the legacy particle location method with high-resolution tracking to improve performance.

When using high-resolution tracking, the default method of locating the cells in which particles are injected is to decompose the cells into subtets and then determines if the particle is present in one of them. Dividing cells into subtets to locate the particles is more accurate and robust than the method used when high-resolution tracking is not active. However, for unsteady particle tracking in polyhedral cells, it can take considerably longer to locate the particles in the domain than with the method used when high-resolution tracking is not enabled. This option is useful when tracking in polyhedral meshes, particularly when unsteady particles that evaporate quickly are injected frequently (for example, gas turbine combustors). Note that it does not improve the particle tracking speed, only the time it takes to inject the particles.

`define/models/dpm/numerics/high-resolution-tracking/particle-relocation/wall-film-relocation-tolerance-scale-factor`

Sets a scaling factor to modify the tolerance for locating Lagrangian wall film (LWF) particles on a film face during the relocation. The default value is 1.0. Increasing the tolerance may help when Ansys Fluent fails to locate LWF particles in such cases.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To improve accuracy and robustness, it is recommended to use the high-resolution tracking of particles from the time they are first injected. If you want to use in your simulation a data file containing particles that were tracked without the high-resolution tracking option, be aware that the cell IDs stored in the file may not be correct. If this is the case, Ansys Fluent needs to locate the cell containing the particle. For cases with many particles, cells, or parallel partitions, this may take considerable time, particularly if any of the particles are outside of the domain and cannot be located. To accelerate this process, you can enable the following text command:

```
define/models/dpm/numerics/high-resolution-tracking/particle-relocation/load-legacy-particles?
Ensure particles are in the expected cells prior to tracking? [no] yes
```

When this option is enabled, the particles that are found outside the associated cells in the Fluent data file will be moved inside these cells prior to particle tracking. Neither the mesh nor the partitioning should be modified prior to the first time the particles are tracked. If you need to perform manual or dynamic mesh adaption, you must first disable this option. This option can be enabled before or after reading the data file into Ansys Fluent.

24.2.4. Drag Laws

There are eight drag laws for the particles that can be selected for each injection. These are specified on the **Physical Models** tab of the **Set Injection Properties** dialog box. See Specifying Injection-Specific Physical Models (p. 2764) for details.

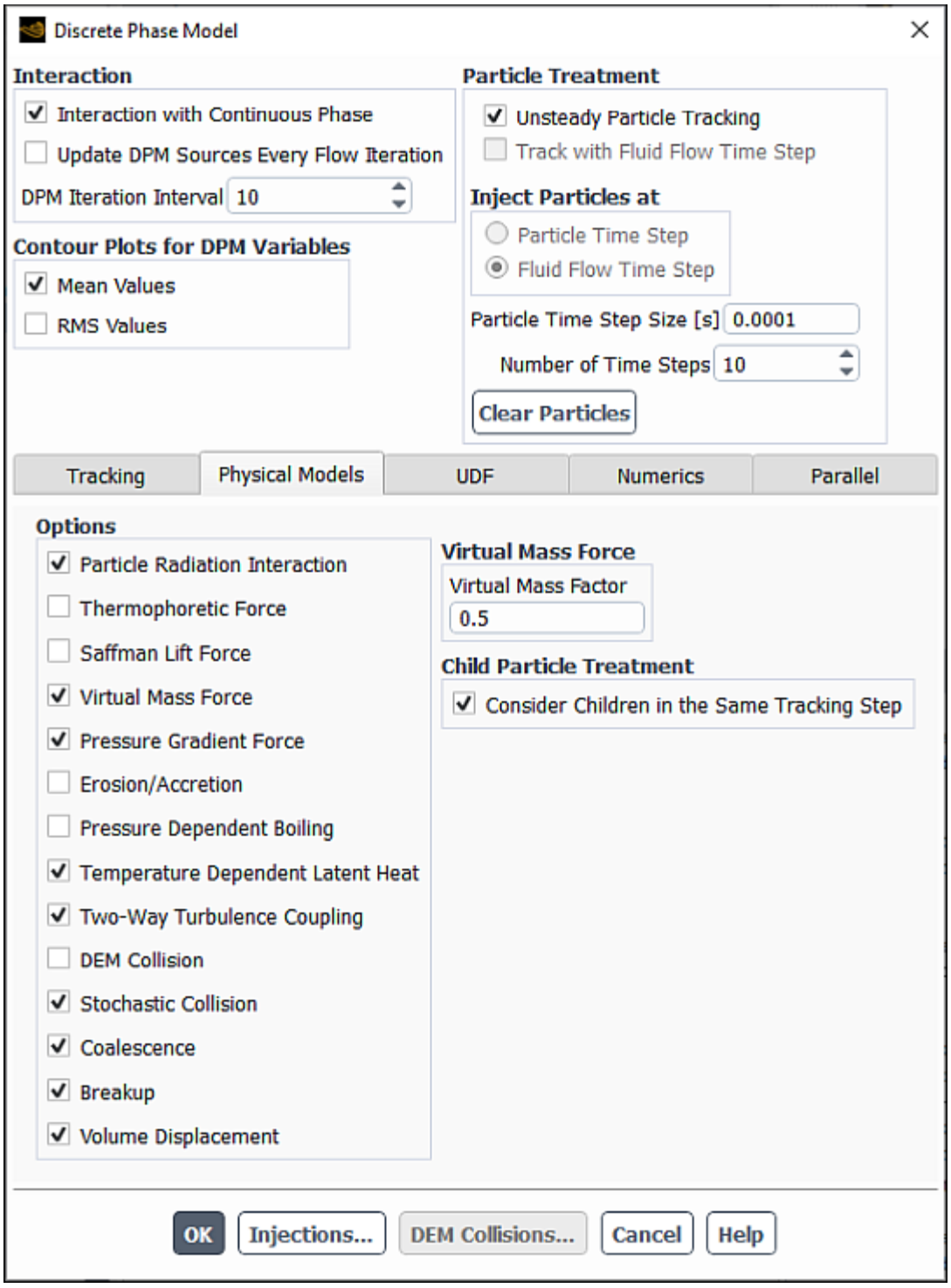
24.2.5. Physical Models for the Discrete Phase Model

This section provides instructions for using the optional discrete phase models available in Ansys Fluent. All of them can be enabled in the **Physical Models** tab of the **Discrete Phase Model** dialog box (Figure 24.6: The Discrete Phase Model Dialog Box - Physical Models Tab (p. 2696)).

Setup → Models → Discrete Phase → Edit...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.6: The Discrete Phase Model Dialog Box - Physical Models Tab



Further details are discussed in the following sections:

24.2.5.1. Including Radiation Heat Transfer Effects on the Particles
24.2.5.2. Including Thermophoretic Force Effects on the Particles
24.2.5.3. Including Saffman Lift Force Effects on the Particles
24.2.5.4. Including the Virtual Mass Force and Pressure Gradient Effects on Particles

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.2.5.5. Monitoring Erosion/Accretion of Particles at Walls
24.2.5.6. Pressure Options for Vaporization Models
24.2.5.7. Considering Pressure Dependence in Boiling
24.2.5.8. Including the Effect of Droplet Temperature on Latent Heat
24.2.5.9. Including the Effect of Particles on Turbulent Quantities
24.2.5.10. Including Collision and Droplet Coalescence
24.2.5.11. Including the DEM Collision Model
24.2.5.12. Including Droplet Breakup
24.2.5.13. Modeling Collision Using the DEM Model
24.2.5.14. Including the Volume Displacement of Particles

24.2.5.1. Including Radiation Heat Transfer Effects on the Particles

If you want to include the effect of radiation heat transfer to the particles (Equation 5.73 in the Theory Guide), you must enable the **Particle Radiation Interaction** option under the **Physical Models** tab, in the Discrete Phase Model Dialog Box (p. 4845) (Figure 24.6: The Discrete Phase Model Dialog Box - Physical Models Tab (p. 2696)). You also must define additional properties for the particle materials (emissivity and scattering factor), as described in Description of the Properties (p. 2817). This option is available only when the P-1 or discrete ordinates radiation model is used.

24.2.5.2. Including Thermophoretic Force Effects on the Particles

If you want to include the effect of the thermophoretic force on the particle trajectories (Equation 12.11 in the Theory Guide), enable the **Thermophoretic Force** option under the **Physical Models** tab, in the Discrete Phase Model Dialog Box (p. 4845). You also must define the thermophoretic coefficient for the particle material, as described in Description of the Properties (p. 2817).

24.2.5.3. Including Saffman Lift Force Effects on the Particles

For sub-micron particles, you can also model the lift due to shear (the Saffman lift force, described in Saffman's Lift Force in the Theory Guide) in the particle trajectory. To do this, enable the **Saffman Lift Force** option under the **Physical Models** tab, in the Discrete Phase Model Dialog Box (p. 4845).

24.2.5.4. Including the Virtual Mass Force and Pressure Gradient Effects on Particles

In cases where the density of the fluid approaches or exceeds the density of the particles (for example liquid flow with gaseous bubbles), it is recommended that you include the Virtual Mass and Pressure Gradient forces in the particle force balance. To do this, enable **Virtual Mass Force** and **Pressure Gradient Force** under the **Physical Models** tab in the Discrete Phase Model Dialog Box (p. 4845). See Other Forces in Fluent Theory Guide for information on the Virtual Mass and Pressure Gradient forces.

24.2.5.5. Monitoring Erosion/Accretion of Particles at Walls

To monitor particle erosion and accretion rates at wall boundaries:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Enable **Erosion/Accretion** in the **Physical Models** tab of the Discrete Phase Model Dialog Box (p. 4845).

   Enabling the **Erosion/Accretion** option will cause the erosion and accretion rates to be calculated at wall boundary faces when particle tracks are updated.

2. For each wall zone, select erosion models and specify erosion model parameters as described in Setting Particle Erosion and Accretion Parameters (p. 2787).

3. If you want to couple particle erosion with dynamic meshes to account for changes in the shape and position of eroded walls, follow the procedure described in Particle Erosion Coupled with Dynamic Meshes (p. 2793).

Note:

The computation of the erosion rate is based on the real wall position, and for cases that involve the DPM rough wall model, variations of the wall orientation are not considered.

24.2.5.6. Pressure Options for Vaporization Models

You can use the text interface command

```
define/models/dpm/options/use-absolute-pressure-for-vaporization?
```

to select whether the absolute pressure or a constant **Operating Pressure** (set in the **Operating Conditions** dialog box) is used in the vaporization rate calculations.

The default and recommended setting is to use the absolute pressure. However, in some incompressible flow cases, when there are large pressure variations in the droplet injection region, selecting the constant operating pressure may stabilize the solution.

See Operating Pressure, Gauge Pressure, and Absolute Pressure (p. 1678) and Equation 8.74 (p. 1678) for definitions of the operating pressure and absolute pressure.

24.2.5.7. Considering Pressure Dependence in Boiling

If the pressure in your simulation differs from atmospheric, you should either modify the boiling point to be consistent with the average pressure in the region where the droplets evaporate, or enable the **Pressure Dependent Boiling** option to change the condition for switching from droplet vaporization (Law 2) to boiling (Law 3). If you modify the boiling point for the droplet material, you should also modify the latent heat value accordingly.

By default Ansys Fluent will switch from the vaporization to the boiling law when the particle temperature has reached the boiling point defined for the droplet material (Equation 12.99 in the Theory Guide). When the **Pressure Dependent Boiling** option is enabled, the switching condition will change to $P_{sat}>P$, where P_{sat} is the saturation vapor pressure at the droplet temperature and P is the domain pressure. If $P_{sat}<P$ while in the boiling law, the model will switch back to the vaporization law. If this option is enabled it is essential to enter the appropriate droplet saturation vapor pressure data to cover the complete pressure/temperature range in your model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When **Pressure Dependent Boiling** is enabled, then the **Temperature Dependent Latent Heat** model automatically applies (see Including the Effect of Droplet Temperature on Latent Heat (p. 2700)).

Setting the **Pressure Dependent Boiling** option has no effect on the multicomponent particles, where switching from the vaporization to the boiling regime is always based on the component saturation vapor pressures (see Multicomponent Particle Definition (Law 7) in the Theory Guide).

In the supercritical pressure regime, failure of the saturation temperature calculation algorithm may indicate that the multicomponent droplet has approached the critical temperature, provided that you have entered accurate vapor pressure data for all components. In such a case, you can use the following text command:

```
define/models/dpm/options/treat-multicomponent-saturation-temperature-
failure?

Dump multicomponent particle mass if the saturation temperature calcu-
lation fails? [no] y
```

When this option is enabled, Ansys Fluent dumps the particle mass into the continuous phase if the saturation temperature calculation fails.

Finally, selection of the **Pressure Dependent Boiling** option is not available with the real-gas models, as the pressure dependence always applies. See Using the Cubic Equation of State Models with the Lagrangian Dispersed Phase Models (p. 1698) for more information.

For pressures higher than the critical pressure of the droplet material ($P>P_c$), a boiling point T_b, defined as the temperature at the saturation vapor pressure ($P=P_{sat}$), cannot be determined as the vapor pressure curve is defined only up to the critical point. Under supercritical pressure conditions ($P>P_c$), the vaporization models are applicable at droplet temperatures below the critical point ($T<T_c$), and the switching condition from vaporization to boiling is never met. For supercritical pressure conditions, you must enter the droplet saturation vapor pressure data for the complete pressure/temperature range up to the critical point T_c.

When **Pressure Dependent Boiling** is enabled, you can use the text user interface command

```
/define/models/dpm/options/allow-supercritical-pressure-vaporization?
```

to enforce the switching from vaporization to boiling even if the boiling point T_b is not calculated from the vapor pressure data.

If the pressure in your model is above critical ($P>P_c$), you must retain the default setting (`yes`) for `allow-supercritical-pressure-vaporization?`.

For subcritical pressure conditions ($P<P_c$), the setting for `allow-supercritical-pressure-vaporization?` will have no effect unless the boiling point calculation fails, which usually indicates that either the vapor pressure data do not cover the complete temperature/pressure range for your model, or inaccurate data have been entered. In that case, if `allow-supercritical-pressure-vaporization?` is set to `no`, the normal boiling point T_b defined for the droplet material will be imposed as a switching condition from vaporization to boiling.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.2.5.8. Including the Effect of Droplet Temperature on Latent Heat

To include the droplet temperature effects on the latent heat as described in Equation 12.92 in the Theory Guide, enable **Temperature Dependent Latent Heat** under the **Physical Models** tab. If you enable this option you must provide accurate temperature-dependent specific heat data for both the droplet and the evaporating species materials.

24.2.5.9. Including the Effect of Particles on Turbulent Quantities

Particles can damp or produce turbulent eddies [109] (p. 5719). To compute turbulence modulation, Ansys Fluent uses the formulation described in [9] (p. 5713) (see Equation (C-4) in [9] (p. 5713)). Damping occurs when the particle diameter is less than one tenth of the turbulent length scale. For larger particle diameters turbulence kinetic energy is produced [54] (p. 5716). Sources to the ε-equation for k- ε based turbulence models or to the ω-equation for k-ω based turbulence models are accounted for.

If you want to consider these effects in the chosen turbulence model, you can enable this using **Two-Way Turbulence Coupling**, under the **Physical Models** tab.

Turbulence modulation due to particles is available for the following turbulence models:

- k-eps model (2-eqn)
- k-omega model (2-eqn), except for the algebraic Reynolds stress sub-model WJ-BSL-EARSM
- Transition k-kl-omega model (3-eqn)
- Transition SST model (4-eqn)
- Scale Adaptive Simulation (SAS)
- Detached Eddy model with Realizable k-eps model (DES)
- Detached Eddy model with SST model (DES)

If a turbulence model other than the models listed above is used, the Ansys Fluent solver will automatically disable the **Two-Way Turbulence Coupling** option.

24.2.5.10. Including Collision and Droplet Coalescence

To include the effect of collisions, as described in Collision and Droplet Coalescence Model Theory in the Theory Guide, select **Stochastic Collision** and **Coalescence** under **Options**.

Note:

Coalescence will appear under **Options** after **Stochastic Collision** has been enabled. If you want to simulate collisions between solid particles, disable the **Coalescence** option.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.2.5.11. Including the DEM Collision Model

The DEM collision model is suitable for simulating granular matter, where such simulations are characterized by a high volume fraction of particles, and the particle-particle interaction is important. See Modeling Collision Using the DEM Model (p. 2701) for details about using this model.

24.2.5.12. Including Droplet Breakup

To model droplet breakup in Ansys Fluent, enable **Breakup** under **Options**. By default, this will enable **Consider Children in the Same Tracking Step** and breakup for all suitable injections. You can then specify the breakup models and parameters (or disable breakup) for each individual injection on the **Physical Models** tab in the **Set Injection Properties** dialog box as described in Breakup (p. 2769).

The **Consider Children in the Same Tracking Step** option collects newly generated child droplets and track them within the same time step. It provides higher accuracy and can lead to faster convergence within a transient flow time step. This option is also automatically enabled if you select **wall-film** as **Discrete Phase BC Type** in the **Wall** dialog box (**DPM** tab).

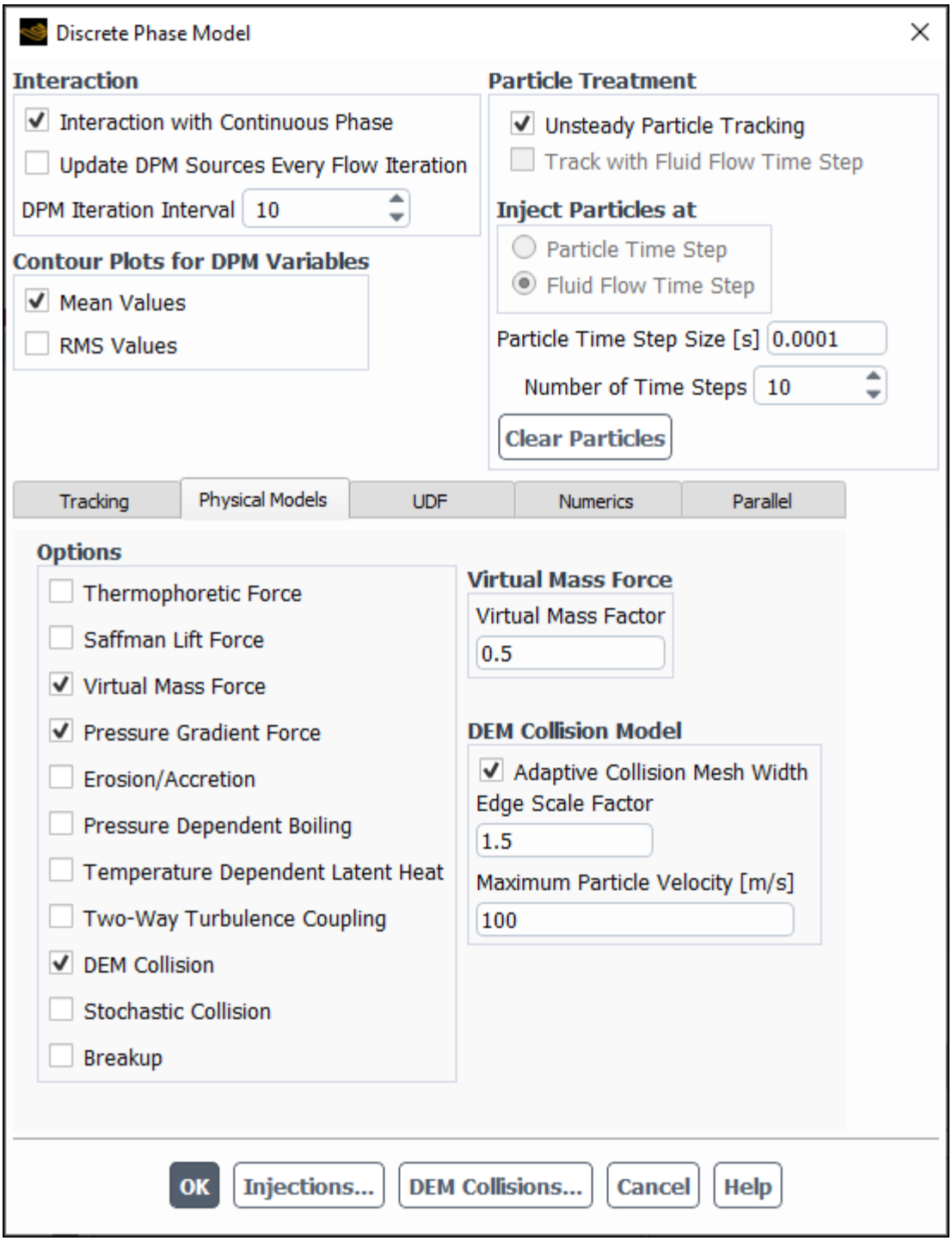
24.2.5.13. Modeling Collision Using the DEM Model

To enable the DEM collision model, select **DEM Collision** under **Options**. A detailed description of this model can be found in Discrete Element Method Collision Model in the Theory Guide.

1. In the **Physical Models** tab of the **Discrete Phase Model** dialog box

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.7: Discrete Phase Model Dialog Box with DEM Collision Model



a. Specify the settings your model requires under **DEM Collision Model**.

b. (Optional) By default, **Adaptive Collision Mesh Width** is enabled. This adjusts the width of the collision mesh to the largest parcel diameter multiplied by the **Edge Scale Factor**. If **Adaptive Collision Mesh Width** is disabled, a fixed **Static Collision Mesh Width** has to be given in the chosen units of length.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. The **Maximum Particle Velocity** limits the maximum particle velocity to a physically plausible range.

2. Set up the injection.

Setup → Models → Discrete Phase → Injections New...

a. In the **Set Injection Properties** dialog box, select a name from the **DEM Collision Partner** drop-down list that will serve as the collision partner. A default name using the particle material will be suggested.

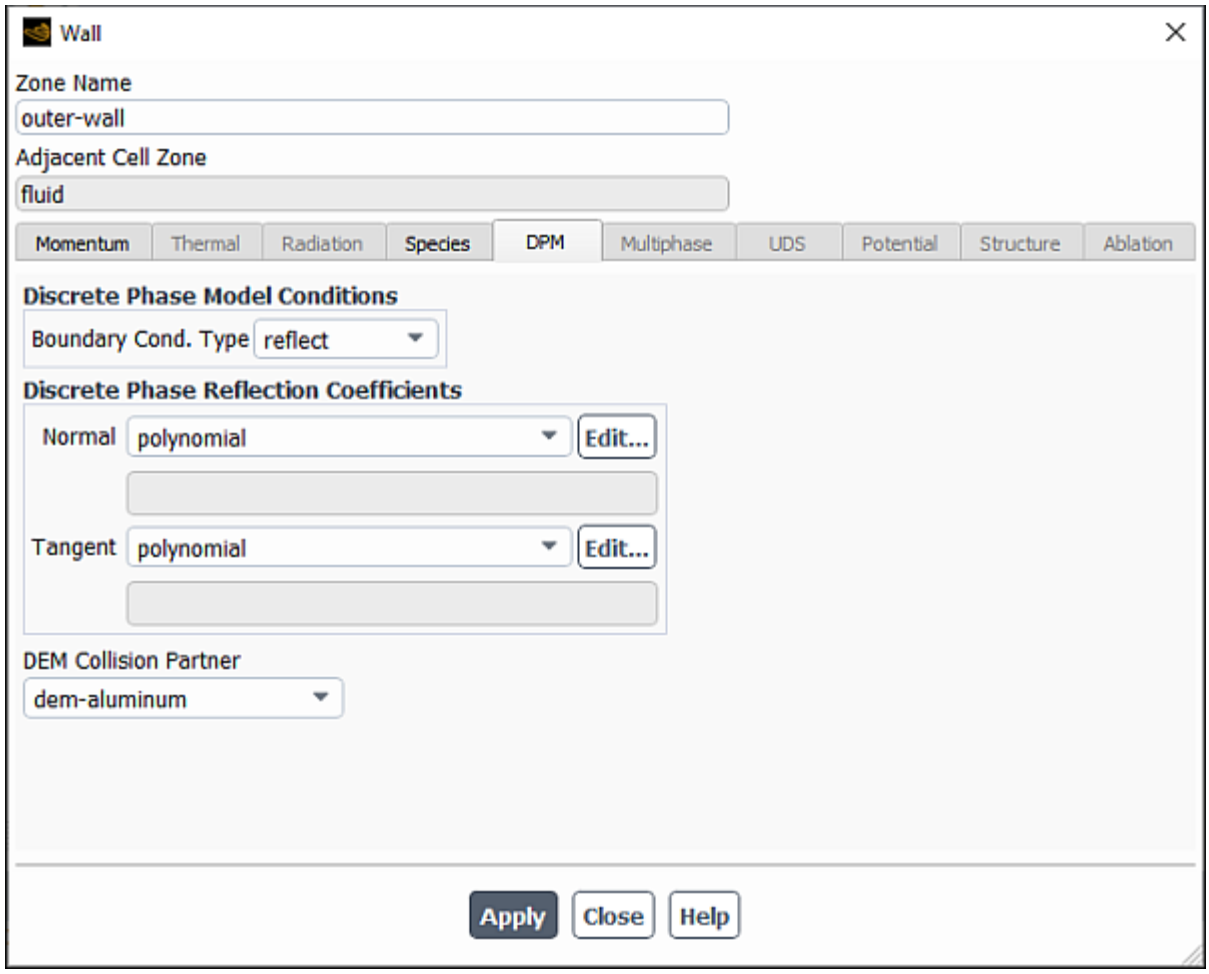
> **Note:**
>
> Selecting none from the drop-down list indicates that the particles released from this injection do not participate in the DEM collision computation.

b. If you want to use the rolling friction collision law (see step 4), under the **Physical Models** tab, enable **Particle Rotation** for those injections whose particles take part in collisions with rolling friction.

3. Set the boundary conditions for the discrete phase as you normally would. If the **Boundary Cond. Type** is **reflect**, select a name from the **DEM Collision Partner** drop-down list to designate the collision partner For example, in Figure 24.8: Wall Boundary Condition for the DEM Model (p. 2704), a wall boundary condition will suggest a wall material name, which will designate the collision partner. The name will have a **dem-** prefix However, if the **DEM Collision Partner** is **none**, the wall will reflect particles like any other DPM particles using the settings for the **Discrete Phase Reflection Coefficients** in the **Normal** and **Tangent** directions. These settings generally apply for particles that are not colliding according to DEM, such as massless particles.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.8: Wall Boundary Condition for the DEM Model

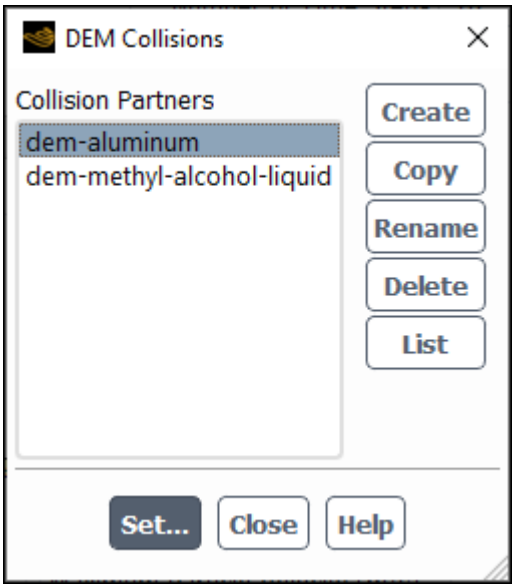


4. Define the particle interaction between the pairs of the collision partners.

   a. Click the **DEM Collisions...** button at the bottom of the **Discrete Phase Model** dialog box. The **DEM Collisions** dialog box (Figure 24.9: Collision Dialog Box (p. 2705)) will appear where you can manage the collision partners. You can **Create**, **Copy**, **Rename**, **Delete**, and **List** collision partners. To define collision laws for a collision partner, select a collision partner from the **Collision Partners** list and click the **Set...** button, or simply double-click the collision partner in the list.

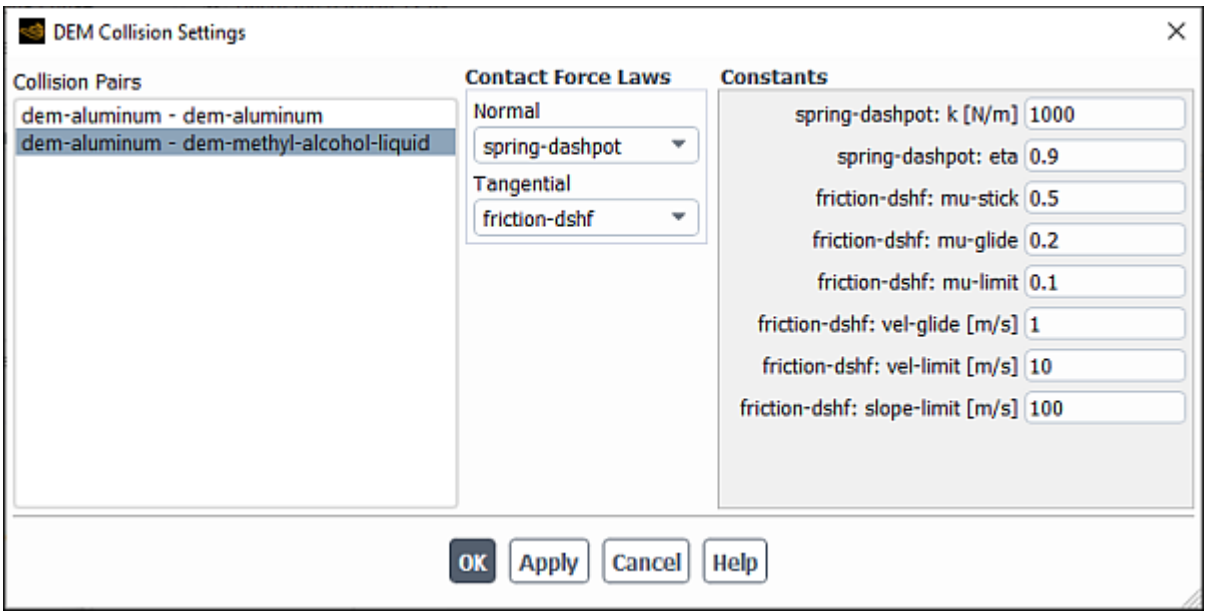
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.9: Collision Dialog Box



b. In the **DEM Collision Settings** dialog box (Figure 24.10: DEM Collision Settings Dialog Box (p. 2705)), a list of all the possible **Collision Pairs** will exist.

Figure 24.10: DEM Collision Settings Dialog Box



i. Select a pair of collision partners from the **Collision Pairs** list.

ii. For this pair, select the **Contact Force Laws** that best describe the collision between these two partners.

The **Normal** contact force laws that exist are:

- **spring**
- **spring-dashpot**

- **hertzian**
- **hertzian-dashpot**

You can also choose to exclude the contact force law by selecting **none**.

For the **Tangential** contact force law (if considered), you can select form the following friction collision laws:

- **friction-dshf**
- **friction-dshf-rolling** (an extension of the **friction-dshf** friction collision law including rolling friction)

Each of the laws is described in the following sections:

- The Spring Collision Law in the *Fluent Theory Guide*
- The Spring-Dashpot Collision Law in the *Fluent Theory Guide*
- The Hertzian Collision Law in the *Fluent Theory Guide*
- The Hertzian-Dashpot Collision Law in the *Fluent Theory Guide*
- The Friction Collision Law in the *Fluent Theory Guide*
- Rolling Friction Collision Law for DEM in the *Fluent Theory Guide*

iii. Enter the **Constants** for the chosen **Contact Force Laws**. Refer to Theory in the Theory Guide for guidance on how to find reasonable values of the force-law constants.

iv. Repeat for all other collision pairs.

v. Click **OK** or **Apply** to apply these settings.

Note:

- The choice of collision laws does not depend on the order in the pair of collision partners.
- Failing to specify a force law for a collision pair implies that respective particles will not collide.
- Rolling friction will only be factored in for a particular DEM particle injection if the **Particle Rotation** option is selected in the **Set Injection Properties** dialog box (**Physical Properties** tab).
- For parallel calculations, the DEM collision model can be optimized by enabling the hybrid optimization method for DPM. This method balances the load across machines, and, within each machine, the hybrid parallel DPM method is used to make sure the load is balanced by multi-threading. For more information, see the description of the **Hybrid Optimization** option in the **Weighting** tab discussion in Partitioning (p. 4319).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.2.5.13.1. Limitations

The following limitations currently apply when using the DEM collision model:

- Axisymmetric geometry cannot be used.
- Dynamic mesh motion cannot be used.
- The Eulerian wall film model cannot be used.
- Shear stress boundary conditions cannot be used at walls.
- Marangoni stress boundary condition cannot be used at walls.
- DEM particles do not transfer heat from particle to particle during contact.

24.2.5.13.2. Numeric Recommendations

To better preserve energy during particle collision, the following settings in the **Numerics** tab are recommended and will be automatically set when DEM Collisions are enabled.

1. Disable **Accuracy Control** to ensure that DEM time stepping remains small.
2. Disable **Automated** under **Tracking Scheme Selection**.
3. Select **implicit** from the **Tracking Scheme** drop-down list.

Refer to Numerics for Tracking of the Particles (p. 2712) for more information about the settings in the **Numerics** tab.

24.2.5.14. Including the Volume Displacement of Particles

In many Lagrangian-Eulerian simulations, the volume fraction of the local particle phase may not be small, and the blocking effect of the particulate phase on the carrier phase may need to be taken into account. See Blockage Effect in the *Fluent Theory Guide* for more information about the volume displacement effect of particles.

To enable the volume displacement effect of particles, select **Volume Displacement** (**Options** group box) in the **Physical Models** tab of the **Discrete Phase Model** dialog box.

Once the blocking effect is enabled, you can set the maximum particle volume fraction used to compute the carrier phase volume fraction in Equation 12.522 in the *Fluent Theory Guide* by issuing the following text command:

```
define/models/dpm/interaction> max-vf-allowed-for-blocking
Maximum DPM volume fraction used in continuous flow (0 1.) [0.95]
```

For high accuracy solution, using the default value of 0.95 is recommended.

The availability of the DPM volume fraction is automatically triggered by the Ansys Fluent solver.

Note that the blocking effect model has the following limitations:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- It is limited to single-phase and homogeneous multiphase (one carrier phase) flows.
- The density-based solver is not supported.

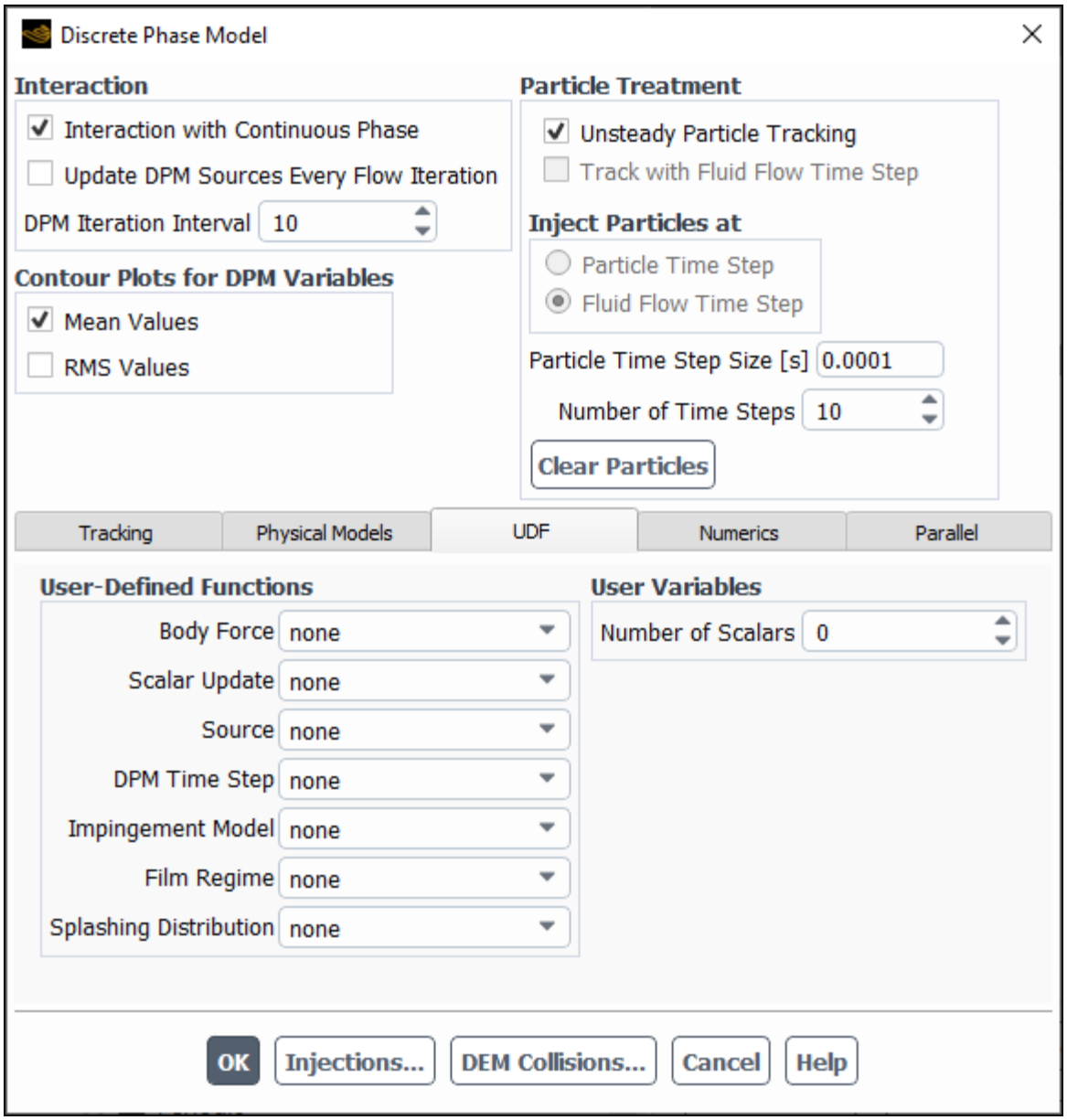
The impact of the blocking effect will be strongest in simulations where large particle volume fractions are present in the computational domain, or where the blockage due to the particulate phase may significantly alter the flow structure of the carrier phase. Typical examples are simulations of a particle spray, where the mesh close to the injection nozzle is refined in such a way that the injection nozzle cross-section is discretized by multiple cells, or jet in cross-flow applications, where the blockage effect of the compact liquid jet may enhance the vortex structure in the wake of the jet.

24.2.6. User-Defined Functions

User-defined functions can be used to customize the discrete phase model to include additional body forces, modify interphase exchange terms (sources), calculate or integrate scalar values along the particle trajectory, and incorporate nonstandard erosion rate definitions. More information about user-defined functions can be found in the Fluent Customization Manual.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.11: The Discrete Phase Model Dialog Box - UDF Tab



In the Discrete Phase Model Dialog Box (p. 4845), under **User-Defined Functions** in the **UDF** tab, there are drop-down lists labeled (see Figure 24.11: The Discrete Phase Model Dialog Box - UDF Tab (p. 2709)):

- **Body Force**
- **Scalar Update**
- **Source**
- **Spray Collide Function**
- **DPM Time Step**
- **Erosion/Accretion** (if **Erosion/Accretion** is enabled under the **Physical Models** tab)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Impingement Model**, **Film Regime** and **Splashing Distribution** (if either **wall-film** is selected for **Discrete Phase BC Type** in the **DPM** tab of the **Wall** dialog box or the **Eulerian Wall Film** model is enabled).

  Note that user-defined Impingement Model and Splashing Distribution will overwrite the impingement/splashing model selected for the Lagrangian or Eulerian wall film boundary condition.

These lists will show available user-defined functions that can be selected to customize the discrete phase model (see Fluent Customization Manual for more information).

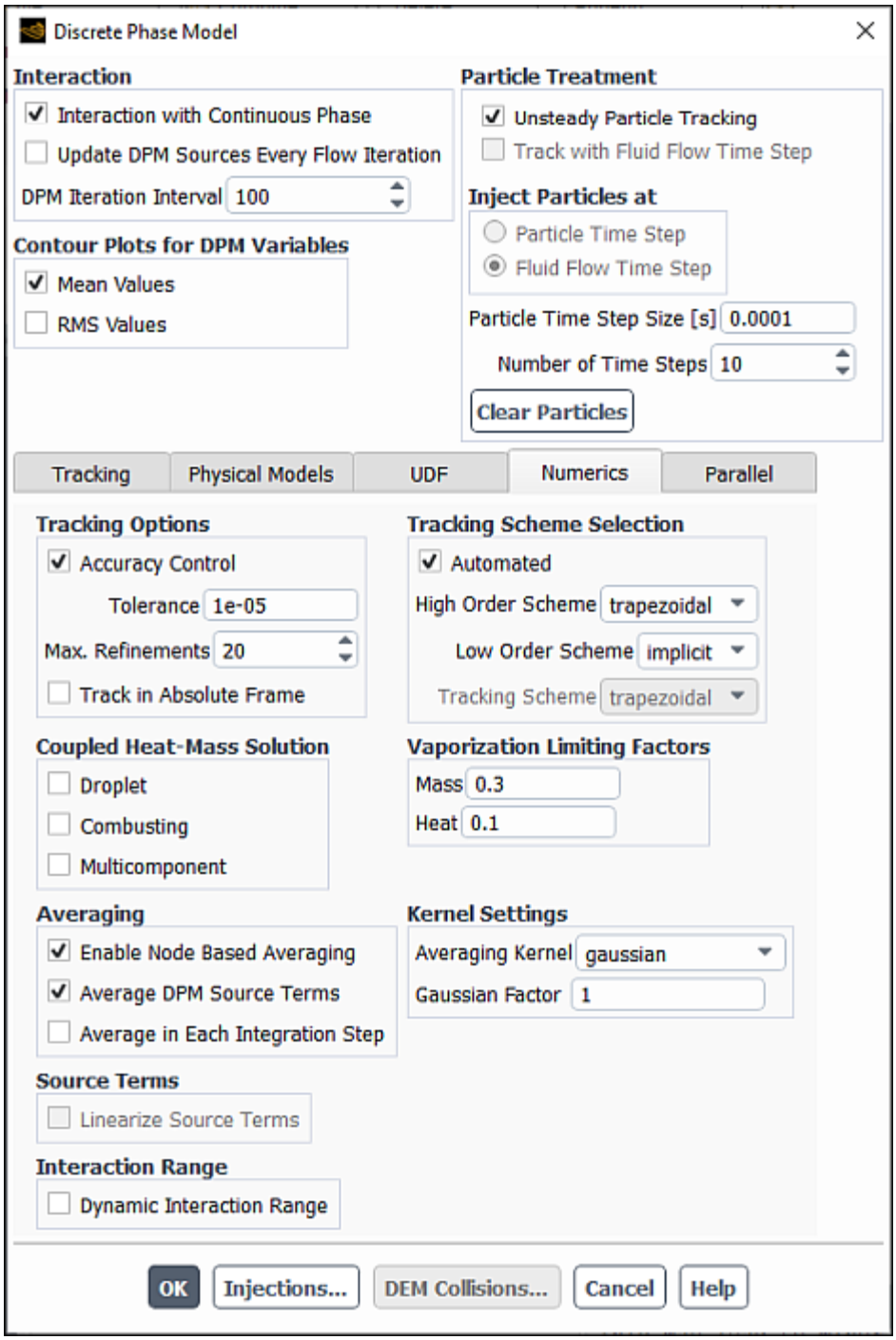
In addition, you can specify a **Number of Scalars** which are allocated to each particle and can be used to store information when implementing your own particle models.

24.2.7. Numerics of the Discrete Phase Model

The underlying physics of the Discrete Phase Model is described by ordinary differential equations (ODE) as opposed to the continuous flow which is expressed in the form of partial differential equations (PDE). Therefore, the Discrete Phase Model uses its own numerical mechanisms and discretization schemes, which are completely different from other numerics used in Ansys Fluent.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.12: The Discrete Phase Model Dialog Box - Numerics Tab



The **Numerics** tab gives you control over the numerical schemes for particle tracking as well as solutions of heat and mass equations (Figure 24.12: The Discrete Phase Model Dialog Box - Numerics Tab (p. 2711)).

Further details are provided in the following sections:

24.2.7.1. Numerics for Tracking of the Particles

24.2.7.2. Including Coupled Heat-Mass Solution Effects on the Particles

24.2.7.3. Tracking in a Reference Frame

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.2.7.4. Node Based Averaging of Particle Data
24.2.7.5. Linearized Source Terms
24.2.7.6. Using the Dynamic Interaction Range
24.2.7.7. Staggering of Particles in Space and Time
24.2.7.8. Packing Limit
24.2.7.9. Under-Relaxing Lagrangian Wall Film Height

24.2.7.1. Numerics for Tracking of the Particles

To solve equations of motion for the particles, the following tracking numerical schemes are available:

implicit

uses an implicit Euler integration of Equation 12.1 in the Theory Guide which is unconditionally stable for all particle relaxation times.

trapezoidal

uses a semi-implicit trapezoidal integration.

analytic

uses an analytical integration of Equation 12.1 of the Theory Guide where the forces are held constant during the integration.

runge-kutta

facilitates a 5th order Runge Kutta scheme derived by Cash and Karp [30] (p. 5714).

For additional details, see Solution Strategies for the Discrete Phase (p. 2827).

To specify a tracking numerical scheme for your simulation, you can either:

- use a single tracking scheme by selecting any of the above numerical schemes from the **Tracking Scheme** drop-down list (**Tracking Scheme Selection** group box), or
- switch between higher order and lower order tracking schemes using an automated selection based on the accuracy to be achieved and the stability range of each scheme. To use this option, you need to perform the following steps:

  1. Enable **Automated** (**Tracking Scheme Selection** group box).

     When this option is enabled, the **Tracking Scheme** drop-down becomes inaccessible, while the **High Order Scheme** and **Low Order Scheme** group boxes become available.

     Important:

     The **Automated** option is available only when **Accuracy Control** is enabled (**Tracking Options** group box).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Select **trapezoidal** or **runge-kutta** as the **High Order Scheme**.

3. Select **implicit** or **analytic** as the **Low Order Scheme**.

During runtime, the solver automatically switches between numerically stable lower order and higher order schemes, which are stable only in a limited range. In situations where the particle is far from hydrodynamic equilibrium, an accurate solution can be achieved very quickly with a higher order scheme, since these schemes need less step refinements for a certain tolerance. When the particle reaches hydrodynamic equilibrium, the higher order schemes become inefficient since their step length is limited to a stable range. In this case, the mechanism switches to a stable lower order scheme and facilitates larger integration steps.

Note:

- When **DEM Collisions** are enabled, Ansys Fluent automatically sets **Tracking Scheme** to **implicit** and disables **Accuracy Control**.
- The equation for rotating particle motion (Equation 12.195 in the *Fluent Theory Guide*) is solved using the Euler implicit discretization scheme.

You can control how accurately the equations need to be solved by using the following settings in the **Tracking Options** group box:

Accuracy Control

enables the solution of equations of motion within a specified tolerance. This is done by computing the error of the integration step and reducing the integration step if the error is too large. If the error is within the given tolerance, the integration step will also be increased in the next steps.

Tolerance

is the maximum relative error that has to be achieved by the tracking procedure. Based on the numerical scheme, different methods are used to estimate the relative error. The implemented Runge-Kutta scheme uses an embedded error control mechanism. The error of the other schemes is computed by comparing the result of the integration step with the outcome of a two step procedure with half the step size.

Max. Refinements

is the maximum number of step size refinements in one single integration step. If this number is exceeded the integration will be conducted with the last refined integration step size.

24.2.7.2. Including Coupled Heat-Mass Solution Effects on the Particles

By default, the particle heat and mass equations are solved in a segregated manner using an implicit Euler integration over the time step used for the trajectory calculation If you enable the **Droplet**, **Combusting**, or **Multicomponent** particles in the **Coupled Heat-Mass Solution** group box, Ansys Fluent will solve the corresponding equations using a coupled ODE solver with error tolerance control. The increased accuracy, however, comes at the expense of increased computational time.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To ensure solution accuracy for the **Droplet** and **Multicomponent** particles, Ansys Fluent will automatically switch to the coupled algorithm during the integration time step when the evaporated mass is greater than the limiting mass change Δm_p (defined in Equation 24.4 (p. 2714)), or when the particle temperature change is greater than the limiting temperature change ΔT_p (defined in Equation 24.5 (p. 2714)).

$$\Delta m_p = m_p a_m \tag{24.4}$$

where m_p is the particle mass (kg) and a_m is the vaporization limiting factor for the mass.

$$\Delta T_p = min\left(\left|T_p - T_\infty\right| a_T, T_p a_T\right) \tag{24.5}$$

where T_p is the particle temperature (K), T_∞ is the bulk temperature (K), and a_T is the vaporization limiting factor for the temperature.

You will enter the **Vaporization Limiting Factors** for **Mass** a_m and **Heat** a_T. The default values of 0.3 and 0.1 have been determined by systematic testing and are recommended for computations.

24.2.7.3. Tracking in a Reference Frame

Particle tracking is related to a coordinate system. With **Track in Absolute Frame** enabled in the **Tracking Options** group box, you can choose to track the particles in the absolute reference frame. All particle coordinates and velocities are then computed in this frame. The forces due to friction with the continuous phase are transformed to this frame automatically.

In rotating flows, it might be appropriate for numerical reasons to track the particles in the relative reference frame. If several reference frames exist in one simulation, then the particle velocities are transformed to each reference frame when they enter the fluid zone associated with this reference frame.

When the impact of particles with walls in multiple rotating reference frames is important, as it is the case with a rotating impeller in a stationary baffled tank, it is necessary to model the flow as a sliding mesh simulation.

24.2.7.4. Node Based Averaging of Particle Data

In general, the presence of DPM parcels may affect the continuous phase fluid flow meaning that discrete phase source terms and particle data such as fluid drag, temperature, volume fraction, and velocity can be integrated into the flow solver. By default in Ansys Fluent, the effects of a particular DPM parcel are applied only to the cells that intersect with the trajectory of that parcel. As an alternative, **Enable Node Based Averaging** is offered which distributes the effects of a DPM parcel even into neighboring cells that share at least a mesh node with the cells intersected by the trajectory. This allows a reduction of grid dependency of DPM simulations, since each parcel's effects on the flow solver are distributed more smoothly across neighboring cells.

You can select **Enable Node Based Averaging** in the **Numerics** tab of the **Discrete Phase Model** dialog box, as shown in Figure 24.12: The Discrete Phase Model Dialog Box - Numerics Tab (p. 2711). For non-DDPM cases, the **Enable Node Based Averaging** option is not available when the **Linearize Source Terms** is selected.

When enabled, the node based averaging is applied to particle velocity, particle temperature, particle volume fraction, and DPM concentration. By default, **Average DPM Source Terms** is also enabled, which applies node averaging to the source terms as well.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When the Dense Discrete Phase Model is being used, the **Average DDPM Variables** option appears to provide node based averaging for the discrete phase specific variables, such as DDPM particle drag. When **Contour Plots for DPM Variables** is enabled (Reporting of Discrete Phase Variables (p. 2869)), these quantities will also be node averaged.

Average in Each Integration Step should be enabled if you find that the flux report is showing discrepancies for mass flow rates of particles injected into and leaving the domain. This option increases computation time so it is not enabled by default.

Once you have selected **Enable Node Based Averaging**, you must also select the **Averaging Kernel** to accumulate the particle related data on the mesh nodes and to redistribute them back to the finite volume cells. The kernel and related settings are specified in the **Kernel Settings** section of the **Discrete Phase Model** dialog box. The following kernels are available:

- **nodes-per-cell**
- **shortest-distance**
- **inverse-distance**
- **gaussian**

  If you use the **gaussian** kernel, you must also specify the **Gaussian Factor**, which determines the width of the Gaussian distribution. The default value is 1.

For details about the averaging and kernel equations, refer to Node Based Averaging in the Theory Guide.

24.2.7.5. Linearized Source Terms

Source terms for discrete phase momentum, energy, and species can be linearized with respect to the cell variable, ϕ:

$$S_{DPM,\phi}=S_{const}+S_{lin}\phi \tag{24.6}$$

You can enable source term linearization with the **Linearize Source Terms** option in the **Numerics** tab of the **Discrete Phase Model** dialog box. For non-DDPM cases, the **Linearize Source Terms** option is not available when the **Enable Node Based Averaging** is selected.

This linearization strongly increases numerical stability for steady flows. For transient flows, it typically allows the use of a larger time step size and larger under-relaxation factors for the DPM model.

If your simulation involves the species transport model, you can disable the linearization of the species source terms for the discrete phase using the following text command:

```
define/models/dpm/interaction/linearized-dpm-species-source-terms?
Linearize the species DPM source terms? [yes] no
```

For non- or partially-premixed combustion cases, the DPM source terms for mixture fractions and inert species (if enabled) can be linearized with respect to the cell variable by enabling the **Linearize Source Terms** option and issuing the following text command:

```
define/models/dpm/interaction/linearized-dpm-mixture-fraction-source-terms? yes
```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note:

As a prerequisite for this linearization, the DPM sources in the mixture fraction equations are cast as a function of the mixture fractions rather than DPM mass. This can be reverted by the following text commands:

```
define/models/dpm/interaction/replace-dpm-mass-source-by-mixture-fraction? no
```

Note that entering **no** will automatically disable the linearization of the DPM mixture fraction source terms.

The source term linearization can be combined with the **Update DPM Sources Every Flow Iteration** option (Options for Interaction with the Continuous Phase (p. 2678)).

Note the following limitation:

- The linearized DPM source terms for mixture fractions and inert species (if enabled) are not compatible with the following UDF macros:
  - DEFINE_DPM_HEAT_MASS
  - DEFINE_DPM_LAW
  - DEFINE_DPM_SOURCE

  The linearization will be automatically disabled if any of the above UDFs is used in your case.

24.2.7.6. Using the Dynamic Interaction Range

DPM cases where the particle diameter may be too large compared to the local mesh cell size can produce an inaccurate solution. To get around this problem, you can enable the **Dynamic Interaction Range** option in the **Interaction Range** group box.

When the **Dynamic Interaction Range** option is enabled, physical models and phase interaction laws (such as drag, fluid temperature, and others) for such large particles are formulated based on freestream conditions of a coarser (or agglomerated) mesh elements. The DPM source terms are distributed over these coarser mesh elements, thus providing more realistic fluid conditions.

This coarser mesh interaction is applied dynamically, that is, the method is only applied if the particle is too large compared to the local cell size at the particle location.

Note the following limitation with the dynamic interaction range model:

- The dense discrete phase model (DDPM) is not supported.
- Two-sided walls (that is, wall / wall-shadow pairs) with fluid zones on both sides are ignored, that is, the coarser mesh elements stretch out over both sides of the two-sided wall, ignoring its presence.

24.2.7.7. Staggering of Particles in Space and Time

In order to obtain a better representation of an injector, the particles can be *staggered* spatially and/or temporally. When particles are staggered spatially, Ansys Fluent randomly samples from the region in which the spray is specified (for example, the sheet thickness in the pressure-swirl atomizer)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

so that as the calculation progresses, trajectories will originate from the entire region. This allows the entire geometry specified in the atomizer to be sampled while specifying fewer streams in the input dialog box, thus decreasing computational expense.

You can enable injection-specific spatial staggering of the particles by selecting the **Stagger Positions** option in the **Stagger Options** group box in the **Point Properties** tab of the **Set Injection Properties** dialog box and then, if applicable, specifying **Stagger Radius** to define the region from which particles are released. For more information, see the appropriate sections for injection-specific point properties.

You can also enable injection-specific spatial staggering of the particles by issuing the following text commands:

- `define/models/dpm/options/stagger-spatially-atomizer-injections?`
- `define/models/dpm/options/stagger-spatially-standard-injections?`

When spatial staggering is enabled for non-atomizer injections, a stagger radius must be specified by using the text command:

`define/models/dpm/options/stagger-radius`

When injecting particles in a transient calculation using a relatively large time step size in relation to the spray event, the particles can clump together in discrete bunches. The clumps do not look physically realistic, though Ansys Fluent calculates the trajectory for each particle as it passes through a cell and the coupling to the gas phase is properly accounted for. To obtain a statistically smoother representation of the spray, the particles can be staggered in time. During the first time step, the particle is tracked for a random percentage of its initial step. This results in a sample of the initial volume swept out by the particle during the first time step and a smoother, more uniform spatial distribution at longer time intervals.

The following setup commands for staggering in time are available in the text user interface (TUI):

- `stagger-temporally?`
- `staggering-factor`

The "staggering factor" is applied to temporal staggering only. It is a constant that multiplies the time step size to give the random sampling interval. The staggering factor controls the percentage of every particle's initial time step that will be sampled. For example, if the staggering factor is 0.5, then the parcels in the injection will be tracked between half and all of their full initial time step. If the staggering factor is 0.1, then the parcels will be tracked between ninety percent and all of their initial time step. If the staggering factor is set to 0.9, the parcels will be tracked between ten percent and all of their initial time step. This allows you to control the amount of smoothing between injections.

The default values for the options in the TUI are spatial staggering for atomizers (including the solid-cone), no temporal staggering and a temporal staggering factor of 1.0. The temporal staggering factor is inactive until the flag for temporal staggering is enabled.

Staggering uses pseudo-random numbers to compute initial particle positions. By default, the initialisation of the random number generator depends on the current iteration and time step numbers, so that the staggered positions of the particles vary between iterations. This helps to distribute

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DPM source terms more smoothly in the domain if only a limited number of injection streams is used.

In certain situations, the dependence of the initial particle position on the iteration and time step number may cause the overall convergence of the simulation to be slower than when the particle injection positions do not vary between iterations. In such cases, you can use the following text commands to make the random numbers independent of the current time step and iteration numbers:

- `define/models/dpm/numerics/re-randomize-every-iteration?`

  `Update randomness every iteration? [yes]` **n**

- `define/models/dpm/numerics/re-randomize-every-timestep?`

  `Update randomness every timestep? [yes]` **no**

When using these text commands, make sure that the number of streams used in every injection is sufficiently large.

24.2.7.8. Packing Limit

In transient simulations of granular phases using the DDPM model, DPM parcels may accumulate to unphysically large values, exceeding the packing limit of granular phases. To avoid such behavior, you can enable **Enhanced Numerics**. This option can be used in combination with the phase-specific option **Volume Fraction Approaching Packing Limit** (see Defining a Granular Discrete Phase (p. 3111) for more information).

24.2.7.9. Under-Relaxing Lagrangian Wall Film Height

As the wall film moves and spreads on the wall faces, under-relaxation is applied to the film height computation. The under-relaxation factor can be specified using the following text command:

`define/models/dpm/numerics/underrelax-film-height`

A lower under-relaxation factor results in a smoother film appearance, especially closer to the film edge. However, it may distort the film shape, especially for cases where high velocity droplets are impinging on the film. The recommended values range between 0.5 (default) and 0.9.

24.3. Setting Initial Conditions for the Discrete Phase

The primary inputs that you must provide for the discrete phase calculations in Ansys Fluent are the initial conditions that define the starting positions, velocities, and other parameters for each particle stream and the physical effects acting on the particle streams, requiring additional particle properties. You will define the initial conditions for a particle/droplet stream by creating an “injection” and assigning properties to it.

The required initial conditions depend on the injection type, while the physical effects are selected by choosing an appropriate particle type. For some injection types you can provide a particle size distribution, like the Rosin-Rammler distribution, see Using the Rosin-Rammler Diameter Distribution Method (p. 2745).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The initial conditions provide the starting values for all of the dependent discrete phase variables that describe the instantaneous conditions of an individual particle, and include the following:

- position (x, y, z coordinates) of the particle
- velocities (u, v, w) of the particle

  Velocity magnitudes and spray cone angle can also be used (in 3D) to define the initial velocities (see Point Properties for Cone Injections (p. 2726)). For moving reference frames, relative velocities should be specified.
- diameter of the particle, d_p
- temperature of the particle, T_p
- mass flow rate of the particle stream that will follow the trajectory of the individual particle/droplet, $\dot{m}_p$ (required only for coupled calculations)
- additional parameters if one of the atomizer models described in Atomizer Model Theory in the Theory Guide is used for the injection

Important:

When an atomizer model is selected, you will not specify initial diameter, velocity, and position quantities for the particles due to the complexities of sheet and ligament breakup. Instead of initial conditions, the quantities you will specify for the atomizer models are global parameters.

These dependent variables (temperature, diameter, and so on) are updated according to the equations of motion (Particle Motion Theory in the Theory Guide) and according to the heat/mass transfer relations applied (Laws for Heat and Mass Exchange in the Theory Guide) as the particle/droplet moves along its trajectory. You can define any number of different sets of initial conditions for discrete phase particles/droplets provided that your computer has sufficient memory.

For the setup of transient particle cases, the point properties for mass flow and velocity can be specified using transient profiles (see Point Properties for Transient Injections (p. 2778)).

For additional information, see the following sections:

24.3.1. Injection Types
24.3.2. Particle Types
24.3.3. Point Properties for Single Injections
24.3.4. Point Properties for Group Injections
24.3.5. Point Properties for Cone Injections
24.3.6. Point Properties for Surface Injections
24.3.7. Point Properties for Volume Injections
24.3.8. Point Properties for Plain-Orifice Atomizer Injections
24.3.9. Point Properties for Pressure-Swirl Atomizer Injections
24.3.10. Point Properties for Air-Blast/Air-Assist Atomizer Injections

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.3.11. Point Properties for Flat-Fan Atomizer Injections
24.3.12. Point Properties for Effervescent Atomizer Injections
24.3.13. Point Properties for File Injections
24.3.14. Point Properties for Condensate Injections
24.3.15. Using the Rosin-Rammler Diameter Distribution Method
24.3.16. Using the Tabulated (Discrete) Diameter Distribution
24.3.17. Creating and Modifying Injections
24.3.18. Defining Injection Properties
24.3.19. Specifying Injection-Specific Physical Models
24.3.20. Specifying Turbulent Dispersion of Particles
24.3.21. Custom Particle Laws
24.3.22. Defining Properties Common to More than One Injection
24.3.23. Point Properties for Transient Injections

24.3.1. Injection Types

Ansys Fluent provides the following types of injections:

- single
- group
- cone (only for 3D cases)
- surface
- volume (only for non-DEM 3D cases with unsteady particle tracking)
- plain-orifice atomizer
- pressure-swirl atomizer
- air-blast-atomizer
- flat-fan-atomizer
- effervescent-atomizer
- file
- condensate

For each non-atomizer injection type, you will specify each of the initial conditions listed in Setting Initial Conditions for the Discrete Phase (p. 2718), the type of particle that possesses these initial conditions, and any other relevant parameters for the particle type chosen.

When defining injection point properties for the following injection types:

- single
- group
- all cone injections
- all atomizer injections

you can interactively preview the current injection position and orientation in the graphics window prior to saving the settings. When you click **Update Injection Display** in the **Set Injection Properties** dialog box or **Set Multiple Injection Properties** dialog box, Ansys Fluent will apply the current settings

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

to the injection display. Once you click **OK** in the **Set Injection Properties** dialog box or **Apply** in the **Set Multiple Injection Properties** dialog box, the injection properties will be saved, and the graphics window will be once more updated. When you open the **Set Injection Properties** dialog box for the existing injection, Ansys Fluent will also automatically display the injection in the graphics window.

The injection position and orientation will not be displayed if you use profiles to specify the injection position and direction. If you use the `DEFINE_DPM_INJECTION_INIT` user-defined function, the displayed injection position and orientation will reflect only the default point properties for the specific injection type.

Use the following guidelines for choosing the injection type:

- Create a **single** injection when you want to specify a single value for each of the initial conditions (Figure 24.13: Particle Injection Defining a Single Particle Stream (p. 2722)).
- Create a **group** injection (Figure 24.14: Particle Injection Defining an Initial Spatial Distribution of the Particle Streams (p. 2722)) when you want to define a range for one or more of the initial conditions (for example, a range of diameters or a range of initial positions).
- To define **cone** injections in 3D problems, create a cone injection and specify a particle release pattern (point, hollow, ring, or solid) (Figure 24.15: Particle Injection Defining an Initial Spray Distribution of the Particle Velocity (p. 2722)).
- Create a **surface** injection to release particles from a surface (either a zone surface or a surface you have defined using **Create** in the **Domain** ribbon tab (**Surface** group box)). A particle stream will be released from each facet of the surface. You can use the **Bounded** and **Sample Points** options in the **Plane Surface** dialog box to create injections from a rectangular mesh of particles in 3D (see Plane Surfaces (p. 3830) for details).
- If you want to create a spray atomizer, you can use one of the predefined atomizer injections (**plain-orifice**, **pressure-swirl**, **air-blast**, **flat-fan**, or **effervescent**).
- Create a **volume** injection if you want to populate a volume zone or bounding geometry with particles. You can either choose one or several volume zones created when generating the mesh for your case or define a bounding geometry such as sphere, cylinder, cone, and hexahedron. See Defining Injection Properties (p. 2756) for details.
- Create a **file** injection to release particles with property settings stored in a file.
- If you are modeling film condensation, create a **condensate** injection to define the vapor-liquid material pair and other modeling settings for the film condensation process.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.13: Particle Injection Defining a Single Particle Stream

Figure 24.14: Particle Injection Defining an Initial Spatial Distribution of the Particle Streams

Figure 24.15: Particle Injection Defining an Initial Spray Distribution of the Particle Velocity

Particle initial conditions (position, velocity, diameter, temperature, and mass flow rate) can also be read from an external file if none of the injection types listed above can be used to describe your injection distribution.

The inputs for setting injections are described in detail in Defining Injection Properties (p. 2756).

24.3.2. Particle Types

When you define a set of initial conditions (as described in Defining Injection Properties (p. 2756)), you will need to specify the type of particle. The particle types available to you depend on the range of physical models that you have defined.

- A **massless** particle is a discrete element that follows the flow and temperature of the continuous phase. As it has no mass, it has no associated physical properties, and no force is exerted on it.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

However, you can assign a User-Defined Law to be applied to the massless particle. The massless particle type is available with all Ansys Fluent models.

- An **inert** particle is a discrete phase element (particle, droplet, or bubble) that obeys the force balance (Equation 12.1 in the Theory Guide) and is subject to heating or cooling via Law 1 (Inert Heating or Cooling (Law 1/Law 6) in the Theory Guide). The inert type is available for all Ansys Fluent models.

- A **droplet** particle is a liquid droplet in a continuous-phase gas flow that obeys the force balance (Equation 12.1 in the Theory Guide) and that experiences heating/cooling via Law 1 followed by vaporization and boiling via Laws 2 and 3 (Droplet Vaporization (Law 2) and Droplet Boiling (Law 3) in the Theory Guide). The droplet type is available when heat transfer is being modeled and at least two chemical species are active or the non-premixed or partially premixed combustion model is active. You should use the ideal gas law to define the gas-phase density (in the **Create/Edit Materials** dialog box, as discussed in Density Inputs for the Incompressible Ideal Gas Law (p. 1631)) when you select the droplet type.

- A **combusting** particle is a solid particle that obeys the force balance (Equation 12.1), and, after an initial phase of inert heating (Law 1), undergoes devolatization (Devolatilization (Law 4)) and then a heterogeneous surface reaction via Law 5 (Surface Combustion (Law 5) in the Theory Guide). Finally, the nonvolatile portion of a combusting particle is subject to inert heating via Law 6. You can also include an evaporating material with the combusting particle by selecting the **Wet Combustion** option in the Set Injection Properties Dialog Box (p. 5608). This allows you to include a material that evaporates and boils via Laws 2 and 3 (Droplet Vaporization (Law 2) and Droplet Boiling (Law 3) in the Theory Guide) before devolatilization of the particle material begins. The combusting type is available when heat transfer is being modeled and at least two chemical species are active or the non-premixed combustion model is active. You should use the ideal gas law to define the gas-phase density (in the Create/Edit Materials Dialog Box (p. 4885)) when you select the combusting particle type.

- A **multicomponent** particle is, as the name implies, a particle containing a mixture of several components or species. The conservation equations of all components, the energy equation, and vapor-liquid-equilibrium at the multicomponent particle surface form a coupled system of differential equations. You should use the volume weighted mixing law to define the particle mixture density (in the Create/Edit Materials Dialog Box (p. 4885)) when you select the particle-mixture material type. Law 7, the multicomponent law (Multicomponent Particle Definition (Law 7) in the Theory Guide) is used for the evaporating and boiling components of the particle.

  When all volatile components are evaporated via Law 7, multicomponent particles can participate in chemical reactions as described in Multicomponent Particles with Chemical Reactions (p. 2409) (Law 8). See Multicomponent Particles with Chemical Reactions in the *Fluent Theory Guide* for reaction types that multicomponent particles can undergo and reaction models that are available in Ansys Fluent.

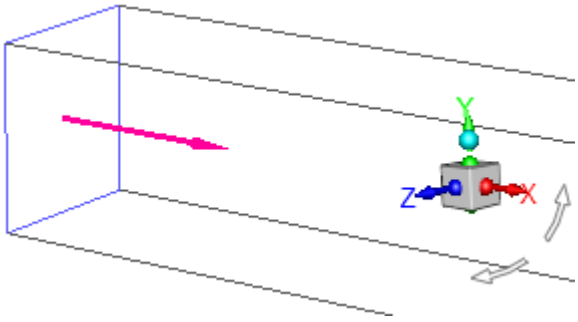
The types of particles and the laws they are subjected to are summarized in the table below.

Particle Type	Description	Laws Activated
Massless	–	–
Inert	inert/heating or cooling	1, 6
Droplet	heating/evaporation/boiling	1, 2, 3, 6

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Particle Type	Description	Laws Activated
Combusting	heating; evolution of volatiles/swelling; heterogeneous surface reaction	1, 4, 5, 6
Multicomponent	multicomponent droplets/particles	7, 8

24.3.3. Point Properties for Single Injections



For a single injection, you will define the following initial conditions for the particle stream under the **Point Properties** heading (in the Set Injection Properties Dialog Box (p. 5608)):

- position

  Set the x, y, and z positions of the injected stream along the Cartesian axes of the problem geometry in the **X-**, **Y-**, and **Z-Position** fields. (**Z-Position** will appear only for 3D problems.)

- velocity

  Set the x, y, and z components of the stream's initial velocity in the **X-**, **Y-**, and **Z-Velocity** fields. (**Z-Velocity** will appear only for 3D problems.)

- angular velocity (available when particle rotation is enabled (see Particle Rotation (p. 2767)))

  Set the x, y, and z components of the stream's initial velocity in the **X-**, **Y-**, and **Z-Angular-Velocity** fields. (Only **Z-Angular-Velocity** will appear for 2D problems, because rotation in 2D occurs in the plane, that is, the rotation vector is normal to the plane.)

- diameter

  Set the initial diameter of the injected particle stream in the **Diameter** field.

- temperature

  Set the initial (absolute) temperature of the injected particle stream in the **Temperature** field.

- mass flow rate

  For coupled phase calculations (see Solution Strategies for the Discrete Phase (p. 2827)), set the mass of particles per unit time that follows the trajectory defined by the injection in the **Flow Rate** field. Note that in axisymmetric problems the mass flow rate is defined per 2π radians and in 2D problems per unit meter depth (regardless of the reference value for length).

- duration of injection

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For unsteady particle tracking (see Steady/Transient Treatment of Particles (p. 2679)), set the starting and ending time for the injection in the **Start Time** and **Stop Time** fields. When In-Cylinder mesh motion is enabled, set the starting and ending crank angles for the injection in the **Start Crank Angle** and **Stop Crank Angle** fields. The values you enter are for one complete engine cycle. In subsequent cycles, the injection will be started and stopped according to Equation 13.30 (p. 1897), where θ_{event} corresponds to the injection start or end crank angle.

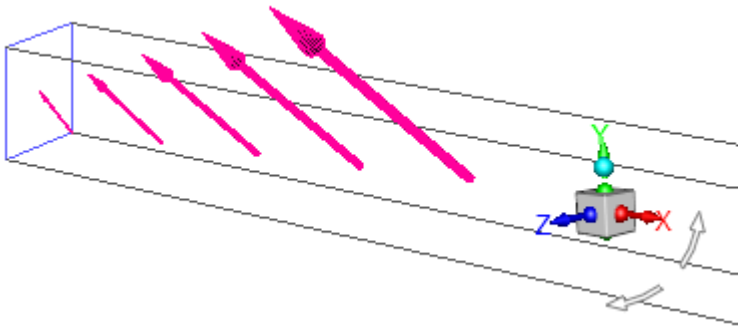
- time delay for reinjecting particles

  If the injection is used only for reinjecting particles (**Reinjection Only** is enabled), you can specify **Reinjection Time Delay**. See The **reinject** Boundary Condition (p. 2784) for details.

For the massless particle type, you will only need to define the position of the injection. The particle injection velocity is set by the solver equal to the velocity of the continuous phase at the injection point.

If you want to randomly distribute the initial placement of the particles, you can enable **Stagger Positions** and specify **Stagger Radius** to define the particle release region. For more information on spatial staggering, see Staggering of Particles in Space and Time (p. 2716). This option is disabled by default.

24.3.4. Point Properties for Group Injections



For group injections, you will define the properties position, velocity, angular velocity (available when particle rotation is enabled (see Particle Rotation (p. 2767))), diameter, temperature, and flow rate for the **First Point** and **Last Point** in the group. That is, you will define a range of values, ϕ_1 through ϕ_N, for each initial condition ϕ by setting values for ϕ_1 and ϕ_N. Ansys Fluent assigns a value of ϕ to the i^{th} injection in the group using a linear variation between the first and last values for ϕ:

$$\phi_i = \phi_1 + \frac{\phi_N - \phi_1}{N-1}(i-1) \tag{24.7}$$

Thus, for example, if your group consists of 5 particle streams and you define a range for the initial x location from 0.2 to 0.6 meters, the initial x location of each stream is as follows:

- Stream 1: x = 0.2 meters
- Stream 2: x = 0.3 meters
- Stream 3: x = 0.4 meters

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Stream 4: $x = 0.5$ meters
- Stream 5: $x = 0.6$ meters

Important:

In general, you should supply a range for only one of the initial conditions in a given group—leaving all other conditions fixed while a single condition varies among the stream numbers of the group. Otherwise you may find, for example, that your simultaneous inputs of a spatial distribution and a size distribution have placed the small droplets at the beginning of the spatial range and the large droplets at the end of the spatial range.

The specified flow rate is defined per particle stream and can also be interpolated using Equation 24.7 (p. 2725). When a Rosin-Rammler sized distribution is specified the total flow rate will be specified.

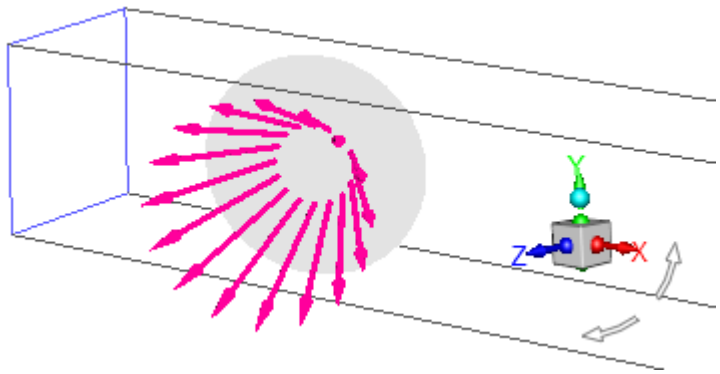
For the massless particle type, you will only need to define the first and last point of the injection group position. The particle velocities are set by the solver equal to the velocity of the continuous phase at the injection points.

If the injections are used only for reinjecting particles (**Reinjection Only** is enabled), you can specify **Reinjection Time Delay**. See The **reinject** Boundary Condition (p. 2784) for details.

Note that you can use a different method for defining the size distribution of the particles, as discussed below.

To randomly distribute the initial placement of the particles, enable **Stagger Positions** and specify **Stagger Radius** to define the particle release region. For more information on spatial staggering, see Staggering of Particles in Space and Time (p. 2716). This option is disabled by default.

24.3.5. Point Properties for Cone Injections



In 3D problems, you can define the following types of particle stream cones:

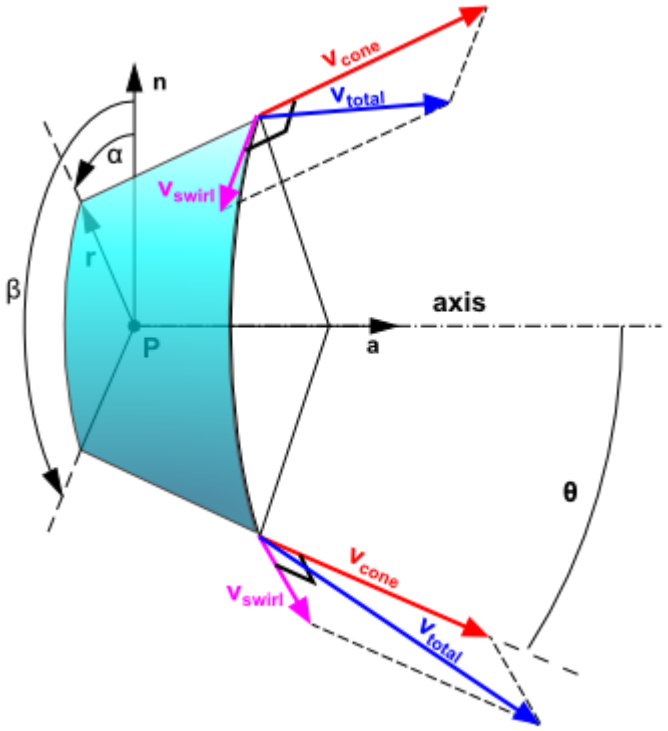
- **point-cone**
- **hollow-cone**
- **ring-cone**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **solid-cone**

The particles are released from a 2D region that may be a point, a circle, or an annulus. Its geometry, location, and orientation as well as the shape of the spray cone are defined by the injection parameters. Once you select a cone injection type from the **Cone Type** drop-down list, (the **Cone Injector Parameters** group box in the **Point Properties** tab of the **Set Injection Properties** dialog box), the parameters specific to the selected injection type will be displayed. These inputs are summarized below. Refer to Figure 24.16: Cone Injector Geometry (p. 2727) for an illustration of the geometry.

Figure 24.16: Cone Injector Geometry



- position

  Set the coordinates of the origin of the spray cone in the **X-**, **Y-**, and **Z-Position** fields (P in Figure 24.16: Cone Injector Geometry (p. 2727))

- diameter

  Set the diameter of the particles in the stream in the **Diameter** field.

- temperature

  Set the temperature of the streams in the **Temperature** field.

- duration of injection

  For unsteady particle tracking (see Steady/Transient Treatment of Particles (p. 2679)), set the starting and ending time for the injection in the **Start Time** and **Stop Time** fields. When In-Cylinder mesh motion is enabled, set the starting and ending crank angles for the injection in the **Start Crank Angle** and **Stop Crank Angle** fields.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- axis

  Set the x, y, and z components of the vector defining the cone's axis in the **X-Axis**, **Y-Axis**, and **Z-Axis** fields. (a in Figure 24.16: Cone Injector Geometry (p. 2727))

- velocity

  Set the velocity magnitude of the particle streams that will be oriented along the specified spray cone in the **Velocity Magnitude** field. (v_{total} in Figure 24.16: Cone Injector Geometry (p. 2727))

  Note:

  This is the total spray velocity magnitude and includes any swirl component in a hollow cone injector.

  $$\vec{v}_{total}=\vec{v}_{cone}+\vec{v}_{swirl}$$

- angular velocity

  Set the angular velocity magnitude of the particle streams that will be oriented along the specified spray cone in the **Angular Velocity Magnitude** field.

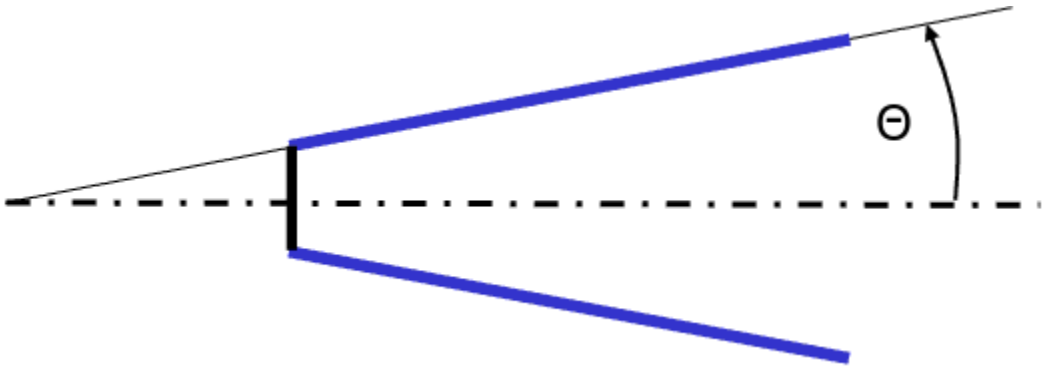
- cone angle

  Set the included half-angle of the spray cone in the **Cone Angle** field. (θ in Figure 24.16: Cone Injector Geometry (p. 2727))

- dispersion angle

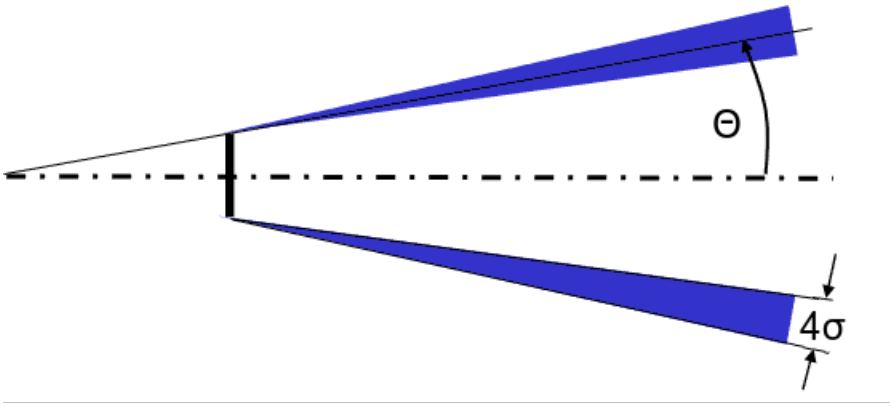
  For **hollow-cone**, you can optionally specify **Dispersion Angle** (default: 0). A non-zero value of the dispersion angle results in a distribution of the spray angle, that is, spray clustered about the mean cone angle using a Gaussian distribution with 0 mean and +/- 2 standard deviation (σ in Figure 24.18: Hollow Cone With Specified Cone Angle And Dispersion Angle (p. 2729)).

  Figure 24.17: Hollow Cone With Specified Cone Angle



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.18: Hollow Cone With Specified Cone Angle And Dispersion Angle



- radii

  For **hollow-cone**, **ring-cone**, and **solid-cone** injections, specify the **Outer Radius**.

  For **ring-cone** injections, specify a nonzero **Inner Radius** to model injectors that do not emanate from a single point. The particles will be distributed about the axis with the specified radius. (r in Figure 24.16: Cone Injector Geometry (p. 2727))

- swirl fraction (hollow cone only)

  Set the swirl fraction, which determines the relative magnitude of the swirl velocity, in the **Swirl Fraction** field.

  $$SF \equiv \frac{|\vec{v}_{swirl}|}{|\vec{v}_{cone}| + |\vec{v}_{swirl}|}$$

  The direction of the swirl component is defined using the right-hand rule about the cone axis (a negative value for the swirl fraction can be used to reverse the swirl direction).

- mass flow rate

  For coupled calculations, set the total mass flow rate for the streams in the spray cone in the **Total Flow Rate** field. Note that for a 3D sector, the flow rate must be appropriate for the sector defined by the **Azimuthal Start Angle** and **Azimuthal Stop Angle**.

- azimuthal angles

  For 3D sectors, set the **Azimuthal Start Angle** and **Azimuthal Stop Angle** to define the sector. (α and β, respectively, in Figure 24.16: Cone Injector Geometry (p. 2727)).

  The **Azimuthal Start Angle** and **Azimuthal Stop Angle** are specified with respect to a reference vector, $\vec{n}$, that is orthogonal to both the cone axis and the global X axis. The reference vector, $\vec{n}$, can be determined as the cross product of the cone axis and the global X axis.

  $$\vec{n} = \vec{a} \times \hat{i}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where $\hat{i}$ is a unit vector along the global X axis. In the case that the cone axis is along the X axis, then $\vec{n}$ is along the Y axis.

- time delay for reinjecting particles

  If the injection is used only for reinjecting particles (**Reinjection Only** is enabled), you can specify **Reinjection Time Delay**. See The **reinject** Boundary Condition (p. 2784) for details.

In addition to the above parameters, you can select the following options:

- **Uniform Massflow Distribution** (**solid-cone** or **ring-cone**): Specifies a uniform spatial distribution of mass over the cross-section of a cone. This is especially important when the mesh cell size is smaller than the particle diameter or another characteristic size. This option is enabled by default.
- **Stagger Positions**: Specifies a random spatial distribution of particle streams.

  For **hollow-cone** injections, you must also specify **Stagger Radius** to define the region over which particle release points are staggered in space. You can also enable the **Stagger in Injection Plane Only** option to restrict the spatial staggering of the initial particle positions to purely the radial direction of the spray. The spatial staggering clusters the initial particle positions about the outer radius of the hollow cone using a Gaussian distribution with 0 mean and +/- 2 standard deviation.

  For more information on spatial staggering, see Staggering of Particles in Space and Time (p. 2716). This option is disabled by default.

The distribution of the velocity directions in the particle streams for the point-cone, ring-cone, and solid-cone injections is random. Furthermore, duplicating this injection may not necessarily result in the same distribution, at the same location.

Important:

For transient calculations, the spatial distribution of streams at the initial injection location is recalculated at each time step. Sampling different possible trajectories allows a more accurate representation of a point, ring, or solid cone using fewer computational parcels. For steady-state calculations, the trajectories are initialized one time and kept the same for subsequent DPM iterations. The trajectories are recalculated when a change in the **Injections** dialog box occurs or when a case and data file are saved. If the residuals and solution change when a small change is made to the injection or when a case and data file are saved, it may mean that there are not enough trajectories being used to represent the point, ring, or solid cone with sufficient accuracy.

Note that you may want to define multiple spray cones emanating from the same initial location in order to specify a known size distribution of the spray or to include a known range of cone angles.

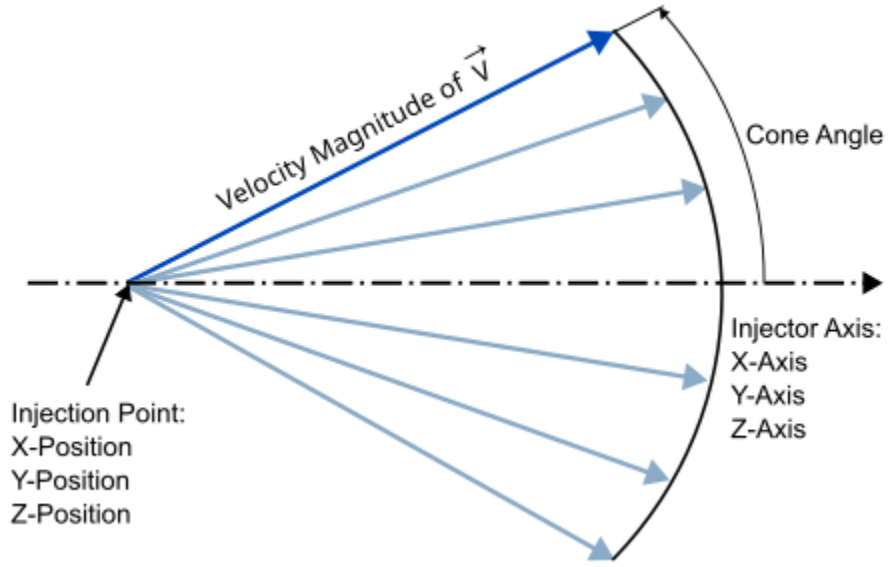
For the massless particle type, you will only need to define position, axis, cone angle, azimuthal angles, and radius. The particle velocities are set by the solver equal to the velocity of the continuous phase at the injection points.

Some additional details about injection point properties and other inputs for different cone injection types are provided below.

- Point injections

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.19: Point Injection

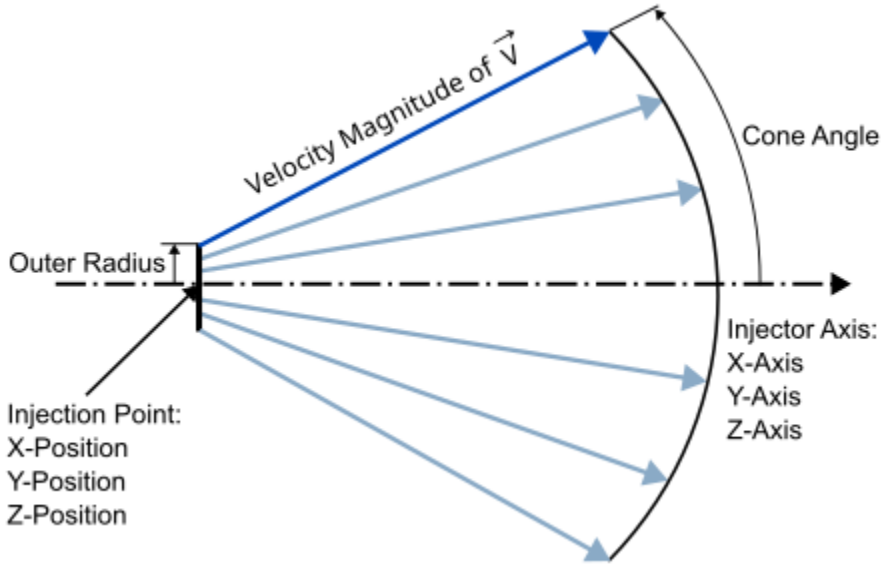


– **Number of Streams** parcels will be injected from the injection point.

– The envelope of the particle trajectories forms a cone with a total opening angle of 2***Cone Angle**.

- Solid injections

Figure 24.20: Solid Injection



– **Number of Streams** parcels will be injected from the injection plane.

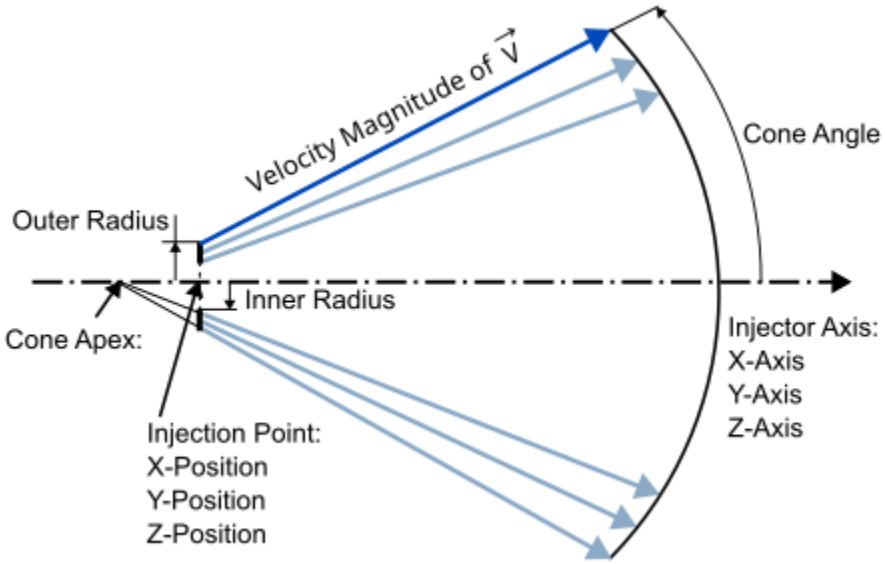
– Parcels start from any radial distance within the injection plane ($0 \le r_p \le r_{outer}$).

– The envelope of the particle trajectories forms a cone with a total opening angle of 2***Cone Angle**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Ring injections

  Figure 24.21: Ring Injection



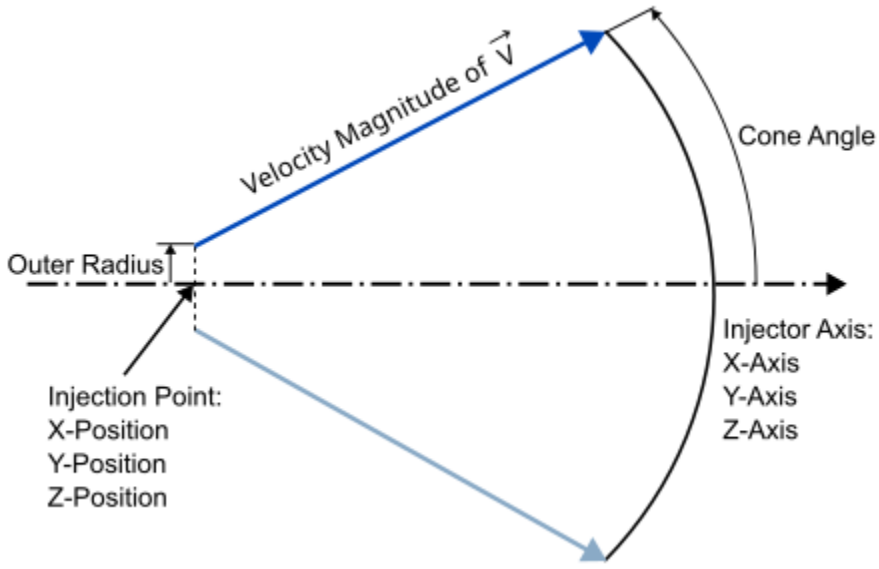
  – **Number of Streams** parcels will be injected from the injection plane.

  – Parcels start from any radial distance within the injection plane ($r_{inner} \leq r_p \leq r_{outer}$).

  – The inner envelope of the particle trajectories results from the position of the cone apex of the outer envelope (which is computed by the solver from the injection point properties) and the inner radius of the injection plane.

- Hollow injections

  Figure 24.22: Hollow Injection



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– **Number of Streams** parcels will be injected from the plane circumference.

– Parcels start from a fixed radial distance (r_p=r_{outer}).

– The outer envelope of the particle trajectories form a cone with a total opening angle of 2***Cone Angle**.

24.3.6. Point Properties for Surface Injections

For surface injections, you will define all the properties described in Point Properties for Single Injections (p. 2724) except for the initial position of the particle streams. The initial positions of the particles will be the location of the data points on the specified surface(s). Note that you will set the **Total Flow Rate** of all particles released from the surface (required for coupled calculations only). If you want, you can scale the individual mass flow rates of the particles by the ratio of the area of the face they are released from to the total area of the surface. To scale the mass flow rates, select the **Scale Flow Rate By Face Area** option under **Point Properties**. For the massless particle type, you will not need to enter any information to define a surface injection. The particle velocities are set by the solver equal to the velocities of the continuous phase at the injection points.

Note that many surfaces have nonuniform distributions of points. If you want to generate a uniform spatial distribution of particle streams released from a surface in 3D, you can create a bounded plane surface with a uniform distribution using the Plane Surface Dialog Box (p. 5580), as described in Plane Surfaces (p. 3830). In 2D, you can create a rake using the Line/Rake Surface Dialog Box (p. 5514), as described in Line and Rake Surfaces (p. 3826).

If you want to inject particles from positions distributed randomly over the selected boundary surfaces, enable **Randomize Starting Points** (**Surface Options** group box).

In addition to the option of scaling the flow rate by the face area, the normal direction of a face can be used for the injection direction. To use the face normal direction for the injection direction, select the **Inject Using Face Normal Direction** option under **Point Properties** (Figure 24.33: Setting Surface Injection Properties (p. 2758)). Once this option is selected, you only need to specify the velocity magnitude of the injection, not the individual components of the velocity magnitude.

Important:

- Only surface injections from boundary surfaces will be moved with the mesh when a sliding mesh, or a moving or deforming mesh is being used.
- For simulations that involve dynamic meshes, sliding meshes, or automatic adaption, only zonal surfaces can be selected.

A nonuniform size distribution can be used for surface injections, as described below.

24.3.6.1. Using the Rosin-Rammler Diameter Distribution Method for Surface Injections

The Rosin-Rammler size distributions described in Using the Rosin-Rammler Diameter Distribution Method (p. 2745) for group injections is also available for surface injections. If you select one of the Rosin-Rammler diameter distributions (**rosin-rammler** or **rosin-rammler-logarithmic**), you will

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

need to specify the following parameters under **Point Properties**, in addition to the initial velocity, temperature, and total flow rate:

- **Min. Diameter**

  This is the smallest diameter to be considered in the size distribution.

- **Max. Diameter**

  This is the largest diameter to be considered in the size distribution.

- **Mean Diameter**

  This is the size parameter, $\overline{d}$, in the Rosin-Rammler equation (Equation 24.9 (p. 2745)).

- **Spread Parameter**

  This is the exponential parameter, n, in Equation 24.9 (p. 2745).

- **Number of Diameters**

  This is the number of diameters in each distribution (that is, the number of different diameters in the stream injected from each face of the surface).

Ansys Fluent will inject streams of particles from each face on the surface, with diameters defined by the Rosin-Rammler distribution function. The total number of injection streams tracked for the surface injection will be equal to the number of diameters in each distribution (**Number of Diameters**) multiplied by the number of faces on the surface.

24.3.6.2. Injecting Wall Film Particles

For unsteady particle tracking, if you want to inject wall film particles from surfaces:

1. For a surface(s) from which wall film particles are released, specify the **wall-film** boundary condition in the **DPM** tab of the **Wall** dialog box. See Setting the Lagrangian Wall Film Model (p. 2802) for details.

2. In the **Set Injection Properties** dialog box, make sure that **surface** is selected as **Injection Type**.

3. In the **Point Properties** tab, enable the **Inject Wallfilm Particles** (**Wallfilm** group box).

   The **Injection Surfaces** multiple-selection list only displays the surfaces with the **wall-film** boundary condition.

4. In the **Injection Surfaces** multiple-selection list, select the surface(s) from which the wall-film particles will be released.

5. Specify other injection properties as appropriate for your case.

The **X-**, **Y-** or **Z-Velocity** will be projected in a directions parallel to each wall surface.

Note that the following options are not supported with the **Inject Wallfilm Particles** option:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Stagger Positions**

  Wall film particles must stay on the wall surface and are not allowed to hover above or below it.

- **Inject Using Face Normal Direction**

  The velocity of the wall-film particles should not be equal to the velocity of the particle release surface.

24.3.7. Point Properties for Volume Injections

For volume injections, you will define all the properties described in Point Properties for Single Injections (p. 2724) except for the initial positions of the particle streams. The initial positions of the particle parcels will be randomly chosen from points uniformly distributed in the volume zone or bounding geometry. See Defining Injection Properties (p. 2756) for details.

By default, the **Total Flow Rate** of the parcel to be injected into the particle release region is to be specified. If you prefer to specify the **Total Mass** of the parcel instead, enable the **Use Mass Instead of Flow Rate** option. If you enable the **Use Volume Fraction Instead of Flow Rate** option and specify **Volume Fraction** (rather than **Total Mass Rate** or **Total Mass**) in the **Set Injection Properties** dialog box (**Point Properties** tab). During simulation, parcels are introduced into the specified zones or bounding shape until either the specified **Volume Fraction** or **Injection Limit per Cell** is reached.

For the massless particle type, no input is required from you to define a volume injection. The particle velocities are set by the solver equal to the velocities of the continuous phase at the injection points.

Important:

If the injection **Stop Time** and **Start Time** are in different fluid flow time steps, then the particles will be injected at once in the **Fluid Time Step** that includes the middle of the injection time interval [**Start Time**, **Stop Time**].

For DDPM cases, the injection **Start Time** and **Stop Time** must be inside the same **Fluid Flow Time Step**; otherwise, no particles will be injected.

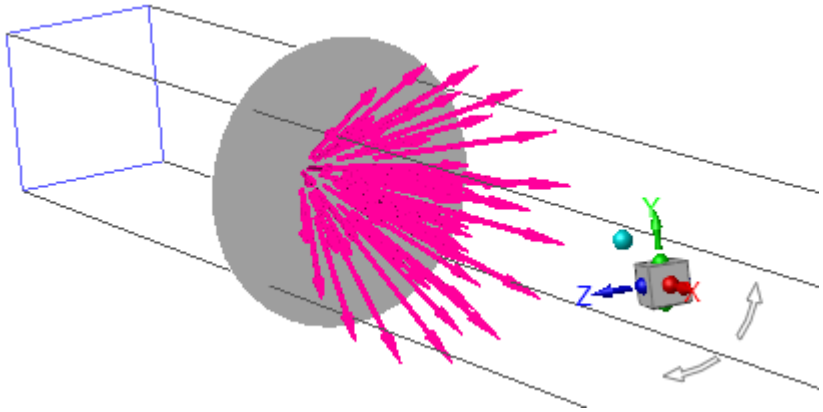
24.3.7.1. Using the Rosin-Rammler Diameter Distribution Method for Volume Injections

The Rosin-Rammler size distributions described in Using the Rosin-Rammler Diameter Distribution Method (p. 2745) for group injections is also available for volume injections. When you select **rosin-rammler** or **rosin-rammler-logarithmic** as **Diameter Distribution**, you need to specify additional point properties parameters described in Using the Rosin-Rammler Diameter Distribution Method for Surface Injections (p. 2733).

Ansys Fluent will inject particle parcels uniformly randomly distributed in either a specified zone or bounding geometry with diameters defined by the Rosin-Rammler distribution function. The total number of injected particle parcels tracked for the volume injection will be equal to the **Number of Diameters** in each distribution multiplied by either the **number-of-starting-points** or the **parcels-per-cell** specified for the zone or **number-of-starting-points** specified for the bounding geometry.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.3.8. Point Properties for Plain-Orifice Atomizer Injections



For a plain-orifice atomizer injection, you will define the following initial conditions under **Point Properties**:

- position

  Set the x, y, and z positions of the injected stream along the Cartesian axes of the problem geometry in the **X-Position**, **Y-Position**, and **Z-Position** fields. (**Z-Position** will appear only for 3D problems).

- axis (3D only)

  Set the x, y, and z components of the vector defining the axis of the orifice in the **X-Axis**, **Y-Axis**, and **Z-Axis** fields.

- temperature

  Set the temperature of the streams in the **Temperature** field.

- mass flow rate

  Set the total mass flow rate for the streams in the atomizer in the **Flow Rate** field. Note that in 3D sectors, the flow rate must be appropriate for the sector defined by the **Azimuthal Start Angle** and **Azimuthal Stop Angle**.

- duration of injection

  For unsteady particle tracking (see Steady/Transient Treatment of Particles (p. 2679)), set the starting and ending time for the injection in the **Start Time** and **Stop Time** fields. When In-Cylinder mesh motion is enabled, set the starting and ending crank angles for the injection in the **Start Crank Angle** and **Stop Crank Angle** fields.

- vapor pressure

  Set the vapor pressure governing the flow through the internal orifice (p_v in Table 12.7: List of Governing Parameters for Internal Nozzle Flow in the Theory Guide) in the **Vapor Pressure** field.

- diameter

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Set the diameter of the orifice in the **Injector Inner Diameter** field (d in Table 12.7: List of Governing Parameters for Internal Nozzle Flow in the Theory Guide).

- orifice length

  Set the length of the orifice in the **Orifice Length** field (L in Table 12.7: List of Governing Parameters for Internal Nozzle Flow in the Theory Guide).

- radius of curvature

  Set the radius of curvature of the inlet corner in the **Corner Radius of Curvature** field (r in Table 12.7: List of Governing Parameters for Internal Nozzle Flow in the Theory Guide).

- nozzle parameter

  Set the constant for the spray angle correlation in the **Constant A** field (C_A in Equation 12.369 in the Theory Guide).

- azimuthal angles

  For 3D sectors, set the **Azimuthal Start Angle** and **Azimuthal Stop Angle**.

- time delay for reinjecting particles

  If the injection is used only for reinjecting particles (**Reinjection Only** is enabled), you can specify **Reinjection Time Delay**. See The **reinject** Boundary Condition (p. 2784) for details.

See The Plain-Orifice Atomizer Model in the Theory Guide for details about how these inputs are used.

If you want to randomly distribute the initial placement of the particles, enable **Stagger Positions** (default). The region over which particle release points will be staggered is defined by the specified orifice diameter. For more information on spatial staggering, see Staggering of Particles in Space and Time (p. 2716).

24.3.9. Point Properties for Pressure-Swirl Atomizer Injections

For a pressure-swirl atomizer injection, you will specify some of the same properties as for a plain-orifice atomizer. In addition to the position, axis (if 3D), temperature, mass flow rate, duration of injection (if unsteady), injector inner diameter, and azimuthal angles (if relevant) described in Point Properties for Plain-Orifice Atomizer Injections (p. 2736), you will need to specify the following parameters under **Point Properties**:

- spray angle

  Set the value of the spray angle of the injected stream in the **Spray Half Angle** field (θ in Equation 12.381 in the Theory Guide).

- pressure

  Set the absolute pressure upstream of the injection in the **Upstream Pressure** field (p_1 in Table 12.7: List of Governing Parameters for Internal Nozzle Flow in the Theory Guide).

- sheet breakup

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Set the value of the empirical constant that determines the length of the ligaments that are formed after sheet breakup in the **Sheet Constant** field ($\ln\left(\frac{\eta_b}{\eta_0}\right)$ in Equation 12.388 in the Theory Guide).

- ligament diameter

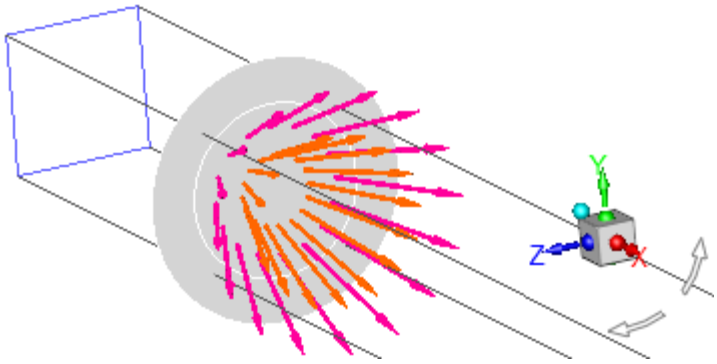
  For short waves, set the proportionality constant that linearly relates the ligament diameter, d_L, to the wavelength that breaks up the sheet in the **Ligament Constant** field (see Equation 12.389—Equation 12.392 in the Theory Guide).

- dispersion angle

  For a smooth distribution of the droplets, the initial velocities is varied within this dispersion angle. A sketch of the **Atomizer Dispersion Angle** for a flat fan atomizer is depicted in Figure 24.23: Flat Fan Viewed from Above and from the Side (p. 2741).

See The Pressure-Swirl Atomizer Model in the Theory Guide for details about how these inputs are used.

24.3.10. Point Properties for Air-Blast/Air-Assist Atomizer Injections



For an air-blast/air-assist atomizer, you will specify some of the same properties as for a plain-orifice atomizer. In addition to the position, axis (if 3D), temperature, mass flow rate, duration of injection (if unsteady), injector inner diameter, and azimuthal angles (if relevant) described in Point Properties for Plain-Orifice Atomizer Injections (p. 2736), you will need to specify the following parameters under **Point Properties**:

- outer diameter

  Set the outer diameter of the injector in the **Injector Outer Diameter** field. This value is used in conjunction with the **Injector Inner Diameter** to set the thickness of the liquid sheet (t in Equation 12.375 in the Theory Guide).

- spray angle

  Set the initial trajectory of the film as it leaves the end of the orifice in the **Spray Half Angle** field (θ in Equation 12.381 in the Theory Guide).

- relative velocity

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Set the maximum relative velocity that is produced by the sheet and air in the **Relative Velocity** field.

- sheet breakup

  Set the value of the empirical constant that determines the length of the ligaments that are formed after sheet breakup in the **Sheet Constant** field ($\ln\left(\frac{\eta_b}{\eta_0}\right)$ in Equation 12.388 in the Theory Guide).

- ligament diameter

  For short waves, set the proportionality constant (C_L in Equation 12.391 in the Theory Guide) that linearly relates the ligament diameter, d_L, to the wavelength that breaks up the sheet in the **Ligament Constant** field.

- dispersion angle

  For a smooth distribution of the droplets, the initial velocities is varied within this dispersion angle. A sketch of the **Atomizer Dispersion Angle** for a flat fan atomizer is depicted in Figure 24.23: Flat Fan Viewed from Above and from the Side (p. 2741).

See The Air-Blast/Air-Assist Atomizer Model in the Theory Guide for details about how these inputs are used.

24.3.11. Point Properties for Flat-Fan Atomizer Injections

The flat-fan atomizer model is available only for 3D models. For this type of injection, you will define the following initial conditions under **Point Properties**:

- arc position

  Set the coordinates of the center point of the arc from which the fan originates in the **X-Center**, **Y-Center**, and **Z-Center** fields (see Figure 24.23: Flat Fan Viewed from Above and from the Side (p. 2741)).

- virtual position

  Set the coordinates of the virtual origin of the fan in the **X-Virtual Origin**, **Y-Virtual Origin**, and **Z-Virtual Origin** fields. This point is the intersection of the lines that mark the sides of the fan (see Figure 24.23: Flat Fan Viewed from Above and from the Side (p. 2741)).

- normal vector

  Set the direction that is normal to the fan in the **X-Fan Normal Vector**, **Y-Fan Normal Vector**, and **Z-Fan Normal Vector** fields.

- temperature

  Set the temperature of the streams in the **Temperature** field.

- mass flow rate

  Set the mass flow rate for the streams in the atomizer in the **Flow Rate** field.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- duration of injection

  For unsteady particle tracking (see Steady/Transient Treatment of Particles (p. 2679)), set the starting and ending time for the injection in the **Start Time** and **Stop Time** fields. When In-Cylinder mesh motion is enabled, set the starting and ending crank angles for the injection in the **Start Crank Angle** and **Stop Crank Angle** fields.

- spray half angle

  Set the initial half angle of the drops as they leave the end of the orifice in the **Spray Half Angle** field.

- orifice width

  Set the width of the orifice (in the normal direction) in the **Orifice Width** field.

- sheet breakup

  Set the value of the empirical constant that determines the length of the ligaments that are formed after sheet breakup in the **Flat Fan Sheet Constant** field (see Equation 12.388 in the Theory Guide).

- dispersion angle

  For a smooth distribution of the droplets, the initial velocities is varied within this dispersion angle. A sketch of the **Atomizer Dispersion Angle** is depicted in Figure 24.23: Flat Fan Viewed from Above and from the Side (p. 2741).

- time delay for reinjecting particles

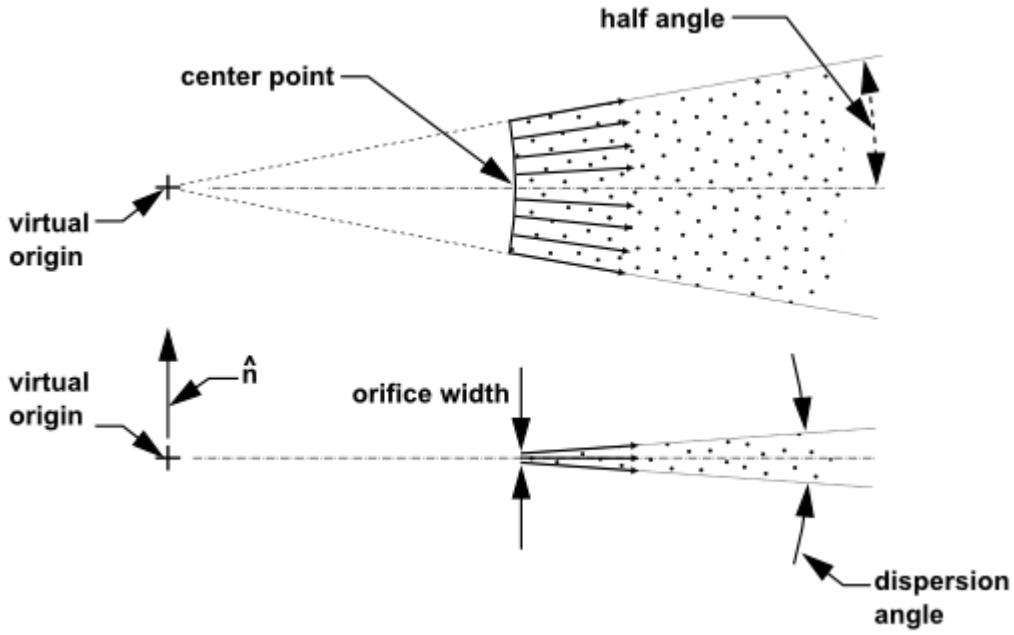
  If the injection is used only for reinjecting particles (**Reinjection Only** is enabled), you can specify **Reinjection Time Delay**. See The **reinject** Boundary Condition (p. 2784) for details.

See The Flat-Fan Atomizer Model in the Theory Guide for details about how these inputs are used.

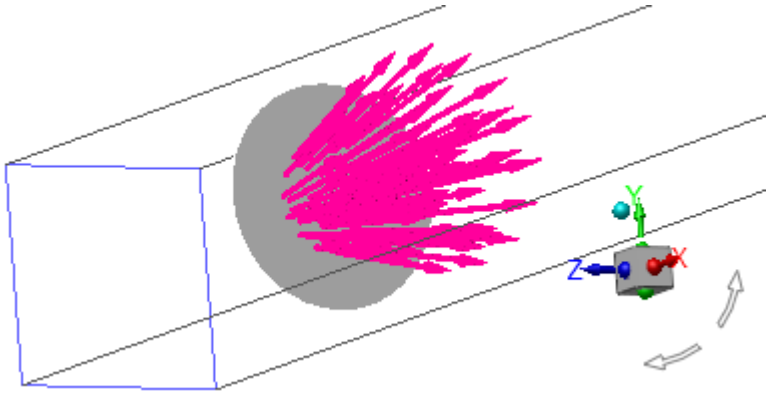
To randomly distribute the initial placement of the particles, enable **Stagger Positions** (default). The orifice width is used as radius to define the region over which particle release points will be staggered in space. For more information on spatial staggering, see Staggering of Particles in Space and Time (p. 2716).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.23: Flat Fan Viewed from Above and from the Side



24.3.12. Point Properties for Effervescent Atomizer Injections



For an effervescent atomizer injection, you will specify some of the same properties as for a plain-orifice atomizer. In addition to the position, axis (if 3D), temperature, mass flow rate (including both flashing and non-flashing components), duration of injection (if unsteady), vapor pressure, injector inner diameter, and azimuthal angles (if relevant) described in Point Properties for Plain-Orifice Atomizer Injections (p. 2736), you will need to specify the following parameters under **Point Properties**:

- mixture quality

  Set the mass fraction of the injected mixture that vaporizes in the **Mixture Quality** field (x in Equation 12.398 in the Theory Guide).

- saturation temperature

  Set the saturation temperature of the volatile substance in the **Saturation Temp.** field.

- droplet dispersion

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Set the parameter that controls the spatial dispersion of the droplet sizes in the **Dispersion Constant** field (C_{eff} in Equation 12.398 in the Theory Guide).

- spray angle

  Set the initial trajectory of the film as it leaves the end of the orifice in the **Maximum Half Angle** field.

See The Effervescent Atomizer Model in the Theory Guide for details about how these inputs are used.

24.3.13. Point Properties for File Injections

File injections can operate in two different modes, depending on the format of the file:

- A steady file can be used in both steady and unsteady particle tracking. See Steady File Format (p. 2742) for details.
- An unsteady file can only be used in unsteady particle tracking. See Unsteady File Format (p. 2743) for details.

An unsteady file can be converted into a steady-state injection file as described in Data Reduction of Samples (p. 2879).

The injection file mode is determined by the file header:

- If the file does not have any header, then it is treated as a steady file for backward compatibility reasons.
- If the file contains a header of the expected format, the information in the header will determine whether the file operates in the steady or unsteady mode as further described.

Important:

Using an unsteady injection file for steady particle tracking will result in an error.

24.3.13.1. Steady File Format

The steady file for a file injection has the following form:

```
(( x y z u v w diameter temperature mass-flow) name )
```

with all of the words denoting numeric parameters in scientific floating-point notation in SI units. All the parentheses are required, but the `name` is optional.

Sample files generated during sampling of trajectories for steady particle tracking as described in Sampling of Trajectories (p. 2873) can also be used as steady injection files since they have a similar file format.

The file can have an optional two-line header automatically generated by particle sampling for steady particle tracking in the following form:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
(z=4 12)
( x  y  z  u  v  w  diameter  t  mass-flow  mass  frequency  time  name)
```

In the first line, you can replace z=4 with an arbitrary label that must not contain any white-space characters. You must not change the rest of the header, but you can add any number of white space characters between its elements.

If the steady file is used in unsteady particle tracking, for every line entry in the file, a new particle parcel is generated for every injection time step in the usual way. By default, injection time steps are identical to the global fluid-flow time steps; however, you can change it in the **Discrete Phase Model** dialog box.

Note:

The information in columns 10 and 11 (mass and frequency) are not used for generating particles. They are only used in a check for consistency with the particle mass specified in column 9. No check for consistency will be made for the particle mass in relation to the particle diameter (column 7). The consistency check can be suppressed by setting the values in columns 10 and 11 to zero

---

24.3.13.2. Unsteady File Format

The unsteady file must have a two-line header of the following format:

```
(z=4 13)
( x  y  z  u  v  w  diameter  t  parcel-mass  mass  n-in-parcel  time  flow-time  name)
```

In the first line, you can replace z=4 with an arbitrary label that must not contain any white-space characters. You must not change the rest of the header, but you can add any number of white space characters between its elements. The header is automatically written by Ansys Fluent into every sampling file for unsteady particle tracking.

The unsteady file for a file injection has the following form:

```
(( x  y  z  u  v  w  diameter  temperature  parcel-mass  mass  n-in-parcel  time  flow-time)  name)
```

Here, every line entry in the file contains a time stamp (flow-time in column 13). At the appropriate simulated flow-time, Ansys Fluent will inject a single particle parcel only once. That means that during the entire time interval that it spans, an unsteady injection file produces as many particle parcels as the number of the data line entries in the file.

Like for the steady file format, all the numbers must be in SI units, and all the parentheses are required, but the name is optional. A file that Ansys Fluent writes from unsteady particle tracking can be used as an injection file in the unsteady mode.

Note:

The information in columns 10 and 11 (mass and n-in-parcel) are not used for generating particles. They are only used in a check for consistency with the particle mass specified in column 9. No check for consistency will be made for the particle mass in

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

relation to the particle diameter (column 7). The consistency check can be suppressed by setting the values in columns 10 and 11 to zero

An unsteady injection file can contain additional columns that can be processed using a `DEFINE_DPM_INJECTION_INIT` UDF. For details, see `DEFINE_DPM_INJECTION_INIT` in the *Fluent Customization Manual*.

Note:

In an unsteady injection file, all entries must be sorted strictly in ascending order of the simulated `flow-time` (column 13). If you use unsteady particle sampling in Ansys Fluent to generate such a file, you must select the **Sort Sample Files** option in the **Sample Trajectories** dialog box.

24.3.13.3. User Input for File Injections

In the **Point Properties** tab in the **Set Injection Properties** dialog box, you can specify the following injection parameters:

Start Time

specifies the simulated time when Ansys Fluent starts to inject particles from this injection.

Stop Time

specifies the simulated time when Ansys Fluent stops to inject particles from this injection.

Start Flow-time in File

(optional) determines the line from which Ansys Fluent starts reading the injection file. All line entries in which `flow-time` (13^{th} column) is less than the specified value will be skipped. For remaining lines, the injection release time is determined as:

Start Time + `flow-time` - **Start Flow-time in File**

The default value of 0 means that Ansys Fluent will read the injection file starting from the first line entry.

Repeat Interval in File

(optional) can be used to put the unsteady injection into a time-periodic mode. Once Ansys Fluent has read the injection file for the specified time period, it rewinds the file and starts reading it again from the line from which it began. This cycling process continues until **Stop Time** is reached. In this way, detailed data for a small time interval can be used to approximate a steady state over longer time periods in an unsteady simulation with unsteady particle tracking. The default value of 0 means no repetition in time.

Note:

For steady injection files, **Start Flow-time in File** and **Repeat Interval in File** are ignored.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.3.14. Point Properties for Condensate Injections

For a condensate injection, you will define only the duration of the injection by setting the starting and ending time for the injection in the **Start Time** and **Stop Time** fields under the **Point Properties** tab in the **Set Injection Properties** dialog box. The duration of the injection determines the time during which film condensation will be initiated on the walls that have the **Film Condensation** option enabled.

24.3.15. Using the Rosin-Rammler Diameter Distribution Method

For liquid sprays, a convenient representation of the droplet size distribution is the Rosin-Rammler expression. The complete range of sizes is divided into an adequate number of discrete intervals; each represented by a mean diameter for which trajectory calculations are performed. If the size distribution is of the Rosin-Rammler type, the mass fraction of droplets of diameter greater than d is given by

$$Y_d = e^{-(d/\bar{d})^n} \tag{24.8}$$

where $\bar{d}$ is the size constant and n is the size distribution parameter.

By default, you will define the size distribution of particles by inputting a diameter for the first and last points and using the linear equation (Equation 24.7 (p. 2725)) to vary the diameter of each particle stream in the group. When you want a different mass flow rate for each particle/droplet size, however, the linear variation may not yield the distribution you need. Your particle size distribution may be defined most easily by fitting the size distribution data to the Rosin-Rammler equation. In this approach, the complete range of particle sizes is divided into a set of discrete size ranges, each to be defined by a single stream that is part of the group. Assume, for example, that the particle size data obeys the following distribution:

Diameter Range (μm)	Mass Fraction in Range
0–70	0.05
70–100	0.10
100–120	0.35
120–150	0.30
150–180	0.15
180–200	0.05

The Rosin-Rammler distribution function is based on the assumption that an exponential relationship exists between the droplet diameter, d, and the mass fraction of droplets with diameter greater than d, Y_d:

$$Y_d = e^{-(d/\bar{d})^n} \tag{24.9}$$

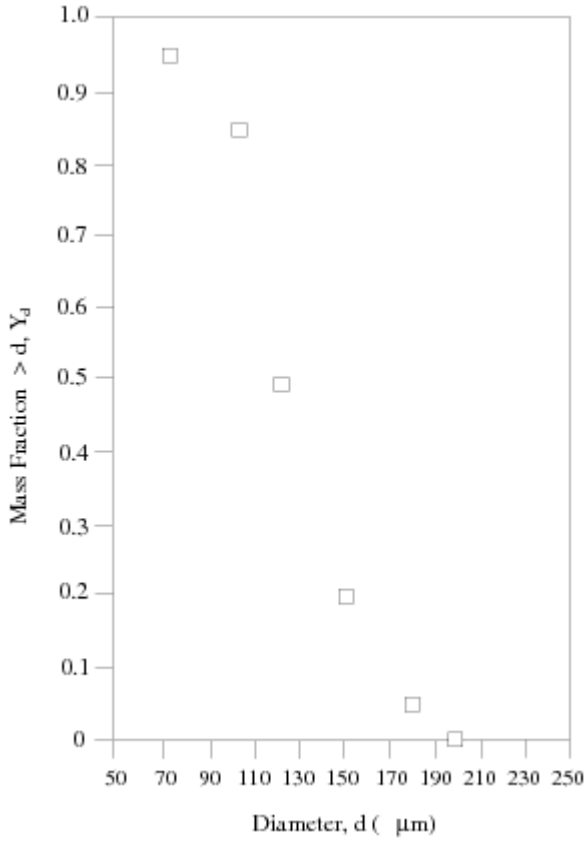
Ansys Fluent refers to the quantity $\bar{d}$ in Equation 24.9 (p. 2745) as the **Mean Diameter** and to n as the **Spread Parameter**. These parameters are specified by you (in the Set Injection Properties Dialog Box (p. 5608) under the **First Point** heading) to define the Rosin-Rammler size distribution. To solve for these parameters, you must fit your particle size data to the Rosin-Rammler exponential equation. To determine these inputs, first recast the given droplet size data in terms of the Rosin-Rammler format. For the example data provided above, this yields the following pairs of d and Y_d:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Diameter, d (μm)	Mass Fraction with Diameter Greater than d, Y_d
70	0.95
100	0.85
120	0.50
150	0.20
180	0.05
200	(0.00)

A plot of Y_d vs. d is shown in Figure 24.24: Example of Cumulative Size Distribution of Particles (p. 2746).

Figure 24.24: Example of Cumulative Size Distribution of Particles



Next, derive values of $\bar{d}$ and n such that the data in Figure 24.24: Example of Cumulative Size Distribution of Particles (p. 2746) fit Equation 24.9 (p. 2745). The value for $\bar{d}$ is obtained by noting that this is the value of d at which $Y_d = e^{-1} \approx 0.368$. From Figure 24.24: Example of Cumulative Size Distribution of Particles (p. 2746), you can estimate that this occurs for $d \approx 131 \mu$m. The numerical value for n is given by

$$n = \frac{\ln\left(-\ln Y_d\right)}{\ln\left(d/\bar{d}\right)} \tag{24.10}$$

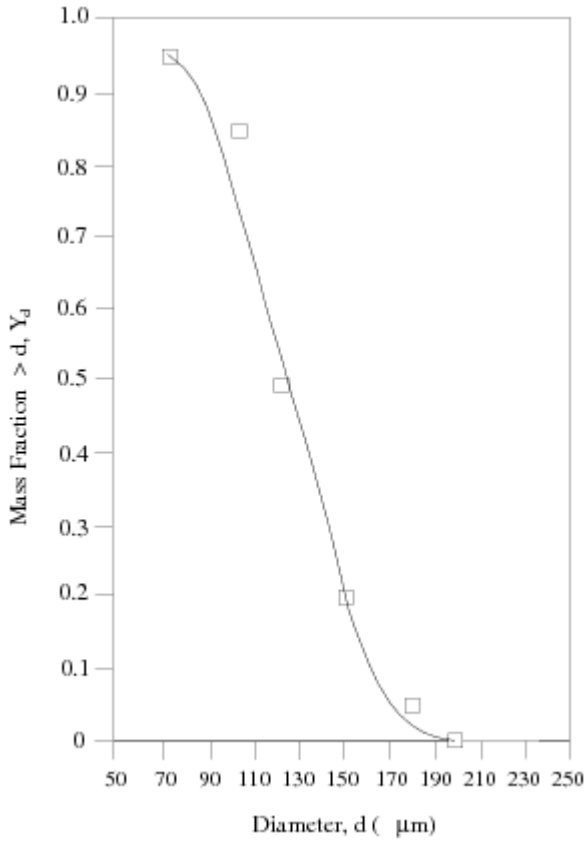
By substituting the given data pairs for Y_d and $d/\bar{d}$ into this equation, you can obtain values for n and find an average. Doing so yields an average value of n = 4.52 for the example data above. The

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

resulting Rosin-Rammler curve fit is compared to the example data in Figure 24.25: Rosin-Rammler Curve Fit for the Example Particle Size Data (p. 2747). You can enter values for $\overline{d}$ and n, as well as the diameter range of the data and the total mass flow rate for the combined individual size ranges, using the Set Injection Properties Dialog Box (p. 5608).

This technique of fitting the Rosin-Rammler curve to spray data is used when reporting the Rosin-Rammler diameter and spread parameter in the **Discrete Phase Summary** dialog box in Summary Reporting of Current Particles (p. 2883).

Figure 24.25: Rosin-Rammler Curve Fit for the Example Particle Size Data



A second Rosin-Rammler distribution is also available based on the natural logarithm of the particle diameter. If in your case, the smaller-diameter particles in a Rosin-Rammler distribution have higher mass flows in comparison with the larger-diameter particles, you may want better resolution of the smaller-diameter particle streams, or "bins". You can therefore choose to have the diameter increments in the Rosin-Rammler distribution done uniformly by $\ln d$.

In the standard Rosin-Rammler distribution, a particle injection may have a diameter range of 1 to 200 μm. In the logarithmic Rosin-Rammler distribution, the same diameter range would be converted to a range of $\ln 1$ to $\ln 200$, or about 0 to 5.3. In this way, the mass flow in one bin would be less-heavily skewed as compared to the other bins.

When a Rosin-Rammler size distribution is being defined for the group of streams, you should define (in addition to the initial velocity, position, and temperature) the following parameters, which appear under the heading for the **First Point**:

- **Total Flow Rate**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This is the total mass flow rate of the N streams in the group. Note that in axisymmetric problems this mass flow rate is defined per 2π radians and in 2D problems per unit meter depth.

- **Min. Diameter**

  This is the smallest diameter to be considered in the size distribution.

- **Max. Diameter**

  This is the largest diameter to be considered in the size distribution.

- **Mean Diameter**

  This is the size parameter, $\overline{d}$, in the Rosin-Rammler equation (Equation 24.9 (p. 2745)).

- **Spread Parameter**

  This is the exponential parameter, n, in Equation 24.9 (p. 2745).

24.3.15.1. The Stochastic Rosin-Rammler Diameter Distribution Method

For atomizer injections, a Rosin-Rammler distribution is assumed for the particles exiting the injector. In order to decrease the number of particles necessary to accurately describe the distribution, the diameter distribution function is randomly sampled for each instance where new particles are introduced into the domain.

The Rosin-Rammler distribution can be written as

$$1-Y=\exp\left[-\left(\frac{D}{\overline{d}}\right)^{n}\right] \tag{24.11}$$

where Y is the mass fraction smaller than a given diameter D, $\overline{d}$ is the Rosin-Rammler diameter and n is the Rosin-Rammler exponent. This expression can be inverted by taking logs of both sides and rearranging,

$$D=\overline{d}\left(-\ln\left(1-Y\right)\right)^{1/n}. \tag{24.12}$$

Given a mass fraction Y along with parameters $\overline{d}$ and n, this function will explicitly provide a diameter, D. Diameters for the atomizer injectors described in Point Properties for Plain-Orifice Atomizer Injections (p. 2736) are obtained by uniformly sampling Y in Equation 24.12 (p. 2748).

24.3.16. Using the Tabulated (Discrete) Diameter Distribution

For cone, surface, and volume injections, when droplet size distribution data is available in a tabular form, you can specify the diameter distribution using a text file.

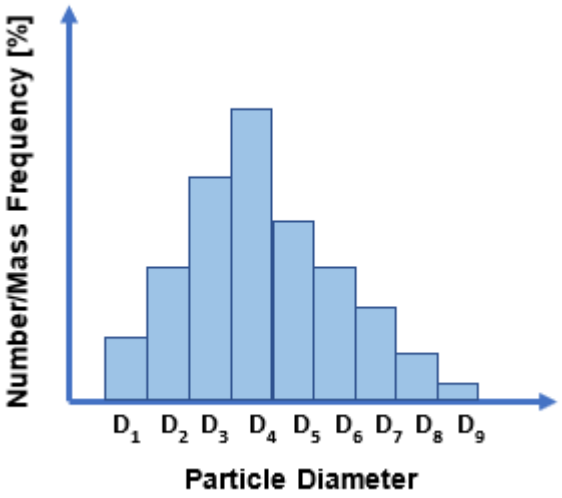
It is assumed that a measured size distribution can be approximated by a discrete number of size intervals (also called "classes"). Each class is defined by the following parameters (see Figure 24.26: Discrete Number/Mass Fraction Distribution (p. 2749)):

- Representative particle size D_i,
- Number fraction $f_n(D_i)$, and/or

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

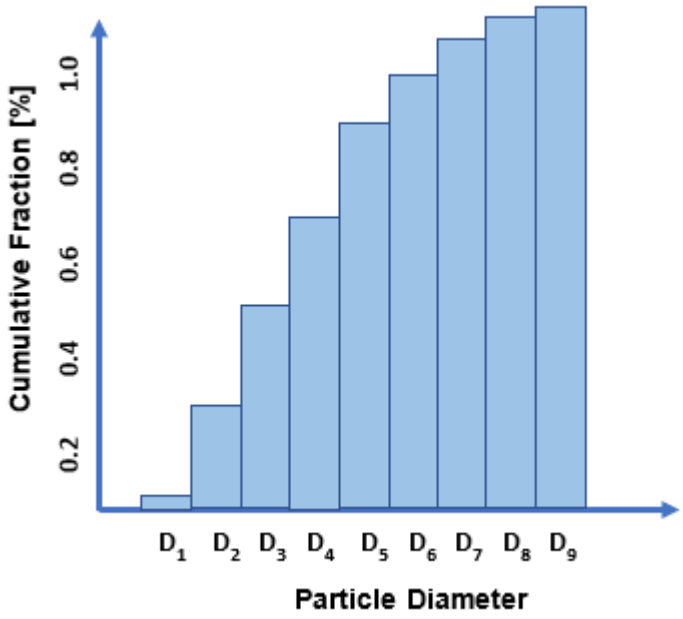
- Mass fraction $f_m(D_i)$

Figure 24.26: Discrete Number/Mass Fraction Distribution



The size distribution data can also be given as a cumulative size distribution, that is, the sum of the frequency/mass distributions (see Figure 24.27: Discrete Cumulative Number/Mass Fraction Distribution (p. 2749)).

Figure 24.27: Discrete Cumulative Number/Mass Fraction Distribution



To specify the droplet size distribution in a tabular format:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

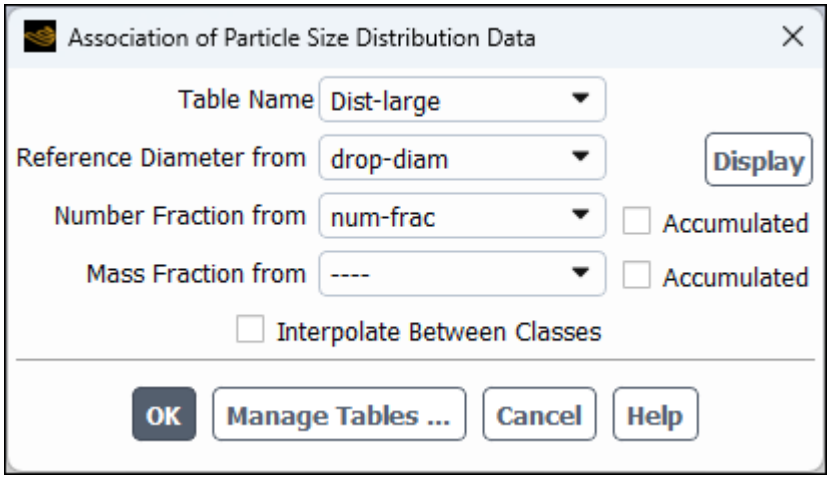
1. In the **Set Injection Properties** dialog box, select **tabulated** from the **Diameter Distribution** drop-down list.

   Note:

   The **tabulated** option is available only for the **cone** and **surface** injection types.

2. In the **Association of Particle Size Distribution Data** dialog box that opens automatically, click **Manage Tables...**.

   Figure 24.28: Association of Particle Size Distribution Data Dialog Box



3. Use the **Table File Manager** dialog box that opens to read a table file containing droplet size distribution data. See Reading Files in Tabular Format (p. 969) for more information on how to read table files.

   Alternatively, you can read a table file as described in Reading Files in Tabular Format (p. 969).

   The size distribution data table is formatted as follows:

   - Row 1: Name of the table
   - Row 2: Header with the column names
   - Row 3 and all consecutive rows: Spray data points

     The spray data points must contain the following columns separated by a space or tab delimiter:

     – Droplet diameter representative for an interval of the spray

       The reference diameters must be sorted in either ascending or descending order.

     – Number fraction and/or mass fraction

       → Cumulative and/or non-cumulative data may be specified.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

→ Number/mass fractions do not need to be normalized. Normalization will be performed by the solver, so that the cumulative and non-cumulative fractions are in the range of 0 to 1.

An example of the data file for the diameter size distribution data is shown below.

```
Dist-large
drop-diam     num-frac     mass-frac     cum-num-frac   cum-vol-frac
0.0001        0.10         1.            0.10           1
0.0002        0.2          1.            0.30           2
0.0003        0.4          1.            0.70           3
0.0004        0.4          1.            1.10           4
0.0005        0.3          1.            1.40           5
0.0006        0.28         1.            1.68           6
0.0007        0.25         1.            1.93           7
0.0008        0.22         1.            2.15           8
0.0009        0.20         1.            2.35           9
0.0010        0.15         1.            2.50           10
…
…
```

In this example, `Dist-large` is the name of the table. The column names are specified in the second line. The spray data points are provided starting from line 3 through the end of the file.

Once the tabulated size distribution has been read into your case, the name of the table and its column names become visible and selectable in the **Association of Particle Size Distribution Data** dialog box. The following data will be stored in your case file:

- Number of diameter classes
- Representative diameters per class
- Number fraction per class and/or mass fraction per class
- Other injection properties (such as the number of streams to be injected and the material density)

4. In the **Association of Particle Size Distribution Data** dialog box, link the size distribution parameters to the tabulated distribution data:

   a. Select the name of your table from the **Table Name** drop down box.

   b. From the **Reference Diameter from**, **Number Fraction from**, and/or **Mass Fraction from** drop-down lists, select the column names where the reference diameter, number fraction, and mass fraction are stored, respectively.

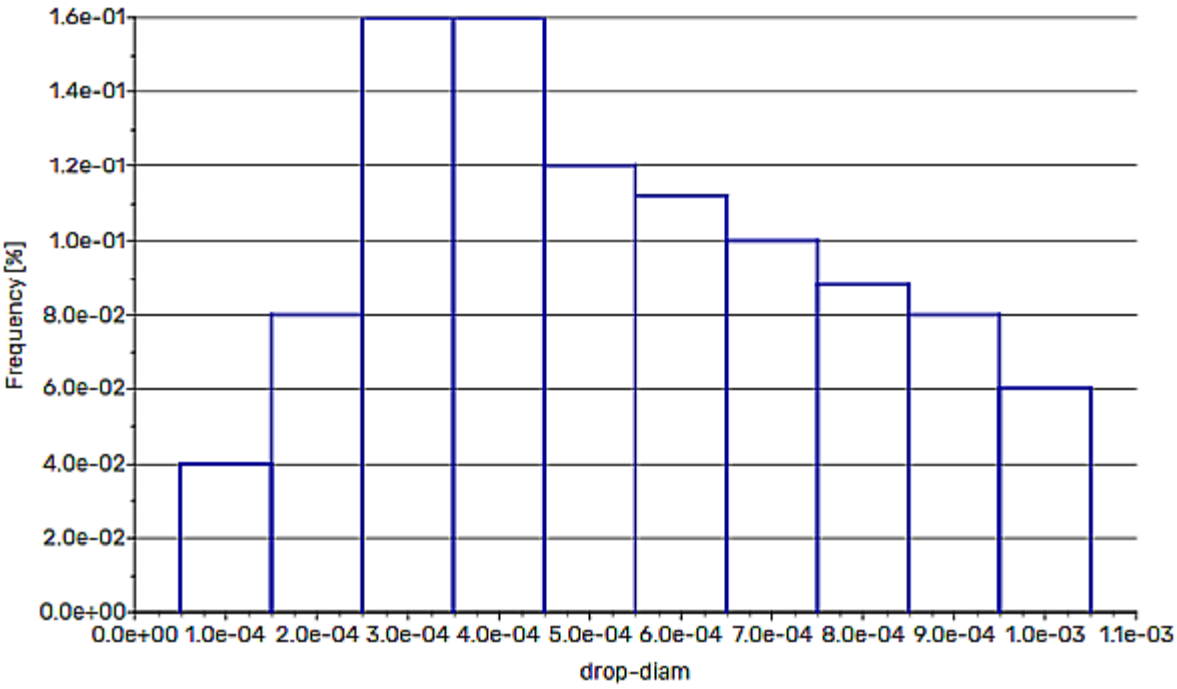
5. If the number fraction or mass fraction is available as cumulative data, select the **Accumulated** check box next to the corresponding variable.

6. Select **Interpolate Between Classes** if you want to generate particle diameters by linearly interpolating between neighboring diameter classes. If this option is not selected, all particles within a class will be assigned the representative class diameter.

7. If you want to visualize the tabulated size distribution of the reference diameter over either mass fraction or number fraction, then make sure that only the column names to be used for abscissa and ordinate are selected and click **Display**. A histogram of the droplet size distribution data is plotted in the graphics window.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.29: Tabulated Size Distribution Histogram of Diameter Over Number Fraction



8. Click **OK** to close the **Association of Particle Size Distribution Data** dialog box.

The particle tracker first assigns particles to different diameter classes based on the supplied number fraction data. In case no number fraction data has been provided, the particle tracker derives the number-per-class information from the mass-per-class data (calculated as the mass fraction per class multiplied by the total injected mass), the material density, and the mass of the individual particle of the class.

The process of assigning a particle to a certain diameter class uses a random number generator.

- For cone injections, to ensure that each class receives at least one representative particle, you must specify a sufficient number of parcels in the **Number of Streams** text entry field in the **Set Injection Properties** dialog box. As a minimum requirement, you must inject at least as many parcels as there are classes. You can obtain a rough estimate for the required number of parcels to be injected by taking the inverse of the smallest number fraction from your data. The solver, however, will tolerate empty diameter classes. The total particle mass to be injected is enforced by assigning a mass to all classes that contain at least one parcel.

- For surface injections, a minimum of one parcel per surface facets will be injected.

- For volume injections, you must specify a sufficient number of starting points in the **number-of-starting-points** or **parcels-per-cell** text entry field. Note that for simulations that involve DDPM, these entry fields are not available since the number of starting points is calculated automatically based on the smallest cell in the mesh. The diameter of the injected parcels is calculated so that five parcels would fit into the smallest cell volume in the mesh, while also satisfying the Injection **Injection Packing Limit per Cell**. If the number or the mass fraction per cell is too small to create at least one parcel in a diameter class, this class will remain empty.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Depending on the data you provide, the particle tracker uses one of the following approaches:

- Only the number fraction per class is specified

  In this scenario, the tracker accumulates the provided number fraction data and scales it to the range [0, 1]. This normalized data is subsequently used to assign a particle to a specific diameter class. For each particle to be injected, the tracker generates a unique random number. The number determines which size class the particle falls into. Based on the size class, the tracker sets the particle diameter. The diameter can either be either a representative diameter specified for the class or an interpolated diameter between neighboring classes to provide a more realistic spray representation.

- Only the mass fraction is specified

  Based on the specified mass fraction, particle material, and representative particle diameter, the approximate number of particles per class is determined. This information is used to derive the number fraction per class. Once this step is completed, the same approach is used as in the first scenario (where only the number fraction per class is specified). The tracker enforces the specified flow rate per class.

- Both the number fraction and mass fraction are specified

  Based on the specified number fraction data, the particle class/diameter assignment is performed as in the first scenario (where only the number fraction per class is specified). In addition, the tracker enforces the specified flow rate per class.

In cases where the number of initial particle positions is so small that the random sampling approach cannot accurately represent the diameter distribution of a spray, you can disable this approach using the following text command:

```
define/models/dpm/injections/properties/set/size-distribution/use-randomized-diameter-sampling?
Use randomized sampling for tabulated diameter distribution? [yes] no
```

When this option is disabled, particles from each tabulated diameter class will be injected from each particle starting position.

24.3.17. Creating and Modifying Injections

You will use the **Injections** branch in the **Outline View** and the Injections Dialog Box (p. 5496) (Figure 24.31: The Injections Dialog Box (p. 2754), Figure 24.30: The Injections Branch in the **Outline View** (p. 2754)) to create, copy, delete, list, read, and write injections.

Setup → Models → Discrete Phase → Injections → New...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.30: The Injections Branch in the Outline View

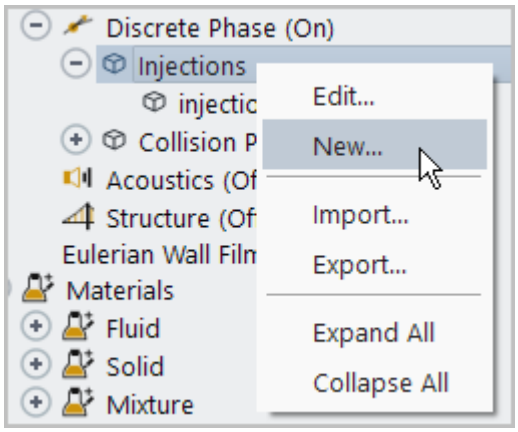
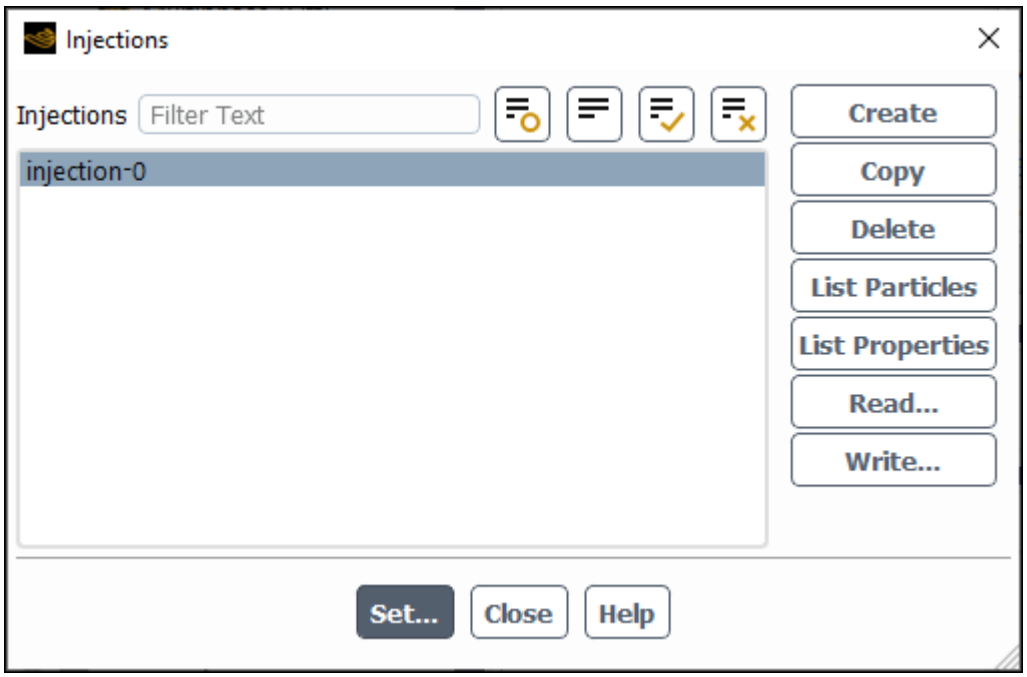


Figure 24.31: The Injections Dialog Box



(You can also click the **Injections...** button in the Discrete Phase Model Dialog Box (p. 4845) to open the **Injections** dialog box.)

Note:

[Linux Only] If you are not part of a video group, a session on a local Linux machine with SLES11, SP4, and OpenGL may terminate abnormally during an injection setup for the discrete phase model case. To avoid this issue, you should restart the Ansys Fluent session with the driver option `-x11` or you should ensure that you are added to the video group on that machine. **Workaround:** Deactivate the injection visualization during the case setup by using the following scheme command:

```
rpsetvar 'dpm/visualize-injections? #f)
```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.3.17.1. Creating Injections

To create an injection, click the **Create** button. The Set Injection Properties Dialog Box (p. 5608) will open automatically to allow you to set the injection properties (as described in Defining Injection Properties (p. 2756)). After the injection is created, the new injection will appear in the **Injections** list, in the **Injections** dialog box.

24.3.17.2. Modifying Injections

To modify an existing injection, select its name in the **Injections** list and click the **Set...** button. The **Set Injection Properties** dialog box will open, and you can modify the properties as needed.

If you have two or more injections for which you want to set some of the same properties, select their names in the **Injections** list and click the **Set...** button. The **Set Multiple Injection Properties** dialog box will open, which will allow you to set the common properties. For instructions about using this dialog box, see Defining Properties Common to More than One Injection (p. 2775).

24.3.17.3. Copying Injections

To copy an existing injection to a new injection, select the existing injection in the **Injections** list and click the **Copy** button. The Set Injection Properties Dialog Box (p. 5608) will open with a new injection that has the same properties as the injection you selected. This is useful if you want to set another injection with similar properties.

24.3.17.4. Deleting Injections

You can delete an injection by selecting its name in the **Injections** list and clicking the **Delete** button.

24.3.17.5. Listing Injection Initial Conditions

To list the initial conditions for the particle streams in the selected injection, click the **List Particles** button. Ansys Fluent reports the initial conditions (in SI units) in the console under various columns:

- The particle stream number is in the column headed `NO`.
- The particle type (`IN` for inert, `DR` for droplet, or `CP` for combusting particle) is in the column headed `TYP`.
- The x, y, and z positions are in the columns headed `(X)`, `(Y)`, and `(Z)`.
- The x, y, and z velocities are in the columns headed `(U)`, `(V)`, and `(W)`.
- The temperature is in the column headed `(T)`.
- The diameter is in the column headed `(DIAM)`.
- The mass flow rate in the column headed `(MFLOW)`.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.3.17.6. Listing Injection Properties

To print the properties of the selected injection(s) in the console window, click the **List Properties** button. Ansys Fluent reports the following basic properties for each selected injection:

- General information (injection name and type, particle type and material, reference frame, and diameter distribution)
- Injection point properties
- Physical models
- Parcel models
- User laws
- UDF Settings

24.3.17.7. Reading and Writing Injections

To transfer information about DPM injections from one case file to another, use the **Read...** and **Write...** buttons. You can write selected injections to a file, which can be read into a different Ansys Fluent session, simplifying the setup of new case files. To write the injection, select the injection from the list, then click **Write...**. The Select File Dialog Box (p. 931) will open where you can enter the name of your injection file. To read in an injection, click **Read...** to open The Select File Dialog Box (p. 931) where you will select the injection file to read in. If the injection that you imported has the same name as that in your current case, then Ansys Fluent will rename the imported injection.

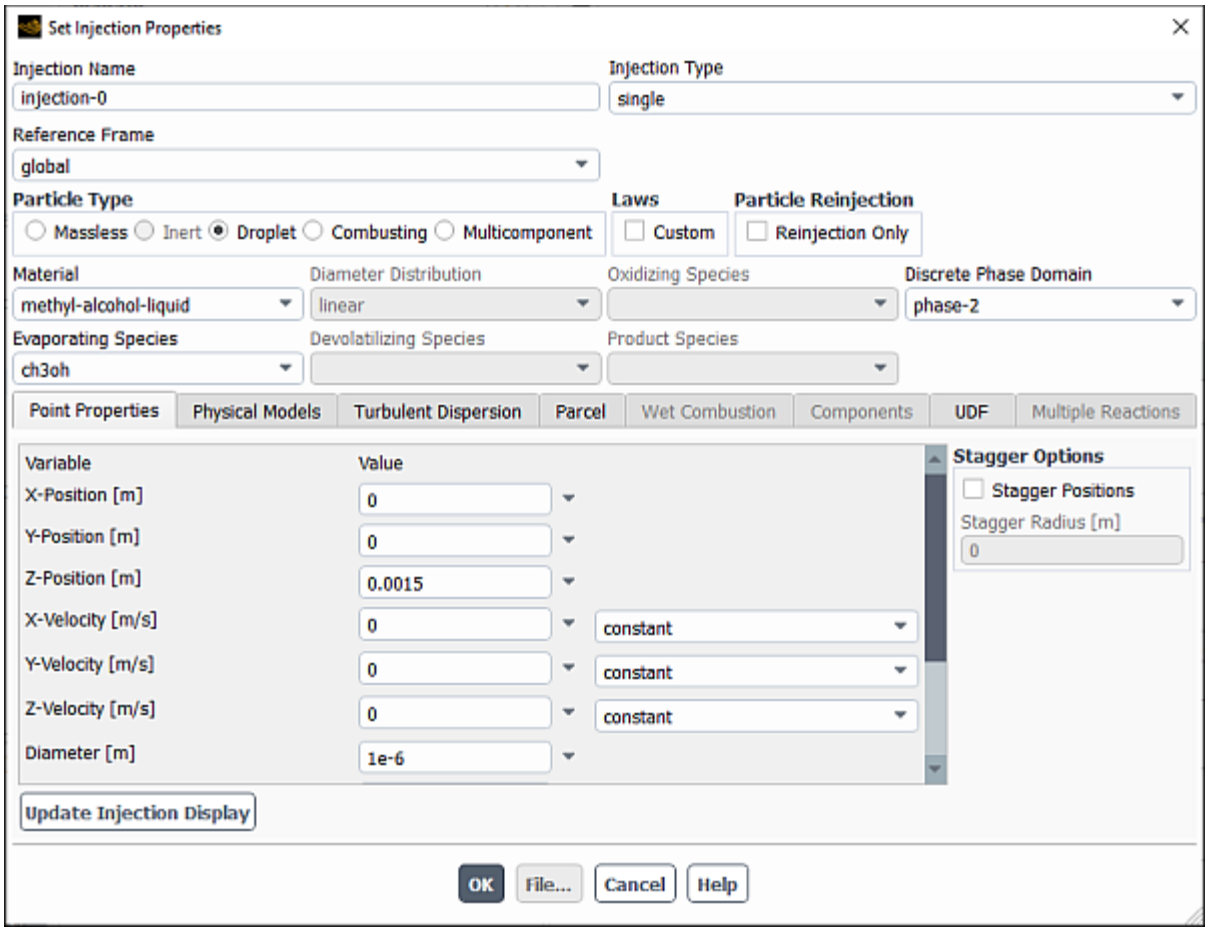
After reading injections, you may need to visit the **Injections** dialog box to modify the settings for the injection material and the DPM laws since the presumed settings may have changed in the current case file setup.

24.3.18. Defining Injection Properties

Once you have created an injection (using the Injections Dialog Box (p. 5496), as described in Creating and Modifying Injections (p. 2753)), you will use the Set Injection Properties Dialog Box (p. 5608) (Figure 24.32: The Set Injection Properties Dialog Box (p. 2757)) to define the injection properties. (Remember that this dialog box will open when you create a new injection, or when you select an existing injection and click the **Set...** button in the **Injections** dialog box.)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.32: The Set Injection Properties Dialog Box



The procedure for defining an injection is as follows:

1. If you want to change the default name of the injection, enter a new name in the **Injection Name** field. This is recommended if you are defining a large number of injections so you can easily distinguish them. When assigning names to your injections, keep in mind the selection shortcut described in Creating and Modifying Injections (p. 2753).

2. Choose the type of injection in the **Injection Type** drop-down list. The injection choices (**single**, **group**, **cone**, **surface**, **volume**, **plain-orifice-atomizer**, **pressure-swirl-atomizer**, **air-blast-atomizer**, **flat-fan-atomizer**, **effervescent-atomizer**, **condensate**, and **file**) are described in Injection Types (p. 2720). Note that if you select any of the atomizer models, you also must set the **Viscosity** and **Droplet Surface Tension** in the **Create/Edit Materials** dialog box.

   Important:

   Note that only surface injections from boundary surfaces will be moved with the mesh when a sliding mesh or a moving or deforming mesh is being used.

3. If you are using a **group**, **cone**, or any of the **atomizer** injections, set the **Number of Streams** in the group, spray cone, or atomizer.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

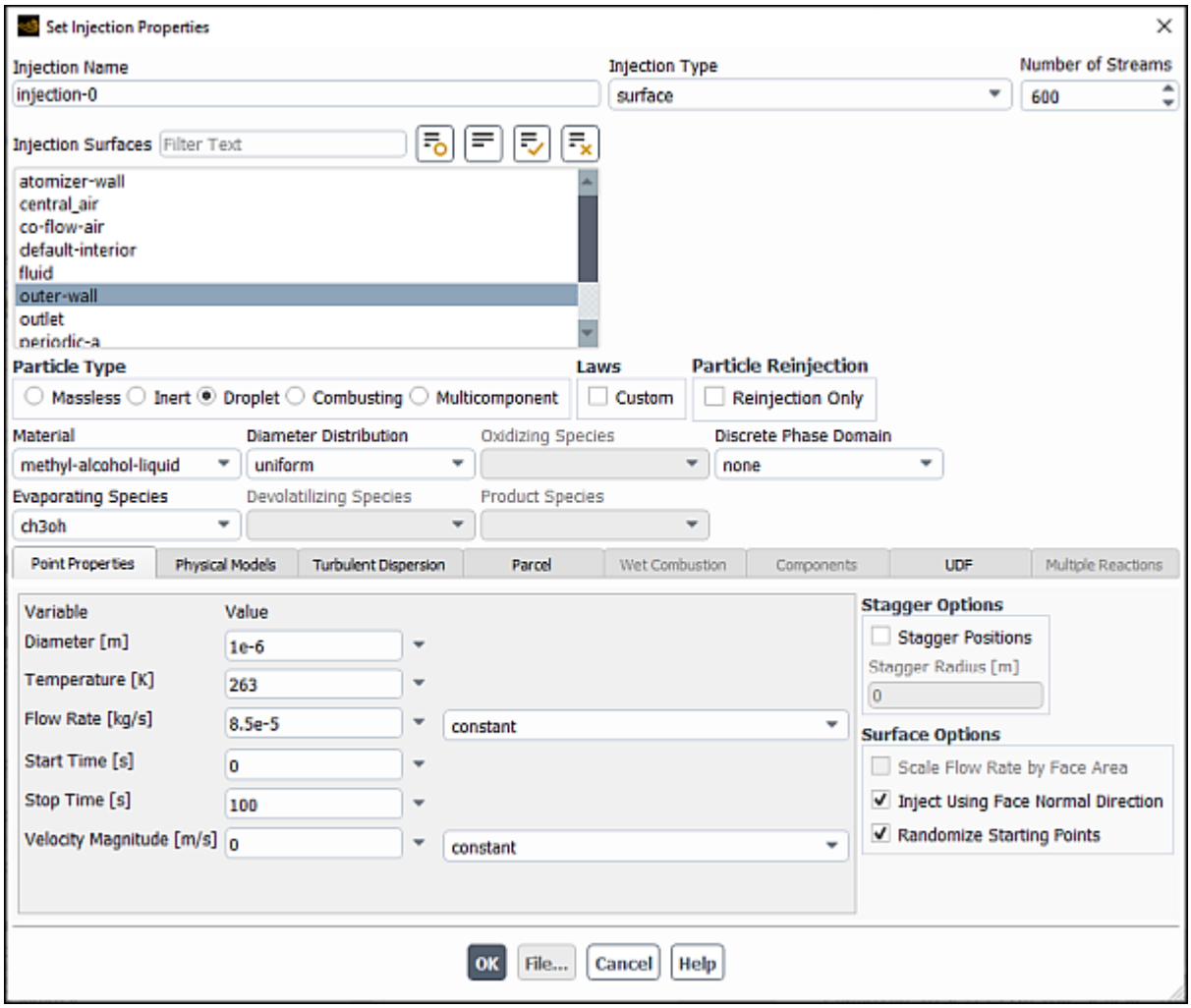
4. If you are defining a **surface** injection, configure the following settings (see Figure 24.33: Setting Surface Injection Properties (p. 2758)):

   a. Choose the surface(s) from which the particles will be released in the **Injection Surfaces** list. Note that for surface injections that use the **Randomize Starting Points** option, you can only select boundary surfaces. Surfaces created in Ansys Fluent, such as iso-surfaces, rake surfaces, and so on, cannot be used.

      By default, particles will be injected from the face center.

   b. If an arbitrary number of particles should be injected from positions distributed randomly over the selected boundary surfaces, select the **Randomize Starting Points** option (**Surface Options** group box).

Figure 24.33: Setting Surface Injection Properties



Note:

The **Randomize Starting Points** and **Scale Flow Rate by Face Area** options are not compatible because not every surface face will be assigned exactly one starting

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

stream when the random distribution is used. When either one of these options is selected, the other becomes unavailable.

c. Specify the **Number of Streams** (to the right of the **Injection Type** drop-down list) in the surface injection.

5. If you are defining a **volume** injection, configure the following settings:

   a. From the **Release From** drop-down list, select how you want to define the particle release region. The following options are available:
      - **zone**
      - **bounding-geometry**

   b. If you have chosen the **zone** specification method, select a suitable fluid zone(s) from which the particles will be released in the **Zones** selection list.

   c. If you have chosen the **bounding-geometry** specification method, select an appropriate geometry to model the injection region from the **Bounding Shape** drop-down list:
      - **sphere**
      - **cylinder**
      - **cone**
      - **hexahedron**

      and specify the **Shape Coordinates**. The inputs are as follows:

      - To define a **sphere**, enter the values for the **Radius** and the coordinates of its center: **X Center**, **Y Center**, and **Z Center**.

      - To define a **cylinder**, enter the value for the **Radius** and the minimum and maximum co-ordinates defining the cylinder axis: (**X-Axis Min**, **Y-Axis Min**, **Z-Axis Min**) and (**X-Axis Max**, **Y-Axis Max**, **Z-Axis Max**).

      - To define a **cone**, enter the value for the **Cone Angle**, the coordinates of the cone apex (**X-Axis Min**, **Y-Axis Min**, **Z-Axis Min**), and the coordinates of a point on the cone axis (**X-Axis Max**, **Y-Axis Max**, **Z-Axis Max**).

      - To define a **hexahedron**, enter the coordinates of two points defining the diagonal of the box: (**X Min**, **Y Min**, **Z Min**) and (**X Max**, **Y Max**, **Z Max**).

   d. Select the **Parcel Specification** from the following options:
      - **number-of-starting-points**
      - **parcels-per-cell** (**zone** release region only)

      and enter a value for the selected specification.

      For the **number-of-starting-points** parcel specification, parcels will be injected into cells at uniformly distributed random points. The specified value determines the number of the starting points.

      For the **parcels-per-cell** parcel specification, the specified value determines the number of starting points of parcels to be injected into each cell of the specified particle release region.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Parcel Specification** drop-down list is not available for DDPM cases with the **Discrete Phase Domain** setting other than **none**.

Note:

For cases that involve the DDPM with the **Discrete Phase Domain** set to anything other than **none**, the number of starting points is calculated automatically based on the smallest cell in the mesh as follows. The diameter of the injected parcels is calculated so that five parcels would fit into the smallest cell volume in the mesh, while satisfying the **Injection Packing Limit per Cell**. The parcel volume is computed based on the parcel diameter (this is not fully applicable to 2D cases because of the special consideration stated in the Note in Steady/Transient Treatment of Particles (p. 2679)). Then as many parcels are injected as the zone(s) or the bounding geometry can accommodate, while also respecting the **Injection Packing Limit per Cell**.

---

e. For DDPM simulations with the **Discrete Phase Domain** set to anything other than **none**, enter a value for **Injection Packing Limit per Cell**, which is the highest particle volume fraction that is possible in a computational cell.

Once the **Injection Packing Limit per Cell** is reached, the cells will stop receiving injected parcels, and the parcel mass will be lost. By default, **Injection Packing Limit per Cell** is set to the theoretical value for monodispersed solid particles of 0.63. If parcels are larger than the cells, no parcels will be injected since the Injection **Injection Packing Limit per Cell** would be exceeded. In DDPM simulations, a full loading or overloading of the fluid cells by setting this value to 1 or greater is not recommended as it will lead to convergence issues or potential application crashes.

You can obtain an estimation of the expected parameters for the injected particle parcels by clicking the **Estimate Parcel Count** button in the **Estimated Particle Parcels** group box. The following values will be reported:

- **Number of Particle Parcels**
- **Total Mass**
- **Parcel Diameter**
- **Parcel Volume**
- **Minimum Cell Volume in Mesh**
- **Averaged Volume Fraction**

6. By default, injection point properties are specified in the global reference frame/coordinate system. If you want to specify the injection point properties in one of the local reference frames that you created earlier, select that frame from the **Reference Frame** drop-down list. This item is not available for **surface**, **volume**, and **condensate** injections.

7. If you are reading the injection from a file, click the **File...** button at the bottom of the Set Injection Properties Dialog Box (p. 5608) and specify the file to be read in the resulting **Select File** dialog box. The parameters in the injection file must be in SI units.

8. As the **Particle Type**, select one of the following:
   - **Massless**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Inert**
- **Droplet**
- **Combusting**
- **Multicomponent**

The available types are described in Particle Types (p. 2722). Note that for the **condensate** injection, the only available particle types are **Droplet** and **Multicomponent**.

9. (unsteady particle tracking only) If you are using this injection only for reinjecting particles as described in The **reinject** Boundary Condition (p. 2784), select **Reinjection Only** (**Particle Reinjection** group box).

10. Choose the material for the particle(s) in the **Material** drop-down list.

    If this is the first time you have created a particle of this type, you can choose from all of the materials of this type defined in the database. If you have already created a particle of this type, the only available material will be the material you selected for that particle. You can define additional materials by copying them from the database or creating them from scratch, as discussed in Setting Discrete-Phase Physical Properties (p. 2814) and described in detail in Using the **Create/Edit Materials** Dialog Box (p. 1599).

    Important:

    Note that you will not choose a **Material** for a **Massless** particle type.

11. If you are defining a **group**, **cone**, **surface**, or **volume** injection and you want to change the default **linear** (**group** injection) or **uniform** (**cone**, **surface**, or **volume** injection) interpolation method for determining the size of the particles, you can select from the following methods in the **Diameter Distribution** drop-down list:
    - **rosin-rammler**
    - **rosin-rammler-logarithmic**.
    - **tabulated** (**cone**, **surface** or **volume** injections only)

    For more information about these methods, see Using the Rosin-Rammler Diameter Distribution Method (p. 2745) and Using the Tabulated (Discrete) Diameter Distribution (p. 2748).

12. If you have created a customized particle law using user-defined functions, enable the **Custom** option under **Laws** and specify the appropriate laws as described in Custom Particle Laws (p. 2774).

13. If you are defining **Droplet** particles, select the gas phase species created by the vaporization and boiling laws (Laws 2 and 3) in the **Evaporating Species** drop-down list.

14. If you are defining **Combusting** particles, configure the following settings:

    a. From the **Devolatilizing Species** drop-down list, select the gas phase species created by the devolatilization law (Law 4).

    b. From the **Oxidizing Species** drop-down list, select the gas phase species that participates in the surface char combustion reaction (Law 5).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. From the **Product Species** drop-down list, select the gas phase species created by the surface char combustion reaction (Law 5).

Note that if the **Combustion Model** for the selected combusting particle material (in the **Create/Edit Materials** dialog box) is the **multiple-surface-reaction** model, then the **Oxidizing Species** and **Product Species** lists will be disabled because the reaction stoichiometry has been defined in the mixture material.

15. In the **Point Properties** tab, specify the point properties (position, velocity, diameter, temperature, and—if appropriate—mass flow rate and any atomizer-related parameters) as described for each injection type in the following sections:
    - Point Properties for Single Injections (p. 2724)
    - Point Properties for Group Injections (p. 2725)
    - Point Properties for Cone Injections (p. 2726)
    - Point Properties for Surface Injections (p. 2733)
    - Point Properties for Volume Injections (p. 2735)
    - Point Properties for Plain-Orifice Atomizer Injections (p. 2736)
    - Point Properties for Pressure-Swirl Atomizer Injections (p. 2737)
    - Point Properties for Air-Blast/Air-Assist Atomizer Injections (p. 2738)
    - Point Properties for Flat-Fan Atomizer Injections (p. 2739)
    - Point Properties for Effervescent Atomizer Injections (p. 2741)
    - Point Properties for Condensate Injections (p. 2745)

16. If you want to set up injection-specific physics models such as drag laws, Brownian motion, and breakup, click the **Physical Models** tab and configure the injection physics models as described in Specifying Injection-Specific Physical Models (p. 2764).

17. If the flow is turbulent and you want to include the effects of turbulence on the particle dispersion, click the **Turbulent Dispersion** tab, enable the **Discrete Random Walk Model** under **Stochastic tracking**, and set the related parameters as described in Specifying Turbulent Dispersion of Particles (p. 2772).

18. If you have enabled **Unsteady Particle Tracking**, you can define settings for how the parcels are released. In the **Parcel** tab, select a **Parcel Release Method** from the drop-down list. The default method is **standard**, in which a single parcel is released per injection stream per time step. Alternatively can choose **constant-number**, **constant-mass**, or **constant-diameter**. Refer to Steady/Transient Treatment of Particles (p. 2679) for details of these methods.

    Note:

    For cases that involve the DDPM with the **Discrete Phase Domain** setting other than **none**, Ansys Fluent uses the **constant-diameter** parcel release method, and the **Parcel** tab is not available.

19. If your combusting particle includes an evaporating material, click the **Wet Combustion** tab, select the **Wet Combustion Model** option, and then select the material that is evaporating/boiling from the particle before devolatilization begins in the **Liquid Material** drop-down list. You should also set the volume fraction of the liquid present in the particle by entering the value of the **Liquid**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fraction. Finally, select the gas phase species created by the evaporating and boiling laws in the **Evaporating Species** drop-down list in the top part of the dialog box.

20. (**Multicomponent** particle type only) In the **Components** tab, configure the following **Multicomponent Settings**:

    a. Specify the **Mass Fraction** of each of the components.

       Note that the sum of the mass fractions should add up to unity, otherwise Ansys Fluent will scale the values such that you have a sum of 1 for the mass fraction, and will prompt you to accept the scaled values.

    b. For each component, you have the following modeling options:

       - To model a species that will be vaporized, select the name of that species from the **Evaporating Species** drop-down list.
       - To model any immiscible non-volatile residue, liquid, or solid component, select **none** from the **Evaporating Species** drop-down list.
       - To model a non-volatile liquid component that forms a mixture with the other liquid components in the particle:

         i. Select a relevant species from the **Evaporating Species** drop-down list.

         ii. For the non-volatile liquid component material, set the **Saturation Vapor Pressure** close to zero (in the **Create/Edit Materials** dialog box).

         The liquid material defined in this way will form a mixture with the liquid components, but will not evaporate.

    c. If you are using the non/partially-premixed combustion models specify the source stream for the evaporating component under **Evaporating Stream**.

    To change the species of the multicomponent droplet:

    a. In the **Create/Edit Materials** dialog box, either define a new droplet materials or copy appropriate droplet-particle materials from the Fluent materials database.

    b. For the mixture material you want to modify, click **Edit...** next to **Mixture Species** (**Property** group box).

    c. In the **Species** dialog box that opens, add the droplet materials that you have copied or defined to the **Selected Species** list and remove unwanted materials.

21. If you want to use a user-defined function to initialize the injection properties, click the **UDF** tab to access the UDF inputs. You can select an **Initialization** function under **User-Defined Functions** to modify injection properties at the time the particles are injected into the domain. This allows the position and/or properties of the injection to be set as a function of flow conditions. More information about user-defined functions can be found in the Fluent Customization Manual.

22. If you have defined more than one particle surface species, for example, carbon (**C<s>**) and sulfur (**S<s>**), you will need to specify the mass fraction of each particle surface species in the combusting particle. To do so, click the **Multiple Reactions** tab, and enter the **Species Mass Fractions**. These

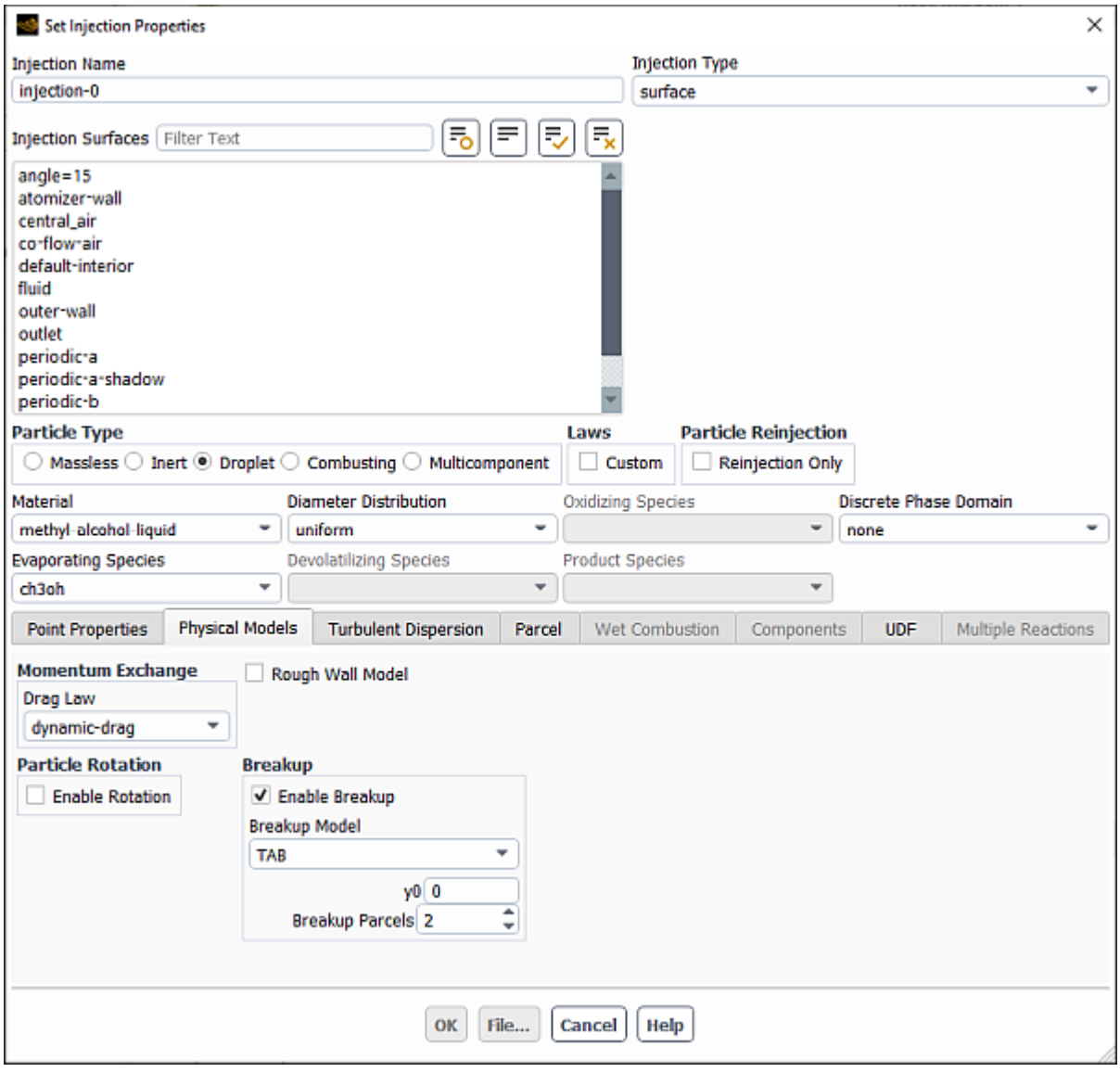
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

mass fractions refer to the combustible fraction of the combusting particle, and should sum to 1. If there is only one surface species in the mixture material, the mass fraction of that species will be set to 1, and you will not specify anything under **Multiple Surface Reactions**.

24.3.19. Specifying Injection-Specific Physical Models

Drag and breakup models can be specified on a per-injection basis, allowing you to specify the most appropriate models for each injection in your model. These per-injection models are specified on the **Physical Models** tab of the **Set Injection Properties** dialog box.

Figure 24.34: Setting Surface Injection Properties



Further details are discussed in the following sections:

24.3.19.1. Drag Laws

24.3.19.2. Heat Transfer Coefficient

24.3.19.3. Particle Rotation

24.3.19.4. Rough Wall Model

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.3.19.5. Brownian Motion Effects

24.3.19.6. Breakup

24.3.19.1. Drag Laws

The following drag laws are always available:

- **spherical**
- **nonspherical**
- **Stokes-Cunningham**
- **high-Mach-number**

For more information about these models, see Particle Force Balance in the Theory Guide.

The **dynamic-drag** law described in Dynamic Drag Model Theory in the Theory Guide is available only when one of the droplet breakup models is used in conjunction with unsteady tracking. See Breakup (p. 2769) for information about enabling the droplet breakup models.

The **Grace** and **Ishii-Zuber** drag laws described in Bubbly Flow Drag Laws in the *Fluent Theory Guide* allow to model the drag coefficient in gas-liquid flows in which the bubbles can have a range of different shapes. Both drag laws are only available if the magnitude and direction of the gravity vector are specified in the **Operating Conditions** dialog box.

The following drag laws:

- **Wen-Yu**
- **Gidaspow**
- **Syamlal-OBrien**
- **Huilin-Gidaspow**
- **Gibilaro**
- **EMMS**
- **Filtered**

are available only when the dense discrete phase model (DDPM) is enabled (Including the Dense Discrete Phase Model (p. 3107)) and the flow regime consists of a dense gas-solid. However, with these models, you cannot verify whether it really is a dense flow or a gas-solid flow. It is up to you to decide. In any case, these drag formulations are suitable for dense gas-solid flows.

No further inputs are required for the following drag laws:

- **spherical**
- **high-Mach-number**
- **dynamic-drag**
- **Wen-Yu**
- **Gidaspow**
- **Syamlal-OBrien**
- **Huilin-Gidaspow**
- **Gibilaro**
- **EMMS**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Filtered**

If the **nonspherical** law is selected, the particle **Shape Factor** (ϕ in Equation 12.52 in the Theory Guide) must be specified. The shape factor value cannot exceed 1.

For the **Stokes-Cunningham** law, the **Cunningham Correction** factor (C_c in Equation 12.55 in the Theory Guide) must be specified.

24.3.19.2. Heat Transfer Coefficient

For DPM cases with heat transfer, Ansys Fluent uses the Ranz-Marshall correlation to compute the heat transfer coefficient (see Inert Heating or Cooling (Law 1/Law 6) in the *Fluent Theory Guide*).

For DPM cases that include inert or combusting particles, or multicomponent particles that do not contain evaporating species, you can choose one of the following models from the **Heat Transfer Coefficient** drop-down list (**Heat Exchange** group box):

constant-HTC

allows you to specify a constant value for the volumetric heat transfer coefficient.

Nusselt-number

allows you to specify a value for the Nusselt number from which the heat transfer coefficient will be computed.

Tomiyama

uses the Tomiyama model, which is suitable for turbulent bubbly flows.

Ranz-Marshall

uses the Ranz-Marshall model described. This is the default heat transfer coefficient model for all particle types.

Hughmark

uses the Hughmark model.

Gunn

uses the Gunn model to compute the heat transfer coefficient. The model is applicable for flows with high volume fraction of the granular phase. This option is available only when the dense discrete phase model (DDPM) is enabled.

See Inert Heating or Cooling (Law 1/Law 6) in the *Fluent Theory Guide* for details about these models

Note the following:

- A value of zero for **constant-HTC** or **Nusselt-number** implies that convective heat transfer is not considered.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The **Heat Transfer Coefficient** drop-down is available when the **Energy Equation** is enabled in the **Energy** dialog box.

24.3.19.3. Particle Rotation

To set-up particle rotation, follow the steps below:

1. In the **Set Injection Properties** dialog box, under the **Physical Models** tab, enable **Particle Rotation**.

   The options related to the particle rotation feature appear in the **Set Injection Properties** dialog box.

2. From the **Rotational Drag Law** drop-down list, select the model for calculating the rotational drag coefficient C_ω. The following options are available:

   Dennis-et-al

   Uses Equation 12.67 in the *Fluent Theory Guide* to calculate the rotational drag coefficient C_ω.

   none (default)

   Implies zero torque, that is, it excludes the influence of the fluid on the rotational particle motion.

3. From the **Magnus Lift Law** drop-down list, select the model for calculating the rotational lift coefficient C_{RL}. The following models are available:

   Oesterle-Bui-Dinh

   Uses Oesterle and Bui Dinh's formulation (Equation 12.18 in the *Fluent Theory Guide*).

   Tsuji-et-al

   Is based on the particle spin rate (Equation 12.19 in the *Fluent Theory Guide*).

   Rubinow-Keller

   Uses a linear dependence of the rotational lift coefficient C_{RL} on the spin rate (Equation 12.21 in the *Fluent Theory Guide*).

   none (default)

   Implies that the Magnus lift will not be included in your simulation.

4. Under the **Point Properties** tab, specify angular velocity components for **single**, **group**, **surface**, and **volume** injections; and angular velocity magnitudes for **cone** injections.

5. For each wall with a **reflect** boundary condition for the discrete phase, specify the friction coefficient (μ in Equation 12.194 and Equation 12.199 in the *Fluent Theory Guide*) under the **DPM** tab of the **Wall** dialog box. For details, see Friction Coefficient (p. 2786).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The particle rotation model options are also accessible through the text user interface, under `define/models/dpm/injections/injection-properties/set/physical-models/particle-rotation`. Note that the `physical-models` command becomes available only after you have selected the injection(s) using the prompts of the `define/models/dpm/injections/properties/set/pick-injections-to-set` text command.

24.3.19.4. Rough Wall Model

To use the rough wall model for walls with the **reflect** boundary condition type:

1. In the **Injection Settings** dialog box, under the **Physical Models** tab, select **Rough Wall Model**.
2. In the **Wall** boundary conditions dialog box for the appropriate wall, under the **DPM** tab, select **reflect** from the **Discrete Phase BC Type** drop-down list.
3. Specify the following wall roughness parameters:

   Ra

   is the wall mean roughness (R_a in Equation 12.201 in the *Fluent Theory Guide*)

   Rq

   is the standard deviation of the roughness structure (R_q in Equation 12.201 in the *Fluent Theory Guide*)

   RSm

   is the mean roughness slope (the average distance between peaks of wall material) (RS_m in Equation 12.201 in the *Fluent Theory Guide*)

   Note:

   Note that the **DPM Wall Roughness Parameters** will only appear in the **Wall** dialog box with the `reflect` DPM boundary condition type if the **Rough Wall Model** is selected in the **Set Injection Properties** dialog box.

For information about the rough wall model, see Rough Wall Model in the *Fluent Theory Guide*.

24.3.19.5. Brownian Motion Effects

For sub-micron particles in laminar flow, you may want to include the effects of Brownian motion (described in Brownian Force in the Theory Guide) on the particle trajectories. To do so, enable the **Brownian Motion** option under the **Physical Models** tab. In order to include Brownian motion effects, you must also select the **Stokes-Cunningham** drag law in the **Drag Law** drop-down list under **Drag Parameters**, and specify the **Cunningham Correction** (C_c in Equation 12.55 in the Theory Guide).

This option is available only when the **Energy** equation is enabled (in the **Models** group).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.3.19.6. Breakup

To enable the modeling of particle breakup for the injection, select **Enable Breakup** and choose the **Breakup Model** (**TAB**, **Wave**, **KHRT**, **SSD**, **Madabhushi** or **Schmehl**) from the drop-down list. A detailed description of these models can be found in Secondary Breakup Model Theory in the Theory Guide.

- For the **TAB** model, you must specify the following values:
  - **y0**: Is the initial distortion at time equal to zero (y_0 in Equation 12.405 in the Theory Guide). The default value ($y_0=0$) is recommended.
  - The number of **Breakup Parcels**: To split the droplet into several child parcels, as described in Velocity of Child Droplets in the Theory Guide. The diameter of the child parcels is sampled from a Rosin-Rammler distribution. This can be switched off in the TUI with the command:

    ```
    /define/models/dpm/spray-model/tab-randomize-breakup-parcel-
    diameter?
    ```

- For the **Wave** model, you must specify the following values:
  - **B0**: Is the constant B_0 in Equation 12.435 in the Theory Guide.
  - **B1**: Is the constant B_1 in Equation 12.437 in the Theory Guide.

  Note:

  You will generally not need to modify the value of **B0**, as the default value 0.61 is acceptable for nearly all cases. A value of 1.73 is recommended for **B1**.

  The **Wave** model implementation has been formulated to deal with the initial breakup of a cylindrical liquid jet. In the Rayleigh regime (that is, at low gas Weber numbers), a cylindrical jet breaks up into droplets whose diameters are larger than that of the jet itself. In this regime, the model assumes that the diameter of the continuous, cylindrical liquid jet has been entered as initial diameter in the injection. Therefore, the droplet diameter can grow beyond the initial diameter. For details, see Droplet Breakup in the *Fluent Theory Guide*.

  If you want to suppress this feature of the Wave model, use the TUI command:

  ```
  /define/models/dpm/spray-model/wave-allow-rayleigh-growth? no
  ```

  Note that this takes effect unconditionally for all injections using the **Wave** or **KHRT** breakup models.

- For the **KHRT** model, you must specify the following values:
  - **B0**: Is the constant B_0 in Equation 12.435 in the Theory Guide.
  - **B1**: Is the constant B_1 in Equation 12.437 in the Theory Guide.
  - **Ctau**: Is the constant C_τ in Equation 12.442 in the Theory Guide.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– **CRT**: Is the constant C_{RT} in Equation 12.443 in the Theory Guide.

– **CL**: :s the constant C_L in Equation 12.438 in the Theory Guide.
The constants **B0** and **B1** are the same as for the Wave model.

For the **Wave** model described above, the **KHRT** model is formulated to deal with the initial breakup of a cylindrical liquid jet. The behavior in the Rayleigh regime is as described above for the **Wave** model and the same TUI command can be used to modify the behavior. In addition, the **KHRT** model uses the Liquid Core Approximation as described in Liquid Core Length in the *Fluent Theory Guide*. To suppress the use of the Liquid Core Approximation you can set the Levich constant, C_L, to zero.

- For the **SSD** model, you must specify the following values:

  – **Critical We**: Is the critical Weber number in Equation 12.444 in the Theory Guide.

  – **Core B1**: Is B in Equation 12.445 in the Theory Guide.

  – **Target Np**: Is the number of droplets given to each child parcel, before scaling is used to give the correct overall mass.

  – **Xi**: Is $\langle \xi \rangle$ in Equation 12.446 in the Theory Guide.

  Note:

  Xi is a negative value: exp(Xi) is a typical factor by which daughter particles are smaller than the original parcel.

- For the **Schmehl** model, you can specify the following settings:

  – **Breakup Parcels**: Is the number of child parcels generated after splitting the droplet as described in Schmehl Breakup Model in the *Fluent Theory Guide*. The default is set to 1. The diameter of the child parcels is sampled from a root normal distribution described in Schmehl Breakup Model in the *Fluent Theory Guide*.

  – **Breakup Max Generation**: Is the maximum number of child parcel generations. The initial injected particle is considered to be generation 0, the child particles created after the first particle breakup are considered to be generation 1, and so on. When this limit is reached, the breakup process stops.

  – **Subsequent Statistical Breakup**: When enabled, the droplet breakup mechanism switches to the subsequent statistical breakup model after the **Breakup Max Generation** is reached. In this model no child parcels are created during the breakup mechanism. Instead, the particle number in the parcel is increased, and the particle diameter is adjusted in such a way as to conserve the full filling mass of the parent parcel.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To set up the **Madabhushi** breakup model, follow the steps below:

  Note:

  – For the Madabhushi breakup model, the most appropriate injection types are single, group, and cone (solid-cone). The recommended type is a solid cone injection.

  – The Madabhushi breakup model is available only with the **dynamic-drag** law (which is automatically selected from the **Drag Law** drop-down list when the Madabhushi breakup model is chosen) or with a drag law specified via a user-defined function (UDF). (See `DEFINE_DPM_DRAG` in the *Fluent Customization Manual* for details.)

  1. In the **Set Injection Properties** dialog box, define the **Point Properties** for the selected injection type (single, group, or solid cone) as follows:

     a. Specify **X-**, **Y-**, and **Z- Position** in such a way that the injected droplets build a column over the midpoint of the nozzle orifice exit.

     b. Set **Diameter** to the jet diameter (nozzle orifice exit diameter).

     c. (single and group injections only) Set **X-**, **Y-**, and **Z- Velocity** to the inlet velocity components in x, y, and z directions, respectively.

     d. (solid cone injections only) Set **Velocity Magnitude** and **X-**, **Y-**, and **Z- Axis** to the inlet velocity magnitude and the direction of the jet, respectively.

     e. Set **Flow Rate** (single and group injections) or **Total Flow Rate** (solid cone injections) to the nozzle orifice flow rate.

     f. (solid cone injections only) Set **Cone angle** to `0` and **Outer Radius** to the jet radius in order to specify a perpendicular jet inflow.

     g. (solid cone injections only) In the **Cone Injector Parameters** group box, select **Uniform Mass Distribution**.

  2. In the **Physical Models** tab, specify the following Madabhushi model settings.

     – For primary breakup based on the Wave breakup model, specify:

       → **B0** and **B1**: Are the same constants as for the Wave model. The recommended values for the Madabhushi model are **B0**= 2.44 and **B1**=10. See Wave Breakup Model for details about these parameters.

       → **Jet Diameter**: Is the nozzle orifice exit diameter.

     – For primary breakup related to the column breakup mechanism, specify:

       → **Ligament Factor**: Adjusts the diameter of the child droplets arising from the column breakup mechanism.

       → **C0**: Is the column breakup time constant that controls how fast the column breakup occurs. The value recommended for the Madabhushi breakup model is **C0**=3.44.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

→ **Column Drag Cd**: Is the constant drag coefficient for calculating the drag force acting on droplets within the liquid column.

– To control the breakup child generation, specify **Breakup Max Generation** and **Subsequent Statistical Breakup** as described above for the **Schmehl** breakup model.

3. From the **Drag Law** drop-down list, select **dynamic-drag**.

The drag is dynamically adapted in the Madabhushi breakup model.

For more information about the Madabhushi breakup model, see Madabhushi Breakup Model in the *Fluent Theory Guide*.

For steady-state simulations, you also must specify an appropriate **Particle Time Step Size** and the **Number of Time Steps**, which will control the spray density. See Options for Interaction with the Continuous Phase (p. 2678) for more information.

Note that you may want to use the dynamic drag law when you use one of the breakup models. See Drag Laws (p. 2695) for information about choosing the drag law.

24.3.20. Specifying Turbulent Dispersion of Particles

As mentioned in Defining Injection Properties (p. 2756), you can enable stochastic tracking for each injection to model turbulent dispersion of particles.

24.3.20.1. Stochastic Tracking

For turbulent flows, if you choose to use the stochastic tracking technique, you must enable the **Discrete Random Walk Model** and specify the **Number of Tries**. Stochastic tracking includes the effect of turbulent velocity fluctuations on the particle trajectories using the DRW model described in Stochastic Tracking in the Theory Guide.

1. Click the **Turbulent Dispersion** tab in the **Set Injection Properties** dialog box.

2. Enable stochastic tracking by turning on the **Discrete Random Walk Model** under **Stochastic Tracking**.

3. Specify the **Number of Tries**:

   Selecting the **Turbulent Dispersion** model tells Ansys Fluent to include turbulent velocity fluctuations in the particle force balance as in Equation 12.22 in the Theory Guide. The trajectory is computed more than once if your input exceeds 1: two trajectory calculations are performed if you enter 2, three trajectory calculations are performed if you enter 3, and so on. Each trajectory calculation includes a new stochastic representation of the turbulent contributions to the trajectory equation.

   When a sufficient number of tries is requested, the trajectories computed will include a statistical representation of the spread of the particle stream due to turbulence. Note that for unsteady particle tracking, the **Number of Tries** is set to 1 if using stochastic tracking.

If you want the characteristic lifetime of the eddy to be random (Equation 12.33 in the Theory Guide), enable the **Random Eddy Lifetime** option. You will generally not need to change the **Time Scale Constant** (C_L in Equation 12.24 in the Theory Guide) from its default value of 0.15, unless

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

you are using the Reynolds Stress turbulence model (RSM), in which case a value of 0.3 is recommended.

Figure 24.35: Mean Trajectory in a Turbulent Flow (p. 2773) illustrates a discrete phase trajectory calculation computed without turbulent dispersion and Figure 24.36: Stochastic Trajectories in a Turbulent Flow (p. 2774) illustrates the "stochastic" tracking (number of tries ≥ 1) option.

When multiple stochastic trajectory calculations are performed, the momentum and mass defined for the injection are divided evenly among the multiple particle/droplet tracks, and are therefore spread out in terms of the interphase momentum, heat, and mass transfer calculations. Including turbulent dispersion in your model can thus have a significant impact on the effect of the particles on the continuous phase when coupled calculations are performed.

Figure 24.35: Mean Trajectory in a Turbulent Flow

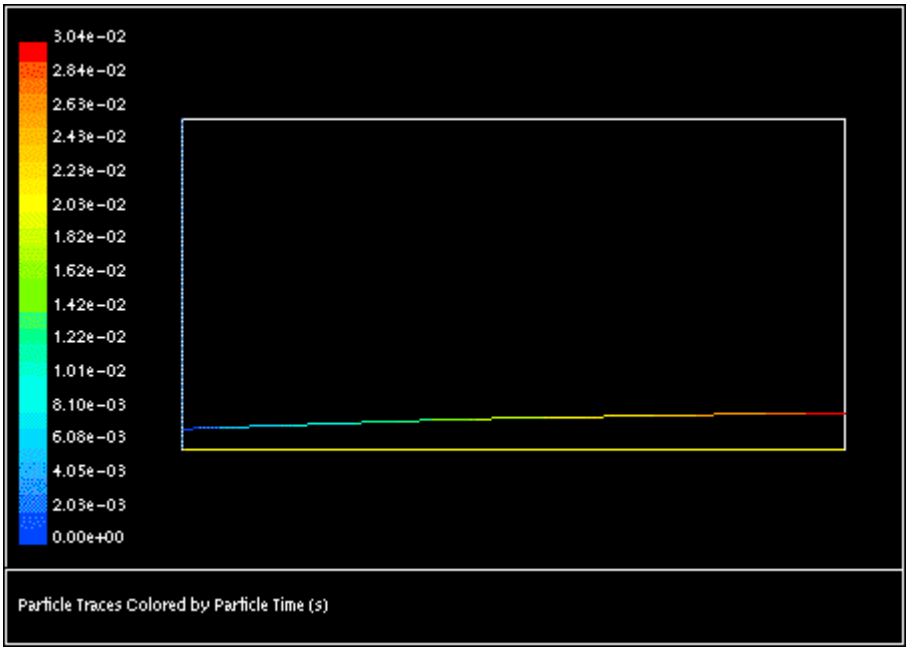
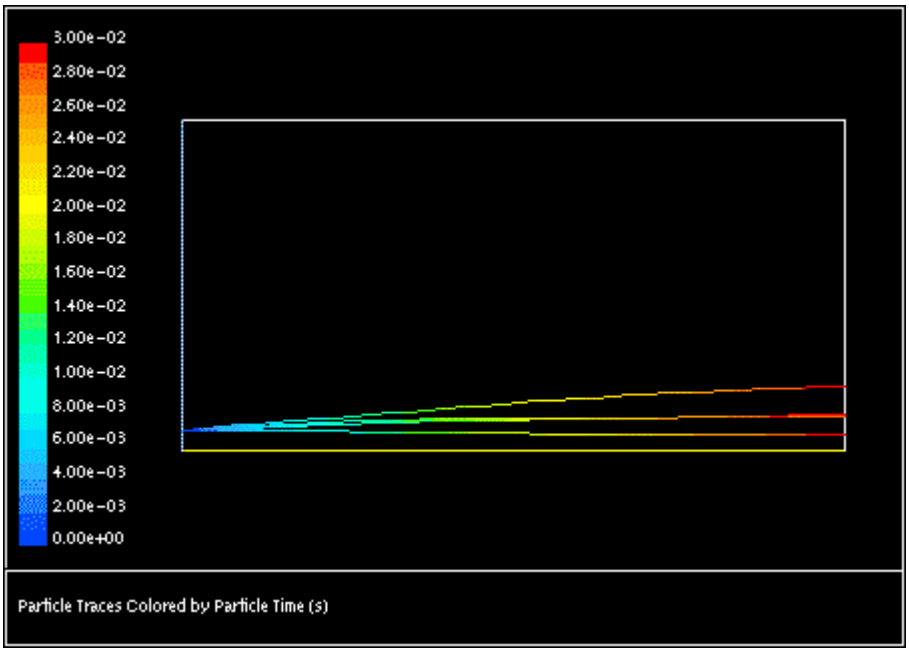


Figure 24.36: Stochastic Trajectories in a Turbulent Flow



24.3.21. Custom Particle Laws

If the standard Ansys Fluent laws do not adequately describe the physics of your discrete phase model, you can modify them by creating custom laws with user-defined functions. More information about user-defined functions can be found in the Fluent Customization Manual. You can also create custom laws by using a subset of the existing Ansys Fluent laws (for example, Laws 1, 2, and 4), or a combination of existing laws and user-defined functions.

Once you have defined and loaded your user-defined function(s), you can create a custom law by enabling the **Custom** option under **Laws** in the Set Injection Properties Dialog Box (p. 5608). This will open the Custom Laws Dialog Box (p. 5429). In the drop-down list to the left of each of the particle laws, you can select the appropriate particle law for your custom law. Each list contains the available options that can be chosen (the standard laws plus any user-defined functions you have loaded).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.37: The Custom Laws Dialog Box

There is a final drop-down list in the Custom Laws Dialog Box (p. 5429) labeled **Switching**. You may want to have Ansys Fluent vary the laws used depending on conditions in the model. You can customize the way Ansys Fluent switches between laws by selecting a user-defined function from this drop-down list (see `DEFINE_DPM_SWITCH` in the Fluent Customization Manual).

An example of when you might want to use a custom law might be to replace the standard devolatilization law with a specialized devolatilization law that more accurately describes some unique aspects of your model. After creating and loading a user-defined function that details the physics of your devolatilization law, you would visit the Custom Laws Dialog Box (p. 5429) and replace the standard devolatilization law (Law 2) with your user-defined function.

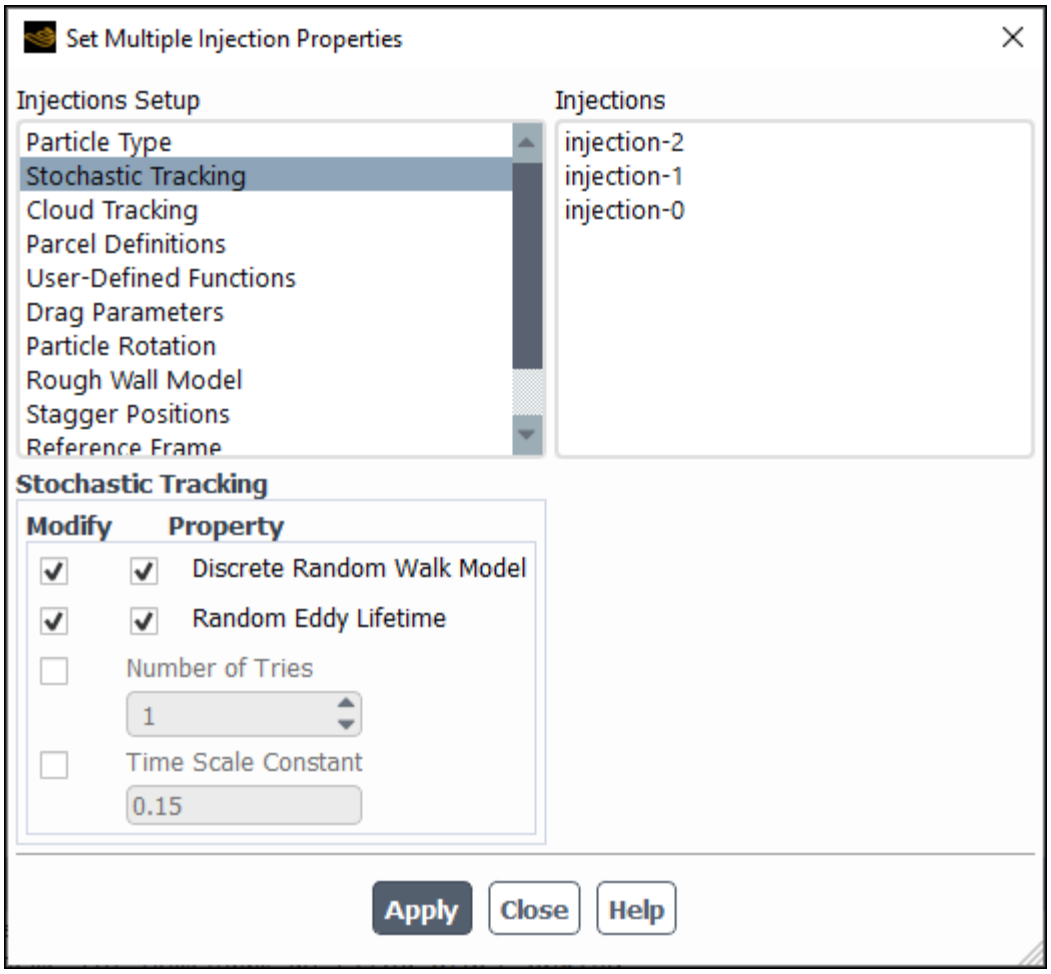
24.3.22. Defining Properties Common to More than One Injection

If you have a number of injections for which you want to set the same properties, Ansys Fluent provides a shortcut so that you do not need to visit the **Set Injection Properties** dialog box for each injection to make the same changes.

As described in Modifying Injections (p. 2755), if you select more than one injection in the **Injections** dialog box, clicking the **Set...** button will open the **Set Multiple Injection Properties** dialog box (Figure 24.38: The Set Multiple Injection Properties Dialog Box (p. 2776)) instead of the **Set Injection Properties** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.38: The Set Multiple Injection Properties Dialog Box



Depending on the type of injections you have selected (single, group, atomizers, and so on), there will be different categories of properties listed under **Injections Setup**. The names of these categories correspond to the headings within the **Set Injection Properties** dialog box (for example, **Particle Type** and **Stochastic Tracking**). Only those categories that are appropriate for all of your selected injections (which are shown in the **Injections** list) will be listed. If all of these injections are of the same type, more categories of properties will be available for you to modify. If the injections are of different types, you will have fewer categories to select from.

Note:

You cannot define any properties as input parameters when you are modifying multiple injections. If you want to specify a property as an input parameter you must use Figure 24.32: The Set Injection Properties Dialog Box (p. 2757).

24.3.22.1. Modifying Properties

To modify a property, perform the following steps:

1. Select the appropriate category in the **Injections Setup** list. For example, if you want to set the same flow rate for all of the selected injections, select **Point Properties**. The dialog box will

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

expand to show the properties that appear under that heading in the **Set Injection Properties** dialog box.

2. Set the property (or properties) to be modified, as described below.

3. Click **Apply**. Ansys Fluent will report the change in the console window.

Important:

You must click **Apply** to save the property settings within each category. If, for example, you want to modify the flow rate and the stochastic tracking parameters, you will need to select **Point Properties** in the **Injections Setup** list, specify the flow rate, and click **Apply**. You would then repeat the process for the stochastic tracking parameters, clicking **Apply** again when you are done.

There are two types of properties that can be modified using the **Set Multiple Injection Properties** dialog box.

The first type involves one of the following actions:

- selecting a value from a drop-down list
- making a selection from a list of radio buttons

The second type involves one of the following actions:

- entering a value in a field
- enabling / disabling an option

Setting the first type of property works the same way as in the **Set Injection Properties** dialog box. For example, if you select **Particle Type** in the **Injections Setup** list, the dialog box will expand to show the portion of the **Set Injection Properties** dialog box where you choose the particle type. You can simply choose the desired type and click **Apply**.

Setting the second type of property requires an additional step. If you select a category in the **Injections Setup** list that contains this type of property, the expanded portion of the dialog box will look like the corresponding part of the **Set Injection Properties** dialog box, with the addition of **Modify** check buttons (see Figure 24.38: The Set Multiple Injection Properties Dialog Box (p. 2776)). To change one of the properties, first turn on the **Modify** check button to its left, and then specify the desired status or value.

For example, if you would like to enable stochastic tracking, first turn on the **Modify** check button to the left of **Stochastic Model**. This will make the property active so you can modify its status. Then, under **Property**, turn on the **Stochastic Model** check button. (Be sure to click **Apply** when you are done setting stochastic tracking parameters.)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you would like to change the value of **Number of Tries**, select the **Modify** check button to its left to make it active, and then enter the new value in the field. Make sure you click **Apply** when you have finished modifying the stochastic tracking properties.

Important:

The setting for a property that has not been enabled with the **Modify** check button is not relevant, because it will not be applied to the selected injections when you click **Apply**. After you enable **Modify** for a particular property, clicking **Apply** will modify that property for all of the selected injections, so make sure that you have the settings the way that you want them before you do this. If you make a mistake, you will have to return to the **Set Injection Properties** dialog box for each injection to fix the incorrect setting, if it is not possible to do so in the **Set Multiple Injection Properties** dialog box.

24.3.22.2. Modifying Properties Common to a Subset of Selected Injections

Note that it is possible to change a property that is relevant for only a subset of the selected injections. For example, if some of the selected injections are using stochastic tracking and some are not, enabling the **Random Eddy Lifetime** option and clicking **Apply** will turn this option on only for those injections that are using stochastic tracking. The other injections will be unaffected.

24.3.23. Point Properties for Transient Injections

Simulations of transient particles often require time dependent injection conditions. Potentially most of the point properties may change over time. One method to accomplish this is the use of an **Initialization** function as described in `DEFINE_DPM_INJECTION_INIT` in the Fluent Customization Manual.

An easier way is to use transient profiles containing one or more variables based on the time or crank angle that can be assigned to various point properties: (Total) **Mass Flow Rate**, **X-, Y-, Z- Velocity**, **Velocity Magnitude,** and **Cone Angle** depending on the injection type selected. See Point Properties for Single Injections (p. 2724) to Point Properties for Effervescent Atomizer Injections (p. 2741) for the description of the individual properties.

Before transient profile variables can be assigned to point properties of injections, a profile file has to be read into Ansys Fluent using the **File**/**Read**/**Profile...** ribbon tab item. In the **Select File** dialog box a **transient profile** in tabular format with extension `.ttab` has to be chosen. Alternatively, you can read this file into Ansys Fluent using the `read-transient-table` text command.

`file` → `read-transient-table`

The profile name should not exceed 63 characters. See Transient Cell Zone and Boundary Conditions (p. 1523) for the format description. The following example illustrates an injection within 3 intervals of 2.5 milliseconds injection time, where the second injection has an elevated velocity.

```
mv-profile 3 13 0
time         mass-flow    velocity
0            0            0.1
0.00999      0            0.1
0.01         0.001        0.1
0.0125       0.001        0.1
0.01251      0            0.1
0.01999      0            0.1
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
0.02        0.001       5
0.0225      0.001       5
0.02251     0           0.1
0.02999     0           0.1
0.03        0.001       0.1
0.0325      0.001       0.1
0.03251     0           0.1
```

For the setting of point properties go to the **Point Properties** tab in the **Set Injection Properties** dialog box.

When transient profiles are loaded, you can choose either a **constant** method (the default) or **profile** from the **Method** drop-down list. If you select **profile**, you also must choose a profile from the corresponding drop-down list that appears in the **Value** column. The variable angle or time from the file is used as an independent interpolation variable and cannot be chosen as a profile.

Important:

All quantities in the profile file, including coordinate values, must be specified in SI units. Specifically, Mass Flow Rate must be specified in kg/s, Velocity in m/s, and Cone Angle in radians. Ansys Fluent does not perform unit conversion when reading profile files.

24.4. Setting Boundary Conditions for the Discrete Phase

When a particle reaches a physical boundary (for example, a wall or inlet boundary) in your model, Ansys Fluent applies a discrete phase boundary condition to determine the fate of the trajectory at that boundary. One of several contingencies may arise:

- The particle may be reflected via an elastic or inelastic collision.
- The particle may escape through the boundary. The particle is lost from the calculation at the point where it impacts the boundary.
- The particle may be trapped at the wall. Nonvolatile material is lost from the calculation at the point of impact with the boundary; volatile material present in the particle or droplet is released to the vapor phase at this point.
- The particle may pass through an internal boundary zone, such as radiator or porous jump.
- The particle may slide along the wall, depending on particle properties and impact angle.
- The particle may form a film (wall film model).
- The particle may be reinjected using location and particle velocity information from a specified injection.

You also have the option of implementing a user-defined function to model the particle behavior when hitting the boundary. More information about user-defined functions can be found in the Fluent Customization Manual.

The boundary condition, or trajectory fate, can be defined separately for each zone in your Ansys Fluent model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For additional information, see the following sections:

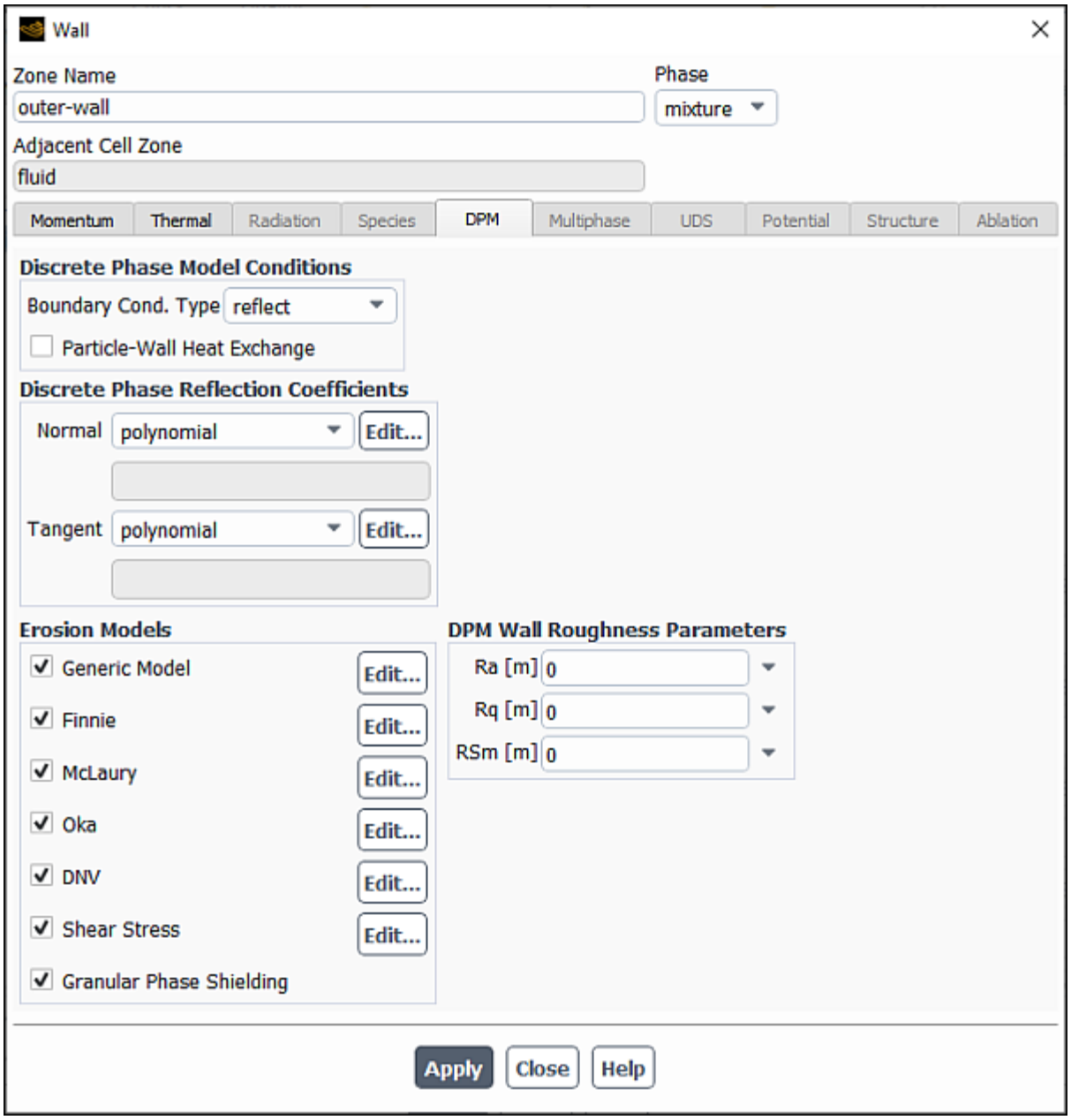
24.4.1. Discrete Phase Boundary Condition Types
24.4.2. Default Discrete Phase Boundary Conditions
24.4.3. Coefficients of Restitution
24.4.4. Friction Coefficient
24.4.5. Particle-Wall Impingement Heat Transfer
24.4.6. Setting Particle Erosion and Accretion Parameters

24.4.1. Discrete Phase Boundary Condition Types

Discrete phase boundary conditions can be set for boundaries in the dialog boxes opened from the **Boundary Conditions** task page. When one or more injections have been defined, inputs for the discrete phase will appear in the dialog boxes. In the **Walls** dialog boxes, you will need to click the **DPM** tab to access the **Discrete Phase Model Boundary Conditions** as shown in Figure 24.39: Discrete Phase Boundary Conditions in the Wall Dialog Box (p. 2781).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.39: Discrete Phase Boundary Conditions in the Wall Dialog Box



The available boundary conditions are:

- **reflect**
- **trap**
- **escape**
- **wall-jet**
- **wall-film**
- **interior**
- **user-defined**

Because you can stipulate any of these conditions at flow boundaries, it is possible to incorporate mixed discrete phase boundary conditions in your Ansys Fluent model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Further details are discussed in the following sections:

24.4.1.1. The reflect Boundary Condition
24.4.1.2. The trap Boundary Condition
24.4.1.3. The escape Boundary Condition
24.4.1.4. The wall-jet Boundary Condition
24.4.1.5. The wall-film Boundary Condition
24.4.1.6. The interior Boundary Condition
24.4.1.7. The reinject Boundary Condition
24.4.1.8. The user-defined Boundary Condition

24.4.1.1. The reflect Boundary Condition

The particle rebounds off the boundary with a change in its momentum. Ansys Fluent provides two models for computing this change:

- For non-rotating particles, the momentum depends on the coefficient of restitution (Equation 12.193 in the *Fluent Theory Guide*).
- For rotating particles, the model proposed by Tsuji et al. 660 is used. The particle rotation model includes frictional effects.

The details on these two models can be found in Wall-Particle Reflection Model Theory in the *Fluent Theory Guide*.

For information on how to set up the parameters for non-rotating and rotating particle models, see Coefficients of Restitution (p. 2785) and Friction Coefficient (p. 2786), respectively.

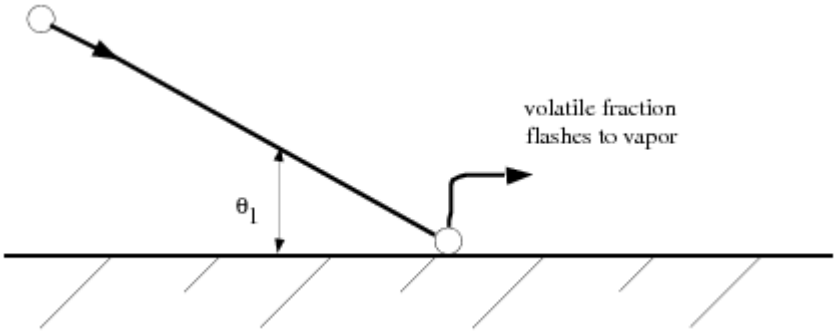
When at least one injection accounts for the rough wall model (the **Rough Wall Model** option is selected under the **Physical Models** tab in the **Set Injections Properties** dialog box), you can specify the wall surface roughness parameters as described in Rough Wall Model (p. 2768).

24.4.1.2. The trap Boundary Condition

The trajectory calculations are terminated and the fate of the particle is recorded as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See Figure 24.40: "Trap" Boundary Condition for the Discrete Phase (p. 2783). In the case of combusting particles, the remaining volatile mass is passed into the vapor phase. No further char combustion occurs. The particle no longer participates in heat transfer.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

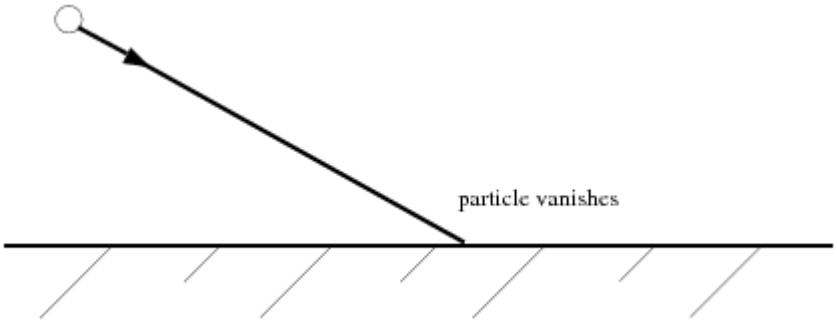
Figure 24.40: "Trap" Boundary Condition for the Discrete Phase



24.4.1.3. The escape Boundary Condition

The particle is reported as having "escaped" when it encounters the boundary in question. Trajectory calculations are terminated. See Figure 24.41: "Escape" Boundary Condition for the Discrete Phase (p. 2783).

Figure 24.41: "Escape" Boundary Condition for the Discrete Phase



24.4.1.4. The wall-jet Boundary Condition

The **wall-jet** type boundary condition is appropriate for high-temperature walls where no significant liquid film is formed, and in high-Weber-number impacts where the spray acts as a jet. The model is not appropriate for regimes where film is important (for example, port fuel injection in SI engines, rainwater runoff, and so on).

A more detailed description of underlying theory is available in Wall-Jet Model Theory in the Theory Guide.

24.4.1.5. The wall-film Boundary Condition

The **wall-film** boundary condition is used for modeling Lagrangian wall films. See Setting the Lagrangian Wall Film Model (p. 2802) for a detailed description of the functionality and options available.

24.4.1.6. The interior Boundary Condition

The **interior** boundary condition means that the particles will pass through the internal boundary. This option is available only for internal boundary zones, such as a radiator or a porous jump.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.4.1.7. The reinject Boundary Condition

The **reinject** boundary condition reintroduces DPM particles into the domain when they reach a certain domain boundary (for example, outlet). The location and velocity at which such recirculated particles are reinjected is taken from a user-specified injection, while all other particle properties remain unchanged.

As an application example, the **reinject** boundary condition type can be useful in simulating a fluidized bed, where usually the details of the external piping and gas-solid separators are not resolved in the computational mesh. In such cases, the **reinject** boundary condition can be used to model the reintroduction of the particles that have left the reactor vessel.

To use this boundary condition:

1. In the **Boundary Conditions** task page, select the zone to which you want to assign the **reinject** boundary condition and click **Edit…**.

2. In the **DPM** tab of the corresponding boundary condition dialog box that opens, select **reinject** from the **Discrete Phase BC Type** drop-down list.

3. In the **Use Settings From** drop-down list, select an injection whose properties will be used for reintroducing particles back to the domain.

   The settings of the selected injection are used to assign every particle that reaches the boundary a new position and new velocity.

   You can either use an existing injection that is also used for injecting new particles or create a separate injection only for specifying a location and initial velocity of reinjected particles. In the latter case, select **Reinjection Only** in the **Set Injection Properties** dialog box (**Particle Reinjection** group box) for that injection, so that it never introduces any particles by itself. Like the particle reinjection option, this option is available only when unsteady particle tracking is enabled.

   Note:

   For postprocessing and reporting, a reinjected particle remains associated with the injection from which it was originally injected.

4. If particles require a significant period of time when they are transported from the outlet to the location of reinjection, you can delay the reinjection by a specified amount of time as follows:

   1. In the **Set Injection Properties** dialog box, make sure that **Reinjection Only** is selected for the injection.

   2. In the **Point Properties** tab, specify **Reinjection Time Delay**.

   Particles that are delayed during reinjection will immediately appear at the new location in graphical postprocessing, but they will not move or participate in any interaction with the continuous phase until their individual delay period has passed.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Interpreting DPM Summary Reports in conjunction with particle reinjection is described in Reinjected Particles (p. 2854)

You can use a `DEFINE_DPM_BC` user-defined function (UDF) to modify the behavior of particle reinjection as shown in Example 5 in the *Fluent Customization Manual*.

24.4.1.8. The user-defined Boundary Condition

It is also possible to use a user-defined function to compute the behavior of the particles at a physical boundary. More information about user-defined functions can be found in the Fluent Customization Manual.

24.4.2. Default Discrete Phase Boundary Conditions

Ansys Fluent makes the following assumptions regarding boundary conditions:

- The **reflect** type is assumed at wall, symmetry, and axis boundaries, with both coefficients of restitution equal to 1.0
- The **escape** type is assumed at all flow boundaries (pressure and velocity inlets, pressure outlets, and so on)
- The **interior** type is assumed at all internal boundaries (radiator, porous jump, and so on)

The coefficient of restitution can be modified only for wall boundaries.

24.4.3. Coefficients of Restitution

A normal or tangential coefficient of restitution equal to 1.0 implies that the particle retains all of its normal or tangential momentum after the rebound (an elastic collision). A normal or tangential coefficient of restitution equal to 0.0 implies that the particle retains none of its normal or tangential momentum after the rebound.

Nonconstant coefficients of restitution can be specified for wall zones with the **reflect** type boundary condition. The coefficients are set as a function of the impact angle, θ_1, in Particle Reflection at Wall in the *Fluent Theory Guide*.

Note that the default setting for both coefficients of restitution is a constant value of 1.0 (all normal and tangential momentum retained).

If you select the **reflect** type at a wall (only), you can define the **Normal** and **Tangent** coefficients of restitution under **Discrete Phase Reflection Coefficients**. You can select a **constant**, **polynomial**, **piecewise-linear**, or **piecewise-polynomial** function from the relevant drop-down lists. The dialog boxes for defining the polynomial, piecewise-linear, and piecewise-polynomial functions are the same as those used for defining temperature-dependent properties. The applied wall heat transfer model assumes that a liquid is getting in contact with the wall. See Defining Properties Using Temperature-Dependent Functions (p. 1616) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.4.4. Friction Coefficient

When at least one injection accounts for particle rotation (the **Enable Rotation** option is selected under the **Physical Models** tab in the **Set Injections Properties** dialog box), you can specify friction coefficients for wall zones with the **reflect** type boundary condition. In the **Wall** dialog box, under the **DPM** tab, you can define the **Friction Coefficient** as a **constant** or as a **polynomial**, **piecewise-linear**, or **piecewise-polynomial** function.

The default value for the **constant** is 0.2. For the piecewise-linear and piecewise-polynomial profiles, the friction coefficient is computed in terms of the relative tangential velocity. The default inputs for these functions are derived from the friction collision law (The Friction Collision Law in the *Fluent Theory Guide*) and are as follows.

- The **piecewise-linear** approximation is defined using the following default points for the relative (particle-to-wall) tangential velocity:
  - Point 1: 0 m/s
  - Point 2: 1 m/s
  - Point 3: 10 m/s
  - Point 4: 210 m/s
- The **piecewise-polynomial** approximation is defined in the three relative (particle-to-wall) tangential velocity ranges as follows:
  - Range 1: 0–1 m/s: a quadratic function
  - Range 2: 1–10 m/s: a constant
  - Range 3: 10–210 m/s: a linear function

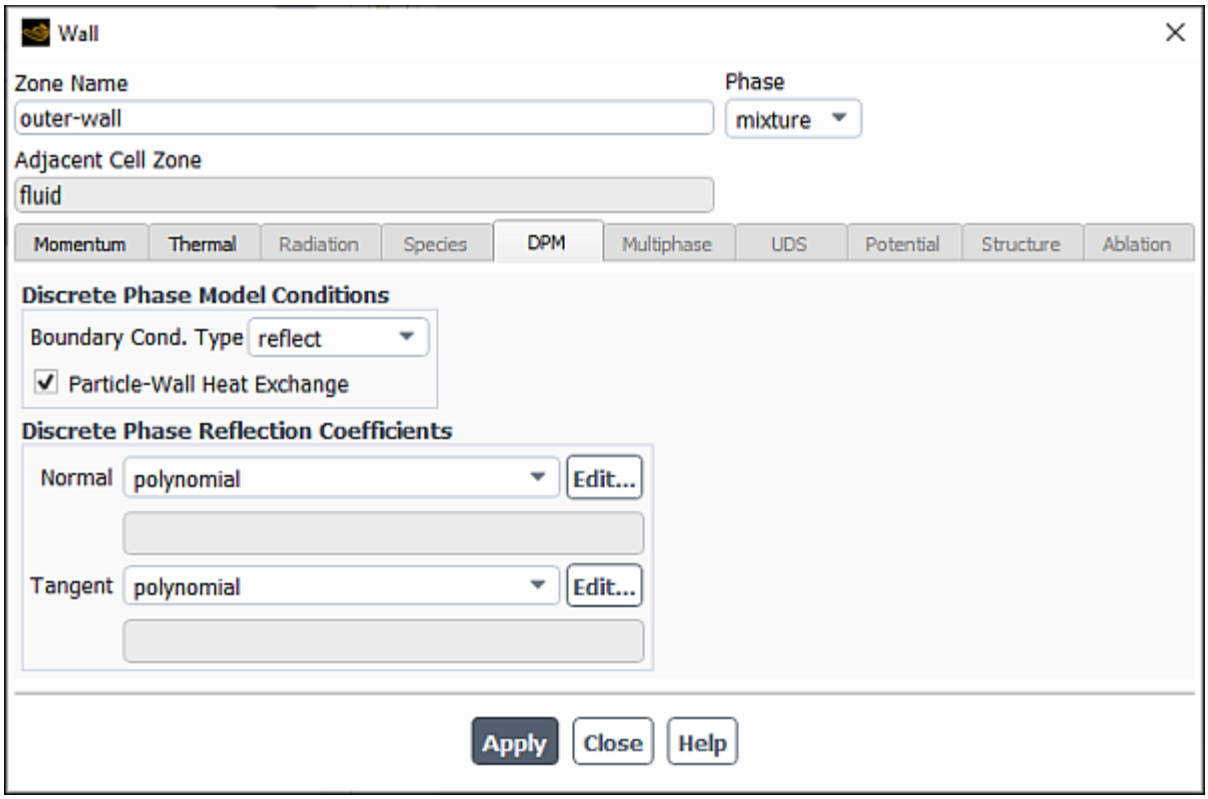
For details about using the **Piecewise-Linear Profile** dialog box and **Piecewise-Polynomial Profile** dialog box see Inputs for Piecewise-Linear Functions (p. 1618) and Inputs for Piecewise-Polynomial Functions (p. 1620), respectively.

24.4.5. Particle-Wall Impingement Heat Transfer

To enable the particle-to-wall heat exchange for the **reflect**, **wall-jet**, or **wall-film** boundary conditions: In the **Wall** dialog box, under the **DPM** tab, enable the **Particle-Wall Heat Exchange** option. The option is available only for wall boundary conditions and unsteady particle tracking when the energy equation is enabled.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.42: The Wall Dialog Box: the Particle-Wall Heat Exchange Option



Note that when a particle is reflected from a wall with the wall film DPM boundary condition, the particle-to-wall heat exchange is calculated directly between the particle and the wall. Any wall film present is not taken into account.

The model is applied for all inert, droplet, and multicomponent particles impinging on the wall. The specific model is not applied for the splashed particles. When the wall film DPM boundary condition is active, the model is applied in the Rebound Regime, and it is assumed that the particles hit the wall irrespective of the presence of a film. In the Splashing regime, the mass fraction that is deposited mixes with any existing film, and the splashed particles retain the impinging droplet temperature.

For combusting particles, the wall heat transfer is calculated only if the **Wet Combustion Model** option is enabled in the **Set Injection Properties** dialog box, and the particle liquid fraction is nonzero, otherwise the **Particle-Wall Heat Exchange** option has no effect.

See Particle–Wall Impingement Heat Transfer Theory in the *Fluent Theory Guide* for the underlying theory and equations of the particle–wall impingement heat transfer model.

24.4.6. Setting Particle Erosion and Accretion Parameters

If the **Erosion/Accretion** option is selected in the **Physical Models** tab of the Discrete Phase Model Dialog Box (p. 4845) (see Monitoring Erosion/Accretion of Particles at Walls (p. 2697)), you can enable erosion models in the **DPM** tab of the Wall Dialog Box (p. 5085) and specify the corresponding erosion

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

rate parameters at the walls. Note that if you are setting up erosion / accretion between a fluid and solid zone, the erosion models must be enabled on the fluid side of the coupled wall.

Important:

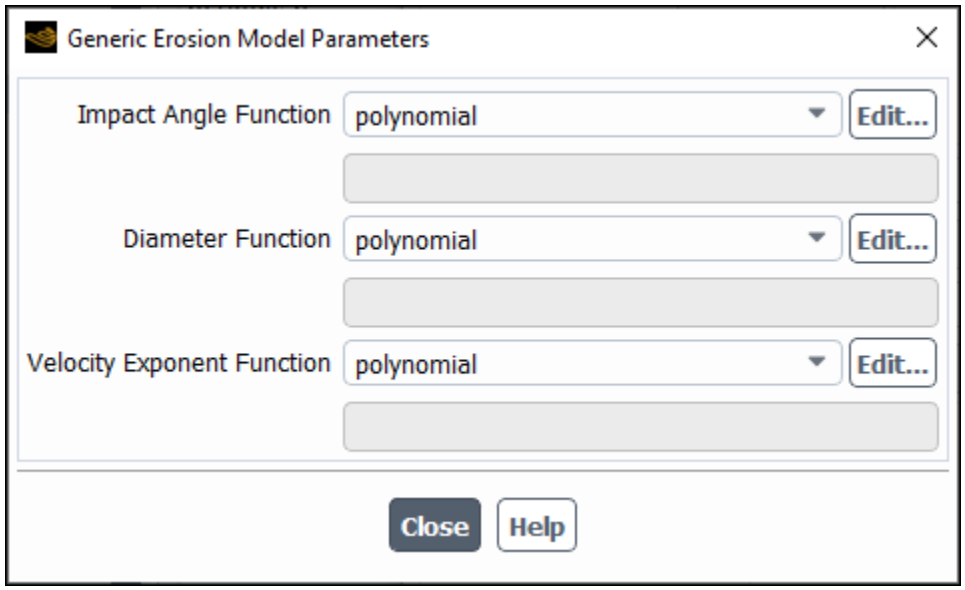
The default erosion rate parameters for each erosion model may not be appropriate for your analysis and must be adjusted based on your own experimental data.

The following models are available in Ansys Fluent:

- **Generic Model**

  The erosion rate of the generic model is defined in Equation 12.346 in the Theory Guide as a product of the mass flux and specified functions for the particle diameter, impact angle, and velocity exponent. To specify the erosion parameters for the Generic model, click **Edit...** next to **Generic Model**.

  Figure 24.43: The Generic Erosion Model Parameters Dialog Box



  In the **Generic Erosion Model Parameters** dialog box, specify the following parameters using a **constant**, **polynomial**, **piecewise-linear**, or **piecewise-polynomial** function:

  – **Impact Angle Function**: $f(\gamma)$ in Equation 12.346 in the *Fluent Theory Guide*.

  – **Diameter Function**: $C(d_p)$ in Equation 12.346 in the *Fluent Theory Guide*.

  – **Velocity Exponent Function**: $b(v)$ in Equation 12.346 in the *Fluent Theory Guide*.

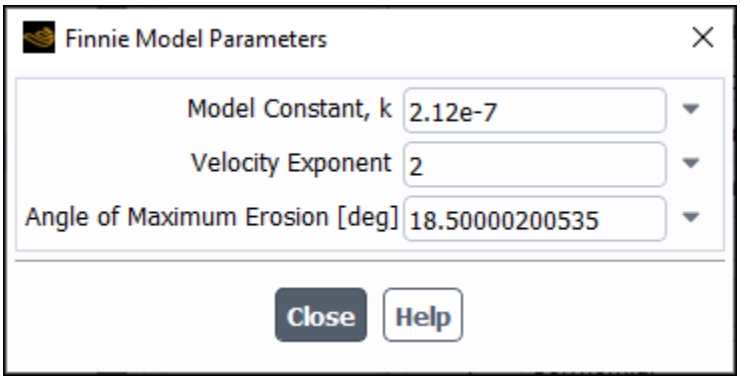
  See Accretion in the Theory Guide for a detailed description of these functions and Defining Properties Using Temperature-Dependent Functions (p. 1616) for details about using the dialog boxes for defining polynomial, piecewise-linear, and piecewise-polynomial functions.

- **Finnie**

  To specify the erosion parameters for the Finnie model, click **Edit...** next to **Finnie**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.44: The Finnie Model Parameters Dialog Box



In the **Finnie Model Parameters** dialog box, specify the following parameters:

– **Model Constant, k**: k in Equation 12.331 in the *Fluent Theory Guide*.

– **Velocity Exponent**: n in Equation 12.331 in the *Fluent Theory Guide*. The default value is 2. The value of the **Velocity Exponent** is generally within the range of 2.3 to 2.5 for metals.

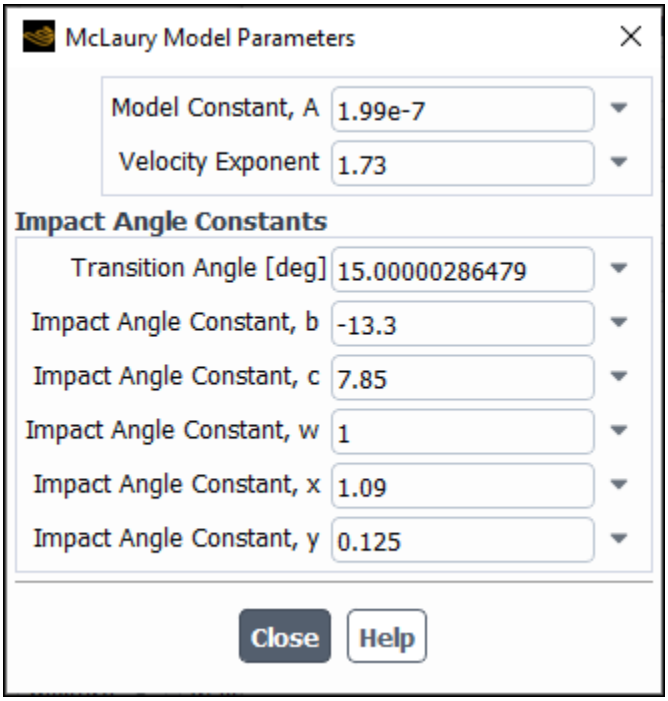
– **Angle of Maximum Erosion**: the impact angle between the approaching particle track and the wall at which the erosion reaches maximum.

For information about the theory behind the Finnie model, see Finnie Erosion Model in the *Fluent Theory Guide*.

- **McLaury**

To specify the erosion parameters for the McLaury model, click **Edit...** next to **McLaury**.

Figure 24.45: The McLaury Model Parameters Dialog Box



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the **McLaury Model Parameters** dialog box, specify the following parameters:

– **Model Constant, A**: A in Equation 12.335 in the *Fluent Theory Guide*.

– **Velocity Exponent**: V in Equation 12.335 in the *Fluent Theory Guide*.

– **Transition Angle**: γ_{lim} in Equation 12.336 and Equation 12.337 in the *Fluent Theory Guide*.

– **Impact Angle Constant, b**: b in Equation 12.336 in the *Fluent Theory Guide*.

– **Impact Angle Constant, c**: c in Equation 12.336 in the *Fluent Theory Guide*.

– **Impact Angle Constant, w**: w in Equation 12.337 in the *Fluent Theory Guide*.

– **Impact Angle Constant, x**: x in Equation 12.337 in the *Fluent Theory Guide*.

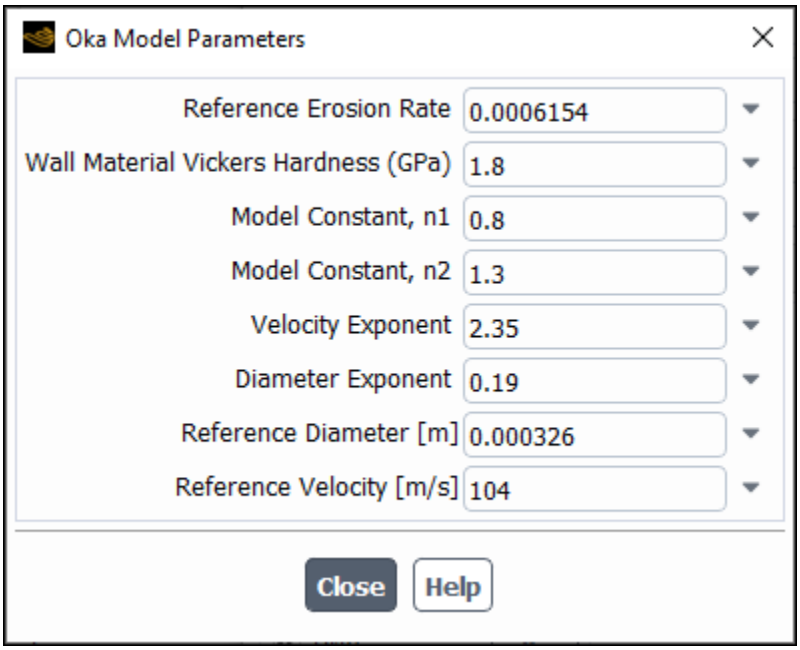
– **Impact Angle Constant, y**: y in Equation 12.337 in the *Fluent Theory Guide*.

See Example of the McLaury Erosion Model Constants in the *Fluent Theory Guide* for more information about the McLaury model parameters. For theoretical background on the McLaury model, see McLaury Erosion Model in the *Fluent Theory Guide*.

- **Oka**

To specify the erosion parameters for the Oka model, click **Edit...** next to **Oka**.

Figure 24.46: The Oka Model Parameters Dialog Box



In the **Oka Model Parameters** dialog box, specify the following parameters:

– **Reference Erosion Rate**: E_{90} in Equation 12.333 in the *Fluent Theory Guide*.

– **Wall Material Hardness**: H_v in Equation 12.334 in the *Fluent Theory Guide*.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– **Model Constant, n1**: n_1 in Equation 12.334 in the *Fluent Theory Guide*.

– **Model Constant, n2**: n_2 in Equation 12.334 in the *Fluent Theory Guide*.

– **Velocity Exponent**: k_2 in Equation 12.333 in the *Fluent Theory Guide*.

– **Diameter Exponent**: k_3 in Equation 12.333 in the *Fluent Theory Guide*.

– **Reference Diameter**: d_{ref} in Equation 12.333 in the *Fluent Theory Guide*.

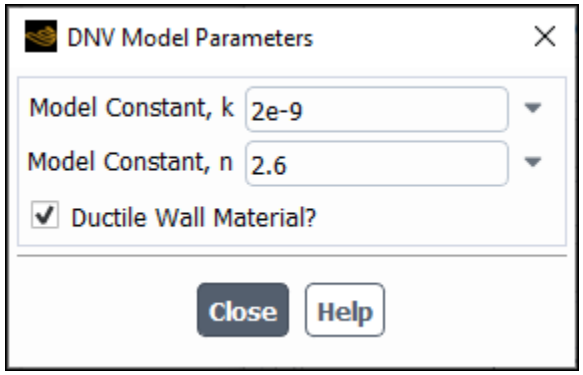
– **Reference Velocity**: V_{ref} in Equation 12.333 in the *Fluent Theory Guide*.

See Example of the Oka Erosion Model Constants in the *Fluent Theory Guide* for an example of the Oka model parameters' values. For more information on the theory behind the Oka model, see Oka Erosion Model in the *Fluent Theory Guide*.

- **DNV**

  To specify the erosion parameters for the DNV model, click **Edit...** next to **DNV**.

  Figure 24.47: The DNV Model Parameters Dialog Box

  In the **DNV Model Parameters** dialog box, specify the following parameters:

  – **Model Constant, k**: k in Equation 12.338 in the *Fluent Theory Guide*.

  – **Model Constant, n**: n in Equation 12.338 in the *Fluent Theory Guide*.

  – **Ductile Wall Material?**: When enabled, specifies that the wall material is ductile.

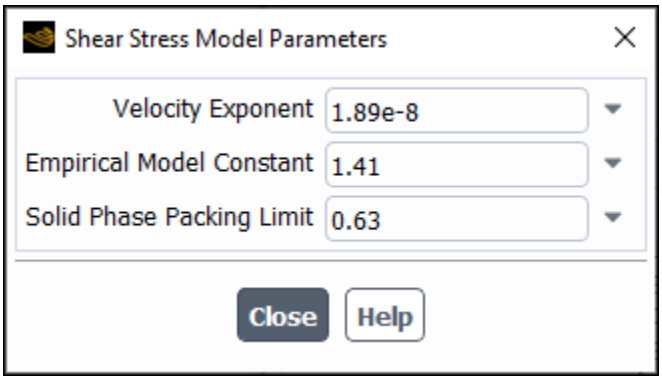
    For information about the theory behind the DNV model, see DNV Erosion Model in the *Fluent Theory Guide*.

- **Shear Stress** (dense flows with the disperse granular phase only)

  To specify the erosion parameters for the shear stress model, click **Edit...** next to **Shear Stress**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.48: The Shear Stress Model Parameters Dialog Box



In the **Shear Stress Model Parameters** dialog box, specify the following parameters:

– **Velocity Exponent**: n in Equation 12.341 in the *Fluent Theory Guide*.

– **Empirical Model Constant**: A in Equation 12.341 in the *Fluent Theory Guide*.

– **Solid Phase Packing Limit**: α_{pack} in Equation 12.341 in the *Fluent Theory Guide*.

This model should be used for dense multiphase flow with a dispersed phase consisting of solid particles. For more information on the theory behind the shear stress model, see Abrasive Erosion Caused by Solid Particles in the *Fluent Theory Guide*.

- **Granular Phase Shielding** (dense flows with the disperse granular phase only)

  The granular phase shielding as given by Equation 12.344 in the *Fluent Theory Guide* is always considered in the shear stress erosion model. If you want to include the shielding effect in other erosion models that are currently enabled, select **Granular Phase Shielding**. This option is available only when **Shear Stress Model Parameters** is selected.

  For more information about the theory behind the wall shielding effects in dense flows, see Wall Shielding Effect in Dense Flow Regimes in the *Fluent Theory Guide*.

By default, all erosion models are enabled.

Important:

- The default constants for the Finnie, Oka, and McLaury models have been tuned to match the experiment described by Oka [108] (p. 5719) in which 326-micron sand particles impact a carbon steel wall at a speed of 104 m/s and therefore are only valid for those specific conditions. You can use the default values as a starting point for your simulation and then adjust them as needed.
- Predictive capabilities of the erosion models are limited and strongly case dependent.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.5. Particle Erosion Coupled with Dynamic Meshes

Under certain circumstances, it is desirable to compute not only the particle-induced wall erosion rates, but also the actual deformation of the material due to erosion. The erosion of material from the surface of the wall may create large cavities that will affect particulate flow and erosion processes and eventually lead to material failure. To account for changes in the wall's shape and position and for more accurate predictions of the erosion rates, Ansys Fluent provides the capability to couple the erosion rate solver with the dynamic mesh model.

Generally, erosion is a slow process, typically far slower than the processes on the fluid side. Ansys Fluent implements the erosion dynamic mesh model using a quasi-steady approach. During each step of the solution process, the flow and particle erosion simulations are performed as steady-state, whereas the mesh position is updated by the dynamic mesh module using the physical time step size.

When erosion is coupled with dynamic meshes, Ansys Fluent computes the mesh deformation of an individual face Δx_{face} as:

$$\Delta x_{face} = ER_{face} * \Delta t_{MM} / \rho_{wm} \tag{24.13}$$

where,

ER_{face} = Wall face specific erosion rate density

Δt_{MM} = Mesh motion time step size

ρ_{wm} = Wall material density

24.5.1. Preliminaries

Before setting up a calculation of erosion coupled with the dynamic mesh model, perform the following steps:

1. Set up a DPM case (including defining fluid properties, setting boundary and cell zone conditions, etc.). For information on setting up a DPM calculation, see Steps for Using the Discrete Phase Models in the *Fluent User's Guide* (p. 2677).

2. Set up injections as described in Creating and Modifying Injections in the *Fluent User's Guide* (p. 2753).

3. Create report definitions to monitor flow variables.

4. Specify convergence criteria and solver controls for your simulation.

5. Initialize the flow field and run the solution.

   Note that you need to compute a large number of particle trajectories in order to provide sufficiently smooth erosion patterns for the dynamic mesh motion algorithm.

6. Enable erosion/accretion on affected walls as described in Monitoring Erosion/Accretion of Particles at Walls (p. 2697) and Setting Particle Erosion and Accretion Parameters in the *Fluent User's Guide* (p. 2787).

   Once erosion/accretion is enabled, the **Erosion Dynamic Mesh** item appears in the tree under the **Setup/Models/Discrete Phase** tree branch.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Perform a calculation for the erosion rates.

> **Note:**
>
> Note that it is not necessary to initialize the flow field and calculate both flow and erosion prior to coupling erosion with dynamic mesh. However, it will speed up the erosion dynamic mesh simulation if you start from a converged flow field and then follow with a first estimation of the erosion rates.

8. Once the solution is completed, select **Steady** in the **Genteral** task page (**Solver** group).

24.5.2. Limitations

Note the following limitations when modeling wall erosion coupled with the dynamic mesh model:

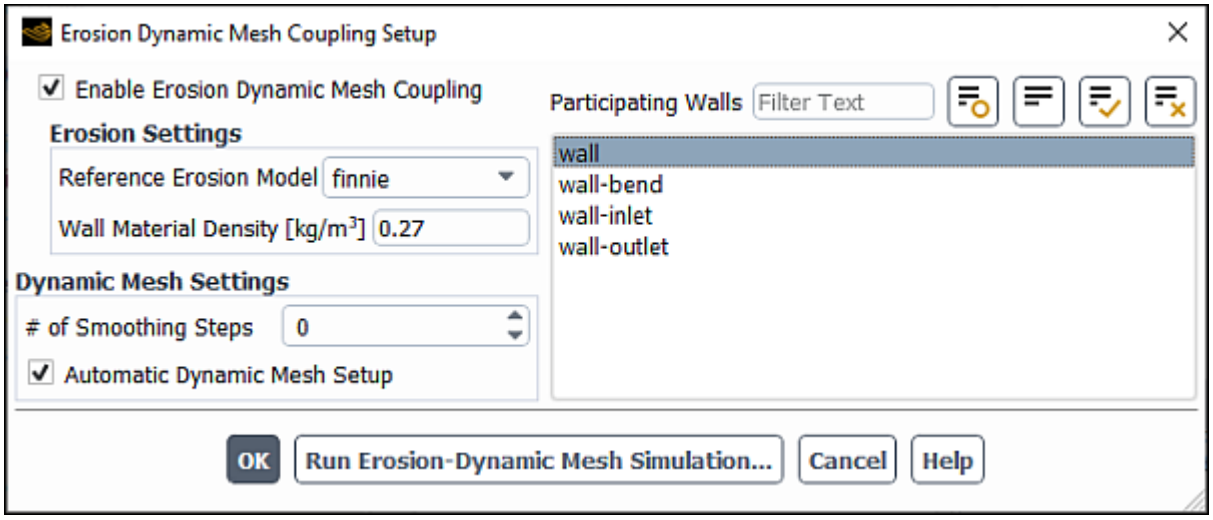
- Erosion coupling with dynamic mesh is available only for steady-state flows due to being implemented via a quasi-steady approach.
- You can run the erosion dynamic mesh simulation via either the **Erosion Dynamic Mesh Coupling Setup** dialog box or the `define/models/dpm/erosion-dynamic-mesh/run-simulation` text command, but not from the **Run Calculation** task page or via the `solve/iterate` text command.

24.5.3. Procedure for the Erosion Coupled with Dynamic Mesh Setup and Solution

1. Open the **Erosion Dynamic Mesh Coupling Setup** dialog box and select **Enable Erosion Dynamic Mesh Coupling**.

   Setup → Models → Discrete Phase → Erosion Dynamic Mesh Edit...

   Figure 24.49: The Erosion Dynamic Mesh Coupling Setup Dialog Box



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. In the **Erosion Settings** group box, select an appropriate **Reference Erosion Model** for your simulation and specify the **Wall Material Density**.

   You can choose from the following erosion models:

   - finnie (default)
   - generic-model
   - mclaury
   - oka
   - dnv

   For information about these models, refer to Setting Particle Erosion and Accretion Parameters in the *Fluent User's Guide* (p. 2787).

   Note that these models will not be available if you are using a UDF to specify the erosion accretion rates as described in `DEFINE_DPM_EROSION` in the *Fluent Customization Manual*. In this case, only **udf** will appear in **Reference Erosion Model**.

   The wall material density is assumed to be identical for all walls involved in the erosion coupled with dynamic mesh simulation. Note that for **Wall Material Density**, you may need to adjust the default value of 2719 kg/m^3, which is the density of aluminum.

3. (dense flows with the disperse granular phase only) If you want to model abrasive erosion rates and wall shielding effects, select **Include Abrasive Erosion Effects**. For more information about these models, refer to Modeling Erosion Rates in Dense Flows in the *Fluent Theory Guide*.

4. In the **Dynamic Mesh Settings** group box, you can:

   - Specify a non-zero value for **# of Smoothing Steps** if you want to apply smoothing to the wall erosion rates in the dynamic mesh module.

     Smoothing the erosion rates is recommended. For cases with a lower particle count, you can increase the **# of Smoothing Steps** to a value greater than 1. Note that tracking a sufficient number of particles in your simulation will ensure no large variations in erosion rate density. If you refine your mesh by a factor of 2 in each direction, you also need to increase the number of particles by a factor of 4 or more.

   - Enable **Automatic Dynamic Mesh Setup** (default)

     The **Automatic Dynamic Mesh Setup** allows for a quick setup of the basic functionalities needed for an erosion dynamic mesh coupled simulation. Selecting this option automatically enables **Dynamic Mesh** and the smoothing and remeshing methods. These methods will ensure that the mesh quality remains acceptable during the simulation. The option also automatically creates a dynamic zone of a **User-Defined** type for each zone participating in the coupled erosion dynamic mesh simulation. The motion of the dynamic zones is defined by the ****erosion**** mesh motion UDF provided by Ansys Fluent.

     For additional information, see the following sections:

     – Specifying the Motion of Dynamic Zones (p. 1904)

     – User-Defined Motion (p. 1913)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– Smoothing Methods (p. 1816)

– Local Cell Remeshing Method (p. 1845)

– Local Remeshing Method (p. 1844)

– Face Region Remeshing Method (p. 1852)

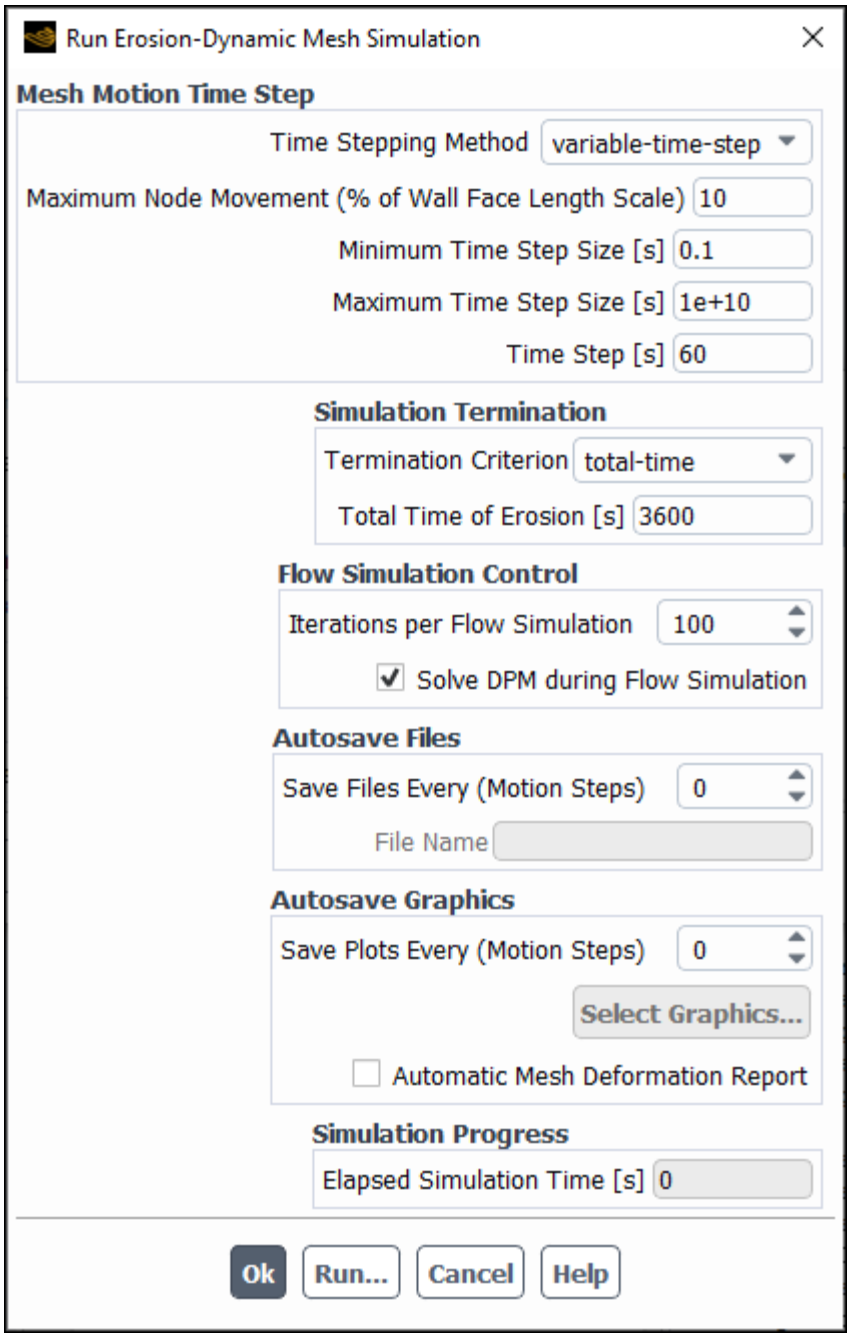
The automatic dynamic mesh setup uses the dynamic mesh methods with reasonable defaults, which may not be optimal for all cases. If you want to modify the dynamic mesh settings after the automatic setup has been completed, you must first disable **Automatic Dynamic Mesh Setup**; otherwise, your changes will be overwritten by the default settings. Before adjusting the dynamic mesh settings, you should become familiar with the **Deform Adjacent Boundary Layer with Zone** and **Feature Detection** options, which are described in Boundary Layer Smoothing Method (p. 1834) and Feature Detection in the *Fluent User's Guide* (p. 1868). Note that if you are setting up erosion / accretion on a moving wall between a fluid and solid zone, you must create a deforming zone for the solid zone that uses the radial basis function smoothing method (for details, see Radial Basis Function Smoothing (p. 1830)).

5. From the **Participating Walls** selection list, select walls that will be deformed due to particle-wall erosion. Note that if you are setting up erosion / accretion between a fluid and solid zone, the fluid side of the coupled wall should be selected under **Participating Walls**, not the solid side.

6. To set controls for the erosion dynamic mesh solver and then run the simulation, click **Run Erosion-Dynamic Mesh Simulation…**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.50: The Run Erosion-Dynamic Mesh Simulation Dialog Box



a. In the **Mesh Motion Time Setup** group box, you can select the time stepping method to compute the mesh deformation from Equation 24.13 (p. 2793):

   - **fixed-time-step**: allows you to specify a constant **Time Step**.
   - **variable-time-step** (default): is automatically determined by the dynamic mesh solver based on the following parameters:

     Maximum Node Movement (% of Wall Face Length Scale)

     the maximum allowed cell deformation per meshing step. It is specified as a fraction (in percent) of the characteristic length, which is derived from the cell wall face

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

area. A default of 10% is an acceptable, conservative value. To speed up the calculation, you can set this parameter to a higher value. A value of 20 to 30%, or even higher, is suitable in most cases. However, because the erosion rate is considered to be constant during each time step, some erosion rate variations may be lost for larger mesh deformations. Trial and error may be required to find an optimal value for your case input.

Minimum/Maximum Time Step Size

upper and lower time limits. If you want the solver to use the time step calculated by the dynamic mesh module, set **Maximum Time Step Size** to a large value.

During the iteration process, the solver adjusts the time step based on the maximum node movement. The solver will gradually increase the time step up to its maximum value, resulting in a faster overall simulation time. Ansys Fluent will report the time step used in the current iteration in the console and in the **Time Step** field.

b. In the **Simulation Termination** group box, specify the **Total Time of Erosion** (in seconds). The simulation stops when the specified time period ends.

c. In the **Flow Simulation Control** group box, you can specify:

- **Iterations per Flow Simulation**: is the number of flow iterations between mesh deformation steps. This number should be sufficiently large to ensure that the flow simulation converges between successive mesh steps.
- (only when **Interaction with Continuous Phase** is enabled) **Solve DPM during Flow Simulation**: enforces a tighter coupling between gas and particle phases. During the flow calculation, the particle tracker will be called regularly at the frequency specified in **DPM Iteration Interval** in the **Discrete Phase Model** dialog box.

d. If you want to automatically save case and data files during the calculation, you can specify a filename and a frequency with which case and data files should be saved for postprocessing or restart purposes in the **Autosave Files** group box. When saving case and data files, Ansys Fluent appends the filename that you have provided with `@time=`*`flow time`* .

e. Specify the autosave plot options in the **Autosave Graphics** group box.

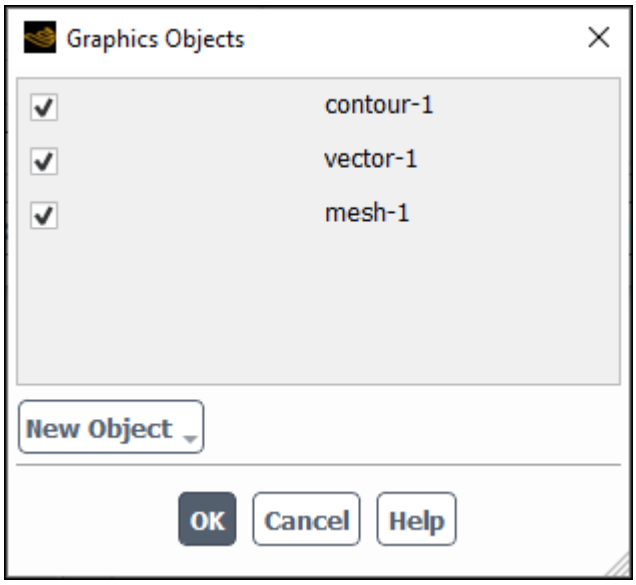
i. If you want to capture images of mesh, contour, or vector plots at the specified intervals for later use (for example, to generate animations of your solution over time), set the frequency at which your plots will be saved in the **Save Plots Every** text entry field and click **Select Graphics...**.

Note:

The **Select Graphics...** button becomes available only after the flow field has been initialized.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.51: The Graphics Objects Dialog Box



A. In the **Graphics Objects** dialog box, from the **New Object** drop-down list, select the object that you want to save:

   - **Mesh...** opens the Mesh Display Dialog Box (p. 4688)
   - **Contours...** opens the Contours Dialog Box (p. 5412)
   - **Vectors...** opens the Vectors Dialog Box (p. 5655)

   Once you create a new object, it will be displayed in the **Graphics Objects** dialog box.

B. Select which graphics objects you want to save as PNG images during the solution run. Similar to case and data files, the image filenames will be appended with `@time=`*`flow time`* .

ii. If you want Ansys Fluent to plot the vertex maximum of the accumulated deformation variable on all walls undergoing mesh deformation, enable **Automatic Mesh Deformation Report**.

When you start iterating, Ansys Fluent automatically creates the **max-accumulated-deformation** report definition and **max-accumulated-deformation-pset** report plot, which will be displayed in the graphics window during the solution run.

7. Click **Run...** in the **Run Erosion-Dynamic Mesh Simulation** dialog box.

Important:

The erosion dynamic mesh simulation must be run only from the **Run Erosion-Dynamic Mesh Simulation** dialog box.

During the coupled erosion dynamic mesh analysis, first, the erosion module computes erosion rates and passes them to the dynamic mesh module. The dynamic mesh module performs a pseudo-steady simulation of the mesh deformation. After that, the solver carries out a steady-state

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

flow calculation for a specified number of flow iterations. If **Solve DPM during Flow Simulation** is enabled, the particle tracking is performed in the specified **DPM Iteration Interval**. Otherwise, the particle tracker is run once after the flow calculation is finished. Whenever the particle tracker is run, the DPM sources are updated. This process is repeated until the specified **Total Time Of Erosion** is reached.

As the solution progresses, Ansys Fluent reports the dynamic mesh time step and the simulation time in the console and the **Time Step** and **Elapsed Simulation Time** fields. If you interrupt the simulation using the **Interrupt Simulation** dialog box, the solver will complete the current flow and erosion iterations before stopping the calculation.

24.5.4. Postprocessing for Erosion Dynamic Mesh Calculations

You can generate graphical plots or alphanumeric reports for the mesh deformation using the following additional postprocessing variable:

- **Accumulated Deformation** (in the **Mesh...** category)

24.6. Modeling Lagrangian Wall Films

The Lagrangian Wall Film (LWF) model can be used to predict the creation and flow of thin liquid films on the surface of walls. This section presents information about the basic functionality of the LWF model.

The LWF model uses DPM particles to represent the wall-film. In the Ansys Fluent LWF model, a liquid droplet or multicomponent particle can impinge upon a boundary surface and form a thin film. The model can be broken down into four major subtopics:

- interaction during the initial impact with a wall boundary
- subsequent tracking on surfaces
- calculation of film variables
- coupling to the gas phase

All inputs and model parameters for the LWF model are specified in the **DPM** tab of the **Wall** dialog box (opened from the **Boundary Conditions** task page). Once you define one or more DPM injections, you will be able to select **wall-film** as **Discrete Phase BC Type** (**Discrete Phase Model Boundary Conditions** group box) and specify all relevant parameters in the **Wall** dialog box.

For additional information, see the following sections:

24.6.1. Limitations on Using the Lagrangian Wall Film Model

24.6.2. Setting the Lagrangian Wall Film Model

24.6.3. Film Condensation Model

24.6.4. Gas-Side Boundary Layer Model

24.6.5. Patching the Wall Film

24.6.1. Limitations on Using the Lagrangian Wall Film Model

- The Lagrangian wall film model is only available with **Unsteady Particle Tracking**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The Lagrangian wall film model is not compatible with hanging node mesh refinement on walls. Though the fluid meshes can be adapted, the refined meshes must not touch walls where the wall film model is enabled.
- The wall film model assumes liquid particle materials. If an inert particle interacts with a wall film boundary, it is assumed to be liquid for the wall film model calculations. For combusting particles, the wall film is applied if **Wet Combustion Model** is enabled and as long as the liquid fraction in the particle is nonzero. If a dry combusting particle interacts with a wall film boundary it will stick to the wall and follow the combusting particle laws.
- The wall film model is not compatible with the Rosseland Radiation model.
- The **Workpile Algorithm** option is not available with the wall film boundary condition. It will be disabled automatically when choosing to simulate a wall film on a wall.
- The Eulerian Wall Film (EWF) model is not compatible with the Lagrangian wall-film boundary condition. If you want to use the EWF model in your simulation, you must first deselect **wall-film** as **Discrete Phase BC Type** in the **Wall** dialog box (**DPM** tab).
- The wall film model can be used with overset meshes only if the film zone is entirely contained within a single overset mesh. Lagrangian wall film particles should not enter the overlap region of background and component meshes. An example of a valid configuration is shown in Figure 24.52: Valid Configuration for the Lagrangian Wall film Model with Overset Mesh (p. 2801).

Figure 24.52: Valid Configuration for the Lagrangian Wall film Model with Overset Mesh

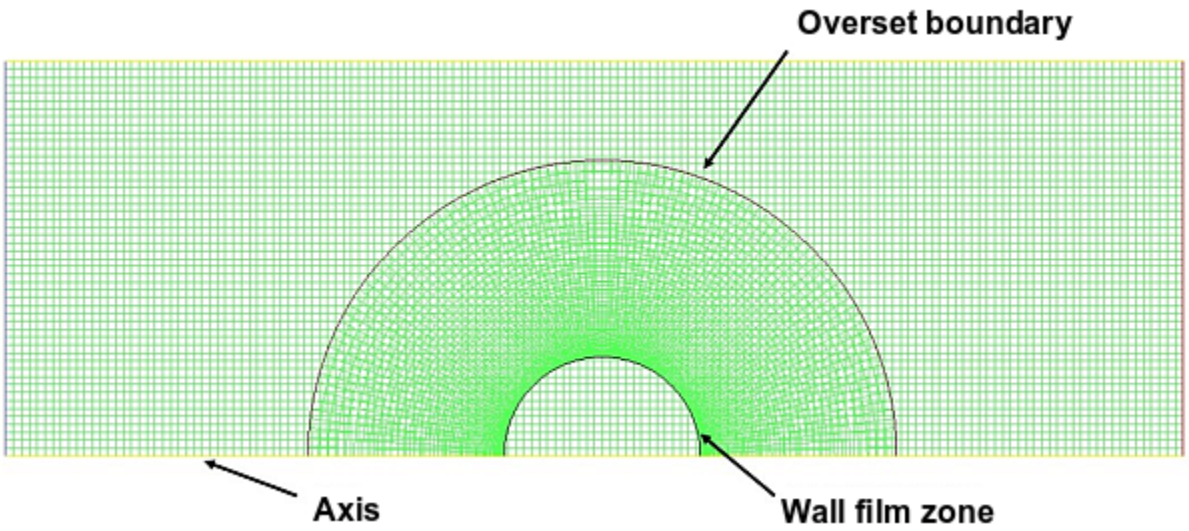
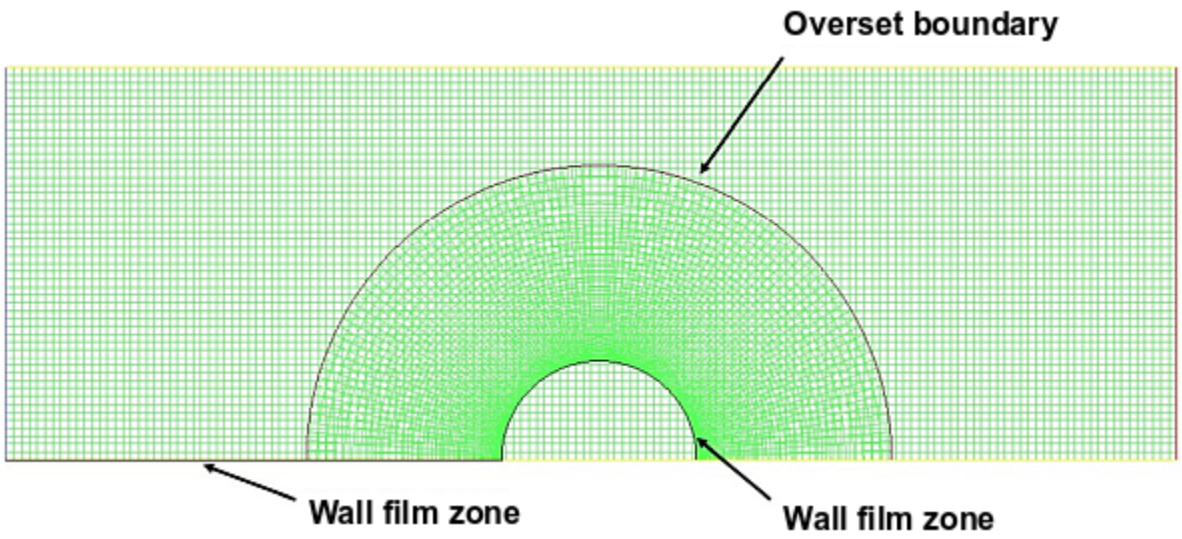


Figure 24.53: Invalid Configuration for the Lagrangian Wall film Model with Overset Mesh (p. 2802) shows an invalid setup in which the wall film zone crosses between the background and component meshes.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.53: Invalid Configuration for the Lagrangian Wall film Model with Overset Mesh



For additional assumptions used by the Lagrangian wall film model, refer to Introduction and Interaction During Impact with a Boundary in the *Fluent Theory Guide*.

Note that the wall film model only applies when the **wall-film** is selected as **Discrete Phase BC Type** in the **Wall** dialog box (**DPM** tab). If wall film particles intersect with a boundary of another type (for example **reflect**), they will be released from the film and become free-stream particles. The following boundaries are exceptions:

- the **trap** boundary condition where the wall film particles are absorbed at the wall with simultaneous release of the momentum, energy, and species (if species are defined) sources
- the **symmetry** boundary condition where the wall film particles are reflected

Special consideration related to wall film particles defined on moving boundaries: if the particles move to another zone, where **wall-film** is not selected as **Discrete Phase BC Type**, then the particles obey the boundary condition defined for this other zone.

24.6.2. Setting the Lagrangian Wall Film Model

For the **wall-film** boundary condition, you can enable and/or set the following models in the **DPM** tab of the **Wall** dialog box:

- Particle-Wall Heat Exchange

  See Particle-Wall Impingement Heat Transfer (p. 2786) for details about this mode.

- Film condensation

  See Film Condensation Model (p. 2806) for details about this mode.

- Gas-side boundary layer model (default)

  See Gas-Side Boundary Layer Model (p. 2809) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Particle erosion (DPM model with the **Erosion/Accretion** option)

  See Setting Particle Erosion and Accretion Parameters (p. 2787) for details about modeling particle erosion.

- Initialize wall film

  See Patching the Wall Film (p. 2809) for details.

- Particle impingement/splashing

  The following options are available in the **Impingement/Splashing Model** drop-down list:

  – **stanton-rutland**: (default) consists of four regimes, namely, stick, rebound, spread, and splash, which are based on the impact energy and wall temperature. See The Stanton-Rutland Model in the *Fluent Theory Guide* for more details.

  – **kuhnke**: consists of four regimes spread, rebound, splash, and dry splash (thermal breakup). See The Kuhnke Model in the *Fluent Theory Guide* for more details. You must specify the following **DPM Wall Roughness Parameters**:

    → **Ra**: is used to compute K_{cold} in Equation 12.249 in the *Fluent Theory Guide*.

    → **Rz**: is used to compute splashed droplets ejection angle in Equation 12.261 in the *Fluent Theory Guide*.

  – **stochastic kuhnke**: is derived from the Kuhnke model and has been developed and tuned for addressing Selective Catalytic Reduction (SCR) modeling applications. It introduces stochastic effects into the critical temperature transition process, and the "partial evaporation" concept for the evaporative splash regime. See The Stochastic Kuhnke Model in the *Fluent Theory Guide* for details. For this model you must specify **Rz** under the **DPM Wall Roughness Parameters**. This parameter will be used to interpolate the constant A from Parameter *A* as a Function of Wall Roughness in the *Fluent Theory Guide*, which will be subsequently used in Equation 12.267 in the *Fluent Theory Guide* to calculate critical Weber number for splashing on a dry wall.

  If you want to model splashing, specify **Number of Splash Drops** (in the **Impingement/Splashing Parameters** group box). A value of zero implies that the splashing model will not be used.

- Regime parameters

  When modeling drop/wall impingement/splashing using the stanton-rutland and kuhnke models, you can calculate the critical transition temperature (T_{crit}) of the fluid that forms the film via one of the following options in the **Regime Parameters** group box:

  – **critical temperature factor**: uses Equation 12.210 and Equation 12.240 in the *Fluent Theory Guide* for the **stanton-rutland** and **kuhnke** splashing models, respectively

  – **calibratable temperature**: uses Equation 12.211 and Equation 12.241 in the *Fluent Theory Guide* for the **stanton-rutland** and **kuhnke** models, respectively

    The **calibratable temperature** may improve the predictions for Selective Catalytic Reduction (SCR) systems.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When modeling drop/wall impingement/splashing using the stochastic kuhnke model, specify the following parameters in the **Regime Parameters** group box:

– **Upper Deposition Limit Offset**: is ΔT_{DL} in Equation 12.264 in the *Fluent Theory Guide*

– **Deposition Delta T**: is ΔT_{lim} in Equation 12.265 in the *Fluent Theory Guide*

– **Laplace Number Constant**: is A_w in Equation 12.266 in the *Fluent Theory Guide*

– **Partial Evaporation Ratio**: is the mass fraction of the impinging liquid spray that vaporizes immediately upon impact when the droplet is in the evaporative splash regime

- Film stripping

  To model film stripping, enable **Particle Stripping** and specify **Critical Shear Stress** (**Stripping Model Parameters** group box). When this value is exceeded, mass will be taken from the film on the face where liquid film exists.

  Optionally, you can use the following text user interface commands to specify:

  – diameter coefficient (F in Equation 17.20 in the Theory Guide (p. 1)):

    ```
    define/models/dpm/stripping-options/diameter-coefficient
    ```

  – mass coefficient (C in Equation 17.21 in the Theory Guide (p. 1)):

    ```
    define/models/dpm/stripping-options/mass-coefficient
    ```

  Important:

  Particle stripping can be modeled only if the film contains a single particle material or a single particle-mixture material. The combusting-particle type is not supported.

  For theoretical information about this option, see Film Stripping in the *Fluent Theory Guide*.

- Film separation

  You can select from the following options from the **Film Separation Model** drop-down list (in the **Separation Model Parameters** group box):

  – **o'rourke**

  – **foucart** (3d only)

  – **friedrich**.

  Details about these models can be found in Separation Criteria in the *Fluent Theory Guide*.

  You can also specify:

  – **Critical Weber Number**: is We_f in Equation 17.7 in the *Fluent Theory Guide*

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– **Film Separation Angle**: is $\theta_{critical}$ in Equation 17.7 in the *Fluent Theory Guide*

Note:

Selecting **wall-film** automatically enables the **Consider Children in the Same Tracking Step** option in the **Discrete Phase Model** dialog box (**Physical Models** tab).

Additional modeling options available via the Text User Interface (TUI) are described in the following sections:

24.6.2.1. Setting the Film-to-Wall Heat Transfer Model

24.6.2.2. Removing the Wall Temperature Limiter for Lagrangian Wall-Film Walls

24.6.2.1. Setting the Film-to-Wall Heat Transfer Model

The default heat transfer model assumes that the heat is transferred from the wall to the film by conduction only. If you want to include convective heat transfer, issue the following text command:

```
define/models/dpm/options/convective-film-heat-transfer?
```

and enter **yes** at the prompt:

```
Enable convection/conduction film to wall heat transfer (else conduction only)? [no] yes
```

By default, the convective heat transfer model uses the turbulent approximation described in Convective Heat Transfer Model in the *Fluent Theory Guide*. To disable it, enter **no** at the next prompt:

```
Enable turbulent film heat transfer approximation? [yes] no
```

This model is recommended for thicker films, films with high velocity, and films of high Prandtl number liquids. In addition, films formed by jet impingement will be represented more accurately with this model since it includes a formulation that specifically takes into account liquid jet impingement processes.

Note:

When enabling the convective wall-film heat transfer model for older Ansys Fluent cases with film particles already existing on the wall, a modified and less accurate formulation will be applied. The current model implementation requires the distance of the film particle from the impingement location to be known. This information is not available for existing film particles. Therefore, if you want to use the convective wall-film heat transfer model for wall-film calculations in older Ansys Fluent cases, it is recommended that you clear existing film particles and recalculate the film.

See Convective Heat Transfer Model in the *Fluent Theory Guide* for more information about the convective wall-film heat transfer model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.6.2.2. Removing the Wall Temperature Limiter for Lagrangian Wall-Film Walls

In Ansys Fluent, the wall temperature is limited by default. The limit is calculated by Equation 12.205 in the *Fluent Theory Guide*. You can remove the wall temperature limit by invoking the following text command:

```
define/models/dpm/options/remove-wall-film-temperature-limiter?
```

You will be asked:

```
Remove the wall film temperature limiter [no]:
```

If you answer **yes**, the wall temperature will not be limited during the simulation, resulting in a smoother wall film appearance. However, it may introduce instabilities to the film formation process, and may have the effect that the wall film completely evaporates off the wall.

If you answer **no** (recommended), you will be prompted to define the temperature difference above the Boiling point:

```
Temperature difference above the liquid film boiling point [50]:
```

You will be then prompted whether you want to enable the reporting of the Leidenfrost temperature on the wall faces:

```
Report the Leidenfrost temperature on wall film faces [yes]
```

Answering **yes** makes the **Leidenfrost Temperature** variable available for postprocessing. For the DPM verbosity level higher than 1 (set by the text command `define/models/dpm/numerics/verbosity`), Ansys Fluent reports in the console for every DPM iteration the percent of the wall area covered by film and percent of film mass where the wall temperature is above the Leidenfrost point.

For more information, see Leidenfrost Temperature Reporting in the *Fluent Theory Guide*.

Detailed information on the wall film model can be found in Lagrangian Wall-Film Model Theory in the *Fluent Theory Guide*.

For a list of limitations that exist with **wall-film** boundary conditions, see Limitations on Using the Lagrangian Wall Film Model (p. 2800).

24.6.3. Film Condensation Model

Condensation can be defined as the transition from the gas phase to the liquid phase as a vapor condenses on a surface. The Ansys Fluent film condensation model is triggered when the partial pressure of vapor species i in the bulk gas mixture exceeds either its vapor pressure at the wall-film or, in case no film exists, at the wall temperature. When these conditions are met, the vapor species will condense and form the liquid film on the wall faces.

The following limitations currently exist in the film condensation model:

- The film particles are stagnant

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

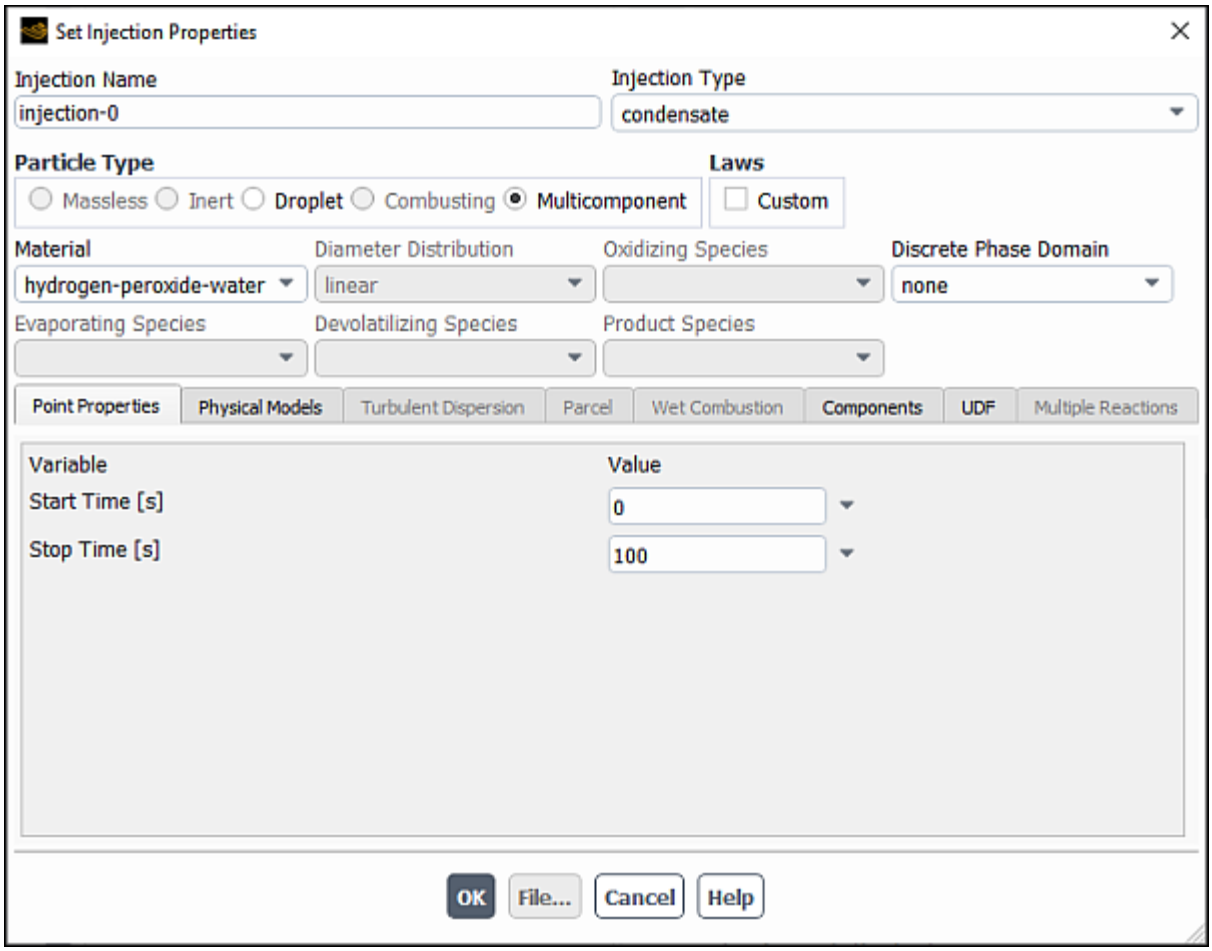
- Only particles of a single material (droplet or multicomponent) can condense on a particular wall face
- At least one species in the gas phase mixture must be non-condensable

To use the film condensation model:

1. Enable the following models:
   - Energy equation
   - Species Transport model
   - **Unsteady Particle Tracking** (in the **Discrete Phase Model** dialog box)
2. If no droplet or multicomponent injection exists in the DPM case setup, create a condensate injection by selecting **condensate** from the **Injection Type** dropdown list in the **Set Injection Properties** dialog box and define the injection properties as appropriate.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.54: The Set Injection Properties Dialog Box: Condensate Injection



Note:

A droplet or multicomponent injection must exist to define the vapor–liquid material pair for the condensation process.

For further information, see Injection Types (p. 2720), Point Properties for Condensate Injections (p. 2745), and Defining Injection Properties (p. 2756).

3. For each droplet and multicomponent injection, specify the condensable species by selecting the **Evaporating Species**.

4. In the **Wall** dialog box, under the **DPM** tab, select **wall-film** from the **Discrete Phase BC Type** drop-down list and enable the **Film Condensation** option for each wall where condensation may occur. If the **Gas-Side Boundary Layer Model** option is disabled, the Fluent solver automatically enables it. For more information, see Gas-Side Boundary Layer Model (p. 2809).

Note:

Film Condensation is available only with the **wall-film** DPM boundary condition.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Enable **Linearized Source Terms** in the **Numerics** tab of the **Discrete Phase Model** dialog box.

   Enabling the **Linearized Source Terms** option helps to stabilize the condensation solution and is strongly advisable.

When the appropriate conditions exist, the evaporating species you have selected will condense on the walls that have the **Film Condensation** option enabled forming either droplet or multicomponent film particles (depending on the **Particle Type** selected in the **Set Injection Properties** dialog box).

For information about the theory behind the Film Condensation model, see Film Condensation in the *Fluent Theory Guide*.

24.6.4. Gas-Side Boundary Layer Model

The Gas-Side Boundary Layer model enables the boundary layer formulations for the mass and heat transfer equations for film vaporization and boiling as described in Film Vaporization and Boiling and Energy Transfer from the Film in the *Fluent Theory Guide*.

The model is enabled by default. To disable it, clear **Gas-Side Boundary Layer Model** in the **DPM** tab of the **Wall** dialog box (**Discrete Phase Model Boundary Conditions** group box). Note that the boundary layer formulations are always used for both condensation and vaporization rates when the Film Condensation model is enabled.

24.6.5. Patching the Wall Film

When a DPM case with the **wall-film** boundary condition is initialized or data is read, you can initialize the Lagrangian wall film (LWF) as follows:

1. In the **DPM** tab of the **Wall** dialog box, enable **Initialize Wall Film** (**Discrete Phase Model Boundary Conditions** group box).

2. Once **Initialize Wall Film** is selected, you can specify the following parameters in the **Wall Film Initial Conditions** group box:

   Film Height

   is the prescribed height of the film. When the film height is prescribed the film mass can be computed by the following equation:

   $$m_f = h_f A \rho_p$$

   where

   h_f = prescribed film height (m)

   A = wall face area (m^2)

   ρ = particle density assigned from the injection selected (kg/m^3)

   Film X, Y, Z Velocity

   are the X, Y, and Z (for 3D flows) components of the film initial velocity that are assigned directly to the film parcels.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Film Temperature

is the initial temperature of the film. This input is available only if **Energy** is enabled.

Minimum Number Of Parcels Per Face

is the minimum number of film parcels that are assigned per face. Note that for highly non-uniform meshes, a value of 1 is recommended.

Minimum Film Parcel Surface Density

is the minimum surface density of the film parcels. This value will be overridden if the number calculated from **Minimum Number Of Parcels Per Face** is larger than the number entered here. For example, if the **Minimum Number Of Parcels Per Face** is specified as 10, and the **Minimum Film Parcel Surface Density** is specified as 1,000,000/m^2, then this value will be overridden for any wall face with a surface area less than $1e^{-5}$ m^2. So, if a wall face surface area = $9e^{-6}$ m^2, then:

$$minimum\ film\ parcel\ surface\ density * surface\ area = 1{,}000{,}000 * 9e^{-6} = 9$$

Because the resulting value is less than 10, the number of parcels assigned to this wall face will be 10.

If the wall face surface area = $1.1e^{-5}$ m^2, then:

$$minimum\ film\ parcel\ surface\ density * surface\ area = 1{,}000{,}000 * 1.1e^{-5} = 11$$

and the number of parcels assigned to this wall face will be 11.

Initialize From Injection

is used to assign the injection material and the corresponding physical properties to the film.

3. Click **Apply All Settings and Initialize Film on Selected Wall(s) Now**.

   All the settings in **Wall** dialog box will be applied, and the Lagrangian wall film on the selected wall zone will be initialized immediately.

Ansys Fluent uses a random number sequence to distribute the parcels on each of the wall faces with the following exception: when one single parcel is assigned to a face, the parcel is placed on the face centroid.

Also, you can reset the LWF to zero from the **Solution Initialization** task page by clicking **Reset LWF**. This will delete the LWF film particles as well as zero out the corresponding face variables without affecting the free stream particles.

In addition, by clicking **Initialize LWF** in the **Solution Initialization** task, you can initialize (or patch) the Lagrangian wall film on all wall zones for which the wall-film boundary condition has been chosen and **Initialize Wall Film** has been selected.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.7. Setting Material Properties for the Discrete Phase

In order to apply the physical models described in earlier sections to the prediction of the discrete phase trajectories and heat/mass transfer, Ansys Fluent requires many physical property inputs.

For additional information, see the following sections:

24.7.1. Summary of Property Inputs

24.7.2. Setting Discrete-Phase Physical Properties

24.7.1. Summary of Property Inputs

Table 24.1: Property Inputs for Inert Particles (p. 2811) – Table 24.5: Property Inputs for Multicomponent Particles (Law 7) (p. 2814) summarize which of these property inputs are used for each particle type and in which of the equations for heat and mass transfer each property input is used. Detailed descriptions of each input are provided in Setting Discrete-Phase Physical Properties (p. 2814).

Table 24.1: Property Inputs for Inert Particles

Property	Symbol
density	ρ_p in Equation 12.1 in the Theory Guide
specific heat	c_p in Equation 12.70 and Equation 12.294
thermal conductivity	k_p in Equation 12.12 and k_l in Equation 12.295
particle emissivity	ε_p in Equation 12.70
particle scattering factor	f in Equation 5.73
thermophoretic coefficient	$D_{T,p}$ in Equation 12.11

Table 24.2: Property Inputs for Droplet Particles

Properties	Symbol
density	ρ_p in Equation 12.1 in the Theory Guide
specific heat	c_p in Equation 12.93 and Equation 12.294
thermal conductivity	k_p in Equation 12.12 and k_l in Equation 12.295
viscosity	μ in Equation 12.402
latent heat	h_{fg} in Equation 12.93, Equation 12.283, Equation 12.284, and Equation 12.301
vaporization temperature	T_{vap} in Equation 12.78
boiling point	T_{bp} in Equation 12.78, Equation 12.99
volatile component fraction	f_{v0} in Equation 12.79, Equation 12.100
binary diffusivity	$D_{i,m}$ in Equation 12.83 and D_i in Equation 12.276
diffusivity reference pressure	P_{ref} in Equation 12.91 in the *Fluent Theory Guide*

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Properties	Symbol
saturation vapor pressure	$p_{sat}(T)$ in Equation 12.81
heat of pyrolysis	h_{pyrol} in Equation 12.508
droplet surface tension	σ in Equation 12.371, Equation 12.401
particle emissivity	ε_p in Equation 12.93, Equation 12.104
particle scattering factor	f in Equation 5.73
composition averaging coefficient	α_Y in Equation 12.186
temperature averaging coefficient	α_T in Equation 12.185
Thermolysis model	
– single-rate	
pre-exponential factor	A in Equation 12.87 (droplet material) or A_i in Equation 12.164 in the *Fluent Theory Guide* (multicomponent particles)
activation energy	E in Equation 12.87 (droplet material) or E_i in Equation 12.164 in the *Fluent Theory Guide* (multicomponent particles)
– constant	
rate constant	C in Equation 12.88 (droplet material) or C_i in Equation 12.165 in the *Fluent Theory Guide* (multicomponent particles)

Table 24.3: Property Inputs for Combusting Particles (Laws 1–4)

Property	Symbol
density	ρ_p in Equation 12.1 in the Theory Guide
specific heat	c_p in Equation 12.70
thermal conductivity	k_p in Equation 12.12
latent heat	H_{lat} in Equation 12.510
vaporization temperature	T_{vap}=T_{bp} in Equation 12.105
volatile component fraction	f_{v0} in Equation 12.106
swelling coefficient	C_{sw} in Equation 12.138
burnout stoichiometric ratio	S_b in Equation 12.145
combustible fraction	f_{comb} in Equation 12.144
heat of reaction for burnout	H_{reac} in Equation 12.159
fraction of reaction heat given to solid	f_h in Equation 12.159
particle emissivity	ε_p in Equation 12.139, Equation 12.159

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Property	Symbol
particle scattering factor	f in Equation 5.73
thermophoretic coefficient	$D_{T,p}$ in Equation 12.11
devolatilization model	
– law 4, constant rate	
– – constant	A_0 in Equation 12.107
– law 4, single rate	
– – pre-exponential factor	A_1 in Equation 12.108
– – activation energy	E in Equation 12.108
– law 4, two rates	
– – pre-exponential factors	A_1, A_2 in Equation 12.111, Equation 12.112
– – activation energies	E_1, E_2 in Equation 12.111, Equation 12.112
– – weighting factors	α_1, α_2 in Equation 12.113
– law 4, CPD	
– – initial fraction of bridges in coal lattice	p_0 in Equation 12.124
– – initial fraction of char bridges	c_0 in Equation 12.123
– – lattice coordination number	$\sigma+1$ in Equation 12.135
– – cluster molecular weight	$M_{w,1}$ in Equation 12.135
– – side chain molecular weight	$M_{w,\delta}$ in

Table 24.4: Property Inputs for Combusting Particles (Law 5)

Property	Symbol
combustion model	
– law 5, diffusion rate	
– – binary diffusivity	$D_{i,m}$ in Equation 12.146 in the Theory Guide
– – diffusivity reference pressure	P_{ref} in Equation 12.91 in the *Fluent Theory Guide*
– law 5, diffusion/kinetic rate	
– – mass diffusion limited rate constant	C_1 in Equation 12.147
– – kinetics limited rate pre-exp. factor	C_2 in Equation 12.148
– – kinetics limited rate activation energy	E in Equation 12.148
– law 5, intrinsic rate	
– – mass diffusion limited rate constant	C_1 in Equation 12.147
– – kinetics limited rate pre-exp. factor	A_i in Equation 12.157
– – kinetics limited rate active energy	E_i in Equation 12.157
– – char porosity	θ in Equation 12.154

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Property	Symbol
– – mean pore radius	$\overline{r}_p$ in Equation 12.156
– – specific internal surface area	A_g in Equation 12.151, Equation 12.153
– – tortuosity	τ in Equation 12.154
– – burning mode	α in Equation 12.158

Table 24.5: Property Inputs for Multicomponent Particles (Law 7)

Property	Symbol
mixture species	selected droplets for components
density	ρ_p in Equation 12.1 of the Theory Guide
specific heat	c_p in Equation 12.168 and Equation 12.294
thermal conductivity	k_p in Equation 12.12 and k_l in Equation 12.295
vapor particle equilibrium	$C_{i,s}$ in Equation 12.162
thermophoretic coefficient	$D_{T,p}$ in Equation 12.11
composition averaging coefficient	α_Y in Equation 12.186
temperature averaging coefficient	α_T in Equation 12.185

24.7.2. Setting Discrete-Phase Physical Properties

24.7.2.1. The Concept of Discrete-Phase Materials

When you create a particle injection and define the initial conditions for the discrete phase (as described in Setting Initial Conditions for the Discrete Phase (p. 2718)), you choose a particular material as the particle's material. All particle streams of that material will have the same physical properties.

Important:

Note that you will not choose a **Material** for a **Massless** particle type in the **Set Injections Properties** dialog box.

Discrete-phase materials are divided into four categories, corresponding to the four types of particles available. These material types are:

- **inert-particle**
- **droplet-particle**
- **combusting-particle**
- **multicomponent-particle**

Each material type will be added to the **Material Type** list in the Create/Edit Materials Dialog Box (p. 4885) when an injection of that type of particle is defined (in the **Set Injection Properties** or **Set Multiple Injection Properties** dialog box, as described in Setting Initial Conditions for the Discrete Phase (p. 2718)). The first time you create an injection of each particle type, you will be able to choose a material from the database, and this will become the default material for that type of

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

particle. That is, if you create another injection of the same type of particle, your selected material will be used for that injection as well. You may choose to modify the predefined properties for your selected particle material, if you want (as described in Modifying Properties of an Existing Material (p. 1601)). If you need only one set of properties for each type of particle, you need not define any new materials; you can simply use the same material for all particles.

Important:

- If you do not find the material you want in the database, you can select a material that is close to the one you want to use, and then modify the properties and give the material a new name, as described in Creating a New Material (p. 1606).
- Note that a discrete-phase material type will not appear in the **Material Type** list in the **Create/Edit Materials** dialog boxes until you have defined an injection of that type of particles. This means, for example, that you cannot define or modify any combusting-particle materials until you have defined a combusting particle injection (as described in Setting Initial Conditions for the Discrete Phase (p. 2718)).

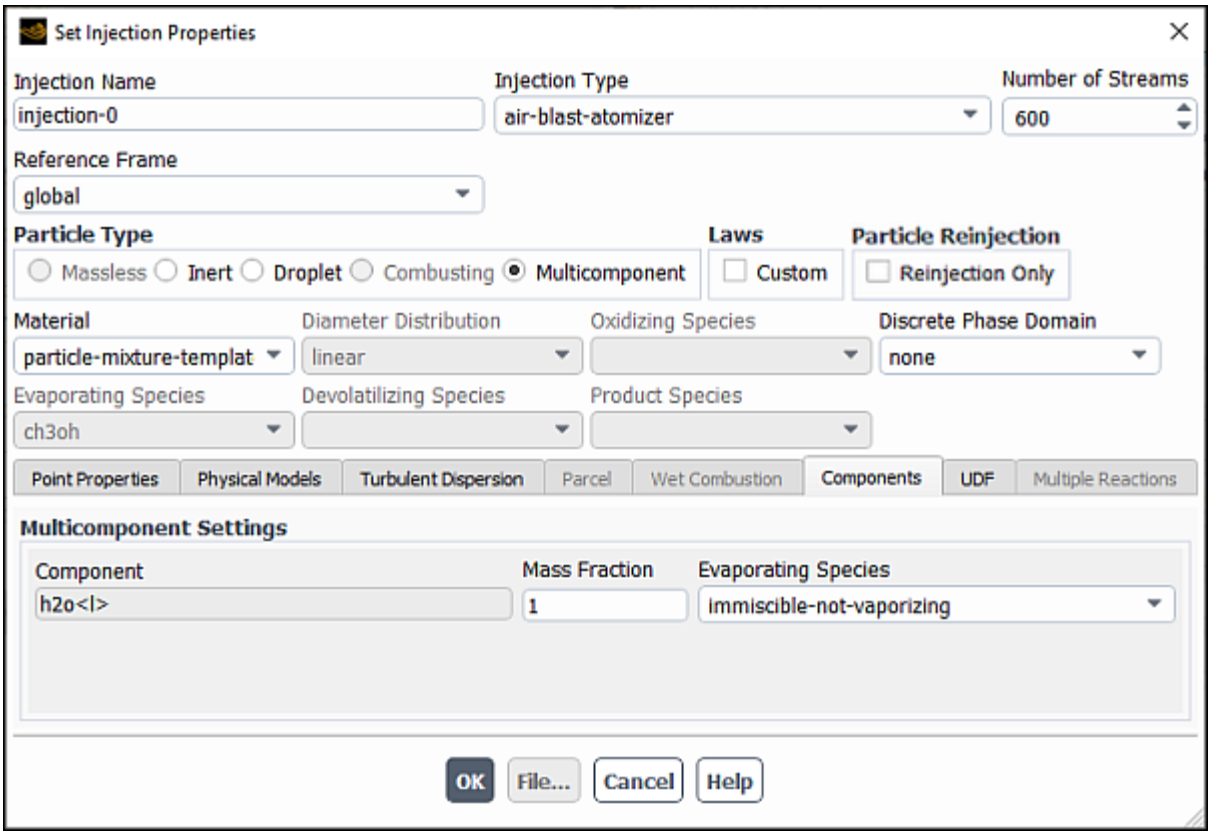
For a **particle-mixture** material type, you must select the species in your mixture. To do this:

1. Click the **Edit...** button next to **Mixture Species** in the **Create/Edit Materials** dialog box (**Properties** group box).
2. In the **Species** dialog box that opens, add your species to the **Selected Species** list.
3. Select **Multicomponent** in the **Set Injection Properties** dialog box (**Particle Type** group box).

   The selected species will now be available, under the **Components** tab (Figure 24.55: The Components Tab (p. 2816)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.55: The Components Tab



24.7.2.1.1. Defining Additional Discrete-Phase Materials

In many cases, a single set of physical properties (density, heat capacity, and so on) is appropriate for each type of discrete phase particle considered in a given model. Sometimes, however, a single model may contain two different types of inert, droplet, combusting particles, or multicomponent particles (for example, heavy particles and gaseous bubbles or two different types of evaporating liquid droplets). In such cases, it is necessary to assign a different set of properties to the two (or more) different types of particles. This is easily accomplished by defining two or more inert, droplet, or combusting particle materials and using the appropriate one for each particle injection.

You can define additional discrete-phase materials either by copying them from the database or by creating them from scratch. See Using the **Create/Edit Materials** Dialog Box (p. 1599) for instructions on using the Create/Edit Materials Dialog Box (p. 4885) to perform these actions.

Important:

Recall that you must define at least one injection (as described in Setting Initial Conditions for the Discrete Phase (p. 2718)) containing particles of a certain type before you will be able to define additional materials for that particle type.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.7.2.2. Description of the Properties

The properties that appear in the Create/Edit Materials Dialog Box (p. 4885) vary depending on the particle type (selected in the **Set Injection Properties** or **Set Multiple Injection Properties** dialog box, as described in Defining Injection Properties (p. 2756) and Defining Properties Common to More than One Injection (p. 2775)) and the physical models you are using in conjunction with the discrete-phase model.

All properties you may need to define for a discrete-phase material are listed below (alphabetically). See Table 24.1: Property Inputs for Inert Particles (p. 2811) – Table 24.4: Property Inputs for Combusting Particles (Law 5) (p. 2813) to see which properties are defined for each type of particle.

Binary Diffusivity

is the mass diffusion coefficient used in the vaporization law (Law 2), $D_{i,m}$ in Equation 12.83 and D_i in Equation 12.276 in the Theory Guide. This input is also used to define the mass diffusion of the oxidizing species to the surface of a combusting particle, $D_{i,m}$, as given in Equation 12.146 in the Theory Guide. (Note that the diffusion coefficient inputs that you supply for the continuous phase are not used for the discrete phase.)

For **Droplet Particle** type **Materials**, select **film-averaged** from the **Binary Diffusivity** drop-down list and specify parameters for the film-averaged model (see Equation 12.89 in the Theory Guide) using the **Film Binary Diffusivity** dialog box. The **film-averaged** model is recommended when accurate temperature-dependent binary diffusivity data are available.

To apply the unity Lewis number model (Equation 12.90 in the *Fluent Theory Guide*) select **unity-lewis-number** from the **Binary Diffusivity** drop-down. The unity Lewis number model is a simplified approach and is not appropriate when the molecular weights of the evaporating species (or oxidizing species for combusting particles) and the gas-phase mixture are very different. The model can be applied, for example, for the evaporation of water, light hydrocarbons, or methanol in air, but is not appropriate for heavy hydrocarbons.

You also have the option of implementing a user-defined function to model the particle binary diffusivity. See `DEFINE_DPM_PROPERTY` in the *Fluent Customization Manual* for more information.

To enable the pressure dependence for the binary diffusivity, in the **Create/Edit Materials** dialog box, under **Properties**, from the **Diffusivity Reference Pressure** drop-down list, select **constant** (see Equation 12.91 in the *Fluent Theory Guide*) and enter a value for the reference pressure P_{ref} that corresponds to the value defined for the **Binary Diffusivity** material property. The pressure dependence option is available with all property methods for the binary diffusivity, including **user-defined**, with the exception of the unity Lewis number model, for which the pressure dependence is already accounted for through the density variable in the model equation.

Boiling Point

is the temperature, T_{bp}, at which the calculation of the boiling rate equation (Equation 12.101 in the Theory Guide) is initiated by Ansys Fluent. When a droplet particle reaches the boiling point, Ansys Fluent applies Law 3 and assumes that the droplet temperature is constant at T_{bp}. The boiling point denotes the temperature at which the particle law transitions from the vaporization law to the boiling law.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For multicomponent particles the boiling point of the components is used only as a reference temperature of the latent heat. Instead, the boiling starts when the sum of the partial component saturation pressures reach the total fluid pressure. The definition of the saturation pressure curve is therefore essential for the boiling of multicomponent particles.

You also have the option of implementing a user-defined function to model the particle boiling point. See `DEFINE_DPM_PROPERTY` in the *Fluent Customization Manual* for more information.

Further information can be found in Defining the Boiling Point and Latent Heat in the *Fluent Theory Guide* and Considering Pressure Dependence in Boiling (p. 2698).

Burnout Stoichiometric Ratio

is the stoichiometric requirement, S_b, for the burnout reaction, Equation 12.145 in the Theory Guide, in terms of mass of oxidant per mass of char in the particle.

Combustible Fraction

is the mass fraction of char, f_{comb}, in the coal particle, that is, the fraction of the initial combusting particle that will react in the surface reaction, Law 5 (Equation 12.144 in the Theory Guide).

Combustion Model

defines which version of the surface char combustion law (Law5) is being used. You can choose from the following options:

- **diffusion-limited** (default)

  The binary diffusivity defined above will be used in Equation 12.146 in the Theory Guide.

- **kinetics/diffusion-limited**

  When the **kinetics/diffusion-limited** limited rate model is selected for the surface combustion model, the Kinetics/Diffusion-Limited Combustion Model Dialog Box (p. 4955) opens where you can enter the **Mass Diffusion Limited Rate Constant** (C_1 in Equation 12.147 in the Theory Guide), **Kinetics Limited Rate Pre-exponential Factor** (C_2 in Equation 12.148), and **Kinetics Limited Rate Activation Energy** (E in Equation 12.148).

  Note that the **Kinetics/Diffusion-Limited Combustion Model** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.

- **intrinsic-model**

  When the **intrinsic-model** is selected for the surface combustion model, the Intrinsic Combustion Model Dialog Box (p. 4955) opens where you can enter the **Mass Diffusion Limited Rate Constant** (C_1 in Equation 12.147 in the Theory Guide), **Kinetics Limited Rate Pre-exponential Factor** (A_i in Equation 12.157), **Kinetics Limited Rate Activation Energy** (E_i in Equation 12.157), **Char Porosity** (θ in Equation 12.154), **Mean Pore Radius** ($\overline{r}_p$ in Equation 12.156), **Specific Internal Surface Area** (A_g in Equation 12.151 Equation 12.153), **Tortuosity** (τ in Equation 12.154), and **Burning Mode, alpha** (α in Equation 12.158).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that the **Intrinsic Combustion Model** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.

- **multiple-surface-reactions**

  When **multiple-surface-reactions** is selected, Ansys Fluent displays the Multiple Surface Reactions Dialog Box (p. 4957) informing you to open the **Reactions** dialog box, where you can review or modify the particle surface reactions that you specified as described in Overview of User Inputs for Modeling Species Transport and Reactions (p. 2356). In addition, you can set property options for the char specific heat and density:

  Composition Dependent Specific Heat

  if this option is enabled, your input for the particle c_p property will be used to determine the specific heat of the volatiles and ash component only. The specific heat of the char will be calculated as a mass-weighted average of the particle surface species specific heat values, and the particle specific heat is finally calculated as the mass average of the char and volatiles+ash fractions. The specific heat of the particle surface species should be defined in the corresponding fluid materials of the Mixture material.

  Composition Dependent Density

  if this option is enabled, your input for the particle Density property will be used to determine the density of the volatiles and ash component. The density of the char will be calculated as a volume-weighted average value of the particle surface species densities, and the particle density is finally calculated as the volume-weighted average of the char and volatiles+ash fractions. The density of the particle surface species should be defined in the corresponding fluid materials of the Mixture material.

  Important:

  If you have not yet defined any particle surface reactions, you must be sure to define them now. See Using the Multiple Surface Reactions Model for Discrete-Phase Particle Combustion (p. 2408) for more information about using the multiple surface reactions model.

  You will notice that the **Burnout Stoichiometric Ratio** and **Heat of Reaction for Burnout** are no longer available in the **Create/Edit Materials** dialog box, as these parameters are now computed from the particle surface reactions you defined in the **Reactions** dialog box.

  Note that the multiple surface reactions model is available only if the **Particle** option for **Reactions** is enabled in the **Species Model** dialog box. See User Inputs for Particle Surface Reactions (p. 2407) for details.

Composition Averaging Coefficient

is the coefficient α_Y in Equation 12.186 in the *Fluent Theory Guide*. To assume bulk gas composition for the physical properties in the particle vaporization and heating rates equations, select **none** from the **Composition Averaging Coefficient** drop-down in the **Create/Edit Materials** dialog box, under **Properties**. To enable property averaging, select **constant** and enter a value

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

between 0 and 1 for the **Composition Averaging Coefficient**. α_Y= 1 corresponds to bulk gas mixture. For α_Y= 0, the composition reverts to the particle surface composition.

Important:

When you define the **Composition Averaging Coefficient** with a **constant** value other than 1, you must also specify **Specific Heat**, **Viscosity** and **Thermal Conductivity** for the evaporating species (in the **Create/Edit Materials** dialog box, under **Properties**, for the corresponding fluid material of the continuous phase mixture). Note that for the definition of the evaporating species properties, the **user-defined** method is not supported and should not be used.

The default and recommended value for the averaging coefficient is 1/3. However, to improve the simulation results, you must provide accurate temperature-dependent data for the vapor material (through the **Create/Edit Materials** dialog box).

Cp

is the specific heat, c_p, of the particle. The specific heat may be defined as a function of temperature by selecting one of the function types from the drop-down list to the right of **Cp**. See Defining Properties Using Temperature-Dependent Functions (p. 1616) for details about temperature-dependent properties. For multicomponent particles, it can be calculated as a mass-weighted value of the specific heat of the droplet component.

You also have the option of implementing a user-defined function to model the particle specific heat. See `DEFINE_DPM_PROPERTY` in the *Fluent Customization Manual* for more information.

A composition-dependent char specific heat option can be enabled if you are using the multiple-surface-reactions model for a combusting particle. For details on enabling this model, see **Combustion Model** (p. 2818).

When you are using the non-premixed or the partially-premixed combustion model in the continuous phase calculation, the specific heat defined for the particle material will be used for the specific heat and enthalpy calculations of the non-volatile/non-reacting particle mass.

Density

is the density of the particulate phase in units of mass per unit volume of the discrete phase. This density is the mass density and not the volumetric density. The density may be defined as a function of temperature by selecting one of the function types from the drop-down list to the right of **Density**. See Defining Properties Using Temperature-Dependent Functions (p. 1616) for details about temperature-dependent properties. For compressible flows, or if any of the real-gas models is enabled, the **compressible-liquid** method is also available. See Compressible Liquid Density Method (p. 1626) for details. For multicomponent particles, it can be calculated as a volume-weighted value of the density of the droplet components.

You also have the option of implementing a user-defined function to model the particle density. See `DEFINE_DPM_PROPERTY` in the *Fluent Customization Manual* for more information.

For a combusting particle that swells during the trajectory calculations, the temperature-dependent density calculation is suspended during the devolatilization law and your input is used to determine the initial particle diameter, $d_{p,0}$ at the start of the devolatilization in Equation 12.138

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

in the *Fluent Theory Guide*. A composition-dependent char density option can be enabled if you are using the multiple-surface-reactions model for a combusting particle. For details on enabling this model, see **Combustion Model** (p. 2818).

Devolatilization Model

defines which version of the devolatilization model, Law 4, is being used.

You can choose the following devolatilization models (as described in Devolatilization (Law 4) in the Theory Guide):

- **constant** (default)

  To use the default constant rate devolatilization model (as described in Equation 12.107 in the Theory Guide), enter the rate constant A_0 in the field below the list.

  For more information, see The Constant Rate Devolatilization Model in the *Fluent Theory Guide*.

- **single rate**

  When the single kinetic rate model is selected, the Single Rate Model Dialog Box (p. 4950) opens where you can enter the **Pre-exponential Factor**, A_1, and the **Activation Energy**, E, to be used in Equation 12.109 in the Theory Guide for the computation of the kinetic rate.

  For more information, see The Single Kinetic Rate Model in the *Fluent Theory Guide*.

- **two-competing-rates**

  When the two competing rates model (**two-competing-rates**) is selected, the Two Competing Rates Model Dialog Box (p. 4952) opens where you can enter, for the **First Rate** and the **Second Rate**, the **Pre-exponential Factor** (A_1 in Equation 12.111 and A_2 in Equation 12.112 in the Theory Guide), **Activation Energy** (E_1 in Equation 12.111 and E_2 in Equation 12.112), and **Weighting Factor** (α_1 and α_2 in Equation 12.113). The constants you specify are used in Equation 12.111 through Equation 12.113.

  For more information, see The Two Competing Rates (Kobayashi) Model in the *Fluent Theory Guide*.

- **cpd-model**

  When the CPD model (**cpd-model**) is selected, the **CPD Model** dialog box opens where you can enter the **Initial Fraction of Bridges in Coal Lattice** (p_0 in Equation 12.124 of the Theory Guide), **Initial Fraction of Char Bridges** (c_0 in Equation 12.123), **Lattice Coordination Number** ($\sigma+1$ in Equation 12.135), **Cluster Molecular Weight** ($M_{w,1}$ in Equation 12.135), and **Side Chain Molecular Weight** ($M_{w,\delta}$ in Equation 12.134).

  For more information, see The CPD Model in the *Fluent Theory Guide*.

Note that the **Single Rate Model**, **Two Competing Rates Model**, and **CPD Model** dialog boxes are modal dialog boxes, which means that you must tend to them immediately before continuing the property definitions.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that by default, the minimum volatiles mass fraction is 0.01. You can lower this value by using the following text command:

```
define/models/dpm/options/lowest-volatiles-mass-fraction
```

Diffusivity Reference Pressure

is the reference pressure P_{ref} for the pressure dependent binary diffusivity in Equation 12.91 in the *Fluent Theory Guide*. To apply the pressure dependence, in the **Create/Edit Materials** dialog box, under **Properties**, from the **Diffusivity Reference Pressure** drop-down list, select **constant** and enter a reference pressure value corresponding to the value for the **Binary Diffusivity** material property. If **none** is selected from the **Diffusivity Reference Pressure** drop-down list, Ansys Fluent assumes no pressure dependence for the binary diffusivity.

Heat of Pyrolysis

is the heat of the instantaneous pyrolysis reaction , h_{pyrol}, that the evaporating/boiling species may undergo when released to the continuous phase. This input represents the conversion of the evaporating species to lighter components during the evaporation process. The heat of pyrolysis should be specified as a positive number for exothermic reaction and as a negative number for endothermic reaction. The default value of zero implies that the heat of pyrolysis is not considered. This input is used in Equation 12.508 in the Theory Guide.

Heat of Reaction for Burnout

is the heat released by the surface char combustion reaction, Law 5 (Equation 12.145 in the Theory Guide). This parameter is specified in terms of heat release (for example, Joules) per unit mass of char consumed in the surface reaction.

Latent Heat

is the latent heat of vaporization, h_{fg}, required for phase change from an evaporating liquid droplet (Equation 12.93, Equation 12.283, Equation 12.284, and Equation 12.301 in the Theory Guide) or for the evolution of volatiles from a combusting particle (Equation 12.139 in the Theory Guide). This input is supplied in units of J/kg in SI units or of Btu/lb_m in British units and is treated as a constant by Ansys Fluent. For the droplet particle, the latent heat value at the boiling point temperature should be used.

You also have the option of implementing a user-defined function to model the particle latent heat. See `DEFINE_DPM_PROPERTY` in the *Fluent Customization Manual* for more information.

Further information can be found in Defining the Boiling Point and Latent Heat in the *Fluent Theory Guide* and Including the Effect of Droplet Temperature on Latent Heat (p. 2700).

React. Heat Fraction Absorbed by Solid

is the parameter f_h (Equation 12.159 in the Theory Guide), which controls the distribution of the heat of reaction between the particle and the continuous phase. The default value of zero implies that the entire heat of reaction is released to the continuous phase.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reaction Model

allows you to select the reaction model for the particle mixture material. You can select either **multicomponent-reactions** or **none** if you do not want to model reactions in particle mixture material. See Multicomponent Particles with Chemical Reactions (p. 2409) for details. This item appears only for cases with reacting multicomponent particles.

Saturation Vapor Pressure

is the saturated vapor pressure, p_{sat}, defined as a function of temperature, which is used in the vaporization law, Law 2 (Equation 12.81 in the Theory Guide). The saturated vapor pressure may be defined as a function of temperature by selecting one of the function types from the drop-down list to the right of its name (see Defining Properties Using Temperature-Dependent Functions (p. 1616) for details about temperature-dependent properties). You may also define the saturated vapor pressure using RGP tables as described in Defining Saturation Properties via RGP Tables (p. 1735). In the case of unrealistic inputs, Ansys Fluent restricts the range of P_{sat} to between 0.0 and the operating pressure. Correct specification of a realistic vapor pressure curve is essential for accurate results from the vaporization model.

You also have the option of implementing a user-defined function to model the particle saturation vapor pressure. See `DEFINE_DPM_PROPERTY` in the *Fluent Customization Manual* for more information.

Swelling Coefficient

is the coefficient C_{sw} in Equation 12.138 in the Theory Guide, which governs the swelling of the coal particle during the devolatilization law, Law 4 (Devolatilization (Law 4) in the Theory Guide). A swelling coefficient of unity (the default) implies that the coal particle stays at constant diameter during the devolatilization process.

You also have the option of implementing a user-defined function to model the particle swelling coefficient. See `DEFINE_DPM_PROPERTY` in the *Fluent Customization Manual* for more information.

Temperature Averaging Coefficient

is the coefficient α_T in Equation 12.185 in the *Fluent Theory Guide*. To assume bulk gas temperature for the physical properties in the particle vaporization and heating rates equations, select **none** from the **Temperature Averaging Coefficient** drop-down in the **Create/Edit Materials** dialog box, under **Properties**. To enable property averaging, select **constant** and enter a value between 0 and 1 for the **Temperature Averaging Coefficient**. α_T= 1 corresponds to bulk gas conditions. For α_T= 0, the temperature reverts to the particle surface temperature.

Important:

When you define the **Temperature Averaging Coefficient** with a **constant** value other than 1, you must also define **Specific Heat**, **Viscosity** and **Thermal Conductivity** for the evaporating species (in the **Create/Edit Materials** dialog box, under **Properties**, for the corresponding fluid material of the continuous phase mixture). Note that for the definition of the evaporating species properties, the **user-defined** method is not supported and should not be used.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The default and recommended value for the averaging coefficient is 1/3. However, to improve the simulation results, you must provide accurate temperature-dependent data for the evaporating vapor material (through the **Create/Edit Materials** dialog box).

Thermal Conductivity

is the thermal conductivity of the particle, k_p. This input is specified in units of W/m-K in SI units or $^{\text{Btu}}/_{\text{ft-h-}^0\text{F}}$ in British units and is treated as a constant by Ansys Fluent.

Thermolysis Model

defines which Thermolysis model is used for the calculation of the mass transfer of the vaporizing species from the droplet to the bulk phase according to a Thermolysis rate equation. You can choose from the following options:

- **single-rate**

  When **single-rate** is selected, the Single Rate Model Dialog Box (p. 4950) opens where you specify the **Pre-exponential Factor** A and the **Activation Energy** E to be used in Equation 12.87 (if you are modeling a droplet material), or the **Pre-exponential Factor** A_i and **Activation Energy** E_i in Equation 12.164 in the *Fluent Theory Guide* (if you are modeling multicomponent particles).

- **secondary-rate**

  (cases with the Lagrangian Wall Film model only) When **secondary-rate** is selected, the Secondary Rate Model Dialog Box (p. 4951) opens where you specify the **Pre-exponential Factor** $A_{particle}$ and **Activation Energy** $E_{particle}$ under the **Particle Thermolysis Rate** and the **Pre-exponential Factor** A_{film} and **Activation Energy** E_{film} under the **Film Thermolysis Rate**. These values will be used in Equation 12.87 in the *Fluent Theory Guide* if you are modeling a droplet material and a film or in Equation 12.164 in the *Fluent Theory Guide* if you are modeling multicomponent particles and a film.

- **constant**

  For the constant rate Thermolysis model, enter the rate constant C to be used in Equation 12.88 (if you are modeling a droplet material), or the rate constant C_i in Equation 12.165 in the *Fluent Theory Guide* (if you are modeling multicomponent particles).

- **none** (default) signifies that the Thermolysis rate model is disabled.

Note that if you are using a multicomponent particle, the Thermolysis model is set for the individual droplet materials. A typical application is the modeling of Selective Catalytic Reduction (SCR) systems where it is recommended to use the single-rate Thermolysis model for the urea-liquid component in the urea-water particle mixture.

Thermophoretic Coefficient

is the coefficient $D_{T,p}$ in Equation 12.11 in the Theory Guide, and appears when the thermophoretic force (which is described in Thermophoretic Force in the Theory Guide) is included in the trajectory calculation (that is, when the **Thermophoretic Force** option is enabled in the **Discrete Phase Model** dialog box). The default is the expression developed by Talbot [158] (p. 5722) (**talbot-diffusion-coeff**) and requires no input from you. You can also define the thermophor-

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

etic coefficient as a function of temperature by selecting one of the function types from the drop-down list to the right of **Thermophoretic Coefficient**. See Defining Properties Using Temperature-Dependent Functions (p. 1616) for details about temperature-dependent properties.

You also have the option of implementing a user-defined function to model the particle thermophoretic coefficient. See `DEFINE_DPM_PROPERTY` in the *Fluent Customization Manual* for more information.

Vaporization Temperature

is the temperature, T_{vap}, at which the calculation of vaporization from a liquid droplet or devolatilization from a combusting particle is initiated by Ansys Fluent. Until the particle temperature reaches T_{vap}, the particle is heated via Law 1, Equation 12.70 in the Theory Guide. This temperature input represents a modeling decision rather than any physical characteristic of the discrete phase.

You also have the option of implementing a user-defined function to model the particle vaporization temperature. See `DEFINE_DPM_PROPERTY` in the *Fluent Customization Manual* for more information.

Vapor-Particle-Equilibrium

is the selected approach for the calculation of the vapor concentration of the components at the surface. This can be Raoult's law (Equation 12.176 in the Theory Guide), the Peng-Robinson real gas model (Equation 12.184 in the Theory Guide), or a user-defined function that defines the equilibrium.

Vaporization/Boiling Model

defines which vaporization/boiling model is used for pure droplets (Law 2) and for multicomponent droplets (Law 7). You can choose from the following options:

- **diffusion-controlled** (default)

  This option will apply Equation 12.80 in the Theory Guide.

- **convection/diffusion-controlled**

  When you select the **convection/diffusion-controlled** model for vaporization, Equation 12.85 in the Theory Guide will be applied for the calculation of the vaporization rate, and Equation 12.97 in the Theory Guide will be applied in the particle heat transfer calculations. This model is recommended when evaporation rates are high. For slowly evaporating droplets both models are expected to give similar results.

  Once you have selected **convection/diffusion-controlled**, you can enable the **Variable Lewis Number Formulation** option in the **Model Options** dialog box to compute the Spalding heat number from Equation 12.96 in the *Fluent Theory Guide*. If the option is disabled Ansys Fluent uses Equation 12.97.

  You can also enable averaging of the Spalding heat transfer term for the convection/diffusion-controlled model according to Equation 12.98 in the *Fluent Theory Guide* by using the TUI command `define/models/dpm/options/vaporization-heat-transfer-averaging`. This option has the net effect of increasing the heat-transfer to the droplet and may

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

produce more realistic results for cases with large temperature differences between droplet and bulk gas.

For both diffusion-controlled and convection/diffusion-controlled models, you can select **Use the Specific Heat of the Evaporating Species in Boiling Law** in the Model Options Dialog Box (p. 4913). This option sets the heat capacity of the gas ($c_{p,\infty}$ in Equation 12.101 in the *Fluent Theory Guide*) to the specific heat of the evaporating species selected in the **Set Injection Properties** dialog box. For the convection/diffusion controlled model with the **Variable Lewis Number Formulation**, this option is selected by default. This ensures consistency with the convection/diffusion controlled model expression for B_T in Equation 12.95 in the *Fluent Theory Guide*.

Volatile Component Fraction

(f_{v0}) is the mass fraction of a droplet particle that may vaporize via Laws 2 and/or 3 (Droplet Vaporization (Law 2) and Droplet Boiling (Law 3) in the *Fluent Theory Guide*). For combusting particles, it is the mass fraction of volatiles that may be evolved via Law 4 (Devolatilization (Law 4) in the Theory Guide).

Note that the non-volatile fraction in the droplet is assumed to be immiscible with the evaporating mass fraction.

When the effect of particles on radiation is enabled (for the P-1 or discrete ordinates radiation model only) in the Discrete Phase Model Dialog Box (p. 4845), you must define the following additional parameters:

Particle Emissivity

is the emissivity of particles in your model, ε_p, used to compute radiation heat transfer to the particles (Equation 12.70, Equation 12.93, Equation 12.104, Equation 12.139, and Equation 12.159 in the Theory Guide) when the P-1 or discrete ordinates radiation model is active. Note that you must enable radiation to particles, using the **Particle Radiation Interaction** option in the Discrete Phase Model Dialog Box (p. 4845). Recommended values of particle emissivity are 1.0 for coal particles and 0.5 for ash [92] (p. 5718).

You also have the option of implementing a user-defined function to model the particle emissivity. See `DEFINE_DPM_PROPERTY` in the *Fluent Customization Manual* for more information.

Particle Scattering Factor

is the scattering factor, f_p, due to particles in the P-1 or discrete ordinates radiation model (Equation 5.73 in the Theory Guide). Note that you must enable particle effects in the radiation model, using the **Particle Radiation Interaction** option in the Discrete Phase Model Dialog Box (p. 4845). The recommended value of f_p for coal combustion modeling is 0.9 [92] (p. 5718).

Note that if the effect of particles on radiation is enabled, scattering in the continuous phase will be ignored in the radiation model.

You also have the option of implementing a user-defined function to model the particle scattering factor. See `DEFINE_DPM_PROPERTY` in the *Fluent Customization Manual* for more information.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When an atomizer injection model and/or the droplet breakup or collision model is enabled in the Set Injection Properties Dialog Box (p. 5608) (atomizers) and/or Discrete Phase Model Dialog Box (p. 4845) (droplet breakup/collision), you must define the following additional parameters:

Droplet Surface Tension

is the droplet surface tension, σ. The surface tension may be defined as a function of temperature by selecting one of the function types from the drop-down list to the right of **Droplet Surface Tension**. See Defining Properties Using Temperature-Dependent Functions (p. 1616) for details about temperature-dependent properties. You also have the option of implementing a user-defined function to model the droplet surface tension. More information about user-defined functions can be found in the Fluent Customization Manual.

Viscosity

is the droplet viscosity, μ_l. The viscosity may be defined as a function of temperature by selecting one of the function types from the drop-down list to the right of **Viscosity**. See Defining Properties Using Temperature-Dependent Functions (p. 1616) for details about temperature-dependent properties. You also have the option of implementing a user-defined function to model the droplet viscosity. More information about user-defined functions can be found in the Fluent Customization Manual.

24.8. Solution Strategies for the Discrete Phase

Solution of the discrete phase implies integration in time of the force balance on the particle (Equation 12.1 in the Theory Guide) to yield the particle trajectory. As the particle is moved along its trajectory, heat and mass transfer between the particle and the continuous phase are also computed via the heat/mass transfer laws (Laws for Heat and Mass Exchange in the Theory Guide). The accuracy of the discrete phase calculation therefore depends on the time accuracy of the integration and upon the appropriate coupling between the discrete and continuous phases when required. Numerical controls are described in Numerics of the Discrete Phase Model (p. 2710). Coupling and performing trajectory calculations are described in Performing Trajectory Calculations (p. 2827). Resetting the Interphase Exchange Terms (p. 2832) and Parallel Processing for the Discrete Phase Model (p. 2888) provide information about resetting interphase exchange terms and using the parallel solver for a discrete phase calculation.

For additional information, see the following sections:

24.8.1. Performing Trajectory Calculations

24.8.2. Resetting the Interphase Exchange Terms

24.8.3. Random Trajectories

24.8.1. Performing Trajectory Calculations

The trajectories of your discrete phase injections are computed when you display the trajectories using graphics or when you perform solution iterations. That is, you can display trajectories without impacting the continuous phase, or you can include their effect on the continuum (termed a coupled calculation). In turbulent flows, trajectories can be based on mean (time-averaged) continuous phase velocities or they can be impacted by instantaneous velocity fluctuations in the fluid. This section describes the procedures and commands you use to perform coupled or uncoupled trajectory calculations, with or without stochastic tracking.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.8.1.1. Uncoupled Calculations

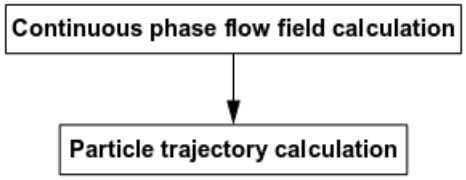
For the uncoupled calculation, you will perform the following two steps:

1. Solve the continuous phase flow field.

2. Plot (and report) the particle trajectories for discrete phase injections of interest.

In the uncoupled approach, this two-step procedure completes the modeling effort, as illustrated in Figure 24.56: Uncoupled Discrete Phase Calculations (p. 2828). The particle trajectories are computed as they are displayed, based on a fixed continuous-phase flow field. Graphical and reporting options are detailed in Postprocessing for the Discrete Phase (p. 2833).

Figure 24.56: Uncoupled Discrete Phase Calculations



This procedure is adequate when the discrete phase is present at a low mass and momentum loading, in which case the continuous phase is not impacted by the presence of the discrete phase.

24.8.1.2. Coupled Calculations

In a coupled two-phase simulation, Ansys Fluent modifies the two-step procedure above as follows:

1. Solve the continuous phase flow field (prior to introduction of the discrete phase).

2. Introduce the discrete phase by calculating the particle trajectories for each discrete phase injection.

3. Recalculate the continuous phase flow, using the interphase exchange of momentum, heat, and mass determined during the previous particle calculation.

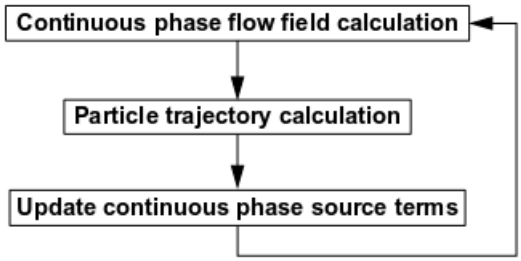
4. Recalculate the discrete phase trajectories in the modified continuous phase flow field.

5. Repeat the previous two steps until a converged solution is achieved in which both the continuous phase flow field and the discrete phase particle trajectories are unchanged with each additional calculation.

This coupled calculation procedure is illustrated in Figure 24.57: Coupled Discrete Phase Calculations (p. 2829). When your Ansys Fluent model includes a high mass and/or momentum loading in the discrete phase, the coupled procedure must be followed in order to include the important impact of the discrete phase on the continuous phase flow field.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.57: Coupled Discrete Phase Calculations



Important:

When you perform coupled calculations, all defined discrete phase injections will be computed. You cannot calculate a subset of the injections you have defined. If there are massless particle injections defined, these will have no effect in the coupled calculation.

24.8.1.2.1. Procedures for a Coupled Two-Phase Flow

If your Ansys Fluent model includes prediction of a coupled two-phase flow, you should begin with a partially (or fully) converged continuous-phase flow field. You will then create your injection(s) and set up the coupled calculation.

For each discrete-phase iteration, Ansys Fluent computes the particle/droplet trajectories and updates the interphase exchange of momentum, heat, and mass in each control volume. These interphase exchange terms then impact the continuous phase when the continuous phase iteration is performed. During the coupled calculation, Ansys Fluent will perform the discrete phase iteration at specified intervals during the continuous-phase calculation. The coupled calculation continues until the continuous phase flow field no longer changes with further calculations (that is, all convergence criteria are satisfied). When convergence is reached, the discrete phase trajectories no longer change either, since changes in the discrete phase trajectories would result in changes in the continuous phase flow field.

The steps for setting up the coupled calculation are as follows:

1. Solve the continuous phase flow field.

2. In the Discrete Phase Model Dialog Box (p. 4845) (Figure 24.1: The Discrete Phase Model Dialog Box - Tracking Tab (p. 2684)), enable the **Interaction with Continuous Phase** option.

3. Set the frequency with which the particle trajectory calculations are introduced in the **DPM Iteration Interval** field. If you set this parameter to 5, for example, a discrete phase iteration will be performed every fifth continuous phase iteration. The optimum number of iterations between trajectory calculations depends upon the physics of your Ansys Fluent model.

   Important:

   Note that if you set this parameter to 0, Ansys Fluent will not perform any discrete phase iterations.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

During the coupled calculation (which you initiate using the Run Calculation Task Page (p. 5207) in the usual manner) you will see the following information in the Ansys Fluent console as the continuous and discrete phase iterations are performed:

```
iter   continuity   x-velocity   y-velocity       k         epsilon       energy    time/it
 314   2.5249e-01   2.8657e-01   1.0533e+00   7.6227e-02   2.9771e-02   9.8181e-03   :00:05
 315   2.7955e-01   2.5867e-01   9.2736e-01   6.4516e-02   2.6545e-02   4.2314e-03   :00:03

  DPM  Iteration  ....
  number  tracked=  9,  number  escaped=  1,  aborted=  0,  trapped=  0,  evaporated  =  8,i
Done.
 316   1.9206e-01   1.1860e-01   6.9573e-01   5.2692e-02   2.3997e-02   2.4532e-03   :00:02
 317   2.0729e-01   3.2982e-02   8.3036e-01   4.1649e-02   2.2111e-02   2.5369e-01   :00:01
 318   3.2820e-01   5.5508e-02   6.0900e-01   5.9018e-02   2.6619e-02   4.0394e-02   :00:00
```

Note that you can perform a discrete phase calculation at any time by using the `solve/dpm-update` text command.

24.8.1.2.2. Stochastic Tracking in Coupled Calculations

If you include the stochastic prediction of turbulent dispersion in the coupled two-phase flow calculations, the number of stochastic tries applied each time the discrete phase trajectories are introduced during coupled calculations will be equal to the **Number of Tries** specified in the Set Injection Properties Dialog Box (p. 5608). For transient particle tracking, the number of tries is set to 1. Input of this parameter is described in Stochastic Tracking (p. 2772). An input of $n \geq 1$ requests n stochastic trajectory calculations for each particle in the injection. When the number of stochastic tracks included is small, you may find that the ensemble average of the trajectories is quite different each time the trajectories are computed. These differences may, in turn, impact the convergence of your coupled solution. For good convergence of a coupled solution, a statistical independent distribution of tracks can be achieved with an adequate number of stochastic tracks and/or a sufficient number of different starting locations of the tracks.

24.8.1.2.3. Under-Relaxation of the Interphase Exchange Terms

When you are coupling the discrete and continuous phases using the calculation procedures noted above, Ansys Fluent applies under-relaxation to the momentum, heat, and mass transfer terms. This under-relaxation serves to increase the stability of the coupled calculation procedure by letting the impact of the discrete phase change only gradually:

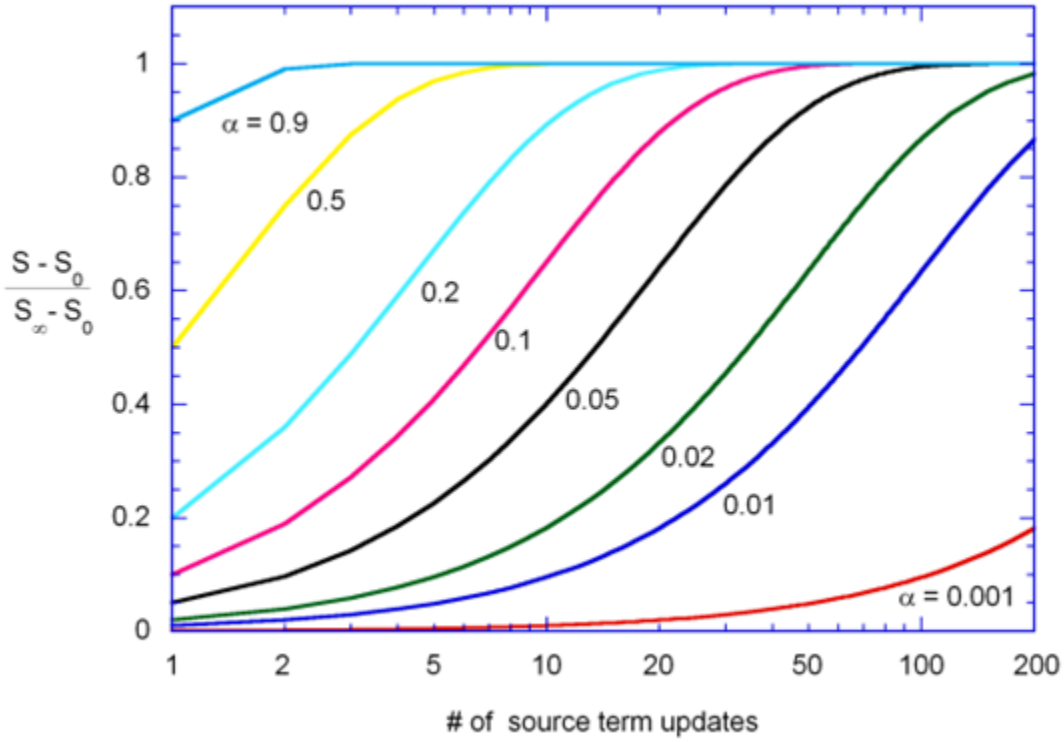
$$E_{new} = E_{old} + \alpha \left(E_{calculated} - E_{old} \right) \tag{24.14}$$

where E_{new} is the exchange term, E_{old} is the previous value, $E_{calculated}$ is the newly computed value, and α is the particle/droplet under-relaxation factor. For α, Ansys Fluent uses a default value of 0.5 for all types of analysis except transient flows with unsteady particle tracking, in which case the default value of 0.9 is used. You can modify α by changing the value in the **Discrete Phase Sources** field under **Under-Relaxation Factors** in the **Solution Controls** task page. You may need to decrease α in order to improve the stability of coupled discrete phase calculations.

Figure 24.58: Effect of Number of Source Term Updates on Source Term Applied to Flow Equations (p. 2831) shows how the source term S, when applied to the flow equations, changes with the number of updates for varying under-relaxation factors α. Each DPM iteration starts with an initial S_0 value for the source term, which is typically zero at the beginning of the calculation. After a number of updates, the source term reaches its final S_∞ value.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.58: Effect of Number of Source Term Updates on Source Term Applied to Flow Equations



Suppose that in a continuous flow simulation with continuous DPM tracking, the DPM under-relaxation factor is chosen to be 0.5, with 20 continuous phase iterations per DPM iteration. From Figure 24.58: Effect of Number of Source Term Updates on Source Term Applied to Flow Equations (p. 2831), we see that approximately 10 source term updates are required for the DPM sources to reach their final values. Therefore, in this example, at least 200 continuous phase iterations are required after any change to the DPM sources (for example, a new injection or a changed DPM mass flow rate), to ensure that the change has taken effect.

For cases with convergence difficulties, reducing the under-relaxation factors is often necessary in order to improve the stability of the coupled DPM simulation. However, for small values of α, many iterations may be needed for the DPM source to reach the final S_∞ value. You can use a text command that linearly ramps up the DPM source term to its maximum S_∞ value. At the end of each iteration i, the source term is computed as:

$$S_i = i\alpha S_{i-1}$$

With this approach, even small values for under-relaxation factors can be applied, and the final S_∞ value can be reached within a reasonable iteration count. For the above example, a minimum of two source term updates is required to reach the final values. Accordingly, it will take only 40 continuous phase iterations to fully account for the effect of the DPM source changes.

To use the linear growth of DPM source term functionality, use the following text commands:

```
define/models/dpm/interaction/linear-growth-of-dpm-source-term?
Change the DPM source term linearly every DPM iteration [no] y
```

Note that the impact of this option is case dependent.

24.8.2. Resetting the Interphase Exchange Terms

If you have performed coupled calculations, resulting in nonzero interphase sources/sinks of momentum, heat, and/or mass that you do not want to include in subsequent calculations, you can reset these sources to zero.

Solution → Initialization → Reset DPM Sources

When you select **Reset DPM Sources**, the sources will immediately be reset to zero without any further confirmation from you.

24.8.3. Random Trajectories

The randomization of particle trajectories is used in the following models for particle injection and tracking:

- random surface injection
- Rosin-Rammler injection diameter distribution
- volume injection
- axial or radial staggering of injections
- atomizer injections
- cone injection direction
- temporal staggering of injections
- random breakup diameters (for example, from the TAB breakup model)
- particle stochastic collision models
- stochastic tracking in turbulent flows
- Brownian motion model

Random number generators used for particle tracking are initialized (that is, seeded on individual particles) and carried with the particles to make results independent of parallel partitioning.

Often there are not enough trajectories to obtain continuous flow sources independent from the seed of the random number generator. Therefore, by default, the initialization of the random number generators is done based on the fluid flow iteration and/or particle time step, thus increasing the variation of starting conditions within the given injection parameters.

You can use the following text commands to modify the default initialization of the random number generators:

- For steady cases, or when computing the same unsteady particle time step multiple times (that is, executing multiple iterations within a flow time step), the initialized injected particles will be repeated with the same random number sequence when you issue the following text command:

  ```
  define/models/dpm/injections/re-randomize-every-iteration?
  Update randomness every iteration? [yes] n
  ```

  This allows to lower the values of the fluid flow residuals because the trajectories and sources will vary by a smaller amount. The downside of this approach is that the result is more dependent on

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

variations in the number of trajectories and on slight changes in the mesh. As an alternative for reducing oscillations in the source terms, the under-relaxation factor of DPM sources can be reduced at the cost of doing more iterations to converge.

- For cases with unsteady particle tracks, you can also use the following text command to inject particles using the same random numbers in each particle time step:

```
define/models/dpm/numerics/re-randomize-every-timestep?
Update randomness every timestep? [yes] n
```

Table 24.6: Randomization of Particle Trajectories in Steady and Unsteady Cases (p. 2833) summarizes the effect of these two settings on cases with unsteady particles.

Table 24.6: Randomization of Particle Trajectories in Steady and Unsteady Cases

Setting / Action	Value / Effect			
`re-randomize-every-iteration`	yes (default)	no	yes	no
`re-randomize-every-timestep`	yes (default)	yes	no	no
Different random number generator initialization when injecting steady particles within the different DPM updates	yes	no	yes	no
Different random number generator initialization when injecting unsteady particles within the same (repeated [a]) particle time step	no	no	yes	no
Same random number generator initialization when injecting unsteady particles in different particle time step	no	no	no, except for different particle time steps within the same DPM update [b]	yes

[a] Particle time steps are tracked repeatedly if more than one DPM update is done within the same flow time step. This happens when the specified **DPM Iteration Interval** is lower than the number of flow iterations actually executed within the flow time step.

[b] This happens when the specified particle time step is smaller than the flow time step.

24.9. Postprocessing for the Discrete Phase

After you have completed your discrete phase inputs and any coupled two-phase calculations of interest, you can display and store the particle trajectory predictions. Ansys Fluent provides both graphical and alphanumeric reporting facilities for the discrete phase, including the following:

- graphical display of the particle trajectories
- summary reports of trajectory fates
- step-by-step reports of the particle position, velocity, temperature, and diameter
- alphanumeric reports and graphical display of the interphase exchange of momentum, heat, and mass

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- optionally, alphanumeric reports and graphical display of various cell-averaged discrete phase field variables
- sampling of trajectories at boundaries and lines/planes
- summary reporting of current particles in the domain
- histograms of trajectory data at sample planes
- display of erosion/accretion rates
- exporting of trajectories to Fieldview and Ensight

This section provides detailed descriptions of each of these postprocessing options.

(Note that plotting or reporting trajectories does not change the source terms.)

For additional information, see the following sections:

24.9.1. Displaying of Trajectories
24.9.2. Particle Tracking Statistics
24.9.3. Summary Reports
24.9.4. Step-by-Step Reporting of Trajectories
24.9.5. Reporting of Current Positions for Unsteady Tracking
24.9.6. Reporting of Interphase Exchange Terms (Discrete Phase Sources)
24.9.7. Reporting of Particle Variables
24.9.8. Reporting of Discrete Phase Variables
24.9.9. Reporting of Unsteady DPM Statistics
24.9.10. Sampling of Trajectories
24.9.11. Histogram Reporting of Samples
24.9.12. Contour Plots of DPM Particle Sampling Results on a Planar Surface
24.9.13. Summary Reporting of Current Particles
24.9.14. Postprocessing of Erosion/Accretion Rates
24.9.15. Assessing the Risk for Solids Deposit Formation During Selective Catalytic Reduction Process

24.9.1. Displaying of Trajectories

When you have defined discrete phase particle injections, as described in Setting Initial Conditions for the Discrete Phase (p. 2718), you can display the trajectories of these discrete particles using the Particle Tracks Dialog Box (p. 5558) (Figure 24.59: The Particle Tracks Dialog Box (p. 2835)).

You can create named particle track definitions and save them for later use. See Creating and Using Contour Plot Definitions (p. 3910) for additional information on graphics object definitions.

*Note that particle track definitions can be included in scenes as long as the **XY Plot** option is not enabled. See Displaying a Scene (p. 3934) for additional information on scenes.*

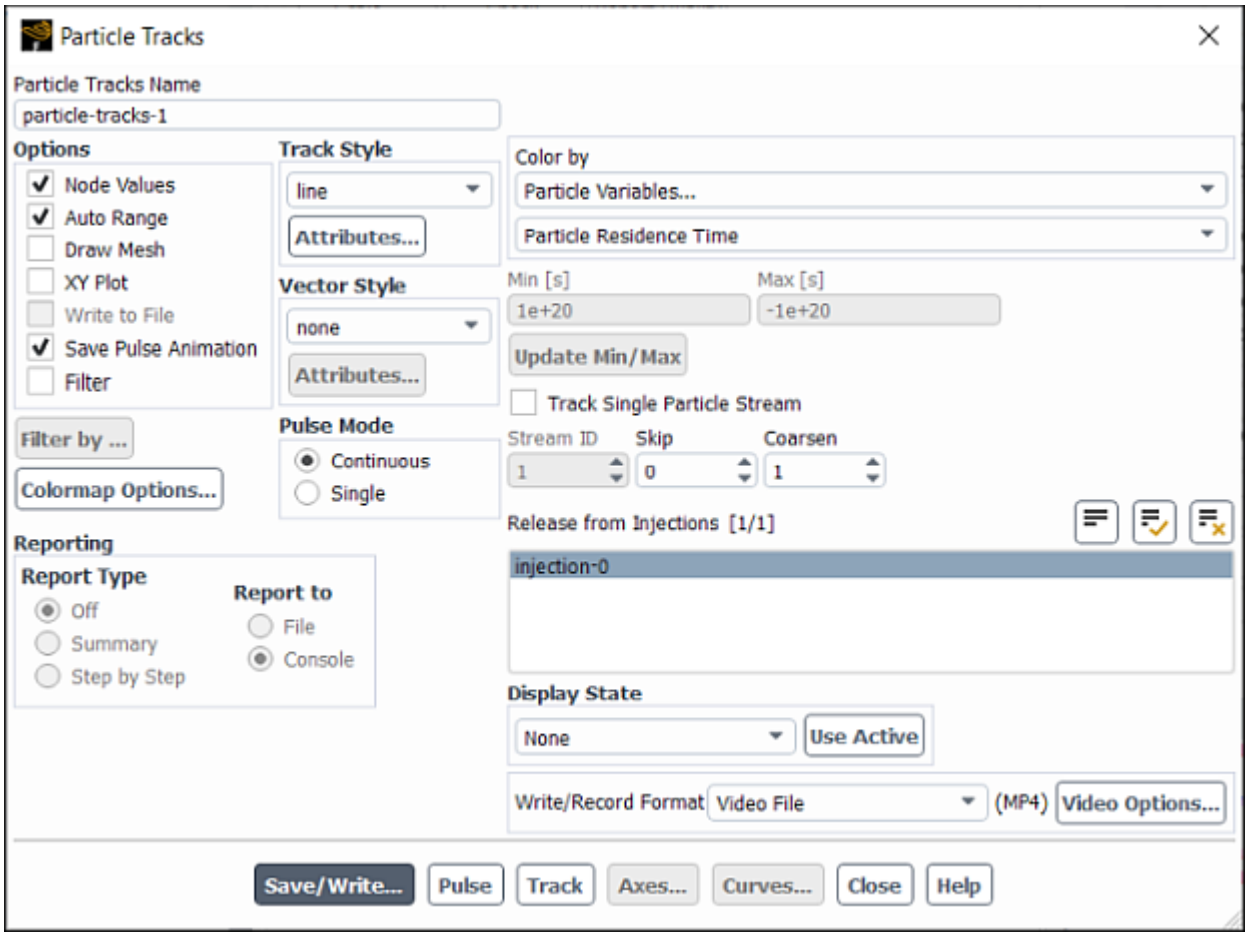
Results → Graphics → Particle Tracks → Edit...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also create a named particle track plot definition and save it for later use.

Results → Graphics → Particle Tracks New...

Figure 24.59: The Particle Tracks Dialog Box



The procedure for drawing trajectories for particle injections is as follows:

1. Select the particle injection(s) you want to track in the **Release from Injections** list. (You can choose to track a specific particle, instead, as described below.)

2. If you have not done this yet, set the **Step Length Factor** and the **Max. Number of Steps** (**Tracking** tab) in the Discrete Phase Model Dialog Box (p. 4845).

   Setup → Models → Discrete Phase Edit...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If stochastic tracking is desired, set the related parameters in the **Set Injection Properties** dialog box, as described in Stochastic Tracking (p. 2772).

Note:

Displaying tracks multiple times at the same solution state will result in identical random tracks. This is because the same seeding is used enabling you to postprocess different variables on the same particle tracks.

3. (optional) Specify which particles you want to display.

   a. If you want to display a single particle stream, enable **Track Single Particle Stream** and specify the particle stream ID number in the **Stream ID** field.

   Note:

   - To determine the stream ID in the injection of interest, use the Injections Dialog Box (p. 5496) to list the injection particle streams, as described in Listing Injection Initial Conditions (p. 2755). The stream ID numbers will be listed in the first column of the data printed in the Ansys Fluent console.
   - For unsteady tracking, all of the particles related to the specified injection stream will be displayed. The displayed data will not correspond to the time history of an individual particle.

   b. (wall film model only) Enable **Free Stream Particles** and/or **Wall Film Particles** if you want to display those types of particles. Note that these options are available only when the Lagrangian wall film model is enabled in the wall boundary conditions dialog box, under the **DPM** tab.

4. Set any of the display options described below.

5. Click the **Display** (or **Save/Display**) button to draw the trajectories or click the **Pulse** button to animate the particle positions. The **Pulse** button will become the **Stop !** button during the animation, and you must click **Stop !** to stop the pulsing.

Note:

Enable **Save Pulse Animation** to save a video or picture files of the pulse animation. When this option is enabled, an additional **Write/Record Format** drop-down list appears that controls what type of file is written when you click **Save/Write...**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A **Continuous** pulse video file will be 5 seconds long, while a **Single** pulse video will be one full cycle.

Important:

- For unsteady particle tracking simulations, clicking **Display** will show only the current location of the particles. Typically, you should select **point** in the **Track Style** drop-down list when displaying transient particle locations since individual positions will be displayed.
- The **Pulse** button option is not available for unsteady tracking.

6. (Optional) Associate a display state with this particle tracks plot for a consistent appearance each time the particle tracks are redisplayed.

   a. Set the display state. *There are two approaches*:

      - Setup the graphics window with your preferred model orientation, zoom level, graphics effects, ruler, and so on, and click **Use Active** to create a display state with these properties.
      - Select a display state from the **Display State** drop-down list (if you already created a display state that you would like to reuse for other graphics objects).

   b. Click **Save/Display** to link the selected display state with this particle tracks plot.

   Refer to Controlling the Display State and Modifying the View (p. 4025) for additional information.

Note:

If you experience an uncharacteristic delay in the displaying of particle tracks (with **Track Style** set to **line** and/or **Vector Style** set to **vector**), disabling the **Hover-Over Probe Values** option in **Preferences** (located in the **Appearance** branch under **Selections**) should restore the display performance.

24.9.1.1. Options for Particle Trajectory Plots

You can include the mesh in the trajectory display, control the style of the trajectories (including the twisting of ribbon-style trajectories), color them by different scalar fields and control the color scale, and coarsen trajectory plots. (More information about these options can be found in Controlling the Particle Tracking Style (p. 2838) and Controlling the Vector Style of Particle Tracks (p. 2841).) You can also choose node or cell values for display. If you are "pulsing" the trajectories, you can control the pulse mode and save an animation of the pulse. Finally, you can generate an XY plot of the

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

particle trajectory data (for example, residence time) as a function of time or path length and save this XY plot data to a file.

Note:

The **Draw Mesh** option settings are not saved with the particle track plot definition. Once the dialog box is closed these settings will revert to being disabled. If you want these settings persisted within the current session, you can use the non-persistent **Particle Tracks** dialog box.

Results → **Graphics** → **Particle Tracks** → **New...**

Plotting particle trajectories can be very time consuming, therefore, to reduce the plotting time, a coarsening factor can be used to reduce the number of points that are plotted. Providing a coarsening factor of n, will result in each n^{th} point being plotted for a given trajectory in any cell. This coarsening factor is specified in the Particle Tracks Dialog Box (p. 5558), in the **Coarsen** field and is only valid for steady-state cases. For example, if the coarsening factor is set to 2, then Ansys Fluent will plot alternate points.

Important:

Note that if any particle or pathline enters a new cell, this point will always be plotted.

To reduce plotting time in transient cases, Ansys Fluent has available an option to skip plotting every n^{th} particle in an injection. Selecting this option is also done in the Particle Tracks Dialog Box (p. 5558) by specifying a nonzero integer in the **Skip** field. For example, if an individual stream is selected and the skip option is set to 1, every other particle will be plotted. If the entire injection is selected with a skip option of 1, every other particle will be plotted for all streams in the injection.

These options are controlled in exactly the same way that pathline-plotting options are controlled. See Options for Pathline Plots (p. 3924) for details about setting the trajectory plotting options mentioned above.

Note that in addition to coloring the trajectories by continuous phase variables, you can also color them according to the discrete phase variables summarized in Reporting of Particle Variables (p. 2861). To display the minimum and maximum values in the domain, click the **Update Min/Max** button.

24.9.1.2. Controlling the Particle Tracking Style

Particle tracking can be displayed as lines (with or without arrows), ribbons, cylinders (coarse, medium, or fine), triangles, spheres, refined spheres, or a set of points. In the **Track Style** drop-down list in the **Particle Tracks** dialog box, you can choose:

- (steady tracking) **line**, **line-arrows**, **point**, **sphere**, **refined-sphere**, **ribbon**, **triangle**, **coarse-cylinder**, **medium-cylinder**, or **fine-cylinder**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- (unsteady tracking) **point**, **sphere**, or **refined-sphere**.

  Important:

  - The main difference between the **sphere**, and the **refined-sphere** option is that the **refined-sphere** is a higher quality sphere that has better performance (frames per second). The performance decreases when the diameter of the sphere is too large (for example, 10 or more).
  - Reflections and shadows are not shown for the **refined-sphere** style.

  Pulsing can be done only on **point**, **sphere**, **refined-sphere**, or **line** styles.

  Once you have selected the track style, click the **Attributes...** button to specify how you would like to display the particle tracks.

- If you are using the **line** or **line-arrows** style, set the **Line Width** in the **Track Style Attributes** dialog box (Figure 24.60: The Track Style Attributes Dialog Box (p. 2839)) that appears when you click the **Attributes...** button. For **line-arrows** you will also set the **Spacing Factor**, which controls the spacing between the particles tracks. The size of the arrow heads can be adjusted by entering a value in the **Scale** text-entry box.

  Figure 24.60: The Track Style Attributes Dialog Box

- If you are using the **point** style, you will set the **Marker Size** in the **Track Style Attributes** dialog box. The thickness of the particle track will be the thickness of the marker.
- If you are using the **sphere** style, you will set the **Diameter**, **scale**, and **Detail** in the **Particle Sphere Style Attributes** dialog box (Figure 24.61: The Particle Sphere Style Attributes Dialog Box (p. 2840)). You have the option of specifying a constant diameter if you enable **Constant** under **Options** and you will then specify the **Diameter**. If you enable **Variable**, you can select a particle variable to estimate the size of the spheres. The spheres are scaled by the factor entered in the **Scale** entry box.

  The best constant diameter to use will depend on the dimensions of the domain, the view, and the particle density. However, an adequate starting point would be a diameter on the order of

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

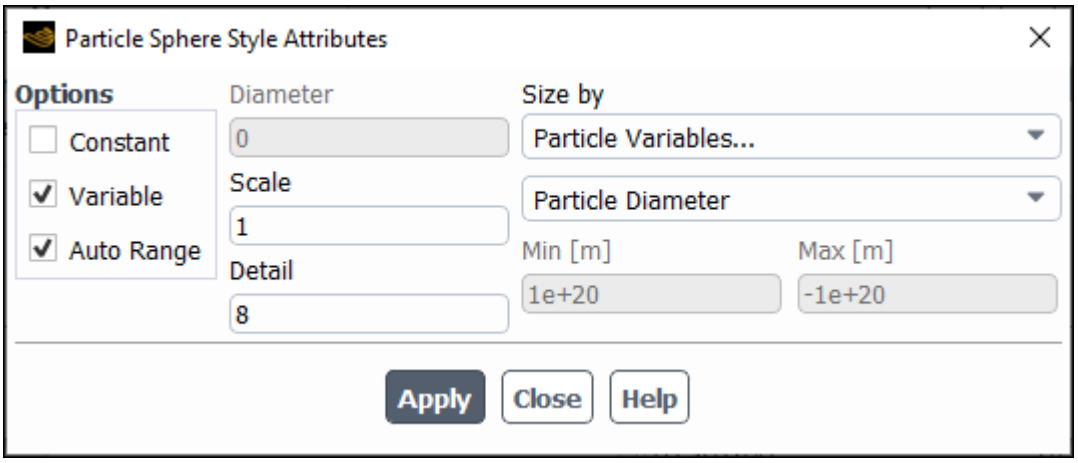
1/4 of the average cell size or 1/4 step size. Units for the **Diameter** field correspond to the mesh dimensional units.

The level of detail applied to the graphical rendering of the spheres can be controlled using the **Detail** field. The level of detail uses integer values ranging from 4 to 50. Note that the performance of the graphical rendering as well as the memory consumption will be better when using a small level of detail, that is, very coarse spheres, such as 6 or 8. The rendering performance significantly decreases with higher levels of detail. You should gradually increase the detail to determine the best-case scenario between performance and quality.

Whenever **Auto Range** is disabled, the spheres are displayed only if they have values between **Min** and **Max**.

Also note that to take full advantage of spherical rendering, lighting should be turned on in the view. The Gouraud setting provides much smoother looking spheres than the Flat setting and better performance than the Phong setting. For more information on lighting, see Adding Lights (p. 3984).

Figure 24.61: The Particle Sphere Style Attributes Dialog Box



- If you are using the **refined-sphere**, you can set the **Diameter** and the **Scale** in Figure 24.61: The Particle Sphere Style Attributes Dialog Box (p. 2840).
- If you are using the **triangle** or any of the **cylinder** styles, you will set the **Width** in the **Track Style Attributes** dialog box. For triangles, the specified value will be half the width of the triangle's base, and for cylinders, the value will be the cylinder's radius.
- If you are using the **ribbon** style, clicking on the **Attributes...** button will open the Ribbon Attributes Dialog Box (p. 5243), in which you can set the ribbon's **Width**. You can also specify parameters for twisting the ribbon tracks. In the **Twist By** drop-down list, you can select a scalar field on which the tracks twisting is based (for example, helicity). Select the desired category in the upper list and then select a related quantity in the lower list. The twisting may not be displayed smoothly because the scalar field by which you are twisting the tracks is calculated at cell centers only (and not interpolated to a particle's position). The **Twist Scale** sets the amount of twist for the selected scalar field. To magnify the twist for a field with very little change, increase this factor; to display less twist for a field with dramatic changes, decrease this factor.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When you click **Compute**, the **Min** and **Max** fields will be updated to show the range of the **Twist By** scalar field.

24.9.1.3. Controlling the Vector Style of Particle Tracks

You can choose to have the particle tracks displayed as vectors. Choose the **Vector Style** from the drop-down list in the **Particle Tracks** dialog box:

- If you select **vector**, the vector will be generated starting in the center of the particle, as shown in Figure 24.62: Particles with the Vector Style (p. 2841).
- If you select **centered-vector**, the midpoint of the vector will appear in the center of the particle, as shown in Figure 24.63: Particles with the Centered Vector Style (p. 2842).
- If you select **centered-cylinder**, the midpoint of the cylinder will appear in the center of the particle, as shown in Figure 24.64: Particles with the Centered Cylinder Style (p. 2843).

Figure 24.62: Particles with the Vector Style

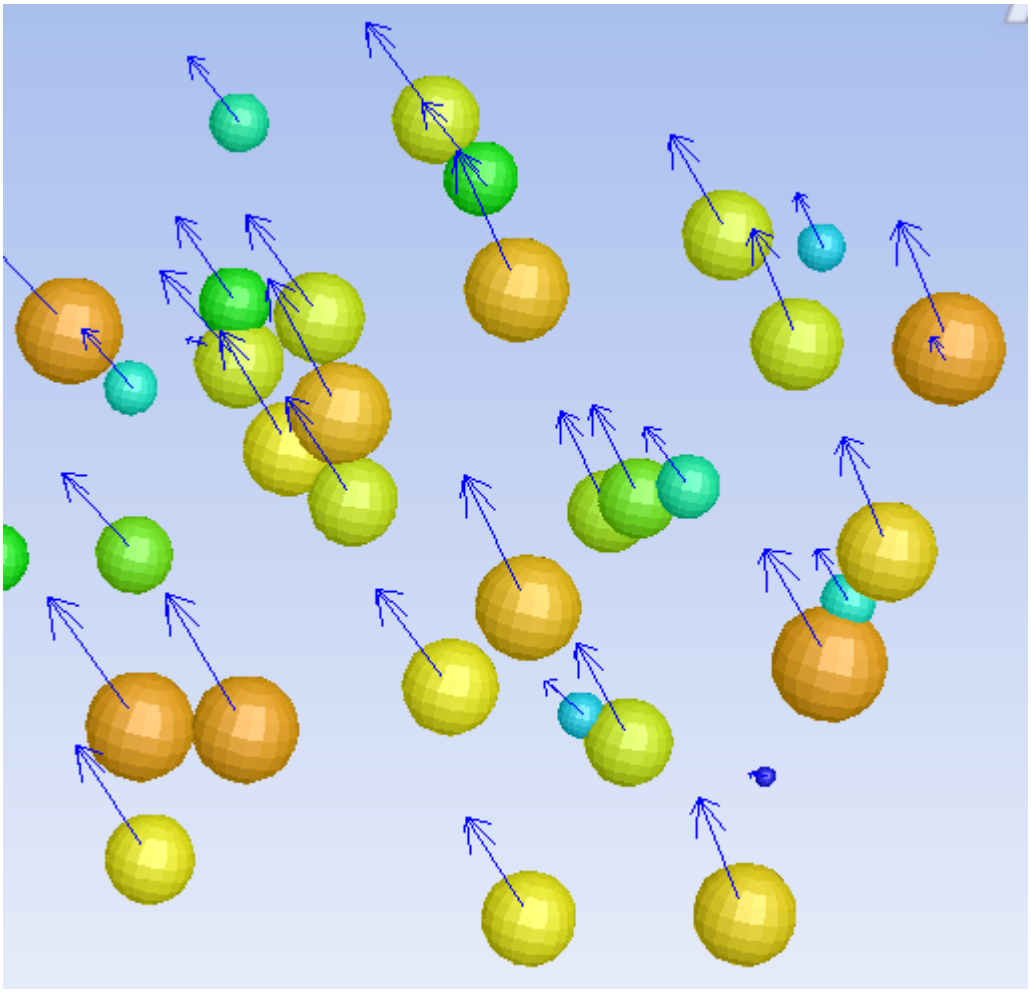
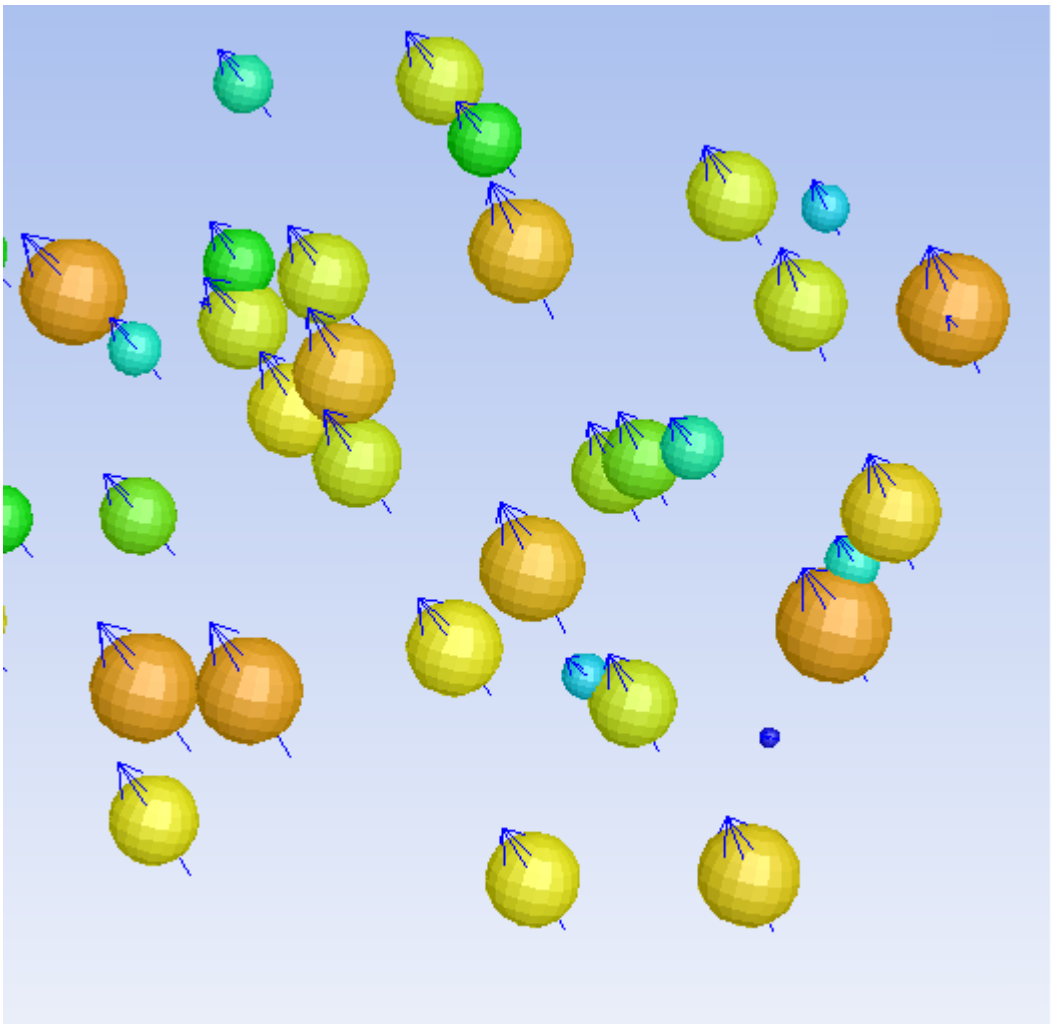
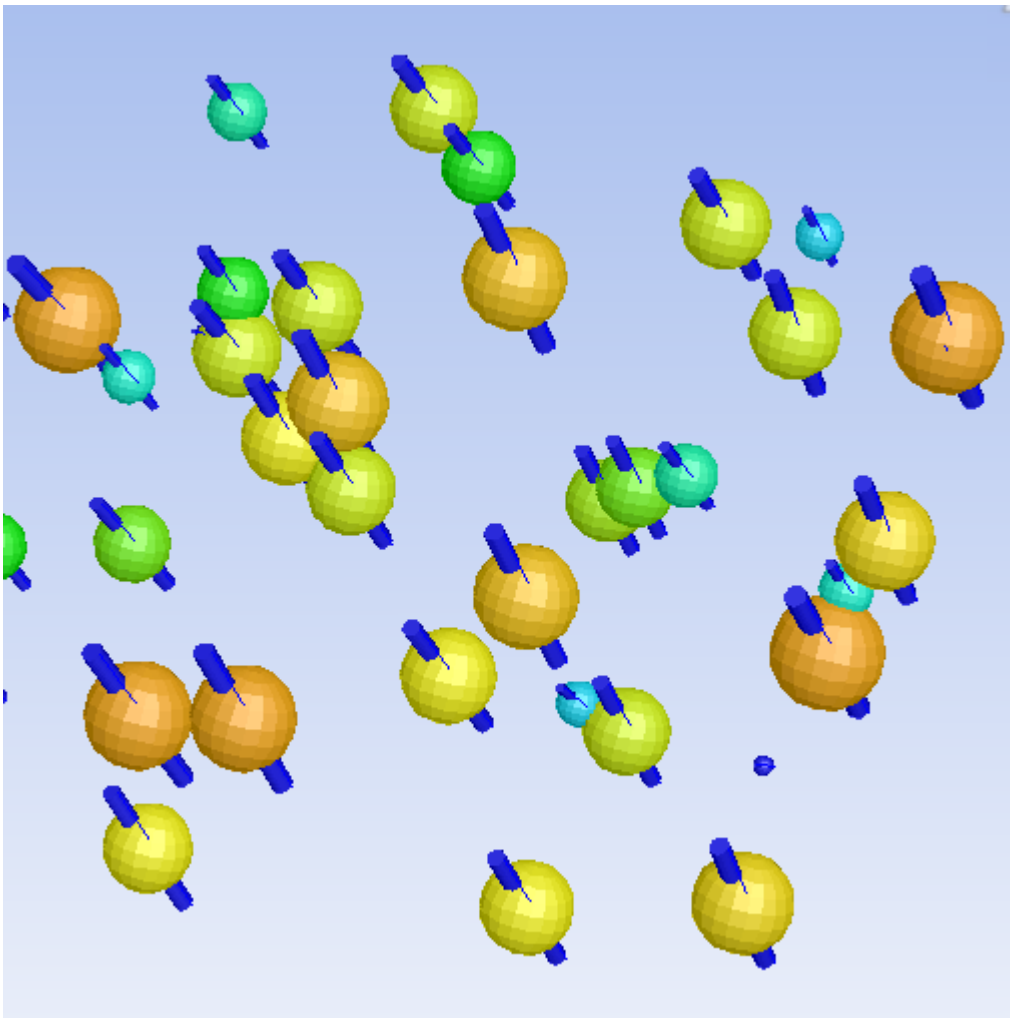


Figure 24.63: Particles with the Centered Vector Style



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.64: Particles with the Centered Cylinder Style

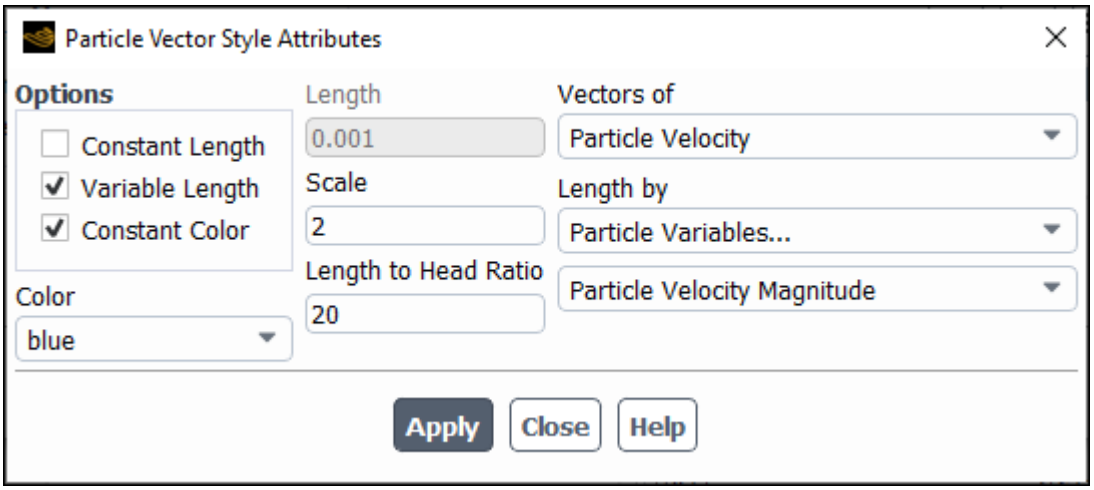


Click the **Attributes...** button to specify how you would like to display the particle tracks. In the **Particle Vector Style Attributes** dialog box (Figure 24.65: The Particle Vector Style Attributes Dialog Box (p. 2844)) you will set the **Length**, **Scale**, and **Length to Head Ratio**. The direction of the vectors is displayed for the selected variable under **Vectors of**. You have the option of specifying a **Constant Length** or a **Variable Length**, which is based on the variable selected under **Length by**. If **Constant Color** is enabled, then all vectors/cylinders are colored by the color selected in the **Color** drop-down list. Otherwise, it is the color selected in the **Particle Tracks** dialog box (seen in the **Mesh Colors** dialog box when **Draw Mesh** is enabled).

Vectors can be scaled by the factor given in the **Scale** entry box. The ratio of vector length to vector head size can be changed in the box **Length to Head Ratio**. In the case of a cylinder, the ratio of the length to the diameter is affected.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.65: The Particle Vector Style Attributes Dialog Box

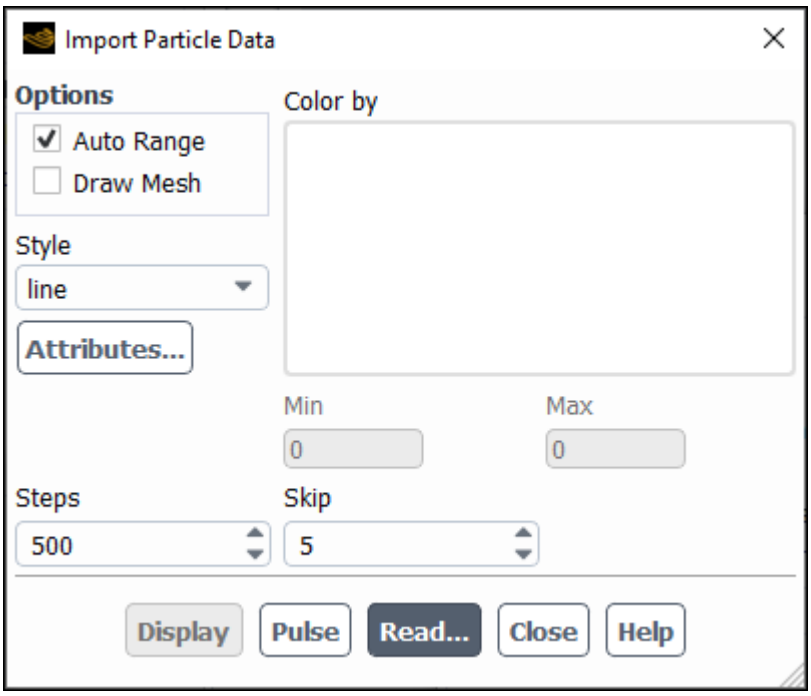


24.9.1.4. Importing Particle Data

For transient simulations, you can use the **Import Particle Data** dialog box (Figure 24.66: The Import Particle Data Dialog Box (p. 2844)) to import particle data to display in the graphics window.

Results → Model Specific → Discrete Phase → Import Particle Data...

Figure 24.66: The Import Particle Data Dialog Box



1. Click **Read...** to display a file selection dialog box where you can enter a file name and a directory that contains the imported data.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Choose from the available import options by selecting **Auto Range** and/or **Draw Mesh** under **Options**. If you prefer to restrict the range of the scalar field, disable the **Auto Range** option and set the **Min** and **Max** values manually beneath the **Color by** list.

3. Choose to color the particle pathlines by any of the scalar fields in the **Color by** list. If you select **COLORBY**, the pathlines will be colored by the quantity that was chosen when the particle data file was created. (See Exporting Steady-State Particle History Data (p. 999))

4. Select a pathline style under **Style**. To set pathline style attributes, click the **Attributes...** button. For more information about the pathline style types, see Controlling the Pathline Style (p. 3925).

5. The value of **Steps** sets the maximum number of steps a particle can advance. A particle will stop when it has traveled this number of steps or when it leaves the domain.

6. If your pathline plot is difficult to understand because there are too many paths displayed, you can "thin out" the pathlines by changing the **Skip** value.

7. Click the **Display** button to draw the pathlines, or click the **Pulse** button to animate the particle positions. The **Pulse** button will become the **Stop !** button during the animation, and you must click **Stop !** to stop the pulsing.

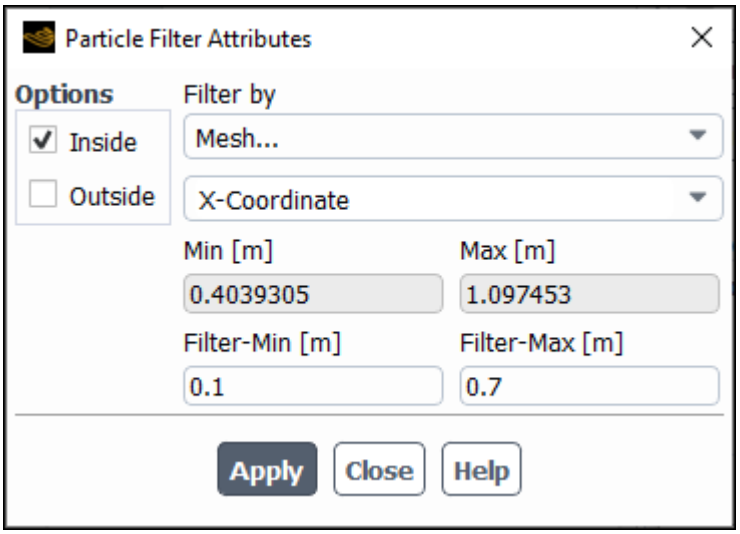
24.9.1.5. Particle Filtering

You can specify how you would like to filter the particles being displayed, by first enabling the **Filter** option in the Options group of the **Particle Tracks** dialog box, then clicking the **Filter by...** button. In the **Particle Filter Attributes** dialog box, select the field variable by which you want to filter, then specify whether you would like to display all the particle tracks **Inside** or **Outside** the **Filter-Min** and **Filter-Max** range, as shown in Figure 24.67: The Particle Filter Attributes Dialog Box (p. 2845).

Note:

All particle variables as well as any field variable except for **Custom Field Functions...** can be used as a filter variable.

Figure 24.67: The Particle Filter Attributes Dialog Box



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.9.1.6. Graphical Display for Axisymmetric Geometries

For axisymmetric problems in which the particle has a nonzero circumferential velocity component, the trajectory of an individual particle is often a spiral about the centerline of rotation. Ansys Fluent displays the r and x components of the trajectory (but not the θ component) projected in the axisymmetric plane.

24.9.2. Particle Tracking Statistics

During two-way coupled DPM calculations or when you do graphical postprocessing of steady particle trajectories (as described in Displaying of Trajectories (p. 2834)), Ansys Fluent prints tracking statistics in the console. For example:

```
DPM Iteration ....
 number tracked = 7, escaped = 4, aborted = 0, trapped = 0, evaporated = 3
Done.
```

The above numbers report events that the numerical parcels may have undergone. They should not be taken for any physically meaningful information since numerical parcels may represent different amounts (particle counts) of the particulate phase.

The description of particle statuses can be found in Trajectory Fates (p. 2848).

By default, only those events that have occurred at least once are reported. This can be changed using the `define/models/dpm/numerics/tracking-statistics` text command.

You can also track particles through the domain without displaying the trajectories by clicking the **Track** button at the bottom of the **Particle Tracks** dialog box. This allows the listing of reports without also displaying the tracks.

24.9.3. Summary Reports

You can request additional detail about the trajectory fates as the particles exit the domain, including the mass flow rates through each boundary zone, mass flow rate of evaporated droplets, and composition of the particles.

1. Follow steps 1 and 2 in Displaying of Trajectories (p. 2834) for displaying trajectories.
2. Select **Summary** as the **Report Type** and click **Display** or **Track**.

> **Important:**
>
> For steady-state simulations, DPM summary data is not stored in the `.dat` file, since it is possible to track particles on single or combinations of injections. Transient simulations store this data since it is accumulated over time starting from initialization.

A detailed report similar to the following example will appear in the console window. (You may also choose to write this report to a file by selecting **File** as the **Report to** option, clicking the **Write...** button (which was originally the **Display** button), and specifying a file name for the summary report file in The Select File Dialog Box (p. 931).)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
number  tracked  =  10,  escaped  =  3,  aborted  =  0,  trapped  =  5,  evaporated  =  2,

Fate          Zone Zone Number                 Elapsed Time (s)     Injection, Index Injection, Index
              Name   Id             Min       Max       Avg   Std Dev             Min             Max
---------- ------- ---- ----- --------- --------- --------- --------- ----------- --- ----------- ----
Evaporated                2 1.770e-03 1.114e-02 6.456e-03 4.686e-03 injection-0   2 injection-0   76
Escaped    outflow    6     3 6.043e-01 7.037e-01 6.471e-01 4.172e-02 injection-1   4 injection-1   72
Trapped       wall    7     5 8.486e-03 1.767e-01 5.030e-02 6.421e-02 injection-1   8 injection-1   75

                                  (*)- Mass Transfer Summary -(*)

Fate           Zone   Zone       Mass Flow (kg/s)
               Name     Id    Initial      Final     Change
---------- ------- ------ ---------- ---------- ----------
Evaporated                  8.333e-02  0.000e+00 -8.333e-02
Escaped     outflow      6  1.167e-01  5.144e-02 -6.523e-02
Trapped        wall      7  2.000e-01  2.400e-02 -1.760e-01
----------                ---------- ---------- ----------
Net                         4.000e-01  7.544e-02 -3.246e-01

                                  (*)- Energy Transfer Summary -(*)

Fate           Zone   Zone      Heat Rate (W)          Change of Heat (W)
               Name     Id    Initial      Final   Sensible     Latent      Total
---------- ------- ------ ---------- ---------- ---------- ---------- ----------
Evaporated                 -3.180e+04  0.000e+00 -3.382e+02  3.214e+04      1.107
Escaped     outflow      6  5.272e+05  6.519e+05 -3.487e+03  1.282e+05      1.523
Trapped        wall      7  4.954e+05  6.993e+05 -1.173e+03  2.051e+05      1.737
----------                ---------- ---------- ---------- ---------- ----------
Net                         9.908e+05  1.351e+06 -4.998e+03  3.654e+05      4.367

                                  (*)- Combusting Particles -(*)

Fate           Zone   Zone   Volatile Content (kg/s)        Char Content (kg/s)
               Name     Id    Initial      Final   %Conv    Initial      Final   %Conv
---------- ------- ------ ---------- ---------- ------- ---------- ---------- -------
Evaporated                  0.000e+00  0.000e+00    0.00  0.000e+00  0.000e+00     100
Escaped     outflow      6  9.333e-03  9.333e-03    0.00  2.133e-02  2.133e-02       0
Trapped        wall      7  9.333e-03  7.485e-01  100.00  2.133e-02  2.133e-02      98
----------                ---------- ---------- ------- ---------- ---------- -------
Net                         1.867e-02  9.333e-03  50.00  4.267e-02  4.267e-02      198

                                  (*)- Multicomponent Droplet -(*)

Fate           Zone   Zone               Species      Species Content (kg/s)
               Name     Id                 Names    Initial      Final   %Conv
---------- ------- ------ -------------------- ---------- ---------- -------
Evaporated                    c5h12-droplet<l>  1.667e-02  0.000e+00  100.00
Evaporated                    c7h16-droplet<l>  3.333e-02  0.000e+00  100.00
Evaporated                             h2o<l>  0.000e+00  0.000e+00    0.00
Escaped    outflow       6    c5h12-droplet<l>  1.667e-02  2.585e-04   98.45
Escaped    outflow       6    c7h16-droplet<l>  0.000e+00  0.000e+00    0.00
Escaped    outflow       6             h2o<l>  3.333e-02  1.134e-02   65.99
Trapped       wall       7    c5h12-droplet<l>  3.333e-02  0.000e+00  100.00
Trapped       wall       7    c7h16-droplet<l>  3.333e-02  0.000e+00  100.00
Trapped       wall       7             h2o<l>  3.333e-02  0.000e+00  100.00
```

The report groups together particles with each possible fate, and reports the number of particles, the time elapsed during trajectories, and the mass and energy transfer This information can be very useful for obtaining information such as where particles are escaping from the domain, where particles are colliding with surfaces, and the extent of heat and mass transfer to/from the particles within the domain. Additional information is reported for combusting particles and multicomponent particles.

For cases with multiple injections, note the following:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you select only one injection from the **Release from Injections** list in the **Sample Trajectories** dialog box, available information is reported for that injection. There may not be any information available for individual flow boundaries unless you issued the text command `/report/dpm-zone-summaries-per-injection?` prior to the particle tracking.
- If you select more than one injection, Fluent produces the DPM summary report for all injections, even if only a subset was selected from the **Release from Injections** list.

Note that for cases with unsteady particle tracking, information about particle parcels still residing within the computational domain is only reported if you use the **Display** button (not the **Track** button). If you require this information but are not interested in the graphical display, you can use the **Display** button but set the **Skip** entry to a very large number and make sure **Draw Mesh** is cleared.

For details, see the following sections:

24.9.3.1. Trajectory Fates
24.9.3.2. Elapsed Time
24.9.3.3. Mass Transfer Summary
24.9.3.4. Energy Transfer Summary
24.9.3.5. Heat Rate and Energy Reporting
24.9.3.6. Combusting Particles
24.9.3.7. Combusting Particles with the Multiple Surface Reaction Model
24.9.3.8. Multicomponent Particles
24.9.3.9. Reinjected Particles
24.9.3.10. Evaporated Mass

24.9.3.1. Trajectory Fates

The possible fates for a particle trajectory are as follows:

- "Escaped" trajectories are those that terminate at a flow boundary for which the "escape" condition is set.
- "Incomplete" trajectories are those that were terminated when the maximum allowed number of time steps — as defined by the **Max. Number of Steps** input in the Discrete Phase Model Dialog Box (p. 4845) (see Numerics of the Discrete Phase Model (p. 2710)) — was exceeded.
- "Incomplete_parallel" may appear as an additional fate for parallel simulations. This means that the number of particle exchanges between partitions has been exceeded. Any remaining particles on the compute nodes are stopped, which is indicated by the number following this fate. Therefore no further source terms from these particles are considered. The number of particle exchanges is limited to avoid very long computational time due to incomplete particles. You can change the default value of 1000 to a value of 20000 with a scheme command. Contact the technical support engineer for this information.
- "Trapped" trajectories are those that terminate at a flow boundary where the "trap" condition has been set.
- "Evaporated" trajectories include those trajectories along which the particles were evaporated within the domain.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- “Aborted” trajectories are particles that have become trapped at interior faces (DPM particles) or edges (Lagrangian wall-film particles) due to discontinuities in the forces on the particle. Make sure you enable interpolation for the flow density, viscosity, wall-film properties, and so on. This may reduce the number of aborted particles if these properties vary spatially.
- “Shed” trajectories are newly generated particles during the breakup of a larger droplet. They appear only if a breakup model is enabled.
- “Coalesced” trajectories are removed particles which have coalesced after particle-particle collisions. They appear only if the coalescence model is enabled.
- “Splashed” trajectories are particles that are newly generated when a particle touches a wall film. Those trajectories appear only if the wall film model is enabled.
- (DPM summary reports only) “Injected” trajectories are sums over all particles injected.

If **Unsteady Particle Tracking** is enabled, the following additional fates may be reported:

- "Transformed" are particle parcels that have been replaced by other parcels (compare with the "Inserted" fate below) or by a representation of the same amount of fluid in the Eulerian formulation.
- "Inserted" are particle parcels that have been added to the simulation by any way other than from an injection. These can include secondary break-up, condensation, any kind of film interaction and model transition between, for example, Eulerian VOF and DPM.
- "In Fluid" particle parcels are free-stream particle parcels that are still travelling in the domain at the current point in time and have not reached any final destiny yet.
- "In Film" particle parcels are Lagrangian wall-film particle parcels that are still inside the computational domain at the current point in time and have not reached any final destiny yet.
- "filmrelease" particle parcels are Lagrangian wall-film particle parcels that are released into the free stream but not assigned a particle diameter based on a physical model (such as stripping or separation). If many such events are reported, you may need to improve the case setup for best possible predictions. This fate may appear in the tracking statistics (see Particle Tracking Statistics (p. 2846)).

In DPM summary reports, "Escaped" and "Trapped" fates are reported separately for every flow boundary. Note that this is not true if there are multiple injections in the case, and the DPM summary report is produced for a single injection (as opposed to a single report for all injections). To record trajectory fates per individual flow boundary for every injection separately, you can use the text command `/report/dpm-zone-summaries-per-injection?` prior to the particle tracking. In a case with unsteady particle tracking, you need to issue this command before the first particle is injected. All other fates are always recorded for every injection separately.

24.9.3.2. Elapsed Time

The number of particles with each fate is listed under the `Number` heading. (Particles that escape through different zones or are trapped at different zones are considered to have different fates, and are therefore listed separately.) The minimum, maximum, and average time elapsed during the trajectories of these particles, as well as the standard deviation about the average time, are listed in the `Min`, `Max`, `Avg`, and `Std Dev` columns. This information indicates how much time the

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

particle(s) spent in the domain before they escaped, aborted, evaporated, or were trapped. Also, on the right side of the report, the injection name and index of the trajectories with the minimum and maximum elapsed times. (You may need to use the scroll bar to view this information.)

Fate	Zone	Zone Number		Elapsed Time (s)				Injection, Index		Injection, Index	
		Name	Id	Min	Max	Avg	Std Dev		Min		Max
Incomplete			2	1.485e+01	2.410e+01	1.947e+01	4.623e+00	injection-0	1	injection-0	0
Escaped	outflow	7	8	4.940e+00	2.196e+01	1.226e+01	4.871e+00	injection-0	9	injection-0	2

24.9.3.3. Mass Transfer Summary

For all droplet or combusting particles with each fate, the total initial and final mass flow rates and the change in mass flow rate are reported in the `Initial`, `Final`, and `Change` columns. With this information, you can determine how much mass was transferred to the continuous phase from the particles.

For unsteady tracking, the report lists the time-integrated mass flow rate of the particle streams that have reached a particular fate at the current flow time. In other words, the report does not include particles that are still being tracked in the domain.

```
                          (*)- Mass Transfer Summary -(*)

Fate              Zone   Zone       Mass Flow (kg/s)
                  Name     Id    Initial      Final     Change
----------- ---------- ------ ---------- ---------- ----------
Incomplete                     1.388e-03  1.943e-04  1.194e-03
Escaped        outflow      7  1.683e-01  7.124e-02 -9.709e-02
```

24.9.3.4. Energy Transfer Summary

This report tells you how much heat was transferred from the particles to the continuous phase. The report is organized in two sections. For steady simulations, there is a `Heat Rate` and a `Change of Heat` section. For unsteady particle tracking, there is an `Energy` and a `Change of Energy` section. The `Heat Rate` and `Energy` sections are the same for all particle types, while the other sections report the change of heat due to the various transfer processes, which differ for each particle type. For steady simulations, the report lists the rate and the change of heat for the particle streams organized according to the particle stream fates. For unsteady tracking, the report lists the time integrated heat rate and change of the particle streams that have reached a particular fate at the current flow time. Note that the report does not include particles that are still being tracked in the domain.

24.9.3.5. Heat Rate and Energy Reporting

For all particles with each fate, the total initial and final heat content are reported in the `Initial` and `Final` columns. The particle heat content H_p is defined as follows:

Inert Particles:

$$H_p = m_p \int_{T_{ref}}^{T_p} C_{p_p} dT \tag{24.15}$$

where:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

m_p = mass flow rate of particles (kg/s)

T_p = temperature of particles (K)

C_{p_p} = heat capacity of particles (J/kg/K)

T_{ref} = reference temperature for enthalpy (K)

Droplet Particles:

$$H_p = m_p[f_v(-H_{lat_{ref}} + H_{pyrol}) + \int_{T_{ref}}^{T_p} C_{p_p} dT] \tag{24.16}$$

where:

H_{pyrol} = heat of pyrolysis (J/kg)

$H_{lat_{ref}}$ = latent heat of evaporation at reference conditions (J/kg)

The latent heat at the reference conditions $H_{lat_{ref}}$ is defined in Equation 12.509 in the Theory Guide.

Combusting Particles:

$$H_p = H_w + H_{dry} \tag{24.17}$$

H_w is the heat content of the evaporating/boiling liquid material if Wet Combustion is selected (otherwise H_w = 0).

$$H_w = f_w m_p \left[\left(-H_{lat_{ref}} + H_{pyrol} \right) + \int_{T_{ref}}^{T_p} C_{p_p} dT \right] \tag{24.18}$$

where:

f_w is the mass fraction of the liquid in the combusting particle

$H_{lat_{ref}}$, H_{pyrol}, and C_{p_p} are properties of the evaporating liquid material

H_{dry} is the heat content of the dry combusting particle and is calculated as

$$H_{dry} = \left(1 - f_w\right) m_p \left[f_v \left(-H_{lat_{ref}} \right) + f_{comb} H_{comb} + \int_{T_{ref}}^{T_p} C_{p_p} dT \right] \tag{24.19}$$

where:

f_v is the volatile fraction

f_{comb} is the combustible fraction

Important:

The `Heat Rate` section of the report is not provided for the multiple surface reactions model.

Multicomponent Particles:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$H_p = m_p \sum_i \left(y_i H_{p_i} \right) \tag{24.20}$$

where:

y_i = mass fraction of component i in particle

H_{p_i} = heat content of component i

and

$$H_{p_i} = \left[-H_{lat_{ref_i}} + H_{pyrol_i} + \int_{T_{ref}}^{T_p} C_{p_{p_i}} dT \right] \tag{24.21}$$

where:

H_{pyrol_i} = heat of pyrolysis for component i (J/kg)

$H_{lat_{ref_i}}$ = latent heat of evaporation at reference conditions for component i (J/kg)

$C_{p_{p_i}}$ = specific heat of component i (J/kg/K)

24.9.3.5.1. Change of Heat and Change of Energy Reporting

This section reports the total heat transferred from the particle to the continuous phase and is analyzed in components of `Sensible` heat, `Latent` heat and heat of `Reaction`. The `Total` change reported equals the difference between the `Initial` and `Final` states of the particle streams. The sensible heat component is reported for all particle types, the latent heat for the droplet, combusting and multicomponent particle, while the heat of reaction is reported for the combusting particle type only. A positive `Change of Heat` denotes that heat is expelled from the continuous phase and absorbed by the particle, while a negative `Change of Heat` denotes heat is released by the particle to the continuous phase.

Steady and Transient Simulations

For steady simulations the report lists the heat rate H_p, while for unsteady tracking the time integrated energy E_p from time 0 to current flow time ts is reported.

$$E_p = \int_0^{ts} H_P(t) dt \tag{24.22}$$

Below is an example of an `Energy Transfer Summary` report for evaporating droplets:

```
                                (*)- Energy Transfer Summary -(*)

Fate               Zone   Zone         Heat Rate (W)             Change of Heat (W)
                   Name     Id     Initial       Final    Sensible      Latent       Total
----------- ------------ ------ ---------- ----------- ----------- ----------- -----------
Evaporated                      -4.530e+04   0.000e+00  -4.750e+02   4.577e+04   4.530e+04
Escaped          utflow      6  -2.339e+05  -1.559e+05  -7.085e+03   8.505e+04   7.797e+04
Trapped            wall      7  -2.176e+05   0.000e+00  -1.058e+03   2.187e+05   2.176e+05
-----------                     ---------- ----------- ----------- ----------- -----------
Net                             -4.353e+05  -4.670e+04  -4.085e+03   3.927e+05   3.886e+05
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Below is an example of an Energy Transfer Summary report for combusting particles:

```
  (*)- Energy Transfer Summary -(*)

Fate           Zone   Zone      Heat Rate (W)          Change of Heat (W)
               Name     Id    Initial      Final   Sensible     Latent   Reaction      Total
---------- ------- ------ ---------- ---------- ---------- ---------- ---------- ----------
Escaped        wall      5  1.697e+05  2.555e+04  3.166e+03  1.034e+00 -1.473e+05      -1.44
Trapped     outflow      6  1.886e+04  1.938e+04  5.731e+02  1.149e-01 -5.370e+01       5.19
----------              ---------- ---------- ---------- ---------- ---------- ----------
Net                        1.886e+05  4.493e+04  3.739e+03  1.149e+00 -1.474e+05      -1.43
```

Important:

In a coupled calculation, for all types of steady flows, the Total Net Change of Heat reported in the Energy Transfer Summary should balance with the opposite of the Sum over all fluid cells of the DPM Sensible Enthalpy Source. If this is not the case, this means that the coupled discrete-continuous phase calculation has not converged, and more DPM phase iterations are required. For more information on coupled calculations, see Performing Trajectory Calculations (p. 2827).

```
                              Sum
  DPM  Sensible  Enthalpy  Source                        (w)
 ------------------------------       --------------------
                           fluid-1                -388937.41
```

24.9.3.6. Combusting Particles

If combusting particles are present, Ansys Fluent will include additional reporting on the volatiles and char converted. These reports are intended to help you identify the composition of the combusting particles as they exit the computational domain.

```
                (*)-  Combusting  Particles  -(*)
Fate          Zone   Zone   Volatile Content (kg/s)        Char Content (kg/s)
              Name     Id    Initial      Final   %Conv    Initial      Final   %Conv
---------- ------ ------ ---------- ---------- ------- ---------- ---------- -------
Incomplete                6.247e-04  0.000e+00  100.00  5.691e-04  0.000e+00  100.00
Escaped    outflow      7  6.758e-04  0.000e+00  100.00  6.158e-04  3.782e-05   93.86
```

The total volatile content at the start and end of the trajectory is reported in the Initial and Final columns under Volatile Content. The percentage of volatiles that has been devolatilized is reported in the %Conv column.

The total reactive portion (char) at the start and end of the trajectory is reported in the Initial and Final columns under Char Content. The percentage of char that reacted is reported in the %Conv column.

24.9.3.7. Combusting Particles with the Multiple Surface Reaction Model

If the multiple surface reaction model is used with combusting particles, Ansys Fluent will include additional reporting on the mass of the individual solid species that constitute the particle mass.

```
(*)- Multiple Surface Reactions -(*)

 Fate         Zone   Zone   Species      Species Content (kg/s)
              Name     Id     Names    Initial      Final   %Conv
```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
--------- ------- ------ --------- ---------- ---------- -------
Escaped   outflow      6      c<s>  6.080e-02  1.487e-06  100.00
Escaped   outflow      6      s<s>  3.200e-03  5.077e-06   99.84
Escaped   outflow      6       cao   0.000e+00  1.153e-03   0.00
Escaped   outflow      6     caso4  9.266e-04  7.776e-03    0.00
Escaped   outflow      6     caco3  8.000e-03  5.260e-03   34.25
```

The total mass of each solid species in the particles at the start and end of the trajectory is reported in the `Initial` and `Final` columns, respectively. The percentage of each species that is reacted is reported in the `%Conv` column. Note that for the solid reaction products (for example, if the mass of a solid species has increased in the particle), the conversion is reported to be 0.

24.9.3.8. Multicomponent Particles

If your simulation includes multicomponent particles, Ansys Fluent generates an additional report for the particle components.

```
                                          (*)- Multicomponent Droplet -(*)

Fate           Zone   Zone              Species   Species Content (kg/s)
               Name     Id                Names    Initial      Final   %Conv
----------- ------- ------ -------------------- ---------- ---------- -------
Evaporated                    c5h12-droplet<l>  1.667e-02  0.000e+00  100.00
Evaporated                    c7h16-droplet<l>  3.333e-02  0.000e+00  100.00
Evaporated                              h2o<l>  0.000e+00  0.000e+00    0.00
Escaped     outflow       6   c5h12-droplet<l>  1.667e-02  2.585e-04   98.45
Escaped     outflow       6   c7h16-droplet<l>  0.000e+00  0.000e+00    0.00
Escaped     outflow       6             h2o<l>  3.333e-02  1.134e-02   65.99
Trapped        wall       7   c5h12-droplet<l>  3.333e-02  0.000e+00  100.00
Trapped        wall       7   c7h16-droplet<l>  3.333e-02  0.000e+00  100.00
Trapped        wall       7             h2o<l>  3.333e-02  0.000e+00  100.00
```

24.9.3.9. Reinjected Particles

Reinjection events are not reported in the tracking statistics (the line that begins with “number tracked = ” printed after every DPM iteration).

In DPM summary reports, a separate fate called "Reinject" lists particles that have reached a domain boundary for which the DPM boundary condition type has been set to **reinject**.

To match the "Net" value, reinjected particles are, in addition, reported with the "Inserted" fate.

Note:

For injection-specific DPM summary reports, note the following:

- The "Reinject" fate is reported for the initial injection that introduced the particle into the domain for the very first time.
- The "Inserted" fate is reported for the injection from which position and velocity data were obtained.

Therefore, where particle reinjection is involved, "Net" and “Injected+Inserted” will not match in a summary report for a single injection.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Summary Reporting When a UDF is Used

When a `DEFINE_DPM_BC` user-defined function (UDF) similar to the one shown in Example 5 in the *Fluent Customization Manual* is used, particles that reach the domain boundary in question will be reported under a fate called "BC-UDF". Similar to the "Reinject" fate, this will be listed for the injection that initially contained that particle.

You can use the "Inserted" fate for individual injections to find out which fraction of the particles was sent to which reinjection location by the UDF.

24.9.3.10. Evaporated Mass

For simulations that involve both evaporating liquid droplets and Lagrangian wall film, information about the fractions of total vapor from the free-stream particles and from evaporating film can be included in the extended summary report as follows:

1. Prior to running your simulation, enable the collection of detailed information about DPM evaporated mass by issuing the following text command:

   ```
   report/calc-exchange-data-on-zone-types
   Available options: ("none" "lagr-wall-film-zones" "cell-zones" "all-zones")
   Details about evaporated mass to be given in DPM summary reports? ["all-zones"]
   ```

   At the prompt shown above, you can specify for which zone types the information about DPM evaporated mass should be collected.

2. (cases with multiple injections only) If you want to generate a DPM summary report for a single injection, enable the collection of more detailed information using the `report/dpm-zone-summaries-per-injection?` text command.

3. Run a simulation.

4. Create an extended discrete phase summary report by using the `report/dpm-extended-summary` text command.

   Additional lines appear at the bottom of the summaries for mass transfer, energy transfer, and multicomponent droplet.

   An example of such lines with information on evaporated mass added to the mass transfer summary report is shown below.

   ```
            (*)- Mass Transfer Summary -(*)

    Fate                        Zone   Zone          Mass (kg)
                                Name     Id    Initial      Final     Change
    ----------- -------------------- ------ ---------- ---------- ----------
   ...
   ...
   ...
    -----------                             ---------- ---------- ----------
    Film                                     0.000e+00 -4.967e-08 -4.967e-08
    Freestream                               0.000e+00 -1.651e-06 -1.651e-06
    Film                          top      4  0.000e+00 -2.947e-09 -2.947e-09
    Film                       bottom      6  0.000e+00 -4.672e-08 -4.672e-08
    Freestr.                    fluid      2  0.000e+00 -1.651e-06 -1.651e-06
   ```

The first two lines report the amount of mass that has evaporated from all liquid wall films (`Film`) and from liquid droplets travelling freely in the surrounding gas (`Freestream`), respectively.

Below these lines, additional lines break down those two categories for the individual fluid zones (for free-stream droplets) and wall zones (for liquid wall film), respectively.

Note:

If you do not want to include this information in the DPM summary report, you can use the following text command to suppress printing the extra lines without disabling the collection and storage of the solution data:

```
report> enable-exch-details-in-dpm-summ-rep?
Details about exchange per zone to be given in DPM summary reports? [yes] no
```

24.9.4. Step-by-Step Reporting of Trajectories

At times, you may want to obtain a detailed, step-by-step report of the particle trajectory/trajectories. Such reports can be obtained in alphanumeric format. This capability allows you to monitor the particle position, velocity, temperature, or diameter as the trajectory proceeds.

The procedure for generating files containing step-by-step reports is listed below:

1. Follow steps 1 and 2 in Displaying of Trajectories (p. 2834) for displaying trajectories. You may want to track only one particle stream at a time, using the **Track Single Particle Stream** option.
2. Select **Step by Step** as the **Report Type**.

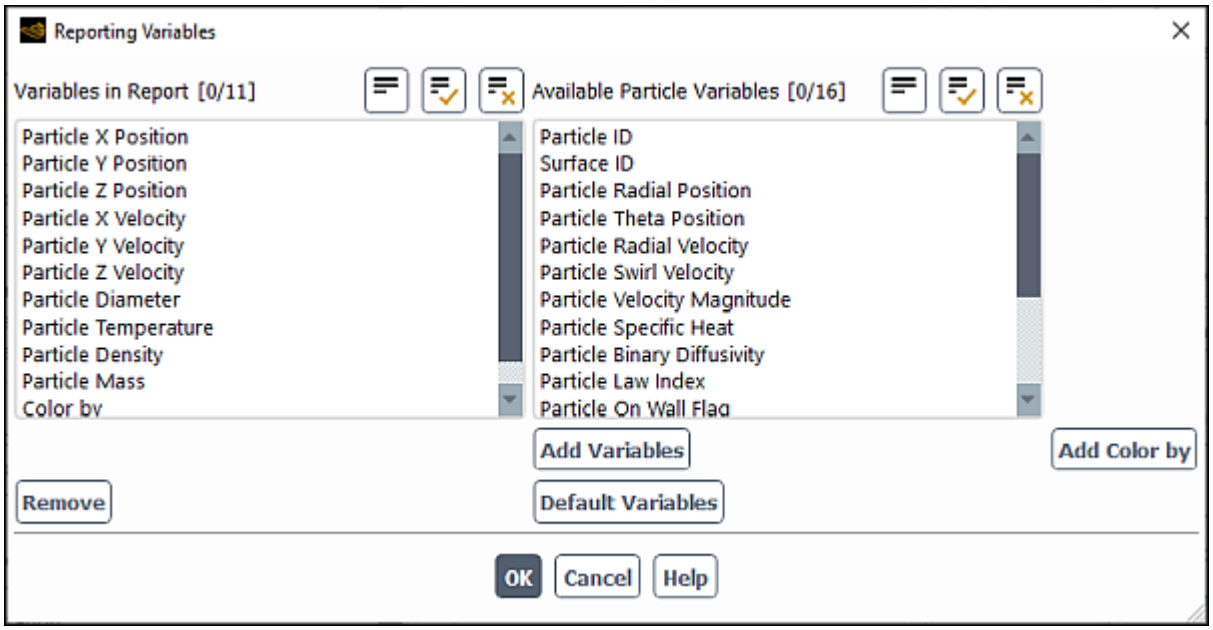
   Important:

   This option is only available for steady-state cases. For transient cases, see Reporting of Current Positions for Unsteady Tracking (p. 2858).

3. Select **File** as the **Report to** option. (The **Track** button will become the **Write...** button.)
4. In the **Significant Figures** field, enter the number of significant figures to be used in the step-by-step report.
5. Click the **Reporting Variables...** button.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. **Figure 24.68: The Reporting Variables Dialog Box**



7. In the **Reporting Variables** dialog box (Figure 24.68: The Reporting Variables Dialog Box (p. 2857)), you can change the variables in the report. The list under **Variables in Report** contains all variables currently reported. The list under **Particle Variables** contains the particle variables that are available for you to select. You can use the following buttons to modify the **Variables in Report** list:

   - **Remove**: Removes selected variables from that list.
   - **Default Variables**: Restores the default list.
   - **Add Variable**: Adds selections to the report using.
   - **Add Color by**: Adds the **Color by** variable to the list of **Variables in Report**, which is the only way to get cell values or customized field functions into the report.

   Important:

   Note that it is possible to select one variable in the **Reporting Variables** dialog box and a variable from the **Color by** drop-down list in the **Particle Tracks** dialog box, however, each variable is reported only once.

8. Click the **Write...** button and specify a file name for the step-by-step report file in The Select File Dialog Box (p. 931).

A detailed report similar to the following example will be saved to the specified file before the trajectories are plotted. (You may also choose to print the report in the console by choosing **Console** as the **Report to** option and clicking **Display** or **Track**, but the report is very long that it is unlikely to be of use to you in that form.)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
 FILE  TYPE: 1
 COLUMNS:  11
 TITLE: TRACK  HISTORY

 COLUMN  TYPE  VARIABLE                        (UNITS)
 ------  ----  --------                       -------
      1     2  ParticleResidenceTime          (s)
      2    10  ParticleXPosition              (m)
      3    10  ParticleYPosition              (m)
      4    10  ParticleZPosition              (m)
      5    10  ParticleXVelocity              (m/s)
      6    10  ParticleYVelocity              (m/s)
      7    10  ParticleZVelocity              (m/s)
      8    10  ParticleDiameter               (m)
      9    10  ParticleTemperature            (K)
     10    10  ParticleDensity                (kg/m3)
     11    10  ParticleMass                   (kg)
-------------------------------------------

  0.00000e+00   5.00000e-02   5.00000e-02   5.00000e-02   2.00000e+01   0.0000  .  .  .
  1.07087e-07   5.00000e-02   5.00000e-02   5.00000e-02   1.23339e+01   2.6696  .  .  .
  2.51617e-07   5.00000e-02   5.00000e-02   5.00000e-02   1.04417e+01   3.3286  .  .  .
 .              .             .             .             .             .
 .              .             .             .             .             .
 .              .             .             .             .             .
```

The default step-by-step report lists the position, velocity, diameter, temperature, density and mass of the particle at selected time steps along the trajectory. In addition, the variable you have selected in the **Color by** list is also included. If you choose **Console** as the **Report to** option, the variable names are written as the header of each column. (You may need to use the scroll bar to view all variables in this column.)

```
Time       X-Position   Y-Position   Z-Position    X-Velocity   Y-Velocity    Z-Veloc
0.000e+00  1.000e-03    3.120e-02    0.000e+00    1.000e+01     5.000e+00     0.000e
1.672e-05  1.168e-03    3.128e-02    0.000e+00    1.010e+01     4.988e+00     0.000e
3.342e-05  1.337e-03    3.137e-02    0.000e+00    1.019e+01     4.977e+00     0.000e
5.010e-05  1.508e-03    3.145e-02    0.000e+00    1.028e+01     4.965e+00     0.000e
6.675e-05  1.680e-03    3.153e-02    0.000e+00    1.038e+01     4.954e+00     0.000e
8.338e-05  1.854e-03    3.161e-02    0.000e+00    1.047e+01     4.942e+00     0.000e
.          .            .            .            .             .             .
.          .            .            .            .             .             .
.          .            .            .            .             .             .
```

If you change the reporting variables, only those selected will appear in the report. The particle time is always reported in the first column. Note that it is possible to select one variable in the **Reporting Variables** dialog box and a variable from the **Color by** drop-down list in the **Particle Tracks** dialog box, however, each variable is reported only once.

When the report is written to a file, a table at the beginning of the file lists all variables selected with the corresponding unit. Thus you can display or export any variable along a particle trajectory to the console or to a file.

Note that the **Coarsen** option affects the step-by-step report.

24.9.5. Reporting of Current Positions for Unsteady Tracking

In transient cases, when using unsteady tracking, you may want to obtain a report of the current positions and states of the particles. Selecting **Current Positions** under **Report Type** in the Particle Tracks Dialog Box (p. 5558) enables the export of the current positions of the particles.

The procedure for generating files containing current position reports is listed below:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Follow steps 1 and 2 in Displaying of Trajectories (p. 2834) for displaying trajectories. You may want to track only one particle stream at a time, using the **Track Single Particle Stream** option.

2. Select **Current Position** as the **Report Type** (**Reporting** group box).

3. Select **File** as the **Report to** option. (The **Track** button will become the **Write...** button.)

4. In the **Significant Figures** field, enter the number of significant figures to be used in the report.

5. To choose the variables to be included in the report, click the **Reporting Variables...** button.

   This button is not available when **Sample File Format** is selected.

6. In the **Reporting Variables** dialog box (Figure 24.68: The Reporting Variables Dialog Box (p. 2857)), you can change the variables in the report. The list under **Variables in Report** contains all variables currently reported. The list under **Particle Variables** contains the particle variables that are available for you to select. You can use the following buttons to modify the **Variables in Report** list:

   - **Remove**: Removes selected variables from that list.
   - **Default Variables**: Restores the default list.
   - **Add Variable**: Adds selections to the report.
   - **Add Color by**: Adds the **Color by** variable to the list of **Variables in Report**, which is the only way to get cell values or customized field functions into the report.

   Important:

   Note that it is possible to select one variable in the **Reporting Variables** dialog box and a variable from the **Color by** drop-down list in the **Particle Tracks** dialog box, however, each variable is reported only once.

7. If you want to generate a file containing the particle current positions in the same file format that is used for particle sampling, select **Sample File Format**.

   The generated file can be used as an injection file to transfer a population of particle parcels from one simulation into another (see below for details). It is also possible to use this file in a case with a different mesh.

8. Click the **Write...** button and specify a file name for the current position report file in The Select File Dialog Box (p. 931).

The default current position report lists the position, velocity, diameter, temperature, density, mass and number in parcel of the particle at selected time steps along the trajectory. In addition, the variable you have selected in the **Color by** list is also included. If you change the reporting variables, only those selected will appear in the report. The particle time is always reported in the first column. It is possible to select one variable in the **Reporting Variables** dialog box and a variable from the **Color by** drop-down list in the **Particle Tracks** dialog box, however, each variable is reported only once.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The output to a file or to the console has the same format as the step-by-step report for steady-state cases.

```
Time       X-Position  Y-Position  Z-Position   X-Velocity   Y-Velocity  Z-Velocity
0.000e+00  1.000e-03   3.120e-02   0.000e+00    1.000e+01    5.000e+00    0.000e+00
1.672e-05  1.168e-03   3.128e-02   0.000e+00    1.010e+01    4.988e+00    0.000e+00
3.342e-05  1.337e-03   3.137e-02   0.000e+00    1.019e+01    4.977e+00    0.000e+00
5.010e-05  1.508e-03   3.145e-02   0.000e+00    1.028e+01    4.965e+00    0.000e+00
6.675e-05  1.680e-03   3.153e-02   0.000e+00    1.038e+01    4.954e+00    0.000e+00
8.338e-05  1.854e-03   3.161e-02   0.000e+00    1.047e+01    4.942e+00    0.000e+00
.          .            .          .            .            .            .
.          .            .          .            .            .            .
.          .            .          .            .            .            .
```

Also listed are the diameter, temperature, density, mass of the particles, number in parcel and the variable selected from the **Color by** list. (You may need to use the scroll bar to view this information.)

```
Time       Diameter    Temperature   Density     Mass        Number      ColorBy
9.999e-04  9.352e-05   3.710e+02     6.840e+02   2.929e-10   2.792e+02   4.783e-02
1.999e-03  7.952e-05   3.710e+02     6.840e+02   1.801e-10   2.792e+02   3.834e-02
3.000e-03  6.660e-05   3.710e+02     6.840e+02   1.058e-10   2.792e+02   2.989e-02
4.001e-03  5.425e-05   3.710e+02     6.840e+02   5.719e-11   2.792e+02   3.719e-02
5.001e-03  4.184e-05   3.710e+02     6.840e+02   2.624e-11   2.792e+02   2.978e-02
.          .           .             .           .           .           .
.          .           .             .           .           .           .
.          .           .             .           .           .           .
```

If you selected **Sample File Format**, then the report file also includes the `flow-time` column. This column contains the current simulated time at which the report was written. When using the file to transfer a population of particle parcels into another simulation, you can use the **Start Time** and **Start Flow-Time in File** file injection point properties in the **Set Injection Properties** dialog box to control the particle injection time. For example, if you set **Start Flow-Time in File** to a value that is only minimally lower than the `flow-time`, then the particle parcels will be injected into the domain at the specified **Start Time**.

24.9.6. Reporting of Interphase Exchange Terms (Discrete Phase Sources)

Ansys Fluent reports the magnitudes of the interphase exchange of momentum, heat, and mass in each control volume in your Ansys Fluent model. You can display these variables graphically, by drawing contours, profiles, and so on. They are all contained in the **Discrete Phase Sources...** category of the variable selection drop-down list that appears in postprocessing dialog boxes:

- **DPM Mass Source**
- **DPM X,Y,Z Momentum Source**
- **DPM Swirl Momentum Source**
- **DPM Turbulent Kinetic Energy Source**
- **DPM Turbulent Dissipation Source**
- **DPM Sensible Enthalpy Source**
- **DPM Enthalpy Source**
- **DPM Burnout**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **DPM Evaporation/Devolatilization**
- **DPM (species) Source**
- **DPM Mixture Fraction Source**
- **DPM Mixture Fraction Secondary Source**
- **DPM Inert Source**

See Field Function Definitions (p. 4167) for definitions of these variables.

Note that these exchange terms are updated and displayed only when coupled calculations are performed. Displaying and reporting particle trajectories (as described in Displaying of Trajectories (p. 2834) and Particle Tracking Statistics (p. 2846)) will not affect the values of these exchange terms.

The exchange terms are reported as the rate occurring in each cell. A unit cell depth is used for 2D cases, and a reference cell depth of 1 radian is used for 2D axisymmetric cases.

24.9.7. Reporting of Particle Variables

In Ansys Fluent, the discrete phase variables listed in the tables below can be used for:

- coloring the particle trajectories
- step-by-step reporting (see Step-by-Step Reporting of Trajectories (p. 2856))
- reporting of the current position and states of unsteady particles (see Reporting of Current Positions for Unsteady Tracking (p. 2858))

- Table 24.7: General Particle Variables (p. 2861)
- Table 24.8: Particle Variables with Heat and Mass Transfer (p. 2864)
- Table 24.9: Particle Variables with the DEM model (p. 2865)
- Table 24.10: Particle Variables with Particle Rotation (p. 2867)
- Table 24.11: Particle Variables with User-Defined Particles (p. 2867)
- Table 24.12: Particle Variables with Particle Breakup (p. 2867)

The following variables are available for all DPM simulations.

Table 24.7: General Particle Variables

Name	Symbol Name in Journals	Units	Description
Particle Residence Time	`particle-resid-time`	time	Time since the moment of particle injection.
Particle ID	`particle-id`	-	Unique (global) particle ID.
Surface ID	`surface-id`	-	ID of the surface from which the particle was released; else -1.
Particle X Position	`particle-x-position`	length	Particle position in the global X-direction.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Symbol Name in Journals	Units	Description
Particle Y Position	particle-y-position	length	Particle position in the global Y-direction.
Particle Z Position	particle-z-position	length	(3d only) Particle position in the global Z-direction.
Particle Radial Position	particle-radial-position	length	(3d only) Particle radial coordinate w.r.t. the cylindrical coordinate system defined for the reference thread.
Particle Theta Position	particle-theta-position	angle	(3d only) Particle theta coordinate w.r.t. the cylindrical coordinate system defined for the reference thread, angle counted starting from a plane containing the Y axis.
Particle X Velocity	particle-x-velocity	velocity	Particle velocity component in the global X-direction.
Particle Y Velocity	particle-y-velocity	velocity	Particle velocity component in the global Y-direction.
Particle Z Velocity	particle-z-velocity	velocity	(3d only) Particle velocity component in the global Z-direction.
Particle Radial Velocity	particle-radial-velocity	velocity	(3d only) Particle radial velocity component w.r.t. the cylindrical coordinate system defined for the reference thread.
Particle Swirl Velocity	particle-swirl-velocity	velocity	(3d only) Particle swirl velocity component w.r.t. the cylindrical coordinate system defined for the reference thread.
Particle Velocity Magnitude	particle-velocity-mag	velocity	$V_{mag} = \sqrt{V_x^2 + V_y^2 + V_z^2}$
Particle Diameter	particle-diameter	length	Particle diameter, d_p.
Particle Density	particle-density	density	Particle mass density, ρ_p. The density may be defined as a function of temperature. For multicomponent particles, it can be calculated as a volume-weighted value of the density of the droplet components.
Particle Specific Heat	particle-Cp	specific-heat	Particle-specific heat capacity defined in material settings for the current particle temperature (or at the reference temperature if physical property averaging is used).
Particle Binary Diffusivity	particle-binary-diffusivity	diffusivity	• For droplet and multicomponent particles: Binary diffusivity of the vapor species at the particle temperature (or at the reference temperature if physical property averaging is used).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Symbol Name in Journals	Units	Description
			• For combusting particles: Binary diffusivity of the oxidant at the continuous-phase temperature.
Particle Mass	particle-mass	mass	$m_p = \frac{\pi}{6}\rho_p d_p^3$
Particle Temperature	particle-temperature	temperature	Particle temperature, T_p (300 K if particle temperature is not solved).
Particle Law Index	particle-law-index	-	Current law index number: • 1: Inert Heating • 2: Droplet Vaporization • 3: Droplet Boiling • 4: Devolatilization • 5: Surface Combustion • 6: Inert Cooling • 7: Multicomponent Particle Definition
Particle On Wall Flag	particle-on-wall	-	True (1) if the particle is a film particle.
Particle Reynolds Number	particle-reynolds-number	-	$Re = \rho_p V_{slip} d_p / \mu$
Particle Time Step	particle-time-step	time	Time step used by the numerical integration scheme.
Particle Number in Parcel	particle-number-in-parcel	mass per time / non-dimensional	• Steady tracking: Mass of particles per time. • Unsteady tracking: Number of particles in parcel (not available for steady tracking).
Tracking Scheme	tracking-scheme	-	• 0: (Euler) implicit • 1: (Euler) trapezoidal • 2: (Exponential) analytic • 3: Runge-Kutta (explicit)
Number of Refinements	number-of-refinements	-	Number of times by which the integration time step size was reduced in the adaptive integration scheme.
Particle Penetration	particle-penetration	length	Distance traveled by the parcel from the injection reference position.
Parcel Mass	parcel-mass	mass	(Unsteady tracking only) Mass of Parcel = Mass of Particle x Number of Particles per Parcel.
Parcel Diameter	parcel-diameter	length	(Unsteady tracking only) Parcel diameter computed from parcel mass and parcel density.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Symbol Name in Journals	Units	Description
Parcel Volume	parcel-volume	volume	(Unsteady tracking only) Volume of Parcel = Volume of Particle x Number of Particles per Parcel.
Particle Current Time	particle-current-time	time	(Unsteady tracking only) Particle time (identical to the current flow time).

The following variables are available if particle heat/mass transfer is enabled.

Table 24.8: Particle Variables with Heat and Mass Transfer

Name	Symbol Name in Journals	Units	Description
Latent Heat	latent-heat	specific-energy	Particle latent heat at the current particle temperature.
Vaporization Limiting Time	vaporization-limiting-time	time	Time scale for achieving fractional changes in both particle mass and temperature according to the **Vaporization Limiting Factors** specified in the **Discrete Phase Model** dialog box. This is available when **Droplet** or **Multicomponent** particles are not enabled in the **Coupled Heat-Mass Solution** group box in the **Discrete Phase Model** dialog box (**Numerics** tab).
Particle Lewis Number	particle-lewis-number	-	(Droplet particles only) $Le = \frac{\lambda}{\rho D_m c_p}$
Particle Nusselt Number	particle-nusselt-number	-	Nusselt Number: Ranz-Marshall $Nu = 2 + 0.6\sqrt{Re_p}\sqrt[3]{Pr}$
Bt Heat Transfer Number	bt-heat-transfer-number	-	(Droplet particles only) Spalding heat transfer number (Equation 12.96 in the *Fluent Theory Guide*).
Bm Mass Transfer Number	bt-mass-transfer-number	-	(Droplet particles only) Spalding mass transfer number (Equation 12.86 in the *Fluent Theory Guide*).
Mass Fraction of *species-n*		-	(Combusting particles with multiple surface reactions only) Mass fraction of the solid species as defined in the **Selected Solid Species** list in the **Species** dialog box.
Particle Mass Fraction of *species-n*		-	(Multicomponent particles only) Mass fraction of the liquid component.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Symbol Name in Journals	Units	Description
Particle Enthalpy		specific-energy	Particle enthalpy at the current particle temperature. For multicomponent particles, it is calculated as a mass-weighted value of the enthalpies of the droplet components.
Particle Liquid Mass Fraction	particle-liquid-mass-fraction	-	• Wet combusting particles: Liquid mass fraction. • Droplet particles: Volatile component fraction.
Particle Volatile Mass Fraction	particle-volatile-fraction	-	• Combusting particles: Mass fraction of volatiles • Droplet particles: Volatile component fraction.
Particle Char Mass Fraction	particle-char-fraction	-	(Combusting particles only) Mass fraction of the char component of the particle.

The following variables are available if the DEM model is enabled.

Table 24.9: Particle Variables with the DEM model

Name	Symbol Name in Journals	Units	Description
Total X Force	force-x-total	force	X component of the total force (body force and volume force) acting on the DEM particle during the DEM collision: • Shows the value from last time step $F_{t,x}$. • Not stored in the `.dat` file.
Total Y Force	force-y-total	force	Y component of the total force (body force and volume force) acting on the DEM particle during the DEM collision: • Shows the value from last time step $F_{t,y}$. • Not stored in the `.dat` file.
Total Z Force	force-z-total	force	Z component of the total force (body force and volume force) acting on the DEM particle during the DEM collision: • Shows the value from last time step $F_{t,z}$. • Not stored in the `.dat` file.
Total Force Magnitude	force-total-mag	force	$F_{t,mag} = \sqrt{F_{t,x}^2 + F_{t,y}^2 + F_{t,z}^2}$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Symbol Name in Journals	Units	Description
Collision X Force	`force-x-coll`	force	X component of the body force acting on the DEM particle during the DEM collision: • Shows the value from last time step. • Not stored in the `.dat` file.
Collision Y Force	`force-y-coll`	force	Y component of the body force acting on the DEM particle during the DEM collision: • Shows the value from last time step. • Not stored in the `.dat` file.
Collision Z Force	`force-z-coll`	force	Z component of the body force acting on the DEM particle during the DEM collision: • Shows the value from last time step. • Not stored in the `.dat` file.
Collision Force Magnitude	`force-coll-mag`	force	$F_{mag} = \sqrt{F_x^2 + F_y^2 + F_z^2}$
Total X Acceleration	`acc-x-total`	acceleration	X component of total acceleration due to the total force during the DEM collision: • Shows the value from last time step. • Not stored in the `.dat` file.
Total Y Acceleration	`acc-y-total`	acceleration	Y component of total acceleration due to total Force during DEM collision: • Shows the value from last time step. • Not stored in the `.dat` file.
Total Z Acceleration	`acc-z-total`	acceleration	Z component of total acceleration due to the total force during the DEM collision: • Shows the value from last time step. • Not stored in the `.dat` file.
Total Acceleration Magnitude	`acc-total-mag`	acceleration	$a_{mag} = \sqrt{a_x^2 + a_y^2 + a_z^2}$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following variables are available if particle rotation is enabled.

Table 24.10: Particle Variables with Particle Rotation

Name	Symbol Name in Journals	Units	Description
Angular X Velocity	`x-angular-velocity`	angular velocity	Particle rotation rate about the X-axis.
Angular Y Velocity	`y-angular-velocity`	angular velocity	Particle rotation rate about the Y-axis.
Angular Z Velocity	`z-angular-velocity`	angular velocity	Particle rotation rate about the Z-axis.
Angular Velocity Magnitude	`angular-velocity-mag`	angular velocity	$\omega_{mag} = \sqrt{\omega_x^2+\omega_y^2+\omega_z^2}$

The following variables are available if user define particles are enabled.

Table 24.11: Particle Variables with User-Defined Particles

Name	Symbol Name in Journals	Units	Description
User Value *i*	`user-value-`*i*	-	User data stored in `TP_USER_REAL(tp, i)`. See `DEFINE_DPM_SCALAR_UPDATE` in the *Fluent Customization Manual* for an example.

The following variables are available if particle breakup is enabled.

Table 24.12: Particle Variables with Particle Breakup

Name	Symbol Name in Journals	Units	Description
Particle Breakup Type	`particle-breakup-type`	-	• `0`: None • `1`: Levich • `2`: TAB • `3`: WAVE • `4`: KHRT • `5`: SSD • `6`: Madabhushi • `7`: Schmehl
Particle Weber Number Gas	`particle-weber-number-gas`	-	Liquid Weber number based on gas density $We = \rho_g V_{slip}^2 d_p / \sigma$
Particle Weber Number Liquid	`particle-weber-number-liquid`	-	Liquid Weber number based on particle density $We = \rho_p V_{slip}^2 d_p / \sigma$
SSD break up time	`ssd-break-up-time`	time	Equation 12.445 in the *Fluent Theory Guide*

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name	Symbol Name in Journals	Units	Description
Normalized TAB displacement	`normalized-TAB-dis-placement`	-	Equation 12.405 in the *Fluent Theory Guide*
Normalized TAB velocity	`normalized-TAB-velo-city`	1/time	Equation 12.409 in the *Fluent Theory Guide*
WAVE break up time	`WAVE-break-up-time`	time	Equation 12.437 in the *Fluent Theory Guide*
WAVE mass in shed drops	`WAVE-mass-in-shed-drops`	mass	Mass of the shed-off droplet during the WAVE breakup process.
WAVE shed drops cutoff mass	`WAVE-shed-drops-cutoff-mass`	mass	Cut-off mass for the shed particles. During WAVE breakup, a new parcel will be created if the particle shed mass becomes larger than the cutoff mass.
Accumulated break up time	`accumu-lated-break-up-time`	time	Equation 12.442 in the *Fluent Theory Guide*
PILCH_ERDMAN deformation time	`PILCH_ERD-MAN-deforma-tion-time`	time	Equation 12.448 in the *Fluent Theory Guide*
PILCH_ERDMAN break up time	`PILCH_ERD-MAN-break-up-time`	time	Equation 12.449 in the *Fluent Theory Guide*
PILCH_ERDMAN total break up time	`PILCH_ERD-MAN-total-break-up-time`	time	Equation 12.455 in the *Fluent Theory Guide*
PILCH_ERDMAN initial weber number	`PILCH_ERD-MAN-ini-tial-weber-number`	-	Particle Weber number at the beginning of the Pilch-Erdman breakup process. It is used to compute the deformation and breakup time of the droplets.
PILCH_ERDMAN initial diameter	`PILCH_ERD-MAN-ini-tial-diamet-er`	length	Particle diameter at the beginning of the Pilch-Erdman breakup process.
MADABHUSHI column break up time	`MADABHUSHI-column-break-up-time`	time	Equation 12.447 in the *Fluent Theory Guide*

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.9.8. Reporting of Discrete Phase Variables

Ansys Fluent reports various discrete phase particle/parcel quantities including erosion/accretion rates, radiation quantities, and (optionally) cell-averaged particle size, velocity, temperature, and so on. You can display these variables graphically, by drawing contours, profiles, and so on. They are accessed in the **Discrete Phase Variables...** category of the variable selection drop-down list that appears in postprocessing dialog boxes.

Several quantities are automatically available depending on the models being used in the simulation. For the cell-averaged quantities to be available, you must first enable **Mean Values** under **Contour Plots for DPM Variables** in the **Discrete Phase Model** dialog box, Discrete Phase Model Dialog Box (p. 4845). RMS quantities are also available for particle velocity and temperature by enabling **RMS Values**.

Note:

Enabling Mean Values to track the cell-averaged variables will increase the memory requirements of your simulation.

The following lists specify those variables that are available automatically and those that require you to enable them

Variables available automatically

- **DPM Erosion Rate (Generic)**
- **DPM Erosion Rate (Finnie)**
- **DPM Erosion Rate (McLaury)**
- **DPM Erosion Rate (Oka)**
- **DPM Erosion Rate (DNV)**
- **DPM Erosion Rate (Wall Shear)**
- **DPM Accretion Rate**
- **DPM Absorption Coefficient**
- **DPM Emission**
- **DPM Scattering**
- **DPM Concentration**
- **DPM Wall X Force**
- **DPM Wall Y Force**
- **DPM Wall Z Force**
- **DPM Wall Normal Pressure**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **DPM (species) Concentration**
- **DPM Collision Rate**

Particle rotation variables (Particle rotation simulations only)

- **Angular X Velocity**
- **Angular Y Velocity**
- **Angular Z Velocity**
- **Angular Velocity Magnitude**
- **Angular Velocity** (in the **Particle Vector Style** dialog box)

Cell-averaged variables available when Mean Values are enabled

- **DPM Volume Fraction**
- **DPM Particles in Cell**
- **DPM Parcels in Cell**
- **DPM Number Density**
- **DPM X, Y, Z Velocity**
- **DPM Diameter**
- **DPM Density**
- **DPM Temperature**
- **DPM Enthalpy**
- **DPM Specific Heat**
- **DPM D20**
- **DPM D30**
- **DPM D32**
- **DPM D43**
- **DPM Granular Temperature**
- **DPM Conc. of (component)**

For the discrete phase variable φ, the event-based time average (mean) in a cell is calculated as:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\overline{\phi} = \frac{\sum_{i=1}^{N} \phi_i t_i n_{p,i}}{\sum_{i=1}^{N} t_i n_{p,i}} \tag{24.23}$$

where

N = number of parcels in the cell

$n_{p,i}$ = number of particles in the i^{th} parcel

t_i = residence time of the parcel

RMS variables available when Mean Values and RMS Values are enabled

- **DPM RMS X, Y, Z Velocity**
- **DPM RMS Temperature**
- **DPM RMS Diameter**

The RMS value of the discrete phase variable φ is calculated as:

$$\phi_{rms} = \sqrt{\sum_{i=1}^{N} (\phi_i - \overline{\phi}_i)^2 t_i n_{p,i} \Big/ \sum_{i=1}^{N} t_i n_{p,i}} \tag{24.24}$$

See Field Function Definitions (p. 4167) for definitions of these variables.

Note that these variables are updated and displayed only when coupled calculations are performed. Displaying and reporting particle trajectories (as described in Displaying of Trajectories (p. 2834) and Particle Tracking Statistics (p. 2846)) will not affect the values of these variables.

Note:

If you select **Enable Node Based Averaging** in the **Numerics** tab of the **Discrete Phase Model** dialog box, any previously calculated DPM variables will be rendered useless and need to be recomputed.

24.9.9. Reporting of Unsteady DPM Statistics

If you are performing a transient simulation, you can include the computation of unsteady time statistics (mean and RMS) for the discrete phase(s) of the transient flow. These are computed on a per-phase basis by event-based time averaging over the discrete phase parcels in the domain. This means that the sampling of the discrete phase particle quantity in a given cell occurs when (and only when) a parcel passes through the cell.

To calculate the unsteady DPM time statistics, enable **Data Sampling for Time Statistics** in the **Run Calculation** task page and enable **DPM Variables** in the **Sampling Options** dialog box when preparing to run your simulation (see Inputs for Time-Dependent Problems (p. 3645)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For a discrete phase variable, ϕ, the event-based time average (mean) in a cell is calculated from Equation 24.25 (p. 2872).

$$\langle \overline{\phi} \rangle = \frac{\sum_{i=1}^{N} \phi_i w_i}{\sum_{i=1}^{N} w_i} \tag{24.25}$$

where N is the number of parcels in the cell, and

$$w_i = n_{p,i} t_i \tag{24.26}$$

Here $n_{p,i}$ is the number of particles in the i^{th} parcel and t_i is the residence time of the i^{th} parcel in the cell. The weighting by residence time is necessary because a parcel may pass through more than one cell within a computational time step and therefore its contribution to the averaged cell quantities depends on the fraction of the time step spent in each cell.

The time-averaged DPM volume fraction is calculated from the individual number of particles in the parcel, the particle volume, and its residence time in the cell in relation to the individual time step.

The RMS value is also available and is computed from Equation 24.27 (p. 2872).

$$\phi_{rms} = \sqrt{\frac{\sum_{i=1}^{N} \left(\phi_i - \langle \overline{\phi} \rangle \right)^2 w_i}{\sum_{i=1}^{N} w_i}} \tag{24.27}$$

Note that these quantities are averaged over all parcels that have passed through the cell and are therefore different from the instantaneous cell-averaged values described in Reporting of Discrete Phase Variables (p. 2869).

Below is a list of the available unsteady statistics when **DPM Variables** is enabled in the **Sampling Options** dialog box. These are accessible by selecting the **Unsteady DPM Statistics...** category in postprocessing or reporting dialog boxes. Note that the availability of some quantities depends on the physics models being used. For definitions of these quantities, refer to the definitions of the instantaneous quantities from which they are derived (see Alphabetical Listing of Field Variables and Their Definitions (p. 4205)).

Unsteady DPM Statistics

- **Mean DPM Volume Fraction**
- **Accum DPM Particles in Cell**
- **Mean DPM X, Y, Z Velocity**
- **Mean DPM Diameter**
- **Mean DPM Density**
- **Mean DPM Temperature**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Mean DPM Granular Temperature**
- **Mean DPM Number Density**
- **Accum DPM Parcels in Cell**
- **Mean DPM D20**
- **Mean DPM D30**
- **Mean DPM D32**
- **Mean DPM D43**
- **RMS DPM Volume Fraction**
- **RMS DPM X, Y, Z Velocity**
- **RMS DPM Diameter**
- **RMS DPM Density**
- **RMS DPM Temperature**
- **RMS DPM Granular Temperature**
- **RMS DPM Number Density**

Explanations for the **Accum**... variables are provided in Alphabetical Listing of Field Variables and Their Definitions (p. 4205).

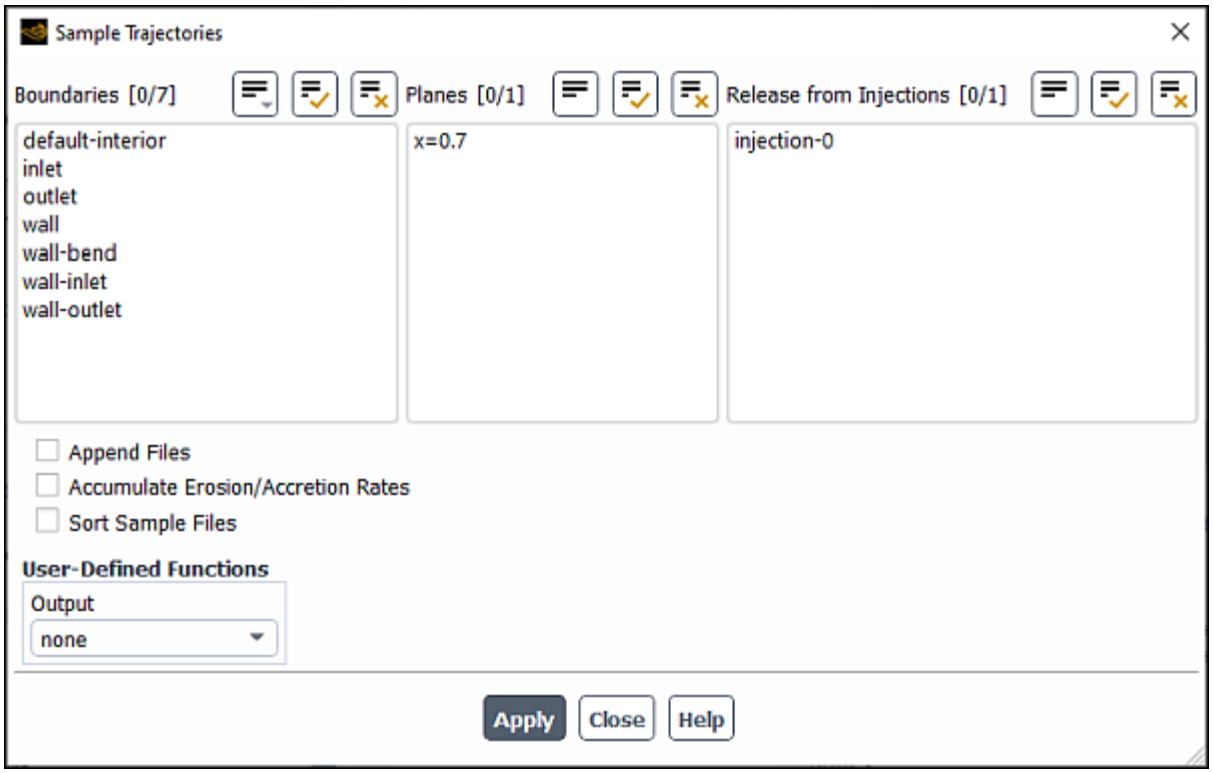
24.9.10. Sampling of Trajectories

Particle states (position, velocity, diameter, temperature, and mass flow rate) can be written to files at various boundaries and planes (lines in 2D) using the Sample Trajectories Dialog Box (p. 5331) (Figure 24.69: The Sample Trajectories Dialog Box (p. 2874)).

Results → Reports → Discrete Phase → Sample → Edit...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.69: The Sample Trajectories Dialog Box



The procedure for generating files containing the particle samples is listed below:

1. Select the injections to be tracked in the **Release From Injections** list.

2. Select the surfaces at which samples will be written. These can be boundaries from the **Boundaries** list or planes from the **Planes** list (in 3D) or lines from the **Lines** list (in 2D).

3. If you want to use the particle sampling file as an unsteady injection file, select **Sort Sample Files**.

   The **Sort Sample Files** option causes the sample files to be sorted in a reproducible manner. For steady particle tracking, files will be sorted first by injection and then by particle ID. For unsteady particle tracking, the files will be sorted strictly by the simulated time (`flow-time` in the 13th column in the sample file) at which the particle passes the sampling plane surface or mesh zone.

4. Click the **Compute** button. Note that for unsteady particle tracking, the **Compute** button will become the **Start** button (to initiate sampling) or a **Stop** button (to stop sampling).

   Clicking the **Compute** button will cause the particles to be tracked and their status to be written to files when they encounter selected surfaces. The file names will be formed by appending `.dpm` to the surface name. The files can be used for:

   - Postprocessing probability distributions of particle properties (see Histogram Reporting of Samples (p. 2875))

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Injecting particles using the **file** injection type

> **Note:**
>
> If you select a face zone in a mesh interface for sampling, Fluent will sample the particles on both of the interface zones and will write out a total of three files. Two of these files correspond to the particles that encounter each respective interface zone from upstream. The third file contains the complete set of particles that have crossed the mesh interface pair (via either interface zone).

For unsteady particle tracking, clicking the **Start** button will open the files and write the file header sections. If the solution is advanced in time by computing some time steps, the particle trajectories will be updated and the particle states will be written to the files as they cross the selected planes or boundaries. Clicking the **Stop** button will close the files and end the sampling.

For stochastic tracking, it may be useful to repeat this process multiple times and append the results to the same file, while monitoring the sample statistics at each update. To do this, enable the **Append Files** option before repeating the calculation (clicking **Compute**). Similarly, you can cause erosion and accretion rates to be accumulated for repeated trajectory calculations by turning on the **Accumulate Erosion/Accretion Rates** option. (See also Postprocessing of Erosion/Accretion Rates (p. 2884).)

The format and the information written for the sample output can also be controlled through a user-defined function, which can be selected in the **Output** drop-down list. See `DEFINE_DPM_OUTPUT` in the *Fluent Customization Manual* for more information.

The generated sample files can be used as injection file in a file injection. Both files use the same file formats which differ between steady and unsteady particle tracking. For details, see Point Properties for File Injections (p. 2742).

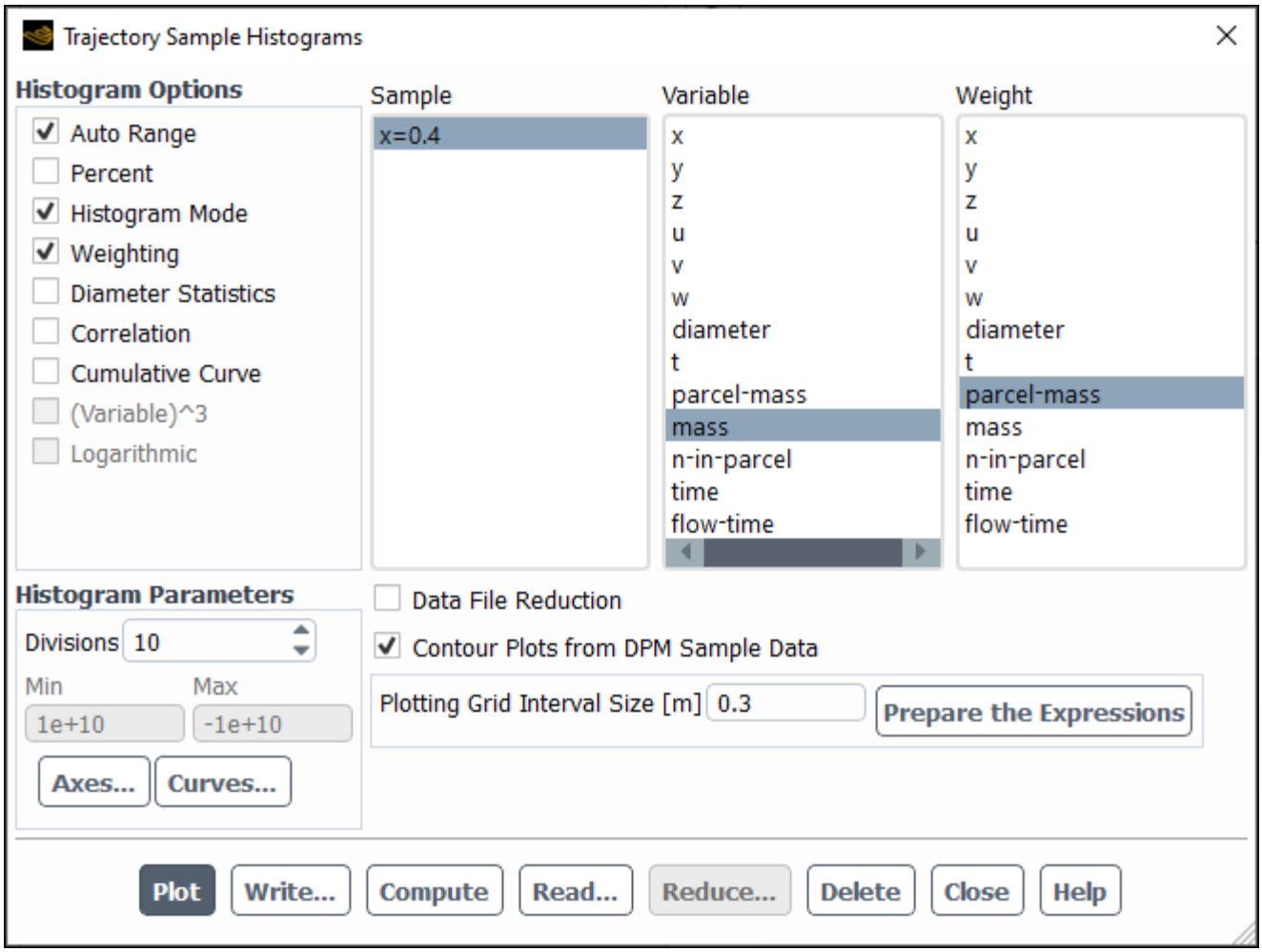
If the number of lines in the sampling file is large, you can reduce it by combining sets of similar line entries into one single entry per such set. The mass represented by that entry will be increased accordingly to ensure mass conservation. For details, see Data Reduction of Samples (p. 2879).

24.9.11. Histogram Reporting of Samples

DPM sample files can be created using the Sample Trajectories Dialog Box (p. 5331) (as described in Sampling of Trajectories (p. 2873)) or by the VOF-to-DPM model transition mechanism. Such a file contains multiple records, each consisting of values for the same set of variables. You can postprocess probability distributions and simple correlation analyses of these variables using the Trajectory Sample Histograms Dialog Box (p. 5333) (Figure 24.70: The Trajectory Sample Histograms Dialog Box (p. 2876)).

Results → Reports → Discrete Phase → Histogram Edit...

Figure 24.70: The Trajectory Sample Histograms Dialog Box



24.9.11.1. Analysis, Investigation, and Reporting of Samples

The basic procedure for plotting a histogram of the probability distribution of a variable read from data in a sample file is listed below:

1. Select a sample file to be read by clicking the **Read...** button. After you read in the sample file, the sample name will appear in the **Sample** list.

2. Select the data sample in the **Sample** list, and then select the variable for which to plot the probability distribution histogram from the **Variable** list.

3. Select the variable by which to weight the file individual line entries from the **Weight** list. To obtain representative information for most intensive variables, such as temperature, velocity, or position, you should select:

   - **mass-flow** for a sample file from steady-state particle tracking
   - **parcel-mass** for a sample file from unsteady particle-tracking.

   If the sampled particle streams / parcels all have the same flow rate / parcel mass, you can clear **Weighting**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. You can specify the number of bins, or intervals, in the plot in the **Divisions** field under **Histogram Parameters**.

5. If you want to plot just some part of the probability distribution, clear **Auto Range** and specify the range of values in the **Min** and **Max** fields under **Histogram Parameters**. You can reset **Min** and **Max** to the complete range by clicking **Compute**.

6. If variable values you are investigating span several orders of magnitude, you can use logarithmic scaling of the bins by selecting **Logarithmic** under **Histogram Options**. Note that this option is available only for the variables that take only positive values. With this option, you should select the logarithmic scaling format for the abscissa and, if necessary, for the ordinate in the **Axes** dialog box that opens by clicking **Axes...** (**Histogram Parameters** group box).

7. Click the **Plot** button at the bottom of the dialog box to display the histogram in the graphics window.

In addition, the following controls are available:

- **Percent**

  By default, the percentages of the total amount measured by the weighting variable will be plotted on the y axis. You can plot the actual number of sampled particle streams / parcels (that is, line entries from the file that fall into each interval on the x axis) by clearing **Percent** under **Histogram Options**. If **Weighting** is selected, the sum of the weights of all entries in each bin will be plotted instead of the number.

- **Histogram Mode**

  To display a probability density function curve instead of the histogram, clear **Histogram Mode** under **Histogram Options**. You can enable the display of data point markers in the plot using the **Curves** dialog box that opens by clicking **Curve...**.

- **Diameter Statistics**

  If you want to display in the console a summary for the selected variables similar to that in Summary Reporting of Current Particles (p. 2883), select **Diameter Statistics** and click **Plot**. Although these statistics are computed for any selected variable in the **Variable** list, it is applicable only to the diameter information.

- **Write...**

  If you want to further postprocess the plot data with other software tools, you can store them in an XY-plot file format using the **Write...** button. If you use the default filename extension `.his`, the file be written with a header. The header lines begin with the comment character `;` and contain the same statistical information that is printed to the Fluent console when you click **Plot**.

- **Delete**

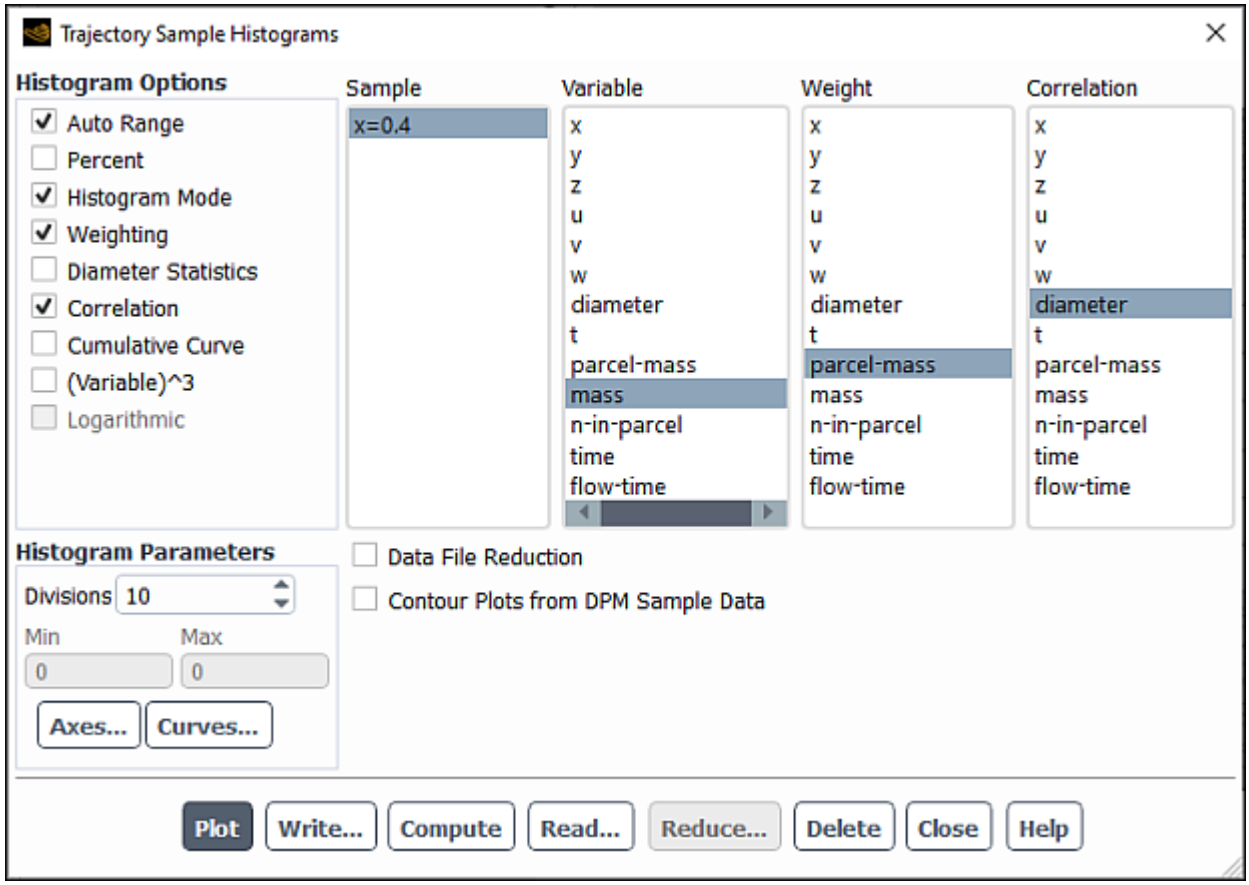
  If you want to remove the currently highlighted sample from the **Sample** list and delete it from the Ansys Fluent session memory, but not the corresponding file, click **Delete**.

- **Correlation**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When investigating the behavior of particles, it is sometimes desirable to know how one particle variable correlates with another particle variable. To facilitate this, the **Correlation** option exists. When you enable this option, an additional column of sampled variables appears, allowing you to choose the correlation variable (see Figure 24.71: The Trajectory Sample Histograms Dialog Box: Correlation (p. 2878)). In this mode, every ordinate value will be a weighted average of the values of the correlation variable in all entries that fall into the respective bin.

Figure 24.71: The Trajectory Sample Histograms Dialog Box: Correlation



- **Cumulative Curve**

  If you want to plot the continuous cumulative distribution function, enable the **Cumulative Curve** option. A cumulative distribution curve is computed of the variable that is selected in the **Variable** list using the weighting options. However, if the **Correlation** option is enabled along with the **Cumulative Curve**, then the cumulative curve of the variable selected in the **Correlation** list is plotted. For a constant particle density, you can plot a correlation curve for the particle mass by selecting the diameter as a **Correlation** variable and enabling **(Variable)^3**. This calculates cubes of the variable chosen under **Correlation** and for which an average is calculated within each bin and used for the ordinate value in the plot as explained above.

- **Contour Plots from DPM Sample Data**

  If you want to use data from a sample file for graphical postprocessing (contour plotting) on the planar surface where the DPM particles were sampled, enable **Contour Plots from DPM Sample Data** and follow steps outlined in Contour Plots of DPM Particle Sampling Results on a Planar Surface (p. 2880).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.9.11.2. Data Reduction of Samples

When you use a sample file as an injection file in a subsequent simulation, it may be desirable to reduce the number of particles injected by the file in order to lower the computational cost. In Ansys Fluent this can be done by reducing the number of lines the sample file contains using a method that is similar to that used to create a histogram. However, unlike a histogram, the weighted entries from the sample file are sorted into intervals for all variables simultaneously, not one variable at a time.

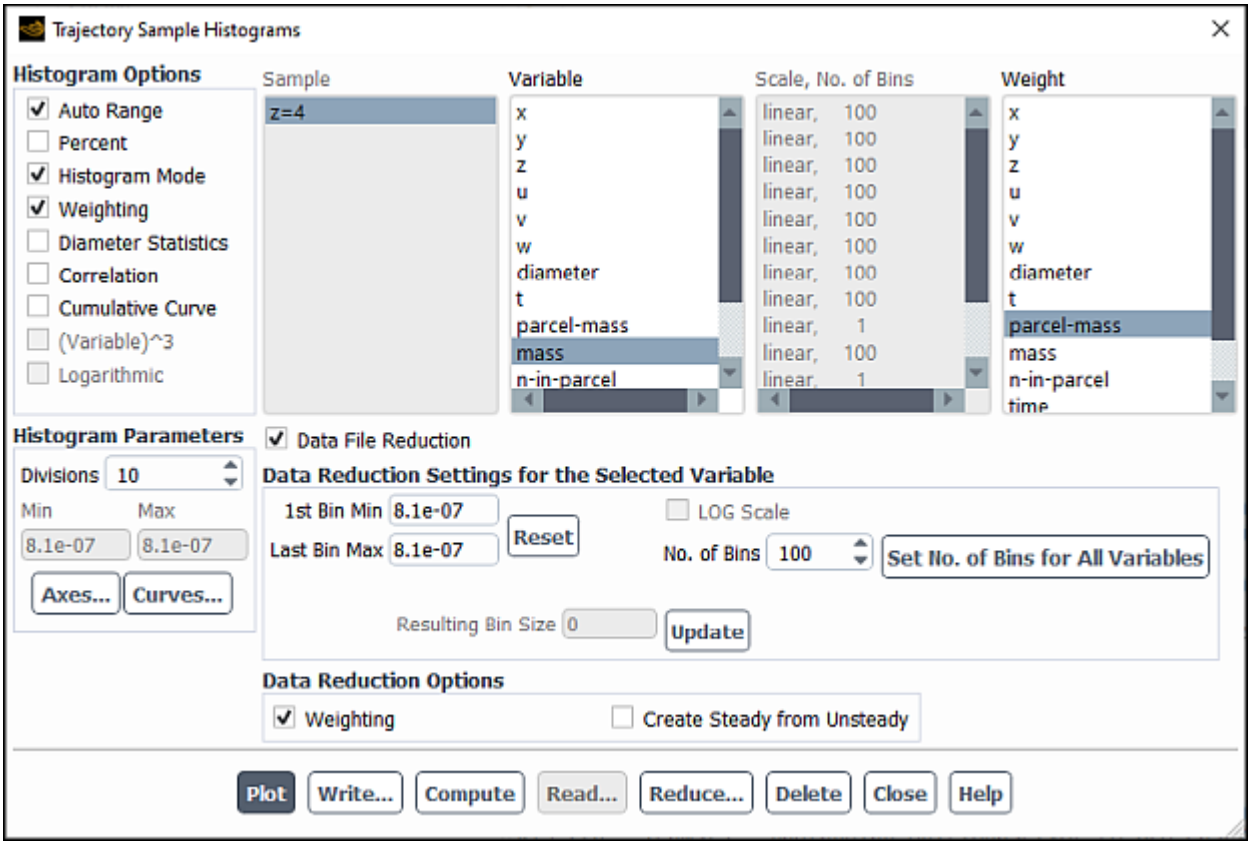
The method first sorts all file entries into groups, or bins, of entries that are sufficiently similar in all properties. The entries that have fallen into the same bin are then averaged to create a single entry. Conservation of mass and volume as well as the number of particles is achieved by appropriate weighting in the averaging process.

To reduce a sample file size:

1. Read a sample file as described in Analysis, Investigation, and Reporting of Samples (p. 2876).
2. Select the data sample in the **Sample** list.
3. Select **Data File Reduction**.

   The **Trajectory Sample Histograms** dialog box will expand to show the **Scale, No. of Bins** list (between the **Variable** and **Weight** lists) and data file reduction inputs.

Figure 24.72: The Trajectory Sample Histograms Dialog Box: Data File Reduction



4. For each variable in the **Variable** list, specify the following parameters in the **Data Reduction in the Selected Variable** group box:

   1st Bin Min , Last Bin Max

   limit the range of values to consider. Note that the complete range for the selected variable is displayed in the **Min** and **Max** fields in the **Histogram Parameters** group box. You can use the **Reset** button to reset **1st Bin Min** and **Last Bin Max** to their original values.

   LOG Scale

   enables the use of a logarithmic scale. The behavior of this option is analogous to the **Logarithmic** option in **Histogram Options** (see Analysis, Investigation, and Reporting of Samples (p. 2876)). By default, the linear scaling is used. This option is available only if all values for the variable in the sample file are positive.

   No. of Bins

   the number of groups into which the specified range for this variable is to be divided. If you want to neglect any variable, set **No. of Bins** to 1. If you want to set the value you specified for all variables, click **Set No. of Bins for All Variable**.

   Once you have specified **1st Bin Min**, **Last Bin Max**, and **No. of Bins**, you can click **Update** to see the **Resulting Bin Size**. If the **LOG Scale** is selected, the value shown in the **Resulting Bin Size** is the ratio for **LOG Scale**.

   You can quickly review the inputs for the scaling and number of bins for each variable in the **Scale, No. of Bins** information list.

5. For a standard DPM sample file generated by Ansys Fluent, you should use the default weighting. However, if you want to reduce data in a file that does not contain the default header and variable columns that are present in a standard DPM sample file, you can select a weighting variable manually from the **Weight** list. If you do not want to use the weight, clear the **Weighting** option in the **Data Reduction Options** group box. Note that in both scenarios, the reduced sample file may not represent the same mass, volume, and flow rates of discrete phase.

6. If the original sample file was generated from unsteady particle tracking, you can reduce the unsteady sample file to a regular steady-state injection file by selecting the **Create Steady from Unsteady** check box in the **Data Reduction Options** group box. This option is available only for sample files from unsteady particle tracking.

7. Once you have configured the data reduction settings for all variables and selected the data reduction options, click **Reduce...**.

8. In The Select File Dialog Box (p. 931) that opens automatically, enter the name of your reduced injection sample file.

   The number of aggregated records will be reported in the Ansys Fluent console.

24.9.12. Contour Plots of DPM Particle Sampling Results on a Planar Surface

You can create contour plots of the spatial distribution of certain particle phase properties from DPM particle sampling on a specific planar surface (cut plane or planar domain boundary face zone). For

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

example, you can create a contour plot of the droplet Sauter mean diameter on a plane orthogonal to the liquid spray direction. In this feature, the planar surface at which the DPM parcel samples are written to a file is divided into squares of a user-specified edge length. The DPM sampling events, which occur when particles cross the plane, are counted in each square. The weighted averages of different quantities are calculated based on the DPM particles sampled in that square. The results are then automatically interpolated into the CFD mesh and made available for graphical postprocessing as named expressions.

The procedural steps for using this feature are shown below.

1. Create a plane cutting through the geometry, or identify a planar domain boundary to be used. (See Plane Surfaces (p. 3830) for details.)

2. Sample particle trajectories on the surface as described in Sampling of Trajectories (p. 2873).

   A DPM sample file with the same name as the surface and a `.dpm` extension will be automatically saved in your working directory.

   Results → Reports → Discrete Phase → Sample → Edit...

3. In the **Trajectory Sample Histograms** dialog box, read the DPM sample file by using the **Read...** button. (See Figure 24.70: The Trajectory Sample Histograms Dialog Box (p. 2876).)

   Results → Reports → Discrete Phase → Histogram → Edit...

   For more information about using the **Trajectory Sample Histograms** dialog box, see Histogram Reporting of Samples (p. 2875).

4. Enable **Contour Plots from DPM Sample Data**.

   The Ansys Fluent solver automatically calculates a default value for **Plotting Grid Interval Size**, which determines the size of the squares into which the sampling plane is divided.

5. If necessary, adjust the **Plotting Grid Interval Size**.

   When specifying the interval size, you should only use values that are larger than the local size of the mesh elements; otherwise, the calculation and interpolation may neglect data points and thus lose fidelity and accuracy. In addition, larger discretisation intervals reduce the risk that too few DPM sampling events fall within any given discretisation square of the plane, and, therefore, the local values for the quantities of interest will be shown as 0.

   Conversely, a very large interval size can affect the accuracy of the calculation of mass flux (steady particle tracking) or mass per unit area (unsteady particle tracking) in regions where the surface either has a boundary or touches the domain boundary. This is because the calculation uses the size of the entire square, although there can be no sampling events outside the plane or domain boundaries.

6. Click **Prepare the Expressions**.

   The Ansys Fluent solver generates named expressions with names that contain the name of the particle sampling surface (represented by <*surfname*> in the named expressions below).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **patt_significance_threshold**: Is the minimum number of DPM sampling events required for statistical evaluation of the local values of quantities defined by the named expressions whose name contain **_signif_** (see below). Each local value will be suppressed if this number is below the specified significance threshold. In this way, values that are based on insufficient statistical data (for example, at the periphery of a spray) are removed from plots.

  The significance threshold is a constant single-valued expression that can be adjusted as desired.

- **<*surfname*>_number_of_events**: Is the number of DPM sampling events (that is, entries in the `*.dpm` file) that have been combined into a single value for every quantity in the local square element of the discretized plane. This number is affected by the value you have entered for **Plotting Grid Interval Size** in the **Trajectory Sample Histograms** dialog box.

  Note that if this number is small, all other values in this location will have uncertainty and should not be used for comparison with experimental results nor for any evaluation of the simulation prediction.

- **<*surfname*>_sufficient_data_flag**: Is a local indicator flag. A value of 1 is shown where the number of DPM sampling events combined into a single local data point exceeds the significance threshold, and a value of 0 is shown elsewhere. You can use this named expression to check if there were enough events in the sampling, and if the **Plotting Grid Interval Size** was sufficiently large.

- **<*surfname*>_signif_sauter_mean_diameter**: Is the local Sauter Mean Diameter [m].

- **<*surfname*>_signif_mass_weighted_mean_temp.**: Is the mass-weighted mean temperature [K].

- **<*surfname*>_signif_mass_weighted_mean_vel._m**: Is the mass-weighted mean velocity magnitude [m/s].

- **<*surfname*>_signif_mass_per_unit_area**: (unsteady particle tracking) Is the amount of DPM mass per unit area that has been calculated in the local square of the discretization plane [kg/m^2].

- **<*surfname*>_signif_mass_flux**: (steady particle tracking) Is the mass flux that has been calculated in the local square of the discretization plane [kg/m^2s].

7. Use the generated named expressions for graphical postprocessing (contour plotting). See Post-processing Expressions (p. 1062) for more information.

Important:

You must make sure that the named expressions are postprocessed only on the planar surface for which they have been prepared, and whose name they bear. Otherwise, the visualisation will display incorrect data.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

24.9.13. Summary Reporting of Current Particles

For many mass-transfer and flow processes, it is desirable to know the mean diameter of the particles. A mean diameter, D_{jk}, is calculated from the particle size distribution using the following general expression [83] (p. 5718):

$$\left(D_{jk}\right)^{j-k} \equiv \frac{\int_0^\infty D^j f(D) dD}{\int_0^\infty D^k f(D) dD} \tag{24.28}$$

where j and k are integers and $f(D)$ is the distribution function (for example, Rosin-Rammler). D_{10}, for example, is the average (arithmetic) particle diameter. The Sauter mean diameter (SMD), D_{32}, is the diameter of a particle whose ratio of volume to surface area is equal to that of all particles in the computation. A summary of common mean diameters is given in Table 24.13: Common Mean Diameters and Their Fields of Application (p. 2883).

Table 24.13: Common Mean Diameters and Their Fields of Application

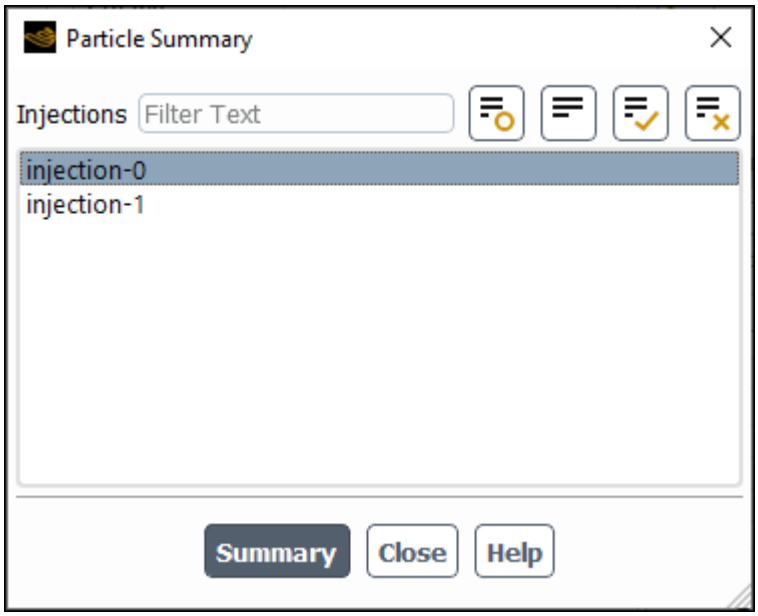
j	k	Order $j+k$	Name	Field of Application
1	0	1	Mean diameter, D_{10}	Comparisons, evaporation
2	0	2	Mean surface diameter, D_{20}	Absorption
3	0	3	Mean volume diameter, D_{30}	Hydrology
2	1	3	Overall surface diameter, D_{21}	Adsorption
3	1	4	Overall volume diameter, D_{31}	Evaporation, molecular diffusion
3	2	5	Sauter mean diameter, D_{32}	Combustion, mass transfer, and efficiency studies
4	3	7	De Brouckere diameter, D_{43}	Combustion equilibrium

Summary information (number, mass, average diameter) for particles currently in the computational domain can be reported using the Particle Summary Dialog Box (p. 5336) (Figure 24.73: The Particle Summary Dialog Box (p. 2884))

Results → Reports → Discrete Phase → Summary Edit...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.73: The Particle Summary Dialog Box



The procedure for reporting a summary for particle injections is as follows:

1. Select the particle injection(s) you want summarized in the **Injections** list.

   Fluent provides a shortcut for selecting injections with names that match a specified pattern. To use this shortcut, enter the pattern in the **Filter Text** box. For example, if you specify `drop*`, all injections that have names beginning with **drop** (for example, **drop-1**, **droplet**) will be listed.

2. Click **Summary** to display the injection summary in the console window.

```
  (*)-  Summary  for  Injection:  injection-0  -(*)

Total  number  of  parcels              :  1862
Total  number  of  particles            :  1.196710e+05
Total  mass                             :  1.128303e-05 (kg)
Maximum  RMS  distance  from  injector  :  7.372527e-01 (m)
Maximum  particle  diameter             :  3.072739e-04 (m)
Minimum  particle  diameter             :  1.756993e-06 (m)
Overall  RR  Spread  Parameter          :  1.446806e+00
Maximum  Error  in  RR  fit             :  1.071220e-01
Overall  RR  diameter            (D_RR):  9.051303e-05 (m)
Overall  mean  diameter          (D_10):  4.663269e-05 (m)
Overall  mean  surface  area     (D_20):  5.344694e-05 (m)
Overall  mean  volume            (D_30):  6.121478e-05 (m)
Overall  surface  diameter       (D_21):  6.125692e-05 (m)
Overall  volume  diameter        (D_31):  7.013570e-05 (m)
Overall  Sauter  diameter        (D_32):  8.030141e-05 (m)
Overall  De  Brouckere  diameter  (D_43):  1.082971e-04 (m)
```

24.9.14. Postprocessing of Erosion/Accretion Rates

You can calculate the erosion and accretion rates in a cumulative manner (over a series of injections) by using the **Sample Trajectories** dialog box. First select an injection in the **Release From Injections** list and compute its trajectory. Then enable the **Accumulate Erosion/Accretion Rates** option, select

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the next injection (after deselecting the first one), and click **Compute** again. The rates will accumulate at the surfaces each time you click **Compute**.

Important:

Both the erosion rate and the accretion rate are defined at wall face surfaces only, so they cannot be displayed at node values.

24.9.15. Assessing the Risk for Solids Deposit Formation During Selective Catalytic Reduction Process

Ansys Fluent provides functionality for assessing the solids deposition risk in the Selective Catalytic Reduction (SCR) process. The computation involves several chemical and hydrodynamic risk factors that are based on thermodynamic data and known chemical pathways, as well as on experimental evidence reported in the literature ([149] (p. 5721), [156] (p. 5722), [22] (p. 5714), [133] (p. 5720), and [176] (p. 5723)). The risk assessment calculation should be invoked after the initial liquid film formation on the surfaces of the SCR device, and should cover a time period over one or more urea injection cycles. The risk variables are defined as dimensionless quantities that vary from 0 to 1, where 0 corresponds to no risk and 1 corresponds to maximum risk. Note that the risk assessment calculation is not a predictive model for urea deposit formation, and the obtained results should be considered only as rough guidance in exploring urea deposit formation trends.

The implementation takes into account the following chemistry risk factors:

- Crystallization risk r_{cryst}

  The limiting urea mass fraction y_{limit} for the onset of crystallization is a function of film temperature T_{film} ([149] (p. 5721) and [133] (p. 5720)). Therefore, the crystallization risk over the time period t_0–t_1 can be calculated as the fraction of time when the urea mass fraction in the liquid film $y_{film,urea}$ exceeds y_{limit}:

$$r_{cryst} = \frac{\int_{t_0}^{t_1} dt}{t_1 - t_0}, \quad \left(y_{film,urea} > y_{limit}(T_{film})\right) \tag{24.29}$$

- Secondary reactions risk r_{sec}

  According to [149] (p. 5721), [156] (p. 5722), and [22] (p. 5714), the formation of biuret and/or cyanuric acid solids deposits starts at T_{min} = 423-433 K and diminishes at T_{max} = 523 K. In addition, the presence of HNCO in the gas phase above the film is required. It follows that the secondary reactions risk over the time period t_0–t_1 can be calculated as the fraction of time when the liquid film is exposed to a non-zero HNCO concentration and the film temperature is between T_{min} and T_{max} as specified above.

$$r_{sec} = \frac{\int_{t_0}^{t_1} dt}{t_1 - t_0}, \quad \left(T_{min} < T_{film} < T_{max}\right) \text{ AND } \left(y_{cell,hnco} > y_{cell,limit}\right) \tag{24.30}$$

In addition, the following hydrodynamic factors have been shown to affect solids deposit formation trends:

- Intense dynamics can be quantified as fluctuations of film height [149] (p. 5721). When the fluctuations are high, the risk of deposit formation is impeded.
- As reported by Smith et al. [149] (p. 5721), thick film and high film velocity indicates low deposit formation risk, while Brack et al. [176] (p. 5723) report that thick film and low heat flux indicate high deposit formation risk.

Accordingly, Ansys Fluent uses the following definitions to quantify the hydrodynamic risk factors:

- Velocity-based risk r_{vel}

$$r_{vel}=y_{\text{film,urea}}(1-h_{d,film}v_{d,film}) \tag{24.31}$$

- Heat-flux based risk $r_{h\text{-}f}$

$$r_{h\text{-}f}=y_{film,urea}h_{d,film}\left(1-hf_{d,film}\right) \tag{24.32}$$

Equation 24.31 (p. 2886) and Equation 24.32 (p. 2886) use the following nomenclature:

$h_{d,film}$ = (Wall Film Height) / (Maximum Film Height)

$v_{d,film}$ =(Wall Film Velocity) / (Maximum Wall Film Velocity)

$hf_{d,film}$ = (Wall Film Heat Flux) / (Maximum Wall Film Heat Flux)

Finally, averaging over the period from t_0 to t_1 and applying appropriate weighting factors, the total risk r_{total} is calculated as:

$$r_{total}=w_{hydro}r_{hydro}+\left(1\text{-}w_{hydro}\right)r_{chem} \tag{24.33}$$

where:

$$r_{chem}=w_{cryst}r_{cryst}+\left(1\text{-}w_{cryst}\right)r_{sec} \tag{24.34}$$

$$r_{hydro}=w_{h\text{-}f}\overline{r_{h\text{-}f}}+w_{vel}\overline{r_{vel}}+\left(1-w_{h\text{-}f}-w_{vel}\right)\overline{y_{\text{film,urea}}}\left(1-\frac{RMS(h_{film})_{prime}}{\overline{h_{film}}}\right) \tag{24.35}$$

The weighting factors w_{hydro}. w_{cryst}, w_{vel}, and $w_{h\text{-}f}$ are user-specified values.

You can access the functionality for assessing the risk of solids deposit formation via the following text commands:

```
define/models/dpm/options/scr-urea-deposition/enable-risk-analysis? yes
```

You will be asked to define the wall zones where the risk assessment should be performed, for example:

```
Wall face zones(1) [()] 8
```

The implementation uses the data sampling and custom field functions in Ansys Fluent, and after enabling the risk assessment capability, the following steps are automatically performed:

- Defining appropriate solution report definitions for calculating the maximum film height, velocity, and heat-flux.
- Defining the required custom field functions for calculating the risk factors.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Enabling data sampling on the custom field functions.

Once the SCR urea deposition risk analysis is enabled, you can set the following parameters and weighting factors (available under `define/models/dpm/options/scr-urea-deposition`):

- `cryst-min-temp` and `cryst-max-temp`: the minimum and maximum film temperatures for the onset of urea crystallization (K), respectively
- `cryst-min-mass-fract`: the minimum urea mass fraction for the onset of urea crystallization as defined in [149] (p. 5721) and [133] (p. 5720)
- `seco-rx-min-hnco`: the minimum HNCO mass fraction in the gas phase above the liquid film for secondary reactions ($y_{cell,limit}$ in Equation 24.30 (p. 2885))
- `seco-rx-min-temp` and `seco-rx-max-temp`: the minimum and maximum film temperatures for secondary reactions (K), respectively (T_{min} and T_{max} in Equation 24.30 (p. 2885))
- `hydrodynamic-risk-weight`: weighting factor for hydrodynamic risk (w_{hydro} in Equation 24.33 (p. 2886))
- `cryst-depo-weight`: weighting factor for crystallization risk (w_{cryst} in Equation 24.34 (p. 2886))
- `velocity-based-risk-weight`: weighting factor for velocity based risk (w_{vel} in Equation 24.35 (p. 2886))
- `heat-flux-based-risk-weight`: weighting factor for heat-flux based risk (w_{h-f} in Equation 24.35 (p. 2886))
- `wall-face-zones`: wall zone list for solids deposit risk prediction

After the time-dependent calculation is completed, the following variables will be available under **Custom Field Functions...** category for post-processing:

- **scr-urea-overall-hydrodynamic-deposition-risk**: is r_{hydro} in Equation 24.35 (p. 2886)
- **scr-urea-overall-chemistry-deposition-risk**: is r_{chem} in Equation 24.34 (p. 2886)
- **scr-urea-total-deposition-risk**: r_{total} in Equation 24.33 (p. 2886)

Additional field variables reported under the **Custom Field Functions...** and **SCR Urea Deposition Risk Analysis...** categories are the following:

- **scr-urea-depo-risk-velocity-based** (in the **Custom Field Functions...** category): r_{vel} in Equation 24.31 (p. 2886)
- **scr-urea-depo-risk-heat-flux-based** (in the **Custom Field Functions...** category): r_{h-f} in Equation 24.32 (p. 2886)
- **SCR Urea Crystallization Risk** (in the **SCR Urea Deposition Risk Analysis...** category): is the chemistry based risk representation r_{cryst} in Equation 24.29 (p. 2885)
- **SCR Urea Secondary Reactions Risk** (in the **SCR Urea Deposition Risk Analysis...** category): is the chemistry based risk representation r_{sec} in Equation 24.30 (p. 2885)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **SCR Overall Urea Chemical Deposition Risk** (in the **SCR Urea Deposition Risk Analysis...** category): is the chemistry based risk representation r_{chem} in Equation 24.34 (p. 2886)

You may ignore the additional variables with the **scr-urea-depo-** prefix in the **Custom Field Functions...** category as they are intermediate quantities with no meaningful values.

24.10. Parallel Processing for the Discrete Phase Model

Ansys Fluent offers three modes of parallel processing for the discrete phase model:

- **Shared Memory** (Linux systems only)

  The **Shared Memory** method is suitable for computations where the machine running the Ansys Fluent host process is an adequately large, shared-memory, multiprocessor machine.

- **Message Passing**

  The **Message Passing** option is suitable for generic distributed memory cluster computing.

- **Hybrid** (default)

  The **Hybrid** option is suitable for modern multicore memory cluster computing.

These options are found under the **Parallel** tab, in the **Discrete Phase Model** dialog box.

Important:

Note the following:

- When tracking particles with the DPM model in combination with any of the multiphase flow models (VOF, mixture, or Eulerian) the **Shared Memory** method cannot be selected. (Note that using the **Message Passing** or **Hybrid** method enables the compatibility of all multiphase flow models with the DPM model.)

- Starting in version 18.2, the **Shared Memory** method is not compatible with serial UDFs when running in serial.

If you want to use **file** injections or to write a DPM summary report directly to a file, your system must meet the following requirements when either the **Message Passing** or **Hybrid** parallel DPM tracking option is selected:

- The machine (for example, cluster node) running the compute-node zero process and the machine running the host process are identical or have the same operating system type (Windows or Linux)

- Both machines can access the same file system through the same path specification

If these conditions are not met, you can still write a DPM summary report to a file by starting transcript, reporting the particle tracking summary to the Fluent console, and then stopping transcript. The summary report will be saved in the transcript file.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

As a workaround for the DPM summary report, you can start transcript, report the particle tracking summary to the Fluent console, and stop transcript. The summary report will be saved in the transcript file

The **Shared Memory** option (Figure 24.74: The Shared Memory Option with Workpile Algorithm Enabled (p. 2890)) is implemented using POSIX Threads (pthreads) based on a shared-memory model. Once the **Shared Memory** option is enabled, you can then select along with it the **Workpile Algorithm** and specify the **Number of Threads**. By default, the **Number of Threads** is equal to the number of compute nodes specified for the parallel computation. You can modify this value based on the computational requirements of the particle calculations. If, for example, the particle calculations require more computation than the flow calculation, you can increase the **Number of Threads** (up to the number of available processors) to improve performance. When using the **Shared Memory** option, the particle calculations are entirely managed by the Ansys Fluent host process. You must make sure that the machine executing the host process has enough memory to accommodate the entire mesh.

Important:

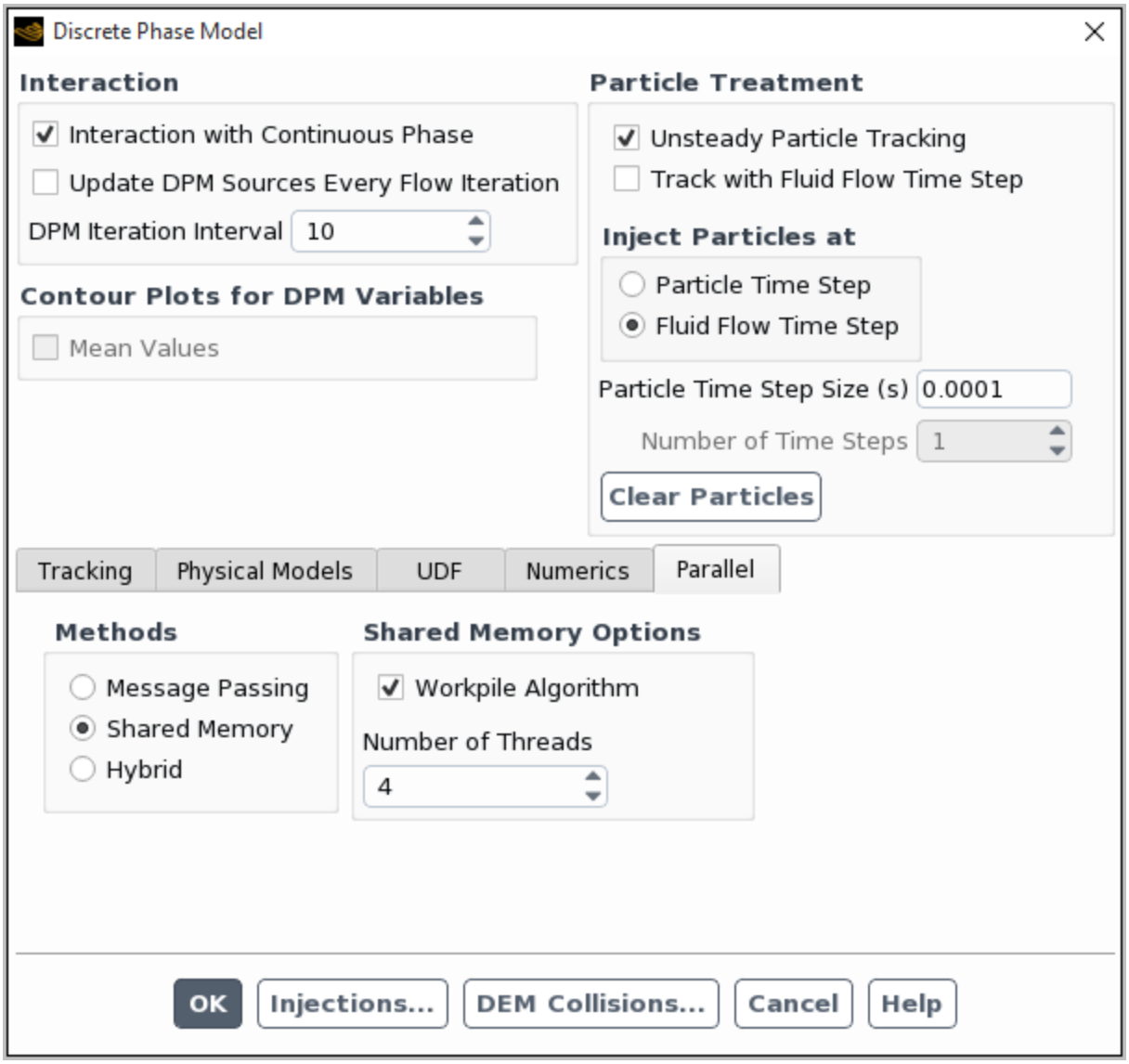
Note that the **Shared Memory** option on Windows-based architectures only provides serial tracking on the host, since the **Workpile Algorithm** is not available due to lack of POSIX Threads on these platforms.

Important:

Note that the **Workpile Algorithm** option is not available with the wall film boundary condition. It will be disabled automatically when choosing to simulate a wall film on a wall.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 24.74: The Shared Memory Option with Workpile Algorithm Enabled



The **Message Passing** option enables cluster computing and also works on shared-memory machines. With this option enabled, the compute node processes perform the particle work on their local partitions. Particle migration to other compute nodes is implemented using message passing primitives. There are no special requirements for the host machine. By default, pathline displays are computed in serial on the host node. Pathline displays may be computed in parallel on distributed memory systems if the **Message Passing** parallel option is selected in the **Discrete Phase Model** dialog box.

The **Hybrid** option combines **Message Passing** and **OpenMP** for a dynamic load balancing without migration of cells, enables multicore cluster computing, and also works on shared-memory machines. With this option enabled, the compute node processes perform the particle calculations on their local partitions. **OpenMP** threads will be spawned, and the number of threads in each Ansys Fluent node process is based on the evaluation of the particle load on the current machine. The maximum number of threads on each machine can be controlled using the **Thread Control** dialog box (see Controlling the Threads (p. 4345) for details). The default value is the number of Ansys Fluent node processes on each machine. Particle migration to other compute nodes is implemented using message passing primitives. There are no special requirements for the host machine. Pathline displays are computed by default in

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

serial on the host node. Pathline displays may also be computed in parallel on distributed memory systems if the **Message Passing** option is selected from the **Methods** list. For a list of limitations that exist with Hybrid parallel method, see Limitations on Using the Hybrid Parallel Method (p. 2676).

Note that for fully coupled flow simulations, the **Hybrid** tracking option may cause slightly different results between runs. The reason for this difference is the non-commutative nature of summations performed with finite-precision floating-point numbers. This becomes more pronounced with the accumulation of particle source term contributions. The order in which particles are tracked and deposit their sources is determined by the execution order of the OpenMP threads. This order varies from run to run, which leads to small changes in the accumulated DPM source terms. Due to the non-linear nature of many physical processes, these differences in the DPM sources may increase and become noticeable as variations in the simulation results.

To avoid the variations between runs, you can issue the following text command:

```
define/models/dpm/parallel/fix-source-term-accumulation-order?
```

Note that using this option may lead to an execution time increase.

When using the Hybrid method, you may optionally enable **Use DPM Domain** under **Hybrid Options**. This option can provide substantially improved load balancing, and thus scalability, at the expense of additional memory overhead. When **Use DPM Domain** is enabled, particle tracking is performed on a separate domain from the root computational domain. Flow and DPM variables are copied between the domains as part of the solution process. By using a separate domain, and thus partitioning strategy, for particle tracking the continuous and discrete phase loads are balanced independently. This allows the load to be shared more equally among the machines regardless of how particles are distributed throughout the computational domain. This is especially beneficial for simulations with non-uniform particle distributions over the computational domain. For limitations associated with the **Use DPM Domain** option, see Limitations on Using the Hybrid Parallel Method (p. 2676). With this option, you may also want to use the following text commands:

- `parallel` → `partition` → `set` → `dpm-load-balancing`

  This text command enables dynamic load balancing. You will be prompted to define a load imbalance threshold (that is, a percentage above which load balancing will occur), as well as the interval of DPM iterations at which you would like load balancing attempted. It is recommended that you retain the default values, as these are appropriate for a wide range of cases.

- `define` → `models` → `dpm` → `parallel` → `expert` → `partition-method-hybrid-2domain`

  This enables a partitioning method that is more granular and can yield faster calculations (especially for cases that are running on a low to moderate number of processors).

You may seamlessly switch among the **Shared Memory** option, the **Message Passing** option, and the **Hybrid** option at any time during the Ansys Fluent session.

In addition to performing general parallel processing of the Discrete Phase Model, you have the option of implementing DPM-specific user-defined functions in parallel Ansys Fluent. For more information about the parallelization of DPM UDFs, see Parallelization of Discrete Phase Model (DPM) UDFs in the Fluent Customization Manual.

When using the **Message Passing** or the **Hybrid** option you can make use of Ansys Fluent's automated load balancing capability by giving an appropriate weight to the particle steps in each cell. In this case

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the number of particle steps in each partition is considered in the load balancing procedure. Further details can be found in Using the Partitioning and Load Balancing Dialog Box (p. 4319).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 25: Modeling Macroscopic Particles

Traditional Lagrangian discrete phase models only apply when particle sizes are small enough to be regarded as point masses within a single cell and when the total particle volume is insignificant within the flow domain volume. In these applications particle-particle and particle-flow interactions are evaluated in terms of impulse, heat, and mass transfer.

However, the total particle volume must be considered when a particle size is larger than several fluid cells because it will affect all the cells within the flow domain volume. Ansys Fluent Macroscopic Particle Model (MPM) predicts the behavior of large (macroscopic) particles and their interaction with the fluid flow, walls, and with other particles.

This chapter provides an overview of how to use the MPM add-on module in Ansys Fluent. For more information about the theoretical background for the MPM approach, see Modeling Macroscopic Particles.

25.1. Overview and Limitations

25.2. Loading the MPM add-on Module

25.3. Setting up Macroscopic Particle Model Simulations

25.4. Modeling Macroscopic Particles

25.1. Overview and Limitations

The MPM is an add-on module based on a suite of UDF functions that models transport of large particles in a fluid flow, particle-particle and particle-wall interactions, as well as particle tracking. The MPM is designed for macroscopic particles that are well-resolved across a particle diameter by at least 20-30 fluid cells.

The MPM is not a general purpose model. The model has been shown to give good results for applications in which macroscopic particles with Reynolds number $Re_p << 1$ move in a flow having a fluid-to-particle density ratio, ρ_f / ρ_p, in the order of 1. For other values of Re_p and ρ_f / ρ_p, the drag prediction may not be accurate; in such cases the MPM may be used when drag is not important.

Note the following limitations when using the MPM:

- Mass transfer and radiation cannot be modeled.
- The MPM is not compatible with moving/deforming meshes.
- Simulations of densely-packed particles are not supported because only one collision event is handled for each particle time step.
- There is no direct interaction with DPM particles (collision).
- Coupling with Multiphase models is possible, but manual settings are required for source terms (mass, momentum, and heat).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Sub-iterations of MPM particle tracks within one time step are not supported.
- The MPM is not compatible with mesh interfaces.

25.2. Loading the MPM add-on Module

The MPM add-on module is installed with the standard installation of Ansys Fluent in a directory called `addons/mpm`. The module can be loaded only when a valid Ansys Fluent case file has been set or read.

Note:

The MPM is only available in 3D.

There are two different ways in which you can load the MPM module into Ansys Fluent:

- Through the ribbon

  In the **Physics** ribbon tab, click **More** (**Models** group), and select **Macroscopic Particle Model...**.

  Physics → **Models** → **More** → **Macroscopic Particle Model...**

  Once the MPM module is loaded into Ansys Fluent, the **Macroscopic Particle Model** dialog box opens where you can enable and set the MPM model as described in Modeling Macroscopic Particles (p. 2895).

- Through the text user interface (TUI)

  Issue the following TUI command in the Fluent console:

  `define` → `models` → `addon-module`

  A list of Ansys Fluent add-on modules is displayed in the console:

```
Fluent Addon Modules:
 0. none
 1. MHD Model
 2. Fiber Model
 3. Fuel Cell and Electrolysis Model
 4. SOFC Model with Unresolved Electrolyte
 5. Population Balance Model
 6. Adjoint Solver
 7. Single-Potential Battery Model
 8. Dual-Potential MSMD Battery Model
 9. PEM Fuel Cell Model
 10.Macroscopic Particle Model
Enter Module Number: [0] 10
```

  Select the Macroscopic Particle Model by entering the module number `10`.

During the loading process, a Scheme library containing the graphical and text user interface, and a UDF library containing a set of user-defined functions (UDFs) for the MPM add-on module are automatically loaded into Ansys Fluent. This process is reported in the Fluent console. Once the MPM module is loaded into Ansys Fluent, the **Macroscopic Particles** model appears under the **Models** tree branch, and the UDF library also becomes visible as a new entry in the **UDF Library Manager** dialog box. By default, the **Macroscopic Particles** model is enabled.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

25.3. Setting up Macroscopic Particle Model Simulations

The following procedure provides an overview of the steps required for setting up and solving an MPM case. Only the steps that are pertinent to MPM are shown here. For information about inputs related to other models that you want to use in conjunction with the MPM, see the appropriate sections for those models in the Ansys Fluent User's Guide (p. 1).

1. Select the **Solution** workspace in the Fluent Launcher.
2. Click **Mesh** (under **Getting Started With...**) to select a 3D mesh file and click **Start With Selected Options**.
3. Scale the mesh, if necessary.
4. For most cases except for cemented particles simulations, select the **Transient** solver from the **Time** list in the **General** task page.
5. Define the boundary conditions and physical properties for fluid flow as usual.
6. (optional) Initialize the flow variables and run the flow simulation until the flow converges.

   This step could be performed to improve solution stability. Using a converged flow solution as a starting point for an MPM simulation can be particularly helpful in situations where there is a strong influence of macroscopic particles on the fluid flow.
7. Use the **Macroscopic Particle Model** dialog box to specify the parameters for the MPM simulation.
8. Initialize either the MPM as described in the sections that follow or the flow field in the usual way.
9. Run the MPM simulation.
10. Save the case and data files, if desired.
11. Review the simulation results by generating plots or alphanumeric reports using the Ansys Fluent postprocessing facilities.

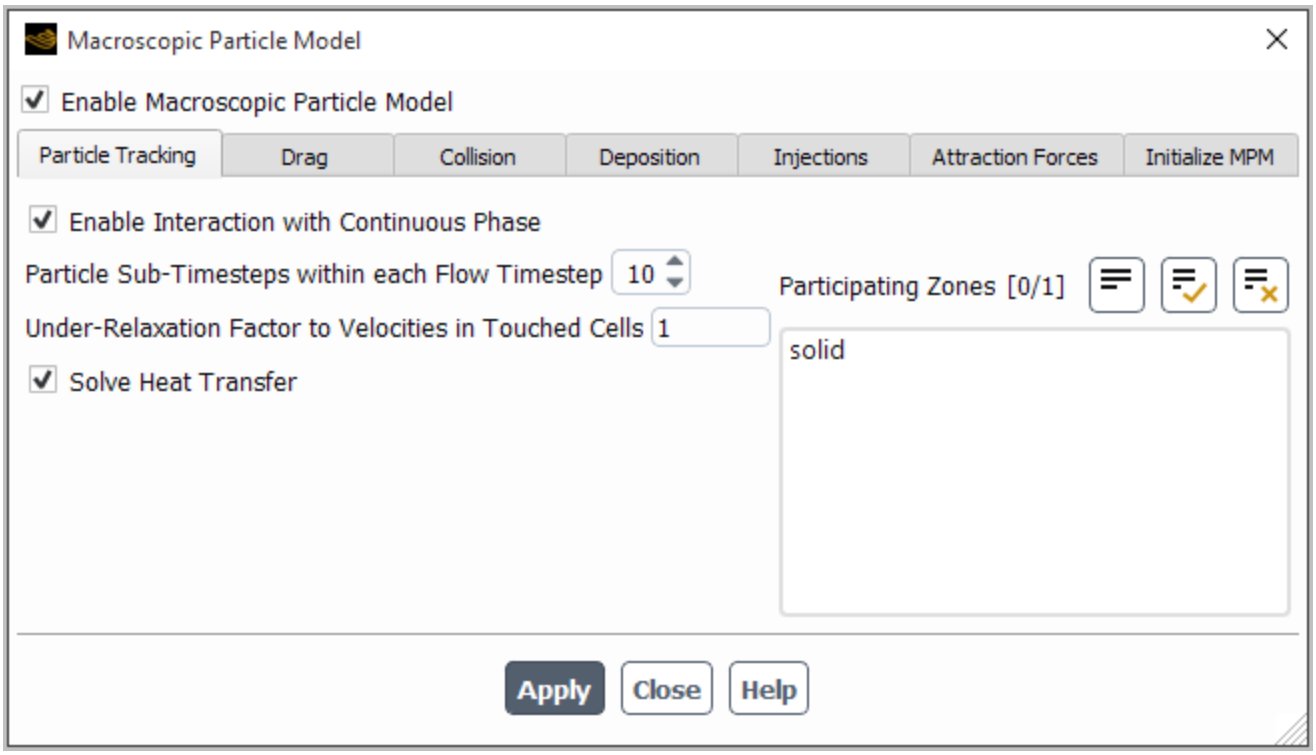
25.4. Modeling Macroscopic Particles

You can set the MPM parameters and select the options that are appropriate for your simulation using the **Macroscopic Particle Model** dialog box that can be accessed either through the ribbon (as described in Loading the MPM add-on Module (p. 2894)) or from the **Setup**/**Models**/**Macroscopic Particle** tree item. From the **Macroscopic Particle Model** dialog box (**Injection** tab), you can access the **Create/Modify Injection** dialog boxes, where you can set the macroscopic particle injection properties (see Specifying Injection Parameters (p. 2903) for details).

Setup → **Models** → **Macroscopic Particle** **Edit...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 25.1: Macroscopic Particle Model Dialog Box (Particle Tracking Tab)



The inputs for the MPM are entered using the following tabs:

Particle Tracking

is where you can define parameters for particle tracking.

Drag

contains drag law options and parameters for the particle drag calculations.

Collision

allows you to enable particle-particle and particle-wall collisions and define the collision parameters.

Deposition

allows you to enable particle deposition on selected surfaces and particles and define the relevant parameters.

Injections

allows you to define macroscopic particle injections.

Attraction Forces

is where you can specify parameters for particle-particle and particle-wall attraction forces.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Initialize MPM

contains controls for initializing the macroscopic particles and displaying particle injections.

Note:

The default settings may not be appropriate for your case. You must provide parameter data for your specific material.

For additional information, see the following sections:

25.4.1. Specifying Particle Tracking Parameters
25.4.2. Specifying the Drag Law
25.4.3. Defining Parameters for Particle-Particle and Particle-Wall Collisions
25.4.4. Specifying Deposition Parameters
25.4.5. Specifying Injection Parameters
25.4.6. Defining Field Forces
25.4.7. Initializing the MPM

25.4.1. Specifying Particle Tracking Parameters

You can use the **Particle Tracking** tab to control the solution process of the MPM trajectory equations using the following settings (see Figure 25.1: Macroscopic Particle Model Dialog Box (Particle Tracking Tab) (p. 2896)):

Particle Zones

contains a selectable list of the cell zones in which you can use the MPM.

Enable Interaction with Continuous Phase

(if selected) enables two-way coupling between particles and fluid flow.

Particle Sub-Timesteps within each Flow Timestep

is a number of sub-time steps for particle tracking in each flow time step.

Under-Relaxation Factor to Velocities in Touched Cells

By default the under-relaxation factor for fixing velocities is set to 1.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solve Heat Transfer

enables solving the energy equation. When this option is enabled, you will need to specify **Initial Temperature** and **Specific Heat** for a particle material in the **Create/Modify Injection** dialog box.

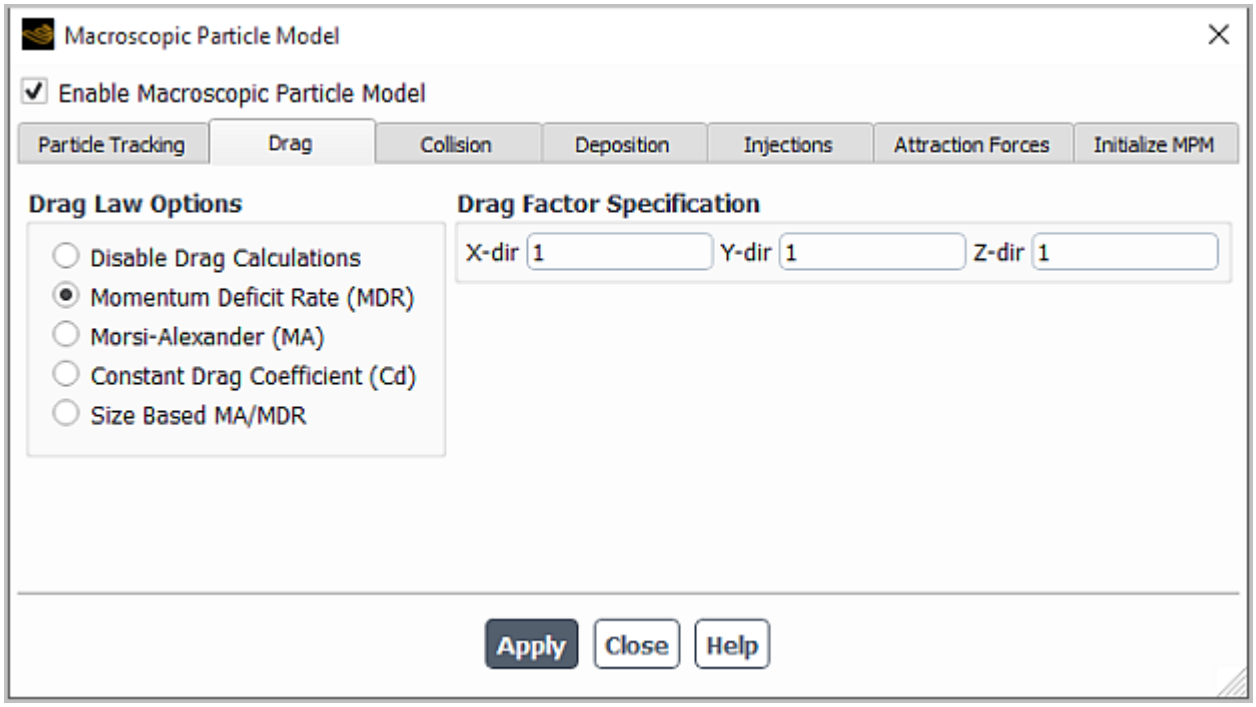
Note:

The particle time step should be chosen so that macroscopic particles do not cross more than one CFD mesh cell.

25.4.2. Specifying the Drag Law

Under the **Drag** tab, you can select an appropriate drag law for your simulation and set the relevant parameters or select **Disable Drag Calculations** if you do not want to model the drag force in the domain.

Figure 25.2: Macroscopic Particle Model Dialog Box (Drag Tab)



In the Ansys Fluent MPM, the following formulations are available for modeling the drag forces between the continuous and macroscopic particle phases:

- **Momentum Deficit Rate (MDR)** (default)

  In this formulation, the momentum exchange coefficient between fluid and solid phases K_{ls} is calculated as:

  $$K_{ls} = f / \Delta t$$

  where f is the drag function, and Δt is the macroscopic particle time step size.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The accuracy of the implicit calculations using the Momentum Deficit Rate model depends on number of cells comprising the macroscopic particle. This drag correlation is more suitable for bigger size particles.

- **Morsi-Alexander (MA)**

  In the Morsi-Alexander based drag correlation 462 adopted in Ansys Fluent, the Reynolds number for the fluid phase is calculated by:

$$Re=\rho_f d_p / \mu_f v_{fp}$$

  where,

  ρ_f = fluid density

  μ_f = fluid viscosity

  d_p = particle diameter

  v_{fp} = relative velocity of the discrete and fluid phases

  The Morsi-Alexander based drag correlation is suitable for smaller size particles.

- **Constant Drag Coefficient (Cd)**

  This option implies that the drag is evaluated using a constant drag coefficient. This correlation is suitable for cases where the particle is moving at a certain specified velocity (or acceleration).

- **Size Based MA/MDR**

  When this option is selected, the Ansys Fluent solver automatically switches between the Momentum Deficit Rate and Morsi-Alexander drag laws based on a critical diameter. This drag formulation is best suited for cases involving wide range of particle sizes in a single simulation. For particles with a diameter below the value that you specify for **Critical Diameter**, the Mosi-Alexander drag law will be used. Otherwise, the Momentum Deficit Rate formulation will be used.

You can define a drag factor in a specific direction (**X-dir**,**Y-dir**,**Z-dir**). A drag factor of 1 means that the virtual mass force equalizes the momentum difference of fluid in cells touched by the particle surface within a single flow time step. As a consequence, the particle sub-time step has to be much smaller (about one-fourth) than the momentum relaxation time τ_∞ 490:

$$\tau_\infty=\frac{\left(\rho_p+\rho_f/2\right)d_p^2}{18\mu}$$

where ρ_p and ρ_f are the particle and fluid densities, respectively, d_p is the particle diameter, and μ is the fluid viscosity.

Besides, the accuracy of the predicted drag depends on the number of cells touched by the particle. At least 20-30 cells are required across the particle diameter.

For multiphase flows, you can select an appropriate fluid velocity field to be used in the drag force calculations. The following options are available in the **Multiphase: Drag Based On** group box:

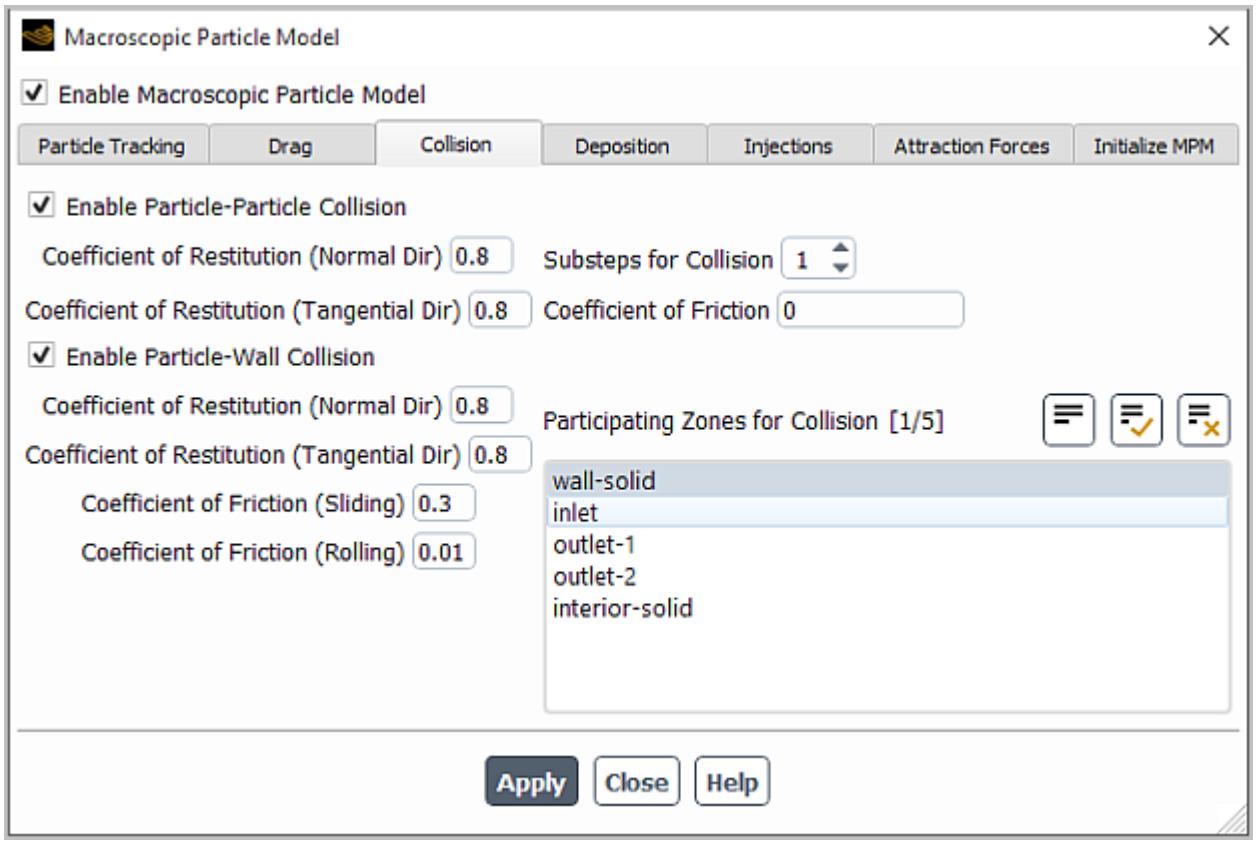
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Primary Phase Velocity**
- **Secondary Phase Velocity**
- **Volume-Weighted Mixture Velocity**

25.4.3. Defining Parameters for Particle-Particle and Particle-Wall Collisions

The **Collision** tab allows you to model the effects of particle-particle and particle-wall collisions.

Figure 25.3: Macroscopic Particle Model Dialog Box (Collision Tab)



To account for collisions between particles:

1. Select **Enable Particle-Particle Collision**.
2. Specify the following parameters:

   Coefficient of Restitution (Normal Dir)

   corresponds to e_n in Equation 13.6.

   Coefficient of Restitution (Tangential Dir)

   corresponds to e_t in Equation 13.7.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Substeps For Collision

is a number of sub-time steps for detecting particle collision in each particle tracking time step.

Coefficient of Friction

corresponds to μ_f in Equation 13.8.

If you also want to account for collisions between particles and walls:

1. Select **Enable Particle-Wall Collision**.
2. From the **Participating Zones For Collision** selection list, select the appropriate walls.
3. In addition to **Coefficient of Restitution (Normal Dir)** and **Coefficient of Restitution (Tangential Dir)** on the selected walls, you can specify the following parameters:

   Coefficient of Friction (Sliding)

   corresponds to μ_f in Equation 13.8.

   Coefficient of Friction (Rolling)

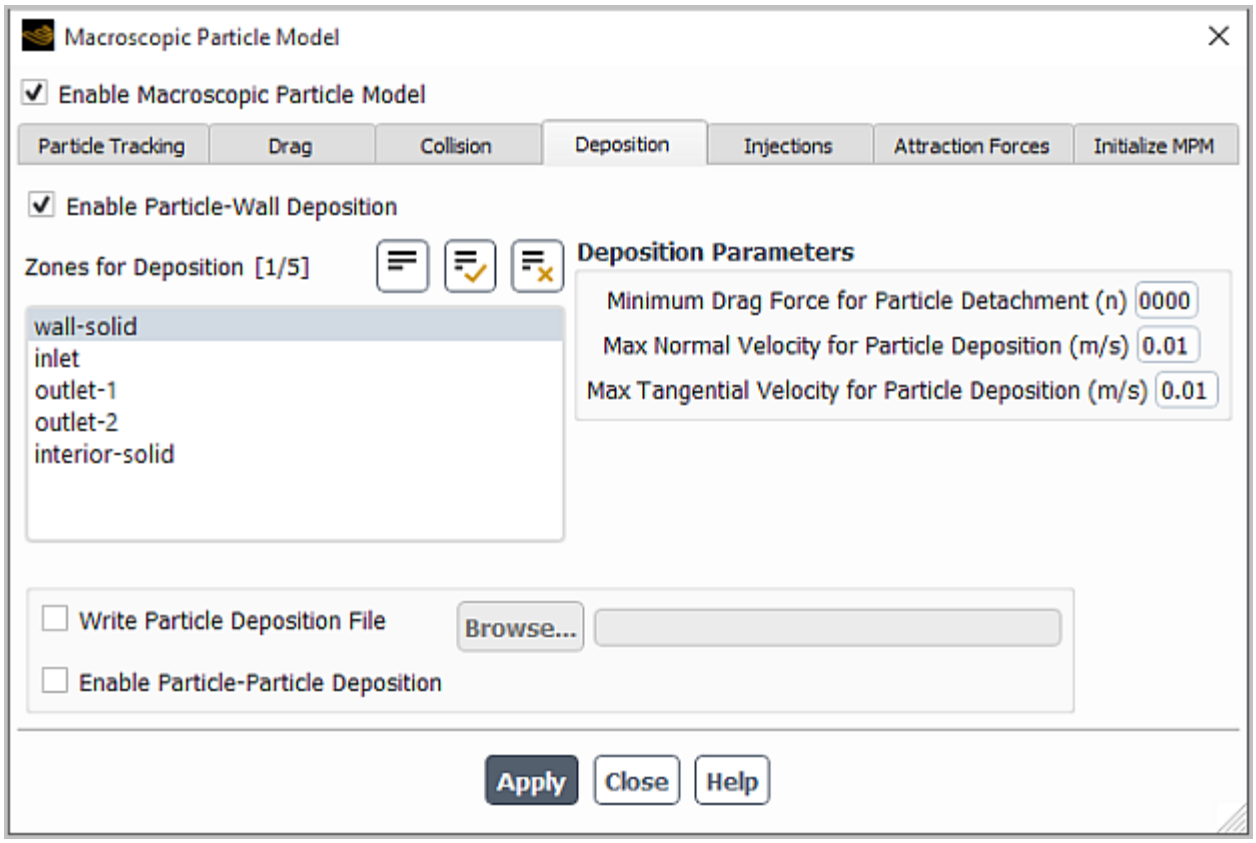
   The value for the rolling coefficient of friction will be factored in the friction coefficient μ_f.

25.4.4. Specifying Deposition Parameters

You can use the **Deposition** tab to specify parameters for macroscopic particle deposition and buildup on selected zones.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 25.4: Macroscopic Particle Model Dialog Box (Deposition Tab)



1. Select **Enable Particle-Wall Deposition**.
2. From the **Zones for Deposition** selection list, select the wall zones on which the macroscopic particles will adhere upon collision.
3. Specify the following deposition parameters:

   Minimum Drag Force for Particle Detachment

   specifies the force at or above which the deposited particles detach from a surface zone.

   Max Normal Velocity for Particle Deposition , Max Tangential Velocity for Particle Deposition

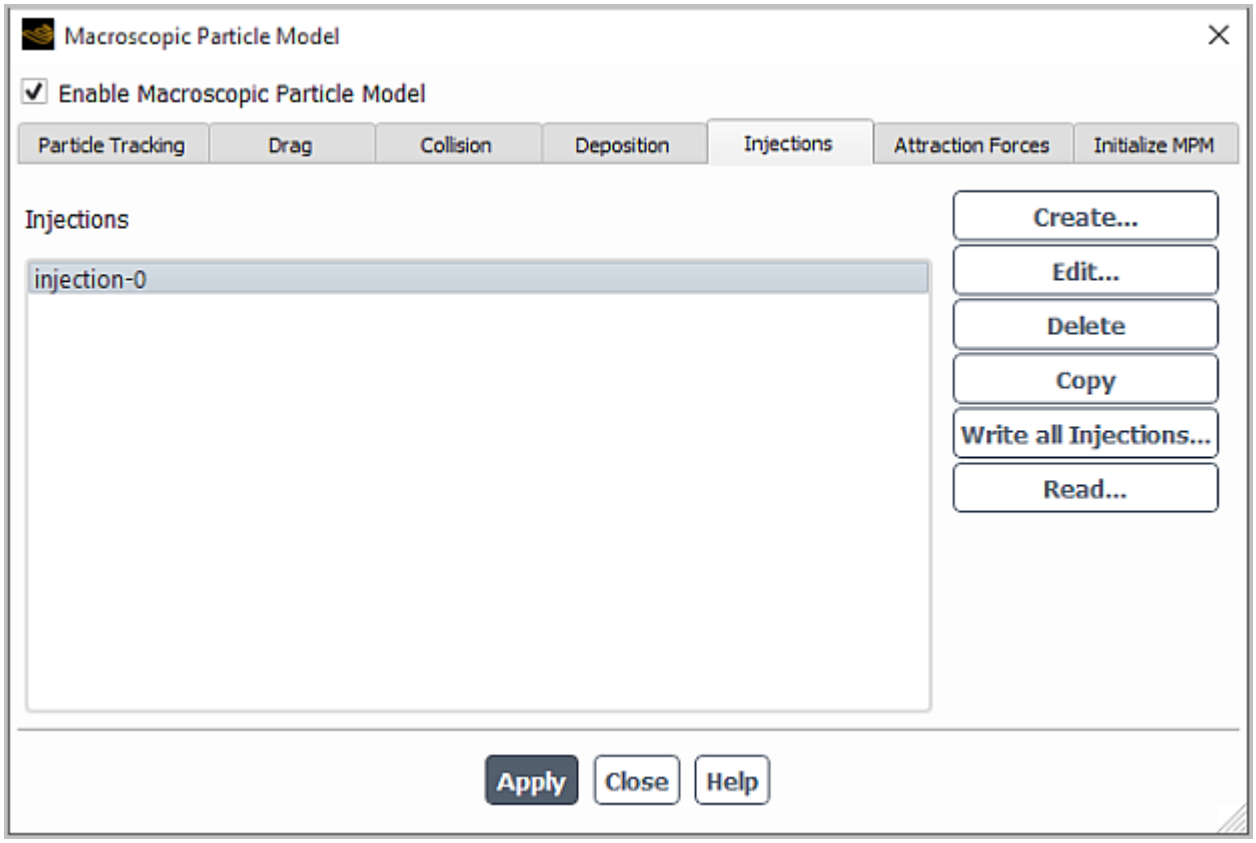
   specify the normal and tangential components of the approach velocity at or below which the particles deposit on the surface.
4. If you want to save particle deposition history data in ASCII format, enable **Write Particle Deposition File**.
5. To account for particle agglomeration, select **Enable Particle-Particle Deposition**. A particle that collides with any previously deposited particles will attach to the particle surface. The deposition parameters that you have specified for particle-wall deposition will be also used for modeling particle-particle deposition.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

25.4.5. Specifying Injection Parameters

You can use the **Injections** tab to create, edit, copy or delete macroscopic particle injections. You can also save all particle injections that you have created to a file for later use or read injections from a previously generated file. The controls under the **Injection** tab are similar to those found in the **Injections** dialog box (see Injections Dialog Box in the *Fluent User's Guide* (p. 5496)) with a few minor differences.

Figure 25.5: Macroscopic Particle Model Dialog Box (Injection Tab)



The procedures for creating, modifying and interacting with injections are also similar to those for the DPM injections. The differences related to the MPM injections will be further emphasized. For more information about using these controls, see Creating and Modifying Injections (p. 2753).

- To create an MPM injection: click **Create...** and set the injection properties using the **Create/Modify Injection** dialog box (see Defining MPM Injection Properties (p. 2904)). After the injection is created, it will appear in the **Injections** selection list, in the **Macroscopic Particle Model** dialog box.
- To edit an existing MPM injection: select it from the **Injections** selection list, click **Edit...**, and in the **Create/Modify Injection** dialog box that will open, modify the injection properties as necessary .
- To copy an existing MPM injection to a new injection: select it from the **Injections** selection list and click **Copy**. The name of the new copied injection appended with _d will appear in the **Injections** list.
- To delete an existing MPM injection: select it from the **Injections** selection list and click **Delete**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To write information about all injections created in a case: click **Write All Injections...** and in the **Select File** dialog box that will open, enter the name for the injection file.

  Note:

  Write All Injections... is not supported for steady flows with cemented particles.

- To read previously generated injection file into your case: click **Read...** and in the **Select File** dialog box that will open, select the injection file to read in. If the injection with the same name already exists in your current case, then Ansys Fluent will rename the imported injection by appending _d to the name.

25.4.5.1. Defining MPM Injection Properties

You can define the properties of an MPM injection using the **Create/Modify Injection** dialog box that can be accessed by clicking **Create...**. The procedure for defining a particle injection is as follows:

1. (Optional) Change the default name of the injection (**`injection-`*`id`***) by entering a new name in the **Injection Name** entry field.
2. From the **Injection Type** drop-down list, select the injection type. In the Ansys Fluent MPM, you can create the following types of injections:
   - `point`: defines an injection released from a single point.
   - `plane`: defines uniformly or randomly distributed injections released from a plane.
   - `packing`: defines an injection of particles packed in a box or cylinder.
   - `from-file`: allows you to read a text file that defines an injection.
3. If you want to model fixed particles, enable **Cemented Particle(s)**.

   Important:

   For steady-state simulations, all particles must be cemented.

4. Specify the particle physical parameters (**Diameter** and **Density**).

   Note:

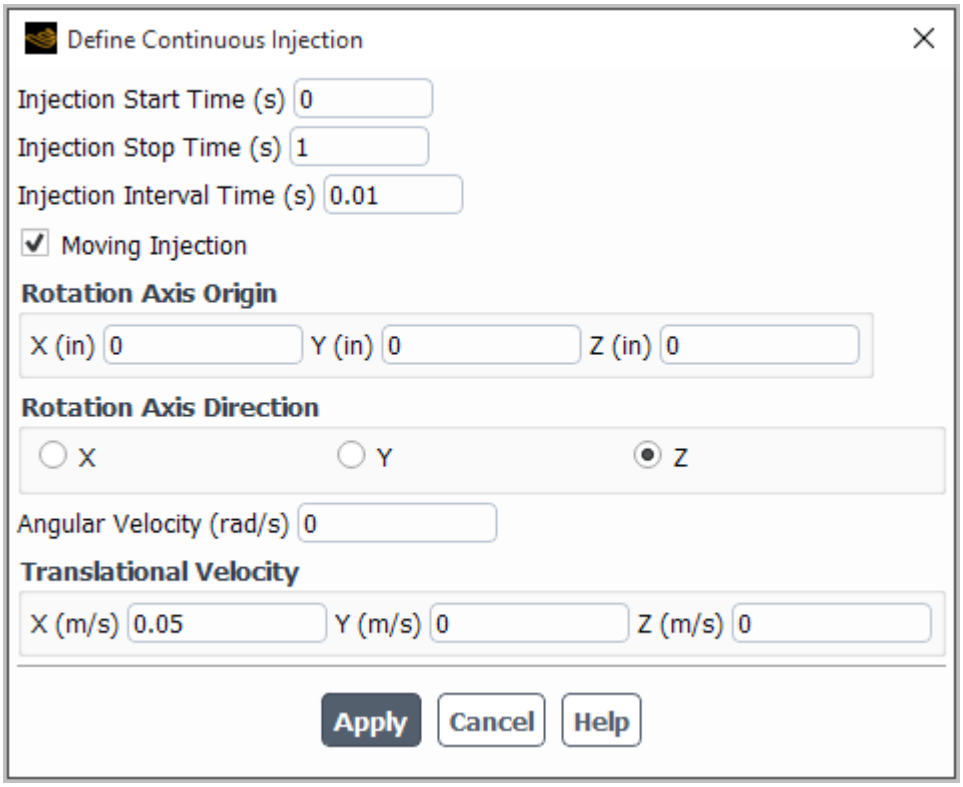
   Note that a material defined in Ansys Fluent cannot be assigned to a particle.

5. Specify the injection type-specific settings as described in the following sections:
   - `point` injections: Inputs for `point` Injections (p. 2906)
   - `plane` injections: Inputs for `plane` Injections (p. 2907)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- `packing` injections: Inputs for `packing` Injections (p. 2909)
- `from-file` injections: Inputs for `from-file` Injections (p. 2910)

6. (Optional) If you want to model a continuous generation of particle injections, select **Enable Continuous Injection** and in the **Define Continuous Injection** dialog box that opens when you click **Set...**, specify the relevant parameters.



You can specify the following settings:

- Injection start, stop, and interval times

  New macroscopic particle will be injected into the domain at regular time intervals specified in **Injection Interval Time** from the **Injection Start Time** to the **Injection Stop Time**.

- (Optional) Moving and/or rotating injection release point parameters (available when **Moving Injection** is enabled):

  – The origin (**X**, **Y**, **Z**) and the direction of the axis (**X**, **Y**, or **Z**) about which the injection is rotating

  – The **Angular Velocity** of the injection location

  – The **X**, **Y**, **Z** components of the **Translational Velocity** of the injection location

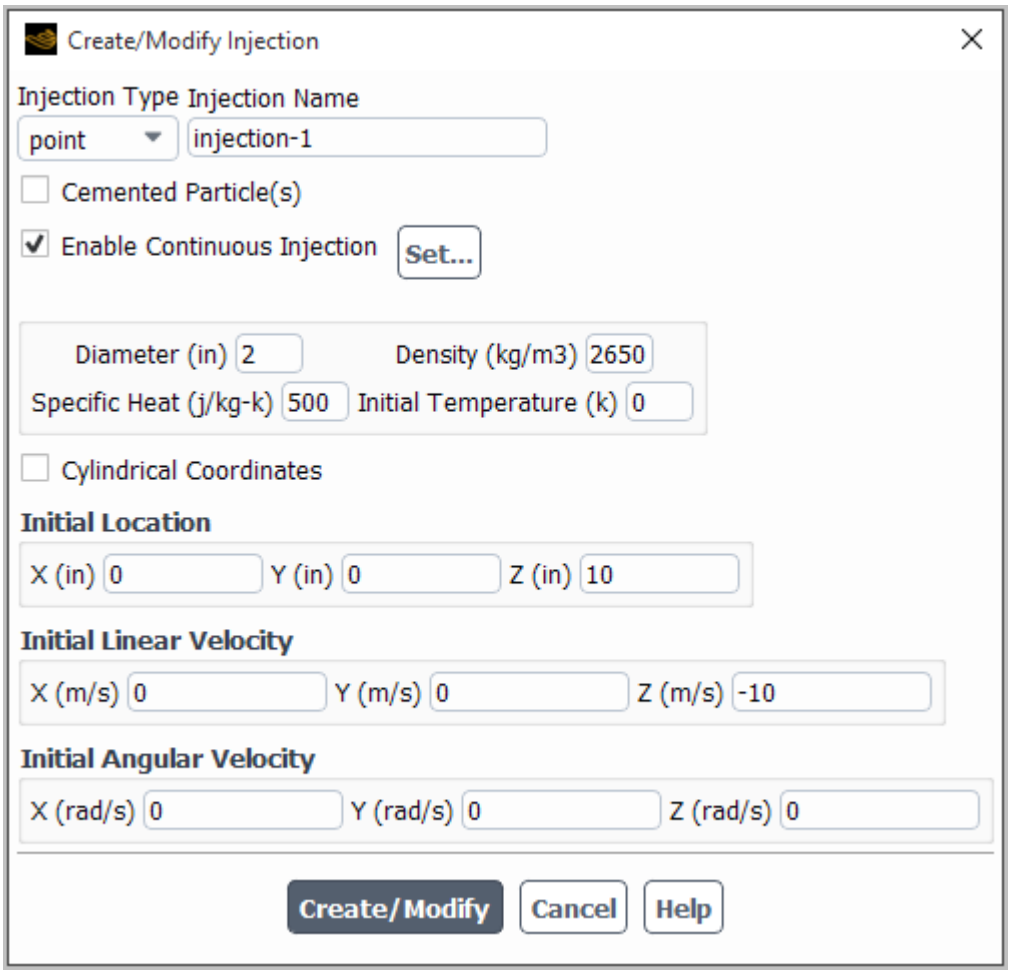
  At each time step, the MPM solver determines a new position of a particle injection based on the injection origin at the previous time step and the specified translational velocities.

7. Click **Create/Modify**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The new injection that you have created appears under the **Injections** selection list.

25.4.5.2. Inputs for `point` Injections



For a point injection of a macroscopic particle, you need to define the following initial conditions in the **Create/Modify Injection** dialog box:

- Particle diameter
- Particle density
- Specific heat of the particle material (only for heat transfer simulations)
- Initial particle temperature (only for heat transfer simulations)
- Initial particle release location:
  - Cartesian coordinates: **X**, **Y**, and **Z**
  - Cylindrical coordinates: **Radius**, **Angle**, and **Axial**
- Particle initial velocity:
  - Cartesian coordinates: **X**, **Y**, and **Z**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

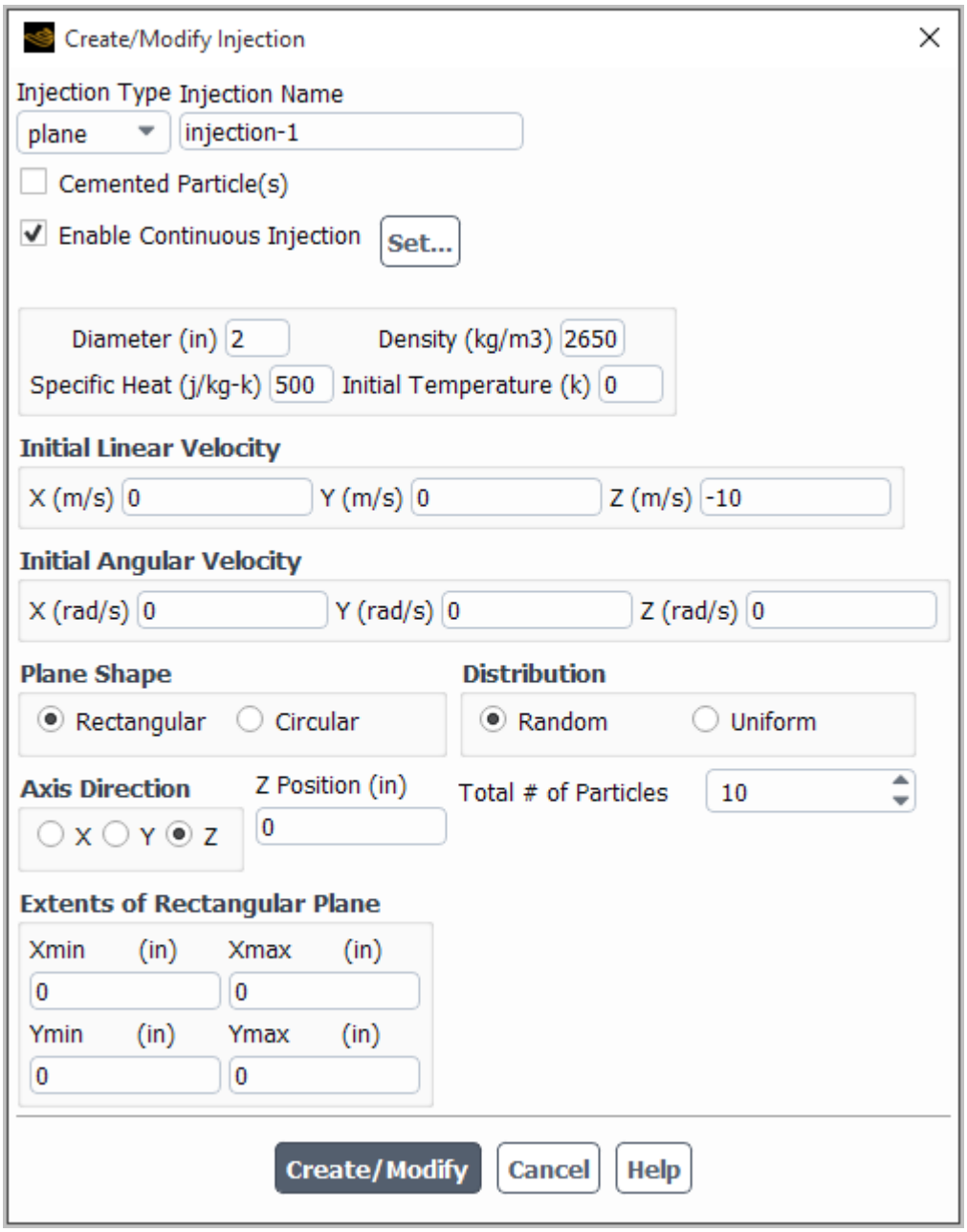
  – Cylindrical coordinates: **Radial**, **Tangential**, and **Axial**

- Particle initial angular velocity

Cylindrical Coordinate System

By default, the global Cartesian coordinate system is used. If you want to specify injection parameters in a local cylindrical coordinate system, enable the **Cylindrical Coordinates** option, and then define the location of its origin (**X**, **Y**, **Z**) and **Axis Direction**.

25.4.5.3. Inputs for `plane` Injections



To define macroscopic particle injections released from a plane:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Specify the particle initial conditions as described in Inputs for `point` Injections (p. 2906), except for the particle release location. Note that for plane injections, velocities and initial position can be specified only in Cartesian coordinates.

2. Define the macroscopic particle injection plane.

   Two options are available:

   - **Rectangular**

     Macroscopic particles will be injected from a rectangular plane surface that is offset from a coordinate plane by a distance specified in **X**, **Y**, or **Z Position** in the selected **X**, **Y**, or **Z** axis direction.

     The minimum and maximum limits of the rectangular surface are defined under the **Extents of Rectangular Plane** group box.

   - **Circular**

     Macroscopic particles will be injected from a circular plane surface that is perpendicular to a selected coordinate direction (**X**, **Y**, or **Z**). The origin and radius of the circle are specified under the **Center Location/Radius of Circular Plane** group box.

3. Define the distribution of the particle release locations.

   Two modeling options are available:

   - **Random**

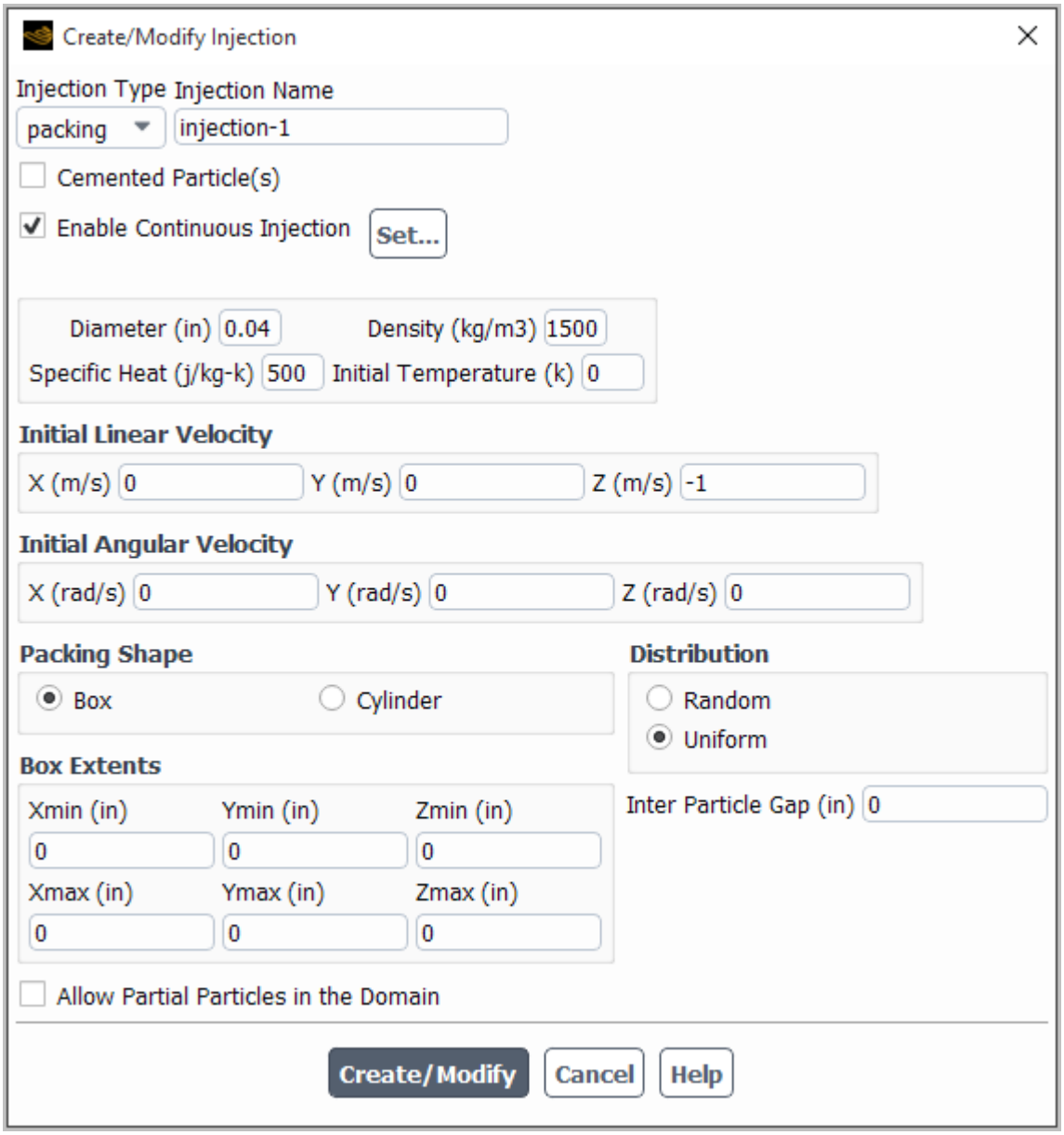
     The MPM will inject particles into a flow domain from random locations on the injection plane. The total number of the released particles is specified in the **Total # of Particles** integer entry field.

   - **Uniform**

     The particles will be injected into a flow domain from points arranged in a rectangular pattern. The number of the release points in two axial directions is specified in the appropriate integer entry fields (for example, **# of Particles in *X*-dir** and **# of Particles in *Y*-dir**).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

25.4.5.4. Inputs for `packing` Injections



To define an injection of macroscopic particles packed in a box or cylinder:

1. Specify particle initial conditions as described in Inputs for `point` Injections (p. 2906), except for the particle release location. Note that for plane injections, velocities and initial position can be specified only in Cartesian coordinates.

2. Define the packing region.

   For a **Box**-type packing, specify the minimum and maximum X, Y, and Z coordinates for the packing bounding box under the **Box Extents** group box.

   For a **Cylinder**-type packing, specify the bounding cylinder axis, minimum and maximum co-ordinates for the cylinder height, cylinder origin and inner and out diameters.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Define the particle distribution in the packing.

   Two modeling options are available:

   - **Random**

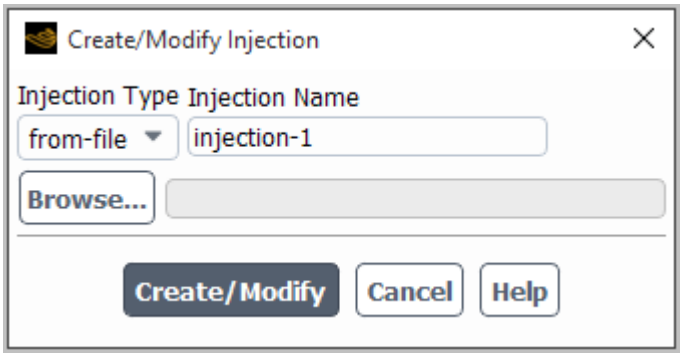
     The initial volume fraction of particles in the packing is specified in the **Particle VOF** entry field.

   - **Uniform**

     The distance between particles in the initial spatially uniform packing is specified in the **Inter Particle Gap** entry field.

4. If you want to allow a part of the particle to be outside of the domain (with the particle center being in the domain), enable **Allow Partial Particles in the Domain**.

25.4.5.5. Inputs for `from-file` Injections



For injections that you want to specify using the `from-file` option, the file format is:

```
number-of-particles
diameter density x-pos y-pos z-pos x-vel y-vel z-vel x-rot y-rot z-rot pstart pstop pinterval cemented
...
```

where the first line specifies the number of particles (`number-of-particles`), followed by `number-of-particles` lines that define parameters for the injection particles. For each particle, you must specify the diameter, density, initial position, linear and angular velocities, start, end, and interval times, and the integer flag (`cemented` in the above file format specification) indicating whether or not the particle is moving. Note that `cemented`=1 indicates a stationary particle, while `cemented`=0 indicates a moving particle. Values describing an injection should be separated by one or more spaces and can be specified using scientific notation (e or E) if needed.

All quantities are in SI units.

An example is shown below:

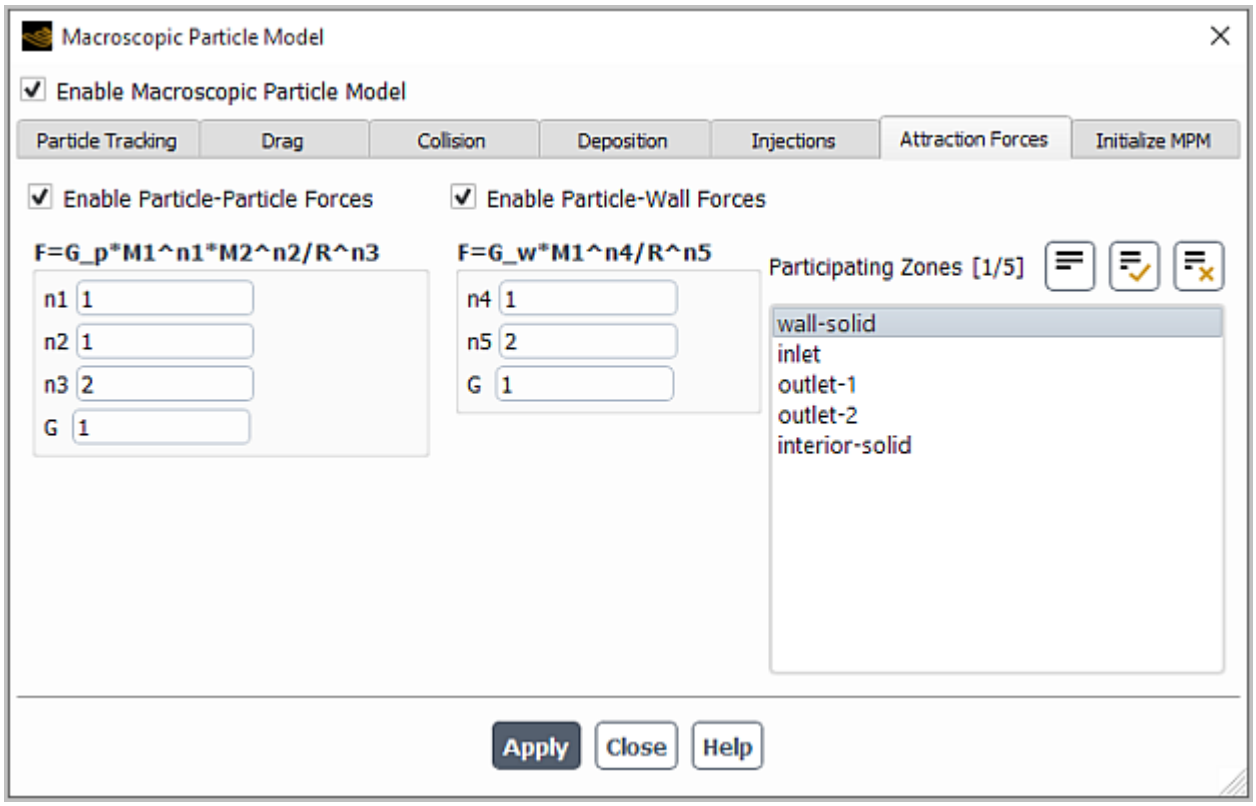
```
1
5e-2 5.08e-2 -1e-8 1e-7 2.4e-1 -2e-4 1e-5 1.1 0.0 0.0 0.0 0.0 1e-2 1e-4 0
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

25.4.6. Defining Field Forces

The **Attraction Forces** tab allows you to include attraction forces between particles and walls in your problem.

Figure 25.6: Macroscopic Particle Model Dialog Box (Attraction Forces Tab)



To define the attraction forces between macroscopic particles:

1. Select **Enable Particle-Particle Forces**.
2. Specify the model constants **n1**, **n2**, **n3**, and **G_p** used in Equation 13.9.

To define the attraction forces between macroscopic particles and walls:

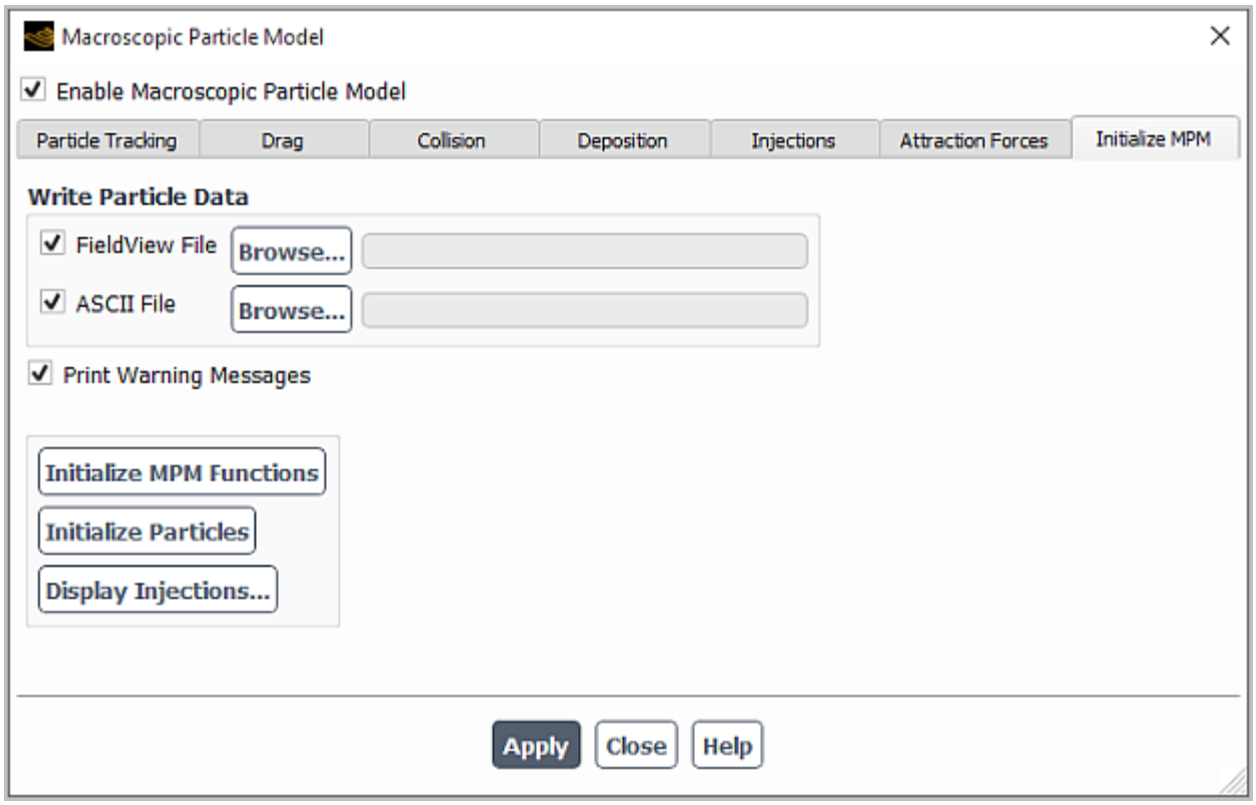
1. Select **Enable Particle-Wall Forces**.
2. From the **Participating Zones** selection list, select the appropriate zones
3. Specify the model constants **n4**, **n5**, and **G_w** used in Equation 13.10.

25.4.7. Initializing the MPM

The **Initialize MPM** tab allows you to apply macroscopic particle related source terms to the continuous phase, initialize macroscopic particles, and display particle trajectories.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 25.7: Macroscopic Particle Model Dialog Box (Initialize MPM Tab)



You can also enable the following solution-related options:

- Saving particle data in a Field View and/or ASCII format
- Printing warning messages related to particle tracking in the Fluent console

Once you have set up the MPM, you need to perform the following steps:

1. Click **Initialize MPM Functions**.

   This will set up macroscopic particle related source terms and apply them to the continuous phase. It will also hook all appropriate UDF functions to the Fluent solver.

   Initializing MPM functions adds the following two commands to the solver that will be executed during simulation:

   - `mpm1`: displays the particle position(s) at each iteration step.
   - `mpm3`: writes the particle position(s) at each iteration step in PNG format.

   You can disable these commands or modify the reporting frequency in the **Execute Command** dialog box (accessible from the **Solution/Calculation Activities/Execute Commands** tree item).

   If you want the solver to report or monitor additional solution quantities during calculation, you can define your own execute command(s) using Fluent TUI commands. For more information, see Executing Commands During the Calculation in the *Fluent User's Guide* (p. 3700).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

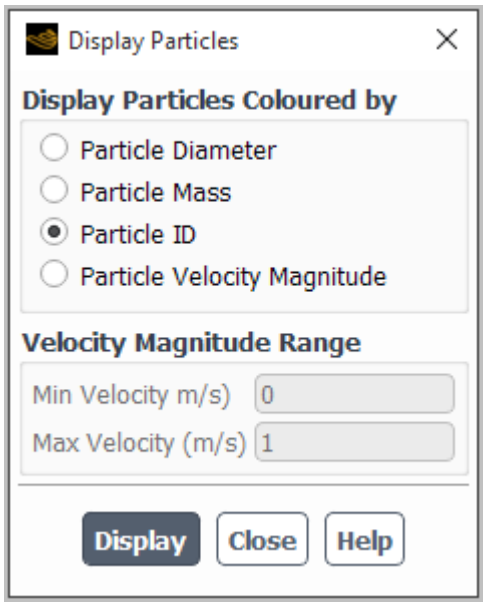
For coupled simulations, once you click **Initialize MPM Functions**, the pressure discretization scheme will automatically switch to **PRESTO!** to provide a more robust solution.

2. Initialize either the macroscopic particles by clicking **Initialize Particles** or the flow field in the usual way

   The macroscopic particle(s) will be introduced into the fluid flow during the next computational time step.

3. Run the simulation with the injected macroscopic particle(s).

Upon completion of the MPM simulation, you can click **Display Injections...** and use the **Display Particles** dialog box to display the computed particle trajectories.



In the **Display Particles** dialog box, you can select the particle coloring option. You can color particle by the following particle variable values:

- diameter
- mass
- particle ID
- velocity magnitude

  For the **Particle Velocity Magnitude** option, you can specify the minimum and maximum range of the velocity magnitude values.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 26: Modeling Multiphase Flows

This chapter discusses the general multiphase models that are available in Ansys Fluent. For information about the various theories behind the general multiphase models in Ansys Fluent, see Multiphase Flows in the Theory Guide.

Information about using the general multiphase models in Ansys Fluent is presented in the following sections:

26.1. Introduction

26.2. Steps for Using a Multiphase Model

26.3. Setting Up the VOF Model

26.4. Setting Up the Mixture Model

26.5. Setting Up the Eulerian Model

26.6. Population Balance Model

26.7. Setting Up the Wet Steam Model

26.8. Solution Strategies for Multiphase Modeling

26.9. Multiphase Case Check

26.10. Postprocessing for Multiphase Modeling

26.1. Introduction

The first step in solving any multiphase problem is to determine which of the regimes described in Multiphase Flow Regimes in the Theory Guide best represents your flow. Model Comparisons in the Theory Guide provides some broad guidelines for determining appropriate models for each regime, and Detailed Guidelines provides details about how to determine the degree of interphase coupling for flows involving bubbles, droplets, or particles, and the appropriate model for different amounts of coupling.

The following sections will guide you through the setup, solution, and postprocessing of multiphase flow models.

26.2. Steps for Using a Multiphase Model

The procedure for setting up and solving a general multiphase problem is outlined below, and described in detail in the subsections that follow. Remember that only the steps that are pertinent to general multiphase calculations are shown here. For information about inputs related to other models that you are using in conjunction with the multiphase model, see the appropriate sections for those models.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See also Additional Guidelines for Eulerian Multiphase Simulations (p. 3065) for guidelines on simplifying Eulerian multiphase simulations.

Important:

Double Precision is recommended for all multiphase cases.

1. Enable the multiphase model you want to use (VOF, mixture, or Eulerian) and specify the number of phases. For the VOF and Eulerian models, specify the volume fraction scheme as well.

   Setup → Models → Multiphase → Edit...

   See Enabling the Multiphase Model (p. 2918) and Choosing Volume Fraction Formulation (p. 2921) for details.

2. Copy the material representing each phase from the materials database.

   Setup → Materials

   If the material you want to use is not in the database, create a new material. See Using the **Create/Edit Materials** Dialog Box (p. 1599) for details about copying from the database and creating new materials. See Modeling Compressible Flows (p. 2931) for additional information about specifying material properties for a compressible phase. It is possible to turn off reactions in some materials by selecting **none** in the **Reactions** drop-down list under **Properties** in the **Create/Edit Materials** dialog box.

   Important:

   If your model includes a particulate (granular) phase, you will need to create a new material for it in the *fluid* materials category (*not* the solid materials category).

3. Define the phases, and specify any interaction between them (for example, surface tension if you are using the VOF model, slip velocity functions if you are using the mixture model, or drag functions if you are using the Eulerian model).

   Setup → Models → Multiphase → Edit...

   See Defining the Phases (p. 2932) – Defining the Phases for the Eulerian Model (p. 3066) for details.

4. (Eulerian model only) If the flow is turbulent, define the multiphase turbulence model.

   Setup → Models → Viscous → Edit...

   See Modeling Turbulence (p. 3089) for details.

5. If body forces are present, enable gravity and specify the gravitational acceleration.

   Setup → Cell Zone Conditions → Operating Conditions...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See Including Body Forces (p. 2933) for details.

6. Specify the boundary conditions, including the secondary-phase volume fractions at flow boundaries and (if you are modeling wall adhesion in a VOF simulation) the contact angles at walls.

   Setup → Boundary Conditions

   See Defining Multiphase Cell Zone and Boundary Conditions (p. 2957) for details.

7. Set any model-specific solution parameters.

   Solution → Methods

   Solution → Controls

   See Setting Time-Dependent Parameters for the Explicit Volume Fraction Formulation (p. 3017) and Solution Strategies for Multiphase Modeling (p. 3184) for details.

8. Initialize the solution and set the initial volume fractions for the secondary phases.

   Solution → Initialization → Patch...

   See Setting Initial Volume Fractions (p. 2974) for details.

9. Calculate a solution and examine the results. Postprocessing and reporting of results are available for each phase that is selected.

   See Solution Strategies for Multiphase Modeling (p. 3184) and Postprocessing for Multiphase Modeling (p. 3211) for details.

Note:

- For information on multiphase model compatibility with other Fluent models, see Appendix A: Ansys Fluent Model Compatibility (p. 5667).
- If the multiphase model is used in combination with a species model, the mass fraction of a species may be displayed incorrectly in Ansys CFD-Post if the volume fraction of a cell is zero. This applies for all Ansys Fluent files written prior to release 19.2.

  For all multiphase, multi-species simulations written in release 19.2 and beyond in the legacy format (`.dat` files), the mass fractions of species will not be available in Ansys CFD-Post, unless you add the mass fraction of a species as an additional quantity to be saved in the data file (through the **Data File Quantities** dialog box).

This section provides instructions and guidelines for using the VOF, mixture, and Eulerian multiphase models.

Information is presented in the following subsections:

26.2.1. Enabling the Multiphase Model

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.2.2. Choosing Volume Fraction Formulation
26.2.3. Solving a Homogeneous Multiphase Flow
26.2.4. Modeling Buoyancy-Driven Multiphase Flow
26.2.5. Modeling Compressible Flows
26.2.6. Defining the Phases
26.2.7. Including Body Forces
26.2.8. Modeling Multiphase Species Transport
26.2.9. Specifying Heterogeneous Reactions
26.2.10. Including Mass Transfer Effects
26.2.11. Defining Multiphase Cell Zone and Boundary Conditions
26.2.12. Setting Initial Conditions

26.2.1. Enabling the Multiphase Model

To enable the VOF, mixture, or Eulerian multiphase model, select **Volume of Fluid**, **Mixture**, or **Eulerian** under **Model** in the **Models** tab of the Multiphase Model Dialog Box (p. 4700).

Setup → **Models** → **Multiphase** → **Edit...**

The dialog box will expand to show the relevant inputs for the selected multiphase model.

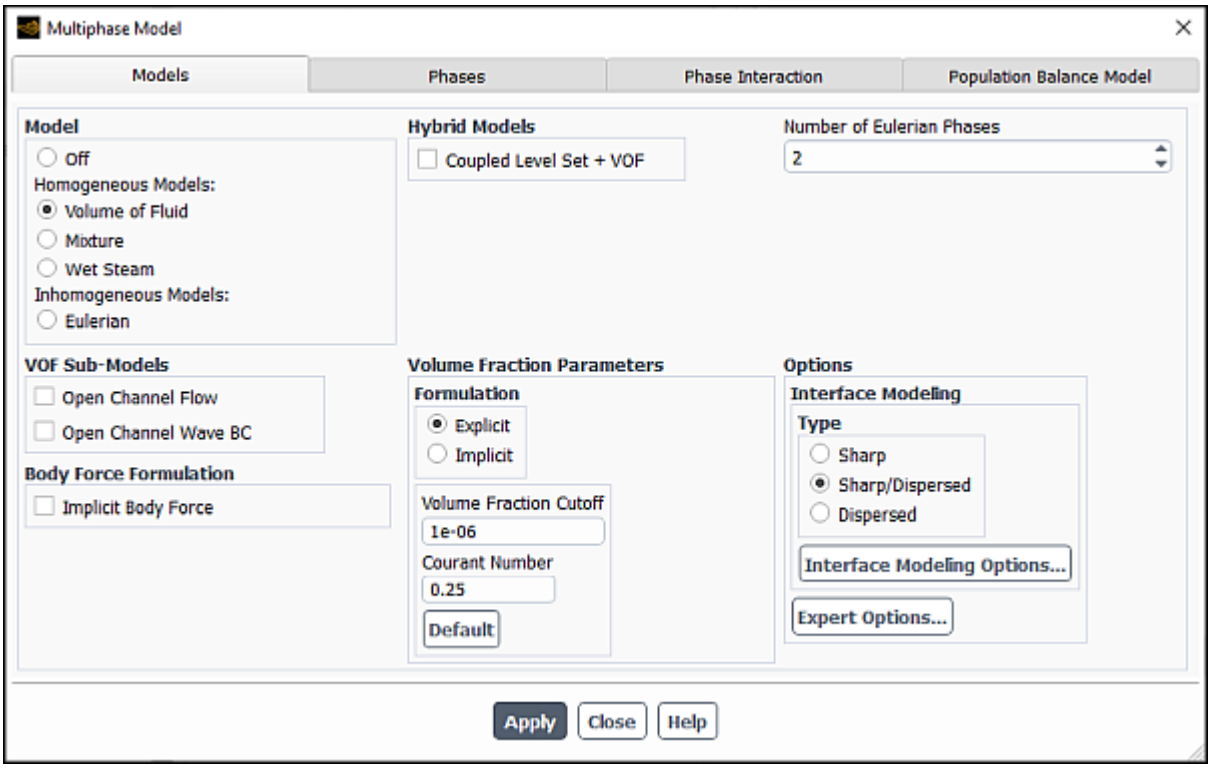
Important:

When setting up your case, if you have made changes in the current tab, you should click the **Apply** button to make them effective before moving to the next tab. Otherwise, the relevant models may not be available in the other tabs, and your settings may be lost.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.2.1.1. Inputs for the VOF Model

Figure 26.1: Multiphase Model Dialog Box for the VOF Model

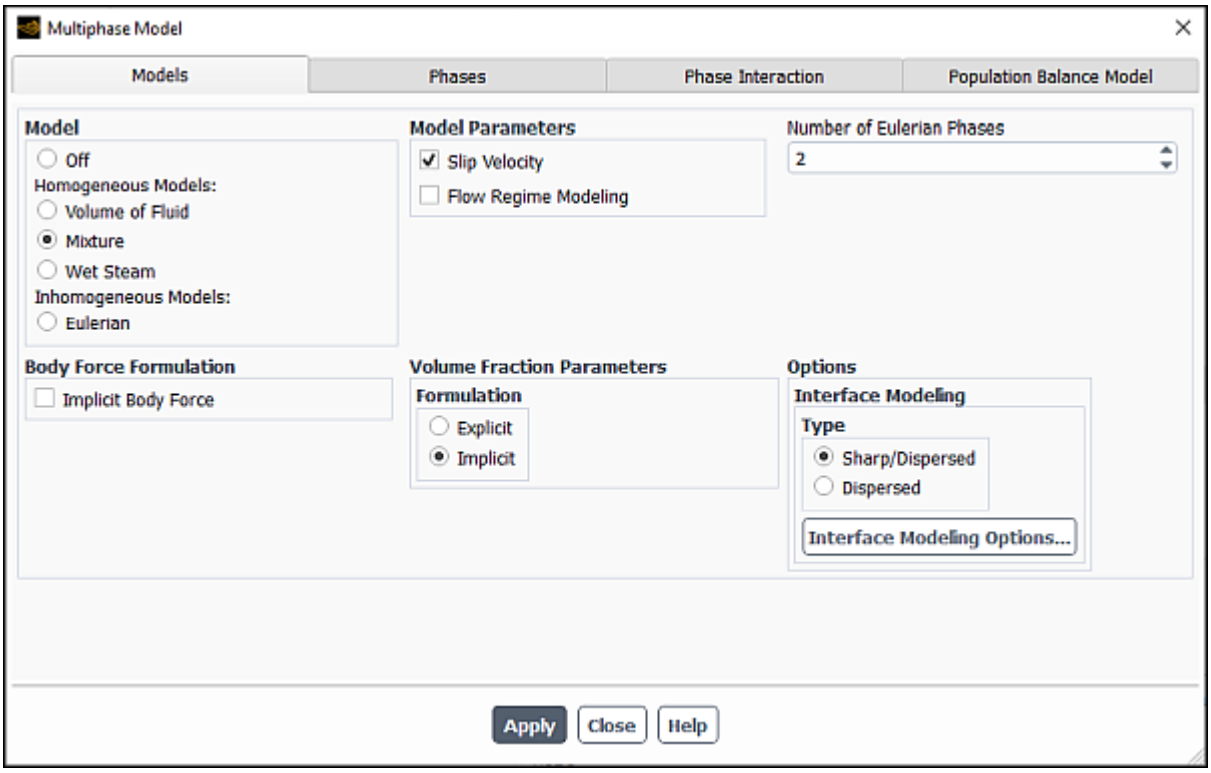


- **Number of Eulerian Phases**
- (optional) **Coupled Level Set + VOF** (see Coupled Level-Set and VOF Model in the Theory Guide)
- volume fraction **Formulation** (**Explicit** or **Implicit**) and associated parameters (see Choosing Volume Fraction Formulation (p. 2921))
- (optional) inclusion of sub-models such as **Open Channel Flow** and **Open Channel Wave BC**
- **Interface Modeling** type and options (see Choosing Volume Fraction Formulation (p. 2921))
- (optional) selection of **Expert Options** (see Expert Options (p. 2926))
- (optional) inclusion of the **Implicit Body Force** formulation (see Including Body Forces (p. 2933))

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.2.1.2. Inputs for the Mixture Multiphase Model

Figure 26.2: Multiphase Model Dialog Box for the Mixture Model

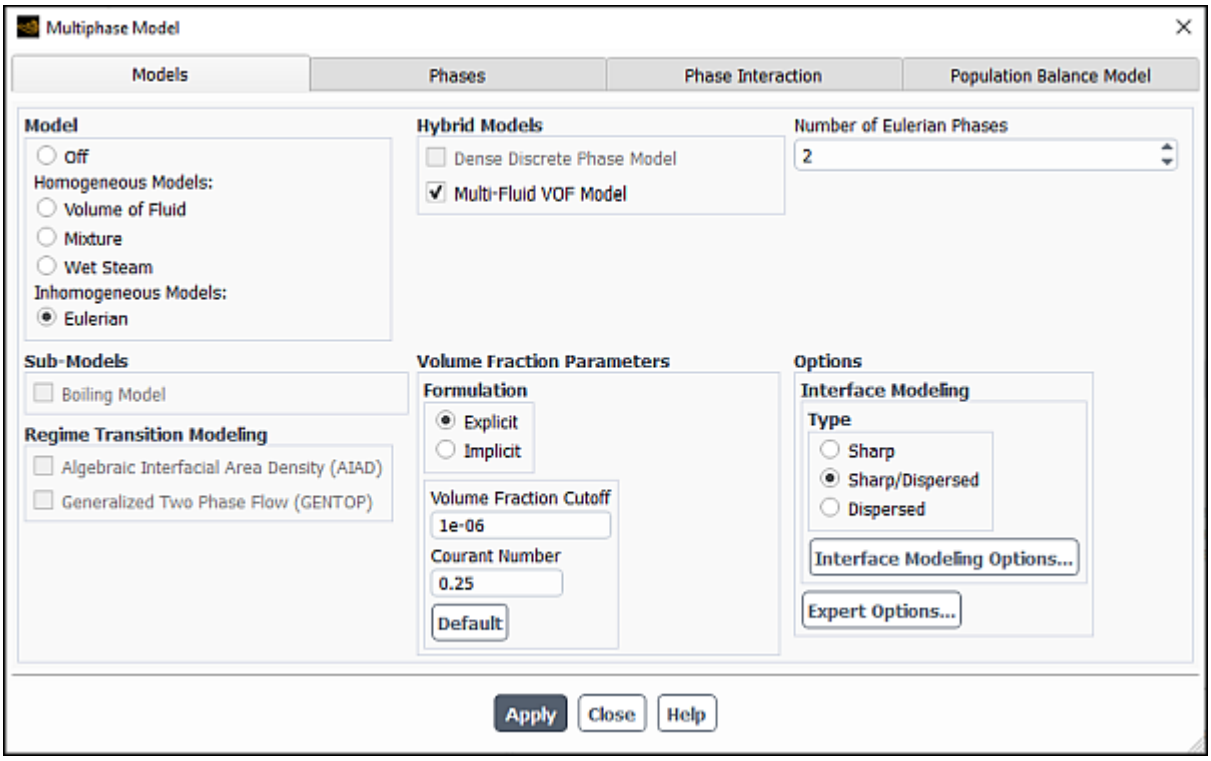


- **Number of Eulerian Phases**
- (optional) inclusion of **Slip Velocity** (see Solving a Homogeneous Multiphase Flow (p. 2929))
- volume fraction **Formulation** (**Explicit** or **Implicit**) and associated parameters (see Choosing Volume Fraction Formulation (p. 2921))
- **Interface Modeling** type and options (see Choosing Volume Fraction Formulation (p. 2921))
- (optional) selection of **Expert Options** (see Expert Options (p. 2926))
- (optional) inclusion of the **Implicit Body Force** formulation (see Including Body Forces (p. 2933))

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.2.1.3. Inputs for the Eulerian Multiphase Model

Figure 26.3: Multiphase Model Dialog Box for the Eulerian Model



- **Number of Eulerian Phases**
- (optional) inclusion of the **Dense Discrete Phase Model**
- (optional) inclusion of the **Boiling Model**
- (optional) inclusion of the **Multi-Fluid VOF Model**
- volume fraction **Formulation** (**Explicit** or **Implicit**) and associated parameters (see Choosing Volume Fraction Formulation (p. 2921))
- **Interface Modeling** type and options (see Choosing Volume Fraction Formulation (p. 2921)). This is only available if the **Multi-Fluid VOF Model** is enabled.
- (optional) selection of **Expert Options** (see Expert Options (p. 2926))

To specify the number of phases for the multiphase calculation, enter the appropriate value in the **Number of Eulerian Phases** field. You can specify up to 20 phases.

26.2.2. Choosing Volume Fraction Formulation

Ansys Fluent supports the following options for volume fraction formulation:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Explicit Formulation

The explicit formulation is non-iterative and is time-dependent. Thus it can only be used with the **Transient** solver. It exhibits better numerical accuracy compared to the implicit formulation. However the time step size is limited by a Courant-based stability criterion.

Implicit Formulation

The implicit formulation is iterative and can be used with either the **Steady** or **Transient** solver. It is well-suited to steady-state applications as the solution information propagates much faster compared to the explicit formulation.

For steady-state flow applications, either a steady-state or a transient simulation may be required, depending on the characteristics of the flow:

- *Steady-state simulation*: If the final steady-state solution is not affected by the initial flow conditions and there is a distinct inflow boundary for each phase.
- *Transient simulation*: If the final steady-state solution is dependent on the initial flow conditions and/or you do not have a distinct inflow boundary for each phase.

For transient flow applications, the implicit formulation may allow you to use a much larger time step size than the explicit formulation due to the implicit formulation's unconditional stability. However, the implicit time-step size may be limited by truncation errors. The implicit formulation produces more numerical diffusion than the explicit formulation when using the **First Order** transient formulation. Therefore, the implicit formulation should be used along with higher-order transient formulations to achieve better numerical accuracy.

To specify the volume fraction formulation to be used for the VOF and Eulerian multiphase models, select the appropriate **Formulation** under **Volume Fraction Parameters** in the **Models** tab of the **Multiphase Model** dialog box.

26.2.2.1. Interface Modeling Type

For the VOF and Mixture multiphase models and for the Eulerian multiphase model with **Multi-Fluid VOF** enabled, you must specify the interface regime that you will be modeling. This will determine the availability of the various volume fraction discretization schemes (Spatial Discretization Schemes for Volume Fraction (p. 2924)). The following options are available for **Interface Modeling Type**:

Sharp

(VOF and Multi-Fluid VOF models only) for when a distinct interface is present between the phases

Dispersed

for when the phases are interpenetrating

Sharp/Dispersed

a hybrid approach for flows consisting of both sharp and dispersed interfaces. This option can also be used to capture mildly sharp interfaces. Mildly sharp interfaces are those that

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

are neither as sharp as would be captured by the schemes available with the **Sharp** option, nor as diffused as would be captured by the schemes available with the **Dispersed** option.

Interfacial Anti-Diffusion

If you are using the Sharp interface modeling type, you can optionally enable the **Interfacial Anti-Diffusion** treatment. This treatment is applied only in interfacial cells and attempts to suppress the numerical diffusion that can arise from the volume fraction advection schemes.

The use of this treatment can have adverse effects on convergence, especially with aggressive settings and large time step size. Therefore, it should be used in cases that use a coarse mesh, that have high aspect-ratio cells, that have large cell-volume jumps in the vicinity of the fluid-fluid interface, or that suffer from excess numerical diffusion.

The strength of the anti-diffusion treatment can be specified using the `define/models/multiphase/interface-modeling-options` text command. The strength can range between `0` (none) and `1` (maximum).

The anti-diffusion treatment can sometimes produce artificial waves due to excessive sharpening. The dynamic anti-diffusion strength option may help in such cases by reducing the anti-diffusion strength in the direction tangential to the interface. Dynamic strength can be enabled using the following text commands.

```
solve/set/multiphase-numerics/advanced-stability-controls/anti-diffusion/
enable-dynamic-strength?

solve/set/multiphase-numerics/advanced-stability-controls/anti-diffusion/enable-dynamic-strength?
Enable anti-diffusion treatment with dynamic strength? [no] yes
```

Dynamic strength is specified as:

$$C = \min\left[(\mathrm{Cos}\theta)^m,\ C_{\max}\right]$$
$$\mathrm{Cos}\theta = \left|\hat{n}\cdot\hat{A}\right| \tag{26.1}$$

where

C = anti diffusion strength

$\hat{n}$ = interface unit normal

$\hat{A}$ = area unit normal

θ = angle between the area and interface unit normals

m = exponent (default = 1.5)

$C_{\max}$ = maximum value of anti-diffusion strength ranging between 0 and 0.95 (default = 0.75)

Values of the exponent and maximum strength can be adjusted using the following text commands, if needed:

`solve/set/multiphase-numerics/advanced-stability-controls/anti-diffusion/set-dynamic-strength-exponent`

Sets the cosine exponent in the dynamic strength treatment (m in Equation 26.1 (p. 2923)).

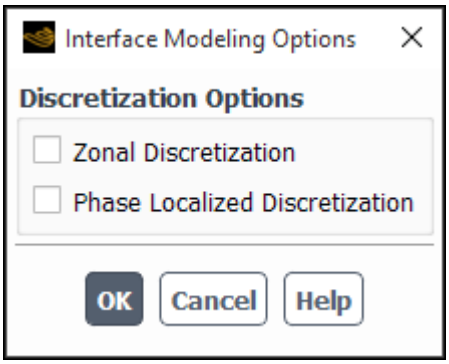
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

solve/set/multiphase-numerics/advanced-stability-controls/anti-diffusion/set-maximum-dynamic-strength

Sets the maximum value of dynamic anti-diffusion strength ($C_{\max}$ in Equation 26.1 (p. 2923)).

Sharp/Dispersed Interface Modeling Options

Selecting **Sharp/Dispersed** will make available some additional discretization options. To access these, click **Interface Modeling Options...** in the **Multiphase Model** dialog box.



These options allow you to adjust the discretization behavior on a per-zone and/or per-phase-pair basis for cases in which it is important to simultaneously model dispersed and sharp interfaces.

Zonal Discretization

This option is appropriate for applications that require either sharp or dispersed interface modeling depending on the cell zone. After enabling **Zonal Discretization** you will need to specify the slope limiter value on the **Multiphase** tab of the **Fluid** dialog box for each cell zone. (See Defining Multiphase Cell Zone and Boundary Conditions (p. 2957))

Phase Localized Discretization

This option is appropriate for applications that require either sharp or dispersed interface modeling depending on the phase-pair comprising the interface. After enabling **Phase Localized Discretization** you will need to specify the slope limiter values for each phase pair in the **Phase Interaction** > **Discretization** tab of the **Multiphase Model** dialog box.

26.2.2.2. Spatial Discretization Schemes for Volume Fraction

You can choose from various spatial discretization schemes for volume fraction depending on the multiphase model in use, the interface regime type, and the volume fraction formulation chosen. The available discretization schemes for each combination of model, formulation, and interface regime are summarized in the following tables:

Table 26.1: Spatial Discretization Schemes for the VOF and Eulerian with Multi-Fluid VOF Models

Interface Modeling Type	Implicit Formulation	Explicit Formulation
Sharp	Compressive BGM (steady state only) Modified HRIC	Geo-Reconstruct CICSAM

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Interface Modeling Type	Implicit Formulation	Explicit Formulation
		Compressive Modified HRIC
Sharp/Dispersed	Compressive Modified HRIC	
Dispersed	First Order Upwind Second Order Upwind QUICK	First Order Upwind QUICK

Table 26.2: Spatial Discretization Schemes for the Eulerian Model without Multi-Fluid VOF

Implicit Formulation	Explicit Formulation
First Order Upwind QUICK Modified HRIC	First Order Upwind QUICK Modified HRIC Compressive

Table 26.3: Spatial Discretization Schemes for the Mixture Model

Interface Modeling Type	Implicit Formulation	Explicit Formulation
Sharp/Dispersed	Compressive Modified HRIC	
Dispersed	First Order Upwind QUICK	

The **Geo-Reconstruct** scheme is an interface tracking scheme based on geometrical information. It is the most accurate scheme, but is more computationally expensive than the other schemes. **Geo-Reconstruct** is the preferred scheme when solving on meshes of poor quality.

The **CICSAM**, **Modified HRIC**, and **Compressive** schemes are interface capturing schemes based on algebraic information.

The Donor-Acceptor discretization is a scheme applicable only to quad- and hex-meshes. As it is considered obsolete, it does not appear by default in the graphical user interface. However, it can be made available if required using the following text command: `solve/set/advanced/show-all-discretization-schemes`. This command can also be used to expose discretization schemes that may be applicable, but are hidden by default according to the tables above.

Important:

For the geometric reconstruction and donor-acceptor schemes, if you are using a conformal mesh (that is, if the mesh node locations are identical at the boundaries where two subdomains meet), you must ensure that there are no two-sided (zero-thickness)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

walls within the domain. If there are, you will need to slit them, as described in Slitting Face Zones (p. 1273).

For more details about the implementations of the volume fraction discretization schemes, refer to Interpolation Near the Interface in the *Fluent Theory Guide*.

26.2.2.3. Volume Fraction Limits

The **Volume Fraction Cutoff** allows you to specify a cutoff limit for the volume fraction values. The value that you provide is used as the lower cutoff for the volume fraction. All volume fraction values in the domain below this cutoff value are set to zero. The upper cutoff is calculated as (1.0 - lower cutoff). All volume fraction values above the upper cutoff value are set to 1.0. The default value is 1e-6, which is the recommended value. Using a higher value may lead to a higher volume imbalance.

Important:

The **Volume Fraction Cutoff** value can be specified only with the **Explicit** formulation for volume fraction.

Note:

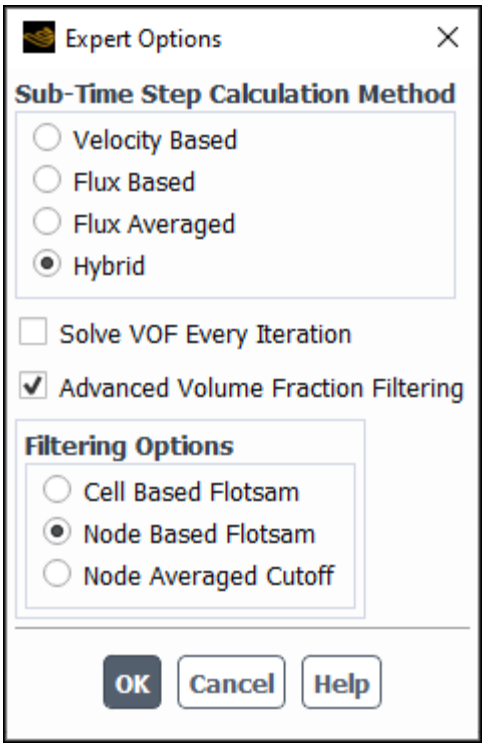
For the **Implicit** formulation, the minimum value allowed is 0 and the maximum allowable value is 1e-6. For the **Explicit** formulation, the minimum value allowed is 0 and the maximum allowable value is 1e-4. A higher cutoff value is available with the **Explicit** formulation because it allows for the local redistribution procedure of partially filled cells, which accounts for volume loss. This treatment is not available with the **Implicit** formulation.

26.2.2.4. Expert Options

When using the **Explicit** formulation, there are several additional options that you can access in the **Expert Options** dialog box that opens by clicking **Expert Options...** in the **Multiphase Model** dialog box (**Models** tab).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.4: Expert Options Dialog Box



Sub-Time Step Calculation Method

This setting controls how the time-scale is computed when determining the time step used for volume fraction.

Solve VOF Every Iteration

This setting controls whether to solve the volume fraction equations once per iteration, or only once per time step.

For a more detailed discussion of how to use these options, refer to Setting Time-Dependent Parameters for the Explicit Volume Fraction Formulation (p. 3017).

In addition, you can enable **Advanced Volume Fraction Filtering** if you want to filter out small values of the volume fraction, which may be purely numerical causing solution instability. Small-scale numerical errors in the volume fraction field may arise due to the various reasons:

- Time truncation errors at large time step size
- Insufficient number of sub-time integration steps
- Limiting and normalization of the volume fraction field

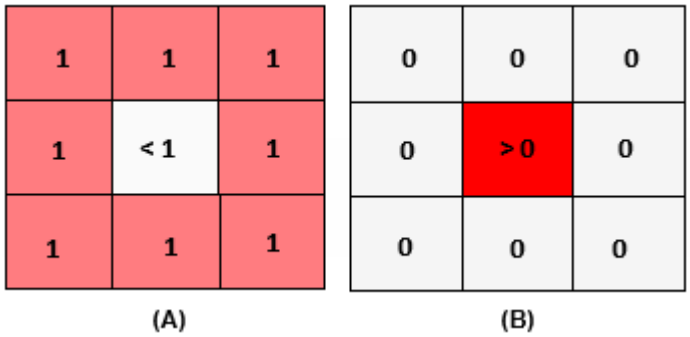
Ansys Fluent uses the flotsam filtering method to identify whether these errors are physical or numerical. The method is based on a principle that an interface cannot be constructed in a single cell.

Figure 26.5: Numerical Flotsams in the Volume Fraction Field (p. 2928) depicts generation of a numerical flotsam. In scenario **A**, the neighboring cells are completely full, while the central cell is not full. In scenario **B**, the neighboring cells are completely void, but the central cell contains a non-

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

zero volume fraction. Both scenarios cannot exist unless there is a mass source/sink present in a single cell.

Figure 26.5: Numerical Flotsams in the Volume Fraction Field



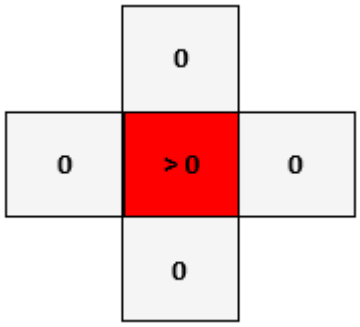
Once a cell is identified as a flotsam, its volume fraction is removed from the cell and distributed to the partial cells (that is, cells with the volume fraction between 0 and 1).

You can select one of the following **Filtering Options**:

Cell Based Flotsam

Flotsams are identified using only face-connected neighbors (see Figure 26.6: The Volume Fraction Field for the Cell Based Flotsam Detection (p. 2928)). This method is rather aggressive, and there is a possibility of filtering out a ligament that passes through a central cell diagonally (in both scenarios **A** and **B** shown in Figure 26.5: Numerical Flotsams in the Volume Fraction Field (p. 2928)).

Figure 26.6: The Volume Fraction Field for the Cell Based Flotsam Detection

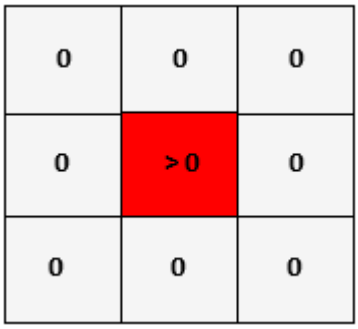


Node Based Flotsam

Flotsams are identified using node-connected neighbors (see Figure 26.7: The Volume Fraction Field for the Node Based Flotsam Detection (p. 2929)). This method is quite conservative, and there is no possibility of filtering out a ligament that passes a central cell diagonally (in both scenarios **A** and **B** shown in Figure 26.5: Numerical Flotsams in the Volume Fraction Field (p. 2928)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.7: The Volume Fraction Field for the Node Based Flotsam Detection

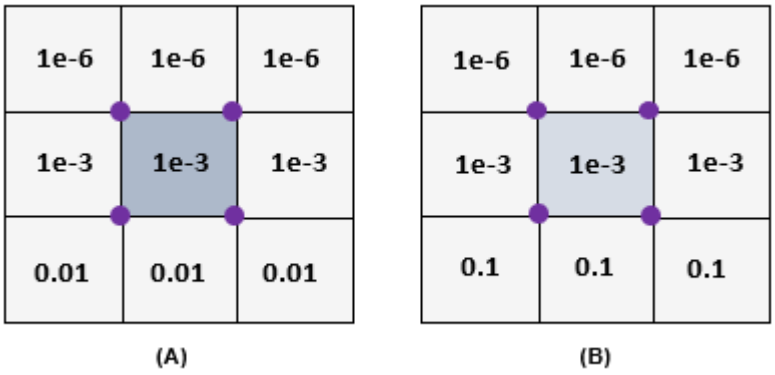


Node Averaged Filtering Method

This method filters out the volume fraction in a cell when the node-averaged volume fraction in the cell falls below a specified cutoff value.

For example, Figure 26.8: The Volume Fraction Field for the Node-Averaged Filtering (p. 2929) shows different scenarios for a cutoff value of 0.01. In the left-hand side (**A**), the central cell qualifies for the filtering criterion because the node-averaged value in the central cell is below 0.01; and therefore, the volume fraction will be removed from the cell and distributed in the neighboring cells. In the right-hand side (**B**), the central cell does not fall under the cutoff criterion, so its volume fraction will remain intact.

Figure 26.8: The Volume Fraction Field for the Node-Averaged Filtering



If you select this method, you can adjust the **Volume Fraction Filtering Cutoff**. The default value is 0.001.

26.2.3. Solving a Homogeneous Multiphase Flow

If you are using the mixture model, you have the option to disable the calculation of slip velocities and solve a homogeneous multiphase flow (that is, one in which the phases all move at the same velocity). By default, Ansys Fluent will compute the slip velocities for the secondary phases, as described in Relative (Slip) Velocity and the Drift Velocity in the Theory Guide. If you want to solve a homogeneous multiphase flow, turn off **Slip Velocity** under **Mixture Parameters**.

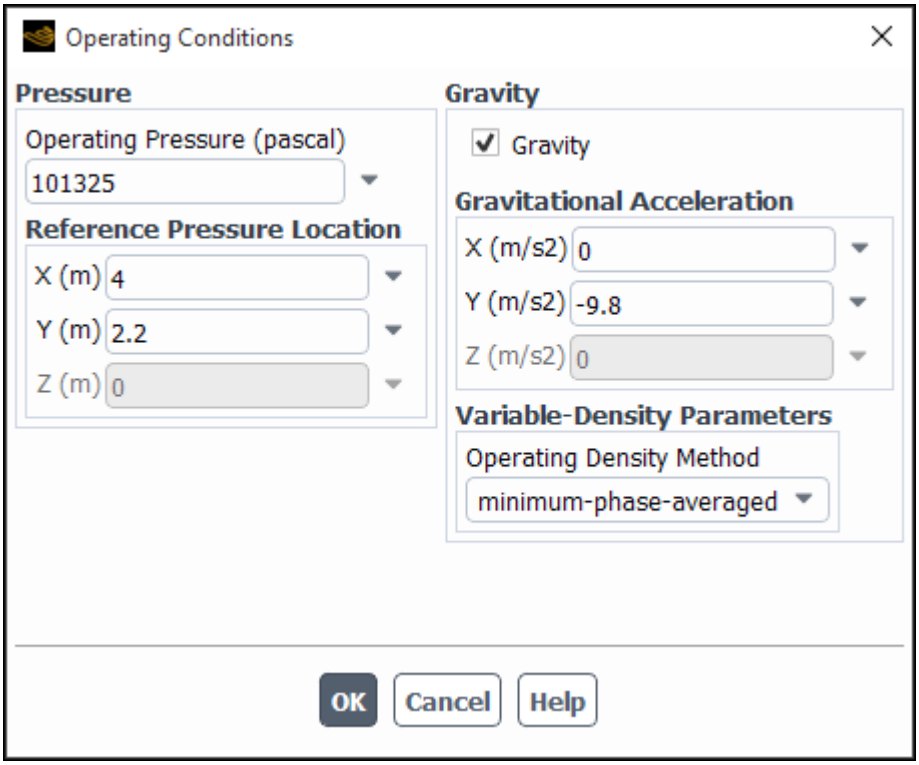
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.2.4. Modeling Buoyancy-Driven Multiphase Flow

26.2.4.1. Setting the Operating Density for a Buoyancy-Driven Multiphase Flow

Ansys Fluent provides the following methods for calculation of operating density for a buoyancy-driven flow. These options appear in the **Operating Density** drop-down list in the **Operating Conditions** dialog box (**Variable-Density Parameters** group box).

Figure 26.9: The Operating Conditions Dialog Box for Multiphase Flows



- **minimum-phase-averaged**: (default) Operating density is taken as the minimum of averaged densities calculated for each phase. For a compressible flow, operating density is internally set to zero.
- **primary-phase-averaged**: Operating density is taken as the averaged density of the primary phase.
- **mixture-averaged**: Operating density is taken as the averaged density of the mixture in the whole domain.
- **user-input**: Allows you to provide your own values for specific needs.

The default method, **minimum-phase-averaged**, is appropriate for most cases, and typically you do not need to change it. It automatically handles the operating density requirements for both incompressible and compressible flows by resolving the correct hydrostatic pressure gradient distribution.

If you are interested only in the pressure gradient (rather than the hydrostatic pressure values), you can use either the **primary-phase-averaged** or **mixture-averaged** method.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.2.4.2. The Boussinesq Approximation in a Multiphase Flow

When using the Boussinesq approximation for density variation in multiphase flows, there are several important items to consider:

- The Boussinesq approximation is only valid for small variations in fluid densities caused by small variations in temperature. Therefore, constant density is assumed, and a momentum source is only added for modeling buoyancy.
- The Boussinesq approximation assumes that small density variations have little effect on inertial terms and strong effects on gravitational terms. Thus, the differences in specific weights between two fluids or layers of fluids is the force driving the flow. (Waves cannot use this approximation.) Two non-dimensional numbers quantify the effect of buoyancy: the Richardson number and the Rayleigh number.
- Operating density should be set to zero.

For additional information on the Boussinesq approximation, see The Boussinesq Model (p. 1388).

26.2.5. Modeling Compressible Flows

If you are using any of the multiphase models for a compressible flow, note the following:

- Ansys Fluent allows the modeling of:
  - Multiple compressible ideal gas phases and/or incompressible/compressible liquids at any inlet
  - A single real gas or real liquid at any inlet
  - Multiple real gases at a single inlet
  - Multiple compressible liquids using either the Fluent built-in compressible liquid density model based on the Tait law (see Compressible Liquid Density Method (p. 1626)) or user-defined functions

  For multiple compressible phases, the fundamental equations for compressible flow remain the same. However, boundary conditions are modified to account for changes in material properties due to compressible fluid mixtures.
- In cases of a mixture of ideal and real gases, the phases must enter the domain through separate inlets.
- When using the VOF model, stability is improved if the primary phase is the compressible ideal gas.
- If you specify the total pressure at a boundary (for example, for a pressure inlet or intake fan), Fluent will use the specified value for temperature at that boundary as total temperature for the compressible phase, and as static temperature for the other phases (which are incompressible).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For each mass-flow inlet, you will need to specify mass flow or mass flux for each individual phase.

  Important:

  Note that if you read a case file that was set up in a version of Fluent previous to 6.1, you will need to redefine the conditions at the mass-flow inlets. See Defining Multiphase Cell Zone and Boundary Conditions (p. 2957) for more information on defining conditions for a mass-flow inlet in multiphase calculations.

Enhanced Compressible Flow Numerics

Compressible flow simulations may have convergence difficulties due to explicit handling of source term accounting for density variation. These simulations could become extremely unstable in the presence of a large variation in pressure and temperature. In Ansys Fluent, an enhanced numerical treatment that provides better stability at startup and during calculation of compressible flows is enabled by default. The treatment controls the rate of change of pressure and temperature between iterations and provides additional diagonal dominance to the matrix of equations.

Although disabling the enhanced numerical treatment is not recommended, you can temporarily turn it off to investigate unstable cases using the following text user interface command:

```
solve/set/multiphase-numerics/compressible-flow/enhanced-numerics
...
enhanced compressible flow numerics? [yes] no
```

See Compressible Flows (p. 1759) for more information about compressible flows.

Alternative Compressible Flow Numerics

A more general formulation for compressible multiphase flows employs a mathematically rigorous treatment of compressible phases at an inlet boundary. Instead of treating the entering incompressible and compressible phases as a pseudo-mixture, this formulation calculates static temperature and pressure using an iterative method based on fundamental thermodynamic relations (such as the Maxwell relations). This approach involves evaluation of the property derivatives, which can be computed either numerically or through analytic expressions. You can enable this alternative treatment via the text command:

```
solve/set/multiphase-numerics/compressible-flow/alternate-bc-formulation
alternate compressible bc formulation? [no] y
use analytical thermodynamic derivatives? [no] y
```

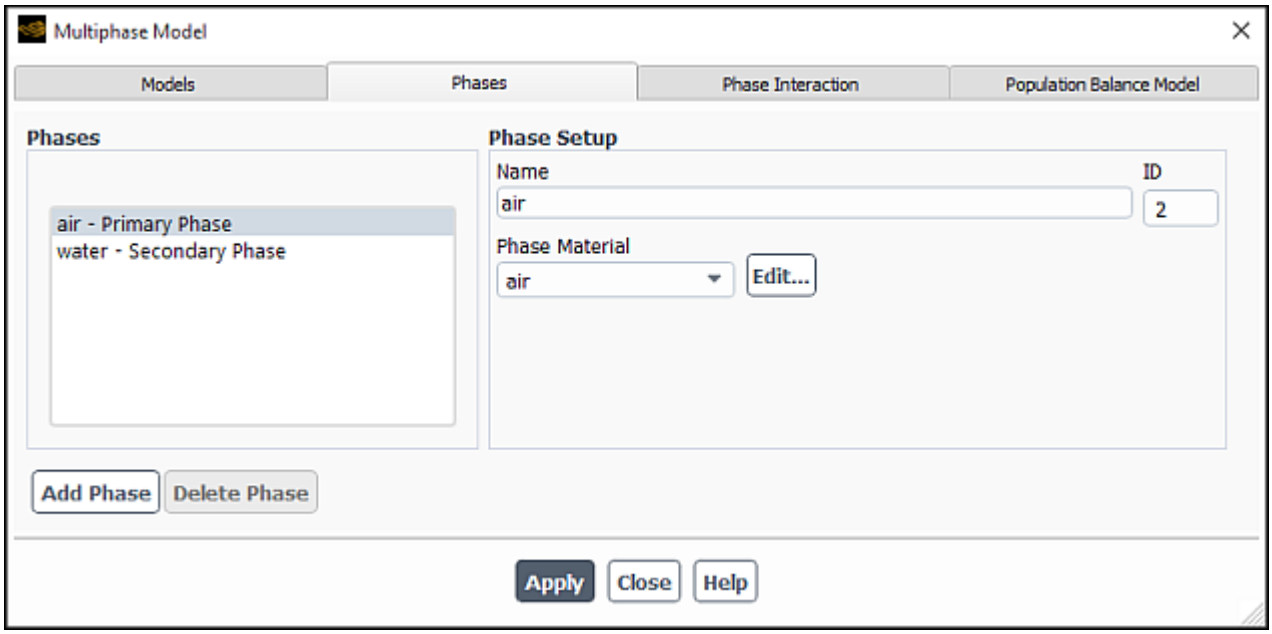
26.2.6. Defining the Phases

You can define the phases (including their material properties) in the **Phases** tab of the **Multiphase Model** dialog box (Figure 26.10: The **Multiphase Model** dialog box - **Phases** tab (p. 2933)).

Setup → Models → Multiphase → Edit...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.10: The Multiphase Model dialog box - Phases tab



Each item in the **Phases** list is one of two types:

- A **Primary-Phase** indicates that the selected item is the primary phase.
- A **Secondary-Phase** indicates that the selected item is a secondary phase.

In the **Phase Setup** group box, you can specify the name of the phase (19 characters maximum) and select its material.

You can specify any interaction between the phases (for example, surface tension and wall adhesion for the VOF model, slip velocity for the mixture model, drag functions for the mixture and the Eulerian models) in the **Phase Interaction** tab of the **Multiphase Model** dialog box.

Instructions for defining mass transfer between phases can be found in Including Mass Transfer Effects (p. 2940).

Instructions for defining heterogeneous reactions for species transport cases are provided in Specifying Heterogeneous Reactions (p. 2937).

Instructions for defining the phases and model-specific phase interactions are provided in the following sections:

- VOF model: Defining the Phases for the VOF Model (p. 3009)
- Mixture model: Defining the Phases for the Mixture Model (p. 3038)
- Eulerian model: Defining the Phases for the Eulerian Model (p. 3066)

26.2.7. Including Body Forces

When large body forces (for example, gravity or surface tension forces) exist in multiphase flows, the body force and pressure gradient terms in the momentum equation are almost in equilibrium while

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the contributions of convective and viscous terms are small in comparison. Segregated algorithms converge poorly unless partial equilibrium of pressure gradient and body forces is taken into account. Ansys Fluent provides an optional "implicit body force" treatment that can account for this effect, making the solution more robust.

The basic procedure involves augmenting the correction equation for the face flow rate, Equation 23.56 in the Theory Guide, with an additional term involving corrections to the body force. This results in extra body force correction terms in Equation 23.54 in the Theory Guide, and allows the flow to achieve a realistic pressure field very early in the iterative process.

To include this body force, enable **Gravity** in the Operating Conditions Dialog Box (p. 4978) and specify the **Gravitational Acceleration**.

Setup → **Cell Zone Conditions** → **Operating Conditions...**

For VOF calculations, you should also enable the **Specified Operating Density** option in the **Operating Conditions** dialog box, and set the **Operating Density** to be the density of the lightest phase. (This excludes the buildup of hydrostatic pressure within the lightest phase, improving the round-off accuracy for the momentum balance.) If any of the phases is compressible, set the **Operating Density** to zero.

Important:

For VOF and mixture calculations involving body forces, it is recommended that you also enable the **Implicit Body Force** treatment for the **Body Force Formulation** in the **Multiphase Model** dialog box. This treatment improves solution convergence by accounting for the partial equilibrium of the pressure gradient and body forces in the momentum equations. See Including Body Forces (p. 2933) for details.

26.2.8. Modeling Multiphase Species Transport

In Ansys Fluent , you can set up a multiphase species transport and volumetric reaction (Modeling Species Transport in Multiphase Flows in the Theory Guide) in a fashion that is similar to setting up a single-phase chemical reaction using the **Species Model** dialog box (for example, Figure 26.11: The Species Model Dialog Box with a Multiphase Model Enabled (p. 2935)).

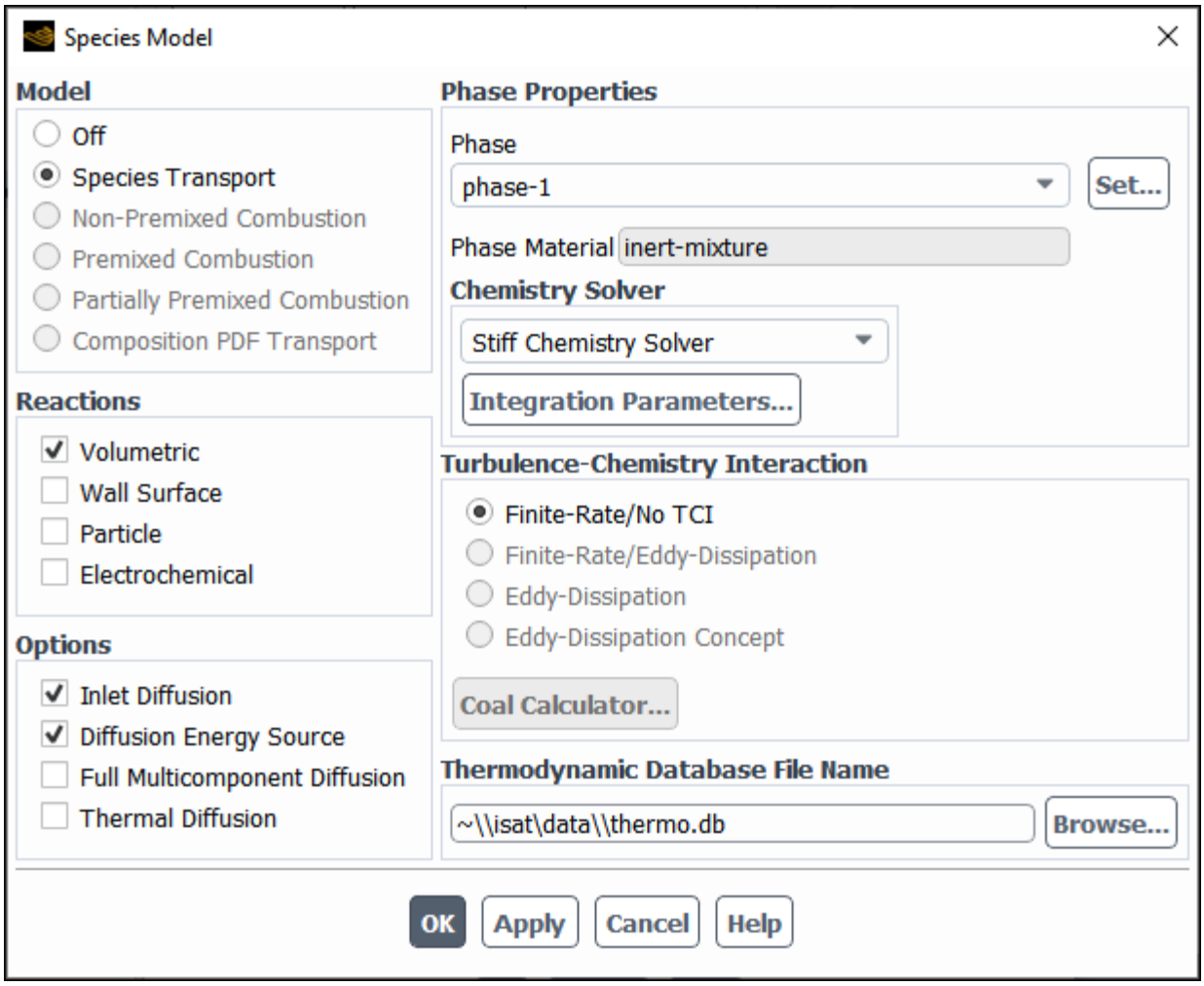
Note:

Note that the **Full Multicomponent Diffusion** option in the **Species Model** dialog box (**Options** group), in not used when modeling multiphase species transport. Instead, you can enable the **Full Multicomponent Diffusion** option for a selected phase in the **Phase Properties** dialog box as further described.

Setup → **Models** → **Species** → **Edit...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

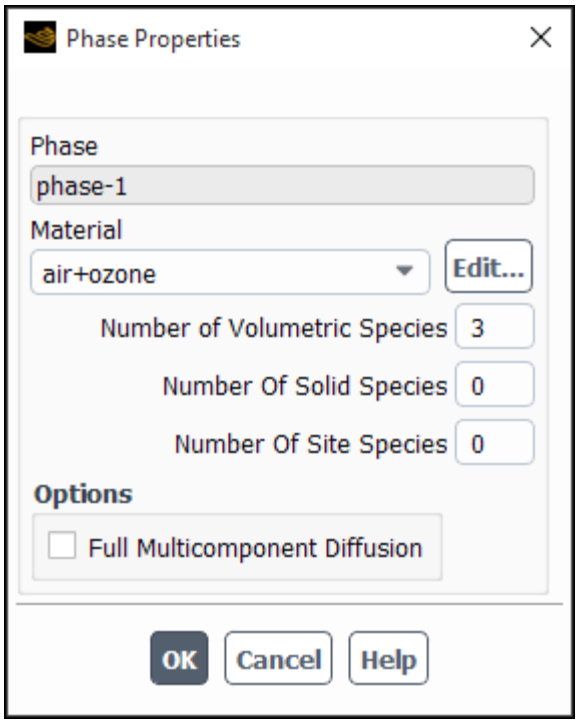
Figure 26.11: The Species Model Dialog Box with a Multiphase Model Enabled



1. Select **Species Transport** under **Model**.
2. Enable **Volumetric** under **Reactions**.
3. If you have already enabled a multiphase model, then you need to select a multi-species phase and assign to it a mixture material. By default, the primary phase is selected in the **Phase** drop-down list, and the fluid material assigned to it is displayed in the **Phase Material** information box (**Phase Properties** group box).
   a. In the **Phase** drop-down list, select a phase for a species mixture material.
   b. Click the **Set...** button to display the **Phase Properties** dialog box (see Figure 26.12: The Phase Properties Dialog Box (p. 2936)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.12: The Phase Properties Dialog Box



c. In the **Material** drop-down list, the materials for each phase already defined in your case are listed at the top followed by the mixture materials from the Fluent database materials. From this list, choose the material that you want to use for a multi-species phase.

d. If you want to use the full multicomponent diffusion model described in Full Multicomponent Diffusion in the *Fluent Theory Guide* for the selected phase, enable the **Full Multicomponent Diffusion** option.

e. Click **OK** to close the **Phase Properties** dialog box.

4. (Eulerian multiphase model only) In the **Species Model** dialog box, select the solver for your simulation from the **Chemistry Solver** drop-down list.

   - If you want to solve finite-rate chemistry, select the **Stiff Chemistry Solver**.

   Note:

   The **Stiff Chemistry Solver** can be used only for one mixture-material phase.

   - If you want to make explicit use of chemistry source terms in the phase-specific species transport equations, without a stiff-chemistry solver, select **None – Explicit Source**. Note that this solver is not recommended for stiff or complex chemistry.

5. (optional) If you selected **Stiff Chemistry Solver** in the previous step, then in the Integration Parameters Dialog Box (p. 4795) (opened by clicking the **Integration Parameters...** button), you can:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- adjust the **ODE Parameters**
- enable the **ISAT** integration method to accelerate chemistry calculations and set up its parameters. See Using ISAT (p. 2457) for additional information.

6. In the **Species Model** dialog box, choose the **Turbulence-Chemistry Interaction** model. Three models are available:

   Finite-Rate/No TCI

   computes only the Arrhenius rate (see Equation 7.21 in the Theory Guide) and neglects turbulence-chemistry interaction.

   Finite-Rate/Eddy-Dissipation

   (for turbulent flows) computes both the Arrhenius rate and the mixing rate and uses the smaller of the two.

7. Click **OK** in the **Species Model** dialog box.

   Once you click **OK**, the mixture material you selected for the multi-species phase will be copied to your case and appear under the **Setup/Materials/Mixture** tree node in the **Outline View**.

When modeling multiphase species transport, additional inputs may also be required depending on your modeling needs. See, for example, Specifying Heterogeneous Reactions (p. 2937) for more information defining heterogeneous reactions, or Including Mass Transfer Effects (p. 2940) for more information on mass transfer effects.

26.2.9. Specifying Heterogeneous Reactions

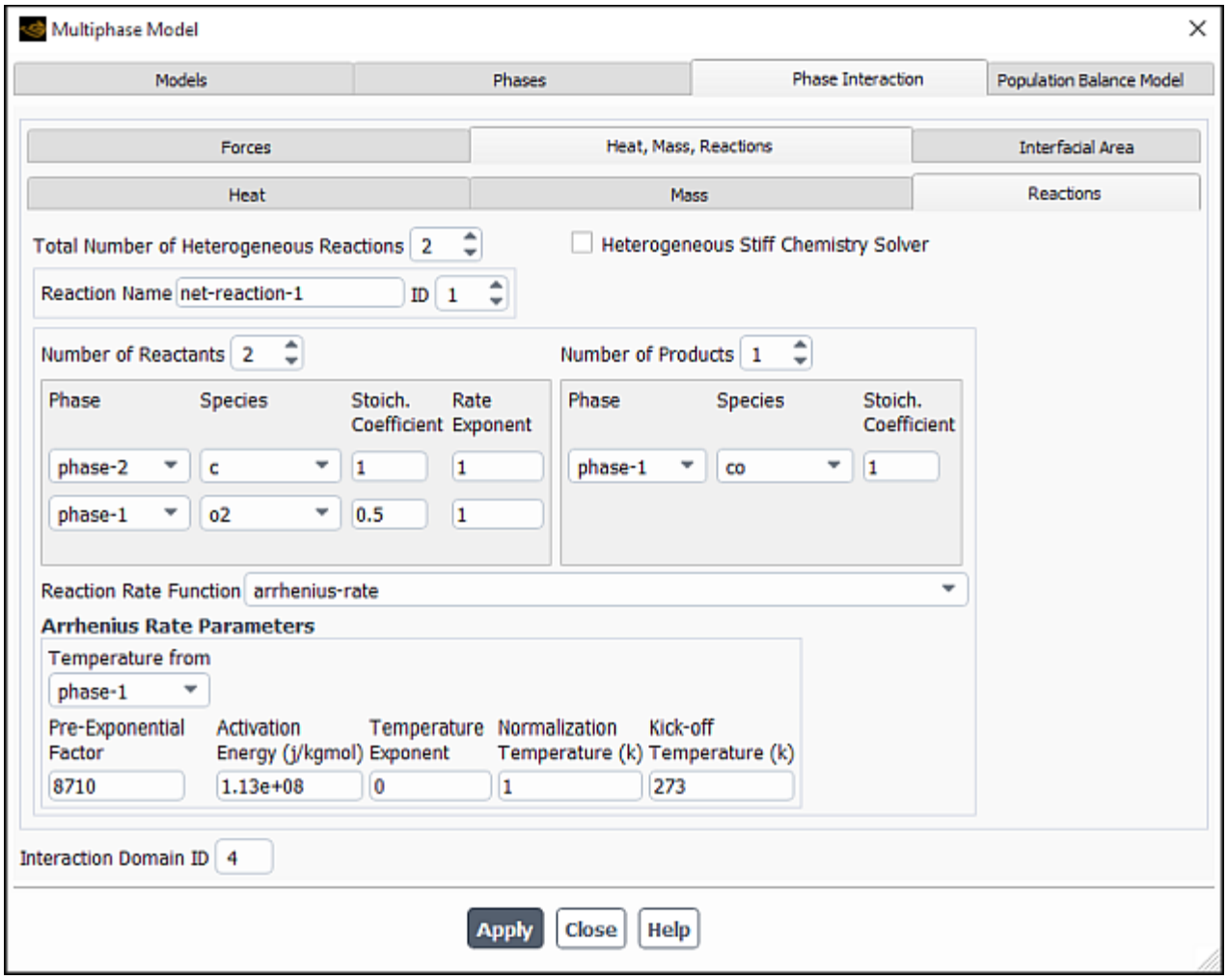
You can define multiple heterogeneous reactions and stoichiometry in the **Reactions** tab of the **Multiphase Model** dialog box (for example, Figure 26.13: The Reactions Tab for Heterogeneous Reactions (p. 2938)).

Setup → Models → Multiphase → Edit...

1. In the **Multiphase Model** dialog box (Figure 26.10: The **Multiphase Model** dialog box - **Phases** tab (p. 2933)), go to the **Phase Interaction** > **Heat, Mass, Reactions** > **Reactions** tab.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.13: The Reactions Tab for Heterogeneous Reactions



2. Set the total number of reactions (volumetric reactions, wall surface reactions, and particle surface reactions) in the **Total Number of Heterogeneous Reactions** field. (Use the arrows to change the value, or type in the value and press **Enter**.)

3. Enable the **Heterogeneous Stiff Chemistry Solver** option if your interphase reaction mechanism contains numerically stiff reactions. This option can improve convergence and is available for transient Eulerian multiphase simulations. When this option is enabled, Ansys Fluent uses a fractional step algorithm where the flow is advanced without reaction sources for a time step, and then the chemistry is integrated point-by-point for the same time step. The stiff chemistry scheme solves all species in all phases coupled. Note that it is possible to include homogeneous (intra-phase) reactions along with the heterogeneous reactions in the **Reactions** tab (instead of in the reaction mechanism in the **Create/Edit Materials** dialog box), and these reactions will be solved with the stiff solver. The stiff ODE solver tolerances can be set using the following text command:

   `solve` → `set` → `heterogeneous-stiff-chemistry`

4. Specify the **Reaction Name** of each reaction that you want to define.

5. Set the **ID** of each reaction you want to define. (Again, if you type in the value be sure to press **Enter**.)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. For each reaction, specify how many reactants and products are involved in the reaction by increasing the value of the **Number of Reactants** and the **Number of Products**. Select each reactant or product in the **Reaction** tab and then set its stoichiometric coefficient in the **Stoich. Coefficient** field. (The stoichiometric coefficient is the constant $\nu'_{i,r}$ or $\nu''_{i,r}$ in Equation 7.20 in the Theory Guide.)

7. For each reaction, indicate the **Phase** and **Species** and the stoichiometric coefficient for each of your reactants and products.

8. For each reaction, use the **Reaction Rate Function** drop-down list to select one of the following:

   none

   if you do not want to include a reaction rate

   population-balance

   is the mass transfer due to nucleation and growth. If neither the primary phase or secondary phase has species associated with it, then the mass transfer is modeled as unidirectional. If the mass transfer process involves reactions or species, the problem must be set up as one involving heterogeneous reaction/mass transfer.

   Important:

   The **population-balance** option for **Reaction Rate Function** should not be used if there are multiple reactions leading to the formation of the secondary phase. In this case, the reaction rate functions should be specified either through the standard dialog boxes or user-defined functions and the growth rate function should be a sum of the individual reaction rates. This would ensure consistency between the individual reaction rates and the total mass transfer from the primary to the secondary phase.

   You can always use `DEFINE_MASS_TRANSFER` or `DEFINE_HET_RXN_RATE` user-defined function types instead of the **population-balance** option to specify mass transfer rates. However, the growth rate function and the mass transfer rates returned from the UDFs need to be consistent with each other.

   arrhenius-rate

   to specify rate exponents for an Arrhenius-type reaction (see Heterogeneous Phase Interaction in the Theory Guide for more information)

   Important:

   This simple form of the Arrhenius rate option may only be used for devolatilization reactions only. Char combustion reaction may be more involved and complicated

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

to be simply casted in this form. Additional diffusion rate formulations may be needed to formulate a complete char (or solid phase) reaction system.

Important:

Note that you can also specify the heterogeneous reaction rates using a user-defined function. A user-defined function is available for an Arrhenius-type reaction with rate exponents that are equivalent to the stoichiometric coefficients. For more information, see `DEFINE_HET_RXN_RATE` in the Fluent Customization Manual.

9. Click **Apply**.

Important:

Ansys Fluent assumes that the reactants are mixed thoroughly *before* reacting together, therefore the heat and momentum transfer is based on this assumption. This assumption can be disabled using a text command. For more information, contact your Ansys Fluent support engineer.

26.2.10. Including Mass Transfer Effects

As discussed in Modeling Mass Transfer in Multiphase Flows in the Theory Guide, mass transfer effects in the framework of Ansys Fluent's general multiphase models (that is, Eulerian multiphase, mixture multiphase, or VOF multiphase) can be modeled in one of three ways:

- Unidirectional constant rate mass transfer
- UDF-prescribed mass transfer
- mass transfer through cavitation, evaporation-condensation, boiling, or generalized species mass transfer

Note:

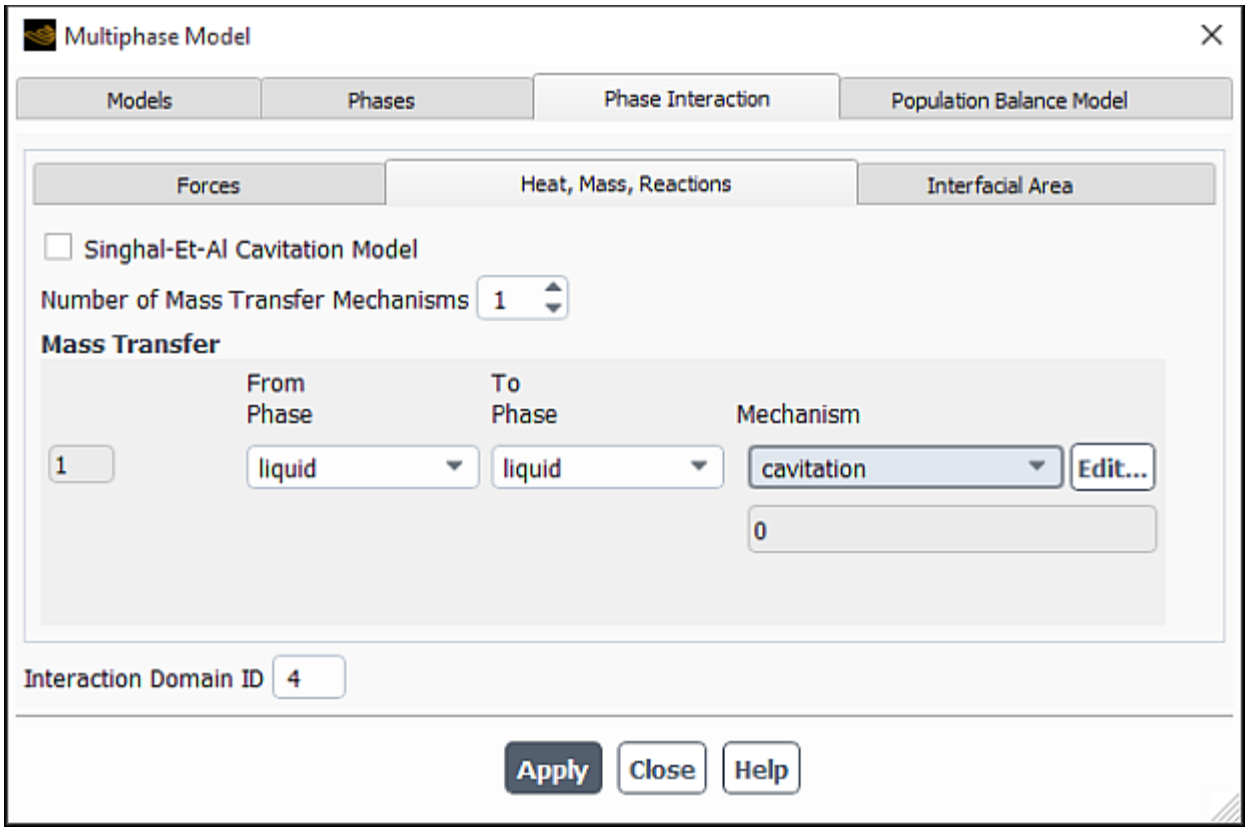
For limitations regarding the cavitation models, see Limitations of the Cavitation Models in the *Fluent Theory Guide*.

You can define mass transfer in a multiphase simulation as a unidirectional constant, using a user-defined function, through population balance, cavitation, evaporation/condensation, or species mass transfer. To do this, go to the **Phase Interaction** > **Heat, Mass, Reactions** > **Mass** tab in the **Multiphase Model** dialog box.

Setup → **Models** → **Multiphase** → **Edit...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.14: The Mass Tab for Mass Transfer



1. Specify the **Number of Mass Transfer Mechanisms**.

   Important:

   In general, you should use only one mass transfer mechanism when you employ one of the available mass transfer models (see Step 7). Although using multiple mass transfer mechanisms is allowed for any pairs of phases (including the same pair of phases), it is not recommended because the mass transfer models have been fundamentally developed for a single mass transfer process. These models do not account for possible interactions between two mass transfer mechanisms in a flow system. In order to account for all relevant physics using UDFs in multiple mass transfer mechanisms, you can use the user-defined option. For additional restrictions when using multiple mass transfer mechanisms, see Step 7.

2. For each mechanism, specify the phase of the source material under **From Phase**. Note that the phase you select for **From Phase** must be a liquid if you plan to select **cavitation**, **evaporation-condensation**, or **boiling** from the **Mechanism** drop-down menu (see Step 7).

3. If species transport is part of the simulation, and the source phase is composed of a mixture material, then specify the species of the source phase mixture material in the corresponding **Species** drop-down list.

4. For each mechanism, specify the phase of the destination material phase under **To Phase**. Note that the phase you select for **To Phase** must be a vapor if you plan to select **cavitation**, **evapor-**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ation-condensation, or **boiling** from the **Mechanism** drop-down menu. If you plan to select **species-mass-transfer** and one of the phases is a gas, it must be specified as the **To Phase** (see Step 7).

5. If species transport is part of the simulation, and the destination phase is composed of a mixture material, then specify the species of the destination phase mixture material in the corresponding **Species** drop-down list.

6. For each pair of phases with mass transfer, select the desired mass transfer mechanism under **Mechanism**. The available mechanisms and their inputs are described in Mass Transfer Mechanisms (p. 2945).

   Ansys Fluent will automatically include the terms needed to model mass transfer in all relevant conservation equations. Another option to model mass transfer between phases is through the use of user-defined sources and their inclusion in the relevant conservation equations. This approach is more involved, but more powerful, allowing you to split the source terms according to a model of your choice.

7. Click **Apply**.

Important:

- Except for the **species-mass-transfer** mechanism, if the same mass transfer model is selected for multiple mass transfer mechanisms, the most recently edited mechanism sub-model, including its parameters, options, and constants (excluding its material properties, such as vaporization pressure or saturation temperature), will apply to all the mass transfer mechanisms.
- When modeling multiple **species-mass-transfer** mechanisms, you can select different species mass transfer models and model options, such as Raoult's Law, Henry's Law, etc. However, because there is a lack of interaction between different species mass transfer models, this should be done with caution. For information on modeling restrictions for interphase species mass transfer, see Interphase Species Mass Transfer in the *Fluent Theory Guide*.
- Momentum, energy, and turbulence are also transported with the mass that is transferred. Ansys Fluent assumes that the reactants are mixed thoroughly *before* reacting together, therefore the heat and momentum transfer is based on this assumption. This assumption can be disabled using a text command. For more information, contact your Ansys Fluent support engineer.

When your model involves the transport of multiphase species, you can define a mass transfer mechanism between species from different phases. If a particular phase does not have a species associated with it, then the mass transfer throughout the system will be performed by the bulk fluid material.

Important:

Including species transport effects in the mass transport of multiphase simulation requires that **Species Transport** be selected in the **Species Model** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setup → **Models** → **Species**

26.2.10.1. Alternative Modeling of Energy Sources

The existing formulation of sensible enthalpy for interphase mass transfer includes the formation enthalpy, which could be problematic for some cases, such as cases that involve species transport and reactions. The alternative treatment, which is available with the VOF and Mixture multiphase models, is based on the explicit modeling of the energy source driven by mass transfer. You can enable the alternative formulation by using the following text command:

```
solve/set/multiphase-numerics/heat-mass-transfer/alternative-energy-
treatment?

enable alternative treatment of handling latent heat source [no] y
```

Energy Source Treatment

The energy source is modeled explicitly as:

$$S_{energy} = -\dot{m}L \tag{26.2}$$

where $\dot{m}$ is the mass transfer rate, and L is latent heat of vaporization. If you are using the **tabular-ptl-sat** method to specify saturation temperature for the Evaporation-Condensation Mechanism (p. 2950) or vaporization pressure for the Cavitation Mechanism (p. 2946), Ansys Fluent will take the latent heat values directly from the tabular data. Otherwise, it is calculated as:

$$L = \left(h_q^s - h_p^s\right) \tag{26.3}$$

Here, h_p^s and h_q^s are saturation enthalpies of phase p and phase q at the saturation temperature, respectively. They are calculated as:

$$h_p^s = h_p^f + \int_{T_{ref,p}}^{T_{sat}} cp_p dT \tag{26.4}$$

$$h_q^s = h_q^f + \int_{T_{ref,q}}^{T_{sat}} cp_q dT \tag{26.5}$$

where,

h_p^f and h_q^f = formation enthalpies of phase p and phase q, respectively

cp_p and cp_q = specific heats of phase p and phase q, respectively

$T_{ref,p}$ and $T_{ref,q}$ = reference temperatures of phase p and phase q as specified in the **Create/Edit Materials** dialog box, respectively.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

T_{sat} = saturation temperature

Note:

For mass transfer mechanisms that do not require explicit input of saturation temperature, such as cavitation and species mass transfer, the saturation temperature is assumed as the reference temperature of the liquid.

Sensible Enthalpy Definition

For mass transfer cases, Ansys Fluent by default uses the following definition of sensible enthalpy:

$$h_p = h_p^f + \int_{T_{ref,p}}^{T_p} cp_p dT \tag{26.6}$$

where,

h_p = sensible enthalpy of phase p

h_p^f = formation enthalpy of phase p

cp_p = specific heat of phase p

T_p = temperature of phase p

$T_{ref,p}$ = reference temperature of phase p as specified in the **Create/Edit Materials** dialog box

The alternative treatment considers the definition of sensible enthalpy that excludes formation enthalpy. Sensible enthalpy for each phase is expressed as:

$$h_p = \int_{T_{ref}}^{T} cp_p dT \tag{26.7}$$

with T_{ref}=298.15 K.

Heat Flux Reporting Strategies

When you use the alternative formulation, the following additional variable become available for postprocessing under the **Phase Interaction...** category:

- **Latent Heat**

  In the variable selection drop-down lists that appear in postprocessing dialog boxes, this quantity will be appended with the ID of the corresponding mass transfer mechanism.

Due to the explicit nature of the alternative formulation, the latent heat source appears as the heat imbalance in heat flux reporting.

If you want to know the heat imbalance due to the latent heat source, you should create a custom-field function for the mass transfer rate multiplied by latent heat, and then report it as a volume integral. Ansys Fluent will report the following value:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\Delta H = \int_V \left(\sum_{i=1}^{N} \dot{m}_i L_i \right) dV \tag{26.8}$$

where,

ΔH = heat imbalance

N = number of mass transfer mechanisms that you defined

$\dot{m}_i$ = mass transfer rate of i^{th} mechanism

L_i = Latent heat of i^{th} mechanism

V = volume

26.2.10.2. Mass Transfer Mechanisms

When including mass transfer effects in a multiphase simulation, you can choose from the following mechanisms for each phase pair that undergoes mass transfer:

- **constant-rate**
- **user-defined**
- **population-balance**
- **cavitation**
- **evaporation-condensation**
- **species-mass-transfer**
- **boiling**

Details for each mechanism are provided in the following sections:

26.2.10.2.1. Constant-Rate Option

26.2.10.2.2. User-Defined Option

26.2.10.2.3. Population-Balance Mechanism

26.2.10.2.4. Cavitation Mechanism

26.2.10.2.5. Evaporation-Condensation Mechanism

26.2.10.2.6. Species-Mass-Transfer Mechanism

26.2.10.2.7. Boiling Mechanism

26.2.10.2.1. Constant-Rate Option

You can use the **constant-rate** option to specify a constant value for unidirectional mass transfer. The **constant-rate** must be entered in units of 1/time unit.

26.2.10.2.2. User-Defined Option

The **user-defined** option allows you to implement a correlation reflecting a model of your choice, through a user-defined function.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.2.10.2.3. Population-Balance Mechanism

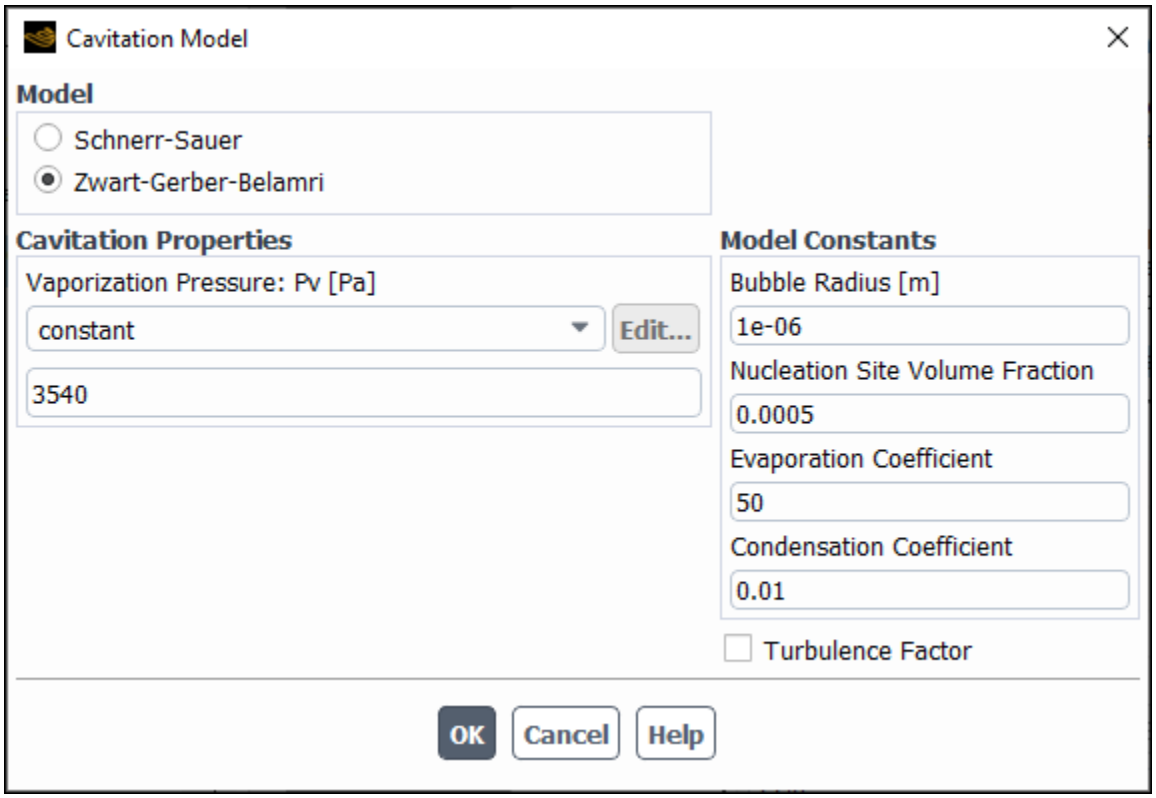
With the **population-balance** mechanism, you can model flow where a number density function is introduced to account for the particle population. With the aid of particle properties (for example, particle size, porosity, composition, and so on), different particles in the population can be distinguished and their behavior can be described. For a comprehensive understanding of this option, refer to Population Balance Model (p. 3131).

26.2.10.2.4. Cavitation Mechanism

The **cavitation** mechanism allows you to select a cavitation model. For a cavitating flow, it is recommended that you select liquid as the primary phase and vapor as the secondary phase.

The cavitation models are available when using mixture, VOF, and Eulerian multiphase models. You are provided with two model options: **Schnerr-Sauer** and **Zwart-Gerber-Belamri**. To open the **Cavitation Model** dialog box, select **cavitation** from the **Mechanism** drop-down list. For information about the cavitation models, refer to Cavitation Models in the Theory Guide.

Figure 26.15: The Cavitation Model Dialog Box



- For the **Schnerr-Sauer** model, specify the **Bubble Number Density** (n_b in Equation 14.588 in the *Fluent Theory Guide*) under **Model Constants**. The default value of 1e11 is suitable for most cases.
- For the **Zwart-Gerber-Belamri** model, specify the **Bubble Radius**, the **Nucleation Site Volume Fraction**, the **Evaporation Coefficient**, and the **Condensation Coefficient** under **Model Constants**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Under **Cavitation Properties**, specify the **Vaporization Pressure** and, if desired, a temperature dependence function. In addition to the methods described in Defining Properties Using Temperature-Dependent Functions (p. 1616) (**constant**, **polynomial**, **piecewise-linear**, **piecewise-polynomial**, or **user-defined**), you can choose one of the following:

  – **rgp-table-sat**

    This option allows you to use to define the vaporization pressure using RGP tables as described in Defining Saturation Properties via RGP Tables (p. 1735).

  – **taylor-approximation**

    The **taylor-approximation** method models the vaporization pressure as linearly varying with temperature about the free-stream value. This approach can improve numerical stability, but is only appropriate when deviations about the free-stream temperature are sufficiently small that higher-order terms can be neglected. The expression for the vaporization pressure using the **taylor-approximation** method is:

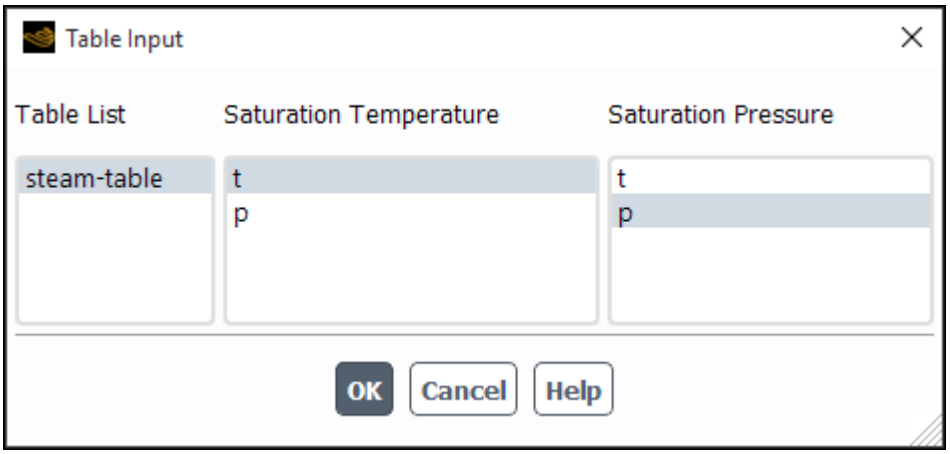
$$P_v = P_v(T_\infty) + C_{thermal} \frac{\rho_l(T_\infty)\rho_v(T_\infty)L(T_\infty)}{T_\infty\left[\rho_l(T_\infty) - \rho_v(T_\infty)\right]}\left(T - T_\infty\right) \tag{26.9}$$

    where T_∞ is the free-stream temperature, L is the latent heat, and $C_{thermal}$ is a user-modifiable coefficient. In the **Taylor Approximation** dialog box, you will specify **Vaporization Pressure** (at the free-stream temperature), **Free Stream Temperature**, and **Thermal Coefficient** ($C_{thermal}$).

  – **tabular-pt-sat**

    This method allows you to specify vaporization pressure using a table that contains data points for saturation temperature and saturation pressure. Saturation pressure will be interpolated based on the static temperature from the solution, which is assumed to be at a saturated state. The table format is described in Reading Files in Tabular Format (p. 969). Once you select **tabular-pt-sat**, the **Table Input** dialog box opens where you can select the appropriate table and the names of the columns from the **Saturation Temperature** and **Saturation Pressure** selection lists.

    Figure 26.16: Table Input for Vaporization Pressure



  – **tabular-ptl-sat**

(available only with Alternative Modeling of Energy Sources (p. 2943)) The method allows you to specify saturation pressure using a table that contains data points for saturation temperature, saturation pressure, and latent heat. Saturation pressure and latent heat will be interpolated based on the static temperature from the solution, which is assumed to be at a saturated state. The table format is described in Reading Files in Tabular Format (p. 969). Once you select **tabular-ptl-sat**, the **Table Input** dialog box opens where you can select the appropriate table and the names of the columns from the **Saturation Temperature**, **Saturation Pressure** and **Latent Heat** selection lists.

Note:

If you are using **tabular-pt-sat** or **tabular-ptl-sat** to specify vaporization pressure, you must first read an appropriate table following the procedure outlined in Reading Files in Tabular Format (p. 969). All table data must be in SI units (saturation temperature in Kelvin (K), saturation pressure in Pascal (Pa), and latent heat in Joule/Kg).

- If you want to include the influence of turbulence on the threshold cavitation pressure, enable **Turbulence Factor**. If desired, you can adjust the value of the **Turbulent Coefficient**. For details about the implementation of the Turbulence Factor, see Turbulence Factor in the *Fluent Theory Guide*.

By default, the minimum vapor pressure limit is set to 1 Pa. The default value for the maximum vapor pressure limit is set to five times the local vapor pressure with consideration of the turbulent and thermal effects for each computational cell and phase. The default and recommended value for the evaporation and condensation coefficients in the Schnerr-Sauer model (F_{vap} and F_{cond} in Equation 14.592 in the *Fluent Theory Guide*) are 1 and 0.2, respectively. You can modify these default values using the following expert text commands:

- `solve/set/multiphase-numerics/heat-mass-transfer/cavitation/min-vapor-pressure`
- `solve/set/multiphase-numerics/heat-mass-transfer/cavitation/max-vapor-pressure-ratio`
- `solve/set/multiphase-numerics/heat-mass-transfer/cavitation/schnerr-cond-coeff`
- `solve/set/multiphase-numerics/heat-mass-transfer/cavitation/schnerr-evap-coeff`

Note:

- If you use multiple cavitation mass transfer mechanisms in your simulation, the value you specify for **Turbulence Factor** will automatically apply to all cavitation mechanisms, thereby overwriting previously existing turbulent coefficients.
- For the **Mixture** multiphase model, in addition to the Schnerr-Sauer and Zwart-Gerber-Belamri models, the Singhal et al. expert cavitation model can be used. See Using the Singhal et al. Expert Cavitation Model (p. 3056) for more information on using this model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Pressure Limiting Method

When calculating the properties of a compressible flow (such as densities and sound speeds), the cavitation model by default limits the absolute pressure to the vapor pressure whenever the absolute pressure drops below the vapor pressure. This default treatment may not predict correct results for a compressible flow because at saturation pressure, the density method behaves as the **incompressible-ideal-gas** law rather than the **ideal-gas** law. In such a case, you can apply a special treatment that allows the pressure to drop to the minimum absolute pressure. The treatment correctly predicts shock, which is not possible with the default treatment. To set the pressure limiting method, use the following text command:

```
solve/set/multiphase-numerics/heat-mass-transfer/cavitation/p-limit-
method
```

The available methods are:

- `local-pvap-min`: Limits the pressure to the local minimum of vapor pressures
- `global-pvap-min`: Limits the pressure to the global minimum of vapor pressures from all cells
- `pabs-min`: Limits the pressure to the absolute minimum pressure specified under solution limits specified in the **Solution Limits** dialog box.

For a cavitating flow, by default, pressure contours are clipped to the pressure specified by the pressure limiting method wherever the pressure drops below the vapor pressure. To see the actual pressure contours, use the following text command:

```
solve/set/multiphase-numerics/heat-mass-transfer/cavitation/dis-
play_clipped_pressure?

display clipped pressure in the post-processing [yes]: no
```

Note:

Clipped pressure values are only used in the computation of properties, while the actual pressure, which is used in the calculation of the mass transfer rate, remains unclipped.

Turbulent Diffusion Treatment

For turbulent flows, if one of the phases participating in cavitation is primary, then Ansys Fluent automatically enables the generalized turbulent diffusion treatment. The treatment adds a diffusion source in the volume fraction equation by assuming the diffusion coefficient as a function of turbulent viscosity of the respective phase. This helps avoid instantaneous solution instability due to cavitation. For more information about this treatment, see Diffusion in VOF Model in the *Fluent Theory Guide*.

The generalized turbulent diffusion treatment can be used for simulating flows with two or more phases. By default, for a flow with more than two phases, the turbulent diffusion acts between primary and secondary phases only. The turbulent diffusion between secondary phases is not accounted for.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you want to use the old turbulent diffusion treatment supported in releases prior to Release 19.2, enter the following text commands:

```
solve/set/multiphase-numerics/heat-mass-transfer/cavitation/turbulent-
diffusion
use generalized treatment of turbulent diffusion [yes] no
use old treatment of turbulent diffusion [yes] yes
```

Unlike the generalized turbulent diffusion treatment, the old treatment is applicable to only two-phase flows. If used for flows with more than two phases, the old treatment unphysically accounts for each secondary phase interacting with the primary phase, despite the secondary non-vapor phases not participating in cavitation.

If your want to disable the turbulent diffusion treatment for cases where the high turbulent viscosity may be a source of an unstable solution, enter the following text commands:

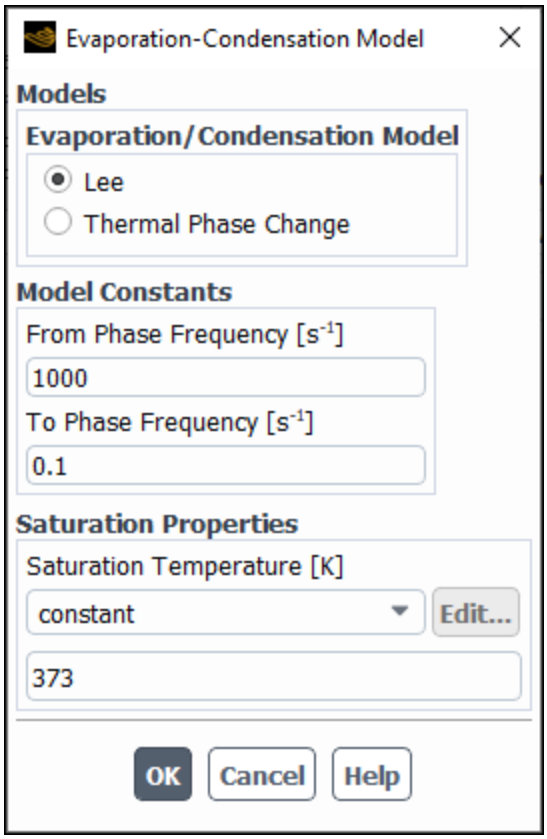
```
solve/set/multiphase-numerics/heat-mass-transfer/cavitation/turbulent-
diffusion
use generalized treatment of turbulent diffusion [yes] no
use old treatment of turbulent diffusion [yes] no
```

26.2.10.2.5. Evaporation-Condensation Mechanism

The **evaporation-condensation** mass transfer mechanism enables you to apply the evaporation-condensation model as the mass transfer mechanism. The VOF and Mixture multiphase models use the Lee model (see Lee Model in the *Fluent Theory Guide*). With the Eulerian multiphase model, you can choose between the Lee model and the Thermal Phase Change model (see Thermal Phase Change Model in the *Fluent Theory Guide*). When you select **evaporation-condensation** under **Mechanism** in the **Phase Interaction** > **Heat, Mass, Reactions** > **Mass** tab of the **Multiphase Model** dialog box, you will be prompted with the **Evaporation-Condensation Model** dialog box. Note that this dialog box could also be accessed by clicking **Edit...** next to the **evaporation-condensation** mechanism drop-down list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.17: The Evaporation-Condensation Model Dialog Box (Eulerian Multiphase Model)



You need to specify the following settings:

1. (Eulerian multiphase model only) Select the evaporation-condensation **Model**. The following models are available:
   - **Lee**
   - **Thermal Phase Change**

2. Specify the **Saturation Temperature** for your flow regime. In addition to the methods described in Defining Properties Using Temperature-Dependent Functions (p. 1616) (**constant**, **polynomial**, **piecewise-linear**, **piecewise-polynomial**, or **user-defined**) you can choose one of the following:

   - **rgp-table-sat**

     This option allows you to use to define the saturated temperature using RGP tables as described in Defining Saturation Properties via RGP Tables (p. 1735).

   - **tabular-pt-sat**

     The **tabular-pt-sat** allows you to specify saturation temperature using a table that contains data points for saturation temperature and saturation pressure. Saturation temperature will be interpolated based on the absolute pressure from the solution, which is assumed to be at a saturated state. The table format is described in Reading Files in Tabular Format (p. 969). Once you select **tabular-pt-sat**, the **Table Input** dialog box opens where you can select the appropriate table and the names of the columns for your simulation from the **Saturation Temperature** and **Saturation Pressure** selection lists.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **tabular-ptl-sat**

  (available only with Alternative Modeling of Energy Sources (p. 2943)) The method allows you to specify saturation temperature using a table that contains data points for saturation temperature, saturation pressure, and latent heat. Saturation temperature and latent heat will be interpolated based on absolute pressure from the solution, which is assumed to be at a saturated state. The table format is described in Reading Files in Tabular Format (p. 969). Once you select **tabular-ptl-sat**, the **Table Input** dialog box opens where you can select the appropriate table and the names of the columns from the **Saturation Temperature**, **Saturation Pressure** and **Latent Heat** selection lists.

  Note:

  - If you use **tabular-pt-sat** or **tabular-ptl-sat** to specify saturation temperature, you must first read an appropriate table following the procedure outlined in Reading Files in Tabular Format (p. 969). All table data must be in SI units (saturation temperature in Kelvin (K), saturation pressure in Pascal (Pa), and latent heat in Joule/Kg).
  - If you use the **tabular-pt-sat**, **tabular-ptl-sat**, **polynomial**, **piecewise-polynomial**, or **piecewise-linear** option for **Saturation Temperature**, the pressure-dependency must be specified in terms of absolute pressure.

3. (Lee model only) Specify the **From Phase Frequency** and **To Phase Frequency** model constants. These values correspond to the coefficient $coeff$ (Equation 14.617 in the Theory Guide). The values are `0.1` by default. However, note that the bubble diameter and accommodation coefficient are usually not very well known, so the appropriate values for a given problem can be very different. It is important to tune the values to match experimental data.

4. (Mixture multiphase model only) For certain models and conditions, you can select the **Semi-Mechanistic** boiling model to model sub-cooled nucleate boiling at low pressure, as described in Including Semi-Mechanistic Boiling (p. 3057).

5. (Thermal Phase Change model only) Specify the following model coefficients:

   - **From Phase Scaling Factor**: C_l in Equation 14.618 in the *Fluent Theory Guide*
   - **To Phase Scaling Factor**: C_v in Equation 14.619 in the *Fluent Theory Guide*

   If you are using the two-resistance option for heat transfer, these values act as multipliers for the phase heat transfer coefficients determined for each phase and the default value of `1` is usually appropriate. If you are using one of the other heat transfer options, then these parameters correspond to the coefficient $coeff$ (Equation 14.617 in the Theory Guide) and should be tuned based on experimental data.

   Note:

   For a phase pair, only one evaporation-condensation mechanism that uses the Thermal Phase Change model can be defined, and it cannot be combined with other mass transfer mechanisms.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.2.10.2.6. Species-Mass-Transfer Mechanism

The **species-mass-transfer** mechanism enables you to model generalized interphase species mass transfer in either the Mixture model or the Eulerian model subject to the following conditions:

- Both phases consist of mixtures with at least two species, and at least one of species is present in both phases.
- The two mixture phases are in contact and separated by an interface.
- Species mass transfer can only occur between the same species from one phase to the other. For example, evaporation/condensation between water liquid and water vapor.
- As in all interphase mass transfer models in Fluent, if a gas mixture is involved in an interphase species mass transfer process, it is always treated as a **To Phase** and the liquid mixture as a **From Phase**.
- For the species involved in the mass transfer, the mass fractions in both phases must be determined by solving transport equations. For example, the mass fractions of water liquid and water vapor in an evaporation/condensation case must be solved directly from the governing equations, rather than algebraically or from the physical constraint relations.

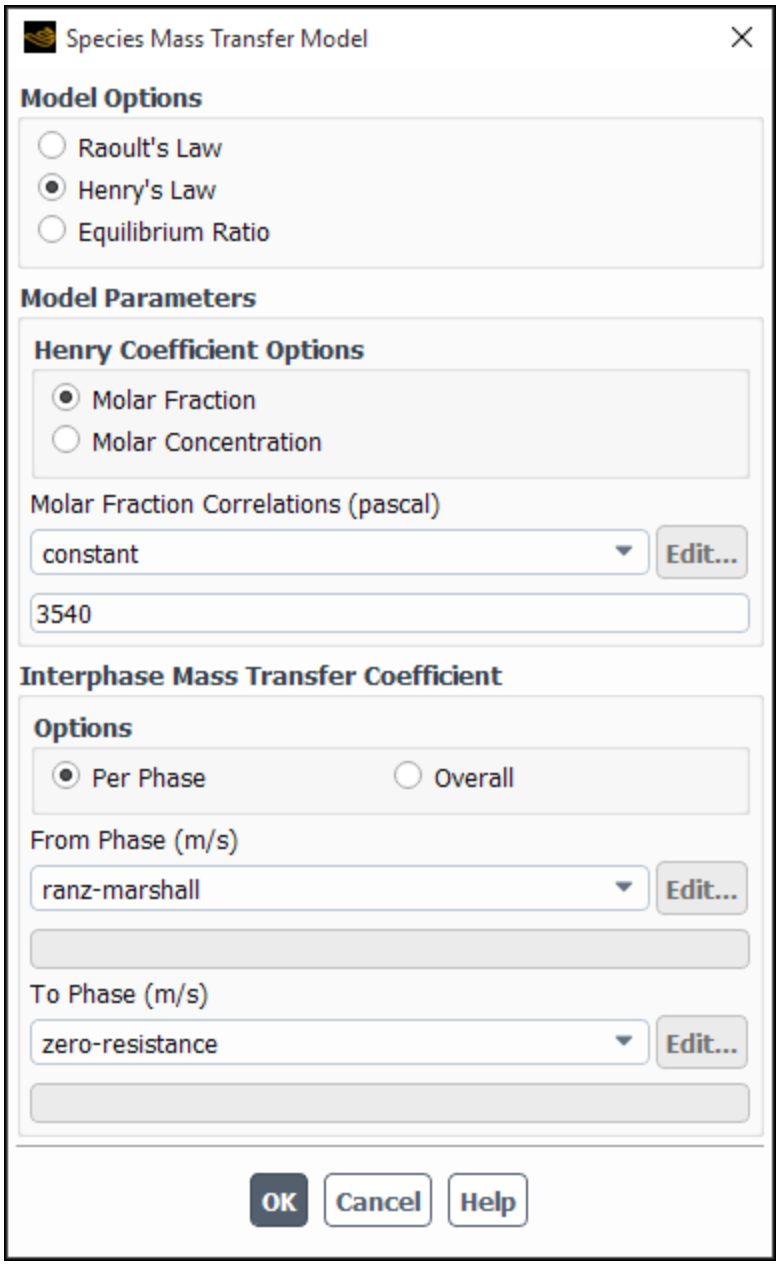
When you select **species-mass-transfer** under **Mechanism** in the **Phase Interaction** > **Heat, Mass, Reactions** > **Mass** tab of the **Multiphase Model** dialog box, you will be prompted with the **Species Mass Transfer Model** dialog box.

Note:

If multiple phase pairs have the **species-mass-transfer** mechanism selected, the same settings in the **Species Mass Transfer Model** dialog box will be used for all phase pairs.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.18: The Species Mass Transfer Model Dialog Box



The steps to configure the species mass transfer model are as follows:

1. Under **Model Options**, choose the sub-model for determining the dynamic equilibrium relations between the same species in a pair of phases.

   Raoult's Law

   Use Raoult's Law as described in Raoult's Law in the *Fluent Theory Guide*. This model is appropriate for gas-liquid cases in which the liquid phase can be modeled as an ideal liquid mixture. If you use Raoult's Law, you need to specify the saturation pressure as a constant, polynomial, piecewise-linear, piece-wise polynomial, or as a user-defined function using the `DEFINE_PROPERTY` macro. You can also specify the saturation

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

pressure using RGP tables as described in Defining Saturation Properties via RGP Tables (p. 1735).

Henry's Law

Use Henry's Law as described in Henry's Law in the *Fluent Theory Guide*. This model is appropriate for cases in which a gas species is dissolved into a liquid mixture phase. If you use Henry's Law you must specify the method for the **Molar Fraction** or **Molar Concentration**. You can choose a constant, **vant-hoff** (to use the Van't Hoff correlation), or a user-defined function using the `DEFINE_MASS_TRANSFER` macro. For the **vant-hoff** model, you will need to specify **Reference Henry Constant** (K_H^0 in Equation 14.658 in the *Fluent Theory Guide*) and **Temperature Dependence** ($\frac{\Delta_{soln}H}{R}$) in the **Vant Hoff's Correlation** dialog box.

Equilibrium Ratio

Use the Equilibrium Ratio method as described in Equilibrium Ratio in the *Fluent Theory Guide*. This model allows you to specify the equilibrium ratio as a constant, or as a user-defined function. If you use **Equilibrium Ratio** you must specify a constant or user-defined function using the `DEFINE_MASS_TRANSFER` macro for **Molar Fraction**, **Molar Concentration**, or **Mass Fraction**.

2. Under **Interphase Mass Transfer Coefficient**, choose whether you want to specify the mass transfer coefficient on a per-phase basis or on all phases:

- If you selected **Per Phase**, you can choose the sub-models for calculating the phase-specific mass transfer coefficients k_p and k_q for the source phase p and q the destination phase, respectively using Equation 14.647 in the *Fluent Theory Guide*. The following models are available in Ansys Fluent:

  constant

  Specify a constant value for the mass transfer coefficient.

  sherwood-number

  Specify a constant value for the Sherwood Number. The mass transfer coefficient is calculated from Equation 14.661 in the *Fluent Theory Guide*.

  ranz-marshall

  Use the Ranz-Marshall model as described in Ranz-Marshall Model in the *Fluent Theory Guide*. The Ranz-Marshall model is based on boundary layer theory for steady flow past a spherical particle. It is applicable under the following flow conditions:

$$0 \leq \mathrm{Re}_q < 200$$
$$0 \leq \mathrm{Sc}_q < 250$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

hughmark

Use the Hughmark model as described in Hughmark Model in the *Fluent Theory Guide*. The Hughmark model extends the Ranz-Marshall model for a wider range of relative Reynolds number, Re_q.

higbie

Use the Higbie model as described in Higbie Model in the *Fluent Theory Guide*. Note the following when using the Higbie method:

- The Higbie method is suitable for modeling mass transfer between gas and liquid phases such as in the simulation of oxygen dissolution in mixing tanks.
- The Higbie correlation is valid only for turbulent flows.
- The primary phase should be modeled as the liquid phase.
- When using the Higbie species-mass-transfer mechanism, **higbie** must be selected for the liquid primary phase, while any available method can be used for the secondary phase.

zero-resistance

Specify zero-resistance mass transfer between the bulk phase and interface.

user-defined

Use a user-defined function for the mass transfer coefficient defined with the `DEFINE_MASS_TRANSFER` macro.

The selection of physically correct mass transfer correlations is highly problem dependent. In the case of mass transfer between a continuous phase and a dispersed phase of approximately spherical particles, the following should be adequate under most situations:

- **ranz-marshall** or **hughmark** for the continuous phase
- **zero-resistance** for the dispersed phase

  Using **zero-resistance** for the dispersed phase implies that the mass transfer occurs very quickly from the interface to the bulk of the dispersed phase. Consequently, the interface concentration and bulk concentration in the dispersed phase are equal. This is usually valid for droplet evaporation or gas absorption/dissolution in bubbles. If there exists significant resistance to the mass transfer on the dispersed phase side, one of the other correlations should be chosen for the dispersed phase.

- If you select **Overall**, you can specify the **Overall Mass Transfer Coefficient** k_{pq} in Equation 14.647 in the *Fluent Theory Guide* either as a **constant** or as a **user-defined** function. These options are the same as those for the **Per Phase** option. The **Overall** option is recommended only if the mass transfer coefficient in each phase is unknown.

For more information about these options, see Mass Transfer Coefficient Models in the *Fluent Theory Guide*.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.2.10.2.7. Boiling Mechanism

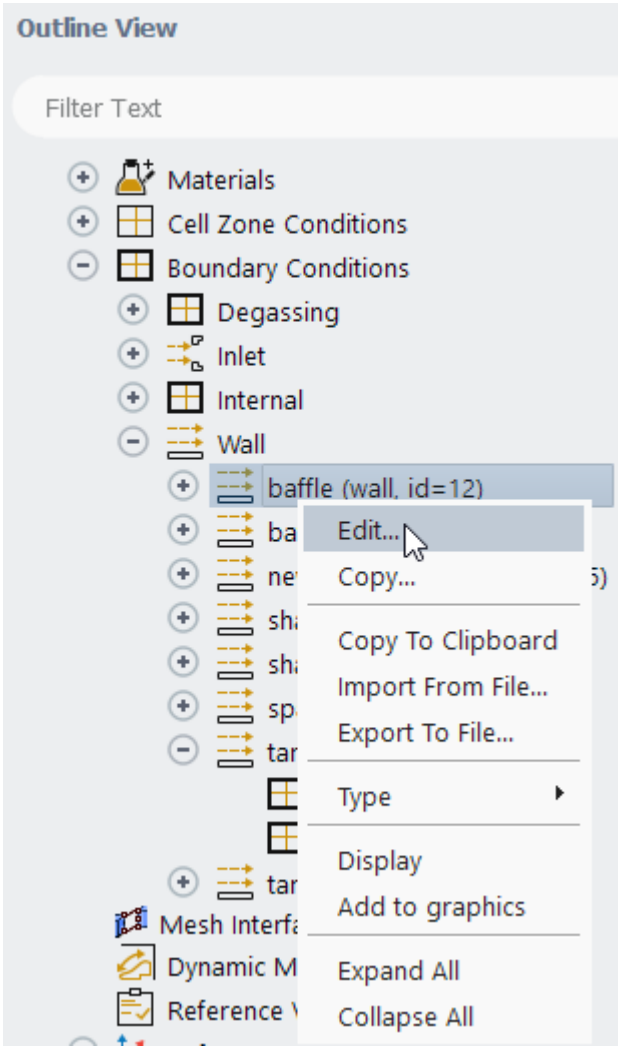
The **boiling** mechanism applies the boiling model as the mass transfer mechanism. This model is only available with the Eulerian multiphase model (when **Boiling** is enabled in the **Multiphase Model** dialog box). For information about the inputs in Figure 26.87: The Boiling Model Dialog Box (p. 3120), refer to Including the Boiling Model (p. 3113).

26.2.11. Defining Multiphase Cell Zone and Boundary Conditions

The procedure for setting multiphase cell and boundary conditions is slightly different than for single-phase models. You will need to set some conditions separately for individual phases, while other conditions are shared by all phases (that is, the mixture), as described in Boundary and Cell Zone Conditions for the Mixture and the Individual Phases (p. 2965).

26.2.11.1. Steps for Setting Boundary Conditions

Setup → Boundary Conditions → <boundary-type> Edit...



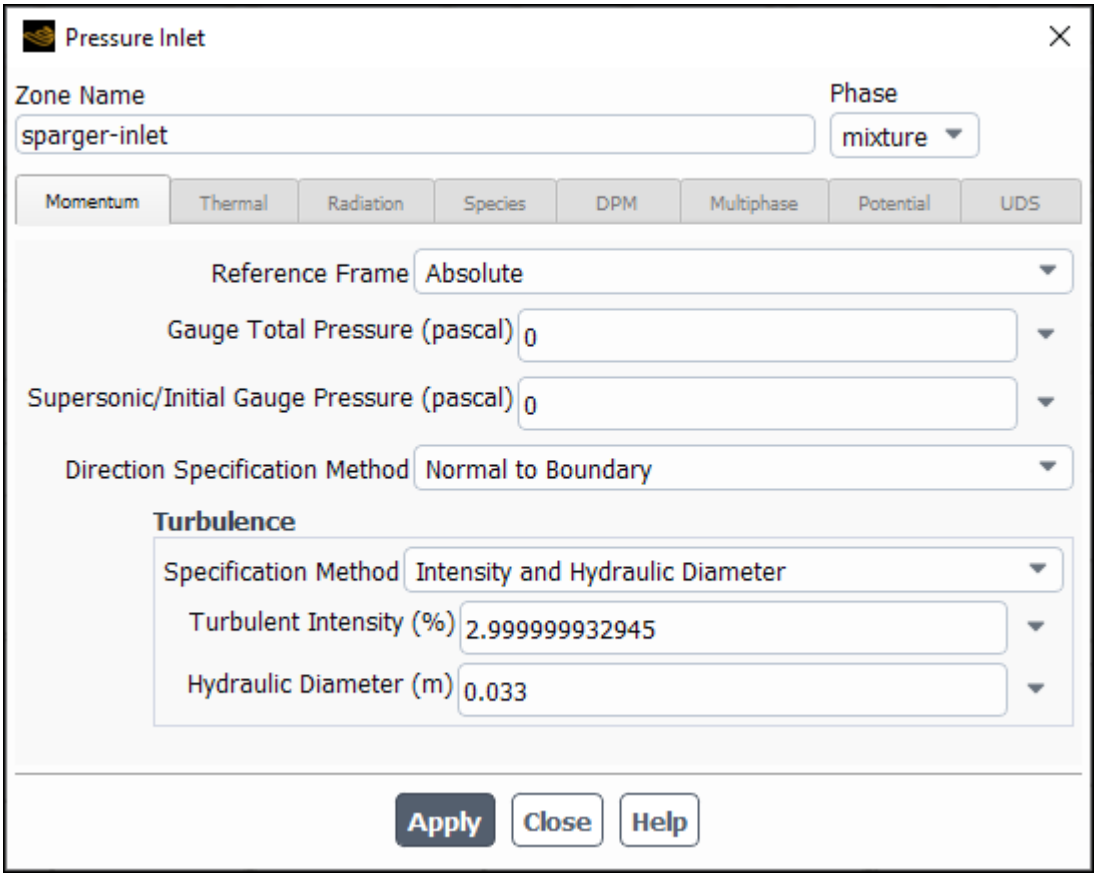
The steps you need to perform for each boundary are as follows:

1. Select the boundary that you want to edit from the Outline View tree, right-click the boundary and click **Edit...**.

2. Set the conditions for the mixture at this boundary, if necessary. (For information about which conditions need to be set for the mixture, refer to Boundary and Cell Zone Conditions for the Mixture and the Individual Phases (p. 2965).)

   a. In the boundary condition dialog box for the selected zone type (for example, the **Pressure Inlet** dialog box for the Eulerian model, shown in Figure 26.19: The Pressure Inlet Dialog Box for a Mixture (p. 2958)), specify the mixture boundary conditions.

   Figure 26.19: The Pressure Inlet Dialog Box for a Mixture



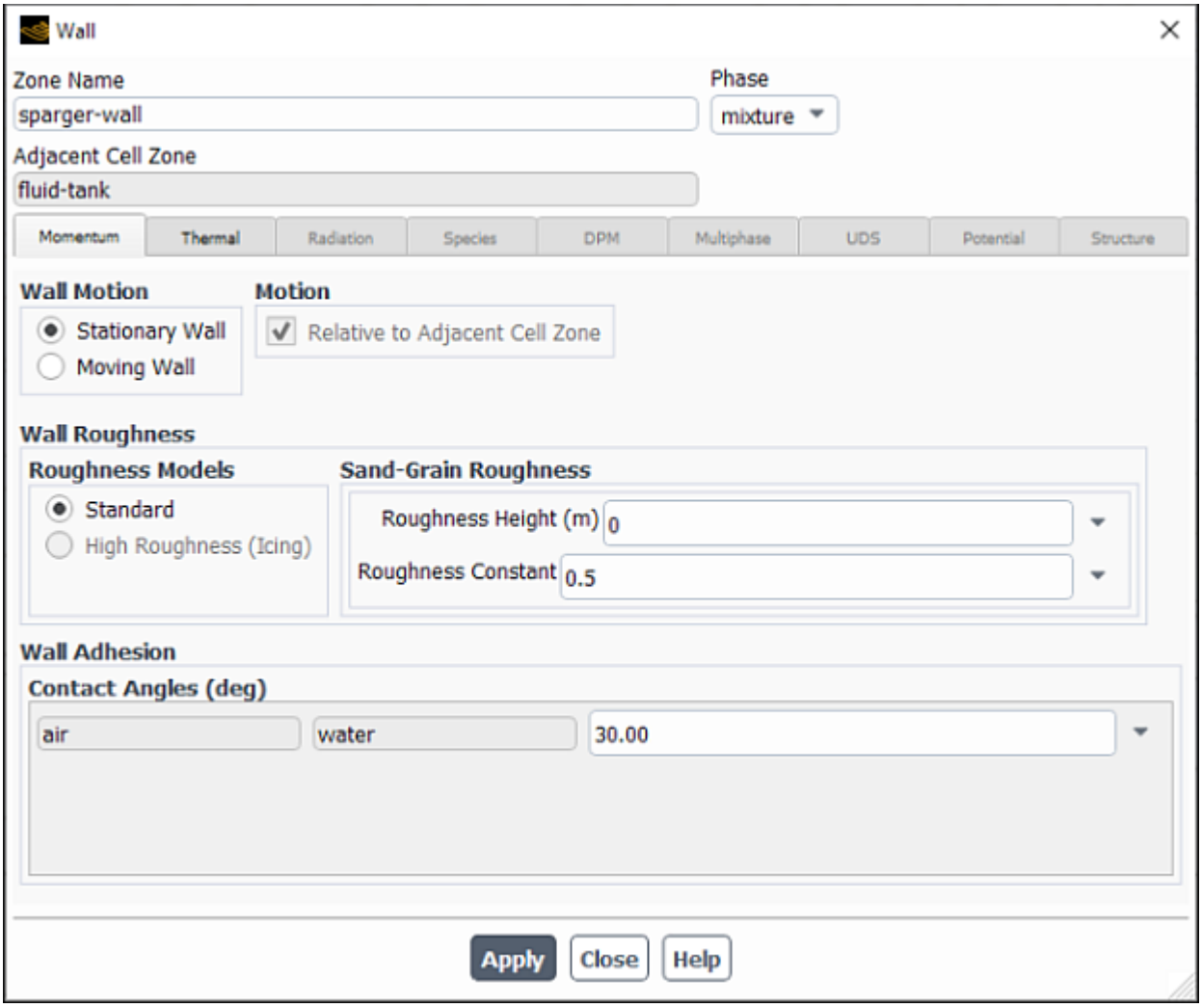
   Note that only those conditions that apply to all phases, as described in Boundary and Cell Zone Conditions for the Mixture and the Individual Phases (p. 2965), will appear in this dialog box.

   For information about inputs related to the relevant boundary conditions, refer to the appropriate sections for those boundary conditions.

   b. If you have enabled the **Wall Adhesion** option in the **Multiphase Model** dialog box (**Phase Interaction** > **Forces**) tab, you need to specify the contact angle at the wall for each pair of phases as a constant (as shown in Figure 26.20: The Wall Dialog Box for a Mixture in a Multiphase Calculation with Wall Adhesion (p. 2959)) or a user-defined function (see the Fluent Customization Manual for more information).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

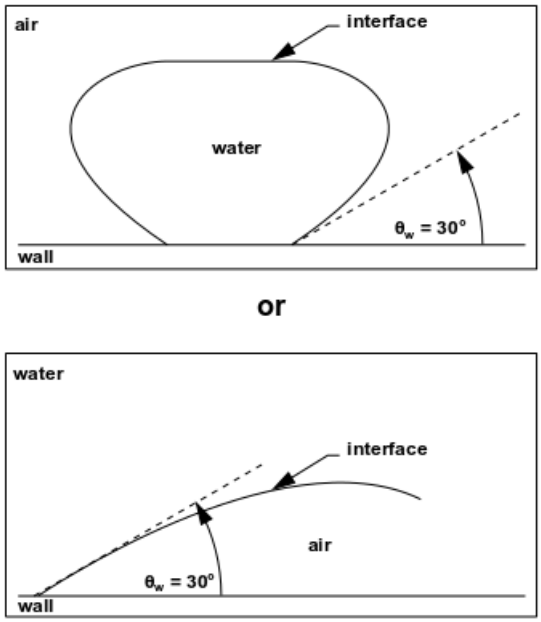
Figure 26.20: The Wall Dialog Box for a Mixture in a Multiphase Calculation with Wall Adhesion



The contact angle (θ_w in Figure 26.21: Measuring the Contact Angle (p. 2960)) is the angle between the wall and the tangent to the interface at the wall, measured inside the phase listed in the left column under **Wall Adhesion** in the **Momentum** tab of the **Wall** dialog box. For example, if you are setting the contact angle between the air and water phases in the **Wall** dialog box shown in Figure 26.20: The Wall Dialog Box for a Mixture in a Multiphase Calculation with Wall Adhesion (p. 2959), θ_w is measured inside the air phase. For more information, refer to Wall Adhesion in the Theory Guide.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.21: Measuring the Contact Angle



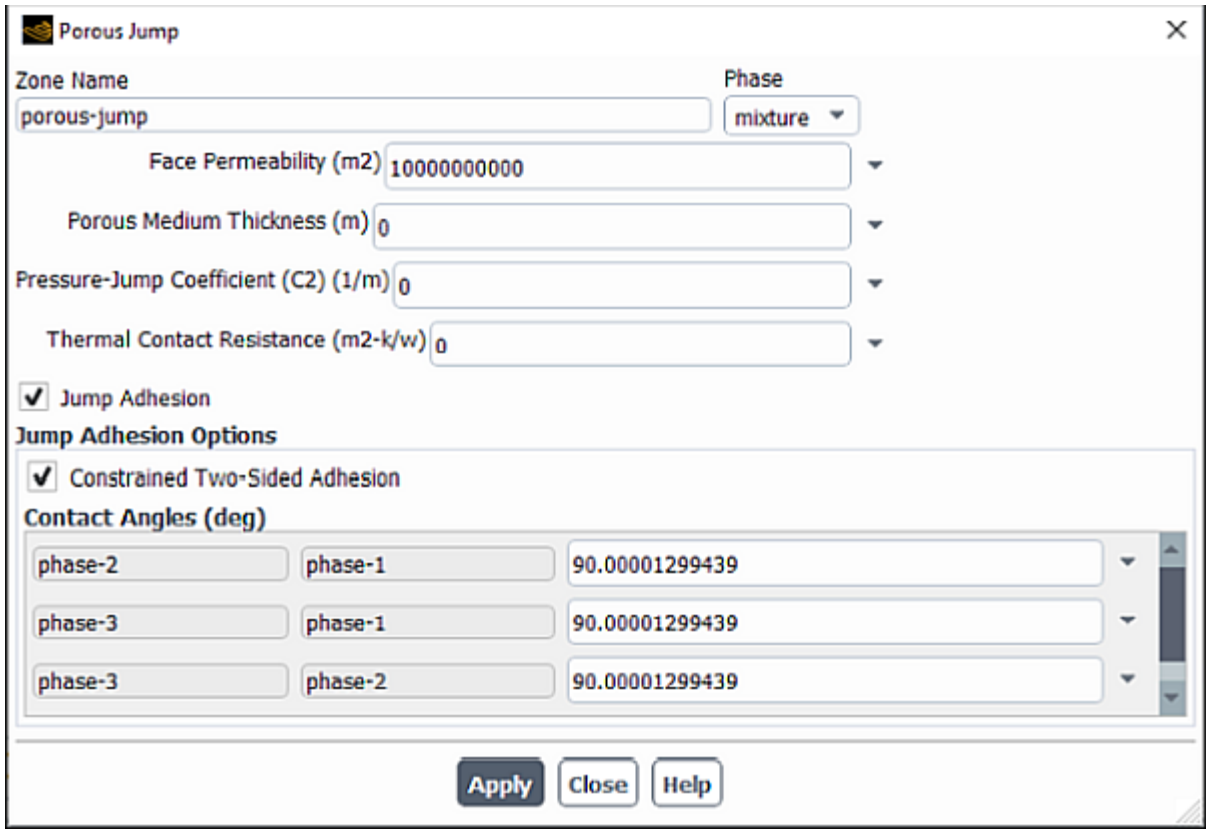
The default value for all pairs is 90 degrees, which means that the interface is normal to the adjacent wall. Note that this is different from not including wall adhesion effects (see Surface Tension in the *Fluent Theory Guide*).

c. Define additional boundary conditions that are specific to the multiphase models.

   i. (VOF model) If you enabled the **Jump Adhesion** option in the **Phase Interaction** > **Forces** tab in the **Multiphase Model** dialog box (**Global Options** group box), specify the contact angle at the porous jump for each pair of phases. When the **Jump Adhesion** option is enabled, it becomes visible at each of the porous jump boundaries. You can enable or disable the **Jump Adhesion** option in the **Porous Jump** dialog boxes and provide the inputs for the contact angle at the desired porous jump boundary, as shown in Figure 26.22: The Porous Jump Dialog Box Displaying Jump Adhesion (p. 2961).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.22: The Porous Jump Dialog Box Displaying Jump Adhesion



The contact angle θ_w is the angle at the porous jump. To constrain the contact angle at the porous jump based on porous or non-porous fluid zones, enable the **Constrained Two-Sided Adhesion** option. Otherwise, if it is disabled, then the forced two-sided adhesion treatment is in effect. For more detail, see Jump Adhesion.

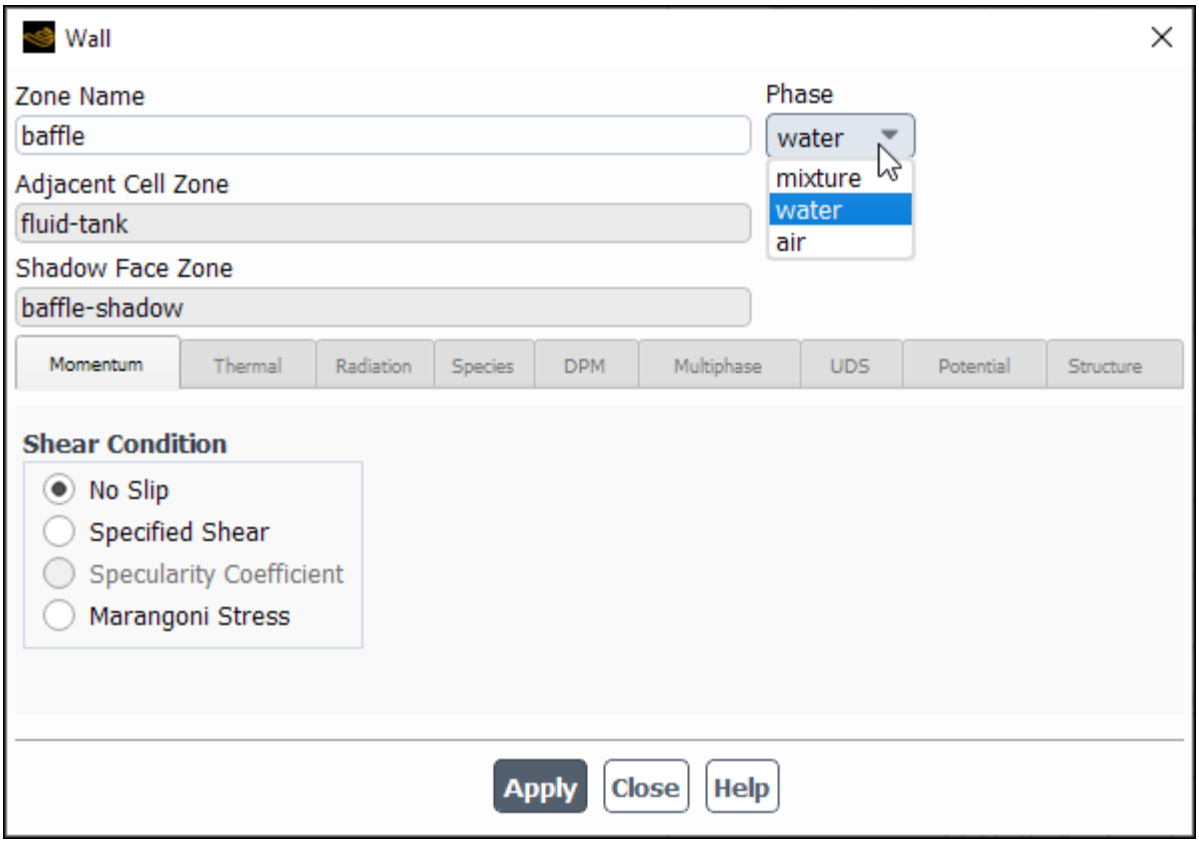
d. Click **OK** once you complete the boundary condition definition for the mixture.

3. Set the conditions for each phase at this boundary, if necessary. (See Boundary and Cell Zone Conditions for the Mixture and the Individual Phases (p. 2965) for information about which conditions need to be set for the individual phases.)

   a. Select the phase from the **Phase** drop-down list, located in the upper-right of each boundary condition dialog box. For example, see Figure 26.23: The Wall Dialog Box for a Phase (p. 2962).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.23: The Wall Dialog Box for a Phase



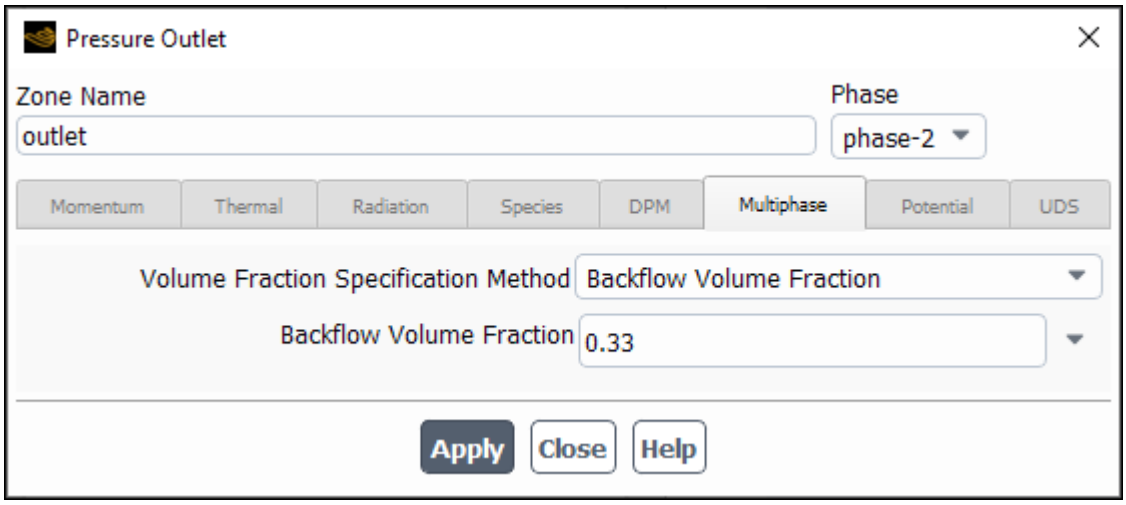
b. Specify the boundary conditions for the phase. Note that only those conditions that apply to the individual phase, as described in Boundary and Cell Zone Conditions for the Mixture and the Individual Phases (p. 2965), will appear in this dialog box.

c. (Secondary phase only) For the pressure outlet, exhaust fan or outlet vent boundary, you can specify the volume fraction specification method as one of the following options:

   - **Backflow Volume Fraction**: You can define the backflow volume fraction as a constant, a profile (see Profiles (p. 1564)), or a user-defined function (see the Fluent Customization Manual).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.24: The Pressure Outlet Dialog Box for a Phase



- **From Neighboring Cell**: This option does not require the specification of the backflow volume fraction, and the volume fraction value is directly taken from the neighboring cells.

  In the case of reverse flow at the pressure outlet boundary, sometimes it becomes tricky to specify the correct values for the backflow volume fraction. Incorrect specification might lead to instability, convergence issues and unphysical behavior. The backflow volume fraction specification from the neighboring interior cells can be a good choice if information at the pressure outlet boundary is not known.

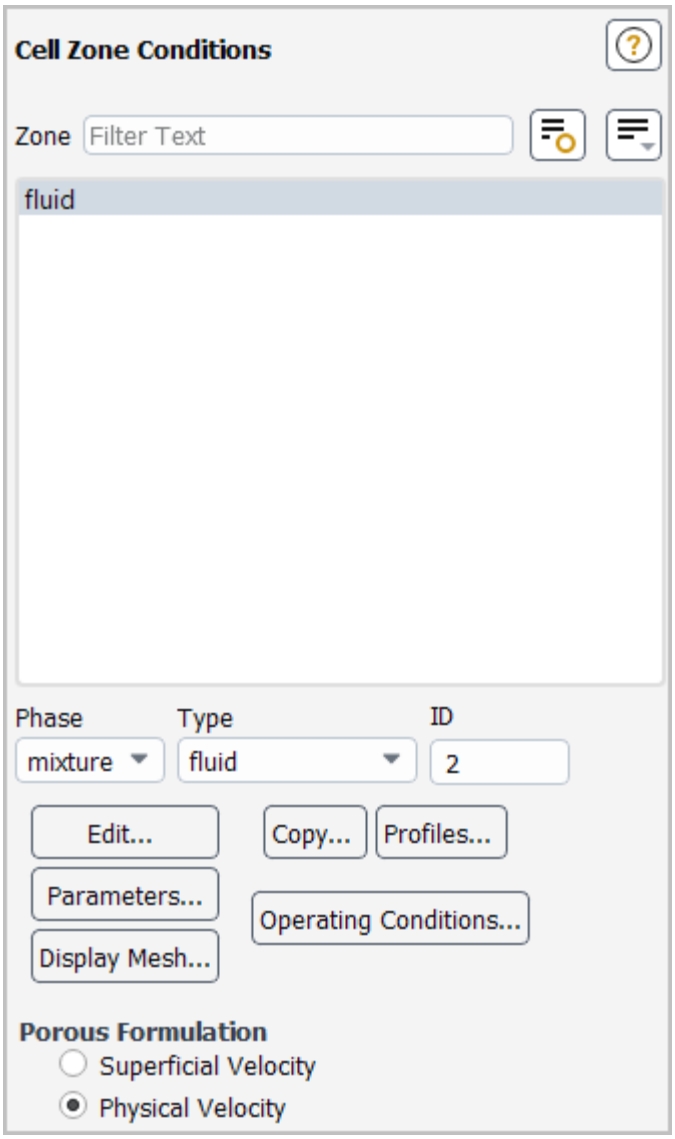
d. Click **Apply** once you complete the boundary condition definition for the phase.

26.2.11.2. Steps for Setting Cell Zone Conditions

Setup → Cell Zone Conditions

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.25: The Cell Zone Conditions Task Page



The steps for specifying the mixture and individual phase cell zone conditions are listed below:

1. Select the cell zone in the **Zone** list in the **Cell Zone Conditions** task page.
2. If needed, set the cell zone conditions for the mixture. (For information about which conditions need to be set for the mixture, refer to Boundary and Cell Zone Conditions for the Mixture and the Individual Phases (p. 2965).)
   a. In the **Phase** drop-down list, select **mixture**.
   b. Make sure that the selected **Type** is appropriate for this zone and click **Edit...**.
   c. In the **Fluid** dialog box, set the cell zone conditions that are applicable to the mixture.
   d. Define additional cell zone conditions that are specific to the multiphase models.
      i. If **Zonal Discretization** is enabled in the **Interface Modeling Options** dialog box, you can specify the **Compressive Scheme Slope Limiter** in the **Multiphase** tab of the **Fluid**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

dialog box. The setting of the slope limiter can be any value between 0 and 2, inclusive. Values of 0, 1, and 2 correspond to first order upwind, second order upwind, and compressive discretization, respectively with intermediate values resulting in a blended formulation. For more details about the slope limiter, refer to The Compressive Scheme and Interface-Model-based Variants in the *Fluent Theory Guide*.

3. If required, set the conditions for each phase. (See Boundary and Cell Zone Conditions for the Mixture and the Individual Phases (p. 2965) for information about which conditions need to be set for the individual phases.)

   a. In the **Phase** drop-down list, select the phase (for example, **water**) and click **Edit...**.

   b. In the **Fluid** dialog box, set the cell zone conditions that are applicable to the individual phase.

For more information about inputs for cell zone conditions, see Fluid Conditions (p. 1320).

26.2.11.3. Boundary and Cell Zone Conditions for the Mixture and the Individual Phases

The boundary conditions you need to specify for the mixture and for the individual phases will depend on which multiphase model you are using.

Details for each model are provided in the following sections:

26.2.11.3.1. VOF Model

26.2.11.3.2. Mixture Model

26.2.11.3.3. Eulerian Model

26.2.11.3.1. VOF Model

If you are using the VOF model, the conditions you need to specify for each type of boundary zone are listed below and summarized in Table 26.4: Phase-Specific and Mixture Conditions for the VOF Model (p. 2966).

- For an inlet vent, intake fan, pressure inlet, or velocity inlet, there are no conditions to be specified for the primary phase. For each secondary phase, you will need to set the volume fraction as a constant, a profile (see Profiles (p. 1564)), or a user-defined function (see the Fluent Customization Manual). All other conditions are specified for the mixture.

- For an exhaust fan, outlet vent, or pressure outlet, there are no conditions to be specified for the primary phase. For each secondary phase, you will need to specify the volume fraction specification method as either **Backflow Volume Fraction** or **From Neighboring Cell** (see step 3. in Steps for Setting Boundary Conditions (p. 2957)). All other conditions are specified for the mixture.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For a mass-flow inlet, you will need to set the mass flow rate, mass flux, or average mass flux for each individual phase. All other conditions are specified for the mixture.

  Important:

  Note that if you read a VOF case that was set up in a version of Ansys Fluent prior to 6.1, you will need to redefine the conditions at the mass-flow inlets.

- For an axis, fan, outflow, periodic, porous jump, radiator, solid, symmetry, or wall zone, all conditions are specified for the mixture. There are no conditions to be set for the individual phases.
- For a wall zone, you can specify the contact angle for the mixture if the wall adhesion option is enabled.
- For a fluid zone, mass sources are specified for the individual phases, and all other sources are specified for the mixture.
  - If the fluid zone is not porous, all other conditions are specified for the mixture.
  - If the fluid zone is porous, you will enable the **Porous Zone** option in the **Fluid** dialog box for the mixture. The porosity inputs (if relevant) are also specified for the mixture. The resistance coefficients and direction vectors, however, are specified separately for each phase. See User Inputs for Porous Media (p. 1343) for the mixture.
  - If **Open Channel Flow** and/or **Open Channel Wave BC** is/are enabled in the **Multiphase Model** dialog box, then the **Numerical Beach** option becomes available under the **Multiphase** tab of the **Fluid** dialog box. To learn how to include numerical beach in your simulation, refer to Numerical Beach Treatment for Open Channels (p. 3005).

See Cell Zone and Boundary Conditions (p. 1299) for details about the relevant conditions for each type of boundary. Note that the pressure far-field boundary is not available with the VOF model.

Table 26.4: Phase-Specific and Mixture Conditions for the VOF Model

Type	Primary Phase	Secondary Phase	Mixture
inlet vent, intake fan, pressure inlet, velocity inlet	nothing	volume fraction	all others
exhaust fan, outlet vent, pressure outlet	nothing	volume fraction specification method	all others
mass-flow inlet	mass flow/flux	mass flow/flux	all others
axis; fan; outflow; periodic; porous jump; radiator; solid; symmetry; wall	nothing	nothing	all others
pressure far-field	not available	not available	not available
fluid	mass source; other porous inputs	mass source; other porous inputs	porous zone; porosity; all others

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.2.11.3.2. Mixture Model

If you are using the mixture model, the conditions you need to specify for each type of boundary zone are listed below and summarized in Table 26.5: Phase-Specific and Mixture Conditions for the Mixture Model (p. 2968).

- For an exhaust fan, outlet vent, or pressure outlet, there are no conditions to be specified for the primary phase. For each secondary phase, you will need to specify the volume fraction specification method as either **Backflow Volume Fraction** or **From Neighboring Cell** (step 3 in Steps for Setting Boundary Conditions (p. 2957). All other conditions are specified for the mixture.

- For an inlet vent, intake fan, or pressure inlet, you will specify for the mixture which direction specification method will be used at this boundary (**Normal to Boundary** or **Direction Vector**). If you select the **Direction Vector** specification method, you will specify the coordinate system (3D only) and flow-direction components for the individual phases. For each secondary phase, you will need to set the volume fraction (as described above). All other conditions are specified for the mixture.

- For a mass-flow inlet, you will need to set the mass flow rate, mass flux, or average mass flux for each individual phase. All other conditions are specified for the mixture.

  Important:

  Note that if you read a mixture multiphase case that was set up in a version of Ansys Fluent previous to 6.1, you will need to redefine the conditions at the mass-flow inlets.

- For a velocity inlet, you will specify the velocity for the individual phases. For each secondary phase, you will need to set the volume fraction (as described above). All other conditions are specified for the mixture.

- For an axis, fan, outflow, periodic, porous jump, radiator, solid, symmetry, or wall zone, all conditions are specified for the mixture. There are no conditions to be set for the individual phases. Outflow boundary conditions are not available for the cavitation model.

- For a fluid zone, mass sources are specified for the individual phases, and all other sources are specified for the mixture.

  If the fluid zone is not porous, all other conditions are specified for the mixture.

  If the fluid zone is porous, you will enable the **Porous Zone** option in the **Fluid** dialog box for the mixture. The porosity inputs (if relevant) are also specified for the mixture. The resistance coefficients and direction vectors, however, are specified separately for each phase. See User Inputs for Porous Media (p. 1343) for details about these inputs. All other conditions are specified for the mixture.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See Cell Zone and Boundary Conditions (p. 1299) for details about the relevant conditions for each type of boundary. Note that the pressure far-field boundary is not available with the mixture model.

Table 26.5: Phase-Specific and Mixture Conditions for the Mixture Model

Type	Primary Phase	Secondary Phase	Mixture
exhaust fan; outlet vent; pressure outlet	nothing	volume fraction specification method	all others
inlet vent; intake fan; pressure inlet	coord. system; flow direction	coord. system; flow direction; volume fraction	dir. spec. method; all others
mass-flow inlet	mass flow/flux	mass flow/flux	all others
velocity inlet	velocity	velocity; volume fraction	all others
axis; fan; outflow (n/a for cavitation model); periodic; porous jump; radiator; solid; symmetry; wall	nothing	nothing	all others
pressure far-field	not available	not available	not available
fluid	mass source; other porous inputs	mass source; other porous inputs	porous zone; porosity; all others

26.2.11.3.3. Eulerian Model

If you are using the Eulerian model, the conditions you need to specify for each type of boundary zone are listed below and summarized in

- Table 26.6: Phase-Specific and Mixture Conditions for the Eulerian Model (for Laminar Flow) (p. 2971)
- Table 26.7: Phase-Specific and Mixture Conditions for the Eulerian Model (with the Mixture Turbulence Model) (p. 2972)
- Table 26.8: Phase-Specific and Mixture Conditions for the Eulerian Model (with the Dispersed Turbulence Model) (p. 2972)
- Table 26.9: Phase-Specific and Mixture Conditions for the Eulerian Model (with the Per-Phase Turbulence Model) (p. 2973)

Note that the specification of turbulence parameters will depend on which of the three multiphase turbulence models you are using, as indicated in

- Table 26.7: Phase-Specific and Mixture Conditions for the Eulerian Model (with the Mixture Turbulence Model) (p. 2972)
- Table 26.8: Phase-Specific and Mixture Conditions for the Eulerian Model (with the Dispersed Turbulence Model) (p. 2972)
- Table 26.9: Phase-Specific and Mixture Conditions for the Eulerian Model (with the Per-Phase Turbulence Model) (p. 2973)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See Turbulence Models in the Theory Guide and Modeling Turbulence (p. 3089) for more information about multiphase turbulence models.

- For an exhaust fan, outlet vent, or pressure outlet, there are no conditions to be specified for the primary phase if you are modeling laminar flow or using the mixture turbulence model (the default multiphase turbulence model), except for backflow total temperature if heat transfer is on.

  For each secondary phase, you will need to specify the volume fraction specification method as either **Backflow Volume Fraction** or **From Neighboring Cell** (step 3 in Steps for Setting Boundary Conditions (p. 2957). If the phase is granular, you will also need to set its backflow granular temperature. If heat transfer is on, you will also need to set the backflow total temperature.

  If you are using the mixture turbulence model, you will need to specify the turbulence boundary conditions for the mixture. If you are using the dispersed turbulence model, you will need to specify them for the primary phase. If you are using the per-phase turbulence model, you will need to specify them for the primary phase and for each secondary phase.

  All other conditions are specified for the mixture.

- For an inlet vent, intake fan, or pressure inlet, you will specify for the mixture which direction specification method will be used at this boundary (**Normal to Boundary** or **Direction Vector**). If you select the **Direction Vector** specification method, you will specify the coordinate system (3D only) and flow-direction components for the individual phases. If heat transfer is on, you will also need to set the total temperature for the individual phases.

  For each secondary phase, you will need to set the volume fraction (as described above). If the phase is granular, you will also need to set its granular temperature.

  If you are using the mixture turbulence model, you will need to specify the turbulence boundary conditions for the mixture. If you are using the dispersed turbulence model, you will need to specify them for the primary phase. If you are using the per-phase turbulence model, you will need to specify them for the primary phase and for each secondary phase.

  All other conditions are specified for the mixture.

- For a mass-flow inlet, you will need to set the mass flow rate, mass flux, or average mass flux for each individual phase. You will also need to specify the temperature of each phase, since the energy equations are solved for each phase.

  For mass-flow inlet boundary conditions, you can specify the slip velocity between phases. When you select a mass-flow inlet boundary for the secondary phase, two options will be available for the **Slip Velocity Specification Method**, as shown in Figure 26.26: Mass-Flow Inlet Boundary Condition Dialog Box (p. 2970) :

  – **Velocity Ratio**

    The value for the **Velocity Ratio** is the secondary phase to primary phase velocity ratio. By default, it is 1.0, which means velocities are the same (no slip). By entering a ratio that is greater than 1.0, you are indicating a larger secondary phase velocity. Otherwise, you can enter a ratio that is less than 1.0 to indicate a smaller secondary phase velocity.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– **Volume Fraction**

If you specify the volume fraction at an inlet, Ansys Fluent will calculate the phase velocities.

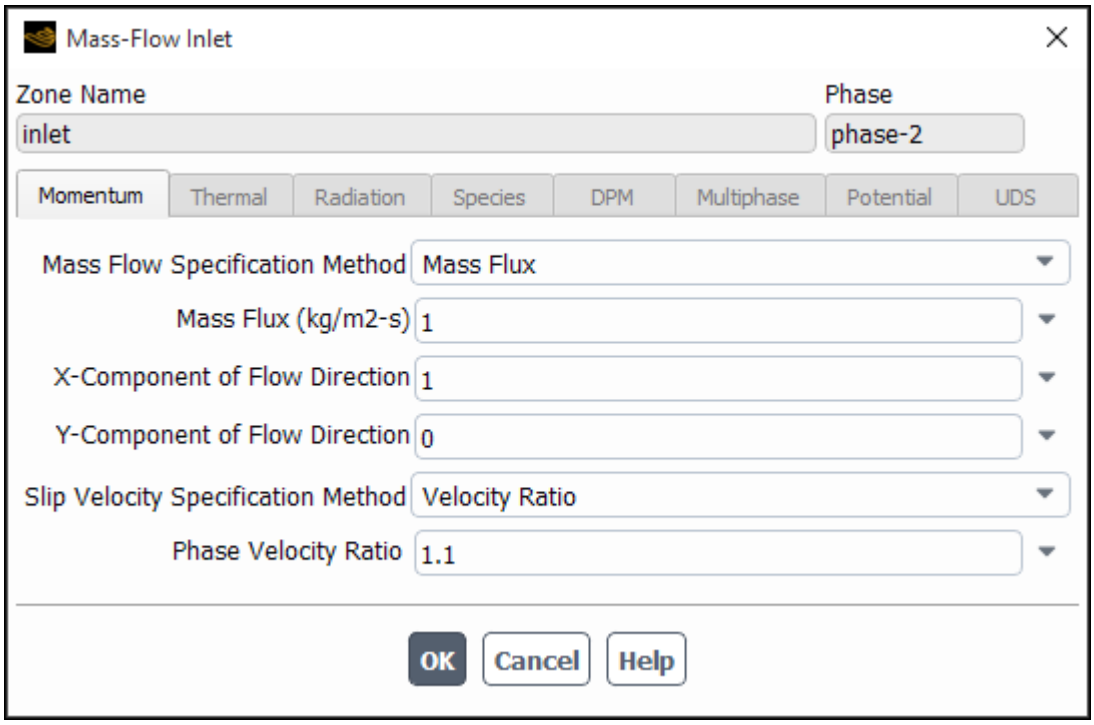
Important:

– If a secondary phase has zero mass flux (that is, the Eulerian model is used to run a single phase case), neither **Velocity Ratio** nor **Volume Fraction** will affect the solution.

– The **Velocity Ratio** and **Volume Fraction** slip velocity methods cannot be mixed for different secondary phases. That is, you should specify one or the other condition for all phases.

The volume fraction is basically the fraction of area covered by that phase.

Figure 26.26: Mass-Flow Inlet Boundary Condition Dialog Box



- For a velocity inlet, you will specify the velocity for the individual phases. If heat transfer is on, you will also need to set the static temperature for the individual phases.

  For each secondary phase, you will need to set the volume fraction (as described above). If the phase is granular, you will also need to set its granular temperature.

  If you are using the mixture turbulence model, you will need to specify the turbulence boundary conditions for the mixture. If you are using the dispersed turbulence model, you will need to specify them for the primary phase. If you are using the per-phase turbulence model, you will need to specify them for the primary phase and for each secondary phase.

  All other conditions are specified for the mixture.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For an axis, outflow, degassing, periodic, solid, or symmetry zone, all conditions are specified for the mixture. There are no conditions to be set for the individual phases.
- For a wall zone, shear conditions are specified for the individual phases. All other conditions are specified for the mixture, including thermal boundary conditions, if heat transfer is on.
- For a fluid zone, all source terms and fixed values are specified for the individual phases, unless you are using the mixture turbulence model or the dispersed turbulence model. If you are using the mixture turbulence model, source terms and fixed values for turbulence are specified instead for the mixture. If you are using the dispersed turbulence model, they are specified only for the primary phase.
  - If the fluid zone is not porous, all other conditions are specified for the mixture.
  - If the fluid zone is porous, you will enable the **Porous Zone** option in the **Fluid** dialog box for the mixture. The porosity inputs (if relevant) are also specified for the mixture. The resistance coefficients and direction vectors, however, are specified separately for each phase. See User Inputs for Porous Media (p. 1343) for details about these inputs. All other conditions are specified for the mixture.

See Cell Zone and Boundary Conditions (p. 1299) for details about the relevant conditions for each type of boundary. Note that the pressure far-field, fan, porous jump, radiator, and mass-flow inlet boundaries are not available with the Eulerian model.

Table 26.6: Phase-Specific and Mixture Conditions for the Eulerian Model (for Laminar Flow)

Type	Primary Phase	Secondary Phase	Mixture
exhaust fan; outlet vent; pressure outlet	(tot. temperature)	volume fraction specification method; gran. temperature (tot. temperature)	all others
inlet vent; intake fan; pressure inlet	coord. system; flow direction (tot. temperature)	coord. system; flow direction; volume fraction; gran. temperature (tot. temperature)	dir. spec. method; all others
velocity inlet	velocity (tot. temperature)	velocity; volume fraction; gran. temperature (tot. temperature)	all others
mass-flow inlet	mass flow rate/flux (temperature)	mass flow rate/flux; velocity ratio/volume fraction; gran. temperature (temperature)	all others
axis; outflow; degassing; periodic; solid; symmetry	nothing	nothing	all others
wall	shear condition	shear condition	all others
pressure far-field; fan; porous jump; radiator	not available	not available	not available

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Type	Primary Phase	Secondary Phase	Mixture
fluid	all source terms; all fixed values; other porous inputs	all source terms; all fixed values; other porous inputs	porous zone; porosity; all

Table 26.7: Phase-Specific and Mixture Conditions for the Eulerian Model (with the Mixture Turbulence Model)

Type	Primary Phase	Secondary Phase	Mixture
exhaust fan; outlet vent; pressure outlet	(tot. temperature)	volume fraction specification method; gran. temperature (tot. temperature)	all others
inlet vent; intake fan; pressure inlet	coord. system; flow direction (tot. temperature)	coord. system; flow direction; volume fraction; gran. temperature (tot. temperature)	dir. spec. method; all others
velocity inlet	velocity (tot. temperature)	velocity; volume fraction; gran. temperature (tot. temperature)	all others
mass-flow inlet	mass flow rate/flux (temperature)	mass flow rate/flux; velocity ratio/volume fraction; gran. temperature (temperature)	all others
axis; outflow; degassing; periodic; solid; symmetry	nothing	nothing	all others
wall	shear condition	shear condition	all others
pressure far-field; fan; porous jump; radiator	not available	not available	not available
fluid	other source terms; other fixed values; other porous inputs	other source terms; other fixed values; other porous inputs	source terms for turbulence; fixed values for turbulence; porous zone; porosity; all

Table 26.8: Phase-Specific and Mixture Conditions for the Eulerian Model (with the Dispersed Turbulence Model)

Type	Primary Phase	Secondary Phase	Mixture
exhaust fan; outlet vent; pressure outlet	turb. parameters (tot. temperature)	volume fraction specification method; gran. temperature (tot. temperature)	all others
inlet vent; intake fan; pressure inlet	coord. system; flow direction; turb. parameters; (tot. temperature)	coord. system; flow direction; volume fraction; gran. temperature (tot. temperature)	dir. spec. method; all others
velocity inlet	velocity; turb. parameters (tot. temperature)	velocity; volume fraction; gran. temperature (tot. temperature)	all others

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Type	Primary Phase	Secondary Phase	Mixture
mass-flow inlet	mass flow rate/flux; turb. parameters (temperature)	mass flow rate/flux; velocity ratio/volume fraction; gran. temperature (temperature)	all others
axis; outflow; degassing; periodic; solid; symmetry	nothing	nothing	all others
wall	shear condition	shear condition	all others
pressure far-field; fan; porous jump; radiator	not available	not available	not available
fluid	momentum, mass, turb. sources; momentum, mass, turb. fixed values; other porous inputs	momentum and mass sources; momentum and mass fixed values; other porous inputs	porous zone; porosity; all

Table 26.9: Phase-Specific and Mixture Conditions for the Eulerian Model (with the Per-Phase Turbulence Model)

Type	Primary Phase	Secondary Phase	Mixture
exhaust fan; outlet vent; pressure outlet	turb. parameters (tot. temperature)	volume fraction specification method; turb. parameters; gran. temperature (tot. temperature)	all others
inlet vent; intake fan; pressure inlet	coord. system; flow direction; turb. parameters (tot. temperature)	coord. system; flow direction; volume fraction; turb. parameters; gran. temperature (tot. temperature)	dir. spec. method; all others
velocity inlet	velocity; turb. parameters (tot. temperature)	velocity; volume fraction; turb. parameters; gran. temperature (tot. temperature)	all others
mass-flow inlet	mass flow rate/flux; tub. parameters (temperature)	mass flow rate/flux; velocity ratio/volume fraction; gran. temperature (temperature)	all others
axis; outflow; degassing; periodic; solid; symmetry	nothing	nothing	all others
wall	shear condition	shear condition	all others
pressure far-field; fan; porous jump; radiator	not available	not available	not available
fluid	momentum, mass, turb. sources; momentum, mass, turb. fixed values; other porous inputs	momentum, mass, turb. sources; momentum, mass, turb. fixed values; other porous inputs	porous zone; porosity; all others

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.2.11.4. Steps for Copying Cell Zone and Boundary Conditions

The steps for copying cell zone and boundary conditions for a multiphase flow are slightly different from those described in Copying Cell Zone and Boundary Conditions (p. 1305) for a single-phase flow. The modified steps are listed below:

1. In the **Cell Zone Conditions** or **Boundary Conditions** task page, click the **Copy...** button. This will open the **Copy Conditions** dialog box.
2. In the **From Cell Zone** or **From Boundary Zone** list, select the zone that has the conditions you want to copy.
3. In the **To Cell Zones** or **To Boundary Zones** list, select the zone or zones to which you want to copy the conditions.
4. In the **Phase** drop-down list, select the phase for which you want to copy the conditions (either **mixture** or one of the individual phases).

   Important:

   Note that copying the boundary conditions for one phase does not automatically result in the boundary conditions for the other phases and the mixture being copied as well. You need to copy the conditions for each phase on each boundary of interest.

5. Click **Copy**. Ansys Fluent will set *all* of the selected phase's (or mixture's) boundary conditions on the zones selected in the **To Cell Zones** or **To Boundary Zones** list to be the same as that phase's conditions on the zone selected in the **From Cell Zone** or **From Boundary Zone** list. (You cannot copy a subset of the conditions, such as only the thermal conditions.)

See Copying Cell Zone and Boundary Conditions (p. 1305) for additional information about copying boundary conditions, including limitations.

26.2.12. Setting Initial Conditions

26.2.12.1. Setting Initial Volume Fractions

Once you have initialized the flow (as described in Initializing the Solution (p. 3627)), you can define the initial distribution of the phases. For a transient simulation, this distribution will serve as the initial condition at $t=0$; for a steady-state simulation, setting an initial distribution can provide added stability in the early stages of the calculation.

You can patch an initial volume fraction for each secondary phase using the Patch Dialog Box (p. 5183).

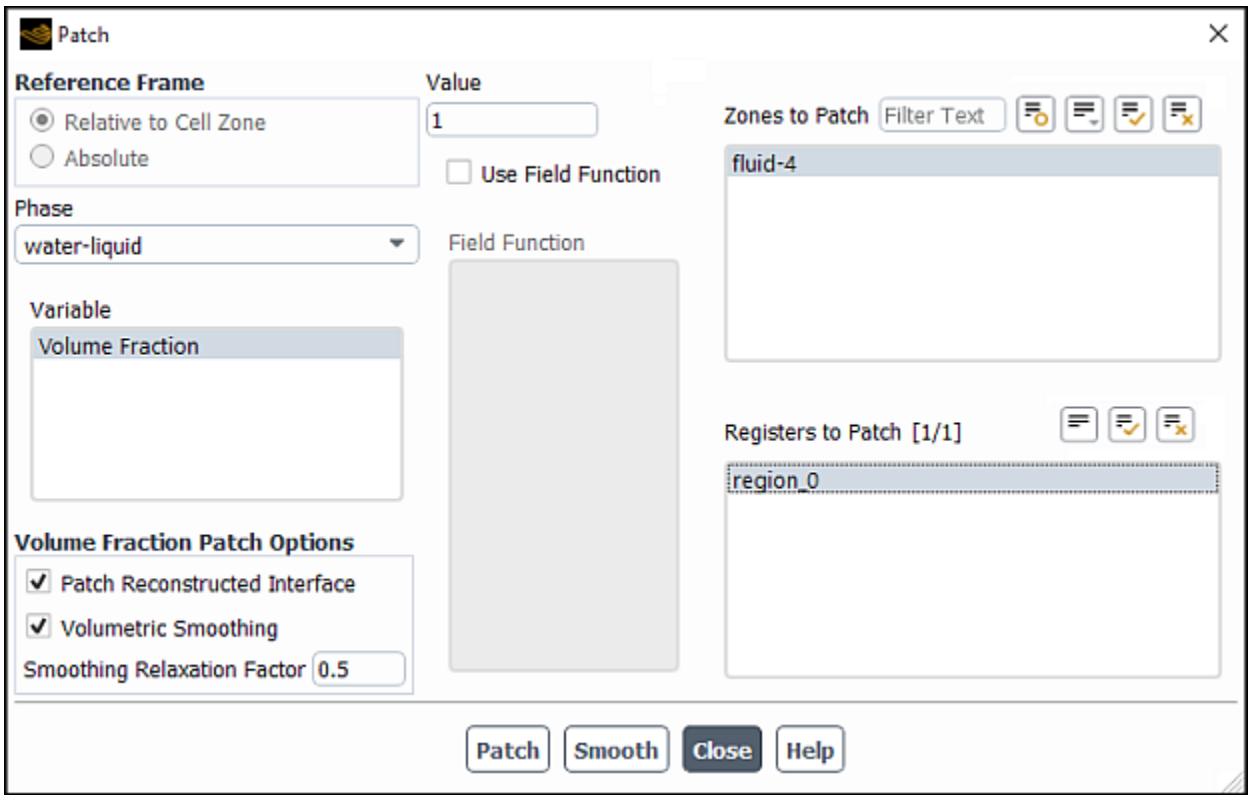
Solution → Initialization → Patch...

If the region in which you want to patch the volume fraction is defined as a separate cell zone, you can simply patch the value there. Otherwise, you can create a cell "register" that contains the appropriate cells and patch the value in the register. See Patching Values in Selected Cells (p. 3630) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.2.12.1.1. Options for Patching Volume Fraction

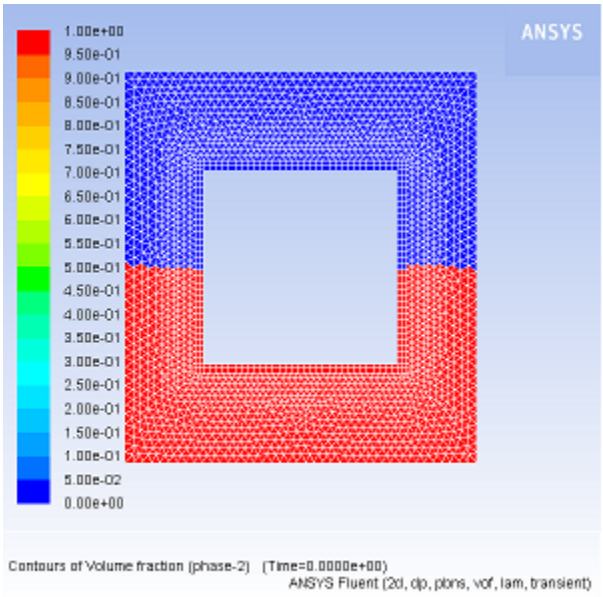
You have additional options for how the volume fraction is patched in a user-defined register. A standard patching procedure will initialize all the cells in the register with the specified volume fraction, without any special treatment for cells that are intersected by the fluid-fluid interface.



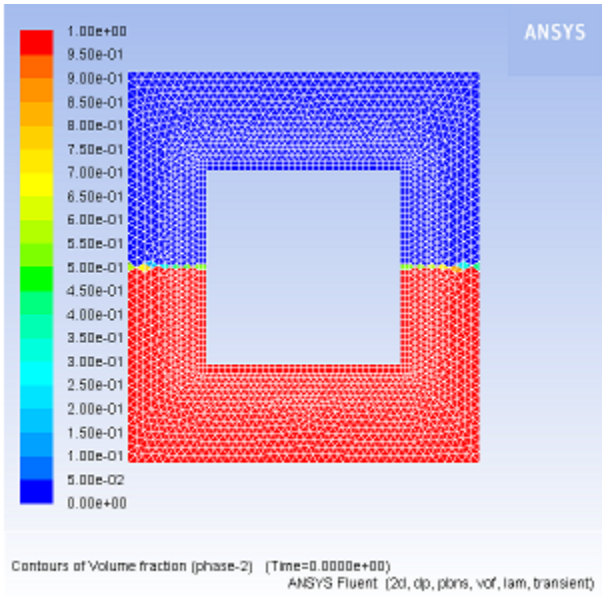
Patch Reconstructed Interface

This option uses piecewise-linear interface reconstruction to identify the fluid-fluid interface and patches the actual volume fraction field in the cells that intersect the interface.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Without Patch Reconstructed Interface



With Patch Reconstructed Interface

Important:

The **Patch Reconstructed Interface** algorithm requires that the register coordinates in the **Region Register** dialog box be well-ordered. That is: **X Min** < **X Max**; **Y Min** < **Y Max**; **Z Min** < **Z Max**. In particular, if you use the **Select Points with Mouse** feature to specify the coordinates, be sure that this condition is met. If the register coordinates are not well-ordered, the register will still be created/displayed as intended; however, the volume fraction value will not be patched within the register.

The **Patch Reconstructed Interface** will be applied only on the last register that was defined, even if multiple registers are selected in the **Patch** dialog box. Therefore, it is recommended that you create and patch registers one at a time.

Volumetric Smoothing

This option smooths the volume fraction field on-demand by taking the volumetric average over the cell neighbors. This can help provide better solution stability at startup. After enabling **Volumetric Smoothing**, you can click **Smooth** to apply the smoothing. Note that the smoothing is applied to the volume fraction field for all phases in the full domain. The **Smoothing Relaxation Factor** controls the degree of smoothing applied. You can specify a value between `0` (no smoothing) and `1` (maximum smoothing). For most cases, the default value is recommended. For fine meshes you may choose to try a larger value to aid solution startup. However, excessively large values can have an undesirable effect during the solution as interface sharpening discretization schemes may deteriorate the initial field.

The **Smooth** command can be used at any time after the solution has been initialized. It can therefore also be used after reading a data file for postprocessing. However, note that the smoothed field will overwrite the original volume fraction field in this case. Also

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

note that you must select a secondary phase variable from the drop-down list in the **Patch** dialog box to expose the **Volumetric Smoothing** option.

26.2.12.2. Setting the Initial Turbulence Field

By default, standard initialization uses domain-averaged values for turbulence variables, which is not ideal for a turbulent flow with multiple phases that have different properties and velocities. Ill-posed initial conditions for turbulence may result in unstable solution, especially at the start of the simulation.

For the VOF and Mixture multiphase models, you can enable the **Localized Turbulence Initialization** option to obtain a better initial guess for turbulence variables, which will promote better startup and run-time stability.

Once you initialize the flow, all turbulent flow variables relevant to the turbulence model will be initialized based on local velocity and local property in each cell. This type of initialization assumes the turbulent intensity of 5% and the turbulent viscosity ratio of 10. You can adjust these values using the following text commands:

```
solve/initialize/mp-localized-turb-init/enable

enable localized initialization of turbulent flow variables? [no] yes

solve/initialize/mp-localized-turb-init/turb-init-parameters

turbulent intensity [0.05]

turbulent viscosity ratio [10]
```

Note that domain-averaged values displayed in the **Initial Values** group box in the **Solution Initialization** task page will be overwritten by the variable turbulence field. You can examine this field by displaying contours of **Turbulence**.

If executing the **Patch** or **Smooth** commands leads to a change in initial conditions, initial values of turbulent flow variables will be automatically recalculated.

26.3. Setting Up the VOF Model

For background information about the VOF model and the limitations that apply, refer to Overview of the VOF Model in the Theory Guide.

This section is organized as follows:

26.3.1. Solving Steady-State VOF Problems

26.3.2. Guidelines for Using the Multiphase Pseudo Time Method

26.3.3. Including Coupled Level Set with the VOF Model

26.3.4. Mesh Adaption with the VOF Model

26.3.5. Modeling Open Channel Flows

26.3.6. Modeling Open Channel Wave Boundary Conditions

26.3.7. Recommendations for Open Channel Initialization

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.3.8. Numerical Beach Treatment for Open Channels
26.3.9. Defining the Phases for the VOF Model
26.3.10. Defining Phase Interaction Terms
26.3.11. Setting Time-Dependent Parameters for the Explicit Volume Fraction Formulation
26.3.12. Modeling Solidification/Melting
26.3.13. Using the VOF-to-DPM Model Transition for Dispersion of Liquid in Gas
26.3.14. Using the DPM-to-VOF Model Transition

26.3.1. Solving Steady-State VOF Problems

For steady-state VOF problems, Ansys Fluent automatically selects **Global Time Step** from the **Pseudo Time Method** list in the **Solution Methods** task page and enables the **Coupled** pressure-velocity coupling scheme, in order to provide better stability and faster convergence. For more information about pseudo time method solutions, see Performing Calculations with a Pseudo Time Method (p. 3673).

26.3.2. Guidelines for Using the Multiphase Pseudo Time Method

When using a **Pseudo Time Method** in multiphase applications, follow the guidelines provided below:

- For the fixed (**User-Specified**) time step method, you must provide a suitable pseudo time step size to the solver. You can use the **Automatic** time step method to estimate the pseudo time step size for the simulation. This can be done by setting the **Verbosity** to 1 and simulating your case for a few iterations. The pseudo time step size will be printed in the Fluent console.
- For the **Automatic** time step method, decreasing the **Time Scale Factor** to a lower value (about 0.3) will provide better solution stability.
- Sometimes, for stiff flow problems, convergence may not be achieved when using the **Automatic** time step method. In such cases, consider switching to the fixed (**User-Specified**) time step method instead.
- For steady-state problems, you should never evaluate convergence based on residuals alone. To better judge convergence, you should also monitor appropriate field variables at a particular location until the value stops changing.
- The **Coupled with Volume Fractions** option is recommended for open channel flow applications.

26.3.3. Including Coupled Level Set with the VOF Model

When using the VOF formulation, you can couple the level set method with it to help overcome some limitations that exist in the interface tracking method of the VOF model and the level set method. To use the coupled level set method with VOF, perform the following:

1. Open the **Multiphase Model** dialog box.

   Setup → **Models** → **Multiphase** → **Edit...**

2. In the **Model** group box (**Models** tab), enable **Volume of Fluid**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. In the **Hybrid Models** group box, enable **Coupled Level Set + VOF** (see Figure 26.1: Multiphase Model Dialog Box for the VOF Model (p. 2919)).

4. (optional) In some cases, spurious currents can arise from the application of the surface tension force in the momentum equation. In these cases you can use the `/define/models/multiphase/coupled-level-set` text command to use a revised formulation of the surface tension force that weights the force towards the heavier phase in the interface cells. You can choose from `none` (the default), `density-correction`, and `heaviside-correction`.

After the **Coupled Level Set + VOF** option is enabled, proceed as you normally would when setting up the VOF model (described in Setting Up the VOF Model (p. 2977)). For theoretical information, refer to Coupled Level-Set and VOF Model in the *Fluent Theory Guide*.

Note:

When using the **Coupled Level Set + VOF** option, the recommended scheme is the geo-reconstruct scheme (see The Geometric Reconstruction Scheme in the *Fluent Theory Guide*).

For additional limitations associated with the level set method, see Limitations in the *Fluent Theory Guide*.

Normally, zero flux of the level set function is set as the default for the boundary conditions. Due to the geometrical re-initialization procedure at each time step, the boundary conditions shall not have any significant effect on the results. For more information, see Re-initialization of the Level-set Function via the Geometrical Method.

26.3.4. Mesh Adaption with the VOF Model

It may be useful to refine and/or coarsen the mesh of your VOF simulation based on the volume fraction of the phases. For information about mesh adaption, see Adapting the Mesh (p. 3773). Note that an easy way to define the settings in the **Manual Mesh Adaption** or **Automatic Mesh Adaption** dialog box for VOF simulations is to use the **Predefined Criteria** drop-down list, as described in VOF Adaption (p. 3795). This list allows you to select from commonly used criteria for adapting the mesh; the **Multiphase...** submenu provides an option for standard VOF cases, as well as options for straightforward or more complex cases that use the VOF-to-DPM model transition mechanism.

26.3.5. Modeling Open Channel Flows

Using the VOF formulation, open channel flows can be modeled in Ansys Fluent. To start using the open channel flow boundary condition, perform the following:

1. Enable **Gravity** and set the gravitational acceleration fields.

   Setup → **General** → **Gravity** → **On**

2. Enable the volume of fluid model.

   a. Open the **Multiphase Model** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setup → **Models** → **Multiphase** → **Edit...**

b. In the **Model** group box (**Models** tab), enable **Volume of Fluid**.

c. In the **Formulation** group box, select either **Implicit** or **Explicit**.

3. In the **VOF Sub-Models** tab, select **Open Channel Flow**.

Note:

The default VOF formulation is set to **Implicit** after enabling the **Open Channel Flow** option. This is done to allow the use of larger time step sizes for such applications.

In order to set specific parameters for a particular boundary for open channel flows, enable the **Open Channel** option in the **Multiphase** tab of the corresponding boundary condition dialog box. Table 26.10: Open Channel Boundary Parameters for the VOF Model (p. 2980) summarizes the types of boundaries available to the open channel flow boundary condition, and the additional parameters needed to model open channel flow. For more information on setting boundary condition parameters, see Cell Zone and Boundary Conditions (p. 1299).

Table 26.10: Open Channel Boundary Parameters for the VOF Model

Boundary Type	Parameter
pressure inlet	Inlet Group ID; Secondary Phase for Inlet; Flow Specification Method; Free Surface Level, Bottom Level; Velocity Magnitude
pressure outlet	Outlet Group ID; Pressure Specification Method; Free Surface Level; Bottom Level
mass-flow inlet	Inlet Group ID; Secondary Phase for Inlet; Free Surface Level; Bottom Level; Mass Flow Rates for the Phases
velocity inlet	Secondary Phase for Inlet; Free Surface Level, Averaged Velocity Inputs/Segregated Velocity Inputs (Available with Open Channel Wave Boundary Condition)
outflow	Flow Rate Weighting

Further details are provided in the following sections:

26.3.5.1. Defining Inlet Groups

26.3.5.2. Defining Outlet Groups

26.3.5.3. Setting the Inlet Group

26.3.5.4. Setting the Outlet Group

26.3.5.5. Determining the Free Surface Level

26.3.5.6. Determining the Bottom Level

26.3.5.7. Specifying the Total Height

26.3.5.8. Determining the Velocity Magnitude

26.3.5.9. Determining the Secondary Phase for the Inlet

26.3.5.10. Determining the Secondary Phase for the Outlet

26.3.5.11. Choosing the Pressure Specification Method

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.3.5.12. Choosing the Density Interpolation Method

26.3.5.13. Open Channel Flow Compatibility with Velocity Inlet

26.3.5.14. Limitations

26.3.5.15. Recommendations for Setting Up an Open Channel Flow Problem

26.3.5.1. Defining Inlet Groups

Open channel systems involve the flowing fluid (the secondary phase) and the fluid above it (the primary phase).

If both phases enter through the separate inlets (for example, `inlet-phase2` and `inlet-phase1`), these two inlets form an inlet group. This inlet group is recognized by the parameter **Inlet Group ID**, which will be same for both the inlets that make up the inlet group. On the other hand, if both the phases enter through the same inlet (for example, `inlet-combined`), then the inlet itself represents the inlet group.

Important:

In three-phase flows, only one secondary phase is allowed to pass through one inlet group.

26.3.5.2. Defining Outlet Groups

Outlet-groups can be defined in the same manner as the inlet groups.

Important:

In three-phase flows, the outlet should represent the outlet group, that is, separate outlets for each phase are not recommended in three-phase flows.

26.3.5.3. Setting the Inlet Group

For pressure inlets and mass-flow inlets, the **Inlet Group ID** is used to identify the different inlets that are part of the same inlet group. For instance, when both phases enter through the same inlet (single face zone), then those phases are part of one inlet group and you would set the **Inlet Group ID** to 1 for that inlet (or inlet group).

In the case where the same inlet group has separate inlets (different face zones) for each phase, then the **Inlet Group ID** will be the same for each inlet of that group.

When specifying the inlet group, use the following guidelines:

- Since the **Inlet Group ID** is used to identify the inlets of the same inlet group, general information such as **Free Surface Level**, **Bottom Level**, or the mass flow rate for each phase should be the same for each inlet of the same inlet group.
- You should specify a different **Inlet Group ID** for each distinct inlet group.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For example, consider the case of two inlet groups for a particular problem. The first inlet group consists of water and air entering through the same inlet (a single face zone). In this case, you would specify an inlet group ID of 1 for that inlet (or inlet group). The second inlet group consists of oil and air entering through the same inlet group, but each uses a different inlet (`oil-inlet` and `air-inlet`) for each phase. In this case, you would specify the same **Inlet Group ID** of 2 for both of the inlets that belong to the inlet group.

26.3.5.4. Setting the Outlet Group

For pressure outlet boundaries, the **Outlet Group ID** is used to identify the different outlets that are part of the same outlet group. For instance, when both phases enter through the same outlet (single face zone), then those phases are part of one outlet group and you would set the **Outlet Group ID** to 1 for that outlet (or outlet group).

In the case where the same outlet group has separate outlets (different face zones) for each phase, then the **Outlet Group ID** will be the same for each outlet of that group.

When specifying the outlet group, use the following guidelines:

- Since the **Outlet Group ID** is used to identify the outlets of the same outlet group, general information such as **Free Surface Level** or **Bottom Level** should be the same for each outlet of the same outlet group.
- You should specify a different **Outlet Group ID** for each distinct outlet group.

  For example, consider the case of two outlet groups for a particular problem. The first outlet group consists of water and air exiting from the same outlet (a single face zone). In this case, you would specify an outlet number of 1 for that outlet (or outlet group). The second outlet group consists of oil and air exiting through the same outlet group, but each uses a different outlet (`oil-outlet` and `air-outlet`) for each phase. In this case, you would specify the same **Outlet Group ID** of 2 for both of the outlets that belong to the outlet group.

Important:

For three-phase flows, when all the phases are leaving through the same outlet, the outlet should consist only of a single face zone.

26.3.5.5. Determining the Free Surface Level

For the appropriate boundary, you need to specify the **Free Surface Level** value. This parameter is available for all relevant boundaries, including pressure outlet, mass-flow inlet, and pressure inlet. The **Free Surface Level**, is represented by y_{local} in Equation 14.42 in the Theory Guide.

$$y_{local} = -\left(\vec{a} \cdot \hat{g}\right) \tag{26.10}$$

where $\vec{a}$ is the position vector of any point on the free surface, and $\hat{g}$ is the unit vector in the direction of the force of gravity. Here a horizontal free surface that is normal to the direction of gravity is assumed.

We can simply calculate the free surface level in two steps:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Determine the absolute value of height from the free surface to the origin in the direction of gravity.

2. Apply the correct sign based on whether the free surface level is above or below the origin.

If the liquid's free surface level lies above the origin, then the **Free Surface Level** is positive (see Figure 26.27: Determining the Free Surface Level and the Bottom Level (p. 2984)). Likewise, if the liquid's free surface level lies below the origin, then the **Free Surface Level** is negative.

You can also specify a transient profiles for a **Free Surface Level** for the relevant open channel boundaries as shown in the example below:

```
/*********************************************************************
Example UDF that demonstrates transient profile for free surface level
*********************************************************************/

#include "udf.h"

#define H 1.5   /* Original Free surface level */
#define T0 0.2  /* Time */

DEFINE_TRANSIENT_PROFILE(fs_level, current_time)
{
    real level;
    if (current_time <= T0)
      level = H - current_time;
    else
      level = H - T0;

    return level;
}
```

26.3.5.6. Determining the Bottom Level

For the appropriate boundary, you need to specify the **Bottom Level** value. This parameter is available for all relevant boundaries, including pressure outlet, mass-flow inlet, and pressure inlet. The **Bottom Level**, is represented by a relation similar to Equation 14.42 in the Theory Guide.

$$y_{bottom} = -\left(\vec{b} \cdot \hat{g}\right) \tag{26.11}$$

where $\vec{b}$ is the position vector of any point on the bottom of the channel, and $\hat{g}$ is the unit vector of gravity. Here we assume a horizontal free surface that is normal to the direction of gravity.

We can simply calculate the bottom level in two steps:

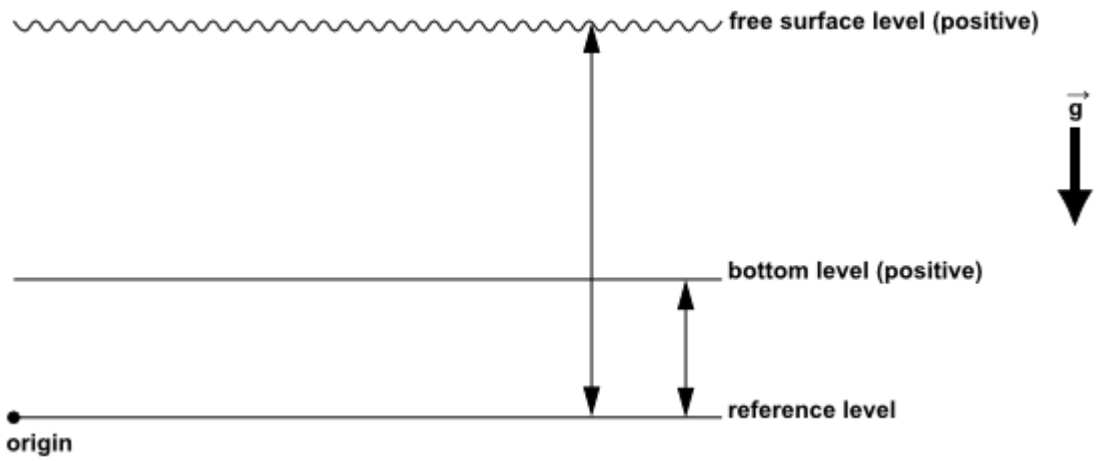
1. Determine the absolute value of depth from the bottom level to the origin in the direction of gravity.

2. Apply the correct sign based on whether the bottom level is above or below the origin.

If the channel's bottom lies above the origin, then the **Bottom Level** is positive (see Figure 26.27: Determining the Free Surface Level and the Bottom Level (p. 2984)). Likewise, if the channel's bottom lies below the origin, then the **Bottom Level** is negative.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.27: Determining the Free Surface Level and the Bottom Level



You can also specify a transient profiles for a **Bottom Level** for the relevant open channel boundaries.

26.3.5.7. Specifying the Total Height

The total height, along with the velocity, is used as an option for describing the flow. The total height is given as

$$y_{tot} = y_{local} + \frac{V^2}{2g} \tag{26.12}$$

where V is the velocity magnitude and g is the gravity magnitude.

26.3.5.8. Determining the Velocity Magnitude

For pressure inlet boundaries, input for **Velocity Magnitude** is required to calculate the dynamic pressure being used in the total pressure calculation.

Note:

The provided **Velocity Magnitude** is not applied to the boundary.

26.3.5.9. Determining the Secondary Phase for the Inlet

In the case of a flow with more than two phases, for pressure inlets and mass-flow inlets, you can select the desired secondary phase from the **Secondary Phase for Inlet** drop-down box.

Important:

Note that only one secondary phase is allowed to pass through one inlet group.

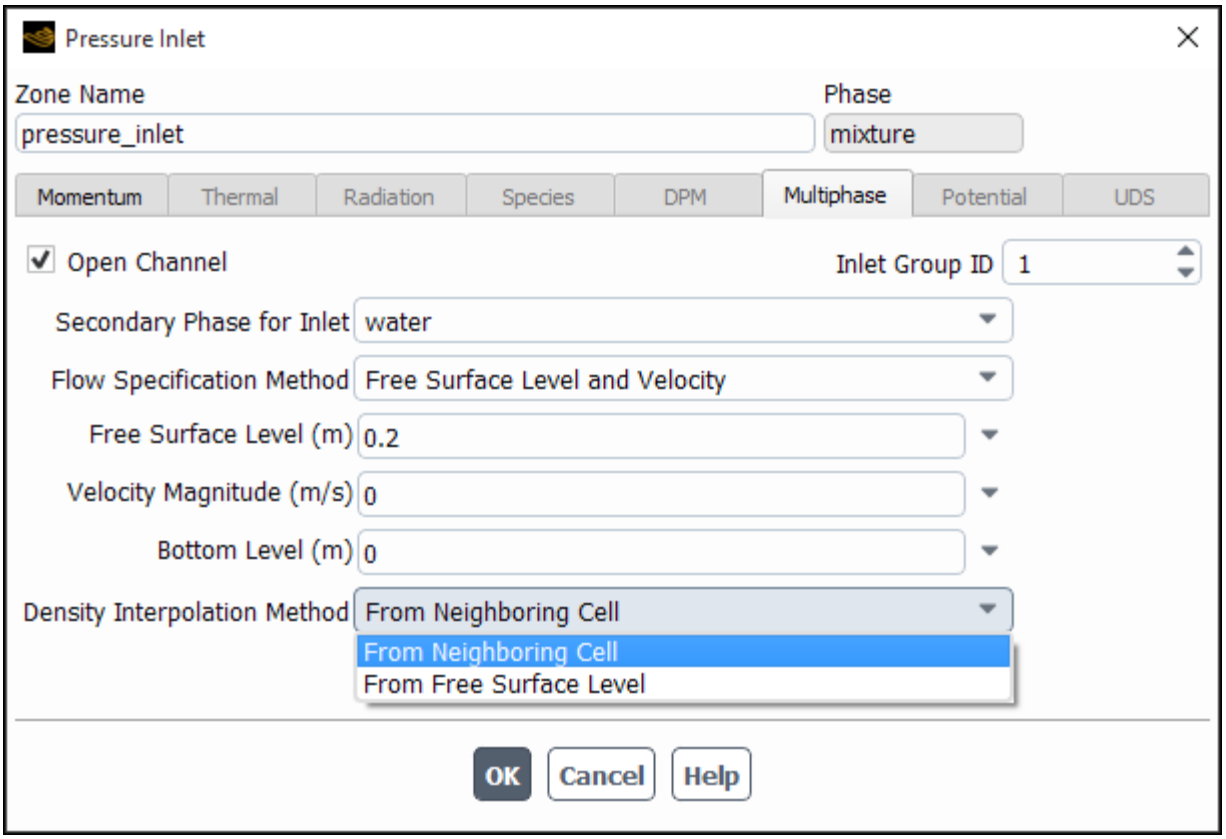
Consider a problem involving a three-phase flow consisting of air as the primary phase, and oil and water as the secondary phases. Consider also that there are two inlet groups:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- water and air
- oil and air

For the former inlet group, you would choose water as the secondary phase. For the latter inlet group, you would choose oil as the secondary phase (as shown in Figure 26.28: Pressure Inlet for Open Channel Flow (p. 2985)).

Figure 26.28: Pressure Inlet for Open Channel Flow



26.3.5.10. Determining the Secondary Phase for the Outlet

For a flow application with more than two phases, if you select **Free Surface Level** for the **Pressure Specification Method** (in the **Pressure Outlet** dialog box, **Multiphase** tab), you must also select the appropriate **Secondary Phase for Level Specification** and specify the **Free Surface Level** and **Bottom Level**. See Determining the Free Surface Level (p. 2982) and Determining the Bottom Level (p. 2983) for details about these parameters. For more information about the available pressure specification methods, refer to Pressure Outlet in the *Fluent Theory Guide*.

The hydrostatic pressure profile is imposed in the selected secondary phase that shares the interface with the primary phase. It is assumed that other secondary phases will not disturb the imposed pressure distribution.

For example, in a three-phase cavitating flow consisting of air, water, and vapor, the vapor is generated in the localized region adjacent to moving bodies. If the pressure outlet boundary is sufficiently far away from the cavitating region, then the hydrostatic assumption in the water phase that shares the interface with air can be safely imposed.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.3.5.11. Choosing the Pressure Specification Method

For a pressure outlet boundary, the outlet pressure can be specified in one of three ways:

- by prescribing the free surface level, such as a hydrostatic pressure profile (available for two-phase flow only)
- by specifying the constant pressure
- by specifying the neighboring cell pressure

Note:

You can also specify a hydrostatic pressure profile at the pressure outlet for axisymmetric open channel flow. However, there are certain limitations as noted in Limitations (p. 2988).

Important:

This option is not available in the case of three-phase flows since the pressure on the boundary is taken from the neighboring cell.

26.3.5.12. Choosing the Density Interpolation Method

For problems involving sub-critical flow, the following options exist for the **Density Interpolation Method**:

- **From Neighboring Cell** is the default option, where the mixture density used in the hydrostatic profile is interpolated using the volume fraction calculated from the neighboring cell.

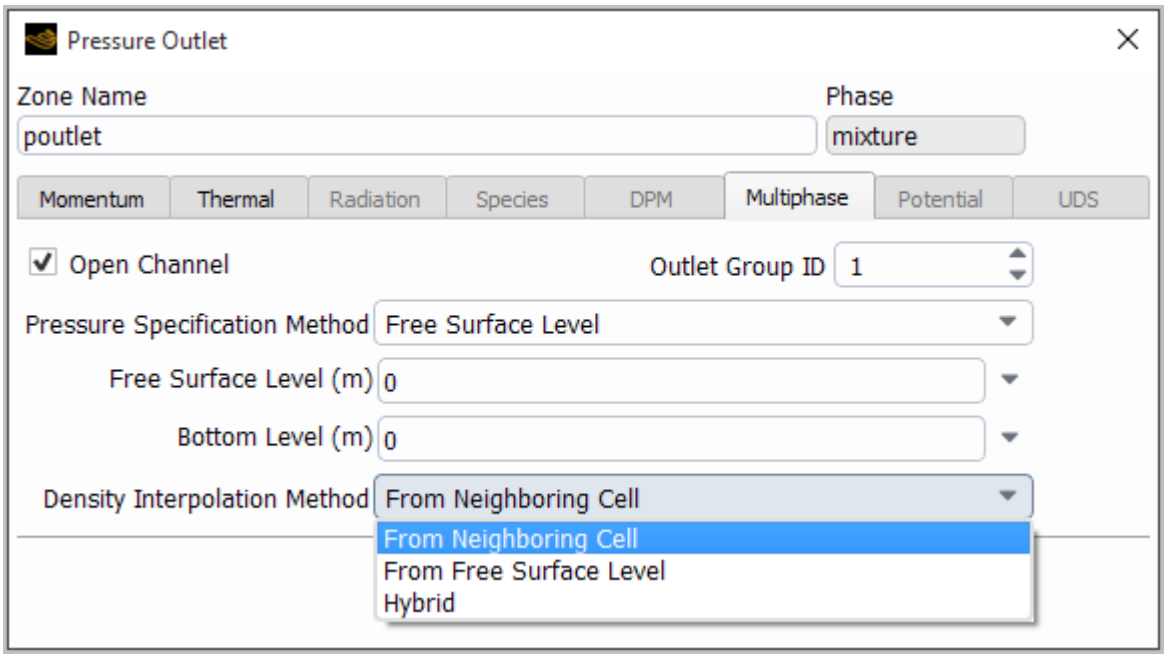
  Note:

  In some cases, this method may cause free surface oscillation near the upstream boundaries. These oscillations are particularly visible in cases where boundaries are close to the object; or if the boundary has unstructured mesh near the free surface level. For such cases, it is recommended that you use the **From Free Surface Level** option under **Density Interpolation Method**.

- **From Free Surface Level** where the mixture density used in the hydrostatic profile is interpolated from the volume fraction calculated from the free surface level.
- **Hybrid** where the mixture density used in the hydrostatic profile is calculated either from the **From Neighboring Cell** or **From Free Surface Level** options, depending on the direction of flux. This method is only available at a pressure outlet.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.29: Density Interpolation Method for Open Channel Flow



26.3.5.13. Open Channel Flow Compatibility with Velocity Inlet

For additional information, see the following sections:

26.3.5.13.1. Velocity Inlet, Open Channel Flow, Steady-State

26.3.5.13.2. Velocity Inlet, Open Channel Flow, Transient

Open channel flow compatibility with velocity inlet is provided in the framework of the open channel wave boundary condition, therefore, the **Open Channel Wave BC** option in the Figure 26.1: Multiphase Model Dialog Box for the VOF Model (p. 2919) must be enabled first.

Velocity inlet compatibility allows you to use either averaged or segregated velocity inputs to moving obstacles, water and air, which is not possible using either the pressure inlet or mass-flow inlet boundary conditions.

Select the **Open Channel Wave BC** option in the **Velocity Inlet** dialog box. For detailed information about the velocity inputs, see Inputs at Velocity Inlet Boundaries (p. 1411).

26.3.5.13.1. Velocity Inlet, Open Channel Flow, Steady-State

You must specify (in the **Multiphase** tab of the **Velocity Inlet** dialog box):

- Secondary phase for the inlet
- Free surface level

26.3.5.13.2. Velocity Inlet, Open Channel Flow, Transient

You must set the **Wave BC Option** to **None** (in the **Multiphase** tab of the **Velocity Inlet** dialog box).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.3.5.14. Limitations

The following list summarizes some issues and limitations associated with the open channel boundary condition.

- The conservation of the Bernoulli integral does not provide the conservation of mass flow rate for the pressure boundary. In the case of a coarser mesh, there can be a significant difference in mass flow rate from the actual mass flow rate. For finer meshes, the mass flow rate comes closer to the actual value. So, for problems having constant mass flow rate, the mass flow rate boundary condition is a better option. The pressure boundary should be selected when steady and non-oscillating drag is the main objective.
- Specifying the top boundary as the pressure outlet can sometimes lead to a divergent solution. This may be due to the corner singularity at the pressure boundary in the air region or due to the inability to specify local flow direction correctly if the air enters through the top locally.
- Only the heavier phase should be selected as the secondary phase.
- In the case of flows with more than two phases, only one secondary phase is allowed to enter through one inlet group (that is, the mixed inflow of different secondary phases is not allowed).
- For axisymmetric flow, pressure and mass-flow inlets do not support open channel boundary conditions.
- Open channel wave boundary conditions for velocity inlets do not support cases with axisymmetric flow.
- Axisymmetric open channel flow is available for two phase flow and the pressure outlet boundary.
- For axisymmetric open channel flow, the direction of gravity must be aligned with the direction of the axis.

26.3.5.15. Recommendations for Setting Up an Open Channel Flow Problem

The following list represents a list of recommendations for solving problems using the open channel flow boundary condition:

- In the cases where the inlet group has a different inlet for each phase of fluid, then the parameter values (such as **Free Surface Level**, **Bottom Level**, and **Mass Flow Rate**) for each inlet should correspond to all other inlets that belong to the inlet group.
- The solution begins with an estimated pressure profile at the outlet boundary.

  In general, you can start the solution by assuming that the level of liquid at the outlet corresponds to the level of liquid at the inlet. The convergence and solution time is very dependent on the initial conditions. When the flow is completely subcritical (upstream and downstream), in marine applications for instance, the above approach is recommended.

  If the final conditions of the flow can be predicted by other means, the solution time can be significantly reduced by using the proper boundary condition.
- In the case of super-critical flow at the outlet, you can use the **From Neighboring Cell** option at the outlet, as it may provide better convergence.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The initialization procedure is very critical in the open channel analysis. Refer to Recommendations for Open Channel Initialization (p. 3001).
- For the initial stability of the solution, a smaller time step size is recommended. You can increase the time step size once the solution becomes more stable.
- For flows that do not make a transition from sub-critical to super-critical, or vice-versa, you can speed-up the solution calculation by specifying the frequency of Froude number updates during run time by setting the following text command: `solve` → `set` → `open-channel-controls`.

  When prompted, set up the following parameter for steady-state flows:

  ```
  /solve/set> open-channel-controls
  Iteration interval for Froude number update [10]
  ```

  When prompted, set up the following parameter for transient flows:

  ```
  /solve/set> open-channel-controls
  Time step interval for Froude number update [1]
  ```

- For pure open channel flow applications, the inlet and outlet boundary conditions are controlled by the Froude number. In certain cases, it is possible to impose a hydrostatic profile at the outlet without any Froude number dependency. You can use the following text command: `solve` → `set` → `open-channel-controls`.

  When prompted, set up the following parameter as shown below:

  ```
  /solve/set> open-channel-controls
  Use Froude number independent boundary condition at outlet? [no] yes
  ```

- Relevant open channel inputs, along with the Froude number, can be reported using the following text command: `define` → `boundary-conditions` → `openchannel-threads`.
- In the case of reverse flow, the pressure outlet boundary behaves as a pressure inlet, and the boundary-specific static pressure is taken as the **Total Pressure**. In this case, the static pressure is calculated from the total pressure. You can use the following text command to fix the boundary-specified static pressure, which helps to suppress reflections from the pressure boundary for certain cases: `solve` → `set` → `open-channel-controls`

  When prompted, set up the following parameter as shown below:

  ```
  /solve/set> open-channel-controls
  Use boundary specified static pressure for backflow? [no] yes
  ```

Open Channel + Translational Moving Reference Frame (MRF)

For high-speed moving hulls, it is generally recommended to ramp up the hull speed from zero to its actual speed for better stability. However, it has been observed that using pressure inlet or velocity inlet boundary conditions to mimic the hull speed causes many convergence issues. The alternative approach is to use a translational MRF with the following solution methodology:

- Set the same speed for the whole domain as for the hull by using a transient profile for a gradual ramping. It is assumed that there is no bulk velocity of fluid.
- Keep the wall boundary conditions stationary in the absolute reference frame.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Specify the inlet boundary condition using one of the following options:
  - Pressure-inlet with the following settings:
    → Select **Relative to Adjacent Cell Zone** for **Reference Frame** (**Momentum** tab of the **Pressure Inlet** dialog box).
    → Use the **udf/profile** option for **Velocity Magnitude** and hook the same transient profile of the hull speed (**Multiphase** tab of the **Pressure Inlet** dialog box.
  - Velocity Inlet with the following settings:
    → Select **Absolute** for **Reference Frame** (**Momentum** tab of the **Velocity Inlet** dialog box).
    → Set **Velocity Magnitude** to 0 m/s
- In the **Solution Initialization** task page, set all velocity components to zero, and initialize the solution in the **Absolute** reference frame.

26.3.6. Modeling Open Channel Wave Boundary Conditions

When modeling open channel wave boundary conditions, many of the variables that are used in open channel flow, also exist for open channel wave boundary conditions. You may have to refer to Modeling Open Channel Flows (p. 2979) for information about some of the settings.

To use the open channel wave boundary condition, perform the following:

1. Enable **Gravity** and set the gravitational acceleration fields.

   Setup → **General** **Gravity** → **On**

2. Enable the **Volume of Fluid** model in the **Multiphase Model** dialog box (**Models** tab).

   Setup → **Models** → **Multiphase** → **Edit...**

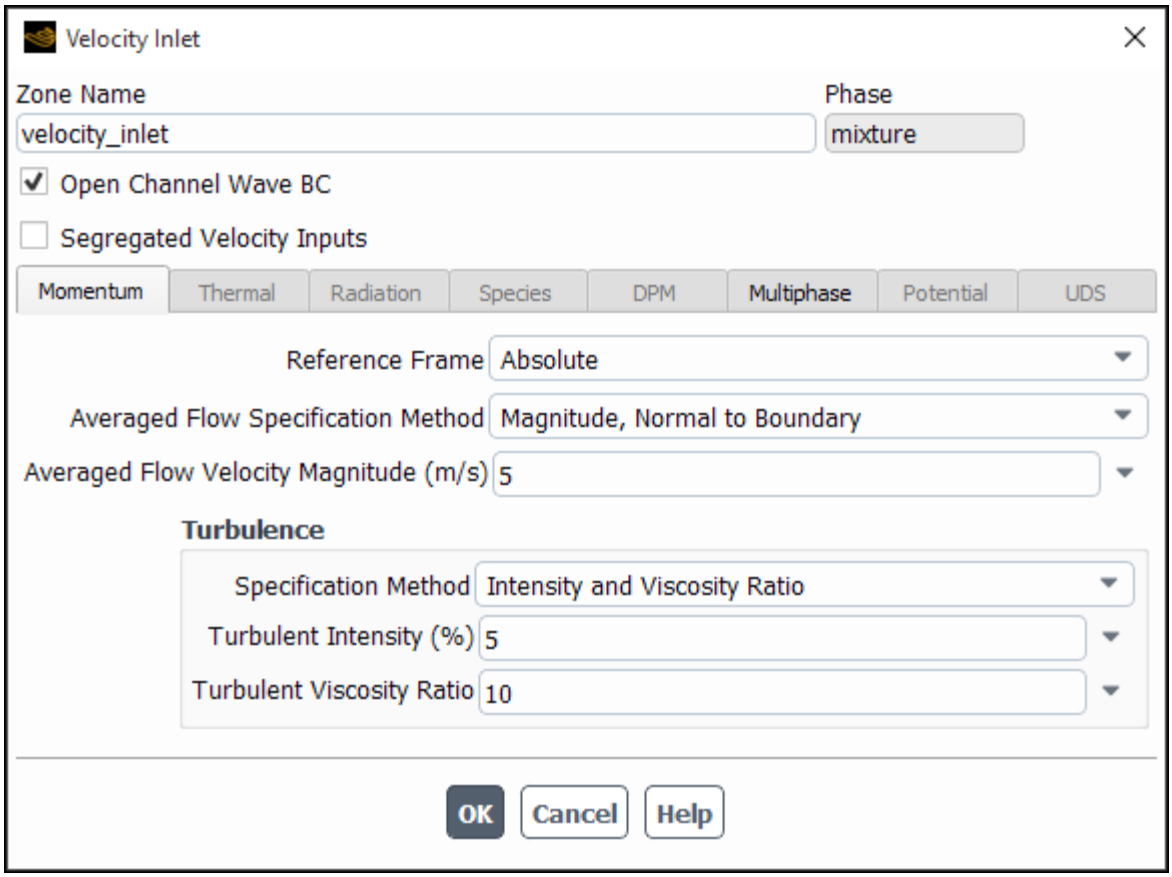
3. In the **Formulation** group box, select either **Implicit** or **Explicit**.

4. In the **VOF Sub-Models** group box, select **Open Channel Wave BC**.

In order to set specific parameters for a particular boundary for open channel wave boundaries, enable the **Open Channel Wave BC** option in the **Velocity Inlet** boundary condition dialog box (Figure 26.30: The Velocity Inlet for Open Channel Wave BC (p. 2991)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.30: The Velocity Inlet for Open Channel Wave BC



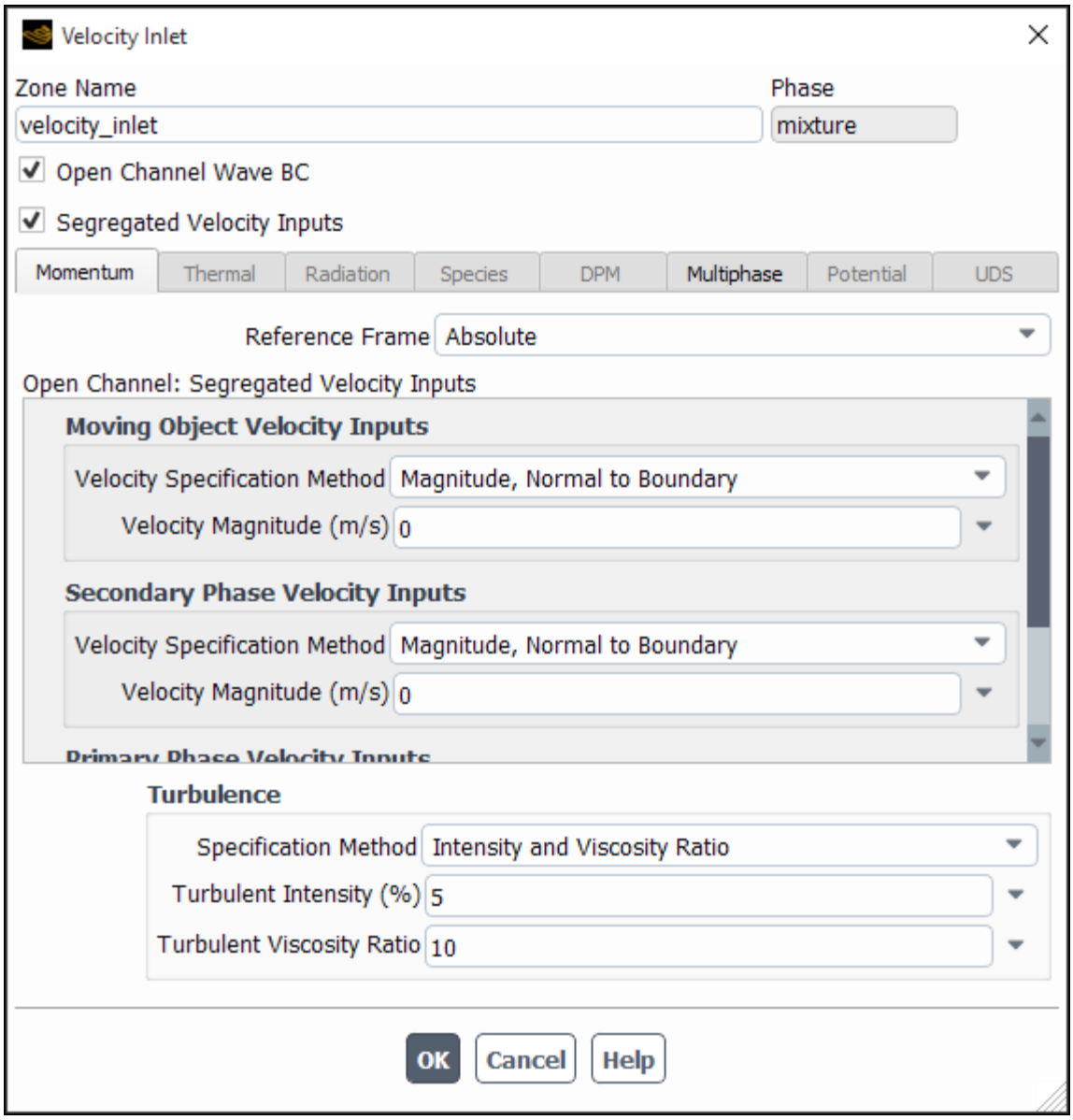
The default method is the **Averaged Flow Specification Method**, where inputs for the moving obstacle and water current are combined. There is no specific treatment for air velocity, which is assumed to be same as the averaged velocity.

Segregated Velocity Inputs

The **Segregated Velocity Inputs** option allows you to prescribe velocity inputs individually for the moving obstacle, Secondary phase (typically water), and Primary phase (typically air).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.31: Segregated Velocity Inputs for Open Channel Wave BC



You can set the moving object and secondary phase velocity specification methods as:

- Magnitude and direction vector
- Magnitude and normal to boundary

You can set the primary phase velocity specification methods as:

- Magnitude and direction vector
- Magnitude and normal to boundary
- Power law and direction vector
- Power law and normal to boundary

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Power Law

$$\frac{V}{V_{ref}} = \left(\frac{z}{z_{ref}}\right)^{\alpha} \tag{26.13}$$

Where z_{ref} is the vertical distance from the instantaneous free surface level at which the reference velocity magnitude, V_{ref} is specified. z is the vertical distance from the free surface level at which, velocity V must be estimated. α is the power law coefficient, which is typically 0.16 for an open sea.

Setting Buffer Layer Height

The buffer layer is assumed to be the layer above the free surface, in which the effect of air is not felt. This option is provided by assuming that the air in the vicinity of free surface blows with the speed of the water and the waves, so that direct effect of air is not felt in the wave propagation.

There are two options for providing the buffer height:

1. **Automatic:** the buffer layer is taken as the specified fraction of the maximum wave height.
2. **User-defined:** you provide the input for the buffer layer height.

The buffer layer can also be defined using the following text command:

```
/solve/set> open-channel-wave-options

/solve/set/open-channel-wave-options>
set-buffer-layer-ht     set-verbosity           stokes-wave-variants

/solve/set/open-channel-wave-options> set-buffer-layer-ht
buffer layer specification method (0 - Automatic, 1 - User-defined)  [0]
fraction of buffer layer height to maximum wave height [0.02]

/solve/set/open-channel-wave-options> set-buffer-layer-ht
buffer layer specification method (0 - Automatic, 1 - User-defined)  [0] 1
enter the value of buffer layer height [0]
```

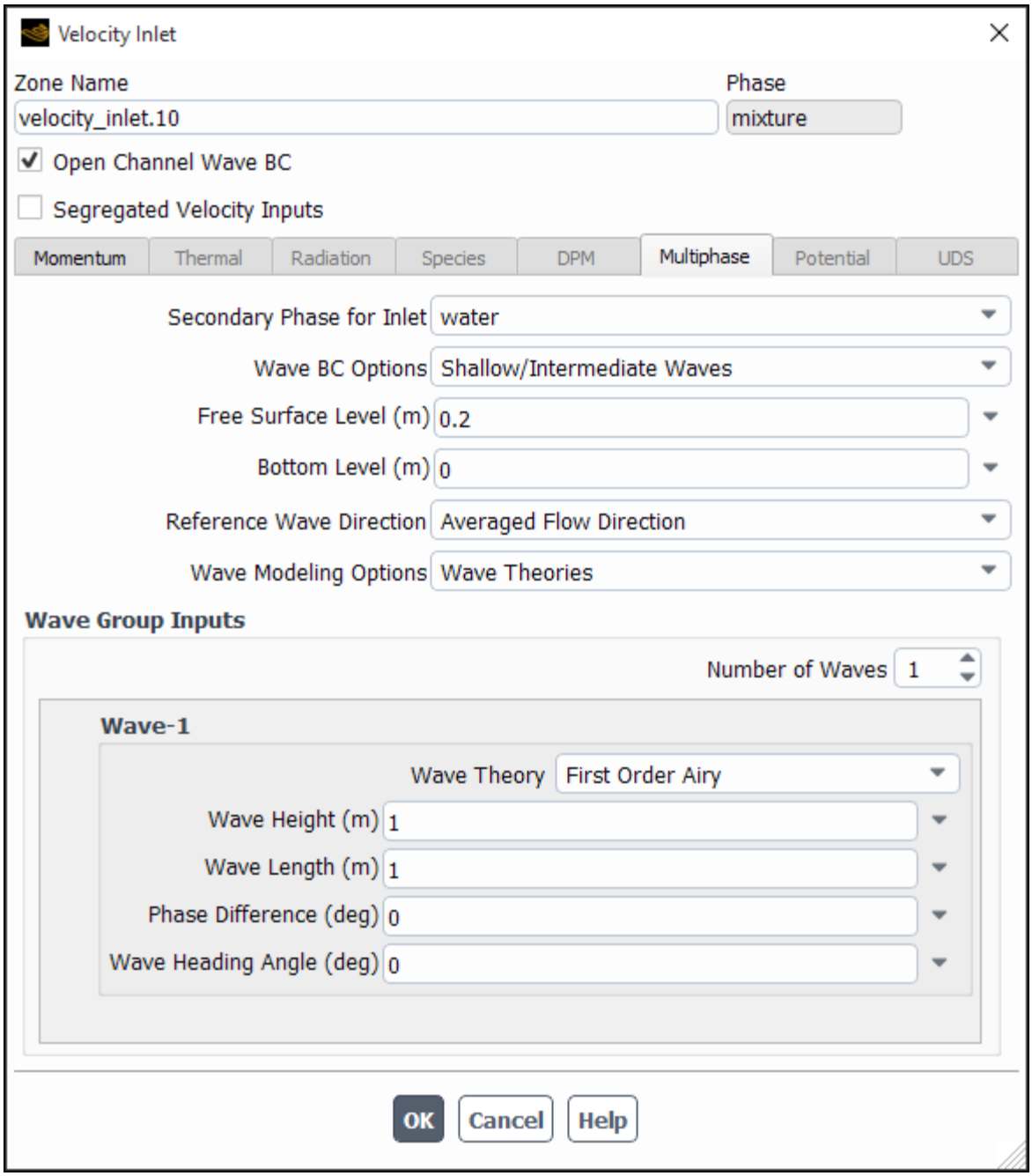
In the **Momentum** tab of the **Velocity Inlet** dialog box, you can enter the averaged flow velocity, which includes components from the flow current and the moving object. You can specify the **Averaged Flow Specification Method** as:

- **Magnitude and Direction**
- **Magnitude and Normal to Boundary**
- **Components**

In the **Multiphase** tab (Figure 26.32: The Velocity Inlet for Open Channel Wave BC (Explicit Formulation) (p. 2994)), you will specify the following:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.32: The Velocity Inlet for Open Channel Wave BC (Explicit Formulation)



- **Secondary Phase for Inlet** specifies the phase to which the wave parameters are applied. In case of a three-phase flow, select the corresponding secondary phase from this list.
- **Wave BC Options** of which you have a choice of **Shallow/Intermediate Waves**, **Short Gravity Waves**, **Shallow Waves**, and **None** (for calm sea or pure open channel flow problems without waves). Information about these waves is available in Open Channel Wave Boundary Conditions in the Theory Guide. Note that the short gravity waves expression is derived under the assumption of infinite liquid height.
- **Free Surface Level** is the same definition as for open channel flow, see Modeling Open Channel Flows (p. 2979).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Bottom Level** is the same definition as for open channel flow, see Modeling Open Channel Flows (p. 2979), and is valid only for shallow or intermediate depth waves. The bottom level is used for calculating the liquid height.
- **Reference Wave Direction** is the direction of wave propagation with zero wave heading angle. You can specify the wave propagation direction as:
  - **Averaged Flow Direction**: In this case, the reference direction is the same as the averaged flow direction.
  - **Direction Vector**
  - **Normal to Boundary**
- **Wave Modeling Options** determines how the waves are modeled in the shallow/intermediate and short gravity wave regimes. You can choose to model the waves using either **Wave Theories** or **Wave Spectrum**. **Wave Theories** is usually selected for simulating regular waves. To simulate random waves, select **Wave Spectrum**.
- Depending on your choice of **Wave BC Options** and **Wave Modeling Options**, you will need to specify the wave parameters as described in the following sections.

Shallow Wave Inputs (p. 2995)
Wave Group Inputs (p. 2996)
Wave Spectrum Inputs (p. 2997)

Shallow Wave Inputs

If you have selected **Shallow Waves** for **Wave BC Options** you can specify the following settings for each wave:

Number of Waves

is the option to set the number of interacting waves (default = 1)

Ideally, shallow waves do not support the principle of superposition. The **Number of Waves** option enables interaction of two different waves originating at unique (non-interfering) locations within the domain. Collision of two counter-propagating solitary waves inside a domain is an example of a shallow wave interaction.

Wave Theory

of which you have a choice of **Fifth Order Solitary** (the default) and **Fifth Order Cnoidal**. Information about the types of wave theory is available in Open Channel Wave Boundary Conditions in the Theory Guide.

Wave Height

is the height difference between a wave crest to the neighboring trough.

Since a solitary wave does not have troughs, the wave height is the distance between a wave crest to mean free surface level.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Wave Length

is the distance between two consecutive zero crossings.

Since solitary waves are derived based on the assumption of infinite wave length, specified wave length is only used to estimate the elliptic function parameter for suitability of the wave theory, and is not used in calculating wave parameters.

Inlet Offset Distance

is the translational distance from the reference point origin in the reference wave propagation direction. This option is used to generate a wave from a location other than the reference frame origin. For a solitary wave, the hump is always generated at the location where:

$$x - x_0 = 0 \tag{26.14}$$

where x and x_0 are spatial coordinates in the reference wave propagation direction.

Wave Heading Angle

is the angle between the direction of the wavefront and the reference wave propagation direction, in the plane of the flow surface. In 2D, there are only two possibilities, zero degree when the wave is in the reference propagation direction, and 180 degree when the wave Is in the direction opposite to the reference propagation direction.

Wave Group Inputs

If you have selected **Wave Theories** for **Wave Modeling Options** you can specify the following settings for each wave:

Number of Waves

is the option to set the number of superposed waves (default = 1)

Wave Theory

of which you have a choice of **First Order Airy** (the default), **Second Order Stokes**, **Third Order Stokes**, **Fourth Order Stokes**, and **Fifth Order Stokes**. Information about the types of wave theory is available in Open Channel Wave Boundary Conditions in the Theory Guide.

Wave Height

is the height difference between a wave crest to the neighboring trough.

Wave Length

is the distance between two consecutive crests, troughs or zero crossings.

Phase Difference

is the phase angle by which one periodic disturbance or wavefront lags behind or precedes another in time or space.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Wave Heading Angle

is the angle between the direction of the wavefront and the reference wave propagation direction, in the plane of the flow surface. In 2D, there are only two possibilities, zero degree when the wave is in the reference propagation direction, and 180 degree when the wave Is in the direction opposite to the reference propagation direction.

Wave Spectrum Inputs

If you have selected **Wave Spectrum** for **Wave Modeling Options** you can specify the following settings for each wave:

Frequency Spectrum Method

specifies the spectrum to use. You can select **Pierson-Moskowitz** (appropriate for fully-developed seas), **Jonswap** (appropriate for fetch-limited seas), or **TMA** (appropriate for fetch-limited, finite-depth seas).

Peak Shape Parameter

controls the shape and amplitude of the frequency peak in the **Jonswap** and **TMA** formulations. This corresponds to γ in Equation 14.101 in the *Fluent Theory Guide*.

Significant Wave Height

is the mean wave height of the largest 1/3 of waves. This corresponds to H_s in Equation 14.100 in the *Fluent Theory Guide*.

Peak Wave Frequency

is the wave frequency corresponding to the highest wave energy. This corresponds to ω_p in Equation 14.100 and Equation 14.101 in the *Fluent Theory Guide*. ω_p can be expressed in terms of the peak wave period, T_p, as $\omega_p = {}^{2\pi}/_{T_p}$.

Minimum/Maximum Wave Frequency

specify the frequency range for the spectrum. Note that the **Peak Wave Frequency** must fall in between **Minimum Wave Frequency** and **Maximum Wave Frequency**.

Number of Frequency Components

specifies the number of components into which the frequency spectrum is divided.

Direction Spreading Method

specifies the method to use for specifying the directional spreading characteristics. You can choose **Unidirectional** (for long-crested waves), **Frequency Independent Cosine Function** (for short-crested waves where the directional component does not depend on frequency), and **Frequency Dependant Hyperbolic Function** (for short-crested waves where the directional component depends on frequency).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Frequency Independent Cosine Exponent

specifies the exponent in the Frequency Independent Cosine Function formulation. This corresponds to s in Equation 14.105 in the *Fluent Theory Guide*.

Mean Wave Heading Angle

specifies the principal wave heading direction.

Angular Spread

specifies the deviation from the mean wave direction for calculating the angular range.

Number of Angular Components

specifies the number of components into which the angular range is divided.

The total number of wave components is the product of **Number of Frequency Components** and **Number of Angular Components**. For modeling random waves, phase differences for the individual wave components are randomly distributed between 0 and 2π.

Note:

Any change in the input for **Number of Wave Components** will change the values of phase difference for each component. Therefore, it is recommended that you start the simulation from a saved case file after setting all the wave spectrum user inputs and that you not change the number of wave components during the simulation.

26.3.6.1. Summary Report and Regime Check

A useful text command used to print out a summary of the open channel wave boundary condition settings is `define/boundary-conditions/open-channel-wave-settings`. The output will depend on whether you have chosen **Wave Theories** or **Wave Spectrum**.

Sample Output for Wave Theories

```
/define/boundary-conditions> open-channel-wave-settings

Wave Input Analysis for Velocity Inlet : Thread ID = 5
*********************************************************

Wave-1 Analysis
***************************************

Current Settings :
------------------
Wave theory : 5th-order-Stokes , Wave regime = Shallow/Intermediate
Wave Height (H) = 0.2160, Wave Length (L) = 1.9400
Liquid Depth (h) = 0.6000, Ursell Number (H*L*L/(h*h*h)) = 3.7636

Mandatory checks for full wave regime within wave breaking limit
----------------------------------------------------------------
Relative Height: H/h = 0.3600 , Maximum theoretical limit = 0.7800
Maximum numerical limit = 0.5500
Relative height within wave breaking limit

Wave Steepness: H/L = 0.1113 , Maximum theoretical limit = 0.1420
Stable numerical limit = 0.1000 , Maximum numerical limit = 0.1200
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
Warning: Wave steepness exceeding the stable numerical limit.
Waves could be stable or unstable in this regime.

Checks for selected wave theory within wave breaking and stability limit
------------------------------------------------------------------------------
Relative height check
H/h = 0.3600 , Min : 0.0000 , Max : 0.5000
Relative height check : successful

Wave Steepness check
H/L = 0.1113 , Min : 0.0000 , Max : 0.1363
Wave steepness check : successful

Ursell Number check
Ur = 3.7636 , Min : 0.0000 , Max : 25.0000
Ursell number check : successful

Wave regime check
h/L = 0.3093 , Min : 0.0600 , Max : 10000.0000
Wave regime check : successful

Summary
----------------------
Checks : passed
Selected wave theory is appropriate for application.
```

Sample Output for Wave Spectrum

```
Wave Input Analysis for Velocity Inlet : Thread ID = 6
*********************************************************

Wave Spectrum Analysis
**************************************

Current Settings :
------------------
Frequency Spectrum : Jonswap
Direction Spreading Method : Unidirectional

Significant Wave Height (Hs) = 5.0000
Peak Wave Frequency (wp) = 1.0000, Peak Time Period (Tp) = 6.2832
Minimum Frequency (wi) = 0.6600, Maximum Frequency (we) = 1.6600

Wave Lengths at Min/Peak/Max frequencies (Li, Lp, Le) = (141.5015, 61.6380, 22.3683)

Recommendation: Set the min and max wave frequencies so that most of the wave energy
  is concentrated in the selected regime of spectrum.
  - Min frequency = 0.5*Peak frequency
  - Max frequency = 2.5*Peak frequency


Wave Regime Check
----------------------
Wave regime type = Short Gravity (Deep)
Information: Validity of short gravity assumption : Liquid Depth > 0.5*Max Wave Length

Sea Steepness Check
----------------------
Peak Time Period (Tp) = 6.2832
Zero Upcrossing Time Period (Tz) = 4.7869

Sea Steepness based on Tp (Sp) = 0.0811, Steepness Limit = 0.0667
Sea Steepness based on Tz (Sz) = 0.1398, Steepness Limit = 0.1000

Warning: Sea Steepness based on peak time period exceeding limit.
Warning: Sea Steepness based on zero upcrossing time period exceeding limit.

Information: High wave steepness of any individual component too could affect the wave pattern,
  as superposition of wave components is based upon lienar wave theory.
```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
Frequency Spectrum Check
-------------------------
Peak Time Period to Wave Ht Sqrt Ratio (r = Tp/sqrt(Hs)) = 2.8099

Message: Selection of Jonswap spectrum is appropriate.

Recommended value of Peak Shape Parameter for r <= 3.6 :  5.0000

Spectrum Resolution check
-------------------------
Number of frequency components = 50
Number of angular components = 1
Total number of wave components = 50

Information: Higher number of wave components would have adverse effect on speed-up
  whereas smaller could affect the accuracy by not resolving the spectrum well.
  Following steps could assist to optimize the number of components
  1. Set number of frequency/angular components in boundary condition panel,
  2. Set TUI command solve> set> open-channel-wave-verbosity to 2 before initialization,
  3. Initialize (It would print Spectrum Energy for each frequency/angle.)
  4. Copy the information to plot Sw vs Omega and S_theta vs theta
  5. Optimize the number of components after repeating steps 1-4
```

26.3.6.2. Transient Profile Support for Wave Inputs

Ansys Fluent supports transient profiles for all wave inputs using user-defined functions as seen by the example below:

```
/************************************************************************
  UDF for defining transient profile wave inputs
 ************************************************************************/

#include "udf..h"

#define H 0.02 /* wave height for both waves : same */
#define LEN1 3. /* wave length for first wave */
#define LEN2 6. /* wave length for second wave*/
#define G 9.81
#define D 10. /* liquid depth */
#define U 1. /* wave current */
#define X 0. /* inlet point */

DEFINE_TRANSIENT_PROFILE(wave_ht, current_time)
{
 real k1 = 2.*M_PI/LEN1;
 real k2 = 2.*M_PI/LEN2;
 real w1 = sqrt(G*k1*tanh(k1*D)) + k1*U;
 real w2 = sqrt(G*k2*tanh(k2*D)) + k2*U;
 real dk = 0.5*(k1 - k2);
 real dw = 0.5*(w1 - w2);
 return (2.*H*cos(dk*X - dw*current_time));
}
```

26.3.6.3. Alternative Stokes Wave Theory Variant

Fluent uses Fenton's formulation for Stokes theories by default. For additional information about the current Stokes theories formulation, see Stokes Wave Theories in the *Fluent Theory Guide*. To revert to the old formulation, provided by Skjebreia and Hendrickson, use the following text command:

```
/solve/set/open-channel-wave-options>
set-buffer-layer-ht     set-verbosity           stokes-wave-variants
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
/solve/set/open-channel-wave-options> stokes-wave-variants
Use Fenton's formulation for Stokes wave theories [yes] no
Activating old formulation (Skjelbreia and Hendrickson) for Stokes wave theories.
```

For additional information on the old formulation, refer to Lin's book [89] (p. 5718).

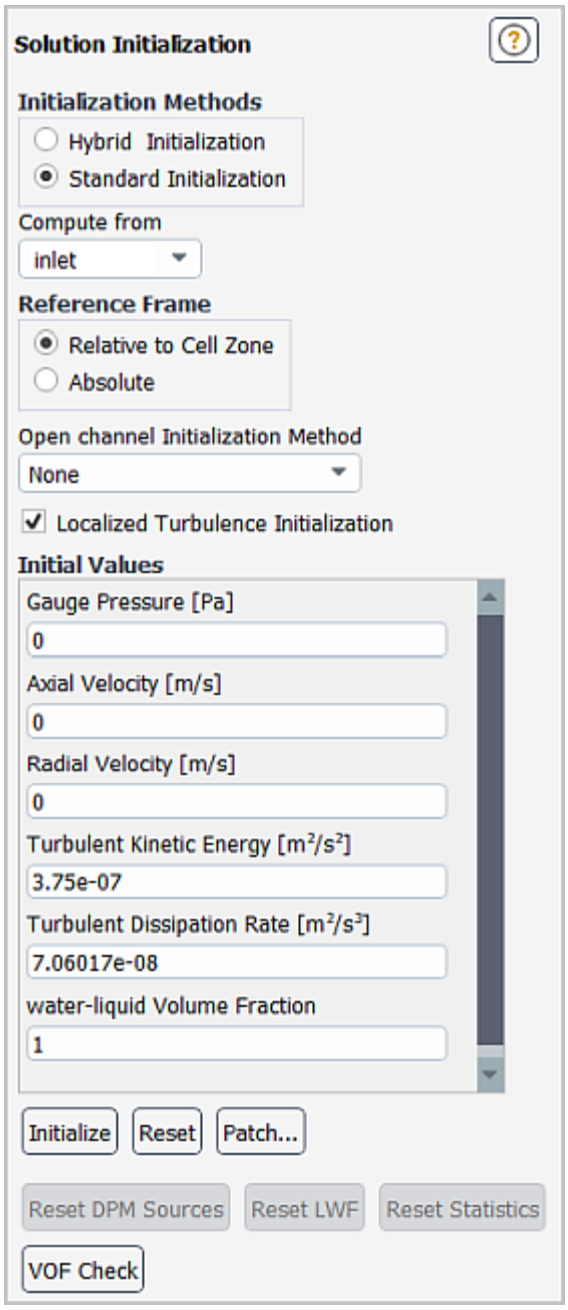
26.3.7. Recommendations for Open Channel Initialization

Once you have selected either the **Open Channel Flow** or the **Open Channel Wave BC** option in the **Multiphase Model** dialog box (**Models** tab), then the **Open Channel Initialization Method** drop-down list appears in the **Solution Initialization** task page.

Solution → **Initialization**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.33: The Solution Initialization Task Page



Select an inlet zone from the **Compute from** drop-down list. You can now make your selection from the **Open Channel Initialization Method** drop-down list. If only the **Open Channel Flow** option was enabled, then you only have a choice of **None** or **Flat**. If you enabled **Open Channel Wave BC**, then your choices are **None**, **Flat**, or **Wavy**. The default initialization method is **None**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you initialize the solution using **None**, it has no effect as it does not use any open channel information from the selected zone. The **Open Channel Initialization Method** comes into effect when you select either **Flat** or **Wavy**.

Important:

This initialization is only valid for pressure-inlets, pressure outlets, and mass-flow inlets for open channel flow and velocity inlets for open channel wave boundary conditions. If the selected inlet zone does not have either open channel flow or open channel wave boundary conditions, Ansys Fluent will report an error message after you initialize the flow with open channel initialization method of **Flat** or **Wavy**.

For open channel initialization from the pressure outlet boundary, the hydrostatic pressure profile based on the **Free Surface Level** is patched in the domain. The volume fraction in the domain is patched based on **Free Surface Level** provided at the pressure outlet boundary. To patch velocity and other variables, the values in the **Solution Initialization** task page will be used.

Important:

The pressure specification methods, with the exception of **Free Surface Level**, are not supported for open channel initialization from the pressure outlet boundary.

Initialization will result in the volume fraction, X, Y, and Z velocities, and pressure being patched in the domain. The volume fraction will be patched in the domain based on the free surface level of the selected zone from the **Compute from** list. The velocities in the domain will be patched assuming the constant value provided for the velocity magnitude in the selected zone.

Important:

If you specify a profile for the velocity magnitude or direction vectors, the initialization will select the value for the velocity magnitude and direction vectors from only one face. Therefore the initialization may be inaccurate. However, generally, open channel inputs for velocity magnitude and direction vectors are constant.

The pressure that is patched is the hydrostatic pressure based on the free surface level specified in the selected zone.

You can use the following text command for open channel automatic initialization:

`solve` → `initialize` → `open-channel-auto-init`

When prompted, set up the following parameters:

boundary thread id

Enter the thread id for the boundary to be selected for open channel automatic initialization.

flat free surface initialization

This option is available for both open channel flow and open channel wave boundary conditions.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

wavy free surface initialization

This option appears only for open channel wave boundary conditions, when flat free surface initialization is not selected.

The steps to be followed for open channel automatic initialization are

1. Compute defaults based on valid open channel boundary thread. This step is required for better initialization of turbulence parameters based on uniform velocity magnitude.

   `solve` → `initialize` → `compute defaults`

2. This would provide information about the selected boundary and type of initialization.

   `solve` → `initialize` → `open-channel-auto-init`

3. Initialize

   `solve` → `initialize` → `initialize`

26.3.7.1. Reporting Parameters for Open Channel Wave BC Option

To report values as wave speed, wave frequency, and time period for individual waves during initialization, you can set the following text command:

`solve` → `set` → `open-channel-wave-options` → `set-verbosity`

When prompted to set `Verbosity for reporting of derived wave inputs during initialization`, enter `1` as shown below:

Note:

You can enter values of 0, 1, or 2. Entering a higher number provides more information.

```
/solve/set/open-channel-wave-options> set-verbosity

Verbosity for reporting of derived wave inputs during initialization
 [0] 1
```

A sample of the resulting output for wave groups after initializing is shown below:

```
Wave-1 : Wave Height = 0.0200, Wave Length = 3.0000
Wave Number = 2.0944, Wave Speed = 2.1642, Wave Frequency = 4.5328
Effective parameters: Wave Speed = 3.1642, Wave Frequency = 6.6272, Time Period = 0.9481

Wave-2 : Wave Height = 0.0200, Wave Length = 6.0000
Wave Number = 1.0472, Wave Speed = 3.0607, Wave Frequency = 3.2052
Effective parameters: Wave Speed = 4.0607, Wave Frequency = 4.2524, Time Period = 1.4776

Time Period based on average effective frequency: 1.1550
```

The resultant output for **Shallow Waves** option is shown below:

```
Wave-1 : Wave Height = 0.2400, Specified Wave Length = 16.0000
Wave Number = 0.5051, Estimated Wave Length = 12.4398
Elliptic Function Parameter (m) : Calculated = 0.9976, Used = 1.0000
Liquid Height = 0.8000, Trough Height = 0.800000
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
Wave Speed = 3.1868, Wave Frequency = 1.6096
Effective parameters: Wave Speed = 4.1868, Wave Frequency = 2.1147
Time Period based on effective frequency: 2.9712
```

26.3.8. Numerical Beach Treatment for Open Channels

In certain applications, it is desirable to suppress numerical reflection near the outlet boundary for wave dampening. To understand the theory involved in this application, refer to Numerical Beach Treatment.

To include numerical beach in your simulation, perform the following:

1. Enable **Gravity** and set the gravitational acceleration fields.

   Setup → **General** **Gravity** → **On**

2. Enable the **Volume of Fluid** model in the **Multiphase Model** dialog box (**Models** tab).

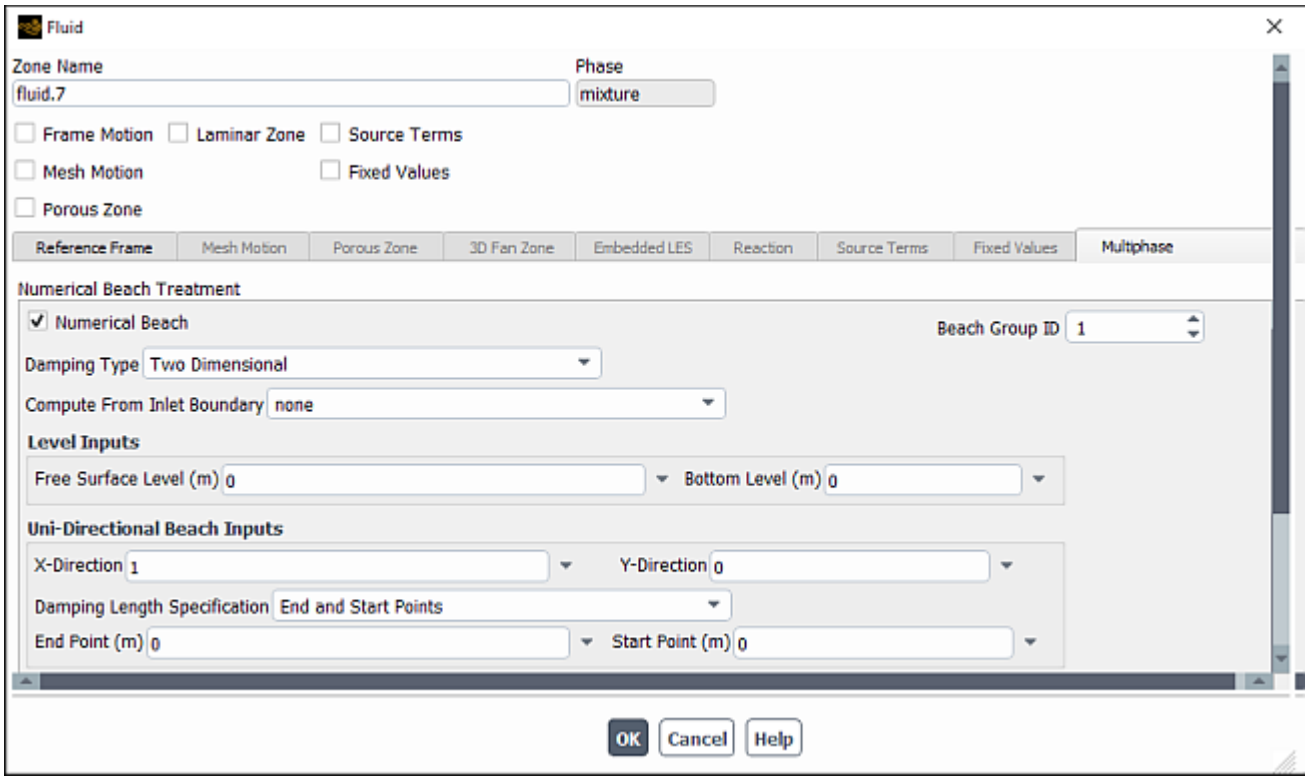
   Setup → **Models** → **Multiphase** → **Edit...**

3. In the **Formulation** group box, select either **Implicit** or **Explicit**.

4. Select **Open Channel Flow** and/or **Open Channel Wave BC**.

In order to set the numerical beach parameters for a fluid zone, go to the **Fluid** dialog box (Figure 26.34: The Fluid Dialog Box to Enable Numerical Beach (p. 3005)).

Figure 26.34: The Fluid Dialog Box to Enable Numerical Beach



In the **Multiphase** tab of the **Fluid** dialog box, enable the **Numerical Beach** option and enter the following:

- **Beach Group ID** represents the cell zones sharing the damping length containing the same input parameters.
- **Multi-Directional Beach** helps to suppress the numerical reflections that arise from multiple open-channel pressure outlet boundaries. This is desirable while modeling oblique waves or when the pressure-outlet boundaries are closer to the zone of interest. (Only available for 3D problems.)
- **Uni-Directional Beach:** should be used to suppress the numerical reflections from a single open channel pressure outlet. This is the case when the flow is generally uni-directional.
- **Damping Type** allows you to choose between **Two Dimensional** and **One Dimensional**.
  - **Two Dimensional** is the damping treatment in the beach and gravity direction.
  - **One dimensional** is the damping treatment in the beach direction.
- **Compute From Inlet Boundary** is set to **none** by default. If there are available open channel boundaries (velocity-inlet, pressure-inlet, and mass-flow-inlet), boundary names are added to the drop-down list. If you select a boundary from the list, the **Level Inputs**, **Uni-Directional Beach Inputs** or **Multi-Directional Beach Inputs** in beach direction, and **Damping Resistance** values will be updated in the interface. You have the option to overwrite the updated inputs with values that are more applicable to your simulation.
- **Level Inputs** is only available for the **Two Dimensional** damping type.
  - **Free Surface Level** is the same definition as for open channel flow, see Modeling Open Channel Flows (p. 2979).
  - **Bottom Level** is the same definition as for open channel flow, see Modeling Open Channel Flows (p. 2979). During the automatic calculation from the velocity inlet boundary for short gravity waves, this parameter is updated under some assumptions, as noted in Solution Strategies (p. 3008). The bottom level is used for calculating the liquid height.

Numerical Beach Inputs

Numerical beach is assumed as a rectangular domain, where its end-plane (outer surface) is an open channel pressure outlet boundary and start-plane is a plane parallel to a pressure outlet. Beach direction is assumed as the direction normal to a pressure outlet. Damping length is the normal distance between the end and start plane.

Uni-Directional Beach Inputs:

X, **Y**, and **Z** are the vector components of the beach outward normal (typically normal to the nearest pressure outlet).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The end point is calculated by taking the dot product of beach direction with any point lying on the end plane. Similarly, the start point is calculated as the dot product of beach direction with any point lying on the start-plane.

Note:

The automatically calculated value of the end point is assumed based on the domain extents for all of the cell zones. Enter your own value if required.

Damping Length Specification is only available if **Open Channel Wave BC** is enabled in the **Multiphase Model** dialog box (**Models** tab). There are two options you can choose from:

- **End Point and Wave Lengths** (default).
- **End and Start Points** are the limits of the damping zone.

Multi-Directional Beach Inputs

- **Number of Beach**

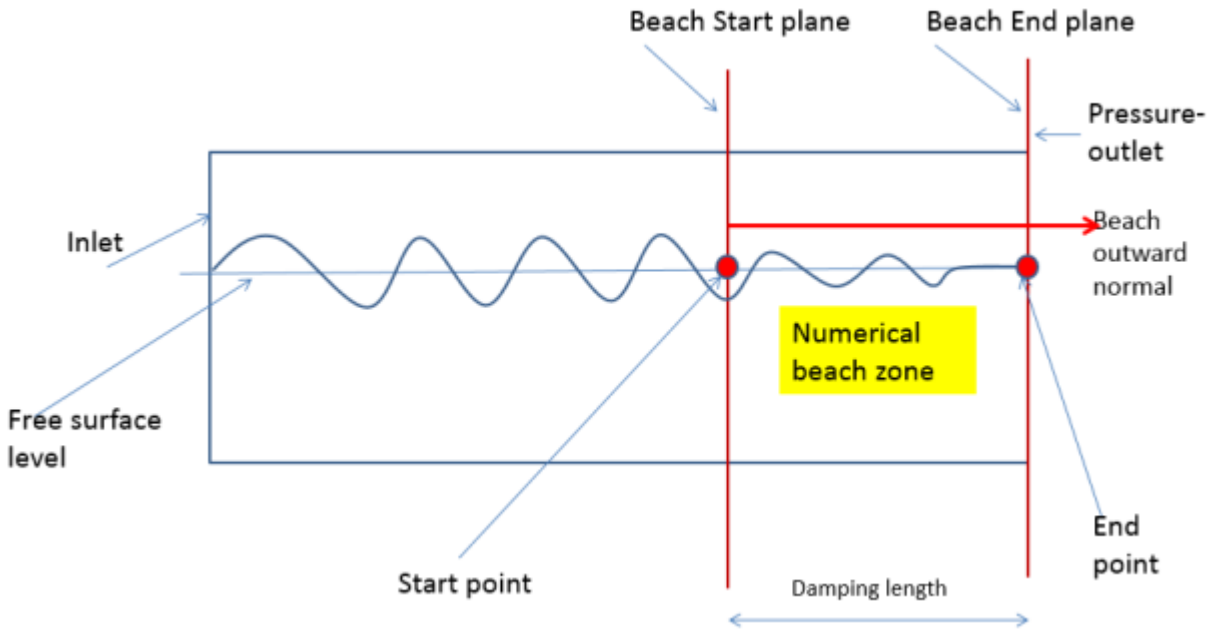
  The default is 2, the maximum is 3.

- **X-Direction**
- **Y-Direction**
- **Z-Direction**
- **End Point**
- **Damping Length**

See Figure 26.35: Numerical Beach Sketch (p. 3008) for a visual representation of the numerical beach.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.35: Numerical Beach Sketch



- **Relative Velocity Resistance Formulation** calculates the source term using relative velocities in the numerical beach zone when using moving/deforming meshes or moving reference frames.
- **Linear Damping Resistance** is the resistance per unit time.
- **Quadratic Damping Resistance** is the resistance per unit length.

26.3.8.1. Solution Strategies

Below are some helpful key points when using the **Numerical Beach** option:

1. The **Compute from Inlet Boundary** drop-down list is a convenient feature as it provides automatic inputs, which you should check to make sure the values are reasonable. Some of the provided values are calculated under certain assumptions:

   a. The bottom level, in the case of short gravity waves, is selected in the velocity inlet dialog box for open channel wave boundary conditions. Since there is no user input for this parameter, Ansys Fluent calculates the bottom level as the value of the free surface level less 2.5 times the wave length. The assumption is that the liquid height is 2.5 times the wave length.

   b. The automatically calculated value of the end point is assumed based on the domain extents for all of the cell zones.

2. For manual inputs of start and end points, you should verify that these points are calculated correctly in the beach direction. To check for their validity, subtracting the start point from the end point should give you a positive value.

3. For manual inputs of **Free Surface Level** and **Bottom Level**, you should verify that these points are calculated correctly in the direction opposite to gravity. To check for their validity, subtracting the bottom level from the free surface level should give you a positive value.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. In case you create separate damping zones, but the damping length is not sufficient to suppress the waves, clubbing of the beach could be done by selecting the same beach ID for other cell zones. In this case, both the cell zones would share the same information.

5. In case of separate damping zones (without clubbing), you should verify that the damping length is more than or equal to the specified number of wave lengths for the calculation of the start point.

6. Damping resistance should be chosen carefully as too much or too little damping could affect the wave profiles in a no-damping zone. The **Compute From Inlet Boundary** option automatically populates the values for damping resistances based on analytical correlations for wave energy. However, you may need to further tune these values for certain cases if the computed values are found to be unsuitable.

7. Steep damping at the beginning of the damping zone could affect the wave profile just before the damping zone.

8. It is recommended that you use a coarse mesh in the damping zone with increased coarseness towards the end of the damping zone.

26.3.9. Defining the Phases for the VOF Model

Instructions for specifying the necessary information for the primary and secondary phases and their interaction in a VOF calculation are provided below.

Important:

In general, you can specify the primary and secondary phases whichever way you prefer. It is a good idea, especially in more complicated problems, to consider how your choice will affect the ease of problem setup. For example, if you are planning to patch an initial volume fraction of 1 for one phase in a portion of the domain, it may be more convenient to make that phase a secondary phase. Also, if one of the phases is a compressible ideal gas, it is recommended that you specify it as the primary phase to improve solution stability.

26.3.9.1. Defining the Primary Phase

To define the primary phase in a VOF calculation, perform the following steps:

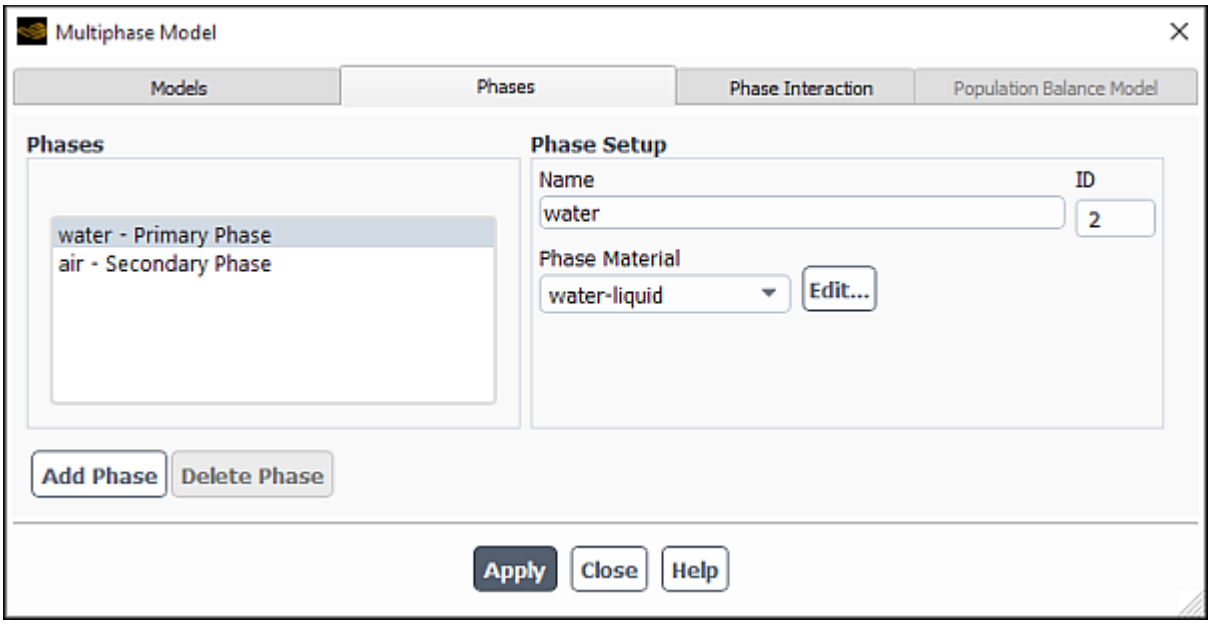
1. In the **Multiphase Model** dialog box, go to the **Phases** tab.

   Setup → Models → Multiphase → Edit...

2. From the **Phases** list, select the phase (for example, phase-1).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.36: Defining the Primary Phase in the Phases Tab



3. In the **Phase Setup** group box, enter a **Name** for the primary phase (for example, `water`).

4. Specify which material the phase contains by choosing the appropriate material in the **Phase Material** drop-down list.

5. Define the material properties for the selected phase material.

   a. Click **Edit...** next to **Phase Material**.

   b. In the **Edit Material** dialog box, check the properties, and modify them if necessary. (See Physical Properties (p. 1597) for general information about setting material properties, Modeling Compressible Flows (p. 2931) for specific information related to compressible VOF calculations, and Modeling Solidification/Melting (p. 3019) for specific information related to melting/solidification VOF calculations.)

   Important:

   If you make changes to the properties, remember to click **Change** before closing the **Edit Material** dialog box.

6. Click **Apply**.

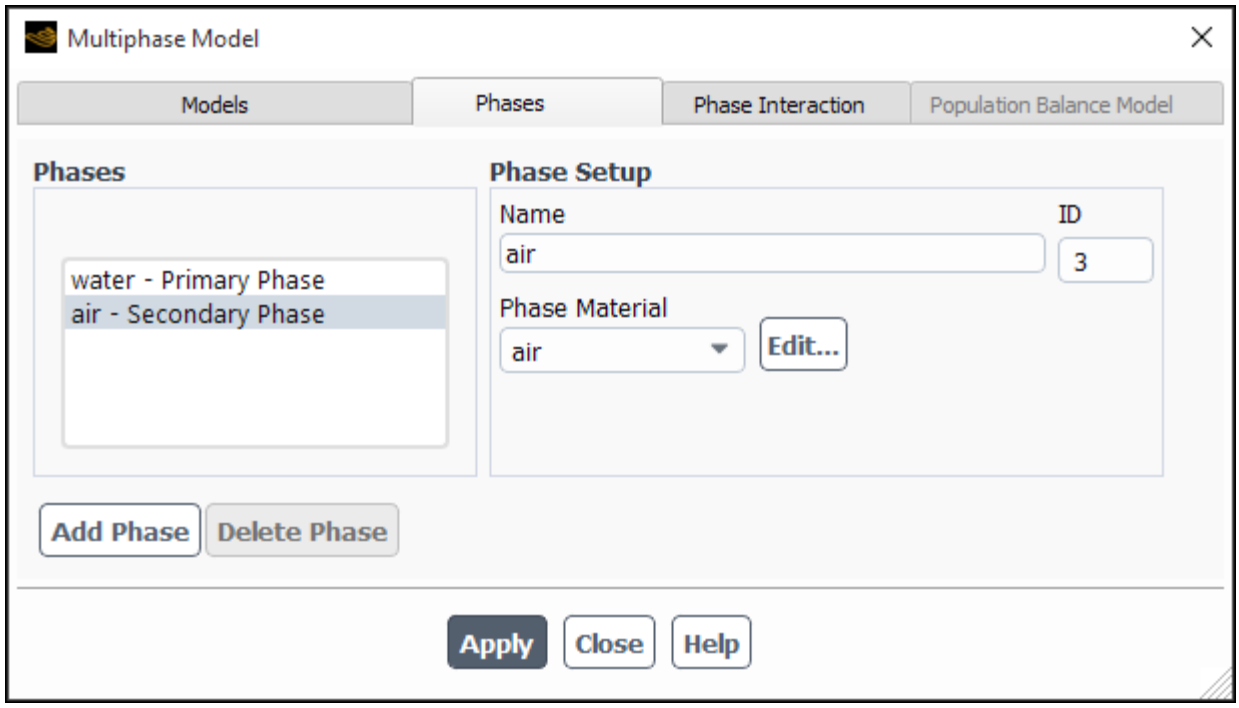
26.3.9.2. Defining a Secondary Phase

The procedure for defining a secondary phase in a VOF calculation is similar to that for the primary phase:

1. Select the phase (for example, **phase-2**) from the **Phases** list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.37: Defining the Secondary Phase in the Phases Tab



2. In the **Phase Setup** group box, enter a **Name** for the phase (for example, **air**).

3. Specify which material the phase contains by choosing the appropriate material in the **Phase Material** drop-down list.

4. Define the material properties for the **Phase Material**, following the procedure outlined above for setting the material properties for the primary phase.

5. Click **Apply**.

26.3.10. Defining Phase Interaction Terms

26.3.10.1. Including Surface Tension and Adhesion Effects

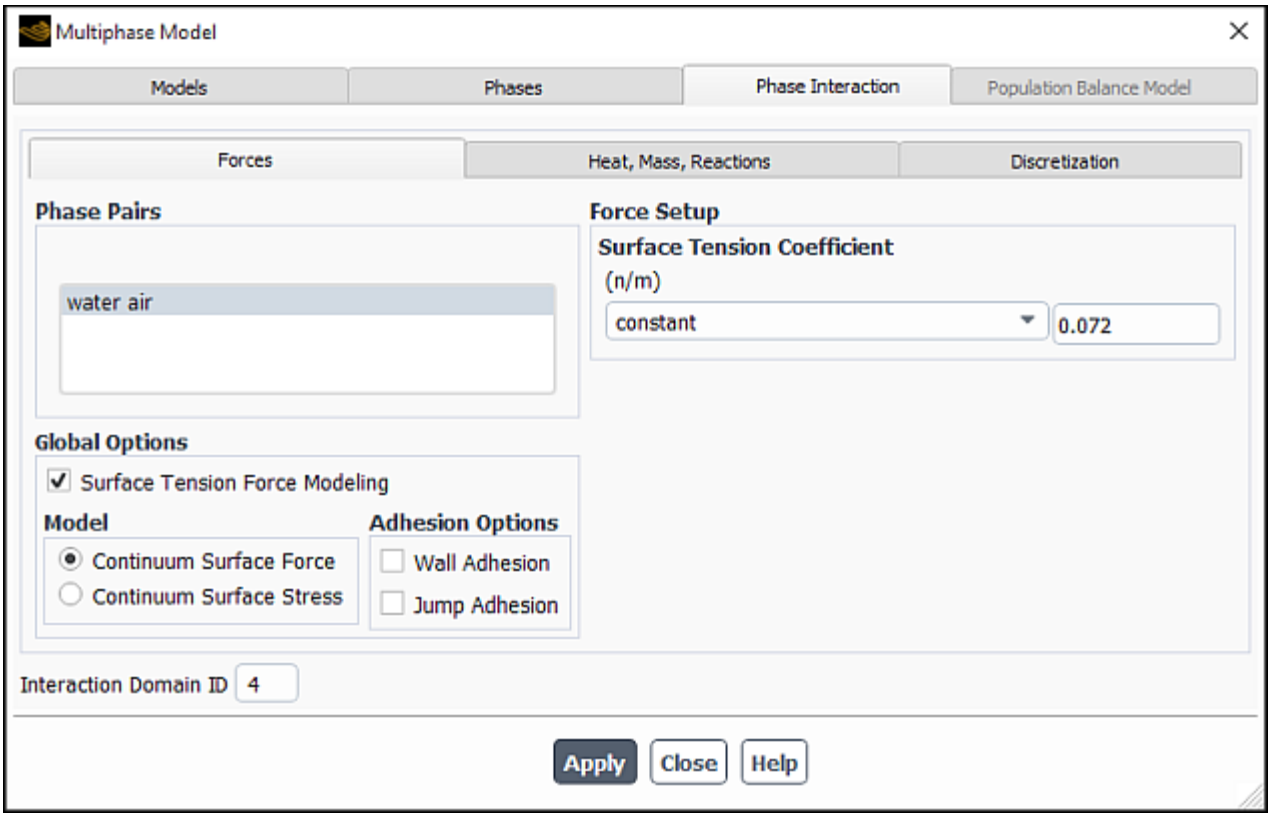
As discussed in When Surface Tension Effects Are Important in the Theory Guide, the importance of surface tension effects depends on the value of the capillary number, Ca (defined by Equation 14.32 in the Theory Guide), or the Weber number, We (defined by Equation 14.33 in the Theory Guide). Surface tension effects can be neglected if $\mathrm{Ca} \gg 1$ or $\mathrm{We} \gg 1$.

You can include the effects of surface tension along the interface between one or more pairs of phases, as described in Surface Tension and Adhesion in the Theory Guide, in the **Phase Interaction** > **Forces** tab of the **Multiphase Model** dialog box (Figure 26.38: The Multiphase Model Dialog Box (Forces Tab) (p. 3012)).

Setup → Models → Multiphase → Edit...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.38: The Multiphase Model Dialog Box (Forces Tab)



Perform the following steps to model surface tension (and, if appropriate, include adhesion) effects along the interface between one or more pairs of phases:

1. In the **Global Options** tab, enable the **Surface Tension Force Modeling** option to include the surface tension method. You can specify the **Surface Tension Coefficients** as **constant**, **polynomial**, **piecewise-linear**, **piecewise-polynomial**, or **user-defined**. You can also use RGP tables to specify **Surface Tension Coefficients** as described in Defining Saturation Properties via RGP Tables (p. 1735).

   Note:

   If **none** is selected, then **Surface Tension Force Modeling** will automatically be disabled.

2. Select the surface tension method that is most applicable to your case. You can choose between **Continuum Surface Force** and **Continuum Surface Stress**. Information about each of the methods is described in Surface Tension in the Theory Guide.

3. For each pair of phases between which you want to include the effects of surface tension, specify an input for the surface tension coefficients using one of the following options:

   - Constant Value
   - Temperature-dependent polynomial, piecewise polynomial, or piecewise linear

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These options are available only while modeling energy.

- User-defined surface tension coefficient

  This can be defined as a function of any space variable or time.

See Surface Tension and Adhesion in the Theory Guide for more information on surface tension, and the Fluent Customization Manual for more information on user-defined functions. All surface tension coefficients are equal to 0 by default, representing no surface tension effects along the interface between the two phases.

Important:

For calculations involving surface tension, it is recommended that you also turn on the **Implicit Body Force** treatment for the **Body Force Formulation** in the **Multiphase Model** dialog box (**Models** tab). This treatment improves solution convergence by accounting for the partial equilibrium of the pressure gradient and surface tension forces in the momentum equations. See Including Body Forces (p. 2933) for details.

---

4. If you want to include wall adhesion, enable the **Wall Adhesion** option and then specify the contact angle at each wall as a boundary condition (as described in Steps for Setting Boundary Conditions (p. 2957)).

5. If you want to include jump adhesion, enable the **Jump Adhesion** option. When **Jump Adhesion** is enabled, you will need to specify the contact angle at the porous jump in the **Porous Jump** dialog box (as described in Defining Multiphase Cell Zone and Boundary Conditions (p. 2957)). The contact angle specification and measurement for jump adhesion is similar to that of wall adhesion (as explained in step 3 above).

Important:

Jump adhesion only supports the cell based smoothing option for surface tension. When the **Jump Adhesion** option is enabled in the **Global Options** group box, the correct settings are automatically set for surface tension. To view the surface tension settings, use the `solve/set/surface-tension` text command.

---

6. Click **Apply**.

Note:

The following limitations exist:

- When the **Wall Adhesion** and **Jump Adhesion** options are enabled, you must specify a nonzero surface tension coefficient.
- **Jump Adhesion** is not available with the **Continuum Surface Stress** model.
- The **Continuum Surface Stress** model and the **Jump Adhesion** option are not available with the **Coupled Level Set + VOF** option.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Several surface tension options are provided through the text user interface (TUI) using the `solve/set/surface-tension` command :

`solve` → `set` → `surface-tension`

The `surface-tension` command prompts you for the following information:

- whether you require node-based smoothing

  The default value is `yes`, indicating that node-based smoothing will be used. Note that if you are reading in a case that was created in versions prior to Ansys Fluent 13, then cell-based smoothing will be used by default.

- the number of smoothings

  The default value is 2, which is suitable for most cases. A higher value can help in reducing spurious currents for highly skewed and non-uniform meshes. Note that for cases created in versions prior to Ansys Fluent version 2020 R1, only single smoothing will be used by default.

- the smoothing relaxation factor

  The default is `1`. This is useful in the cases where volume fraction smoothing causes a problem (for example, liquid enters through the inlet with wall adhesion on).

- whether you want to use volume fraction gradients at the nodes for curvature calculations

  With this option, Ansys Fluent uses volume fraction gradients directly from the nodes to calculate the curvature for surface tension forces. The default is `yes` which produces better results with surface tension compared to gradients that are calculated at the cell centers.

When **Surface Tension Force Modeling** is enabled, the following additional variables become available for postprocessing under the **Phases...** category:

- **Smoothed VOF Gradient-dX**, **Smoothed VOF Gradient-dY**, and **Smoothed VOF Gradient-dZ**
- **Smooth VOF Gradient Magnitude**

See Field Function Definitions (p. 4167) for their definitions.

Important:

- Note that the calculation of surface tension effects will be more accurate if you use a quadrilateral or hexahedral mesh in the area(s) of the computational domain where surface tension is significant. If you cannot use a quadrilateral or hexahedral mesh for the entire domain, then you should use a hybrid mesh, with quadrilaterals or hexahedra in the affected areas.

  Ansys Fluent uses node based smoothing for smoothed volume fraction and node based gradients for smoothed curvature calculation to provide better accuracy and robustness.

- Pressure jump caused by the surface tension is discontinuous in nature, and also it acts locally at the interface. These numerical difficulties are overcome in an approximate

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

manner by considering the smoothed distribution of the volume fraction field within the finite interfacial width. Smoothing procedures, in general, are mesh dependent. Therefore, the amount and nature of the smoothing procedure could have a significant effect on the results for surface tension cases.

26.3.10.2. Discretizing Using the Phase Localized Compressive Scheme

Most of the interface capturing or tracking schemes, which are common in literature, can simulate either diffused interface modeling *or* sharp interface modeling. There are a number of applications where you may be interested in modeling diffused *and* sharp interfaces in different regimes, which results in the need for a unique discretization procedure to handle such applications.

Below are some examples of such applications:

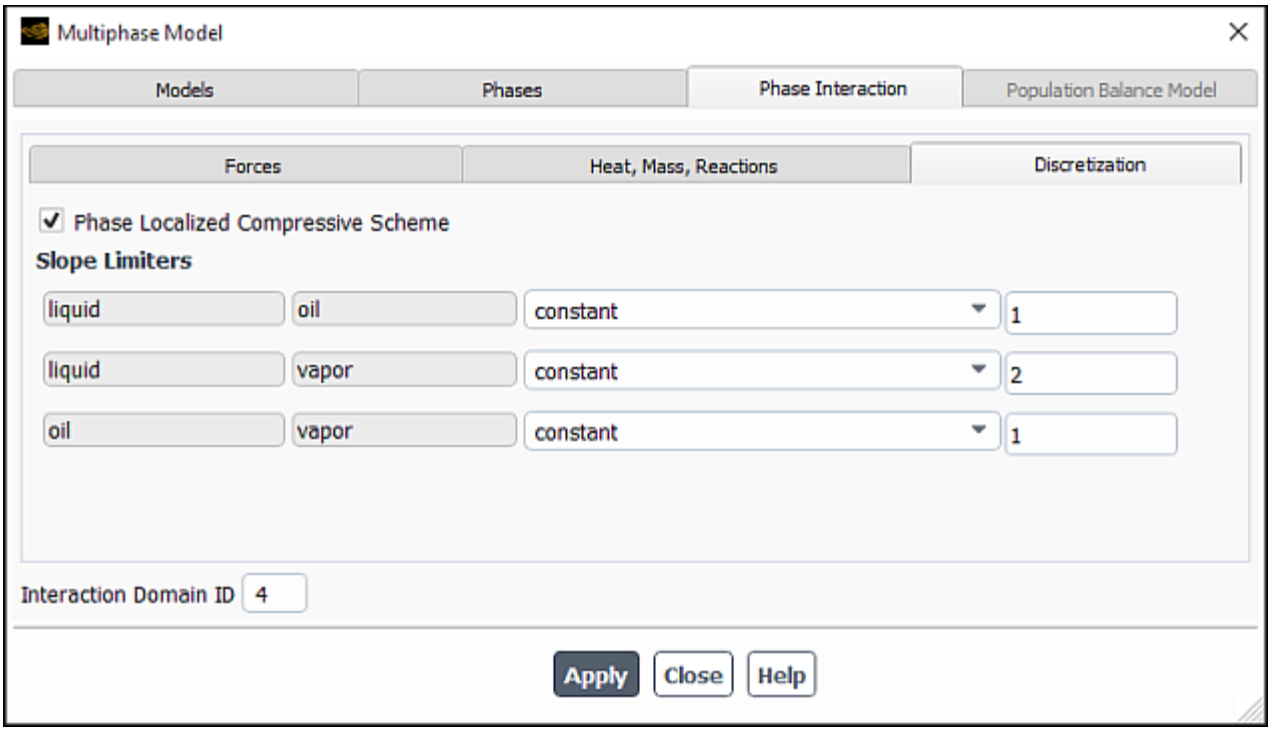
- **Air bubble rise in water-solid slurry:** Diffused interface modeling is required for the water and solid, and sharp interface modeling for the air bubble is desirable. A sharpening scheme would cause undesirable effects for the modeling of the diffused interface between water and solids and a diffusive scheme would not be able to maintain the sharp interface for bubble rising in the slurry.
- **Evaporation/condensation in a tank partially filled with water and being heated at the bottom:** Diffused interface modeling is required for the liquid-vapor and sharp interface modeling for the water-air regime is desirable. A sharpening scheme, would not be able to simulate the diffused phenomena of evaporation/condensation for the liquid- vapor phase and a diffusive scheme would not be able to maintain the sharp interface between water and air.
- **Air jet penetrating through layer of liquids:** You may be interested in diffused jet modeling and sharp interface modeling between the liquid layers. A sharpening scheme would not be able to maintain the continuous air stream and a diffusive scheme would not be able to maintain the sharp interfaces between the layers of liquids.

Using the Modified HRIC and Compressive schemes for such applications would result in the HRIC scheme producing undesirable sharpening of the dispersed phases and undesirable diffusion of the continuous phases, whereas the Compressive scheme would produce undesirable sharpening of the dispersed phases.

Therefore, the phase localized compressive scheme is particularly useful in cases where the desirable behavior is such that you have diffused modeling of dispersed phases and sharp modeling of continuous phases. In Ansys Fluent, diffusive and anti-diffusive discretization procedures can be used across the distinct interfaces, which share a pair of phases. This functionality is provided through the compressive discretization scheme, where the degree of diffusion or sharpness is controlled through the value of the slope limiters. The theory used is described in The Compressive Scheme and Interface-Model-based Variants.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.39: The Multiphase Model Dialog Box for the VOF Model (Discretization Tab)



To use the phase localized compressive scheme, perform the following steps:

1. Click the **Discretization** tab.

   Note:

   The **Discretization** tab is available for the VOF model, Mixture multiphase model, and Eulerian multiphase model with **Mulit-Fluid VOF** only when the **Phase Localized Discretization** option is enabled in the **Interface Modeling Options** dialog box. (Interface Modeling Type (p. 2922)).

2. Enable **Phase Localized Compressive Scheme**.

   Important:

   When this option is enabled, the **Compressive** spatial discretization scheme is automatically selected for the **Volume Fraction** in the **Solution Methods** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. For each pair of phases, specify the value of the slope limiter. You can enter a value of 0, 1, or 2, or any value between 0 and 2. Refer to Table 26.11: Slope Limiter Discretization Scheme (p. 3017) to equate each value of the slope limiter with a discretization scheme.

Table 26.11: Slope Limiter Discretization Scheme

Slope Limiter Value β	Scheme
0	first order upwind
1	second order reconstruction bounded by the global minimum/maximum of the volume fraction
2	compressive
$0<\beta<1$ and $1<\beta<2$	blended: where a value between 0 and 1 means blending of the first order and second order and a value between 1 and 2 means blending of the second order and compressive scheme

When using the **Phase Localized Compressive Scheme**, note the following:

- The minimum limit for the slope limiter is 0, whereas the maximum is 2.
- The slope limiter can be interpreted as the degree of compression/anti-diffusion, where 0 demonstrates the minimum compression and 2 demonstrates the maximum compression.
- For interfaces sharing the diffused phases, a slope limiter value of 2 should not be used. A value between 0 and 1 is recommended.
- For interfaces sharing the continuous phases, any value between 0 and 2 can be used depending on the application.
- For interfaces sharing the continuous and diffused phases, any value between 0 and 2 can be used based on the application.
- If you want variable discretization behavior for an interface between two phases, you can do so via the `DEFINE_PROPERTY` user-defined function.

Note:

– If the interface has a transition from sharp to diffused modeling, you should not experience any problems.

– If the interface has a transition from diffused to sharp modeling, this transition should be smooth by gradually varying the value of the slope limiter within some transition zone.

26.3.11. Setting Time-Dependent Parameters for the Explicit Volume Fraction Formulation

If you are using the **Explicit** volume fraction formulation in Ansys Fluent, an explicit solution for the volume fraction is obtained either once each time step or once each iteration, depending upon your

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

inputs to the model. You also have control over the time step size used for the volume fraction calculation.

You can access the controls for the volume fraction in the **Expert Options** dialog box that opens by clicking **Expert Options...** in the Multiphase Model Dialog Box (p. 4700) (**Models** tab).

For the time-dependent calculation for the VOF model, you need to specify the following settings:

Sub-Time Step Calculation Method

When Ansys Fluent uses the **Explicit** formulation, the time step size used for the volume fraction calculation will not be the same as the time step size used for the rest of the transport equations. Ansys Fluent will refine the time step size for VOF automatically, based on your input for the maximum **Courant Number** allowed near the free surface. The Courant number is a dimensionless number that compares the time step size in a calculation to the characteristic time of transit of a fluid element across a control volume.

The characteristic transit time for a fluid element across the control volume represents the time taken by the fluid to empty out of the cell. This transit time is taken as the smallest of such time in the region near the fluid interface. A sub time step size, for use in the VOF calculations, is computed based on this characteristic time, and the maximum allowed **Courant Number** set in the Multiphase Model Dialog Box (p. 4700) (**Models** tab). For example, if the maximum allowable **Courant Number** is `0.25` (default), the computed sub time step size will have a maximum value equal to a quarter of the minimum transit time for any cell near the interface.

In Fluent, the sub time step size for VOF calculations can be computed using the following options:

- **Velocity Based:** The sub time step size is estimated by the cell size and the fluid velocity normal to the interface:

$$\triangle t_v = C\frac{\triangle x}{v_{fluid}} \tag{26.15}$$

  where, C is the Courant Number, v_{fluid} is the relative velocity of the fluid, and $\triangle x$ is the cell size.

- **Flux Based**: For this option, the sub time step size is estimated based on the cell volume and the summation of outgoing fluxes in the cell:

$$\triangle t_f = C\frac{V}{\sum_{cell} U_f} \tag{26.16}$$

  where, C is the Courant Number, V is the volume, and U_f are the outgoing fluxes.

- **Flux Averaged**: The sub time step size is estimated by volumetric averaging the calculations of the **Flux Based** method for the neighboring cells.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Hybrid**: This is the default method for calculating sub time step size. The sub time step size is estimated by blending the **Velocity Based** and **Flux Averaged** calculation methods.

Note:

- The **Flux Based** calculation method is the most conservative sub-time step size estimation, while the **Velocity Based** calculation is the most aggressive. **Flux Averaged** and **Hybrid** methods fall in between for most cases.

  Aggressive sub time step calculation methods generally save time during interface reconstruction. However, the aggressive method can lead to increased number of iterations due to larger transient errors being introduced while integrating over the sub time steps for the interface reconstruction.

- During the simulation, the **Global Courant Number [Explicit VOF Criteria]** is printed in the console. It is calculated using the previously computed sub-time step Δt_{sub} as:

$$C_{global} = C\frac{\Delta t_{global}}{\Delta t_{sub}}$$

  where C is the maximum allowed Courant number, and Δt_{global} is the global time step.

Solve VOF Every Iteration

By default, Ansys Fluent will solve the volume fraction equation(s) once for each time step. This means that the convective flux coefficients appearing in the other transport equations will not be completely updated each iteration, since the volume fraction fields will not change from iteration to iteration.

When Ansys Fluent solves these equations every iteration, the convective flux coefficients in the other transport equations will be updated based on the updated volume fractions at each iteration. This choice is the less stable of the two, and requires more computational effort per time step than the default choice.

Important:

If you are using sliding meshes, or dynamic meshes with layering and/or remeshing, using the **Solve VOF Every Iteration** option will yield more accurate results, although at a greater computational cost.

26.3.12. Modeling Solidification/Melting

If you are including melting or solidification in your VOF calculation, note the following:

- It is possible to model melting or solidification in a single phase or in multiple phases.
- For phases that are *not* melting or solidifying, you must set the latent heat (L), liquidus temperature ($T_{liquidus}$), and solidus temperature ($T_{solidus}$) to zero.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See Modeling Solidification and Melting (p. 3219) for more information about melting and solidification.

26.3.13. Using the VOF-to-DPM Model Transition for Dispersion of Liquid in Gas

For the analysis of dispersion of liquid in gas, two principal multiphase flow methodologies can be applied:

- In the Euler-Lagrange approach, discrete droplets are tracked through the domain. The particle tracker uses physical properties of individual droplets in order to account for the exchange of mass, momentum, heat, species (and so on) with the continuous phase. The gas volume displacement by the droplets is ignored. This approach is relatively computationally inexpensive because it allows the mesh to be coarser than the size of a droplet. The secondary breakup can be modeled relatively accurately using known breakup models. However, in the dense region of a spray, simulation accuracy may suffer from neglecting the volume displacement. Also, in regions where the spray does not consist of discrete spherical droplets, special models must be used to predict the primary breakup of the initial contiguous liquid jet. These models must be derived from experimental data based upon the operating conditions for each injector.

- In the volume-of-fluid (VOF) approach, the volume fraction of the liquid is stored in each cell. The gas-liquid interface can be tracked by explicit discretization schemes, such as geometric reconstruction. The VOF model requires a much higher mesh resolution and a much smaller time step size. The phase boundary around every droplet must be resolved by a mesh that is significantly finer than the finest droplets. This allows for better predictions of the primary breakup. The volume displacement is inherently accounted for, which can be important for the dense region of the dispersed spray. However, this method is expensive and requires large HPC resources.

Ansys Fluent provides the capability to combine the two approaches described above by using a VOF-to-DPM model transition mechanism. The initial jet and primary breakup are predicted using the VOF model on a sufficiently fine mesh, while the voluminous dilute region of the dispersed spray is simulated by the DPM. The VOF-to-DPM model transition algorithm automatically finds compact liquid masses that have detached from the liquid core in the VOF solution. It then assesses their eligibility for the VOF-to-DPM model transition. If the liquid lump satisfies user-specified criteria, such as lump size and asphericity (that is, relative or normalized deviation from the shape of a perfect sphere), it is removed from the resolved liquid in the VOF simulation and converted to a particle parcel in the Lagrangian formulation. Any local solution-adaptive mesh refinement used to resolve the gas-liquid interface is automatically reverted, so that the Lagrangian particle parcel can be placed in a single large cell. If the volume of a liquid lump is much larger than the volume of the coarsened mesh cell, then the lump will be converted into as many Lagrangian particle parcels as the number of cells spanned by the lump such that the volume of each parcel will be equal to its corresponding cell volume. In early stages after the conversion, the group of particle parcels representing the liquid droplet will travel together, but as they proceed through the domain, their paths may gradually diverge.

Converting liquid lumps to Lagrangian parcels does not impose volume displacement on the continuous-phase VOF flow simulation. In order to avoid spurious momentum sources, a volume of gas with the same volume as the liquid lump is created in the VOF simulation to maintain the volume conservation. The mass source that is equivalent to the mass of the gas volume will affect the overall mass balance.

This approach can be used for simulating liquid atomization in gas turbines, internal combustion engines, and other similar applications.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Such simulations usually require high spatial and temporal resolution to capture all the relevant details of a primary atomization process of a liquid jet. Resource limitations will restrict this approach to detailed analyses in regions near the nozzle. For the complete simulation of a spray system or process, it is recommended that all information about the Lagrangian DPM particle parcels be captured and transferred into a separate simulation that only uses the Euler/Lagrange approach because it allows a coarser mesh and larger time step size to be used.

This can be achieved, for example, by using DPM particle sampling on either a cut plane or the outlet boundary of the domain of the detailed VOF-to-DPM simulation. For unsteady particle tracking, the sampling feature writes information into a file for every particle parcel that crosses the sampling plane or boundary. The file can then be used as an unsteady injection file for another unsteady particle tracking simulation. Ansys Fluent will recreate each particle parcel at the same location and instance in simulated time (`flow-time` in the injection file).

An alternative to using DPM particle sampling for creating such a file is to use the VOF-to-DPM built-in functionality as described below. If you provide a filename, the solver will write a new file containing information for each liquid lump that has been converted. The file has the same format as a DPM sample file for unsteady particle tracking. Therefore, it can be read into a separate DPM simulation as an unsteady injection file.

26.3.13.1. Limitations on Using the VOF-to-DPM Model Transition

- The VOF-to-DPM model transition feature cannot be used with the following:
  - Steady-state flow
  - Multi-fluid VOF
  - Steady DPM particle tracking
  - The Level Set model
- The identification of individual lumps and the calculation of the lump properties do not account for special boundary conditions such as symmetry and periodicity (including periodic non-conformal interfaces). A liquid lump that touches a symmetry boundary is treated as if it was attached to a wall. A lump that intersects a periodic boundary is treated as multiple independent lumps. In both cases, the values calculated for the lump properties, particularly the asphericity, will be affected.

Important:

The VOF-to-DPM model transition mechanism is available only for 3D cases.

26.3.13.2. Setting up the VOF-to-DPM Model Transition

To use the VOF-to-DPM Model Transition capability, follow the steps below:

1. When generating the mesh for your analysis, avoid high-aspect-ratio cells. Adjust the mesh resolution so that four or five levels of adaption will be sufficient for resolving the finest structures when you are using the VOF calculation.
2. If appropriate, define the automatic solution-adaptive mesh refinement.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Domain → **Adapt** → **Automatic...**

For example, if you just want to make sure that the phase boundary is resolved with the highest possible mesh resolution, you can use cell registers that apply the **Curvature** method to the phase volume fraction in order to coarsen all cells with a value below 1.e-10 and refine all cells with a value above 1.e-08. Make sure to use a small **Frequency** (such as 1 or 2).

If the resolution of the initial mesh is not uniform, you can specify a minimum edge length in the **General Adaption Controls** dialog box in order to avoid excessively small cells (and therefore high CFL numbers), or when multiphase-specific time stepping is used, to avoid excessively small time-steps. If a sufficient number of refinement levels is specified, the adapted mesh at the liquid/gas interface will have little variation in cell volume, regardless of the local resolution in the initial mesh.

Note that an easy way to set up the adaption for VOF simulations is to use the **Predefined Criteria** drop-down list, as described in VOF Adaption (p. 3795). This list allows you to select from commonly used criteria for adapting the mesh; the **Multiphase...** submenu provides options for VOF, including **VOF-to-DPM [Generic]** (which uses fairly straightforward cell registers) and **VOF-to-DPM [Advanced]** (which uses more complex expressions).

If you use hanging nodes to create the initial mesh, make sure that you use the default polyhedral unstructured mesh adaption (PUMA) for the automatic solution-adaptive mesh refinement, because the hanging node adaption method will skip two cell layers on the coarse side of the hanging nodes. To select PUMA, use the following text command:

```
mesh/adapt/set/method  2
```

3. In the **Discrete Phase Model** dialog box, select **Unsteady Particle Tracking**.

   Physics → **Models** → **Discrete Phase...**

4. Create a DPM injection of any type as usual and specify a very large value for **Start Time**. This will guarantee that no particles will be injected at the specified location.

   Setup → **Models** → **Discrete Phase** → **Injections** **New...**

5. In the **Models** tab of the **Multiphase Model** dialog box, configure the following settings.

   Physics → **Models** → **Multiphase...**

Tab	Setting	Value
Model	**Volume of Fluid**	(Selected)
Volume Fraction Parameters	**Formulation**	**Explicit**
Interface Modeling	**Type**	**Sharp**
Body Force Formulation	**Implicit Body Force**	(Selected)

6. In the **Phase Interaction** tab of the **Multiphase Model** dialog box, define the correct surface tension for the gas-liquid boundary.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. In the **Solution Methods** task page, configure the following solution parameters.

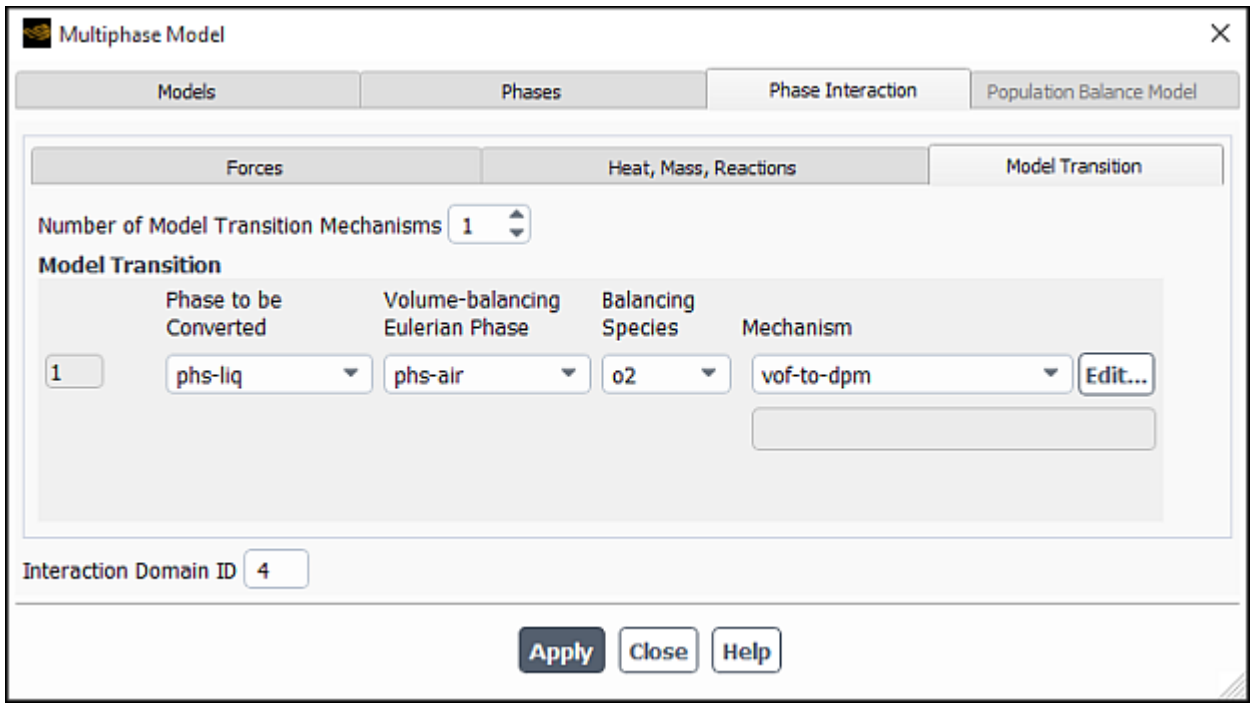
   Solution → Solution → Methods...

Group Box	Setting	Value
Pressure-Velocity Coupling	**Scheme**	**PISO**
Spatial Discretization	**Volume Fraction**	**Geo-Reconstruct**
	Pressure	**PRESTO!**, **Body Force Weighted**, or **Modified Body Force Weighted**

8. If you selected **PISO** for the **Pressure-Velocity Coupling** scheme, increase the **Pressure** and **Momentum** Under-Relaxation Factors to values close to 1 (for example, 0.98) in the **Solution Controls** task page.

   Solution → Controls → Controls...

9. In the **Phase Interaction** > **Model Transition** tab, define VOF-to-DPM model transition.



   a. Specify the **Number of Model Transition Mechanisms**.

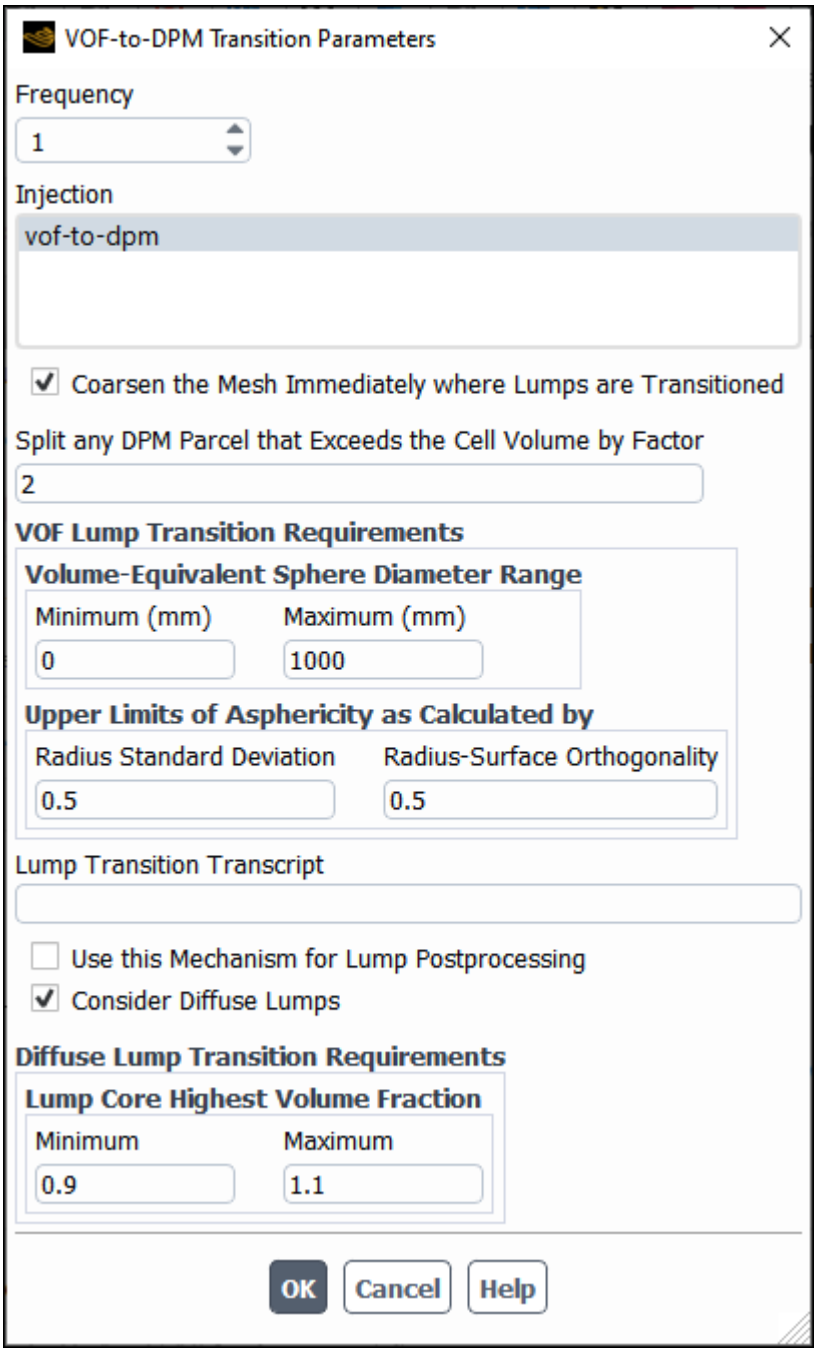
   b. For each mechanism, select the liquid phase from the **Phase to be Converted** drop-down list and also the gas phase from the **Volume-balancing Eulerian Phase** drop-down lists.

      Lagrangian particle parcels do not have any volume displacement effect on any VOF phase. Therefore, in order to maintain volume conservation in the VOF solution, when liquid from the VOF solution is converted into one or more Lagrangian particle parcel(s), an equal volume of the selected gas phase is placed into the VOF solution as a replacement for the converted liquid.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. For each pair of phases with model transition, select **vof-to-dpm** from the **Mechanism** drop-down list.

Figure 26.40: The VOF-to-DPM Transition Parameters Dialog Box



d. In the **VOF-to-DPM Transition Parameters** dialog box that opens automatically, specify the parameters for the model transition:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Frequency

specifies the frequency at which the VOF-to-DPM model transition mechanism is run. That is, this parameter denotes the number of time steps between consecutive VOF-to-DPM model transition passes for the individual model transition mechanism.

Injection

is a list of available injections. Make sure you select the injection that you created earlier for the VOD-to-DPM model transition. All newly created particle parcels will be assigned to the selected injection.

Coarsen the Mesh Immediately where Lumps are Transitioned

specifies whether the mesh should be coarsened when a liquid lump is converted to a DPM particle parcel and removed from the VOF solution. The coarsening is achieved by reverting all previously dynamically added mesh refinements. This option is selected by default and recommended.

Split any DPM Parcel that Exceeds the Cell Volume by Factor

specifies a ratio used for determining the number of DPM parcels into which the liquid lump is divided. If the liquid lump volume does not exceed the volume of the coarsened cell multiplied by this factor, the lump is converted into one parcel containing one particle. Otherwise, the lump is split into as many parcels as there are cells covered by the original liquid lump. To conserve mass, the number of particles per parcel is reduced from unity such that the sum of all parcels resulting from the split represents exactly the mass of the original liquid lump.

The splitting of a single lump into multiple particle parcels may be needed if there is a risk that DPM source terms may cause instabilities in the continuous phase flow solution. This may happen for cases with evaporation or boiling and a fine background mesh.

As long as you do not encounter any stability issues in the continuous phase flow solution, you can set a very high value for this factor in order to prevent lumps from splitting into multiple parcels. However, if the factor is set relatively low, a lump may split into multiple parcels. These parcels are mutually independent and therefore may disperse. Splitting parcels does not change the global size distribution, but they may affect the spatial distribution of parcels of different sizes throughout the spray.

VOF Lump Transition Requirements

contains parameters for specifying conversion eligibility limits.

Volume-Equivalent Sphere Diameter Range

sets the **Minimum** and **Maximum** diameter values for a volume-equivalent sphere. Liquid droplets found in the VOF solution whose volume is outside this range will not be converted to Lagrangian particle parcels.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Upper Limits of Asphericity as Calculated by

Radius Standard Deviation

specifies the maximum asphericity calculated from the normalized radius standard deviation. This quantity is calculated for every liquid lump found in the VOF solution as follows. For every facet of the lump surface (gas-liquid phase interface), the distance between the facet center and the lump center of gravity is calculated. The individual distance values are weighted by the size of the individual lump boundary facet. Their standard deviation is computed and then normalized by the average radius.

This quantity is zero for perfect spheres and increasingly greater than zero the more the shape deviates from a sphere. All droplets for which the asphericity value is below the specified maximum may be automatically elected for conversion. The default value for **Radius Standard Deviation** is 0.5.

Radius-Surface Orthogonality

specifies the maximum asphericity calculated from the average radius-surface orthogonality. This quantity is calculated for every liquid lump found in the VOF solution as follows. For every facet of the lump surface (gas-liquid phase interface), a vector from the lump's center of gravity to the center of the lump boundary facet is computed. This vector is normalized and then used in a dot product with the facet unit normal to yield a measure of a relative orthogonality ranging from 0 to 1. For a perfect sphere, the value would all be 1. A facet area-weighted average of these values is then subtracted from 1 to obtain the asphericity value.

For almost spherical droplets, this quantity is significantly closer to 0 than the standard deviation-based asphericity. For more drastically non-spherical lumps, it produces values comparable to those obtained with the standard deviation-based asphericity. The default value for **Radius-Surface Orthogonality** is 0.5.

Lump Conversion Transcript

(optional) specifies a name for the file where Ansys Fluent will save the DPM particle variables corresponding to each VOF lump that is converted. The file contains one line for each converted lump, regardless of whether a lump was converted into one or multiple DPM parcels. The file is written in the same format that is used for particle sampling (see Sampling of Trajectories (p. 2873)).

As with any sample file written by unsteady particle tracking, this file can be used as an unsteady injection file. This makes it possible to calculate the small dense region of a spray using the VOF-to-DPM model transition mechanism and the much larger dilute region in a pure DPM calculation that uses the lump conversion transcript as an injection file. For better control over the number of the numerical particle parcels, the injection file can be run through the file reduction tool as described in Data Reduction of Samples (p. 2879).

If you use a `DEFINE_DPM_OUTPUT` UDF to write the particle sampling file, the same UDF will be used to write the lump conversion transcript file. The UDF gives you control over the file content and it even allows you to modify the DPM particle parcels generated

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

from the liquid lumps during the VOF-to-DPM model transition. For details, see DEFINE_DPM_OUTPUT in the *Fluent Customization Manual*.

You can use the DEFINE_DPM_OUTPUT UDF to add additional DPM variables to the output file in extra columns. If you then use that lump conversion transcript file as an unsteady injection file in a separate simulation, you can use a DEFINE_DPM_INJECTION_INIT UDF to read the additional columns and use the data to initialize the additional DPM variables. For details, see DEFINE_DPM_INJECTION_INIT in the *Fluent Customization Manual*.

Use this Mechanism for Lump Postprocessing

when selected, Ansys Fluent uses the mechanism settings for the calculation of the postprocessing lump detection quantities. See Postprocessing for VOF-to-DPM Model Transition Calculations (p. 3029) for information about postprocessing lump detection quantities.

Consider Diffuse Lumps

when selected, distinct liquid lumps that for some numerical reason started to be smeared out in the mesh before they detached fully from the contiguous liquid jet will be converted to Lagrangian particle parcels. You can specify the following **Diffuse Lump Transition Requirements** in the **Lump Core Highest Volume Fraction** group box:

Minimum

controls the conversion of distinct liquid lumps. If the highest VOF liquid volume fraction within the lump is below the specified value, the lump will not be converted. The default value is 0.9.

Maximum

can be used to restrict the conversion to diffused lumps only. If the highest VOF liquid volume fraction within the lump is above the specified value, the lump will not be converted. The default value is 1.1. Note that although the volume fraction cannot exceed unity, the default value of 1.1 is used to prevent roundoff errors.

The VOF-to-DPM solver finds the highest VOF liquid volume fraction within each eligible lump that meets all other criteria. Only if this value lies within the specified range will the lump be converted to the DPM Lagrangian particle parcel.

Refer to Best Practice Guidelines for Considering Diffuse Lumps (p. 3028) for tips for using the **Consider Diffuse Lumps** option.

The liquid lumps having dimensions within the specified lump diameter range and falling below both the asphericity criteria will be elected for conversion to DPM parcels.

In order to see typical asphericity values for the droplets in your simulation, you can, for example, create contour plots of the asphericity field functions on an iso-surface of a volume fraction of 0.5.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.3.13.3. Best Practice Guidelines for Considering Diffuse Lumps

The **Consider Diffuse Lumps** option (see Setting up the VOF-to-DPM Model Transition (p. 3021)) can be useful in the following scenario.

In industrial VOF-to-DPM calculations, some diffused VOF liquid structures may not be converted to Lagrangian parcels although they have obviously detached from the main liquid jet. If the automatic solution-adaptive mesh refinement has been set up such that it keeps every cell that contains any liquid (or is in close vicinity to such a cell) refined, then these liquid lumps can greatly increase computational time.

The following strategy helps to minimize the occurrence of such diffused liquid lumps.

In the **Phase Interaction** dialog box, under the **Model Transition** tab, define the following three **vof-to-dpm** mechanisms. The mechanisms will always be processed in the order in which they appear in the user interface.

- Mechanism I:

  1. Allow tiny droplets only by setting **Minimum** (in the **Volume-Equivalent Sphere Diameter Range** group) to 0 and **Maximum** to a drop size that the mesh is too coarse to resolve (similar to the average length of a typical cell near the gas-liquid interface).

  2. Allow any droplet shape by setting both Asphericity criteria to large values.

  3. In the **Lump Core Highest Volume Fraction** group box, set the **Minimum** to a very low value (for example, 0.1), and keep the default value of 1.1 for **Maximum**.

- Mechanism II:

  1. Restrict the size and shape of the droplets to values appropriate for your application. This mechanism is the one that converts the lumps of your interest.

  2. In the **Lump Core Highest Volume Fraction** group box, use the default values of 0.9 and 1.1 for **Minimum** and **Maximum**, respectively.

- Mechanism III:

  1. Allow any size and any shape of the droplets.

  2. Allow smeared-out lumps only by setting **Minimum** to a low value (for example, 0.1) and **Maximum** to a somewhat reduced value (for example, 0.8 or, at most, 0.9) in the **Lump Core Highest Volume Fraction** group box.

     This mechanism will detect and convert all separated contiguous lumps of non-zero liquid volume fraction that do not contain any highly concentrated (close to 100% volume fraction) liquid regions.

If you use a different injection in each of these mechanisms, then in the future, you will be able to easily determine which particles were generated by which model transition mechanism.

Note that structures of cells with non-zero liquid volume fraction that do not detach completely from the initial jet (that is, they remain connected to the initial jet by a bridge of non-zero liquid volume fraction) will be regarded as part of the jet and will not receive any special treatment.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Should such structures form and have an impact on the fidelity of the simulation, then that is an indication that either the VOF interface tracking algorithm or the discretization scheme cannot keep the gas-liquid interface sufficiently sharp. In such situations, you should double-check your case setup and make sure that the VOF Courant number remains around or below unity. If you really need to change the volume fraction cut-off for lump detection, you can use the `define/phases/set-domain-properties/interaction-domain/model-transition/model-transition` text command to modify the value for `Lump Periphery Lower Liquid Volume Fraction Limit`.

26.3.13.4. Postprocessing for VOF-to-DPM Model Transition Calculations

The following additional postprocessing variables will become available for postprocessing under the **Lump Detection...** category:

- **Lump ID**
- **Lump Diameter**
- **Lump Density**
- **Lump Asph. by Rad. Std. Dev.**
- **Lump Asph. by Orthogonality**
- **Lump X, Y, Z Coordinate**
- **Lump X, Y, Z Velocity**
- **Lump Temperature**

When the energy equation is enabled, the following additional variables are available for postprocessing under the **Lump Detection...** category:

- **Lump Temperature**
- **Lump Enthalpy**
- **Lump Pressure**

26.3.14. Using the DPM-to-VOF Model Transition

In some applications, a liquid phase appears both in a dispersed form (droplets) and in a continuous form (pool or wall-film). Depending on the flow, droplets may hit the free liquid surface or wall-film and merge into it. Typical examples are rainwater falling on a vehicle windscreen or entering ventilation intake ducts, lubrication oil injected into a bearing chamber, or pre-filming liquid fuel atomisation.

For modeling such applications, using both the discrete phase model (for simulating the droplets) and the VOF multiphase model (for simulating the liquid pool / thick wall-film) is recommended. Where the wall-film is thin, it can be modeled using the Lagrangian wall-film model, which is part of the discrete phase model (DPM).

Where Langrangian DPM particles travelling in the gas phase hit the gas-liquid interface in the VOF formulation, particles are usually expected to merge into the VOF liquid phase. To achieve this, the

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DPM-to-VOF model transition mechanism couples the VOF and DPM solutions allowing to accurately capture the process of DPM droplets merging into the liquid pool / film.

In addition, the DPM-to-VOF model transition mechanism can couple the Lagrangian wall-film model with the VOF solution in both directions:

- Where the film height exceeds a threshold, the Lagrangian wall film is transitioned into the VOF formulation
- Where the VOF liquid attached to a wall forms a sufficiently thin film, it can be transitioned back into the Lagrangian wall-film formulation

26.3.14.1. Setting up the DPM-to-VOF Model Transition

Note:

Using the double-precision version of the Ansys Fluent solver is recommended when simulating the DPM-to-VOF model transition.

1. Enable and set up the Discrete Phase Model and the VOF multiphase model as appropriate for your case.

   Note:

   - In the **Discrete Phase Model** dialog box, make sure to enable **Unsteady Particle Tracking**.
   - Setting **DPM Iteration Interval** to a non-zero value is recommended.

2. Assign materials with compatible liquid properties to a DPM injection and one of the VOF phases.

   - If you plan to use the droplet or multi-component particle type for any DPM injection, you need to perform the following steps:

     a. Enable the Species model.

     b. Assign a gaseous mixture material to the primary VOF phase.

     c. Assign a liquid material (either single-component or mixture) to the secondary VOF phase. The droplet particle material and the VOF liquid phase material must match in all material properties, and if they are multicomponent mixtures, the material properties of all their individual components must also match.

   - It is possible to use a multi-component mixture in the VOF liquid phase (with a matching multicomponent particle material in the DPM injection) while using a pure single-component material for the VOF gas phase. For that, you must make sure to assign the liquid phase mixture material to the primary phase. Note that such a configuration does not permit to simulate DPM particles evaporating into the gas phase.

3. Open the **Phase Interaction** tab in the **Multiphase Model** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setup → Models → Multiphase → Edit...

4. In the **Model Transition** tab, set the **Number of Model Transition Mechanisms**.

5. For each mechanism, select:

   - the VOF liquid phase from the **Transitioning Phase** drop-down list
   - the other VOF phase (preferably gaseous) from the **Volume-Balancing Eulerian Phase** drop-down lists

     The volume of this phase will be used to introduce the VOF liquid.

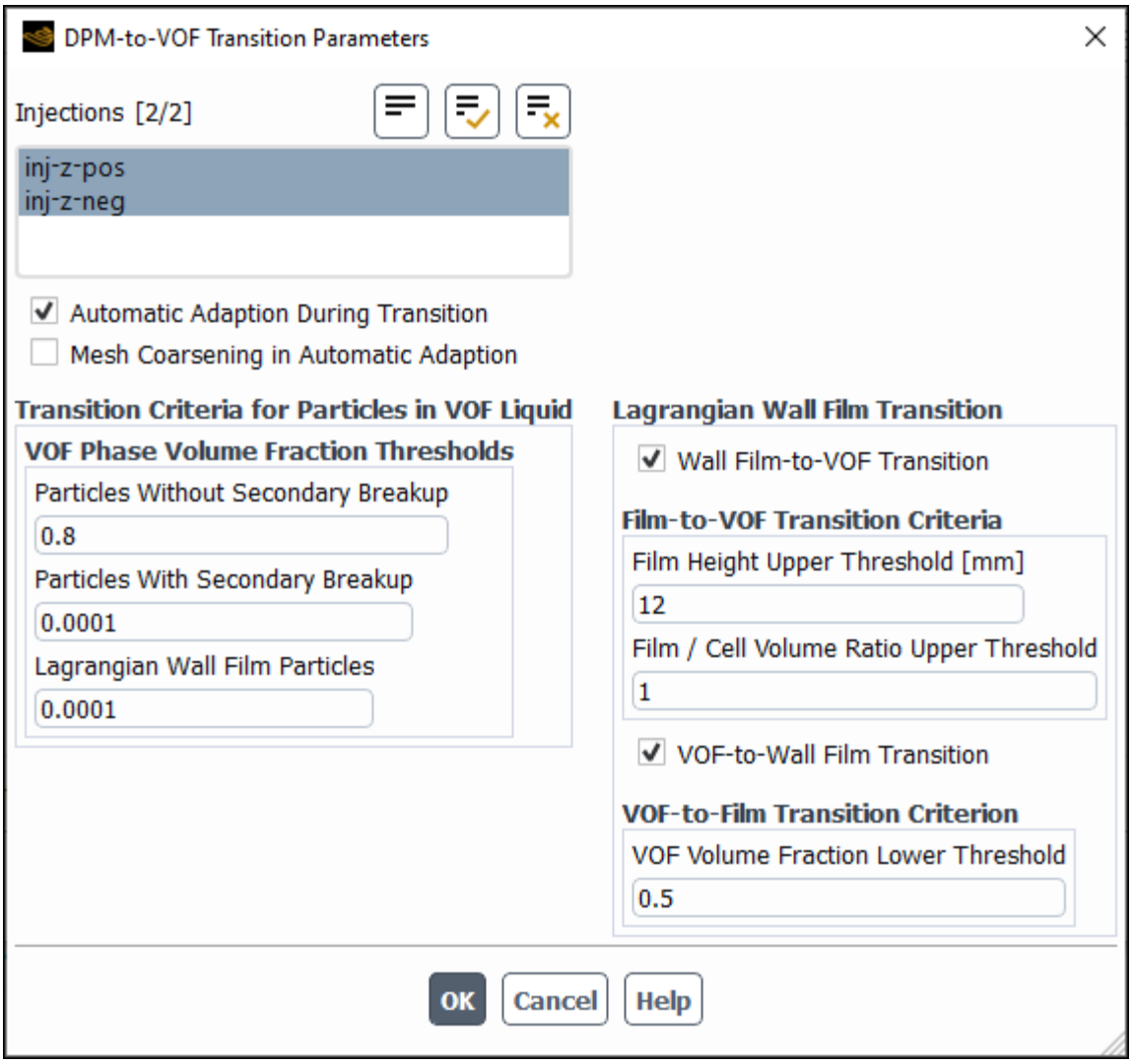
     The DPM particles do not have any volume-displacing effect on Eulerian phases. Therefore, when a DPM particle representing a liquid droplet is to be converted into VOF liquid, strict mass conservation of all phases would require a local volume source in the Eulerian formulation. This would result in unrealistic local pressure surges equivalent to local momentum sources pointing radially outwards in all directions away from the location of the particle conversion.

     To avoid these non-physical and destabilizing effects on the numerical solution, when the VOF liquid is introduced in the amount represented by the DPM particle, an equal (by volume) amount of the VOF gas phase is removed.

6. For each pair of phases with model transition, select **dpm-to-vof** from the **Mechanism** drop-down list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.41: The DPM-to-VOF Transition Parameters Dialog Box



7. In the **DPM-to-VOF Transition Parameters** dialog box that opens automatically, specify the model transition parameters.

   a. In the **Injections** multiple-selection list, select one or more DPM injections.

      Whenever a particle allocated to any of the selected injections reaches a cell with a volume fraction of the Eulerian **Transitioning Phase** exceeding the thresholds specified in **Transition Criteria for Particles in VOF Liquid** as further described, it undergoes a transition.

   b. If you use automatic mesh adaption, enable **Automatic Adaption During Transition** (recommended).

      This option makes sure that the new shape of the VOF gas-liquid interface is correctly resolved by the refined mesh for the adaption criteria activated in the **Manage Adaption Criteria** dialog box.

      If no mesh adaption is applied during the model transition, the new VOF gas-liquid phase interface will be placed in cells that may be refined by the automatic adaption in subsequent

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

time steps. That adaption would then refine cells containing the gas-liquid interface. This would result in a diffuse representation of the interface in the VOF solution, which is not desirable.

Note that this option may consume a significant amount of time. This is because every automatic adaption pass adds or removes at most one level of refinement. Multiple levels of refinement may be allowed by the adaption criteria. The DPM-to-VOF mechanism continues the adaption cycle until the mesh no longer changes.

c. Unless needed for special reasons, keep the **Mesh Coarsening in Automatic Adaption** option disabled. Otherwise, depending on the exact criteria for mesh refinement and coarsening, the mesh cell count may never stop changing in the automatic adaption iterative cycling, which will greatly increase the computational time. In that case, a built-in limit prevents the infinite loop.

d. In the **Transition Criteria for Particles in VOF Liquid** group box, specify the **VOF Phase Volume Fraction Thresholds**. As soon as a DPM particle enters a computational cell in which the local volume fraction exceeds the respective threshold, the trajectory calculation of the particle is stopped, and the particle is converted into the equivalent amount of the selected **Transitioning Phase**. You can specify the following three thresholds:

   - **Particles Without Secondary Breakup**

     It is recommended that you use a high value for this parameter (usually between 0.7 and 0.9). Converting DPM particles without secondary breakup only when they come into contact with the VOF liquid core at high volume fractions helps avoid artificial entrapment of VOF gas between the existing VOF liquid and the new VOF liquid formed by the model transition. The default is 0.9.

     In the case of diffuse VOF phase boundaries (several layers of cells having VOF liquid volume fractions between zero and one) it may be preferable to reduce this threshold in order to avoid the coexistence of the VOF liquid and the Lagrangian (DPM) particles.

   - **Particles With Secondary Breakup**

     For particles with a secondary breakup model enabled, the model transition should be triggered as soon as the particle enters a cell with even a small VOF liquid volume fraction. Otherwise, such particles may undergo non-physical secondary break-up. This is so because the density of the local mixture of all VOF phases is used to calculate, for example, the particle Weber number, which in many secondary breakup models controls the size of the child droplets arising from a breakup event. This would result in excessively small child droplets, which would affect both the accuracy and progress of the simulation.

   - **Langrangian Wall Film Particles**

     The transition of Lagrangian wall film particles generates, or adds to, the VOF liquid that is in contact with the wall. Therefore, there is no generic risk of gas being entrapped between pre-existing and newly formed VOF liquid. For that reason, and since the coexistence of the Lagrangian wall film and the VOF liquid is usually not desirable, a small value is recommended for this threshold. The default is 0.0001.

e. (optional) If you want the Lagrangian wall film to be also transitioned to the Eulerian VOF formulation, enable **Wall Film-to-VOF Transition** (**Lagrangian Wall Film Transition** group

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

box) and specify the **Film Height Upper Threshold** in the **Film-to-VOF Transition Criteria** group box. The Lagrangian wall film will be transitioned to the VOF liquid where the smoothed film height exceeds this threshold.

f. (optional) If you want the wall-adjacent VOF liquid to be transitioned back into the Lagrangian formulation, enable **VOF-to-Wall Film Transition** and specify the **VOF Volume Fraction Lower Threshold** (**VOF-to-Film Transition Criterion** group box).

The VOF liquid in wall-adjacent mesh cells that contain volume fractions lower than this threshold will be transitioned back to the Lagrangian wall-film formulation if the surrounding cells do not contain any VOF liquid. The allowed values range between 0 and 0.9. The default value is 0.5.

If the **Wall Film-to-VOF Transition** option is enabled, you also need to specify **Film / Cell Volume Ratio Upper Threshold**. This value should be greater by at least 0.1 than that specified for the **VOF Volume Fraction Lower Threshold**. The default is the maximum of 1.0.

The Lagrangian wall film transition will be restricted to places where both criteria are fulfilled:

- the film height exceeds the **Film Height Upper Threshold**
- the film volume exceeds the cell volume multiplied by the value of the **Film / Cell Volume Ratio Upper Threshold**

This way, oscillating transitioning back and forth between Lagrangian and VOF formulations is minimised.

8. Click **OK** in the **DPM-to-VOF Transition Parameters** dialog box, and then click **Apply** in the **Multiphase Model** dialog box.

The tracking of a DPM particle is stopped as soon as the transition criteria are met, and the particle is scheduled for the model transition. All such particles are then transitioned into equivalent amounts of the selected VOF phase before the transport equation solver for the continuous phase is started for the current timestep.

The DPM-to-VOF model transition mechanism conserves mass and momentum of the transitioning phase regardless of the sizes of the particles and the cells. Particles that are much smaller than the cell are usually transitioned into the same cell, while particles that are larger than the cell are transitioned by introducing VOF liquid and removing equal amounts of VOF gas in multiple cells that form a hemispherically-shaped phase boundary around the location of the particle at the time when the model transition mechanism is triggered.

The figures below show changes in the gas-liquid interface and in the mesh in the process of the DPM-to-VOF model transition and automatic mesh adaption.

Figure 26.42: DPM Particles at the Gas-Liquid Interface (p. 3035) displays a planar gas-liquid surface and three large Lagrangian DPM particles. The center points of the particles have just reached the phase boundary and the particles are about to be converted.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.42: DPM Particles at the Gas-Liquid Interface

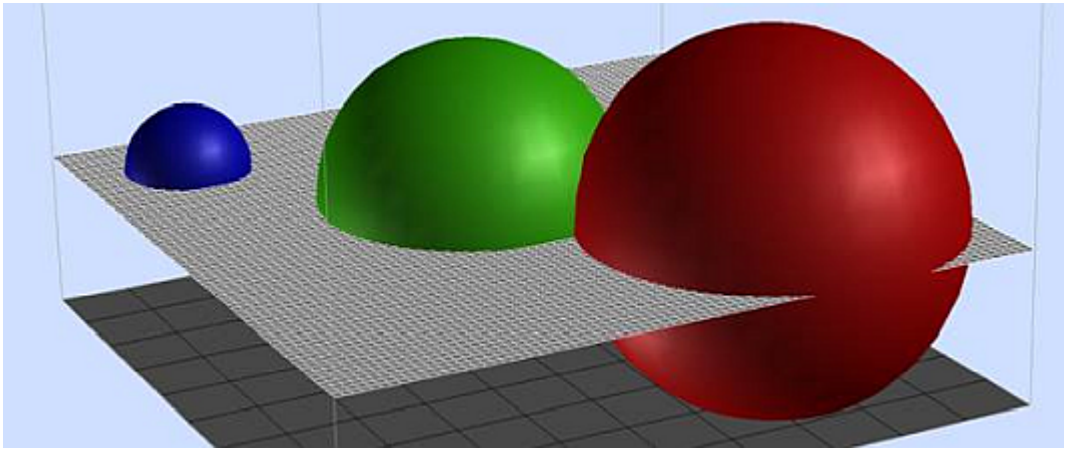


Figure 26.43: Surface Mesh Before the Model Transition is Triggered (p. 3035) shows the surface mesh of the surrounding box. The phase boundary has been resolved by the solution-adaptive mesh refinement using appropriate criteria.

Figure 26.43: Surface Mesh Before the Model Transition is Triggered

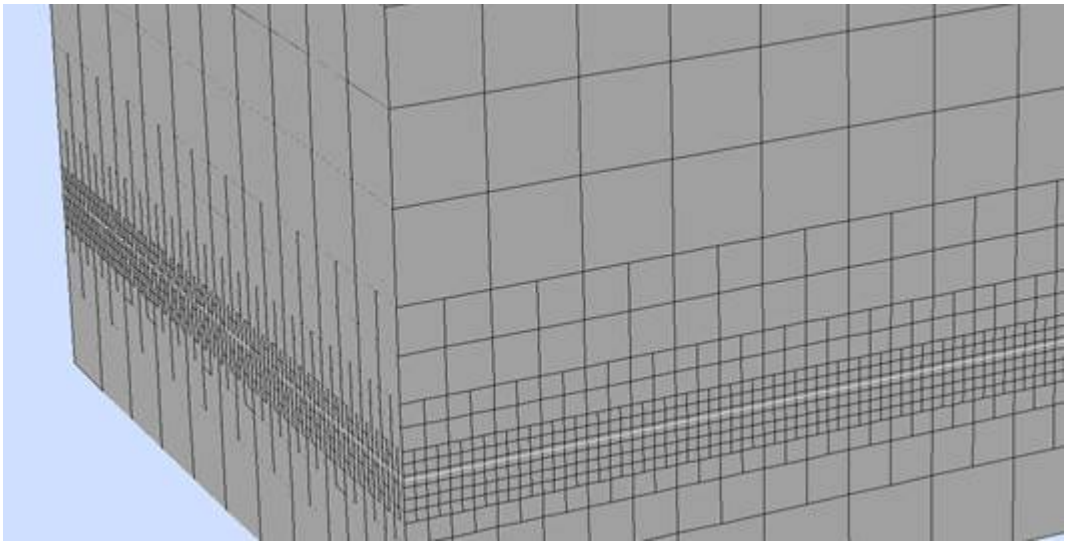
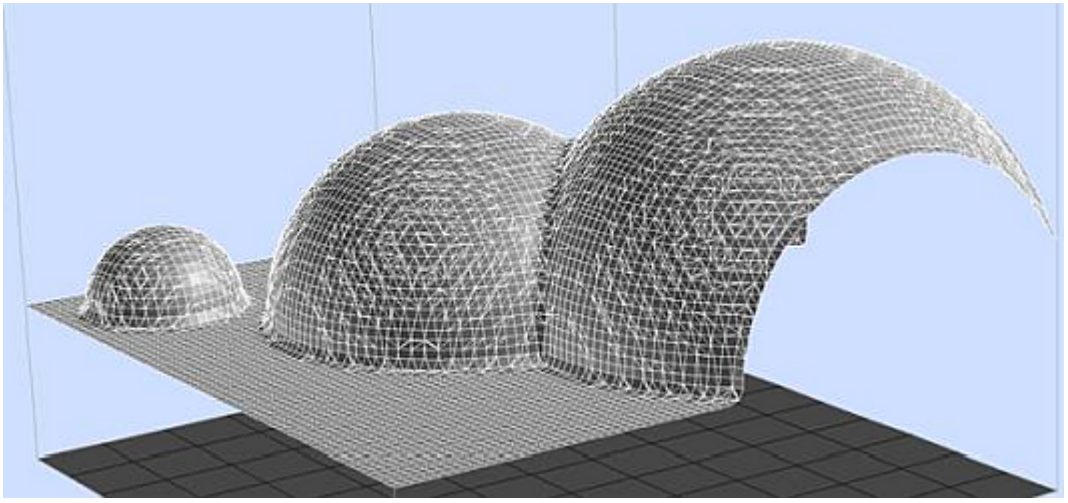


Figure 26.44: Gas-Liquid Interface after the Model Transition is Triggered (p. 3036) shows the VOF phase interface right after the model transition has been triggered.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.44: Gas-Liquid Interface after the Model Transition is Triggered



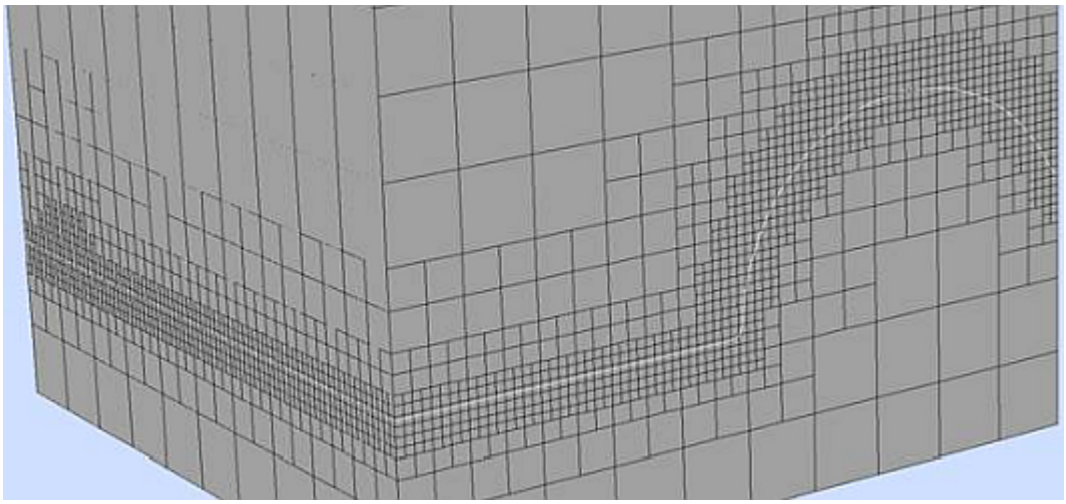
For the smallest particle, a hemisphere has been formed. Its volume is equal to that of the Lagrangian DPM particle. This includes the part of the particle that has already penetrated the VOF liquid.

The other two particles overlapped, which was neglected in the calculation of the particle trajectories. This has been taken into account by the DPM-to-VOF model transition mechanism to ensure total conservation of mass.

Figure 26.45: Surface Mesh After the Mesh Adaption and Phase Transition (p. 3037) demonstrates the surface mesh of the same surrounding box as in Figure 26.43: Surface Mesh Before the Model Transition is Triggered (p. 3035) after the DPM particles have been transitioned to VOF liquid. During the model transition, automatic adaption has been triggered as many times as necessary, so that the maximum mesh refinement has been reached, and the new gas-liquid boundary has been formed as a result. At the same time, the mesh coarsening has removed the mesh refinement in the location inside the large particle where an interface no longer exists after the model transition.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.45: Surface Mesh After the Mesh Adaption and Phase Transition



Note:

The DPM-to-VOF model transition mechanism is independent of the phase materials. It may, therefore, be possible to apply the same model transition mechanism to scenarios with exchanged gas and liquid phases, for example, for gas bubbles modeled as Lagrangian DPM particles that reach a free liquid surface. However, no experience has been gathered with the application of this transition mechanism to such a scenario.

In addition, the volume-balancing approach would result in more recognizable mass losses in the liquid phase as VOF liquid would be removed and replaced by VOF gas wherever a Lagrangian DPM particle reaches the VOF gas-liquid interface. This may be appropriate if the process of displacing liquid by accumulating gas is to be modeled, but it is not acceptable, for example, for the analysis of a bubble column or an aerated stirred tank reactor where conservation of the liquid is expected.

For such applications, it is recommended to either approximate the free liquid surface with the **No Slip** shear condition and **escape** DPM wall boundary conditions or use a user-defined function to stop particle tracking for particles representing gas bubbles that have reached the free liquid surface.

26.3.14.2. Limitations

In the current implementation of the DPM-to-VOF model transition, no splashing or any other kind of interaction between a droplet and a film or pool of liquid is considered. Whenever the center of a Lagrangian DPM particle reaches a cell that fulfills the applicable criteria, the amount of liquid represented by the particle is assumed to stick to the existing VOF liquid.

In addition to momentum and mass of the transitioning phase, heat and composition are also conserved. However, as explained above, in the **Volume-Balancing Eulerian Phase**, conservation of mass (or volume) and other quantities associated with the mass is not fulfilled.

The DPM-to-VOF model transition is not compatible with:

- Steady-state flow

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Steady particle tracking
- 2D Fluent mode
- Eulerian multiphase model with **Multi- Fluid VOF** enabled
- Shared-memory parallel DPM tracking
- **Use DPM Domain** option of the **Hybrid** parallel DPM tracking mode

26.4. Setting Up the Mixture Model

For background information about the mixture model and the limitations that apply, refer to Overview in the Theory Guide.

For additional information, see the following sections:

26.4.1. Defining the Phases for the Mixture Model
26.4.2. Including Mixture Drift Force
26.4.3. Using the Flow Regime Modeling
26.4.4. Using the Singhal et al. Expert Cavitation Model
26.4.5. Including Semi-Mechanistic Boiling

26.4.1. Defining the Phases for the Mixture Model

Instructions for specifying the necessary information for the primary and secondary phases and their interaction for a mixture model calculation are provided below.

26.4.1.1. Defining the Primary Phase

The procedure for defining the primary phase in a mixture model calculation is the same as for a VOF calculation. See Defining the Primary Phase (p. 3009) for details.

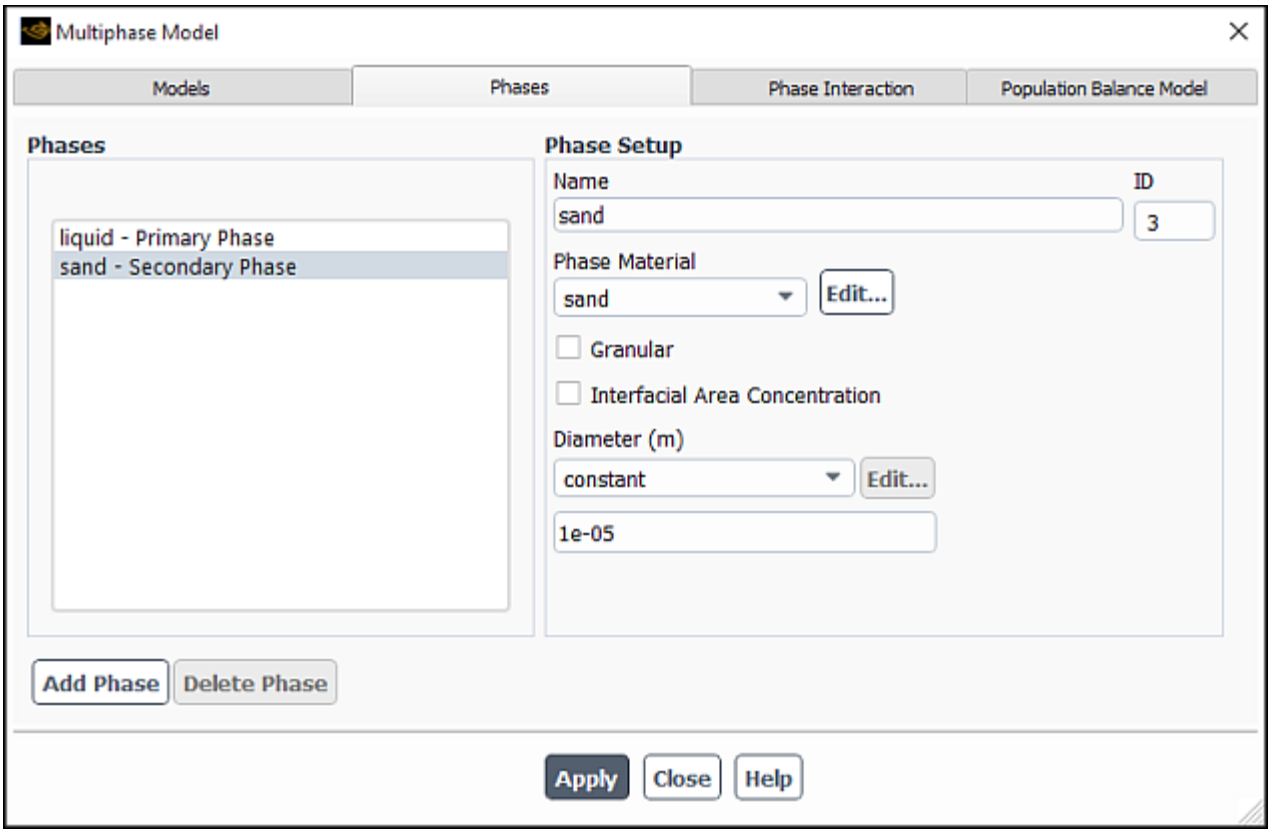
26.4.1.2. Defining a Non-Granular Secondary Phase

To define a non-granular (that is, liquid or vapor) secondary phase in a mixture multiphase calculation, go to the **Phases** tab of the **Multiphase Model** dialog box and perform the following steps:

1. Select the phase (for example, **phase-2**) in the **Phases** list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.46: Defining the Secondary Phase for the Mixture Model



2. In the **Phase Setup** group box, enter a **Name** for the phase (for example, **sand**).

3. Specify which material the phase contains by choosing the appropriate material in the **Phase Material** drop-down list.

4. Define the material properties for the **Phase Material**, following the same procedure you used to set the material properties for the primary phase (see Defining the Primary Phase (p. 3009)). For a particulate phase (which must be placed in the fluid materials category, as mentioned in Steps for Using a Multiphase Model (p. 2915)), you need to specify only the density; you can ignore the values for the other properties, since they will not be used.

5. Specify the **Diameter** of the bubbles, droplets, or particles of this phase (d_p in Equation 14.166 in the Theory Guide). You can specify a constant value, or use a user-defined function. See the Fluent Customization Manual for details about user-defined functions. Note that when you are using the mixture model without slip velocity, this input is a characteristic length scale.

6. Click **Apply**.

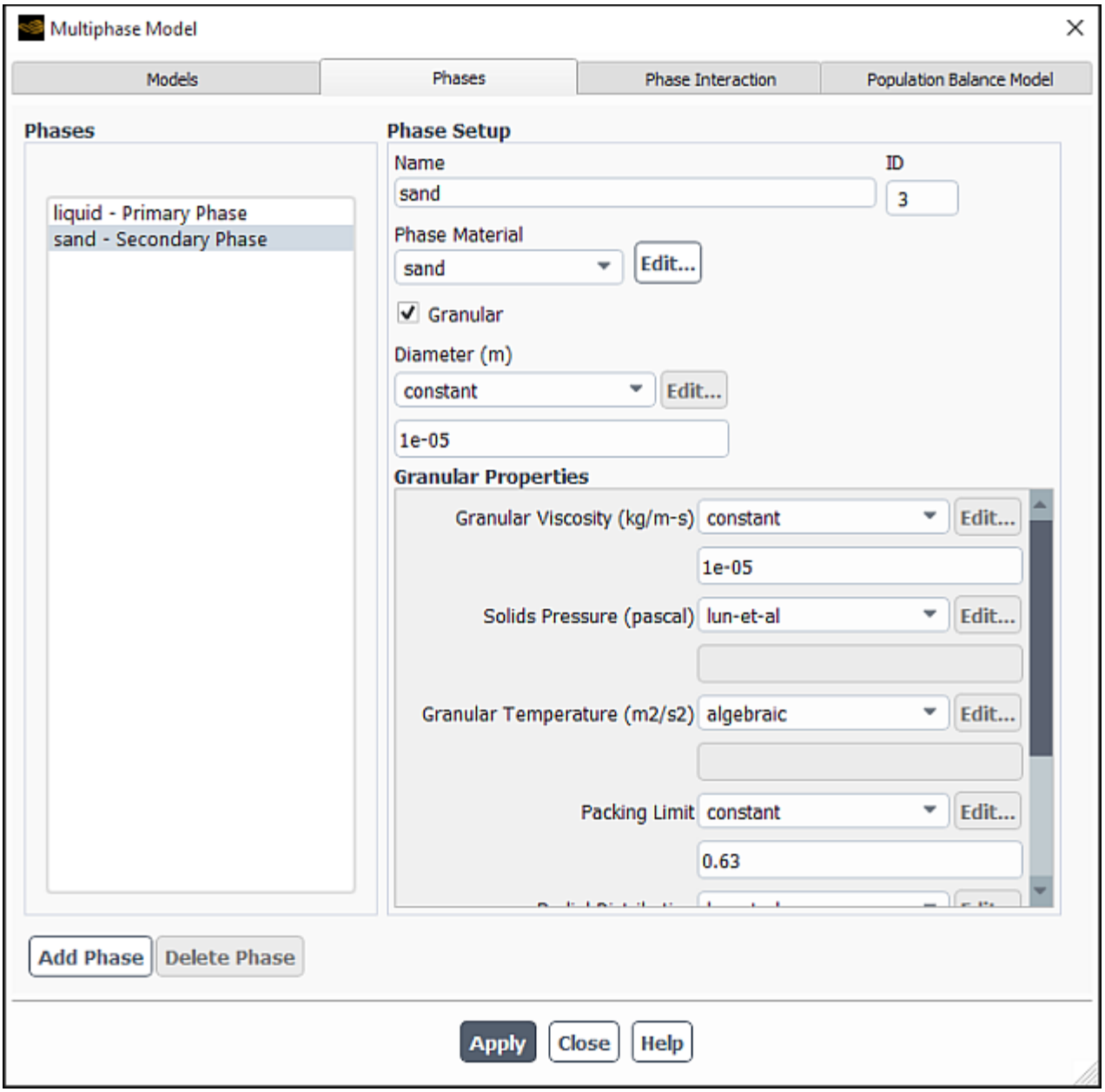
26.4.1.3. Defining a Granular Secondary Phase

To define a granular (that is, particulate) secondary phase in a mixture model multiphase calculation, perform the following steps:

1. Select the phase (for example, **phase-2**) in the **Phases** list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.47: Defining a Granular Phase in the Mixture Model



2. In the **Phase Setup** group box, enter a **Name** for the phase (for example, **sand**).

3. Specify which material the phase contains by choosing the appropriate material in the **Phase Material** drop-down list.

4. Define the material properties for the **Phase Material**, following the same procedure you used to set the material properties for the primary phase (see Defining the Primary Phase (p. 3009)). For a granular phase (which must be placed in the fluid materials category, as mentioned in

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Steps for Using a Multiphase Model (p. 2915)), you need to specify only the density; you can ignore the values for the other properties, since they will not be used.

Important:

Note that all properties for granular flows can be defined by user-defined functions (UDFs).

See the Fluent Customization Manual for details about user-defined functions.

5. Enable the **Granular** option.

6. Specify the **Diameter** of the particles. You can select **constant** in the drop-down list and specify a constant value, or select **user-defined** to use a user-defined function. See the Fluent Customization Manual for details about user-defined functions.

7. In the **Granular Properties** group box, specify the following properties of the particles of this phase:

Granular Viscosity

specifies the method for computing the kinetic ($\mu_{s,kin}$) and collisional ($\mu_{s,col}$) components of the granular viscosity (Equation 14.139 in the *Fluent Theory Guide*). Selecting **constant**, **syamlal-obrien** (Equation 14.141 in the *Fluent Theory Guide*), or **gidaspow** (Equation 14.142 in the *Fluent Theory Guide*) will use these expressions for the kinetic portion of the viscosity and will calculate the collisional portion of the viscosity from Equation 14.140 in the *Fluent Theory Guide*. Alternatively, you can select **user-defined** to use a user-defined function. Note that if you select **user-defined**, your user-defined function must include both the kinetic portion and the collisional portion of the viscosity in the value it returns.

Frictional Pressure

specifies the pressure gradient term, $\nabla P_{friction}$, in the granular-phase momentum equation. Choose **none** to exclude frictional pressure from your calculation, **johnson-et-al** to apply Equation 14.371 in the Theory Guide, **syamlal-obrien** to apply Equation 14.268 in the Theory Guide, **based-ktgf**, where the frictional pressure is defined by the kinetic theory [38] (p. 5715). The solids pressure tends to a large value near the packing limit, depending on the model selected for the radial distribution function. You must hook a user-defined function when selecting the **user-defined** option. See the Fluent Customization Manual for information on hooking a UDF.

Frictional Modulus

is defined as

$$G=\frac{\partial P_{friction}}{\partial \alpha_{friction}} \tag{26.17}$$

with $G \geq 0$, which is the **derived** option. You can also specify a **user-defined** function for the frictional modulus.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Friction Packing Limit

specifies a threshold volume fraction at which the frictional regime becomes dominant. The default value is 0.61.

Granular Temperature

specifies temperature for the solids phase and is proportional to the kinetic energy of the random motion of the particles. Choose either the **algebraic**, the **constant**, or **user-defined** option.

Solids Pressure

specifies the pressure gradient term, ∇p_s, in the granular-phase momentum equation. Choose either the **lun-et-al**, the **syamlal-obrien**, the **ma-ahmadi**, or the **user-defined** option.

Radial Distribution

specifies a correction factor that modifies the probability of collisions between grains when the solid granular phase becomes dense. Choose either the **lun-et-al**, the **syamlal-obrien**, the **ma-ahmadi**, the **arastoopour**, or a **user-defined** option.

Elasticity Modulus

is defined as

$$G=\frac{\partial P_s}{\partial \alpha_s} \tag{26.18}$$

with $G \geq 0$.

Choose either the **derived** or **user-defined** options.

Packing Limit

specifies the maximum volume fraction for the granular phase. For monodispersed spheres, the packing limit is about 0.63, which is the default value in Ansys Fluent. In polydispersed cases, however, smaller spheres can fill the small gaps between larger spheres, so you may need to increase the maximum packing limit.

8. Click **Apply**.

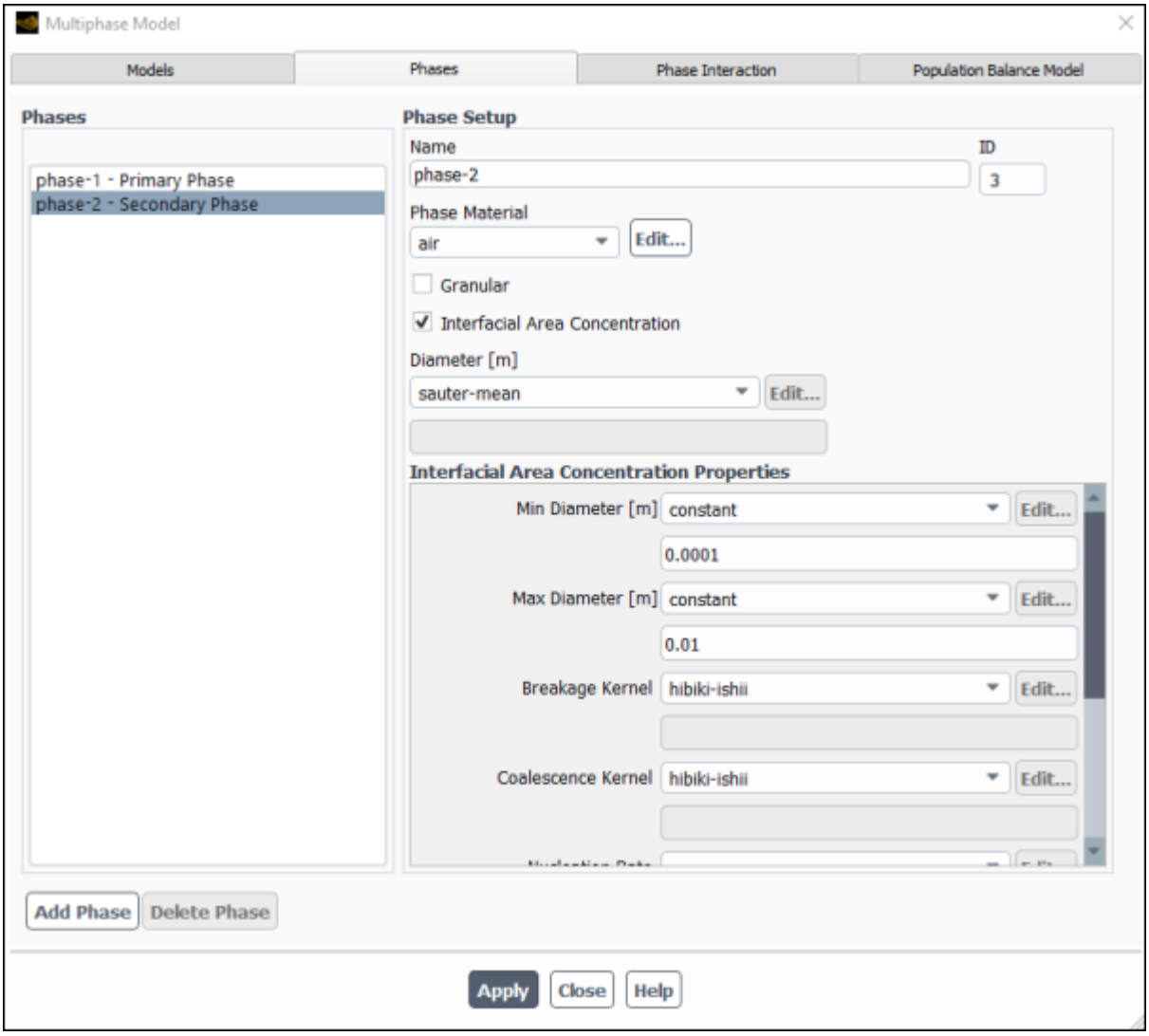
26.4.1.4. Defining the Interfacial Area Concentration via the Transport Equation

If you want to solve the transport equation for interfacial area concentration of the secondary phase in the mixture model (Interfacial Area Concentration in the *Fluent Theory Guide*, perform the following steps:

1. In the **Phases** tab of the **Multiphase Model** dialog box, select the phase (for example, **phase-2**) from the **Phases** list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.48: The Multiphase Model Dialog Box Displaying the Interfacial Area Concentration Settings



2. In the **Phase Setup** group box, enter a **Name** for the phase (for example, `water-vapor`).

3. Specify which material the phase contains by choosing the appropriate material in the **Phase Material** drop-down list.

4. Define the material properties for the **Phase Material**.

5. Enable the **Interfacial Area Concentration** option. Make sure the **Granular** option is disabled for the **Interfacial Area Concentration** option to be visible in the interface.

6. Specify the **Diameter** of the particles or bubbles. You can select **constant** in the drop-down list and specify a constant value, or select **user-defined** to use a user-defined function. See the Fluent Customization Manual for details about user-defined functions. The **Diameter** recommended setting is **sauter-mean**, allowing for the effects of the interfacial area concentration values to be considered for mass, momentum and heat transfer across the interface between phases.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. In the **Interfacial Area Concentration Properties** group box, specify the following properties of the particles of this phase:

Min/Max Diameter

are the limits of the bubble diameters.

Coalescence Kernel, Breakage Kernel

allow you to specify the coalescence and breakage kernels. You can select **none**, **constant**, **hibiki-ishii**, **ishii-kim**, **yao-morel**, or **user-defined**. The three options, **hibiki-ishii**, **ishii-kim** and **yao-morel** are described in detail in Interfacial Area Concentration in the Theory Guide.

In addition to specifying the **hibiki-ishii**, **ishii-kim**, and **yao-morel** as the coalescence and breakage kernels, you can also tune the properties of the three models by using the `/define/phases/iac-expert/hibiki-ishii-model`, `/define/phases/iac-expert/ishii-kim-model`, and `/define/phases/iac-expert/yao-morel-model` text commands.

For each of the three models you can specify the parameters listed in Table 26.12: Parameters for the Coalescence and Breakage Kernels (p. 3044)

Table 26.12: Parameters for the Coalescence and Breakage Kernels

Hibiki-Ishii	Ishii-Kim Model	Yao-Morel Model
`Coefficient Gamma_c`	`Coefficient Crc`	`Coefficient K_c1`
`Coefficient K_c`	`Coefficient Cwe`	`Coefficient K_c3`
`Coefficient Gamma_b`	`Coefficient C`	`Coefficient K_b1`
`Coefficient K_b`	`Coefficient Cti`	`alpha_max`
`alpha_max`	`alpha_max`	

These values are discussed in greater detail in Interfacial Area Concentration in the Theory Guide.

Nucleation Rate

is a source term for the interfacial area concentration that models the rate of formation of the dispersed phase. You can choose from **constant** or **user-defined**.

Surface Tension

specifies the surface tension at the liquid-air interface. You can select either the **hibiki-ishii** or the **ishii-kim** model.

Critical Weber Number

will need to be specified if you selected **ishii-kim** or **yao-morel** for the **Breakage Kernel**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Dissipation Function

gives you the option to choose the formula which calculates the dissipation rate used in the **hibiki-ishii** and **ishii-kim** models. You can choose amongst **constant**, **wu-ishii-kim**, **fluent-ke**, and **user-defined** for the dissipation function.

The **wu-ishii-kim** option uses a simple algebraic correlation for ε:

$$\varepsilon = f_{TW}\left(1/\ 2D_h\right)v_m^3 \tag{26.19}$$

where

$$f_{TW} = \frac{0.316}{\left[(1-\alpha)Re_m\right]^{0.25}}$$

and

$$Re_m = \frac{\rho_m v_m D_h}{\mu_m}$$

where ρ_m, v_m, μ_m, and D_h are the mixture density, mixture velocity, mixture molecular viscosity, and hydraulic diameter of the flow path.

When you select the **wu-ishii-kim** model, you will set an additional input for **Hydraulic Diameter**.

Hydraulic Diameter

is the value used in Equation 26.19 (p. 3045), should you use the **wu-ishii-kim** formulation.

8. Click **Apply**.

Note:

When solving a steady-state problem, the preferred setting for the **Under-Relaxation Factor** is 1.0, as the interfacial area equation for the boiling models is currently under-relaxed using a locally defined pseudo time step. If you want extra explicit under-relaxation, you may set the value of the **Under-Relaxation Factor** to less than one, this may be done only in case of serious convergence problems with the interfacial area transport equation. To improve convergence you can switch to a pseudo time step for the interfacial area concentration only, using the `define/phases/iac-expert/iac-pseudo-time-step` text command and set the local pseudo time step to less than 1.

26.4.1.5. Defining the Algebraic Interfacial Area Concentration

If you have chosen not to solve the transport equation for Interfacial Area Concentration as described in Defining the Interfacial Area Concentration via the Transport Equation (p. 3042), then you can select an algebraic model to estimate the interfacial area from the secondary phase diameter specified in the **Phases** tab of the **Multiphase Model** dialog box. To choose an algebraic interfacial area model perform these steps.

1. In the **Multiphase Model** dialog box, go to the **Phase Interaction** > **Interfacial Area** tab.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Select the desired algebraic model from the **Interfacial Area** drop-down list:

   ia-symmetric

   considers both the primary and secondary phase volume fractions in estimating the interfacial area.

   ia-particle

   considers the secondary phase volume fraction in estimating the interfacial area.

   ia-gradient

   considers the volume fraction gradient at the interface between two phases in estimating the interfacial area.

   ia-hybrid

   uses different algebraic interfacial area models for different regimes. This item is available only for the regime flow modeling. See Using the Flow Regime Modeling (p. 3048) for more information about this method.

   user-defined

   See `DEFINE_EXCHANGE_PROPERTY` in the Fluent Customization Manual.

In addition to these models, Ansys Fluent provides the Gradient-Symmetric model, which is available via the text user interface. You can use the following text commands to enable this model:

```
solve/set/multiphase-numerics/heat-mass-transfer/area-density/ia-grad-sym?

switch to ia-grad-symmetric option? [yes] yes
```

See Algebraic Models in the *Fluent Theory Guide* for details about the theory behind algebraic interfacial area models.

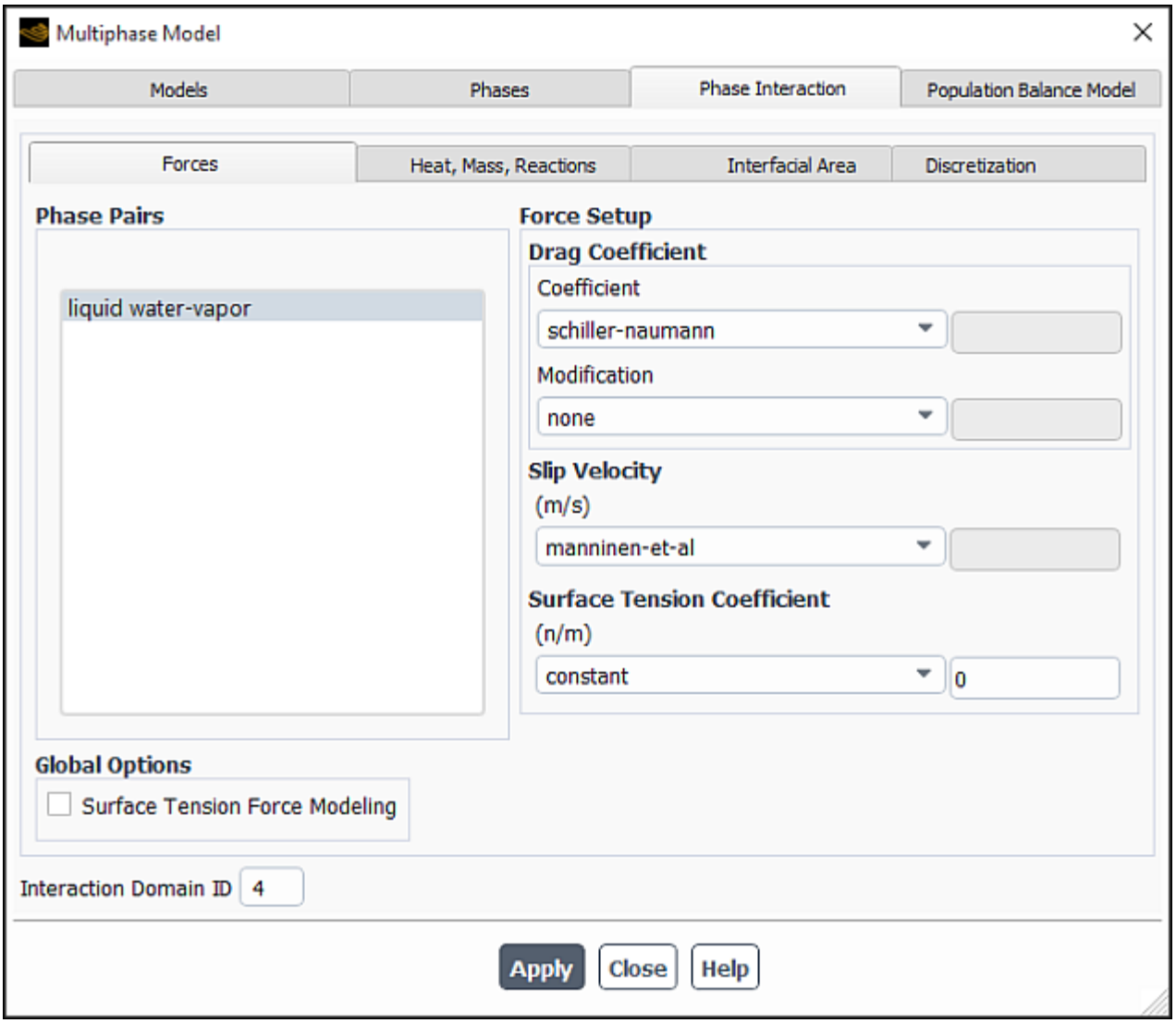
26.4.1.6. Defining Drag Between Phases

For mixture multiphase flows with slip velocity, you can specify the drag function to be used in the calculation.

To specify drag laws, go to the **Phase Interaction** > **Forces** tab in the **Multiphase Model** dialog box (Figure 26.49: The Multiphase Model Dialog Box for the Mixture Model (Forces Tab) (p. 3047)) and make the appropriate selections in the **Drag Coefficient** group box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.49: The Multiphase Model Dialog Box for the Mixture Model (Forces Tab)



The drag coefficients available here are a subset of those discussed in Specifying the Drag Function (p. 3075).

In addition, for cases that involve flow regime modeling, the **drag-hybrid** method is available as described in Using the Flow Regime Modeling (p. 3048).

Relative (Slip) Velocity and the Drift Velocity in the Theory Guide provides theoretical background information.

26.4.1.7. Defining the Slip Velocity

If you are solving for slip velocities during the mixture calculation, you can modify the slip velocity definition in the **Phase Interaction** > **Forces** tab in the **Multiphase Model** dialog box (Figure 26.49: The Multiphase Model Dialog Box for the Mixture Model (Forces Tab) (p. 3047)).

Under **Slip Velocity**, you can specify the slip velocity function for each secondary phase with respect to the primary phase by choosing the appropriate item in the adjacent drop-down list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Select **manninen-et-al** (the default) to use the algebraic slip method of Manninen et al. [95] (p. 5718), described in Relative (Slip) Velocity and the Drift Velocity in the Theory Guide.
- Select **none** if the secondary phase has the same velocity as the primary phase (that is, no slip velocity).
- Select **user-defined** to use a user-defined function for the slip velocity. See the Fluent Customization Manual for details.

26.4.1.8. Including Surface Tension and Wall Adhesion Effects

You can include the effects of surface tension along the interface between one or more pairs of phases as described in Including Surface Tension and Adhesion Effects (p. 3011). For more information about the theoretical background, see Surface Tension and Adhesion in the *Fluent Theory Guide*.

Note:

- The surface tension option in recommended for cases with sharp/dispersed interface modeling type.
- Jump adhesion is not supported with the Mixture Multiphase model.

26.4.2. Including Mixture Drift Force

If you are solving for slip velocity and using one of the turbulence models in your Mixture multiphase calculation, you can also include the effects of the slip velocity on the momentum and turbulence equations. To include these effects, enable **Mixture Drift Force** in the Viscous Model Dialog Box (p. 4715). The turbulent dispersion effect will be included in the slip velocity equation (Equation 14.135 in the *Fluent Theory Guide*). Note that the inclusion of these terms can slow down convergence noticeably. If you are looking for additional accuracy, you may want to compute a solution first without these sources, and then continue the calculation with these terms included. In most cases these terms can be neglected.

26.4.3. Using the Flow Regime Modeling

26.4.3.1. Limitations for the Flow Regime Modeling

The following limitation exists with the flow regime modeling:

- The flow regime modeling is supported only with the algebraic slip model, which is valid for low Stokes numbers.

26.4.3.2. Setting the Flow Regime Modeling

To use the flow regime modeling framework, follow the steps outlined below. Only the steps that are pertinent to the flow regime modeling are listed here. For the theory behind the flow regime modeling, see Flow Regime Modeling in the *Fluent Theory Guide*.

1. In the **Models** tab of the **Multiphase Model** dialog box, set up the following models and options.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setup → Models → Multiphase Edit...

a. In the **Model** group box, enable **Mixture**.

b. In the **Model Parameters** group box, enable **Slip Velocity**.

Once you have selected **Slip Velocity**, the **Flow Regime Modeling** option appears in the **Model Parameters** group box.

c. Enable **Flow Regime Modeling**.

Once you have enables **Flow Regime Modeling**, Ansys Fluent automatically enables the following options:

- **Sharp/Dispersed** interface modeling type in the **Options** group box
- **Regime Based Discretization** in the **Interface Modeling Options** dialog box.

Note:

Although the **Dispersed** interface modeling type is available, using this option in your simulation may yield less accurate results for some analyses. For instance, the free surface evolution may not be captured well with this option.

---

d. Click the **Interface Modeling Options...** button and in the **Interface Modeling Options** dialog box, specify the appropriate settings.

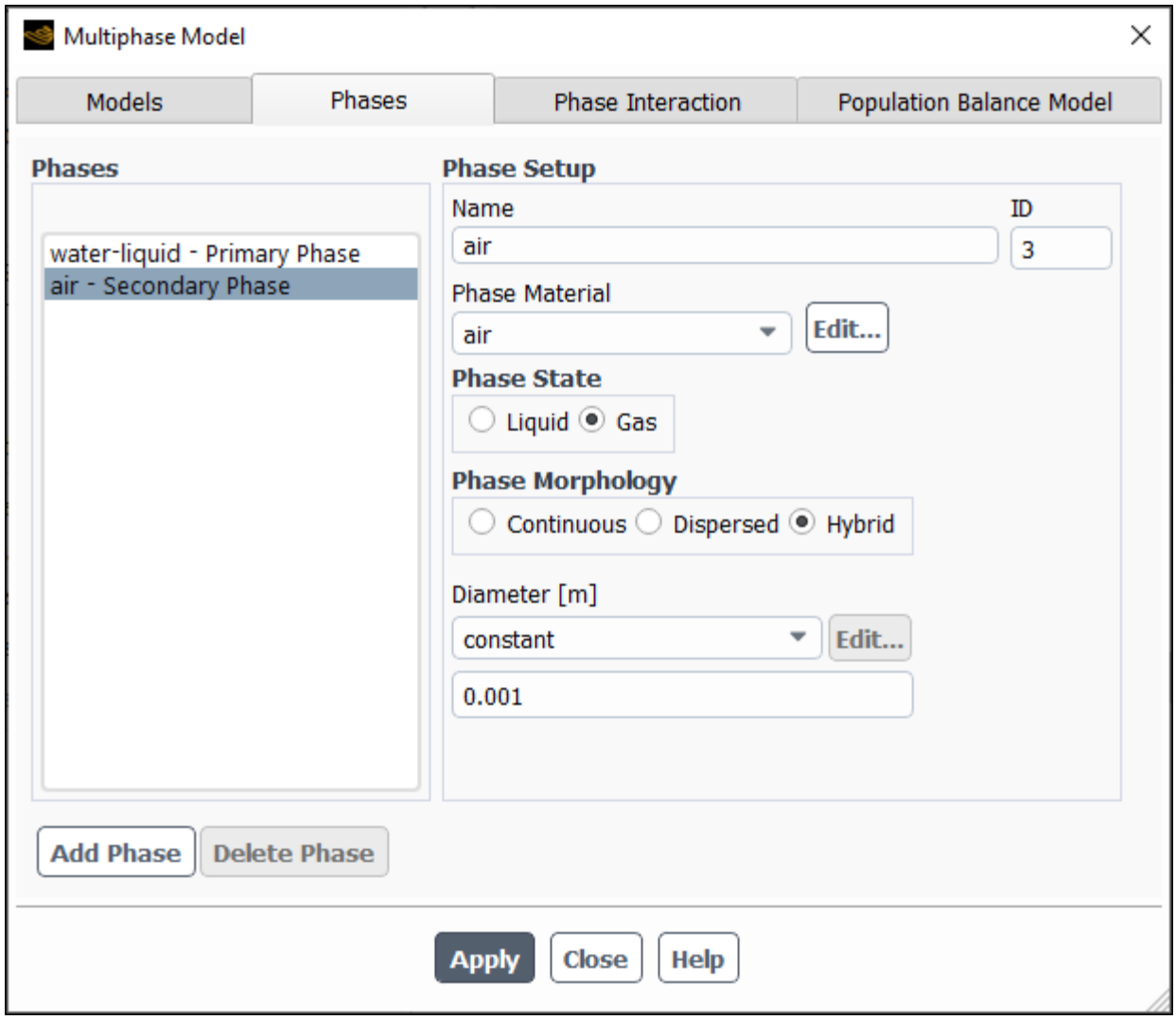
The default values for both the **Bubbly/Droplet Flow Limiter** and **Free Surface Flow Limiter** are 1.5, which should be suitable for most cases. You can modify these values as needed for your analysis. See The Compressive Scheme and Interface-Model-based Variants in the *Fluent Theory Guide* for more information about the slope limiters.

2. Click **Apply**.

3. In the **Phases** tab, define the phases for your simulation.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.50: The Multiphase Model Dialog Box - Hybrid Phase Morphology



a. Define the primary and secondary phases as appropriate.

b. For each phase, select either **Liquid** or **Gas** from the **Phase State** group box.

c. Select the **Phase Morphology** from the following options:

   - **Continuous**
   - **Dispersed** (available for secondary phases only)
   - **Hybrid**

   See Flow Regime Modeling in the *Fluent Theory Guide* for details about phase morphologies.

d. For phase with either the **Dispersed** or **Hybrid** morphology, specify **Diameter** of the particles, droplets, or bubbles (depending on the flow regime).

4. Click **Apply**.

5. In the **Phase Interaction** tab, define the interaction between the phases.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For different phase pair interactions, different methods and defaults are available, depending on the phase morphologies. The possible combinations of the phase-pair morphologies are:

- Continuous-Continuous
- Continuous-Dispersed
- Dispersed-Dispersed
- Hybrid-Continuous
- Hybrid-Dispersed
- Hybrid-Hybrid

The methods and default settings available for each phase pair morphology are described below.

a. In the **Forces** tab, set the interfacial forces between the phase pairs.

   i. Select the drag coefficient and slip velocity from the **Drag Coefficient** and **Slip Velocity** drop-down lists.

   The default settings for the drag coefficient and slip velocity for different phase-pair morphology combinations are listed in the table below.

Phase-Pair Morphology	Drag Coefficient	Slip Velocity
Continuous-Continuous	**zero-resistance**	**none**
Continuous-Dispersed	**schiller-naumann** (default) **symmetric** **morsi-alexander** **ishii-zuber** **none** **constant** **user-defined**	**manninen-et-al** (default) **none** **user-defined**
Dispersed-Dispersed	-	-
Hybrid-Continuous Hybrid-Dispersed Hybrid-Hybrid	**drag-hybrid**	**manninen-et-al** (default) **none** **user-defined**

   Note the following:

   - The **zero-resistance** method (available only for Continuous-Continuous systems) is equivalent to an infinite drag, which forces the velocities of the continuous phases to

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

be equal. See Phase Pair Interaction for Hybrid Morphology in the *Fluent Theory Guide* for more information.

- If you do not want to consider the drag force or slip velocity, choose **none**.
- The **drag-hybrid** method uses different drag coefficient methods for different flow regimes. You can select other than the default drag coefficient methods as further described. For more information, see Flow Regime Modeling in the *Fluent Theory Guide*.

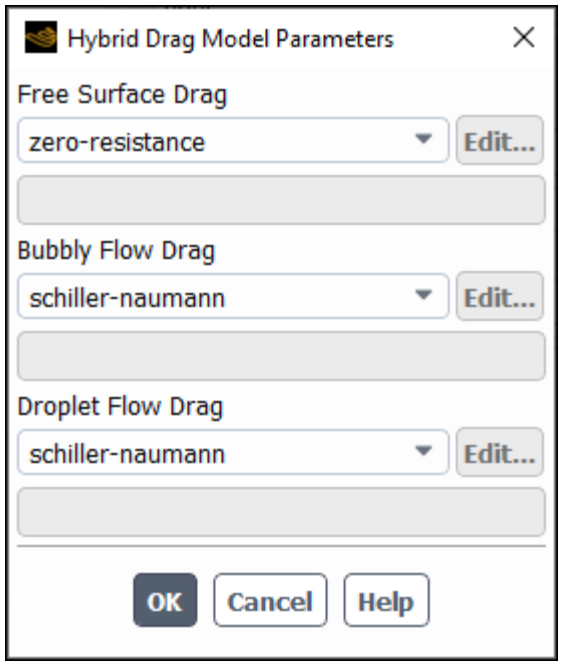
For more information about the drag coefficients, see Defining Drag Between Phases (p. 3046).

For more information about the slip velocity methods, see Defining the Slip Velocity (p. 3047).

ii. For the **drag-hybrid** method, you can modify the selection of the drag coefficients for the free surface, bubbly, and droplet flow regimes.

A. Click **Edit...** next to the selected **drag-hybrid** drag coefficient.

Figure 26.51: The Hybrid Drag Model Parameters Dialog Box



B. In the **Hybrid Drag Model Parameters** dialog box that opens, select the appropriate draft coefficient methods from the **Free Surface Drag**, **Bubbly Flow Drag**, and **Droplet Flow Drag** drop-down lists.

The available methods for different flow regimes are listed in the table below.

Flow Regime	Drag Coefficient
Free Surface Drag	**zero-resistance**
Bubbly Flow Drag	**schiller-naumann** (default)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Flow Regime	Drag Coefficient
Droplet Flow Drag	**symmetric** **morsi-alexander** **ishii-zuber** **constant** **user-defined**

iii. Click **Apply**.

b. In the **Interfacial Area** tab, define the interfacial area parameters. See Algebraic Interfacial Area Density Method for Flow Regime Blending in the *Fluent Theory Guide* for more information.

i. For each pair of phases, select the interfacial area method.

The available methods for different phase pair morphologies are listed in the table below.

Phase-Pair Morphology	Interfacial Area Methods
Continuous-Continuous	**ia-gradient** **user-defined**
Continuous-Dispersed	**ia-symmetric** (default) **ia-particle** **user-defined**
Dispersed-Dispersed	**ia-symmetric** (default) **user-defined**
Hybrid-Continuous Hybrid-Dispersed Hybrid-Hybrid	**ia-hybrid**

For information about these methods, see Algebraic Models in the *Fluent Theory Guide*.

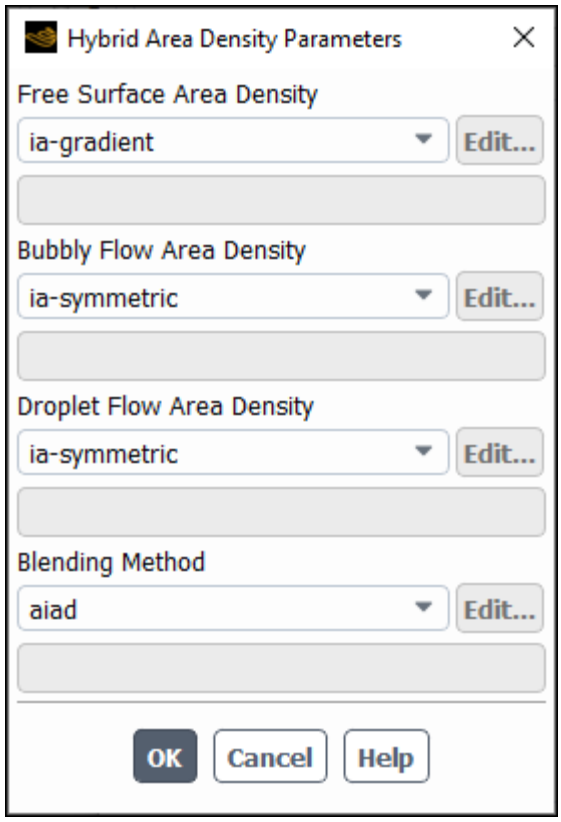
For the theory behind the **ia-hybrid** method, see Algebraic Interfacial Area Density Method for Flow Regime Blending in the *Fluent Theory Guide*.

ii. For the **ia-hybrid** method, you can modify the selection of the area density parameters for the free surface, bubbly, and droplet flow regimes.

A. Click **Edit...** next to the selected **ia-hybrid** interfacial area.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.52: The Hybrid Area Density Parameters Dialog Box



B. In the **Hybrid Area Density Parameters** dialog box that opens, select the appropriate area density methods from the **Free Surface Area Drag**, **Bubbly Flow Area Density**, and **Droplet Flow Area Density** drop-down lists.

The available methods for different flow regimes are listed in the table below.

Flow Regime	Area Density
Free Surface Area Density	**ia-gradient** **user-defined**
Blending Method	**aiad** **user-defined**

C. Select the **Blending Method** from the following options:

aiad

is the algebraic interfacial area density (AIAD) method described in Algebraic Interfacial Area Density Method for Flow Regime Blending in the *Fluent Theory Guide*.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

user-defined

allows you to provide your own blending factors for the flow regime modeling. Refer to `DEFINE_VECTOR_EXCHANGE_PROPERTY` in the *Fluent Customization Manual* for details.

Note:

The default values for the AIAD method parameters are suitable for most simulations. If needed, you can adjust these values using the following text commands:

`define/models/multiphase/flow-regime-modeling/aiad-parameters/critical-vf`

Specifies the critical volume fraction of either gas or liquid for calculating the blending factors for the bubbly and droplet flows ($\alpha_{i,crit}$ and $\alpha_{j,crit}$ in Equation 14.178 in the *Fluent Theory Guide*).

`define/models/multiphase/flow-regime-modeling/aiad-parameters/delta-grad`

Specifies the critical gradient fraction for determining the intermediate transition width for the free surface blending factor (β in Equation 14.183 in the *Fluent Theory Guide*).

`define/models/multiphase/flow-regime-modeling/aiad-parameters/delta-vf`

Specifies the volume fraction width for the intermediate transition (δ_α in Equation 14.178 in the *Fluent Theory Guide*).

`define/models/multiphase/flow-regime-modeling/aiad-parameters/ncells-fs`

Specifies the cell number across the interface for determining the interfacial width for free surface flow (n in Equation 14.182 in the *Fluent Theory Guide*).

See Alphabetical Listing of Field Variables and Their Definitions (p. 4205) for their definitions.

---

c. Click **Apply**.

26.4.3.3. Solution Strategies

Since the algebraic slip model is applicable only to low Stokes number (~1) regimes, you should examine the postprocessing results of the **Relaxation Time** field variable, which indicates the validity of the algebraic slip model.

The slip velocity treatment is explicit; therefore, lowering the slip velocity under-relaxation factor can improve convergence.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.4.3.4. Postprocessing for Flow Regime Modeling Simulations

The following additional postprocessing variables will become available for postprocessing under the **Phase Interaction..**. category:

- **Slip X-velocity**
- **Slip Y-velocity**
- **Slip Z-velocity** (3D only)
- **Flow Regime Blending Factors Fcc**, **Fcd**, **Fdc**, and **Fdd**
- **Relaxation Time**

See Field Function Definitions (p. 4167) for their definitions.

26.4.4. Using the Singhal et al. Expert Cavitation Model

In addition to the **Schnerr-Sauer** and **Zwart-Gerber-Belamri** mechanisms described in Cavitation Mechanism (p. 2946), you can also use the Singhal et al. expert model for special mixture multiphase cases to model the effects of cavitation. For the theory behind this model, refer to Expert Cavitation Model in the *Fluent Theory Guide*.

The procedure for setting up the Singhal et al. cavitation model is presented below:

1. Issue the following text command:

```
solve/set/advanced/singhal-et-al-cavitation-model
Enable? [no] yes
```

2. Enable the **Singhal-Et-Al Cavitation Model** option that now appears in the **Multiphase Model** dialog box under the **Phase Interaction** > **Heat, Mass, Reactions** tab.
3. Specify the following parameters to be used in the calculation of mass transfer due to cavitation:
   - **Vaporization Pressure**: The default value of p_{sat} is 3540 Pa, the vaporization pressure for water at ambient temperature. Note that p_{sat} and the surface tension are properties of the liquid, depending mainly on temperature.
   - **Surface Tension Coefficient**
   - **Non-Condensable Gas Mass Fraction**: The mass fraction of dissolved gases, which depends on the purity of the liquid.

When multiple species are included in one or more secondary phases, or the heat transfer due to phase change must be taken into account, the mass transfer mechanism must be defined *before* enabling the cavitation model. It may be noted, however, that for cavitation problems, at least *two* mass transfer mechanisms are defined:

- mass transfer from liquid to vapor.
- mass transfer from vapor to liquid.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To disable Singhal et al. cavitation model:

1. Disable the **Singhal-Et-Al Cavitation Model** option in the **Heat, Mass, Reaction** tab.
2. Issue the following text command:

```
solve/set/advanced/singhal-et-al-cavitation-model
Enable? [yes] no
```

26.4.5. Including Semi-Mechanistic Boiling

26.4.5.1. Overview and Limitations for the Semi-Mechanistic Boiling Model

The semi-mechanistic boiling model can be used only with the following models and conditions:

- Mixture multiphase
- Turbulent kinetic energy-based models (such as k-ε or k-ω)
- Activated gravity along with zero operating density
- Surface tension coefficient (specified in the **Phase Interaction** > **Forces** tab of the **Multiphase Model** dialog box)
- Boiling walls with either temperature or conjugate heat transfer boundary conditions

> **Note:**
>
> If your case has a boiling wall with heat flux boundary condition, you can add a small thickness of solid and apply heat flux to the solid boundary. Thus you will be able to apply heat flux boundary condition indirectly at the boiling wall through conjugate heat transfer boundary condition.

- Binary mixtures with the fixed mixture composition only (pseudo-fluid approach). All the properties and saturation data for the pseudo-fluid must be provided according to the mixture fixed composition.

For the theoretical background on the semi-mechanistic boiling model, see Semi-Mechanistic Boiling Model in the *Fluent Theory Guide*.

26.4.5.2. Using the Semi-Mechanistic Boiling Model

To use the semi-mechanistic boiling model, follow the steps below:

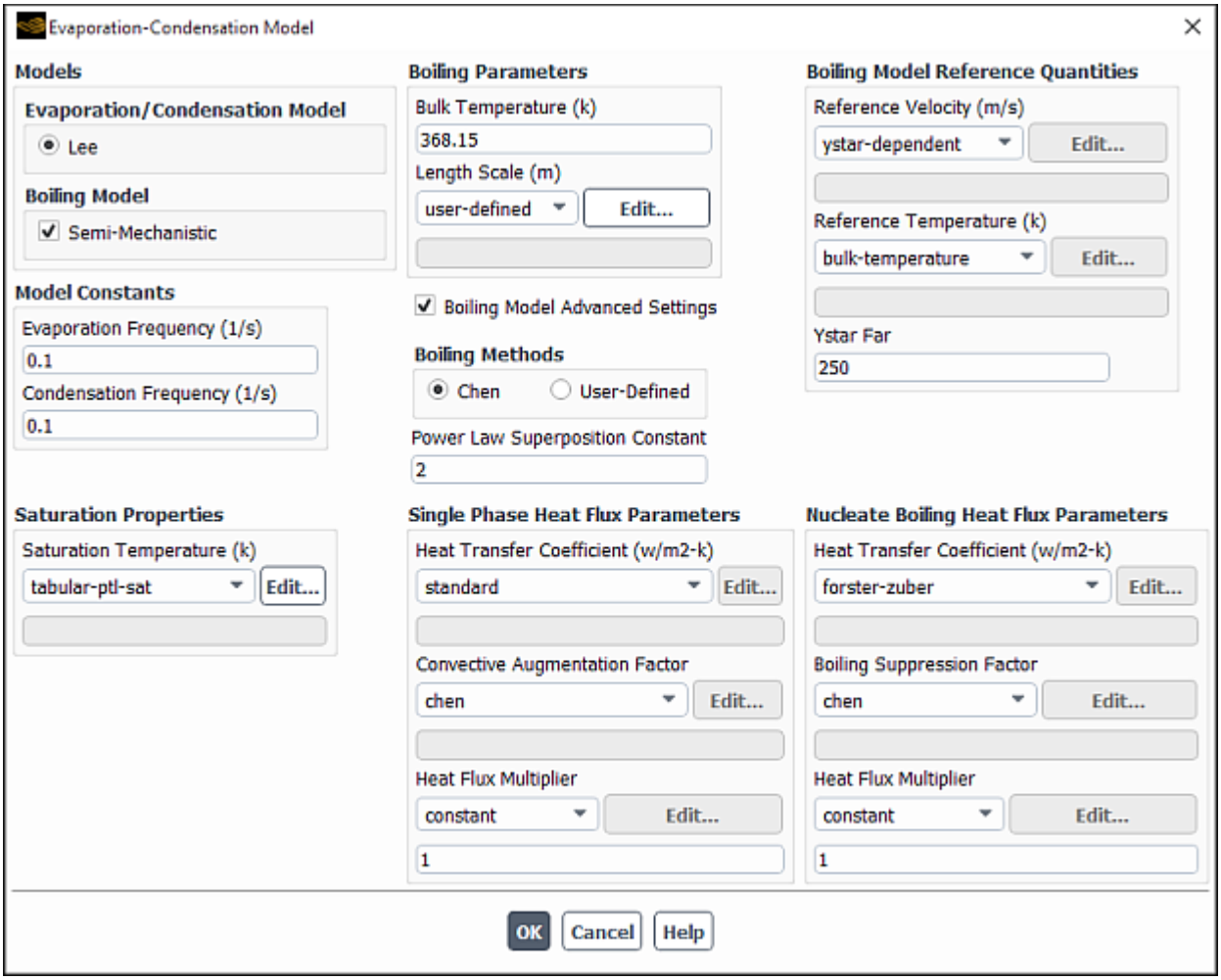
1. In the **Multiphase Model** dialog box under the **Phase Interaction** > **Heat, Mass, Reactions** tab, specify the **Number of Mass Transfer Mechanisms** and then select the liquid and vapor phases in the **From Phase** and **To Phase** drop-down lists, respectively.
2. Select **evaporation-condensation** from the **Mechanism** drop-down list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. In the **Evaporation-Condensation Model** dialog box that opens, select **Semi-Mechanistic** as the **Boiling Model**.

   The **Evaporation-Condensation Model** dialog box expands to reveal additional **Boiling Parameters**.

Figure 26.53: The Evaporation-Condensation Model Dialog Box



Note:

For the **Semi-Mechanistic** boiling model, the **Saturation Temperature** can be only specified using either the **tabular-pt-sat** or **tabular-ptl-sat** option.

4. In the **Boiling Parameters** group box, specify the following parameters:

Bulk Temperature

is a temperature of bulk fluid away from the wall. Typically, the inlet temperature of the coolant is used as the bulk temperature.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Length Scale

is used in the calculation of the reference Reynolds number. It is typically taken to be the hydraulic diameter.

5. To access additional boiling settings for advanced control, select **Boiling Model Advanced Settings** to reveal more options.

6. In the **Boiling Model Reference Quantities** group box, you can specify the following:

Reference Velocity

is the velocity of the fluid away from a heated wall at some reference location. This quantity is used to calculate the reference Reynolds number $Re_{l,ref}$ in Equation 14.639 in the *Fluent Theory Guide*. You can choose one of the following specification methods:

ystar-dependent

(default) calculates the reference velocity away from the boiling wall at $y^* = 250$ using standard wall functions for turbulence. Here, y^* is a normalized wall distance. You can adjust its value in the **Ystar Far** field.

constant

specifies a constant value for the reference velocity.

user-defined

uses a user-defined function for the reference velocity calculation.

Reference Temperature

is the temperature of fluid away from a heated wall at some reference location (T_{ref} in Equation 14.639 in the *Fluent Theory Guide*). Fluid properties used for the reference Reynolds number computation are calculated at the reference temperature. You can choose one of the following specification methods:

bulk-temperature

(default) takes the reference temperature from a user-defined **Bulk Temperature** value.

ystar-dependent

calculates the reference temperature at y^*=250 using standard wall functions for turbulence. In this case, a value specified for **Bulk Temperature** serves as the minimum reference temperature.

user-defined

uses a user-defined function for the reference temperature calculation.

7. In the **Boiling Methods** group box, you can select:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chen

(default) calculates the boiling heat flux as a weighted superposition of single-phase heat flux (forced convective) and nucleate boiling heat flux. See Semi-Mechanistic Boiling Model in the *Fluent Theory Guide* for details about this method.

User-Defined

uses a framework provided by Ansys Fluent to hook user-defined correlations for all of the relevant inputs.

8. Enter the value for the **Power Law Superposition Constant** (n in Equation 14.627 in the *Fluent Theory Guide*). This constant is used to blend single phase and nucleate boiling heat transfer coefficients based on asymptotic power law. The default value is 2. The typical range is from 1 to 3.

9. In the **Single Phase Heat Flux Parameters** group, you can set:

   - **Heat Transfer Coefficient**: is h_{sp} in Equation 14.628 in the *Fluent Theory Guide*. The following specifications methods are available:

     – **standard**: Uses a built-in treatment of the heat transfer coefficient for a liquid phase.

     – **constant**

     – **user-defined**

   - **Convective Augmentation Factor**: specifies the single-phase heat flux augmentation factor due to the bubble agitation. The following specification methods are available:

     – **chen**: (default) Uses the correlations proposed by Chen. See Semi-Mechanistic Boiling Model in the *Fluent Theory Guide* for more information.

     – **constant**

     – **user-defined**

     This input parameter is available only when **Chen** is selected in the **Boiling Methods** group box.

   - **Heat Flux Multiplier**: is an additional factor to adjust a single-phase heat flux (M_{sp} in Equation 14.626 in the *Fluent Theory Guide*). This factor can be used when the single phase-heat flux is not predicted well due to an under-developed turbulent flow, uncertainties in surface roughness, interaction of thermal and momentum boundary layers when starting from different points or other factors. You can choose a **constant** or **user-defined** specification method. The default value is 1.

10. In the **Nucleate Boiling Heat Flux Parameters** group, you can set:

    - **Heat Transfer Coefficient**: The specification methods are:

      – **forster-zuber**: Uses the nucleate boiling heat transfer coefficient proposed by Forster and Zuber (Equation 14.629 in the *Fluent Theory Guide*).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - **constant**
  - **user-defined**

- **Boiling Suppression Factor**: Accounts for the nucleate boiling suppression due to the effect of forced convection. The presence of bubbles due to local boiling at heated walls augments forced convection and suppresses nucleate boiling. The following specification methods are available:
  - **chen-steiner**: (default) Uses the modification proposed by Steiner (Equation 14.635 in the *Fluent Theory Guide* to Equation 14.637 in the *Fluent Theory Guide*). See Semi-Mechanistic Boiling Model in the *Fluent Theory Guide* for more information.
  - **chen**: Uses Chen's correlation $S=S_{fc}$ (Equation 14.637 in the *Fluent Theory Guide*). See Semi-Mechanistic Boiling Model in the *Fluent Theory Guide* for more information.
  - **constant**
  - **user-defined**

  This input parameter is available only when **Chen** is selected in the **Boiling Methods** group box.

- **Heat Flux Multiplier**: is an additional factor to adjust a nucleate boiling heat flux (M_{nb} in Equation 14.626 in the *Fluent Theory Guide*). Boiling correlations are obtained for a specific pressure and temperature range for certain materials. This additional heat flux multiplier allows you to tune in heat transfer coefficient when these correlations are used in generalized sense. You can choose a **constant** or **user-defined** specification method. The default value is 1.

11. Postprocess the solution results.

    When the calculation is complete, the following additional field variables are available in the **Wall Fluxes...** category.

    - **SMB Single Phase Heat Flux**
    - **SMB Nucleate Boiling Heat Flux**

You can use the `DEFINE_MASS_TR_PROPERTY` UDF to define input parameters for the semi-mechanistic boiling model. See `DEFINE_MASS_TR_PROPERTY` in the *Fluent Customization Manual* for details.

26.4.5.3. Cell Zone Specific Boiling

By default, boiling is enabled in all the cell zones. When boiling is expected only in certain zones, you can disable boiling in other zones to make the calculation faster. This can be done by clearing **Boiling Zone** in the **Fluid** dialog box (**Multiphase** tab).

26.4.5.4. Expert Options for the Semi-Mechanistic Boiling Model

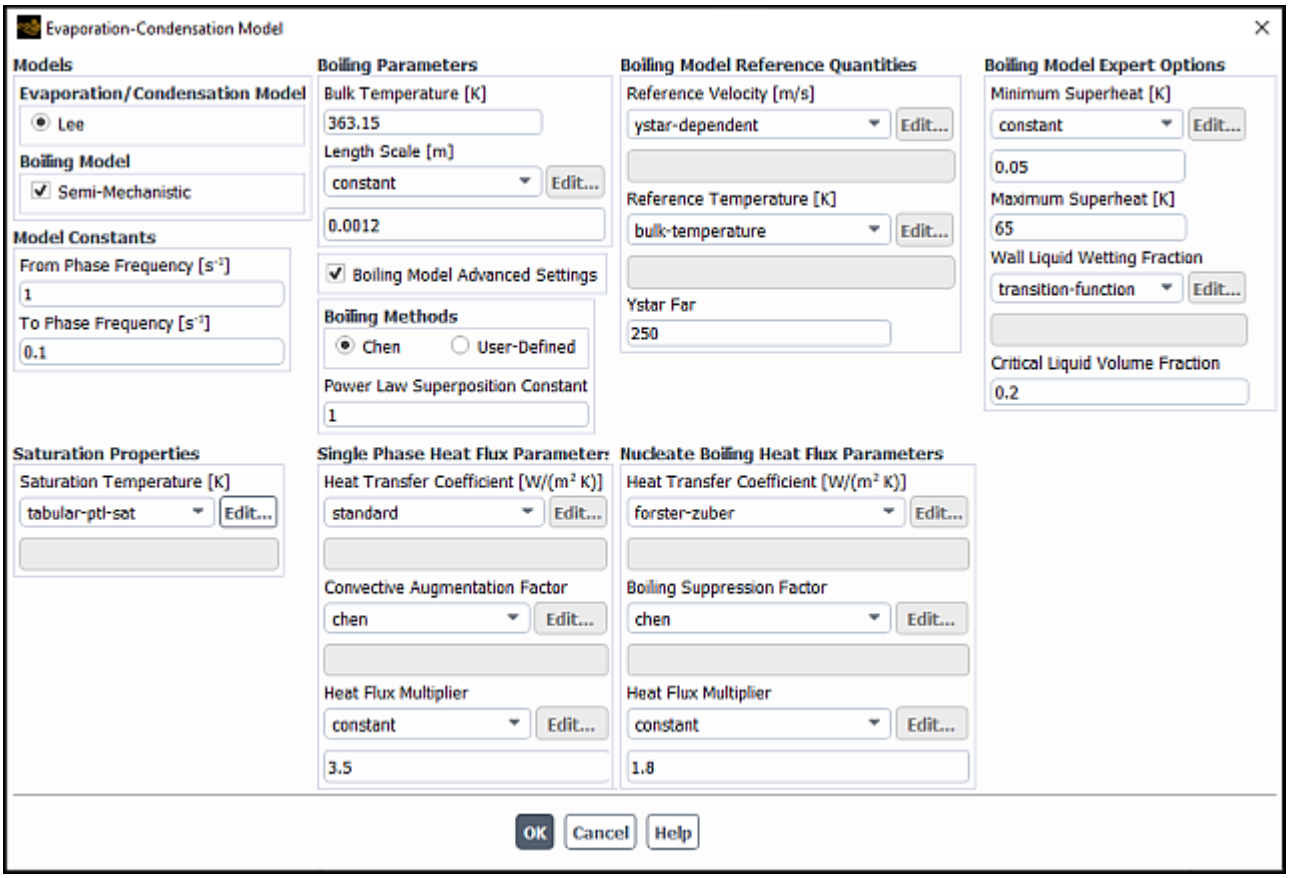
You can access expert options for the semi-mechanistic boiling model by issuing the following text commands:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
solve/set/multiphase-numerics/heat-mass-transfer/boiling/show-expert-
options?

show expert options in semi-mechanistic boiling model? [yes] yes
```

The evaporation-condensation model dialog box expands to show boiling model expert options.

Figure 26.54: Boiling Model Expert Options



You can adjust the following parameters:

Minimum Superheat

is the minimum superheat required for the onset of nucleate boiling. You can define the **Minimum Superheat** as:

- **constant**
- **hsu**
- **user-defined**

The **hsu** method uses the expression proposed by Hsu [67] (p. 5716):

$$\Delta T_{\min} = \frac{4\sigma T_{sat} h_l}{k_l \rho_{lv} L}\left[1+\sqrt{1+\frac{k_l \rho_{lv} L \Delta T}{2\sigma T_{sat} h_l}}\right]$$

$$\rho_{lv} = \rho_l - \rho_v$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where

$\Delta T_{\min}$ = minimum superheat

h_l = single phase heat transfer coefficient for liquid

k_l = thermal conductivity

The remaining notation is the same as in Semi-Mechanistic Boiling Model in the *Fluent Theory Guide*.

Maximum Superheat

is the maximum superheat used in the nucleate boiling heat flux expressions.

Wall Liquid Wetting Fraction

is the area fraction of the wall wetted by liquid. Since a wall may be completely wetted by liquid even if the neighboring cell liquid volume fraction is below 1, this is a crucial parameter for the proper estimation of the heat flux.

You can choose one of the following methods:

transition-function

(default) is the wetting fraction that controls the transition from a completely wet to a completely dry wall state. When the liquid volume fraction in the wall neighboring cells exceeds a value of the **Critical Liquid Volume Fraction** that you specified, the wall is assumed to be completely wetted by liquid. The wetting fraction f is calculated as:

$$f = \begin{cases} 1 - 0.5e^{-n(r-1)} & r > 1 \\ 0.5r^n & r \le 1 \end{cases}$$

$$r = \frac{\alpha_l}{\alpha_{l,cr}}$$

where

α_l = volume fraction of liquid

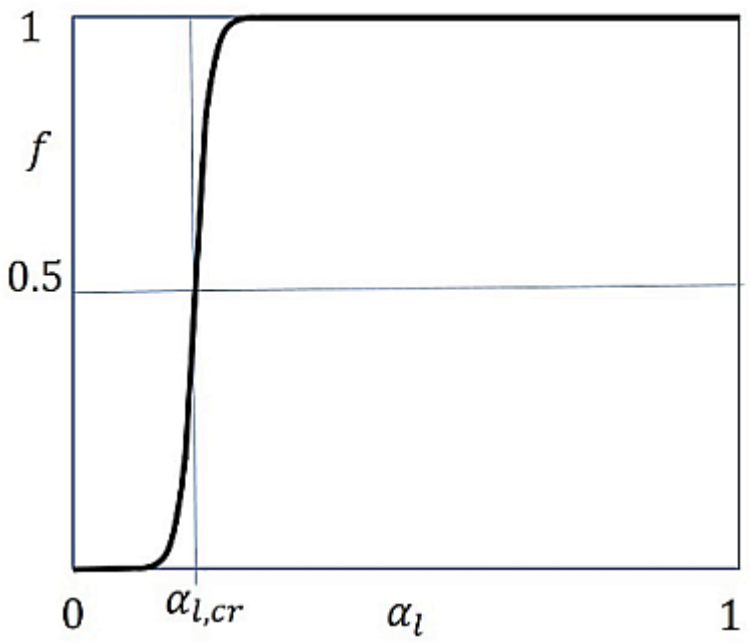
$\alpha_{l,cr}$ = **Critical Liquid Volume Fraction** (default = 0.2)

n = transition constant (default = 20)

The dependency of the wetting function f on the liquid volume fraction α_l is shown in Figure 26.55: Transition Function vs. Volume Fraction of Liquid (p. 3064).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.55: Transition Function vs. Volume Fraction of Liquid



wall-neighbor

calculates the wetting fraction through the neighboring cell volume fractions.

constant

allows you to specify the wall liquid wetting fraction as a constant value.

user-defined

allows you to hook a user-defined function that specifies custom values for the wall liquid wetting fraction.

26.4.5.5. Solution Strategies for the Semi-Mechanistic Boiling Model

If you face difficulties in convergence for the energy equation while using the conjugate heat transfer boundary condition, you can use the following strategies:

- It is recommended to use meshes with y+ > 5 for better stability.
- Use low order schemes for turbulence kinetic energy and momentum.
- Reduce the relaxation factor for turbulence kinetic energy to a value between 0.5-0.75.
- Reduce the relaxation factor for heat flux to a value between 0.3-0.5.
- In case of steady-state simulations that use a pseudo time method with automatic time-scale, reduce the fluid **Time Scale Factor** to 0.1.
- For a large change in heat flux, you can modify the heat flux relaxation factor using the following text command:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
solve/set/multiphase-numerics/heat-mass-transfer/boiling/heat-flux-
relaxation-factor

heat flux relaxation factor for semi-mechanistic boiling model [0.8]
0.3
```

The heat flux relaxation factor will be modified as follows:

$$q = q^* + f_{heat-flux}(q - q^*)$$

where q and q^* are the new and old heat fluxes, respectively, and $f_{heat-flux}$ is the heat flux relaxation factor that you specified.

Note:

To carry out validations against experimental data, you should first run simulations under non-boiling conditions. If the single-phase heat flux differs from the available data, you should adjust the single-phase heat flux multiplier prior to carrying out the boiling simulations. There is no unique choice of nucleate boiling correlations; therefore, the default nucleate boiling correlation is not guaranteed to work under all the operating conditions. Ansys Fluent provides flexibility either to tune in the existing methods or to hook user-defined properties for all supported boiling model inputs.

26.5. Setting Up the Eulerian Model

For background information about the Eulerian model and the limitations that apply, refer to Overview of the Eulerian Model in the Theory Guide.

For additional information, see the following sections:

26.5.1. Additional Guidelines for Eulerian Multiphase Simulations

26.5.2. Defining the Phases for the Eulerian Model

26.5.3. Modeling Turbulence

26.5.4. Including Heat Transfer Effects

26.5.5. Using an Algebraic Interfacial Area Model

26.5.6. Using the Algebraic Interfacial Area Density (AIAD) Model

26.5.7. Using the Generalized Two Phase Flow (GENTOP) Model

26.5.8. Including the Dense Discrete Phase Model

26.5.9. Including the Boiling Model

26.5.10. Setting Up Polydisperse Boiling

26.5.11. Including the Multi-Fluid VOF Model

26.5.1. Additional Guidelines for Eulerian Multiphase Simulations

Once you have determined that the Eulerian multiphase model is appropriate for your problem (as described in Choosing a General Multiphase Model in the Theory Guide), you should consider the

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

computational effort required to solve your multiphase problem. The required computational effort depends strongly on the number of transport equations being solved and the degree of coupling. For the Eulerian multiphase model, which has a large number of highly coupled transport equations, computational expense will be high. Before setting up your problem, try to reduce the problem statement to the simplest form possible.

Instead of trying to solve your multiphase flow in all of its complexity on your first solution attempt, you can start with simple approximations and work your way up to the final form of the problem definition. Some suggestions for simplifying a multiphase flow problem are listed below:

- Use a hexahedral or quadrilateral mesh (instead of a tetrahedral or triangular mesh).
- Reduce the number of phases.

You may find that even a very simple approximation will provide you with useful information about your problem.

See Eulerian Model (p. 3205) and Setting Solution Limits (p. 3621) for more solution strategies for Eulerian multiphase calculations.

26.5.2. Defining the Phases for the Eulerian Model

Instructions for specifying the necessary information for the primary and secondary phases and their interaction for an Eulerian multiphase calculation are provided below.

26.5.2.1. Defining the Primary Phase

The procedure for defining the primary phase in an Eulerian multiphase calculation is the same as for a VOF calculation. See Defining the Primary Phase (p. 3009) for details.

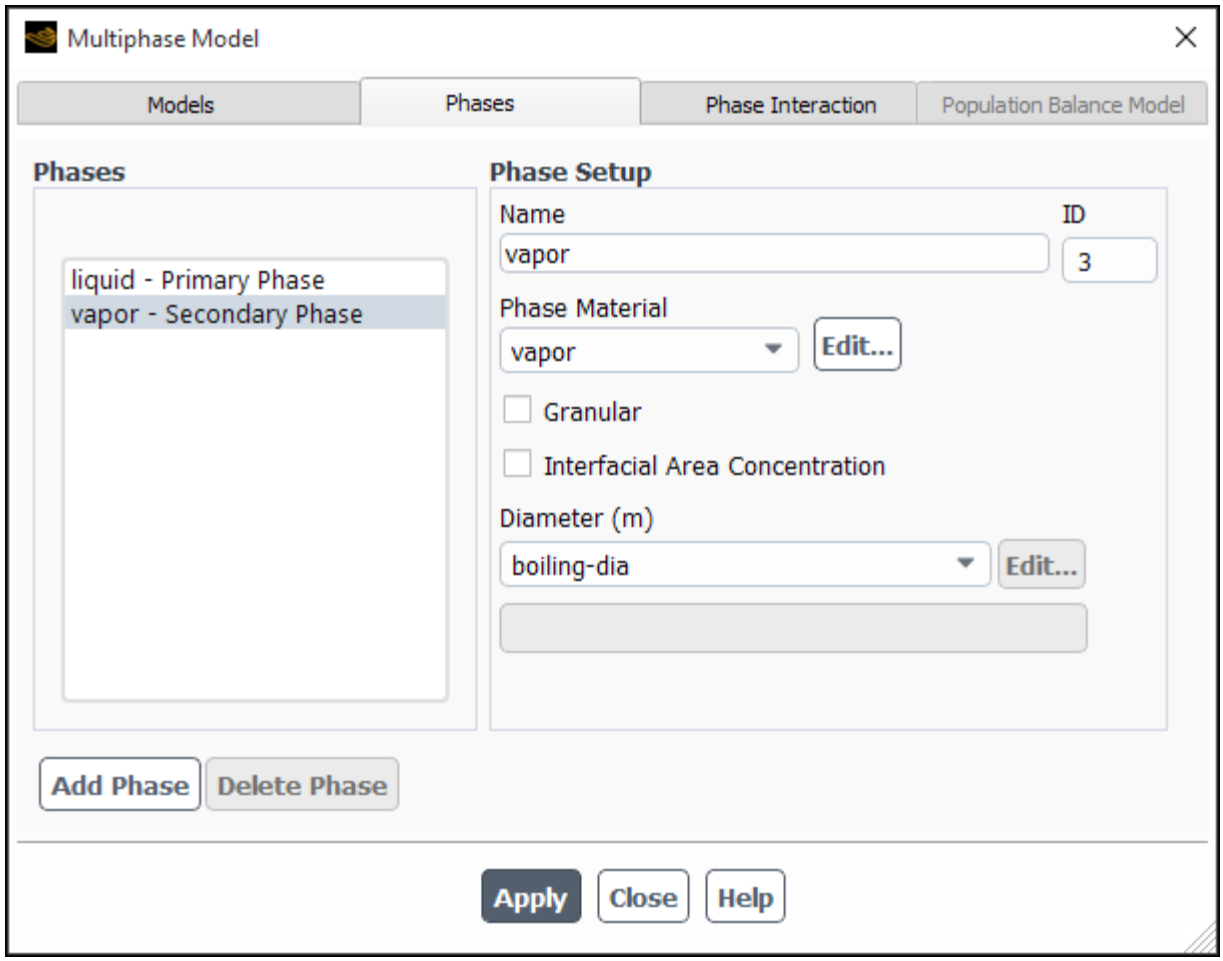
26.5.2.2. Defining a Non-Granular Secondary Phase

To define a non-granular (that is, liquid or vapor) secondary phase in an Eulerian multiphase calculation, go to the **Phases** tab in the **Multiphase Model** dialog box and perform the following steps:

1. Select the phase (for example, **phase-2**) in the **Phases** list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.56: The Multiphase Model Dialog Box for a Non-Granular Phase (Phases Tab)



2. In the **Phase Setup** group box, enter a **Name** for the phase (for example, `vapor`).

3. Specify which material the phase contains by choosing the appropriate material in the **Phase Material** drop-down list.

4. Define the material properties for the **Phase Material**, following the same procedure you used to set the material properties for the primary phase (see Defining the Primary Phase (p. 3009)).

5. Specify the **Diameter** of the bubbles or droplets of this phase. You can specify a constant value, or use a user-defined function. See the Fluent Customization Manual for details about user-defined functions.

6. Click **Apply**.

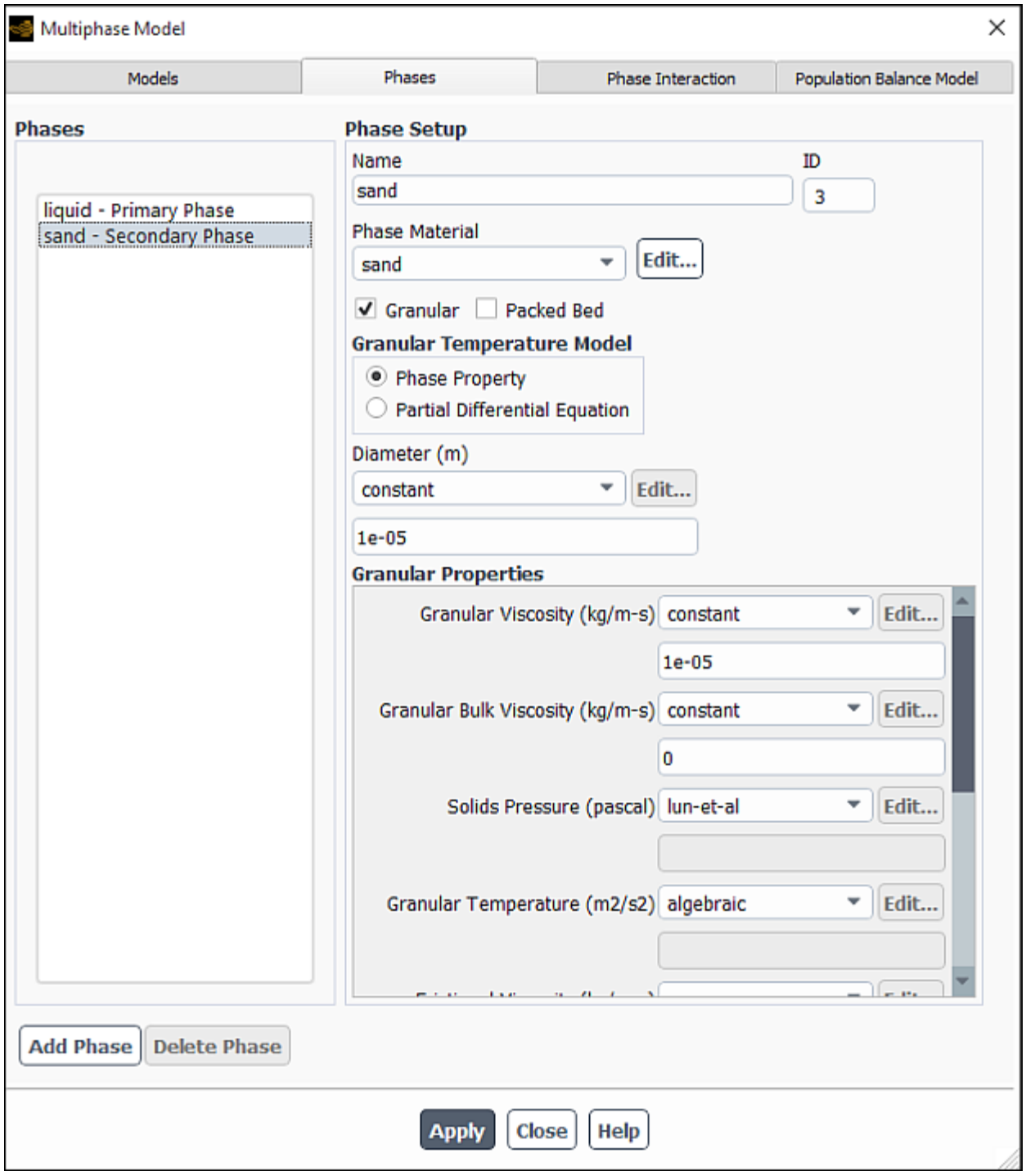
26.5.2.3. Defining a Granular Secondary Phase

To define a granular (that is, particulate) secondary phase in an Eulerian multiphase calculation, go to the **Phases** tab in the **Multiphase Model** dialog box and perform the following steps::

1. Select the phase (for example, **phase-2**) in the **Phases** list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.57: The Multiphase Model Dialog Box for a Granular Phase (Phases Tab)



2. In the **Phase Setup** group box, enter a **Name** for the phase (for example, **sand**).

3. Specify which material the phase contains by choosing the appropriate material in the **Phase Material** drop-down list.

4. Define the material properties for the **Phase Material**, following the same procedure you used to set the material properties for the primary phase (see Defining the Primary Phase (p. 3009)). For a granular phase (which must be placed in the fluid materials category, as mentioned in

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Steps for Using a Multiphase Model (p. 2915)), you need to specify only the density; you can ignore the values for the other properties, since they will not be used.

Important:

Note that all properties for granular flows can be defined by user-defined functions (UDFs).

See the Fluent Customization Manual for details about user-defined functions.

5. Enable the **Granular** option.

6. (optional) Enable the **Packed Bed** option if you want to freeze the velocity field for the granular phase. Note that when you select the packed bed option for a phase, you should also use the fixed velocity option with a value of zero for all velocity components for all interior cell zones for that phase. Using fixed velocity in radial direction as a constant value (other than zero) does not ensure continuity. With the fixed velocity option, both velocity and pressure are fixed in a zone. Since face area in the radial direction is a function of radial distance from the axis, it will result in a mass conservation problem due to flux imbalance.

   Using zero fixed velocity in the radial direction does not cause any problems related to mass conservation.

7. Specify the **Granular Temperature Model**. Choose either the default **Phase Property** option or the **Partial Differential Equation** option. See Granular Temperature in the Theory Guide for details.

8. Specifies the **Diameter** of the particles. You can select **constant** in the drop-down list and specify a constant value, or select **user-defined** to use a user-defined function. See the Fluent Customization Manual for details about user-defined functions.

9. In the **Granular Properties** group box, specify the following properties of the particles of this phase:

   Granular Viscosity

   specifies the method for computing the kinetic ($\mu_{s,kin}$) and collisional ($\mu_{s,col}$) components of the granular viscosity (Equation 14.362 in the *Fluent Theory Guide*). Selecting **constant**, **syamlal-obrien** (Equation 14.364), or **gidaspow** (Equation 14.365) will use these expressions for the kinetic portion of the viscosity and will calculate the collisional portion of the viscosity from Equation 14.363 in the *Fluent Theory Guide*. Alternatively, you can select **user-defined** to use a user-defined function. Note that if you select **user-defined**, your user-defined function must include both the kinetic portion and the collisional portion of the viscosity in the value it returns.

   Note:

   If **filtered** is selected for **Drag Coefficient**, then **Granular Viscosity** is set automatically to **filtered**. The granular viscosity will be calculated as described in The Filtered Two-Fluid Model in the *Fluent Theory Guide*.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Granular Bulk Viscosity

specifies the solids bulk viscosity (λ_q in Equation 14.195 in the Theory Guide). You can select **constant** (the default) in the drop-down list and specify a constant value, select **lun-et-al** to compute the value using Equation 14.366 in the Theory Guide, or select **user-defined** to use a user-defined function.

Note:

If **filtered** is selected for **Drag Coefficient**, then **Granular Bulk Viscosity** is set to zero automatically.

Frictional Viscosity

specifies a shear viscosity based on the viscous-plastic flow ($\mu_{s,fr}$ in Equation 14.362 in the Theory Guide). By default, the frictional viscosity is neglected, as indicated by the default selection of **none** in the drop-down list. If you want to include the frictional viscosity, you can select **constant** and specify a constant value, select **schaeffer** to compute the value using Equation 14.367 in the Theory Guide, or select **user-defined** to use a user-defined function.

Angle of Internal Friction

specifies a constant value for the angle ϕ used in Schaeffer's expression for frictional viscosity (Equation 14.367 in the Theory Guide). This parameter is relevant only if you have selected **schaeffer** or **user-defined** for the **Frictional Viscosity**.

Frictional Pressure

specifies the pressure gradient term, $\nabla P_{friction}$, in the granular-phase momentum equation. Choose **none** to exclude frictional pressure from your calculation, **johnson-et-al** to apply Equation 14.371 in the Theory Guide, **syamlal-obrien** to apply Equation 14.268 in the Theory Guide, **based-ktgf**, where the frictional pressure is defined by the kinetic theory [38] (p. 5715). The solids pressure tends to a large value near the packing limit, depending on the model selected for the radial distribution function. You must hook a user-defined function when selecting the **user-defined** option. See the Fluent Customization Manual for information on hooking a UDF.

Frictional Modulus

is defined as

$$G=\frac{\partial P_{friction}}{\partial \alpha_{friction}} \tag{26.20}$$

with $G \geq 0$, which is the **derived** option. You can also specify a **user-defined** function for the frictional modulus.

Friction Packing Limit

specifies a threshold volume fraction at which the frictional regime becomes dominant. The default value is 0.61.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Granular Conductivity

specifies the solids granular conductivity (k_{Θ_s} in Equation 14.374 in the Theory Guide). You can select **syamlal-obrien** to compute the value using Equation 14.375 in the Theory Guide, select **gidaspow** to compute the value using Equation 14.376 in the Theory Guide, or select **user-defined** to use a user-defined function. Note that in the algebraic model, shown in Figure 26.57: The Multiphase Model Dialog Box for a Granular Phase (**Phases** Tab) (p. 3068), the granular conductivity is not required in the computation of the granular temperature. This has been obtained by neglecting convection and diffusion in the transport equation, Equation 14.374 in the Theory Guide [157] (p. 5722).

Granular Temperature

specifies temperature for the solids phase and is proportional to the kinetic energy of the random motion of the particles. Choose the **algebraic**, **constant**, **dpm-averaged**, or **user-defined** option.

Note:

- The **dpm-averaged** method is only available when using the Dense Discrete Phase Model (DDPM).
- If **filtered** is selected for **Drag Coefficient**, then **Granular Temperature** is set automatically to zero.

Solids Pressure

specifies the pressure gradient term, ∇p_s, in the granular-phase momentum equation. Choose either the **lun-et-al**, the **syamlal-obrien**, the **ma-ahmadi**, **none**, or a **user-defined** option.

Note:

If **filtered** is selected for **Drag Coefficient**, then **Solids Pressure** is set automatically to **filtered**. The solid pressure will be calculated as described in The Filtered Two-Fluid Model in the *Fluent Theory Guide*.

Radial Distribution

specifies a correction factor that modifies the probability of collisions between grains when the solid granular phase becomes dense. Choose either the **lun-et-al**, the **syamlal-obrien**, the **ma-ahmadi**, the **arastoopour**, or a **user-defined** option.

Note:

If **filtered** is selected for **Drag Coefficient**, then **Radial Distribution** is set automatically to **filtered**. In the simulation, the radial distribution will be set to zero since the filtered two-fluid model is not based on kinetic theory.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Elasticity Modulus

is defined as

$$G=\frac{\partial P_s}{\partial \alpha_s} \tag{26.21}$$

with $G \geq 0$.

Packing Limit

specifies the maximum volume fraction for the granular phase. For monodispersed spheres, the packing limit is about 0.63, which is the default value in Ansys Fluent. In polydispersed cases, however, smaller spheres can fill the small gaps between larger spheres, so you may need to increase the maximum packing limit.

10. Click **Apply**.

26.5.2.4. Defining the Interfacial Area Concentration

When using the Eulerian multiphase model, you can choose to solve a transport equation for the interfacial area, which allows for a distribution of secondary phase diameter, or you can use an algebraic model to compute the interfacial area from a specified diameter. To use an algebraic model, refer to Using an Algebraic Interfacial Area Model (p. 3096). To solve the transport equation (Interfacial Area Concentration in the *Fluent Theory Guide*), perform the following steps:

1. In the **Multiphase Model** dialog box, go to the **Phase Interaction** > **Interfacial Area** tab.
2. Select the phase (for example, **phase-2**) in the **Phases** list.
3. In the **Phase Setup** group box, enter a **Name** for the phase.
4. Specify which material the phase contains by choosing the appropriate material in the **Phase Material** drop-down list.
5. Define the material properties for the **Phase Material**.
6. Enable the **Interfacial Area Concentration** option. Make sure the **Granular** option is disabled for the **Interfacial Area Concentration** option to be visible in the interface.
7. Specify the **Diameter** of the particles or bubbles. You can select **constant** in the drop-down list and specify a constant value, or select **user-defined** to use a user-defined function. See the Fluent Customization Manual for details about user-defined functions. The **Diameter** recommended setting is **sauter-mean**, allowing for the effects of the interfacial area concentration values to be considered for mass, momentum and heat transfer across the interface between phases.
8. In the **Interfacial Area Concentration Properties** group box, specify the following properties of the particles of this phase:

   Coalescence Kernel and Breakage Kernel

   allows you to specify the coalescence and breakage kernels. You can select **none**, **constant**, **hibiki-ishii**, **ishii-kim**, **yao-morel**, or **user-defined**. The three options, **hibiki-ishii**, **ishii-kim** and **yao-morel** are described in detail in Interfacial Area Concentration in the Theory Guide.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In addition to specifying the **hibiki-ishii**, **ishii-kim**, and **yao-morel** as the coalescence and breakage kernels, you can also tune the properties of the three models by using the `/define/phases/iac-expert/hibiki-ishii-model`, `/define/phases/iac-expert/ishii-kim-model`, and `/define/phases/iac-expert/yao-morel-model` text commands.

For each of the three models you can specify the parameters listed in Table 26.13: Parameters for the Coalescence and Breakage Kernels (p. 3073)

Table 26.13: Parameters for the Coalescence and Breakage Kernels

Hibiki-Ishii Model	Ishii-Kim Model	Yao-Morel Model
`Coefficient Gamma_c`	`Coefficient Crc`	`Coefficient K_c1`
`Coefficient K_c`	`Coefficient Cwe`	`Coefficient K_c3`
`Coefficient Gamma_b`	`Coefficient C`	`Coefficient K_b1`
`Coefficient K_b`	`Coefficient Cti`	`alpha_max`
`alpha_max`	`alpha_max`	

These values are discussed in greater detail in Interfacial Area Concentration in the Theory Guide.

Nucleation Rate

is a source term for the interfacial area concentration that models the rate of formation of the dispersed phase. You can choose from **constant** or **user-defined**. If the **Boiling Model** option is enabled, you can also select **yao-morel**. The **yao-morel** option is described in Yao-Morel Model in the Theory Guide.

Critical Weber Number

will need to be specified if you selected **ishii-kim** or **yao-morel** for the **Breakage Kernel**.

Dissipation Function

gives you the option to choose the formula which calculates the dissipation rate used in the **hibiki-ishii** and **ishii-kim** models. You can choose amongst **constant**, **wu-ishii-kim**, **fluent-ke**, and **user-defined** for the dissipation function.

The **wu-ishii-kim** option uses a simple algebraic correlation for ε:

$$\varepsilon = f_{TW}\left(1/\ 2D_h\right)v_m^3 \tag{26.22}$$

where

$$f_{TW} = \frac{0.316}{\left[(1-\alpha)Re_m\right]^{0.25}}$$

and

$$Re_m = \frac{\rho_m v_m D_h}{\mu_m}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where ρ_m, v_m, μ_m, and D_h are the mixture density, mixture velocity, mixture molecular viscosity, and hydraulic diameter of the flow path.

When you select the **wu-ishii-kim** model, you will set an additional input for **Hydraulic Diameter**.

Hydraulic Diameter

is the value used in Equation 26.22 (p. 3073), should you use the **wu-ishii-kim** formulation.

Min/Max Diameter

allow you to specify the minimum and maximum diameters of the secondary phase to which the interfacial area concentration model is applied, preventing some computing anomalies to spread beyond control.

9. Click **Apply**.

Note:

When solving a steady-state problem, the preferred setting for the **Under-Relaxation Factor** is 1.0, as the interfacial area equation for the boiling models is currently under-relaxed using a locally defined pseudo time step. If you want extra explicit under-relaxation, you may set the value of the **Under-Relaxation Factor** to less than one, this may be done only in case of serious convergence problems with the interfacial area transport equation. To improve convergence you can switch to a pseudo time step for the interfacial area concentration only, using the `define/phases/iac-expert/iac-pseudo-time-step` text command and set the local pseudo time step to less than 1.

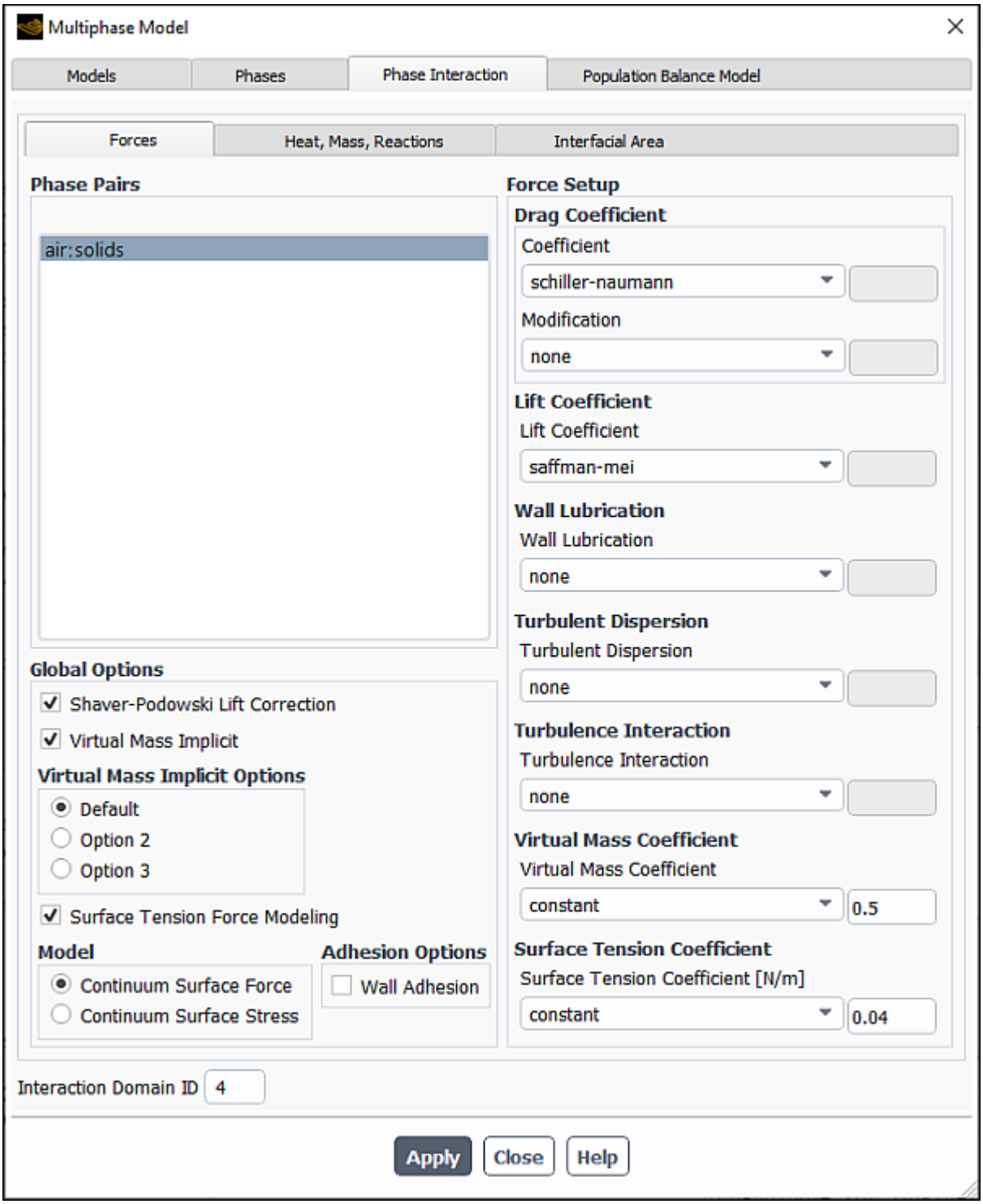
26.5.2.5. Defining the Interaction Between Phases

For both granular and non-granular flows, you need to specify the drag function to be used in the calculation of the momentum exchange coefficients. You can also specify lift forces, wall lubrication forces (for non-granular flows only), turbulent dispersion forces, surface tension effects, and virtual mass force. For granular flows, you will also need to specify the restitution coefficient(s) for particle collisions.

To specify these parameters, go to the **Phase Interaction** > **Forces** tab in the **Multiphase Model** dialog box and select appropriate options and values in the **Global Options** and **Force Setup** group boxes.

Setup → Models → Multiphase → Edit...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



26.5.2.5.1. Specifying the Drag Function

Ansys Fluent allows you to specify a drag function for each pair of phases.

For each pair of phases, in the **Drag coefficient** group box, select the one of the following options from the **Coefficient** drop-down list.

- Select **ishii** to use the fluid-fluid drag function described by Equation 14.516 in the Theory Guide. This option is available when the boiling model is enabled.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Select **schiller-naumann** to use the fluid-fluid drag function described by Equation 14.215 in the Theory Guide. The Schiller and Naumann model is the default method, and it is acceptable for general use in all fluid-fluid multiphase calculations. When using the homogeneous population balance models you can also select **schiller-naumann-pb** in which case the interfacial area will be calculated directly from the population balance variables.

- Select **morsi-alexander** to use the fluid-fluid drag function described by Equation 14.219 in the Theory Guide. The Morsi and Alexander model is the most complete, adjusting the function definition frequently over a large range of Reynolds numbers, but calculations with this model may be less stable than with the other models.

- Select **symmetric** to use the fluid-fluid drag function described by Equation 14.225 in the Theory Guide. The symmetric model is recommended for flows in which the secondary (dispersed) phase in one region of the domain becomes the primary (continuous) phase in another. For example, if air is injected into the bottom of a container filled halfway with water, the air is the dispersed phase in the bottom half of the container; in the top half of the container, the air is the continuous phase. The **symmetric** drag law is the default method for the **Multi-Fluid VOF Model**, which is available with Eulerian multiphase model.

- Select **grace** to use the fluid-fluid drag function described by Equation 14.230 in the Theory Guide. The Grace model is recommended for gas-liquid flows in which the bubbles can have a range of shapes such as spherical, elliptical, or cap.

  In the **Grace Swarm Correction** dialog box that opens automatically when you first select this model, you can specify **Coefficient: Cexp** (C_{exp} in Equation 14.230 in the *Fluent Theory Guide*) as **constant** or **user-defined**. Depending on the flow regime and the bubble size, the following values should be used:

  – Sparsely distributed fluid particles

    In flows with sparsely distributed fluid particles, C_{exp} in Equation 14.230 in the *Fluent Theory Guide* is zero (default).

  – Densely distributed fluid particles

    → Since small bubbles tend to rise slowly in high gas volume fractions due to an increase in the effective mixture viscosity, a negative C_{exp} should be used. The Ishii-Zuber correlation uses an exponent of -1 for this limit. A value of -0.5 has also been used successfully by some investigators.

    → Large bubbles, on the other hand, tend to rise faster at high gas volume fractions because they are dragged along by the wakes of other bubbles. This is modeled by using a positive C_{exp}. The Ishii-Zuber correlation uses an exponent of 2 for this regime. A value of 4 has also been used successfully by some researchers in the past

  See Grace et al. Model in the *Fluent Theory Guide* for more information.

- Select **tomiyama** to use the fluid-fluid drag function described by Equation 14.236 in the Theory Guide. Like the Grace model, the Tomiyama model is recommended for gas-liquid flows in which the bubbles can have a range of shapes such as spherical, elliptical, or cap.

- Select **ishii-zuber** to use the fluid-fluid drag function described in Ishii-Zuber Drag Model in the *Fluent Theory Guide*. The model automatically accounts for swarms of bubbles moving to-

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

gether in high gas volume fractions. Like the Grace and Tomiyama drag models, the Ishii-Zuber drag law is recommended for gas-liquid flows in which bubbles may have a range of shapes such as spherical, elliptical, or cap.

- Select **anisotropic** to use the fluid-fluid drag function described in Multi-Fluid VOF Model in the Theory Guide. The **anisotropic** drag law is recommended for free surface modeling. It is based on higher drag in the normal direction to the interface and lower drag in the tangential direction to the interface.

- Select **universal-drag** for bubble-liquid and/or droplet-gas flow when the characteristic length of the flow domain is much greater than the averaged size of the droplets/bubbles. It is suitable for flows in which the bubbles/droplets may have a range of shapes. The universal drag law is described using Equation 14.247 in the Theory Guide. When **universal-drag** is selected, you will need to set a value for the **Surface Tension Coefficient**. This value will apply to the primary phase and the secondary phase.

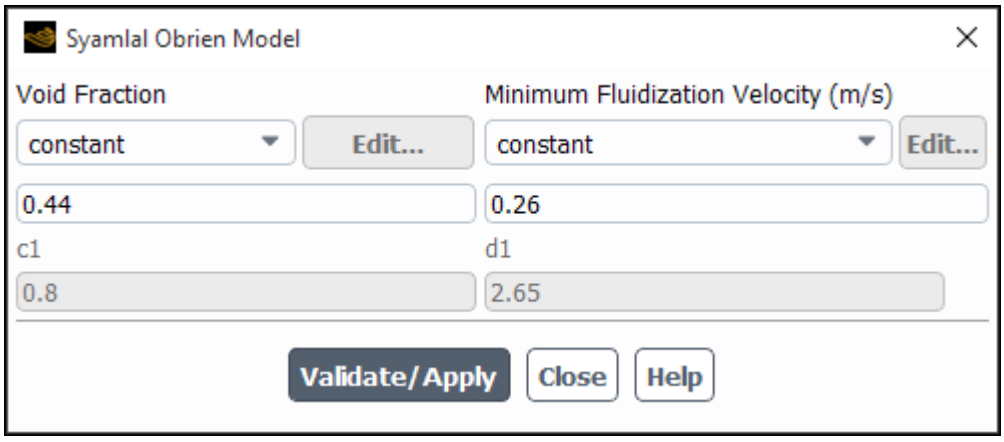
- Select **wen-yu** to use the fluid-solid drag function described by Equation 14.282 in the Theory Guide. The Wen and Yu model is applicable for dilute phase flows, in which the total secondary phase volume fraction is significantly lower than that of the primary phase. When using the homogeneous population balance models you can also select **wen-yu-pb** in which case the interfacial area will be calculated directly from the population balance variables.

- Select **gidaspow** to use the fluid-solid drag function described by Equation 14.284 in the Theory Guide. The Gidaspow model is recommended for dense fluidized beds.

- Select **syamlal-obrien** to use the fluid-solid drag function described by Equation 14.269 in the Theory Guide. The Syamlal-O'Brien model is recommended for use in conjunction with the Syamlal-O'Brien model for granular viscosity.

- Select **syamlal-obrien-para** to use the parameterized formulation of the Syamlal-O'Brien model described by Equation 14.279 in the Theory Guide. This model addresses under/over-prediction of bed expansion that can arise with **syamlal-obrien**. When you first select this model, you will be presented with the **Syamlal Obrien Model** dialog box.

Figure 26.58: Syamlal Obrien Model Dialog Box



There are two inputs:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Void Fraction

the volume fraction of the gas phase in the bed at the minimum fluidization condition

Minimum Fluidization Velocity

the expected or experimentally-determined nominal minimum fluidization velocity of the gas phase

Once you have input these values, click **Validate/Apply** to compute the coefficients **c1** and **d1** used in Equation 14.280 in the *Fluent Theory Guide*. A message will also be printed to the text user interface summarizing the fluid and flow properties used in the computation and the resulting values. You should check that these property values are consistent with the properties set elsewhere.

Important:

If you change property values in the **Create/Edit Materials** dialog box or diameter for the secondary phase in the **Phases** tab, you must initialize the flow field or run at least one iteration before computing c1 and d1. Otherwise, incorrect properties will be used in the computation.

Note that the **syamlal-obrien-para** model is appropriate only for Geldart Group B particles and is only applicable to cases involving a single secondary phase.

- Select **syamlal-obrien-symmetric** to use the solid-solid drag function described by Equation 14.292 in the Theory Guide. The symmetric Syamlal-O'Brien model is appropriate for a pair of solid phases.
- Select **huilin-gidaspow** to use the fluid-solid drag function described by Equation 14.286 in the Theory Guide. This option provides a better blending function for the Gidaspow model when moving from the dense packing limit to the dilute flow limit.
- Select **gibilaro** to use the fluid-solid drag function described by Equation 14.288 in the Theory Guide. This option is used for circulating fluidized beds.
- Select **emms** drag model for gas-solid flow where heterogeneous meso-scale flow structures are dominant and such structures cannot be adequately resolved using fine grid resolution in commercial scale fluidized bed simulation. The fluid-solid drag function is described by Equation 14.290 in the *Fluent Theory Guide*. The EMMS drag model is applicable only to monodispersed two-phase granular flows without axisymmetric swirl.

  Note that this option is not available for non-granular flows.
- Select **filtered** drag model for gas-solid flow cases with coarse meshes. The fluid-solid drag function is computed by Equation 14.290 and Equation 14.488 in the *Fluent Theory Guide*. Once you click **Apply**, the following settings are automatically applied to the granular phase properties in the **Phases** tab:
  - **filtered** is selected for **Granular Viscosity**, **Solid Pressure**, and **Radial Distribution**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the simulation, the radial distribution will be set to zero since the filtered two-fluid model is not based on kinetic theory.

– **Granular Bulk Viscosity** and **Granular Temperature** are set to a constant value of zero.

For information about how these parameters are calculated, see The Filtered Two-Fluid Model in the *Fluent Theory Guide*.

In case of convergence issues, you can first run the case with homogeneous drag before switching to the filtered model.

The filtered drag model is applicable only to monodispersed two-phase flows involving gas and granular phases without axisymmetric swirl and is well suited for predicting flow hydrodynamics in fluidized beds.

- Select **constant** to specify a constant value for the drag function, and then specify the value in the text field.
- Select **user-defined** to use a user-defined function for the drag function (see the Fluent Customization Manual for details).
- If you want to temporarily ignore the interaction between two phases, select **none**.

26.5.2.5.1.1. Drag Modification

When using the Eulerian or Mixture multiphase models, you can optionally specify a drag modification term for the selected drag law. The drag modification term acts as a multiplier for the drag coefficient computed from the models detailed in Specifying the Drag Function (p. 3075). You can specify the drag modification term individually for each pair of primary-secondary phases. By default, **Modification** is set to **none**. To define drag modification, for each pair of phases, select one of the following options from the **Modification** drop-down list:

- **constant**: Uses a specified constant value for the drag modification factor.
- **brucato**: Uses the Brucato et al. correlation described by Brucato et al. Correlation in the Theory Guide. This modification is appropriate for dilute gas-liquid flows and solid-liquid flows.
- **wall-drag-enhancement**: Uses the wall drag modification described in Near-Wall Drag Enhancement in the *Fluent Theory Guide*. The modification is useful for improved predictions of the slip velocity in near-wall regions and boiling flow simulations.
- **user-defined**: Uses a user-defined function for the drag modification factor (see `DEFINE_EXCHANGE_PROPERTY` in the Fluent Customization Manual for details).

For additional details on the implementation of the drag modification, see Drag Modification in the Theory Guide.

26.5.2.5.2. Specifying the Restitution Coefficients (Granular Flow Only)

For granular flows, you need to specify the coefficients of restitution for collisions between particles (e_{ls} in Equation 14.292 and e_{ss} in Equation 14.345 in the Theory Guide). In addition to specifying the restitution coefficient for collisions between each pair of granular phases, you will also specify the restitution coefficient for collisions between particles of the same phase.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Perform the following steps:

1. In the **Phases** list, select the granular phase pair.

2. For each pair of granular phases, specify a constant **Restitution Coefficient**. All restitution coefficients are equal to 0.9 by default.

26.5.2.5.3. Including the Lift Force

For both granular and non-granular flows, it is possible to include the effect of lift forces ($\vec{F}_{lift}$ in Equation 14.298 in the Theory Guide) on the secondary phase particles, droplets, or bubbles. These lift forces act on a particle, droplet, or bubble mainly due to velocity gradients in the primary-phase flow field. In most cases, the lift force is insignificant compared to the drag force, so there is no reason to include it. If the lift force is significant (for example, if the phases separate quickly), you may want to include this effect.

Important:

Note that the lift force will be more significant for larger particles, but the Ansys Fluent model assumes that the particle diameter is much smaller than the interparticle spacing. Therefore, the inclusion of lift forces is not appropriate for closely packed particles or for very small particles.

To include the effect of lift forces, for each pair of phases, select the appropriate specification method from the **Lift Coefficient** drop-down list. Note that, since the lift forces for a particle, droplet, or bubble are due mainly to velocity gradients in the primary-phase flow field, you will not specify lift coefficients for pairs consisting of two secondary phases; lift coefficients are specified only for pairs consisting of a secondary phase and the primary phase.

- Select **none** (the default) to ignore the effect of lift forces.

- Select **constant** to specify a constant lift coefficient, and then specify the value in the text field.

- Select **moraga** to use the Moraga lift model (Moraga Lift Force Model in the Theory Guide). The Moraga lift model is applicable to spherical solid particles, drops, and bubbles.

- Select **saffman-mei** to use the Saffman-Mei lift model (Saffman-Mei Lift Force Model in the Theory Guide). The Saffman-Mei lift model is applicable to spherical solid particles, and to drops and bubbles that are not significantly distorted.

- Select **legendre-magnaudet** to use the Legendre—Magnaudet lift model (Legendre-Magnaudet Lift Force Model in the Theory Guide). The Legendre-Magnaudet model is applicable to small diameter spherical fluid particles, though it can be applied to non-distorted liquid drops and bubbles. It accounts for momentum transfer between the flow around the particle and the inner recirculation flow inside the fluid particle caused by fluid friction/stresses at the fluid interface.

- Select **tomiyama** to use the Tomiyama lift model (Tomiyama Lift Force Model in the Theory Guide). The Tomiyama model is applicable to larger-scale deformable bubbles in the ellipsoidal and spherical cap regimes. Its main feature is the prediction of the cross-over point in bubble size at which particle distortion causes a reversal in the sign of the lift force.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Select **hessenkemper-et-al** to use the Hessenkemper et al. lift model (Hessenkemper et al. Lift Force Model in the *Fluent Theory Guide*). The Hessenkemper et al. lift model is applicable to small diameter spherical fluid particles and large-scale deformable fluid particles in the ellipsoidal and spherical cap regimes.
- Select **user-defined** to use a user-defined function for the lift coefficient (see the Fluent Customization Manual for details).

26.5.2.5.4. Including the Lift Correlation

When the lift force is included in your simulation, you can enable the **Shaver-Podowski Lift Correlation** option (**Global Options** group box). This option improves the accuracy of the prediction of the void peak near the wall and overall robustness when the turbulent dispersion and wall lubrication are used in your case. For more information about this option, see Shaver-Podowski Correction in the *Fluent Theory Guide*.

26.5.2.5.5. Including the Wall Lubrication Force

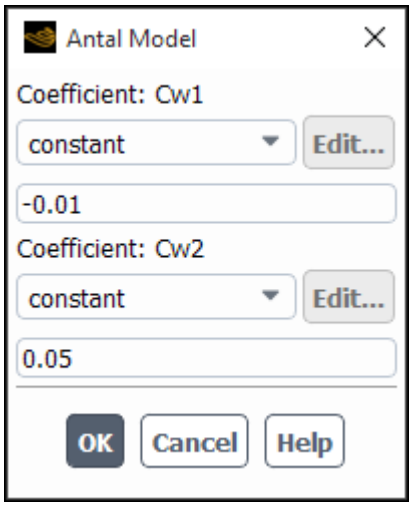
For liquid-gas bubbly flows using the Eulerian multiphase model, you can include the effect of wall lubrication forces ($\vec{F}_{wl}$ in Equation 14.194 in the Theory Guide) on the secondary phase bubbles. These forces tend to push the bubbles away from walls at small wall distances. For details on how wall lubrication is modeled in Ansys Fluent see Wall Lubrication Force in the Theory Guide.

To include the effect of wall lubrication forces, for each pair of phases, select the appropriate specification method from the **Wall Lubrication** drop-down list. Note that you will specify the wall lubrication only for pairs consisting of a secondary phase and the primary phase.

- Select **none** (the default) to ignore the effect of wall lubrication forces.
- Select **lubchenko** to use the Lubchenko model (Lubchenko Model in the *Fluent Theory Guide*). This model is available only either **burns-et-al** or **lopez-de-bertodano** is selected for the **Turbulent Dispersion**. If lift is considered in your simulation, you should use the **shaver-podowski** lift correction.
- Select **antal-et-al** to use the Antal et al. model (Antal et al. Model in the Theory Guide). You can edit the model parameters in the **Antal Model** dialog box (Figure 26.59: Antal et al. Model Dialog Box (p. 3082)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.59: Antal et al. Model Dialog Box



The following model parameters are available:

Coefficient: Cw1

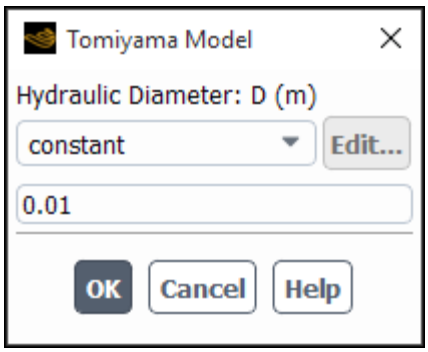
Specify the constant C_{w1} in Equation 14.320 in the Theory Guide. You can enter a constant value or specify a user-defined function defined using the `DEFINE_EXCHANGE_PROPERTY` macro (`DEFINE_EXCHANGE_PROPERTY`).

Coefficient: Cw2

Specify the constant C_{w2} in Equation 14.320 in the Theory Guide. You can enter a constant value or specify a user-defined function defined using the `DEFINE_EXCHANGE_PROPERTY` macro (`DEFINE_EXCHANGE_PROPERTY`).

- Select **tomiyama** to use the Tomiyama model (Tomiyama Model in the Theory Guide). This model is only applicable for pipe geometries. You can enter the Hydraulic Diameter for your geometry in the **Tomiyama Model** dialog box (Figure 26.60: Tomiyama Model Dialog Box (p. 3082)).

Figure 26.60: Tomiyama Model Dialog Box



The following model parameters are available:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

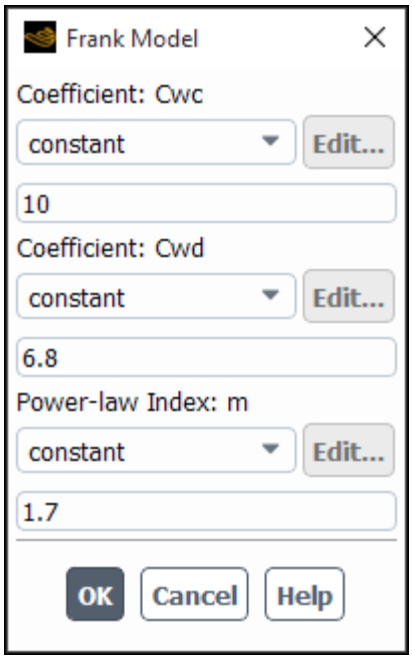
Hydraulic Diameter: D (m)

Specify the hydraulic diameter, D, in Equation 14.322 in the Theory Guide.

> **Important:**
>
> The hydraulic diameter must be entered in units of meters.

- Select **frank** to use the Frank et al. model (Frank Model in the Theory Guide). You can edit the model parameters in the **Frank Model** dialog box (Figure 26.61: Frank Model Dialog Box (p. 3083)).

Figure 26.61: Frank Model Dialog Box



The following model parameters are available:

Coefficient: Cwc

Specify the cutoff coefficient, C_{wc}, in Equation 14.324 in the Theory Guide. You can enter a constant value or specify a user-defined function defined using the `DEFINE_EXCHANGE_PROPERTY` macro (`DEFINE_EXCHANGE_PROPERTY`).

Coefficient: Cwd

Specify the damping coefficient, C_{wd}, in Equation 14.324 in the Theory Guide. You can enter a constant value or specify a user-defined function defined using the `DEFINE_EXCHANGE_PROPERTY` macro (`DEFINE_EXCHANGE_PROPERTY`).

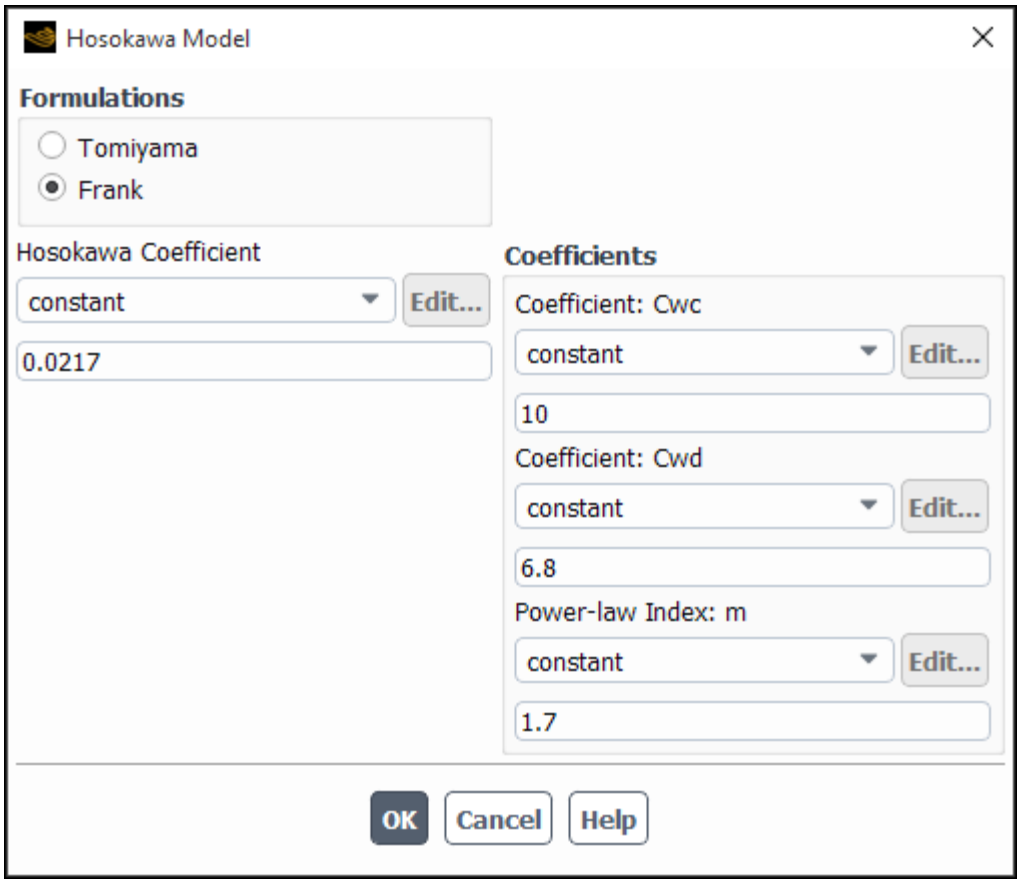
Power-law Index: m

Specify the power law constant, m, in Equation 14.324 in the Theory Guide. You can enter a constant value or specify a user-defined function defined using the `DEFINE_EXCHANGE_PROPERTY` macro (`DEFINE_EXCHANGE_PROPERTY`).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Select **hosakawa** to use the Hosokawa model (Hosokawa Model in the Theory Guide). You can edit the model parameters in the **Hosokawa Model** dialog box (Figure 26.62: Hosokawa Model Dialog Box (p. 3084)).

Figure 26.62: Hosokawa Model Dialog Box



The following model parameters are available:

Formulations

Select which formulation of the Hosokawa model to use. You can select either **Frank** or **Tomiyama**.

Hosokawa Coefficient

Specify the coefficient of the Eotvos number in Equation 14.325 in the Theory Guide. You can enter a constant value or specify a user-defined function defined using the `DEFINE_EXCHANGE_PROPERTY` macro (`DEFINE_EXCHANGE_PROPERTY`).

Coefficients

Specify the model parameters associated with the formulation you selected under **Formulations**.

- Select **user-defined** to use a user-defined function for the wall lubrication coefficient (see `DEFINE_EXCHANGE_PROPERTY` in the *Fluent Customization Manual* for details).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

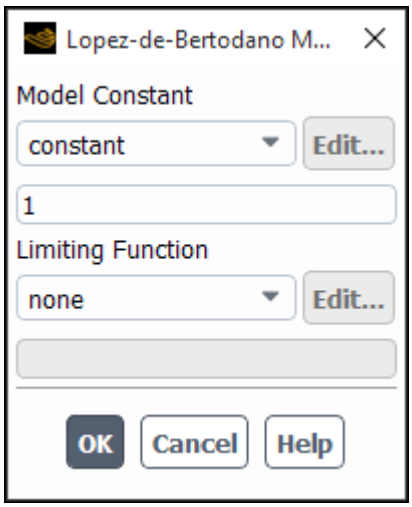
26.5.2.5.6. Including the Turbulent Dispersion Force

For turbulent flows using the Eulerian multiphase model, you can include the effects of turbulent dispersion force ($\vec{F}_{td}$ in Equation 14.194 in the Theory Guide). The turbulent dispersion force acts as a turbulent diffusion in dispersed flows. For details on how turbulent dispersion is modeled in Ansys Fluent see Turbulent Dispersion Force in the Theory Guide.

To include the effects of turbulent dispersion force, for each pair of phases, select the appropriate specification method from the **Turbulent Dispersion** drop-down list. Note that you will specify the turbulent dispersion only for pairs consisting of a secondary phase and the primary phase.

- Select **none** (the default) to ignore the effects of turbulent dispersion.
- Select **lopez-de-bertodano** to use the Lopez de Bertodano model (Lopez de Bertodano Model in the Theory Guide). You can edit the model parameters in the **Lopez de Bertodano Model** dialog box (Figure 26.63: Lopez de Bertodano Model Dialog Box (p. 3085)).

  Figure 26.63: Lopez de Bertodano Model Dialog Box

  The following model parameters are available:

  Model Constant

  Specify the coefficient, C_{TD}, in Equation 14.332 in the Theory Guide. You can enter a constant value or specify a user-defined function defined using the `DEFINE_EXCHANGE_PROPERTY` macro (`DEFINE_EXCHANGE_PROPERTY`).

  Limiting Function

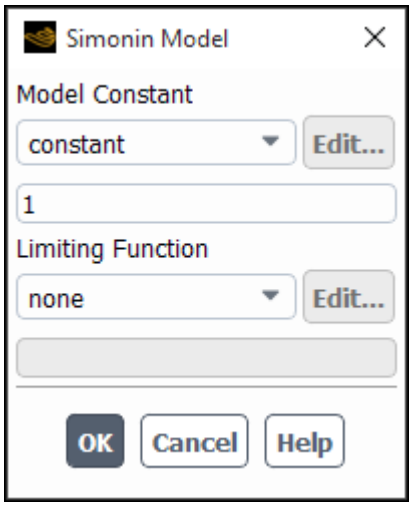
  Specify the limiting function to use. You may select **none**, the **standard** limiting function, or a **user-defined** function using the `DEFINE_EXCHANGE_PROPERTY` macro (`DEFINE_EXCHANGE_PROPERTY`). See Limiting Functions for the Turbulent Dispersion Force in Fluent Theory Guide for details on the limiting functions.

- Select **simonin** to use the Simonin model (Simonin Model in the Theory Guide).

  You can specify the model parameters in the **Simonin Model** dialog box (Figure 26.64: Simonin Model Dialog Box (p. 3086)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.64: Simonin Model Dialog Box



The following model parameters are available:

Model Constant

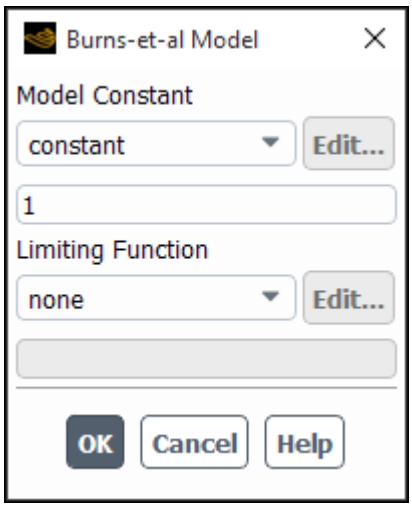
Specify the coefficient, C_{TD}, in Equation 14.334 in the Theory Guide. You can enter a constant value or specify a user-defined function defined using the DEFINE_EXCHANGE_PROPERTY macro (DEFINE_EXCHANGE_PROPERTY).

Limiting Function

Specify the limiting function to use. You may select **none**, the **standard** limiting function, or a **user-defined** function using the DEFINE_EXCHANGE_PROPERTY macro (DEFINE_EXCHANGE_PROPERTY). See Limiting Functions for the Turbulent Dispersion Force in Fluent Theory Guide for details on the limiting functions.

- Select **burns-et-al** to use the Burns et al. model (Burns et al. Model in the Theory Guide). You can edit the model parameters in the **Burns-et-al Model** dialog box (Figure 26.65: Burns et al. Model Dialog Box (p. 3086)).

Figure 26.65: Burns et al. Model Dialog Box



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following model parameters are available:

Model Constant

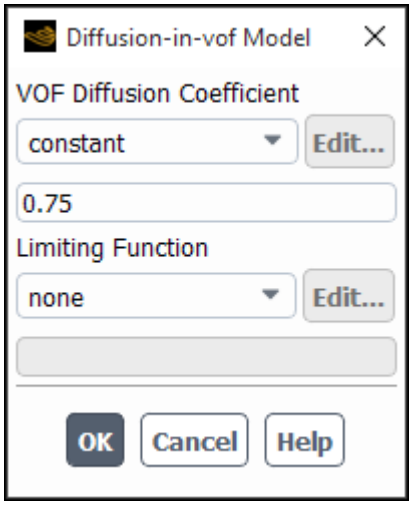
Specify the coefficient, C_{TD}, in Equation 14.337 in the Theory Guide. You can enter a constant value or specify a user-defined function defined using the DEFINE_EXCHANGE_PROPERTY macro (DEFINE_EXCHANGE_PROPERTY).

Limiting Function

Specify the limiting function to use. You may select **none**, the **standard** limiting function, or a **user-defined** function using the DEFINE_EXCHANGE_PROPERTY macro (DEFINE_EXCHANGE_PROPERTY). See Limiting Functions for the Turbulent Dispersion Force in Fluent Theory Guide for details on the limiting functions.

- Select **diffusion-in-vof** to use the Diffusion in VOF model (Diffusion in VOF Model in the Theory Guide). You can edit the model parameters in the **Diffusion-in-vof Model** dialog box (Figure 26.66: Diffusion—in—vof Model Dialog Box (p. 3087)).

Figure 26.66: Diffusion—in—vof Model Dialog Box



The following model parameters are available:

VOF Diffusion Coefficient

Specify the coefficient, σ_q, in Equation 14.340 in the Theory Guide.

Limiting Function

Specify the limiting function to use. You may select **none**, the **standard** limiting function, or a **user-defined** function using the DEFINE_EXCHANGE_PROPERTY macro (DEFINE_EXCHANGE_PROPERTY). See Limiting Functions for the Turbulent Dispersion Force in Fluent Theory Guide for details on the limiting functions.

- Select **user-defined** to use a user-defined function for the turbulent dispersion (see DEFINE_VECTOR_EXCHANGE_PROPERTY in the *Fluent Customization Manual* for details).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.5.2.5.7. Including Surface Tension and Wall Adhesion Effects

As discussed in When Surface Tension Effects Are Important in the Theory Guide, the importance of surface tension effects depends on the value of the capillary number, Ca (defined by Equation 14.32 in the Theory Guide), or the Weber number, We (defined by Equation 14.33 in the Theory Guide). Surface tension effects can be neglected if Ca $\gg 1$ or We $\gg 1$.

Important:

Note that the calculation of surface tension effects will be more accurate if you use a quadrilateral or hexahedral mesh in the area(s) of the computational domain where surface tension is significant. If you cannot use a quadrilateral or hexahedral mesh for the entire domain, then you should use a hybrid mesh, with quadrilaterals or hexahedra in the affected areas. Ansys Fluent also offers an option to use VOF gradients at the nodes for curvature calculations on meshes when more accuracy is desired. For more information, see Surface Tension and Adhesion in the Theory Guide.

If you want to include the effects of surface tension along the interface between one or more pairs of phases, as described in Surface Tension and Adhesion in the Theory Guide, refer to Including Surface Tension and Adhesion Effects (p. 3011).

26.5.2.5.8. Including the Virtual Mass Force

For both granular and non-granular flows, it is possible to include the "virtual mass force" ($\vec{F}_{vm}$ in Equation 14.343 in the Theory Guide) that is present when a secondary phase accelerates relative to the primary phase. The virtual mass effect is significant when the secondary phase density is much smaller than the primary phase density (for example, for a transient bubble column).

To include the effect of the virtual mass force perform the following steps:

1. For each phase pair, select the method for specifying the **Virtual Mass Coefficient**, C_{vm}, in Equation 14.343 in the *Fluent Theory Guide*. You can choose from the following methods:

   none

   Disable virtual mass modeling for the phase pair.

   constant

   Specify a constant value for the coefficient. The default value of `0.5` is typical.

   user-defined

   Use a user-defined function created with the `DEFINE_EXCHANGE_PROPERTY` macro. See `DEFINE_EXCHANGE_PROPERTY` in the *Fluent Customization Manual* for details.

2. If desired, you can enable the **Virtual Mass Implicit** method.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When using the **Virtual Mass Implicit** method, the **Default** option uses the complete term in Equation 14.343. You can also choose **Option 2** or **Option 3**, which use locally truncated forms of the virtual mass terms in cells where divergence is detected.

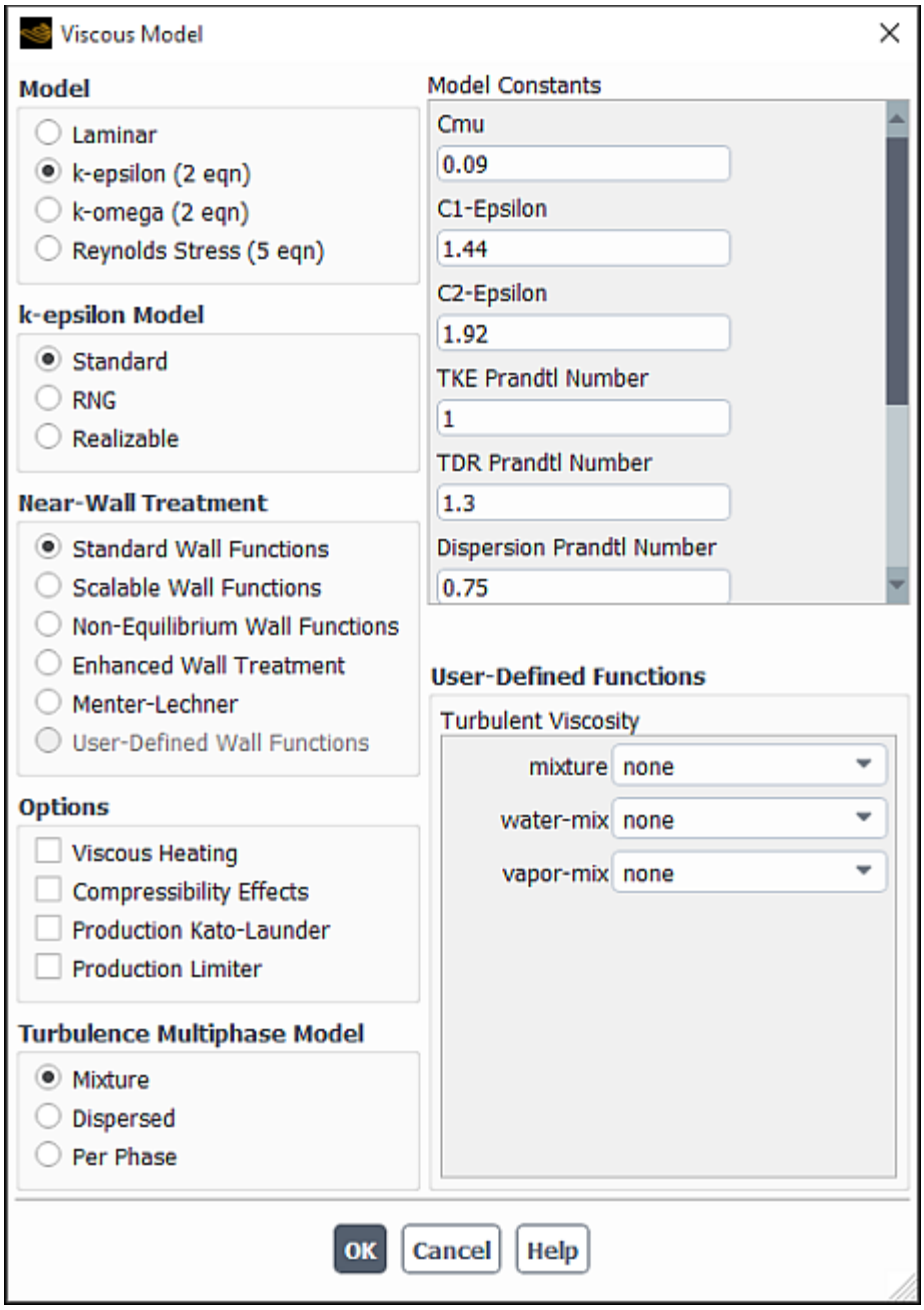
The **Virtual Mass Implicit** method is recommended for steady-state coupled simulations as it offers improved convergence in such cases. The recommended approach is to begin the simulation using **Option 2**. Once the solution has converged enough to establish the basic flow field, switch back to **Default** to include the complete virtual mass term.

26.5.3. Modeling Turbulence

If you are using the Eulerian model to solve a turbulent flow, you will need to choose one of turbulence models described in Turbulence Models in the Theory Guide in the Viscous Model Dialog Box (p. 4715) (Figure 26.67: The Viscous Model Dialog Box for an Eulerian Multiphase Calculation (p. 3090)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.67: The Viscous Model Dialog Box for an Eulerian Multiphase Calculation



The procedure is as follows:

1. Select **k-epsilon**, **k-omega**, or **Reynolds Stress** under **Model**.

2. Select the desired **k-epsilon Model**, **k-omega Model**, or **Reynolds-Stress Model** and any other related parameters, as described for single-phase calculations in Steps in Using a Turbulence Model (p. 2081).

3. Under **Turbulence Multiphase Model** or **RSM Multiphase Model**, indicate the desired multiphase turbulence model (see Turbulence Models in the Theory Guide for details about each):

   - Select **Mixture** to use the mixture turbulence model. This is the default model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Select **Dispersed** to use the dispersed turbulence model. This model is applicable when there is clearly one primary continuous phase and the rest are dispersed dilute secondary phases.
- Select **Per Phase** to use a k- ε or k- ω turbulence model for each phase. This model is appropriate when the turbulence transfer among the phases plays a dominant role.

26.5.3.1. Including Turbulence Interaction Source Terms

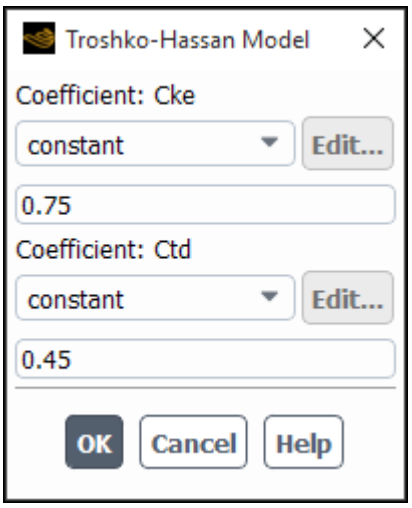
By default, interphase turbulence source terms are not included in the calculation. If you want to include these source terms, you can define them in the **Forces** tab of the Multiphase Model Dialog Box (p. 4700).

Setup → Models → Multiphase → Edit...

For each pair of phases, select the desired correlation for the **Turbulence Interaction** (**Forces Setup** group box):

- select **none** to omit turbulence interaction source terms. This is the default.
- select **troshko-hassan** to use the Troshko-Hassan model described in Troshko-Hassan in the *Fluent Theory Guide*. You can edit the model parameters in the **Troshko-Hassan Model** dialog box (Figure 26.68: Troshko-Hassan Model Dialog Box (p. 3091)).

  Figure 26.68: Troshko-Hassan Model Dialog Box

  The following model parameters are available:

  Coefficient: Cke

  Specify the coefficient, C_{ke}, in the equations in Troshko-Hassan in the *Fluent Theory Guide*.

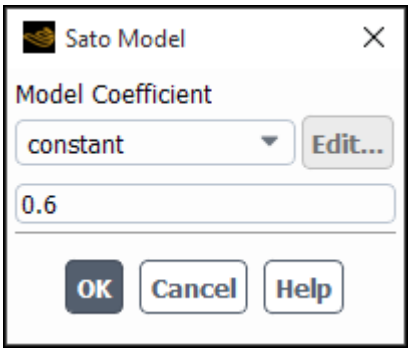
  Coefficient: Ctd

  Specify the coefficient, C_{td}, in the equations in Troshko-Hassan in the *Fluent Theory Guide*.

- select **sato** to use the Sato model described in Sato in the *Fluent Theory Guide*. You can edit the model parameters in the **Sato Model** dialog box (Figure 26.69: Sato Model Dialog Box (p. 3092)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.69: Sato Model Dialog Box



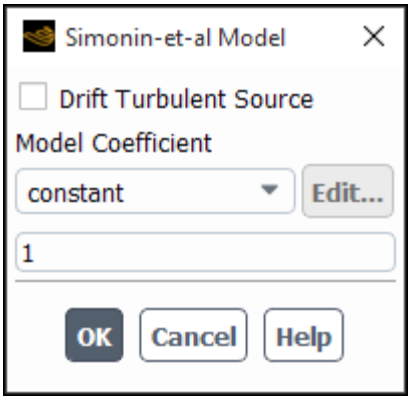
The following model parameters are available:

Model Coefficient

Specify the coefficient, $C_{\mu,p}$, in the equations in Sato in the *Fluent Theory Guide*.

- select **simonin-et-al** to use the Simonin et al. model described in Simonin et al. in the *Fluent Theory Guide*. This option is available only when **Dispersed** or **Per Phase** is selected in the **Viscous Model** dialog box. You can edit the model parameters in the **Simonin-et-al Model** dialog box (Figure 26.70: Simonin-et-al Model Dialog Box (p. 3092)).

Figure 26.70: Simonin-et-al Model Dialog Box



The following model parameters are available:

Drift Turbulent Source

include the portion the turbulent kinetic energy source term arising from the drift velocity.

Model Coefficient

Specify the coefficient, C_s, in the equations in Simonin et al. in the *Fluent Theory Guide*.

If you decide to enable turbulence interaction for multiple phase pairs, it is recommended that you avoid mixing the different models and that you select the same model for each phase pair with turbulence interaction enabled.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that the inclusion of these terms can slow down convergence noticeably. If you are looking for additional accuracy, you may want to compute a solution first without these sources, and then continue the calculation with these terms included. In most cases these terms can be neglected.

26.5.3.2. Customizing the k-ε Multiphase Turbulent Viscosity

If you are using the k- ε multiphase turbulence model, a user-defined function can be used to customize the turbulent viscosity for each phase. This option will enable you to modify μ_t in the k- ε model. For more information, see the Fluent Customization Manual.

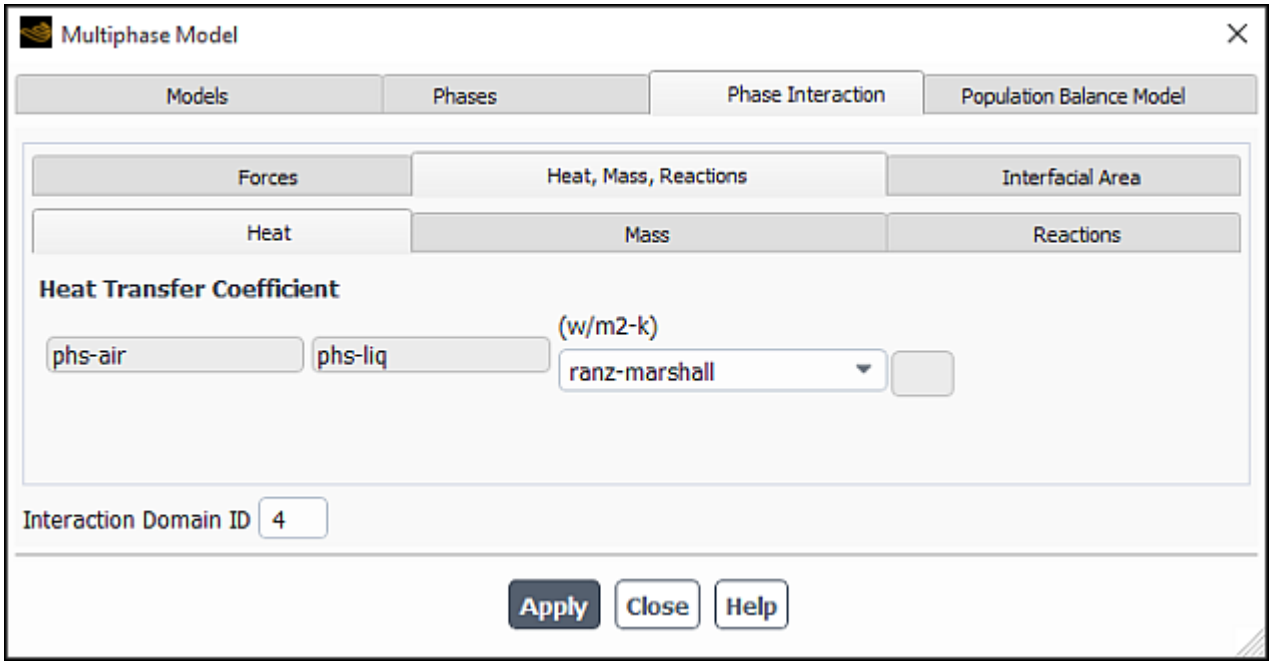
In the **Viscous Model** dialog box, under **User-Defined Functions**, select the appropriate user-defined function in the **Turbulent Viscosity** drop-down list.

26.5.4. Including Heat Transfer Effects

To define heat transfer in a multiphase Eulerian simulation, after you have enabled the energy equation in the **Energy** dialog box, you need to visit the **Phase Interaction** > **Heat, Mass, Reactions** > **Heat** tab in the **Multiphase Model** dialog box.

Setup → **Models** → **Multiphase** **Edit...**

Figure 26.71: The Multiphase Model Dialog Box for Heat Transfer



1. Select the desired correlation for the **Heat Transfer Coefficient**. See The Heat Exchange Coefficient in the *Fluent Theory Guide* for more information. The available choices are as follows and depend on what other models you have enabled:

   constant-htc

   allows you to specify a constant value for the volumetric heat transfer coefficient. See Constant in the *Fluent Theory Guide* for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

nusselt-number

allows you to specify a value for the Nusselt number from which the heat transfer coefficient will be computed. See Nusselt Number in the *Fluent Theory Guide* for details.

gunn

is frequently used for Eulerian multiphase simulations involving a granular phase. See Gunn Model in the *Fluent Theory Guide* for details.

ranz-marshall

is frequently used for Eulerian multiphase simulations not involving a granular phase. See Ranz-Marshall Model in the *Fluent Theory Guide* for details.

hughmark

is an extension of Ranz-Marshall to a wider range of Reynolds Number. See Hughmark Model in the *Fluent Theory Guide* for details.

tomiyama

is frequently used for Eulerian multiphase simulations of bubbly flows with relatively low Reynolds number. See Tomiyama Model in the *Fluent Theory Guide* for details.

none

allows you to ignore the effects of heat transfer between the two phases.

user-defined

allows you to implement a correlation reflecting a model of your choice, through a user-defined function.

fixed-to-sat-temp

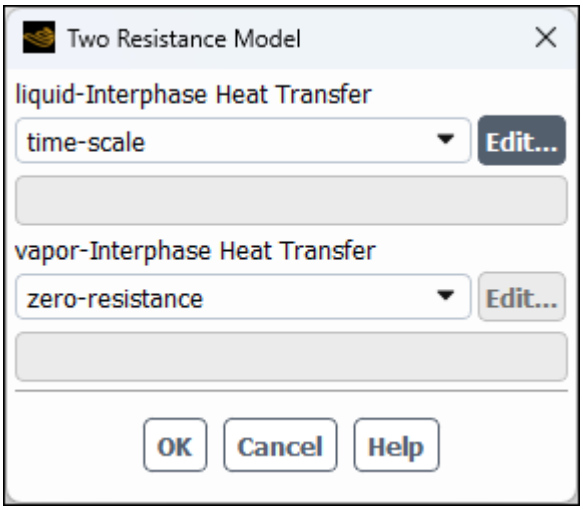
models heat transfer when all of the heat transfer to a phase interface is used in mass transfer. It assumes that the temperature at the **To-Phase** in the mass transfer is equal to the saturation temperature. Note that this model is available only when **Boiling** is disabled in the **Models** tab of the **Multiphase Model** dialog box.

two-resistance

opens the **Two Resistance Model** dialog box where you can independently specify the heat transfer coefficient correlations for the two phases.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.72: The Two Resistance Model Dialog Box



In addition to heat transfer coefficient models described above, the following sub-models of the two-resistance model are available:

time-scale

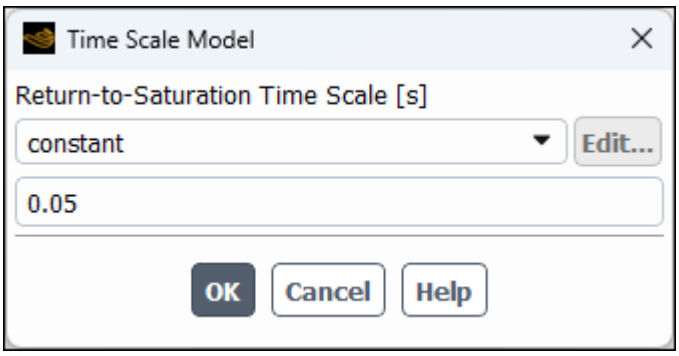
models the gas-interface heat transfer coefficient. It assumes that the gas phase retains the saturation temperature by rapid evaporation or condensation.

zero-resistance

when selected for one of the phases, the phase temperature will be equal to the interfacial temperature. Note that you should not select this option for both interphase heat transfer coefficients.

When **time-scale** is selected, the **Time Scale Model** dialog box opens automatically where you need to specify a positive non-zero value for the **Return-to-Saturation Time Scale** (δt in Equation 14.396 in the *Fluent Theory Guide*).

Figure 26.73: The Time Scale Model Dialog Box



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This option is recommended when using the evaporation-condensation mass transfer model. See Two-Resistance Model in the *Fluent Theory Guide* for details.

Note:

When using multiple boiling mechanisms (such as in boiling coupled with the in-homogeneous PBM), the **two-resistance** heat transfer option is not recommended for phase pairs with the boiling mass transfer option as it may result in incorrect solution. Instead, use the two-resistance boiling framework that is enabled via the Text User Interface (TUI) as described in Including the Boiling Model (p. 3113).

2. Set the appropriate thermal boundary conditions. You will specify the thermal boundary conditions for each individual phase on most boundaries, and for the mixture on some boundaries. See Cell Zone and Boundary Conditions (p. 1299) for more information on boundary conditions, and Eulerian Model (p. 2968) for more information on specifying boundary conditions for a Eulerian multiphase calculation.

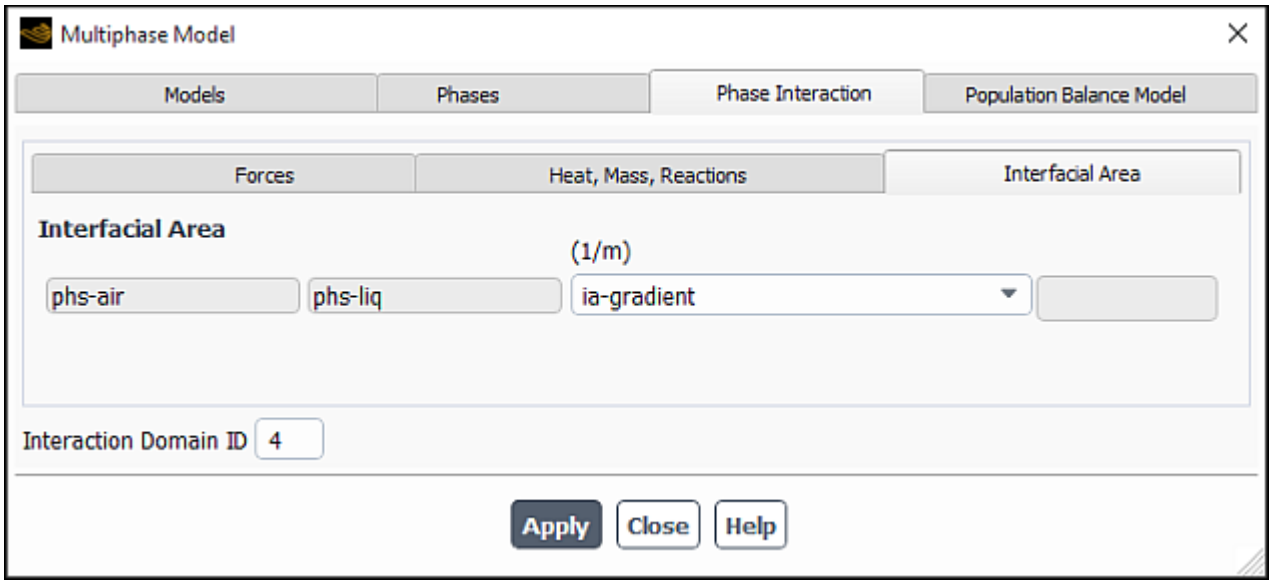
See Description of Heat Transfer in the Theory Guide for more information on heat transfer in the framework of a Eulerian multiphase simulation and the available models.

26.5.5. Using an Algebraic Interfacial Area Model

If you have chosen not to solve the transport equation for Interfacial Area Concentration (Defining the Interfacial Area Concentration (p. 3072)), you can select an algebraic model to estimate the interfacial area from the secondary phase diameter specified in the **Interfacial Area** tab of the **Multiphase Model** dialog box. To choose an algebraic interfacial area model perform these steps.

1. In the **Multiphase Model** dialog box, go to the **Phase Interaction** > **Interfacial Area** tab box (for example, Figure 26.74: The Multiphase Model Dialog Box for Interfacial Area (p. 3096)).

Figure 26.74: The Multiphase Model Dialog Box for Interfacial Area



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. In the **Interfacial Area** tab, select the desired algebraic model for the **Interfacial Area**. Note the following regarding the available choices:

   ia-symmetric

   considers both the primary and secondary phase volume fractions in estimating the interfacial area.

   ia-particle

   considers only the secondary phase volume fraction in estimating the interfacial area.

   ia-gradient

   considers the volume fraction gradient at the interface between two phases in estimating the interfacial area.

   Additional options are available if you have enabled one of the boiling models. See Including the Boiling Model (p. 3113).

See Interfacial Area Concentration in the *Fluent Theory Guide* for details about the algebraic interfacial area models.

26.5.6. Using the Algebraic Interfacial Area Density (AIAD) Model

26.5.6.1. Limitations

The Algebraic Interfacial Area Density (AIAD) model is available only within the Eulerian multiphase framework and cannot be used with the following models:

- Dense Discrete Phase Model (DDPM)
- Boiling model

26.5.6.2. Procedure for Setting the AIAD Model

To use the Algebraic Interfacial Area Density (AIAD) model, follow the steps outlined below. Only the steps that are pertinent to the AIAD modeling are listed here. For the theory behind the AIAD model, see Algebraic Interfacial Area Density (AIAD) Model in the *Fluent Theory Guide*.

1. In the **Multiphase Model** dialog box, select **Eulerian** (in the **Model** group box) and **Algebraic Interfacial Area Density (AIAD)** (in the **Regime Transition Modeling** group box).

   Setup → **Models** → **Multiphase** **Edit...**

   Important:

   Only two Eulerian phases are required for the AIAD model. However, if you want to model phase entrainment, then at least one additional secondary phase must be defined in your simulation.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Phase entrainment and absorption are modeled through the **aiad-entrained-absorption** mass transfer mechanism as further described. For example, if you want to model droplet entrainment, define gas as a primary phase, liquid as a secondary continuous phase, and droplets as a secondary entrained phase. Bubble entrainment can be modeled in a similar manner. In bubble entrainment modeling, define liquid as a primary continuous phase, gas as a secondary continuous phase, and bubbles as a secondary entrained phase. The AIAD model implementation requires both secondary phases to be of the same material.

2. Click **Apply**.

   Important:

   When setting up your case, if you have made changes in the current tab, you should click the **Apply** button to make them effective before moving to the next tab. Otherwise, the relevant models may not be available in the other tabs, and your settings may be lost.

   Once the AIAD model is applied, Ansys Fluent automatically selects the following options in the **Multiphase Model** dialog box:

   - **Multi-Fluid VOF Model** (**Hybrid Models** group box)
   - **Sharp/Dispersed** interface modeling type (**Options** group box)
   - **Phase Localized Discretization** (Interface **Modeling Options** dialog box)

   Note:

   Although these options are displayed in the **Models** tab, they cannot be accessed when the **Algebraic Interfacial Area Density (AIAD)** is selected.

3. In the **Phases** tab, set the phases for your simulation.

   a. Define the primary and secondary phases as appropriate. Note that **AIAD Continuous Phase Treatment (Primary)** is automatically selected for the primary phase.

   b. For one of the secondary phases, assign a different phase material than that of the primary phase and then select the **AIAD Continuous Phase Treatment (Secondary)** option.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

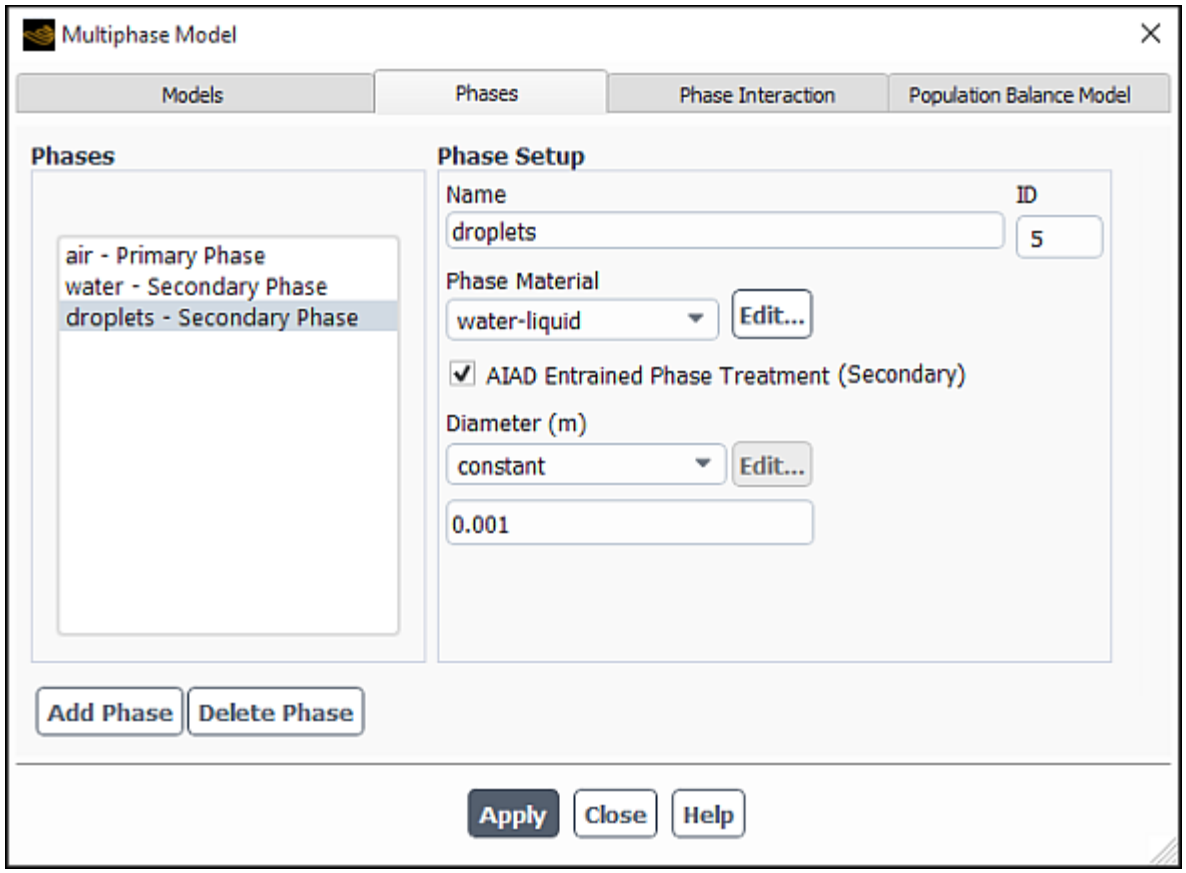
Figure 26.75: Defining a Secondary Phase with the AIAD Continuous Phase Treatment

c. If you are modeling phase entrainment, then define the other secondary phase as follows:

   i. Assign the same phase material as for the AIAD secondary continuous phase you defined in the previous step.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.76: Example of Defining a Secondary Entrained Phase for Droplets



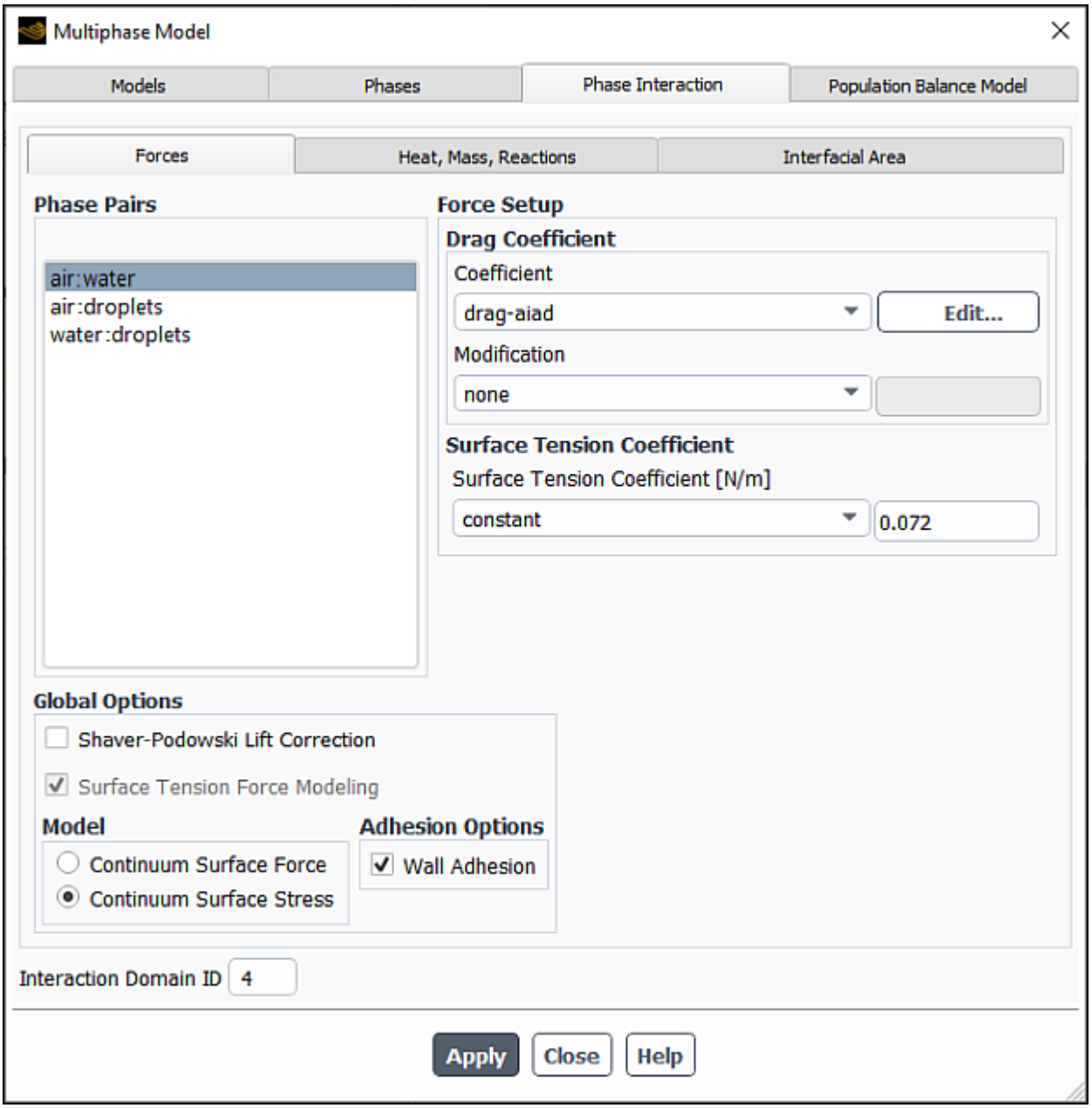
ii. Select the **AIAD Entrained Phase Treatment (Secondary)** option.

iii. Enter the bubble diameter in the **Diameter** field.

d. Click **Apply**.

4. In the **Phase Interaction** tab, define the interaction between the phases.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



a. In the **Forces** tab, set the interfacial forces between the phase pairs.

   Note the following when using the AIAD model:

   - For the pair of the primary and AIAD secondary continuous phases, you can define only the drag and surface tension coefficients.
   - The pair of the primary and secondary entrained phases has no restrictions in terms of setting interfacial forces.
   - The two phases of the same material (AIAD continuous and entrained) interact via entrainment and deposition. No forces should be defined for such phase pairs.
   - Once the **Algebraic Interfacial Area Density (AIAD)** model is applied, Ansys Fluent automatically selects the following options:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

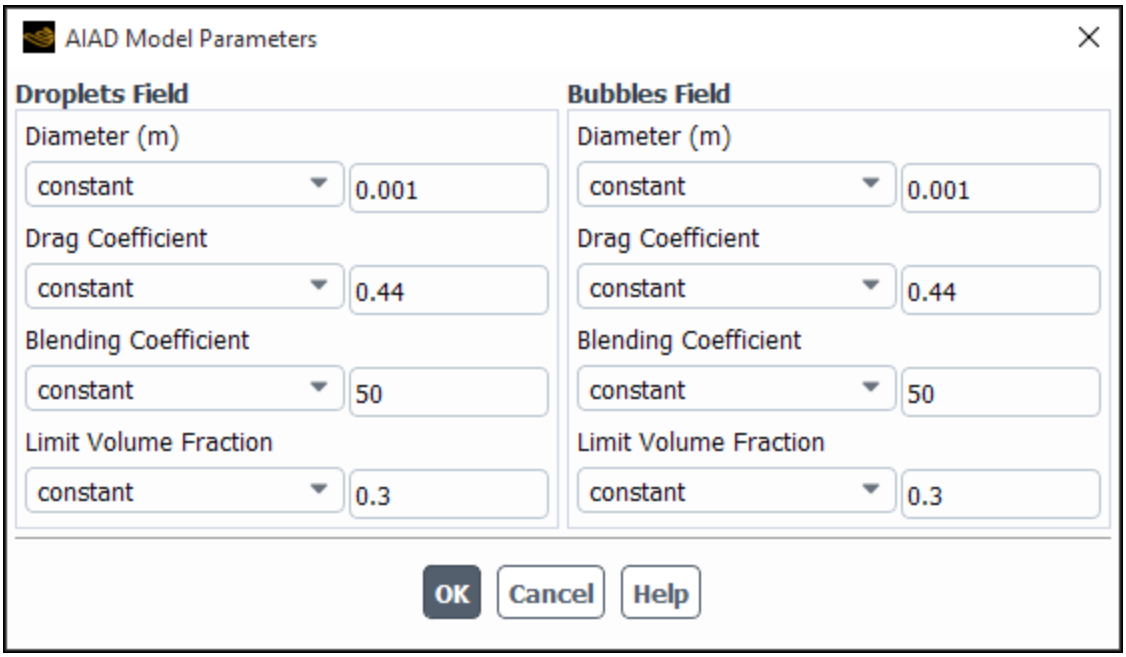
– **Surface Tension Force Modeling** and **Continuum Surface Stress** (**Global Options** group box)

  Although the **Surface Tension Force Modeling** option cannot be overridden, you can select a surface tension and adhesion methods suitable for your simulation.

– **drag-aiad** for the drag **Coefficient** for the pair of the primary and AIAD secondary continuous phases

  You can click **Edit...** next to the **Coefficient** text entry field and adjust values of the AIAD model parameters in the **AIAD Model Parameters** dialog box.

Figure 26.77: The AIAD Model Parameters Dialog Box



You can specify the following parameters in the **Droplets Field** and **Bubbles Field** group boxes:

→ **Diameter**: d_B and d_D in Equation 14.458 and Equation 14.459 in the *Fluent Theory Guide*

→ **Drag Coefficient**: $C_{D,B}$ and $C_{D,D}$ in Equation 14.470 in the *Fluent Theory Guide*

  You can specify the drag coefficients for both droplet and bubbles files using either constant values or the **user-defined** option and use the **DEFINE_EXCHANGE_PROPERTY** user-defined function to define these inputs. See section "DEFINE_EXCHANGE_PROPERTY" in the Fluent Customization Manual for details.

→ **Blending Coefficient**: a_D and a_B in Equation 14.455 in the *Fluent Theory Guide* and Equation 14.456 in the *Fluent Theory Guide*, respectively

→ **Limit Volume Fraction**: $\alpha_{D,limit}$ and $\alpha_{B,limit}$ in Equation 14.455 in the *Fluent Theory Guide* and Equation 14.456 in the *Fluent Theory Guide*, respectively

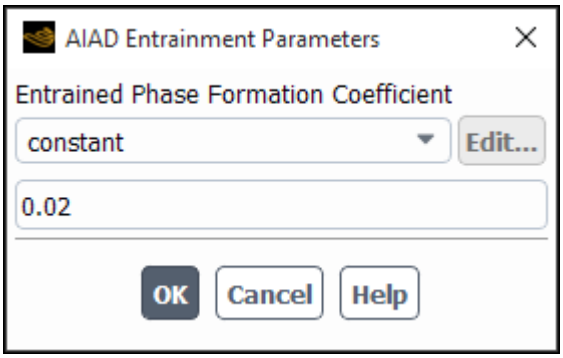
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The default values for **Drag Coefficient**, **Blending Coefficient**, and **Limit Volume Fraction** are suitable for most cases that involve the AIAD model.

5. If you are modeling phase entrainment, then in the **Heat, Mass, Reactions** tab, Ansys Fluent automatically selects the **aiad-entrained-absorption** mechanism for mass transfer from the AIAD secondary continuous phase to the entrained secondary phase. The mass transfer rate is positive when the entrainment mechanism is dominant and negative when the absorption mechanism is dominant.

   If you want to adjust a value for C_{ent} in Equation 14.478 in the *Fluent Theory Guide*), click **Edit...** next to the selected mechanism and enter a new value in the **Entrained Phase Formation Coefficient** text entry field in the **AIAD Entrainment Parameters** dialog box.

   Figure 26.78: The AIAD Entrainment Parameters Dialog Box

   For more information about the **aiad-entrained-absorption** mass transfer mechanism, see Modeling Entrainment-Absorption in the *Fluent Theory Guide*.

   Note:

   The **aiad-entrained-absorption** mechanism cannot be removed unless the **AIAD Entrained Phase Treatment (Secondary)** option is cleared for the relevant phase in the **Phases** tab.

6. In the **Interfacial Area** tab, select the interfacial area method for phase pairs.

   Ansys Fluent automatically assigns the **ia-aiad** interfacial area algebraic model for the AIAD phase pair. This is the only model available for the AIAD phase pair. There are no restrictions for the other phase pairs.

7. If you are using the entrained phase in your simulation, you can set up the population balance model in the **Population Balance Model** tab to predict the growth of the droplet phase and entrained particle distribution. You can use the aggregation and breakage kernels from and to the entrained phase.

   For information on how to set up the population balance simulation, see Population Balance Model (p. 3131).

8. In the **Viscous Model** dialog box, the **ANSYS Fluent** solver has automatically enables the following models and options:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- k-omega SST turbulence model
- **Turbulence Damping** (**Turbulence Damping Options** group box)

  This option affects only the continuous phases (primary and AIAD secondary continuous phases)
- **Per Phase** (**Options** group box)
- **Subrgid Turbulence Contribution (AIAD)** (**Regime Transition Modeling** group box)

  This option adds an extra source to the turbulent kinetic energy. For more information, see Modeling Sub-grid Wave Turbulence Contribution (SWT) in the *Fluent Theory Guide*.

These models and options are best suitable for simulations that involve the AIAD model; however, you can override these settings (except for the **Per Phase** option) for your current simulation.

9. Ansys Fluent automatically enables gravity for the AIAD model. Make sure that the gravitational field is correctly specified in the **General** task page.
10. Due to the nature of the problems that involve the AIAD model, the **Transient** solver has been automatically selected for the AIAD model. However, you can use the steady-state solver for your current AIAD case. Selecting **Global Time Step** from the **Pseudo Time Method** list (in the **Solution Methods** task page) is recommended for such cases, which is available with the **Coupled** scheme for the pressure-velocity coupling.
11. In the **Solution Controls** task page, the **Under-Relaxation Factors** have been changed to more conservative values as recommended for the AIAD model. Depending on your problem and the convergence behavior of your simulation, you can adjust these values to better suit your specific case.

26.5.6.3. Solution Strategies

If a convergence problem arises when solving a case that involves the phase entrainment model, you can try running the simulation without entrainment, and then continuing running with the **AIAD Entrained Phase Treatment (Secondary)** option, which automatically adds the **aiad-entrainment-absorption** mass transfer mechanism. You can also lower the **Vaporization Mass** relaxation factor (default: 1.0). This can be reduced to as low as 0.2, depending of the complexity of the problem.

To improve accuracy, you can increase the under-relaxation factors for your AIAD simulation depending on the convergence behavior of the calculation.

26.5.7. Using the Generalized Two Phase Flow (GENTOP) Model

26.5.7.1. Limitations

The GENTOP model is available only within the Eulerian multiphase framework and cannot be used with the following models:

- Multi-fluid VOF model

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

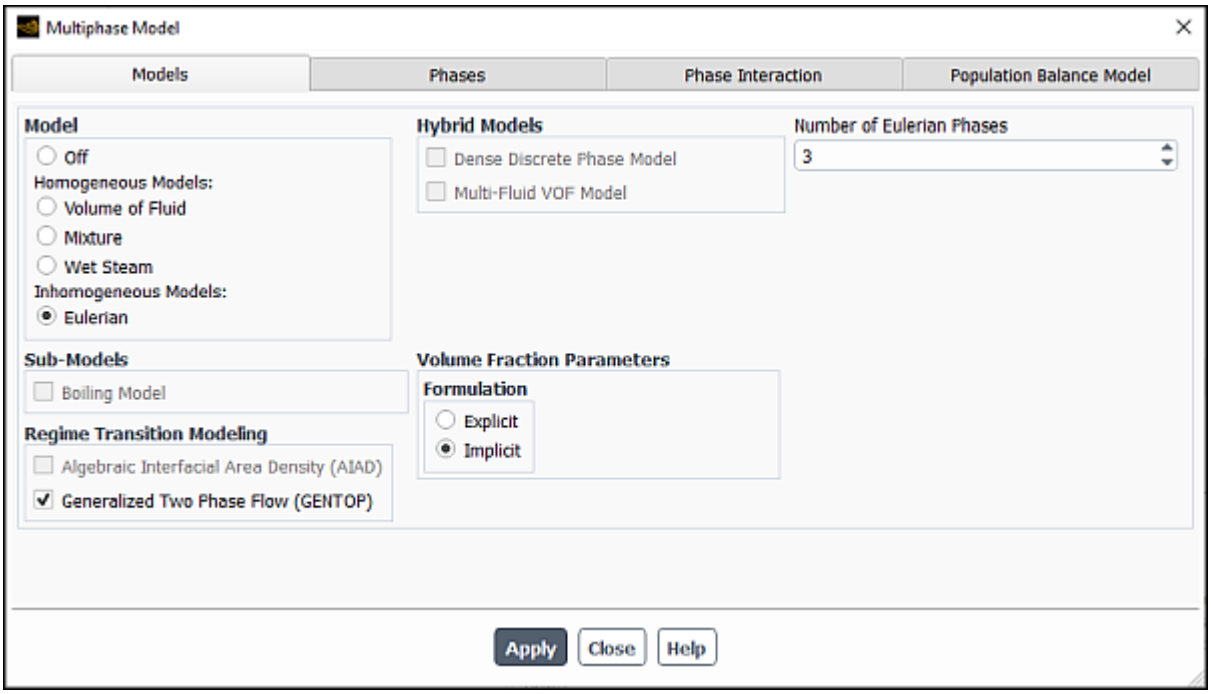
- Dense discrete phase model (DDPM)
- Boiling model
- Species transport model

26.5.7.2. Steps for Using the GENTOP Model

The procedure for setting up a flow regime transition simulation using the GENTOP model is described below. Note that only steps that are pertinent to the GENTOP model are shown. For theoretical information about the GENTOP model, see Generalized Two Phase (GENTOP) Flow Model in the *Fluent Theory Guide*.

1. In the **Multiphase Model** dialog box, select the **Eulerian** inhomogeneous model and **Generalized Two Phase Flow (GENTOP)** (**Regime Transition Modeling** group box).

   Setup → **Models** → **Multiphase** **Edit...**



   Ansys Fluent automatically makes the following changes:

   - Adds three Eulerian phases

     You can include more than two secondary phases in your simulation.

   - In the **Heat, Mass, Reactions** tab, automatically selects the **gentop-complete-coalescence** mechanism for transferring mass from secondary non-GENTOP phases to the GENTOP phase.
   - Applies the following settings to the population balance model:

     – Enables the **Inhomogeneous Discrete** method

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– Defines **Bins** for all relevant active secondary phases. For the GENTOP phase, only one bin with the largest diameter size is assigned.

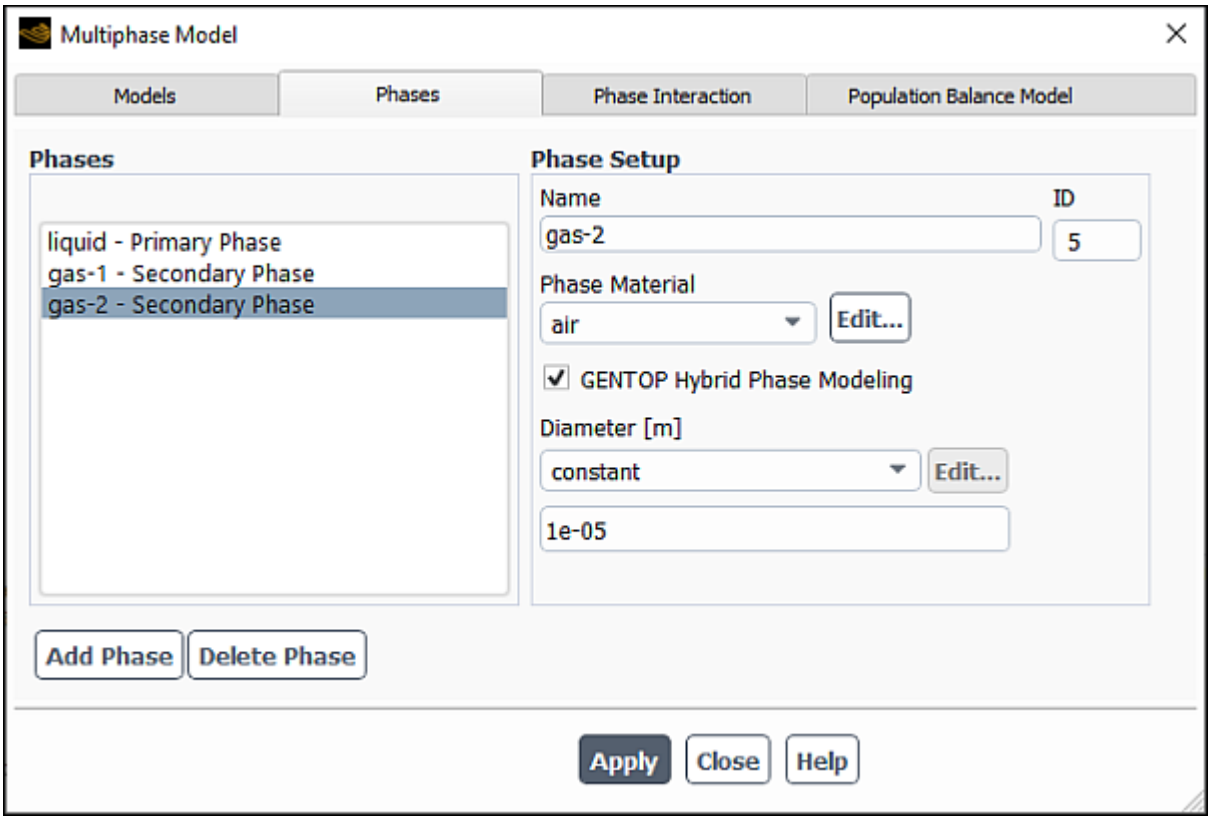
– Enables the Aggregation and Breakage kernels to allow the growth to occur between the active secondary phases and the GENTOP phase.

Note that these settings cannot be modified.

2. Specify the **Calibration Coefficient** (c_{clust} in Equation 14.483 in the *Fluent Theory Guide*). The default value of 1.0 is acceptable for most regime transition / mixing problems.

3. Click **Apply**.

4. In the **Phases** tab, define the phases for your simulation.



Note the following:

- For one of the secondary phases, you must select **GENTOP Hybrid Phase Modeling**.
- All the secondary phases must share the same material, which must be different than that of the primary phase.

5. Click **Apply**.

6. In the **Phase Interaction** tab, define the interaction between the phases.

   a. In the **Forces** tab, set the interfacial forces between the phase pairs.

      When setting the primary-GENTOP phase pair, note the following:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Only **ishii-zuber**, **grace**, and **universal-drag** are available for the **Drag Coefficient** specification.
- There is no limitations on using lift, wall lubrication, and virtual mass models as well as all options for the surface tension coefficient that are available in Ansys Fluent.
- Turbulent dispersion and turbulence interaction models are not available.

For other phase pairs, no limitation exists on using the Ansys Fluent force interaction models.

b. Click **Apply**.

c. In the **Population Balance Model** tab, check and modify the available settings as needed (such as the number of bins, breakage and aggregation kernels and submodels, and so on).

d. Click **Apply**.

e. Proceed with the simulation as usual.

26.5.7.3. Solution Strategies

If any convergence issues arise, you can try to lower the relaxation factors and follow the solution strategies for improving convergence for Eulerian flows outlined in Eulerian Model (p. 3205).

You can also try using the **Coupled** solver to improve solver robustness.

To improve accuracy, you can decrease the default under-relaxation factors, depending on the convergence behavior of the calculation.

26.5.8. Including the Dense Discrete Phase Model

If you are using the Eulerian multiphase model (Setting Up the Eulerian Model (p. 3065)), you have the option of including the dense discrete phase mode (DDPM) (Dense Discrete Phase Model in the Theory Guide).

Important:

- Enabling this model automatically enables the DPM model. You will notice that **Interaction with Continuous Phase** in the **Discrete Phase Model** dialog box is enabled.
- The dense discrete phase model is available only with the **Eulerian** multiphase model. The limitations that currently apply to DPM and Eulerian multiphase models also apply to the dense discrete phase model. See Limitations of the Eulerian Model in the *Fluent Theory Guide* and Limitations (p. 2674) for limitations that exist with the DPM and Eulerian multiphase models.

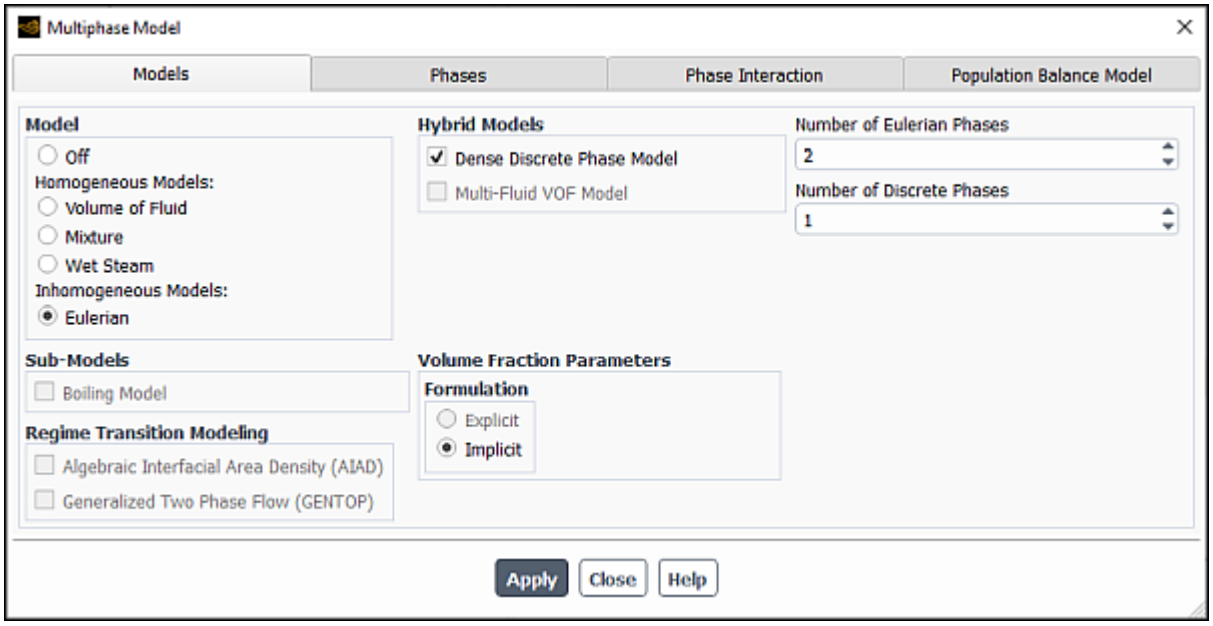
The required work flow when using the dense discrete phase model is as follows:

1. In the **Multiphase Model** dialog box, enable the DDPM (**Models** tab) (Figure 26.79: The Dense Discrete Phase Model - Models Tab (p. 3108)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setup → **Models** → **Multiphase** → **Edit...**

Figure 26.79: The Dense Discrete Phase Model - Models Tab



a. Select **Dense Discrete Phase Model** in the **Hybrid Models** group box.

b. Specify the **Number of Discrete Phases** that are present in your case.

c. Click **Apply**.

Once the DDPM is enabled, Ansys Fluent automatically applies the following changes in the **Multiphase Model** dialog box:

- **averaged-discrete-phase-drag** is selected for the **Drag Coefficient** (**Phase Interaction** > **Forces** tab).

  averaged-discrete-phase-drag corresponds to K_{DPM} in Equation 14.498 in the *Fluent Theory Guide* and indicates that the interaction between discrete and fluid phases will be computed based on averaged values of fluid drag of all particles crossing the fluid cell. That is, at each time step, the drag is evaluated for each particle using the drag law that you have selected in the **Set injection Properties** dialog box and then averaged based on the residence time of each particle in the cell.

- (cases with energy transfer) **averaged-discrete-phase-heat** is selected for the **Heat Transfer Coefficient** (**Phase Interaction** > **Heat, Mass, Reactions** > **Heat** tab).

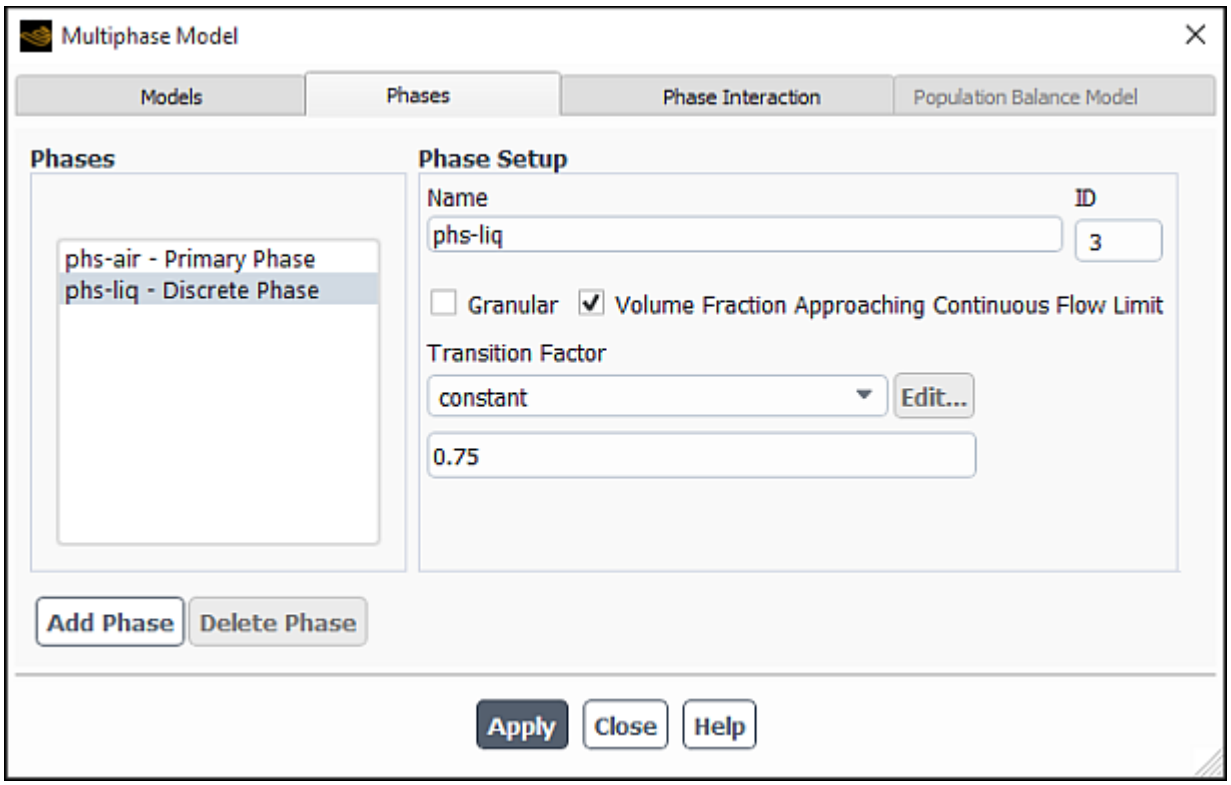
  averaged-discrete-phase-heat corresponds to the heat exchange coefficient h_{DPM} in Equation 14.500 in the *Fluent Theory Guide* and indicates that the interaction between the discrete and fluid phases will be computed based on the averaged heat transfer between the fluid and all particles crossing the fluid cell. The heat transfer law that is used to evaluate the heat exchange is the law that you have selected in the **Set injection Properties** dialog box. The contribution of each particle to the averaged heat transfer is based on the residence time of the particle in the cell.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. In the **Phases** tab, define the phases.

   a. Define the discrete phase, by selecting the phase from the **Phases** selection list that is labeled **Discrete Phase**. Then set up the properties in the **Phase Setup** group box (for example, Figure 26.80: The Multiphase Model Dialog Box for DDPM (p. 3109)).

   Figure 26.80: The Multiphase Model Dialog Box for DDPM



   Note that for non-granular flow, no additional inputs are required here (since the material is set automatically in the background, and the diameter is part of the solution).

   b. To prevent particle concentration from becoming unphysically high, enable **Volume Fraction Approaching Continuous Flow Limit** and specify **Transition Factor**. The criterion of transition from the standard method to a special treatment is specified as **Transition Factor** multiplied by the theoretical close-packing limit for mono-sized spheres ($\pi/3\sqrt{2} \approx 3/4$). If the particle volume fraction exceeds this transition volume fraction, Ansys Fluent will apply a special treatment to the particle momentum equation. For more information, refer to Dense Discrete Phase Model in the *Fluent Theory Guide*.

   The default value for the **Transition Factor** is 0.75, giving a transition volume fraction of 0.5625 (=0.75*3/4). Note that the **Transition Factor** is not limited to a range between 0 and 1. In other words, you can specify values outside this range. For example, a value of 1.2 will give a transition volume fraction of 0.9. You can also specify locally variable values using the `DEFINE_PROPERTY` user-defined macro, depending on the local particle size distribution, as an example.

   See `DEFINE_PROPERTY` UDFs in the Fluent Customization Manual for information about the `DEFINE_PROPERTY`.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. Define the primary and secondary phases, as described in Defining the Phases for the Eulerian Model (p. 3066).

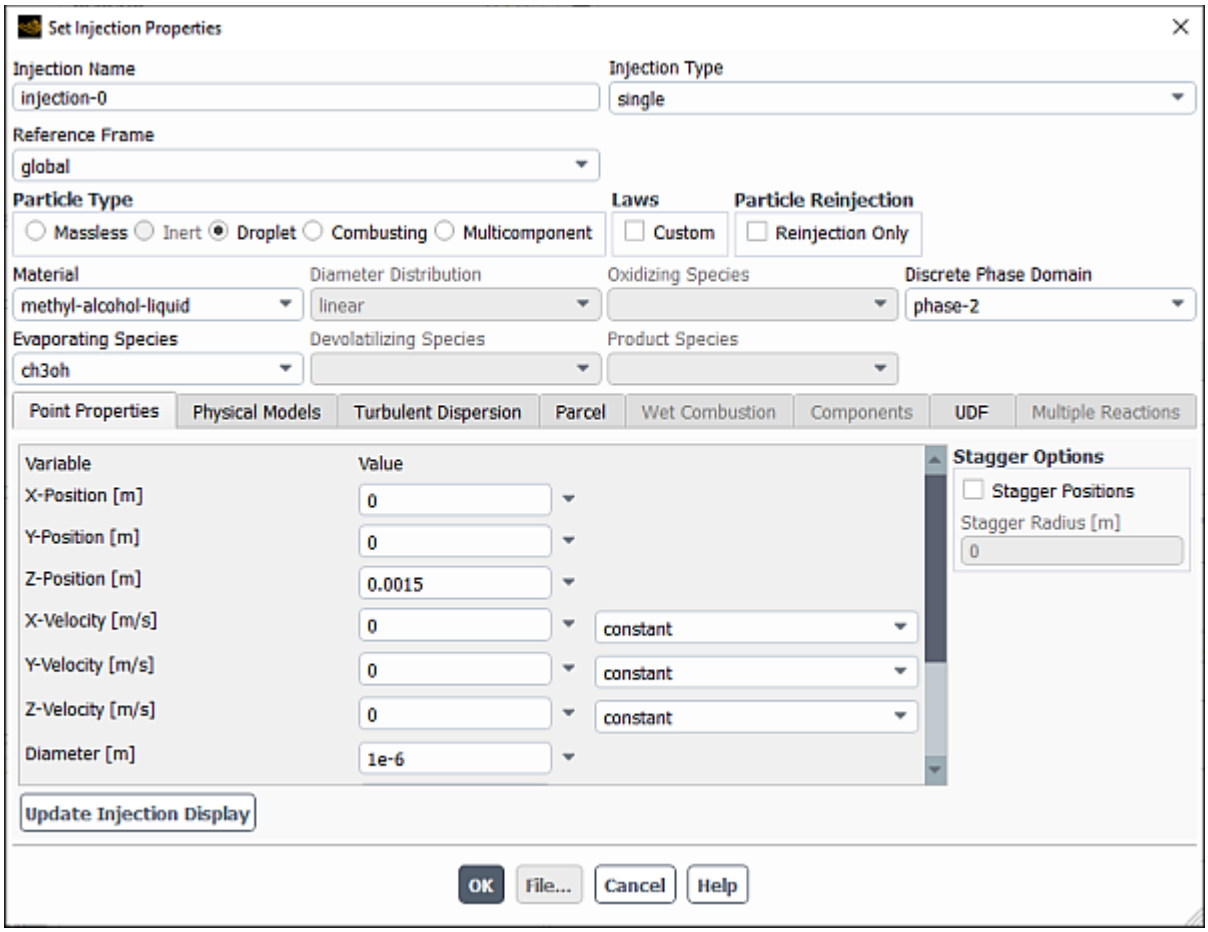
3. Define the injections using the **Injections** dialog box.

Setup → Models → Discrete Phase → Injections → New...

a. Create a new injection by clicking the **Create** button in the **Injections** dialog box, or edit an existing injection by selecting the injection from the **Injections** list and clicking the **Set...** button.

b. In the **Set Injection Properties** dialog box that opens (Figure 26.81: The Set Injection Properties Dialog Box (p. 3110)), specify properties for each injection as described in Setting Initial Conditions for the Discrete Phase (p. 2718)

Figure 26.81: The Set Injection Properties Dialog Box



c. Select the discrete phase from the **Discrete Phase Domain** drop-down list.

4. (cases with the energy equation enabled) By default, Ansys Fluent uses the heat exchange coupling formulation described in Equation 14.499 in the *Fluent Theory Guide* to model the energy interaction with the primary phase. One part of the energy transfer is provided via Q_{DPM}, and the rest is included in the source term $S_{DPM,explicit}$.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For simulations with poly-dispersed particles that use the default heat exchange coupling, it is recommended that you enable the enhanced Eulerian interphase exchange using the following text command:

```
define/models/dpm/interaction/ddpm-enhanced-inter-phase-exchange?
Apply enhanced inter-phase exchange? [no] yes
```

As an alternative to the default coupling formulation, you can use the following text command options for the energy interaction with the primary phase:

- Energy coupling purely via the source terms:

  ```
  define/models/dpm/interaction/ddpm-energy-coupling-via-source-term?
  Establish energy coupling in DDPM via source term? [no] yes
  ```

  With this option enabled, the interaction is achieved purely through an explicit source term $S_{DPM,explicit}$ in Equation 14.499 in the *Fluent Theory Guide*. The heat exchange term Q_{DPM} is not used for this coupling. This option can accelerate the solution for flows with either low to moderate particle load or small particles.

- Energy coupling via the linearized source terms:

  ```
  define/models/dpm/interaction/linearized-dpm-source-terms?
  Linearize the DPM source terms? [no] yes
  ```

  This option may help stabilize the solution in cases where it is unstable due to the high particle volume fraction and/or the high particle-to-fluid density ratio. This coupling can be used only when the energy coupling via the source terms is enabled. Otherwise, the default coupling formulation will be used for all injections with the **Discrete Phase Domain** set to anything other than **none**, and the source term linearization will be applied only to non-DDPM injections.

5. Define the material properties for each injection.

   Setup → **Materials**

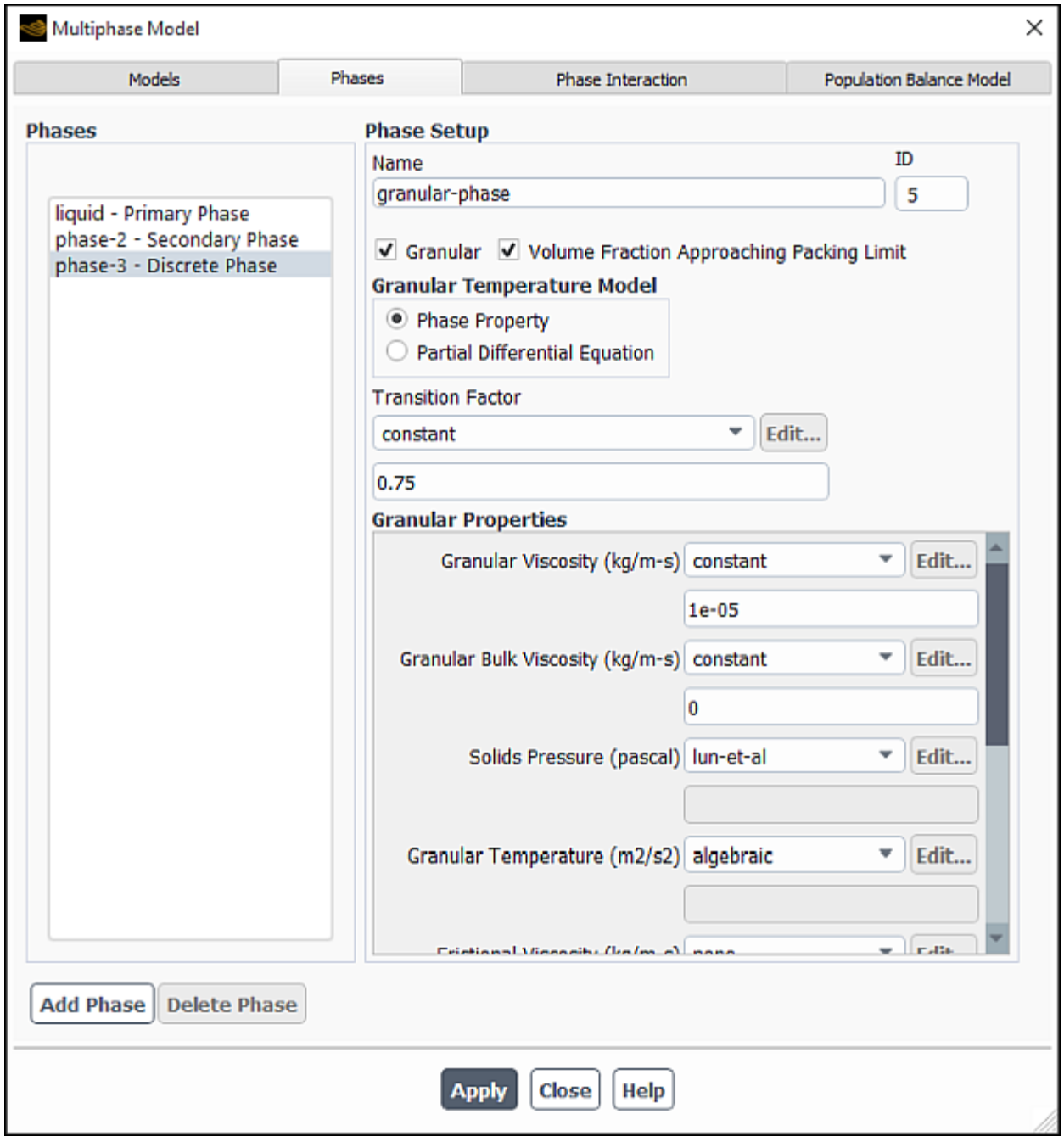
26.5.8.1. Defining a Granular Discrete Phase

To define a granular (that is, particulate) discrete phase in an Eulerian multiphase calculation, perform the following steps:

1. Select the phase (for example, Figure 26.82: The Multiphase Model Dialog Box for a Granular Phase (p. 3112)) in the **Phases** list. Then set up the properties in the **Phase Setup** group box (for example, (Figure 26.82: The Multiphase Model Dialog Box for a Granular Phase (p. 3112))

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.82: The Multiphase Model Dialog Box for a Granular Phase



2. Enter a **Name** for the phase.

3. Enable the **Granular** option.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Enable **Volume Fraction Approaching Packing Limit** to prevent the unlimited accumulation of particles, which are operating at packing limit conditions.

   Note:

   To reduce excessive accumulation of particles in cells, you can also enable the **Enhanced Numerics** option in the **Discrete Phase Model** dialog box (**Numerics** tab). Enhanced numerics is applied to particles of granular phases only. This option can be used with or without the **Volume Fraction Approaching Packing Limit** option.

5. Specify the **Transition Factor** as either a **constant** or a **user-defined** function. The default value for the **Transition Factor** is 0.75. The transition criterion is based on the local particle volume fraction of the given discrete phase and is specified as a factor multiplied by the maximum packing limit (also a user-specified value). For example, for a typical granular phase with a maximum packing limit of 0.63, the transition volume fraction is the product of 0.75 and 0.63, which is equal to 0.4725.

You can now define all other fields of the discrete phase in a similar manner to that described in Defining a Granular Secondary Phase (p. 3067).

26.5.9. Including the Boiling Model

If you are using the Eulerian multiphase model (Setting Up the Eulerian Model (p. 3065)), you have the option of including the **Boiling Model** (see Wall Boiling Models in the Theory Guide).

26.5.9.1. Limitations of the Boiling Model

Note the following limitations:

- The boiling model is only available with
  - **Eulerian** multiphase model
  - **Pressure-Based** solver
  - Turbulent flow
- The boiling model is not compatible with:
  - Species transport model
  - Standard moment, quadrature moment, and QMOM population balance methods
  - Simulations that involve granular secondary phases

26.5.9.2. Procedure for Setting the Boiling Model

The required work flow when using the boiling model is as follows:

1. Choose one of the turbulence models that is available with the Eulerian multiphase model.

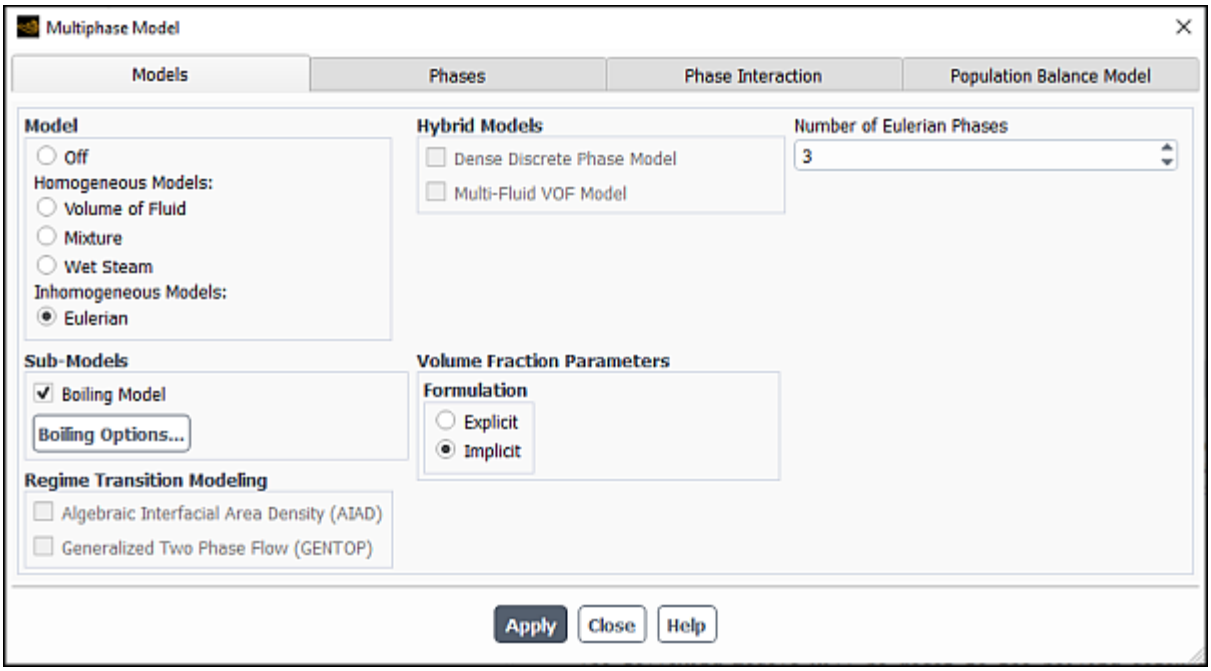
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setup → **Models** → **Viscous** → **Edit...**

2. Open the **Multiphase Model** dialog box (Figure 26.83: The Boiling Model (p. 3114)).

Setup → **Models** → **Multiphase** → **Edit...**

Figure 26.83: The Boiling Model



a. Enable **Boiling Model** in the **Sub-Models** group box.

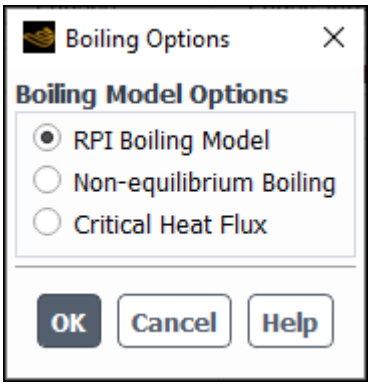
Once the boiling model is enabled, Ansys Fluent automatically applies the default and recommended settings shown in the table below.

Parameter	Setting
Gravity	Enabled
Surface Tension Coefficient	0.07 N/m
Drag Coefficient	**ishii**
Lift Coefficient	**tomiyama**
Secondary phase **Diameter**	**boiling-dia**
Mass transfer **Mechanism** for primary-secondary phase pairs	**boiling**
Heat Transfer Coefficient	**two-resistance**

b. Click **Boiling Options...**, and in the **Boiling Options** dialog box that opens (see Figure 26.84: The Boiling Options Dialog Box (p. 3115)), you can select from the boiling model

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.84: The Boiling Options Dialog Box



RPI Boiling Model

is where the total heat flux from the wall to the liquid is partitioned into three components, namely the convective heat flux, the quenching heat flux, and the evaporative heat flux. Details about this model can be found in RPI Model in the Theory Guide.

Non-equilibrium Boiling

is a modification to the RPI model in order to model different boiling regimes like DNB and critical heat flux. Details about this model can be found in Non-equilibrium Subcooled Boiling in the Theory Guide.

Critical Heat Flux

is where the critical heat flux condition is characterized by a sharp reduction of local heat transfer coefficients and the excursion of wall surface temperatures. Details about this model can be found in Critical Heat Flux in the Theory Guide.

c. Set the total **Number of Eulerian Phases** that are present in your case.

This can consist of two phases: liquid and vapor, which are directly involved in boiling mass transfer; or it can include "non-boiling phases" or "species". If a system consists of three phases: liquid, vapor, and air, then the **Number of Eulerian Phases** will be 3, where air is the non-boiling phase.

Note:

- To include the liquid volume fraction effects, use the `solve/set/multiphase-numeric/boiling-parameters/liquid-vof-factor` text command. When this option is enabled, the heat transfer coefficients of boiling models will be multiplied by the local liquid volume fraction.
- To include the thin film effects described in [86] (p. 5718), use the `solve/set/multiphase-numeric/boiling-parameters/thin-film` text command.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Make sure the energy option is enabled.

   Note:

   The energy option will be automatically turned on when the boiling model is enabled.

   Setup → Models → Energy On

4. Make sure that **Gravity** is enabled and set the **Operating Pressure** in the **Operating Conditions** dialog box.

5. Open the **Create/Edit Materials** dialog box and specify the material properties for the liquid, vapor, solid, and any other phases that may exist.

   Setup → Materials → Fluid → Create/Edit...

   Important:

   For the liquid and vapor phases, the **Standard State Enthalpy** must be specified, as it is used in the computation of the **Latent Heat**.

6. In the **Phases** tab, define the phases.

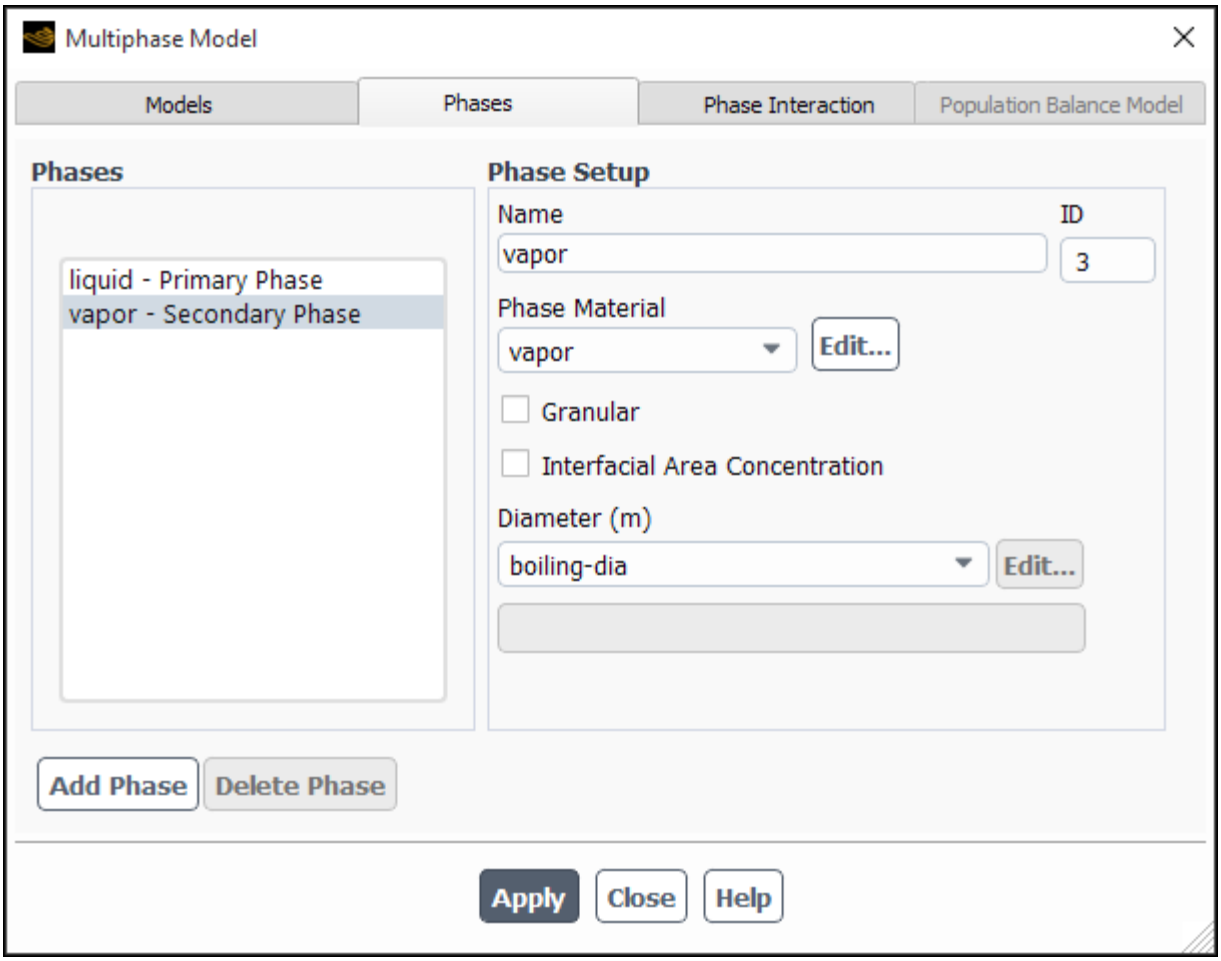
   a. Define the liquid as the primary phase by selecting the phase from the **Phases** selection list that is labeled **Primary Phase** and specifying phase parameters in the **Phase Setup** group box.

   b. Define the vapor as the secondary phase.

      In the **Multiphase Model** dialog box, the **Diameter** is automatically set to **boiling-dia** by default, but you have the option of setting it up as a **constant** value or a **user-defined** function, as shown in Figure 26.85: The Multiphase Model Dialog Box for the Boiling Model (Phases Tab) (p. 3117).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.85: The Multiphase Model Dialog Box for the Boiling Model (Phases Tab)

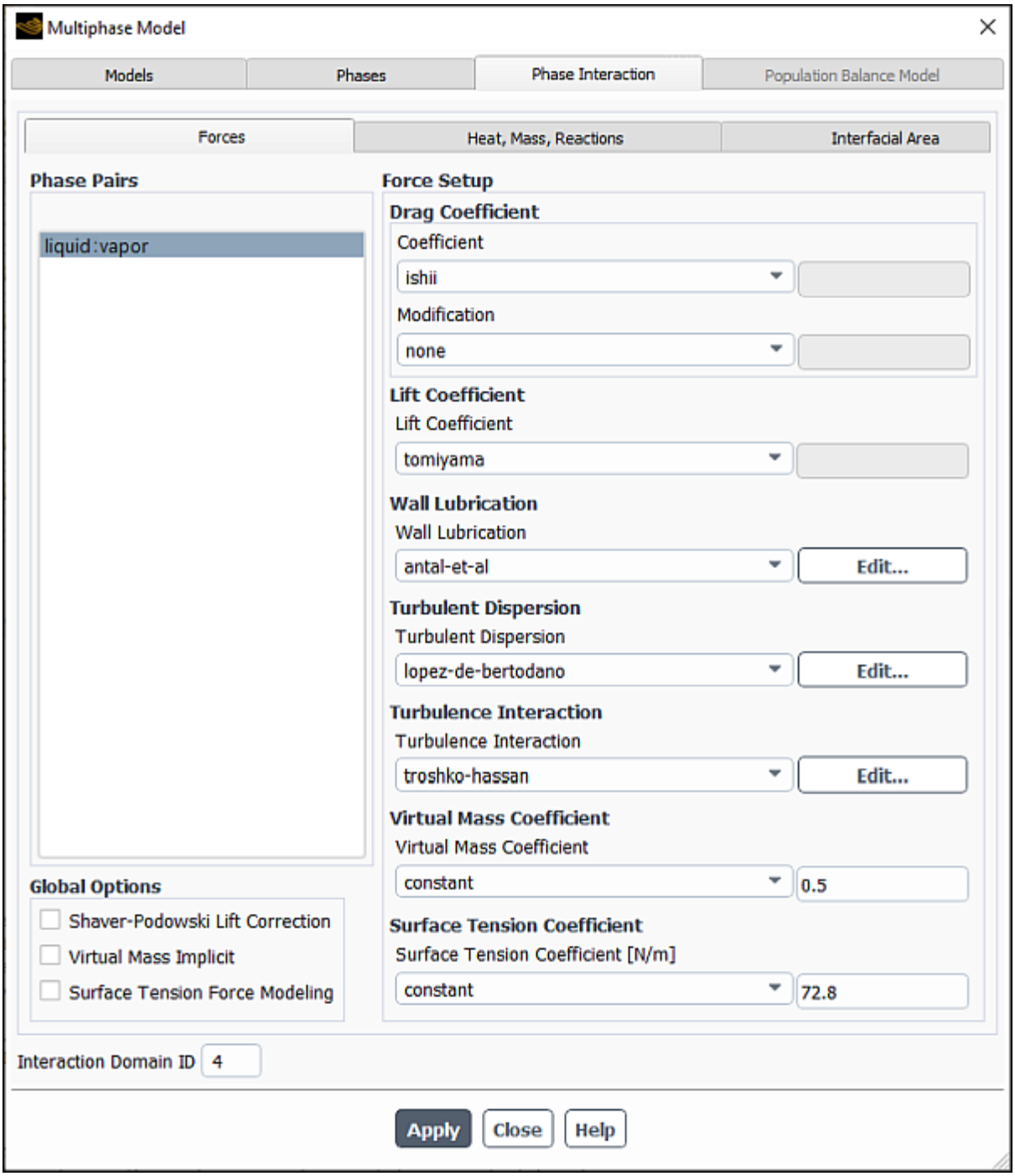


7. In the **Phase Interaction** > **Forces** tab, configure the following settings:

   a. In the **Drag Coefficient** group box, you can select other than the default and recommended **ishii** drag specification method from the **Coefficient** drop-down list. See Specifying the Drag Function (p. 3075) for available drag options and their definitions.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.86: Defining Forces in the Forces Tab



b. From the **Lift Coefficient** drop-down list, you can select other than the default **tomiyama** lift coefficient model as appropriate for your simulation (see Including the Lift Force (p. 3080) for available options).

c. From the **Wall Lubrication** drop down list, select the wall lubrication model for your case (see Including the Wall Lubrication Force (p. 3081) for details).

   For boiling flows, **antal-et-al** is typically chosen.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

d. From the **Turbulent Dispersion** drop-down list, select the turbulent dispersion force model (see Including the Turbulent Dispersion Force (p. 3085) for details).

   For boiling flows, **lopez-de-bertodano** is typically chosen. See Including the Turbulent Dispersion Force (p. 3085) for details.

e. From the **Turbulent Interaction** drop-down list, select the desired turbulence correlation (see Including Turbulence Interaction Source Terms (p. 3091) for details).

   For boiling flows, **troshko-hassan** is typically chosen. See Including Turbulence Interaction Source Terms (p. 3091) for details.

f. The default **constant** value of 0.07 N/m for the **Surface Tension Coefficient** is suitable for most cases. You can adjust the **Surface Tension Coefficient** if desired.

g. Click **Apply**.

8. In the **Phase Interaction** > **Heat, Mass, Reactions** > **Mass** tab, configure the following settings:

   a. Specify the **Number of Mass Transfer Mechanisms**.

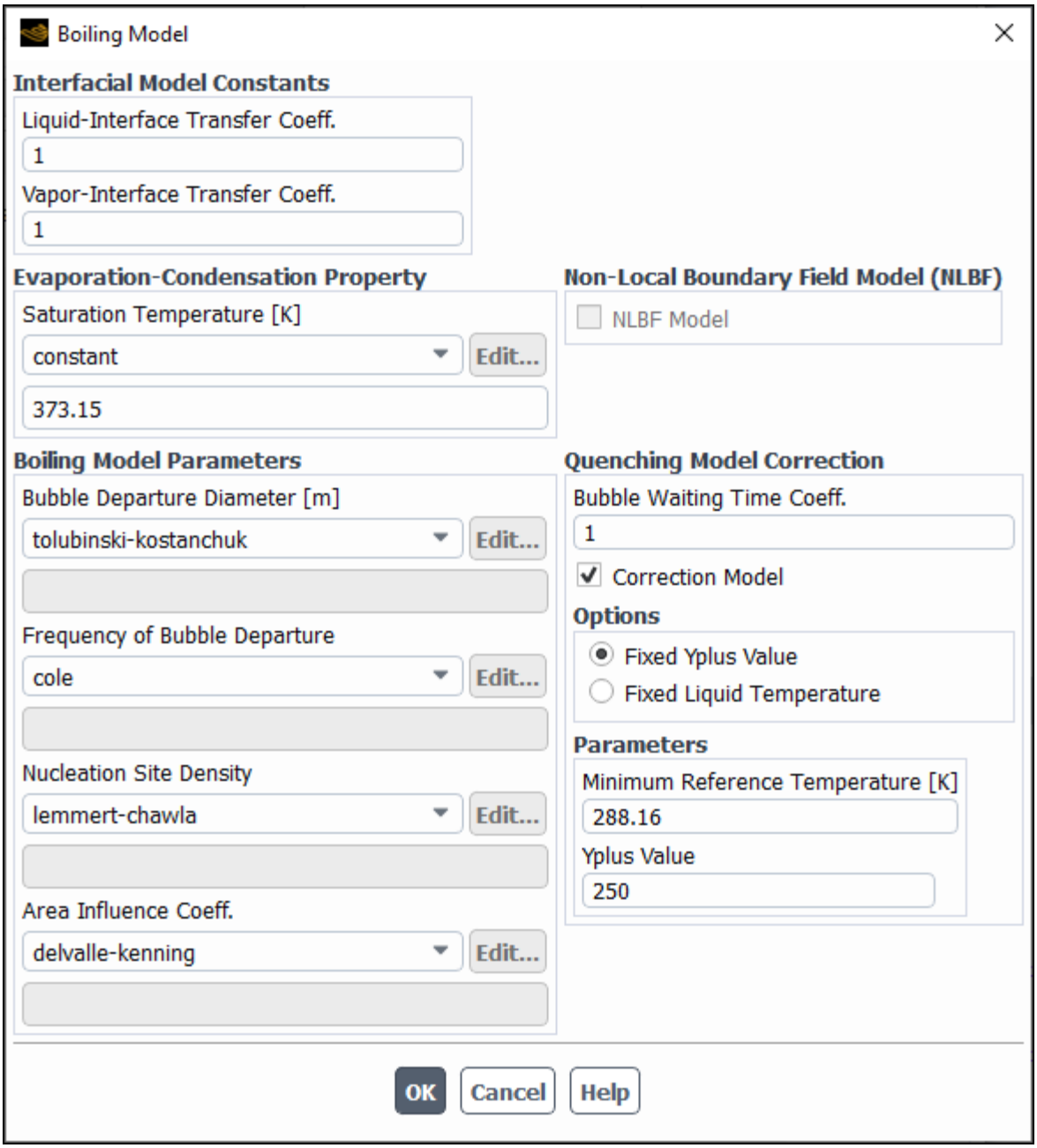
   b. Make sure the transfer is always from the liquid to the vapor phase.

   c. Make sure that **boiling** is selected under **Mechanism**.

   d. In the **Boiling Model** dialog box that opens (Figure 26.87: The Boiling Model Dialog Box (p. 3120)), set the following boiling model parameters and the quenching model corrections:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.87: The Boiling Model Dialog Box



i. Specify the **Interfacial Model Constants** and the **Saturation Temperature**. The default values for the liquid and vapor interface transfer coefficients are 1.

Note:

- If you use the **polynomial**, **piecewise-polynomial**, or **piecewise-linear** option for **Saturation Temperature**, the pressure-dependency must be specified in terms of absolute pressure.
- You may specify **Saturation Temperature** using RGP tables as described in Defining Saturation Properties via RGP Tables (p. 1735).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

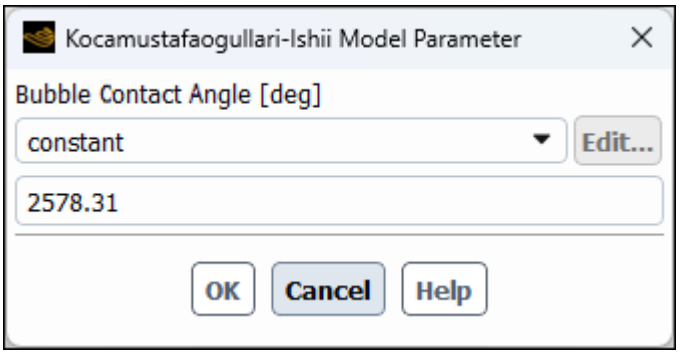
- If you selected the **Non-equilibrium Boiling** or **Critical Heat Flux** in the **Multiphase Model** dialog box (**Models** tab), then you need to specify a value for the **Vapor-Interface Transfer Coeff.** parameter.

ii. Under the **Boiling Model Parameters**, select the **Bubble Departure Diameter** that best describes your model from the following options:

- **tolubinski-kostanchuk** (default, see Equation 14.518 in the *Fluent Theory Guide*)
- **unal** (see Equation 14.520 in the *Fluent Theory Guide*)
- **kocamustafaogullari-ishii** (see Equation 14.516 in the *Fluent Theory Guide*)

When you select **kocamustafaogullari-ishii**, the **Kocamustafaogullari-Ishii Model Parameter** dialog box opens automatically where you can adjust the value of the of the **Bubble Contact Angle**.

Figure 26.88: The Kocamustafaogullari-Ishii Model Parameter Dialog Box



- **constant**
- **user-defined**

iii. Specify the **Frequency of Bubble Departure**. You can choose the **cole** option (default, see Equation 14.514 in the *Fluent Theory Guide*), or enter a **constant** value.

iv. Specify the **Nucleation Site Density**. You can choose between:

- **lemmert-chawla** (default)
- **kocamustafaogullari-ishii**
- **user-defined**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This quantity is usually represented by a correlation based on the wall superheat described in Equation 14.515 in the Theory Guide.

Note:

It is recommended that when using the **kocamustafaogullari-ishii** option, it should be used for both the **Bubble Departure Diameter** and **Nucleation Site Density**.

v. Specify the **Area Influence Coeff.** The area of influence is based on the bubble departure diameter and the nucleate site density, as defined in Equation 14.511 in the Theory Guide. You can choose from the following options:

- **delvalle-kenning** (default, see Equation 14.512 in the *Fluent Theory Guide*
- **constant**
- **user-defined**

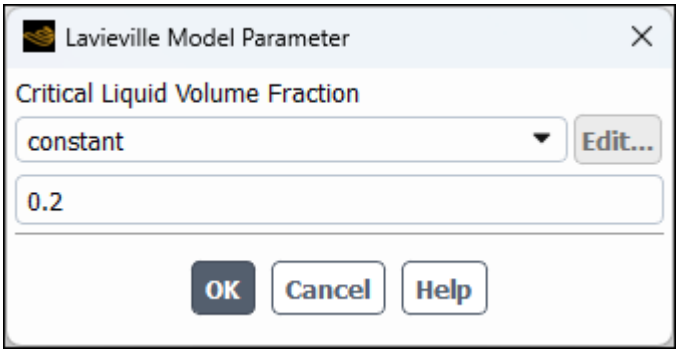
vi. If you have selected either the **Non-equilibrium Boiling** or **Critical Heat Flux** boiling option in the **Multiphase Model** dialog box (**Models** tab), you can select the **Boiling Blending Function**, which determines the partitioning of the wall heat flux between the liquid phase and the vapor phase, and specify its parameters. For more information, see Non-equilibrium Subcooled Boiling and Wall Heat Flux Partition in the *Fluent Theory Guide*

You can choose from the following options:

- **lavieville-et-al**: Is the blending function by Lavieville et al. (see Equation 14.527 in the *Fluent Theory Guide*).

  When you select **lavieville-et-al**, the **Lavieville Model Parameters** dialog box opens automatically where you can adjust the value of the **Critical Liquid Volume Fraction** ($\alpha_{l,crit}$ in Equation 14.527 in the *Fluent Theory Guide*).

  Figure 26.89: The Lavieville Model Parameters Dialog Box

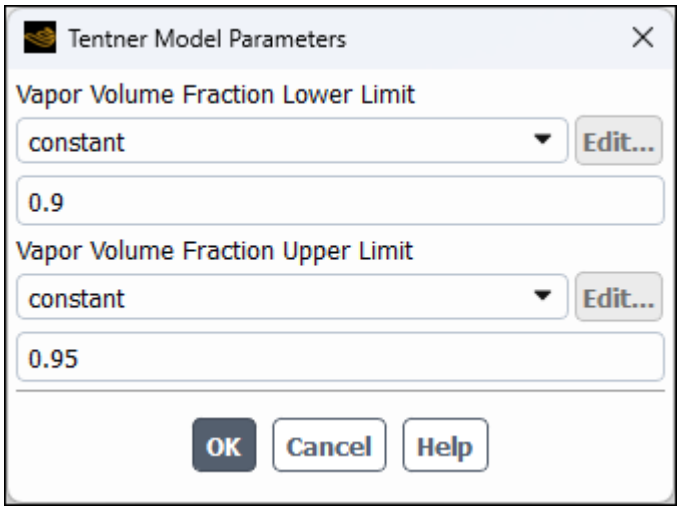


- **tentner-et-al** (default): Is the boiling blending function by Tentner et al. (see Equation 14.528 in the *Fluent Theory Guide*). This option is available only for the **Critical Heat Flux** boiling model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When you select **tentner-et-al**, the **Tentner Model Parameters** dialog box opens automatically.

Figure 26.90: The Tentner Model Parameters Dialog Box



You can adjust the values of **Vapor Volume Fraction Lower Limit** and **Vapor Volume Fraction Upper Limit** ($\alpha_{v,1}$ and $\alpha_{v,2}$ in Equation 14.528 in the *Fluent Theory Guide*, respectively).

- **ioilev-et-al**: Is the boiling blending function by Ioilev et al. (see Equation 14.529 in the *Fluent Theory Guide*). This option is available only for the **Critical Heat Flux** boiling model.

  The input parameters for the **ioilev-et-al** boiling bending function are similar to those for **tentner-et-al**. When you select **ioilev-et-al**, the **Ioilev Model Parameters** dialog box opens automatically where you can adjust the values of **Vapor Volume Fraction Lower Limit** and **Vapor Volume Fraction Upper Limit** ($\alpha_{v,1}$ and $\alpha_{v,2}$ in Equation 14.529 in the *Fluent Theory Guide*, respectively)

- **user-defined**: Allows you to implement and use your own boiling bending function using the `DEFINE_BOILING_PROPERTY` user-defined function (UDF). Refer `DEFINE_BOILING_PROPERTY` in the *Fluent Customization Manual* for details about this UDF.

vii. If you use a mesh with y+ < 30, select **NLBF Model** in the **Non-Local Boundary Field Model (NLBF)** group box. This option helps prevent unphysical local temperature fields at the wall and achieve a less mesh dependent solution. This option is not compatible with the quenching model correction. If the **Correction Model** is enabled, you must disable it to use the **NLBF Model**.

  The non-local boundary field (NLBF) model uses liquid temperatures from the bulk of the flow to determine the average temperature. This is done by collecting data from points on lines that are normal to the walls and extend into the interior of the domain. The points are equidistantly positioned on each line. The sampled temperature data is processed by the second order averaging algorithm that takes into account the temperature gradients and the sample point positions within relevant cells.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The average temperature is used in the calculations of quenching and bubble departure diameters.

You need to specify the following NLBF model parameters:

- **Maximum Line Length per Wall Element**: Is the maximum value of the length of each line where the sampling points are located
- **Number of Points per Line**: Is the number of the sample points on each line

Note:

- The points outside of the mesh are not accounted for in averaging the field data.
- In dynamic mesh cases, the geometry of the mesh is automatically recalculated when mesh events occur.
- When the calculation that involves the NLBF model is complete, an additional field variable called **NLBF Averaged Temperature** will be available for postprocessing under the **Temperature...** category at a mixture level.

viii. In the **Quenching Model Correction** group box, specify the below settings. Note that the quenching model correction is not compatible with **NLBF Model**. If the **NLBF Model** is enabled, you must disable it to use the quenching model correction.

Quenching model correction addresses the quenching term in the wall heat flux partition (described in Wall Heat Flux Partition in the Theory Guide). The quenching term in the wall heat flux partition models the cyclic averaged transient energy transfer related to liquid filling the wall vicinity after the bubble detachment with a period T and it is expressed as:

$$\dot{q}_q = C_{wt} \frac{2k_l}{\sqrt{\pi \gamma_l T}} \left(T_w - T_l\right) A_b \tag{26.23}$$

where k_l is the liquid heat conductivity, T is the periodic time, C_{wt} is a coefficient introduced to correct the waiting time between departures of consecutive bubbles, and $\gamma_l = k_l \Big/ \left(\rho_l C_{pl}\right)$ is the liquid phase diffusivity.

From Equation 26.23 (p. 3124), you can see that the quenching model is strongly dependent on T_l, which results in grid-dependent solutions. To remedy this, two approaches have been adopted in Ansys Fluent:

- fixed y+ value
- fixed liquid temperature

Note:

When the calculated temperature in regions away from the walls is lower than the **Operating Temperature** (set in the **Boussinesq Parameters** group box

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

in the **Operating Conditions** dialog box), the solver uses the operating temperature for the quenching correction.

A. Specify **Bubble Waiting Time Coefficient**, which is C_{wt} in Equation 26.23 (p. 3124). The default value is 1, however you can modify this value as needed, but it can only be specified as a constant.

B. Enable the **Correction Model** if you want to achieve a certain level of grid-independence in your solution. If this option is disabled, Equation 26.23 (p. 3124) calculates the quench flux term without corrections.

C. Select **Fixed Yplus Value** if you want to use the logarithmic form of the wall functions to estimate the liquid temperature T_l at a fixed Yplus value of 250, as proposed by Egorov and Mentor [40] (p. 5715), instead of using the liquid temperature values in the near-wall cells. Enter the **Minimum Reference Temperature**, which limits the lowest value of the liquid temperature. It should not be lower than the liquid inlet temperature. Specify a **Yplus Value**. It is set to 250 by default.

D. If you select **Fixed Liquid Temperature**, you can choose from **standard**, **constant**, or **user-defined** for the **Liquid Reference Temperature**.

The **standard** method calculates the **Liquid Reference Temperature** as:

$$T_l = \min(T_{Sat}, T^*) \tag{26.24}$$

$$T^* = (1-\theta)\min(\overline{T}_L, T_{Sat}) + \theta T_{Sat} \tag{26.25}$$

$$\theta = \max\left(0, \min\left(1, \frac{\Phi_v - \Phi_{v,LB}}{\Phi_{v,UB} - \Phi_{v,LB}}\right)\right) \tag{26.26}$$

where

T_{Sat} = saturation temperature

T^* = intermediate temperature

$\overline{T}_L$ = average or bulk temperature of the liquid

Φ_v = vapor volume fraction

$\Phi_{v,LB}$ and $\Phi_{v,UB}$ = lower and upper limits for the vapor volume fraction, equal to 0.05 and 0.3, respectively

θ = linear blending function

You will also need to enter the **Minimum Reference Temperature**. The liquid temperature used to compute the quenching flux should usually be between the inlet liquid temperature and the saturation temperature.

Note:

To learn how to hook boiling parameter user-defined functions, refer to `DEFINE_BOILING_PROPERTY` in the Fluent Customization Manual.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

e. Click **Apply**.

9. In the **Phase Interaction** > **Heat, Mass, Reactions** > **Heat** tab, specify the heat transfer coefficient for each phase pair. See Including Heat Transfer Effects (p. 3093) for available options.

   a. For cases with multiple boiling mechanisms (such as in boiling coupled with the inhomogeneous PBM), you can increase the robustness and accuracy of the solution by using the two-resistance boiling framework, which is invoked by the following text command:

   ```
   solve/set/multiphase-numerics/heat-mass-transfer/boiling/two-resistance-boiling-framework?
   Allow generalized two-resistance framework for boiling? [no] yes
   ```

   Once the generalized two-resistance boiling framework is enabled, the two resistance model with the default and recommended interphase heat transfer coefficients for the liquid and vapor phases is automatically assigned to each boiling phase pair. Note that with the two-resistance boiling framework, the only option that appears in the **Heat Transfer Coefficient** drop-down list for each boiling phase pair is **two-resistance**. You can modify the selection of the interphase heat transfer coefficient for the liquid phase in the **Two Resistance Model** dialog box.

   b. For all other cases: Select the desired correlation from the **Heat Transfer Coefficient** drop-down box.

   For boiling flows, **ranz-marshall** is usually chosen.

   c. Click **Apply**.

10. In the **Phase Interaction** > **Interfacial Area** tab, configure the following settings:

    a. For each phase pair, you can select from the following formulations:
    - **ia-symmetric** (see Equation 14.208 in the Theory Guide)
    - **ia-particle** (see Equation 14.207 in the Theory Guide)
    - **ia-ishii** (see Equation 14.211 in the Theory Guide). This option is only available with the RPI model.
    - **user-defined** (see Example 4- Custom Interfacial Area in the Fluent Customization Manual)

    For boiling flows, **ia-symmetric** or **ia-particle** are usually chosen.

    b. Click **Apply**.

11. Set up the conditions at inlets, outlets, and thermal conditions for walls.

Setup → Boundary Conditions

The wall boiling models are compatible with three different wall boundaries: isothermal wall, specified heat flux, and specified heat transfer coefficient (coupled wall boundary).

Note:

The boiling models do not apply to thin walls.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12. Choose **Coupled** as the pressure-velocity coupling scheme.

Solution → **Methods**

Note:

Although **Phase Coupled SIMPLE** is also available, it is generally less robust and is not recommended for use in steady cases involving boiling or mass transfer.

---

13. Adjust the solution controls for your simulation.

The following solution strategies are recommended for boiling model simulations:

Solution → **Controls**

- **Courant Number**: use a value between 1 and 20 is recommended. As a first try, use a value of 10.
- **Explicit Relaxation Factors**: use the default values of 1 for **Momentum** and **Pressure**.
- **Vaporization Mass**: use an under-relaxation factor between 0.5 and 1. As a first try, use a value of 1.
- **Volume Fraction**: use an under-relaxation factor between 0.3 and 0.5.
- **Turbulent Kinetic Energy**: use an under-relaxation factor between 0.3 and 0.8.
- **Turbulent Viscosity**: use an under-relaxation factor between 0.5 and 1.0.
- **Energy**: use an under-relaxation factor between 0.5 and 0.8.

26.5.9.3. Postprocessing for the Boiling Model

The following additional postprocessing variables are available in the **Phase Interaction...** category for generating contour plots for the mixture:

- **Saturation Temperature**
- **Liquid Subcooling**
- **Wall Superheat**
- **Bubble Departure Diameter**
- **Bubble Departure Frequency**
- **Bubble Waiting Time**
- **Bubble Area Fraction**
- **Bubble Nucleation Site Density**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

With the exception of **Saturation Temperature** and **Liquid Subcooling**, these postprocessing variables are best visualized for heated walls. See Field Function Definitions (p. 4167) for a complete list of field functions and their definitions.

26.5.10. Setting Up Polydisperse Boiling

The procedure for setting up a polydisperse boiling simulation is similar to that for setting up a boiling simulation outlined in Including the Boiling Model (p. 3113). Only the differences are highlighted here.

1. In the **Models** tab, set the total **Number of Eulerian Phases** to 3.
2. In the **Phases** tab, define, specify the liquid material as the primary phase, and the same vapor material as the secondary phases.

   Important:

   Small and large bubbles formed during the boiling of the liquid phase will be designated to two separate secondary phases. Therefore, the material assigned to these two secondary phases must be identical.

3. In the **Phase Interaction** > **Heat, Mass, Reactions** > **Mass** tab, configure the following settings:
   a. Set the **Number of Mass Transfer Mechanisms** to 2.
   b. Make sure the transfer is always from the liquid to the vapor phase.
   c. For both phase pairs, select **boiling** under **Mechanism**.

      Note:

      Although the bulk mass transfer (evaporation and condensation) is considered identical between the liquid and the two vapor phases, nucleation happens only for the phase which contains the smallest bin.

   d. In the **Population Balance Model** tab, set up the population balance model (see Population Balance Model (p. 3131)) and configure the following settings.
      i. Enable **Phenomena**.
      ii. Enable **Nucleation Rate** and **Growth Rate**.

      Once these options are enabled, Ansys Fluent automatically selects the following options:

      - **pb-boiling** for both **Nucleation Rate** and **Growth Rate**

        See Coupling Between the RPI Boiling Model and the Homogeneous or Inhomogeneous Discrete PBM in the *Fluent Theory Guide* for more information about the **pb-boiling** method.

      - **sauter-mean** for **Diameter** for both secondary phases (**Phases** tab)

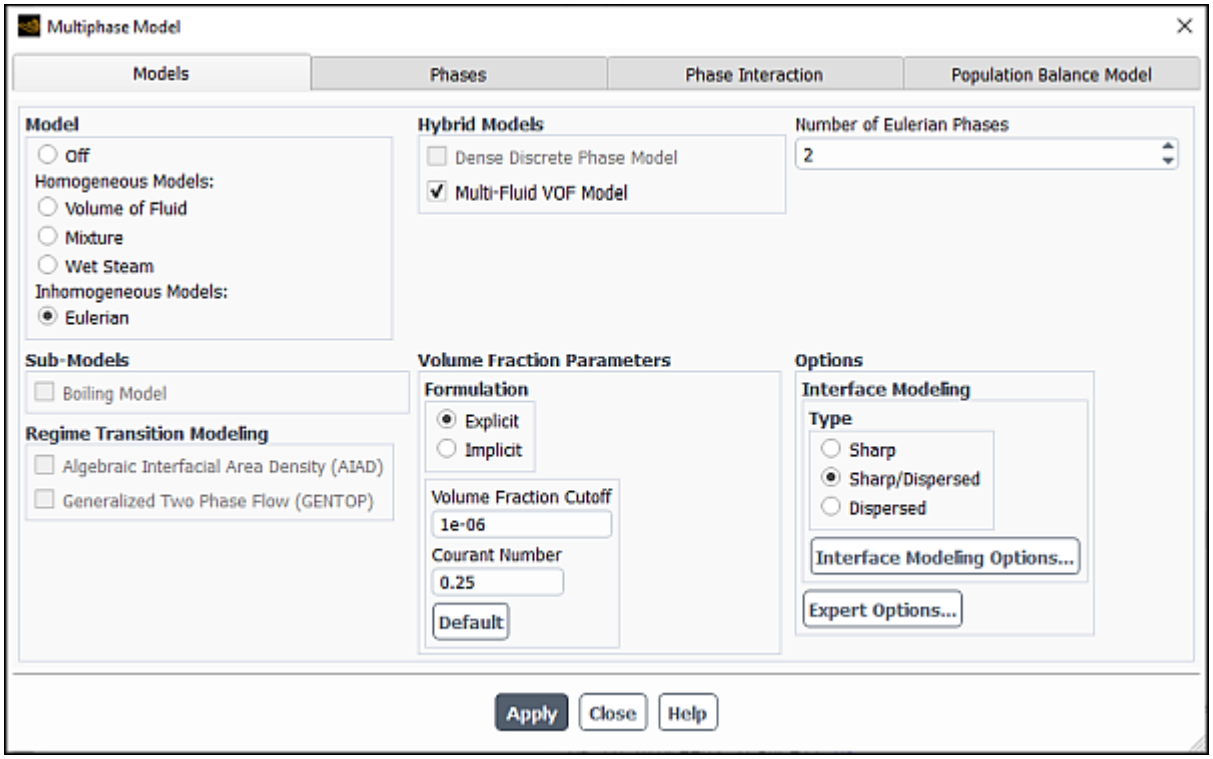
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This determines how the individual bin masses are distributed to ensure consistency of mass balance between phases and bins.

iii. Optionally, enable and set up the aggregation and breakage kernels.

26.5.11. Including the Multi-Fluid VOF Model

After you have selected the Eulerian multiphase model (Setting Up the Eulerian Model (p. 3065)), you can enable the **Multi-Fluid VOF Model** (Multi-Fluid VOF Model in the Theory Guide).



Note the following limitations:

- This model is only available with the **Eulerian** multiphase model.
- You cannot use this model with the **Dense Discrete Phase Model**.

After enabling **Multi-Fluid VOF**, the **Interface Modeling Options** become available, as described in Interface Modeling Type (p. 2922). You can select from **Sharp**, **Dispersed**, or hybrid **Sharp/Dispersed** interface modeling to expose the most appropriate discretization schemes for your simulation. You can also specify some additional time-advancement settings as described in Setting Time-Dependent Parameters for the Explicit Volume Fraction Formulation (p. 3017).

If you want to use the anisotropic drag law, perform the following steps:

1. Define the drag law in the **Phase Interaction** > **Forces** tab.

   a. For each pair of phases, select the appropriate drag law from the **Drag Coefficient** drop-down list.

      The available drag laws depend on your selection for **Interface Modeling**:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Sharp** interface modeling supports only **symmetric**, **anisotropic-drag**, **user-defined**, and **none**.
- **Sharp/Dispersed** interface modeling supports all the relevant drag laws described in Specifying the Drag Function (p. 3075).
- **Dispersed** interface modeling does not support **anisotropic-drag**, but supports all other relevant drag laws (see Specifying the Drag Function (p. 3075)).

With the **Sharp** and **Sharp/Dispersed** interface modeling, you can use **anisotropic-drag** when there is higher drag in the normal direction to the interface and lower drag in the tangential direction to the interface. For details about this drag law, refer to Multi-Fluid VOF Model in the Theory Guide.

Note:

Anisotropic drag applicable to free surface modeling is only compatible with the **Mixture** turbulence model.

To specify the input parameters for anisotropic drag, you can use the `solve/set/mp-mfluid-aniso-drag` text command. The options for an `Anisotropic Drag Method` of 0 (which is based on the symmetric drag), are as follows:

```
Anisotropic Drag Method
  [0]

Normal Interfacial Drag Friction Factor
  [1000000]

Tangential Interfacial Drag Friction Factor
  [10]

Length scale
  [0.0001]
```

The options for an `Anisotropic Drag Method` of 1 are as follows:

```
Anisotropic Drag Method
  [0] 1

Viscosity option
  [2]

Normal Interfacial Drag Friction Factor
  [1000000]

Tangential Interfacial Drag Friction Factor
  [10]

Length scale
  [0.0001]
```

b. Click **Apply**.

2. If you have enabled **Phase Localized Discretization** in the **Models** tab of the **Multiphase Model** dialog box, select the **Phase Localized Compressive Scheme** in the **Discretization** tab and click **Apply** to save your settings. For information about applying the various schemes, refer to Discretizing Using the Phase Localized Compressive Scheme (p. 3015).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.6. Population Balance Model

The population balance model is available with the mixture and Eulerian multiphase models.

This chapter provides basic instructions to set up and solve population balance problems in Ansys Fluent. This chapter describes the following:

26.6.1. Population Balance Model Setup
26.6.2. Solution Strategies
26.6.3. Postprocessing for the Population Balance Model
26.6.4. UDFs for Population Balance Modeling
26.6.5. DEFINE_HET_RXN_RATE Macro

26.6.1. Population Balance Model Setup

This section describes the appropriate inputs for setting the population balance model. Using the double-precision version of Ansys Fluent when solving population balance problems is highly recommended.

Important:

Note the following limitation of the population balance model: The model can be used only on one secondary phase, even if your problem includes additional secondary phases. A three-phase gas-liquid-solid case can be modeled, where the population balance model is used for the gas phase and the solid phase acts as a catalyst. However, if you are using the **Inhomogeneous Discrete**, more than one secondary phase can be used. The properties of the secondary phases selected for that method should be the same for consistency.

For more information, see the following sections:

26.6.1.1. Enabling the Population Balance Model
26.6.1.2. Defining Population Balance Boundary Conditions
26.6.1.3. Defining Population Balance Cell Zones Conditions
26.6.1.4. Specifying Population Balance Solution Controls
26.6.1.5. Coupling With Fluid Dynamics
26.6.1.6. Specifying Interphase Mass Transfer Due to Nucleation and Growth
26.6.1.7. Size Calculator

26.6.1.1. Enabling the Population Balance Model

The below procedure for setting up a population balance problem includes only those steps necessary for the population balance model itself. You will need to set up other models, boundary conditions, and so on, as usual. See the Ansys Fluent User's Guide (p. 1) for details.)

1. Start the double-precision version of Ansys Fluent.

2. Enable either the mixture or Eulerian multiphase model.

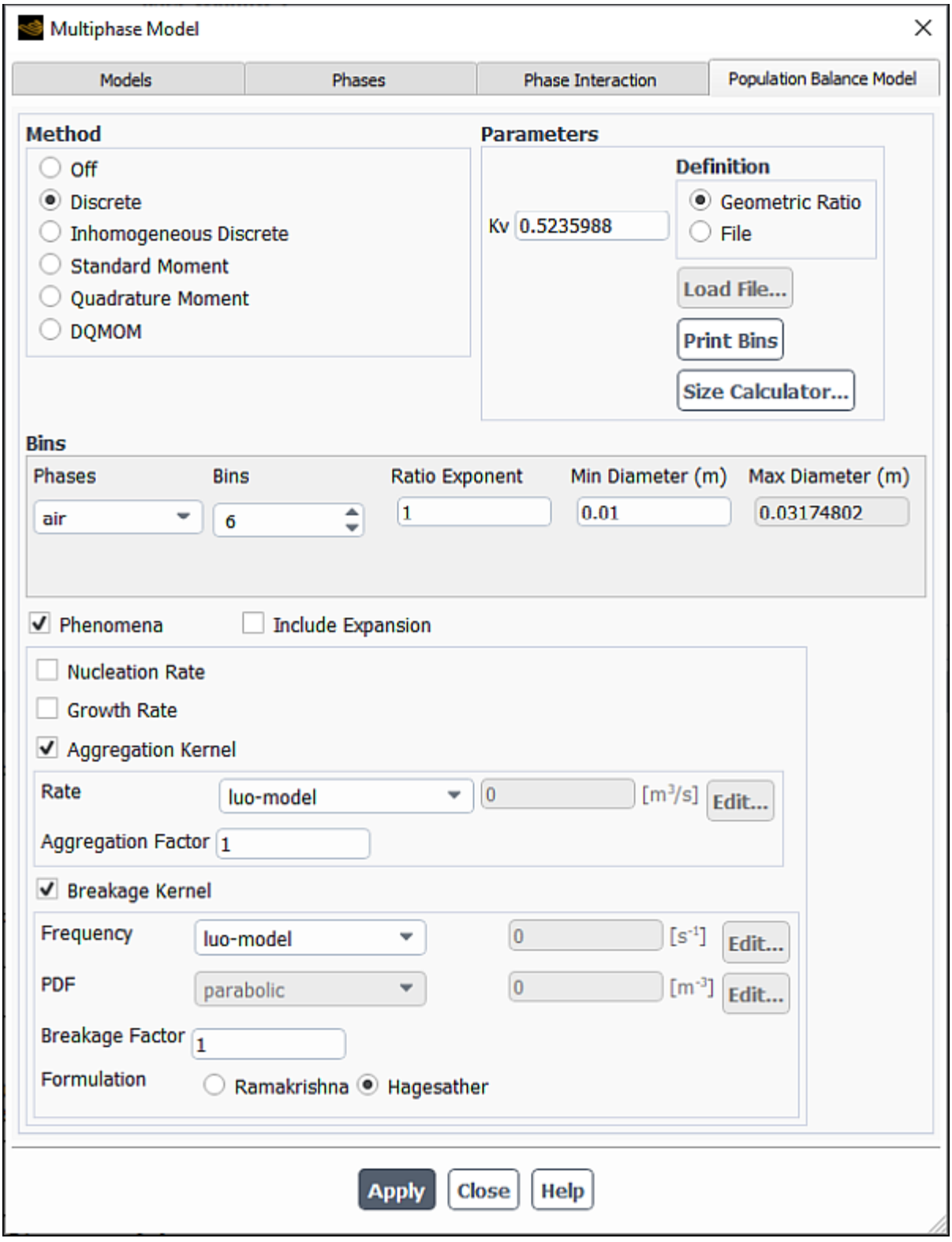
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. In the **Multiphase Model** dialog box, open the **Population Balance Model** tab (Figure 26.91: The **Multiphase Model** Dialog Box - **Population Balance Model** Tab (p. 3133)).

   Setup → **Models** → **Population Balance** **Edit...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.91: The Multiphase Model Dialog Box - Population Balance Model Tab



4. Specify the population balance method under **Method**.

   - If you select **Discrete**, you will need to specify the following parameters:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Kv

specifies the value for the particle volume coefficient K_v (as described in Particle Growth). By default, this coefficient has a value of $\pi / 6$.

Definition

can be specified as a **Geometric Ratio** or as a **File**. If **Geometric Ratio** is selected, then the **Ratio Exponent** must be specified. If **File** is selected, you will click the **Load File...** button and select the bin size file that you want loaded.

You can input the diameter through the text file, with each diameter listed on a separate line, starting from the smallest to the largest diameter (one entry per line). Hence, you are not limited by the choices specified in the dialog box.

Bins

specifies the number of particle size bins used in the calculation. The maximum number of bins allowed is 50.

Ratio Exponent

specifies the exponent q used in the discretization of the growth term volume coordinate (see Numerical Method).

Min Diameter

specifies the minimum bin size $L_0 \equiv \left(V_0 / K_v\right)^{1/3}$.

Max Diameter

displays the maximum bin size, which is calculated internally.

To display a list of the bin sizes in the console window, click **Print Bins**. The bin sizes will be listed in order of size, from the largest to the smallest. This option is only available when the **Geometric Ratio Definition** is selected.

- If you select **Inhomogeneous Discrete** under **Method**, you will specify the same parameters as for the **Discrete** model. Additionally, you can include more than one secondary phase in the bin definition. Enter the total number of **Active Secondary Phases** in your simulation.

  Note:

  While reading bins through the **Load File...** option for the **Inhomogeneous Discrete** model, the corresponding phase name must be included, for example `(("air-1" (0.1 0.2 0.3)))`

- If you select **Standard Moment** under **Method**, you will specify the number of **Moments** under **Parameters**.

- If you selected **Quadrature Moment** under **Method**, you will set the number of moments to either 4, 6 or 8 under **Parameters**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you selected **DQMOM** under **Method**, you will select the **DQMOM Phases** from the list. You will also specify the following **Parameters**:

  Max Size

  specifies the maximum size of the particle.

  Min Size

  specifies the minimum size of the particle.

  Reference Length

  is the reference particle size. Normally, the averaged size of the particle group should be sufficient.

  Min VOF

  is the minimum VOF, where the total volume fraction of the particle phases (participating in DQMOM computations) is below the minimum value; the source terms caused by the breakage and coalescence are not computed in that cell for the DQMOM and VOF equations.

  Max VOF Change/Time Step

  is the maximum VOF change in percentage for each DQMOM phase per time step, in order to smooth the convergence progress.

  Generate DQMOM Values

  enables you to generate DQMOM values from PDF, CDF, or Overall Moments files. Each of the file formats and the way to generate the values are discussed in Generated DQMOM Values (p. 3145).

  Note:

  The DQMOM method is restricted to a four-phase system, of which three secondary phases are directly involved in the DQMOM computation. Unsteady simulations are required to model breakage and coalescence and a well defined initial field is recommended, in which the abscissas are distinct. Only growth, breakage, and aggregation are the available phenomena. No nucleation is considered.

  You can use the built-in size calculator to evaluate bubble sizes and/or droplet size limits. You can open the **Size Calculator** dialog box by clicking the **Size Calculator...** button and obtain the recommendations as described in Size Calculator (p. 3157). The tool provides estimated values based on best practices and engineering recommendations.

5. Select the secondary phase from the **Phase** drop-down list for which you want to apply the population balance model parameters.

6. For all population balance methods, you can enable the following under **Phenomena** :

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Nucleation Rate

enables you to specify the nucleation rate ($\mathrm{particles}/\mathrm{m}^3\text{–s}$). You can select **constant** or **user-defined** from the drop-down list. If you select **constant**, specify a value in the adjacent field. If you have a user-defined function (UDF) that you want to use to model the nucleation rate, you can choose the **user-defined** option and specify the appropriate UDF.

Note:

This option is not available when using the **Inhomogeneous Discrete** method.

Growth Rate

enables you to specify the particle growth rate (m/s). You can select **constant** or **user-defined** from the drop-down list. If you select **constant**, specify a value in the adjacent field. If you have a user-defined function (UDF) that you want to use to model the growth rate, you can choose the **user-defined** option and specify the appropriate UDF.

Note:

This option is not available when using the **Inhomogeneous Discrete** method.

Aggregation Kernel

enables you to specify the aggregation kernel (m^3/s) and specify the **Aggregation Factor**.

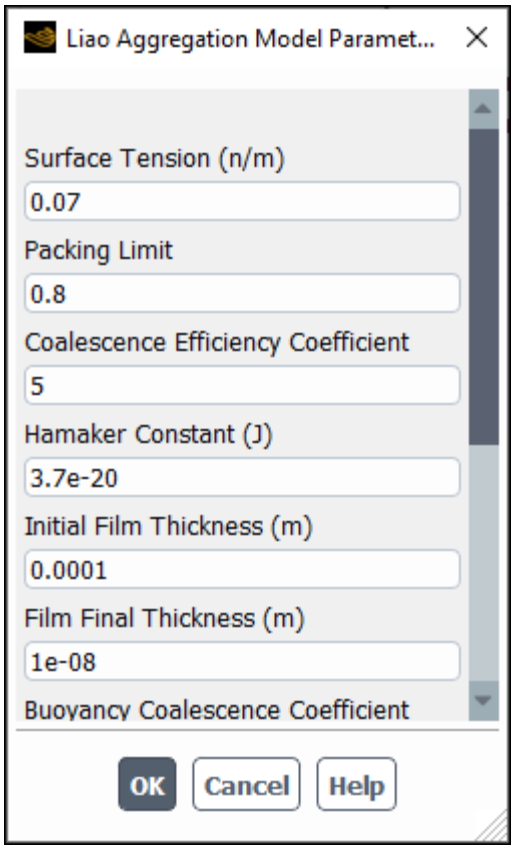
The **Aggregation Factor** (F_A in Equation 14.737 in the *Fluent Theory Guide*) is used for the calibration of the aggregation kernel.

For **Rate**, you can select **constant**, **liao-aggregation-model**, **luo-model**, **free-molecular-model**, **turbulent-model**, **prince-blanch-model**, or **user-defined** from the drop-down list:

- If you select **constant**, specify a value in the adjacent field.
- If you select **liao-aggregation-model**, the **Liao Aggregation Model Parameters** dialog box will open automatically. Note that because this is a modal dialog box, you must tend to it immediately.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.92: The Liao Aggregation Model Parameters Dialog Box



You can specify the following parameters:

- **Surface Tension**
- **Packing Limit**: Is the maximum packing limit, $\alpha_{g,max}$ in Equation 14.764 in the *Fluent Theory Guide*.
- **Coalescence Efficiency Coefficient**: λ_{eff} in Equation 14.771 in the *Fluent Theory Guide*.
- **Hamaker Constant**: A_H in Equation 14.776 in the *Fluent Theory Guide*.
- **Initial Film Thickness**
- **Final Film Thickness**
- **Buoyancy Coalescence Coefficient**: Is C_{buoy} in Equation 14.771 in the *Fluent Theory Guide*.
- **Turbulence Coalescence Coefficient**: Is C_{turb} in Equation 14.770 in the *Fluent Theory Guide*.
- **Eddy Coalescence Coefficient**: Is C_{eddy} in Equation 14.778 in the *Fluent Theory Guide*.
- **Shear Coalescence Coefficient**: Is C_{shear} in Equation 14.777 in the *Fluent Theory Guide*.

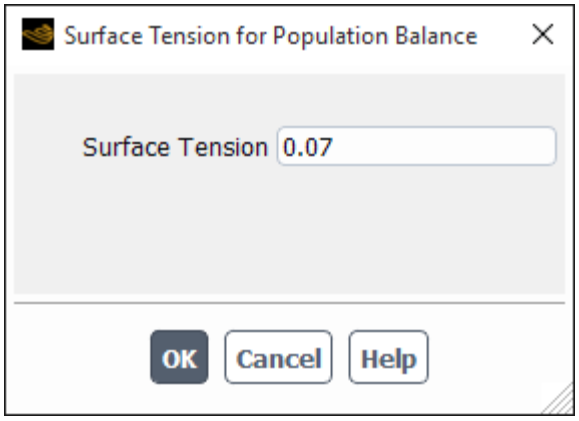
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– **Wake Coalescence Coefficient**: Is C_{wake} in Equation 14.780 in the *Fluent Theory Guide*.

For more information about this model and its parameters, see Liao Aggregation Kernel in the *Fluent Theory Guide*.

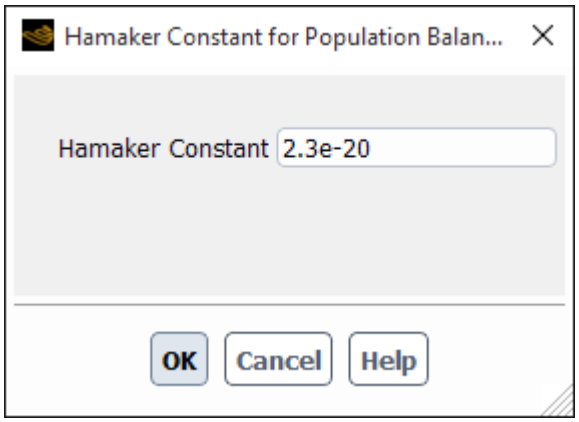
- If you select **luo-model**, the **Surface Tension for Population Balance** dialog box will open automatically to enable you to specify the surface tension (see Figure 26.93: The Surface Tension for Population Balance Dialog Box (p. 3138)). The aggregation rate for the model will then be calculated based on Luo's aggregation kernel (as described in Particle Birth and Death Due to Breakage and Aggregation).

Figure 26.93: The Surface Tension for Population Balance Dialog Box



- If you select **free-molecular-model**, then Equation 14.746 is applied.
- If you select **turbulent-model**, the **Hamaker Constant for Population Balance** dialog box will open automatically to enable you to specify the Hamaker constant (see Figure 26.94: The Hamaker Constant for Population Balance Dialog Box (p. 3138)). More information about this model is available in Turbulent Aggregation Kernel.

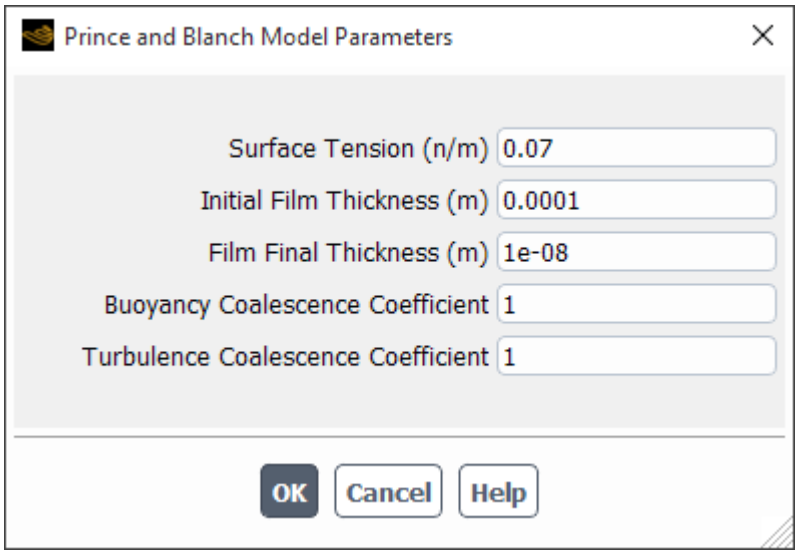
Figure 26.94: The Hamaker Constant for Population Balance Dialog Box



- If you select **prince-blanch-model**, the **Prince and Blanch Model Parameters** dialog box will open automatically. Note that because this is a modal dialog box, you must tend to it immediately.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.95: The Prince and Blanch Model Parameters Dialog Box



You can specify the following parameters:

– **Surface Tension**

– **Initial Film Thickness**: Is h_o in Equation 14.756 in the *Fluent Theory Guide*.

– **Final Film Thickness**: Is h_f in Equation 14.756 in the *Fluent Theory Guide*.

– **Buoyancy Coalescence Coefficient**: Is F_{CB} in Equation 14.762 in the *Fluent Theory Guide*.

– **Turbulence Coalescence Coefficient**: Is F_{CT} in Equation 14.759 in the *Fluent Theory Guide*.

More information about this model is available in Prince and Blanch Aggregation Kernel.

- If you have a user-defined function (UDF) that you want to use to model the aggregation rate, you can choose the **user-defined** option and specify the appropriate UDF.

Breakage Kernel

enables you to specify the particle breakage rate ($\text{particles}/\text{m}^3\text{–s}$) and the **Breakage Factor**.

The **Breakage Factor** (F_B in Equation 14.704 in the *Fluent Theory Guide*) is used for the calibration of the breakage kernel.

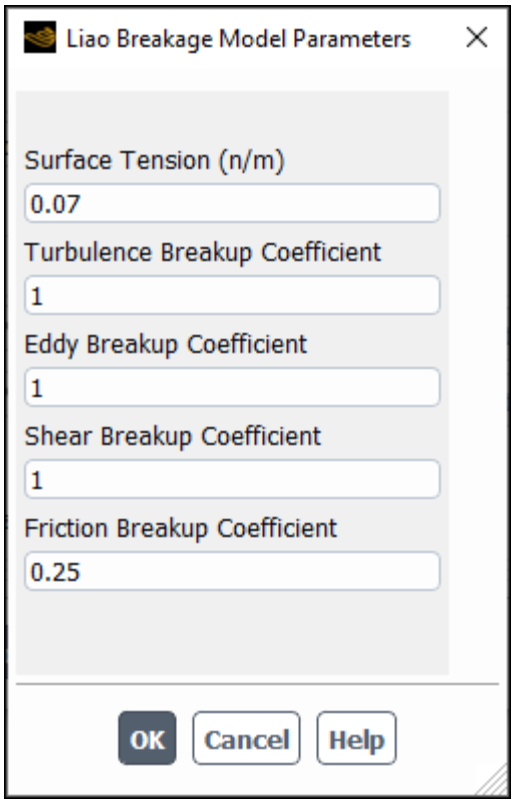
For **Frequency**, you can select **constant**, **liao-breakage-model**, **luo-model**, **lehr-model**, **ghadiri-model**, **laakkonen-model**, **martinez-bazan-model**, or **user-defined** from the **Frequency** drop-down list:

- If you select **constant**, specify a value in the adjacent field.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you select **liao-breakage-model**, the **Liao Breakage Model Parameters** dialog box will open automatically. Note that because this is a modal dialog box, you must tend to it immediately.

  Figure 26.96: Liao Breakage Model Parameters Dialog Box



  You can specify the following parameters:

  – **Surface Tension**

  – **Turbulence Breakup Coefficient**: Is B_{turb} in Equation 14.720 in the *Fluent Theory Guide*.

  – **Eddy Breakup Coefficient**: Is B_{eddy} in Equation 14.722 in the *Fluent Theory Guide*.

  – **Shear Breakup Coefficient**: Is B_{shear} in Equation 14.721 in the *Fluent Theory Guide*.

  – **Friction Breakup Coefficient**: Is B_{fric} in Equation 14.723 in the *Fluent Theory Guide*.

  For more information about this model and its parameters, see Liao Breakage Kernel in the *Fluent Theory Guide*.

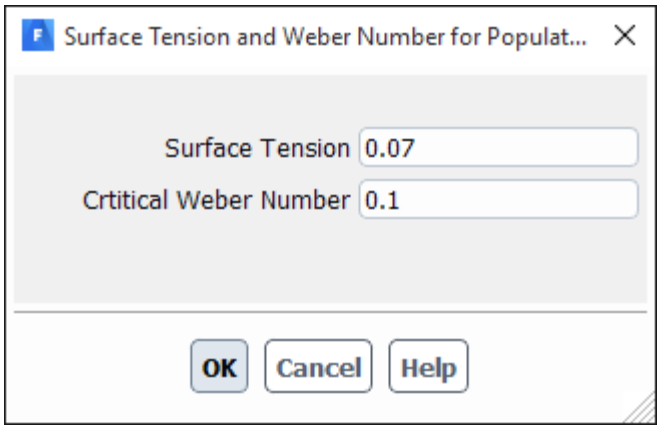
- If you select **luo-model**, the **Surface Tension for Population Balance** dialog box will open automatically to enable you to specify the surface tension (see Figure 26.93: The Surface Tension for Population Balance Dialog Box (p. 3138)). The frequency used in the breakage rate will then be calculated based on Luo's breakage kernel (as described in Particle Birth and Death Due to Breakage and Aggregation).

- If you select **lehr-model**, the **Surface Tension and Weber Number** dialog box will open automatically to enable you to specify the surface tension and critical Weber number (see

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

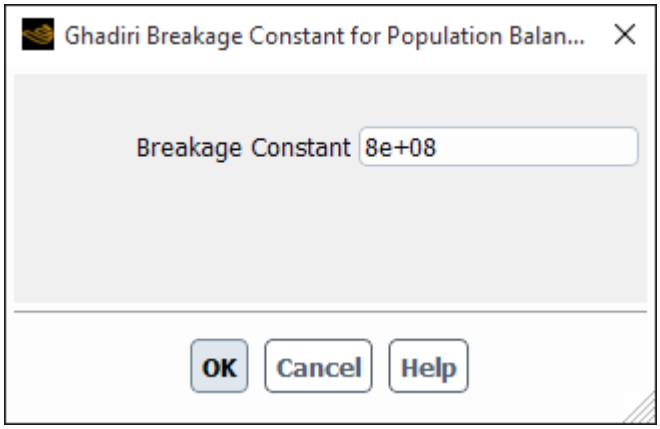
Figure 26.97: The Surface Tension and Weber Number Dialog Box (p. 3141)). The frequency used in the breakage rate will then be calculated based on Lehr's breakage kernel (as described in Particle Birth and Death Due to Breakage and Aggregation).

Figure 26.97: The Surface Tension and Weber Number Dialog Box



- If you select **ghadiri-model**, the **Ghadiri Breakage Constant for Population Balance** dialog box will open automatically to enable you to specify the breakage constant (see Figure 26.98: The Ghadiri Breakage Constant for Population Balance Dialog Box (p. 3141)). The frequency will then be calculated based on Ghadiri's breakage kernel (as described in Particle Birth and Death Due to Breakage and Aggregation).

Figure 26.98: The Ghadiri Breakage Constant for Population Balance Dialog Box



- If you select **laakkonen-model**, the **Surface Tension for Population Balance** dialog box will open automatically to enable you to specify the **Surface Tension** and the constant **C2** (Laakkonen Breakage Kernels). The frequency will then be calculated based on Laakkonen's breakage kernel (as described in Laakkonen Breakage Kernels).
- When you select **martinez-bazan-model**, the Martinez-Bazan model is used to model the breakage rate. The Martinez-Bazan frequency predicts the breakage frequency of parent bubbles/particles based on the local values of a maximum stable bubble size as described in Martinez-Bazan Breakage Kernel in the *Fluent Theory Guide*. This model is widely used in stirred tank reactors.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

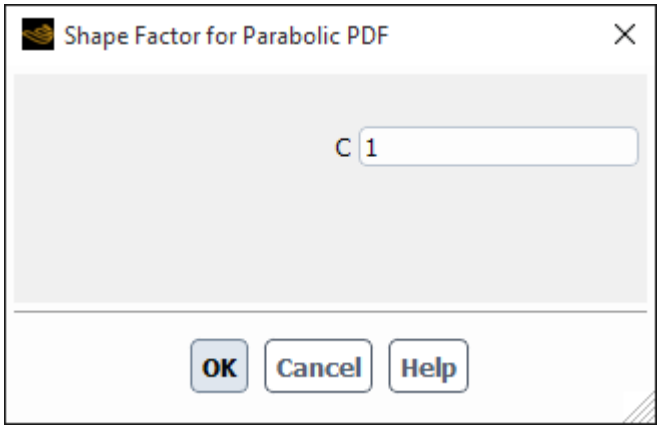
- If you have a user-defined function (UDF) that you want to use to model the frequency for the breakage rate, you can choose the **user-defined** option and specify the appropriate UDF.

If you selected **constant**, **ghadiri-model**, **laakkonen-model**, **martinez-bazan-model**, or **user-defined** for **Frequency**, then you can specify the probability density function used to calculate the breakage rate by making a selection in the **PDF** drop-down list. You can select the following models for the breakage **PDF**:

- **parabolic**
- **laakkonen**
- **generalized**
- **inverted-u-pdf**
- **user-defined**

- If you select **parabolic**, the **Shape Factor for Parabolic PDF** dialog box will open automatically to enable you to specify the shape factor **C** (see Figure 26.99: The Shape Factor for Parabolic PDF Dialog Box (p. 3142)). The PDF used in the breakage rate will then be calculated according to Equation 14.724 (as described in Particle Birth and Death Due to Breakage and Aggregation).

Figure 26.99: The Shape Factor for Parabolic PDF Dialog Box

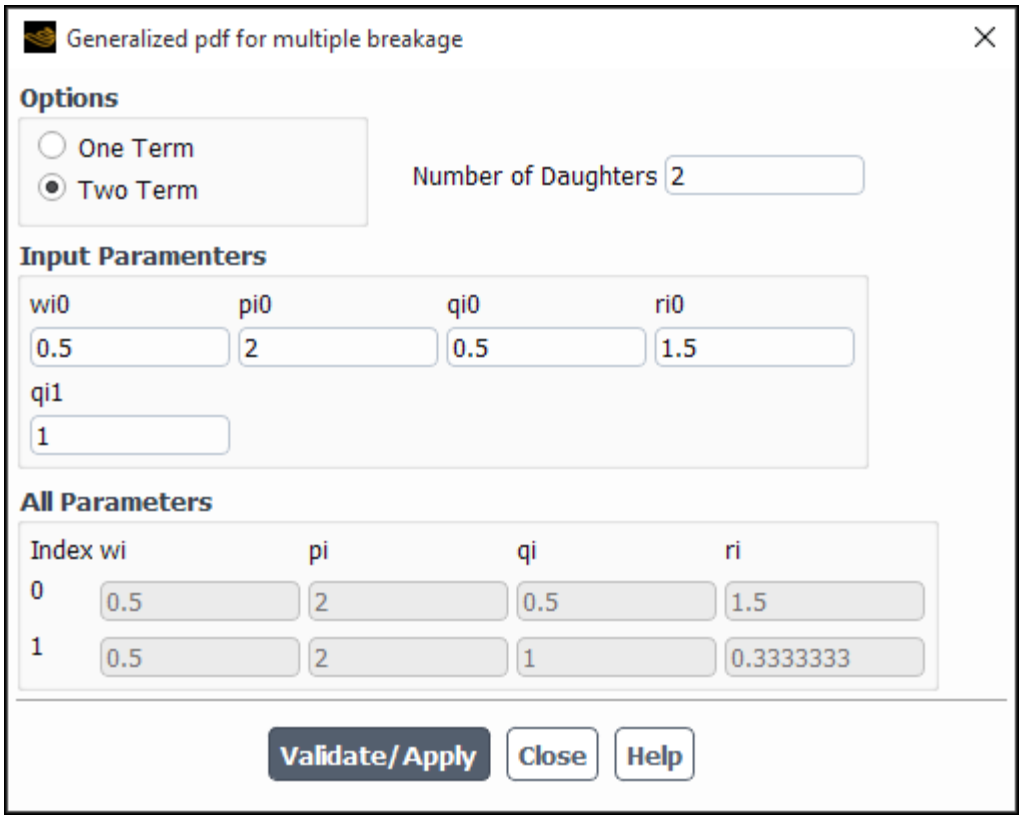


- When you select **laakkonen**, the daughter PDF is modeled using the Laakkonen model as described in Laakkonen Breakage Kernels in the *Fluent Theory Guide*.

- If you select **generalized**, the **Generalized pdf for multiple breakage** dialog box will open automatically (Figure 26.100: The Generalized pdf for multiple breakage Dialog Box (p. 3143)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.100: The Generalized pdf for multiple breakage Dialog Box



Perform the following steps in the **Generalized pdf for multiple breakage** dialog box:

a. Select either **One Term** or **Two Term** from the **Options** list. Your selection will determine whether i in Equation 14.730 is 0 or 1, respectively.

b. Enter a value for the averaged **Number of Daughters**. It can be any real number (including non-integers, such as 2.5), as long as it is not less than 2.

c. Define the parameter(s) for Equation 14.730 in the **Input Parameters** group box. When **One Term** is selected from the **Options** list, you must enter a value for **qi0**. When **Two Term** is selected from the **Options** list, you must enter values for **wi0**, **pi0**, **qi0**, **ri0**, and **qi1**. For information about appropriate values for these parameters to result in the daughter distributions shown in Table 14.6: Daughter Distributions, see Table 14.7: Daughter Distributions (cont.).

Note:

For the equal-size generalized pdf breakage distribution, the value for **qi0** should be set to `1e20`.

---

d. Click the **Validate/Apply** button to save the settings. The text boxes in the **All Parameters** group box will be updated, using the values you entered in the **Input Parameters** group box, as well as values derived from the constraints shown in Equation 14.731 – Equation 14.733.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

e. Verify that the values in the **All Parameters** group box represent your intended PDF before clicking **Close**.

- When you select **inverted-u-pdf**, the inverted distribution PDF is used to model bubble breakage as described in Inverted U-PDF in the *Fluent Theory Guide*. This model can produce more physical results in simulations of certain impeller-agitated mixing tanks.
- If you have a user-defined function (UDF) that you want to use to model the PDF for the breakage rate, you can choose the **user-defined** option and specify the appropriate UDF. See UDFs for Population Balance Modeling (p. 3163) for details about UDFs for the population balance model.

Choose between the default **Hagesather** formulation and the **Ramakrishna** formulation. Detailed information about these two methods can be found in Breakage Formulations for the Discrete Method.

Note:

Since most standard breakage/aggregation kernels depend on the primary phase turbulence, Ansys Fluent assumes a volume fraction cutoff of 0.8 or less of secondary phases (that is, 0.2 or more of a primary phase) for breakage or aggregation to be enabled. This cutoff volume fraction can be modified by the following text command:

```
define/models/multiphase/population-balance/phenomena/break-
age-aggregation-vof-cutoff
```

7. (Discrete, QMOM, and Inhomogeneous Discrete models only) If you want to account for bubble expansion due to large changes in hydrostatic pressure in compressible flows, enable **Include Expansion**. The bubble expansion is modeled as a growth term in the Population Balance equations (Equation 14.703). This can be relevant in applications such as deep-water drilling. For more information, see Particle Growth.

Note:

The secondary phase must be modeled as compressible. This option is currently available for Discrete and QMOM only.

8. Specify the boundary conditions for the solution variables.

Setup → Boundary Conditions

See Defining Population Balance Boundary Conditions (p. 3149) below.

9. Specify the initial guess for the solution variables.

Solution → Initialization

10. Solve the problem and perform relevant postprocessing functions.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

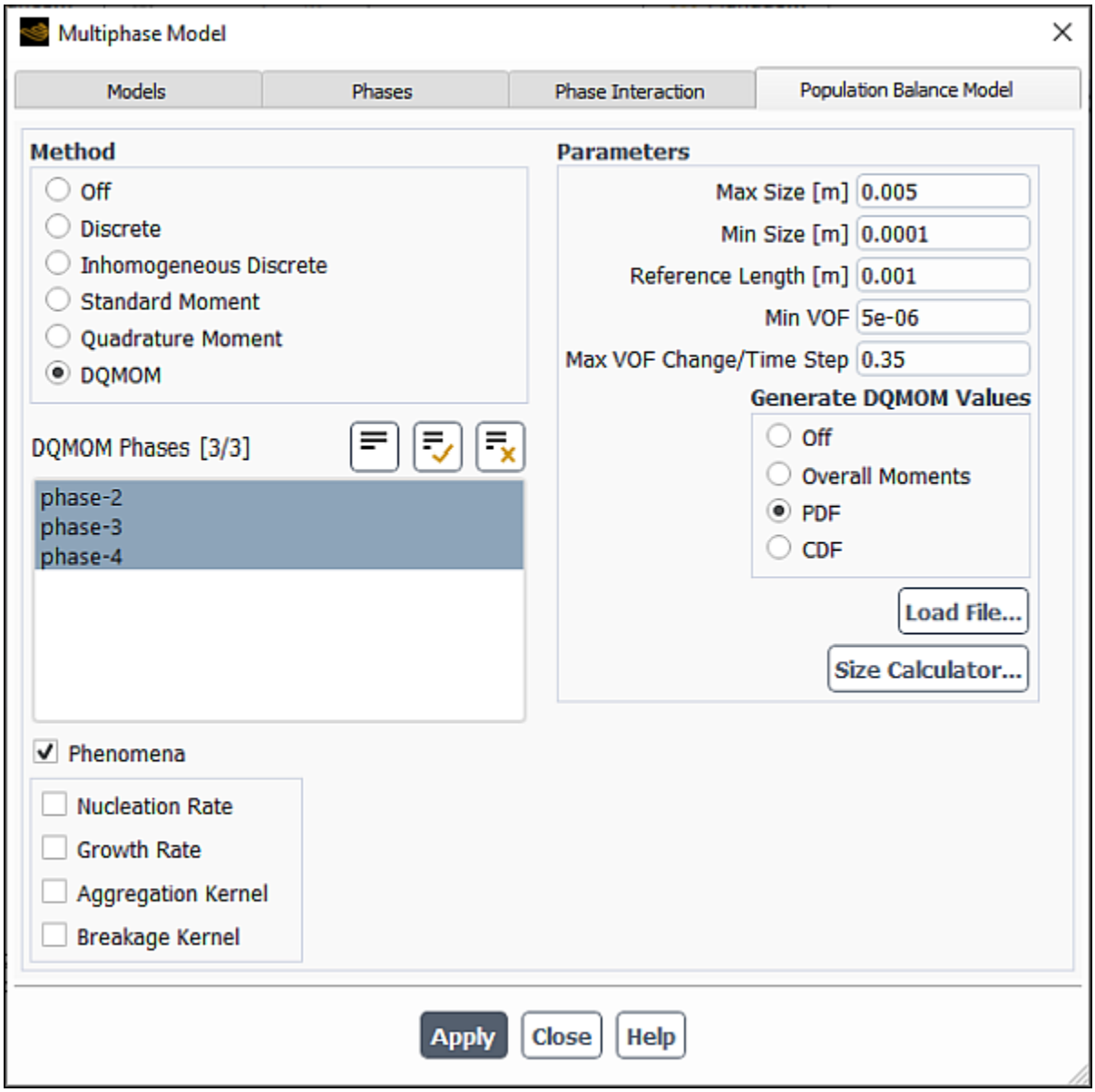
Solution → Run Calculation

See Postprocessing for the Population Balance Model (p. 3160) for details about postprocessing.

26.6.1.1.1. Generated DQMOM Values

When using the **DQMOM** model, you have the option of generating DQMOM values from three different file formats. To do so, select **Overall Moments**, **PDF** or **CDF** under **Generate DQMOM Values** and then click the **Load File...** button. The **Select File** dialog box will open where you will select the appropriate file. Clicking **OK** in the **Select File** dialog box will result in the file being read and calculations carried out. The DQMOM values will be printed in the console.

Figure 26.101: The Population Balance Model Tab for the DQMOM Model



DQMOM Values Produced From PDF, CDF Files, or Overall Moments for the Particles

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Three quadrature points are assumed, namely QP0, QP1, and QP2 (see Figure 26.102: DQMOM Values Produced From a PDF File (p. 3146)).
- `Length`, `Volume Fraction`, and `DQMOM-m4` values are given. The latter two can be used for initial fields of VOF and DQMOM as well as boundary conditions.
- For verification purposes, the first six moments are also given together with the total volume fraction of all particles from the PDF or CDF file. It is your responsibility to make sure that these values are correct, especially the total volume fraction.
- The definition of the first six moments of the particles is length based for the overall moments, described in The Quadrature Method of Moments (QMOM).
- In PDF or CDF format, the resultant volume fraction is normally given as unity. If you want the real particle volume fraction to be reflected in the mixture, the second column of the PDF or CDF data file need to be multiplied by the value of the real volume fraction. Another way is to multiply the values of the generated DQMOM volume fraction and DQMOM-m4 using the real particle volume fraction.

Figure 26.102: DQMOM Values Produced From a PDF File

```
 First 6 Moments computed from PDF:

       m0 = 1.730757e+13
       m1 = 5.442254e+08
       m2 = 2.802300e+04
       m3 = 1.909819e+00  (Total Volume Fraction is 9.999800e-01)
       m4 = 1.533388e-04
       m5 = 1.374813e-08

 DQMOM values produced from PDF:

                    QP0                 QP1                 QP2
 Length (m)         1.050580e-04        5.154987e-05        1.282842e-05
 Volume Fraction    5.452821e-01        4.433921e-01        1.130576e-02
 DQMOM-m4 (m)       5.728627e-05        2.285681e-05        1.450350e-07
```

PDF File Format

- The probability density function (PDF) is defined by the probability distribution of particles in terms of the volume fraction over the particle length (namely the diameter of the particle).
- The integration of PDF over all possible particle length (normally from 0 to the maximum diameter) shall give a value of unity or the real value of the volume fraction of all participating particles.
- The following file format is required (as shown below):
  - An integer number specified in the first line, indicating the number of data pairs to follow
  - The data in the 1st column specifying the length or diameter of particles in ascending order in meters (m)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– The data in the 2nd column specifying the probability density function. Be aware that the integration of the PDF over the length shall result in a value of volume fraction for that particular particle length range

```
37
5e-6 0.000058e6
10e-6 0.000271e6
15e-6 0.000669e6
20e-6 0.001264e6
25e-6 0.002062e6
30e-6 0.003055e6
35e-6 0.004228e6
40e-6 0.005549e6
45e-6 0.006972e6
50e-6 0.008439e6
55e-6 0.00988e6
60e-6 0.011217e6
65e-6 0.012366e6
70e-6 0.013253e6
75e-6 0.013811e6
80e-6 0.013996e6
85e-6 0.013789e6
90e-6 0.013199e6
95e-6 0.012268e6
100e-6 0.011062e6
105e-6 0.009668e6
110e-6 0.00818e6
115e-6 0.006694e6
120e-6 0.00529e6
125e-6 0.004033e6
130e-6 0.002962e6
135e-6 0.002093e6
140e-6 0.00142e6
145e-6 0.000925e6
150e-6 0.000576e6
155e-6 0.000344e6
160e-6 0.000196e6
170e-6 0.000055e6
180e-6 0.000012e6
190e-6 0.000002e6
200e-6 0.
210e-6  0.
```

CDF File Format

- The cumulative density function (CDF) is defined as the integration of PDF over all possible particles up to the length L, resulting in a value of volume fraction for all particles less than length L.
- The value of the CDF at the maximum particle length/diameter shall be unity, or the real value of the volume fraction of all particles in the mixture.
- The following file format is required (as shown below):
  - An integer number specified in the first line, indicating the number of data pairs to follow
  - The data in the 1st column specifying the length or diameter of particles in ascending order in meters (m)
  - The data in the 2nd column specifying the cumulative density function in terms of the volume fraction of particles

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– $CDF \leq 1$ at the maximum particle length

```
37
5e-6 0
10e-6 0.109e-2
15e-6 0.16e-2
20e-6 0.175e-2
25e-6 0.208e-2
30e-6 0.304e-2
35e-6 0.614e-2
40e-6 1.5e-2
44e-6   3.e-2
45e-6   3.44e-2
50e-6 6.573e-2
55e-6 11.19e-2
60e-6 17.11e-2
65e-6 24.17e-2
70e-6 32.007e-2
75e-6 40.243e-2
80e-6 48.397e-2
85e-6 56.453e-2
90e-6 63.823e-2
95e-6 70.53e-2
100e-6 76.073e-2
105e-6 81e-2
110e-6 85.057e-2
115e-6 88.187e-2
120e-6 90.89e-2
125e-6 92.897e-2
130e-6 94.437e-2
135e-6 95.543e-2
140e-6 96.523e-2
145e-6 97.173e-2
150e-6 97.717e-2
155e-6 98.007e-2
160e-6 98.363e-2
170e-6 98.88e-2
180e-6 99.16e-2
190e-6 99.327e-2
200e-6 100.e-2
```

Overall Moments File Format

- The third option is to specify the first six moments for all particles as shown below.
- The following file format is required (as shown below):
  - An integer number specified in the first line, indicating the number of data (moments) to follow. By default, this shall be 6
  - six moments from moment-0 to moment-5 are given in that order. The definition of the first six moments is based on length as described in The Quadrature Method of Moments (QMOM)
  - As a check for moments, $Moment\text{-}3(m3) = {}^{1}\!/_{(K_v)(volume_fraction)}$. In the values given below, volume fraction is assumed to be unity and K_v is assumed to be ${}^{\pi}\!/_{6}$

```
6
1.120556e+013
4.022475e+008
2.523370e+004
1.909857e+000
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
1.611191e-004
1.498663e-008
```

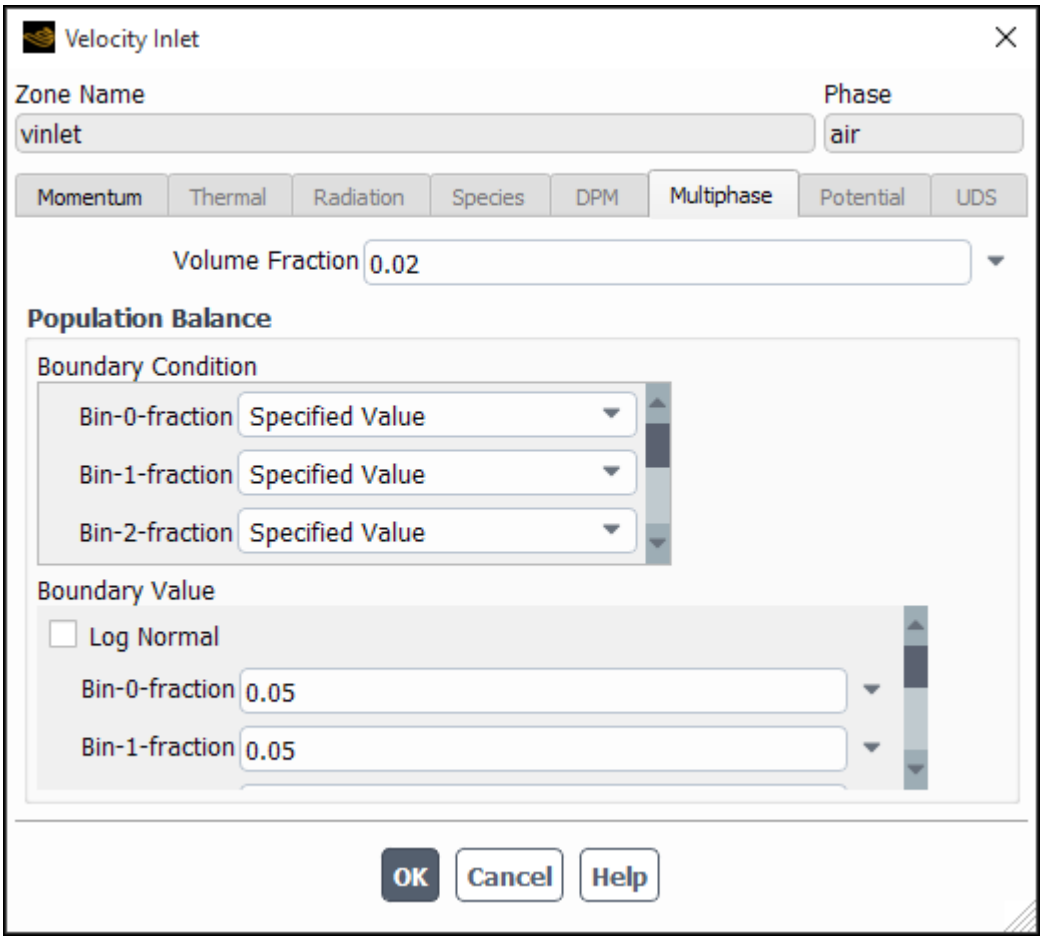
26.6.1.2. Defining Population Balance Boundary Conditions

To define boundary conditions specific to the population balance model, use the following procedure:

1. In the **Boundary Conditions** task page, select the secondary phase(s) in the **Phase** drop-down list and then open the appropriate boundary condition dialog box (for example, Figure 26.103: Specifying Inlet Boundary Conditions for the Population Balance Model (p. 3149)).

   Setup → Boundary Conditions

   Figure 26.103: Specifying Inlet Boundary Conditions for the Population Balance Model



2. In the **Multiphase** tab, under **Boundary Condition**, select the type of boundary condition for each bin (for the discrete method) or moment (for SMM and QMOM) as either **Specified Value** or **Specified Flux**.

   Note that the boundary condition variables (for example, **Bin-0**) are labeled according to the following:

   bin/moment **-** i^{th} bin/moment

where the i^{th} *bin/moment* can range from **0** (the first bin or moment) to $N-1$, where N is the number of bins/moments that you entered in the **Population Balance Model** tab.

3. Under **Population Balance Boundary Value**, enter a value or a flux as appropriate.

   - If you selected **Specified Value** for the selected boundary variable, enter a value in the field adjacent to the variable name. This value will correspond to the variable f_i in Equation 14.786 (for the discrete method) or m_k in Equation 14.801 (for SMM or QMOM). If you are using either of the discrete methods and have selected **Specified Value** for all bins, you can optionally specify a log-normal distribution and have Fluent automatically initialize the bin fractions accordingly to a log-normal distribution (see Initializing Bin Fractions With a Log-Normal Distribution (p. 3150)).

   - If you selected **Specified Flux** for the selected boundary variable, enter a value in the field adjacent to the variable name. This value will be the spatial particle volume flux dV / dx_i.

26.6.1.2.1. Initializing Bin Fractions With a Log-Normal Distribution

When using one of the discrete methods with Specified Value selected as the boundary condition for all bins, you can have Fluent populate the bin fractions according to a log-normal distribution characterized by a mean and standard deviation that you supply. For details of the log-normal distribution, refer to The Log-Normal Distribution.

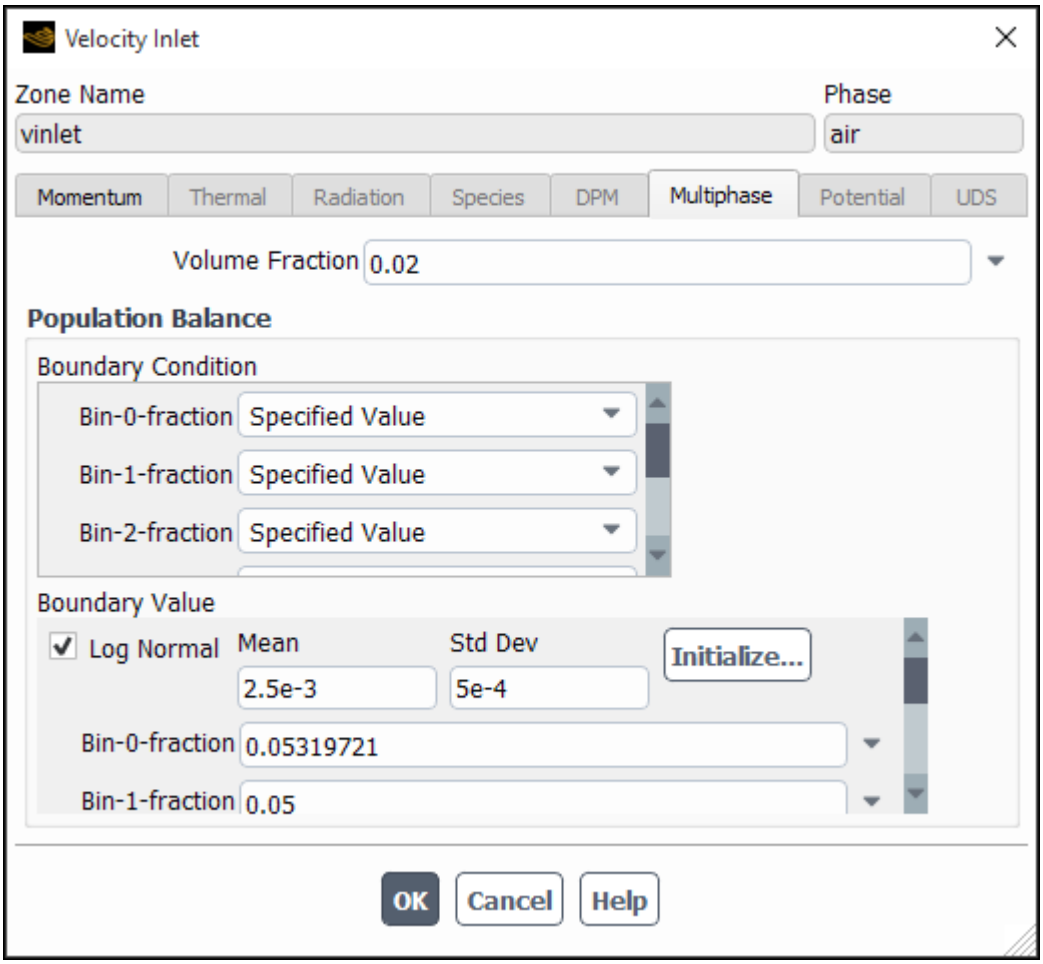
Important:

The log-normal initialization feature is only meaningful and should only be used when **Specified Value** is selected for all bins under **Boundary Condition**.

To use the log-normal distribution, perform the following steps.

1. Enable **Log Normal** in the **Boundary Value** group box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Enter values for the **Mean** and **Std Dev** of the desired distribution.

3. Click **Initialize...**

26.6.1.3. Defining Population Balance Cell Zones Conditions

For the **Discrete** and **Inhomogeneous Discrete** population balance models, you can specify source terms for bin fractions. For general information about defining source terms, see Defining Mass, Momentum, Energy, and Other Sources (p. 1382). For additional details, see Procedure for Defining Sources (p. 1383) and Discrete Bin Fraction Sources for the Population Balance Model (p. 1387).

26.6.1.4. Specifying Population Balance Solution Controls

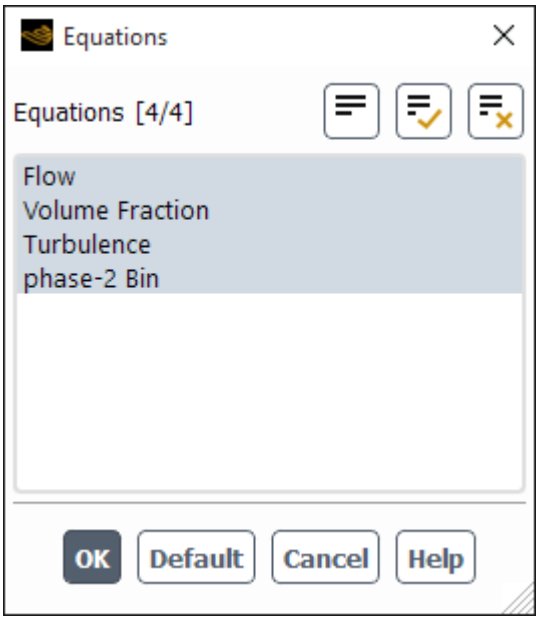
In the **Equations** dialog box (Figure 26.104: The Equations Dialog Box (p. 3152)), equations for each bin (for example, **phase-2 Bin**) will appear in the **Equations** list.

Solution → **Controls** **Equations...**

The default value under **Under-Relaxation Factors** (in the **Solution Controls** task page) for the population balance equations is `0.5`, and the default **Discretization** scheme (in the **Solution Methods** task page) is **First Order Upwind**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.104: The Equations Dialog Box



26.6.1.5. Coupling With Fluid Dynamics

To couple population balance modeling of the secondary phase(s) with the overall problem fluid dynamics, a Sauter mean diameter (d_{32} in Equation 14.808) may be used to represent the particle diameter of the secondary phase. The Sauter mean diameter is defined as the ratio of the third moment to the second moment for the SMM and QMOM. For the discrete method, it is defined as

$$d_{32}=\frac{\sum N_i L_i^3}{\sum N_i L_i^2} \tag{26.27}$$

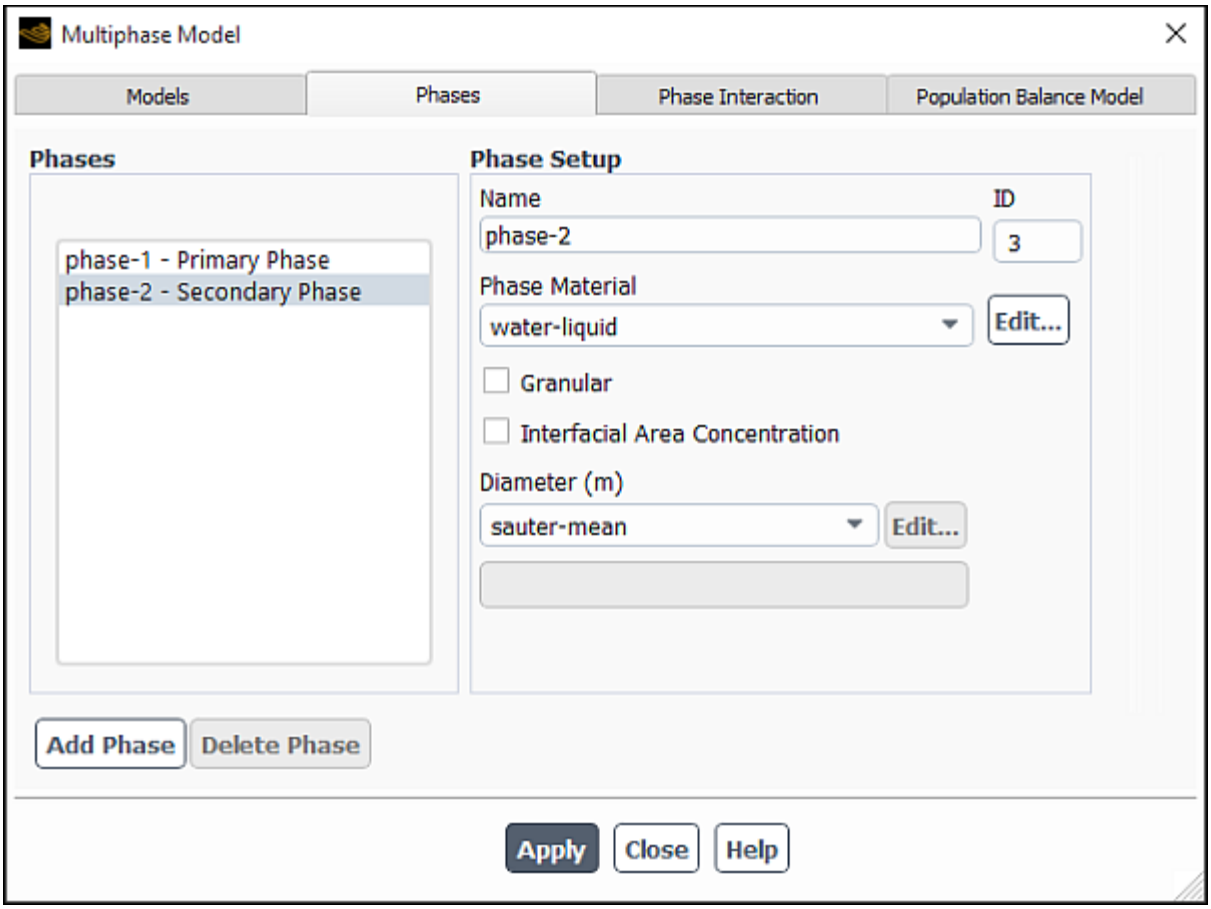
To specify the Sauter mean diameter as the secondary phase particle diameter, open the **Phases** tab

Setup → Models → Multiphase → Edit...

In the **Multiphase Model** dialog box (**Phases** tab) (for example, Figure 26.105: Setting the Secondary Phase for Hydrodynamic Coupling (p. 3153)), select **sauter-mean** from the **Diameter** drop-down list under **Properties**. Note that a constant diameter or user-defined function may also be used.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.105: Setting the Secondary Phase for Hydrodynamic Coupling



26.6.1.6. Specifying Interphase Mass Transfer Due to Nucleation and Growth

In applications that involve the creation, dissolution, or growth of particles (such as crystallization), the total volume fraction equation for the particulate phase will have source terms due to these phenomena. The momentum equation for the particulate phase will also have source terms due to the added mass. In Ansys Fluent, the mass source term can be specified using the UDF hook DEFINE_HET_RXN_RATE, as described in DEFINE_HET_RXN_RATE Macro (p. 3170), or using the **Phase Interaction** tab, described below.

As an example, in crystallization, particles are created by means of nucleation ($\dot{n}_0$), and a growth rate (G) can also be specified. The mass transfer rate of formation (in $\mathrm{kg/m^3\text{–}s}$) of particles of all sizes is then

$$\begin{aligned} \dot{m} &= 3\rho K_v \int_0^\infty L^2 G n(L) dL \\ &= \tfrac{1}{2}\rho K_a \int_0^\infty L^2 G n(L) dL \end{aligned} \tag{26.28}$$

For the discrete method, the mass transfer rate due to growth can be written as

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\begin{aligned} \dot{m} &= \rho \int_0^\infty G_v n(L) dL \\ &= \rho \int_0^\infty G_v n(V) dV \\ &= \rho \sum_i G_{v,i} N_i \end{aligned} \tag{26.29}$$

If the nucleation rate is included in the total mass transfer, then the mass transfer becomes

$$\dot{m} = \rho V_0 \dot{n}_0 + \sum_i \rho G_{v,i} N_i \tag{26.30}$$

Important:

For the discrete method, the sources to the population balance equations must sum to the total mass transfer rate. To access the sources, you can use the macro `C_PB_DISCI_PS (cell, thread, i)`.

See UDFs for Population Balance Modeling (p. 3163) for more information about macros for population balance variables.

For the SMM, only a size-independent growth rate is available. Hence, the mass transfer rate can be written as

$$\dot{m} = \frac{1}{2} \rho K_a G m_2 \tag{26.31}$$

For the QMOM, the mass transfer rate can be written as

$$\dot{m} = \frac{1}{2} \rho K_a \sum_i L_i^2 w_i G(L_i) \tag{26.32}$$

For both the SMM and QMOM, mass transfer due to nucleation is negligible, and is not taken into account.

Important:

Note that for crystallization, the primary phase has multiple components; at the very least, there is a solute and a solvent. To define the multicomponent multiphase system, you will need to activate **Species Transport** in the **Species Model** dialog box for the primary phase after activating the multiphase model. The rest of the procedure for setting up a species transport problem is identical to setting up species in single phase. The heterogeneous reaction is defined as:

$$liquid(solvent) \rightarrow crystal(solute)$$

When the population balance model is activated, mass transfer between phases for non-reacting species (such as boiling) and heterogeneous reactions (such as crystallization) can be done automatically, in lieu of hooking a UDF.

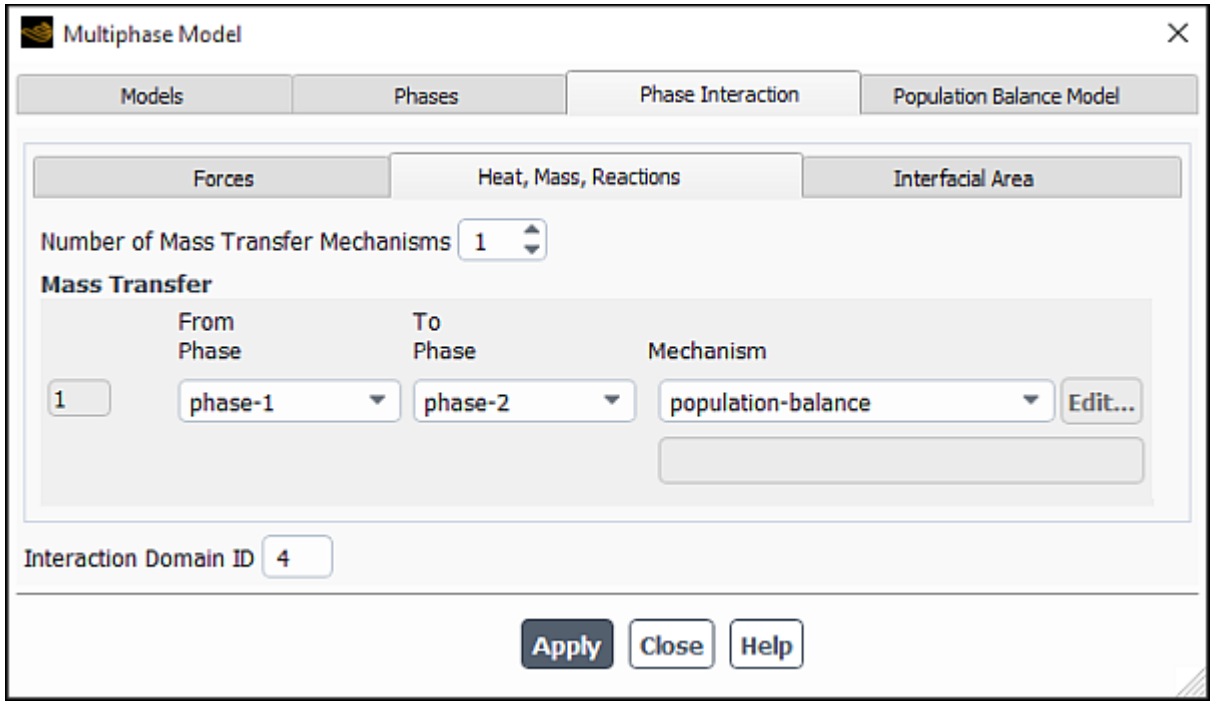
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For simple unidirectional mass transfer between primary and secondary phases due to nucleation and growth phenomena of non-reacting species, configure the following settings:

1. In the **Multiphase Model** dialog box, open the **Phase Interaction** > **Heat, Mass, Reactions** > **Mass** tab (Figure 26.106: The Phase Interaction Tab for Non-reacting Species (p. 3155)).

   Setup → Models → Multiphase → Edit...

   Figure 26.106: The Phase Interaction Tab for Non-reacting Species



2. Specify the **Number of Mass Transfer Mechanisms** involved in your case.

3. For each mechanism, specify the phase of the source material under **From Phase** and the phase of the destination material phase under **To Phase**.

4. From the **Mechanism** drop down list, select one of the mechanisms:

   none

   if you do not want any mass transfer between the phases.

   constant-rate

   for a fixed, user-specified rate.

   user-defined

   if you hooked a UDF describing the mass transfer mechanism.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

population-balance

for an automated method of mass transfer, not involving a UDF. The nucleation and the growth rates calculated by the population balance kernels are used for mass transfer.

Note:

For the **Inhomogeneous Discrete** population balance model, where there is more than one secondary phase, you can select `population-balance` as the mechanism of mass transfer between the solvent phase (say for crystallization) and each of the solute phases defined under the **Inhomogeneous Discrete** population balance model.

---

5. Click **Apply** to save the settings.

For heterogeneous reactions, configure the following settings:

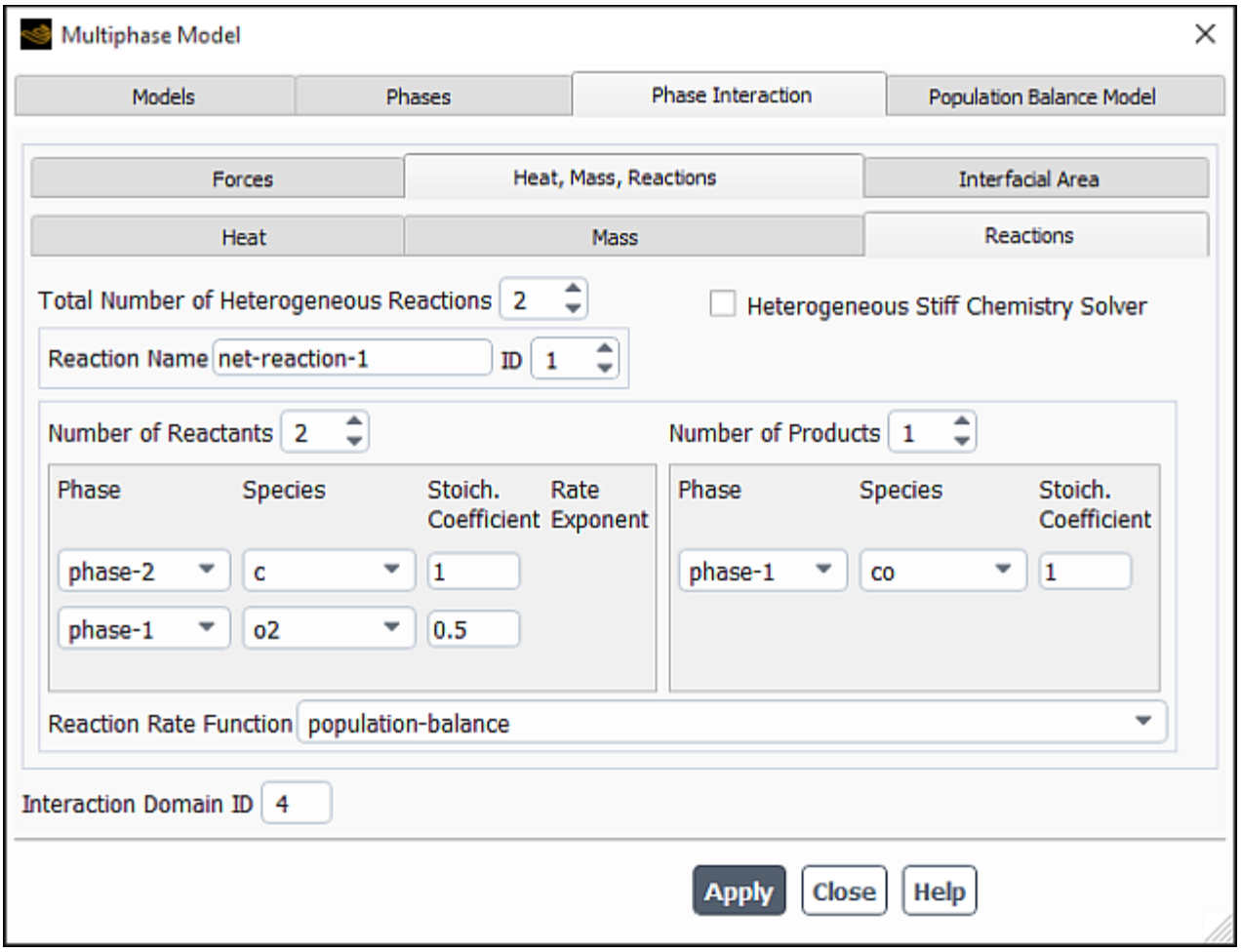
1. For the primary phase, activate the **Species Transport** model.
2. Open the **Phase Interaction** > **Heat, Mass, Reactions** > **Reactions** tab in the **Multiphase Model** dialog box (Figure 26.107: The Reactions Tab for a Heterogeneous Reaction (p. 3157)).

   Setup → **Models** → **Multiphase** **Edit...**

3. Under the **Reactions** tab, specify the stoichiometry for the reactant and the product.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.107: The Reactions Tab for a Heterogeneous Reaction



4. Select `population-balance` as the **Reaction Rate Function**.

5. Click **OK** to save the settings.

Either this method or the use of the UDF, described in `DEFINE_HET_RXN_RATE` Macro (p. 3170), will produce the same results.

For the **Inhomogeneous Discrete** population balance model involving nucleation and growth, you can select `population-balance` as the **Reaction Rate Function** for each heterogeneous reaction you have set up. To learn how to set up reactions, go to Specifying Heterogeneous Reactions (p. 2937) in the User's Guide (p. 1).

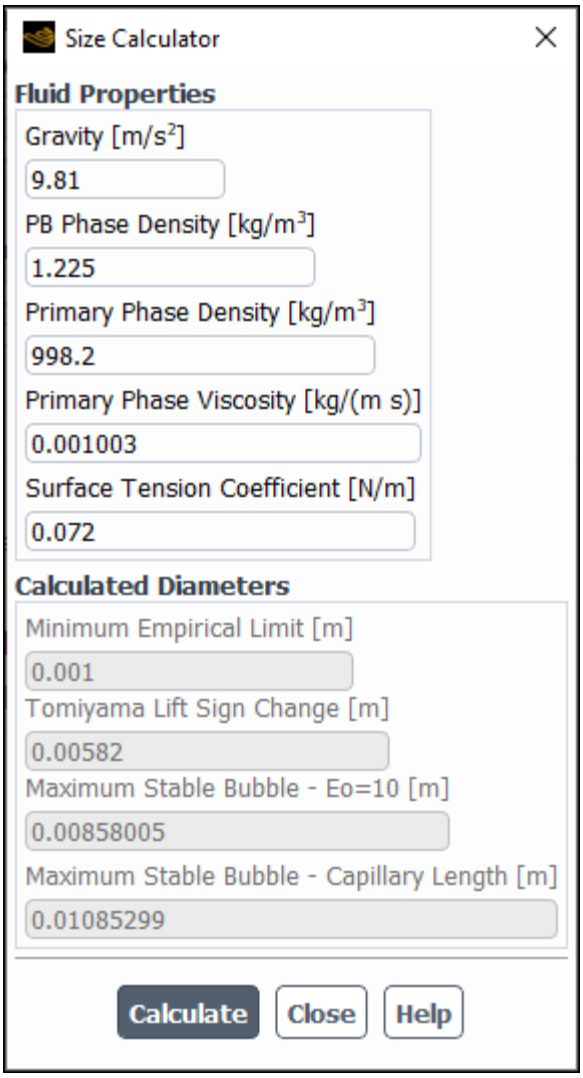
26.6.1.7. Size Calculator

The size calculator provides estimation for appropriate bubble sizes and/or droplet size limits. To use this tool:

1. Click **Size Calculator...** in the **Population Balance Model** tab (**Parameters** group).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.108: The Size Calculator Dialog Box



2. In the **Size Calculator** dialog box that opens, adjust the following parameters in the **Fluid Properties** group box:

   - **Gravity**
   - **PB Phase Density**
   - **Primary Phase Density**
   - **Primary Phase Viscosity**
   - **Surface Tension Coefficient**

   Note the following:

   - When you open the dialog box for the first time, the entry fields in the **Fluid Properties** group box will be pre-populated with the case fluids information if constant properties are used. Otherwise, it will be pre-populated with gas-liquid information under standard pressure/temperature conditions. You need to review these values and modify them, as necessary.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Gravity** refers to the gravity magnitude, which is computed based on the operating conditions specification.

3. Click **Calculate**.

   In the **Calculated Diameters** group box, the solver gives the following fluid particle size estimations:

   - For bubbly flows:
     - **Minimum Empirical Limit**: This size is based on the minimum particle size limit for typical experimental acquisition methods.
     - **Lift Sign Inversion**: This size is calculated using the sign inversion from Equation 14.306 in the *Fluent Theory Guide*.
     - **Maximum Stable Bubble – Eo = 10**: This size is calculated using the modified Eötvös number calculated from Equation 14.308 in the *Fluent Theory Guide*.
     - **Maximum Stable Bubble – Capillary Length**: This size is calculated as 4 times the capillary length. It is based on the maximum diameter before a bubble disintegrates or starts breaking down as given by Ishii and Zuber ("Drag Coefficient and Relative Velocity in Bubbly, Droplet or Particulate Flows" in the *Fluent Theory Guide*). Typically, interfacial forces are empirically limited to this size.
   - For droplets flows:
     - **Minimum Empirical Limit**: Same as for bubbly flows.
     - **Maximum Stable Droplet – We=12**: This size is calculated using a Weber number of 12.0, and it represents the maximum diameter at which droplets start disintegrating.
     - **Maximum Stable Droplet – Taylor Limit**: This size is calculated using twice the capillary length, and it should be used if the stability of the droplet in the flow is governed by Taylor instabilities.

   The recommended values will be also printed in the Fluent console.

   You can use these recommendations as inputs to the Population Balance model.

26.6.2. Solution Strategies

In simulations that involve the QMOM population balance model, Ansys Fluent sometimes suppresses the calculation of sources in cells of very low volume fraction to avoid potential convergence problems. However, in certain cases this has been found to be counterproductive because the computation of sources in all cells can be beneficial for a robust solution. To invoke the calculation of all the sources regardless of the local volume fraction, you can use the following text command:

```
define/models/multiphase/population-balance/expert/qmom/retain-qmom-sources-for-low-vof?
Retain QMOM sources? [no] yes
```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.6.3. Postprocessing for the Population Balance Model

Ansys Fluent provides postprocessing options for displaying, plotting, and reporting various particle quantities, which include the main solution variables and other auxiliary quantities.

26.6.3.1. Population Balance Solution Variables

26.6.3.2. Reporting Derived Population Balance Variables

26.6.3.1. Population Balance Solution Variables

Solution variables that can be reported for the population balance model are:

- **Bin-i fraction** (discrete method only), where i is N-1 bins/moments.
- **Number density of Bin-i fraction** (discrete method only)
- **Diffusion Coef. of Bin-i fraction/Moment-i**
- **Sources of Bin-i fraction/Moment-i**
- **Moment-i** (SMM and QMOM only)
- **Abscissa-i** (QMOM method only)
- **Weight-i** (QMOM method only)

Bin-i fraction (f_i) is the total volume fraction for the i^{th} size bin when using the discrete method.

Number density of Bin-i fraction is the number density (N_i) in $\text{particles}/\text{m}^3$ for the i^{th} size bin.

Moment-i is the i^{th} moment of the distribution when using the standard method of moments or the quadrature method of moments.

Important:

- Though the diffusion coefficients of the population variables (for example, **Diffusion Coef. of Bin-i fraction/Moment-i**) are available, they are set to zero because the diffusion term is not present in the population balance equations.
- In the Inhomogeneous Discrete population balance method involving 'n' bins, bin-0 and bin-(n-1) correspond to the largest and smallest bin sizes respectively.
- The sources of bin i available for post-processing represent the total of sources due to nucleation, growth, breakage, and coalescence phenomena and do not include any source terms specified for bins in the cell zone condition dialog box (for example, the **Fluid** dialog box).

26.6.3.2. Reporting Derived Population Balance Variables

Two options are available in the **Results** ribbon tab that allow you to report computed moments and number density on selected surfaces or cell zones of the domain.

26.6.3.2.1. Computing Moments

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

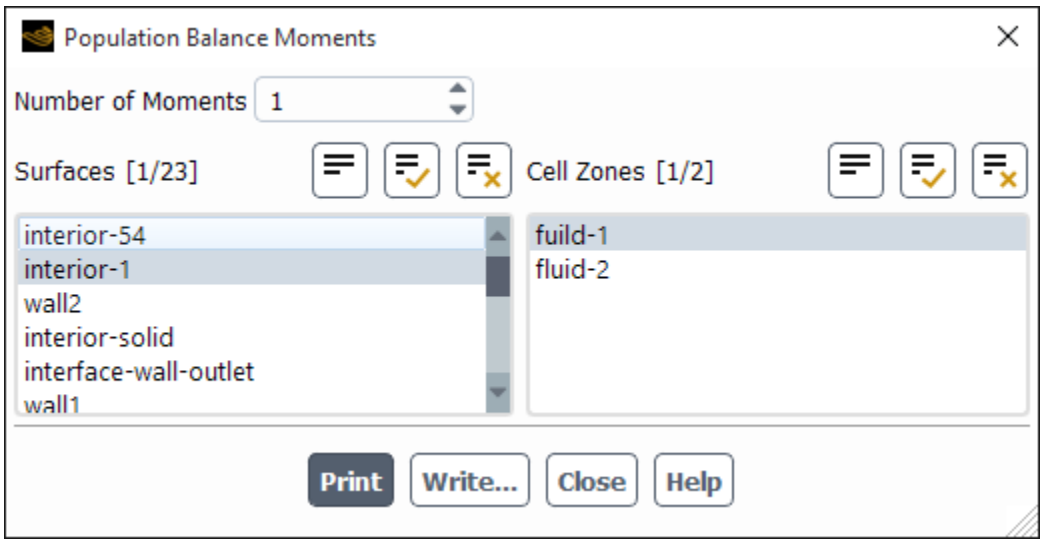
26.6.3.2.2. Displaying a Number Density Function

26.6.3.2.1. Computing Moments

You can compute moments for the population balance model using the **Population Balance Moments** dialog box (Figure 26.109: The Population Balance Moments Dialog Box (p. 3161)).

Results → **Model Specific** → **Population Balance** → **Moments...**

Figure 26.109: The Population Balance Moments Dialog Box



The steps for computing moments are as follows:

1. For the discrete method, specify the **Number of Moments**. For the SMM and QMOM, the number of moments is set equal to the number of moments that were solved, and therefore cannot be changed.

2. For a surface average, select the surface(s) on which to calculate the moments in the **Surfaces** list.

3. For a volume average, select the volume(s) in which to calculate the moments in the **Cell Zones** list.

4. Click **Print** to display the moment values in the console window.

5. To save the moment calculations to a file, click **Write...** and enter the appropriate information in the resulting **Select File** dialog box. The file extension should be `.pb`.

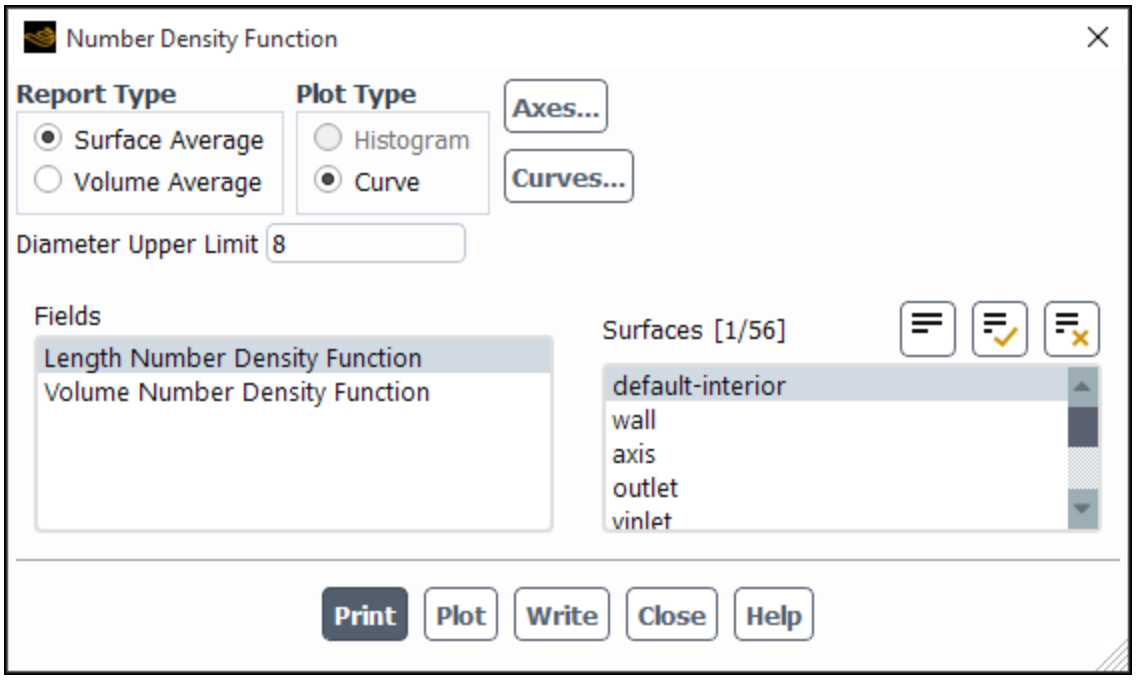
26.6.3.2.2. Displaying a Number Density Function

You can display the number density function for the population balance model using the **Number Density Function** dialog box (Figure 26.110: The Number Density Function Dialog Box (p. 3162)).

Results → **Model Specific** → **Population Balance** → **Number Density...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.110: The Number Density Function Dialog Box



The steps for displaying the number density function are as follows:

1. Specify the **Report Type** as either a **Surface Average** or a **Volume Average**.

2. Under **Plot Type**, specify how you would like to display the number density function data.

   Histogram

   displays a histogram of the discrete number density (N_i). The number of divisions in the histogram is equal to the number of bins specified in the **Population Balance Model** tab of the **Multiphase Model** dialog box. This option is available only with the discrete method.

   Curve

   displays a smooth curve of the number density function.

3. (Quadrature Moment method only) Specify **Diameter Upper Limit** to reconstruct the length or volume-based number density function based on the moments predicted by QMOM (see Reconstructing the Particle Size Distribution from Moments in the *Fluent Theory Guide* for more information).

   The moments are calculated by integrating the area under the number density curve. The accuracy of the calculations is dependent on the integration limit. Typically, this is expressed in terms of the mean of the distribution and is specified in the **Diameter Upper Limit** field. The default value of 1.5 means that the maximum diameter value considered is 1.5 times the mean of the distribution.

4. In the **Fields** list, select the data to be plotted.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Discrete Number Density

(N_i) is the number of particles per unit volume of physical space in the i^{th} size bin plotted against particle diameter size i. This option is available only with the discrete method.

Length Number Density Function

($n(L)$) is the number of particles per unit volume of physical space per unit particle length plotted against particle diameter.

Volume Number Density Function

($n(V)$) is the number of particles per unit volume of physical space per unit particle volume plotted against particle volume.

5. Choose the cell zones on which to plot the number density function data in the **Cell Zones** list.
6. Click **Plot...** to display the data.
7. (optional) Click **Print** to display the number density function data in the console window.
8. Click **Write** to save the number density function data to a file. The **Select File** dialog box will open, where you can specify a name and save a file containing the plot data. The file extension should be `.pbd`.

26.6.4. UDFs for Population Balance Modeling

This section contains the following sections:

26.6.4.1. Population Balance Variables
26.6.4.2. Population Balance DEFINE Macros
26.6.4.3. Hooking a Population Balance UDF to Ansys Fluent

26.6.4.1. Population Balance Variables

The macros listed in Table 26.14: Macros for Population Balance Variables Defined in `sg_pb.h` (p. 3163) can be used to return `real` variables associated with the population balance model. The variables are available in both the pressure-based and density-based solvers. The macros are defined in the `sg_pb.h` header file, which is included in `udf.h`.

Table 26.14: Macros for Population Balance Variables Defined in `sg_pb.h`

Macro	Argument Types	Returns
`C_PB_DISCI`	`cell_t c, Thread *t, int i`	(f_i) the total volume fraction for the i^{th} size bin
`C_PB_SMMI`	`cell_t c, Thread *t, int i`	i^{th} moment
`C_PB_QMOMI`	`cell_t c, Thread *t, int i`	i^{th} moment, where i=0,1,2,3,4,5

Macro	Argument Types	Returns
C_PB_QMOMI_L	cell_t c, Thread *t, int i	abscissa L_i, where i=0,1,2
C_PB_QMOMI_W	cell_t c, Thread *t, int i	weight w_i, where i=0,1,2
C_PB_DISCI_PS	cell_t c, Thread *t, int i	net source term to i^{th} size bin
C_PB_SMMI_PS	cell_t c, Thread *t, int i	net source term to i^{th} moment
C_PB_QMOMI_PS	cell_t c, Thread *t, int i	net source term to i^{th} moment

26.6.4.2. Population Balance DEFINE Macros

This section contains descriptions of DEFINE macros for the population balance model. Definitions of each DEFINE macro are contained in the udf.h header file.

26.6.4.2.1. DEFINE_PB_BREAK_UP_RATE_FREQ

26.6.4.2.2. DEFINE_PB_BREAK_UP_RATE_PDF

26.6.4.2.3. DEFINE_PB_COALESCENCE_RATE

26.6.4.2.4. DEFINE_PB_NUCLEATION_RATE

26.6.4.2.5. DEFINE_PB_GROWTH_RATE

26.6.4.2.1. DEFINE_PB_BREAK_UP_RATE_FREQ

You can use the DEFINE_PB_BREAK_UP_RATE_FREQ macro if you want to define the breakage frequency using a UDF. The function is executed at the beginning of every time step.

26.6.4.2.1.1. Usage

```
DEFINE_PB_BREAK_UP_RATE_FREQ(name, cell, thread, d_1)
```

Argument Type	Description
char name	UDF name
cell_t cell	Cell index
Thread *thread	Pointer to the secondary phase thread associated with d_1
real d_1	Parent particle diameter or length
Function returns	
real	

There are four arguments to DEFINE_PB_BREAK_UP_RATE_FREQ: name, cell, thread, and d_1. You will supply name, the name of the UDF. cell, thread, and d_1 are variables that are passed by the Ansys Fluent solver to your UDF.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.6.4.2.1.2. Example

Included below is an example of a UDF that defines a breakage frequency (see Particle Birth and Death Due to Breakage and Aggregation) that is based on the work of Tavlarides [123], such that

$$g(v)=C_1\frac{\varepsilon^{1/3}}{(1+\alpha)d^{2/3}}\exp\left(-C_2\frac{\sigma(1+\alpha)^2}{\rho_d\varepsilon^{2/3}d^{5/3}}\right) \tag{26.33}$$

where C_1 and C_2 are constants with values of 0.00481 and 0.08, respectively, ε is the turbulence energy dissipation rate of the primary phase, d is the parent diameter, σ is the surface tension, α is the volume fraction of the dispersed phase, and ρ_d is the density of the dispersed phase.

```
/************************************************************************
 UDF that computes the particle breakage frequency
 ************************************************************************/

 #include "udf.h"
 #include "sg_pb.h"
 #include "sg_mphase.h"

 DEFINE_PB_BREAK_UP_RATE_FREQ(break_up_freq_tav, cell, thread, d_1)
 {
     real epsi, alpha, f1, f2, rho_d;
     real C1 = 0.00481, C2 = 0.08, sigma = 0.07;
     Thread *tm = THREAD_SUPER_THREAD(thread);/*passed thread is phase*/
     epsi = C_D(cell, tm);
     alpha = C_VOF(cell, thread);
     rho_d = C_R(cell, thread);
     f1 = pow(epsi, 1./3.)/((1.+alpha)*pow(d_1, 2./3.));
     f2 = -(C2*sigma*(1.+alpha)*(1.+alpha))/(rho_d*pow(epsi,2./3.)*pow(d_1, 5./3.));
     return C1*f1*exp(f2);
 }
```

26.6.4.2.2. DEFINE_PB_BREAK_UP_RATE_PDF

You can use the DEFINE_PB_BREAK_UP_RATE_PDF macro if you want to define the breakage PDF using a UDF. The function is executed at the beginning of every time step.

26.6.4.2.2.1. Usage

```
DEFINE_PB_BREAK_UP_RATE_PDF(name, cell, thread, d_1, thread_2, d_2)
```

Argument Type	Description
char name	UDF name
cell_t cell	Cell index
Thread *thread	Pointer to the secondary phase thread associated with d_1
real d_1	Parent particle diameter or length
Thread *thread_2	Pointer to the secondary phase thread associated with d_2
real d_2	Diameter of one of the daughter particles after breakage; the second daughter particle diameter is calculated by conservation of particle volume

Function returns

real

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

There are six arguments to DEFINE_PB_BREAK_UP_RATE_PDF: name, cell, thread, d_1, thread_2, and d_2. You will supply name, the name of the UDF. cell, thread, d_1, thread_2, and d_2 are variables that are passed by the Ansys Fluent solver to your UDF.

Note:

thread and thread_2 are the same for the Discrete, QMOM and SMM models. They may be the same or different depending on whether d_1 and d_2 belong to the same phase or different phases for the Inhomogeneous model.

26.6.4.2.2.2. Example

Included below is an example of a UDF that defines a breakage PDF (see Particle Birth and Death Due to Breakage and Aggregation) that is parabolic, as defined in Equation 14.724.

```
/************************************************************************
 UDF that computes the particle breakage PDF
 ************************************************************************/

 #include "udf.h"
 #include "sg_pb.h"
 #include "sg_mphase.h"

 DEFINE_PB_BREAK_UP_RATE_PDF(break_up_pdf_par, cell, thread, d_1, thread_2, d_2)
 {
     real pdf;
     real kv = M_PI/6.;

     real C = 1.0;

     real f_2, f_3, f_4;
     real V_prime = kv*pow(d_1,3.);
     real V = kv*pow(d_2,3.);

     f_2 = 24.*pow(V/V_prime,2.);
     f_3 = -24.*(V/V_prime);
     f_4 = 6.;

     pdf = (C/V_prime) + ((1.-C/2.)/V_prime)*(f_2 + f_3 + f_4);

     return 0.5*pdf;
 }
```

26.6.4.2.3. DEFINE_PB_COALESCENCE_RATE

You can use the DEFINE_PB_COALESCENCE_RATE macro if you want to define your own particle aggregation kernel. The function is executed at the beginning of every time step.

26.6.4.2.3.1. Usage

```
DEFINE_PB_COALESCENCE_RATE(name, cell, thread, d_1, thread_2, d_2)
```

Argument Type	Description
char name	UDF name
cell_t cell	Cell index
Thread *thread	Pointer to the secondary phase thread associated with d_1
Thread *thread_2	Pointer to the secondary phase thread associated with d_2

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Argument Type	Description
`real d_1, d_2`	Diameters of the two colliding particles

Function returns

`real`

There are six arguments to `DEFINE_PB_COALESCENCE_RATE`: `name`, `cell`, `thread`, `d_1`, `thread_2`, and `d_2`. You will supply `name`, the name of the UDF. `cell`, `thread`, `d_1`, and `d_2` are variables that are passed by the Ansys Fluent solver to your UDF. Your UDF will need to return the `real` value of the aggregation rate.

Note:

`thread` and `thread_2` are the same for the Discrete, QMOM and SMM models. They may be the same or different depending on whether `d_1` and `d_2` belong to the same phase or different phases for the Inhomogeneous model.

26.6.4.2.3.2. Example

Included below is an example UDF for a Brownian aggregation kernel. In this example, the aggregation rate is defined as

$$a(L,\lambda)=a(V,V')=\beta_0\frac{(L+\lambda)^2}{L\lambda}$$

where $\beta_0=1\times10^{-17}\ \mathrm{m^3/s}$.

```
/************************************************************************
 UDF that computes the particle aggregation rate
 ************************************************************************/

 #include "udf.h"
 #include "sg_pb.h"
 #include "sg_mphase.h"

 DEFINE_PB_COALESCENCE_RATE(aggregation_kernel,cell,thread,d_1,thread_2,d_2)
 {
      real agg_kernel;
      real beta_0 = 1.0e-17 /* aggregation rate constant */
      agg_kernel = beta_0*pow((d_1+d_2),2.0)/(d_1*d_2);
      return agg_kernel;
 }
```

26.6.4.2.4. `DEFINE_PB_NUCLEATION_RATE`

You can use the `DEFINE_PB_NUCLEATION_RATE` macro if you want to define your own particle nucleation rate. The function is executed at the beginning of every time step.

26.6.4.2.4.1. Usage

```
DEFINE_PB_NUCLEATION_RATE(name, cell, thread)
```

Argument Type	Description
`char name`	UDF name

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Argument Type	Description
`cell_t cell`	Cell index
`Thread *thread`	Pointer to the secondary phase thread

Function returns

`real`

There are three arguments to `DEFINE_PB_NUCLEATION_RATE`: `name`, `cell`, and `thread`. You will supply `name`, the name of the UDF. `cell` and `thread` are variables that are passed by the Ansys Fluent solver to your UDF. Your UDF will need to return the `real` value of the nucleation rate.

26.6.4.2.4.2. Example

Potassium chloride can be crystallized from water by cooling. Its solubility decreases linearly with temperature. Assuming power-law kinetics for the nucleation rate,

$$\dot{n}_0 = K_n \left(S-1\right)^{N_n}$$

where $K_n = 4\times10^{10}$ particles/m^3-s and $N_n = 2.77$.

```
/************************************************************************
 UDF that computes the particle nucleation rate
 ************************************************************************/

 #include "udf.h"
 #include "sg_pb.h"
 #include "sg_mphase.h"
 DEFINE_PB_NUCLEATION_RATE(nuc_rate, cell, thread)
 {
      real J, S;
      real Kn = 4.0e10; /* nucleation rate constant */
      real Nn = 2.77; /* nucleation law power index */
      real T,solute_mass_frac,solvent_mass_frac, solute_mol_frac,solubility;
      real solute_mol_wt, solvent_mol_wt;

      Thread *tc = THREAD_SUPER_THREAD(thread); /*obtain mixture thread */
      Thread **pt = THREAD_SUB_THREADS(tc);  /* pointer to sub_threads */
      Thread *tp = pt[P_PHASE];     /* primary phase thread */

      solute_mol_wt = 74.55; /* molecular weight of potassium chloride */
      solvent_mol_wt = 18.; /* molecular weight of water */
      solute_mass_frac = C_YI(cell,tp,0);
      /* mass fraction of solute in primary phase (solvent) */

      solvent_mass_frac = 1.0 - solute_mass_frac;
      solute_mol_frac = (solute_mass_frac/solute_mol_wt)/
      ((solute_mass_frac/solute_mol_wt)+(solvent_mass_frac/solvent_mol_wt));

           T = C_T(cell,tp);  /* Temperature of primary phase in Kelvin */

          solubility = 0.0005*T-0.0794;
      /* Solubility Law relating equilibrium solute mole fraction to Temperature*/

    S = solute_mol_frac/solubility; /* Definition of Supersaturation */
     if (S == 1.)
          {
           J = 0.;
          }
     else
          {
           J = Kn*pow((S-1),Nn);
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
        }
    return J;
}
```

Important:

Note that the solubility and the chemistry could be defined in a separate routine and simply called from the above function.

26.6.4.2.5. DEFINE_PB_GROWTH_RATE

You can use the DEFINE_PB_GROWTH_RATE macro if you want to define your own particle growth rate. The function is executed at the beginning of every time step.

26.6.4.2.5.1. Usage

```
DEFINE_PB_GROWTH_RATE(name, cell, thread,d_i)
```

Argument Type	Description
char name	UDF name
cell_t cell	Cell index
Thread *thread	Pointer to the secondary phase thread
real d_i	Particle diameter or length

Function returns

real

There are four arguments to DEFINE_PB_GROWTH_RATE: name, cell, thread, and d_i. You will supply name, the name of the UDF. cell, thread, and d_i are variables that are passed by the Ansys Fluent solver to your UDF. Your UDF will need to return the real value of the growth rate.

26.6.4.2.5.2. Example

Potassium chloride can be crystallized from water by cooling. Its solubility decreases linearly with temperature. Assuming power-law kinetics for the growth rate,

$$G=K_g\left(S-1\right)^{N_g}$$

where $K_g=2.8\times10^{-8}$ m/s and $N_g=1$.

```
/************************************************************************
 UDF that computes the particle growth rate
 ************************************************************************/

 #include "udf.h"
 #include "sg_pb.h"
 #include "sg_mphase.h"
 DEFINE_PB_GROWTH_RATE(growth_rate, cell, thread,d_1)
 {
      /* d_1 can be used if size-dependent growth is needed */
      /* When using SMM, only size-independent or linear growth is allowed */
      real G, S;
      real Kg = 2.8e-8; /* growth constant */
```

```
      real Ng = 1.; /* growth law power index */
      real T,solute_mass_frac,solvent_mass_frac, solute_mol_frac,solubility;
      real solute_mol_wt, solvent_mol_wt;

      Thread *tc = THREAD_SUPER_THREAD(thread); /*obtain mixture thread */
      Thread **pt = THREAD_SUB_THREADS(tc);  /* pointer to sub_threads */
      Thread *tp = pt[P_PHASE];     /* primary phase thread */

      solute_mol_wt = 74.55; /* molecular weight of potassium chloride */
      solvent_mol_wt = 18.; /* molecular weight of water */
      solute_mass_frac = C_YI(cell,tp,0);
      /* mass fraction of solute in primary phase (solvent) */

      solvent_mass_frac = 1.0 - solute_mass_frac;
      solute_mol_frac = (solute_mass_frac/solute_mol_wt)/
      ((solute_mass_frac/solute_mol_wt)+(solvent_mass_frac/solvent_mol_wt));

           T = C_T(cell,tp);  /* Temperature of primary phase in Kelvin */
   solubility = 0.0005*T-0.0794;
      /* Solubility Law relating equilibrium solute mole fraction to Temperature*/

      S = solute_mol_frac/solubility; /* Definition of Supersaturation */
      if (S == 1.)
        {
           G = 0.;
        }
   else
        {
           G = Kg*pow((S-1),Ng);
        }
   return G;
}
```

Important:

Note that the solubility and the chemistry could be defined in a separate routine and simply called from the above function.

26.6.4.3. Hooking a Population Balance UDF to Ansys Fluent

After the UDF that you have defined using `DEFINE_PB_BREAK_UP_RATE_FREQ`, `DEFINE_PB_BREAK_UP_RATE_PDF`, `DEFINE_PB_COALESCENCE_RATE`, `DEFINE_PB_NUCLEATION_RATE`, or `DEFINE_PB_GROWTH_RATE` is interpreted or compiled, the name that you specified in the `DEFINE` macro argument (for example, `agg_kernel`) will become visible and selectable in the appropriate drop-down list under **Phenomena** in the **Population Balance Model** tab in the **Multiphase Model** dialog box (Figure 26.91: The **Multiphase Model** Dialog Box - **Population Balance Model** Tab (p. 3133)).

26.6.5. `DEFINE_HET_RXN_RATE` Macro

This section discusses the `DEFINE_HET_RXN_RATE` macro:

26.6.5.1. Description

26.6.5.2. Usage

26.6.5.3. Example

26.6.5.4. Hooking a Heterogeneous Reaction Rate UDF to Ansys Fluent

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.6.5.1. Description

You need to use DEFINE_HET_RXN_RATE to specify reaction rates for heterogeneous reactions. A heterogeneous reaction is one that involves reactants and products from more than one phase. Unlike DEFINE_VR_RATE, a DEFINE_HET_RXN_RATE UDF can be specified differently for different heterogeneous reactions.

During Ansys Fluent execution, the DEFINE_HET_RXN_RATE UDF for each heterogeneous reaction that is defined is called in every fluid cell. Ansys Fluent will use the reaction rate specified by the UDF to compute production/destruction of the species participating in the reaction, as well as heat and momentum transfer across phases due to the reaction.

A heterogeneous reaction is typically used to define reactions involving species of different phases. The bulk phase can participate in the reaction if the phase does not have any species (that is, the phase has fluid material instead of mixture material). Heterogeneous reactions are defined in the **Phase Interaction** > **Heat, Mass, Reactions** > **Reactions** tab.

26.6.5.2. Usage

DEFINE_HET_RXN_RATE (name,c,t,r,mw,yi,rr,rr_t)

Argument Type	Description
char name	UDF name
cell_t c	Cell index
Thread *t	Cell thread (mixture level) on which heterogeneous reaction rate is to be applied
Hetero_Reaction *r	Pointer to data structure that represents the current heterogeneous reaction (see sg_mphase.h)
real mw[MAX_PHASES][MAX_SPE_EQNS]	Matrix of species molecular weights. mw[i][j] will give molecular weight of species with ID j in phase with index i. For phase that has fluid material, the molecular weight can be accessed as mw[i][0].
real yi[MAX_PHASES][MAX_SPE_EQNS]	Matrix of species mass fractions. yi[i][j] will give molecular weight of species with ID j in phase with index i. For phase that has fluid material, yi[i][0] will be 1.
real *rr	Pointer to laminar reaction rate
real *rr_t	Currently not used. Provided for future use.

Function returns

void

There are eight arguments to DEFINE_HET_RXN_RATE: name, c, t, r, mw, yi, rr, and rr_t. You will supply name, the name of the UDF. c, t, r, mw, yi, rr, and rr_t are variables that are passed by the Ansys Fluent solver to your UDF. Your UDF will need to set the values referenced by the real pointer rr.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.6.5.3. Example

The following compiled UDF, named `arrh`, defines an Arrhenius-type reaction rate. The rate exponents are assumed to be same as the stoichiometric coefficients.

```
#include "udf.h"

 static const real Arrhenius = 1.e15;
 static const real E_Activation = 1.e6;
 #define SMALL_S 1.e-29

 DEFINE_HET_RXN_RATE(arrh,c,t,hr,mw,yi,rr,rr_t)
 {
    Domain **domain_reactant = hr->domain_reactant;
    real *stoich_reactant = hr->stoich_reactant;
    int *reactant = hr->reactant;
    int i;
    int sp_id;
    int dindex;
    Thread *t_reactant;
    real ci;
    real T = 1200.; /* should obtain from cell */

    /* instead of compute rr directly, compute log(rr) and then take exp */

    *rr = 0;
    for (i=0; i < hr->n_reactants; i++)
      {
          sp_id = reactant[i]; /* species ID to access mw and yi */
          if (sp_id == -1) sp_id = 0; /* if phase does not have species,
                       mw, etc. will be stored at index 0 */
          dindex = DOMAIN_INDEX(domain_reactant[i]);
                     /* domain index to access mw & yi */
          t_reactant = THREAD_SUB_THREAD(t,dindex);

           /* get conc. */
          ci = yi[dindex][sp_id]*C_R(c,t_reactant)/mw[dindex][sp_id];
          ci = MAX(ci,SMALL_S);
          *rr += stoich_reactant[i]*log(ci);
       }

       *rr += log(Arrhenius + SMALL_S) -
                   E_Activation/(UNIVERSAL_GAS_CONSTANT*T);

    /* 1.e-40 < rr < 1.e40 */
    *rr = MAX(*rr,-40);
    *rr = MIN(*rr,40);

    *rr = exp(*rr);
 }
```

26.6.5.4. Hooking a Heterogeneous Reaction Rate UDF to Ansys Fluent

After the UDF that you have defined using `DEFINE_HET_RXN_RATE` is interpreted or compiled (see the Fluent Customization Manual for details), the name that you specified in the `DEFINE` macro argument (for example, `arrh`) will become visible and selectable under **Reaction Rate Function** in the **Phase Interaction** > **Heat, Mass, Reactions** > **Reactions** tab of the **Multiphase Model** dialog box. (Note you will first need to specify the **Total Number of Reactions** greater than **0**.)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.7. Setting Up the Wet Steam Model

You can choose the wet steam model (see Wet Steam Model Theory in the Theory Guide) by opening the **Multiphase Model** dialog box and selecting the **Wet Steam** option. It is available for both the pressure-based and density-based solvers.

Setup → Models → Multiphase → Edit...

Figure 26.111: The Multiphase Model Dialog Box with the Wet Steam Model Selected (Pressure-Based)

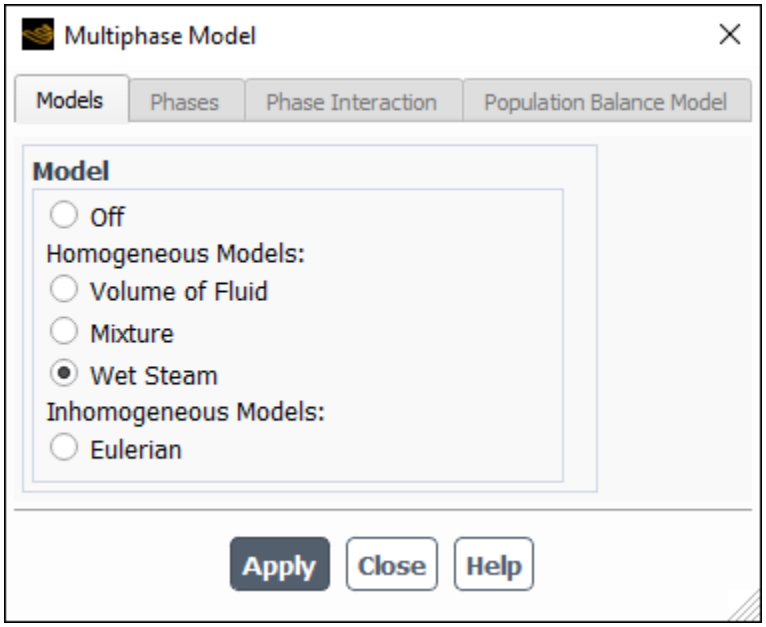
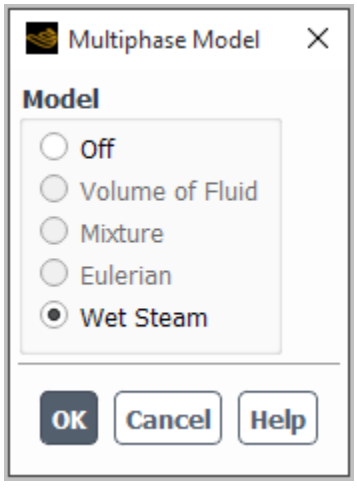


Figure 26.112: The Multiphase Model Dialog Box with the Wet Steam Model Selected (Density-Based)



The following options are available:

1. You can set the droplet growth formula that is to be used by the wet steam model using the TUI command:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
define/models/multiphase/wet-steam/set/droplet-growth-rate
```

You have the option of using the default Young's formula or switching to Hill's formula as well as setting any modeling parameters. For details, see Phase Change Model in the *Fluent Theory Guide*.

2. You can set the virial Equation of State (EOS) that is to be used by the wet steam model using the TUI command:

   ```
   define/models/multiphase/wet-steam/set/virial-equation
   ```

   You have the option of using the default EOS developed by Vukalovich or switching to the EOS proposed by Young. The selected option will also define the associated equations for the thermodynamic properties of steam. For details, see Built-in Thermodynamic Wet Steam Properties in the *Fluent Theory Guide*.

3. Alternative to the virial EOS and build-in steam properties, you have an option to use Real Gas Property (RGP) tables to define the thermodynamic state and properties of steam. For details, see Real Gas Property (RGP) Table Files for the Wet Steam Model in the *Fluent Theory Guide*. Creating RGP files is described in Using Real Gas Property (RGP) Table Files (p. 1734). Once the proper RGP file is created, you can load it into Fluent by selecting:

   User-Defined → **User-Defined** → **Read Table...**

   or using the TUI command:

   ```
   file/table-manager/read-rgp-file
   ```

   For the Wet Steam model, there is no need to create the RGP material from the table or embed the RGP data into the case file, so both options may keep the default setting. These options can be controlled via the **Settings...** button in the **Table File Manager** dialog box.

   The loaded RGP table can now be linked to the Wet Steam model using the TUI command:

   ```
   define/models/multiphase/wet-steam/set/rgp-tables
   ```

4. You can set the option to compute the stagnation condition values using the TUI command:

   ```
   define/models/multiphase/wet-steam/set/stagnation-conditions
   ```

   For details, see Computing Stagnation Conditions for the Wet Steam Model in the *Fluent Theory Guide*.

5. You have the option to switch off the phase change model in specific cell zones using the TUI command: `define/models/multiphase/wet-steam/set/zonal-phase-change`

   This option can be used to prevent condensation from forming until a certain stage in a multi-stage steam turbine. In cell zones where the phase change model is switched off, the source terms of the transport equations for the wet steam model will be set to zero during the solution process. For details, see Wet Steam Flow Equations in the *Fluent Theory Guide* and Phase Change Model in the *Fluent Theory Guide*.

Solution settings and strategies for the wet steam model can be found in Wet Steam Model (p. 3208). Postprocessing variables are described in Model-Specific Variables (p. 3211).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The wet steam model is compatible with General Turbo Interfaces (GTI) that are used to simulate flows in rotating and stationary components of turbomachines, in particular steam turbines. Details of different GTI types are described in Blade Row Interaction Modeling (p. 1956).

This section includes information about using your own property functions and data with the wet steam model.

26.7.1. Using User-Defined Thermodynamic Wet Steam Properties
26.7.2. Writing the User-Defined Wet Steam Property Functions (UDWSPF)
26.7.3. Compiling Your UDWSPF and Building a Shared Library File
26.7.4. Loading the UDWSPF Shared Library File
26.7.5. UDWSPF Example

26.7.1. Using User-Defined Thermodynamic Wet Steam Properties

Ansys Fluent allows you to use your own property functions and data with the wet steam model. This is achieved with user-defined wet steam property functions (UDWSPF).

These user-defined functions are written in the C programming language and there is a certain programming format that must be used so that you can build a successful library that can be loaded into the Ansys Fluent code.

The following is the procedure for using the user-defined wet steam property functions (UDWSPF):

1. Define the wet steam equation of state and all related thermodynamic and transport property equations.
2. Create a C source code file that conforms to the format defined in this section.
3. Start Ansys Fluent and set up your case file in the usual way.
4. Turn on the wet steam model.
5. Compile your UDWSPF C functions and build a shared library file using the text user interface.

   `define` → `models` → `multiphase` → `wet-steam` → `compile-user-defined-wetsteam-functions`
6. Load your newly created UDWSPF library using the text user interface.

   `define` → `models` → `multiphase` → `wet-steam` → `load-unload-user-defined-wetsteam-library`
7. Run your calculation.

26.7.2. Writing the User-Defined Wet Steam Property Functions (UDWSPF)

Creating a UDWSPF C function library is reasonably straightforward:

- The code must contain the `udf.h` file inclusion directive at the beginning of the source code. This allows the definitions for `DEFINE` macros and other Ansys Fluent functions to be accessible during the compilation process.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The code must include at least one in the UDF's `DEFINE` functions (that is `DEFINE_ON_DEMAND`) to be able to use the compiled UDFs utility.
- Any values that are passed to the solver by the UDWSPF or returned by the solver to the UDWSPF are assumed to be in SI units.
- You must use the principle set of user-defined wet steam property functions in your UDWSPF library, as described in the list that follows. These functions are the mechanism by which your thermodynamic property data is transferred to the Ansys Fluent solver.

The following lists the user-defined wet steam property function names and arguments, as well as a short description of their functions. Function inputs from the Ansys Fluent solver consist of one or more of the following variables: T = temperature (K), P = pressure (Pa), and ρ = vapor-phase density (kg/m^3).

- `void wetst_init(Domain *domain)`

  This will be called when you load the UDWSPF. You use it to initialize wet steam model constants or your own model constants. It returns nothing.

- `real wetst_satP(real T)`

  This is the saturated pressure function, which takes on temperature in K and returns saturation pressure in Pa.

- `real wetst_satT(real P, real T)`

  This is the saturated temperature function, which takes on pressure in Pa and a starting guess temperature in K and returns saturation temperature in K.

- `real wetst_eosP(real rho, real T)`

  This is the equation of state, which takes on vapor density in kg/m^3 and Temperature in K and returns pressure in Pa.

- `real wetst_eosRHO(real P, real T)`

  This is the equation of state, which takes on pressure in Pa and temperature in K and returns vapor density in kg/m^3.

- `real wetst_cpv(real T, real rho)`

  This is the vapor specific heat at constant pressure, which takes on temperature in K and vapor density in kg/m^3 and returns specific heat at constant pressure in J/kg/K.

- `real wetst_cvv(real T, real rho)`

  This is the vapor specific heat at constant volume, which takes on temperature in K and vapor density in kg/m^3 and returns specific heat at constant volume in J/kg/K.

- `real wetst_hv(real T,real rho)`

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This is the vapor specific enthalpy, which takes on temperature in K and vapor density in kg/m^3 and returns specific enthalpy in J/Kg.

- `real wetst_sv(real T, real rho)`

  This is the vapor specific entropy, which takes on temperature in K and vapor density in kg/m^3 and returns specific entropy in J/Kg/K.

- `real wetst_muv(real T, real rho)`

  This is the vapor dynamic viscosity, which takes on temperature in K and vapor density in kg/m^3 and returns viscosity in kg/m/s.

- `real wetst_ktv(real T, real rho)`

  This is the vapor thermal conductivity, which takes on temperature in K and vapor density in kg/m^3 and returns thermal conductivity in W/m/K.

- `real wetst_rhol(real T)`

  This is the saturated liquid density, which takes on temperature in K and returns liquid density in kg/m^3.

- `real wetst_cpl(real T)`

  This is the saturated liquid specific heat at constant pressure, which takes on temperature in K and returns liquid specific heat in J/kg/K.

- `real wetst_mul(real T)`

  This is the liquid dynamic viscosity, which takes on Temperature in K and returns dynamic viscosity in kg/m/s.

- `real wetst_ktl(real T)`

  This is the liquid thermal conductivity, which takes on temperature in K and returns thermal conductivity in W/m/K.

- `real wetst_surft(real T)`

  This is the liquid surface tension, which takes on Temperature in K and returns surface tension N/m.

At the end of the code you must define a structure of type `WS_Functions` whose members are pointers to the principle functions listed previously. The structure is of type `WS_Functions` and its name is `WetSteamFunctionList`.

```
UDF_EXPORT  WS_Functions  WetSteamFunctionList  =
 {
     wetst_init,     /*initialization  function*/
     wetst_satP,     /*Saturation  pressure*/
     wetst_satT,     /*Saturation  temperature*/
     wetst_eosP,     /*equation  of  state*/
     wetst_eosRHO,   /*equation  of  state*/
     wetst_hv,       /*vapor  enthalpy*/
     wetst_sv,       /*vapor  entropy*/
     wetst_cpv,      /*vapor  isobaric  specific  heat*/
```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
    wetst_cvv,      /*vapor  isochoric  specific  heat*/
    wetst_muv,      /*vapor  dynamic  viscosity*/
    wetst_ktv,      /*vapor  thermal  conductivity*/
    wetst_rhol,     /*sat.  liquid  density*/
    wetst_cpl,      /*sat.  liquid  specific  heat*/
    wetst_mul,      /*sat.  liquid  viscosity*/
    wetst_ktl,      /*sat.  liquid  thermal  conductivity*/
    wetst_surft     /*liquid  surface  tension*/
};
```

26.7.3. Compiling Your UDWSPF and Building a Shared Library File

This section presents the steps you will need to follow to compile your UDWSPF C code and build a shared library file. This process requires the use of a C compiler. For more details on compiler requirements, see Compilers in the *Fluent Customization Manual*.

Important:

To use the UDWSPF you will need to first build the UDWSPF library by compiling your UDWSPF C code and then loading the library into the Ansys Fluent code.

The UDWSPF shared library is built in the same way that the Ansys Fluent executable itself is built. Internally, a script called `Makefile` is used to invoke the system C compiler to build an object code library that contains the native machine language translation of your higher-level C source code. This shared library is then loaded into Ansys Fluent (either at run time or automatically when a case file is read) by a process called *dynamic loading*. The object libraries are specific to the computer architecture being used, as well as to the particular version of the Ansys Fluent executable being run. The libraries must, therefore, be rebuilt any time Ansys Fluent is upgraded, when the computer's operating system level changes, or when the job is run on a different type of computer.

The general procedure for compiling UDWSPF C code is as follows:

1. Place the UDWSPF C code in your working directory (that is, where your case file resides).
2. Launch Ansys Fluent.
3. Read your case file into Ansys Fluent.
4. You can now compile your UDWSPF C code and build a shared library file using the commands provided in the text command interface (TUI):
   a. Select the `define/models/multiphase/wet-steam` menu item.

      `define` → `models` → `multiphase` → `wet-steam`

      If the Wet Steam model has not been enabled yet, use the following TUI commands to enable it:

      ```
      define/models/multiphase/wet-steam> enable?

      Enable wet steam model? [no] yes
      ```

   b. Select the `compile-user-defined-wetsteam-functions` option.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. Enter the compiled UDWSPF library name.

   The name given here is the name of the directory where the shared library (for example, `libudf`) will reside. For example, if you press **Enter** then a directory should exist with the name `libudf`, and this directory will contain library file called `libudf`. If, however, you type a new library name such as `mywetsteam`, then a directory called `mywetsteam` will be created and it will contain the library `libudf`.

d. Continue on with the procedure when prompted.

e. Enter the C source file names.

   Important:

   Ideally you should place all of your functions into a single file. However, you can split them into separate files if desired.

f. Enter the header file names, if applicable. If you do not have an extra header file, then press **Enter** when prompted.

   Ansys Fluent will then start compiling the UDWSPF C code and put it in the appropriate architecture directory.

26.7.4. Loading the UDWSPF Shared Library File

To load the UDWSPF library, perform the following steps:

- Go to the `define/models/multiphase/wet-steam` menu item in the text user interface.

  `define` → `models` → `multiphase` → `wet-steam`

- Select the `load-unload-user-defined-wetsteam-library` option and follow the procedure when prompted.

  If the loading of the UDWSPF library is successful, you will see a message similar to the following:

```
Opening user-defined wet steam library "libudf"...
Library "libudf/lnamd64/2d/libudf.so" opened

Setting material properties to Wet-Steam...

Initializing user defined material properties...
```

26.7.5. UDWSPF Example

This section describe a simple UDWSPF. You can use this example as the basis for your own UDWSPF code. For approximate calculations at low pressure, the simple ideal-gas equation of state and constant isobaric specific heat is assumed and used. The properties at the saturated liquid line and the saturated vapor line used in this example are similar to the one used by Ansys Fluent.

```
/************************************************************************/
 /* User Defined Wet Steam Properties:
```

```
  EOS    : Ideal Gas Eq.
  Vapor Sat. Line : W.C.Reynolds tables (1979)
  Liquid Sat. Line: E. Eckert & R. Drake book (1972)

  Use ideal-gas EOS with Steam properties
  to model wet steam condensation in low pressure nozzle
  Author: L. Zori
  Date : Jan. 29 2004
*/
/***********************************************************************/
#include "udf.h"
#include "stdio.h"
#include "ctype.h"
#include "stdarg.h"

/*Global Constants for this model*/
real ws_TPP = 338.150 ;
real ws_aaa = 0.01 ;
real cpg = 1882.0;/* Cp-vapor at low-pressure region*/

DEFINE_ON_DEMAND(I_do_nothing)
{
   /* This is a dummy function to allow us to use */
   /* the Compiled UDFs utility      */
}



void wetst_init(Domain *domain)
{
   /*
     You must initialize these material property constants.
     they will be used in the wet steam model in fluent
   */
   ws_Tc = 647.286  ;/*Critical Temp. */
   ws_Pc = 22089000.00 ;/*Critical Pressure */
   mw_f  = 18.016 ;/*fluid droplet molecular weight (water) */
   Rgas_v  = 461.50 ;/*vapor Gas Const*/
}

real wetst_satP(real T)
{
   real psat;
   real SUM=0.0;
   real pratio;
   real F;
   real a1 = -7.41924200 ;
   real a2 = 2.97210000E-01;
   real a3 = -1.15528600E-01;
   real a4 = 8.68563500E-03;
   real a5 = 1.09409899E-03;
   real a6 = -4.39993000E-03;
   real a7 = 2.52065800E-03;
   real a8 = -5.21868400E-04;
   if (T > ws_Tc) T = ws_Tc ;
   F  = ws_aaa*(T - ws_TPP)  ;
   SUM = a1 + F*(a2+ F*(a3+ F*(a4+ F*(a5+ F*(a6+ F*(a7+ F*a8)))))) ;
   pratio = (ws_Tc/T - 1.0)*SUM;
   psat  = ws_Pc *exp(pratio) ;
  return psat; /*Pa */
}



real
wetst_satT(real P, real T)
{
   real tsat;
   real dT, dTA,dTM,dP,p1,p2,dPdT;
  real dt = 1.e-4;
   int i ;
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
     for (i=0; i<25; ++i)
      {
        if (T > ws_Tc) T = ws_Tc-0.5;

        p1= wetst_satP(T)  ;
        p2= wetst_satP(T+dt) ;
        dPdT = (p2-p1)/dt;

        dP = P - p1 ;

        dT = dP/dPdT  ;

        dTA = fabs(dT);
        T = T + dT;
        if (fabs(dT)<TEMP_eps*T) break;
        }
     tsat = T;
   return tsat; /*K */
}

real
wetst_eosP(real rho, real T)
{
   real P;

   P = rho* Rgas_v * T ;

  return P; /*Pa */
}



real
wetst_eosRHO(real P, real T)
{
    real rho;

    rho = P/(Rgas_v * T) ;

    return rho; /*kg/m3 */
}


real
wetst_cpv(real T, real rho)
{
    real cp;

    cp = cpg ;

    return cp; /* (J/Kg/K) */
}

real
wetst_cvv(real T, real rho)
{
   real cv;

   cv = wetst_cpv(T,rho) - Rgas_v ;

   return cv; /* (J/Kg/K) */
}

real
wetst_hv(real T,real rho)
{
   real h;

   h = T* wetst_cpv(T,rho) ;

  return h; /* (J/Kg) */
```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
}



real
wetst_sv(real T, real rho)
{
   real s ;

   real TDatum=288.15;
   real PDatum=1.01325e5;
   s=wetst_cpv(T,rho)*log(T/TDatum)+
   Rgas_v*log(PDatum/(Rgas_v*T*rho));

 return s; /* (J/Kg/K) */
}


real
wetst_muv(real T, real rho)
{
   real muv;

   muv=1.7894e-05 ;

   return muv; /* (Kg/m/s) */
}

real
wetst_ktv(real T, real rho)
{
   real ktv;

   ktv=0.0242 ;

   return ktv; /* W/m/K */
}



real
wetst_rhol(real T)
{
   real rhol;

   real SUM = 0.0;
   int ii;
   int i;
   real rhoc = 317.0;
   real D[8];

   D[0] =  3.6711257 ;
   D[1] = -2.8512396E+01 ;
   D[2] =  2.2265240E+02 ;
   D[3] = -8.8243852E+02 ;
   D[4] =  2.0002765E+03 ;
   D[5] = -2.6122557E+03 ;
   D[6] =  1.8297674E+03 ;
   D[7] = -5.3350520E+02 ;
   if (T > ws_Tc) T = ws_Tc ;
   for(ii=0;ii 8;++ii)
     {
        i = ii+1 ;
        SUM += D[ii] * pow((1.0 - T/ws_Tc), i/3.0);
     }
   rhol = rhoc*(1.0+SUM);

   return rhol; /* (Kg/m3) */
}
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
real
wetst_cpl(real T)
{
   real cpl;

  real a1= -36571.6 ;
   real a2= 555.217 ;
   real a3= -2.96724 ;
   real a4= 0.00778551;
   real a5= -1.00561e-05;
   real a6= 5.14336E-09;

   if (T > ws_Tc) T = ws_Tc ;
   cpl = a1 + T*(a2+ T*(a3+ T*(a4+ T*(a5+ T*a6)))) ;

   return cpl; /* (J/Kg/K) */
}


real
wetst_mul(real T)
{
   real mul;

   real a1= 0.530784;
   real a2= -0.00729561;
   real a3= 4.16604E-05 ;
   real a4= -1.26258E-07;
   real a5= 2.13969E-10;
   real a6= -1.92145E-13;
   real a7= 7.14092E-17;

   if (T > ws_Tc) T = ws_Tc ;
   mul = a1 + T*(a2+ T*(a3+ T*(a4+ T*(a5+ T*(a6+ T*a7))))) ;

   return mul; /* (Kg/m/s) */
}

real
wetst_ktl(real T)
{
 real ktl;

   real a1= -1.17633;
   real a2= 0.00791645;  real a3= 1.48603E-05;
   real a4= -1.31689E-07;
   real a5= 2.47590E-10;
   real a6= -1.55638E-13;

   if (T > ws_Tc) T = ws_Tc ;
   ktl = a1 + T*(a2+ T*(a3+ T*(a4+ T*(a5+ T*a6)))) ;

 return ktl; /* W/m/K */
}

real
wetst_surft(real T)
{
   real sigma;

   real Tr;
   real a1= 82.27  ;
   real a2= 75.612;
   real a3= -256.889 ;
   real a4= 95.928;

   if (T > ws_Tc) T = ws_Tc ;
   Tr  = T/ws_Tc ;
   sigma = 0.001*(a1 + Tr*(a2+ Tr*(a3+ Tr*a4))) ;
```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
   return sigma ;/* N/m */
}

/* do not change the order of the function list */
UDF_EXPORT WS_Functions WetSteamFunctionList =
  {
  wetst_init,     /*initialization  function*/
  wetst_satP,     /*Saturation  pressure*/
  wetst_satT,     /*Saturation  temperature*/
  wetst_eosP,     /*equation  of  state*/
  wetst_eosRHO,   /*equation  of  state*/
  wetst_hv,       /*vapor  enthalpy*/
  wetst_sv,       /*vapor  entropy*/
  wetst_cpv,      /*vapor  isobaric  specific  heat*/
  wetst_cvv,      /*vapor  isochoric  specific  heat*/
  wetst_muv,      /*vapor  dynamic  viscosity*/
  wetst_ktv,      /*vapor  thermal  conductivity*/
  wetst_rhol,     /*sat.  liquid  density*/
  wetst_cpl,      /*sat.  liquid  specific  heat*/
  wetst_mul,      /*sat.  liquid  viscosity*/
  wetst_ktl,      /*sat.  liquid  thermal  conductivity*/
  wetst_surft     /*liquid  surface  tension*/
 };
/************************************************************************/
```

26.8. Solution Strategies for Multiphase Modeling

Important:

Double Precision is recommended for all multiphase cases.

The following topics are discussed:

26.8.1. General Solution Strategies

26.8.2. Model-Specific Solution Strategies

26.8.1. General Solution Strategies

General strategies that are applicable to multiphase models are discussed in the following sections:

26.8.1.1. Coupled Solution for Eulerian Multiphase Flows

26.8.1.2. Coupled Solution for VOF and Mixture Multiphase Flows

26.8.1.3. Selecting the Pressure-Velocity Coupling Method

26.8.1.4. Controlling the Volume Fraction Coupled Solution

26.8.1.5. Default and Stability Controls

26.8.1.6. Heat Transfer and Radiative Flux Distribution for Non-Eulerian Multiphase Models

26.8.1.7. Steady-State Solution Strategies

26.8.1.1. Coupled Solution for Eulerian Multiphase Flows

In multiphase flow, the phasic momentum equations, the shared pressure, and the phasic volume fraction equations are highly coupled. Traditionally, these equations have been solved in a segregated fashion using some variation of the SIMPLE algorithm to couple the shared pressure with the mo-

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

mentum equations. This is attained by effectively transforming the total continuity into a shared pressure. The Ansys Fluent **Phase Coupled SIMPLE** algorithm has been successfully implemented and solves a wide range of multiphase flows. However, coupling the linearized system of equations in an implicit manner would offer a more robust alternative to the segregated approach.

One of the fundamental problems is that the resulting matrix is not symmetric and that the continuity constraint may contribute to a zero diagonal block, making the solution difficult to obtain. One way to circumvent this problem is to use direct solvers, but these are too expensive for large industrial cases. In addition, we need to avoid a zero diagonal, resulting from the continuity constraint, and like the segregated solver, we need to construct a pressure correction equation. In multiphase, we also have the additional problem of the vanishing phase, which for the coupled solver is important to ensure some continuity in the coefficients. Like the **Phase Coupled**, we use a Rhie and Chow type of scheme to calculate volume fluxes and to provide proper coupling between velocity and pressure, thereby avoiding unphysical oscillations.

Consider a single-phase system and let us denote the velocity correction components in the three Cartesian directions by u', v', and w' with p' denoting the shared pressure correction. These are discrete variables and can be expressed in the form p', U'. The linear system that is generated by the single-phase coupled solver is of the form

$$\begin{pmatrix} A_p & C_U \\ B_U & A_U \end{pmatrix} \begin{pmatrix} p' \\ U' \end{pmatrix} = \begin{pmatrix} S_p \\ S_U \end{pmatrix} \tag{26.34}$$

For a notation in component form $\left(p', u', v', w'\right)$

$$\begin{pmatrix} A_{pp} & C_u & C_v & C_w \\ B_u & A_{uu} & A_{uv} & A_{uw} \\ B_v & A_{vu} & A_{vv} & A_{vw} \\ B_w & A_{wu} & A_{wv} & A_{ww} \end{pmatrix} \begin{pmatrix} p' \\ u' \\ v' \\ w' \end{pmatrix} = \begin{pmatrix} S_p \\ S_u \\ S_v \\ S_w \end{pmatrix} \tag{26.35}$$

Now let us consider a multiphase system of n-phases and denote the phasic velocity correction components in the three Cartesian directions by u_k', v_k', and w_k' where the subscript k represents the phase notation, p' denotes the shared pressure correction and α_k' denotes the volume fraction correction (Ansys Fluent can solve in both correction form for velocity and volume fraction and non-correction form). For simplicity the matrix will be shown for two phases. The vector solution is of the form $\left(p', u_1', u_2', v_1', v_2', w_1', w_2', \alpha_2'\right)$ or in a shorter notation $\left(p', U', V', W', \alpha_2'\right)$. The linear system would be an extension of the one generated by the coupled solver shown by Equation 26.34 (p. 3185).

$$\begin{pmatrix} A_p & C_U & C_V & C_W & D_{\alpha_2'} \\ B_U & A_{UU} & A_{UV} & A_{UW} & D_U \\ B_V & A_{VU} & A_{VV} & A_{VW} & D_V \\ B_W & A_{WU} & A_{WV} & A_{WW} & D_W \\ E_p & E_U & E_V & E_W & A_{\alpha_2'} \end{pmatrix} \begin{pmatrix} p' \\ U' \\ V' \\ W' \\ \alpha_2' \end{pmatrix} = \begin{pmatrix} S_p \\ S_U \\ S_V \\ S_W \\ S_{\alpha_2'} \end{pmatrix} \tag{26.36}$$

This system can be easily generalized to n phases. The components of this matrix are also matrices.

For large problems, we need to resort to iterative solvers. The Ansys Fluent AMG Coupled solver with an ILU smoother has proved to be a robust method. Most coupled solvers also need a pseudo stepping method, adding more diagonal dominance to the matrix. Our method here is to use under-

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

relaxation factors for momentum, which is equivalent to time stepping in steady flows. Similar to that of the single phase, we have introduced a steady Courant Number instead of an under-relaxation for velocities. Having this control is important when using second order numerical schemes in the convective terms.

For the sake of simplicity, input parameters for the **Coupled** solver are similar to the single-phase solver. We have the options for solving the whole system including volume fraction, or to treat the volume fraction solution in a segregated manner while preserving the pressure-velocity coupling for all phases.

Important:

Equations in multiphase are more strongly linked than single phase and generally may need more under-relaxation, hence using the same values as single phase may not be ideal. A low Courant number would stabilize the solution.

See Selecting the Pressure-Velocity Coupling Method (p. 3187) for information about applying the various algorithms.

26.8.1.2. Coupled Solution for VOF and Mixture Multiphase Flows

We have the option of solving the multiphase system for VOF and mixture multiphase models in the following ways:

- Solving the continuity and momentum equations in a coupled manner (see Coupled Algorithm in the Theory Guide) and solving the volume fraction equation in a segregated manner.
- Solving the volume fraction equation in a coupled manner along with the continuity and momentum equations.

Solving the volume fraction equation in a coupled manner requires discretization of the volume fraction equation in the correction form along with the discretization of the continuity and momentum equation, as discussed in Coupled Algorithm in the Theory Guide.

The volume fraction equation could be represented in the discretized form as

$$\sum_k \sum_j a_{ij}^{\alpha u_k} u_{kj} + \sum_j a_{ij}^{\alpha p} p_j + \sum_j a_{ij}^{\alpha\alpha} \alpha_j = b_i^{\alpha} \tag{26.37}$$

The overall system of equations after being transformed to the correction form could be represented as

$$\sum_j [A]_{ij} \vec{X}_j = \vec{B}_i \tag{26.38}$$

For a 2D case and two-phase flow, the system could be expanded as follows:

$$A_{ij} = \begin{bmatrix} a_{ij}^{pp} & a_{ij}^{pu} & a_{ij}^{pv} & a_{ij}^{p\alpha} \\ a_{ij}^{up} & a_{ij}^{uu} & a_{ij}^{uv} & a_{ij}^{u\alpha} \\ a_{ij}^{vp} & a_{ij}^{vu} & a_{ij}^{vv} & a_{ij}^{v\alpha} \\ a_{ij}^{\alpha p} & a_{ij}^{\alpha u} & a_{ij}^{\alpha v} & a_{ij}^{\alpha\alpha} \end{bmatrix} \tag{26.39}$$

and the unknown and residual vectors have the form

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\vec{X}_j = \begin{bmatrix} p'_i \\ u'_i \\ v'_i \\ \alpha'_i \end{bmatrix} \tag{26.40}$$

$$\vec{B}_i = \begin{bmatrix} -r_i^p \\ -r_i^u \\ -r_i^v \\ -r_i^\alpha \end{bmatrix} \tag{26.41}$$

where

A_{ij} = coefficient matrix

$\vec{X}_j$ = solution vector

$\vec{B}_i$ = residual vector

p' = pressure correction

u', v' = velocity corrections

α' = volume fraction correction

26.8.1.3. Selecting the Pressure-Velocity Coupling Method

The settings that are available in the **Solution Methods** task page (see Figure 26.113: The Solution Methods Task Page Displaying The Pressure-Velocity Coupling Options (p. 3189)) for solving the coupled system of equations arising in multiphase flows are:

Schemes

- **Phase Coupled SIMPLE** (Eulerian multiphase)
- **PISO** (VOF and mixture)
- **SIMPLE** (VOF and mixture)
- **SIMPLEC** (VOF and mixture)
- **Coupled** (all multiphase models)

Options

- **Coupled with Volume Fractions** (all multiphase models except the mixture model with the **Slip Velocity** option and the wet steam model)
- **Solve N-Phase Volume Fraction Equations** (mixture and Eulerian models)

The **PISO**, **SIMPLE**, and **SIMPLEC** schemes apply to the VOF and mixture models and are discussed in Pressure-Velocity Coupling in the Theory Guide.

The **Phase Coupled SIMPLE** (PC-SIMPLE) is an extension of the SIMPLE algorithm [114] (p. 5719) to multiphase flows. The velocities are solved coupled by phases in a segregated fashion. Fluxes are reconstructed at the faces of the control volume and then a pressure correction equation is built

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

based on total continuity. The coefficients of the pressure correction equations come from the coupled per phase momentum equations. This method has proven to be robust and it is the only method available for all previous versions of Ansys Fluent.

The **Coupled** scheme (also known as **Multiphase Coupled** in previous Ansys Fluent versions) solves all equations for phase velocity corrections and shared pressure correction simultaneously [50] (p. 5715). These methods incorporate the lift forces and the mass transfer terms implicitly into the general matrix. This method works very efficiently in steady-state situations, or for transient problems when a larger time step size is required.

The **Coupled with Volume Fractions** option (also known as **Full Multiphase Coupled** in previous Ansys Fluent versions) couples velocity corrections, shared pressure corrections, and the correction for volume fraction simultaneously. Theoretically, it should be more efficient, however it may have some drawbacks in robustness and CPU time usage. The robustness issue stems from the lack of control of the solution of the volume fraction equation. The continuity constraint (sum of all volume fractions equals 1, and individual values limited between zero and one) cannot be enforced exactly during inner solver iterations, and slight variations from the physical limits may lead to divergence. Research is ongoing in this area to improve the method. The method is advantageous for heterogeneous mass transfer when a low Courant number is given; it also works well in dilute situations.

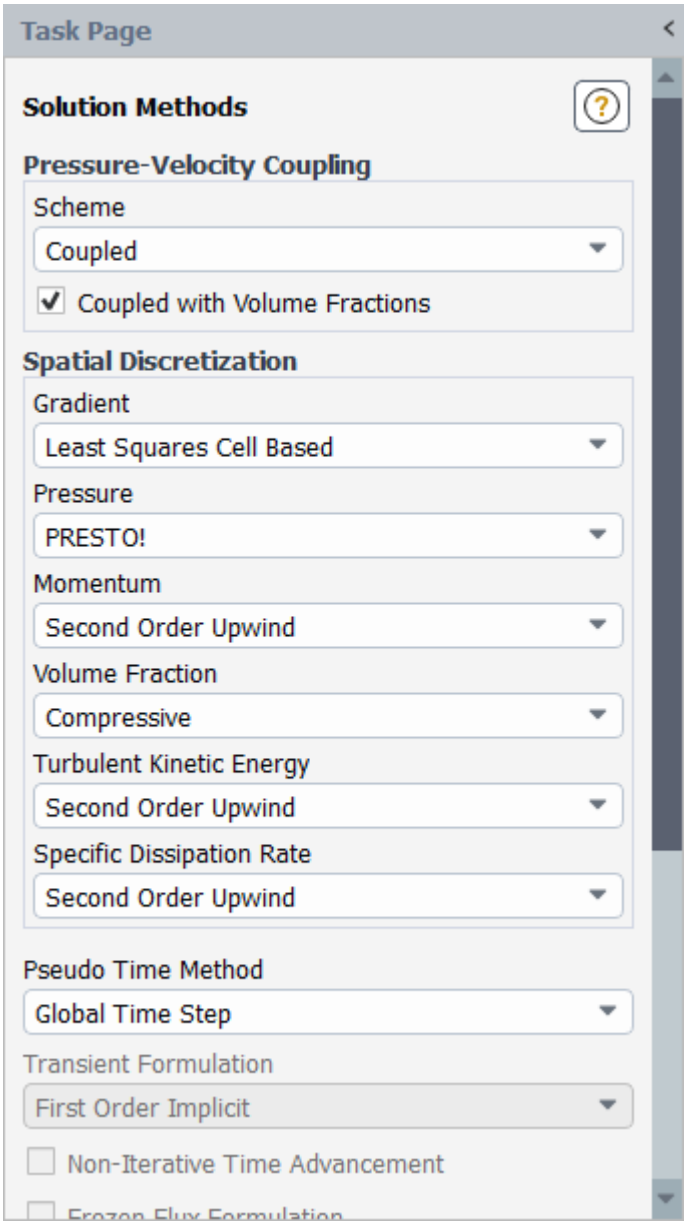
The Volume Fraction Coupling Method aims to achieve a faster steady-state solution compared to the segregated method of solving equations. It may not be a suitable option for transient applications due to the significant overhead in CPU time compared to the segregated method, unless it is run with a larger time step size.

Note:

The **Coupled with Volume Fractions** option is available in the interface after you have selected **Coupled** from the **Scheme** drop-down list for **Pressure-Velocity Coupling**. For steady-state cases, **Global Time Step** will be automatically selected from the **Pseudo Time Method** list when you enable the **Coupled with Volume Fractions** option for the VOF and mixture models.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.113: The Solution Methods Task Page Displaying The Pressure-Velocity Coupling Options



26.8.1.3.1. Limitations and Recommendations of the Coupled with Volume Fraction Options for the VOF and Mixture Models

The coupled with volume fractions option has the following limitations:

- It is not available when **Slip Velocity** is enabled for the **Mixture** multiphase model.
- It is not supported when using the Singhal-Et-Al cavitation model.
- It is not supported when the **Explicit** formulation for volume fraction is selected.

Recommended uses of the coupled with volume fractions option:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Selecting global time step for the pseudo time method is recommended for steady-state calculations.
- It is recommended that you use lower under-relaxation factors for momentum for higher order schemes.
- For marine applications, it is recommended that you use a low-order variant or hybrid treatment for the Rhie-chow face flux interpolation (see High-Order Rhie-Chow Face Flux Interpolation (p. 3203)).

26.8.1.3.2. Solving N-Phase Volume Fraction Equations

For a multiphase flow, the sum of the phase volume fractions must always equal 1. By default, Fluent enforces this constraint by solving the volume fraction equations for each of the secondary phases and then setting the volume fraction of the primary phase to the complement.

In certain cases, where poor convergence and/or mass imbalance are observed, enabling **Solve N-Phase Volume Fraction Equations** may improve solutions, although in practice the results obtained from using this option are case dependent. If selected, this option will solve all volume fraction equations, including both primary and secondary phases. The resulting phase volume fractions are then scaled in order to satisfy the requirement of summing to 1. This approach is more computationally expensive than the default approach and, in general, should not be necessary for the simulation.

Note:

The **Solve N-Phase Volume Fraction Equations** option is not available for the **Volume of Fluid** model or **Eulerian** with **Multi-Fluid VOF Model** enabled.

26.8.1.4. Controlling the Volume Fraction Coupled Solution

When using the **Coupled with Volume Fractions** scheme, you will need to specify the following in the **Solution Controls** task page (see Figure 26.114: The Solution Controls Task Page Displaying the Coupled Volume Fraction Method for the VOF and Mixture Models (p. 3192)):

- **VOF and Mixture Multiphase Model:**
  - For steady-state cases using a pseudo time method (global time step), specify the **Volume Fraction Courant Number**, and the **Pseudo Time Explicit Relaxation Factors**. The **Pseudo Time Explicit Relaxation Factors** are described in Global Time Step Method Settings (p. 3676).

    Note:

    The **Pseudo Time Explicit Relaxation Factors** for the **Volume Fraction** is set to 0.5 by default.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

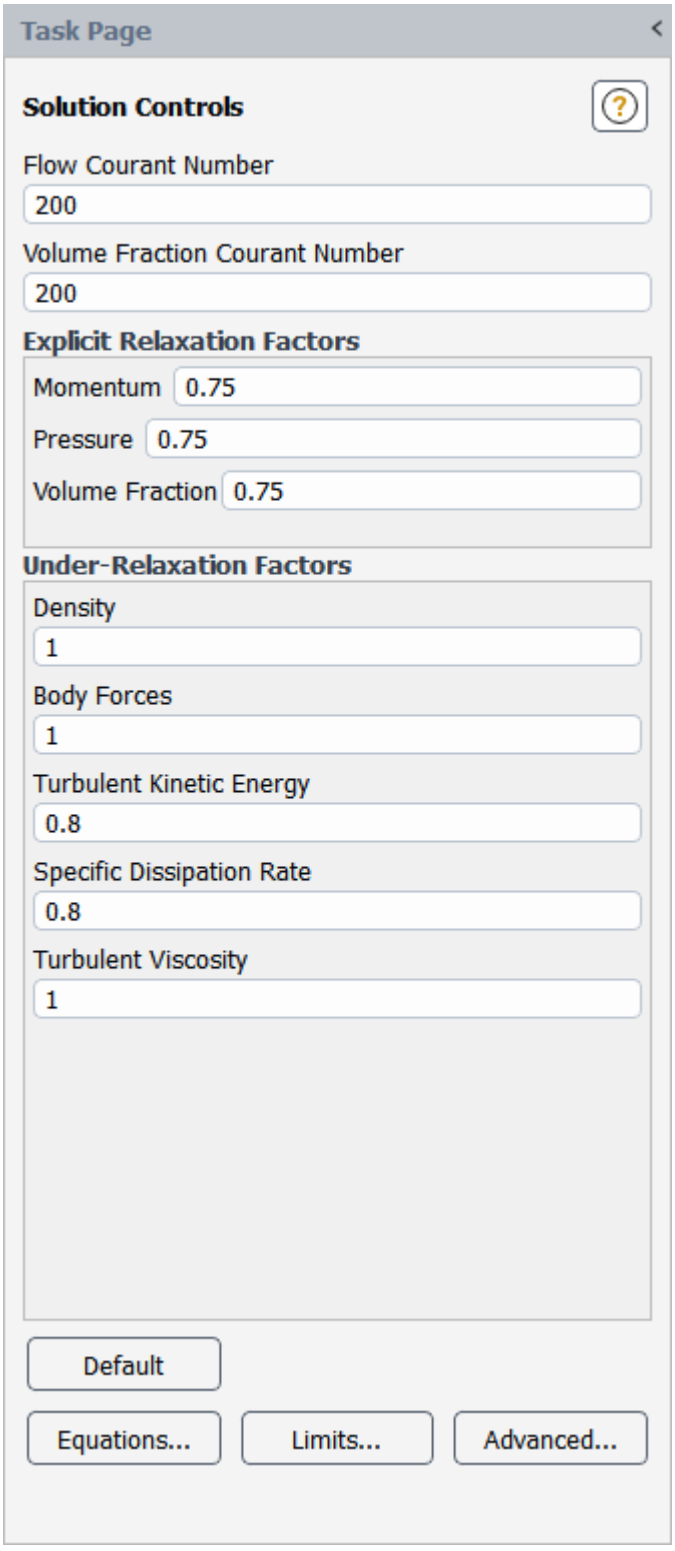
– For transient cases or steady-state cases with the pseudo time method turned off, enter the **Flow Courant Number**, the **Volume Fraction Courant Number**, **Explicit Relaxation Factors**, and the **Under-Relaxation Factors**.

Note:

By default, the **Volume Fraction** is set to 0.75 for transient cases and 0.5 for steady-state cases in the **Explicit Relaxation Factors** group box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.114: The Solution Controls Task Page Displaying the Coupled Volume Fraction Method for the VOF and Mixture Models



- **Eulerian Multiphase Model:**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– For steady-state cases using a pseudo time method (global time step), specify the **Pseudo Time Explicit Relaxation Factors** as described in Global Time Step Method Settings (p. 3676).

Note:

The **Pseudo Time Explicit Relaxation Factors** for the **Volume Fraction** is set to 0.5 by default.

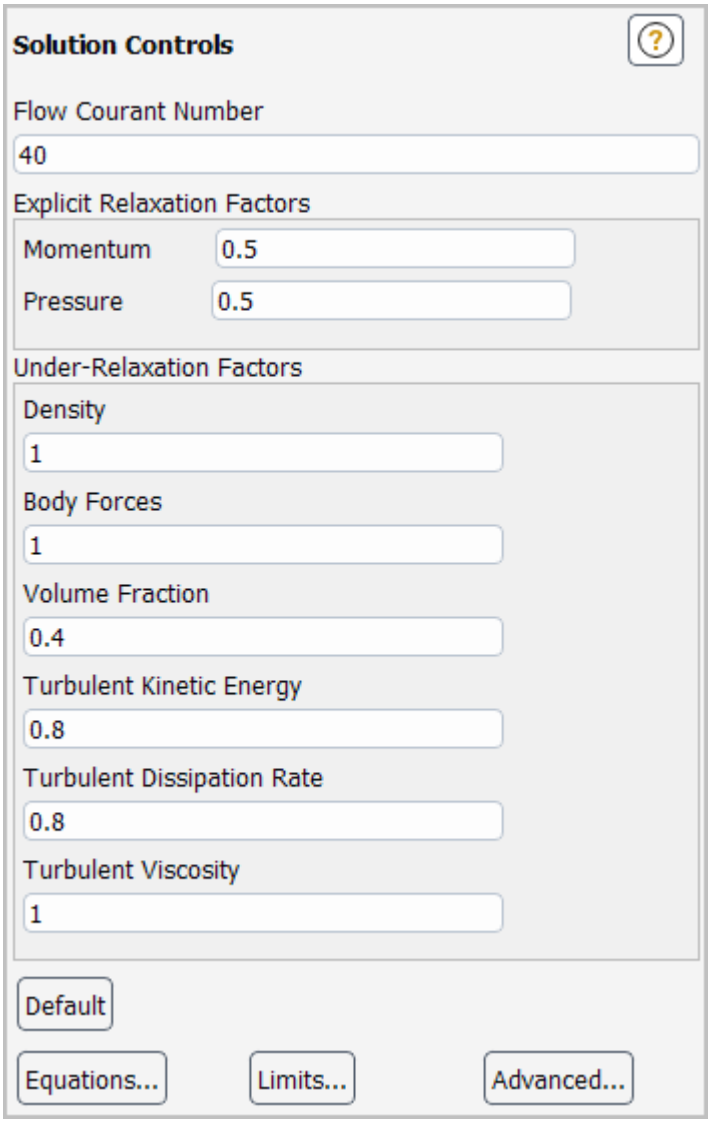
– For transient cases or steady-state cases with the pseudo time method turned off, enter the **Flow Courant Number**, the **Explicit Relaxation Factors**, and the **Under-Relaxation Factors**.

Note:

The under-relaxation for **Volume Fraction** is set by using an implicit **Under-Relaxation Factor**, rather than a **Volume Fraction Courant Number**, and is 0.5 by default.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 26.115: The Solution Controls Task Page Displaying the Coupled Volume Fraction Method for the Eulerian Multiphase Model



26.8.1.5. Default and Stability Controls

26.8.1.5.1. Default Controls

Ansys Fluent provides you with the tools to manage the version-specific defaults.

Starting from version 2020-R1, for newly created cases, the new multiphase defaults will be applied automatically. If you read in an old case file saved in any version prior to Ansys Fluent 2020 R1, the existing settings will be respected.

If you want to apply the new multiphase defaults to the pre 2020-R1 case, you can issue the text user interface (TUI) command as shown in the example below. The information about the new settings will be printed in the Ansys Fluent console. You can review these settings prior to executing the changes.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
solve/set/multiphase-numerics/default-controls/
recommended-defaults-for-existing-cases

The multiphase flow default changes integrated in Fluent version 2020-R1 are
as follows:
 Operating Conditions :
  - In case of gravity, the operating density method is changed to minimum
    of phase averaged densities.
 Surface Tension :
 - For surface tension force modeling, the number of smoothings for volume
   fraction is changed to 2.
 Initialization :
 - The Patch Reconstructed Interface and Volumetric Smoothing options are
   enabled.
 - The Volumetric Smooting Relaxation Factor is changed to 0.5.
 - These settings are applied when executing the Patch and Smooth commands.
 Solver Methods :
 - The unstructured variant of PRESTO is enforced irrespective of a mesh type.

Do you want to execute these default changes? [no] yes
```

To revert some of the new defaults to the pre-R20.1 default settings, you can use the text commands shown in the example below:

```
Revert to the old default of operating density method? [no] yes
  - the operating density method has changed to mixture-averaged.
Revert to the old default of volume fraction smoothings for surface tension? [no]
yes
  - the number of volume fraction smoothings has changed to 1.
Revert to the old variant of PRESTO for cases using structured mesh? [no] yes
  - the structured variant of PRESTO has been enabled for structured mesh.
```

26.8.1.5.2. VOF Solution Stability Controls

The VOF model is numerically more challenging compared to other models due to resolution of sharp discontinuities in the flow system. Body force and pressure gradient imbalances near free surface may lead to unphysical acceleration of the lighter phase. Diffusing these discontinuities adversely affects the interface accuracy; therefore, it may not be an efficient approach.

Complex VOF applications may also suffer from solution instabilities due to several other reasons:

- Improper settings
- Too large time step size
- Higher order oscillations
- Improper convergence
- Poor mesh
- Meshing events where interpolated quantities do not satisfy continuity

Ansys Fluent offers the following options for controlling the solution process:

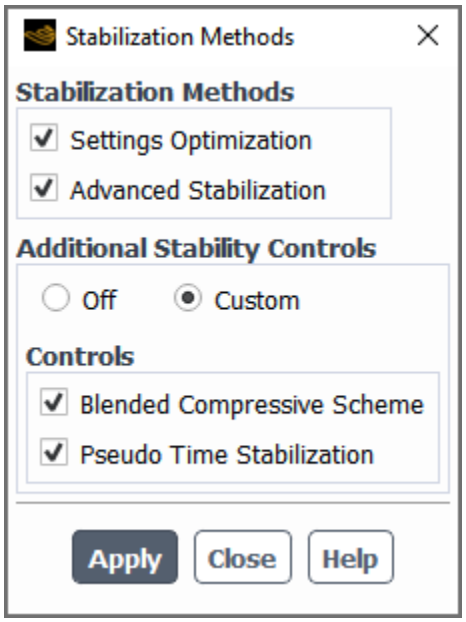
- Stabilization methods
- Velocity limiting treatment

To access the stabilization methods:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the **Solution Methods** task page, click the **Stabilization Methods...** button (**VOF Solution Stability Controls** group box).

   Figure 26.116: The Stabilization Methods Dialog Box



2. In the **Stabilization Methods** dialog box that opens, you can enable the following methods:

   Settings Optimization

   makes the basic optimization changes in the case setup depending on the type of physics to promote better stability.

   Advanced Stabilization

   enables advanced numerical methods and treatments that are important for solution stability. You can use this option as a first step to improve solution stability without significantly compromising its accuracy.

   Additional Stabilization Controls

   contains controls that facilitates better solution stability with aggressive settings by adding diffusion in space and time (**Off** by default). You can use this option as a second step of solution stabilization. When **Custom** is selected, you can enable the following option:

   Blended Compressive Scheme

   applies the following settings:
   - **Sharp/Dispersed** interface modeling type
   - **Compressive** volume fraction spatial discretization
   - Slope limiter = 1.5

   Pseudo Time Stabilization

   uses the following methods:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- False time stepping

  This method provides additional stability for buoyancy-driven flows in the steady-state pseudo time method mode by increasing the diagonal dominance using the false time step size.
- Momentum stabilization

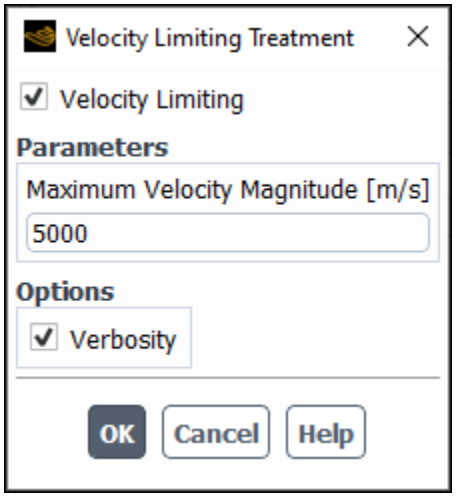
This item is available only for the steady-state solver.

Pseudo time stabilization options are available via the Text User Interface (TUI) as described in Text User Interface for VOF Stability Controls (p. 3197).

To access the velocity limiting treatment:

1. In the **Solution Methods** task page, click the **Velocity Limiting Treatment...** button (**VOF Solution Stability Controls** group box).

   Figure 26.117: The Velocity Limiting Treatment Dialog Box

2. In the **Velocity Limiting Treatment** dialog box that opens, enable **Velocity Limiting**.

   Unphysical velocity spikes may arise from solution instabilities. Most often, the velocity spikes are very local in nature and subside as the iterations progress. However, sometimes these spikes may grow and can cause sudden divergence. The velocity limiting treatment can increase the solution stability by curbing these spikes, which are numerical and local in nature.

3. In the **Parameters** group box, you can adjust the **Maximum Velocity Magnitude**.

   This limit can be roughly taken as 2-3 times of system bulk velocity.

4. If you want the number of cells where the limiting is applied to be printed every time-step, enable **Verbosity**. Note that excessive limiting can adversely affect the convergence and may pose a question on the validity of results.

26.8.1.5.3. Text User Interface for VOF Stability Controls

The solution stabilization controls can be also set in the Text User Interface (TUI). The TUI also provides additional options.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Additional stabilization controls can be executed using the following text commands:

  – `solve/set/multiphase-numerics/solution-stabilization/execute-additional-stability-controls?`

  – `solve/set/multiphase-numerics/solution-stabilization/execute-advanced-stabilization?`

  – `solve/set/multiphase-numerics/solution-stabilization/additional-stabilization-controls/execute-settings-optimization?`

- For complex VOF and mixture multiphase cases, Ansys Fluent provides advanced options for improving stability via the following text commands:

  `solve/set/multiphase-numerics/advanced-stability-controls/`

  The available controls are:

  – Pseudo time stability controls

    These controls are available for the **Coupled** pressure-velocity coupling scheme with **Global Time Step** selected from the **Pseudo Time Method** list. They can be accessed via the `solve/set/multiphase-numerics/advanced-stability-controls/pseudo-time/` TUI menu:

    You can use the following options:

    `solve/set/multiphase-numerics/advanced-stability-controls/pseudo-time/false-time-step-linearization?`

    Provides additional stability for buoyancy-driven flows in the steady-state pseudo time method mode by increasing the diagonal dominance using the false time step size.

    `solve/set/multiphase-numerics/advanced-stability-controls/pseudo-time/smoothed-density-stabilization-method?`

    Smooths the cell density near the interface, therefore avoiding unphysical acceleration of lighter phase in the vicinity of interface. The default number of density smoothings is 2. In case of very large unphysical velocities across the interface, you can increase this number when prompted with `Number of density smoothings`.

    For automatic pseudo time step calculations, you can use the advanced method for better solution stability. The method is available only when the **Automatic** time step method is used for a pseudo time step calculation (see Pseudo Time Settings for the Calculation (p. 3680) for details). It is enabled by issuing the following text command:

    `solve/set/multiphase-numerics/advanced-stability-controls/pseudo-time/auto-dt-advanced-controls/enable?`

    `Enable advanced automatic time-stepping? [no]` **`yes`**

    In addition to time scales described in Global Time Step Method in the *Fluent Theory Guide*, this method considers viscous and surface tension time scales using the following correlations.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\Delta t_{visc} = \frac{L^2}{\upsilon_{avg}} \tag{26.42}$$

$$\triangle t_{surf} = 0.01\sqrt{\frac{\rho_{avg} L^3}{2\pi\sigma}} \tag{26.43}$$

In Equation 26.42 (p. 3199) and Equation 26.43 (p. 3199), the following nomenclature is used:

$\triangle t_{visc}$ = viscous time scale

$\triangle t_{surf}$ = surface tension time scale

L = length scale based on the user-selected method

υ_{avg} = averaged kinematic viscosity

ρ_{avg} = averaged fluid density

σ = surface tension coefficient

You can use the following controls available via the TUI to adjust the advanced method settings:

solve/set/multiphase-numerics/advanced-stability-controls/pseudo-time/auto-dt-advanced-controls/dt-factor-max

Sets the maximum limit for increasing the size of the pseudo time step. Default: 1.2.

solve/set/multiphase-numerics/advanced-stability-controls/pseudo-time/auto-dt-advanced-controls/dt-factor-min

Sets the minimum limit for increasing the pseudo time step size. Default: 0.25.

solve/set/multiphase-numerics/advanced-stability-controls/pseudo-time/auto-dt-advanced-controls/dt-init-limit

Sets the maximum pseudo time step size for the first iteration. Default: 0.01s.

The time step size during the first iteration may not be optimal due to lack of solution data or time-step estimation methods for all possible scenarios. A very large time step size at the start can distort the whole solution; therefore, limiting the initial time step size can help to smooth the start of the simulation.

solve/set/multiphase-numerics/advanced-stability-controls/pseudo-time/auto-dt-advanced-controls/dt-max

Sets the maximum pseudo time step size for the entire simulation. Default: 1000s.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

solve/set/multiphase-numerics/advanced-stability-controls/pseudo-time/auto-dt-advanced-controls/max-velocity-ratio

Sets the maximum velocity ratio (maximum velocity to average velocity) above which the solver freezes the time step size for better solution stability. Default: 100.

Note:

Since the predicted time step size depends heavily on the length scale, you should specify a representative **Length Scale Method** (in the **Run Calculation** task page, **Pseudo Time Settings** group box) based on your problem.

– Equation order controls

These controls are accessed via the `solve/set/multiphase-numerics/advanced-stability-controls/equation-order/` TUI menu.

By default, Ansys Fluent solves the flow at the start of each iteration (except for explicit VOF problems where the flow is solved right after solving the volume fraction equation). You can use the `solve-flow-last?` text option (under the `equation-order` TUI menu) to solve the flow equation at the end of each iteration. This may improve the behavior at the start of the new time step if the solution does not converge properly.

– Pressure velocity coupling controls

These controls are accessed via the `solve/set/multiphase-numerics/advanced-stability-controls/p-v-coupling/` TUI menu

The following options are available:

→ `coupled-vof/`

By default, Ansys Fluent performs the linearization of the buoyancy terms with respect to the volume fraction while solving the momentum equation. This is critical for fast convergence. However, for certain unstable cases, using the linearization in all regions except near the free surface (the blended treatment of the buoyancy force linearization) may provide better stability. This can be done via the `buoyancy-force-linearization?` text command (under the `coupled-vof/` TUI menu).

→ `pressure-interpolation/`

You can use the modified variant of the **Body Force Weighted** pressure interpolation scheme for improved robustness and convergence behavior in general. This option is available via the `modified-bfw-scheme` text commend and is highly recommended for flows with a high viscosity ratio where the original body force weighted scheme has severe limitations. Note that this scheme may not offer the same level of accuracy for very highly rotating flows as PRESTO.

→ `rhie-chow-flux/`

For collocated grids, an accurate calculation of flux at the cell faces is extremely important in order to avoid pressure and velocity checkerboarding. However, sometimes higher-order

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

interpolations become a source of instability in the vicinity of free surface or for a highly-skewed mesh. If the solution becomes unstable while using the higher-order momentum discretization, try switching to the low-order flux calculation (rather than the low-order momentum discretization) by using the `low-order-rhie-chow?` text option (under the `rhie-chow-flux` TUI menu). This option uses the low-order velocity interpolation in the flux calculation.

→ `skewness-correction/`

You can use the `limit-pressure-correction-gradient?` option (under the `skewness-correction/` TUI menu) to limit the pressure correction gradient term for better stability on a skewed mesh while using skewness corrections with the **PISO**, **SIMPLEC**, or **Fractional-Step** pressure-velocity coupling scheme.

26.8.1.6. Heat Transfer and Radiative Flux Distribution for Non-Eulerian Multiphase Models

For Eulerian multiphase models, the energy equation is solved in the phase domain and therefore, heat flux can be reported based on the mixture domain or any of the phases individually. However, for the VOF and Mixture multiphase models, the energy equation is discretized in the phase domains but solved only in the Mixture domain after summing over the individual phases. Boundary heat fluxes for flow boundaries are stored at the phase domain but wall heat fluxes are only stored at the mixture domain. This approach of modeling wall heat fluxes using mixture domain properties has certain deficiencies in proper accounting and reporting of phasic wall heat fluxes.

The phasic wall heat fluxes for the VOF and Mixture multiphase models can be calculated using an approach which makes use of the linearized coefficients at the phase level. These coefficients are then accumulated in the mixture domain to be used in the energy equation. This phasic approach corrects the reporting of wall heat fluxes at the wall boundaries.

The phasic approach can activated using the following text command:

```
/solve/set/multiphase-numerics/energy/phasic-wall-heat-flux-form?
```

Note that the default is **no** meaning the phasic formulation is not used. To enable, respond **yes** to the prompt, and then you may initialize or continue the run with the phasic formulation active.

26.8.1.7. Steady-State Solution Strategies

For steady-state problems that either involve a closed domain or have no separate inflow/outflow boundaries for all the phases, mass conservation cannot be guaranteed with the pseudo time method set to global time step. It might be better to run these problems with the transient solver using few iterations per time step to achieve faster results.

26.8.2. Model-Specific Solution Strategies

Strategies that are applicable to specific multiphase models are discussed in the following sections:

26.8.2.1. VOF Model

26.8.2.2. Mixture Model

26.8.2.3. Eulerian Model

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.8.2.4. Wet Steam Model

26.8.2.1. VOF Model

Several recommendations for improving the accuracy and convergence of the VOF solution are presented here.

26.8.2.1.1. Setting the Reference Pressure Location

The site of the reference pressure can be moved to a location that will result in less round-off in the pressure calculation. By default, the reference pressure location is the center of the cell at or closest to the point (0,0,0). You can move this location by specifying a new **Reference Pressure Location** in the Operating Conditions Dialog Box (p. 4978).

Setup → **Cell Zone Conditions** → **Operating Conditions...**

The position that you choose should be in a region that will always contain the least dense of the fluids (for example, the gas phase, if you have a gas phase and one or more liquid phases). This is because variations in the static pressure are larger in a more dense fluid than in a less dense fluid, given the same velocity distribution. If the zero of the relative pressure field is in a region where the pressure variations are small, less round-off will occur than if the variations occur in a field of large nonzero values. Thus in systems containing air and water, for example, it is important that the reference pressure location be in the portion of the domain filled with air rather than that filled with water.

26.8.2.1.2. Pressure Interpolation Scheme

For all VOF calculations, you should use the body-force-weighted pressure interpolation scheme or the PRESTO! scheme.

Solution → **Controls**

By default, Ansys Fluent uses an unstructured variant of the PRESTO pressure scheme for all types of meshes. For cases created in versions prior to Ansys Fluent 2020 R1, the structured variant of the PRESTO scheme will be used for structured meshes.

To enable the unstructured variant of the PRESTO scheme, use the following text command:

```
solve/set/vof-numerics
```

When prompted with

```
Use unstructured variant of PRESTO pressure scheme? [no]
```

Answer `yes`.

Note:

You do not need to use this text command option for cases that involve unstructured meshes since the unstructured variant of the PRESTO scheme is automatically enabled for such cases.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.8.2.1.3. Discretization Scheme Selection

The availability of discretization schemes for volume fraction is summarized in Spatial Discretization Schemes for Volume Fraction (p. 2924).

If you are modeling a sharp interface regime on meshes that have high aspect ratio cells, large jumps in cell size, or high skewness, interface capturing schemes such as **Compressive** and **CICSAM** will not produce very sharp results. For such cases, you should use the **Geo-Reconstruct** scheme.

For meshes of good quality, it is recommended that you use the **Compressive** scheme, which is available in both explicit and implicit formulations and is applicable to both sharp and dispersed flow regimes. The **Compressive** scheme also provides better accuracy compared with **CICSAM** and **Modified HRIC** for most cases. When used with the implicit formulation and the second order time scheme, the **Compressive** scheme produces quite a sharp interface.

When using the **Compressive** spatial discretization scheme, it is recommended that you use step-wise sharpening after the flow transitions from the diffused zone to the sharpening zone. In other words, you would want to transition from first order to second order, followed by a transition from second order to compressive. You would want to take this approach if the flow has a smooth transition from a sharp interfacial zone to a diffused interfacial zone. However, if the flow has a nonuniform transition from a diffused interfacial zone to a sharp interfacial zone, this might create unphysical sharpening of the interface if not handled properly, especially for transient cases. For example, transitioning from a slope limiter of 2 (compressive) to a slope limiter of 0 or 1 (first order or second order) might be acceptable, but a first order to compressive transition might create unphysical sharpening of the interface.

If you are solving a problem with the **evaporation-condensation** mass transfer mechanism enabled (Including Mass Transfer Effects (p. 2940)) it is recommended that you use one of the diffusive interface capturing schemes such as **QUICK**, **HRIC**, or **Phase Localized Compressive** (Discretizing Using the Phase Localized Compressive Scheme (p. 3015)).

Important:

The **BGM** scheme produces a sharp interface, which may result in poor convergence in some cases. In such situations, we recommend you use a low value for the VOF under-relaxation. In addition, you can start with the **Compressive** or **Modified HRIC** scheme and then switch to the **BGM** scheme.

26.8.2.1.4. High-Order Rhie-Chow Face Flux Interpolation

In VOF modeling, using a high-order discretization scheme for the momentum transport equations may reduce the stability of the solution compared to cases using first-order discretization. In such situations, there are a couple of recommendations:

1. Use a low-order variant of the Rhie-Chow face flux interpolation. This is enabled using the following text command:

   `solve` → `set` → `numerics`

   You will be asked

   ```
   disable high order Rhie-Chow flux? [no]
   ```

to which you will respond yes.

2. Use a hybrid treatment of high-order Rhie-Chow face flux interpolation. This is enabled using the following text command:

   solve → set → vof-numerics

   You will be asked

   ```
   Use hybrid treatment for high order Rhie-Chow flux?  [no]
   ```

   to which you will respond yes.

   Hybrid treatment allows you to use a high-order variant of the Rhie-Chow face flux interpolation everywhere inside the domain, except in the vicinity of the interface where a low-order variant is used for the interpolation of the face flux. This treatment could be helpful to get better convergence without compromising much of the accuracy.

26.8.2.1.5. Treatment of Unsteady Terms in Rhie-Chow Face Flux Interpolation

For cases using Moving Mesh/Dynamic Mesh/Multiple Reference Frame models, Ansys Fluent does not consider the unsteady terms for Rhie-Chow face flux interpolation by default. Accounting for the unsteady terms helps to achieve better solution stability, especially when using a lower time step size with the PRESTO scheme.

To enable forced treatment of unsteady terms in Rhie-Chow face flux interpolation, use the following text command:

solve → set → vof-numerics

You will be asked

```
Use forced treatment of unsteady terms in Rhie-Chow flux? [no]
```

to which you will respond yes.

26.8.2.1.6. Pressure-Velocity Coupling and Under-Relaxation for the Time-dependent Formulations

Another change that you should make to the solver settings is in the pressure-velocity coupling scheme and under-relaxation factors that you use. The PISO scheme is recommended for transient calculations in general. Using PISO allows for increased values on all under-relaxation factors, without a loss of solution stability. You can generally increase the under-relaxation factors for all variables to 1 and expect stability and a rapid rate of convergence (in the form of few iterations required per time step). For calculations on tetrahedral or triangular meshes, an under-relaxation factor of 0.7–0.8 for pressure is recommended for improved stability with the PISO scheme.

Solution → Controls

As with any Ansys Fluent simulation, the under-relaxation factors will need to be decreased if the solution exhibits unstable, divergent behavior with the under-relaxation factors set to 1. Reducing the time step size is another way to improve the stability.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

26.8.2.1.7. Under-Relaxation for the Steady-State Formulation

If you are using the steady-state implicit VOF scheme, the under-relaxation factors for all variables should be set to values between 0.2 and 0.5 for improved stability.

26.8.2.2. Mixture Model

26.8.2.2.1. Setting the Under-Relaxation Factor for the Slip Velocity

You should begin the mixture calculation with a low under-relaxation factor for the slip velocity. A value of 0.2 or less is recommended. If the solution shows good convergence behavior, you can increase this value gradually.

26.8.2.2.2. Calculating an Initial Solution

For some cases (for example, cyclone separation), you may be able to obtain a solution more quickly if you compute an initial solution without solving the volume fraction and slip velocity equations. Once you have set up the mixture model, you can temporarily disable these equations and compute an initial solution.

Solution → **Controls**

In the Equations Dialog Box (p. 5166), deselect **Volume Fraction** and **Slip Velocity** in the **Equations** list. You can then compute the initial flow field. Once a converged flow field is obtained, turn the **Volume Fraction** and **Slip Velocity** equations back on again, and compute the mixture solution.

26.8.2.3. Eulerian Model

26.8.2.3.1. Calculating an Initial Solution

To improve convergence behavior, you may want to compute an initial solution before solving the complete Eulerian multiphase model. There are multiple methods you can use to obtain an initial solution for an Eulerian multiphase calculation:

- Set up and solve the problem using the mixture model (with slip velocities) instead of the Eulerian model. You can then enable the Eulerian model, complete the setup, and continue the calculation using the mixture-model solution as a starting point.
- Set up the Eulerian multiphase calculation as usual, but compute the flow for only the primary phase. To do this, deselect **Volume Fraction** in the **Equations** list in the Equations Dialog Box (p. 5166). Once you have obtained an initial solution for the primary phase, turn the volume fraction equations back on and continue the calculation for all phases.
- Use the mass-flow inlet boundary condition to initialize the flow conditions. It is recommended that you set the value of the volume fraction close to the value of the volume fraction at the inlet.
- At the beginning of the solution, a smaller time step size is recommended in order to reach convergence.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If using the volume fraction explicit scheme, do not start with a large Courant number at the beginning of your run.
- If using the volume fraction explicit scheme, start a run with a smaller time step size and then increase it. Alternatively, this could be done by using multiphase-specific time stepping, which would increase the time step size based on the input parameters.
- For problems involving a free surface or sharp interfaces between the phases, using the **symmetric** drag law (available in the **Forces** tab of the **Multiphase Model** dialog box) is recommended.
- Multiphase-specific time stepping is not recommended for compressible flows.

Important:

You should *not* try to use a single-phase solution obtained without the mixture or Eulerian model as a starting point for an Eulerian multiphase calculation. Doing so will not improve convergence, and may make it even more difficult for the flow to converge.

26.8.2.3.2. Temporarily Ignoring Lift and Virtual Mass Forces

If you are planning to include the effects of lift and/or virtual mass forces in a steady-state Eulerian multiphase simulation, you can often reduce stability problems that sometimes occur in the early stages of the calculation by temporarily ignoring the action of the lift and the virtual mass forces. Once the solution without these forces starts to converge, you can interrupt the calculation, define these forces appropriately, and continue the calculation.

26.8.2.3.3. Using W-Cycle Multigrid

For problems involving a packed-bed granular phase with very small particle sizes (on the order of 10 μm), convergence can be obtained by using the W-cycle multigrid for the pressure. In the **Multigrid** tab, under **Fixed Cycle Parameters** in the Advanced Solution Controls Dialog Box (p. 5168), you may need to use higher values for **Pre-Sweeps**, **Post-Sweeps**, and **Max Cycles**. When you are choosing the values for these parameters, you should also increase the **Verbosity** to 1 in order to monitor the AMG performance; that is, to make sure that the pressure equation is solved to a desired level of convergence within the AMG solver during each global iteration. See Defining the Phases for the Eulerian Model (p. 3066) for more information about granular phases, and The V and W Cycles in the Theory Guide and Modifying Algebraic Multigrid Parameters (p. 3741) for details about multigrid cycles.

26.8.2.3.4. Including the Anisotropic Drag Law

When using the anisotropic drag law (Including the Multi-Fluid VOF Model (p. 3129)), it is recommended that you start the solution with a lower anisotropy ratio. After your solution has run for some time, you can then increase the ratio by reducing the friction factor in the tangential direction. Note that You can also start the solution with the symmetric drag law, then change to the anisotropic drag law.

Using a smaller under-relaxation for pressure and momentum may also help in convergence for cases with a higher anisotropy ratio.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If the flow for a particular phase is important in both directions (normal and tangential to the interface), use a lower anisotropy ratio, between 100-1000. A higher anisotropy ratio might cause an unstable solution for such cases. For a higher anisotropy ratio of more than 1000, a smaller under-relaxation for pressure and momentum is recommended. When using the coupled multiphase solver, if the solution is unstable with a higher anisotropy ratio, then reducing the courant number may be beneficial. `Anisotropic Drag Method` [1], with `Viscosity option` [2] is recommended for a higher viscosity ratio.

26.8.2.3.5. Using Reference Density

In the Eulerian multiphase model, the volume fraction and continuity equations (see Equation 14.203 in the *Fluent Theory Guide*) can be represented either in the mass or volumetric form by choosing a reference density method using the following text command:

```
solve/set/mp-reference-density
```

The available reference methods are listed in the below table.

Reference Density Method	Volume Fraction Equation Representation	Continuity Equation Representation	Phase Reference Density Calculation
`0` (default)	Mass form	Volumetric form	Averaged density of the phase
`1`	Mass form	Volumetric form	Cell density of the phase
2	Mass form	Mass form	1
3	Volumetric form	Volumetric form	Cell density of the phase

There is not much difference between Methods `0` and `1` for incompressible flows. Method 2, in general, may not be very stable at larger time steps and, therefore, is not recommended. Method 3 is similar to the Mixture multiphase model and may provide better stability in certain cases.

Note that both the volume fraction and continuity equations remain mass conservative regardless of their representation. Different representations affect only the solution stability and convergence behavior.

26.8.2.3.6. Controlling NITA Solution Options via the Text Interface

For the Eulerian multiphase model, the Non-Iterative Time Advancement (NITA) scheme uses PISO as a pressure velocity coupling method, where the pressure correction equation is solved in two steps named predictor and corrector.

For the face pressure calculation, the Eulerian multiphase model uses PRESTO as a default method for both the predictor and corrector steps.

You can change the face pressure interpolation method using the following text command:

```
solve/set/multiphase-numerics/nita-controls/face-pressure-options
```

To select Body Force Weighted as a face pressure interpolation method for both predictor and corrector steps, enter **`yes`** at the `enable body-force-weighted for face pressure calculation?` prompt.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To select the Second Order method as a face pressure interpolation method for both predictor and corrector steps:

1. Enter **no** at the `enable body-force-weighted for face pressure calculation?` prompt.
2. When prompted with `face pressure calculation method for predictor step`, choose Second Order (**1**).

To select the Second Order method as a face pressure interpolation method for the corrector step only:

1. Enter **no** at the `enable body-force-weighted for face pressure calculation?` prompt.
2. When prompted with `face pressure calculation method for predictor step`, retain the default selection of PRESTO (**0**).
3. When prompted with `face pressure calculation method for corrector step`, choose Second Order (**1**) .

When PRESTO is used for the face pressure calculation, you will be asked whether you want to exclude transient terms. Because the impact of fluxes on the face pressure is inversely proportional to the time step size, which may lead to numerical problems as the time step size decreases, the exclusion of transient terms is recommended for very small time steps when using PRESTO:

```
exclude transient terms in face pressure calculation? [no] yes
```

26.8.2.4. Wet Steam Model

26.8.2.4.1. Boundary Conditions, Initialization, and Patching

When you use the wet steam model (described in Wet Steam Model Theory in the Theory Guide, and Setting Up the Wet Steam Model (p. 3173)), the following two field variables will show up in the inflow, outflow boundary dialog boxes, and in the **Solution Initialization** task page and **Patch** dialog boxes.

- **Liquid Mass Fraction** (or the wetness factor)

  In general, for dry steam entering flow boundaries the wetness factor is zero.

- **Log10 (Droplets Per Unit Volume)**

  In general this value is set to zero, indicating zero droplets entering the domain.

26.8.2.4.2. Solution Limits for the Wet Steam Model

When you select the wet steam model for the first time, a message is displayed indicating that the **Minimum Static Temperature** should be adjusted to 273 K since the accuracy of the built-in steam data is not guaranteed below a value of 273 K. If you use your own steam property functions, you can adjust this limit to whatever is permissible for your data.

To adjust the temperature limits, go to the **Solution Limits** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution → Controls → Limits...

The default maximum wetness factor or liquid mass fraction (β) is set to 0.1. In general, during the convergence process, it is common that this limit will be reached, but eventually the wetness factor will drop below the value of 0.1. However, in cases where the limit must be adjusted, you can do so using the text user interface.

`define → models → multiphase → wet-steam → set → max-liquid-mass-fraction`

Important:

Note that the maximum wetness factor should not be set beyond 0.2 since the present model assumes a low wetness factor. When the wetness factor is greater than 0.1, the solution tends to be less stable due to the large source terms in the transport equations. Thus, the maximum wetness factor has been set to a default value of 0.1, which corresponds to the fact that most nozzle and turbine flows will have a wetness factor less than 0.1.

26.8.2.4.3. Solution Strategies for the Wet Steam Model

If you face convergence difficulties while solving wet steam flow, try the following solver settings:

1. Lower the under-relaxation factor for the wet steam equation below the current set value. The default value is 0.8 for the density-based solver and 0.6 for the pressure-based solver. The under-relaxation factor can be found on the **Solution Controls** task page.

   Solution → Controls → Controls

2. Solve for an initial solution with no condensation. Once you have obtained a proper initial solution, turn on the condensation.

   To turn condensation on or off, go to the **Solution Controls** task page.

   Solution → Controls → Controls

   In the **Equations** dialog box, deselect **Wet Steam** in the **Equations** list. When doing so, you are preventing condensation from taking place while still computing the flow based on steam properties. Once a converged flow field is obtained, turn the **Wet Steam** equation back on again and compute the mixture solution.

If you are still unable to obtain a converged solution, try lowering the flow Courant number when using the density-based solver or the coupled pressure-based solver with the pseudo time method turned off. For the segregated pressure-based solver, try lowering the under-relaxation factor for the momentum equations. These parameters also can be found in the **Solution Controls** task page. For the coupled pressure-based solver with the pseudo time method turned off, try lowering the value for the **Time Scale Factor** available in the **Run Calculation** task page:

Solution → Run Calculation

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Lastly, you can try to use the first-order discretization schemes for the solution, available on the **Solution Methods** task page.

Solution → **Solution** → **Methods**

If you have convergence difficulties for a turbomachinery case using a Mixing Plane (MP) interface, it is recommend that you first obtain a solution using the No Pitch-Scale (NPS) interface type. Once you have a converged flow field, switch the MP interface back on again and then attempt to compute the solution. Details of different GTI types are described in Blade Row Interaction Modeling (p. 1956).

26.9. Multiphase Case Check

You can check the multiphase case setup using the multiphase check function available via the text user interface (TUI) command:

```
/report/mphase-summary
```

This function prints for you a summary of useful settings, important model information, and recommendations for choosing case parameters. Your case will be checked for compliance in the following categories:

- `MODELING`
- `MESHING`
- `PHASES`
- `PHASE INTERACTION`
- `BOUNDARY CONDITIONS`
- `CELL ZONE CONDITIONS`
- `OPERATING CONDITIONS`
- `SOLVER METHODS/CONTROLS`
- `ADVANCED SOLVER NUMERICS`
- `MULTIPHASE NUMERICS`
- `SOLUTION LIMITS`
- `MONITORS/RESIDUALS`
- `INITIALIZATION`
- `RUN CALCULATION`
- `POST PROCESSING`

Important:

To execute this command, a case and valid data (solution or initialization) are required. If the case has been modified, you need to either reinitialize or rerun the case.

When prompted for `verbosity`, you can set the level of detail for the report. A value of **`0`** (the default) corresponds to a basic report, while a value of **`1`** corresponds to a more detailed report.

In each category, established rules and built-in best practices will be available, along with recommended changes to your current settings. At your discretion, you may choose to apply the recommendations,

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

or keep your current settings. Refer to the appropriate sections in the Fluent User's Guide (p. 1) for more information about the recommended changes.

For VOF cases, you can generate a summary report for the case setup using the **VOF Check** button that appears in the **Solution Initialization** task page after you initialize your case. The action of this button is similar to the `report/mphase-summary` text command with the verbosity set to 0.

26.10. Postprocessing for Multiphase Modeling

Each of the three general multiphase models provides a number of additional field functions that you can plot or report. You can also report flow rates for individual phases for all three models, and display velocity vectors for the individual phases in a mixture or Eulerian calculation.

Information about these postprocessing topics is provided in the following subsections:

26.10.1. Model-Specific Variables

26.10.2. Displaying Velocity Vectors

26.10.3. Reporting Fluxes

26.10.4. Reporting Forces on Walls

26.10.5. Reporting Flow Rates

26.10.1. Model-Specific Variables

When you use one of the general multiphase models, some additional field functions will be available for postprocessing, as listed in this section. Most field functions that are available in single phase calculations will be available for either the mixture or each individual phase, as appropriate for the general multiphase model and specific options that you are using. See Field Function Definitions (p. 4167) for a complete list of field functions and their definitions. Displaying Graphics (p. 3869) and Reporting Alphanumeric Data (p. 4097) explain how to generate graphics displays and reports of data.

Note:

Datasets of steady or unsteady statistics created using the Zone-Specific Sampling Options Dialog Box (p. 5233) will include the phase name within the dataset name. For example, to visualize contours of the average volume fraction of CO2 in a mixture, you would select **Unsteady Statistics...** and **Mean-volume-fraction-co2-dataset** in the **Contours of** drop-down lists.

26.10.1.1. VOF Model

For VOF calculations you can generate graphical plots or alphanumeric reports of the following additional item:

- **Volume fraction** (in the **Phases...** category)

  This item is available for each phase.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The variables that are not phase specific are available (for example, variables in the **Pressure...** and **Velocity...** categories) and represent mixture quantities. Thermal quantities will be available only for calculations that include the energy equation.

26.10.1.2. Mixture Model

For calculations with the mixture model, you can generate graphical plots or alphanumeric reports of the following additional items:

- **Diameter** (in the **Properties...** category)

  This item is available only for secondary phases.

- **Volume fraction** (in the **Phases...** category)

  This item is available only for secondary phases.

- **Interfacial Area Concentration** (in the **Interfacial Area Concentration...** category)

  This item is available only for secondary phases.

The variables that are not phase specific are available (for example, variables in the **Pressure...** category) represent mixture quantities. Thermal quantities will be available only for calculations that include the energy equation.

26.10.1.3. Eulerian Model

For Eulerian multiphase calculations you can generate graphical plots or alphanumeric reports of the following additional items:

- **Diameter** (in the **Properties...** category)

  This item is available only for secondary phases.

- **Granular Conductivity** (in the **Properties...** category)

  This item is available only for granular phases.

- **Granular Pressure** (in the **Granular Pressure...** category)

  This item is available only for granular phases.

- **Granular Temperature** (in the **Granular Temperature...** category)

  This item is available only for granular phases.

- **Volume fraction** (in the **Phases...** category)

  This item is available only for secondary phases.

- **Interfacial Area Concentration** (in the **Interfacial Area Concentration...** category)

  This item is available only for secondary phases.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The availability of turbulence quantities will depend on which multiphase turbulence model you used in the calculation. Thermal quantities will be available (on a per-phase basis) only for calculations that include the energy equation.

More advanced options for the mixture phase are available under the **Phases...** category, allowing you to select from a list of variables to postprocess. To access the entire list in the graphical user interface, type the following text command:

`solve` → `set` → `advanced` → `retain-temporary-solver-mem`

This will expose a list under the **Phases** category for the mixture phase, one of which will be the **Phase ID**. Selecting this option allows you to plot contours of phase IDs for the volume fraction, which will facilitate phase distribution display when more than two phases are present for free surface calculations.

Note:

The text command for retaining the temporary solver memory is incompatible with automatic adaption in serial and parallel for all multiphase models.

Important:

This option is available for all the multiphase models. However, note that only cell values should be plotted for this option. Make sure that the **Node Values** option is not selected as it will show the wrong phase ID contours at the interface.

26.10.1.4. Multiphase Species Transport

For calculations using species transport with either of the multiphase models, you can generate graphical plots or alphanumeric reports of the following additional items:

- **Mass Fraction of *`species-n`*** (in the **Species...** category)

  This item is available for each species.

- **Mole Fraction of *`species-n`*** (in the **Species...** category)

  This item is available for each species.

- **Molar Concentration of *`species-n`*** (in the **Species...** category)

  This item is available for each species.

- **Lam. Diff Coeff of *`species-n`*** (in the **Species...** category)

  This item is available for each species.

- **Eff. Diff. Coeff. of *`species-n`*** (in the **Species...** category)

  This item is available for each species.

- **Enthalpy of *`species-n`*** (in the **Species...** category)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This item is available for each species.

- **Relative Humidity** (in the **Species...** category).
- **Turbulent Rate of Reaction-n** (in the **Reactions...** category)

  This item is available for each species.
- **Rate of Reaction** (in the **Reactions...** category).
- **Mass Transfer Rate *n*** (in the **Phase Interaction...** category)

  This item is available for each mass transfer mechanism that you defined.
- **Saturation Vapor Pressure *n*** (in the **Phase Interaction...** category)

  This item is available for each cavitation mass transfer mechanism that you defined.
- **Mass Concentration of *species-n* (Phase Level)** (in the **Species...** category)

  This item is available for all the phases that have species components.
- **Mass Concentration of *species-n* (Mixture Level)** (in the **Species...** category)

  This item is available for all the phases that have species components.

Thermal quantities will be available only for calculations that include the energy equation.

Computing Mass of a Component in a Phase

The volume integral is always calculated at the mixture level for the whole cell volume. Therefore, you can use a **Volume Integral** of **Mass Concentration of *species-n* (Mixture Level)** to compute the mass of a phase component n. The reported value will be calculated as:

$$m_{n,p}=\rho_p y_{n,p} \alpha_p V$$

where

ρ_p = density of phase p

$y_{n,p}$ = mass fraction of component n in phase p

$y_{n,p}$ = mass fraction of component n in phase p

α_p = volume fraction of phase p

V = cell volume

26.10.1.5. Wet Steam Model

Ansys Fluent provides a wide range of postprocessing information related to the wet steam model.

The wet steam related items can be found in **Wet Steam....** category of the variable selection drop-down list that appears in the postprocessing dialog boxes.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Liquid Mass Fraction**
- **Liquid Mass Generation Rate**
- **Log10 (Droplets Per Unit Volume)**
- **Log10 (Droplets Nucleation Rate)**
- **Steam Density (Gas-Phase)**
- **Liquid Density (Liquid-Phase)**
- **Mixture Density**
- **Saturation Ratio**
- **Saturation Pressure**
- **Saturation Temperature**
- **Subcooled Vapor Temperature**
- **Droplet Surface Tension**
- **Droplet Critical Radius (microns)**
- **Droplet Average Radius (microns)**
- **Droplet Growth Rate (microns/s)**

26.10.1.6. Dense Discrete Phase Model

For postprocessing, both the DPM and the Eulerian multiphase capabilities are retained. In addition to usual DPM postprocessing (Postprocessing for the Discrete Phase (p. 2833)), you can display, for example, vector plots of the particle's velocity field. Make sure to select the discrete phase from the **Phase** drop-down list. For transient simulations that include the dense discrete phase model, you can display the following when the **Unsteady Statistics...** category is selected:

- **Mean Velocity**
- **Mean Volume Fraction**
- **Mean Phase Diameter**
- **RMSE Velocity**
- **RMSE Volume Fraction**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **RMSE Phase Diameter**

Important:

For the **Unsteady Statistics...** category to appear in the postprocessing dialog boxes, make sure that **Data Sampling for Time Statistics** is enabled in the **Run Calculation** task page, and that you have performed the calculation.

26.10.2. Displaying Velocity Vectors

For mixture and Eulerian calculations, it is possible to display velocity vectors for the individual phases using the **Vectors** dialog box.

Results → Graphics → Vectors → Edit...

To display the velocity of a particular phase, select **Velocity** in the **Vectors of** drop-down list, and then select the desired phase in the **Phase** drop-down list. You can also choose **Relative Velocity** to display the phase velocity relative to a moving reference frame. To display the mixture velocity $\vec{v}_m$ (relevant for mixture model calculations only), select **Velocity** (or **Relative Velocity** for the mixture velocity relative to a moving reference frame), and **mixture** as the **Phase**. Note that you can color vectors by values of any available variable, for any phase you defined. To do so, make the appropriate selections in the **Color by** and following **Phase** drop-down lists.

26.10.3. Reporting Fluxes

When you use the **Flux Reports** dialog box to compute fluxes through boundaries, you will be able to specify whether the report is for the mixture or for an individual phase.

Results → Reports → Fluxes → Edit...

Select **mixture** in the **Phase** drop-down list at the bottom of the dialog box to report fluxes for the mixture, or select the name of a phase to report fluxes just for that phase.

When mass transfer is included in multiphase simulations, Ansys Fluent by default uses the following sensible enthalpy definition for the reporting of the enthalpies and total heat transfer rate:

$$H(T) = H(T_{ref}) + \int_{T_{ref}}^{T} C_p(T)\,dT$$

where T is the phase temperature for the Eulerian multiphase model or the mixture temperature for all other multiphase models, T_{ref} is the reference temperature, $H(T_{ref})$ is the formation enthalpy at T_{ref}, and Cp is the specific heat. The values for $H(T_{ref})$, Cp, and T_{ref} are those you specified in the **Create/Edit Materials** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The mixture enthalpy is calculated as a mass-weighted average of the phase enthalpies.

Note:

- When the Boiling model is used as a mass transfer mechanism, the Ansys Fluent post-processing tools may report the heat transfer rate imbalance that is equal to boiling heat flux (in the absence of any external heat source). This is not an indicator of any problem because the reported imbalance is due to explicit handling of boiling heat fluxes.
- When using the alternative formulation for modeling energy sources due to mass transfer, you can refer to Alternative Modeling of Energy Sources (p. 2943) for heat flux reporting options and sensible enthalpy definition.

26.10.4. Reporting Forces on Walls

For Eulerian calculations, when you use the **Force Reports** dialog box to compute forces or moments on wall boundaries, you will be able to specify the individual phase for which you want to compute the forces.

Results → Reports → Forces Edit...

Select the name of the desired phase in the **Phase** drop-down list on the left side of the dialog box.

26.10.5. Reporting Flow Rates

You can obtain a report of mass flow rate for each phase (and the mixture) through each flow boundary using the `report/fluxes/mass-flow` text command:

`report` → `fluxes` → `mass-flow`

When you specify the phase of interest (the mixture or an individual phase), Ansys Fluent will give you the option to list each zone, followed by a summary of the mass flow rate through that zone for the specified phase, or will summarize the mass flow rate for all zones. An example is shown below, demonstrating how to list the mass flow rate for all zones.

```
/report/fluxes> mf
(mixture water air)
domain id/name [mixture] air
all boundary/interior zones [yes]
Write to File? [no]

                           air
                Mass Flow Rate               (kg/s)
------------------------------ --------------------
            spiral-press-outlet           -1.2330244
                pressure-outlet           -9.7560663
               spiral-vel-inlet            0.6150589
                          walls                    0
                 velocity-inlet            4.9132133
```

```
               --------------- --------------------
                           Net           -5.4608185
```

Note:

The mass flow rate is calculated through the simulation domain or the specified zones. However, if you specify only the interior zone(s), the net mass flow rate may show as 0, because these internal boundaries are skipped when Ansys Fluent computes the net influx and outflux.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 27: Modeling Solidification and Melting

This chapter describes how you can model solidification and melting in Ansys Fluent. For information about the theory behind the model, see Solidification and Melting in the Theory Guide. Information about using the model is organized into the following sections:

27.1. Setup Procedure

27.2. Procedures for Modeling Continuous Casting

27.3. Modeling Thermal and Solutal Buoyancy

27.4. Solution Procedure

27.5. Postprocessing

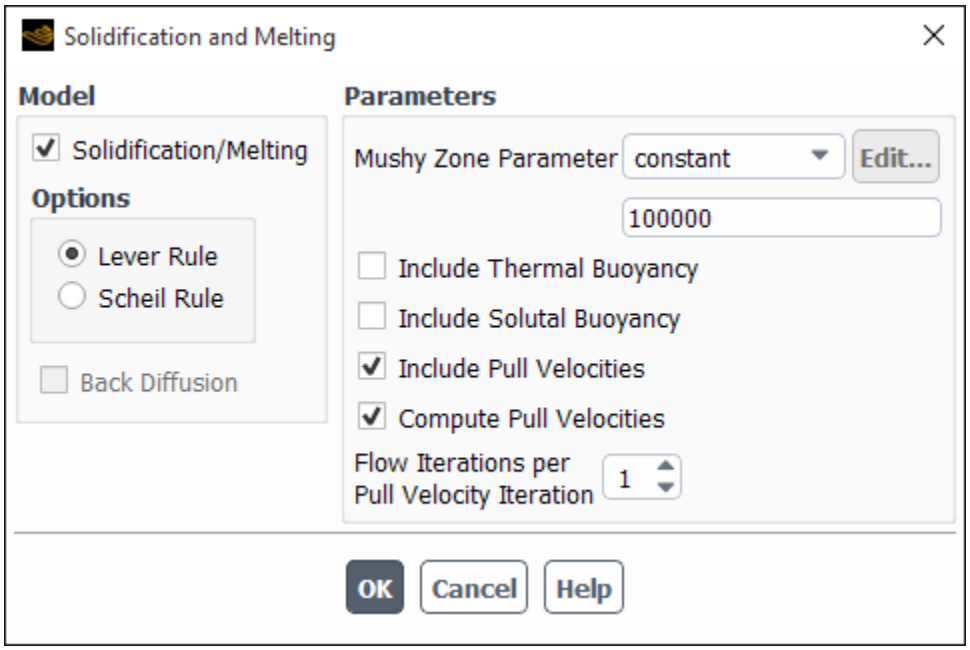
27.1. Setup Procedure

The procedure for setting up a solidification/melting problem is described below. (Note that this procedure includes only those steps necessary for the solidification/melting model itself; you will need to set up other models, boundary conditions, and so on, as usual.)

1. To use the solidification/melting model, enable the **Solidification/Melting** option in the **Solidification and Melting** dialog box (Figure 27.1: The Solidification and Melting Dialog Box (p. 3219)).

   Setup → **Models** → **Solidification & Melting** **Edit...**

   Figure 27.1: The Solidification and Melting Dialog Box



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Fluent will automatically enable the energy equation, so you do not have to visit the **Energy** dialog box before turning on the solidification/melting model.

2. Under **Parameters**, specify the value of the **Mushy Zone Parameter** (A_{mush} in Equation 15.6) as a **constant**, or as a **user-defined** function. Refer to `DEFINE_SOLIDIFICATION_PARAMS` in the Fluent Customization Manual for detailed information about the user-defined function.

   Values between 10^4 and 10^7 are recommended for most computations. The higher the value of the **Mushy Zone Parameter**, the steeper the damping curve becomes, and the faster the velocity drops to zero as the material solidifies. Very large values may cause the solution to oscillate as control volumes alternately solidify and melt with minor perturbations in liquid volume fraction.

3. If you want to include the pull velocity in your simulation (as described in Momentum Equations and Pull Velocity for Continuous Casting in the Theory Guide), enable the **Include Pull Velocities** option under **Parameters**.

4. If you are including pull velocities and you want Ansys Fluent to compute them (using Equation 15.22) based on the specified velocity boundary conditions, as described in Pull Velocity for Continuous Casting in the Theory Guide, enable the **Compute Pull Velocities** option and specify the number of **Flow Iterations Per Pull Velocity Iteration**.

   Important:

   It is not necessary to have Ansys Fluent compute the pull velocities. See Procedures for Modeling Continuous Casting (p. 3222) for information about other approaches.

   The default value of 1 for the **Flow Iterations Per Pull Velocity Iteration** indicates that the pull velocity equations will be solved after each iteration of the solver. If you increase this value, the pull velocity equations will be solved less frequently. You may want to increase the number of **Flow Iterations Per Pull Velocity Iteration** if the liquid fraction equation is almost converged (that is, the position of the liquid-solid interface is not changing very much). This will speed up the calculation, although the residuals may jump when the pull velocities are updated.

5. Under **Options**, select either **Lever Rule** or **Scheil Rule**. See Species Equations in the Theory Guide for details.

   Important:

   The **Lever Rule** and **Scheil Rule** options are available only when **Species Transport** is enabled in the **Species Model** dialog box.

6. If you select **Scheil Rule**, then you can enable **Back Diffusion**. Enter either a **constant** or a **user-defined** function to specify the value of the **Back Diffusion Parameter** (γ in Equation 15.19). Refer to `DEFINE_SOLIDIFICATION_PARAMS` in the Fluent Customization Manual for detailed information about the user-defined function. Note that the value for the **Back Diffusion Parameter** must be between 0 and 1.

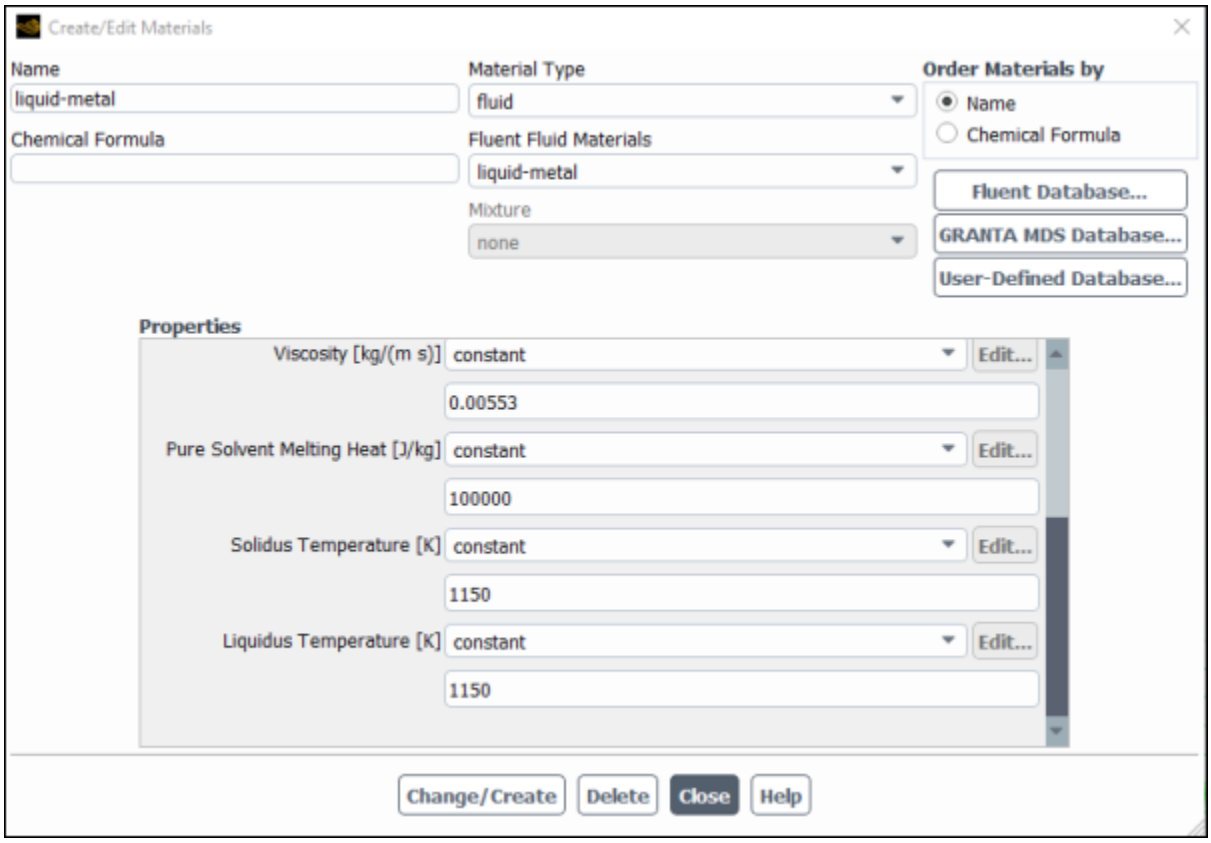
7. In the Create/Edit Materials Dialog Box (p. 4885) (Figure 27.2: The Create/Edit Materials Dialog Box for Melting and Solidification (p. 3221)), specify the **Pure Solvent Melting Heat** (L in Equation 15.4),

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solidus Temperature ($T_{solidus}$ in Equation 15.3), and **Liquidus Temperature** ($T_{liquidus}$ in Equation 15.3) for the material being used in your model.

Setup → **Materials**

Figure 27.2: The Create/Edit Materials Dialog Box for Melting and Solidification



Important:

For solidification to occur, **Pure Solvent Melting Heat** must be set to a positive non-zero value.

If you are solving for species transport, you need to specify properties for the mixture, including the method by which the **Solidus Temperature** and the **Liquidus Temperature** are calculated. The default method is the **mixing-law** (Equation 15.8 and Equation 15.9 in the Theory Guide), in which the solidus temperature and the liquidus temperature are calculated from the parameters provided for each solute (such as the slope of the liquidus line or partition coefficient). However, a **user-defined** function of type `DEFINE_PROPERTY` can be used to specify both of these temperatures. See the Fluent Customization Manual for examples of `DEFINE_PROPERTY`.

Important:

It is highly recommended that you use the same method for specifying the **Solidus Temperature** and the **Liquidus Temperature**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When defining the mixture, you will also specify the **Mass Diffusivity** ($D_{i,m,\text{liq}}$ in Equation 15.15 and Equation 15.18) and the **Eutectic Temperature** (T_{Eut} in Equation 15.10), as well as the **Pure Solvent Melting Heat** (L in Equation 15.4) and the **Pure Solvent Melting Temperature** (T_{melt} in Equation 15.8 and Equation 15.9). Note that the solvent is the last species listed under **Selected Species** in the **Species** dialog box.

For each solute, you have to specify the **Slope of Liquidus Line** (m_i in Equation 15.8 and Equation 15.9 in the Theory Guide) with respect to the concentration of the solute, the **Partition Coefficient** (K_i in Equation 15.8), the **Eutectic Mass Fraction** ($Y_{i,\text{Eut}}$ in Equation 15.10), and, if **Lever Rule** is selected in the **Solidification and Melting** dialog box, the coefficient for **Diffusion in Solid** ($D_{i,m,\text{sol}}$ in Equation 15.15). It is not necessary to specify m_i, K_i, $Y_{i,\text{Eut}}$, and $D_{i,m,\text{sol}}$ for the solvent.

8. Set the boundary conditions.

   Setup → **Boundary Conditions**

   In addition to the usual boundary conditions, consider the following:

   - If you want to account for the presence of an air gap between a wall and an adjacent solidified region (as described in Contact Resistance at Walls in the Theory Guide), specify a nonzero value, a profile, or a user-defined function for **Contact Resistance** (R_c in Equation 15.23) under **Thermal Conditions** in the Wall Dialog Box (p. 5085).
   - If you want to specify the gradient of the surface tension with respect to the temperature at a wall boundary, you can use the **Marangoni Stress** option for the wall **Shear Condition**. See Marangoni Stress (p. 1466) for details.
   - If you want Ansys Fluent to compute the pull velocities during the calculation, note how your specified velocity conditions are used in this calculation (see Pull Velocity for Continuous Casting in the Theory Guide).

Procedures for Modeling Continuous Casting (p. 3222) contains additional information about modeling continuous casting. See Solution Procedure (p. 3224) and Postprocessing (p. 3225) for information about solving a solidification/melting model and postprocessing the results.

27.2. Procedures for Modeling Continuous Casting

As described in Momentum Equations and Pull Velocity for Continuous Casting in the Theory Guide, you can include the pull velocities in your solidification/melting calculation to model continuous casting. There are three approaches to modeling continuous casting in Ansys Fluent:

- Specify constant or variable pull velocities.

  To use this approach (the default), do not enable the **Compute Pull Velocities** option.

  If you use this approach, you will need to patch constant values or custom field functions for the pull velocities, after you initialize the solution.

  Solution → **Initialization** → **Patch...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

See Patching Values in Selected Cells (p. 3630) for details about patching values. Note that it is acceptable to patch values for the pull velocities in the entire domain, because the patched values will be used only if the liquid fraction, β, is less than 1.

- Have Ansys Fluent compute the pull velocities (using Equation 15.22) during the calculation, based on the specified velocity boundary conditions.

  To use this approach, enable the **Compute Pull Velocities** option. This method is computationally expensive, and is recommended only if the pull velocities are strongly dependent on the location of the liquid-solid interface.

  If you have Ansys Fluent compute the pull velocities, then there are no additional inputs or setup procedures beyond those presented in Setup Procedure (p. 3219).

- Have Ansys Fluent compute the pull velocities just once, and then use those values for the remainder of the calculation.

  To use this approach, perform one iteration with Ansys Fluent computing the pull velocities, and then turn off the **Compute Pull Velocities** option and continue the calculation. For the remainder of the calculation, Ansys Fluent will use the values computed for the pull velocities at the first iteration.

27.3. Modeling Thermal and Solutal Buoyancy

When the effects of thermal and solutal buoyancy are present, a flow can be induced inside the domain due to the effect of gravity on the variable density of the medium. In the case of multi-component solidification problems, the density variation takes place due to temperature changes and also due to species concentration gradients near the liquid-solid interface. The flow due to buoyancy with solidification and melting can be modeled in Ansys Fluent using the thermal and solutal buoyancy options.

For more information on the theory behind buoyancy induced flow in solidification and melting problems, see Thermal and Solutal Buoyancy in the Theory Guide.

The procedure for setting up a solidification/melting problem is described in Setup Procedure (p. 3219). To include thermal and solutal buoyancy effects, perform the following steps:

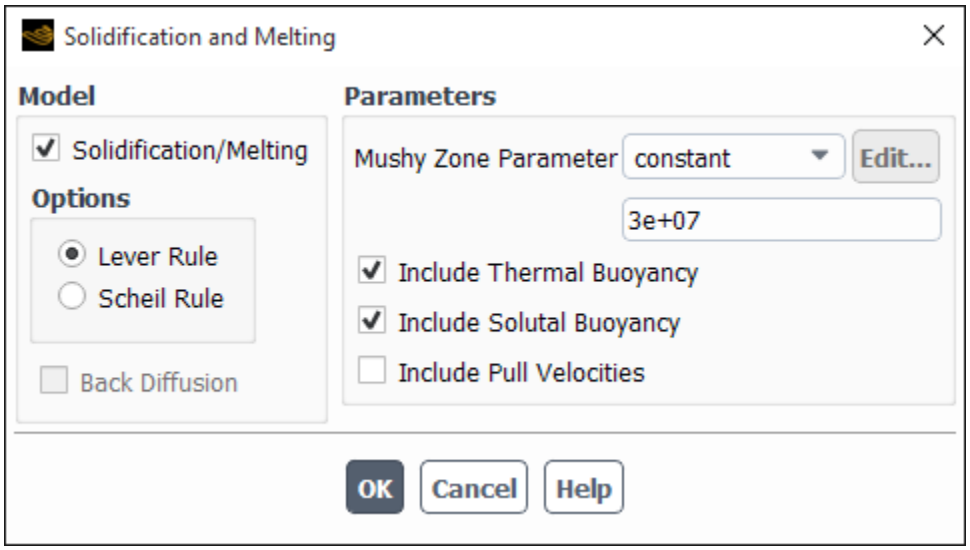
1. Enable the **Include Thermal Buoyancy** and **Include Solutal Buoyancy** options in the **Solidification and Melting** dialog box (Figure 27.3: The Solidification and Melting Dialog Box (p. 3224)).

   Note:

   The **Include Thermal Buoyancy** and **Include Solutal Buoyancy** options are available only when solidification is modeled with species transport.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 27.3: The Solidification and Melting Dialog Box



2. Define the operating conditions and properties for modeling thermal buoyancy as described in Buoyancy-Driven Flows and Natural Convection (p. 1387).

3. To model solutal buoyancy, specify a value for the **Solutal Expansion Coefficient** for all the species except the last one in the mixture in the **Create/Edit Materials** dialog box.

Note:

By default, the eutectic mass fraction of the solute is used as the reference species mass fraction of the solute for the calculation of the body force due to solutal buoyancy. Therefore, no additional input is required. However, in certain applications, it is not always desirable to use the default values of the reference mass fraction. For such cases, the solute mass fraction values can be entered through text user interface as follows:

```
 define/models/solidification-melting? yes
Include Thermal Buoyancy? yes
Include Solutal Buoyancy? yes
Use reference mass fraction of solutes? yes
Reference mass fraction of the species-i "value"
```

27.4. Solution Procedure

Before solving the coupled fluid flow and heat transfer problem, you may want to patch an initial temperature or solve the steady conduction problem as an initial condition. The coupled problem can then be solved as either steady or transient. Because of the nonlinear nature of these problems, however, in most cases a transient solution approach is preferred.

You can specify the under-relaxation factor applied to the liquid fraction equation in the Solution Controls Task Page (p. 5163).

Solution → Controls

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Specify the desired value in the **Liquid Fraction Update** field under **Under-Relaxation Factors**. This sets the value of α_β in the following equation for updating the liquid fraction from one iteration (n) to the next ($n+1$):

$$\beta_{n+1}=\beta_n+\alpha_\beta\Delta\beta \tag{27.1}$$

where $\Delta\beta$ is the predicted change in liquid fraction.

In many cases, there is no need to change the default value of α_β. If, however, there are convergence difficulties, reducing the value may improve the solution convergence. Convergence difficulties can be expected in steady-state calculations, continuous casting simulations, simulations involving multicomponent solidification, and simulations where a large value of the mushy zone constant is used.

27.5. Postprocessing

For solidification/melting calculations, you can generate graphical plots or alphanumeric reports of the following items depending on which other models are enabled in the simulation. These quantities are available in the **Solidification/Melting...** category of the variable selection drop-down list that appears in postprocessing dialog boxes:

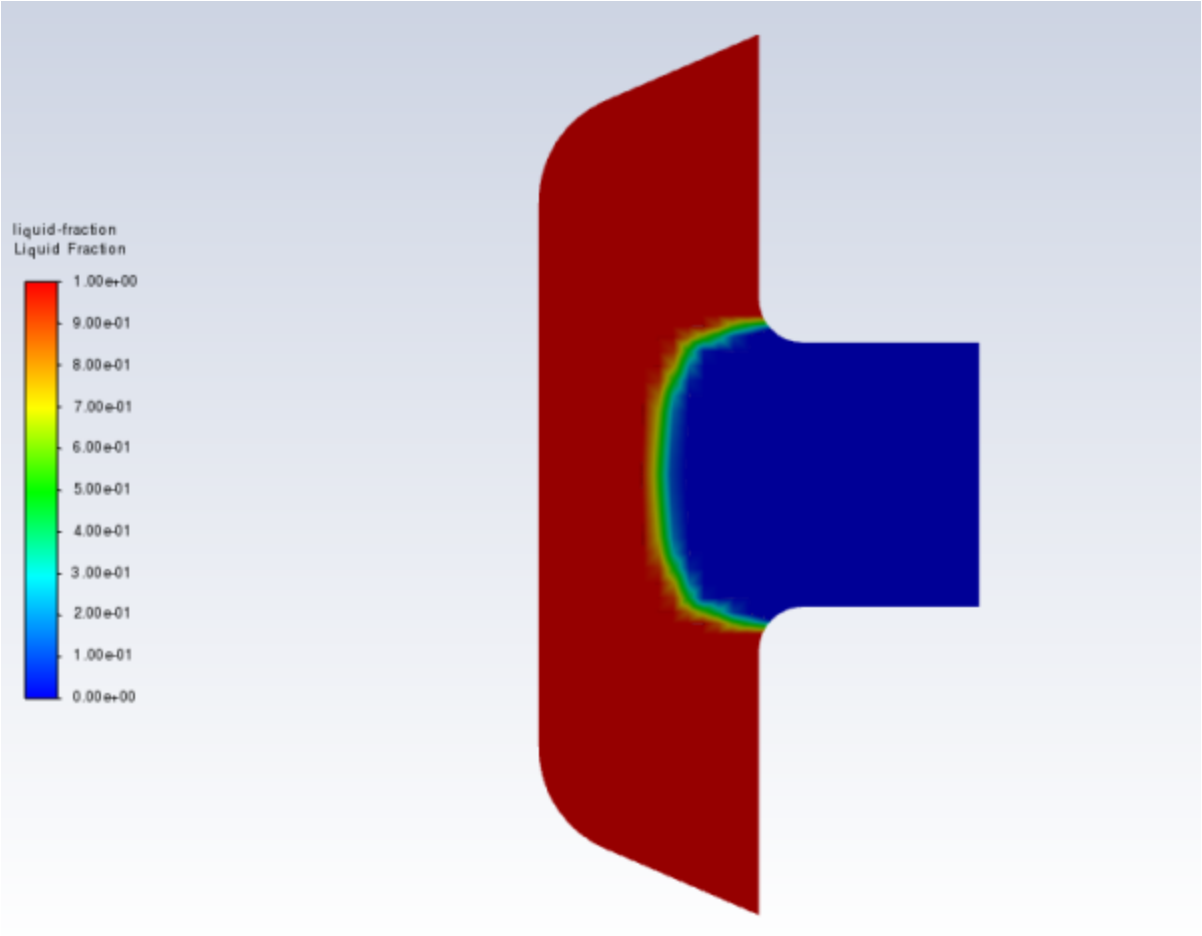
- **Liquid Fraction**
- **Contact Resistivity**
- **Pull Velocity** (**X**, **Y**, **Z**, **Axial**, **Radial**, and **Swirl** components)
- **Liquidus Temperature**
- **Solidus Temperature**

The **Liquid Fraction** and **Contact Resistivity** solution variables are available for all solidification/melting simulations. The **Pull Velocity** components are available only if you are including pull velocities (either computed or specified) in the simulation. **Liquidus Temperature** and **Solidus Temperature** are available only if the **Species** model is set up to perform a multi-component solidification/melting simulation. See Field Function Definitions (p. 4167) for a complete list of field functions and their definitions. Displaying Graphics (p. 3869) and Reporting Alphanumeric Data (p. 4097) explain how to generate graphics displays and reports of data.

Figure 27.4: Liquid Fraction Contours for Continuous Crystal Growth (p. 3226) shows filled contours of liquid fraction for a continuous crystal growth simulation.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 27.4: Liquid Fraction Contours for Continuous Crystal Growth



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 28: Modeling Fluid-Structure Interaction (FSI) Within Fluent

This chapter provides details about how to model and simulate one-way and two-way fluid-structure interaction (FSI) completely within a Fluent session. This is referred to as "intrinsic FSI", as all of the structural calculations are performed by Fluent; note that you can also use extrinsic solvers for FSI problems, incorporating the structural data into your Fluent fluid simulation through mapping or system coupling, as described in Mapping Data for Fluid-Structure Interaction (FSI) Applications (p. 1013) and Performing System Coupling Simulations Using Fluent (p. 4647), respectively. The theoretical aspects of intrinsic FSI calculations are presented in The Structural Model for Intrinsic Fluid-Structure Interaction (FSI) in the Theory Guide. The following sections provide details on running intrinsic FSI simulations:

28.1. Overview and Limitations

28.2. Setting Up an Intrinsic Fluid-Structure Interaction (FSI) Simulation

28.1. Overview and Limitations

Intrinsic fluid-structure interaction (FSI) capabilities allow you to simulate either one-way or two-way FSI problems completely within a Fluent session. One-way FSI problems assume that only the fluid side will have an impact on the solid side through the fluid force that acts on the structure. Since there is no feedback information from the solid side, for one-way FSI simulations you can either compute the fluid flow and structural deformation simultaneously, or compute the structural side independently after the fluid simulation is completed. However, if the solid domain influences the fluid side of the FSI simulation, then you must model it as a two-way FSI simulation by enabling mesh motion for the fluid domain through a dynamic mesh.

The solution of an FSI problem requires the solution of both the fluid flow problem as well as the structural problem. The solution of the structural equations is limited by following:

- The structural model only supports the following cell types: in 2D, quadrilateral and/or triangular cell types; in 3D, hexahedral, tetrahedral, wedge, and/or pyramid cell types. The structural model does not support polyhedral cells. This applies only to the solid zones.

  Note:

  If you use the meshing mode of Fluent to generate a hexcore mesh, the resulting mesh will not be suitable for the structural model, as such meshes contain small numbers of polyhedral cells.

- You are not allowed to replace the mesh once an FSI solution has been initialized and/or started. This is due to the fact that the new mesh will be partitioned and the subsequent data migration is not supported in FEA cases. Re-partitioning fluid zones is allowed, however, since it does not affect solid zone partitioning.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The fluid and solid zones must be separated by two-sided walls (that is, wall / wall-shadow pairs).
- At least one solid zone must be present in the domain in order to enable the structural model.
- The following dynamic mesh options are not supported for intrinsic FSI problems:
  - **In-Cylinder**
  - **Six DOF**
  - **Contact Detection**
- The **Intrinsic FSI** type can only be selected in the **Dynamic Mesh Zones** dialog box for the side of a two-sided wall (that is, the wall or wall-shadow) that is immediately adjacent to the fluid cell zone (as indicated by the **Adjacent Cell Zone** field in the **Wall** dialog box).
- For a wall that is adjacent to a solid zone or between a solid and fluid zone:
  - The only user-defined functions (UDFs) that are supported for displacement and force are `DEFINE_SOURCE_FE`, `DEFINE_WALL_NODAL_DISP` and `DEFINE_WALL_NODAL_FORCE`, respectively (as described in `DEFINE_WALL_NODAL_DISP` and `DEFINE_WALL_NODAL_FORCE`). All other boundary condition profiles or UDFs (such as `DEFINE_PROFILE`) will produce an error message.
  - The **Shell Conduction** option is not supported.
- The structural model is **not** compatible with:
  - mesh adaption
  - the mesh morpher/optimizer
  - British or centimeter-gram-second (CGS) units
  - FMG initialization
  - adaptive time stepping
- The structural model is not available when running Fluent under Ansys Workbench.
- The **Linear Elasticity** structural model is only appropriate when the stress loading does not exceed the yield strength of the solid material.
- The **Linear Elasticity** structural model is compatible with the 2D **Axisymmetric** or **Axisymmetric Swirl** option.
- Postprocessing variables related to structures and the Structural Model are only available on solid cell zones. Therefore, any computations made on variables available under **Structure...** should only include solid zones to ensure they are evaluated correctly.
- For solid structural solutions, there is no special treatment for periodic boundary conditions. The structural solver will treat such boundary as stress free.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

28.2. Setting Up an Intrinsic Fluid-Structure Interaction (FSI) Simulation

The procedure for setting up an intrinsic FSI simulation is described below. Note that only steps that are pertinent to the structural model are listed here. You will also need to define the other settings (such as other material properties, boundary conditions, other models) as usual. For information about inputs related to other Ansys Fluent models that you are using in conjunction with the intrinsic FSI simulation, refer to the appropriate sections for those models.

Important:

Double Precision is recommended for all intrinsic FSI simulations.

---

1. Define the settings in the **General** task page.

   Setup → General

   a. Define the time dependence of the simulation: one-way FSI problems can be **Steady** or **Transient**, whereas two-way FSI problems must be **Transient**. Note that modeling two-way FSI is only necessary when the deformation of the solid zone is significant enough to affect the fluid flow.

   b. You can enable the **Gravity** option and define the coordinates for the acceleration if you would like the structural model to account for gravitational forces when calculating the deformation of the solid cell zone.

2. Define a solid zone.

   Setup → Cell Zone Conditions

   Note that you can enable the **Frame Motion** option in the **Solid** dialog box and define a rotational velocity for the reference frame if you would like the structural model to account for rotational forces when calculating the deformation of the solid cell zone. For details, see Defining Zone Motion (p. 1328) and Modeling Flows with Moving Reference Frames (p. 1769).

3. For a two-way FSI problem, it is recommended that you run the calculation at this point to obtain a converged solution for the fluid, and then proceed with the following steps to set up and run the structural model.

4. In the **Structural Model** dialog box, select a variation of the structural model.

   - The **Linear Elasticity** model enables structural calculations for the solid cell zone such that the internal load is linearly proportional to the nodal displacement, and the structural stiffness matrix remains constant.

   - The **Nonlinear Elasticity** model allows you to model geometrically nonlinear elasticity, in order to simulate large deformation in solid cell zones.

   - The structural model allows you to model linear or nonlinear thermo-elasticity, in order to simulate the effects of thermal load on solid structure deformation. To set up such a simulation, choose the **Thermal Effects** option in the **Structural Model** dialog box.

     Thermoelasticity also can be enabled via the text user interface using the following command:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

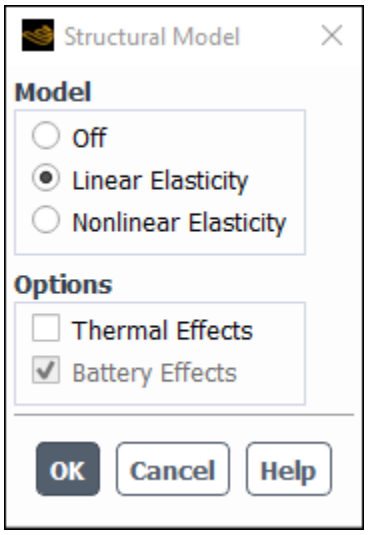
`define` → `models` → `structure` → `controls` → `thermal-effects?`

Note that thermal effects are only available for the linear or nonlinear structural model if the energy equation is enabled.

- For Newman's P2D battery cases that include swelling effects, you can model linear or non-linear elasticity (see Inputs for the Newman's P2D Model (p. 3342)). After selecting an elasticity model in the **Structural Model** dialog box, the **Battery Effects** option is enabled automatically and cannot be disabled.

Setup → **Models** → **Structure** **Edit...**

Figure 28.1: The Structural Model Dialog Box



5. Make sure that the material used in the solid cell zone is properly defined, by setting the **Youngs Modulus** and **Poisson Ratio** fields in the **Properties** group box of the **Create/Edit Materials** dialog box.

   Setup → **Materials**

   Thermo-elasticity requires you to set up two additional parameters per solid material used:

   - Starting Temperature [K]

     The starting temperature specified in the **Materials** dialog box will be constant by default. If your problem requires a non-constant starting temperature for thermo-elastic analysis, the following procedure is recommended:

     a. First perform an initial calculation to solve for the temperature field within the solid zones of your problem (for more information see Modeling Conductive and Convective Heat Transfer (p. 2163)).

     b. Re-assign the temperature field calculated in the previous step as the non-constant starting temperature, by entering the following TUI command in the console:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
/define/models/structure/expert/starting-t-re-initialization?
```

When prompted with `Starting T-field re-initialization method (0 1) [0]`, enter `1`. This will perform re-initialization of the new starting temperature field for structural analysis.

The new starting temperature field can be visualized by displaying contours of **Structure** and **Starting Structure Temperature** within the Contours Dialog Box (p. 5412).

Note that after re-initializing the starting temperature field, the starting temperature will no longer be available within the **Materials** dialog box.

- Thermal Expansion [1/K]

Note that the **Youngs Modulus**, **Poisson Ratio**, and **Thermal Expansion** can also be defined using a user-defined function (UDF) by interpreting or compiling a suitable UDF. For details, see DEFINE_PROPERTY UDFs.

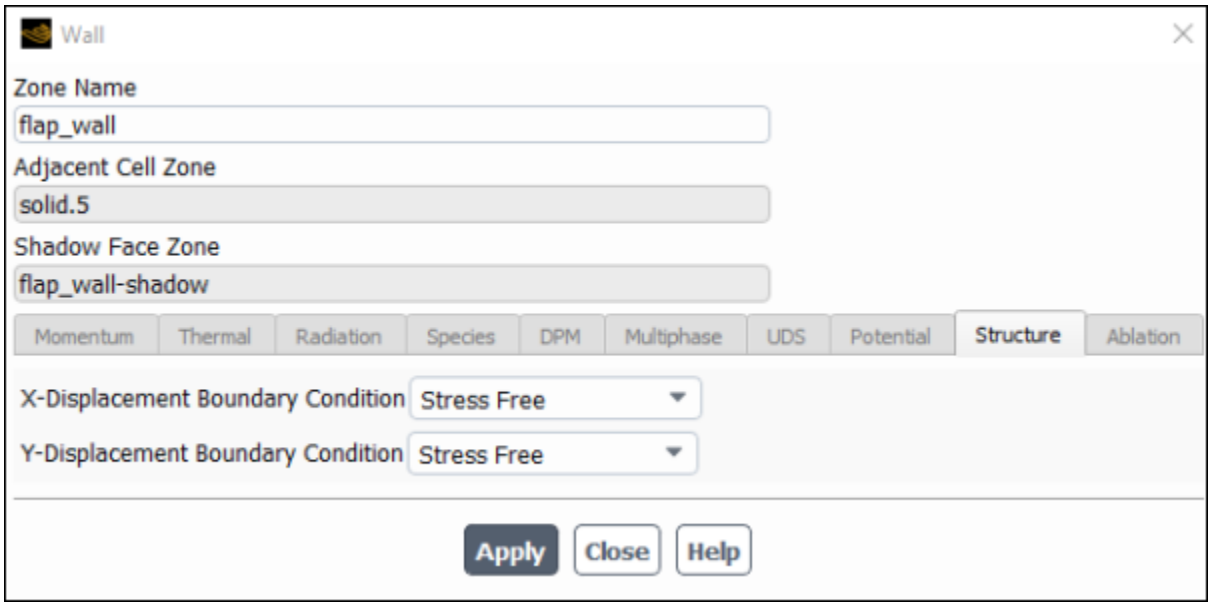
6. If you plan to define volumetric body force as a source term for a solid cell zone using a user-defined function (UDF), interpret or compile a suitable UDF. For details, see DEFINE_SOURCE_FE.

7. If you plan to define the displacement or force applied to the nodes of a wall adjacent to the solid zone using a user-defined function (UDF), interpret or compile a suitable UDF. For details, see DEFINE_WALL_NODAL_DISP or DEFINE_WALL_NODAL_FORCE, respectively.

8. For every wall that is immediately adjacent to the solid zone, define the displacement boundary condition settings in the **Structure** tab of the **Wall** dialog box.

Setup → Boundary Conditions

Figure 28.2: The Structure Tab of the Wall Dialog Box



The walls will be one-sided when they are only adjacent to a solid cell zone, and will be two-sided (that is, a wall / wall-shadow pair) when they are adjacent to both a solid and a fluid cell zone. Note

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

that for two-sided walls, the **Structure** tab is only available for one side of the wall / wall-shadow pair; which one will depend on how the mesh was set up.

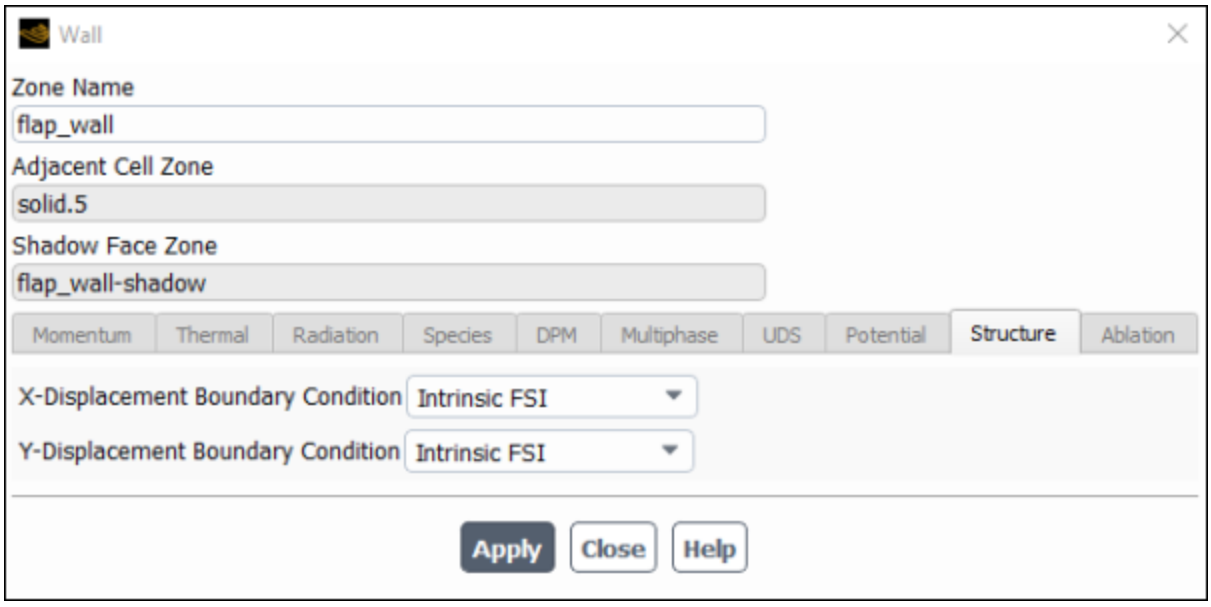
You must select one of the following from the **X-**, **Y-**, and (for 3D cases) **Z-Displacement Boundary Condition** drop-down lists, in order to define how the displacement is calculated for the nodes in that particular direction:

- **Stress Free** specifies that the displacement is not affected by stress loads from the fluid flow.
- **Node X-**, **Node Y-**, and (for 3D cases) **Node Z-Force** specifies that the displacement results from a specified force applied on the nodes, which is defined using the **Node X-**, **Node Y-**, and (for 3D cases) **Node Z-Force** field.
- **Node X-**, **Node Y-**, and (for 3D cases) **Node Z-Displacement** applies a specified displacement on the nodes, which is defined using the **X-**, **Y-**, and (for 3D cases) **Z-Displacement** field.
- **Face Pressure** specifies that the displacement results from a specified pressure load applied on the faces, which is defined using the **Face Pressure** field.
- **Intrinsic FSI** specifies that the displacement results from pressure loads exerted by the fluid flow on the faces. Note that this selection is only available for a zone that is part of a wall / wall-shadow pair.

Note:

You can specify the nodal force (**Node X-Force**, **Node Y-Force** and **Node Z-Force**), displacement (**Node X-Displacement**, **Node Y-Displacement** and **Node Z-Displacement**) as well as the **Face Pressure** using a constant value, an input parameter, an interpreted or compiled user-defined function (see `DEFINE_WALL_NODAL_DISP` or `DEFINE_WALL_NODAL_FORCE` in the Fluent Customization Manual), or you can use an expression (see Fluent Expressions Language (p. 1037)).

Figure 28.3: The Wall Dialog Box for a Two-Sided Wall



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In a typical Fluent simulation, symmetry boundary conditions are used for boundaries on the solid zone and the same strategy can be applied for intrinsic FSI Fluent simulations. Alternatively, the plane of symmetry, when aligned with Cartesian planes, could be modeled using a wall boundary condition. For example, when the symmetry plane is the XY plane, you should assign the x- and y-displacement to **Stress Free** and then assign the z-displacement to 0

Note:

When solving a structural problem, boundary conditions that allow for structural rigid body motion can lead to an ill-posed problem. That is, if it is at all possible that any motion can lead to a scenario with no stresses, then the problem cannot be solved. Fluent can analyze the setup and will provide a warning when such motion is not prevented by boundary conditions (except when disconnected regions belong to a single solid zone). Such problematic configurations should therefore be avoided.

9. For two-way FSI simulations, define dynamic mesh properties to allow the mesh to handle the deformation of the solid zone. This includes the following:

   - Enable dynamic mesh **Smoothing**, and select either the **Diffusion**, **Linearly Elastic Solid**, or **Radial Basis Function** method in the Mesh Method Settings Dialog Box (p. 5114).
   - For cases with strong FSI coupling (such as when the fluid and solid densities are comparable or for large deformations), enable the **Implicit Update** option in the **Dynamic Mesh** task page and define appropriate settings in the **Implicit Update** tab of the **Options** dialog box. For details, see Setting Dynamic Mesh Modeling Parameters (p. 1814) and Implicit Update Settings (p. 1888).
   - Define an **Intrinsic FSI** dynamic mesh zone for the side of a two-sided wall (that is, the wall or wall-shadow) that is immediately adjacent to the fluid cell zone (as indicated by the **Adjacent Cell Zone** field in the **Wall** dialog box). For further details, see Intrinsic FSI Motion (p. 1917). You should create stationary dynamic mesh zones for all the boundaries that are not deforming.

Setup → Dynamic Mesh

10. For transient simulations, you can make a selection from the **Structure Transient Formulation** drop-down list in the Solution Methods Task Page (p. 5157) to specify the direct time integration method used to solve the finite element semi-discrete equation of motion. You can select from the **Newmark** method (default) or the **Backward Euler** method. For further details about these methods, see Dynamic Structural Systems in the Theory Guide.

Solution → Methods

11. You can revise the following settings if you find that the defaults do not produce satisfactory results:

    - You can specify the solution method used by the linear solver for the structural model calculations. By default, the bi-conjugate gradient stabilized (BCGSTAB) method is used. This method offers a good balance of speed and robustness. If divergence is detected with BCGSTAB, then the generalized minimal residual (GMRES) method will be used for an iteration as a fallback, and you will be informed in the console that a "fe-structure" equation is being stabilized to enhance linear solver robustness. If you find that your residual level / drop is not satisfactory, you can try increasing

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the maximum number of inner iterations from the default of 500 by using the following text command:

`define` → `models` → `structure` → `controls` → `max-iter`

If the BCGSTAB method continues to produce unsatisfactory residuals with a higher number of inner iterations or if you repeatedly see console messages that say the GMRES fallback is being used, you can change the solution to the GMRES method. Note that GMRES is the most robust method, but it is also more demanding in terms of memory usage and solver time. With GMRES, it is recommended that you start with the maximum iterations set to 50. Also available is the conjugate gradient (CG) method: while in some cases it is the least robust method, it can result in faster calculations, as it takes advantage of the symmetry of the matrix used in the linear systems of equations. To change the solution method, use the following text command:

`define` → `models` → `structure` → `controls` → `amg-stabilization`

- For transient simulations, you can revise the numerical damping factor for the structural model calculations (that is, the amplitude decay factor γ in Equation 16.16 in the Theory Guide) by using the following text command:

  `define` → `models` → `structure` → `controls` → `numerical-damping-factor?`

- You can enable an explicit fluid-structure interaction force by using the following text command:

  `define` → `models` → `structure` → `expert` → `explicit-fsi-force?`

- You can enable the inclusion of operating pressure into the fluid-structure interaction force by using the following text command:

  `define` → `models` → `structure` → `expert` → `include-pop-in-fsi-force?`

- You can enable the inclusion of a viscous fluid-structure interaction force by using the following text command:

  `define` → `models` → `structure` → `expert` → `include-viscous-fsi-force?`

- When the deformation of a structure is mainly due to bending, the displacement can be underestimated when a linear interpolation is used for the displacement, especially on rather coarse meshes. This is often referred to as *shear locking*, as the representation of the bending requires the shearing of linear elements which actually absorbs a part of the energy put into the bending.

  The effects of shear locking can be corrected by invoking the enhanced strain element. It is available for quad cells in 2D and hex cells in 3D.

  The enhanced strain element can be enabled in the TUI using the command:

  `define` → `models` → `structure` → `controls` → `enhanced-strain?`

  and by subsequently answering `yes` at the prompt.

- To enable Rayleigh damping for unsteady simulations (Rayleigh Damping), enter the following text user interface (TUI) command:

  `define` → `models` → `structure` → `controls` → `unsteady-damping-rayleigh?`

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following damping coefficients are available for solid materials:

- **Mass Proportional Damping** [1/s]
- **Stiffness Proportional Damping** [s]

12. If necessary, revise the default convergence criteria for the **x- y-**, and (for 3D cases) **z-displacement** residual equations in the Residual Monitors Dialog Box (p. 5599).

Solution → **Monitors** → **Residual** **Edit...**

13. (Optional) Create structural point surfaces at points of interest within the solid zone, which you can use for monitoring and reporting data. Note that for two-way FSI simulations, a structural point surface does not necessarily remain fixed in space, but will continue to represent its original cell as the solid zone moves / deforms.

Results → **Surfaces** **New** → **Structural Point...**

For further details, see Structural Point Surfaces (p. 3825).

To monitor a field variable at such a structural point surface, create a report definition. For example, you could monitor and plot the vertex average of the displacement of the nodes that surround a structural point surface:

Solution → **Report Definitions** **New** → **Surface Report** → **Vertex Average...**

For further details, see Creating Report Definitions (p. 4098).

14. Initialize the solution. Note that for standard initialization, you can specify the initial **X**, **Y**, and (for 3D cases) **Z Displacement** in the Solution Initialization Task Page (p. 5179).

Solution → **Initialization**

Note:

In order to ensure the quality for structural model files, only the Common Fluids Format (CFF) is supported when writing case and data files when the structural model is enabled.

15. (Optional) For two-way FSI simulations, set up solution animations.

You can capture results for the transient simulation as it calculates the solution and then later display how the fluid flow and/or the solid shape (node displacement and/or mesh) changes over time. You should narrow the min-max range, since the contour changes may be small. See Animating Graphics (p. 4051) for more information.

Results → **Animations** → **Scene Animation** **Edit...**

16. Run the calculation.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution → **Run Calculation**

Tip:

For one-way FSI simulations, the calculation time will be faster if you first run the calculation with only the fluid equations enabled in the **Equations** dialog box, and then run it with only the structure equation enabled.

Solution → **Controls** → **Equations...**

For one-way or two-way FSI, it is also recommended that you solve the steady flow problem first, because it allows you to easily discern and resolve any convergence issues that are not related to the fluid-structure interaction.

17. View the results by displaying contours of the following field variables (in the **Structure...** category, also available for displaying vectors):

    - **Total Displacement**
    - **X**, **Y**, and **Z Displacement** for the displacement values
    - **Sigma XX**, **YY**, **XY**, **ZZ**, **YZ**, and **XZ** for the stress tensor values
    - **von Mises Stress** for material failure analysis (being automatically calculated once selected here for postprocessing)

    Results → **Graphics** → **Contours** → **New...**

    Note:

    - You should confirm that the stress loading does not exceed the yield strength of the solid material, as this is an assumption of the **Linear Elasticity** structural model.
    - While contours of the **Structures...** field variables are only displayed in solid cell zones, a few displays do not follow this paradigm: contours based on expressions or custom field functions, minimum/maximum computations, and XY plots.

28.2.1. Using Intrinsic FSI With Non-Conformal Interfaces

In many FSI applications, meshes for solid zones are prepared differently from fluid zones. Typically, the mesh quality for a finite element analysis is not as critical as the mesh quality for a computational

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

fluid dynamics analysis. Since it is common to have a non-conformal mesh at the fluid-solid interface, support for non-conformal interfaces for one-way and two-way FSI calculations is desirable.

Note:

It is assumed that the mesh for the solid side surface can be either quadrilateral or triangular in 3D, and that ANSYS Fluent calculates the intersection facets (that is, the faces imprinted from, and interested with, the other surface) at the non-conformal interface.

Note:

Solid-solid non-conformal interfaces are not supported; only fluid-solid interfaces are supported.

To use non-conformal interfaces in your intrinsic FSI simulation, perform the following steps:

1. Read your mesh into an Ansys Fluent session.
2. Create your mesh interface(s).

   Setup → **Mesh Interfaces** **New...**

3. Enable the **Structural Model**.

   Setup → **Models** → **Structure** **Edit...**

   Choose the appropriate intrinsic FSI structural model (**Linear** or **Nonlinear**)

4. Set up the intrinsic FSI boundary condition and dynamic mesh settings.

   Setup → **Boundary Conditions**

   Setup → **Dynamic Mesh**

   Define intrinsic FSI boundary condition and dynamic mesh settings on those wall boundaries that were automatically created as part of defining the mesh interface(s).

5. Set up the rest of the intrinsic FSI problem as you normally would, as described in Setting Up an Intrinsic Fluid-Structure Interaction (FSI) Simulation (p. 3229).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 29: Modeling Eulerian Wall Films

The Eulerian Wall Film (EWF) model can be used to predict the creation and flow of thin liquid films on the surface of walls. This chapter presents information about the basic functionality of the Eulerian Wall Film (EWF) model. Additional information about the model is provided in the following sections:

29.1. Limitations

29.2. Overview of Using the Eulerian Wall Film Model

29.3. Setting Eulerian Wall Film Model Options

29.4. Setting Eulerian Wall Film Solution Controls

29.5. Setting Eulerian Wall Film Boundary, Initial, and Source Term Conditions

29.6. Coupling of Eulerian Wall Film with the VOF Multiphase Model

29.7. Postprocessing the Eulerian Wall Film

For more information about Eulerian Wall Film model theory, see Eulerian Wall Films in the Theory Guide (p. 1). For more information about setting boundary conditions for liquid films at wall boundaries, see Wall Film Conditions for Walls (p. 1480).

29.1. Limitations

The following limitations exist for the Eulerian Wall Film model:

- The Eulerian Wall Film model is available for 3D geometries only.
- Many models (for example, VOF multiphase flow or radiation) will not interact correctly with the film model without first modifying the boundary conditions using UDFs.
- Wall-film behavior at junctions with baffles (zero-thickness walls) may lead to unphysical results due to the local mesh topology (wall, wall-shadow) and should be used with care, that is, isolated film walls must not share a common edge.
- The wall film can only be single-component, but not multi-component.
- The Eulerian Wall Film model is not compatible with the following models and features:
  - Non-conformal interfaces
  - Overset meshes
  - Mesh operations (such as fuse, zone extrusion, zone type change, zone activation, and others)

    Note that cell/face separation and cell/face merge are compatible with the EWF model. See Overview of Using the Eulerian Wall Film Model (p. 3240) for details.
  - Moving and deforming meshes

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that moving or rotating boundaries are compatible with the EWF model.

– Lagrangian Wall Film (LWF) model

When EWF is enabled in the **Eulerian Wall Film** dialog box, then the **wall-film** option will not appear in the **Discrete Phase BC Type** drop-down list in the **Wall** dialog box (**DPM** tab).

29.2. Overview of Using the Eulerian Wall Film Model

This section describes the typical workflow of using the Eulerian Wall Film (EWF) model to simulate thin liquid films.

The use of the Eulerian Wall Film model involves the following steps:

1. Enable and setup relevant Ansys Fluent models that are compatible with the Eulerian Wall Film model. Ansys Fluent models that are not compatible with the Eulerian Wall Film model are outlined in Limitations (p. 3239).

2. Enable the Eulerian Wall Film model and set the model option as described in Setting Eulerian Wall Film Model Options (p. 3241). Note that certain options are only available when the relevant Ansys Fluent models are enabled.

3. Set the wall film solution controls as described in Setting Eulerian Wall Film Solution Controls (p. 3248).

4. For each wall where you want to model the Eulerian wall film, assign appropriate boundary conditions as described in Setting Eulerian Wall Film Boundary, Initial, and Source Term Conditions (p. 3252).

5. If necessary, solve the flow field first to establish a sound flow field before enabling the wall film solution.

6. Initialize and solve the film flow. Note that the wall film model can be initialised along with the flow field initialisation, or over an existing flow field from the **Eulerian Wall Film** dialog box (**Model Options and Setup** tab) as described in Setting Eulerian Wall Film Model Options (p. 3241).

7. Examine the film solution results. For information about available wall film postprocessing variables, refer to Postprocessing the Eulerian Wall Film (p. 3261).

Performing Mesh and Zone operations during wall film solution process:

Any mesh and zone operations, such as **Make Polyhedra**, **Combine**, and **Separate**, should be performed prior to setting up the Eulerian Wall Film model whenever possible. These operations impact the wall film model in the following ways:

- For the **Make Polyhedra** operation, the Eulerian Wall Film model and all assigned wall film conditions will be disabled before the polyhedral mesh conversion and enabled automatically after the operation is complete. All existing wall film data will be discarded.

- For the cell zone **Merge** and **Separate** operations, the Eulerian Wall Film model and all assigned wall film conditions will be disabled and will require manually enabling afterwards. Warnings will be issued before and during the operations. All existing wall film data will be discarded.

- For the face zone **Merge** and **Separate** operations, the Eulerian Wall Film model will remain enabled. However, assigned wall films on walls involved in the face merge and separation will be disabled and

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

will need to be manually enabled afterwards. The film model must be re-initialised after the face operation is complete. All existing wall film data will be discarded.

Mesh Adaption during wall film solution process:

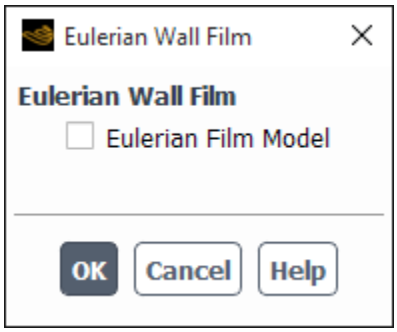
The Eulerian Wall Film model is compatible with the mesh adaption. Wall film data is preserved during the adaption process. However, certain wall film model setups, such the VOF coupling transition criteria, may need to be adjusted to reflect the changes in mesh resolution.

29.3. Setting Eulerian Wall Film Model Options

You can enable the Eulerian Wall Film model by selecting **Eulerian Wall Film** from the **Models** task page.

Setup → Models → Eulerian Wall Film Edit...

*This opens the **Eulerian Wall Film** dialog box.*



Once you open the Eulerian Wall Film Dialog Box (p. 4870), you can select the **Eulerian Wall Film** check box to enable the model so that you can use it in your simulation. Enabling the model expands the dialog box to reveal additional model options and solution controls.

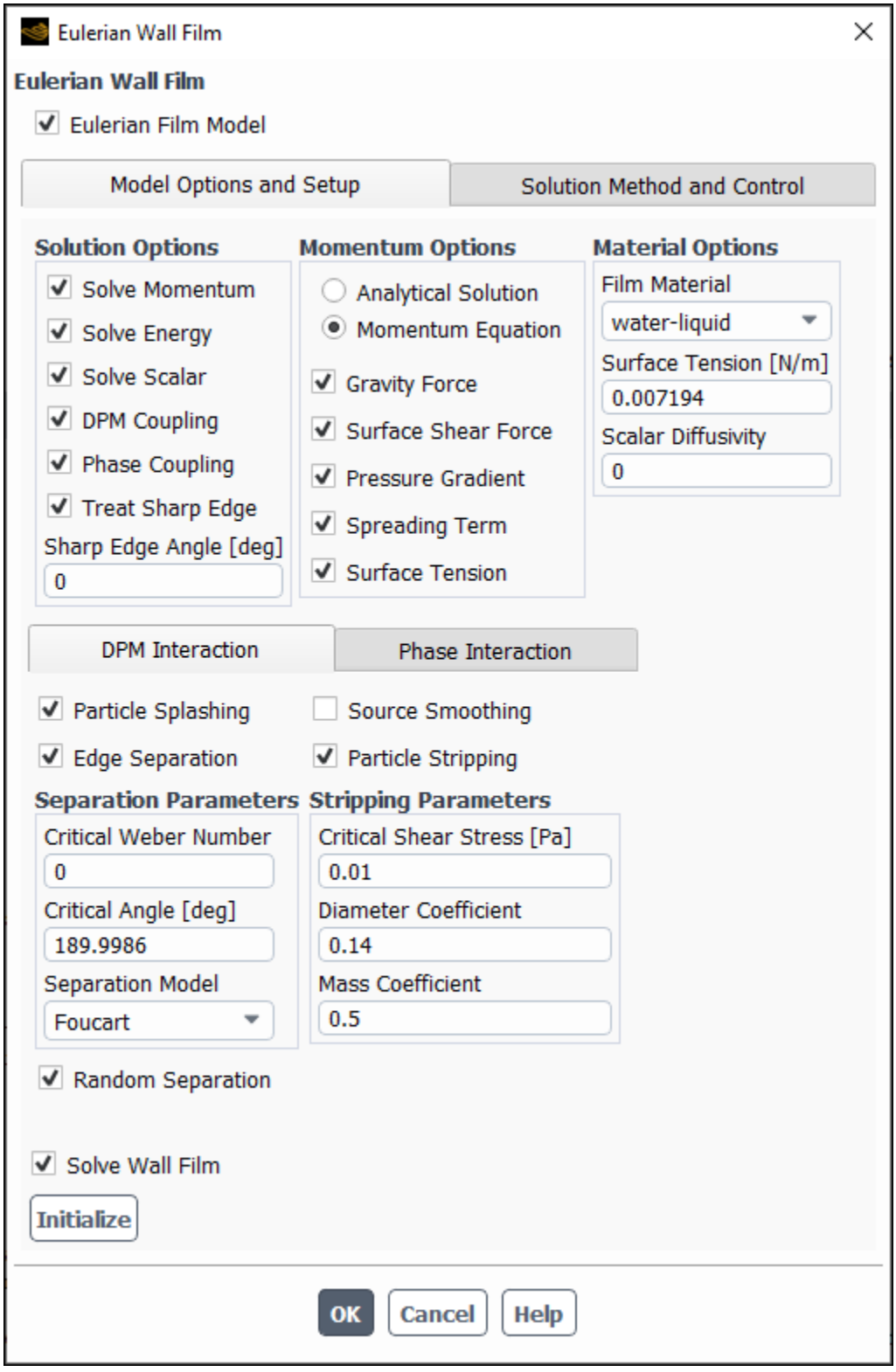
Important:

If you want to modify your mesh in Fluent, you must first disable the Eulerian Wall Film model. Once you save the modified mesh, you must re-enable the Eulerian Wall Film model and re-initialize your case.

You can set general Eulerian Wall Film model options in the **Model Options and Setup** tab of the **Eulerian Wall Film** dialog box. This tab contains controls for specific solution, discrete phase model (DPM), and material options for the Eulerian Wall Film model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 29.1: The Eulerian Wall Film Dialog Box - Model Options and Setup tab (with DPM Interaction)



In the **Model Options and Setup** tab, the following options and controls are available:

- **Solution Options**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solve Momentum

specifies whether film velocity is solved for the wall film. When the option is disabled, film velocity is fixed at the existing values. When the option is enabled, you can choose to solve film velocity using the method specified in the **Momentum Options** group box as further described.

Solve Energy

specifies whether the energy equation (see Equation 17.3 in the *Fluent Theory Guide*) is solved for the wall film or not.

Solve Scalar

specifies whether the passive scalar transport equation Equation 17.38 is solved for the wall film or not. When the **Solve Scalar** option is enabled, you can specify **Scalar Diffusivity** in the **Material Options** group box.

DPM Coupling

enables the interaction of discrete phase particles with the film model (see DPM Collection in the *Fluent Theory Guide*). When this option is enabled, you can adjust the DPM coupling settings in the **DPM Interaction** tab as further described. This option is only available when the DPM model is enabled.

Phase Coupling

allows you to account for phase changes between the film material (liquid) and the gas species (vapor) (see Coupling of Wall Film with Mixture Species Transport in the *Fluent Theory Guide*), compute the secondary phase collection efficiency on a wall surface, and enable the interaction of the EWF model with the VOF multiphase model (see Coupling of Eulerian Wall Film with the VOF Multiphase Model in the *Fluent Theory Guide*). When this option is enabled, you can adjust the phase coupling settings in the **Phase Interaction** tab as further described. This option is only available when the **VOF**, **Eulerian**, or **Mixture** (with **Slip Velocity**) multiphase model or the species model is enabled.

Treat Sharp Edge

accounts for the effects of sharp edges. When this option is enabled, you can specify **Sharp Edge Angle**. Note that when the edge angle is smaller than the **Sharp Edge Angle**, the film wall edge is treated as a boundary edge (namely, the film becomes detached from the wall, rather than bending around the edge and attaching to the wall). If the Discrete Phase Model has been enabled along with the Eulerian Wall Film model, then the film wall edge is treated as a separation edge.

- **Momentum Options**

  When you enable the **Solve Momentum** option in the **Solution Options** group box, you can select the method for solving the film velocity:

  – **Analytical Solution**: Applies film velocity computed from the external flow shear and gravity as described in Obtaining Film Velocity Without Solving the Momentum Equations in the *Fluent Theory Guide*.

  – **Momentum Equation**: Enables the solution of film momentum equation (see Equation 17.2).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For the **Momentum Equation** method, you can select the following terms for inclusion in the calculation:

Gravity Force

is the second term on the right hand side of Equation 17.2, and is responsible for accelerating the film in the direction of gravity component that is parallel to the wall.

Surface Shear Force

is the third term on the right hand side of Equation 17.2, and is responsible for accelerating the film in the direction of the external flow.

Pressure Gradient

is the first term on the right hand side of Equation 17.2, and is the term accelerating the film in the direction opposing the gradient in external pressure. For example, if a film on the surface of a wing is being modeled and there is a high pressure region at the leading edge with low pressure on the top of the wing, this term will tend to move a uniform film towards the low pressure region on the top of the wing.

Spreading Term

is the P_h term on the right hand side of Equation 17.2, and is responsible for spreading. This term will accelerate the flow in the direction opposing the gradient in height, moving the film towards regions of lower thickness. This term becomes available only when both **Pressure Gradient** and **Gravity Term** are selected.

Surface Tension

is the P_σ term on the right hand side of Equation 17.2. This term becomes available only when **Pressure Gradient** is selected.

- **Material Options**

  You can select **Film Material** and also, when the **Solve Momentum** option is enabled in the **Solution Options** group box, you can specify the following parameters:

  – Wall film **Surface Tension**

  – **Scalar Diffusivity** (available with the **Solve Scalar** option only)

  Note:

  – The liquid film can only be a single-component fluid.

  – The type of material for a wall film must be **Fluid**, **Droplet Particle**, or **Inert Particle** (fluid).

    You can define density, viscosity, specific heat, thermal conductivity, vaporization temperature, and vapor saturation pressure of a wall film material or film vapor using one of the following methods:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

→ constant

→ polynomial (including piecewise linear and piecewise polynomial)

→ compressible liquid (for fluid density)

→ user-defined (not available for fluid specific heat)

→ default-method (for film vapor saturation pressure)

- DPM settings in the **DPM Interaction** tab (available when **DPM Coupling** is enabled in the **Solution Options** group box)

  These settings define the interaction between the discrete phase and the film(see DPM Collection in the *Fluent Theory Guide*). You can select:

  – **Particle Splashing**

    When this option is enabled, you can specify splashing options in the **Wall Film** tab of the **Wall** dialog box. For details, see Setting Eulerian Wall Film Boundary, Initial, and Source Term Conditions (p. 3252).

  – **Source Smoothing**

    When this option is enabled, smoothing is applied to the discrete phase impingement source terms to reduce the impact of spotty source distributions. By default, the smoothing level of 2 and the smoothing factor of 0.5 are applied. You can adjust these settings by using the `define/models/eulerian-wallfilm/model-options` text command.

  – **Edge Separation** and specify the following settings under the **Separation Parameters** group box:

    → **Critical Weber Number**

    → **Critical Angle**

    → **Separation Model**

    → **Random Separation**

      When this option is enabled, the locations at which the newly spawned particles are injected (due to film separation) are randomly selected along the edge at which the separation takes place.

    For details about these settings, see Film Separation in the *Fluent Theory Guide*.

  – **Particle Stripping** and specify the following settings under the **Stripping Parameters** group box:

    → **Critical Shear Stress**

    → **Diameter Coefficient**

    → **Mass Coefficient**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For details about these settings, see Film Separation in the *Fluent Theory Guide*.

Note:

The following considerations should be taken into account when the Eulerian Wall Film model is used with the DPM:

– When the **DPM Coupling** option is used with the Eulerian wall film model, at least one injection must be created for wall film formation.

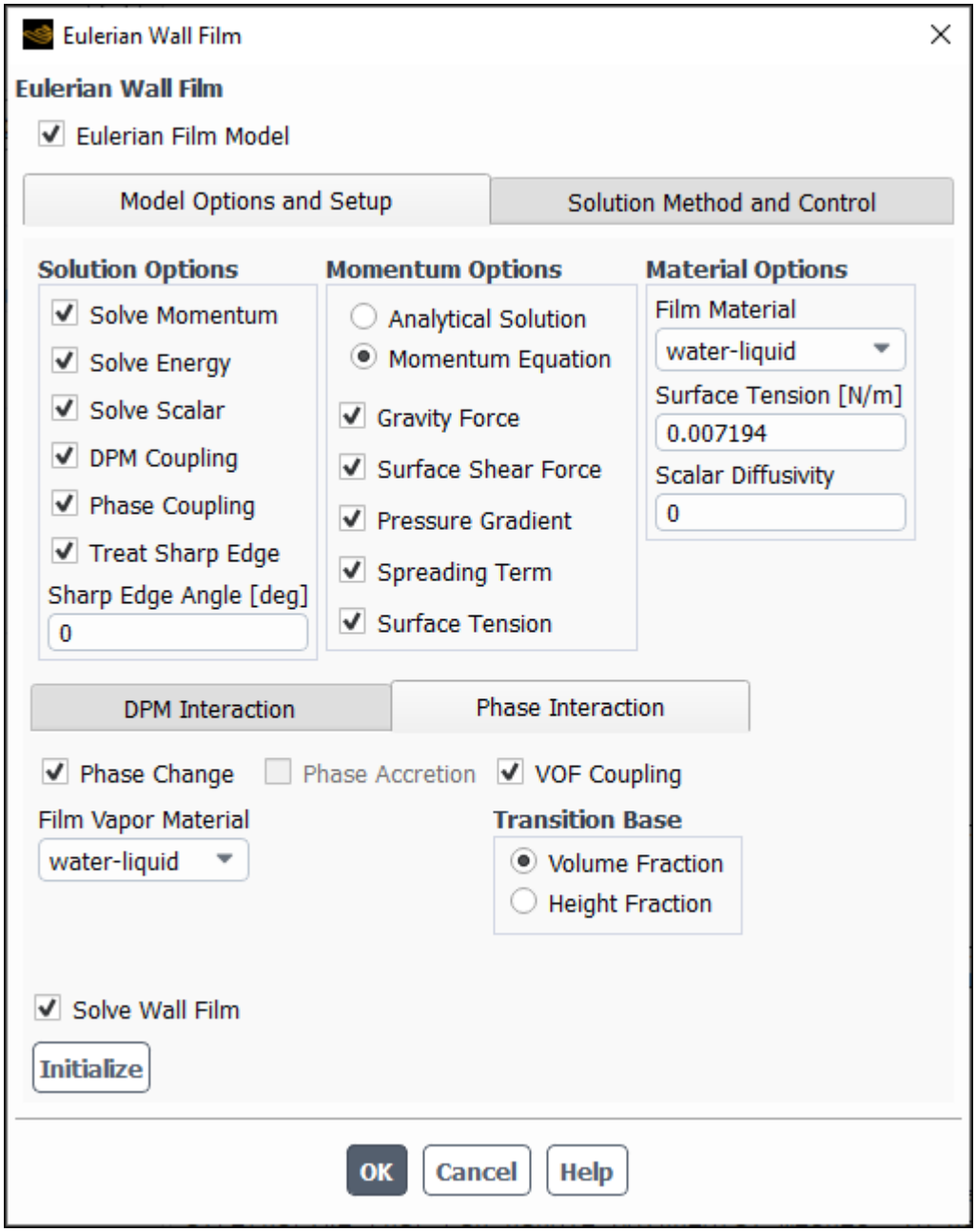
– The injection material and film material must have the same properties. The name of each injection that is used to form the wall film must contain the name of the material assigned to the wall film and an arbitrary suffix separated by a hyphen (-). For example, for a film material with the name 'water-liquid', a valid name for the injection material may be 'water-liquid-particle'.

– When the **Edge Separation** and/or **Particle Stripping** options are enabled, an injection is automatically created with the name 'ewf-strip-sep-injection' and an auto-assigned material called 'film_material_name-pcle' to track the separated and/or stripped particles.

- Phase interactions settings in the **Phase Interaction** tab (available when **Phase Coupling** is enabled in the **Solution Options** group box).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 29.2: The Eulerian Wall Film Dialog Box - Model Options and Setup tab (with Phase Interaction)



These settings account for the effect of the interaction of the wall film with Eulerian and Mixture multiphase flow (see Secondary Phase Accretion in the *Fluent Theory Guide*). You can select:

– **Phase Change** and select the **Film Vapor Material**.

Note:

The secondary phase that is intended for the film material cannot be a mixture of species.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When **Phase Change** is selected, you can also specify phase change parameters—in the **Wall Film** tab of the **Wall** dialog box. For details, see Setting Eulerian Wall Film Boundary, Initial, and Source Term Conditions (p. 3252). This option is only available when the species model is enabled.

– **Phase Accretion** and specify **Phase Concentration** and **Phase Velocity** in the **Phase Accretion Parameters** group box. The **Phase Accretion** option enables you to compute the secondary phase collection efficiency on a wall surface. This option is available only when the **Eulerian** or **Mixture** (with **Slip Velocity**) multiphase model is enabled.

  For details about these settings, see Secondary Phase Accretion in the *Fluent Theory Guide*.

– **VOF Coupling** and select the EWF/VOF model transition criterion in the **Transition Base** group box:

  → **Volume Fraction**: Is the ratio of the film liquid volume to the film wall volume in the neighboring cell.

  → **Height Fraction**: Is the ratio of the film half-thickness to the center height of the neighboring cell .

If you want to skip the wall film solution during the gas phase solution, but keep the variables and setup active, enable **Solve Wall Film**.

Note:

The wall film cannot be solved without first initializing the wall film model (using the **Initialize** button) to initialize the wall film variables and prepare the solver for the solution procedure.

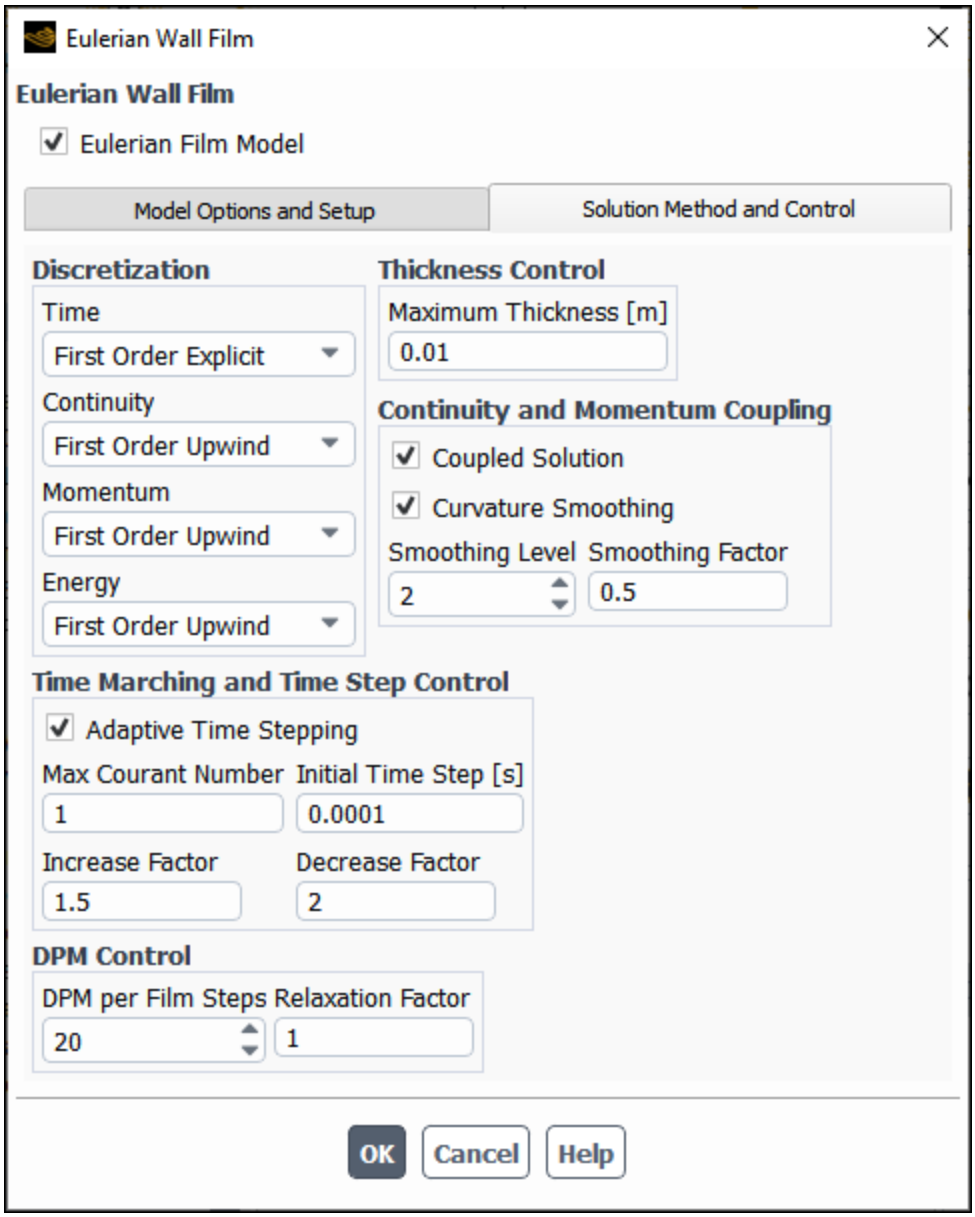
You can also use the `define/models/eulerian-wallfilm/model-options` text command to define the settings and the discretization methods for the Eulerian Wall Film model.

29.4. Setting Eulerian Wall Film Solution Controls

You can set solution controls for the Eulerian Wall Film model in the **Solution Method and Control** tab of the **Eulerian Wall Film** dialog box. This tab contains controls for specific temporal and spatial discretization options for the Eulerian Wall Film model.

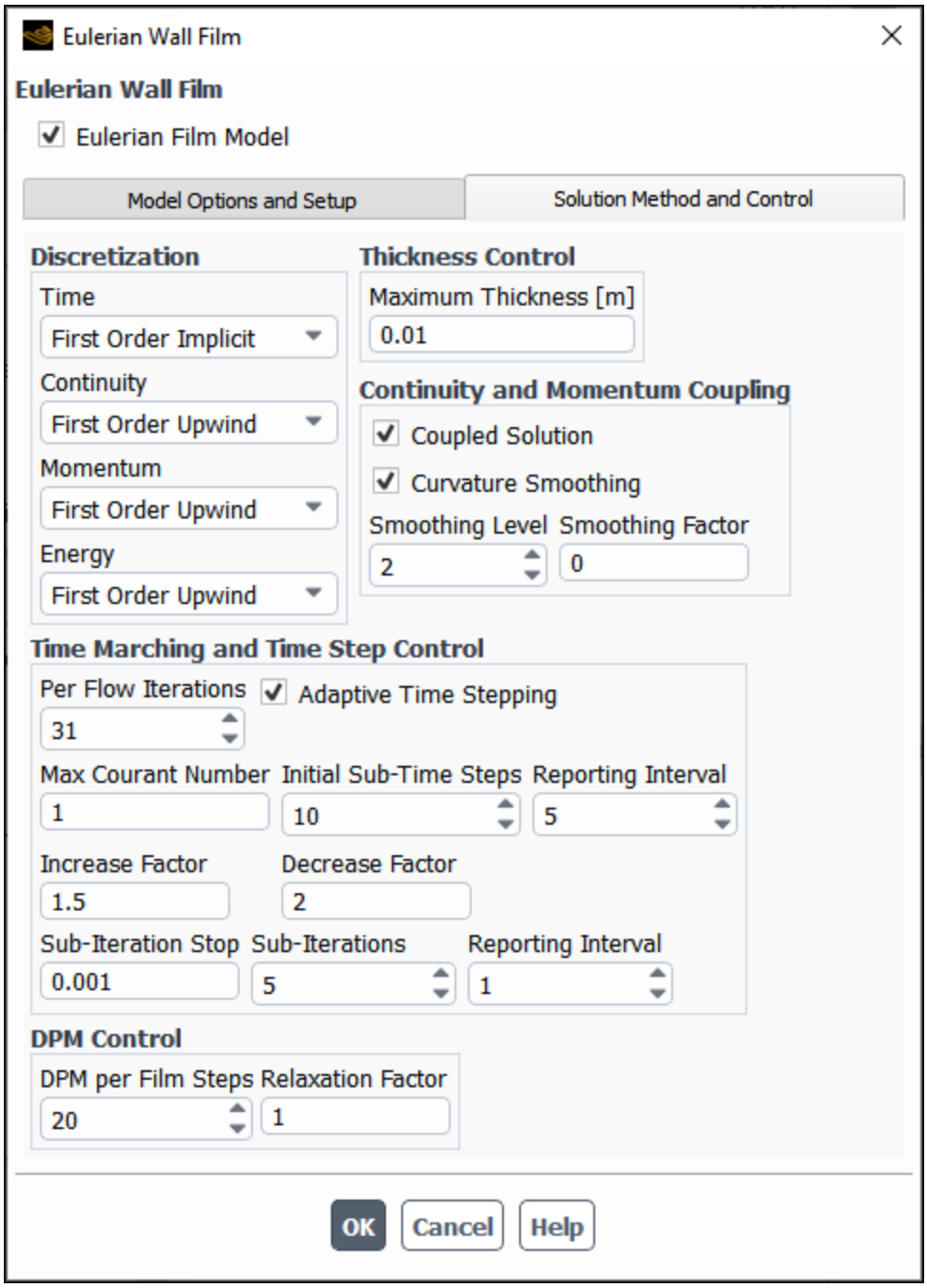
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 29.3: Eulerian Wall Film Solution Controls (Steady Flow)



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 29.4: Eulerian Wall Film Solution Controls (Unsteady Flow)



In the **Solution Method and Control** tab, you can set the following:

- Select the temporal and spatial (continuity, momentum, energy, and passive scalar) discretization methods under **Discretization**.

  For the **Time** discretization scheme, the available options are: **First Order Explicit**, **High Order Explicit**, **First Order Implicit**, and **Second Order Implicit**. See Temporal Differencing Schemes in the *Fluent Theory Guide* for theoretical information about theses options.

  The default discretization order for **High Order Explicit** is 2. You can increase the order to 3 or 4 by issuing the following text command:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
define/models/eulerian-wallfilm/solution-options
First Order Excplicit Time Discretization? [no]
High Order Excplicit Time Discretization? [yes]
 Explicit Scheme Order (2 - 4) [2] 4
```

For the **Continuity**, **Momentum**, **Energy**, and **Scalar** discretization schemes, the available options are: **First Order Upwind** and **Second Order Upwind**. See Spatial Differencing Schemes in the *Fluent Theory Guide* for theoretical information about theses options.

- Specify the **Maximum Thickness** for the film.

  The **Maximum Thickness** setting will limit the film thickness by removing material from the film where this value is exceeded.

- (If **Solve Momentum** is selected in the **Model Options and Setup** tab) Select the **Coupled Solution** option.

  When **Surface Tension** is selected in the **Model Options and Setup** tab, you can select the **Curvature Smoothing** option and specify the **Smoothing Level** and **Smoothing Factor**. See Coupled Solution Approach in the *Fluent Theory Guide* for details about these controls.

- (Steady state cases only (Figure 29.3: Eulerian Wall Film Solution Controls (Steady Flow) (p. 3249)) Set either of the following:

  – Specify the film model specific **Time Step** (the Eulerian Wall Film model is always transient)

  – Select the **Adaptive Time Stepping** option and specify:

  → **Max Courant Number**

  → **Initial Time Step**

  → Film time step **Increase Factor** and **Decrease Factor**

  Note that both **Increase Factor** and **Decrease Factor** must be larger than 1.

  See Steady Flow in the *Fluent Theory Guide* for more information.

- (Transient cases only (see Figure 29.4: Eulerian Wall Film Solution Controls (Unsteady Flow) (p. 3250)) Define the following:

  – The film solution interval **Per Flow Iterations**

  Within each flow time step, the film solution is called at the first flow iteration and subsequently at the flow iteration intervals defined by **Per Flow Iterations**. If you have entered **0**, the film solution is called at the end of the flow time step.

  – Either specify **Sub-Time Steps** per main flow time step or select the **Adaptive Time Stepping** option to set:

  → **Max Courant Number**

  → **Initial Sub-Time Step**

  → Film time step **Increase Factor** and **Decrease Factor**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that both **Increase Factor** and **Decrease Factor** must be larger than 1.

  - Specify **Reporting Interval** for film calculation reporting

    Note that the first and the last sub-time steps are always reported.

- (Cases with either first or second order implicit time discretization only) Specify the following:
  - The value of **Sub-Iteration Stop** below which the film sub-iteration stops
  - The number of film **Sub-Iterations**
  - The **Reporting Interval** for film sub-iteration reporting.

    Note that the first and the last sub-iterations are always reported.
- (If **DPM Coupling** is selected in the **Model Options and Setup** tab) Specify how often the DPM phase is calculated for the film by entering a value for the **Film Steps per DPM Step** and set **Relaxation Factor**. These controls are only available if **Interaction with Continuous Phase** is disabled in the **Discrete Phase Model** dialog box.

29.5. Setting Eulerian Wall Film Boundary, Initial, and Source Term Conditions

In the **Wall** dialog box, under the **Wall Film** tab, you can set boundary, initial, and source term conditions for liquid films on specified wall boundaries. When you select the **Eulerian Film Wall** option, Fluent solver designates the wall as a film wall. Wall film equations are only solved on designated film walls.

Note that the wall film edges connected to symmetry or periodic flow boundaries will also be treated as symmetry and periodic for the film flow. Film wall edges that are not connected to other designated film walls, symmetry boundaries, or periodic flow boundaries will be treated as film flow outlets where the wall film liquid exits the designated film walls.

The inputs are grouped into the following tabs:

- **Boundary Type**
- **Source Terms**
- **Phase Change**
- **Surface Contact**
- **DPM Interaction**
- **VOF Interaction**

For additional information, see the following sections:

29.5.1. Specifying the Boundary Type
29.5.2. Setting the Source Terms
29.5.3. Setting the Phase Change
29.5.4. Setting the Surface Contact

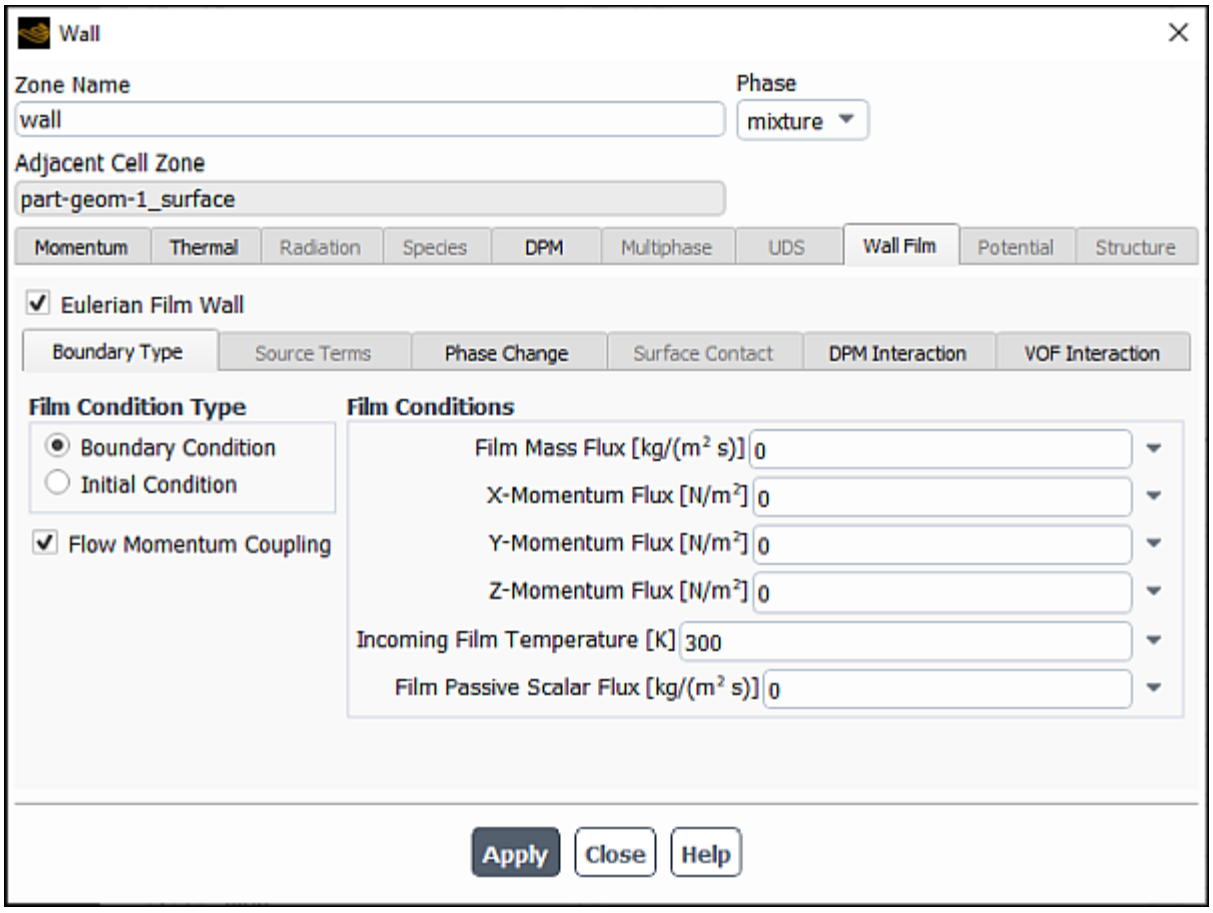
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

29.5.5. Setting the DPM interaction
29.5.6. Setting the VOF interaction

29.5.1. Specifying the Boundary Type

In the **Boundary Type** tab, you can select the film condition type and specify film conditions.

Figure 29.5: Wall Dialog Box - Boundary Type Tab



The following types of film conditions exist in Ansys Fluent:

- **Boundary Condition**: Is where film mass, momentum, energy and scalar quantities are introduced to the wall film solution domain.

  You can enter values for the following quantities:

  – **Film Mass Flux**

  – **X-Momentum Flux**, **Y-Momentum Flux**, and **Z-Momentum Flux**

  – **Incoming Film Temperature**

  – **Film Passive Scalar Flux**

  For moving wall simulations, you can specify the film momentum flux relative to the wall on which liquid film is defined by selecting **Relative Film Momentum Flux**. Note that this option is only

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

available when **Moving Wall** is selected in the **Wall** dialog box and/or **Frame Motion** is selected in the **Fluid** dialog box.

the film momentum flux or initial velocity inputs will be relative to the wall on which liquid film is defined.

- **Initial Condition**: Is the initial state of wall film.

  You can set values for the following quantities:

  – **Film Height**

  – **X-Velocity**, **Y-Velocity**, and **Z-Velocity** components

  – **Film Temperature**

  – **Film Passive Scalar**

  For moving wall simulations, you can specify the initial velocity relative to the wall on which liquid film is defined by selecting **Relative Initial Film Velocity**. Note that this option is only available when **Moving Wall** is selected in the **Wall** dialog box and/or **Frame Motion** is selected in the **Fluid** dialog box.

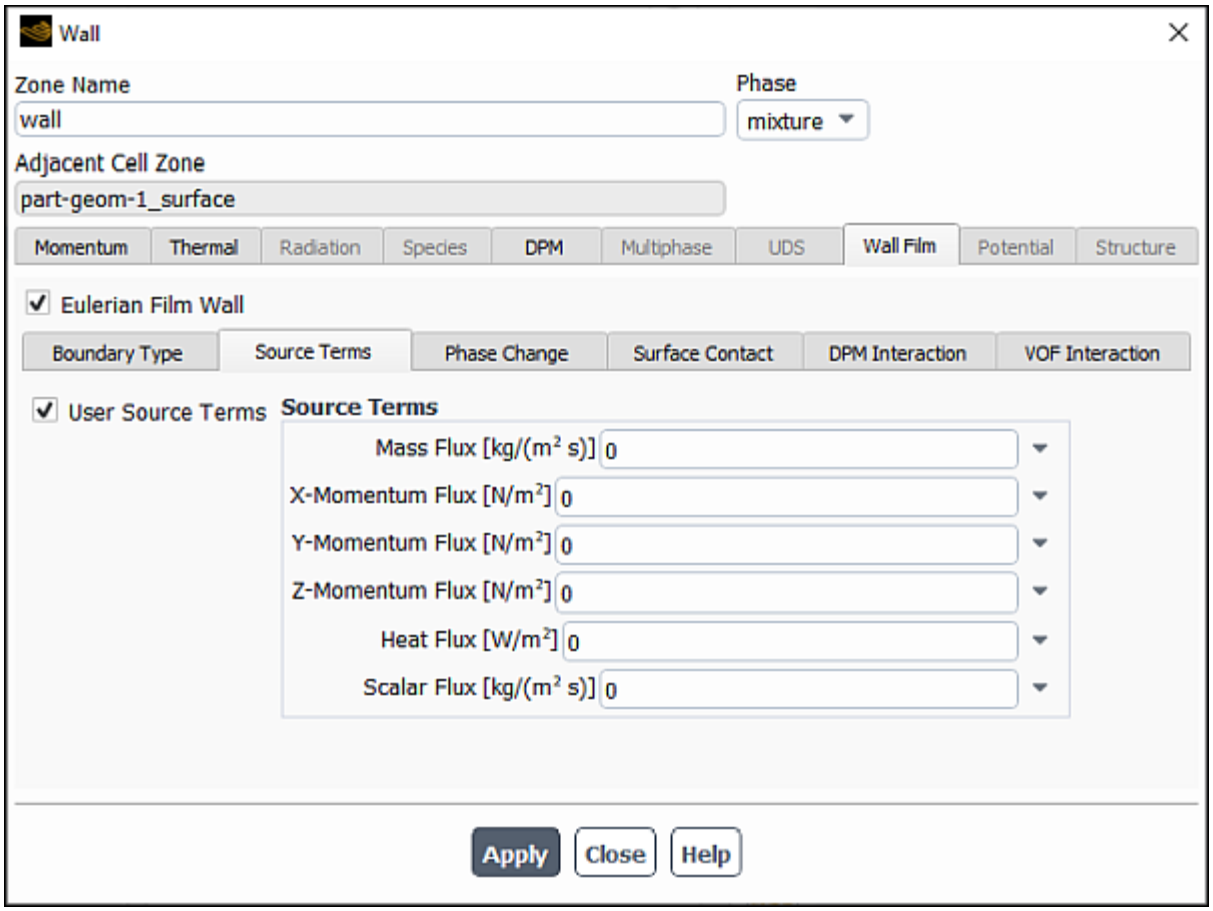
For both boundary and initial conditions, if you select the **Flow Momentum Coupling** option, the liquid film and the gas flow shares the same velocity at the interface of the liquid-gas interface using a two-way coupling. If you clear this option, the coupling between the liquid film and the gas flow occurs only one-way, namely, while the gas flow impacts the film flow, the film flow does not impact the bulk of the gas flow.

29.5.2. Setting the Source Terms

The **Sources Terms** tab is available only with **Initial Condition**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 29.6: Wall Dialog Box - Sources Terms Tab



If you select the **User Source Terms** option, you can also specify additional source terms to the film continuity, momentum, energy, and passive scalar equations by setting values for the following quantities:

- **Mass Flux**
- **X-Momentum Flux**, **Y-Momentum Flux**, and **Z-Momentum Flux**
- **Heat Flux**
- **Scalar Flux**

29.5.3. Setting the Phase Change

When the **Phase Change** option is selected in the **Eulerian Wall Film** dialog box, you can enable the **Film Phase Change** for the selected film wall, and then select the **Phase Change Model** and specify the model parameters in the **Phase Change** tab.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 29.7: Wall Dialog Box - Phase Change Tab

The following phase change models are available:

- **diffusion-balance**

  For the **diffusion-balance** model, you can specify **Condensation Constant** and **Vaporization Constant**.

- **wall-boundary-layer**

  No user input is required for this model.

- **user-defined**

  For the **user-defined** model, you can specify **Condensation Rate** and **Vaporization Rate** as constants, parameters, or profile user-defined functions (UDFs). Note that condensation and vaporization rates must be specified as positive values in kg/s. For more information on profile UDFs, the separate Fluent Customization Manual.

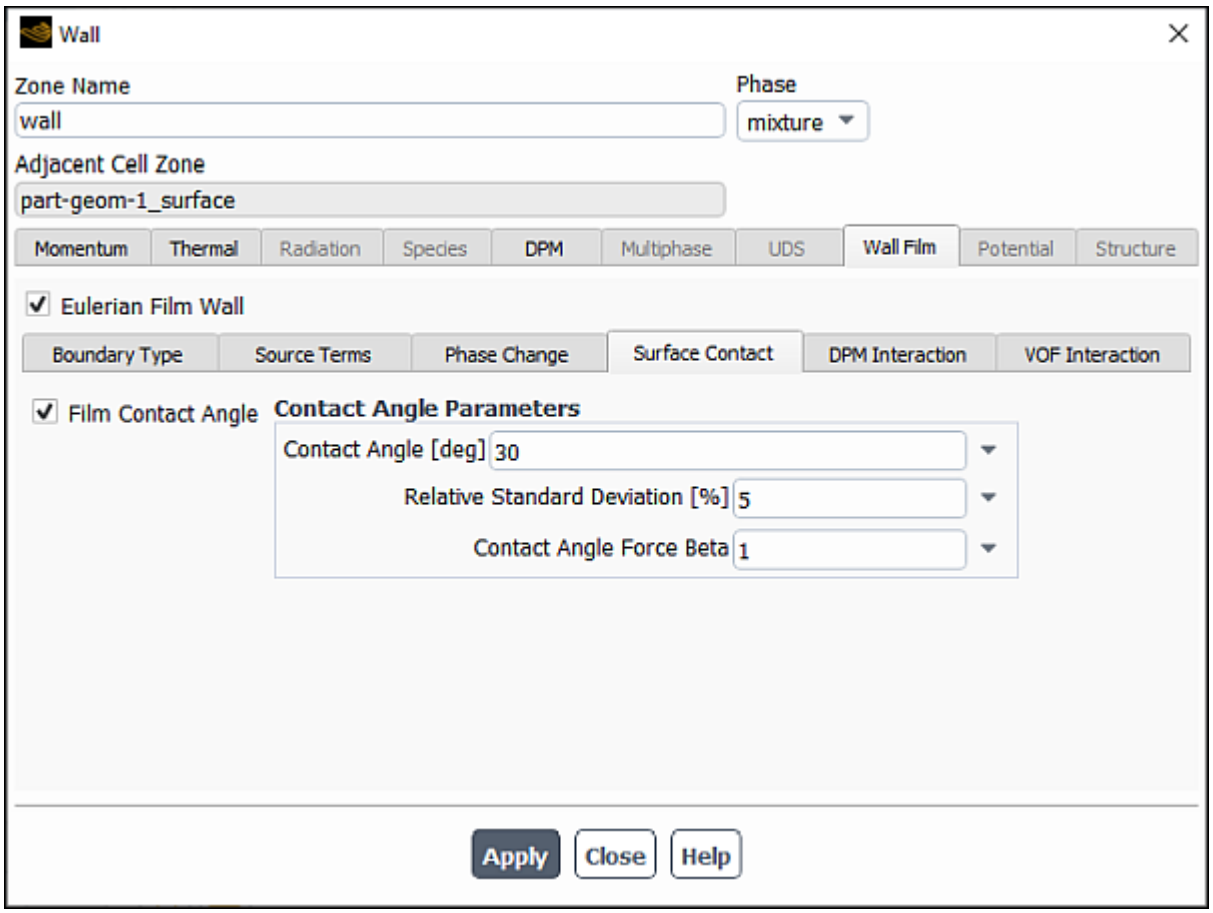
For more information on the phase change models see Coupling of Wall Film with Mixture Species Transport in the *Fluent Theory Guide*.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

29.5.4. Setting the Surface Contact

For cases with the initial condition, if you select the **Surface Tension** option in the **Eulerian Wall Film** dialog box, then you can enable the partial wetting model for the selected film wall by selecting the **Film Contact Angle** option in the **Surface Contact** tab.

Figure 29.8: Wall Dialog Box - Surface Contact Tab



You can then specify the following parameters in the **Contact Angle Parameters** group box:

- Mean **Contact Angle** (θ_m in Partial Wetting Effect in the *Fluent Theory Guide*)

  The input range for the mean **Contact Angle** is 0 to 180 degrees. You can also use a user-defined profile UDF. The UDF must return a contact angle value within the above range in radian. For more information on profile UDFs, the separate Fluent Customization Manual.

- **Relative Standard Deviation** (α in Partial Wetting Effect in the *Fluent Theory Guide*)

  The input for the **Relative Standard Deviation** should be in the range of 0 to 50%.

- **Contact Angle Force Beta** (β in Equation 17.29 in the *Fluent Theory Guide*)

  The input range for **Contact Angle Force Beta** is 0 to 10.

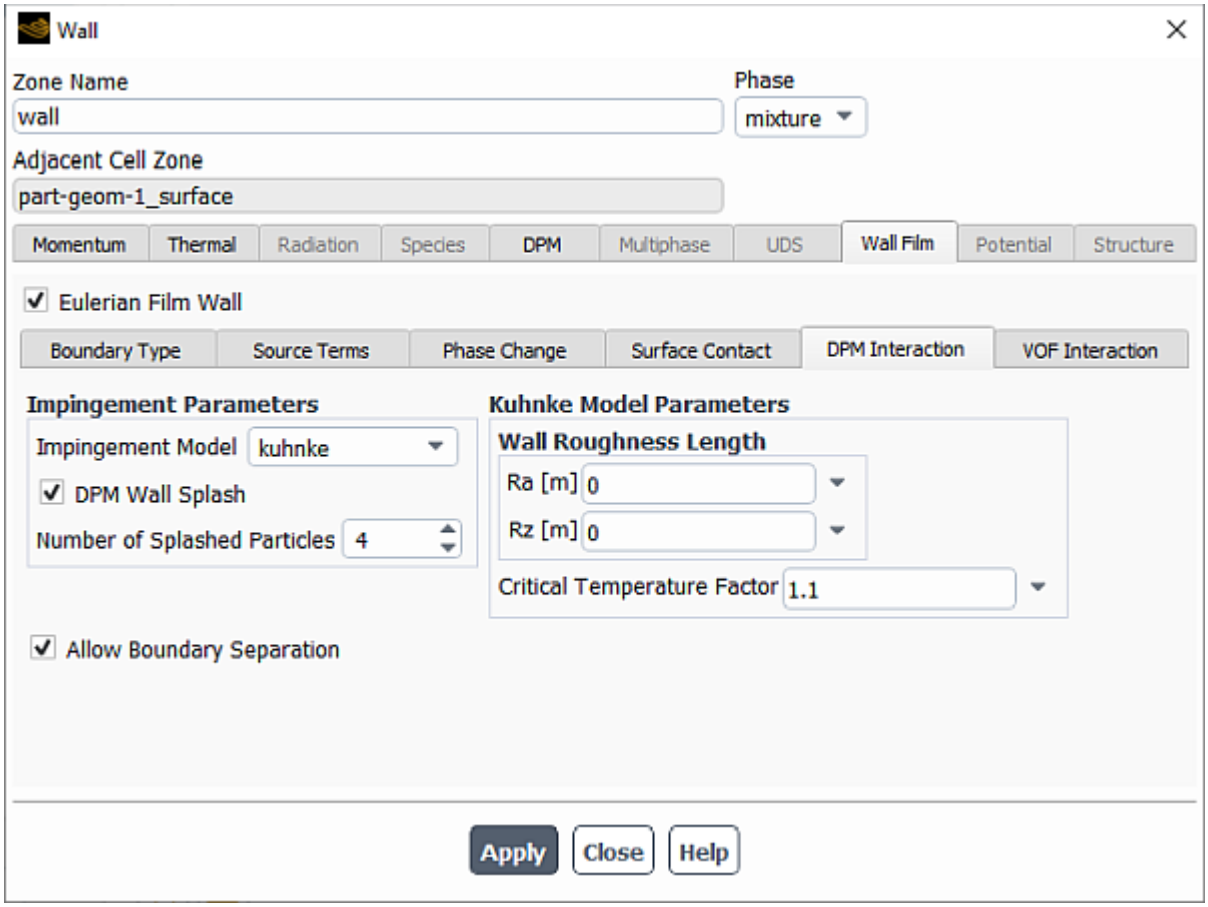
For more information on modeling the film partial wetting effect, see Partial Wetting Effect in the *Fluent Theory Guide*.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

29.5.5. Setting the DPM interaction

When the **DPM Coupling** option is selected in the **Eulerian Wall Film** dialog box, you can select a particle-wall impingement model and specify relevant model parameters in the **DPM Interaction** tab of the **Wall** dialog box.

Figure 29.9: Wall Dialog Box - DPM Interaction Tab



The following impingement models are available in Ansys Fluent:

- **stanton-rutland** (see The Stanton-Rutland Model in the *Fluent Theory Guide* for details)
- **kuhnke** (see The Kuhnke Model in the *Fluent Theory Guide* for details)

  For the Kuhnke model, you must specify the following model parameters:

  – **Ra**: The wall mean roughness that is used to compute K_{cold} in Equation 12.249 in the *Fluent Theory Guide*

  – **Rz**: The average surface roughness that is used to compute splashed particle reflection angle distribution Equation 12.261 in the *Fluent Theory Guide*.

  – **Critical Temperature Factor**: The dimensionless variable that is used to determine the impingement regime transition temperature in Equation 12.239 in the *Fluent Theory Guide*.

- **stochastic kuhnke** (see The Stochastic Kuhnke Model in the *Fluent Theory Guide* for details)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For the stochastic Kuhnke model, specify the following parameters in the **Kuhnke Model Parameters** group box:

- **Rz** under the **Wall Roughness Length**: This parameter will be used to interpolate the constant A from Parameter *A* as a Function of Wall Roughness in the *Fluent Theory Guide*, which will be subsequently used in Equation 12.267 in the *Fluent Theory Guide* to calculate critical Weber number for splashing on a dry wall.
- **Upper Deposition Limit Offset**: Is ΔT_{DL} in Equation 12.264 in the *Fluent Theory Guide*.
- **Deposition Delta T**: Is ΔT_{lim} in Equation 12.265 in the *Fluent Theory Guide*.
- **Laplace Number Constant**: Is A_w in Equation 12.266 in the *Fluent Theory Guide*.
- **Partial Evaporation Ratio**: Is the mass fraction of the impinging liquid spray that vaporizes immediately upon impact when the droplet is in the evaporative splash regime.

Note:

You can also add an impingement effect via UDFs as described in Hooking `DEFINE_IMPINGEMENT` UDFs, Hooking `DEFINE_FILM_REGIME` UDFs, and Hooking `DEFINE_SPLASHING_DISTRIBUTION` UDFs in the *Fluent Customization Manual*. Note that UDFs settings will override those for the impingement/splashing model selected for the Eulerian wall film boundary condition.

In addition, when **Particle Splashing** is selected in the **Eulerian Wall Film** dialog box, you can select **DPM Wall Splash** and specify **Number of Splashed Particles**.

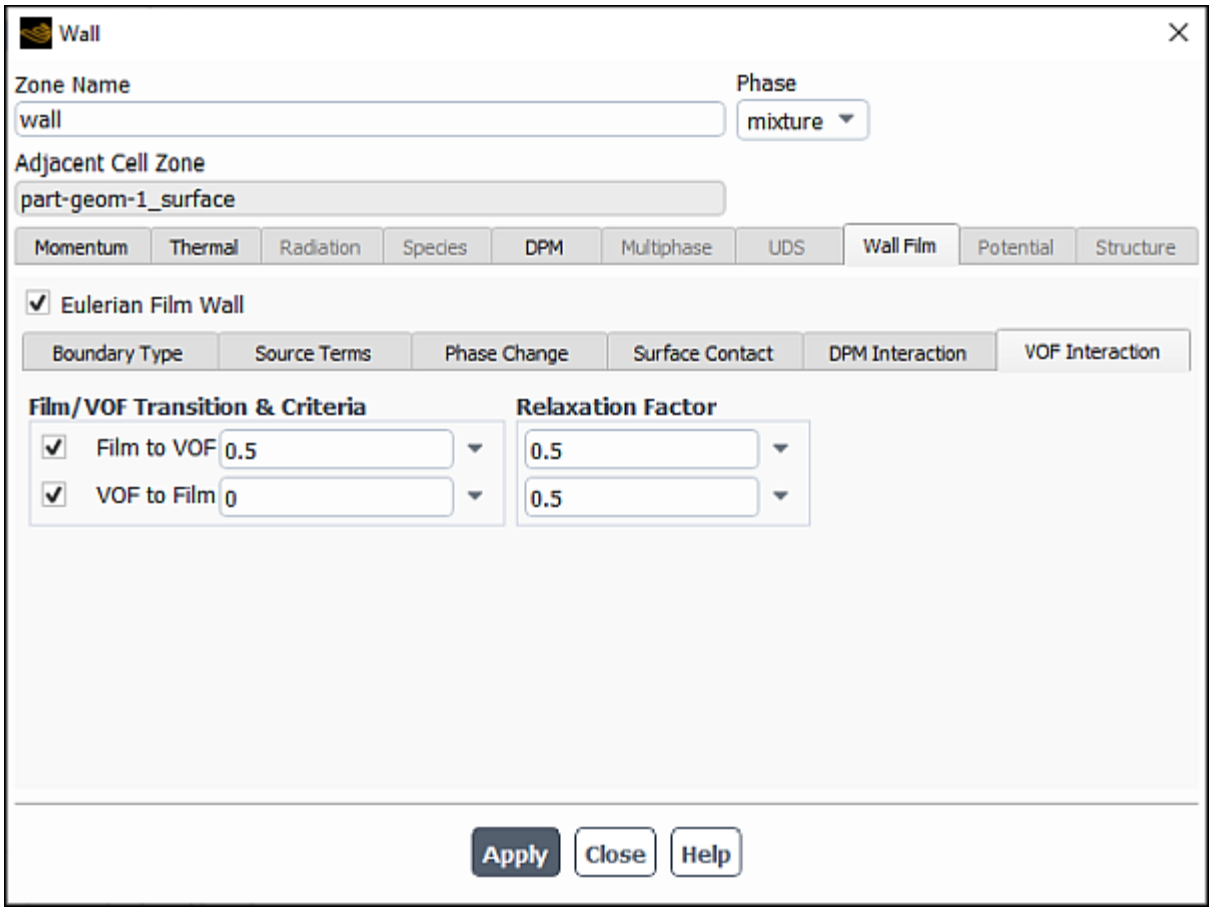
When the **Edge Separation** option is enabled in the **Eulerian Wall Film** dialog box, you can select the **Allow Boundary Separation** option to allow a separation at disconnected internal film wall boundaries.

29.5.6. Setting the VOF interaction

When the **VOF Coupling** option is selected in the **Eulerian Wall Film** dialog box (**Phase Interaction** tab), you can enable and set the **Film/VOF Transition & Criteria** and specify the **Relaxation Factor** in the **VOF Interaction** tab.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 29.10: Wall Dialog Box - VOF Interaction Tab



The **Film to VOF** and **VOF to Film** transition criteria are fraction values between 0 and 1 of either **Volume Fraction** or **Height Fraction** as specified in the **Eulerian Wall Film** dialog box (**Phase Interaction** tab, **Transition Base** group box). The **Film to VOF** transition criterion must be larger than the **VOF to Film** transition criterion. A weighted film thickness threshold is computed based on the specified transition criterion value and used to determine the onset of the transition.

The Film-to-VOF transition occurs when either the film thickness is larger than the **Film to VOF** threshold, or the film thickness exceeds **Maximum Thickness** specified in the **Eulerian Wall Film** dialog box (**Solution Method and Control** tab).

The VOF-to-Film transition occurs when the equivalent film thickness (computed from the relevant fraction values) is smaller than the **VOF to Film** threshold.

The thickness threshold values for the **Film to VOF** and **VOF to Film** transitions are displayed in the Fluent console once these settings are applied and during the model initialization.

The relaxation factors are values in the range of 0 to 1 (default: 0.5). They are used to control the release of transferred source terms and improve the stability of the solution process. In general, higher transition criteria tend to make the solution unstable and, therefore, may require smaller relaxation factors.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

29.6. Coupling of Eulerian Wall Film with the VOF Multiphase Model

To couple the Eulerian Wall Film and the VOF multiphase models, follow these steps:

1. Enable and set up the Eulerian wall film model and the VOF multiphase model as appropriate for your case.
2. In the **Eulerian Wall Film** dialog box, enable **Phase Coupling** (**Solution Options** group box) as shown in Figure 29.2: The Eulerian Wall Film Dialog Box - Model Options and Setup tab (with Phase Interaction) (p. 3247).
3. In the **Phase Interaction** tab, enable **VOF Coupling**.
4. In the **Transition Base** group box, select either **Volume Fraction** or **Height Fraction** transition criteria type. See Setting Eulerian Wall Film Model Options (p. 3241) for more information about these options.
5. In the **Wall** dialog box, go to the **Wall Film** tab.
6. In the **VOF Interaction** tab that becomes available after **VOF Coupling** is enabled, enable **Film VOF Coupling** and set parameters for the EWF and VOF model coupling as described in Setting Eulerian Wall Film Boundary, Initial, and Source Term Conditions (p. 3252).

For theoretical information about the EWF/VOF coupling, see Coupling of Eulerian Wall Film with the VOF Multiphase Model in the *Fluent Theory Guide*.

29.7. Postprocessing the Eulerian Wall Film

When using the Eulerian Wall Film model, the following additional variables will be available for post-processing (see Field Function Definitions (p. 4167) for their definitions):

- **Film Thickness**
- **Film Mass**
- **Film Temperature** (when **Solve Energy** is enabled)
- **Film X-Velocity**
- **Film Y-Velocity**
- **Film Z-Velocity**
- **Film Velocity Magnitude**
- **Film Surface X-Velocity**
- **Film Surface Y-Velocity**
- **Film Surface Z-Velocity**
- **Film Surface Velocity Magnitude**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Film Surface Temperature** (when **Solve Energy** is enabled)
- **Film Passive Scalar** (when **Solve Scalar** is enabled)
- **Film Density** (when film material density is non-constant)
- **Film Effective Pressure**
- **Film Coverage** (when **Solve Momentum** is enabled)
- **Film Courant Number**
- **Film Weber Number**
- **Film Stripped Mass** (when **Particle Stripping** is enabled)
- **Film Stripped Diam** (when **Particle Stripping** is enabled)
- **Film DPM Mass Source** (when **DPM Coupling** is enabled)
- **Film DPM Energy Source** (when **DPM Coupling** and **Solve Energy** are enabled)
- **Film DPM X-Momentum Source** (when **DPM Coupling** is enabled)
- **Film DPM Y-Momentum Source** (when **DPM Coupling** is enabled)
- **Film DPM Z-Momentum Source** (when **DPM Coupling** is enabled)
- **Film Separated Mass** (when **Edge Separation** is enabled)
- **Film Separated Diam** (when **Edge Separation** is enabled)
- **Film Separation Rate** (when **Edge Separation** is enabled)
- **Film Phase Change Rate** (when **Phase Change** is enabled)
- **Film Outflow Mass**
- **Film Secondary Phase Mass** (when **Phase Accretion** is enabled)
- **Film Secondary Phase Collection Coef** (when **Phase Accretion** is enabled)
- **Film VOF Equivalent Thickness** (when **VOF Coupling** is enabled)
- **Film Total Equivalent Thickness** (when **VOF Coupling** is enabled)
- **Film Volume Fraction** (when **VOF Coupling** is enabled)
- **Film Total Equivalent Volume Fraction** (when **VOF Coupling** is enabled)
- **Film Height Fraction** (when **VOF Coupling** is enabled)
- **Film Total Equivalent Height Fraction** (when **VOF Coupling**
- **Film VOF Mass Exchange** (when **VOF Coupling** is enabled)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Film VOF Exchanged Mass** (when **VOF Coupling** is enabled)

You can report on the mass and energy fluxes for the Eulerian Wall Film model using the **Flux Reports** dialog box (see Generating a Flux Report (p. 4136)), for domain boundaries such as inlets, outlets, and so on, as well as wall boundaries.

Results → Reports → Fluxes **Edit...**

The **Film Mass Flow Rate** and **Film Heat Transfer Rate** are available as options in the **Flux Reports** dialog box when the **Eulerian Wall Film** model is enabled.

Note:

The energy flux report only accounts for the heat transfer rate through the external boundary. It does not consider heat transfer at gas-liquid interfaces or heat transfer due to phase change or film-DPM interaction.

You can also report on the mass flow rate and heat transfer rate using the User Interface (TUI) as follows:

- For mass flow rate, enter the text command `report/fluxes/film-mass-flow`.
- For heat transfer rate, enter the text command `report/fluxes/film-heat-transfer`.

You will be prompted for the boundary face zone IDs for which the report will be generated, and whether or not you want to save the reported values to a file. Ansys Fluent will either print the report in the console or write it to the file you have specified.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 30: Modeling Electric Potential Field and Electrochemistry Models

This chapter provides details about how to model and simulate the electric potential field and electrochemistry models (electrolysis and H2 pump model and lithium-ion battery model) in Ansys Fluent. The theoretical aspects of the electric potential and electrochemistry models are presented in Electric Potential and Electrochemistry Models in the *Fluent Theory Guide*. Additional information about these models is provided in the following sections:

30.1. Simulating the Electric Potential Field

30.2. Simulating the Lithium-ion Battery

30.3. Setting the Electrolysis and H2 Pump Model

30.4. Postprocessing Electric Potential Field and Li-ion Battery Quantities

30.1. Simulating the Electric Potential Field

The Ansys Fluent electric potential model allows for predicting the electric potential field and its effects. The electric potential model can be used separately (for example, to compute electric static force for charged particles in a DPM calculation) or in combination with other Ansys Fluent models.

The electric potential solver has been integrated with the electrochemical reaction model (Electrochemical Reactions in the *Fluent Theory Guide*). When you enable the electrochemistry in your case, Ansys Fluent automatically enables the electric potential solver. However, if you want to use the electric potential model along with other Ansys Fluent models, you need to manually enable it as described in Setting Up the Electric Potential Model (p. 3266).

Using the electric potential solver, you can simulate the electric potential field in both fluid and solid zones.

When defining the boundary conditions for the electric potential field, you can specify either potential or current density at the external walls.

The following sections contain details about the Electric Potential model. Only information for setting the Electric Potential model is provided here.

30.1.1. Limitation of the Electric Potential Model

30.1.2. Setting Up the Electric Potential Model

30.1.1. Limitation of the Electric Potential Model

The following limitation exist for the electric potential model:

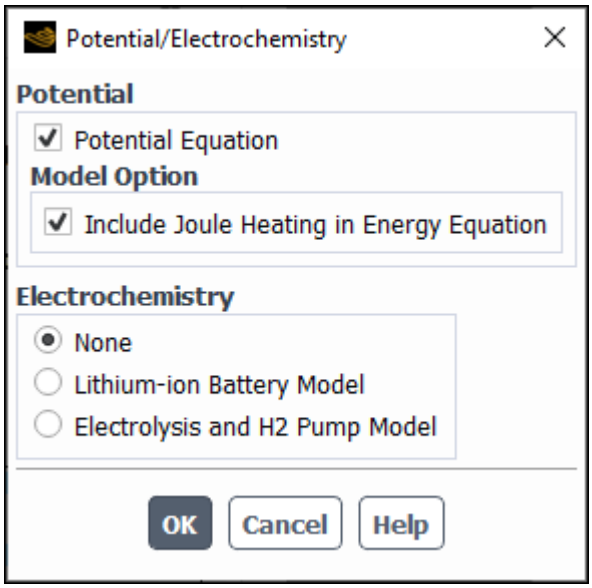
- The electric potential model cannot be used in simulations with walls that have a shell conduction boundary condition.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

30.1.2. Setting Up the Electric Potential Model

The procedure for setting up the electric potential model is described below. Note that only steps that are pertinent to electric potential field modeling are listed here. You will also need to define the other settings (such as other material properties, boundary conditions, other models) as usual. For information about inputs related to other Ansys Fluent models that you are using in conjunction with the electric potential model, refer to the appropriate sections for those models.

1. In the **Potential/Electrochemistry** dialog box, which is opened from the **Setup** → **Model** → **Potential/Electrochemistry** tree item, enable the **Potential Equation** option.



   Once the **Potential Equation** is enabled, the electric conductivity material property for fluid, solid, and mixture materials defined in your case will appear in the **Create/Edit Materials** dialog box.

2. If you want to include Joule heating in the energy equation (Equation 5.1 in the *Fluent Theory Guide*), under the **Model Option** group box, enable **Include Joule Heating in Energy Equation**.

   Note:

   The **Include Joule Heating in Energy Equation** option appears in the **Potential/Electrochemistry** dialog box only when **Energy Equation** is enabled in the **Energy** dialog box.

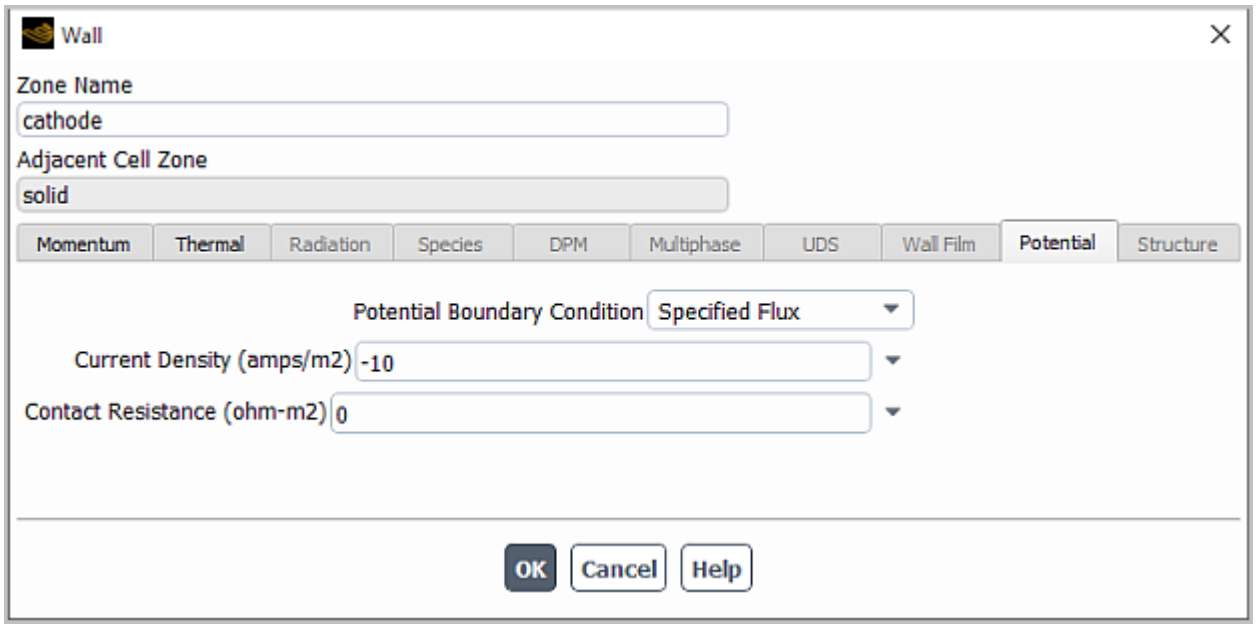
3. In the **Create/Edit Materials** dialog box, under **Properties**, specify **Electric Conductivity** for the conductive materials defined in your simulation.

   Setup → **Materials**

4. Define boundary conditions on conductive walls.

   Setup → **Boundary Conditions**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



a. Select one of the following options from the **Potential Boundary Condition** drop-down list:

   - **Specified Flux** and specify **Current Density**
   - **Specified Value** and specify **Potential**
   - (two-sided internal walls only) **Coupled**

     The coupled boundary condition you specify on one wall will be automatically applied to both walls in the wall / wall-shadow pair.

   When you use decoupled potential boundary conditions for two-sided internal walls (**Specified Flux** or **Specified Value**), you can define different boundary conditions on the wall and wall-shadow.

b. Specify **Contact Resistance** (R in Equation 18.3 in the *Fluent Theory Guide*).

5. If required, specify electric potential conditions on other boundaries in the **Potential** tab of the boundary conditions dialog boxes. Note that the default setting, zero current flux, is suitable for most problems and needs not be changed.

6. If you want to fix a value of potential or specify a potential source in a cell zone, you can do it by setting the **Fixed Values** or **Source Terms** in the appropriate cell zone condition dialog box.

30.2. Simulating the Lithium-ion Battery

The Lithium-ion Battery model allows you to simulate the detailed physics governing the charging and discharging processes. The theoretical aspects of the Lithium-ion Battery model are presented in Lithium-ion Battery Model Theory in the *Fluent Theory Guide*.

Note that the Lithium-ion Battery model is designed to simulate detailed physics occurring on a very small electrode scale. If you are interested in the thermal aspect of the whole battery cell or a battery

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

pack, consider a different modeling approach such as the dual potential MSMD battery model. See Using the MSMD-Based Battery Models (p. 3300) for details.

The following sections contain details about the Lithium-ion Battery model. Only information for setting the Lithium-ion Battery model is provided here.

30.2.1. Limitations of the Detailed Lithium-ion Battery Model

30.2.2. Setting Up the Lithium-ion Battery Model

30.2.1. Limitations of the Detailed Lithium-ion Battery Model

- The electrode cell zone must be solid, and the electrolyte cell zone must be fluid.
- The model can be used only for full electrode simulations; that is, the model must include a positive electrode zone, an electrolyte zone and a negative electrode zone. Half-cell models are currently not supported.

30.2.2. Setting Up the Lithium-ion Battery Model

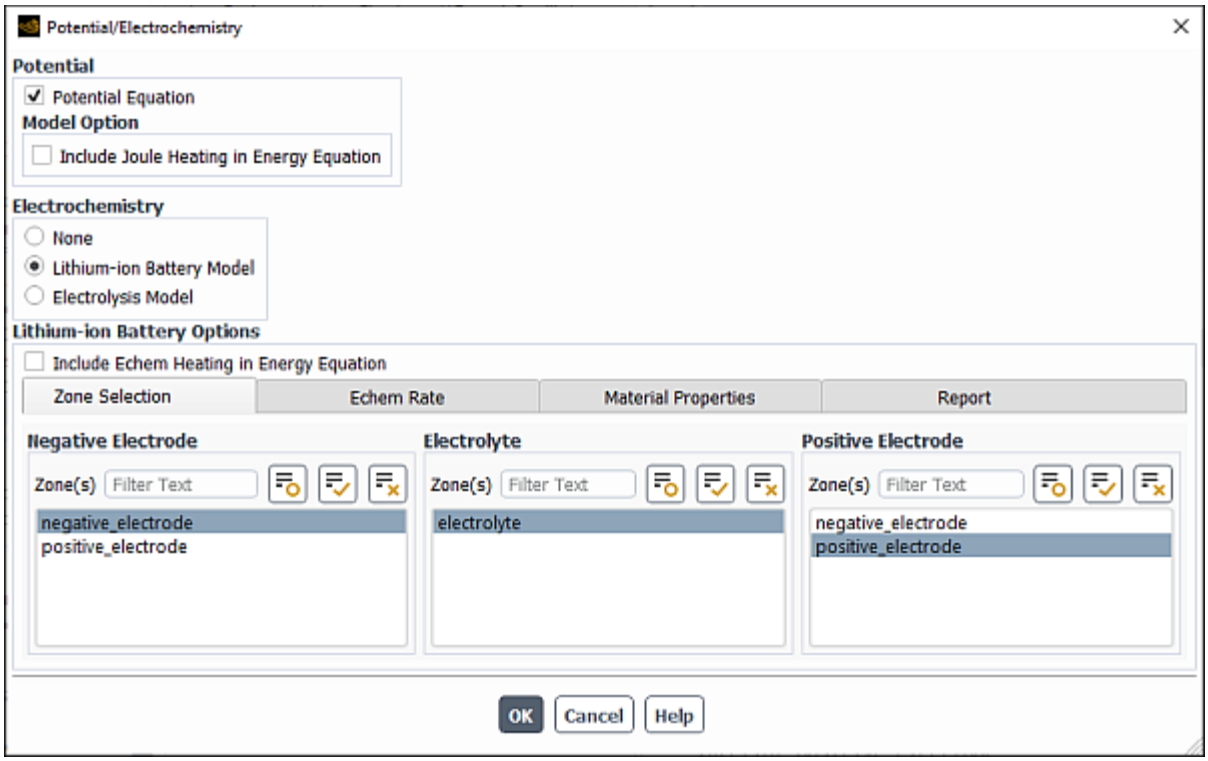
1. In the **Potential/Electrochemistry** dialog box, enable the **Potential Equation** (**Potential** group box) and the **Lithium-ion Battery Model** (**Electrochemistry** group box).

   Setup → **Models** → **Potential/Electrochemistry** **Edit...**

   The dialog box expands to reveal the parameters for the detailed lithium-ion battery model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 30.1: The Potential/Electrochemistry Dialog Box - Lithium-ion Battery Model



Note:

Once the **Lithium-ion Battery Model** is enabled,Ansys Fluent automatically switches to the transient solver.

The inputs are grouped into the following tabs:

- **Zone Selection**
- **Echem Rate**
- **Material Properties**
- **Report**

You need to visit each tab and specify the lithium-ion battery model parameters for your analysis.

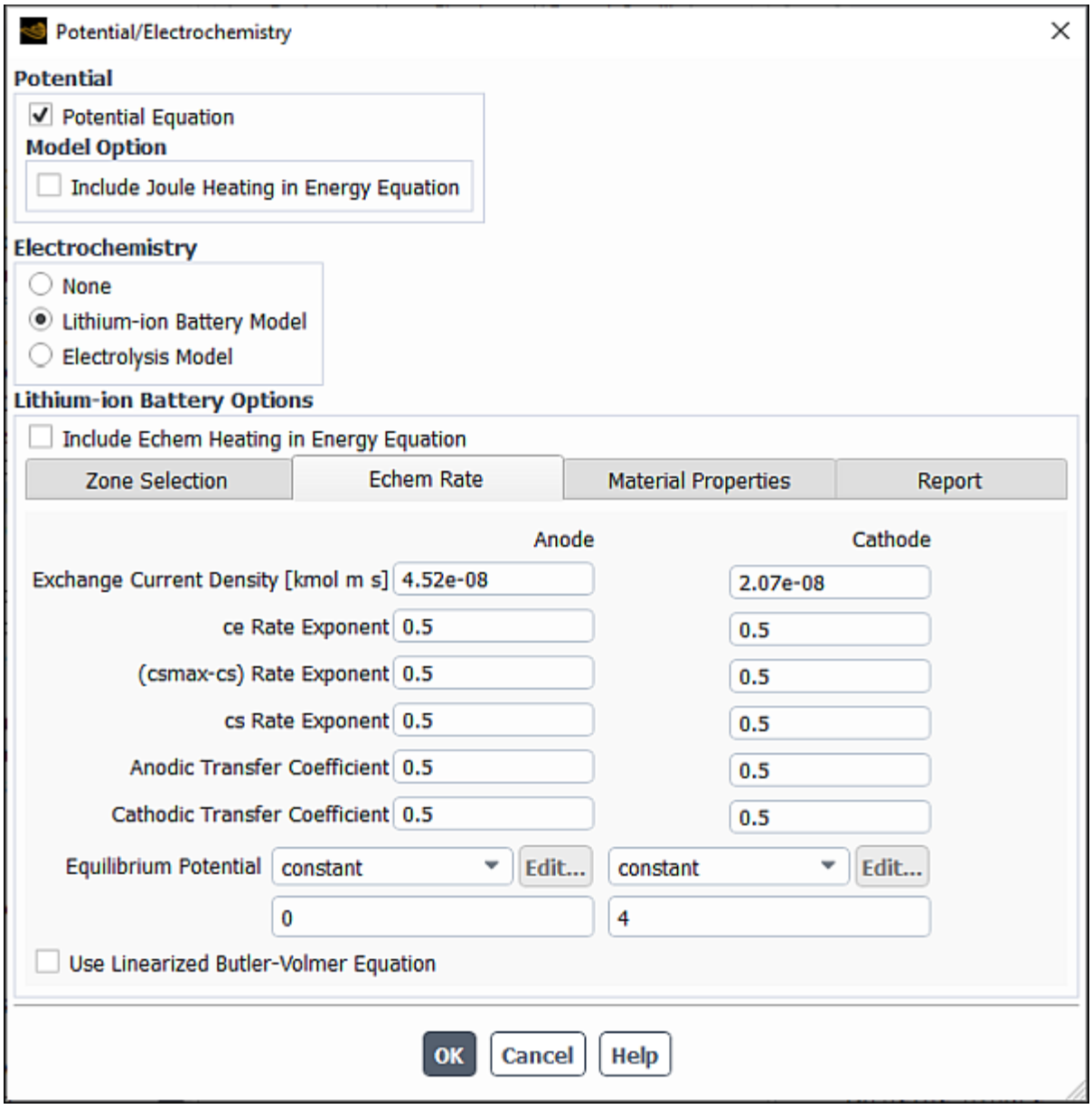
2. If you want to consider the electro-chemical reaction heat in the energy equation, select **Include Echem Heating in Energy Equation**.

3. In the **Zone Selection** tab (see Figure 30.2: The **Potential/Electrochemistry** Dialog Box - Echem Rate Tab (p. 3270)), select appropriate zones from the **Positive Electrode**, **Electrolyte Zone**, and **Negative Electrode** selection lists.

4. In the **Echem Rate** tab, define the Butler-Volmer rate parameters for the cathode and anode:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 30.2: The Potential/Electrochemistry Dialog Box - Echem Rate Tab



- **Exchange Current Density**: is i_0 in Equation 18.9 in the *Fluent Theory Guide*.
- **ce Rate Exponent**: is α_1 in Equation 18.9 in the *Fluent Theory Guide*.
- **(csmax-cs) Rate Exponent**: is α_2 in Equation 18.9 in the *Fluent Theory Guide*.
- **cs Rate Exponent**: is α_3 in Equation 18.9 in the *Fluent Theory Guide*.
- **Anodic Transfer Coefficient**: is α_a in Equation 18.9 in the *Fluent Theory Guide*.
- **Cathodic Transfer Coefficient**: is α_c in Equation 18.9 in the *Fluent Theory Guide*.
- **Equilibrium Potential**: is U in Equation 18.9 in the *Fluent Theory Guide*.

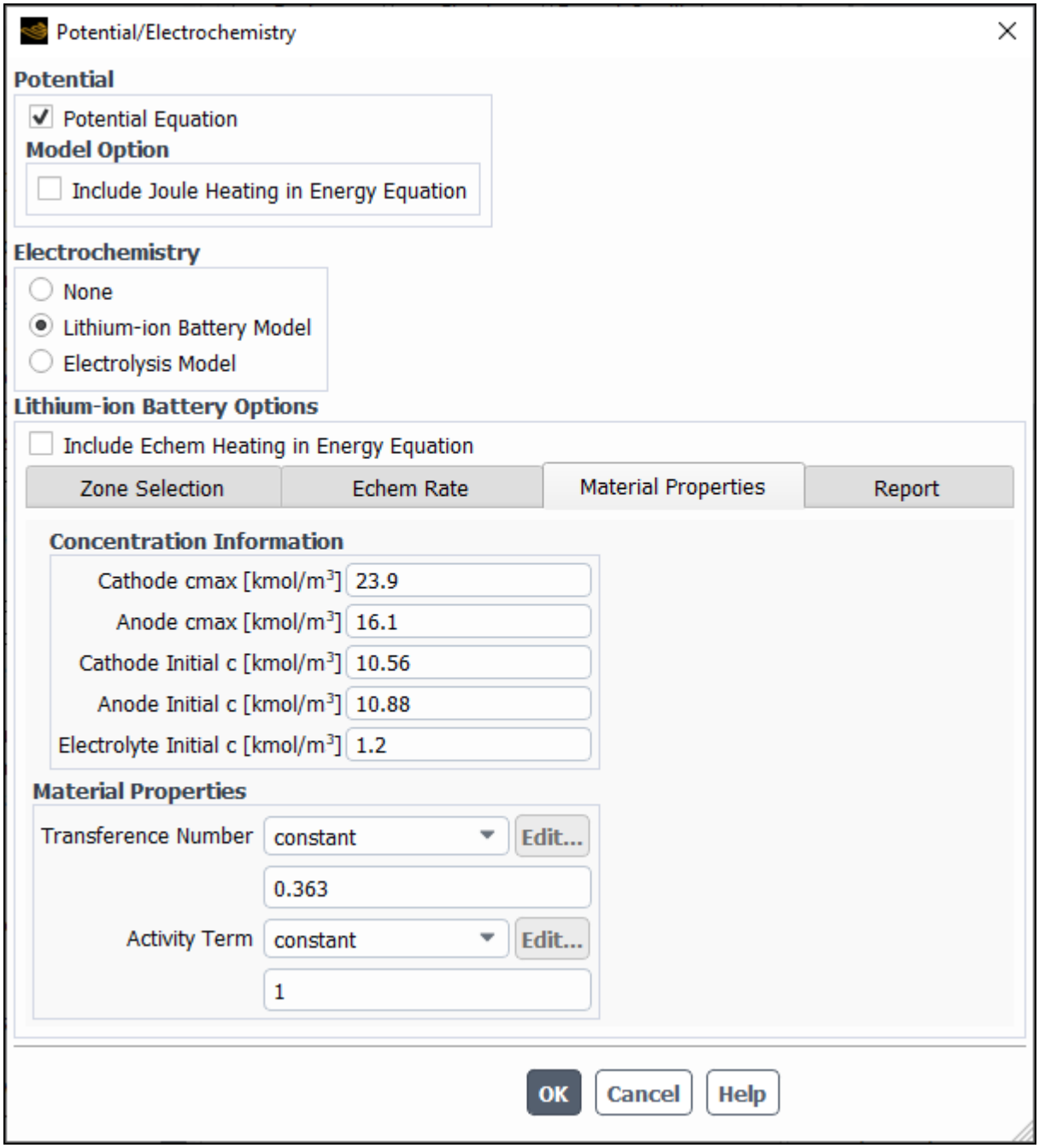
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To customize **Equilibrium Potential**, you can use the `DEFINE_EC_KINETICS_PARAMETER` user-defined function (see the section on `DEFINE_EC_KINETICS_PARAMETER` in the Ansys Fluent Customization Manual for details).

If you want to use the linearized form of the Butler-Volmer equation (Equation 18.10 in the *Fluent Theory Guide*), select **Use Linearized Butler-Volmer Equation**.

5. In the **Material Properties** tab, define the lithium concentration information and the material properties:

Figure 30.3: The Potential/Electrochemistry Dialog Box - Material Properties Tab



You can specify the following properties:

- **Concentration Information**:
  - **Cathode cmax**: a maximum value of lithium concentration in the cathode (used in Equation 18.9 and Equation 18.10 in the *Fluent Theory Guide*).
  - **Anode cmax**: a maximum value of lithium concentration in the anode (used in Equation 18.9 and Equation 18.10 in the *Fluent Theory Guide*).
  - **Cathode Initial c**: lithium concentration in the cathode at the beginning of discharge (c in Equation 18.4 in the *Fluent Theory Guide*).
  - **Anode Initial c**: lithium concentration in the anode side at the beginning of discharge (c in Equation 18.4 in the *Fluent Theory Guide*)
  - **Electrolyte Initial c**: lithium-ion concentration in the electrolyte (c in Equation 18.4 in the *Fluent Theory Guide*)
- **Material Properties**:
  - **Transient Number** is the lithium ion transference number t^+ in Equation 18.8 in the *Fluent Theory Guide*.
  - **Activity Term** is the term $1+\frac{\mathrm{d}\left(\ln f_{\pm}\right)}{\mathrm{d}\left(\ln c\right)}$ in Equation 18.8 in the *Fluent Theory Guide*.

6. In the **Create/Edit Material** dialog box, define materials for the negative electrode, positive electrode, and electrolyte. Make sure to specify proper values for **Electrical Conductivity** and **Lithium Diffusivity** for each material.

   Physics → **Materials** → **Create/Edit...**

7. In the **Cell Zone Conditions** task page, assign appropriate materials to different zones. The electrode cell zone must be solid, and the electrolyte cell zone must be fluid.

   Physics → **Zones** → **Cell Zones**

8. In the **Potential** tab of the **Wall** dialog boxes, define electric potential boundary conditions for the anode and cathode walls.

   Physics → **Zones** → **Boundaries**

   Select one of the following options from the **Potential Boundary Condition** drop-down list:

   - **Specified Flux** and specify **Current Density**
   - **Specified Value** and specify **Potential**

   For the anode, select **Specified Value** for **Potential Boundary Condition**. The default value of **0** for **Potential** is usually adequate.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For the cathode, when specifying **Current Density**, a negative value means current coming out of the domain, while a positive value means current flowing into the domain.

Note:

There is no need to define any boundary condition for lithium.

9. In the **Residual Monitors** dialog box, set the convergence conditions for the **potential** and **lithium** equations. Usually you need to tighten the convergence criteria. For example, set **`1.0e-10`** for the **potential** field, and **`1e-12`** for the **lithium** concentration field.

   Solution → **Reports** → **Residuals...**

10. (optional) If there is no flow field in the simulation, disable the **Flow** and **Energy** equations in the **Equations** dialog box (accessed from the **Solution Controls** task page).

    Solution → **Controls** → **Controls...**

11. Initialize the flow field.

    Note:

    The lithium and potential fields will be automatically initialized according to the information you provided in the **Potential/Electrochemistry** dialog box. No input is required from you for the two equations.

12. In the **Solution Controls** task page, adjust the under-relaxation factor for **Faradaic Interface Current** if necessary.

    Solution → **Controls** → **Controls...**

    Depending on the non-linearity of your problem, you can steer the solution by adjusting the value of the **Faradaic Interface Current** under-relaxation factor. If you are using a constant value for the **Equilibrium Potential** in the Butler-Volmer rate, you can use the default value of 0.01 or even a larger value. If you use a UDF to define equilibrium potential as a function of lithium concentration, then the coupling between potential and lithium concentration equation will be strong. Sometimes you may need to use a small value (as low as 0.001) in order to obtain a stable solution.

13. Once the lithium-ion battery model is enabled, Ansys Fluent automatically generates two report definitions:
    - `li-ion-battery-soc`
    - `li-ion-battery-capacity`

    You can include these reports in report files or report plots (for example by right-clicking a report definition in the **Outline View** and selecting **Include In** > **Report Plots** > **New...**). In order for these report definitions to be computed correctly, make sure to correctly set lithium concentration at 0% and 100% SOC conditions (**CLI at 0% SOC** and **CLI at 100% SOC** in the **Report** tab).

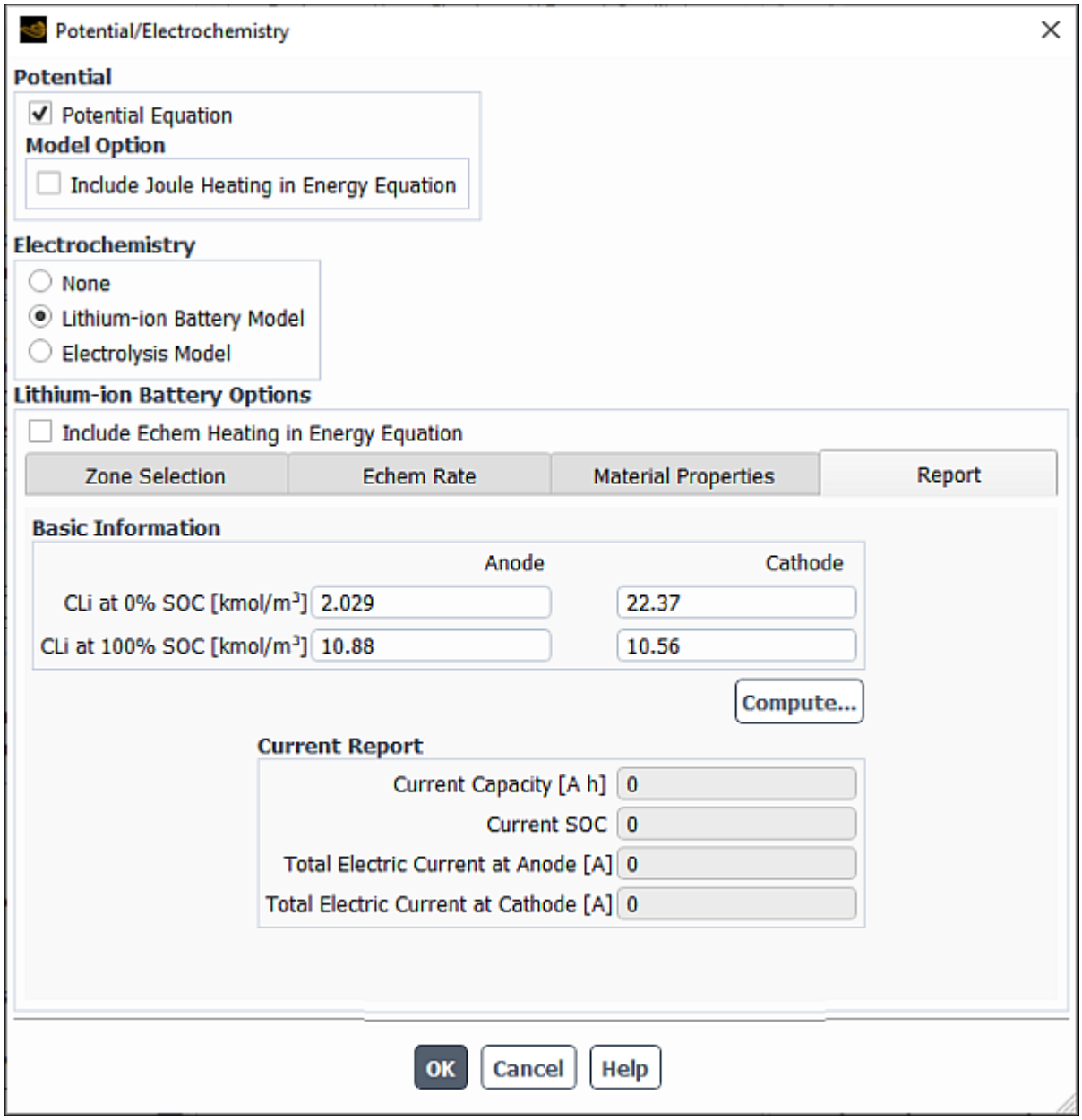
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

14. Run the simulation.

    If you use a low value for the **Faradaic Interface Current** under-relaxation factor, you may need to specify a large number for **Max Iterations/Time Step** in the **Run Calculation** task page. You can experiment with different numbers to find an appropriate value for your specific application. You can check the total current at the anode and cathode. The goal is that the two current values must be adequately balanced out during every time step in the simulation.

15. In the **Report** tab of the **Potential/Electrochemistry** dialog box, check the current balance at the SEIs in the negative and positive electrodes.

Figure 30.4: The Potential/Electrochemistry Dialog Box - Report Tab



a. Specify the following values under **Anode** and **Cathode**:

   - **CLI at 0% SOC**: Lithium concentration at 0% SOC condition

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **CLI at 100% SOC**: Lithium concentration at 100% SOC condition

b. Click **Compute...**.

In the **Current Report** group box, the solver reports battery's current capacity, state of charge (SOC) and total current at anode and cathode SEIs.

Total current at anode and total current at cathode should be close enough to ensure good results. If they do not match, the solution has not fully converged. You may want to increase **Max Iterations/Time Step** and rerun your simulation.

30.3. Setting the Electrolysis and H2 Pump Model

The electrolysis and H2 pump model can be used to simulate PEM electrolysis, and alkaline electrolysis. For more information about the theory behind the electrolysis and H2 pump model, see Electrolysis and H2 Pump Model in the *Fluent Theory Guide*.

The following sections contain details about the electrolysis and H2 pump model. Only information for setting the electrolysis and H2 pump model is provided here.

30.3.1. Geometry Definition for the Electrolysis and H2 Pump Model

30.3.2. Workflow for Using the Electrolysis and H2 Pump Model

30.3.3. Setting up the Electrolysis and H2 Pump Model

30.3.4. Solution Strategies for the Electrolysis and H2 Pump Model

30.3.1. Geometry Definition for the Electrolysis and H2 Pump Model

Since there is a number of different physical zones associated with the electrolysis device, the following regions must be present in the electrolysis model mesh:

- Anode flow channel
- Anode gas diffusion layer (porous layer)
- Anode catalyst layer
- Membrane layer
- Cathode catalyst layer
- Cathode gas diffusion layer (porous layer)
- Cathode flow channel

In addition, the following zones must be identified, if present in the electrolysis model mesh:

- Anode current collector (solid)
- Cathode current collector (solid)

30.3.2. Workflow for Using the Electrolysis and H2 Pump Model

The following describes an overview of the procedure required in order to use the electrolysis and H2 pump model in Ansys Fluent.

1. Start Ansys Fluent.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Read the case or mesh file.
3. Scale the mesh, if necessary.
4. Enable and set up electrolysis and H2 pump model as described in Setting up the Electrolysis and H2 Pump Model (p. 3276).
5. Define material properties.

   In the **Create/Edit Materials** dialog box, under **Properties**, you must specify **Electrical Conductivity** and **Electrolyte Conductivity** for the collector, porous, catalyst, and electrolyte materials defined in your simulation. The material electrical and electrolyte conductivities can be specify as constant, a function of temperature in forms of piecewise-linear, piecewise-polynomial or polynomial, an expression, or as a user-defined function using the `DEFINE_PROPERTY` macro.
6. Set the operating conditions.
7. Set the boundary conditions for the flow.

   Note:

   Ansys Fluent automatically sets boundary conditions for the electric potential field. They are suitable for most cases and should not be modified.
8. Start the calculations.
9. Save the case and data files.
10. Process your results.

Note:

The **Potential/Electrochemistry** dialog box greatly simplifies the input of parameters and boundary conditions, but it does not replace the boundary conditions interface. Therefore, it is a good practice to start the setup with the **Potential/Electrochemistry** dialog box and do the finishing steps for boundary conditions afterwards.

30.3.3. Setting up the Electrolysis and H2 Pump Model

1. In the **Potential/Electrochemistry** dialog box enable **Potential Equation** (in the **Potential** group box), and **Electrolysis and H2 Pump Model** (in the **Electrochemistry** group box).

   Setup → **Models** → **Potential/Electrochemistry** **Edit...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Once the electrolysis and h2 pump model has been enabled, Ansys Fluent automatically enables the mixture multiphase phase and specie transport models.

Note:

The electrolysis and H2 pump model uses the mixture multiphase model to consider multiphase phenomena. It cannot be used with VOF or Eulerian multiphase model.

The dialog box expands to reveal the parameters for the detailed electrolysis and H2 pump model. The model settings are organized into the following tabs:

Model

contains the general model settings to define problems that involve the electrolysis and H2 pump model.

Parameters

is where you specify parameters for solving the model equations.

Anode

allows you to select the appropriate anode zones and specify their properties.

Electrolyte

allows you to select the appropriate electrolyte/membrane zones and specify their properties.

Cathode

allows you to select the cathode zones and specify their properties.

Electrical Tabs

is where you can select anode and cathode tabs.

Customization

is where you can customize electrolysis model parameters by hooking your user-defined functions (UDFs).

Advanced

is where you can specify contact resistivity at the interface between cell zones.

2. If you want to consider the electro-chemical reaction heat in the energy equation, select **Include Echem Heating in Energy Equation** (**Potential** group box).

3. Identify the relevant zones for the current collectors, flow channels, gas diffusion layers (porous layers), catalyst layers, and the electrolyte (membrane) and specify all relevant parameters.

Refer to the following sections for more information:

30.3.3.1. Specifying Model Options (Model Tab)

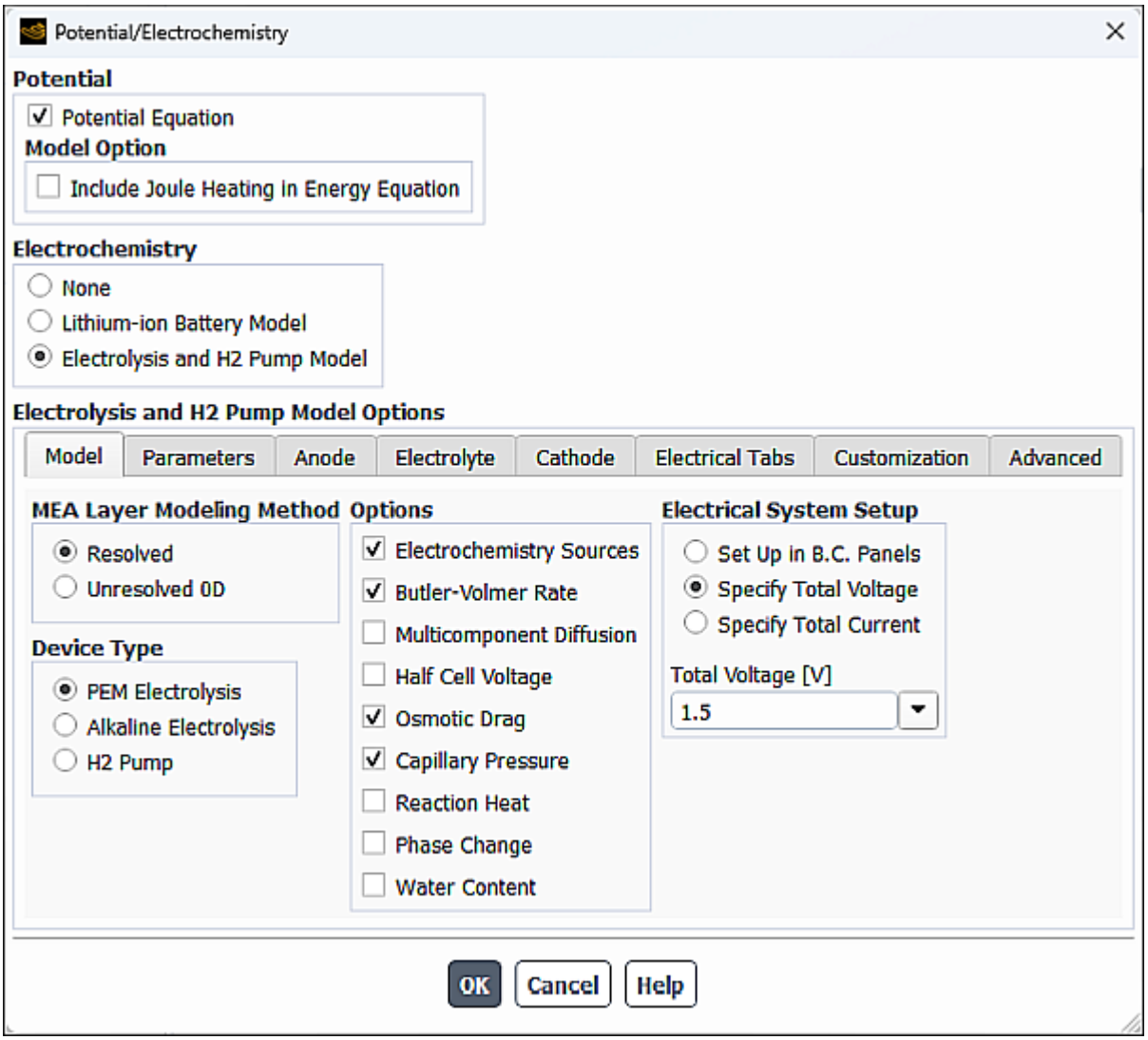
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

30.3.3.2. Specifying Model Parameters (Parameters Tab)
30.3.3.3. Specifying Anode Properties (Anode Tab)
30.3.3.4. Specifying Electrolyte/Membrane Properties (Electrolyte Tab)
30.3.3.5. Specifying Cathode Properties (Cathode Tab)
30.3.3.6. Setting the External Electrical Tabs (Electrical Tabs Tab)
30.3.3.7. (Customization Tab)
30.3.3.8. Setting Advanced Options (Advanced Tab)

30.3.3.1. Specifying Model Options (Model Tab)

The **Model** tab of the **Potential/Electrochemistry** dialog box allows you to enable or disable various general options and specify relevant settings when solving an electrolysis problem.

Figure 30.5: The Potential/Electrochemistry Dialog Box (Model Tab)



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

MEA Layer and H2 Pump Method

In the **MEA Layer and H2 Pump Method** group box, you can select the modeling approach for the membrane electrode assemblies (MEA) zones (the catalyst layers and the membrane). The following options are available:

- **Resolved**

  Refer to Resolved Modeling Approach in the *Fluent Theory Guide* for details about this approach.

- **Unresolved 0D**

  Refer to Unresolved 0D Modeling Approach in the *Fluent Theory Guide* for details about this approach.

Device Type

In the **Device Type** group box, you can select from the following electrolysis processes you want to model:

- **PEM Electrolysis**
- **Alkaline Electrolysis**
- **H2 Pump**

The main difference between the two is that different electrochemistry is applied in the calculation.

Options

You can use the following options:

Electrochemistry Sources

allows the electrolysis and H2 pump model to take electrochemistry effects into account. If you are only interested in the basic potential field throughout the electrolysis and H2 pump model, you can disable the **Electrochemistry Sources** option to suppress most effects of the electrolysis and H2 pump model. You may also disable the effect of these sources to obtain an initial fluid flow and then enable it and continue your simulation.

Butler-Volmer Rate

(default) is used to compute the transfer currents inside the catalyst layers. If this option is disabled, the Tafel approximation is used (Equation 20.7 and Equation 20.8).

Multicomponent Diffusion

is used to compute the gas species mass diffusivity using the full multicomponent diffusion method (Full Multicomponent Diffusion in the *Fluent Theory Guide*).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Half Cell Voltage

enables the calculations of the anode and cathode half-cell potentials using the Nernst equations (Equation 20.13 and Equation 20.14 in the *Fluent Theory Guide*). If this option is disabled, the constant open-circuit voltage is assumed.

Osmotic Drag

when enabled, includes the osmotic drag effects into calculation.

Capillary Pressure

when enabled, includes the capillary pressure effects in the calculation. Capillary pressure facilitates the transport of the gaseous species produced in the catalyst layer through the porous layer into the flow channel.

Reaction Heat

when enabled, takes into account the heat generated by the electrolysis reactions.

Phase Change

when enabled, takes into account the phase change between liquid water and water vapor.

Water Content

when enabled, solves the transport equation of water content using Equation 18.20 and Equation 18.21 in the *Fluent Theory Guide*.

If you want to account for the effect of the membrane hydration level on the electrolyte conductivity, you can define electrolyte conductivity of the electrolyte material using the **DEFINE_PROPERTY** user-defined function. This option is only available for the resolved approach.

Electrical System Setup

In the **Electrical System Setup** group box, you can select one of the following methods to set up the electrical potential boundary conditions at external wall tabs:

Set Up in B.C. Panels

when enabled, Ansys Fluent uses your electrical potential boundary conditions that you specified in the relevant boundary condition dialog boxes.

Specify Total Voltage

allows you to specify **Total Voltage**, which is the cell or stack voltage (V). You have the option to specify the **Total Voltage** as a constant (default) or expression. See Creating and Using Expressions (p. 1049) for more information about the expression specification method.

The **Potential Boundary Condition** and **Electrolyte Potential Boundary Condition** specified in the boundary condition dialog boxes (**Potential** tab) for anode and cathode are ignored. Ansys Fluent automatically sets the potential to zero at the cathode tabs and applies a fixed value of **Total Voltage** at the anode tabs.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Specify Total Current

allows you to specify **Total Current**, which is the total current (Amps) generated by the cell or stack. You have the option to specify the **Total Current** as a constant (default) or expression. See Creating and Using Expressions (p. 1049) for more information about the expression specification method.

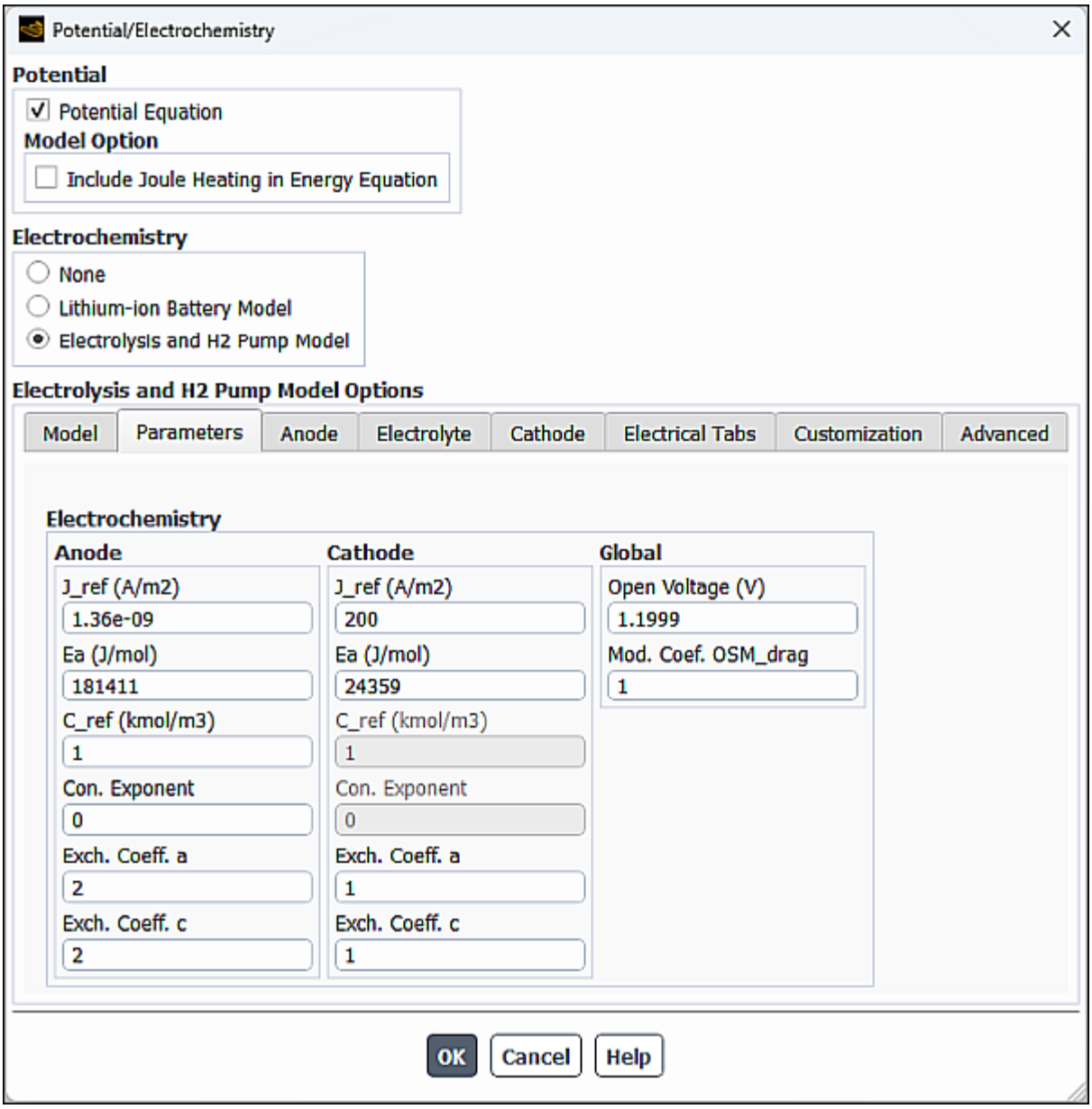
The **Potential Boundary Condition** and **Electrolyte Potential Boundary Condition** specified in the boundary condition dialog boxes (**Potential** tab) for anode and cathode are ignored. Ansys Fluent automatically sets the potential to zero at the cathode tabs and applies a constant flux (current density) at the anode tabs.

30.3.3.2. Specifying Model Parameters (Parameters Tab)

In the **Parameters** tab of the **Potential/Electrochemistry** dialog box, you can set the electrochemistry parameters and model parameters on the anode and cathode sides.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 30.6: The Potential/Electrochemistry Model Dialog Box (Parameters Tab)



In the **Electrochemistry** group box, you can specify the following parameters or leave the default values. on the anode and cathode sides:

J_ref

is the reference exchange current density j_{an}^{ref} and j_{cat}^{ref} in Equation 20.9 and Equation 20.10 in the *Fluent Theory Guide*.

Ea

is the permeation activation energy Ea in Equation 20.9 and Equation 20.10 in the *Fluent Theory Guide*.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

C_ref

is the reference concentration $[A]^{ref}$ and $[C]^{ref}$ in Equation 20.5 and Equation 20.6 in the *Fluent Theory Guide* in units of 1 kg-mol/m^3. Note that for the **PEM Electrolysis** and **H2 Pump** device types, this parameter is available only for anode, while for the **Alkaline Electrolysis** device type, it is available only for cathode.

Con. Exponent

is the concentration dependence γ_{an} and γ_{cat} in Equation 20.5 and Equation 20.6 in the *Fluent Theory Guide*. Note that for the **PEM Electrolysis** and **H2 Pump** device types, this parameter is available only for anode, while for the **Alkaline Electrolysis** device type, it is available only for cathode.

Exch. Coeff. a , Exch. Coeff. c

are the transfer coefficients α_{an}^{an} and α_{cat}^{an} in Equation 20.5 and Equation 20.6 in the *Fluent Theory Guide*.

Std. State E0

are the reversible potentials E_{an}^{0} and E_{cat}^{0} in Equation 20.13 and Equation 20.14 in the *Fluent Theory Guide*. This field appears only when **Half Cell Potentials** option is selected in the **Model** tab.

Entropy

is the reaction entropy ΔS_{an} and ΔS_{cat} in Equation 20.13 and Equation 20.14 in the *Fluent Theory Guide*.

In addition, you can specify the following **Global** parameters:

Open Voltage

is the constant value of open-circuit voltage assigned to anode half-cell potential U_{an}^{0} in Equation 20.11 in the *Fluent Theory Guide*. When **Open Voltage** is specified, the cathode half-cell potential U_{cat}^{0} is set to zero. This parameter appears only when **Half Cell Voltage** is disabled in the **Model** tab.

Standard Temperature

is the reference standard state temperature T^0 used for half-cell potentials computations in the Nernst equations (Equation 20.13 and Equation 20.14 in the *Fluent Theory Guide*). This parameter appears only when **Half Cell Voltage** is enabled in the **Model** tab.

Standard Pressure

is the reference standard state pressure p^0, which is used for half-cell potentials computations in the Nernst equations (Equation 20.13 and Equation 20.14 in the *Fluent Theory Guide*). This parameter appears only when **Half Cell Voltage** is enabled in the **Model** tab.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Vaporization Rate

is the evaporation rate (γ_{evap} in Equation 18.22 in the *Fluent Theory Guide*). This parameter appears only when **Phase Change** is enabled in the **Model** tab.

Condensation Rate

is the condensation rate (γ_{cond} in Equation 18.22 in the *Fluent Theory Guide*). This parameter appears only when **Phase Change** is enabled in the **Model** tab.

Mod. Coef. OSM_drag

is the constant γ in Equation 18.22 in the *Fluent Theory Guide*. The default value is 1.0 for PEM and alkaline electrolysis, and 0.001 for the H2 pump. This parameter appears only when **Osmotic Drag** is enabled in the **Model** tab.

30.3.3.3. Specifying Anode Properties (Anode Tab)

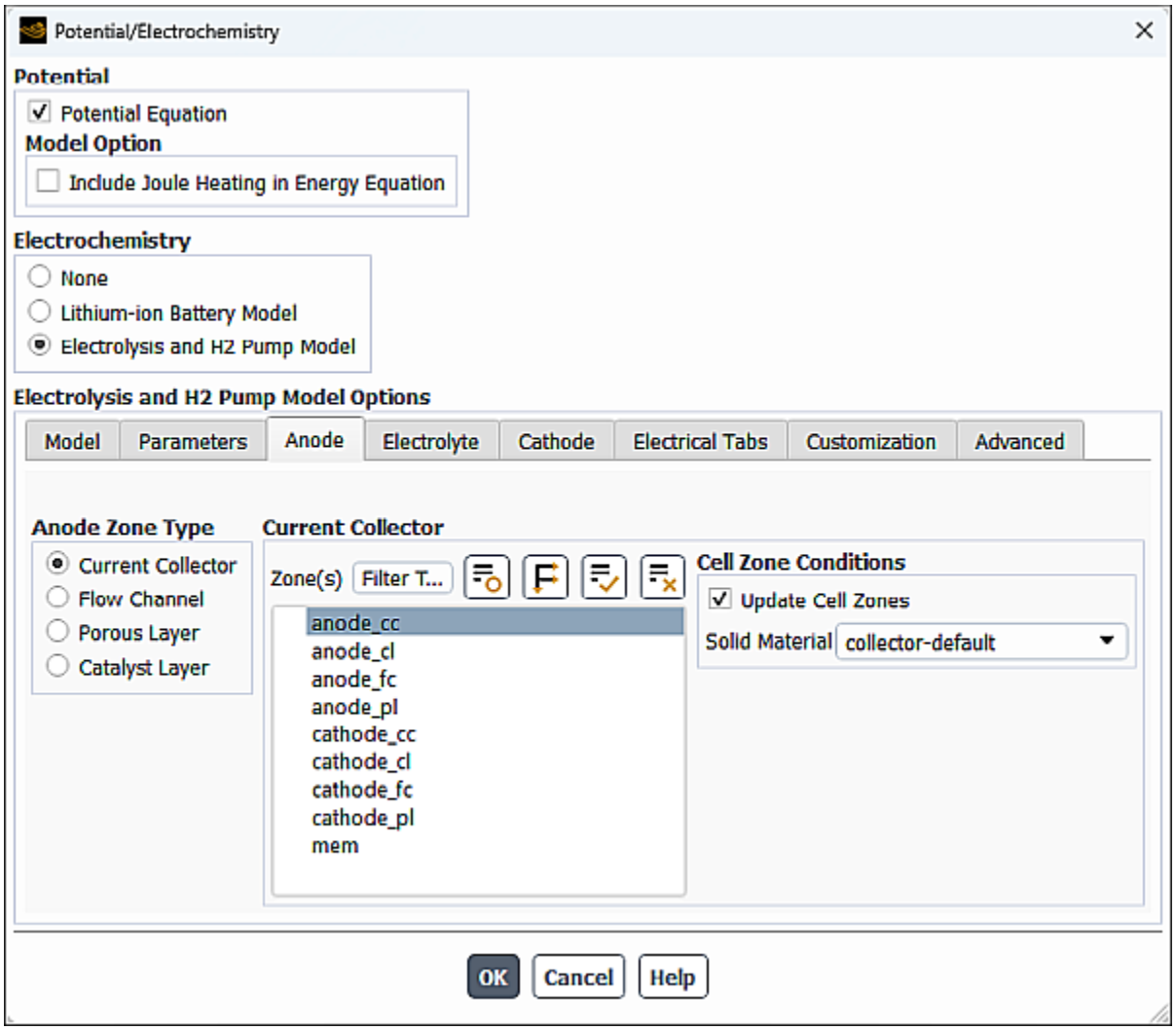
In the **Anode** tab of the **Potential/Electrochemistry** dialog box, you can specify zones and properties of the following zones for the anode portion of the electrolysis and H2 pump model:

- Current collector
- Flow channel
- Porous layer
- Catalyst layer (resolved approach only)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

30.3.3.3.1. Specifying Current Collector Properties for the Anode

Figure 30.7: The Anode Tab of the Potential/Electrochemistry Dialog Box for the Current Collector



1. In the **Anode Zone Type** group box, select **Current Collector**.

2. From the **Zone(s)** multiple-selection list, select a corresponding zone(s).

3. From the **Solid Material** drop-down list (**Cell Zone Conditions** group box), select the appropriate material from the default Ansys Fluent materials (collector-default, porous-default, catalyst-default, and electrolyte-default). The default values of electrical conductivity and electrolyte conductivity for the default Ansys Fluent materials are shown in the table below.

Material Property	collector-default	porous-default	catalyst-default	electrolyte-default
Electrical conductivity (S/m)	1e6	1e4	1e4	1e-16

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

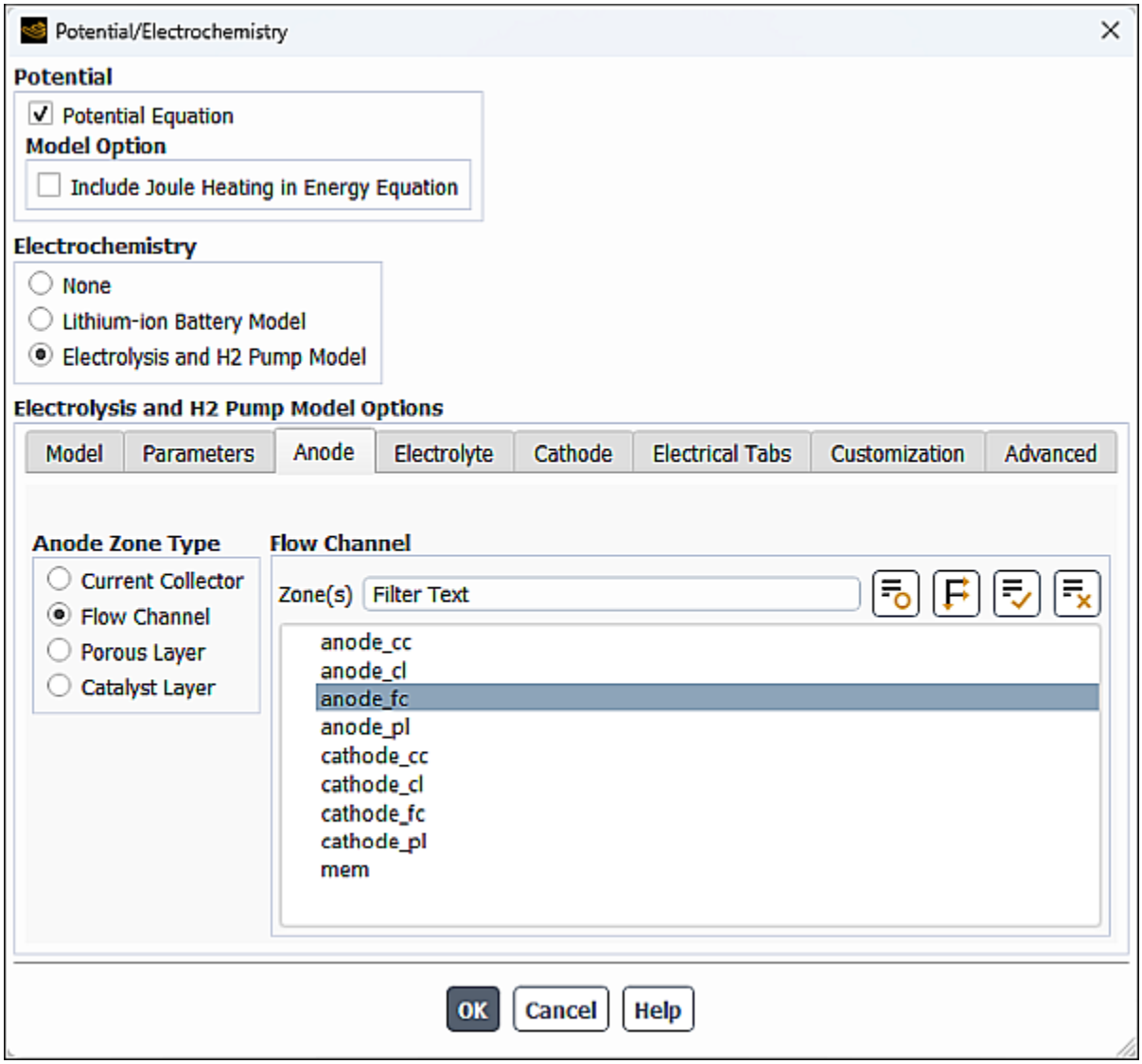
Material Property	collector-default	porous-default	catalyst-default	electrolyte-default
Electrolyte conductivity (S/m)	1e-16	1e-16	2.3677	2.3677

You can use the **Create/Edit Materials** dialog box to customize solid material properties.

The cell zone conditions you specify are applied to all selected zones in the **Zone(s)** list. If you want to set the cell zone conditions for each zone individually (using the **Cell Zone Conditions** task page), you should disable the **Update Cell Zones** option.

30.3.3.3.2. Specifying Flow Channel Properties for the Anode

Figure 30.8: The Anode Tab of the Potential/Electrochemistry Dialog Box for the Flow Channel



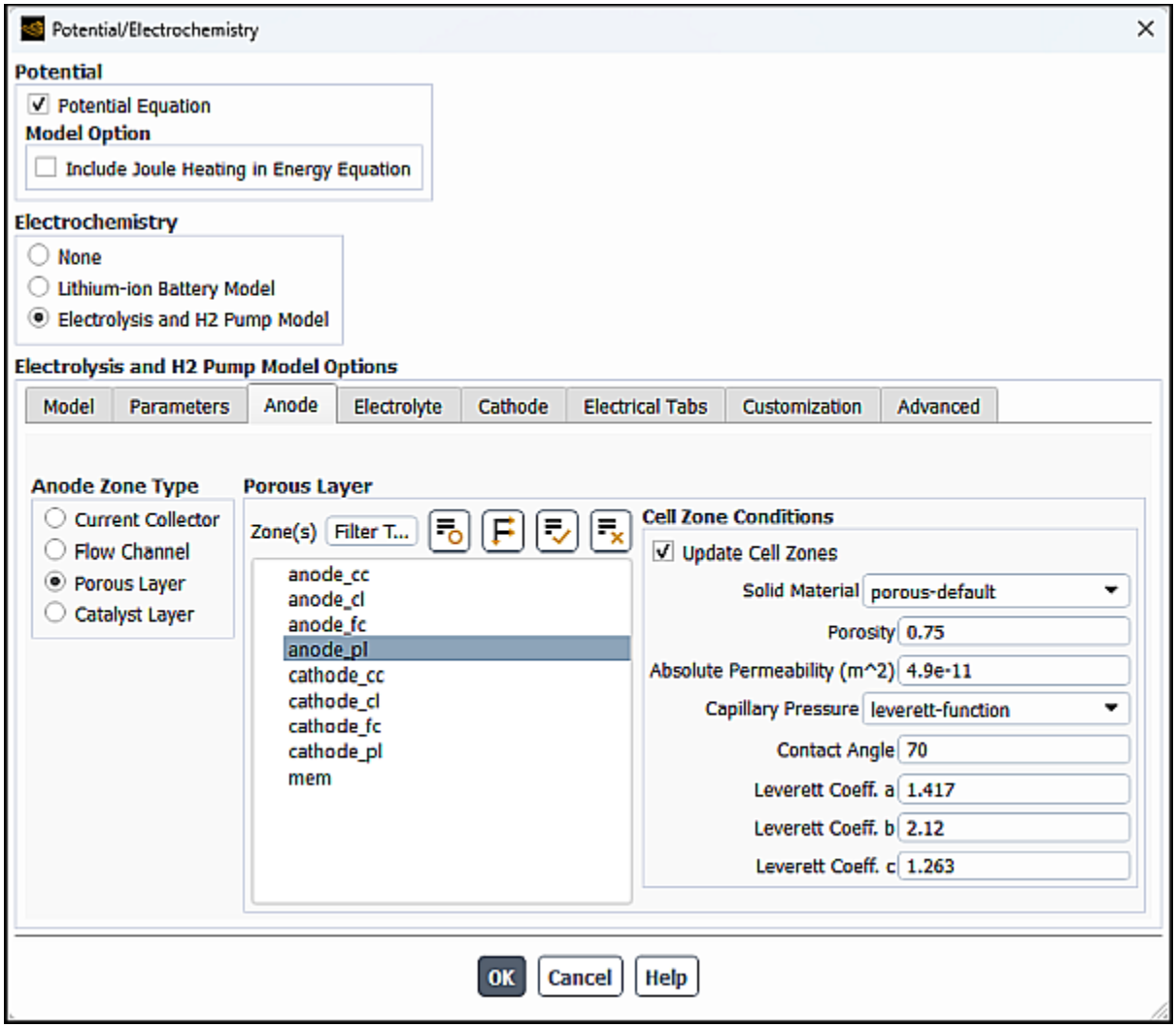
1. In the **Anode Zone Type** group box, select **Flow Channel**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. From the **Zone(s)** multiple-selection list, select an appropriate zone(s).

30.3.3.3.3. Specifying Porous Layer Properties for the Anode

Figure 30.9: The Anode Tab of the Potential/Electrochemistry Dialog Box for the Porous Layer



1. In the **Anode Zone Type** group box, select **Porous Electrode**.

2. From the **Zone(s)** selection list, select a corresponding zone(s).

3. Select the solid material as described in Specifying Current Collector Properties for the Anode (p. 3285).

4. Specify values for the following cell zone parameters:

   Porosity

   is ε in Equation 18.17 in the *Fluent Theory Guide*.

Absolute Permeability

is the inverse of viscous resistance α in Equation 7.4 (p. 1336).

Capillary Pressure

allows you to choose a capillary pressure definition method. You can choose **leverett-function** or your own user-defined function. See `DEFINE_CAPILLARY_PRESSURE` in the *Fluent Customization Manual* for more information.

If you choose **leverett-function**, you need to define the following parameters:

Contact Angle

is Θ in Equation 20.35.

Leverett Func. Coeff_a

is the Leverett function coefficient a in Equation 20.37 (default = 1.417).

Leverett Func. Coeff_b

is the Leverett function coefficient b in Equation 20.37 (default = 2.12).

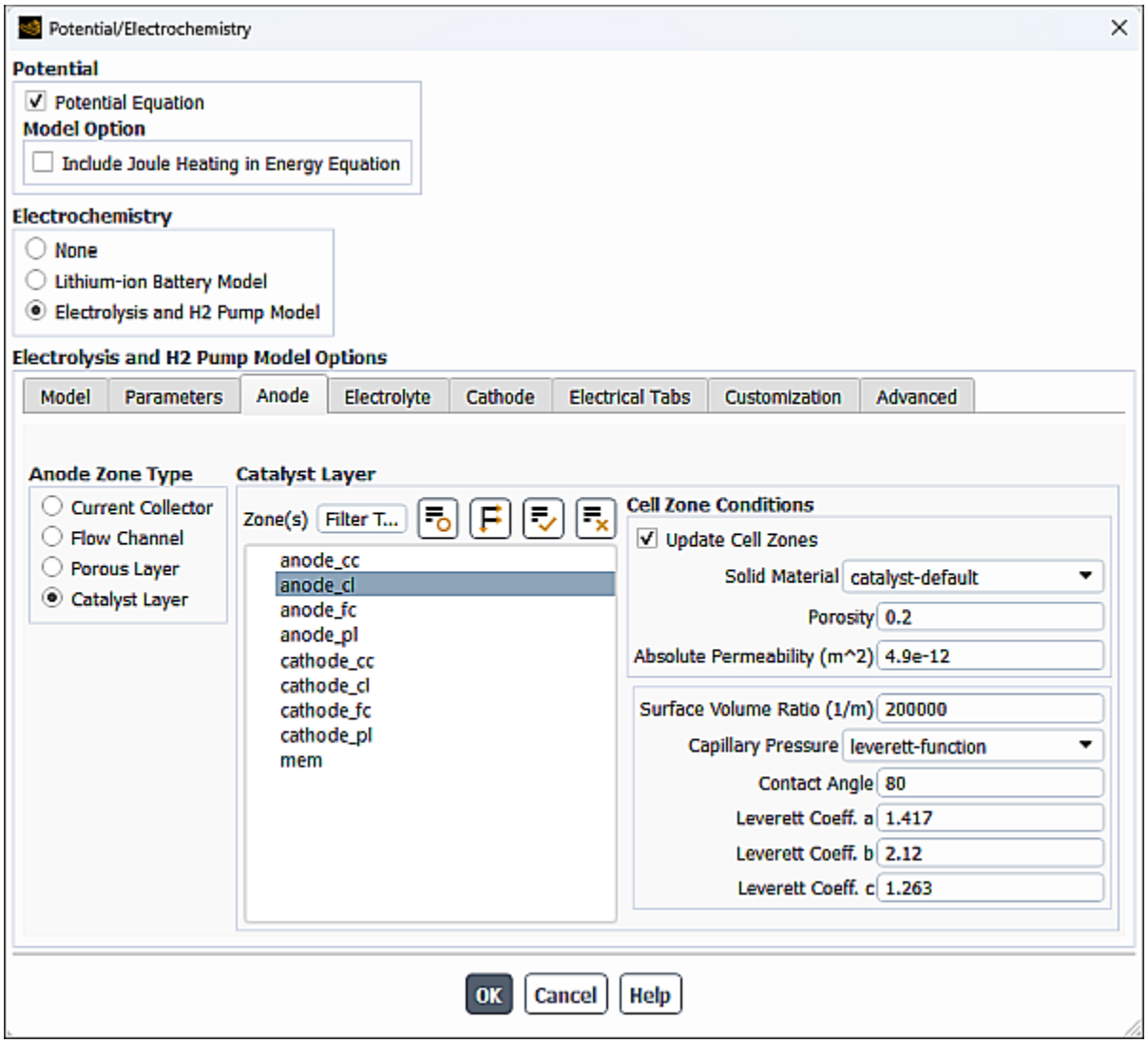
Leverett Func. Coeff_c

is the Leverett function coefficient c in Equation 20.37 (default = 1.263).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

30.3.3.3.4. Specifying Catalyst Layer Properties for the Anode

Figure 30.10: The Anode Tab of the Potential/Electrochemistry Dialog Box with the Catalyst Layer Selected - Resolved Approach



- Resolved Approach

  Settings for the **Catalyst Layer** are similar to those for the **Porous Layer** described in Specifying Porous Layer Properties for the Anode (p. 3287).

  For the catalyst layer, in addition to the cell zone parameters defined in Specifying Porous Layer Properties for the Anode (p. 3287), you can also specify the following parameter:

  Surface Volume Ratio

  is the specific active surface area (ζ_{an} in Equation 20.5 for the anode and ζ_{cat} in Equation 20.6 in the *Fluent Theory Guide* for the cathode).

- Unresolved Approach

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can specify the following parameters:

Surface Volume Ratio

is the specific active surface area of the anode and cathode, respectively (ξ_{an} Equation 18.28 in the *Fluent Theory Guide* and ξ_{cat} in Equation 18.29 in the *Fluent Theory Guide*).

Layer Thickness

is the thickness of the anode catalyst layer (δ_{an} Equation 18.28 in the *Fluent Theory Guide* for the anode catalyst layer, and δ_{cat} in Equation 18.29 in the *Fluent Theory Guide* for the cathode catalyst layer).

Catalyst Electrical Conductivity

is the electrical conductivity of the catalyst layer (σ_{an} and σ_{cat} in Equation 18.27 in the *Fluent Theory Guide* for the anode and cathode, respectively).

30.3.3.4. Specifying Electrolyte/Membrane Properties (Electrolyte Tab)

In the **Electrolyte** tab of the **Potential/Electrochemistry** dialog box, you can specify settings of the electrolyte/membrane portion of the electrolysis device. Settings are different for the resolved and unresolved MEA layer methods. Further details are provided in the following sections:

30.3.3.4.1. Resolved MEA Layer Method

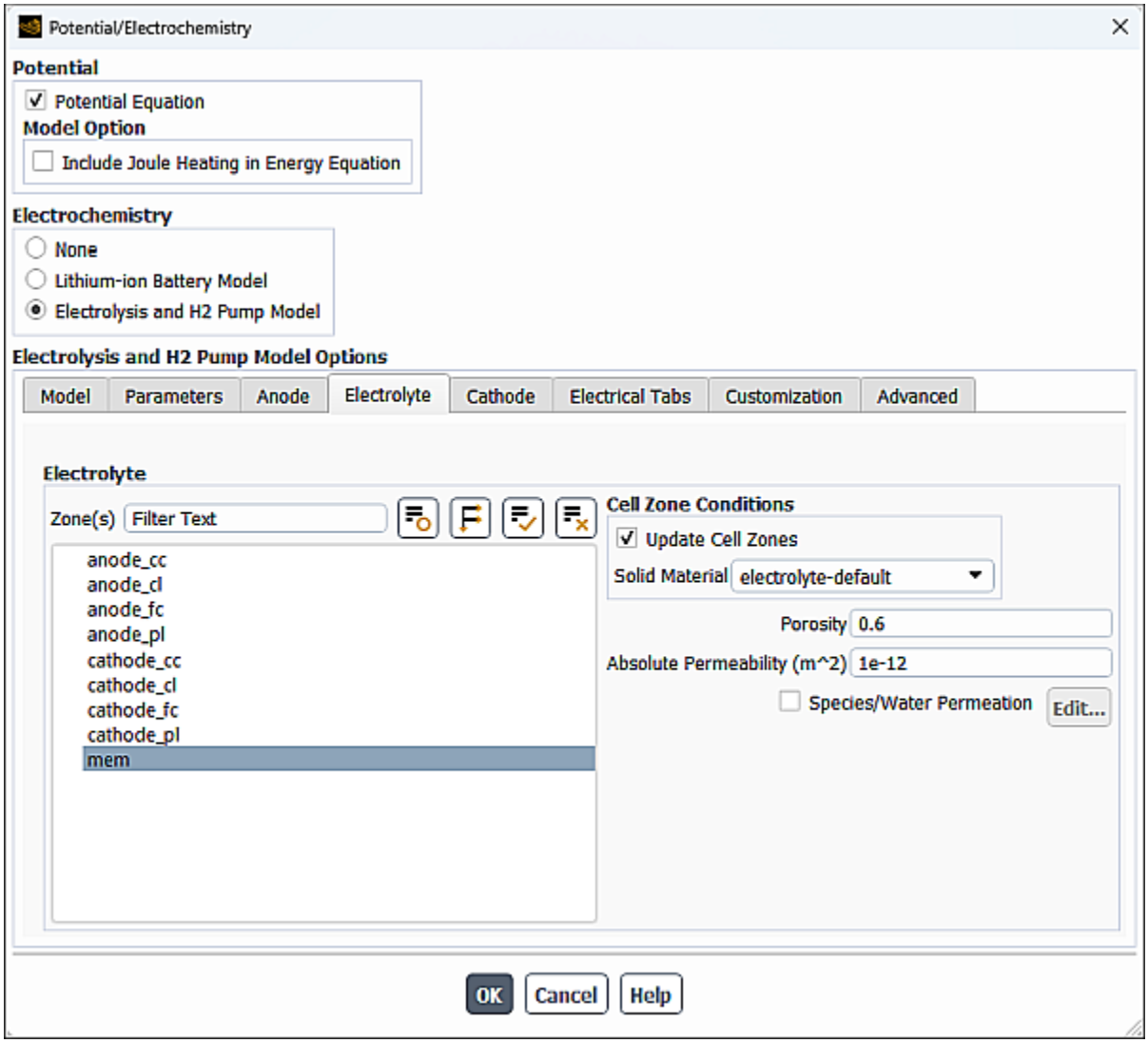
30.3.3.4.2. Unresolved MEA Layer Method

30.3.3.4.1. Resolved MEA Layer Method

When **Resolved** is selected as the **MEA Layer and H2 Pump Method** (**Model** tab), you can specify zones and properties of the electrolyte/membrane portion of the electrolysis device in the **Electrolyte** tab of the **Potential/Electrochemistry** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 30.11: The Electrolyte Tab of the Potential/Electrochemistry Dialog Box - Resolved MEA Layer



1. From the **Zone(s)** selection list, select a corresponding zone(s).

2. Select the solid material as described in Specifying Current Collector Properties for the Anode (p. 3285).

3. Specify the following parameters for the selected cell zone:

   Porosity

   is ε in Equation 18.17 in the *Fluent Theory Guide*.

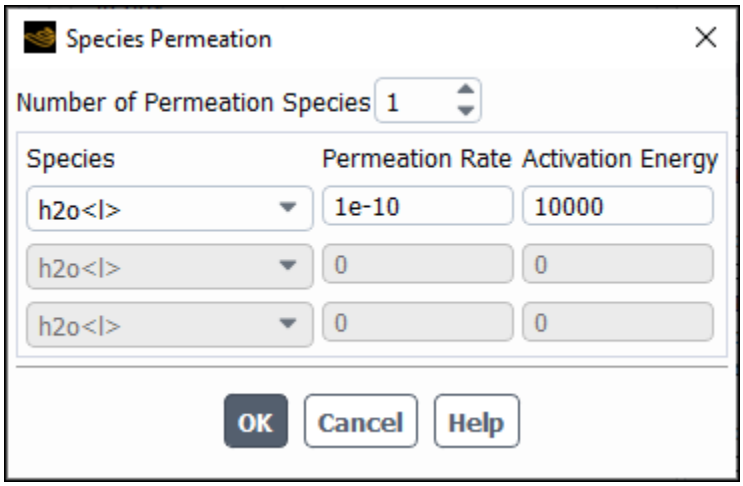
   Absolute Permeability

   is the inverse of viscous resistance α in Equation 7.4 (p. 1336).

4. If you want to account for species or liquid water crossing the membrane from the higher to the lower pressure side, enable **Species/Water Permeation** and click **Edit...**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 30.12: The Species Permeation Dialog Box



a. In the **Species Permeation** dialog box that opens, enter the **Number of Permeation Species**. Use the arrows to change the value, or type in the value and press **Enter**.

b. Select each species you want to consider in the **Species** drop-down list and specify the following parameters:

Permeation Rate

is the permeation rate constant α in Equation 18.24 in the *Fluent Theory Guide*.

Activation Energy

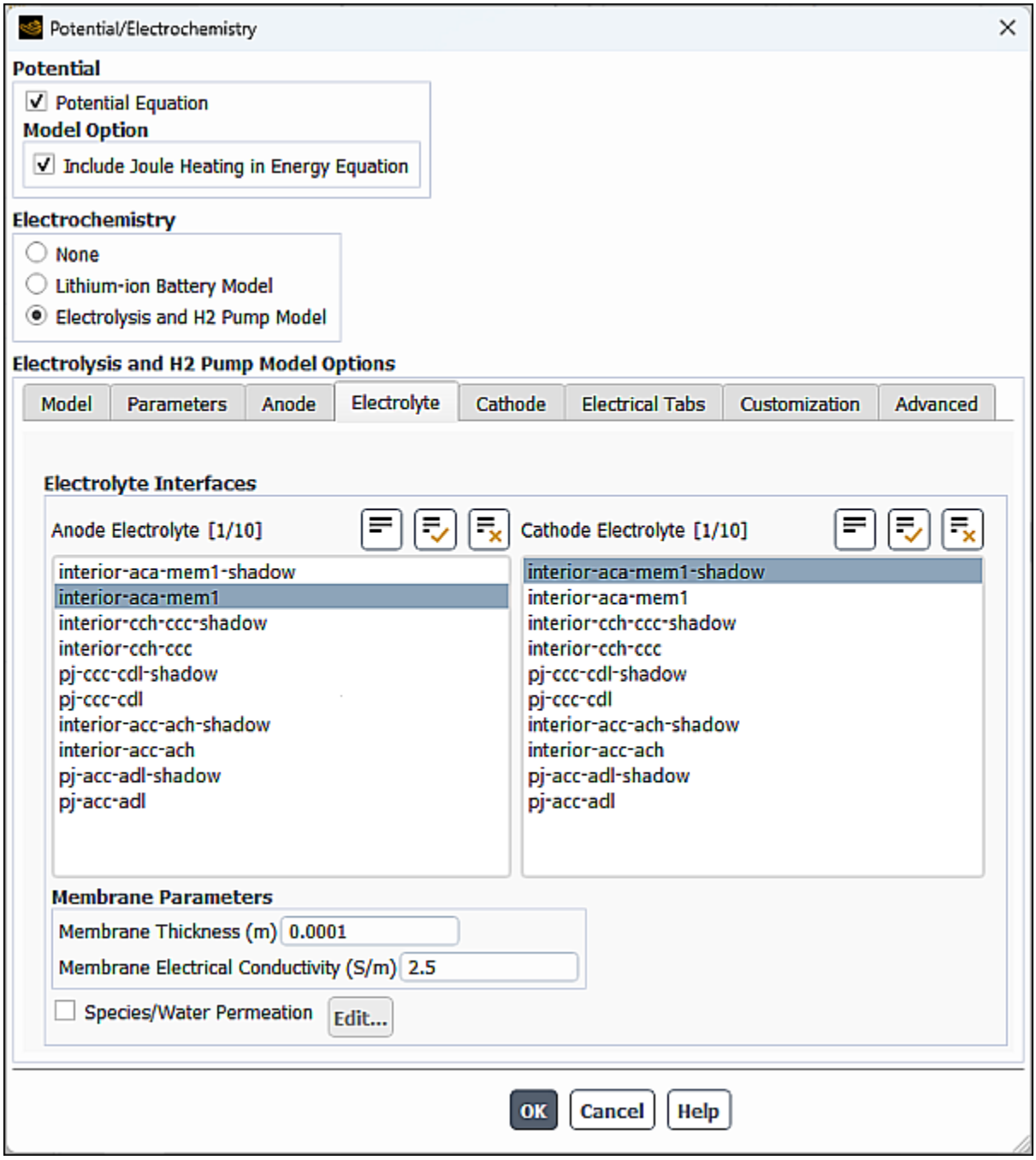
is the permeation activation energy Ea in Equation 18.24 in the *Fluent Theory Guide*.

30.3.3.4.2. Unresolved MEA Layer Method

When **Unresolved 0D** is selected as the **MEA Layer and H2 Pump Method** (**Model** tab), you can specify electrolyte interfaces and MEA parameters of the electrolyte/membrane portion of the electrolysis device in the **Electrolyte** tab of the **Potential/Electrochemistry** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 30.13: The Electrolyte Tab of the Potential/Electrochemistry Dialog Box - Unresolved MEA Layer



1. In the **Anode Electrolyte** and **Cathode Electrolyte** selection lists, select pairs of wall/wall shadow. The solver uses the selected pairs to determine where the electrochemistry sources are added.

2. In the **MAE Parameters** group box, specify the following parameters:

   Membrane Thickness

   is the conductivity of the membrane (δ_{ele} in Equation 18.27 in the *Fluent Theory Guide*).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Membrane Electrical Conductivity

is the conductivity of the membrane (σ_{ele} in Equation 18.27 in the *Fluent Theory Guide*).

3. If you want to account for species or liquid water crossing the membrane from the higher to the lower pressure side, enable **Species/Water Permeation** and specify the species permeation as described in Resolved MEA Layer Method (p. 3290).

30.3.3.5. Specifying Cathode Properties (Cathode Tab)

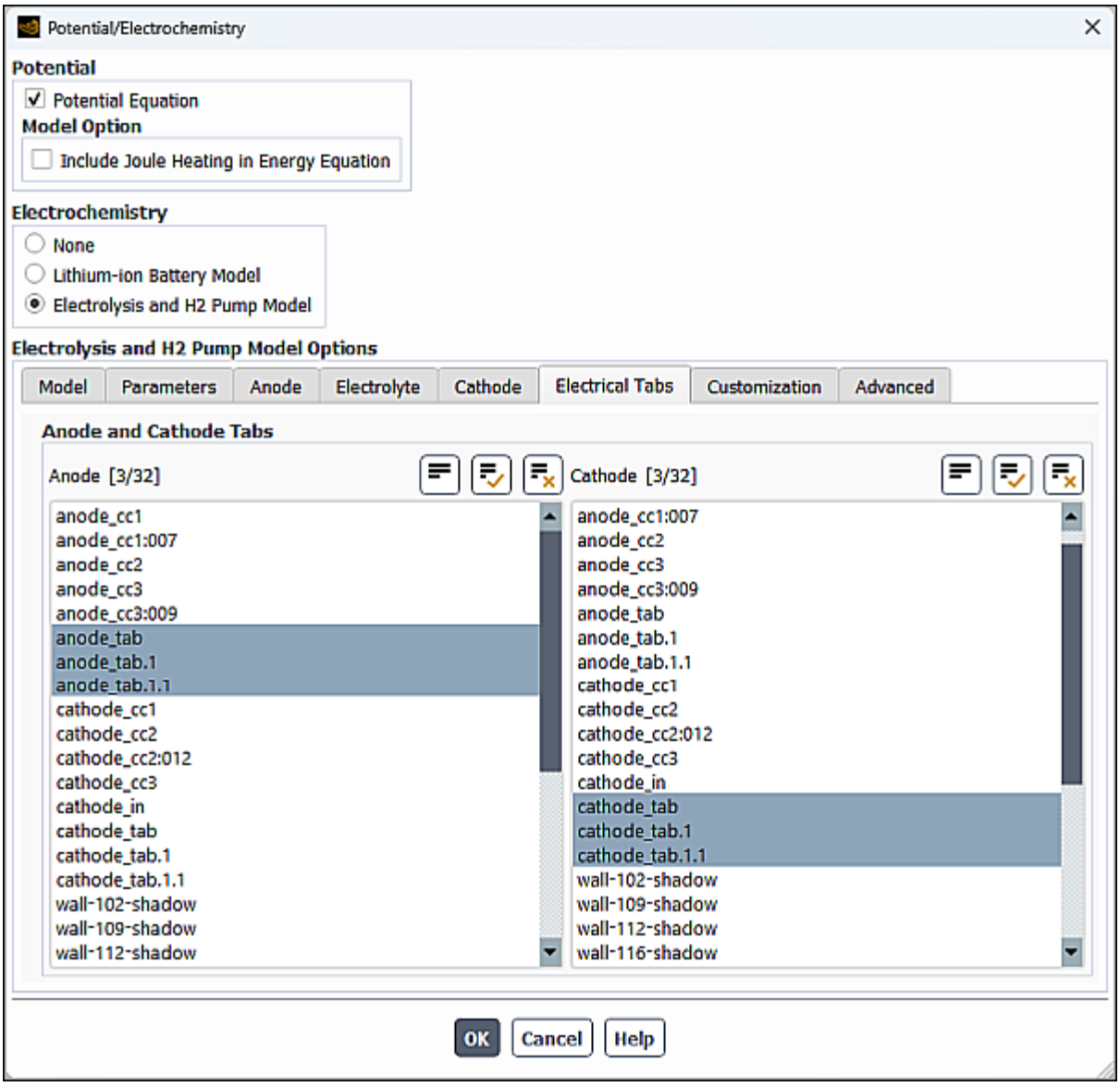
The procedures for specifying the properties for the current collector, flow channel, porous layer, and catalyst layer on the cathode side are similar to those on the anode side. For specific steps, refer to Specifying Anode Properties (Anode Tab) (p. 3284) and substitute “cathode” for “anode” where appropriate.

30.3.3.6. Setting the External Electrical Tabs (Electrical Tabs Tab)

In the **Electrical Tabs** tab of the **Potential/Electrochemistry** dialog box, you can specify **Anode and Cathode Tabs**. The anode and cathode tabs are the external wall surfaces where electrical potential boundary conditions are applied using the method specified in the **Electrical System Setup** group box (**Model** tab).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 30.14: The Electrical Tabs Tab of the Potential/Electrochemistry Dialog Box



30.3.3.7. (Customization Tab)

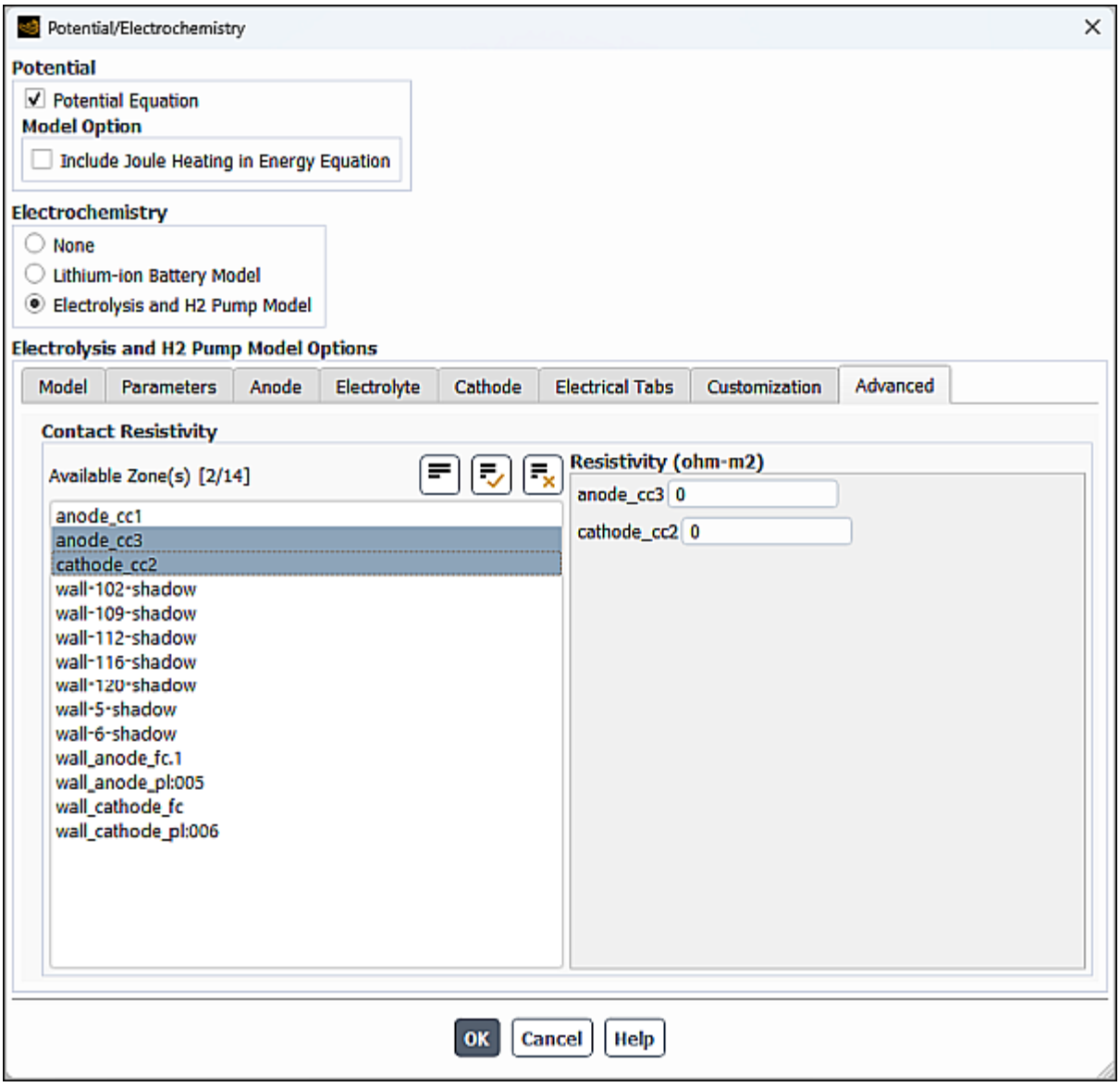
In the **Customization** tab of the **Potential/Electrochemistry** dialog box, you can use the `DEFINE_ELECTROLYSIS_ECHEM_RATE` and `DEFINE_ELECTROLYSIS_RELATIVE_PERMEABILITY` user-defined functions to customize the transfer currents and relative permeability. See `DEFINE_ELECTROLYSIS_ECHEM_RATE` and `DEFINE_ELECTROLYSIS_RELATIVE_PERMEABILITY` in the *Fluent Customization Manual* for more information.

30.3.3.8. Setting Advanced Options (Advanced Tab)

In the **Advanced** tab of the **Potential/Electrochemistry** dialog box, you can specify **Contact Resistivity** at the interface between cell zones.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 30.15: The Advanced Tabs Tab of the Potential/Electrochemistry Dialog Box



1. From the **Available Zone(s)** selection list, select any number of corresponding interfaces. These are face zones over which a jump in electrical potential is caused by imperfect conduction.

2. Specify a value for the **Resistivity** for each specified zone.

   Note that at porous jump and wall/wall-shadow interfaces, the contact resistivity is applied at each face. Therefore, you need to specify the resistivity only at one of the interface walls, otherwise the actual resistance will be doubled at the interface.

30.3.4. Solution Strategies for the Electrolysis and H2 Pump Model

It may be useful to disable **Include Joule Heating in Energy Equation** (**Potential** group box) and **Electrochemistry Sources** (**Model** tab) in the **Potential/Electrochemistry** dialog box for the first few (approximately 10-20) iterations after initialization. This allows the two potentials to adjust from their initial values to more physical values, avoiding the possibility of extreme electrochemical reactions and electric currents that would in turn adversely impact the solution.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

30.4. Postprocessing Electric Potential Field and Li-ion Battery Quantities

For electric potential field or Li-ion battery calculations, you can generate graphical plots or alphanumeric reports of the following items. These quantities are available in the variable selection drop-down lists that appear in postprocessing dialog boxes.

- Electric potential field:
  - In the **Properties...** category:
    - → **Electric Conductivity**
  - In the **Potential...** category:
    - → **Electric Potential**
    - → **Electric Conductivity**
    - → **Electric Current Magnitude**
    - → **Joule Heat Source**
    - → **X Current**
    - → **Y Current**
    - → **Z Current**
    - → **dPotential/dx**
    - → **dPotential/dy**
    - → **dPotential/dz**
- Li-ion battery (in the **Lithium...** category):
  - **Lithium Concentration**
  - **Magnitude of Lithium Mass Flux Vector**
  - **X Lithium Mass Flux**
  - **Y Lithium Mass Flux**
  - **Z Lithium Mass Flux**
- Electrolysis (in the **Potential...** category):
  - **Electrolyte Potential**
  - **Over Potential** (mixture only)
  - **Transfer Current** (mixture only)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Osmotic Drag** (mixture only)
- **Water Phase Change** (phases only)
- **Water Content** (mixture only)
- **Capillary Pressure** (mixture only)

For the electric potential field, you can generate vector plots of the electric current fields by selecting **Electric Current** in the **Vectors of** drop-down list in Vectors Dialog Box (p. 5655).

For the Li-ion battery model, you can generate vector plots of the lithium mass flux by selecting **Lithium Mass Flux** in the **Vectors of** drop-down list in Vectors Dialog Box (p. 5655).

See Field Function Definitions (p. 4167) for the definitions of these quantities.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 31: Modeling Batteries

This chapter provides information about using the battery models available in Ansys Fluent. Additional information about the model is provided in the following sections:

31.1. Introduction

31.2. Using the MSMD-Based Battery Models

31.1. Introduction

The application of lithium ion batteries has been rapidly expanding from electric appliances and electronic devices to hybrid electric vehicles (HEVs) and electric vehicles (EVs), due to their high energy density. The main concerns when designing a Li-ion battery are its performance, life, and safety. The Ansys Fluent battery models allow simulating a single battery cell or a battery pack using CFD technology to study their thermal and electrochemical behavior.

Note that the Fluent Tutorials provides tutorials that illustrate how to use the Ansys Fluent battery models.

This chapter contains the following information:

31.1.1. Overview

31.1.2. General Procedure

31.1.1. Overview

Ansys Fluent has adopted the multi-scale multi-domain (MSMD) methodology in battery modeling. In this method, the whole battery is treated as an orthotropic continuum; thus, the mesh is no longer constrained by the micro-structure of the battery. Two potential equations are solved in the battery domain. To fit various analysis needs, the model includes three electrochemical submodels of different levels of complexity:

- Newman, Tiedemann, Gu, and Kim (NTGK) and Direct Current Internal Resistance (DCIR) model
- Equivalent Circuit model (ECM)
- Newman's Pseudo-2D (P2D) model

The model offers you the flexibility to study the physical and electrochemical phenomena that extend over many length scales in battery systems of various arrangements.

The MSMD method is a comprehensive method that allows the distributed heat source treatment to be accounted in a simulation. Moreover, Ansys Fluent also provides several uniform heat source treatment within the battery model framework, including CHT coupling, FMU-CHT coupling and the Circuit Network method (see Battery Solution Methods in the *Fluent Theory Guide*).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.1.2. General Procedure

The following describes an overview of the procedure required in order to use the Battery Model in Ansys Fluent.

1. Start Ansys Fluent.
2. Read the mesh file.
3. Scale the grid, if necessary.
4. Load the module and use the **Battery Model** dialog box to define the battery model parameters.
5. Define material properties.
6. Set the operating conditions.
7. Set the boundary conditions.
8. Start the calculations.
9. Save the case and data files.
10. Process your results.

Important:

Note that the majority of this manual describes how to set up the Ansys Fluent Battery Model using the graphical user interface. You can also perform various tasks using the text user interface.

31.2. Using the MSMD-Based Battery Models

This section describes how to use Ansys Fluent modeling capabilities based on the multi-scale multi-domain (MSMD) method. Only the procedural steps related to battery modeling are shown here. For information about inputs related to other models used in conjunction with the battery model, see the appropriate sections for those models in the Ansys Fluent Fluent User's Guide (p. 1). For more details on the modeling approach, refer to Battery Model in the *Fluent Theory Guide*.

Information about using the model is presented in the following sections:

31.2.1. Limitations
31.2.2. Geometry Definition
31.2.3. Setting up the Battery Model
31.2.4. Using Parameter Estimation Tools
31.2.5. Initializing the Battery Model
31.2.6. Modifying Material Properties
31.2.7. Solution Controls for the MSMD Battery Model
31.2.8. Predefined Report Definitions for the Battery Model

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.2.9. Postprocessing the MSMD Battery Model

31.2.1. Limitations

Note the following limitations when using the MSMD Battery model:

- For battery pack simulations, all batteries in the pack must be identical initially, having the same battery type with the same initial conditions.
- For battery pack simulations, all batteries in the pack must have the nPmS connection pattern; that is, n batteries connected in parallel, and then m such units connected in series.
- A conductive zone must be continuous. Disconnected zones cannot be grouped as one entity.
- System positive and negative tabs must be continuous surfaces. Disconnected faces cannot be grouped together as one entity.
- The positive and negative tabs must be connected by either the real tab/busbar conductive volumes or the virtual connections defined through a file.

31.2.2. Geometry Definition

In the MSMD and Cell Network frameworks, the negative and positive current collectors, anode, and cathode are not resolved, and the battery is regarded as a homogeneous body. The computational mesh does not have to be adjusted to all layers present in the battery domain. The following zones have to be identified when creating the battery mesh:

- Cell
- Tab
- Busbar (for battery pack cases with real battery connections)

Electro-chemical reactions occur in cell zones, which are defined in the **Active Components** selection list in the **Battery Model** dialog box (**Conductive Zones** tab). Tab and busbar zones only conduct electric current and are defined in the **Passive Components** selection list. See Specifying Conductive Zones (p. 3317) for details.

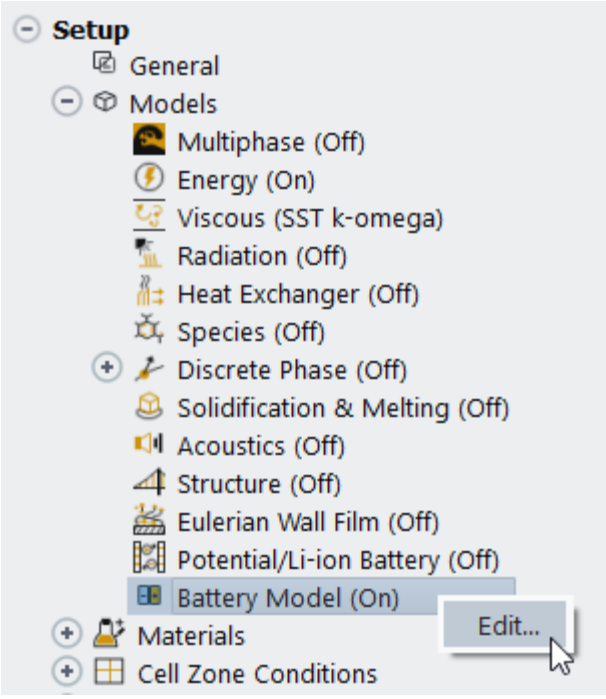
31.2.3. Setting up the Battery Model

To enable the battery model, in the **Outline View** under the **Models** branch, right-click **Battery Model** and click **Edit...** in the menu that opens.

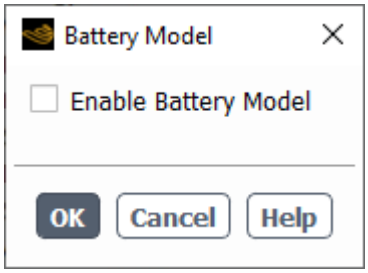
Setup → Models → Battery Model → Edit...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.1: The Battery Model Option in the Outline View



In the **Battery Model** dialog box that opens, select the **Enable Battery Model** check box to activate the model.



The **Battery Model** dialog box expands to reveal additional model options and solution controls.

The inputs for the battery model are entered using the following tabs:

Model Options

contains the general model settings to define problems using the battery model.

Conductive Zones

allows you to select zones for the active, tab and busbar components.

Electric Contacts

allows you to define the contact surface and external connectors.

Model Parameters

is where you specify the parameters to be used for solving the model equations.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UDF

is where you can hook your user-defined functions.

Advanced Options

allows you to quickly check the selected electrochemical submodel behavior in a standalone mode, and also to use various advanced modeling capabilities. See Specifying Advanced Options (p. 3351) for more information.

Note:

When setting up the model, you can click the **Apply** button to save your input values and your choices for your model. To restore the last saved settings for all tabs, click the **Reset** button.

For additional information, see the following sections:

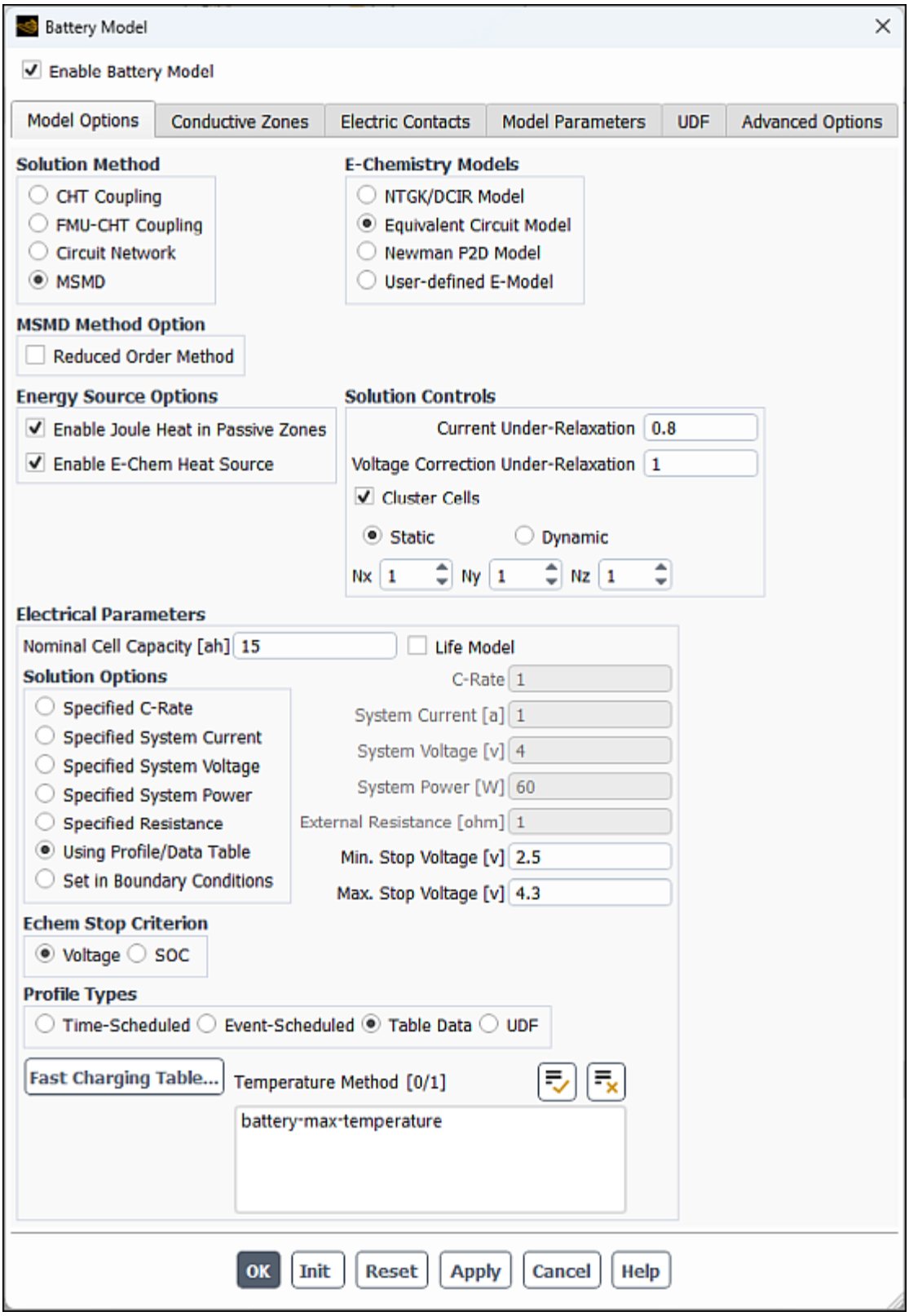
31.2.3.1. Specifying Battery Model Options
31.2.3.2. Specifying Conductive Zones
31.2.3.3. Specifying Electric Contacts
31.2.3.4. Specifying Battery Model Parameters
31.2.3.5. Hooking User-Defined Functions
31.2.3.6. Specifying Advanced Options
31.2.3.7. Specifying External and Internal Short-Circuit Resistances
31.2.3.8. Setting a Battery Swelling Case

31.2.3.1. Specifying Battery Model Options

In the **Model Options** tab, you can select the electrochemical battery submodel and set various model options, solution controls, and electrical parameters.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.2: The Battery Model Dialog Box (Model Options Tab)



1. Select the solution method.

   For **Solution Method**, if you select:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **CHT coupling**: A uniform heat source will be distributed within each battery's active zone. You need to specify the heat source directly. None or one potential equation will be solved, depending on whether Joule heating in passive zones is included or not. See CHT Coupling Method in the *Fluent Theory Guide* for more information about this method.
- **FMU-CHT coupling**: A uniform heat source will be distributed within each battery's active zone. The heat source is computed from the functional mockup unit (FMU). None or one potential equation will be solved, depending on whether Joule heating in passive zones is included or not. See FMU-CHT coupling method in the *Fluent Theory Guide* for more information about this method.
- **Circuit Network**: A uniform heat source will be distributed within each battery's active zone, the heat source is computed from an electric circuit network in Fluent. None or one potential equation will be solved, depending on whether Joule heating in passive zones is included or not. See Circuit Network Solution Method in the *Fluent Theory Guide* for more information about this method.
- **MSMD**: If the **Reduced Order Method** option is not enabled in the **MSMD Method Option** group box, two potential equations will be solved during each iteration in the simulation. See MSMD Solution Method in the *Fluent Theory Guide* for more information about this method.

2. If you want to obtain a faster solution, select **Reduced Order Method** in the **MSMD Method Option** group box. This will enable the solver to run faster using the reduced order method (ROM). You can use this option if the two conditions listed in Reduced Order Solution Method (ROM) are met. Note that, prior to using the ROM, you must first run a simulation without this option for at least three time steps. Ansys Fluent will use these results as reference potential fields for the ROM method.

3. If you have selected **Circuit Network** or **Reduced Order Method**, set **Number of Sub-Steps/Time Step**. Using this value, you can specify the ratio of the time steps for thermal and electro-chemistry calculations. For example, if the **Number of Sub-Steps/Time Step** is set to **10**, the solver will use one tenth of the CFD time step for the electro-chemistry calculation.

   Important:

   The internal short circuit treatment requires two potential equations to be solved. For this reason, it can only be used with the MSMD solution method, but not with the ROM method.

4. Select the electrochemistry model. The following submodels are available in Ansys Fluent under the **E-Chemistry Models** group box:

   NTGK/DCIR Model

   is an empirical model. This is the simplest and the least expensive model. It also requires the fewest model inputs. The model uses the same NTGK method as that used by the Single-Potential Empirical model. However, in this case, it solves two potential equations continuously instead of one potential equation with a jump condition at the separator over all conductive zones. The model cannot accurately capture the battery's dynamic (inertial) response.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Equivalent Circuit Model

is a semi-empirical model. It has limited capability for predicting the battery dynamic behavior.

Newman P2D Model

is the most comprehensive physics-based battery model. It is computationally expensive, and it requires extensive user input of battery-related fundamental information.

User-defined E-Model

is the user-specified electrochemical submodel. To be able to use this option in the Fluent MSMD method framework, you must provide the `void user_defined_echem_model()` function in the customizable `cae_user.c` source code file.

5. Optionally, you can enable the following **Energy Source Options**:

   - **Enable Joule Heat in Passive Zones** (default) in order to include the Joule heating source in the thermal energy Equation 19.3 in the battery passive conducting zones, including tabs and busbars,
   - **Enable Joule Heat in Active Zones** (default) in order to include the Joule Heating source in the thermal energy equation Equation 19.3.
   - **Enable E-Chem Heat Source** (default) in order to include the heat source due to electrochemistry in the thermal energy equation Equation 19.3.

6. Specify the following settings in the **Solution Controls** group box:

   a. Set **Current Under-Relaxation Factor**. The volumetric current density used in the two potential equations are under-relaxed according to this factor. The default value of `0.8` may be sufficient for most cases. The values in the range of 0.8–1.0 are recommended.

   b. Set **Voltage Correction Under-Relaxation Factor**. The default value of 1.0 is acceptable for most cases. You should reduce this value only if the convergence difficulty occurs during the solution of the potential equations.

   c. (Equivalent Circuit Model and Newman P2D Model) Select the **Cluster Cells** check box to enable cell clustering and then define it using one of the following methods:

      - **Static**

        When you select **Static**, you can specify the number of clusters $\mathbf{N_x}$, $\mathbf{N_y}$, and $\mathbf{N_z}$ in X, Y, and Z directions respectively. The solver will divide the active zone of every battery cell into $\mathbf{N_x}$ x $\mathbf{N_y}$ x $\mathbf{N_z}$ clusters and solve the electrochemical sub-model in each cluster using the cluster-averaging information.

        The static clustering method is purely geometry-based. Clusters are formed before a simulation is run and they remain fixed in the simulation.

        The default values for $\mathbf{N_x}$, $\mathbf{N_y}$, and $\mathbf{N_z}$ are 1, 1, and 1, respectively. For the initial run, the default settings are recommended.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Dynamic**

  When you select **Dynamic**, you can define the **Number of Clusters** and specify the **Target Variable**, which could be **voltage** or **temperature**. You can also define your own target variable by writing a `DEFINE_BATTERY_CLUSTER` UDF. See `DEFINE_BATTERY_CLUSTER` in the *Fluent Customization Manual* for details.

  The dynamic clustering is done before the beginning of every time step and the clusters are formed dynamically during the simulation. The solver finds the range of the target variable in the active zone of a battery cell, and then divide this range into the specified number of clusters. Each control volume in the battery cell is grouped into a cluster according to its target variable value.

7. In the **Electrical Parameters** group box, specify the following operating conditions:

   - Set the **Nominal Cell Capacity** (the capacity of the battery cell).

   - If you want to model the battery capacity changes over time and over repeated discharging and charging cycles, enable the **Life Model** option and then set up the life model as described in Setting the Life Model (p. 3308).

   - For the **Solution Options**, you can:

     – Select **Specified C-Rate** and then specify **C-Rate**, which is the hourly rate at which a battery is discharged. A positive value means discharging C-rate and a negative value means charging C-rate. In this case, the total current at the cathode tabs is fixed as the product of C-Rate and Nominal Capacity, while the electrical potential is anchored at zero on the anode tabs. Note that the computation of current from the C-rate is always based only on the single battery's nominal capacity and does not account for the battery's connection information. For example, in a 2P2S module case, if you want to completely drain the fully charged module in one hour, you need to set the C-rate to 2.

     – Select **Specified System Current** and then specify **System Current** applied at the cathode tabs. A positive value means discharging current and a negative value means charging current.

     – Select **Specified System Voltage** and then specify **System Voltage** applied to the cathode tab. The anode tab has a voltage of 0 V.

     – Select **Specified System Power** and then specify **System Power**. If you specify the total power output from your battery, the Fluent solver will ensure that the product of the system current and voltage equals the system power. A positive value means discharging power output and a negative value means charging power input.

     – Select **Specified Resistance** and then specify **External Resistance** in an external short-circuit simulation. If you specify the external electric resistance in Ohm, the Fluent solver will ensure that the ratio of the system voltage to the system current equals the specified external resistance.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - Select **Set in Boundary Conditions** and then set the UDS boundary conditions directly, for example, the voltage value or the current value (specified flux), using the **Boundary Conditions** task page in Ansys Fluent for the specific face zone.

  - Select **Using Profile/Data Table** and then specify a profile or data table as described in Specifying a Profile or Data Table (p. 3311).

- For **Echem Stop Criterion**, you can:

  - Select the **Voltage** method and specify **Min Stop Voltage** and **Max Stop Voltage**, or

  - Select the **SOC** method and specify **Min Stop SOC** and **Max Stop SOC**.

  In a battery simulation, the stop criterion is checked after every time step. For the voltage method, battery's cell voltage is checked against the minimum and maximum stop voltage. For the SOC method, the SOC level in a battery is checked. Once the stop criterion is met, the simulation stops automatically.

31.2.3.1.1. Setting the Life Model

If you have selected the **Life Model** option (**Electrical Parameters** group box), you must specify the following life model parameters:

Calender Time

is the storage time (t in Equation 19.45 in the *Fluent Theory Guide*). You can use any units that are consistent with the kinetics quantities you specify in the **Calendar Life Parameters** dialog box (see Figure 31.3: The Calendar Life Parameters Dialog Box (p. 3309)).

Cycle Number

is the number of cycles that the battery has experienced so far (N_{cycle} in Equation 19.46 in the *Fluent Theory Guide*).

Operation Temperature

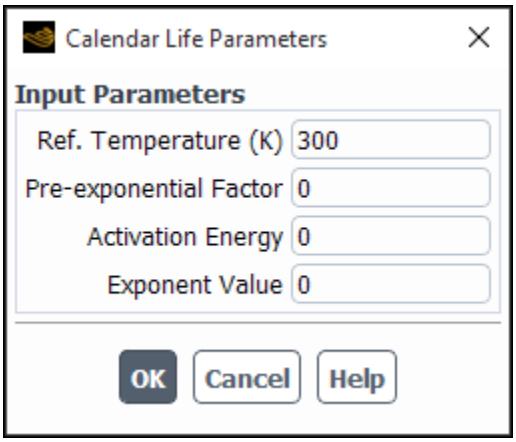
is the average temperature to which the battery has been exposed over time and over its previous discharging/charging cycle (T in Equation 19.45 and Equation 19.46 in the *Fluent Theory Guide*). Note that this temperature is not the same as the battery temperature during CFD simulations.

Calendar Life Parameters

are the calendar-life-related parameters that can be specified in the **Calendar Life Parameters** dialog box that opens when you click the **Calendar Life - Input Parameters...** button.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.3: The Calendar Life Parameters Dialog Box



The following **Input Parameters** are available:

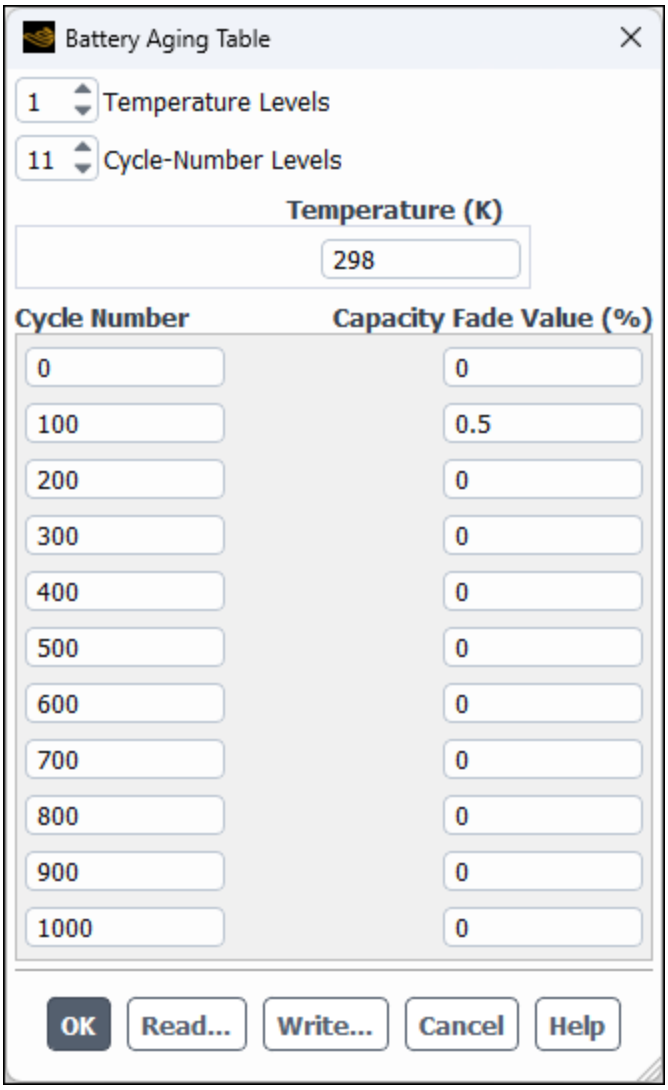
- **Ref. Temperature**: Is the reference temperature under which these kinetics parameters are obtained (T_o in Equation 19.45 in the *Fluent Theory Guide*).
- **Pre-exponential Factor**: Is A in Equation 19.45 in the *Fluent Theory Guide*.
- **Activation Energy**: Is E in Equation 19.45 in the *Fluent Theory Guide*).
- **Exponent Value**: Is the time exponent factor (α in Equation 19.45 in the *Fluent Theory Guide*).

Battery Aging Table

is a 2D lookup table for capacity fade factors at different temperatures and cycle numbers. All values are entered in the **Battery Aging Table** dialog box that opens by clicking the **Cycle Life - Battery Aging Table...** button (for example Figure 31.3: The Calendar Life Parameters Dialog Box (p. 3309)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.4: The Battery Aging Table Dialog Box - Cycle Life



For the NTGK model, you can specify the following inputs in the **Battery Aging Table** dialog box:

- **Temperature Levels**
- **Cycle-Number Levels**
- Values of **Temperature**
- Values of **Cycle-Number**
- Capacity fade values for each cycle-temperature pair

You can specify up to 20 **Temperature Levels** and **Cycle-Number Levels**.

You can use the **Write...** button to store the 2D lookup table to a file in a text format, which you can use later by clicking the **Read...** button and using the **Select File** dialog box that opens. You can also generate your own 2D lookup text file and read it into Ansys Fluent. Both the compact scheme and the plain text format can be used when reading or writing a table from/to a file. Refer to Using Tables in the Battery Model (p. 3315) for more information about these table formats. The capacity fade values will be automatically populated in the **Battery Aging Table** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.2.3.1.2. Specifying a Profile or Data Table

If you have select **Using Profile/Data Table** (**Solution Options** group box), you can use one of the following options in the **Profile Types** group box:

- **Time-Scheduled**
- **Event-Scheduled**
- **Table Data**

  This option can be used to specify a fast charging electric current profile as a 2D lookup table. Further details are provided below.

- **UDF**

  You can use the **UDF** option to define the electric load type and value using a `DEFINE_BATTERY_ELOAD_PROFILE` UDF. The **UDF** option is used if the time-scheduled and event-scheduled profile cannot cover the profile you want to use. See `DEFINE_BATTERY_ELOAD_PROFILE` in the *Fluent Customization Manual* for information about `DEFINE_BATTERY_ELOAD_PROFILE`.

For the **Time-Scheduled** or **Event-Scheduled** options, you must provide time-dependent or event-dependent input (respectively) for the C-rate, current, voltage, or power. The time-scheduled and event-scheduled profiles enable you to change the electric load type and electric value in the simulation on the fly. When either the **Time-Scheduled** or **Event-Scheduled** profile type is selected, you can browse to the ASCII text file with either the `.dat` or `.txt` file extension that you generated beforehand. The inputs for the **Time-Scheduled** and **Event-Scheduled** profiles are described below.

Time-Scheduled Profile Specification

The **Time-Scheduled** profile file must be organized in three columns:

1. Time
2. Electric load value
3. Electric load type

   The electric load type is defined by an integer number:

   0: C-rate

   1: Current

   2: Voltage

   3: Power

   4: External electric resistance

For example, the below time-scheduled profile:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
0        15    1
500      15    1
500.1    60    3
1000     60    3
1000.1   -1    0
1500     -1    0
```

specifies the following battery charging scenario:

1. The battery is discharged at 15 A for the first 500 seconds.
2. The battery discharge continues at 60 W for the next 500 seconds.
3. The battery is charged at a constant 1 C-rate for the last 500 seconds.

Note:

- For current, power, or C-rate, a positive value means discharging and a negative value means charging.
- The unit of the variables in the input file is always SI.

Ansys Fluent solver uses linear interpolation of your time-scheduled profile data in the problem simulation. If you want to capture a sudden change in the profile accurately, use smaller time gaps to describe the profile.

Event-Scheduled Profile Specification

The **Event-Scheduled** profile file must be organized in the following five columns:

1. Electric load type

   Just as in the time-scheduled profile, the electric load type is defined by an integer number:

   0: C-rate

   1: Current

   2: Voltage

   3: Power

   4: External electric resistance

2. Electric load value
3. Forwarding condition

   The forwarding condition (FC) is defined by an integer number:

   1: Time > FC_Value

   2: I < FC_Value (discharging current minimum)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3: I > FC_Value (charging current maximum)

4: V < FC_Value (system voltage minimum)

5: V > FC_Value (system voltage maximum)

6: P < FC_Value (discharging power minimum)

7: P > FC_Value (charging power maximum)

8: Time Duration > FC_Value (Time Duration)

4. Forwarding condition value

5. Stop flag

   The stop flag is an integer value of 0 or 1, where 0 specifies that the run should continue to the next event entry line, and 1 specifies that the run should end.

For example, the below event-scheduled profile:

```
0    1    4  3.5    0
0    0    8  300    0
1  -15    5  4.1    0
2  4.1    3   -1    1
```

specifies the following scenario:

1. The battery is discharged at 1 C rate until the voltage reaches 3.5 V.
2. The battery is at rest for 300 seconds.
3. The battery is charged at constant current 15 A until the voltage reaches 4.1 V.
4. The battery keeps charging at 4.1 V until the current is greater -1 A. The simulation stops after reaching this point.

Note:

- For current, power, or C-rate, a positive value means discharging and a negative value means charging.
- The unit of the variables in the input file is always SI.

Any time-scheduled profile can be rewritten in an event-scheduled profile format, but not vice-versa. The example shown in the time-scheduled profile can be redefined in an event-scheduled profile format as follows.

```
1    15    1    500    0
3    60    1   1000    0
0    -1    1   1500    1
```

Both profile types can be read into Ansys Fluent. You can choose the most convenient one to use in your specific application.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

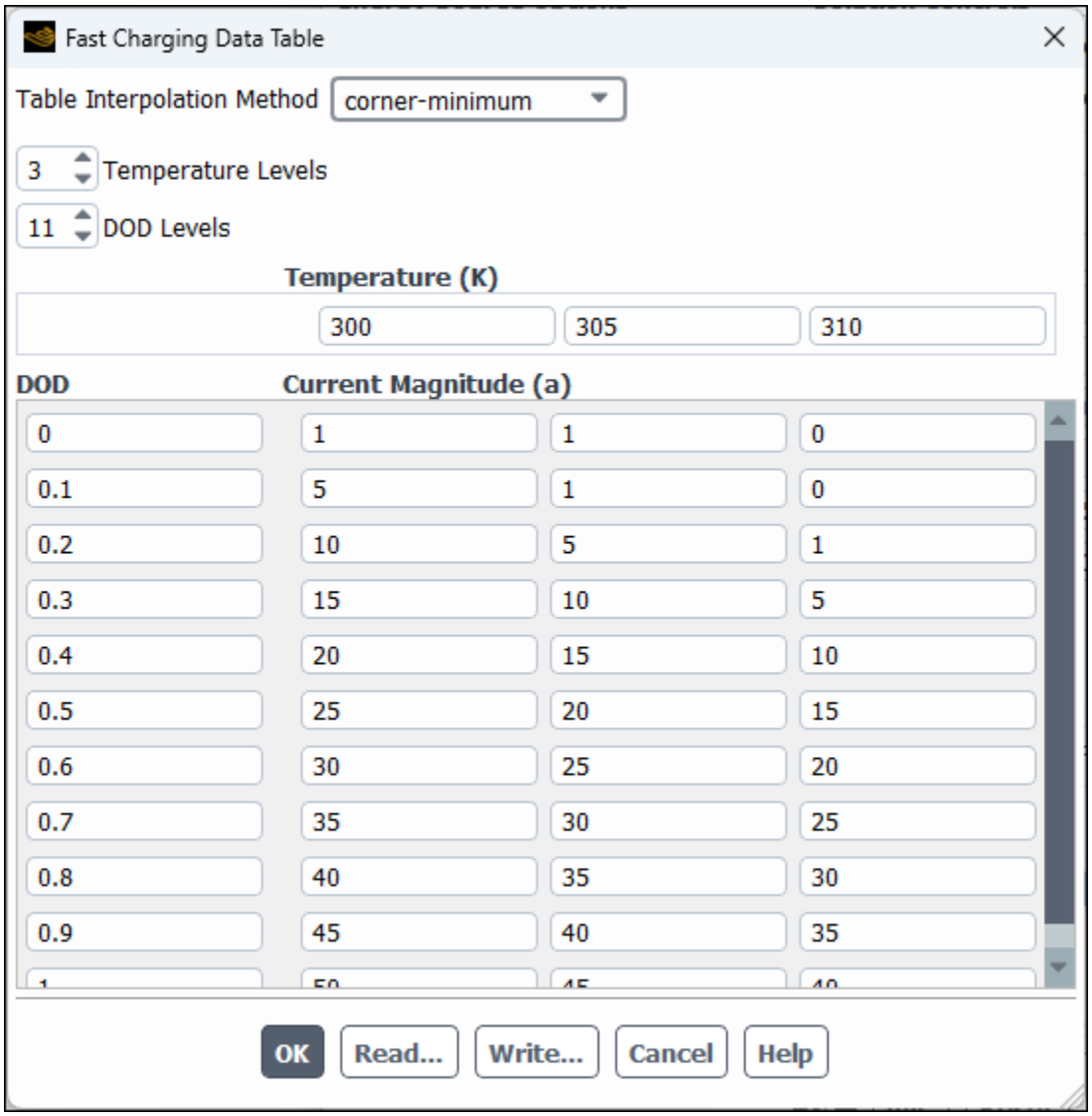
Table Data Specification

The fast charging 2D lookup table defines the charging current as a function of battery's average DOD and temperature. To use this specification method follow the steps below:

1. Create one or more report definitions of temperature.
2. In the **Temperature Method** list, select the report definition(s) to be used in the lookup table.

   The lookup table will be searched for all selected temperature definitions, and the minimum value will be used as the charging current.
3. Click **Fast Charging Table...**

Figure 31.5: The Fast Charging Data Table Dialog Box



4. In the **Fast Charging Data Table** dialog box that opens, specify the following inputs.
   a. Select the **Table Interpolation Method** from the following options:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **bilinear**

  The bilinear interpolation method is used to find charging current value given a DOD and a temperature value.

- **staircase-floor**

  For a given temperature, the current changes as a staircase and the lower DOD value is used. Linear interpolation is used for temperature.

- **staircase-ceiling**

  The DOD ceiling value is used. Linear interpolation is used for temperature.

- **corner minimum**

  The lowest conner value is used from the lookup table.

The interpolated value for charging current are different for different methods. The example below shows the charging current data for two temperature levels of and two DOD levels.

	T=300 K	T=310 K
DOD=0.2	10	5
DOD=0.3	20	10

The interpolated value of charging current for T=305K and DOD=0.25 will be:

- Bilinear: Charging Current = ((20+10)/2+(5+10)/2)/2= 11.25
- Staircase-floor: Charging Current = (10+5)/2 =7.5
- Staircase-ceiling: Charging Current = (20+10)/2 = 15
- Corner-minimum: Charging Current = 5

b. Enter values for the following table parameters:

- **Temperature Levels**
- **DOD Levels**
- Values of **Temperature**
- Values of **DOD**
- values of **Current Magnitude** for each DOD level and temperature

31.2.3.1.3. Using Tables in the Battery Model

2D and 3D lookup tables are used in the battery model to specify various quantities as tabular functions of other solution variables when the tabular input option is available. Lookup tables are tab- or space-delimited ASCII text files that support the following formats:

- Plain text format (files with extension `.txt`)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A 2D table must contain the following entries:

Row 1 – Header keyword "`x-values`"

Row 2 - x-value points

Row 3 – Header keyword "`y-values`"

Row 4 - y-values points

Row 5 – Header keyword "`table-values`"

Row 6 and all consecutive rows - table input data

An example of the 2D table for capacity fade factors at two temperature levels and three cycles-number levels is shown below:

```
x-values
   300   310
y-values
     0   100   200
table-values
     0     0
   0.1  0.15
   0.2  0.25
```

A 3D table is used only for specifying internal resistance data. For each electric current level, a 2D table of resistance values at different temperature and DOD levels must be provided. The table must include the following data:

– Keyword "`z-value`" followed by a current value at the first electric current level

– 2D table for the first electric current level. see the description above of 2D tables

– Keyword "`z-value`" followed by a current value at the second electric current level

– 2D table for the second electric current level

– Etc.

An example of the 3D table is shown below:

```
z-value  -10
x-values
   300   310
y-values
     0   0.5   1.0
table-values
   111   112
   121   122
   131   132

z-value  0
x-values
   300   310
y-values
     0   0.5   1.0
table-values
   211   212
   221   222
   231   232
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
z-value  10
x-values
   300   310
y-values
     0   0.5   1.0
table-values
   311   312
   321   322
   331   332
```

- Compact scheme format (files with extension other than `.txt`, for example `.tab`)

  In this format, each data group is enclosed in parentheses, and the input values within each group are separated by spaces and/or tabs.

  In the following example, the same 2D table for capacity fade factors shown above is written in the compact format:

  ```
  ((300 310) (0 100 200) ((0  0) (0.1  0.15) ( 0.2 0.25)))
  ```

  The first group of numbers `(300 310)` contains x-value points, which are the temperature levels. The second group of numbers `(1 100 200)` contains y-value points, which are the cycle numbers. The third group contains table entry values where each sub-group in parentheses represents a row of data in the table.

  The 3D table containing internal resistance data shown in the example above can be written in the compact format as follows:

  ```
  ((-10 0 10) (((300 310) (0 0.5 1.0) ((112 112) (121 122) (131 132)))
               ((300 310) (0 0.5 1.0) ((212 212) (221 222) (231 232)))
               ((300 310) (0 0.5 1.0) ((312 312) (321 322) (331 332)))) 0)
  ```

  The first group of data `(-10 0 10)` contains three electric current values. The second group includes three 2D lookup tables for the three electrical current levels. In each 2D table, the temperature array is listed first, followed by the DOD array and then by the resistance values for each temperature-DOD pair. The last digit in the list indicates the interpolation method. `0` denotes the bilinear interpolation, which must always be used for this particular table.

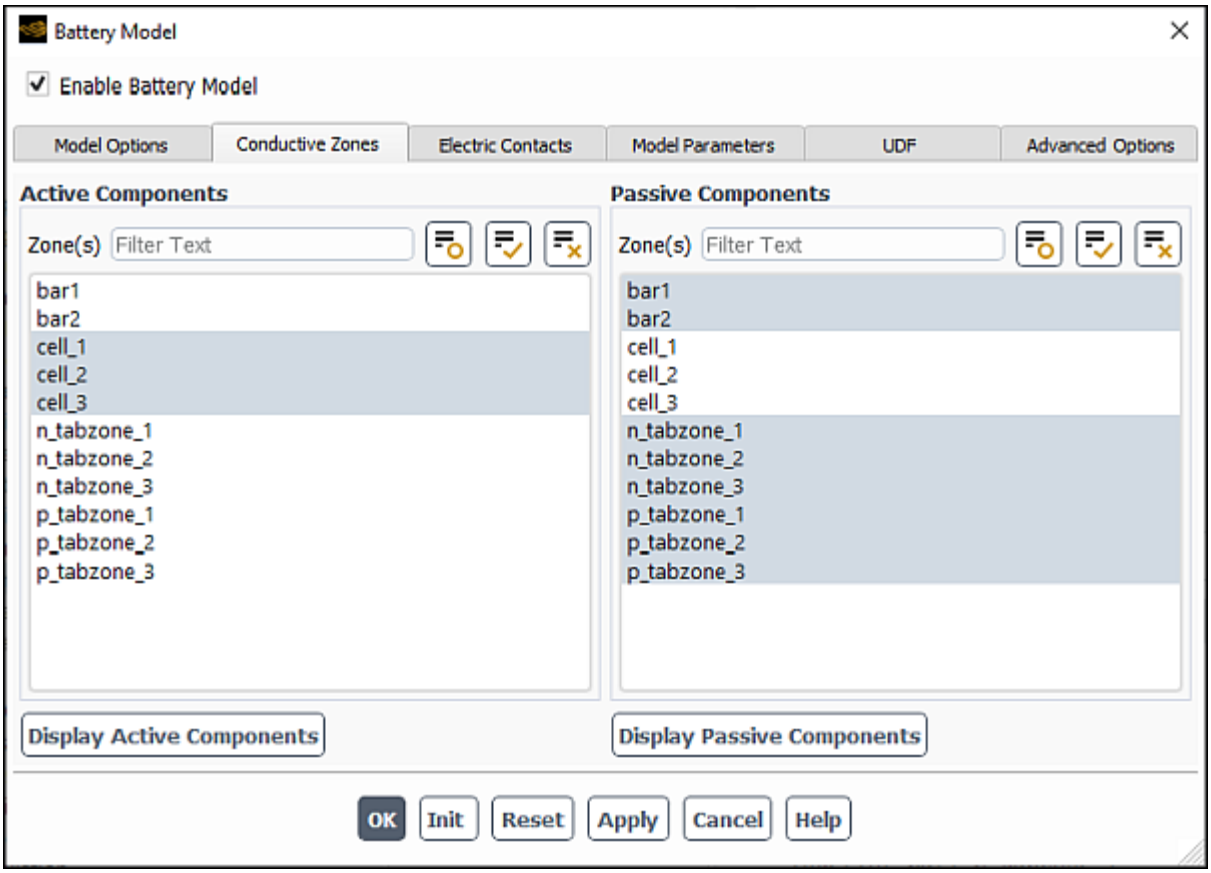
Usually if you transfer table data from one simulation to another, you can use the compact scheme format. If you prepare table data manually through a file, the plain text format is more readable and may be a better choice.

31.2.3.2. Specifying Conductive Zones

The **Conductive Zones** tab of the **Battery Model** dialog box allows you to specify the active, tab, and busbar zones.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.6: The Battery Model Dialog Box (Conductive Zones Tab)



In the **Conductive Zones** tab, select the appropriate zones for the **Active Components** and **Passive Components**. Passive components include battery tab zones and busbars used to connect tabs of different batteries. You can visualize all active or passive zones by using the **Display Active Components** or **Display Passive Components** button, respectively. This opens the **Mesh Display** dialog box where all relevant components are selected automatically.

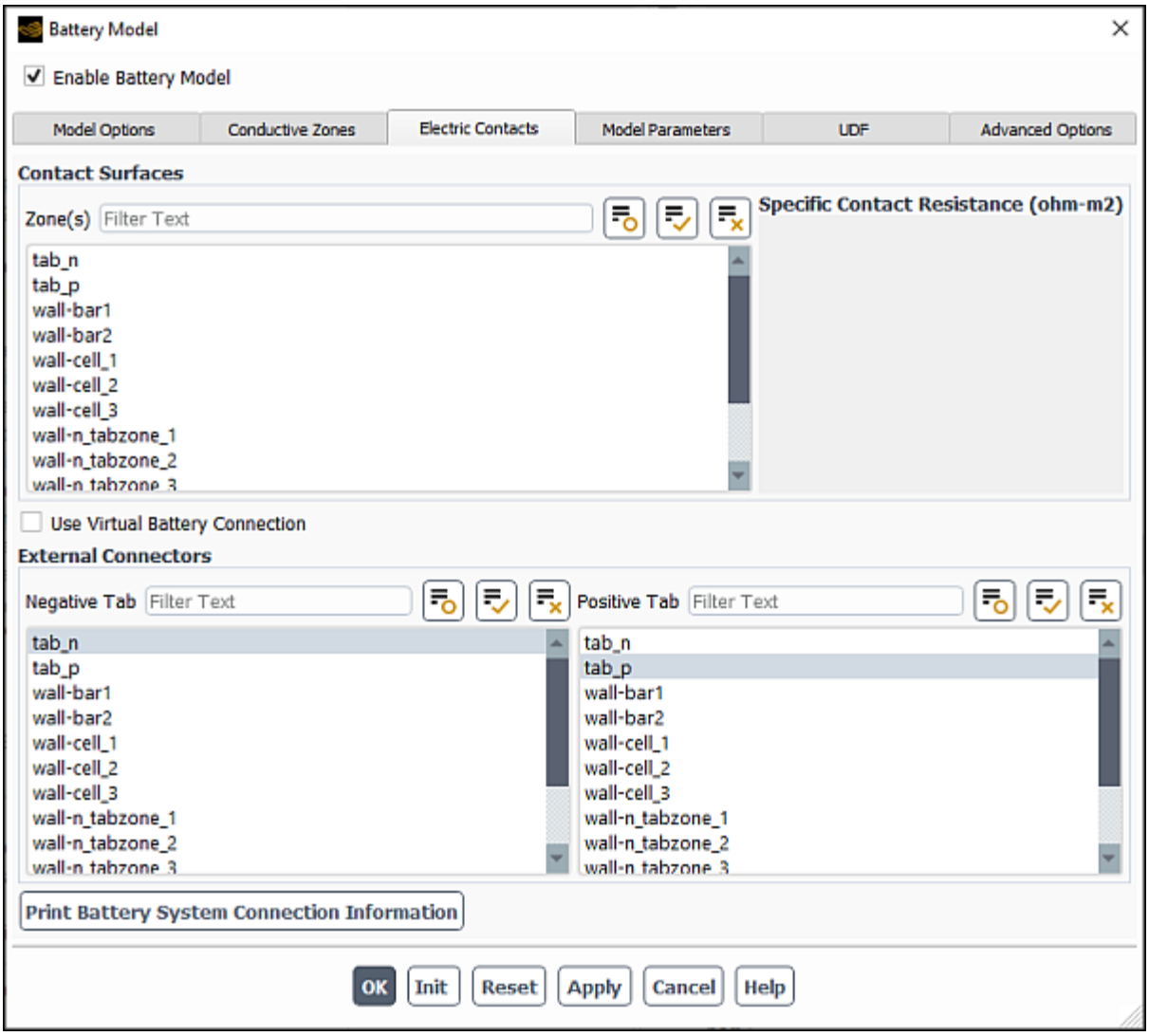
Battery cells are modeled as active zones; and battery tabs and busbars are modeled as passive zones. Electrochemical reactions occur only in the active zones. The potential field is solved in both active and passive zones.

31.2.3.3. Specifying Electric Contacts

The **Electric Contacts** tab of the **Battery Model** dialog box allows you to set the properties of the **Contact Surfaces** and the **External Connectors**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.7: The Battery Model Dialog Box (Electric Contacts Tab)



If you want to consider contact resistance, select zones for the **Contact Surfaces** and set the **Specific Contact Resistance** for any selected contact surface.

When using real connections, define the system negative and positive tabs under **External Connectors**.

When using the virtual connection option in a battery pack simulation, select the **Use Virtual Battery Connection** check box and in the **Specify connection file** text-entry box that appears, enter the filename containing the virtual connection definition.

The following input formats can be used:

- tab surface-based format
- active zone volume-based format

The tab surface-based format for the connection definition file is as follows:

```
mP 2
nS 2
tab_N_1P1S  tab_P_1P1S
```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

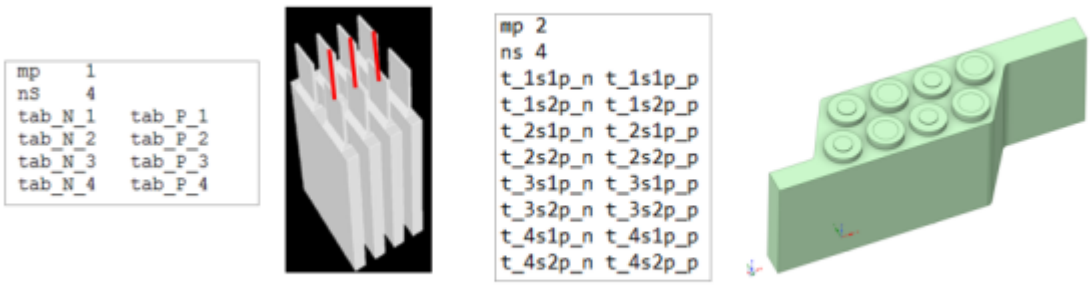
```
tab_N_2P1S  tab_P_2P1S
tab_N_1P2S  tab_P_1P2S
tab_N_2P2S  tab_P_2P2S
```

where:

- The first line specifies the number of parallel batteries in a stage (mP).
- The second line specifies the number of stages in series in a pack (nS).
- The remaining lines contain pairs of the face names of the negative and positive tabs for each battery. The battery data are listed in the following order: 1P1S ... mP1S to 1PnS ... mPnS.

For the above battery connection file example, the battery solver establishes the 2P2S connection pattern.

The following figure shows the different resulting connection patterns for the different connection data.



Once all conductive zones and the positive and negative tabs have been defined, Ansys Fluent automatically detects the battery network connection.

The active zone volume-based format for the connection definition is as follows:

```
mP 1
nS 4
cell_1
cell_2
cell_3
cell_4
```

where:

- The first line specifies the number of parallel batteries in a stage (mP).
- The second line specifies the number of stages in series in a pack (nS).
- The remaining lines contain the names of the active battery zones in the order of: 1P1S ... 1PnS to mP1S ... mPnS.

You should use one line only for each battery. If a battery active zone includes multiple cell volumes, list all of them in one line separated by space, for example:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
mP 1
nS 4
cell_1_a   cell_1_b    cell_1_c
cell_2
cell_3_a   cell_3_b    cell_3_c
cell_4
```

Note:

The active zone volume-based connection file can only be used for cases that involve the **Circuit Network** solution method without the Joule heating effects. This is because the electric current flow passage is not determined in a battery pack. Consequently, the potential field, and therefore the Joule heating effect, cannot be solved.

When you import either format of the connection file, Ansys Fluent automatically detects the type of the format. If the active zone volume-based format is used, the solver enables the **Circuit Network** solution method and disables the Joule heating effects. The zones in the active and passive components listed under the **Conductive Zones** tab are updated automatically according to the information provided in the connection file.

After you have defined all the conductive zones or modified zone related data, click **Print Battery System Connection Information**, and review the battery connection information printed in the Ansys Fluent console window for any errors. If necessary, re-define the connections in the **Conductive Zones** and **Electric Contacts** tabs.

31.2.3.4. Specifying Battery Model Parameters

The **Model Parameters** tab of the **Battery Model** dialog box allows you to specify the submodel-specific parameters. The **Model Parameters** tab contains only the entry fields for parameters related to either the solution method or electrochemical submodels you have chosen in the **Model Option** tab. A detailed description of inputs for each submodel is given in the sections that follow.

31.2.3.4.1. Inputs for the CHT Coupling Method

31.2.3.4.2. Inputs for the FMU-CHT Coupling Method

31.2.3.4.3. Inputs for the NTGK/DCIR Model

31.2.3.4.4. Inputs for the Equivalent Circuit Model

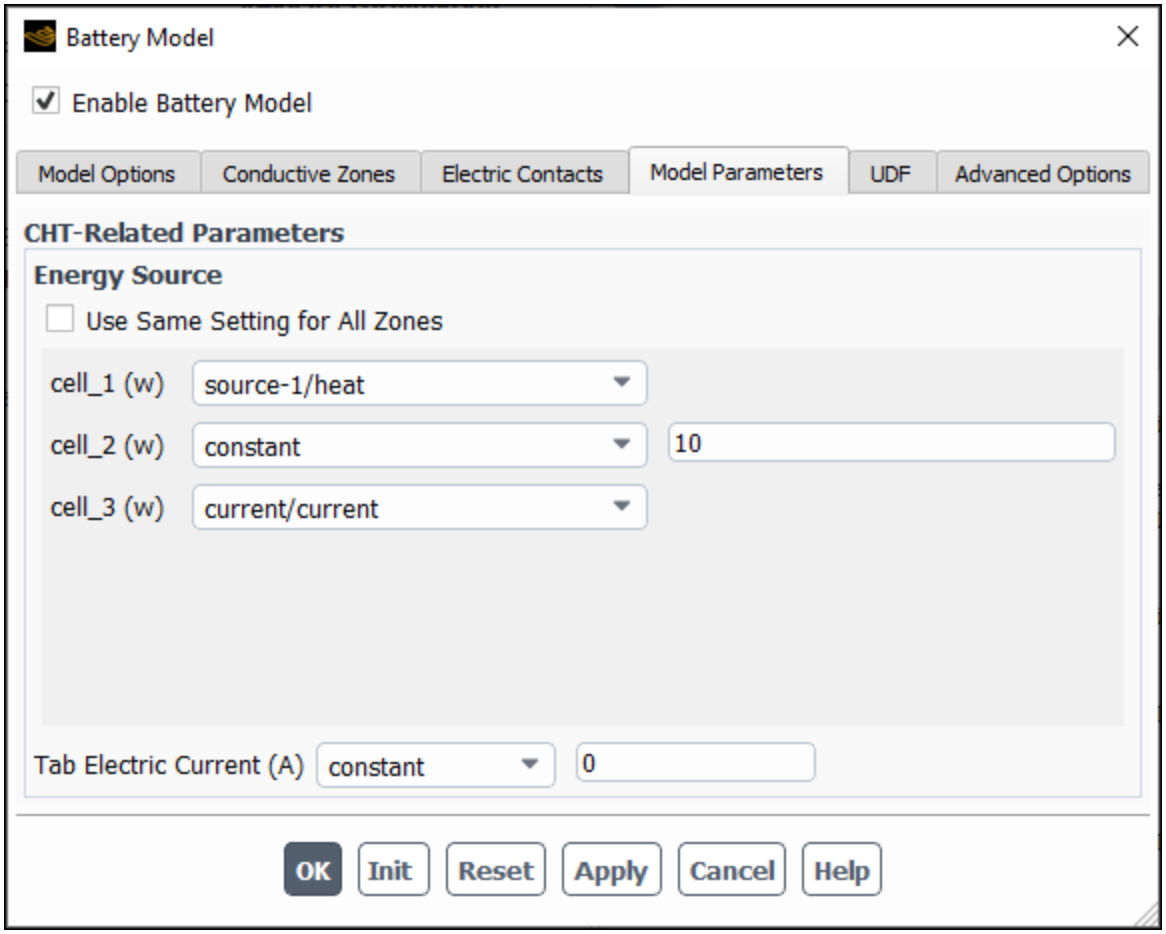
31.2.3.4.5. Inputs for the Newman's P2D Model

31.2.3.4.6. Input for the User-Defined E-Model

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.2.3.4.1. Inputs for the CHT Coupling Method

Figure 31.8: Model Parameters Tab—CHT Coupling Method



You need to specify the following **CHT-Related Parameters**:

- total **Energy Source** for each battery active zone
- (if you want to consider the Joule heating in passive zones) **Tab Electric Current**

For both heat source and tab electric current, you can use either **constant** data type or any of the imported transient profiles.

If **constant** is selected, you need to specify a value in the adjacent field.

To use transient profiles for energy sources, you first need to read all profile files into your simulation as outlined in Reading Profile Files (p. 964). Once the profiles have been imported into your analysis, their names automatically appear in the drop-down lists for each active zone and for tab electric current. The profile file format conforms to the Ansys Fluent built-in transient profiles described in Transient Cell Zone and Boundary Conditions (p. 1523). The below example shows the file format of a tabular transient profile (`.ttab`).

```
energy-source  2  4  0
time       energy-source
0          5.0
100        5.0
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
100.1      10.0
10000      10.0
```

In this example, the first line contains the profile name, followed by the number of fields, the number of lines, and the flag indicating whether the profile is periodic.

The second line contains all the field names. One of the field names must be `time` for the profile to be correctly recognized.

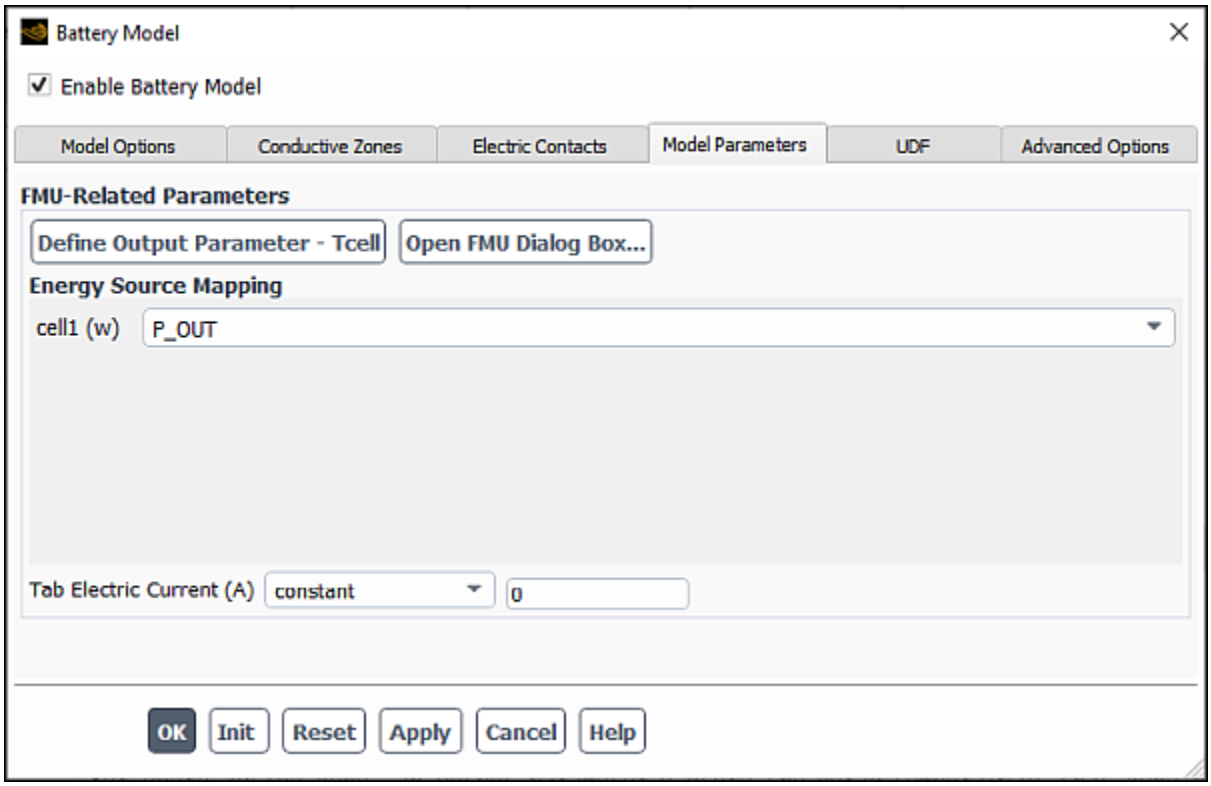
The remaining rows contain the profile data.

All the imported profiles will be saved within the Ansys Fluent case file.

If in a pack simulation, the same heat source is used for all active zones, you can select **Use Same Settings for All Zones** and define only one set of data. Ansys Fluent will automatically update all zones with the same cell information.

31.2.3.4.2. Inputs for the FMU-CHT Coupling Method

Figure 31.9: Model Parameters Tab—FMU-CHT Coupling Method



1. In the **FMU-Related Parameters** group box, click the **Define Output Parameter- Tcell** button.

   Fluent creates an output parameter, the average zone temperature, for each active zone.

2. Click **Open FMU Dialog Box...** and then use the cell average temperatures that have been generated in the previous step to define the FMU input parameters in the **Parameters** dialog box. Refer to Coupled Simulations in Ansys Fluent with Functional Mock-up Unit (FMU) Files (p. 1028) for details on the FMU file importing procedure.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

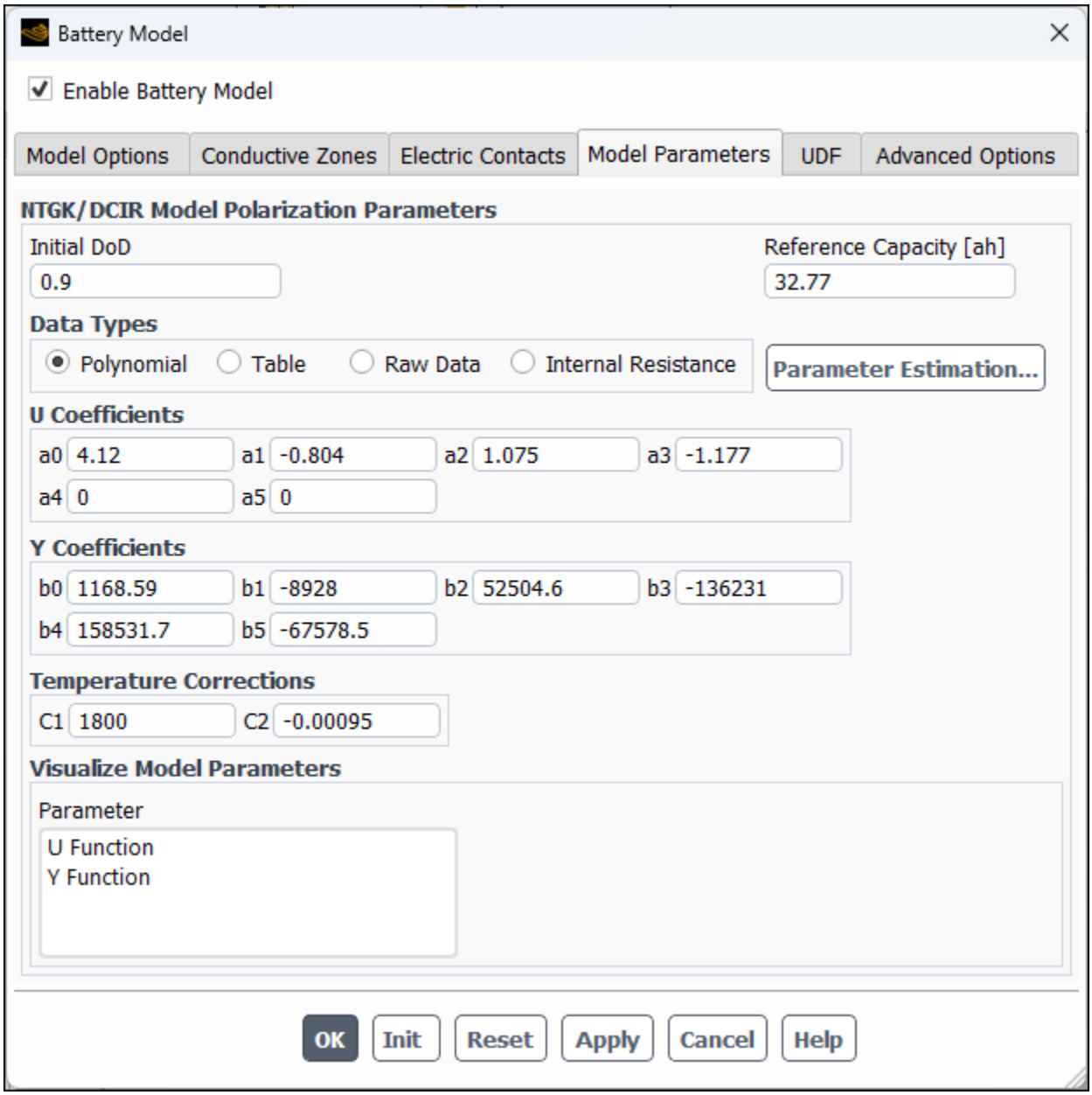
3. Once the FMU file importing is complete, you can define the following parameters:

   - For each battery active zone, under **Energy Source Mapping**, select an appropriate FMU output parameters that have been imported in the previous step to be an energy source term.

   - If the Joule heating is considered, specify **Tab Electric Current**. Similar to the **CHT Coupling** solution method, you can choose either **constant** or any of the imported transient profile. See Inputs for the CHT Coupling Method (p. 3322) for more information.

31.2.3.4.3. Inputs for the NTGK/DCIR Model

Figure 31.10: Model Parameters Tab—NTGK/DCIR Model



For the **NTGK/DCIR Model**, define the following operating conditions:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Specify the following NTGK model parameters:

   - **Initial DoD**: Is the battery initial DOD at t=0. The default value of `0` for **Initial DoD** indicates the fully-charged state of the battery cell.
   - **Reference Capacity**: Is the capacity of the battery that is used in experiments to obtain the model parameters (Q_{ref} in Equation 19.5). Nominal and reference capacities may have different values.

2. Specify U and Y model parameters in Equation 19.7 using one of the following:

   - **Polynomial** data type
   - **Table** data type
   - **Raw Data** data type
   - **Parameter Estimation** tool

   or directly define the battery internal resistance by using the **Internal Resistance** data type.

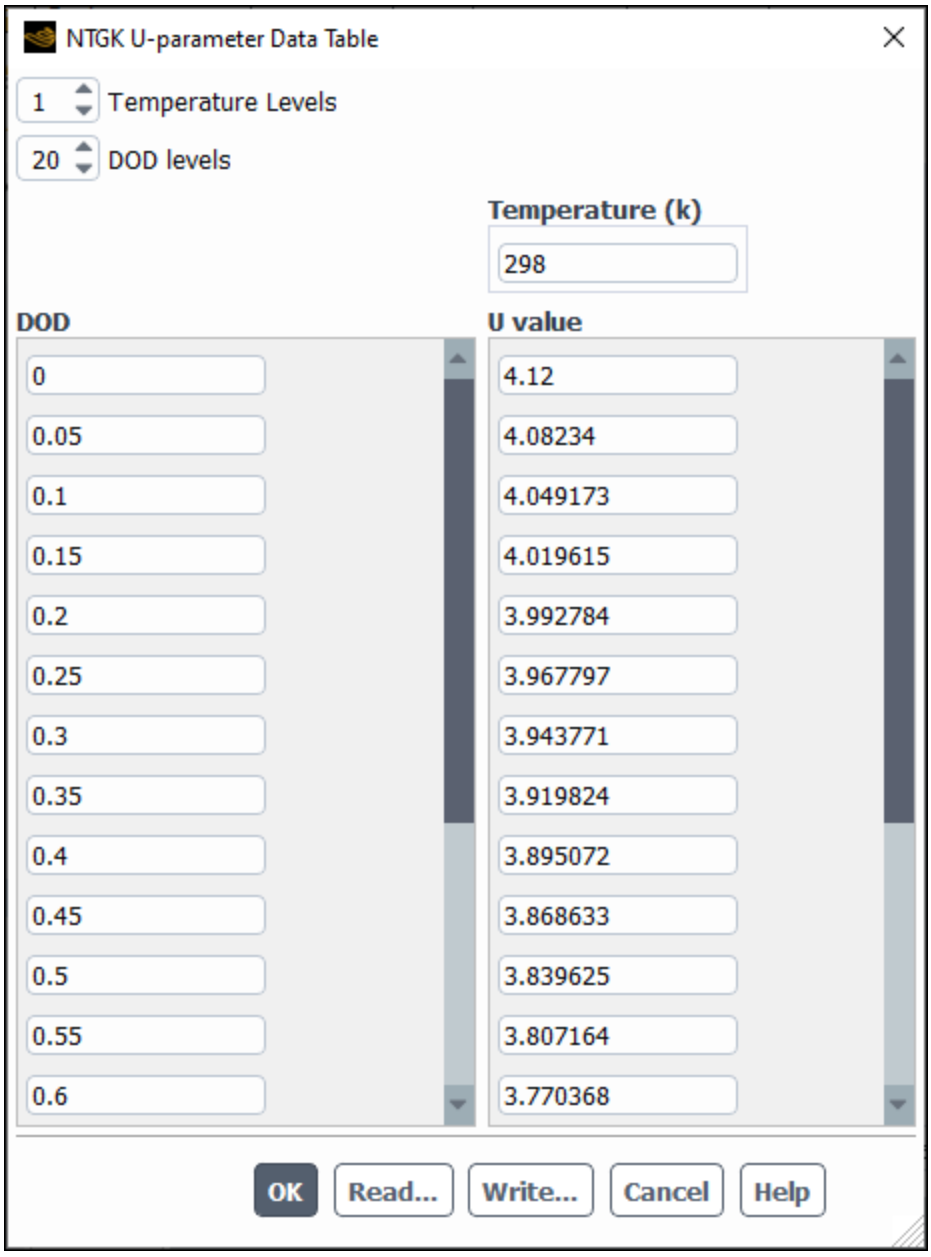
3. If you selected **Polynomial** in the **Data Types** group box, enter polynomial coefficients for U, Y, and temperature correction in Equation 19.7 directly in the **U Coefficients** and **Y Coefficients** group boxes.

4. If you selected **Table** in the **Data Types** group box, define a 2D lookup table for each NTGK model parameter.

   Tabular data are entered in the corresponding parameter data table dialog box that opens by clicking the relevant button (**U Table...** or **Y Table...**) in the **NTGK Parameter Data Table** group box (see Figure 31.11: NTGK U-parameter Data Table Dialog Box (p. 3326)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.11: NTGK U-parameter Data Table Dialog Box



For each NTGK model parameter, you can specify up to 50 DOD levels and 20 temperature levels. For each DOD-temperature pair, you must enter the corresponding model parameter value. Each table can be exported to or imported from a file by clicking **Write** or **Read**, respectively, in the parameter data table dialog box. Both the compact scheme and the plain text format can be used when reading or writing a table from/to a file. Refer to Using Tables in the Battery Model (p. 3315) for more information about these table formats.

Note:

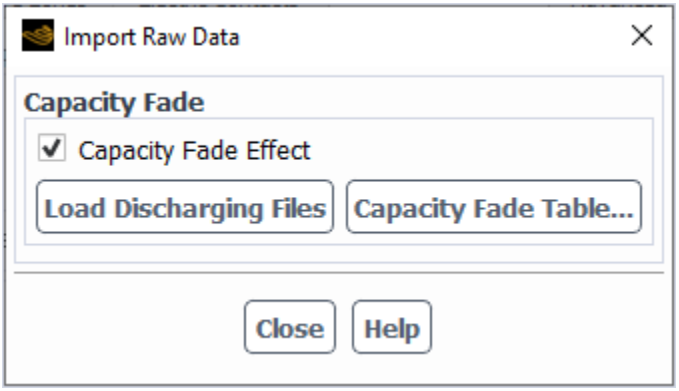
The model parameters for the **NTGK Model Polarization Parameters** are based on battery cell polarization test curves. Obtaining model parameters (other than the default values) can be dependant on your battery configuration and material

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

properties. The default values are taken from Kim's paper 313. If you model a battery having a different design, you must provide a different set of battery parameters.

5. If you selected **Raw Data** in the **Data Types** group box, you need to import a set of testing data (discharging curves under different temperature and C-rate) by clicking the **Import Testing Data** button.

Figure 31.12: The Import Raw Data Dialog Box



In the **Import Raw Data** dialog box that opens, you can import testing data files by clicking the **Load Discharging Files** button and then using the **Select File** dialog box to read the files into your Fluent case.

If you want to include the capacity fade effect into your simulation, enable **Capacity Fade Effect**. After you import the testing data, the capacity fade table is populated automatically. You can review the table by clicking **Capacity Fade Table...**, but you cannot modify it.

Each capacity fade factor in the capacity fade table is computed from the test data as:

$$capacity\ \ fade\ \ factor = discharging\ \ time * Crate \big/ 3600$$

After you import test data files, no other information is needed. The Ansys Fluent solver stores the `U~I` relationship from the test data and will use it during the battery simulation.

Note:

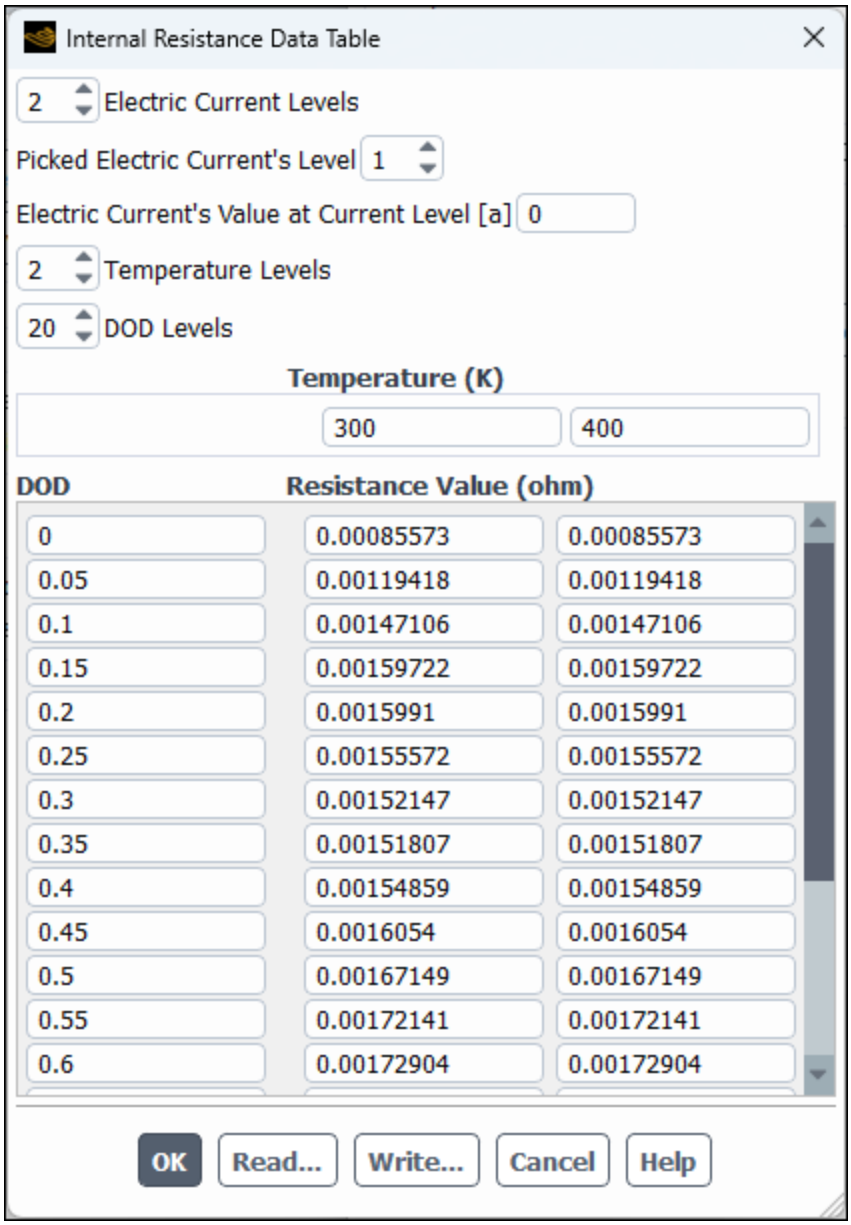
The raw data treatment depends on whether the capacity fade effect is included or not. If you enable or disable the **Capacity Fade Effect** option, you need to load the data files again to re-process it.

6. If you selected **Internal Resistance** in the **Data Types** group box, you need to define the internal resistance table and specify other parameters in the **DCIR Method Settings** group box.

   a. Click the **Internal Resistance Table...** button.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.13: The Internal Resistance Data Table Dialog Box



b. In the **Internal Resistance Data Table** dialog box that opens, enter the table data.

   i. Specify the number of **Electric Current Levels**.

   ii. Set the **Selected Current Level** for which you want to define internal resistance points.

   iii. Enter a value for the **Current** for the selected current level.

   iv. Specify values for the following table parameters:

   - **Temperature Levels**
   - **DOD Levels**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Values of **Temperature**
- Values of **DOD**
- Values of **Current Magnitude** for each DOD level and temperature

  A positive value means discharging current and a negative value means charging current. In this way, you can use different internal resistance when simulating charging and discharging.

v. Repeat steps 2 through 4 for each current level.

You can write a table data into a file or fill the table by importing data from a file. Both the compact scheme and the plain text format can be used when reading or writing a table from/to a file. Refer to Using Tables in the Battery Model (p. 3315) for more information.

c. If you want to include the open circuit potential, enable **U Table**, click the **U Table...** button, and in the **NTGK Parameter Data Table** dialog box that opens (see Figure 31.11: NTGK U-parameter Data Table Dialog Box (p. 3326)), enter U values for each DOD-temperature pair.

Note:

If the U-table is not provided, you can use only either the current or C-rate electrical load condition. In addition, only single battery or pure serial battery connection can be simulated.

---

d. (cases with no fast charging) If you want to limit current in your simulation, perform the following steps:

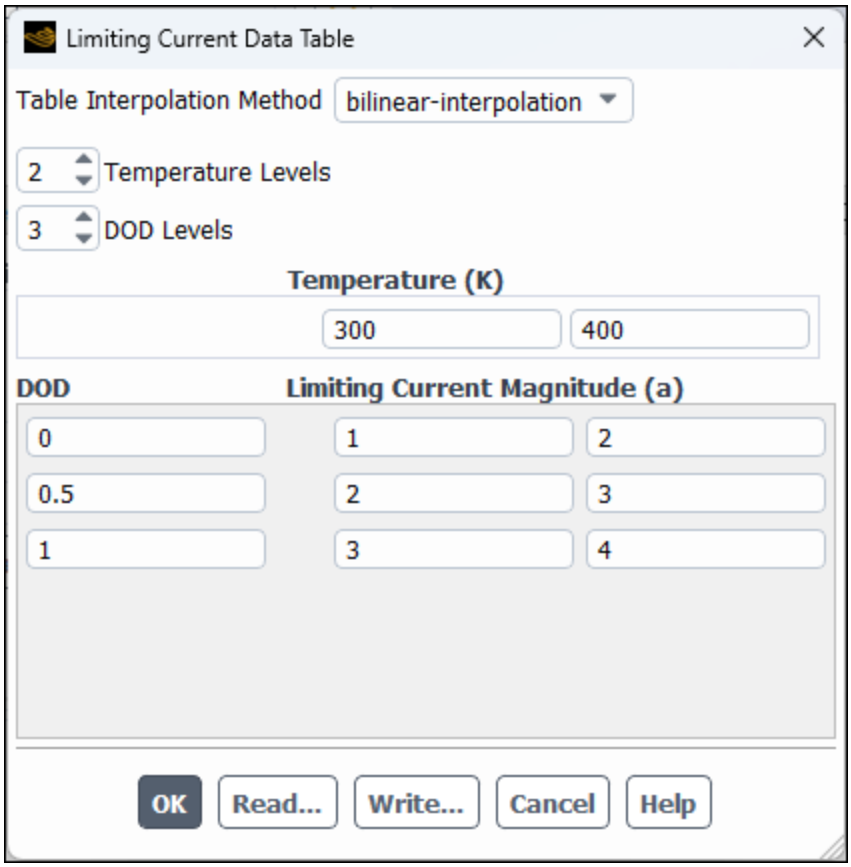
i. Enable **Limiting Current**.

ii. In the **Temperature Method** list, select the report definition(s) to be used in the lookup table.

iii. Click the **Current Limiting Table...** button.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.14: The Limiting Current Data Table Dialog Box



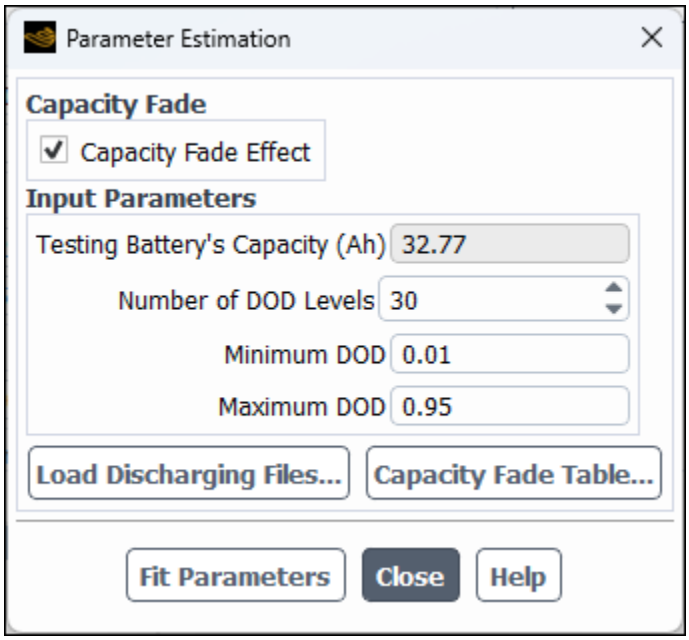
iv. In the **Limiting Current Data Table** dialog box that opens, specify a 3D table of the limiting current values.

The inputs are similar to those in the **Fast Charging Data Table** dialog box described in Specifying a Profile or Data Table (p. 3311).

7. If you want to use the parameter estimation tool to fit U and Y from testing data, click the **Parameter Estimation...** button.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.15: The Parameter Estimation Dialog Box for the NTGK Model



a. In the **Parameter Estimation** dialog box that opens, you need to specify the following **Input Parameters**:

   - **Testing Battery's Capacity**

     This value is copied from the **Reference Capacity** (**Model Parameters** tab). You need to verify it is correct; otherwise, go back to the **Model Parameters** tab to modify it.

   - **Number of DOD Levels**

     The number of DOD levels is a number between 10 and 20. The default 20 is a good choice for all cases. The required discharging data are battery's discharging curves at different constant C-rates. You can use testing data at different temperatures. For each C-rate, you must provide a separate file in text format. At least two input files with data at different C-rates are required to get meaningful fitting results.

   - **Minimum DOD**: Minimum battery's DOD level used in the fitting. The default value is 0.01, which is suitable for most cases.

   - **Maximum DOD**: Maximum battery's DOD level used in the fitting. The default value is 0.95, which is suitable for most cases.

   - Corresponding discharging data (as further described)

b. If you want to consider the capacity fade effect in your simulation, enable **Capacity Fade Effect**.

   After the parameter fitting process is complete, the capacity fade table is populated automatically. You can review the table by clicking **Capacity Fade Table...**, but you cannot modify it. The fitting results (U and Y parameters) are dependent upon whether the capacity fade effect is considered or not. If you have modified the state of **Capacity Fade Effect**, you need to re-fit the model parameters.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. To import all the testing files, click the **Load Discharging Files** button and use the **Select File** dialog box that opens to read multiple testing data files. The selected testing files will be listed at the bottom of the **Select File** dialog box. If you accidentally select the wrong file, you can use the **Remove** button to remove it from the selected testing file list.

An example of an input file is shown below.

```
Crate 1.0
Temperature 300.0
1.0000e+00   4.1097e+00
2.0000e+00   4.1094e+00
3.0000e+00   4.1092e+00
...
3.6000e+01   4.1010e+00
...
```

The first line of the file starts with `Crate` followed by a C-rate value separated by a space. The second line starts with `Temperature` followed by a value. The lines that follow contain pairs of space-separated time-voltage values.

d. After testing files are imported, click the **Fit Parameters** button to run the NTGK parameter estimation tool.

If you provide testing data files at different temperatures, the parameter estimation tool will perform the fitting for each temperature level. Ansys Fluent will report the fitted results in the console and automatically pass them in a table data type to the parameter data table dialog boxes. The **Table** data type will be automatically selected in the **Battery Model** dialog box. This will allow you to consider the temperature-dependent effect.

If you use testing data from only one temperature level, the fitting results in both polynomial and table data types will be filled in the **Battery Model** dialog box. In this case, the **Polynomial** data type is the default data type. However, you can use either data type in your simulation.

e. (optional) Visualize and check the resulting fitted curves of the battery discharge data at different temperatures and C-rates to detect any issues with the fitting process.

Once the fitting is complete, the **Parameter Estimation** dialog box expands to reveal the **Visualize Fitting Results** group box containing the **Temperature** and **C rate** selection lists. You can display curves of the fitted discharge data and the experimental data from your input files at different temperatures and C-rates by selecting a pair of desired values from the **Temperature** and **C rate** lists. The corresponding curves will be automatically plotted in the graphics window.

f. Click **Close** in the **Parameter Estimation** dialog box.

Note:

The NTGK parameter estimation tool can also be accessed through the text user interface (TUI) as described in Using the Parameter Estimation Tool for the NTGK Model in the TUI (p. 3391).

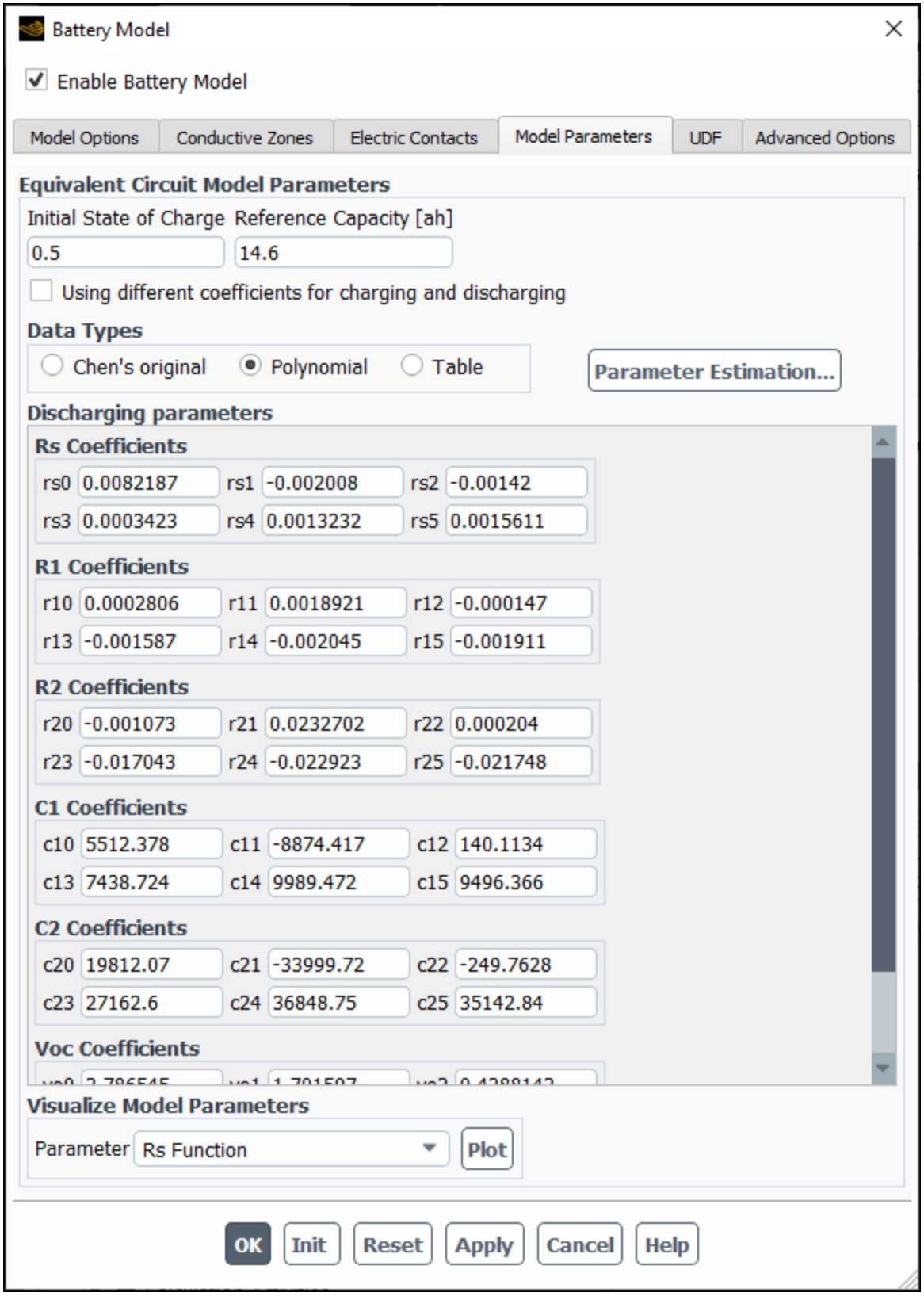
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. (**Polynomial** or **Table** data type only) You can visualize the resulting U and Y parameters varying with DOD by selecting **U Function** or **Y Function** from the **Parameter** list (**Visualize Model Parameters** group box) and clicking the **Plot** button. The resulting curves at temperatures you provided will be plotted in the graphics window.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.2.3.4.4. Inputs for the Equivalent Circuit Model

Figure 31.16: Model Parameters Tab—Equivalent Circuit Model



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For the Equivalent Circuit Model (ECM), define the following operating conditions:

1. Specify the following **Equivalent Circuit Model** parameters:

   - **Initial State of Charge**: Is the battery initial SOC at t=0. The default value of 1 for **Initial State of Charge** indicates the fully-charged state of the battery cell.
   - **Reference Capacity**: Is the capacity of the battery that is used in experiments to obtain model parameters (Q_{ref} in Equation 19.9). Note that nominal and reference capacities may have different values.

2. Specify resistors' resistances (R_s, R_1, and R_2), capacitors' capacitances (C_1 and C_2), and the open circuit voltage (V_{oc}) using one of the following:

   - **Chen's original** data type
   - **Polynomial** data type
   - **Table** data type
   - **Parameter Estimation** tool

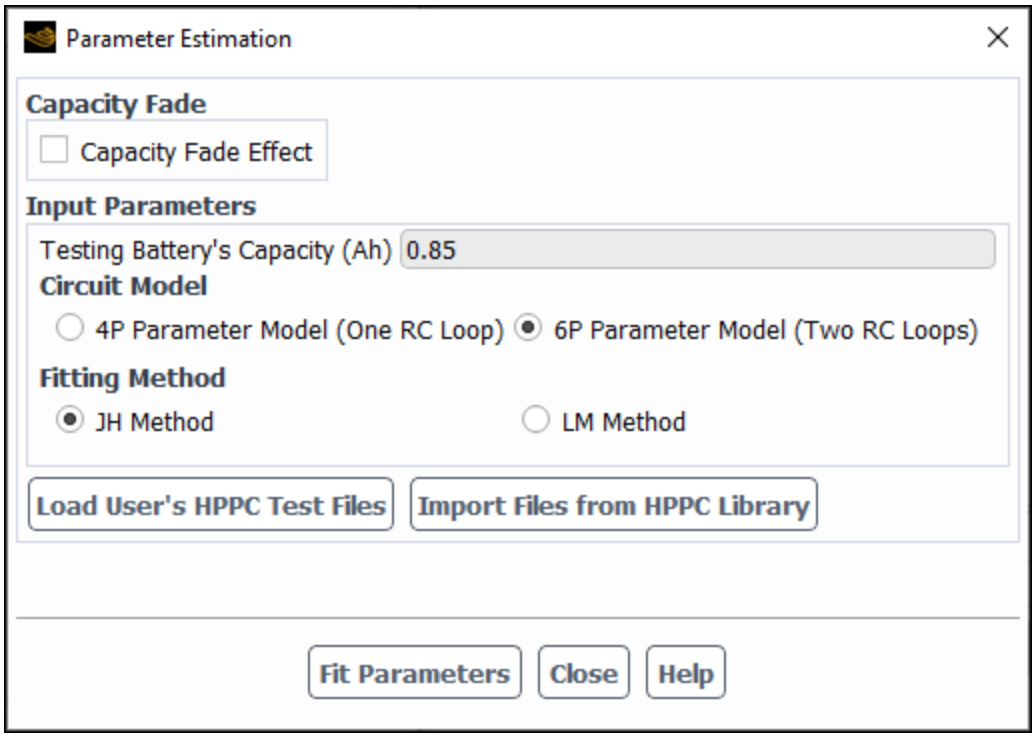
   Note:

   The values of model parameters for the **Equivalent Circuit Model** are battery specific and need to be specified before conducting a simulation. The default values are taken from Chen (see ECM Model for details). If you model a battery of different design, you should specify a different set of battery parameters.

3. If you selected **Chen's original** in the **Data Types** group box, you need to specify all coefficients in Equation 19.11.
4. If you selected **Polynomial** in the **Data Types** group box, you need to directly specify polynomial coefficients in Equation 19.10.
5. If you selected **Table** in the **Data Types** group box, you need to define a 2D lookup table for each ECM parameter. See Inputs for the NTGK/DCIR Model (p. 3324) for more information on how to define model parameters in a table data type.
6. (optional) You can enable the **Use Different Coefficient for Charging and Discharging** option and specify the coefficients for charging parameters.
7. If you want to use the parameter estimation tool to fit the parameters from testing data, click the **Parameter Estimation** button.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.17: The Parameter Estimation Dialog Box for the ECM



a. In the **Parameter Estimation** dialog box that opens, you need to specify the following **Input Parameters**:

   - **Testing Battery's Capacity**

     This value is copied from **Reference Capacity** (**Model Parameters** tab). You need to verify it is correct; otherwise, go back to the **Model Parameters** tab to modify it.

   - **Circuit Model** (**4P Parameter Model (One RC Loop)** or **6P Parameter Model (Two RC Loop)**)

   - **Fitting Method** (**JH** or **LM Method**)

     The Jiang-Hu (JH) method uses only the testing data from the relaxation time period, and the Levenberg-Marquardt (LM) method uses the testing data from both the pulse and relaxation time periods. The JH method uses the approach from paper [69] (p. 5717), and the LM method uses the general Levenberg-Marquardt method.

   - Corresponding HPPC testing data (as further described)

b. If you want to consider the capacity fade effect in your simulation, enable **Capacity Fade Effect**.

   If this option is enabled, you need to import HPPC testing files for the discharging curves. The format of the discharging curve is the same as used for the NTGK parameter estimation tool. After you load files for the discharging curves, the capacity fade table is populated automatically. You can review the table by clicking **Capacity Fade Table...**, but you cannot modify it.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. Load HPPC files into your case.

The required testing data are battery's HPPC data at different SOC and temperature. For each SOC and temperature level, you must provide a separate file in text format.

At least one valid SOC level testing data is required per temperature level. For good simulation results, the range of SOC levels used in the testing should cover the SOC range simulated in the CFD runs.

The file format must follow the format as shown in the below example:

```
SOC 1.0
I   14.6
Temperature 300.0
1.0000e-01    4.2292e+00
2.0000e-01    4.2292e+00
...
1.1800e+01    4.1041e+00
...
```

The first line of the file starts with `SOC` followed by a SOC value separated by a space. The second line contains the pulse current information (`I` followed by a pulse current value). The third line provides temperature information (keyword `Temperature` followed by a value). The remaining lines contain pairs of space-separated time-voltage values. The input data must include data from the rest period, the pulse period, and the relaxation period in the testing.

You can load HPPC files in one of the following ways:

- If you have data from the battery's HPPC test, you can load them into your case by clicking the **Load User's HPPC Test Files** button.

  i. In the **Select File** dialog box that opens, browse to the directory where the test files reside.

  ii. Select all the files obtained from different temperature and SOC levels you want to use in the parameter estimation tool (use **Ctrl** or **Shift** to select multiple files).

  The selected testing files will be automatically added to the list of the files at the bottom of the **Select File** dialog box. If you accidentally selected the wrong file, you can use the **Remove** button to remove it from the selected testing file list.

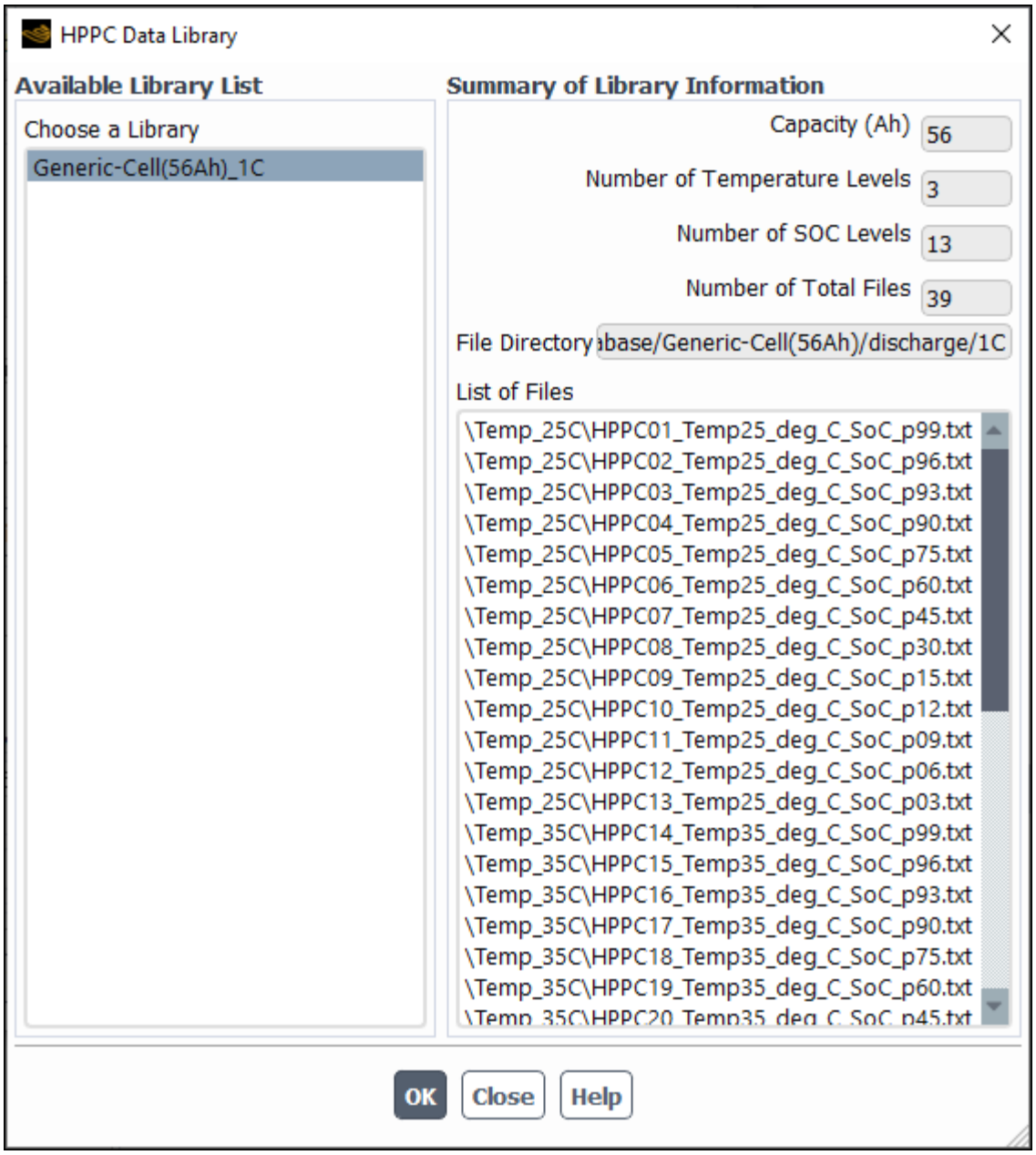
  iii. Click **OK**.

  See The Select File Dialog Box (p. 931) for details.

- You can load test files from the HPPC library provided by Ansys Fluent by clicking the **Import Files from HPPC Library** button.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.18: The HPPC Data Library Dialog Box



In the **HPPC Data Library** dialog box that opens, you can select a library from the **Available Library List**. The related information is displayed in the **Summary of Library Information** group box. Clicking **OK** will import the library files into your case.

See The HPPC Library (p. 3340) for more information.

d. After testing files are imported, click the **Fit Parameters** button to run the ECM parameter estimation tool. The fitting results will be passed to the **Battery Model** dialog box automatically.

   If you provide testing data files at different temperatures, the parameter estimation tool will perform the fitting for each temperature level. Ansys Fluent will report the fitted results in the console and automatically pass them in a table data type to the parameter data

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

table dialog boxes. The **Table** data type will be automatically selected in the **Battery Model** dialog box (**Model Parameters** tab). This will allow you to consider the temperature-dependent effect.

If you use testing data from only one temperature level, the fitting results in both polynomial and table data types will be transferred to the **Battery Model** dialog box and to the parameter data table dialog boxes, respectively. In this case, the **Polynomial** data type will be automatically selected as default in the **Battery Model** dialog box. However, you can use either data type in your simulation.

If your testing data contains both discharging and charging curves, the ECM parameter estimation tool will be run twice—once for the discharging curves, and once for the charging curves. After the fitting is complete, the **Use Different Coefficient for Charging and Discharging** button will be automatically selected, and the fitting results for each subset of data will be automatically transferred to the **Battery Model** dialog box and displayed in the corresponding **Discharging Parameters** and **Charging Parameters** group boxes (**Model Parameters** tab).

Note:

- Because the parameter estimation tool attempts to calculate six or four ECM model parameters from a non-linear data regression process for each SOC curve, the testing data for each SOC curve should be sufficiently large in order to obtain a good fitting.
- The parameter estimation tool always fits the experimental data into either six parameters used in the Chen's circuit shown in Figure 19.1: Electric Circuits Used in the ECM Model or four parameters in the reduced Chen's circuit with only one RC loop.
- To obtain optimal fitting of HPPC testing data, you can try different combinations of the circuit model and the fitting method. The six-parameter circuit model has more physics, but the four-parameter circuit model is generally more robust. As for the fitting method, the JH method is in general more robust than LM method. You can start with the six-parameter model, and then, if the fitting results are not satisfactory, switch to the four parameter model.

---

e. (optional) Visualize and check the resulting fitted curves of the battery HPPC data at different temperatures and SOC to detect any issues with the fitting process.

Once the fitting is complete, the **Parameter Estimation** dialog box expands to reveal the **Visualize Fitting Results** group box containing the **Temperature** and **State of Charge** selection lists. You can display curves of the fitted data and the experimental data from your input files at specific temperatures and C-rates by selecting a pair of values from the **Temperature** and **State of Charge** lists. The corresponding curves will be automatically plotted in the graphics window.

When the data contains both discharging and charging curves, you can select either **Discharging Data** or **Charging Data** to visualize the corresponding subset of data. The **Temperature** and **State of Charge** lists will be updated accordingly.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note the following when you are using the LM fitting method:

- The LM method is a non-linear fitting method. To have a better fitting result with this method, the JH method is always run first to provide an initial condition for the fitting parameters. When you plot fitted curves, the fitting result from the JH method will also be plotted for your reference.
- If the fitting fails, a warning message will be printed in the console window, and the fitted curve for the LM method will not be plotted.

f. Click **Close** in the **Parameter Estimation** dialog box.

Note:

The ECM parameter estimation tool can also be accessed through the text user interface (TUI) as described in Using the Parameter Estimation Tool for the ECM Model in the TUI (p. 3392).

---

8. (**Chen's original**, **Polynomial**, or **Table** data type only) You can visualize the resulting resistors' resistances (R_s, R_1, and R_2), capacitors' capacitances (C_1 and C_2), and the corresponding time constants (R_1*C_1 and R_2*C_2) varying with DOD by selecting an appropriate function from the **Parameter** drop-down list (**Visualize Model Parameters** group box) and clicking the **Plot** button. The resulting curves will be plotted in the graphics window.

31.2.3.4.4.1. The HPPC Library

Ansys Fluent provides a small HPPC library that you can use for importing HPPC test data into your case. Currently, the library contains only a single battery, called `Generic-Cell`. In future releases, the library will be expanded to include commonly used batteries. The purpose of the `Generic-Cell` library is to show the folder structure of test files as further described.

The Ansys Fluent HPPC library is located in the following directory:

path`/fluent24.2.0/addons/msmdbatt/hppc_database/`

where *path* is the Ansys Fluent installation directory (for example, `C:\Program Files\ANSYS Inc\v242\fluent\fluent24.2.0`).

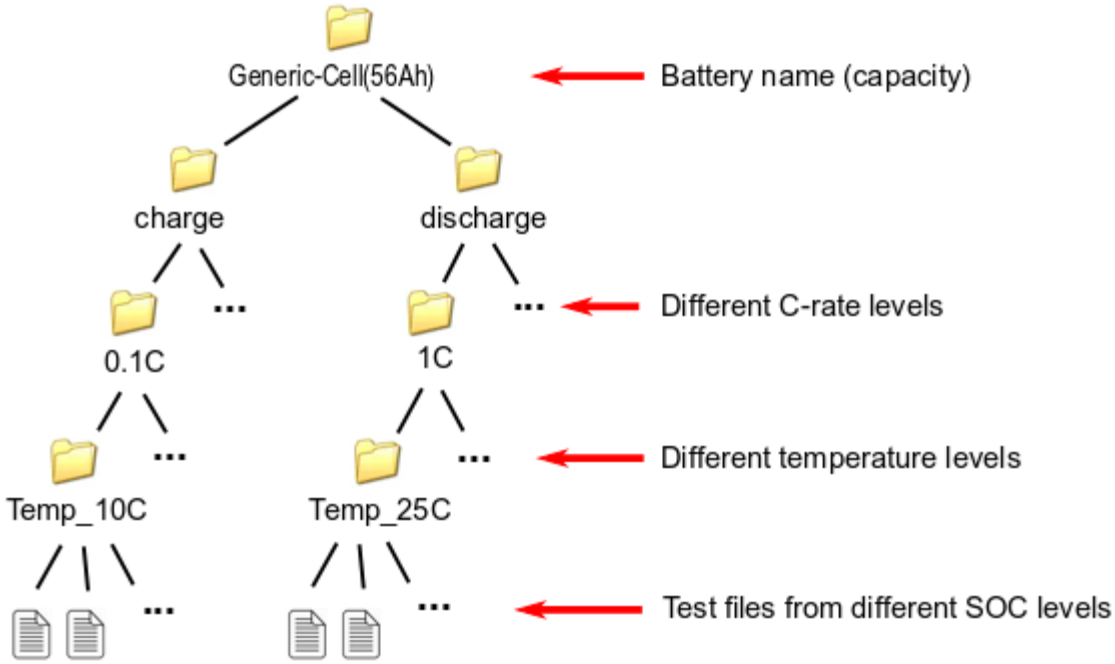
In a standard HPPC test, a pulse of current is applied to a battery at a particular temperature and SOC level, and the voltage response of the battery is measured. The voltage response curve depends on:

- the nature of the pulse (discharging or charging)
- the magnitude of the current
- the temperature level
- the SOC level of the battery

To reflect such testing procedure, the test files are organized in the folder structure as shown in Figure 31.19: Overview of the Folder Structure in the HPPC Library (p. 3341).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.19: Overview of the Folder Structure in the HPPC Library



The naming convention is as follows:

- The battery name can be any string. The capacity of the battery must be put in the parenthesis with a number followed by the unit "Ah".
- The names of the first-level folder must be `discharge` and `charge`.
- The number of the second- and third-level folders and their names can be arbitrary.

You can create your own library following the outlined conventions and put it in the `hppc_database` folder. The name of your library will appear in the **HPPC Data Library** dialog box. This way, the HPPC library can be expanded with the proprietary data and shared between colleagues within an organization.

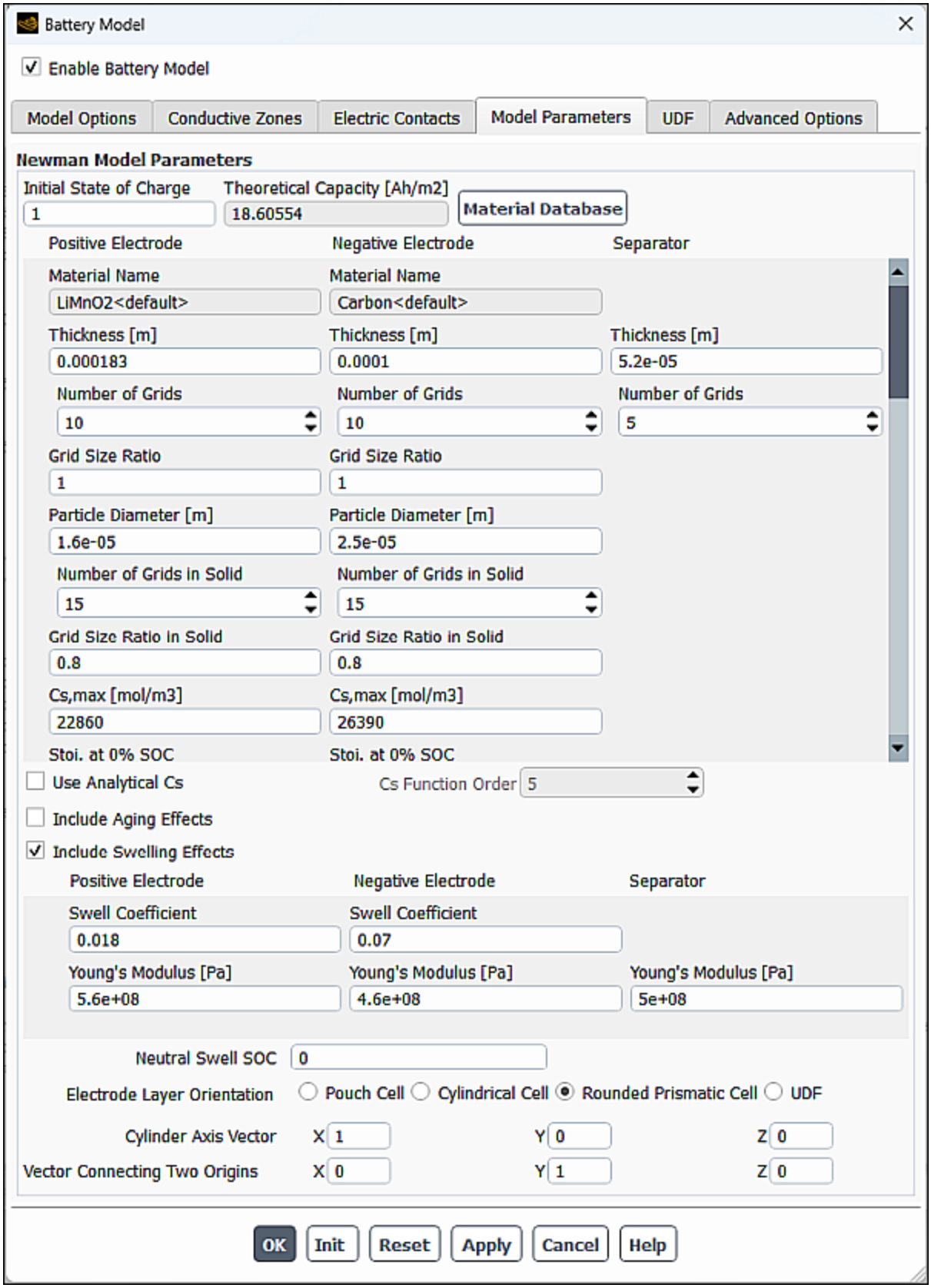
Note:

Currently, the parameter estimation tool can only take data from the `discharge` branch and only handle data from one C-rate level. Data from different C-rate levels are treated as different battery libraries. You should select the library with the C-rate level close to that at which the battery is operating in your application. The `charge` branch is not used so far. However, support for the `charge` branch is expected in future releases as the solver capability will be expanded in Ansys Fluent.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.2.3.4.5. Inputs for the Newman's P2D Model

Figure 31.20: Model Parameters Tab—Newman's P2D Model



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Theoretical Capacity is computed automatically from the Newman input parameters you specify and reported in the **Model Parameters** tab.

Specify the following settings in the **Model Parameters** for the Newman's P2D model:

1. Specify **Initial State of Charge**. The default value of 1 for **Initial State of Charge** indicates the fully-charged state of the battery cell.

2. Specify the following parameters in the **Positive Electrode**, **Negative Electrode**, and **Separator** columns:

   Thickness

   are l_p, l_n, and l_s in Equation 19.30.

   Number of Grids

   is a number of grid points used to discretize the positive electrode, negative electrode, and separator zones.

   Grid Size Ratio

   is used to create a biased mesh. The default value of 1 corresponds to a uniform mesh.

   Particle Diameter

   is the particle diameter of the active material.

   Number of Grids in Solid

   is the number of grid points used to discretize the sphere of particles.

   Grid Size Ratio in Solid

   is used to create a biased mesh for the solid phase. The default value of 1 corresponds to a uniform mesh.

   Cs,max

   is the maximum concentration of Lithium in the solid phase (used in Equation 19.20).

   Stoi. at 0% SOC

   is the stoichiometry at 0% SOC that is used to compute theoretical capacity and initial lithium concentration in the electrode.

   Stoi. at 100% SOC

   is the stoichiometry at 100% SOC that is used to compute theoretical capacity and initial lithium concentration in the electrode.

   Init. Cs

   is the initial lithium concentration in the electrode.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Init. Ce

is the initial lithium concentration in the electrolyte phase.

VOF of Electrolyte

is the volume fraction of the electrolyte material in the electrode, ε_e in Equation 19.21.

Filler Fraction

is the volume fraction of the filler material in the electrode, ε_f in Equation 19.28.

Ds

is the diffusion coefficient of lithium ion in the electrode at reference temperature T_{ref}=25°C, $D_{s,ref}$ in Equation 19.21.

Activation Energy E_d

is E_d in Equation 19.21.

Brugg

is the Bruggeman tortuosity exponent, β in Equation 19.21.

Conductivity

is the electric conductivity, σ in Equation 19.21.

Ref. Rate Constant

is the rate constant at reference temperature T_{Ref}=25°C, $k_{m,ref}$ in Equation 19.21.

Activation Energy E_r

is E_r in Equation 19.21.

Trans. Coef. a

is the charge transfer coefficient at anode, α_a in Equation 19.20.

Trans Coef. c

is the charge transfer coefficient at cathode, α_c in Equation 19.20.

OCP

is the open circuit potential of the electrode material, U in Equation 19.19. Usually, it is a function of state of lithiation (sol) $c_s/c_{s,\max}$. It can be defined by a piece-wise linear function or a UDF.

De

is the diffusion coefficient of Li+ in the electrolyte phase, D_e in Equation 19.21.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

t+ Factor

is the transference number of lithium ion, t_+ in Equation 19.21.

Ionic Cond

is the ionic conductivity, k in Equation 19.22.

Activity Term

Is the activity related term, $\frac{d\ln f_{\pm}}{d\ln c_e}$ in Equation 19.23.

You can specify the open circuit potentials, the diffusion coefficients, the conductivities, the transference number of lithium ion, and the activity related term as a **constant**, a **piecewise-linear** function, or a **polynomial** function. Alternatively, you can specify a user-defined function (UDF) for these parameters. For more information, see `DEFINE_BATTERY_PROPERTY` in the *Fluent Customization Manual*.

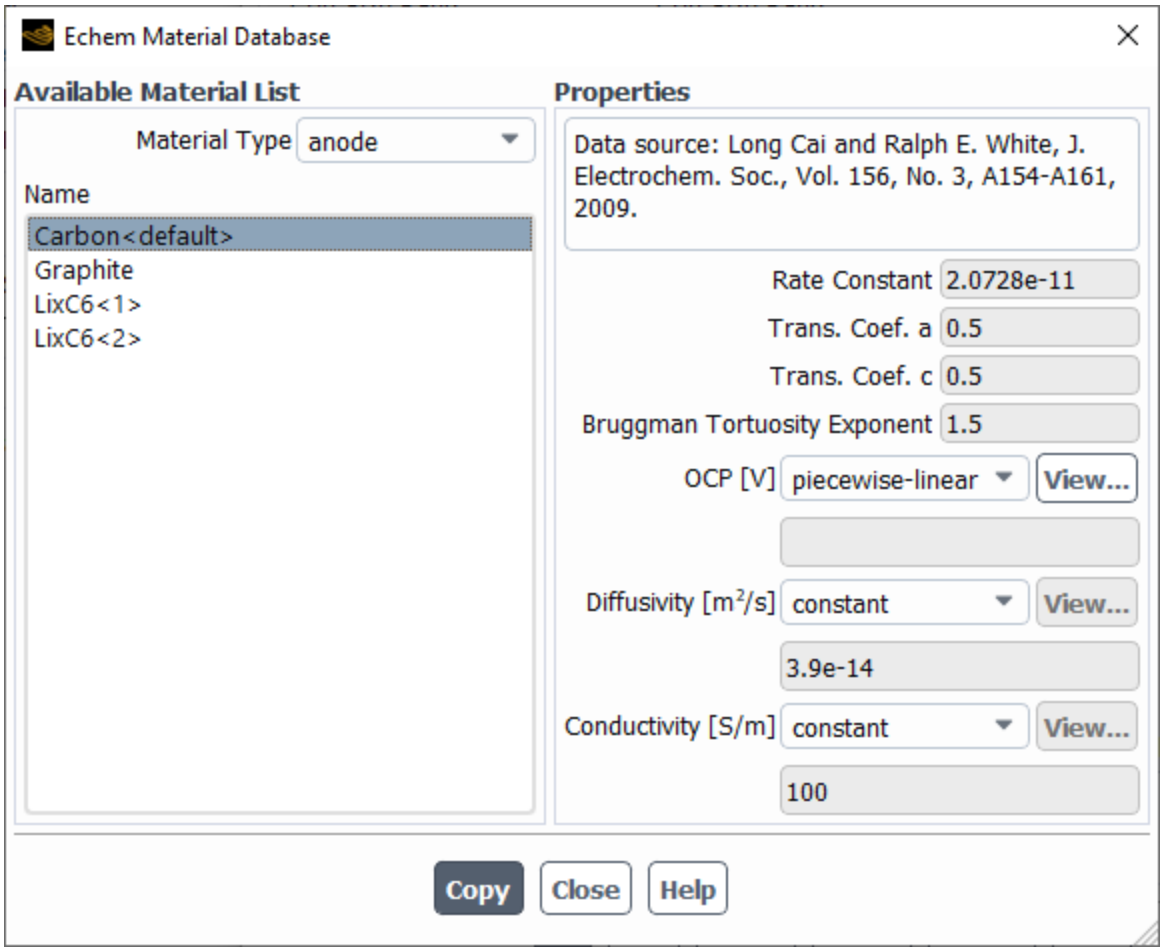
Some of the above inputs are related to the material properties. You can use the Ansys Fluent echem material database that offers commonly-used anode, cathode, and electrolyte materials.

3. If you want to import material properties from the material database:

   a. Click the **Material Database** button.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.21: The Echem Material Database Dialog Box



b. In the **Echem Material Database** dialog box that opens, select the **Material Type** from the following options:
   - **anode**
   - **electrolyte**
   - **cathode**

c. From the **Name** selection list, select the material that you want to copy.

   The properties of the selected material are displayed in the **Properties** group box for information only.

d. Click **Copy**.

   The names and the properties of the selected materials are copied to the correspondent fields in the **Newman Model Parameters** group box.

Note:

The Ansys Fluent echem material database is saved in an ASCII file located at `<installation directory>/fluent/fluent24.2.0/addons/msmd-`

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

`batt/lib/echem_database.scm`, where `<installation directory>` is where you installed the software. You can modify the data in the database or add more materials into that file if you want to build up your own material database.

4. If you want to use the reduced order method to solve the lithium diffusion equation in solid [90], select the **Use Analytical Cs** check box and enter the value for the **Cs Function Order**.

   By default, Ansys Fluent discretizes the lithium conservation equation and solves it numerically.

5. If you want to include aging effects into the Newman's P2D model simulation as steady-state input profiles:

   a. Select the **Include Aging Effects** check box.

   b. (optional) Import customized aging effect profiles by either entering the file name in the **Import Aging Effects Profile** text entry field or using the **Browse...** button to select it.

      Once you have imported the aging effects profile, they will be automatically stored with your case file.

      Note:

      If the aging effects profiles have been generated by running the physics-based battery life model in the standalone mode as described in Simulating the Aging Process of a Battery (Standalone Mode) (p. 3356), these profiles are automatically available in your case, and you do not need to import them into your case.

   c. From the **Select Aging Effects Profiles** drop-down list, select a profile for the battery aging conditions.

   The battery aging information (such as SEI concentration, surface film resistance, and porosity) stored in the selected profile will be used in the Newman's P2D model calculation. For theoretical background about the physics-based battery life model, see Physics-Based Battery Life Model in the *Fluent Theory Guide*.

6. If you want to consider the deformation of the battery during charge and discharge:

   a. Enable **Include Swelling Effects**.

      The dialog box expands to reveal the swelling modeling parameters.

   b. Specify the following parameters in the **Positive Electrode**, **Negative Electrode**, and **Separator** columns below the **Include Swelling Effects** check box:

      Swell Coefficient

      is the volume growth rate of the active material particles with respect to the normalized particle lithium concentration, ω in Equation 19.69 in the *Fluent Theory Guide*.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Young's Modulus

is E in Equation 19.72, Equation 19.79, and Equation 19.80 in the *Fluent Theory Guide*.

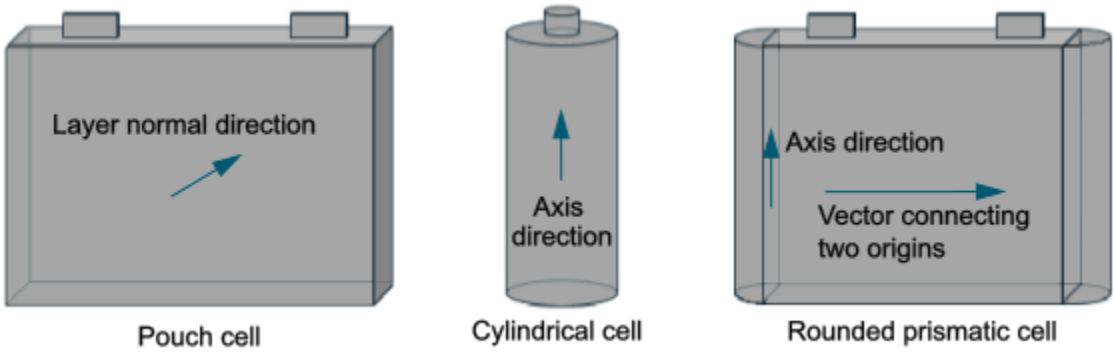
Neutral Swell SOC

is the battery's state of charge when swelling is considered neutral.

When the swelling effects are included, **Thickness**, **Particle Diameter**, **VOF of Electrolyte**, and **Filler Fraction** for positive electrode, negative electrode, and, when applicable, separator are specified at the **Neutral Swell SOC**. During the echem model swelling calculations, their true values will fluctuate around the values you entered.

c. Specify the **Electrode Layer Orientation** ($\underline{N}$ in Equation 16.38 in the *Fluent Theory Guide*).

Figure 31.22: Electrode Layer Orientation



The following orientation types are available (see Figure 31.22: Electrode Layer Orientation (p. 3348)):

- **Pouch Cell**

  For the **Pouch Cell** geometry type, you need to specify the **X**, **Y**, **Z** components of the **Layer Normal Vector**, which is the unit vector normal to the battery electrode sandwich layer surface.

- **Cylindrical Cell**

  For the **Cylindrical Cell** geometry type, you need to specify the **X**, **Y**, **Z** components of the **Cylinder Axis Vector**.

- **Rounded Prismatic Cell**

  For the **Rounded Prismatic Cell** geometry type, you need to specify the **X**, **Y**, **Z** components of two vectors - the **Cylinder Axis Vector** and the **Vector Connecting Two Origins** of the prismatic cell.

- **UDF**

  This option allows you to specify an electrode layer orientation for more complex battery geometry via the `DEFINE_BATTERY_SWELL_LAYER_N` user-defined function. See

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DEFINE_BATTERY_SWELL_LAYER_N in the *Fluent Customization Manual* for more information.

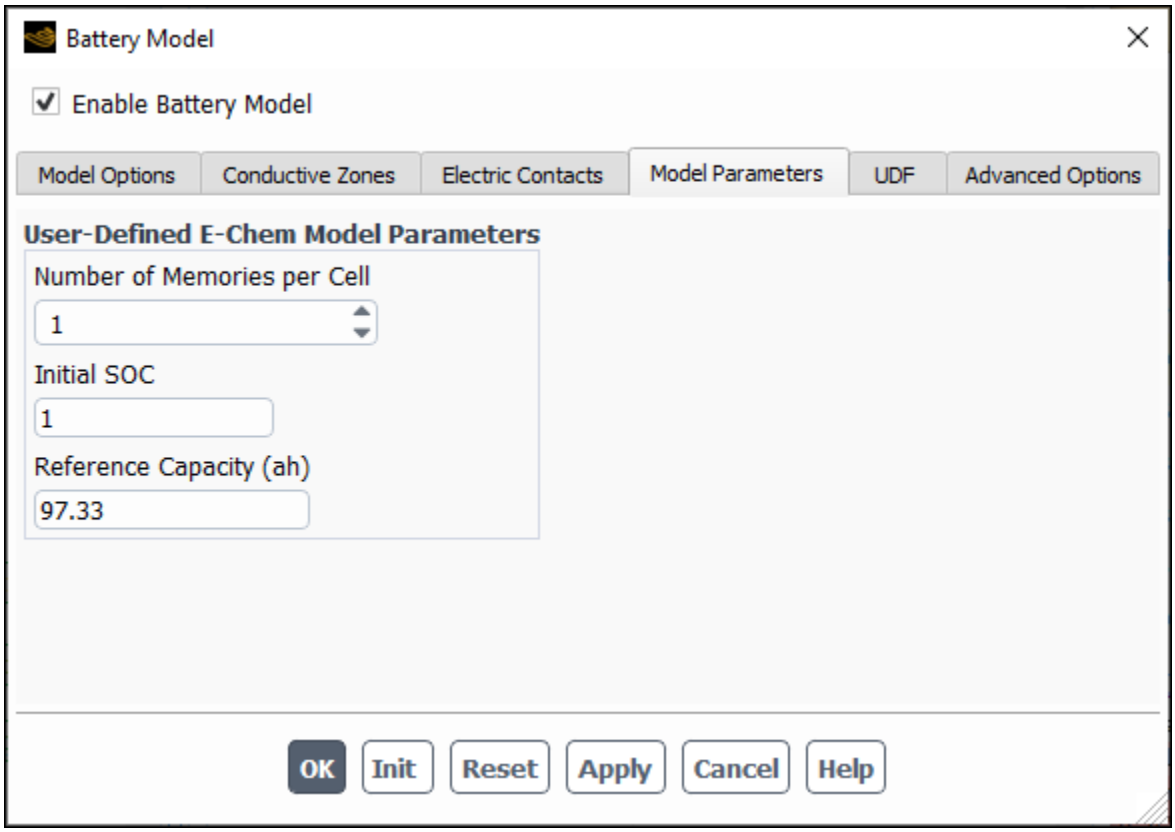
For steps on setting a swelling simulation, see Setting a Battery Swelling Case (p. 3389).

Note:

Model parameters are battery specific. The default values are taken from Cai & White [90]. To simulate a battery of different design, you should provide a user-specified set of battery parameters.

31.2.3.4.6. Input for the User-Defined E-Model

Figure 31.23: Model Parameters Tab—User-Defined E-Model



The following parameters are available under **User-Defined E-Chem Model Parameters**:

Number of Memories per Cell

is the number of submodel parameters that need to be saved per computational cell during a simulation.

Initial SOC

is the value of the battery initial state of charge.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

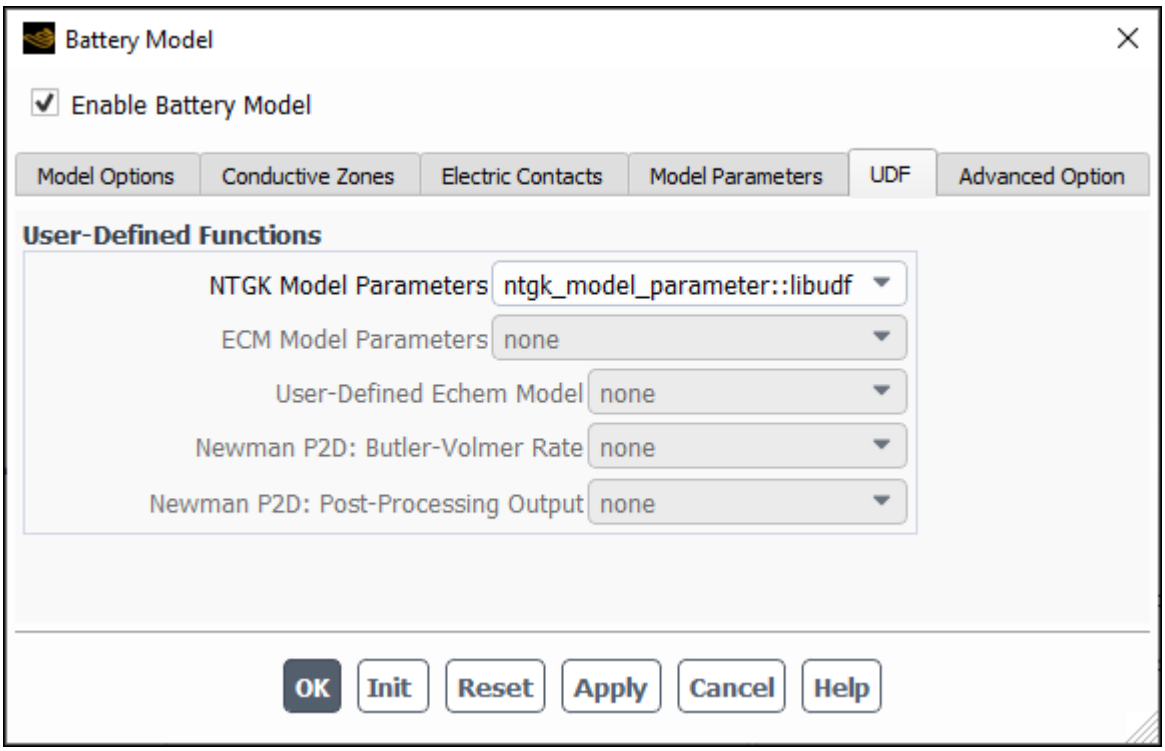
Reference Capacity

is the battery capacity (Ah).

31.2.3.5. Hooking User-Defined Functions

In the **UDF** tab, you can customize various battery model parameters by hooking up your user-defined functions (UDFs).

Figure 31.24: The Battery Model Dialog Box (UDF Tab)

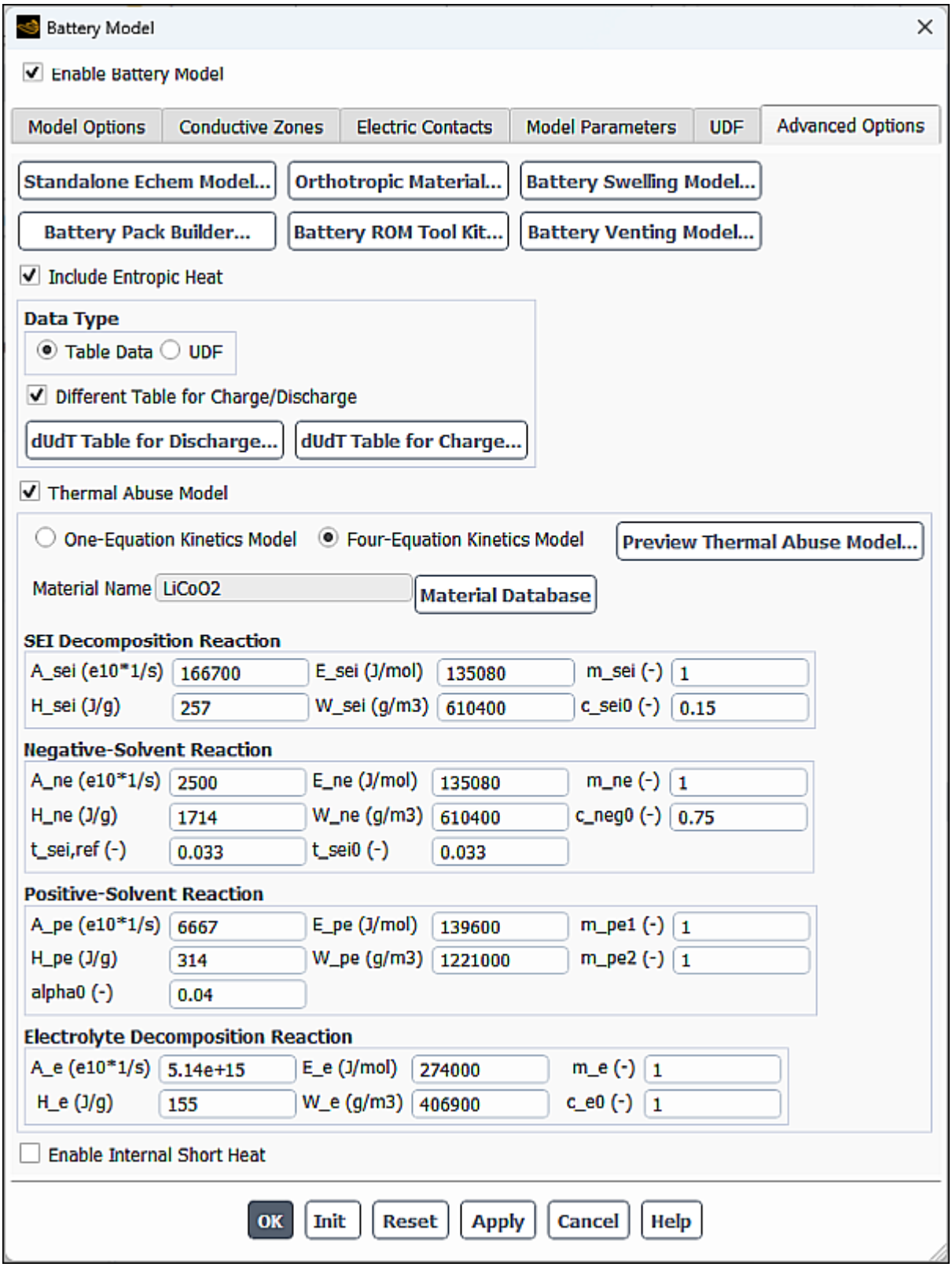


Refer to Battery Model `DEFINE` Macros in the *Fluent Customization Manual* for a list of all available UDFs for specific battery models, their detailed descriptions and examples on how to use them.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.2.3.6. Specifying Advanced Options

Figure 31.25: The Battery Model Dialog Box (Advanced Options Tab)



Using the **Advanced Options** tab of the **Battery Model** dialog box, you can:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Check the selected electrochemical model behavior before running the CFD case by performing a quick simulation of the electrochemical model in a standalone mode
- Run the physics-based battery life model to simulate the aging process of a battery through electric load cycles in a standalone mode
- Use the Battery Pack Builder tool to construct a battery pack from existing Ansys Fluent simulations of the battery module and cold plate
- Create LTI-ROM or SVD-ROM training data and load SVD-ROM results into Ansys Fluent for post-processing
- Define orthotropic thermal conductivity for all battery active zones
- (NTGK and ECM models only) Enable and set the empirical-based battery swelling model for an FSI coupled simulation
- Enable and set the battery venting model
- Include the entropic heat term in the battery thermal analysis when any of the electro-chemical submodels is used
- Perform thermal abuse analysis

For additional information, see the following sections:

31.2.3.6.1. Running the Standalone Echem Model
31.2.3.6.2. Simulating the Aging Process of a Battery (Standalone Mode)
31.2.3.6.3. Using the Battery Pack Builder Tool
31.2.3.6.4. Using the Battery ROM Tool Kit
31.2.3.6.5. Defining Orthotropic Thermal Conductivity
31.2.3.6.6. Using the Empirical-Based Battery Swelling Model
31.2.3.6.7. Battery Venting Model
31.2.3.6.8. Including the Entropic Heat Effects
31.2.3.6.9. Using the Thermal Abuse Model

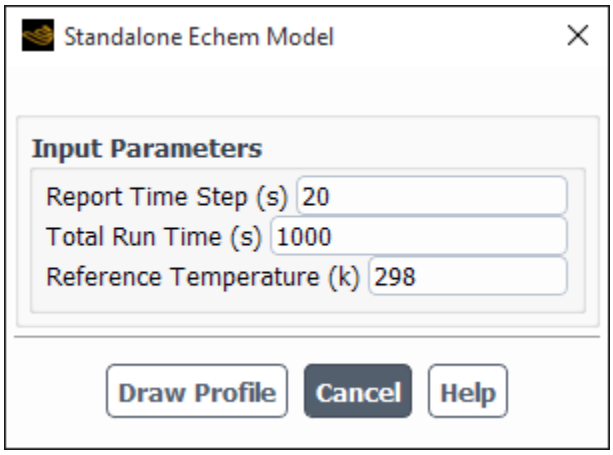
31.2.3.6.1. Running the Standalone Echem Model

Prior to running a coupled electrochemical-thermal simulation, you should perform a quick analysis of your electrochemical model by simulating it in a standalone mode. The standalone model calculation uses the operating conditions that you have specified in the **Model Options** tab (such as electric load type and value) and the echem model parameters that you have defined in the **Model Parameters** tab. This preliminary analysis will allow you to confirm the electrochemical behavior of the selected submodel and, if necessary, adjust the model settings.

To run the echem model in a standalone mode:

1. Click **Run Echem Model Standalone...**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



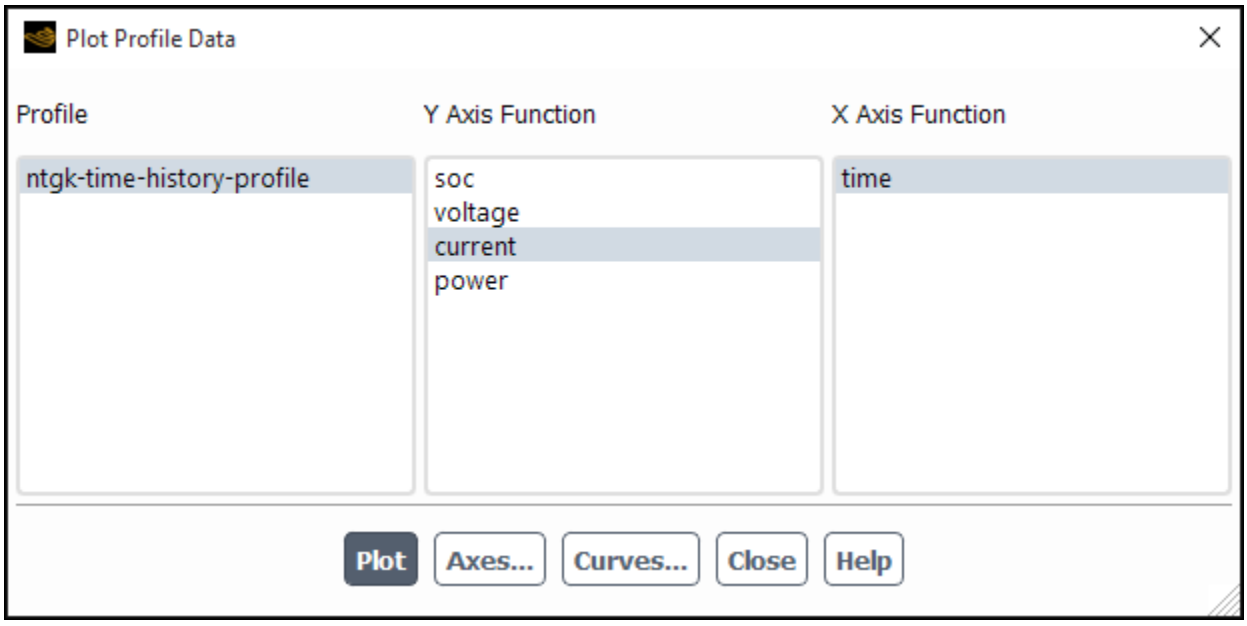
2. In the **Standalone Echem Model** dialog box that opens, specify the following parameters:

   - **Report Time Step**: The time interval at which the results will be saved and reported.
   - **Total Run Time**: The total time for running the stand-alone model simulation.
   - **Reference Temperature**: The temperature level at which you want to simulate your electrochemical model.
   - (Neman's P2D model with battery swelling effects only) **Mechanical Load Type**: The external mechanical load that causes electrode elastic deformation (see Modeling Swelling in the E-Chem Standalone Mode in the *Fluent Theory Guide* for more information). You can select and specify the following load types:
     - **External Pressure**: The external normal force exerted on the battery electrode layer end, P_{ext} in Equation 19.78 in the *Fluent Theory Guide*. A positive value represents compression.
     - **Total Deformation**: The percentage of the total thickness change of the battery electrode layer, the right-hand side of Equation 19.79 in the *Fluent Theory Guide*.
     - **External Stiffness**: The effective stiffness of external constraint applied to the battery electrode layer, E_{ext} in Equation 19.80 in the *Fluent Theory Guide*.

3. Initiate the electrochemical model simulation by clicking **Draw Profile**.

   Once the calculation is complete, the **Plot Profile Data** dialog box opens.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Using the **Plot Profile Data** dialog box, visualize and analyze the results of the standalone echem model simulation. Different profiles are available for different electrochemical models.

   - For the NTGK model, a time history profile, called `ntgk-time-history-profile`, is generated after the calculation is complete. You can draw the X-Y plots of the following variables as a function of time:

     – **soc**: the battery state of charge (SOC)

     – **voltage**: the battery voltage

     – **current**: the battery electric current

     – **power**: the battery power output

   - For the ECM model, a time history profile, called `ecm-time-history-profile`, will be generated. In addition to the variables available for the NTGK model, you can also draw the X-Y plots of the following variables as a function of time:

     – **v1**: V_1 in Equation 19.9

     – **v2**: V_2 in Equation 19.9

   - For the Newman P2D model, the following two profiles will be generated:

     – Time history profile `p2d-time-history-profile`

     – Spatial field profile called `p2d-field-profile`

     For the `p2d-time-history-profile` profile, you can plot the time history profile for the following variables:

     – **current** (see above)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **voltage** (see above)
- **power** (see above)
- **ce_x=0**
- **ce_x=lp**
- **ce_x=lps**
- **ce_x=lpsn**
- **cs_x=0**
- **cs_x=lp**
- **cs_x=lps**
- **cs_x=lpsn**
- **phis_x=0**
- **phis_x=lp**
- **phis_x=lps**
- **phis_x=lpsn**
- **phie_x=0**
- **phie_x=lp**
- **phie_x=lps**
- **phie_x=lpsn**

where

ce is the lithium-ion concentration in electrolyte phase at different locations

x=0 denotes the location of the outside face of the positive electrode

x=lp denotes the interface of the positive electrode and separator

x=lps denotes the interface between the separator and negative-electrode

x=lpsn denotes the outside face of the negative electrode

cs is the lithium concentration at the surface of the active material in electrode at different locations

phis is the electric potential in the electrode solid phase at different locations

phie is the electric potential in the electrolyte liquid phase at different locations

For the `p2d-field-profile` profile, you can plot the spatial profiles of the following variables at the end of the simulation:

- **phi_e**: the electric potential in electrolyte phase

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **phi_s**: the electric potential in electrode phase
- **ce**: the lithium-ion concentration in electrolyte phase
- **cs_center**: the lithium concentration in the center of the active electrode particles
- **cs_average**: the average lithium concentration in the active electrode particles
- **cs_surf**: the lithium concentration at the surface of the active electrode particles

Note:

SI units are always used for all the variables in the X-Y plot.

31.2.3.6.2. Simulating the Aging Process of a Battery (Standalone Mode)

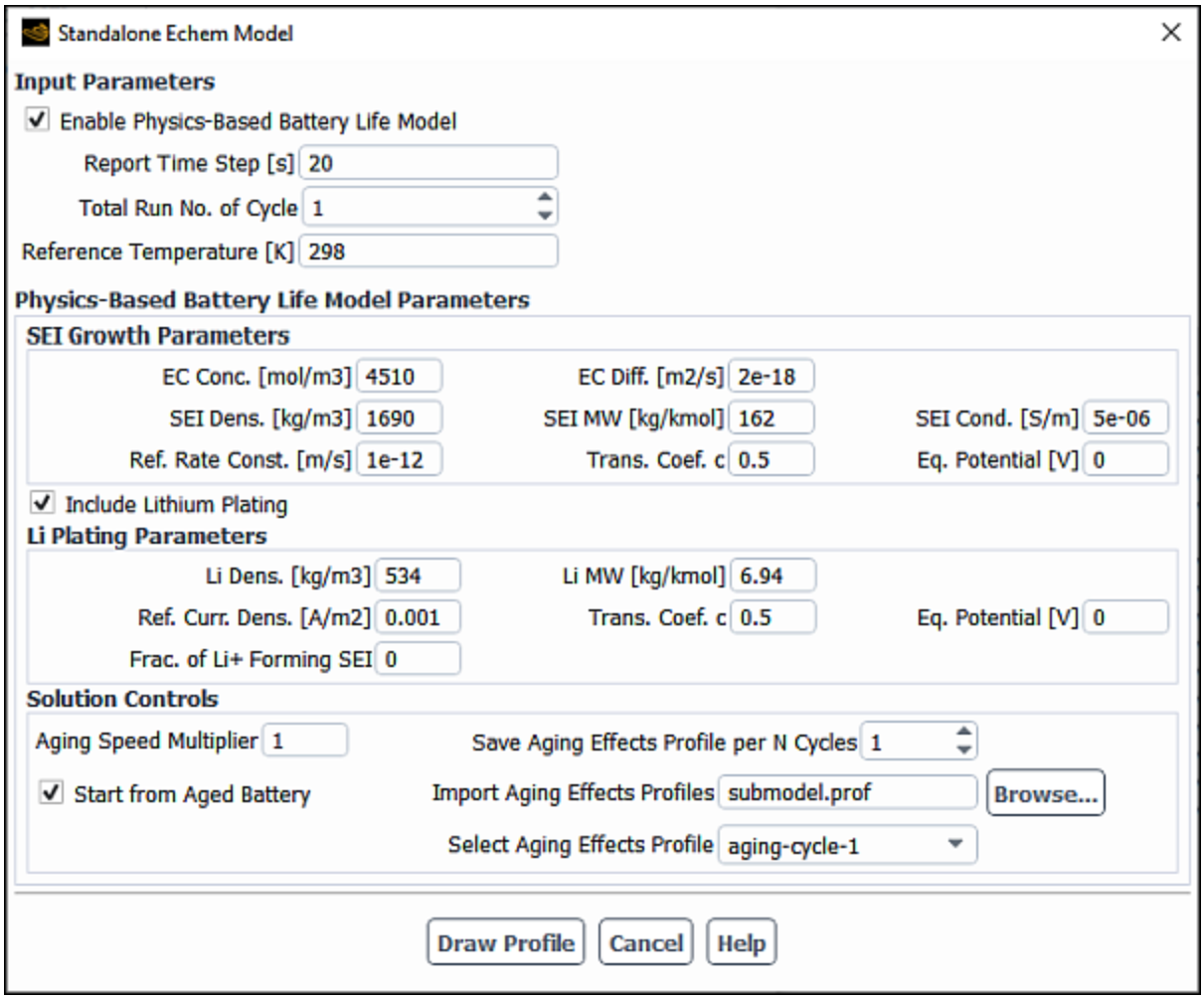
In practical Newman's P2D model simulations of battery aging through electric load cycles, the physics-based battery life model is solved on the electrode scale in the standalone mode. Further, the standalone simulation results can be included in a 3D CFD simulation to account for the battery aging effects as described in Inputs for the Newman's P2D Model (p. 3342). For more information about the theory behind the physics-based battery life model, see Physics-Based Battery Life Model in the *Fluent Theory Guide*.

To run the physics-based battery life model in the standalone mode:

1. In the **Model Options** tab of the **Battery Model** dialog box, select **Newman P2D Model** as the E-chemistry model.
2. In the **Solution Options** group box, select **Using Profile**.
3. In the **Profile Types** group box, select either **Time-Scheduled** or **Event-Scheduled** and specify a profile file to define the boundary conditions of a single electric load cycle.
4. In the **Advanced Options** tab, click **Run Echem Model Standalone...**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.26: The Standalone Echem Model Dialog Box (Battery Life Model)



5. In the **Standalone Echem Model** dialog box, select **Enable Physics-Based Battery Life Model**. The dialog box expands to reveal relevant model inputs in the **Physics-Based Battery Life Model Parameters** group box.

6. In addition to specifying **Report Time Step** and **Reference Temperature** (see Running the Standalone Echem Model (p. 3352) for their definitions), specify **Total Run No. of Cycle**, which is the number of times the electric load profile you provided earlier is automatically looped over during the simulation.

   Note:

   Prior to using a large number of cycles, you should run a test with a small number of cycles (for example, less than 20) as a sanity check and an estimation of computational time.

7. Set up the **Physics-Based Battery Life Model Parameters**.

   a. In the **SEI Growth Parameters** group box, you can specify:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

EC Conc.

is the concentration of ethylene carbonate (EC) in the bulk electrolyte; c_{EC} in Equation 19.58 in the *Fluent Theory Guide*.

EC Diff.

is the diffusivity of EC; D_{EC} in Equation 19.58 in the *Fluent Theory Guide*.

SEI Dens.

is the density of solid-electrolyte-interface (SEI); ρ_{SEI} in Equation 19.60-Equation 19.62 in the *Fluent Theory Guide*.

SEI MW

is the molecular weight of SEI; M_{SEI} in Equation 19.60-Equation 19.62 in the *Fluent Theory Guide*.

SEI Cond.

is the ionic conductivity of SEI; k_{SEI} in Equation 19.56 in the *Fluent Theory Guide*.

Ref. Rate Const.

is the rate constant of SEI formation reaction; κ_{SEI} in Equation 19.61 in the *Fluent Theory Guide*.

Trans. Coef. c.

is the cathodic charge transfer coefficient of SEI formation reaction; $\alpha_{c,SEI}$ in Equation 19.56 in the *Fluent Theory Guide*.

Eq. Potential

is the equilibrium potential of SEI formation reaction; η_{SEI} in Equation 19.56 in the *Fluent Theory Guide*.

b. If you want to include the effects of lithium plating in your simulation, select **Include Lithium Plating** and then specify the following **Lithium Plating Parameters**:

Li Dens.

is the density of plated lithium; ρ_{Li} in Equation 19.60-Equation 19.62 in the *Fluent Theory Guide*.

Li MW

is the molecular weight of plated lithium; M_{Li} in Equation 19.60-Equation 19.62 in the *Fluent Theory Guide*.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ref. Curr. Dens.

is the transfer current density of lithium deposition reaction; i_{Li} in Equation 19.57 in the *Fluent Theory Guide*.

Trans. Coef. c.

is the cathodic charge transfer coefficient of lithium deposition reaction; $\alpha_{c,Li}$ in Equation 19.57 in the *Fluent Theory Guide*.

Eq. Potential

is the equilibrium potential of lithium deposition reaction; η_{Li} in Equation 19.57 in the *Fluent Theory Guide*.

Frac. of Li forming SEI

is the fraction of deposited lithium ions that becomes SEI; β in Equation 19.54 and Equation 19.55 in the *Fluent Theory Guide*.

c. In the **Solution Controls** group box, specify controls you want to use in your simulation.

i. Enter the **Aging Speed Multiplier**, which will be applied as a scaling factor to the cycle-to-cycle increment of the SEI and plated lithium concentrations after each cycle.

ii. In the **Save Aging Effects Profile per N Cycles** number entry box, specify the cycle interval at which the aging effects profiles are saved.

iii. If you want to start your simulation from an aged battery rather than from a new one, select **Start from Aged Battery**.

iv. Enter the name of the file that contains the aging effects profiles in the **Import Aging Effects Profile** text entry field or use the **Browse...** button to select it. See below for more information about aging effects profile files.

Note:

When you select **Start from Aged Battery**, you do not need to import your customized profile file each time you run a battery simulation in the standalone mode. Once imported, the aging effects profile is stored within the case file until deleted manually or overwritten by importing a profile with the same name.

---

v. From the **Aging Effects Profiles** drop-down list, select a profile for the simulation initial conditions.

8. Initiate the electrochemical model simulation by clicking **Draw Profile**.

Once the simulation is complete, the aging effects profiles generated in a standalone simulation will be automatically loaded into Ansys Fluent and can be visualized and analyzed using the

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Plot Profile Data dialog box that opens. See Running the Standalone Echem Model (p. 3352) for more information about the **Plot Profile Data** dialog box.

The following variables generated by the physics-based battery life model are saved in the time-history profile called `p2d-time-history-profile` and in the cycle-history profile called `aging-cycle-history-profile`:

- `cs_sei_x=lps`: Concentration of SEI (mol/m^3)
- `cs_li_x=lps`: Concentration of plated lithium (mol/m^3)
- `r_film_x=lps`: Surface resistance of the anode particle (Ohm-m^2)
- `theoretical_capacity`: Theoretical capacity of the battery electrode layer (Ah/m^2)

Here, `x=lps` denotes the interface between the separator and the negative-electrode.

Note:

The cycle-history profile saves the battery aging variables at the end of each electric load cycle, that is, whenever the standalone simulation reaches the end of the **Time-Scheduled** or **Event-Scheduled** profile you specified in the **Battery Model** dialog box.

Aging Effects Profiles

The 1D spatial results of the physics-based battery life model are saved in aging effects profiles, which are Ansys Fluent built-in format profiles (see Profiles (p. 1564) for more information about profile files). In the standalone mode, all the generated aging effects profiles are saved in the file called `aging-field-profile.prof`. The profiles are named `aging-cycle-`*`number`*, where *`number`* is the cycle number. Each profile contains the following field variables sampled at the cycle end:

- `x`: Coordinate of the anode grid center (m)
- `cs_sei`: Concentration of SEI (mol/m^3)
- `cs_li`: Concentration of plated lithium (mol/m^3)
- `eta_sei`: Overpotential of the SEI formation reaction (V)
- `eta_li`: Overpotential of the lithium deposition reaction (V)
- `j_sei`: Current flux at the active material pore-wall due to SEI formation reaction (A/m^2)
- `j_li`: Current flux at the active material pore-wall due to lithium deposition reaction (A/m^2)
- `delta_film`: Surface film thickness of the anode particles (m)
- `r_film`: Surface film electric resistance of the anode particles (Ohm-m^2)
- `por_e`: Electrolyte porosity (non-unit)

Apart from postprocessing, the aging effects profiles can be used in the following two scenarios:

- **Scenario A**: The profile can be used as initial conditions in the standalone simulation when you select the **Start from Aged Battery** option in the **Standalone Echem Model** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Scenario B**: The profile can be used as model inputs of the E-chemistry submodel calculations in the CFD simulation when you select **Include Aging Effects** in the **Model Parameters** tab. In this scenario, the aging effects are included into the Newman's P2D model using the profile as a frozen-in-time snapshot. See Inputs for the Newman's P2D Model (p. 3342) for details.

You can also create your customized aging effects profiles based on experimental data and import them into case. A customized profile should conform to the format of the Ansys Fluent built-in profiles described in Profiles (p. 1564) and may contain different field variables based on different scenarios:

- For Scenario A, a customized profile must at least contain the following variables:
  - `x`
  - `cs_sei`
- For Scenario B, a customized profile must at least contain the following variables:
  - `x`
  - `c_sei`
  - `r_film`
  - `por_e`

The customized profile can optionally contain `cs_li`, which is otherwise treated as zero.

Note:

The x-coordinate of an aging effects profile (that is, the mesh point number and anode thickness) can be different from the Newman's P2D model mesh. A mapping is automatically performed in such a case.

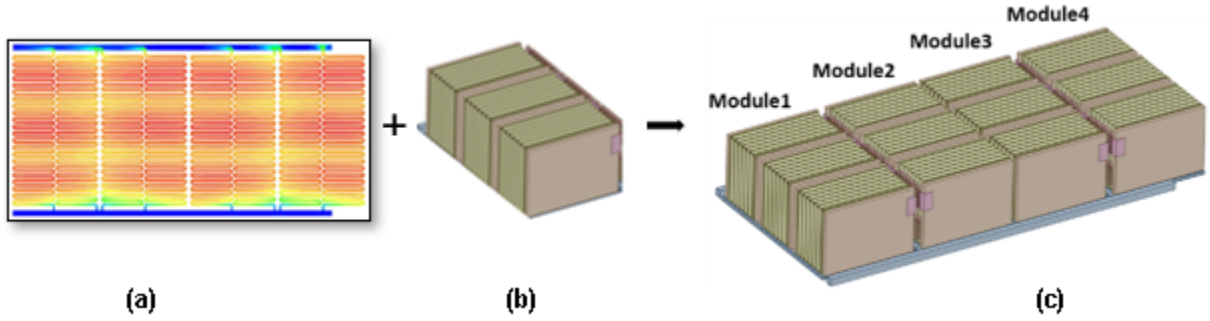
31.2.3.6.3. Using the Battery Pack Builder Tool

Many industrial applications use a battery pack in which multiple battery modules are connected in either series or parallel. Within a pack, modules are usually laid out in some repeated pattern. In addition, a cold plate may be attached to the batteries. You can use the brute force method to build the whole pack geometry and then perform a simulation of the battery pack in Ansys Fluent. However, before running the pack simulation, separate simulations of individual battery module and a cold plate are often required. The Battery Pack Builder tool allows you to construct a battery pack directly within Ansys Fluent from existing Ansys Fluent simulations of the battery module and cold plate.

Figure 31.27: Battery Pack with a Cold Plate (p. 3362) shows an example of a battery pack where the cold plate (a) and a single module (b) have been already simulated separately in Ansys Fluent. The Battery Pack Builder constructs the pack geometry (c) within Ansys Fluent.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

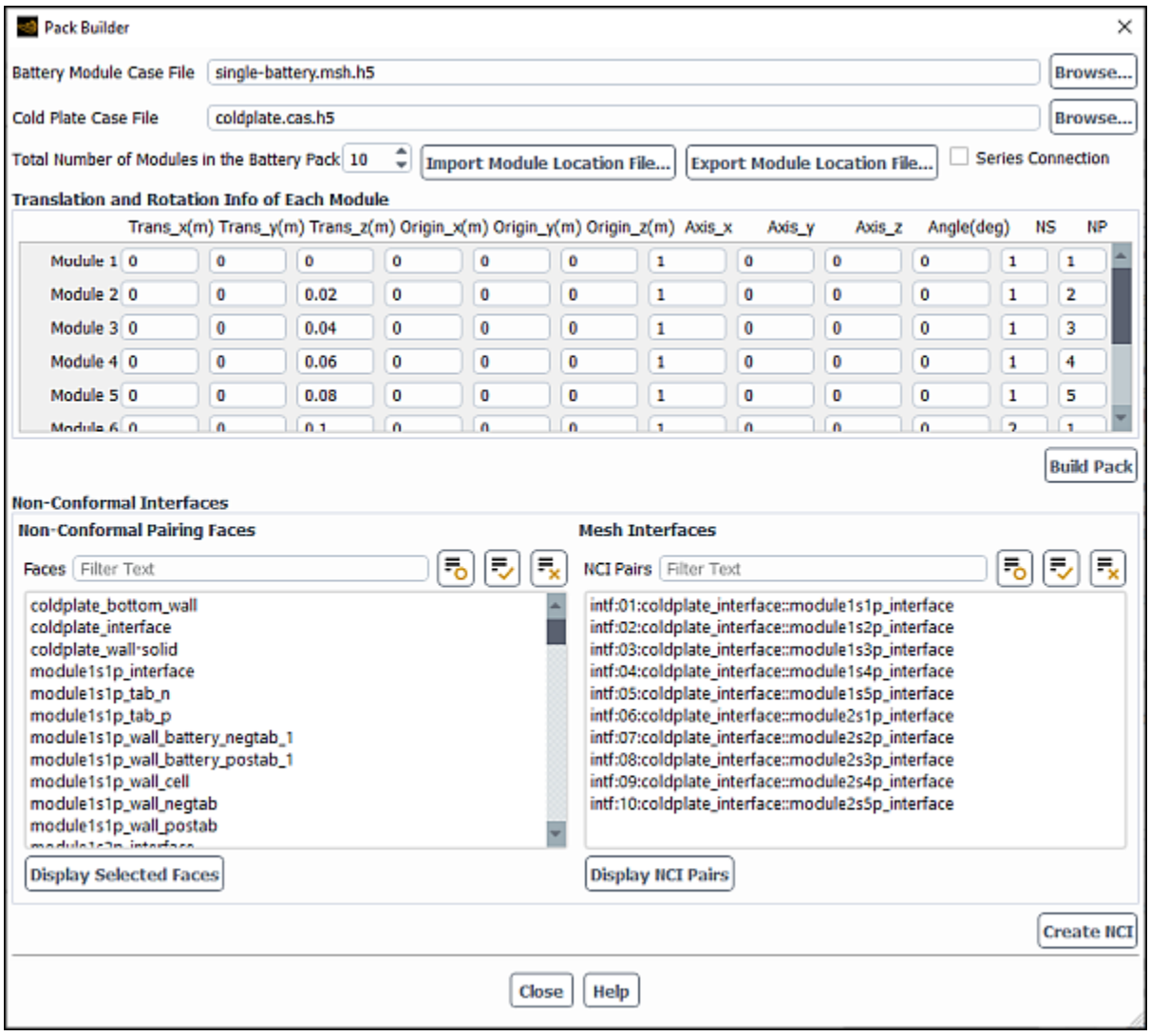
Figure 31.27: Battery Pack with a Cold Plate



To use the Batter Pack Builder tool:

1. In the **Advanced Options** tab of the **Battery Model** dialog box, click the **Battery Pack Builder...** button to open the **Pack Builder** dialog box.

Figure 31.28: The Pack Builder Dialog Box



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. In the **Pack Builder** dialog box, specify the following information:

   - **Battery Module Case File**: The name of the Ansys Fluent case file for the module simulation.

     The battery module problem setup must be complete (that is, the electric load condition, the model selection, and the battery connections are properly defined) and ready for simulation in Ansys Fluent.

   - **Cold Plate Case File**: The name of the Ansys Fluent case file for the cold plate simulation.

     The problem setup must be complete. The plate geometry must be properly oriented. If the data file of the cold plate case is available, the data file will be loaded automatically.

   - Location information for each module.

     This can be done in two different ways:

     – You can manually specify the **Total Number of Modules in the Battery Pack** and enter the location and connection information for each module in the **Translation and Rotation Info of Each Module** group box.

       Each module position and orientation is defined with respect to the original module geometry location by the translational and rotational movements. For the module translational movement, you need to specify the translation distances in the X, Y, and Z directions (**Tran_x**, **Tran_y**, **Tran_z**). For the module rotation, you need to specify the rotational origin (**Orgin_x**, **Origin_y**, **Origin_z**), rotational axis (**Axix_x**, **Axis_y**, **Axis_z**), and the and rotational angle (**Angle**). Each module is placed in a new location, first by a rotation movement and then by a translation movement.

       For the general module connection in a pack (the **Series Connection** option is disable), you also need to specify the connection pattern (**NS** and **NP**).

     – You can import a file in a text format with location information for each module by clicking the **Import Module Location File...** button and then using the **Select File** dialog box.

       An example of the location file is shown below:

```
(
(0  0     0  0  0  0  1  0  0  0  1  1)
(0  0  0.02  0  0  0  1  0  0  0  2  1)
(0  0  0.04  0  0  0  1  0  0  0  3  1)
...
)
```

       Each line contains 12 numbers that describe the translation distances in the X, Y, and Z directions, the coordinates of the rotation origin, the rotation axis direction, the rotational angle for each battery module, and the connection information of the module in a pack, NS and NP. The numbers can be separated by a space or tab delimiter. An extra outer parenthesis pair is needed for the convenience of file reading.

       In the above example, the three modules are connected in series. If they are connected in parallel, then the last two columns will contain the following data instead of the values shown above:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
... 1  1)
... 1  2)
... 1  3)
```

Once the location file is imported into your case, the relevant fields in the **Transition and Rotation Info of Each Module** group box are automatically populated. You can edit each field as appropriate for your case.

If you want to export the module location information into a separate file, click **Export Module Location File...**.

Series Connection: Select this option if all modules are connected in series. In this case, you do not need to define the module connection data.

3. Click the **Build Battery Pack** button to construct the battery pack.

The tool loads battery modules one by one and places them in the correct location. After all modules are loaded, the tool loads the cold plate. The battery pack builder will overwrite the previously existing geometry and replace it with the combined pack geometry, which will be displayed in the graphics window once the building process is complete.

For the convenience of zone identification in the constructed battery pack, all faces and zones in the model are renamed. For any entity in a specific module, the prefix `module#` (where # is the module number for the series connection and 1s1p, 1s2p, and so on for the general connection) is used, while for the cold plate, the prefix `coldplate` is used.

After the build operation is completed, the pack connection information is printed in the Ansys Fluent console, as in the following example:

```
Number of module series stages = 2; Number of modules in parallel per series stage = 3
Number of battery series stages = 8; Number of batteries in parallel per series stage = 1
```

In this example, the battery pack has the 2S3P module connection pattern, and each module has the 8s1p connection pattern.

Note the following:

- It is your responsibility to align the modules and the cold plate correctly. You must define the location of each module properly, so that the touching faces between the battery module and cold plate are in contact in the same plane.
- During the battery pack building, different modules and a cold-plate are appended to the pack sequentially. Depending on the module case size, this may take a long time to complete. To help you estimate the loading time, some timing information is printed in the console. For example:

```
Done with Module 6. Total elapsed time = 235.6866s, loading time for this module = 65.208097s
Done with cold-plate. Total elapsed time = 341.55449s, loading time of cold-plate = 105.86753s
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Create non-conformal interfaces between the battery modules and cold plate by selecting all the pairing faces from the **Non-Conformal Pairing Faces** multiple-selection list and then clicking the **Create NCI** button.

   Note:

   If no faces are selected in the **Non-Conformal Pairing Faces** multiple-selection list, then all the faces in that list will be used in the search of pairing faces. This process can be expensive for a large list of faces. Therefore, selecting the pairing faces is highly recommended to speed up this process.

   The tool automatically detects the touching faces, pairs them, and generates non-conformal interfaces between them. The pairing process uses the one-to-one interface creation method. All the generated non-conformal interfaces are listed in the **Mesh Interfaces** list.

   You can display the non-conformal pairing faces or interfaces in the graphics window by selecting them from the corresponding multiple-selection lists and clicking **Display Selected Faces** or **Display NCI Pairs**, respectively.

5. Click **Close** to close the **Pack Builder** dialog box.

6. Prior to running the simulation in Ansys Fluent, check the model setup and do necessary adjustments. Make sure the correct solver (steady or unsteady), all appropriate models (for example, energy equation) are used, and the correct electric load condition for the pack is applied.

7. Run the battery pack simulation.

   You can freeze the flow field if it remains unchanged (as in a liquid cooling system) in the pack simulation to speed up the calculation. In this case, you can initialize the battery model only by clicking the **Init** button in the **Battery Model** dialog box to start the simulation.

Note the following limitation that exist with the Battery Pack Builder tool:

- The module case file that is used to build a battery pack cannot be built from the Battery Pack Builder tool.

31.2.3.6.4. Using the Battery ROM Tool Kit

Simulation of a battery thermal management system can be performed accurately using computational fluid dynamics (CFD). However, CFD models can be too large and too slow for a system-level transient thermal analysis. Reduced order models (ROMs), such as LTI-ROM and SVD-ROM, can be used in the system-level analysis with minimum loss of accuracy if certain conditions are satisfied.

In the LTI-ROM [70] (p. 5717), the battery thermal problem is regarded as a system that processes a set of input signals (simply called inputs) and yields a set of output signals (simply called outputs). For a thermal problem, the inputs are the power (or the heat source), and the outputs are the temperature rise at user-specified locations or the average temperature rise across a zone. The benefit of the Linear and Time Invariant (LTI) system is that it is completely characterized by its impulse response, and the output of an arbitrary input function can be readily computed as the convolution of the input function and the system impulse response function.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A thermal system becomes an LTI system under constant flow rate, constant material properties, and linear boundary conditions. The temperature and heat flux boundary conditions are linear, and the radiation boundary condition (if present) is nonlinear. Constant density and constant properties are fairly accurate assumptions for water-cooling battery systems. For air-cooling battery systems, these assumptions can still be valid for the range of temperature variation under consideration.

For such an LTI thermal system, given an arbitrary time history of the battery heat source, the temperature evolution at a point can be obtained solely from the impulse response function at that point (that is, from the history of the temperature increase for a given unit amount of the heat source applied to the battery cell at "zero" time). The impulse response is obtained by performing a CFD calculation. In practice, instead of using an impulse function, a unit step function is applied, and the step response is monitored. The impulse response function is the time derivative of the step response function.

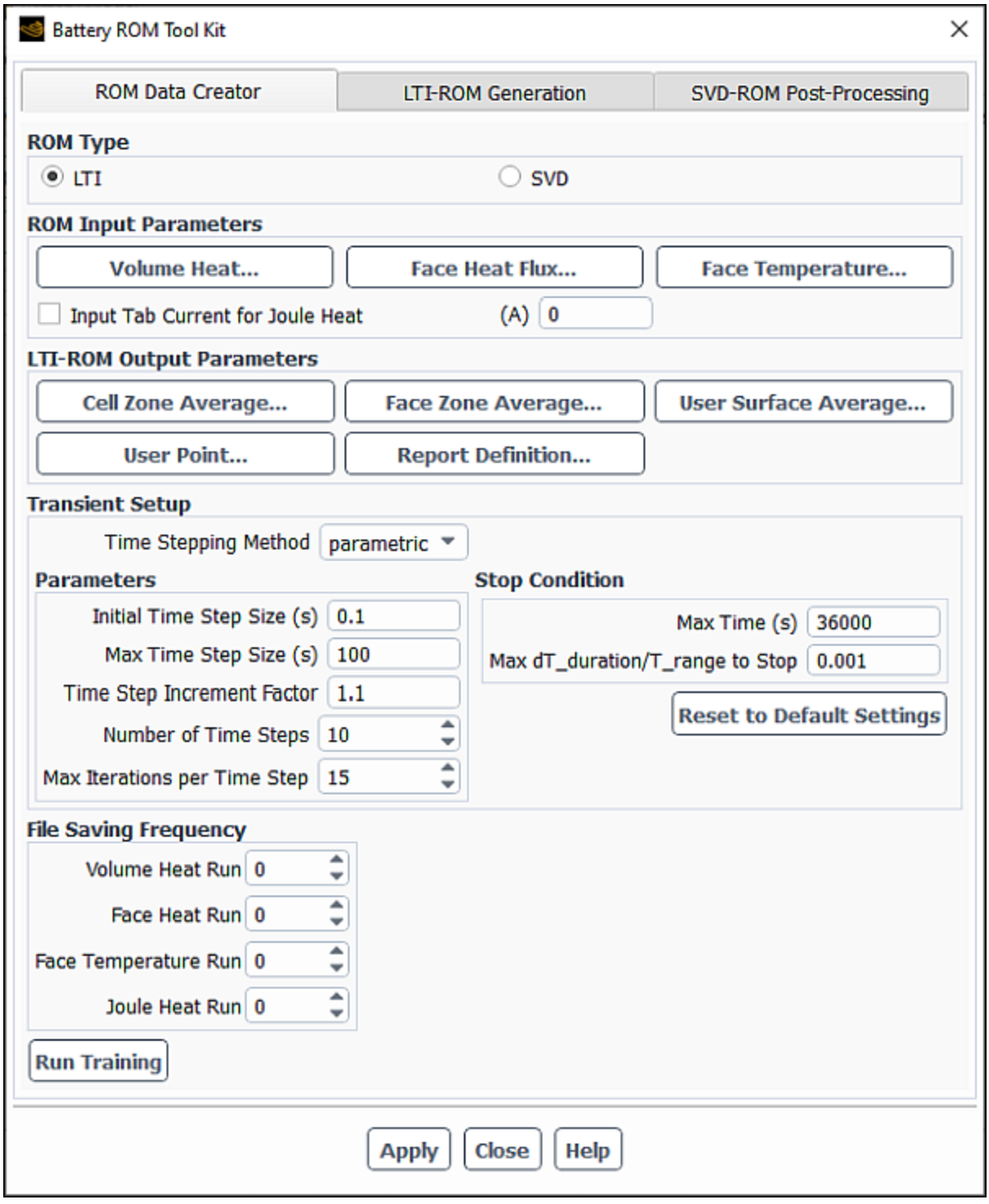
The LTI-ROM can calculate only cell-average temperature or temperature at a user-specified point. To obtain the temperature field, in theory, you can monitor the response function at each control volume and use the LTI-ROM for all the control volumes. However, the computational cost of such an approach would be too high, making it impractical. The dimension reduction method [12] (p. 5713), [68] (p. 5717) can be used for such cases to reduce the simulation time. In this method, Ansys Fluent first runs a CFD analysis to obtain the step response of the temperature distribution. Then, the singular value decomposition (SVD) is used to separate the spatial and temporal contributions of the temperature field. The dynamics of the temperature response is approximated in a largely reduced dimension space, which is formed by keeping only the first several leading singular values and their corresponding orthonormal left singular vectors. The LTI-ROM is then applied in such a reduced dimension space. Temperature at any physical space location can be obtained by a linear combination of the temperature points in the reduced subspace.

Similar to the LTI-ROM, the battery SVD-ROM tool kit in Ansys Fluent creates only the step response functions. The SVD-ROM files generated by Ansys Fluent can be used to run simulations in Ansys Twin Builder. Contact your technical engineer for detailed information and tutorials on using SVD files generated by Ansys Fluent in Ansys Twin Builder.

To use the battery ROM tool kit, click the **Battery ROM Tool Kit…** button in the **Advanced Options** tab of the **Battery Model** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.29: The Battery ROM Tool Kit Dialog Box



In the **Battery ROM Tool Kit** dialog box that opens, you can:

- Create ROM training data using the **ROM Data Creator** tab
- Compute coefficient matrices of the state space equations using the **LTI-ROM Generation** tab.
- Do postprocessing of data obtained from the SVD-ROM tool using the **SVD-ROM Post-Processing** tab

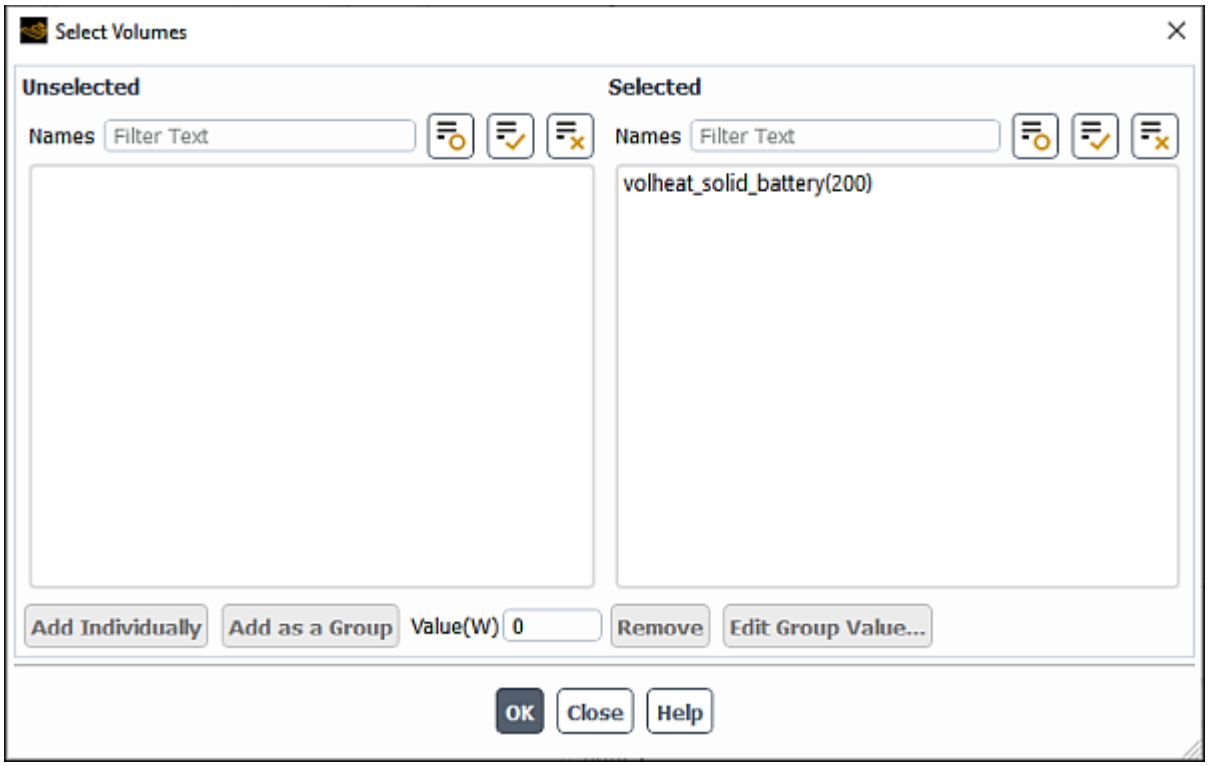
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.2.3.6.4.1. Create ROM Training Data (ROM Data Creator Tab)

In the **ROM Data Creator** tab (see Figure 31.29: The Battery ROM Tool Kit Dialog Box (p. 3367)), you can create ROM training data as follows:

1. In the **ROM Type** group box, select either **LTI** or **SVD** (**ROM Data Creator** tab).
2. Wherever applicable, specify the possible heating sources using the controls in the **ROM Input Parameters** group box.
   a. Define the volumetric heat in zones in the **Select Volumes** dialog box that opens by clicking the **Volume Heat...** button.

Figure 31.30: The Select Volumes Dialog Box



   i. Enter a heat source value in the **Value** text entry box.
   ii. In the **Unselected** multiple-selection list, select the volume zone(s) where you want to define a volumetric source.
   iii. To define the selected zones as a single group parameter, click **Add as a Group**; to define the selected zones as individual parameters, click **Add Individually**.

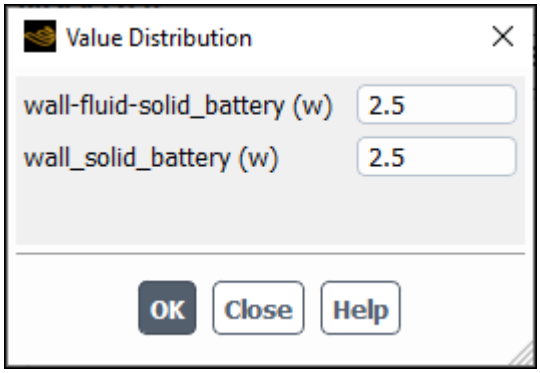
   The selected zones will appear in the **Selected** multiple-selection list. The heat source value you specified will be assigned to each individual parameter. For a single group parameter, the heat source value will be equally distributed among all the volume entities in the group.

   iv. To remove the parameter from the **Selected** multiple-selection list, select it and click **Remove**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

v. (group parameters only) If you want to change the heat distribution among the volume entities in the group, select that group in the **Selected** multiple-selection list and click **Edit Group Value...**. In the **Value Distribution** dialog box that opens, specify a source value for each zone.

Figure 31.31: The Value Distribution Dialog Box



b. In a similar manner, if necessary, define the following ROM input parameters:

- Heat fluxes through a surface—in the **Select Faces** dialog box that opens by clicking the **Face Heat Flux...** button
- Temperature at a surface—in the **Select Faces** dialog box that opens by clicking the **Face Temperature...** button

3. If you have selected **Enable Joule Heat in Passive Zones** in the **Model Options** tab, you can enable **Input Tab Current for Joule Heat** if you want to solve the potential equation and specify the boundary condition for the current in the text entry box.

4. (LTI-ROM only) Using the dialog boxes that can be accessed via the **LTI-ROM Output Parameters** group box, select the geometric entities of a mesh (that is, volumes, surfaces, lines, and points) and/or report definitions for which you want to obtain the following quantities after completing the training run:
   - average temperatures (for volumes, surfaces, and lines)
   - temperature at a point (for points)
   - temperature response for a specific variable (for report definitions)

   The controls in the LTI-ROM output dialog boxes are similar to those in the ROM Input dialog boxes described above.

   a. To select cell zones, click **Cell Zone Average...**.

      i. In the **Select Cell Zones** dialog box, select the cell zones in the same way it is done when defining **ROM Input Parameters** (as described above).

      ii. (group parameters only) To view the cell zones included in the group parameter, select it in the **Selected** multiple-selection list and click **Show Group Entities...**. All the cell zones will be listed in the **List Entities** dialog box that opens.

   b. In a similar manner, you can select the following output entities:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Face zones—in the **Select Face Zones** dialog box that opens by clicking the **Face Zone Average...** button
- User-created surfaces—in the **Select Surfaces** dialog box that opens by clicking the **User Surface Average...** button
- User-created points—in the **Select Point Surfaces** dialog box that opens by clicking the **User Point...** button
- Report definitions—in the **Select Report-Definitions** dialog box that opens by clicking the **Report Definition...** button

5. (SVD-ROM only) Select the zones for which you want to export temperature information in the **Select Exported Temperature Zones** dialog box that opens by clicking the **Exported Temperature Zone...** button (**SVD-ROM Output Parameters** group box).

6. In the **Transient Setup** group box, specify the parameters and the stop condition to control the time stepping during the training run:

   a. Select either **parametric** (default) or **file** as **Time Stepping Method**.

   b. If you have selected the **parametric** method, specify the following solution parameters and stop conditions.

      In the **Parameters** group box, specify:

      Initial Time Step Size

      is the initial time step size in the calculation.

      Max Time Step Size

      is the maximum time step size.

      Time Step Increment Factor

      is the factor by which the time step is increased during the solution.

      Number of Time Steps

      is the number of time steps that use the same time step size.

      Max Iterations per Time-Step

      is the maximum number of iterations per time step.

      In the **Stop Condition** group box, specify:

      Max Time

      is the maximum time of the transient simulation.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Max dT_duration/T_range to Stop

is a ratio of the temperature change over the specified time duration against the temperature range. This ratio is checked for all the monitored temperatures. If they all fall bellow the specified threshold defined by this parameter, a training run stops even before the maximum time is reached.

You can use the **Reset to Default Settings** button to reset the parameters and stop condition to default values.

c. If you selected **file** as the **Time Stepping Method**, then you need to specify a filename with time stepping information or use the **Browse...** button to select it.

The file must contain the following three columns:

- time step
- number of time steps
- maximum iterations per time step

Each line specifies a time marching event. The varying time step is controlled by different time marching events in the file.

An example of a file with time-stepping information is shown below:

```
1      5     10
1.2    5     10
3      5     10
...
10     5     10
```

7. If you want to save the simulation files during the training run (for example, for debugging purposes), you can use the below controls (in the **File Saving Frequency** group box) to specify the frequency with which the files should be saved for each type of input parameters.
   - **Volume Heat Run**
   - **Face Heat Run**
   - **Face Temperature Run**
   - **Joule Heat Run** (if the **Input Tab Current for Joule Heat** is enabled)

   For example, if you have defined four volume heat input parameters and set the **Volume Heat Run** file saving frequency to be 1, all four simulation files will be saved after the training run. If you set **Volume Heat Run** to be 2, only two simulation files will be saved. By defaults, the simulation files are not saved.

8. Click **Run Training**.

   The battery tool kit runs all the transient Ansys Fluent simulations sequentially for each defined ROM input parameter. The number of transient training runs equals to the total number of the ROM input parameters.

   Every training simulation starts with the initial time step and runs the specified number of time steps. Then the time step is increased by the time step increment factor, and the solution runs the specified number of time steps again. This process continues until the total time reaches the specified **Max Time** or the stop condition is met.

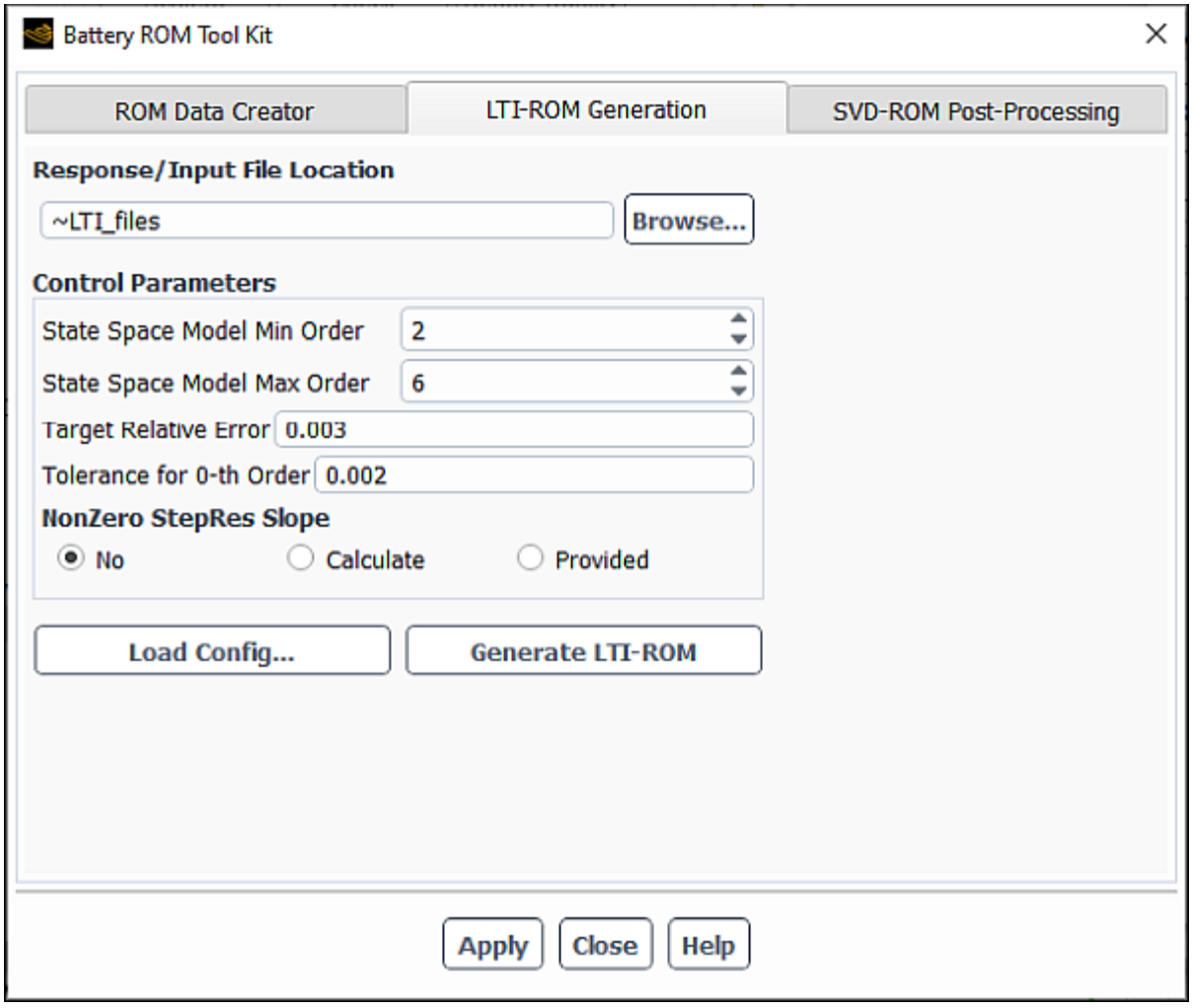
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

After completing all solution runs, the results are saved in your work directory in a folder called `LTI` for the LTI-ROM or SVD for the SVD-ROM. You can then import these files into Ansys Twin Builder to run the ROM simulation. Contact your technical engineer for detailed information and tutorials on using LTI and SVD files generated by Ansys Fluent in Ansys Twin Builder. For the LTI-ROM, you can use files generated directly in the LTI-ROM Generation tool.

31.2.3.6.4.2. Computing Coefficient Matrices of the State Space Equations (LTI-ROM Generation Tab)

In a system simulation, an LTI-ROM model is usually solved by state space equations. Ansys Fluent provide a tool to calculate the coefficient matrices of the state space equations.

Figure 31.32: The Battery ROM Tool Kit Dialog Box - LTI-ROM Generation Tab



You can use the **LTI-ROM Generation** tab to perform the calculation as follows:

1. In the **Response/Input File Location** text entry box, specify the directory path where the response files are stored either manually or by using the **Browse...** button.

2. In the **Control Parameters** group box, specify the following parameters:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

State Space Model Min Order

is the minimum order of the state space model for each element. A higher order yields more accurate results, but the computation time will also increase. Note that if zero is entered, then a contribution of cross heating from other volumes may still be ignored depending on the value you specify for **Tolerance for 0th Order**.

State Space Model Max Order

is the maximum order of the state space model for each element.

Target Relative Error

is the relative error that determines the order of the state space model fitted. In general, a smaller error creates a higher order state space model.

Tolerance for 0-th Order

allows you to ignore contributions from some inputs (for example, negligible contributions from certain cross-heating) if they are less than the specified tolerance multiplied by the largest contribution. This can significantly reduce the size of the overall model.

NonZero StepRes Slop

is an indicator of how the end slope of the response curves is handled. You can select from the following options:

- **No**
- **Calculate**
- **Provided**

For instance, in a case with all adiabatic boundary conditions, the step response will never reach a steady state condition, and the curve will have a slope. You can let the solver calculate it by selecting **Calculate** or provide a specific value by selecting **Provided**.

3. Click **Generate LTI-ROM**.

The LTI-ROM tool will use all the necessary files (including `Input_PortNames.txt`, `Input_StepInputValues.txt`, and all the response files) from the folder specified in the **Response/Input File Location** field, run the fitting process, and generate the matrices of the state space equation. The resulting matrices will be saved in the `Thermal_ROM_Matlab.m` file in the same location. This file can be used in any system simulation software that solves a state space model. The GUI settings you specified will be automatically saved in a file called `Input_ROMconfig.txt`, which can be read back in a later session using the **Load Config...** button.

Contact your technical engineer for detailed information and tutorials for the LTI-ROM generator tool.

31.2.3.6.4.3. Postprocessing of the SVD-ROM Data (SVD-ROM Post-Processing Tab)

You can use the **SVD-ROM Post-Processing** tab to visualize the 3D temperature field from an SVD-ROM simulation in Ansys Twin Builder:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the **SVD-ROM Post-Processing** tab of the **Battery ROM Took Kit** dialog box, enter the name of the solution file in the **One of Basis File** text entry field or use the **Browse...** button to select it.

2. Enter the name of the **Transient Coefficient File** or use the **Browse...** button to select it.

3. You can display the reconstructed temperature field for each snapshot time and create animation from the ROM results. Contact your technical engineer for detailed information and tutorials about these functionalities.

31.2.3.6.5. Defining Orthotropic Thermal Conductivity

A battery is made of a rolled or stacked sandwich structure of layered electrodes. Usually, the material properties in the in-plane direction of the electrode are different from those in the cross-plane direction. For example, the thermal conductivity in the in-plain direction is typically much larger than that in the cross-plane direction.

In battery applications, a battery module or pack may contain many batteries. In Ansys Fluent, you can define the orthotropic material for each battery in two different ways:

- Define the orthotropic thermal conductivity in the **Create/Edit Materials** dialog box. For example, you can define the orthotropic thermal conductivity of the battery material in cylindrical coordinates as described in Cylindrical Orthotropic Thermal Conductivity (p. 1652). In this case, you need to create a separate solid material for each battery where each material has a unique set of the origin and cylindrical axis. This is a tedious and time-consuming process, which can lead to input errors.

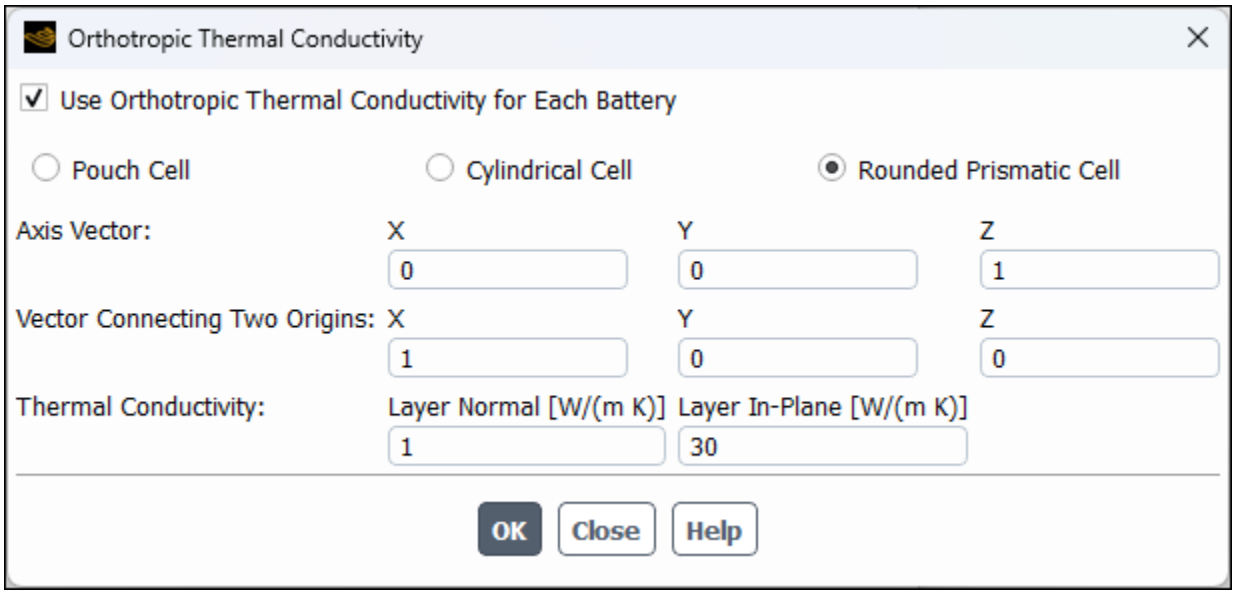
- Use the Ansys Fluent built-in tool that provides a fast and convenient method for defining orthotropic thermal conductivity property for each battery active zone globally.

To use the Ansys Fluent built-in tool for defining orthotropic thermal conductivity for all battery active zones:

1. In the **Battery Model** dialog box, click the **Orthotropic Material...** button (**Advanced Options** tab) to open the **Orthotropic Thermal Conductivity** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.33: The Orthotropic Thermal Conductivity Dialog Box



2. In the **Orthotropic Thermal Conductivity** dialog box, enable **Use Orthotropic Thermal Conductivity for Each Battery**.

   The dialog box will expand to reveal the related settings.

3. Choose your battery cell type from the following options (see Figure 31.22: Electrode Layer Orientation (p. 3348)):

   - **Pouch Cell**
   - **Cylindrical Cell**
   - **Rounded Prismatic Cell**

4. Specify the cell orientation information:

   - For the **Pouch Cell** battery type, you need to specify the **X**, **Y**, **Z** components of the **Layer Normal Direction**.
   - For the **Cylindrical Cell** battery type, you need to specify the **X**, **Y**, **Z** components of the **Axis Vector**.
   - For the **Rounded Prismatic Cell** battery type, you need to specify the **X**, **Y**, **Z** components of two vectors - the **Axis Vector** and the **Vector Connecting Two Origins** of the prismatic cell.

5. Specify the **Layer Normal** and **Layer In-Plane** components for **Thermal Conductivity**.

6. Click **OK** to close the **Orthotropic Thermal Conductivity** dialog box.

   The tool will determine the origin of each battery active zone and apply the orthotropic material properties to all the active zones.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you use the Battery Pack Builder tool, then material property settings specified in this way will be carried over to each battery module. That is, if a battery pack is constructed from a module that uses the orthotropic thermal conductivity, then the material properties specified in the **Orthotropic Thermal Conductivity** dialog box will be automatically applied to all batteries in all other modules. The cylindrical origin and axis of batteries in other modules will be updated automatically according to the module location and its orientation.

Note:

- The orientation detection method requires each battery to have a single solid active zone. If this requirement is not met, the detected orientation may be incorrect.
- If multiple batteries are used in a simulation, the orientation of all the batteries should be the same.

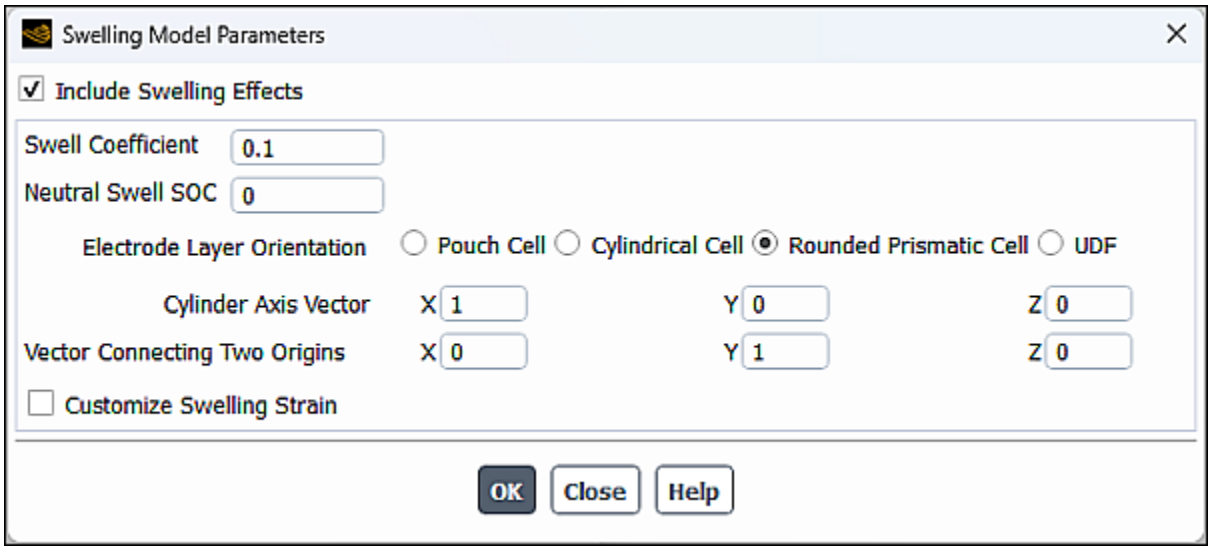
31.2.3.6.6. Using the Empirical-Based Battery Swelling Model

For the NTGK and the ECM model, the empirical-based battery model can be used in an FSI coupled simulation (see Empirical-Based Swelling Model in the *Fluent Theory Guide*).

To include the swelling effect, perform the following steps:

1. Make sure that either **NTGK/DCIR Model** or **Equivalent Circuit Model** is selected in the **E-Chemistry Models** group box of the **Battery Model** dialog box (**Model Options** tab).
2. In the **Advanced Options** tab, click **Battery Swelling Model...**.

   Figure 31.34: The Swelling Model Parameters Dialog Box



3. In the **Swelling Model Parameters** dialog box that opens, enable **Include Swelling Effects**.
4. Specify the following settings:
   - **Swell Coefficient**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Neutral Swell SOC**
- **Electrode Layer Orientation**

These controls are similar to those for the Newman's P2D model. See Inputs for the Newman's P2D Model (p. 3342) for more information about these settings.

5. If you want to use your own battery swelling model, enable **Customize Swelling Strain** and select the name of your compiled `DEFINE_BATTERY_SWELL_STRAIN` user-defined function (UDF) in the **Swelling Strain UDF** drop-down list. See `DEFINE_BATTERY_SWELL_STRAIN` in the *Fluent Customization Manual* for details.

Note:

- The orientation detection method requires each battery to have a single solid active zone. If this requirement is not met, the detected orientation may be incorrect.
- If multiple batteries are used in a simulation, the orientation of all the batteries should be the same.

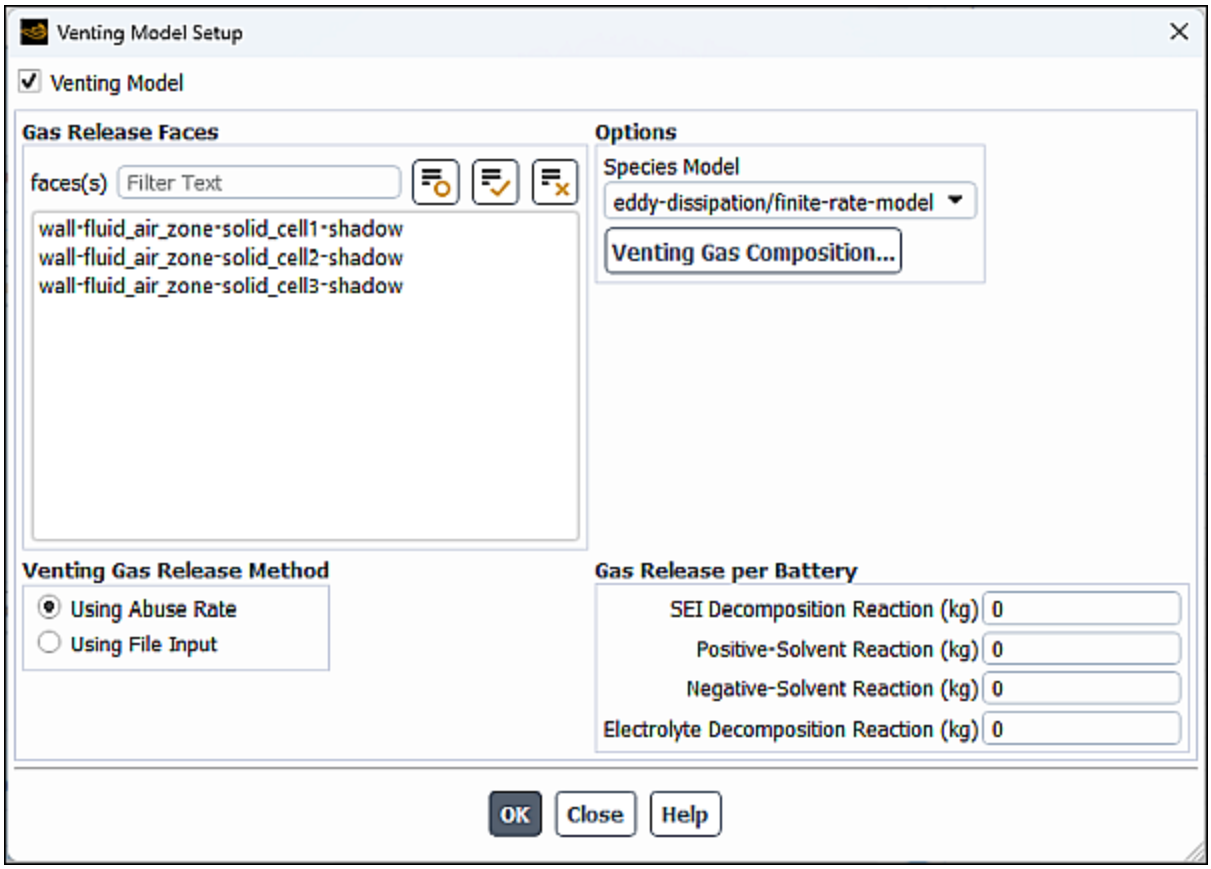
31.2.3.6.7. Battery Venting Model

To use the Ansys Fluent battery venting model described in Battery Venting Model in the *Fluent Theory Guide* in your simulation, perform the following steps:

1. In the **Advanced Options** tab, click **Battery Venting Model...**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.35: The Venting Model Setup Dialog Box



2. In the **Venting Model Setup** dialog box that opens, enable **Venting Model**.

3. From the **Gas Release Faces** selection list, select the face zones from which venting gas is released. Only faces between a battery active zone and a fluid zone are listed as potential gas release faces.

4. In the **Options** group box, you can specify the venting model options for your simulation.

   - For cases that do not involve the multiphase model, you can select one of the following from the **Species Model** drop-down list:

     – **species-transport-w/o-reaction**

     – **eddy-dissipation-model**

     – **eddy-dissipation/finite-rate model**

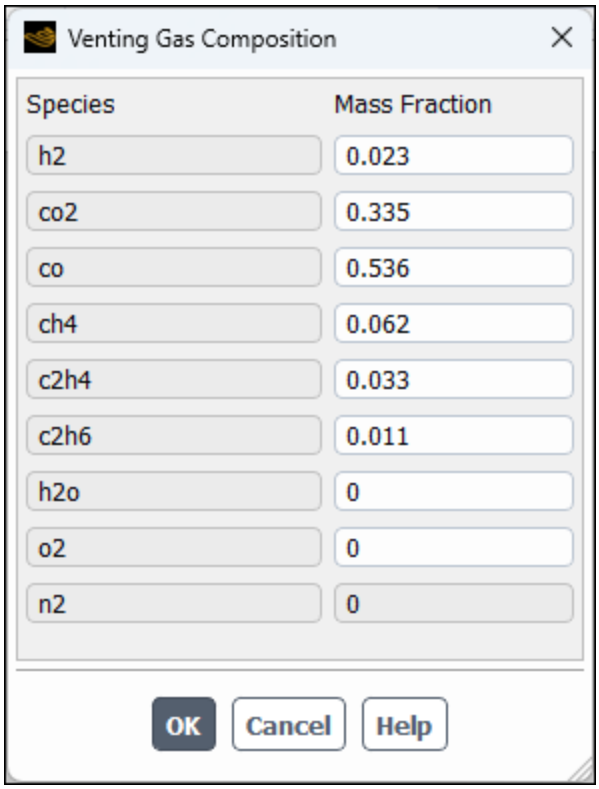
     If the post combustion is not considered, you can choose the **species-transport-w/o-reaction** option. If combustion of the venting gas is important, you can choose either **eddy-dissipation-model** or **eddy-dissipation/finite-rate model**. As for the difference between these two models, refer to Volumetric Reactions in the *Fluent Theory Guide* for details.

   - For cases that involve the multiphase model, you can use the **Phase of Venting Gas** drop-down list to select the phase into which the venting gas is released in the multiphase simulation.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. If you want to modify the venting gas composition, click **Venting Gas Composition...** and in the **Venting Gas Composition** dialog box that opens, adjust the mass fraction of each species in the venting gas mixture.

Figure 31.36: The Venting Gas Composition Dialog Box



Note:

To change the venting gas mixture itself, refer to Material Types and Databases (p. 1598) for details.

6. Select the **Venting Gas Release Method** from the following options:

   - **Using Abuse Rate**
   - **Using File Input**

   If you have selected **Using Abuse Rate**, then you need to specify the following parameters in the **Gas Release per Battery** group box:

   - For the **One-Equation Kinetics Model** thermal abuse model:
     - **Released Gas Mass**: is m_0 in Equation 19.42 in the *Fluent Theory Guide*
   - For the **Four-Equation Kinetics Model** thermal abuse model:
     - **SEI Decomposition Reaction**: is $m_{sei,0}$ in Equation 19.43 in the *Fluent Theory Guide*
     - **Positive-Solvent Reaction**: is $m_{pe,0}$ in Equation 19.43 in the *Fluent Theory Guide*

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– **Negative-Solvent Reaction**: is $m_{ne,0}$ in Equation 19.43 in the *Fluent Theory Guide*

– **Electrolyte Decomposition Reaction**: is $m_{e,0}$ in Equation 19.43 in the *Fluent Theory Guide*

If you have selected **Using File Input**, then you need to specify the following:

- **Activation Temperature**: the temperature level at which the gas venting is activated
- The name of the file with the venting gas release rate information in the **Specify File** text entry field. Alternatively, the file can be provided using the **Select File** dialog box, which is opened by clicking the **Browse...** button. The file should contain three columns of data separated by spaces or tabs:

  – time in second

  – venting gas temperature in Kelvin

  – venting gas release rate in kg/s

  An example of an input file is shown below:

```
0       400       1e-3
10      400       1e-3
10.1    450       2e-3
100     450       2e-3
```

Once the solution is completed, in addition to the battery variables listed in Postprocessing the MSMD Battery Model (p. 3396), the following variable becomes available for postprocessing of simulations that involve the venting model (in the **Battery Variables...** category):

- **Venting Gas Mass Source**

This variable can be used to display where the venting gas is added in the domain. The volume integral of **Venting Gas Mass Source** gives the total venting gas release rate in kg/s.

Note:

The venting valve model is not implemented in the current release. For information on how to model the venting valve in a simulation, contact your support engineer at Ansys Fluent for guidance.

31.2.3.6.8. Including the Entropic Heat Effects

When any of the electrochemical submodels is used in the calculation, you can include entropic heat in the heat source calculation (Equation 19.8, Equation 19.13, and Equation 19.30 in the *Fluent Theory Guide*). To do so, follow the steps below.

1. Enable **Include Entropic Heat**.

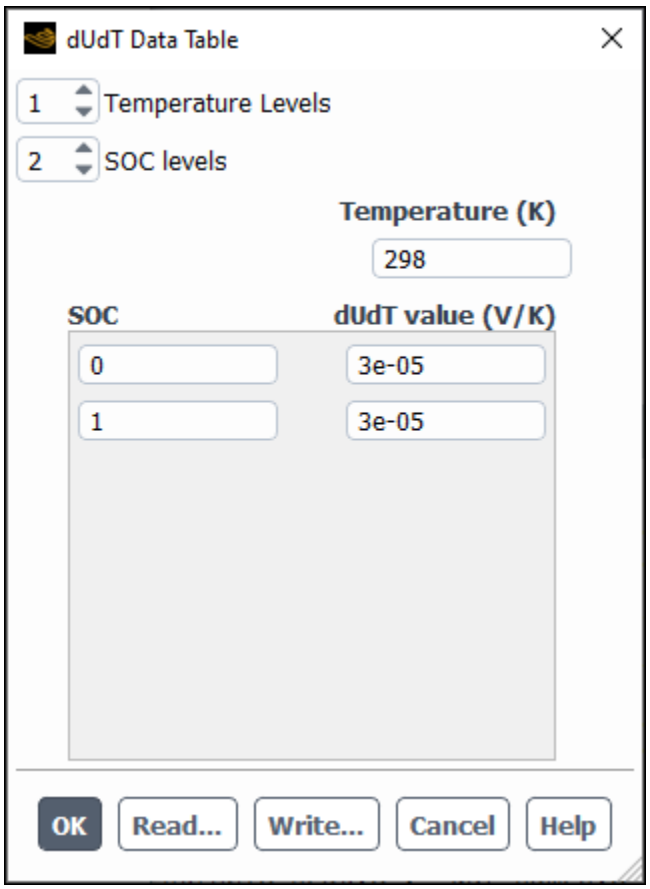
2. In the **Data Type** group box, select the specification method for the dU/dT term in the entropic heat equations (Equation 19.8, Equation 19.13, and Equation 19.30 in the *Fluent Theory Guide*) from the following options:
   - **Data Table**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **UDF**

3. If you have selected **Data Table**, click **dUdT Table...** to define a lookup table for dU/dT at different temperature and SOC levels in the **dUdT Data Table** dialog box that opens.

Figure 31.37: The dUdT Data Table Dialog Box



The **dUdT Data Table** dialog box is similar to the **Battery Aging Table** dialog box in operation (see Setting the Life Model (p. 3308)).

You can specify the following inputs:

- **Temperature Levels**
- **Soc Levels**
- Values of **Temperature**
- Values of **SOC**
- dUdT values for each SOC-temperature pair

If you want to specify different tables for discharging and charging, do the following:

a. Enable **Different Table for Charge/Discharge**.

b. Click **dUdT Table for Discharge...** and specify data for battery discharging in the **dUdT Data Table** dialog box that opens.

c. Similarly, click **dUdT Table for Charge...** and specify data for battery charging.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. If you have selected **UDF**, you need to compile a `DEFINE_BATTERY_ENTROPIC_HEAT` user-defined function and hook it using the **Entropic Heat UDF** dropdown list. See `DEFINE_BATTERY_ENTROPIC_HEAT` in the *Fluent Customization Manual* for details.

31.2.3.6.9. Using the Thermal Abuse Model

To model the thermal abuse, select the **Thermal Abuse Model** option. The **Battery Model** dialog box expands to show additional model parameters (see Figure 31.25: The Battery Model Dialog Box (Advanced Options Tab) (p. 3351)).

You can select the following options to model thermal runaway reactions:

- **One-Equation Kinetics Model**
- **Four-Equation Kinetics Model**

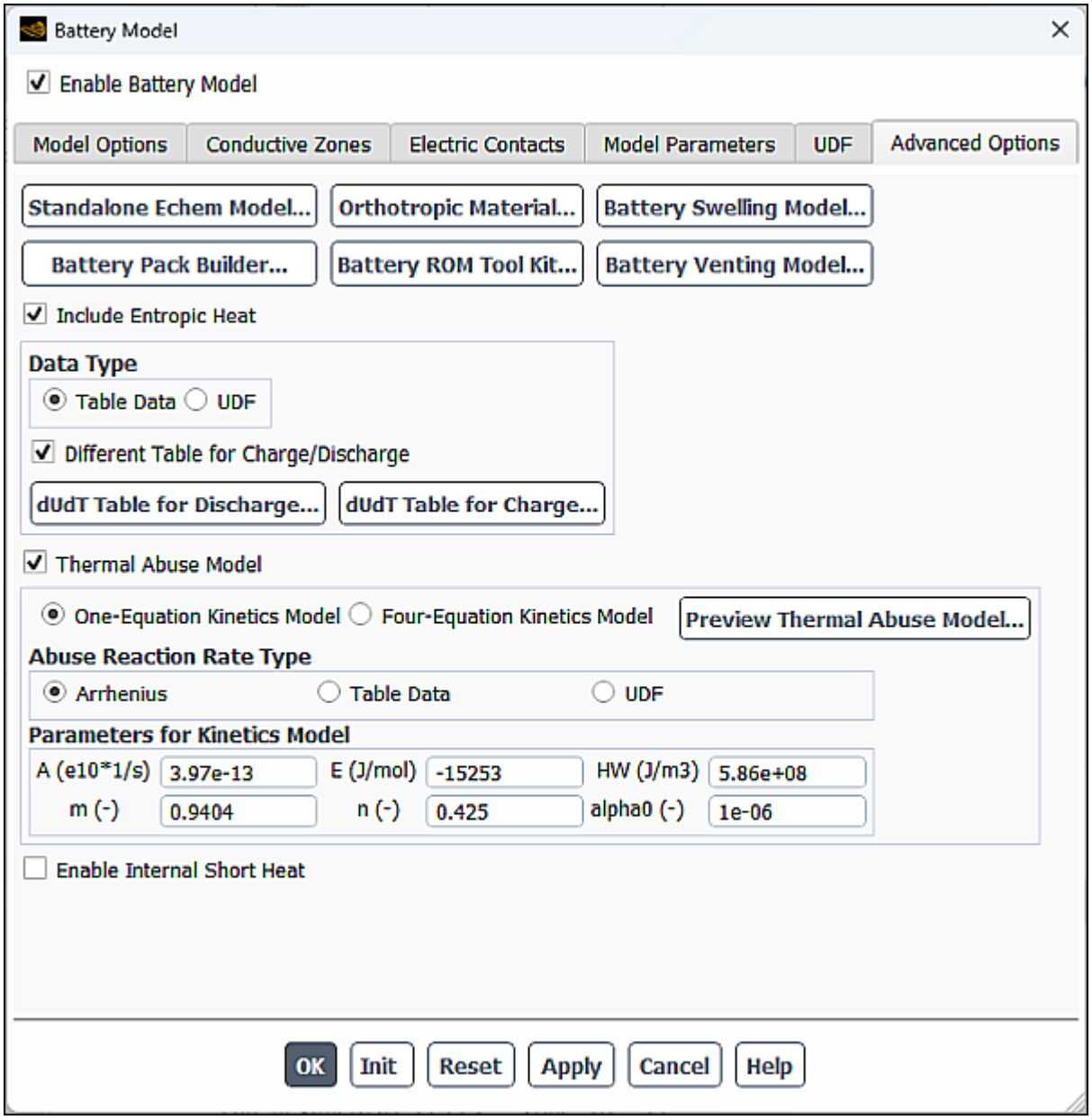
Further details on these models are provided below.

For theoretical information about the thermal abuse model, see Thermal Abuse Model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.2.3.6.9.1. One-Equation Kinetics Model

Figure 31.38: The Battery Model Dialog Box - One-Equation Kinetics Model



For the **One-Equation Kinetics Model**, you can select the **Abuse Reaction Rate Type** from the following options:

- **Arrhenius**
- **Table Data**
- **UDF**

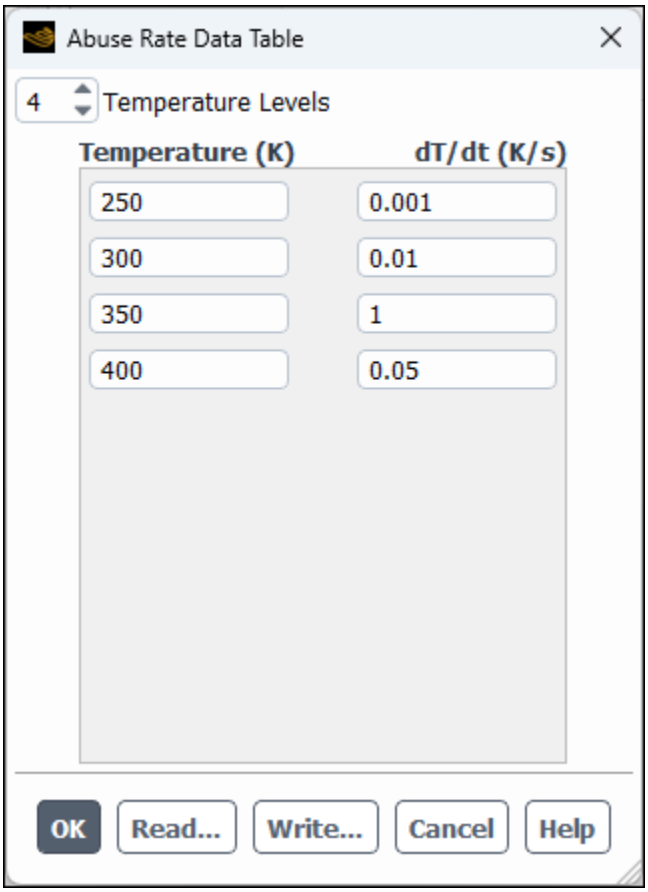
If you have selected **Arrhenius**, you need to specify the battery constants A, E, HW, m, and n used in Equation 19.33 through Equation 19.34, as well as the initial value for α (**Parameters for Kinetics Model** group box).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you have selected **Table Data**, you need to specify:

- **Initial Reaction Progress Variable**: Is the initial value of α in Equation 19.33 and Equation 19.34.
- **Specific Heat Release, HW**: Is HW in Equation 19.34.
- Data for the abuse reaction rate table where the temperature increase rate is defined as a function of the temperature. You need to enter the following inputs in the **Abuse Reaction Rate Table** dialog box that opens by clicking the **Abuse Reaction Rate Table...** button:

  Figure 31.39: The Abuse Rate Data Table Dialog Box



  – **Temperature Levels**

  – **Temperature**

  – **Rate dT/dt**

  The controls in the **Abuse Reaction Rate Table** dialog box are similar to those in the **Battery Aging Table** dialog box described in Setting the Life Model (p. 3308).

If you have selected **UDF**, you need to specify **Initial Reaction Progress Variable** and **Specific Heat Release, HW**, and select the name of your compiled `DEFINE_BATTERY_ABUSE_RATE` UDF from the **Abuse Reaction Rate UDF** drop-down box. The UDF will be used to compute

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$\frac{d\alpha}{dt}$ in Equation 19.33 and Equation 19.34. See `DEFINE_BATTERY_ABUSE_RATE` in the *Fluent Customization Manual* for details.

31.2.3.6.9.2. Four-Equation Kinetics Model

When you select **Four-Equation Kinetics Model**, you can specify the battery constants A, E, H, W, and m used in Equation 19.35) through Equation 19.39, as well as the initial values for c_{sei}, c_{ne}, α, and c_e.

You can include the internal short circuit reaction (Equation 19.40 in the *Fluent Theory Guide*) to consider the possible initiation of an internal short circuit and its propagation in a battery by selecting the **Enable Internal Short Heat** option. Once this option is enabled, you need to specify the following model constants for the reaction in the **Internal Short Heat Reaction** group box (see Figure 31.25: The Battery Model Dialog Box (Advanced Options Tab) (p. 3351)):

A_ec

is a pre-exponential factor, A_{ec} in Equation 19.40 in the *Fluent Theory Guide*.

E_ec

is the activation energy of the reaction, E_{ec} in Equation 19.40 in the *Fluent Theory Guide*.

H_ec

is the heat of the short-circuit electrochemical reaction, H_{ec} in Equation 19.39 in the *Fluent Theory Guide*.

T_trigger

is the user-specified trigger temperature, $T_{trigger}$ in Equation 19.41 in the *Fluent Theory Guide*.

Soc

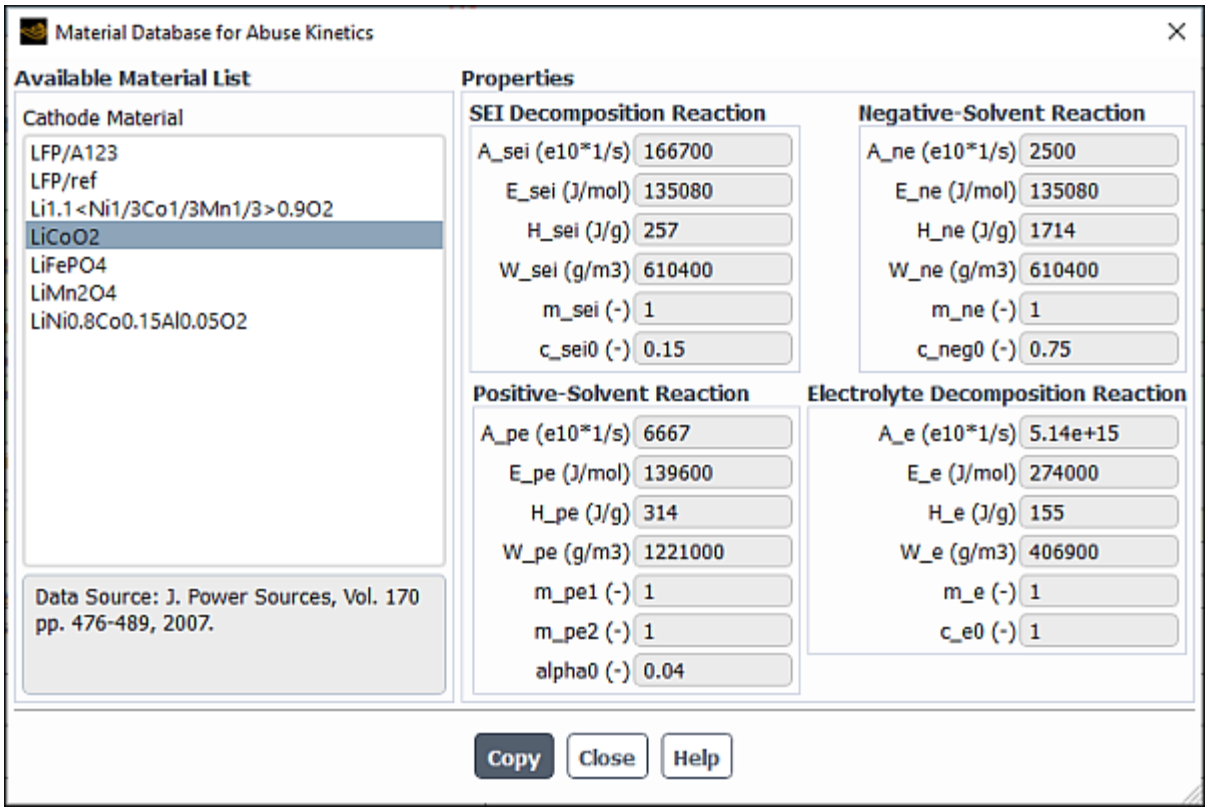
is the progress variable for the internal short reaction, Soc in Equation 19.40 in the *Fluent Theory Guide*.

If you are using the Four-Equation Kinetics model, you can import kinetics data of the abuse reactions from the Ansys Fluent database using the below steps:

1. Click the **Material Database** button next to **Material Name**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.40: The Material Database for Abuse Kinetics Dialog Box



2. In the **Material Database for Abuse Kinetics** dialog box, select the appropriate cathode material from the **Available Material List**.

   The reaction properties are displayed in the **Properties** group box.

3. Import kinetics data by clicking **Copy**.

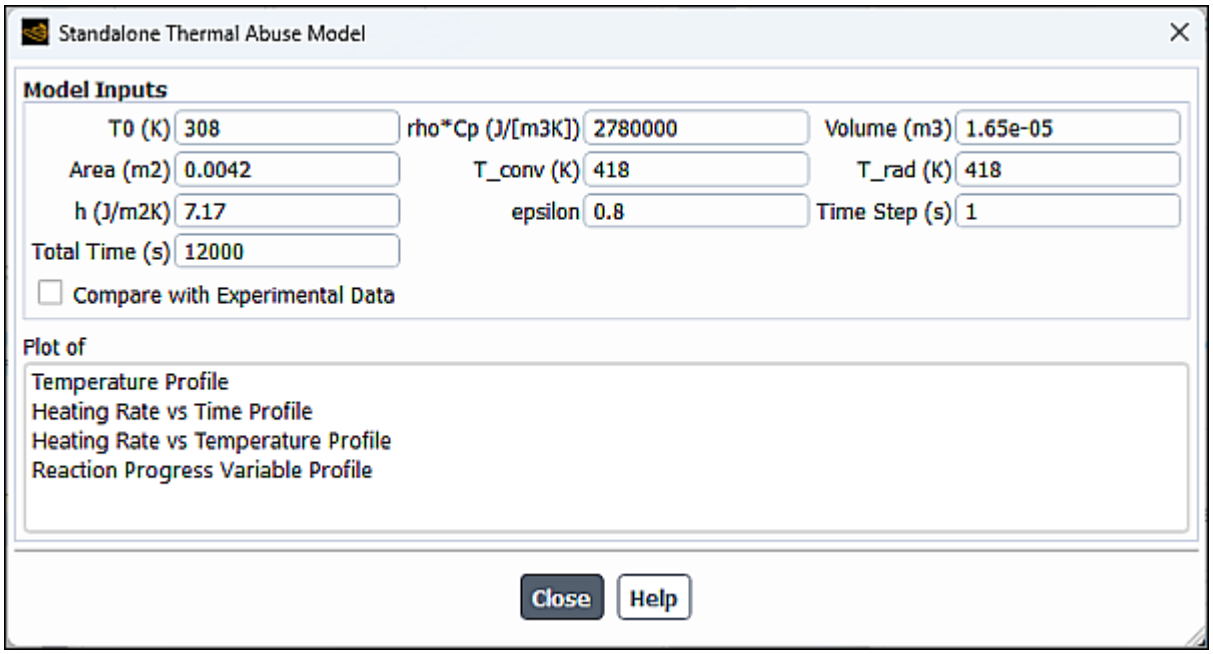
   The name of the copied material is displayed in the **Material Name** text box, and the kinetics data is automatically copied to the correspondent reaction group boxes in the **Advanced Options** tab of the **Battery Model** dialog box.

31.2.3.6.9.3. Standalone Thermal Abuse Model

Before running a CFD simulation, you can run the standalone thermal abuse model to compute the abuse reaction behavior and display the results in the graphics windows. This tool can be used to quickly assess the abuse model you are using in your simulation. To use this tool, click the **Preview Thermal Abuse Model...** button, and in the **Standalone Thermal Abuse Model** dialog box that opens, specify the following parameters in the **Model Inputs** group box:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 31.41: The Standalone Thermal Abuse Model Dialog Box



- **T0**: Is the initial temperature
- **rho*Cp**: Is the volumetric heat capacity
- **Volume**: Is the battery volume
- **Area**: Is the battery external area
- **T_conv**: Is the environment temperature for convective heat transfer
- **T_rad**: Is the environment temperature for radiation heat transfer
- **h**: Is the heat transfer coefficient between battery and its surrounding environment
- **epsilon**: Is the battery surface emissivity
- **Time Step**: Specifies the time step size used in the calculation
- **Total Time**: Specifies the total calculation time

If you want to compare the temperature prediction with experimental data, enable the **Compare with Experimental Data** option and specify the name of the ARC experimental data file in the **File Name** text box. Alternatively, the file can be selected using the **Select File** dialog box, which is opened by clicking the **Browse...** button.

Once you entered the model inputs, you can click one of the plots (**Temperature Profile**, **Heating Rate vs Time Profile**, **Heating Rate vs Temperature Profile**, or **Reaction Progress Variable Profile**) in the **Plots of** group box to display the simulation results in the graphics window. Note that the standalone solver automatically performs computations in the background once you click one of the available plots. After reviewing the results, you can adjust the rate parameter if needed.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.2.3.6.9.4. Running the Thermal Abuse Model Only

For both **One-Equation Kinetics Model** and **Four-Equation Kinetics Model**, you can select **Run Thermal Abuse Model Only (Battery Model is Turned off)** if you want to solve the energy equation (Equation 19.3) alone. Otherwise, the two battery potential equations (Equation 19.4) will also be solved.

Note:

- The kinetics parameters in thermal abuse models are specific to the battery material. The default settings may not be appropriate for your battery. You must provide parameter data for your own battery.
- For the One-Equation thermal abuse model, you can use the Ansys Fluent parameter estimation tool available through the TUI to compute the kinetics parameters from the battery oven testing data. Once the fitting is complete, the abuse rate data are passed to the thermal abuse model in both **Arrhenius** and **Table Data** formats, and the following text entry fields are automatically populated with fitted results:
  - **Arrhenius** type: parameters in the **Parameters for Kinetics Model** group box (**Battery Model** dialog box)
  - **Table Data** type: parameters in the **Abuse Rate Data Table** dialog box

  The experimental conditions used in the fitting are passed to the **Standalone Thermal Abuse Model** dialog box. You can run the standalone abuse model to evaluate the fitting quality and adjust the Arrhenius parameters or the table data to better match the experimental data.

  For details, see Using Parameter Estimation Tools (p. 3390).

31.2.3.7. Specifying External and Internal Short-Circuit Resistances

External Short-Circuit

To define the external short-circuit: In the **Battery Model** dialog box, under the **Model Options** tab, select **Specified Resistance** and specify a value for **External Resistance**. See Specifying Battery Model Options (p. 3303) for details.

Internal Short-Circuit

The intensity of the internal short-circuit is determined by the value of volumetric contact resistance r_c / a (External and Internal Electric Short-Circuit Treatment. In the Ansys Fluent battery model, this value is saved in a predefined user-defined memory allocation, Short Circuit Resistance. You can modify the value of this user-defined memory allocation to simulate the internal short circuit.

By default, the contact resistance is set to a very large number, so that $j_{short}=0$ and $\dot{q}_{short}=0$, that is, there is no internal short.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can simulate an internal short-circuit using either of the following two ways:

- **Patching**: Patch the contact resistance to the short circuit zone. The short circuit zone can be defined by creating a region cell register using the **Region Register** dialog box. See Region in the *Fluent User's Guide* (p. 3852) for more details.
- **Using UDF**: Use `DEFINE_ADJUST` or `DEFINE_ON_DEMAND UDFs` to specify the contact resistance using the UDM macro, `C_BATTERY_SHORT_R(c,t)`. The contact resistance could be a function of location and time. See the Fluent Customization Manual for details.

The internal short-circuit model is available only for the **MSMD** solution method.

31.2.3.8. Setting a Battery Swelling Case

The procedure for setting up a battery simulation that involves swelling is described below. Note that only steps that are pertinent to modeling swelling are listed here. You will also need to define the other settings (such as other material properties, boundary conditions, other models) as usual. For information about inputs related to other Ansys Fluent models that you are using in your simulation, refer to the appropriate sections for those models.

1. In the **Model Options** tab of the **Battery Model** dialog box, make sure that either **MSMD** or **Circuit Network** is selected as the **Solution Method**, and **Newman P2D Model**, **NTGK/DCIR Model**, or **Equivalent Circuit Model** is selected as the **E-chemistry Model**.
2. Specify the battery swelling model parameters.
   a. For NTGK or ECM battery model cases, follow the steps for including swelling effects outlined in Using the Empirical-Based Battery Swelling Model (p. 3376).
   b. For Newman's P2D battery cases, specify the swelling model parameters described in Inputs for the Newman's P2D Model (p. 3342).

      Once the battery swelling model is enabled, the Young's modulus of the battery active zone material will be automatically set to the average of the battery swelling model's inputs for the positive electrode, negative electrode, and separator.
3. Setup the structure model as described in Setting Up an Intrinsic Fluid-Structure Interaction (FSI) Simulation (p. 3229).

   The **Battery Effects** option will be automatically selected in the **Structural Model** dialog box.
4. Define the boundary conditions for all the battery wall zones.

   See Setting Up an Intrinsic Fluid-Structure Interaction (FSI) Simulation (p. 3229) for details.
5. If you want to simulate the flow around the battery module deformed due to swelling, set up the dynamic mesh model.
   a. In the **Dynamic Mesh** task page, select **Smoothing** as **Mesh Methods**.
   b. In the **Mesh Method Settings** dialog box (that opens by clicking **Settings...**), select **Linearly Elastic Solid** for the smoothing method.
   c. Create dynamic mesh zones.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

i. In the **Dynamic Mesh** task page, click **Create/Edit...** to open the **Dynamic Mesh Zones** dialog box.

ii. For each zone where considerable deformation due to the battery swelling effects is expected, select **Intrinsic FSI** for **Type** in the **Dynamic Mesh Zones** dialog box.

Note:

- You can keep the remaining settings at their default values.
- It is recommended that you create dynamic mesh zones for all fluid zone adjacent to the battery surface.

iii. Click **Create**.

See Using Dynamic Meshes (p. 1812) for more information.

31.2.4. Using Parameter Estimation Tools

The parameters for the NTGK, ECM, and one-equation thermal abuse models are battery specific. In theory, they can be fitted from some standard battery's testing data. Ansys Fluent provides parameter estimation tools for computing parameters for these models. The tool can be accessed from the graphical user interface (GUI) and text user interface (TUI).

31.2.4.1. Using Parameter Estimation Tools in the GUI

31.2.4.2. Using Parameter Estimation Tools in the TUI

31.2.4.1. Using Parameter Estimation Tools in the GUI

For the NTGK/DCIR and ECM models, you can access the parameter estimation tool from the **Battery Model** dialog box as described in the following sections:

- NTGK Model: Inputs for the NTGK/DCIR Model (p. 3324)
- Equivalent Circuit Model: Inputs for the Equivalent Circuit Model (p. 3334)

31.2.4.2. Using Parameter Estimation Tools in the TUI

To access the parameter estimation tool for the NTGK, ECM, and one-equation thermal abuse models in the text user interface (TUI), issue the following text commands:

```
define/models/battery-model/parameter-estimation-tool
Parameter Estimation for Model:
 1: NTGK Model
 2: ECM Model
 3: Thermal Abuse Model
Model option:  [1]
```

After you select a model option in the TUI, you will be prompted for information specific to the selected model.

For additional information, see the following sections:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

31.2.4.2.1. Using the Parameter Estimation Tool for the NTGK Model in the TUI
31.2.4.2.2. Using the Parameter Estimation Tool for the ECM Model in the TUI
31.2.4.2.3. Using the Parameter Estimation Tool for the Thermal Abuse Model in the TUI

31.2.4.2.1. Using the Parameter Estimation Tool for the NTGK Model in the TUI

The NTGK model parameters are fitted as functions of DOD and temperature. To fit the NTGK model parameters, you need to have battery's discharging curves at different constant C-rates. You can use testing data at different temperatures.

To use the parameter estimation tool for the NTGK model:

1. At the prompt, enter the testing data filenames.

   The file format will be displayed in the Fluent console.

   See Inputs for the NTGK/DCIR Model (p. 3324) for an example of an input file.

   Note:

   - The input file order does not matter.
   - You can use basic wildcards such as "*" and "?" to specify multiple files.
   - Once you specified all file names, you can enter **`quit`** (or a substring, such as **`q`**) or press **Enter** at the prompt to exit the file name specification loop.

2. For each temperature level, enter the temperature value and provide files containing testing data at the specified temperature.

3. After you specify input file names for different C-rate testing data, enter responses to the following prompts:

   - `Battery capacity`: Specifies capacity of the tested battery. Its value will be automatically transferred and displayed in the **Reference Capacity** text entry box in the **Battery Model** dialog box (**Model Parameters** tab).
   - `Number of DOD-levels`: Is used in the fitting. The default value of 30 DOD levels is usually sufficient.
   - `Include capacity fade effect`: Enables the capacity fade model.

If you use testing data from only one temperature level, the fitting results in both polynomial and table data types will be transferred to the **Battery Model** dialog box and to the parameter data table dialog boxes. In this case, the **Polynomial** data type will be automatically selected in the **Battery Model** dialog box, respectively. You can use either data type in your simulation.

The following is an example of using the parameter estimation tool for fitting the NTGK model parameters:

```
Parameter Estimation for Model:
 1: NTGK Model
 2: ECM Model
```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
 3: Thermal Abuse Model
Model option:  [1] 1

-- Make sure every input file has this format --
        Crate     1.0
        Temperature   300.0
        time_1    voltage_1
        time_2    voltage_2
        ...       ...
------------------------------------------------

 file name for curve (wildcards "*" and "?" can be used within the file names) [] ntgk-*.dat
The following 15 new curves have been added:
ntgk-3C-290K.dat
ntgk-3C-300K.dat
ntgk-2C-310K.dat
ntgk-4C-300K.dat
ntgk-5C-310K.dat
ntgk-1C-310K.dat
ntgk-4C-310K.dat
ntgk-2C-290K.dat
ntgk-5C-290K.dat
ntgk-3C-310K.dat
ntgk-1C-290K.dat
ntgk-2C-300K.dat
ntgk-5C-300K.dat
ntgk-4C-290K.dat
ntgk-1C-300K.dat
 file name for curve (wildcards "*" and "?" can be used within the file names) [] ()
Battery capacity (Ah) [14.6]
Number of DOD-levels [20]
Include Capacity Fade Effect? [yes] no
```

For more information, see Inputs for the NTGK/DCIR Model (p. 3324).

31.2.4.2.2. Using the Parameter Estimation Tool for the ECM Model in the TUI

To fit the ECM model parameters, you need to have battery's Hybrid Pulse Power Characterization (HPPC) testing data at different SOC levels. Similar to the NTGK model, you can use testing data at different temperatures.

To use the parameter estimation tool for the ECM model:

1. At the prompt, specify the HPPC testing data file names.

   For each SOC level, you must provide a separate file with battery's testing data points in text format. At least one valid SOC level testing data is required per temperature level. For good simulation results, the range of SOC levels used in the testing should cover the SOC range simulated in the CFD runs.

   The file format will be displayed in the Fluent console.

   See Inputs for the Equivalent Circuit Model (p. 3334) for an example of an input file.

   Note:

   - The input file order does not matter.
   - You can use basic wildcards such as "*" and "?" to specify multiple files.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Once you specified all file names, you can enter **quit** (or a substring, such as **q**) or press **Enter** at the prompt to exit the file name specification loop.

2. After you specify input file names, enter responses to the following prompts:

   - `Battery capacity`: Specifies capacity of the tested battery. Its value will be automatically transferred and displayed in the **Reference Capacity** text entry box in the **Battery Model** dialog box (**Model Parameters** tab).
   - `Circuit Model`:
     - Enter **1** if you want to use the 4P parameter model (one RC loop circuit)
     - Enter **2** if you want to use the 6P parameter model (two RC loop circuit).
   - `Fitting Method`:
     - Enter **1** if you want to use the JH method

       The JH method uses HPPC testing data only from the relaxation period.
     - Enter **2** if you want to use the LM method.

       The LM method uses data from both pulse and relaxation periods.

3. Although the parameter estimation tool does not require the testing battery's capacity, you must specify this value.

4. Specify whether the capacity fade effect is considered or not. If your answer is **yes**, you also need to specify a set of testing data files for discharging curves at different discharging rates (as for the NTGK fitting process).

If you provide testing data files at different temperatures, the parameter estimation tool will perform the fitting for each temperature level. Ansys Fluent will report the fitted results in the console and automatically pass them in a table data type to the parameter data table dialog boxes. The **Table** data type will be automatically selected in the **Battery Model** dialog box (**Model Parameters** tab). This will allow you to consider the temperature-dependent effect.

The following is an example of using the parameter estimation tool for fitting the ECM model parameters.

```
Parameter Estimation for Model:
 1: NTGK Model
 2: ECM Model
 3: Thermal Abuse Model
Model option:  [2] 2

-- Make sure every input file has this format --
       SOC       0.6
       I         3.153
       Temperature   300.0
       time_1    voltage_1
       time_2    voltage_2
       ...       ...
-----------------------------------------------
  where SOC: soc level
```

```
          I: pulse current

 file name for curve (wildcards "*" and "?" can be used within the file names) [] soc??-???k.dat
The following 30 new curves have been added:
soc02-310k.dat
soc09-290k.dat
soc10-290k.dat
soc05-290k.dat
soc02-290k.dat
soc06-300k.dat
soc01-290k.dat
soc05-300k.dat
soc02-300k.dat
soc08-300k.dat
soc03-310k.dat
soc05-310k.dat
soc03-290k.dat
soc04-290k.dat
soc10-310k.dat
soc03-300k.dat
soc07-310k.dat
soc04-310k.dat
soc08-290k.dat
soc09-310k.dat
soc01-300k.dat
soc10-300k.dat
soc01-310k.dat
soc07-300k.dat
soc06-290k.dat
soc06-310k.dat
soc04-300k.dat
soc08-310k.dat
soc07-290k.dat
soc09-300k.dat
 file name for curve (wildcards "*" and "?" can be used within the file names) [] ()
Battery capacity (Ah) [14.6]
Circuit Model:
 1: 4P parameter model (one RC loop)
 2: 6P parameter model (two RC loops)
 Circuit model option:  [2]
Fitting Method:
 1: Jiang-Hu Method (JH)
 2: Levenberg-Marquardt Method (LM)
 Fitting method option:  [2]
Include Capacity Fade Effect? [yes] no
```

For more information, see Inputs for the Equivalent Circuit Model (p. 3334).

31.2.4.2.3. Using the Parameter Estimation Tool for the Thermal Abuse Model in the TUI

Kinetics data used in the thermal abuse model are battery specific. Reaction kinetics parameters in Equation 19.33 can be fitted by using testing data (for example, Oven test or ARC). Temperature vs. time data collected from experiment for a battery cell under thermal abuse condition can be used as input in the parameter estimation tool.

The following thermal balance equation is used to describe the heating process and in the kinetics parameter fitting of the temperature-time data:

$$V\rho C_p\frac{dT}{dt} = VHA\cdot\exp\left[-\frac{E}{RT}\right]\alpha^m(1-\alpha)^n - A_{ext}h(T-T_0) - A_{ext}\sigma\varepsilon(T^4-T_{inf}^4) \tag{31.1}$$

where:

V = volume

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A_{ext} = external area

ρ = material density

C_p = specific heat

h = heat transfer coefficient

σ = Stefan-Boltzmann constant

ε = surface emissivity

H = heat of reaction

A = pre-exponential factor

E = activation energy

α = reaction progress variable

T = temperature

T_0 = ambient temperature

T_{inf} = enclosure temperature for radiation

m and n = reaction order parameters

The equation will be printed in the console for your reference.

You will be prompted to specify file names for temperature testing data and values for the variables used in Equation 31.1 (p. 3394). For good fitting results, it is recommended that you use multiple sets of testing data obtained under different heating conditions, such as different oven temperature or different battery initial temperature.

After running the parameter estimation tool for the one-equation thermal abuse model, Ansys Fluent will obtain and pass the following parameter to the **Battery Model** dialog box (**Advanced Options** tab): **A**, **E**, **HW**, **m**, and **n**. These parameters will also be printed in the console.

31.2.5. Initializing the Battery Model

There are two ways to initialize the battery model:

- Initialize the flow field from the **Solution Initialization** task page. The battery model will be initialized also.
- Initialize the battery model in isolation by clicking the **Init** button in the **Battery Model** dialog box. Other flow variables remain untouched.

  Use the second method if you want to freeze the flow field and use a different battery model to simulate the battery field. When switching from one battery submodel to another, the battery model must be re-initialized. The **Init** button in the **Battery Model** dialog box provides a convenient way to accomplish this if you do not want to initialize the flow field.

31.2.6. Modifying Material Properties

When using the MSMD battery models in Ansys Fluent, you must define the electrical conductivity of the cell, tab, and busbar materials.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For active zone materials, you usually need to use the **defined-per-uds** method, specifying different values of electrical conductivity σ_+ and σ_- for the positive and negative electrodes, respectively. For passive zone materials, usually you can use the **constant** method to define electrical conductivity.

31.2.7. Solution Controls for the MSMD Battery Model

When you use the battery models, two potential equations are solved in addition to other fluid dynamic equations, depending on the type of simulation. The two potential equations (**Potential Phi+** and **Potential Phi-**) are listed in the **Equations** dialog box, where they can be enabled/disabled during the solution process.

Solution → **Controls** → **Equations...**

Also, if required, define advanced settings the **Advanced Solution Controls** dialog box.

Setup → **Controls** → **Advanced...**

31.2.8. Predefined Report Definitions for the Battery Model

In battery applications, battery terminal voltage, terminal current and maximum temperature of battery zones are always of great interest. To assist you in reporting solution data during battery model simulations, the following report definitions are automatically created in Ansys Fluent:

- **battery-max-temperature**
- **battery-terminal-voltage** (**Circuit Network** and **MSMD** solution methods only)
- **battery-terminal-current** (**Circuit Network** and **MSMD** solution methods only)

You can use these predefined report definitions to create report files and report plots as described in Creating Report Files (p. 4123) and Creating Report Plots (p. 4126).

Note that these report definitions are not available for export of copying.

31.2.9. Postprocessing the MSMD Battery Model

You can perform postprocessing using standard Ansys Fluent quantities and variables related to the battery model under the **Battery Variables...** category. When using the battery model, the following additional variables will be available for postprocessing:

- For the **CHT Coupling** and **FMU-CHT Coupling** solution methods:
  - **Potential Field**
  - **Current Magnitude**
  - **Electrical Conductivity**
  - **Total Heat Source**
  - **X Current Density in Battery**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - **Y Current Density in Battery**
  - **Z Current Density in Battery**

  Total Heat Source is always available. The other variables are available only if the **Enable Joule Heat in Passive Zones** option is selected in the **Model Option** tab.

- For the **Circuit Network** solution method:
  - **Potential Field**
  - **Current Magnitude**
  - **Electrical Conductivity**
  - **Network Current**
  - **Network Voltage**
  - **Network State of Charge**
  - **Total Heat Source**
  - **X Current Density in Battery**
  - **Y Current Density in Battery**
  - **Z Current Density in Battery**

  Network Current, **Network Voltage**, **Network State of Charge**, and **Total Heat Source** are always available, while the other variables are available only if the **Enable Joule Heat in Passive Zones** option is selected in the **Model Options** tab.

- For the **MSMD** solution method:
  - **Cathode Potential**
  - **Cell Cluster ID**
  - **Cell Voltage**
  - **Current Magnitude**
  - **Cathode Electric Conductivity**
  - **Anode Electrical Conductivity**
  - **Passive Zone Electrical Conductivity**
  - **Passive Zone Potential**
  - **State of Charge**
  - **Joule Heat Source**
  - **Echem Heat Source**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Entropic Heat Source**
- **Short-Circuit Heat Source**
- **Total Heat Source**
- **Short Current Source**
- **Echem Current Source**
- **Total Current Source**
- **Short-Circuit Resistance**
- **X Current Density for phi+**
- **Y Current Density for phi+**
- **Z Current Density for phi+**
- **X Current Density for phi-**
- **Y Current Density for phi-**
- **Z Current Density for phi-**
- **X Current Density in Battery**
- **Y Current Density in Battery**
- **Z Current Density in Battery**
- **Newman P2D: Average ce in Cathode** (Newman's P2D model only)
- **Newman P2D: Average ce in Separator** (Newman's P2D model only)
- **Newman P2D: Average ce in Anode** (Newman's P2D model only)
- **Newman P2D: Average cs in Cathode** (Newman's P2D model only)
- **Newman P2D: Average cs in Anode** (Newman's P2D model only)
- **Newman P2D: Average phis in Cathode** (Newman's P2D model only)
- **Newman P2D: Average phis in Anode** (Newman's P2D model only)
- **Newman P2D: Average phis in electrolyte** (Newman's P2D model only)
- **Newman P2D: Average phi_s-phi_e at anode-separator interface** (Newman's P2D model)
- **Li-Insertion-Induced Electrode Layer Thickness Change Ratio** (ϵ_{swell} in Equation 19.77 in the *Fluent Theory Guide*; Newman's P2D model only)
- **X Component of Battery Layer Normal Vector** (Newman's P2D model only)
- **Y Component of Battery Layer Normal Vector** (Newman's P2D model only)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - **Z Component of Battery Layer Normal Vector** (Newman's P2D model only)
  - **Total Electrode Layer Thickness Change Ratio** (left-hand side of Equation 19.79 or Equation 19.80 in the *Fluent Theory Guide*; Newman's P2D model only)
  - **Projected Stress in Electrode Layer Normal Direction** (Newman's P2D model only)

- In any of the solution methods, if the thermal abuse model is used, the following variables are also available for postprocessing:
  - For the one-equation thermal abuse model:
    - → **Thermal-Abuse Heat Source**
    - → **Abuse: alpha**
  - For the four-equation thermal abuse model:
    - → **Thermal-Abuse Heat Source**
    - → **Abuse: Csei**
    - → **Abuse: Cpe**
    - → **Abuse: Cne**
    - → **Abuse: Ce**
    - → **Thermal-Abuse Heat Source: SEI Decomposition**
    - → **Thermal-Abuse Heat Source: Negative-Solvent**
    - → **Thermal-Abuse Heat Source: Positive-Solvent**
    - → **Thermal-Abuse Heat Source: Electrolyte Decomposition**
    - → **Thermal-Abuse Heat Source: Internal Short** (with the **Enable Internal Short Option** only)

In addition, you can generate vector plots of the following postprocessing variables:

- **current-density-jp**: Is the electric current vector for the potential field on the positive electrode. It is available only for the MSMD solution method.
- **current-density-jn**: Is the electric current vector for the potential field on the negative electrode. It is available only for the MSMD solution method.
- **current-density-j**: Is the electric current vector in the battery conductive zones. It is available if at least one potential equation is solved.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **layer-nornal-direction**: Is the vector of battery layer normal direction. It is only available when either the battery swelling model or the orthotropic thermal conductivity model is used where the battery layer orientation information is needed.

Note:

For reviewing simulation results in CFD Post for cases involving UDS, export the solution data as CFD-Post-compatible file (`.cdat`) in Fluent and then load this file into CFD-Post. This will ensure that the full variable set is available for post processing in CFD-Post.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 32: Modeling Fuel Cells

The Ansys Fluent Fuel Cell Modules Chapter provides information about the background and the usage of two separate add-on fuel cell modules for Ansys Fluent. The available Ansys Fluent add-on fuel cell modules are:

- PEMFC Model - allows you to model polymer electrolyte membrane fuel cells (PEMFC) with (or without) micro-porous layers. This model is a new and recommended model for simulating energy conversion processes in a PEM fuel cell. For more information, see PEMFC Model Theory.
- Fuel Cell and Electrolysis Model - allows you to model polymer electrolyte membrane fuel cells (PEMFC), solid oxide fuel cells (SOFC), and solid oxide electrolysis cells (SOEC) with Ansys Fluent. This model is sometimes referred to as the Resolved Electrolyte model. Note that the PEMFC sub-model can be used to model both the low-temperature and high-temperature PEM fuel cells. For more information, see Fuel Cell and Electrolysis Model Theory.
- SOFC With Unresolved Electrolyte Model - allows you to model solid oxide fuel cells (SOFC). For more information, see SOFC Fuel Cell With Unresolved Electrolyte Model Theory.

Additional information about the model is provided in the following sections:

32.1. Using the PEMFC Model

32.2. Using the Fuel Cell and Electrolysis Model

32.3. Using the Solid Oxide Fuel Cell With Unresolved Electrolyte Model

32.1. Using the PEMFC Model

The procedure for setting up and solving fuel cell problems using the PEMFC model is described in detail in this section. Refer to the following sections for more information:

32.1.1. Overview and Limitations

32.1.2. Geometry Definition for the PEMFC Model

32.1.3. Installing the PEMFC Model

32.1.4. Loading the PEMFC Module

32.1.5. Workflow for Using the PEMFC Module

32.1.6. Setting Up the PEMFC Module

32.1.7. PEMFC Model Boundary Conditions

32.1.8. Solution Guidelines for the PEMFC Model

32.1.9. Postprocessing the PEMFC Model

32.1.10. User-Accessible Functions

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.1.1. Overview and Limitations

The Ansys Fluent PEMFC model is composed of several user-defined functions (UDFs) and a graphical user interface. The potential fields are solved as user-defined scalars. The capillary pressure, the water content (water in dissolved phase), and liquid saturation in the gas channels are also solved as user-defined scalars. The electrochemical reactions occurring in the catalyst layers are modeled through various source terms while other model parameters are handled through the user interface. The PEMFC model can be used in parallel Ansys Fluent as well.

Note the following limitation when using the PEMFC model:

- The anisotropic species diffusivity option is not compatible with the PEMFC model.

32.1.2. Geometry Definition for the PEMFC Model

Due to the fact that there is a number of different physical zones associated with the fuel cell, the following regions must be present in the fuel cell mesh (see Figure 20.1: Schematic of a PEM Fuel Cell):

- Anode flow channel
- Anode gas diffusion layer
- Anode micro-porous layer (optional)
- Anode catalyst layer
- Membrane layer (solid)
- Cathode catalyst layer
- Cathode micro-porous layer (optional)
- Cathode gas diffusion layer
- Cathode flow channel

The following zones have to be identified, if present in the fuel cell mesh:

- Anode current collector (solid)
- Cathode current collector (solid)
- Coolant channel

32.1.3. Installing the PEMFC Model

The PEMFC model is provided as an add-on module with the standard Ansys Fluent licensed software. The module is installed with the standard installation of Ansys Fluent in a directory called `addons/pem-fc` in your installation area. The PEMFC module consists of a UDF library and a pre-compiled Scheme library, which must be loaded and activated before calculations can be performed.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.1.4. Loading the PEMFC Module

The PEMFC module can be loaded only after a valid Ansys Fluent case file has been set or read.

There are two different ways in which you can load the PEMFC module into Ansys Fluent:

- Through the ribbon

  In the **Physics** ribbon tab, click **More** (**Models** group), and select **PEM Fuel Cell Model...**.

  Physics → **Models** → **More** → **PEM Fuel Cell Model...**

  Once the PEMFC module is loaded into Ansys Fluent, the **PEM Fuel Cell Model** dialog box opens where you can enable and set the PEMFC model as described in Setting Up the PEMFC Module (p. 3404).

- Through the text user interface (TUI)

  The module The text command to load the add-on module is

  `define` → `models` → `addon-module`

  A list of Ansys Fluent add-on modules is displayed:

```
> /define/models/addon-module
Fluent  Addon Modules:
     0. None
     1. MHD Model
     2. Fiber Model
     3. Fuel Cell and Electrolysis Model
     4. SOFC Model with Unresolved Electrolyte
     5. Population Balance Model
     6. Adjoint Solver
     7. Single Potential Battery Module
     8. MSMD Battery Model
     9. PEM Fuel Cell Model
Enter Module Number: [0] 9
```

  Select the PEM Fuel Cell Model by entering the module number 9.

During the loading process, a Scheme library (containing the graphical and text user interface) and a UDF library (containing a set of user defined functions) are loaded into Ansys Fluent.

32.1.5. Workflow for Using the PEMFC Module

The PEMFC model can be used to model polymer electrolyte membrane fuel cells (PEMFC). The following describes an overview of the procedure required in order to use the PEMFC model in Ansys Fluent.

1. Start Ansys Fluent.

   You must start Ansys Fluent in 3D double-precision mode. Note that the PEMFC model is only available in 3D.

2. Read the case or mesh file.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Scale the mesh, if necessary.
4. Use the **PEM Fuel Cell Model** dialog box to define the fuel cell model parameters.
5. Define material properties.
6. Set the operating conditions.
7. Set the boundary conditions.
8. Start the calculations.
9. Save the case and data files.
10. Process your results.

Important:

- The **PEM Fuel Cell Model** dialog box greatly simplifies the input of parameters and boundary conditions, but it does not replace the boundary conditions interface. Therefore, it is a good policy to start the setup with the **PEM Fuel Cell Model** dialog box and do the finishing steps for boundary conditions afterwards.
- Note that the majority of this chapter describes how to set up the Ansys Fluent PEMFC model using the graphical user interface. You can also perform various tasks using the text user interface. For more information, see Using the PEMFC Text User Interface.

32.1.6. Setting Up the PEMFC Module

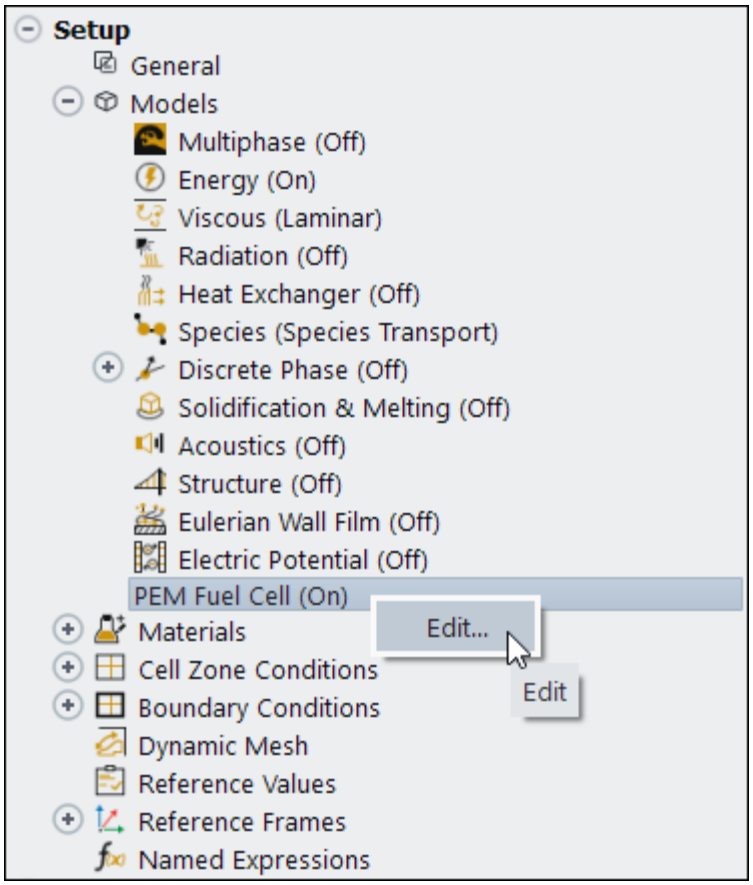
Once the module has been loaded, in order to set fuel cell model parameters and assign properties to the relevant regions in your fuel cell, you need to access the fuel cell graphical user interface (the **PEM Fuel Cell Model** dialog box).

The **PEM Fuel Cell Model** dialog box can be accessed either through the ribbon (as described in Loading the PEMFC Module (p. 3403)) or from the **Setup/Models/PEM Fuel Cell** tree item.

Setup → Models → PEM Fuel Cell **Edit...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 32.1: The PEMFC Option in the Outline View



If the PEMFC model is not yet enabled, you can enable it by selecting **Enable PEMFC** in the **PEM Fuel Cell Model** dialog box.

Using the **PEM Fuel Cell Model** dialog box, you can identify the relevant zones for the current collectors, flow channels, gas diffusion layers, micro porous layers, catalyst layers, and the electrolyte (membrane). You can specify the following inputs using the **PEM Fuel Cell Model** dialog box. Optional inputs are indicated as such.

1. Set the appropriate options for the fuel cell model.
2. Set the various parameters for the fuel cell model.
3. Select the appropriate zones and specify the properties on the anode side.
4. Select the appropriate zones and specify the properties of the electrolyte/membrane.
5. Select the appropriate zones and specify the properties on the cathode side.
6. Provide input for advanced features such as contact resistivities, coolant channel properties, or stack management settings (optional).
7. Set solution controls such as under-relaxation factors (optional).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. Provide input to assist reporting (optional).

Note:

The default values of the fuel cell model parameters and zone properties are determined based on various data available in the literature (419, 476, 506, 586, 663 718, (listed in the Bibliography in the *Fluent Theory Guide*), and others).

Refer to the following sections for more information:

32.1.6.1. Specifying Model Options (Model Tab)

32.1.6.2. Specifying Model Parameters (Parameters Tab)

32.1.6.3. Specifying Anode Properties (Anode Tab)

32.1.6.4. Specifying Electrolyte/Membrane Properties (Electrolyte Tab)

32.1.6.5. Specifying Cathode Properties (Cathode Tab)

32.1.6.6. Setting the External Electrical Tabs (Electrical Tabs Tab)

32.1.6.7. Setting Advanced Properties (Advanced Tab)

32.1.6.8. Customizing the PEM Fuel Cell Module

32.1.6.9. Reporting on the Solution (Reports Tab)

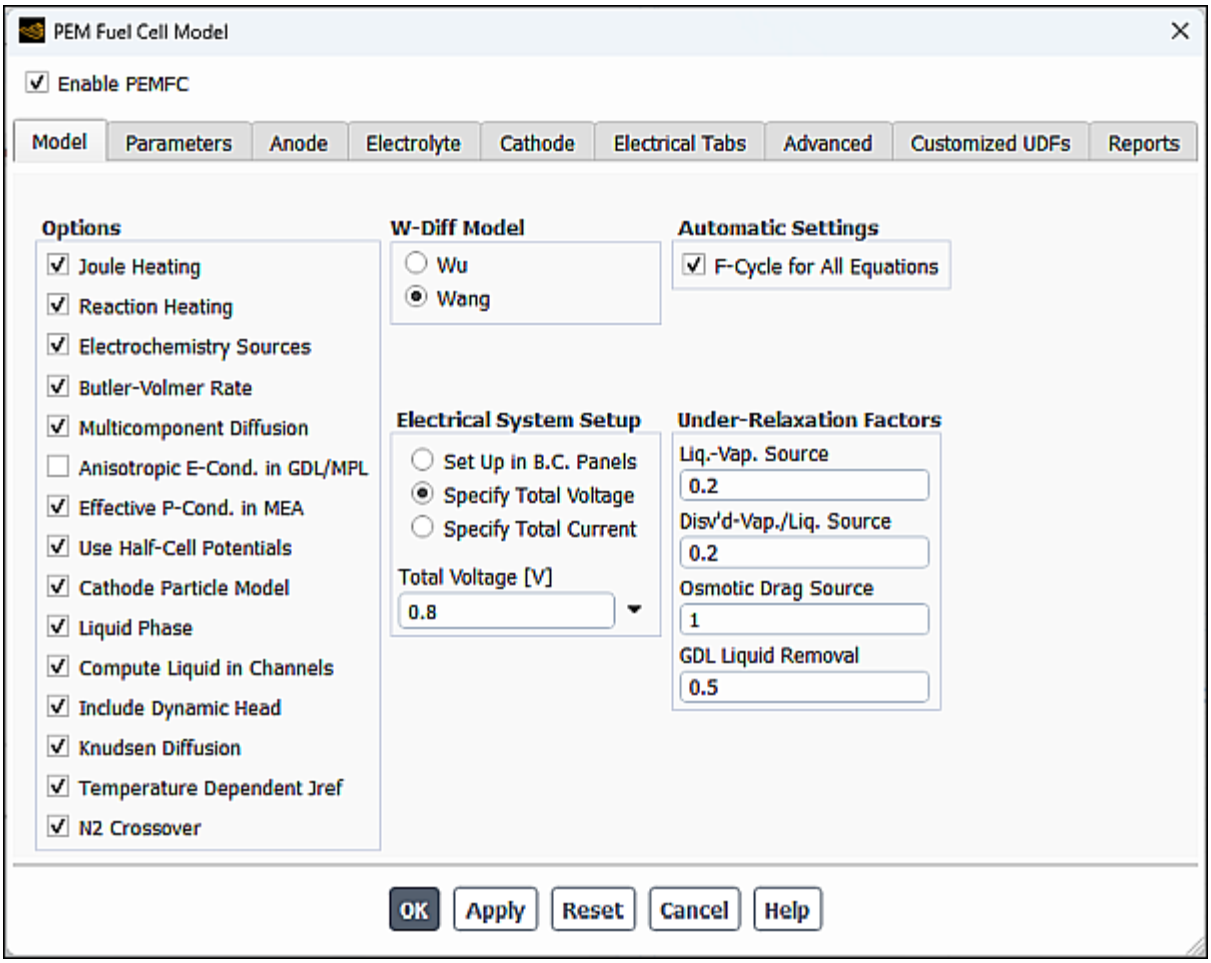
32.1.6.10. User-Defined Functions Hooked to the PEMFC Module

32.1.6.1. Specifying Model Options (Model Tab)

The **Model** tab of the **PEM Fuel Cell Model** dialog box allows you to select or cancel the selection of various options when solving a fuel cell problem.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 32.2: The Model Options in the PEM Fuel Cell Model Dialog Box



Options

Several fuel cell model options are available including:

Joule Heating

takes into account ohmic heating. This option includes the i^2/σ terms in the energy source term in the calculations. See Table 20.1: Volumetric Heat Source Terms for the list of additional volumetric sources in the thermal energy equation.

Reaction Heating

takes into account the heat generated by the electrochemical reactions.

Electrochemistry Sources

allows the PEMFC model to take electrochemistry effects into account. If you are only interested in the basic flow field throughout the fuel cell, you can turn off the **Electrochemistry Sources** option in order to suppress most effects of the PEMFC model. You may also turn off the effect of these sources in order to obtain a fluid flow initially, and then turn it back on.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Butler-Volmer Rate

(the default) is used to compute the transfer currents inside the catalyst layers. If this option is turned off, the Tafel approximation (Equation 20.7 and Equation 20.8) is used.

Liquid Phase

takes into account liquid phase calculations. Use this option if you are solving for liquid transport in the fuel cell.

Multicomponent Diffusion

is used to compute the gas species mass diffusivity using the full multicomponent diffusion method as described in Equation 20.47, as opposed to the default option that uses Equation 20.45.

Anisotropic E-cond. in GDL/MPL

is used to model the typically non-isotropic electrical conductivity. It is applicable only for porous electrodes (gas diffusion layers and micro porous layers).

Due to the fibrous structure of the porous material that is used for the electrodes (or gas diffusion layer), the electrical conductivity is typically non-isotropical, with the cross-plane components being orders of magnitude smaller than the in-plane components. This can be modeled using the **Anisotropic E-cond in GDL/MPL** setting. When this option is enabled, the **Electrical Conductivity** for the solid material used in the porous electrodes is no longer used. Instead, you need to specify, for this solid material, the electrical conductivity by choosing one of the three non-isotropical options for the UDS diffusivity (`UDS-0`). The three options are: **anisotropic**; **orthotropic**; and **cyl-orthotropic**. For more information about these UDS Diffusivity options, refer to the Ansys Fluent User's Guide (p. 1).

For example, to use this feature, perform the following steps:

- Select the **Anisotropic E-cond in GDL/MPL** option in the **Model** tab of the **PEM Fuel Cell Model** dialog box.
- In the **Create/Edit Materials** dialog box, select `defined-per-uds` for **UDS Diffusivity** for the solid material that is to be used for the porous electrode.
- Select one of the three options for UDS-0: **anisotropic**; **orthotropic**; or **cyl-orthotropic** and set the appropriate values.

Important:

Note that, in this case, the **Electrical Conductivity** for *this* solid material is ignored.

Effective P-Cond. in MEA

enables the computation of the electrolyte phase conductivity σ_{mem} with consideration of ionomer volume fraction and tortuosity in the catalyst layers using Equation 20.48. If this option is disabled, then $\varsigma_a / \tau_a = \varsigma_c / \tau_c = 1$ and $\beta_{an} = \beta_{ca} = \beta_{mem}$.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Use Half-Cell Potentials

enables the calculations of the anode and cathode half-cell potentials U^0_{an} and U^0_{cat} using the Nernst equations (Equation 20.13 and Equation 20.14). If this option is disabled, the constant open-circuit voltage is assumed, the U^0_{an} set to zero, and U^0_{cat} is set to a value specified for **V_Open** in the **Parameters** tab.

Cathode Particle Model

enables the cathode particle model. For information about this model, see The Cathode Particle Model.

Liquid Phase

enables the liquid water transport equations in the computations. By default, this option is enabled.

Compute Liquid in Channels

enables the solution of the liquid saturation equation in the gas channels, which helps in predicting the pressure drop increase. When this option is enabled, the viscous resistance in the gas channel is treated as a source term and is added directly to the momentum equations. By default, this option is disabled, and the equation is not solved.

Include Dynamic Head

(if enabled) includes the dynamic pressure in the gas channel in Equation 20.35.

Knudsen Diffusion

(if enabled) includes the effect of Knudsen diffusion in the calculation of the effective gas diffusivity using Equation 20.47 in the *Fluent Theory Guide*.

Temperature Dependent Jref

(when enabled) takes into account the temperature dependency of the reference exchange current density (Equation 20.9 and Equation 20.10 in the *Fluent Theory Guide*). When this option is enabled (default), you can specify **Activation Energy for J_ref** and **Reference Temperature for J_ref** for the anode and cathode catalyst layers as further described.

N2 Crossover

(when enabled) includes the effect of the N_2 crossover flux across the membrane (Equation 20.10 in the *Fluent Theory Guide*). When this option is enabled, you must specify **N2 Crossover factor** in the **Parameters** tab (**Other Parameters** group box). This option is disabled by default.

Ice Phase

(when enabled) includes the effect of the ice phase (Equation 20.41 in the *Fluent Theory Guide*). When this option is enabled, you must specify **Desublimation Rate** and **Sublimation Rate** in the **Parameters** tab (**Other Parameters** group box). This option is available only for transient simulations and disabled by default.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

W-Diff Model

You can select one of the following formulations for the function $f(\lambda)$ in Equation 20.50:

Wu

computes $f(\lambda)$ using Wu's formulation (Equation 20.51).

Wang

computes $f(\lambda)$ using Wang's formulation (Equation 20.52).

Electrical System Setup

You can select one of the following methods to set up the electrical potential (UDS0) boundary conditions at external wall tabs:

Set Up in B.C. Panels

When enabled, Ansys Fluent uses your electrical potential boundary conditions that you specified in the relevant boundary condition dialog boxes for the anode and cathode terminals. See PEMFC Model Boundary Conditions (p. 3440) for more information.

Specify Total Voltage

Allows you to specify **Total Voltage**, which is the cell or stack voltage (V). You have the option to specify the **Total Voltage** as a constant (default) or expression. See Creating and Using Expressions (p. 1049) for more information about the expression specification method.

The electrical potential UDS0 conditions in the boundary condition dialog boxes are ignored. Ansys Fluent automatically sets the potential to zero at the anode tabs and applies a fixed value of **Total Voltage** at the cathode tabs.

Specify Total Current

Allows you to specify **Total Current**, which is the total current (Amps) generated by the cell or stack. You have the option to specify the **Total Current** as a constant (default) or expression. See Creating and Using Expressions (p. 1049) for more information about the expression specification method.

The electrical potential UDS0 conditions in the boundary condition dialog boxes are ignored. Ansys Fluent automatically sets the potential to zero at the anode tabs and applies a constant flux (current density) at the cathode tabs. The current density is computed by dividing the **Total Current** by the **Electrolyte Projected Area** (specified in the **Reports** tab).

Under-Relaxation Factors

You can use the **Under-Relaxation Factors** fields to influence the solution process.

Liquid-Vapor

The source term S_{gl} in Equation 20.33 usually requires under-relaxation. Since the S_{gl} is computed explicitly as a source term for the capillary pressure equation (Equation 20.33), the under-relax-

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ation factor usually must be a small value in order to obtain convergence. You can change the default value for the under-relaxation factor by changing the value for **Liquid-Vapor**.

Disv'd-Vapor/Liquid

The source terms S_{ld} and S_{gd} in Equation 20.25 usually require under-relaxation also. You can change the default value for the under-relaxation factor by changing the value for **Disv'd-Vapor/Liquid**.

Osmotic Drag Source

You can also specify an under-relaxation factor for the osmotic drag term, namely the left-hand side term of Equation 20.25 by changing the value for **Osmotic Drag Source**.

GDL Liquid Removal

The liquid water flux that goes out of the gas diffusion layer (GDL) at the GDL-Channel interface (Equation 20.35) can also be under-relaxed using **GDL Liquid Removal**.

Automatic Settings

The following parallel multigrid solver control parameter is available:

F-cycle for All Equations

sets the multigrid cycle type to F cycle for all equations that are being solved. This control overrides the equation cycle settings in the **Advanced Solution Controls** dialog box.

Note:

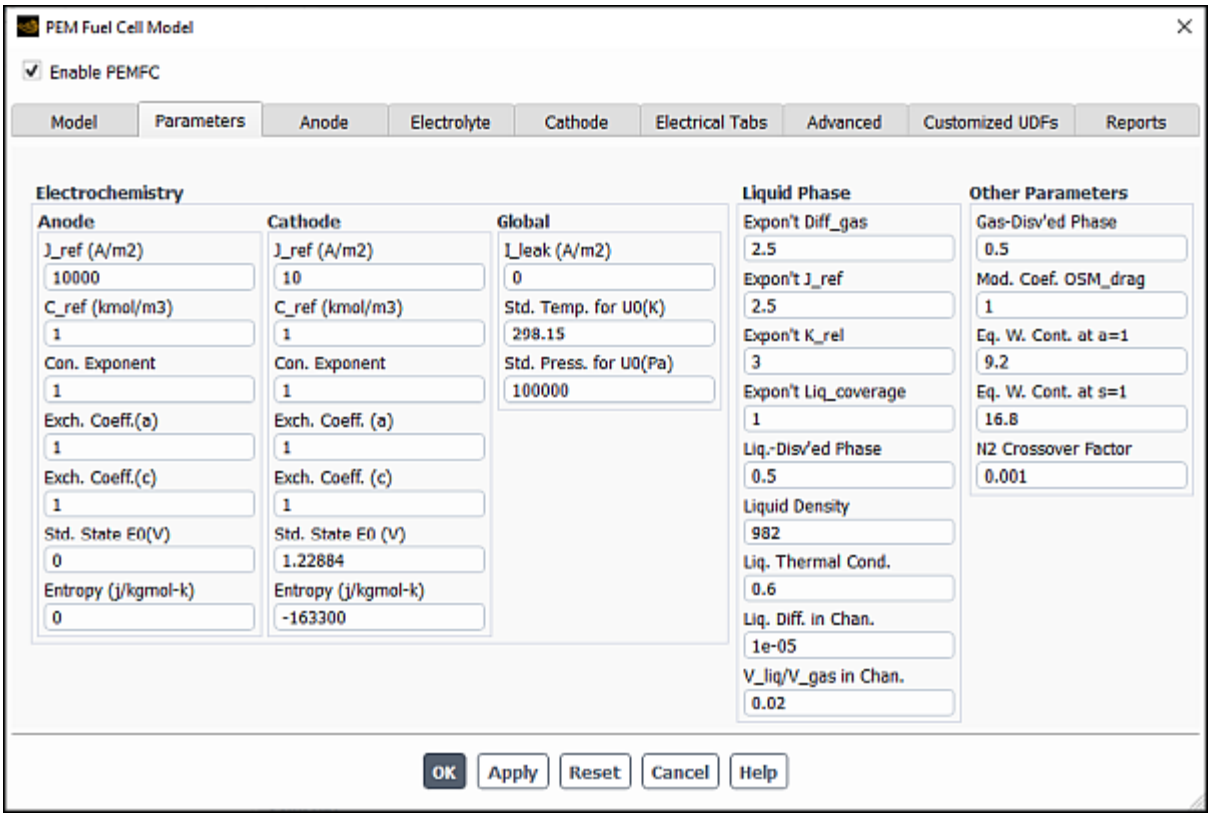
Using F-cycle for PEMFC computations is the best practice approach.

32.1.6.2. Specifying Model Parameters (Parameters Tab)

You can use the **Parameters** tab of the **PEM Fuel Cell Model** dialog box to specify the electrochemistry parameters for the PEMFC model, reference diffusivities for the reactants and other model parameters.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 32.3: The Parameters Tab of the PEM Fuel Cell Model Dialog Box



Electrochemistry

There are various parameters under **Electrochemistry** in the **PEM Fuel Cell Model** dialog box. For both the anode and the cathode, you can set the following parameters or leave the default values:

J_ref

corresponds to j_{an}^{ref} and j_{cat}^{ref}, the reference exchange current density from Equation 20.9 and Equation 20.10.

C_ref

corresponds to the reference concentration ($[H_2]_{ref}$ and $[O_2]_{ref}$) with units of 1 kgmol/m^3 (see Equation 20.5 and Equation 20.6).

Con. Exponent

corresponds to γ, the concentration dependence from Equation 20.5.

Exch. Coeff. (a) , Exch. Coeff. (c)

are the transfer coefficients α_{an} and α_{cat} from Equation 20.5 and Equation 20.6, respectively.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Std. State E0

are the reversible potentials E^0_{an} and E^0_{cat} in Equation 20.13 and Equation 20.14, respectively. This field appears only when **Use Half-Cell Potentials** option is selected under the **Model** tab.

Entropy

is the reaction entropy ΔS_{an} and ΔS_{cat} in Equation 20.13 and Equation 20.14.

V_Open

corresponds to the constant value assigned to cathode half-cell potential U^0_{an} and U^0_{cat}. This parameter appears only when **Use Half-Cell Potentials** option is not selected under the **Model** tab.

I_leak

is the leakage current density (A/m^2). It is used to compute the total leakage current I_l (in Equation 20.55 through Equation 20.57) by:

$$I_l = I_{leak} A_e$$

where A_e is the electrolyte projected area (specified under the **Reports** tab). When leakage through the electrolyte occurs, the fuel cell generates less current especially for cases with low values of fuel or air utilization. Note that you can also specify the total leakage current through the user-defined function `Leakge_Current()`. For more information, see User-Accessible Functions (p. 3443).

Std. Temp. for U0 (K)

is the reference standard state temperature which is used for half-cell potentials computations in the Nernst equations (Equation 20.13 and Equation 20.14). This parameter appears only when the **Use Half-Cell Potentials** option is selected under the **Model** tab.

Std. Press. for U0 (Pa)

is the reference standard state pressure which is used for half-cell potentials computations in the Nernst equations (Equation 20.13 and Equation 20.14). This parameter appears only when the **Use Half-Cell Potentials** option is selected under the **Model** tab.

Reference Diffusivity

The parameters in the **Reference Diffusivity** group box appear only if the **Multicomponent Diffusion** option is turned off in the **Model** tab. These parameters correspond to the species mass diffusivity D^0_i from Equation 20.45.

Liquid Phase

When the **Liquid Phase** option is selected under the **Model** tab, you can specify the following parameters that appear in the **Liquid Phase** group box:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Expon't Diff_gas

corresponds to r_s from Equation 20.45 for multiphase PEMFC calculations.

Expon't J_ref

is the exponent γ_j that is used to modify R_{an} and R_{ca} according to Equation 20.38 to account for liquid blockage to the reaction surface in Equation 20.5 through Equation 20.8.

Expon't K_rel

is the exponent b that is used to compute the relative permeability in K_r (Equation 20.31).

Expon't Liq_coverage

is the exponent θ that is used to compute the phase-change rates S_{gd} and S_{ld} (Equation 20.26 and Equation 20.27).

Liq.-Disv'ed Phase

is the mass exchange rate constant γ_{ld} that is used to compute the rate of mass change between liquid and dissolved phases S_{ld} (Equation 20.27).

Liquid Density , Liq. Thermal Cond.

are the density ρ_l and thermal conductivity k_l of liquid water, respectively. Note that the effective thermal conductivity in porous media is computed as:

$$k_{eff}=sk_l+\left(1-s\right)\left[\varepsilon k_{gas}+\left(1-\varepsilon\right)k_{solid}\right] \tag{32.1}$$

Liq. Diff. in Chan.

is the D_{liq} in Equation 20.39. This parameter appears only when the **Compute Liquid In Channel** option is selected under the **Model** tab.

V_liq/V_gas in Chan.

is the liquid to gas velocity ratio in the channel (χ in Equation 20.40). This parameter appears only when the **Compute Liquid In Channel** option is selected under the **Model** tab.

Other Parameters

Under **Other Parameters**, you can specify the following:

Gas-Disv'ed Phase

is the mass exchange rate constant γ_{gd} that is used to compute the rate of mass change between gas and dissolved phases S_{gd} (Equation 20.26).

Mod. Coef. OSM_drag

is the constant η_{osm} with the default value of 1.0 that is used to generalize the default osmotic drag coefficient n_d (Equation 20.53).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Eq. W. Cont. at a=1

is the equilibrium water content at the water activity of 1, $\lambda_{a=1}$, that is used in computing the equilibrium water content λ_{eq} (Equation 20.28).

Eq. W. Cont. at s=1

is the equilibrium water content at water saturation of 1, $\lambda_{s=1}$, that is used in computing the equilibrium water content λ_{eq} (Equation 20.28). This item appears only if the **Liquid Phase** option is selected under the **Model** tab.

N2 Crossover Factor

is the N_2 crossover scaling factor (α_{N_2} in Equation 20.23 in the *Fluent Theory Guide*, that is used in compute the N_2 crossover flux across the membrane). This item appears only if the **N2 Crossover** option is selected in the **Model** tab (**Options** group).

Desublimation Rate

is the ice phase desublimation rate (γ_{desub} in Equation 20.42 in the *Fluent Theory Guide*. This item appears only if the **Ice Phase** option is selected in the **Model** tab (**Options** group).

Sublimation Rate

is the ice phase sublimation rate (γ_{sub} in Equation 20.42 in the *Fluent Theory Guide*. This item appears only if the **Ice Phase** option is selected in the **Model** tab (**Options** group).

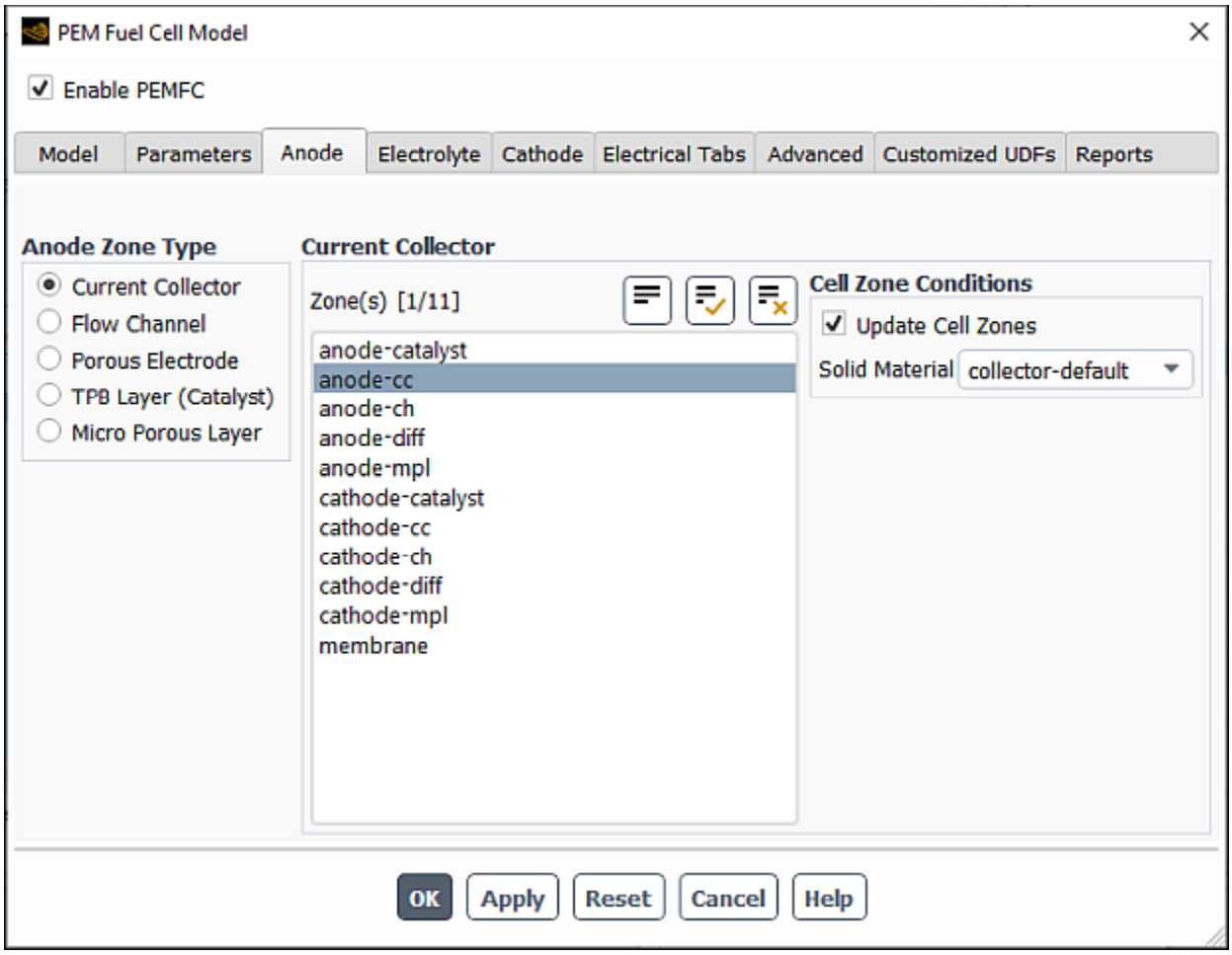
32.1.6.3. Specifying Anode Properties (Anode Tab)

You can use the **Anode** tab of the **PEM Fuel Cell Model** dialog box to specify zones and properties of the current collector, the flow channel, the diffusion layer, the micro porous layer (optional), and the catalyst layer for the anode portion of the fuel cell.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.1.6.3.1. Specifying Current Collector Properties for the Anode

Figure 32.4: The Anode Tab of the PEM Fuel Cell Model Dialog Box with Current Collector Selected



1. Under **Anode Zone Type**, select **Current Collector**.

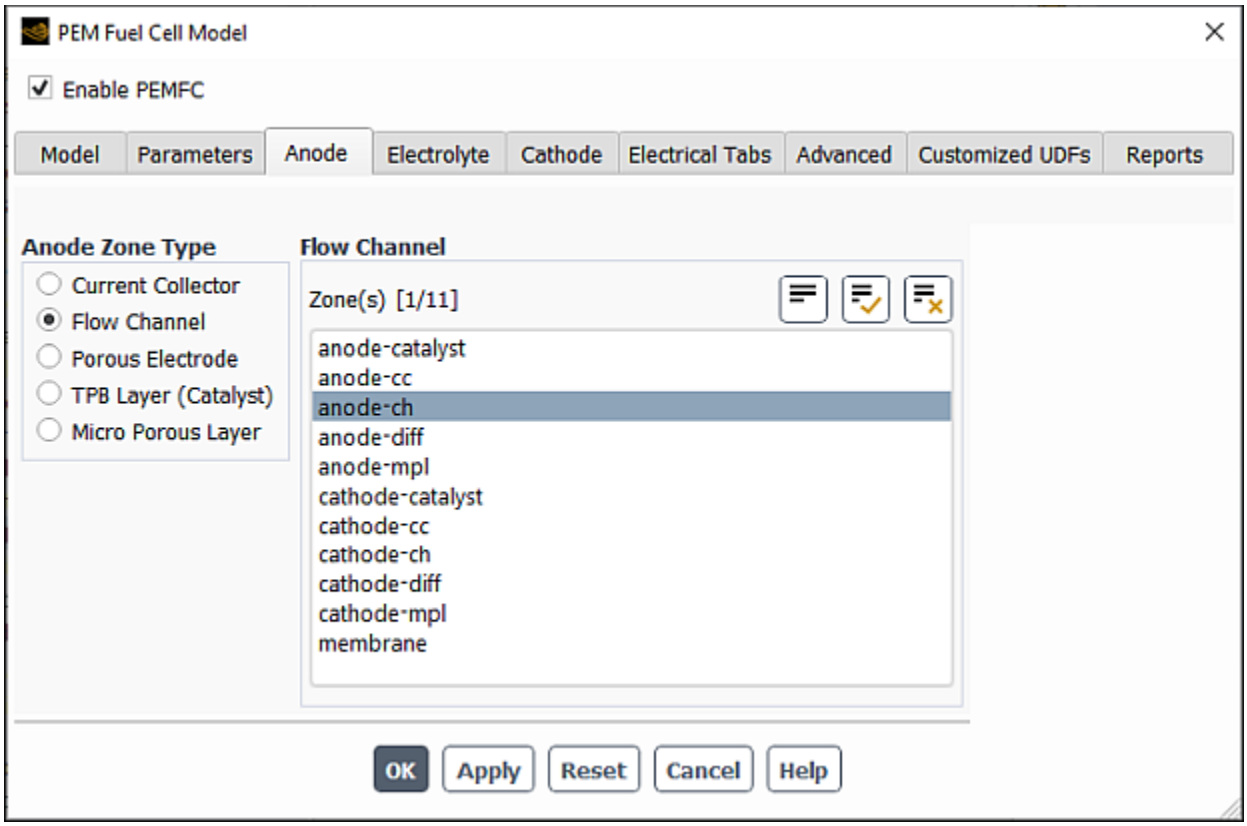
2. From the **Zone(s)** selection list, select a corresponding zone. If you are modeling a fuel cell stack, then you must select *all* zones of a particular type as a group.

3. From the **Solid Material** drop-down list, select the appropriate material. You can use the **Create/Edit Materials** dialog box to customize solid materials. Note that for the **Electrical Conductivity**, you can only select a constant value in the **Create/Edit Materials** dialog box. The solid electrical conductivity value is the diffusivity of the solid phase potential in the solid zones.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.1.6.3.2. Specifying Flow Channel Properties for the Anode

Figure 32.5: The Anode Tab of the PEM Fuel Cell Model Dialog Box with Flow Channel Selected

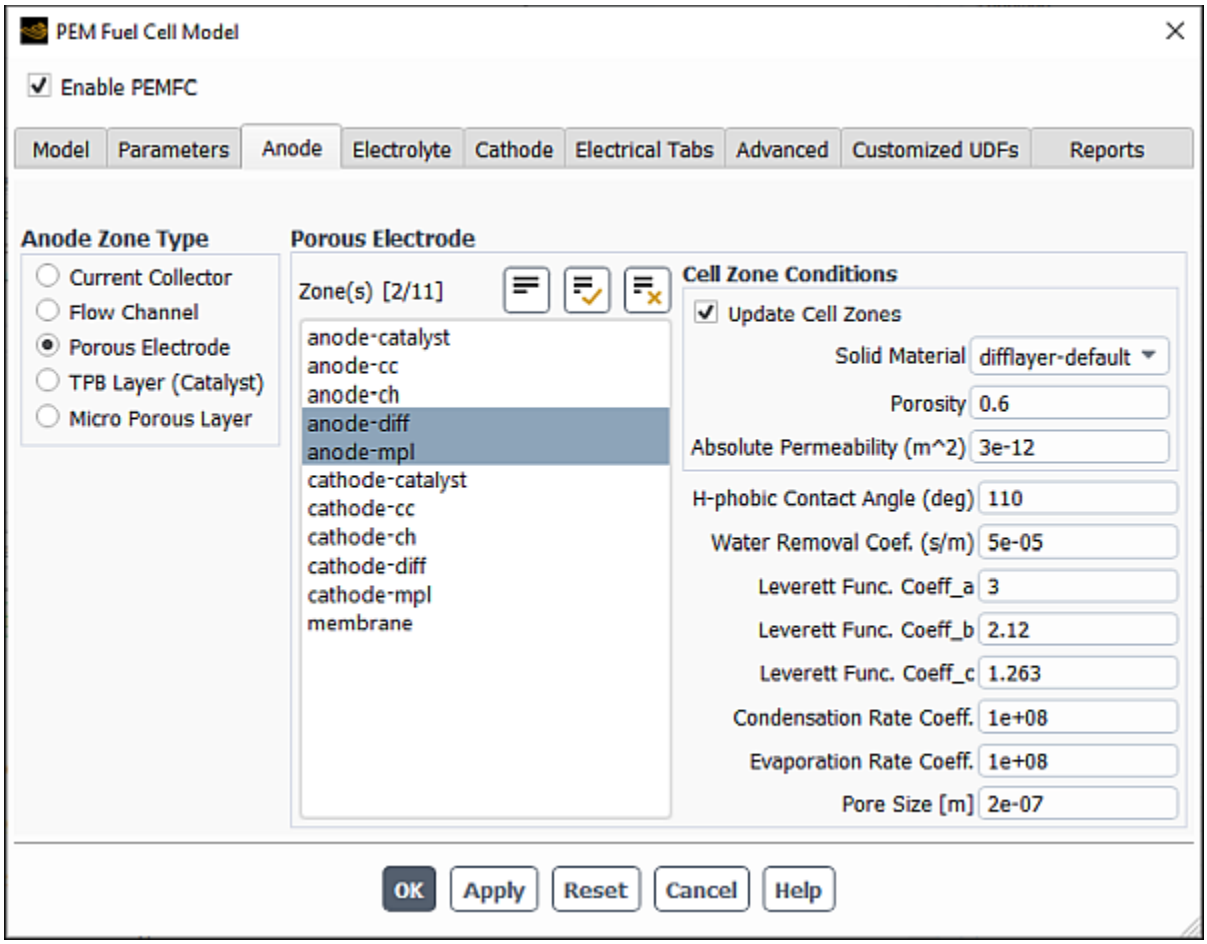


1. Under **Anode Zone Type**, select **Flow Channel**.
2. From the **Zone(s)** selection list, select an appropriate zone.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.1.6.3.3. Specifying Porous Electrode Properties for the Anode

Figure 32.6: The Anode Tab of the PEM Fuel Cell Model Dialog Box with Porous Electrode Selected



1. Under **Anode Zone Type**, select **Porous Electrode**.

2. From the **Zone(s)** selection list, select a corresponding zone. If you are modeling a fuel cell stack, then you must select *all* zones of a particular type as a group.

3. From the **Solid Material** drop-down list, select the appropriate material. You can use the **Create/Edit Materials** dialog box to customize solid materials. Note that for the **Electrical Conductivity**, you can only select a constant value in the **Create/Edit Materials** dialog box. The solid electrical conductivity value is the diffusivity of the solid phase potential in the solid zones.

4. Specify values for the following parameters:

   Porosity

   is ε in Equation 20.45.

   Absolute Permeability

   is K in Equation 20.30.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

H-phobic Contact Angle

is the hydrophobic contact angle θ_c in Equation 20.36.

Water Removal Coef.

is Θ in Equation 20.35.

Leverett Func. Coeff_a

is the Leverett function coefficient a in Equation 20.37 (default = 1.417).

Leverett Func. Coeff_b

is the Leverett function coefficient b in Equation 20.37 (default = 2.12).

Leverett Func. Coeff_c

is the Leverett function coefficient c in Equation 20.37 (default = 1.263).

Condensation Rate Coeff.

is γ_c in Equation 20.34

Evaporation Rate Coeff.

is γ_e in Equation 20.34

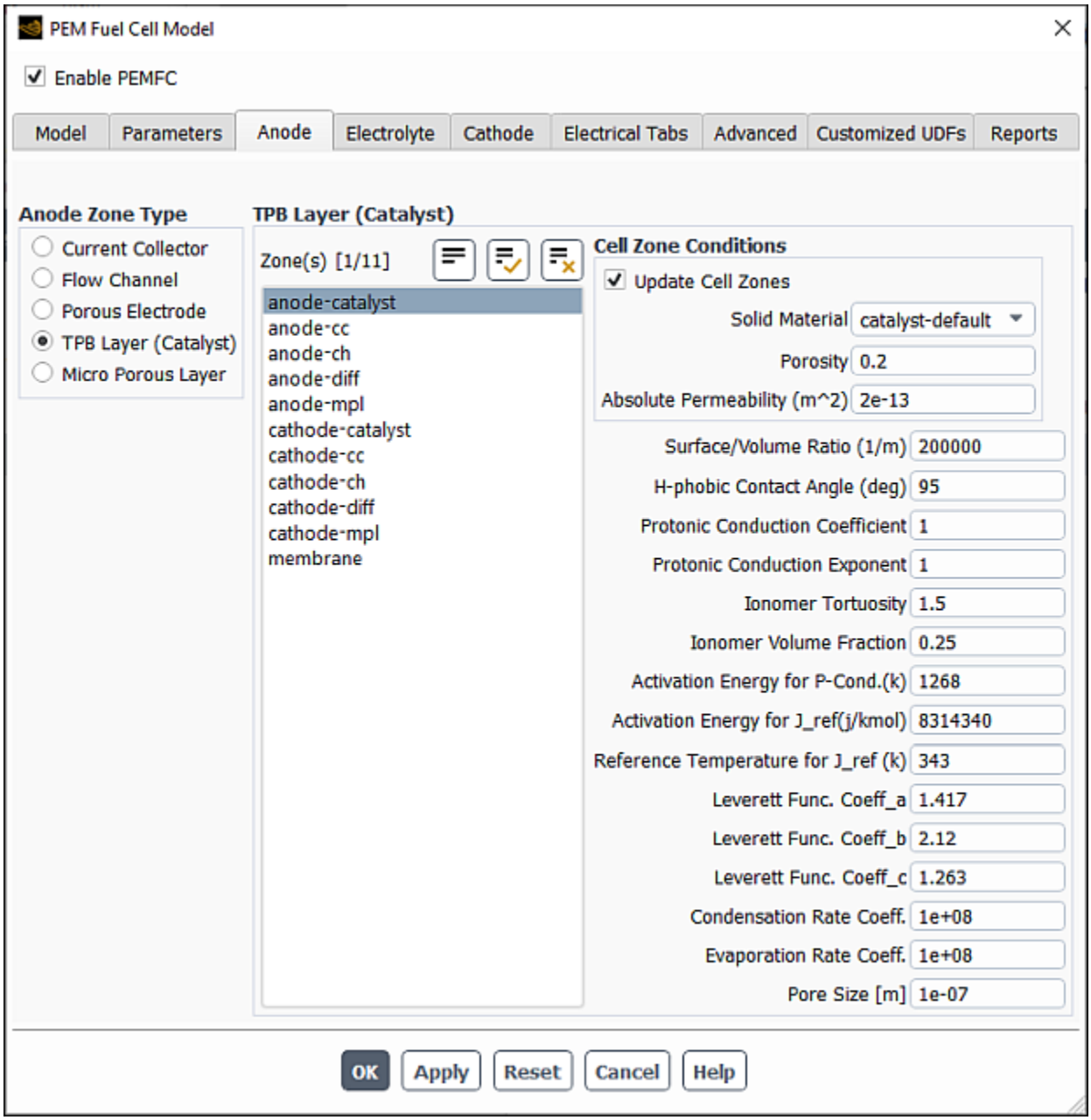
Pore Size

is the average pore size, r_e in Equation 20.105

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.1.6.3.4. Specifying Catalyst Layer Properties for the Anode

Figure 32.7: The Anode Tab of the PEM Fuel Cell Model Dialog Box with TPB Layer (Catalyst) Selected



1. Under **Anode Zone Type**, select **TPB Layer (Catalyst)**.

2. From the **Zone(s)** selection list, select a corresponding zone. If you are modeling a fuel cell stack, then you must select *all* zones of a particular type as a group.

3. From the **Solid Material** drop-down list, select the appropriate material. You can use the **Create/Edit Materials** dialog box to customize solid materials. Note that for the **Electrical Conductivity**, you can only select a constant value in the **Create/Edit Materials** dialog box. The solid electrical conductivity value is the diffusivity of the solid phase potential in the solid zones.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Specify values for the following parameters:

Porosity

is ε in Equation 20.45.

Absolute Permeability

is K in Equation 20.30.

Surface/Volume Ratio

is the specific active surface area ζ in Equation 20.5.

H-phobic Contact Angle

is the hydrophobic contact angle θ_c in Equation 20.36.

Protonic Conduction Coefficient

is the model constant β_{an} in Equation 20.49 (available only when the **Effective P-Conductivity in MEA** option is selected under the **Model** tab).

Protonic Conduction Exponent

is the model constant ω_i in Equation 20.48 (available only when the **Effective P-Conductivity in MEA** option is selected under the **Model** tab).

Ionomer Tortuosity

is the τ_a in Equation 20.49 (available only when the **Effective P-Conductivity in MEA** option is selected under the **Model** tab).

Ionomer Volume Fraction

is the ς_a in Equation 20.49 (available only when the **Effective P-Conductivity in MEA** option is selected under the **Model** tab).

Activation Energy for P-Cond

is the E_i in Equation 20.48 (available only when the **Effective P-Conductivity in MEA** option is selected under the **Model** tab).

Activation Energy for J_ref

is the E_{an} in Equation 20.9 (available only when **Temperature Dependent Jref** is enabled in the **Model** tab (**Options** group box)).

Reference Temperature for J_ref

is the T_{an}^{ref} in Equation 20.9 (available only when **Temperature Dependent Jref** is enabled in the **Model** tab (**Options** group box)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Leverett Func. Coeff_a

is the Leverett function coefficient a in Equation 20.37 (default = 1.417).

Leverett Func. Coeff_b

is the Leverett function coefficient b in Equation 20.37 (default = 2.12).

Leverett Func. Coeff_c

is the Leverett function coefficient c in Equation 20.37(default = 1.263).

Condensation Rate Coeff.

is γ_c in Equation 20.34

Evaporation Rate Coeff.

is γ_e in Equation 20.34

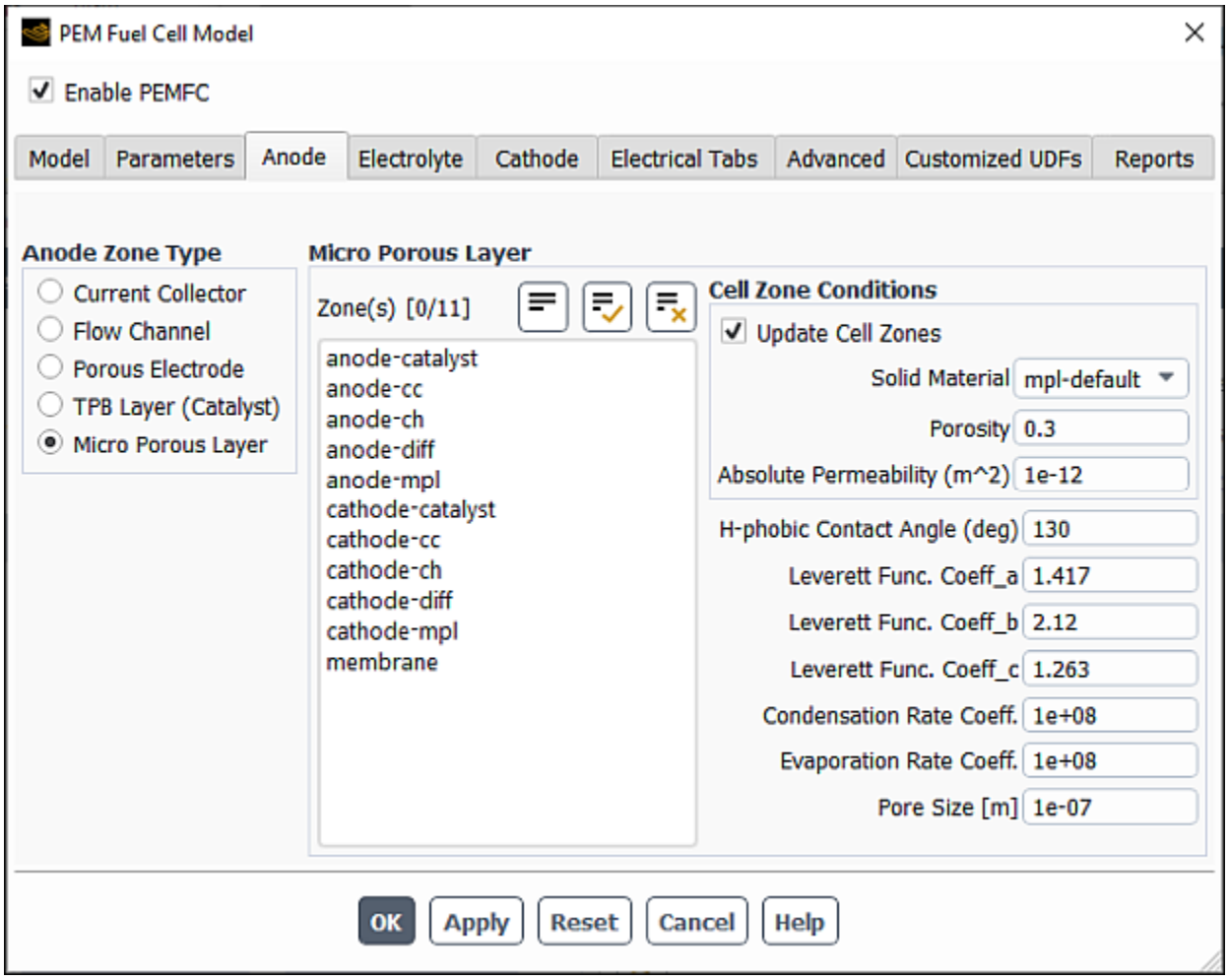
Pore Size

is the average pore size, r_e in Equation 20.105

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.1.6.3.5. Specifying Micro Porous Layer (Optional) Properties for the Anode

Figure 32.8: The Anode Tab of the PEM Fuel Cell Model Dialog Box with Micro Porous Layer Selected



1. Under **Anode Zone Type**, select **Micro Porous Layer**.

2. From the **Zone(s)** selection list, select a corresponding zone. If you are modeling a fuel cell stack, then you must select *all* zones of a particular type as a group.

3. From the **Solid Material** drop-down list, select the appropriate material. You can use the **Create/Edit Materials** dialog box to customize solid materials. Note that for the **Electrical Conductivity**, you can only select a constant value in the **Create/Edit Materials** dialog box. The solid electrical conductivity value is the diffusivity of the solid phase potential in the solid zones.

4. Specify values for the following parameters:

   Porosity

   is ε in Equation 20.45.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Absolute Permeability

is K in Equation 20.30.

H-phobic Contact Angle

hydrophobic contact angle θ_c in Equation 20.36.

Leverett Func. Coeff_a

is the Leverett function coefficient a in Equation 20.37 (default = 1.417).

Leverett Func. Coeff_b

is the Leverett function coefficient b in Equation 20.37 (default = 2.12).

Leverett Func. Coeff_c

is the Leverett function coefficient c in Equation 20.37 (default = 1.263).

Condensation Rate Coeff.

is γ_c in Equation 20.34

Evaporation Rate Coeff.

is γ_e in Equation 20.34

Pore Size

is the average pore size, r_e in Equation 20.105

32.1.6.3.6. Specifying Cell Zone Conditions for the Anode

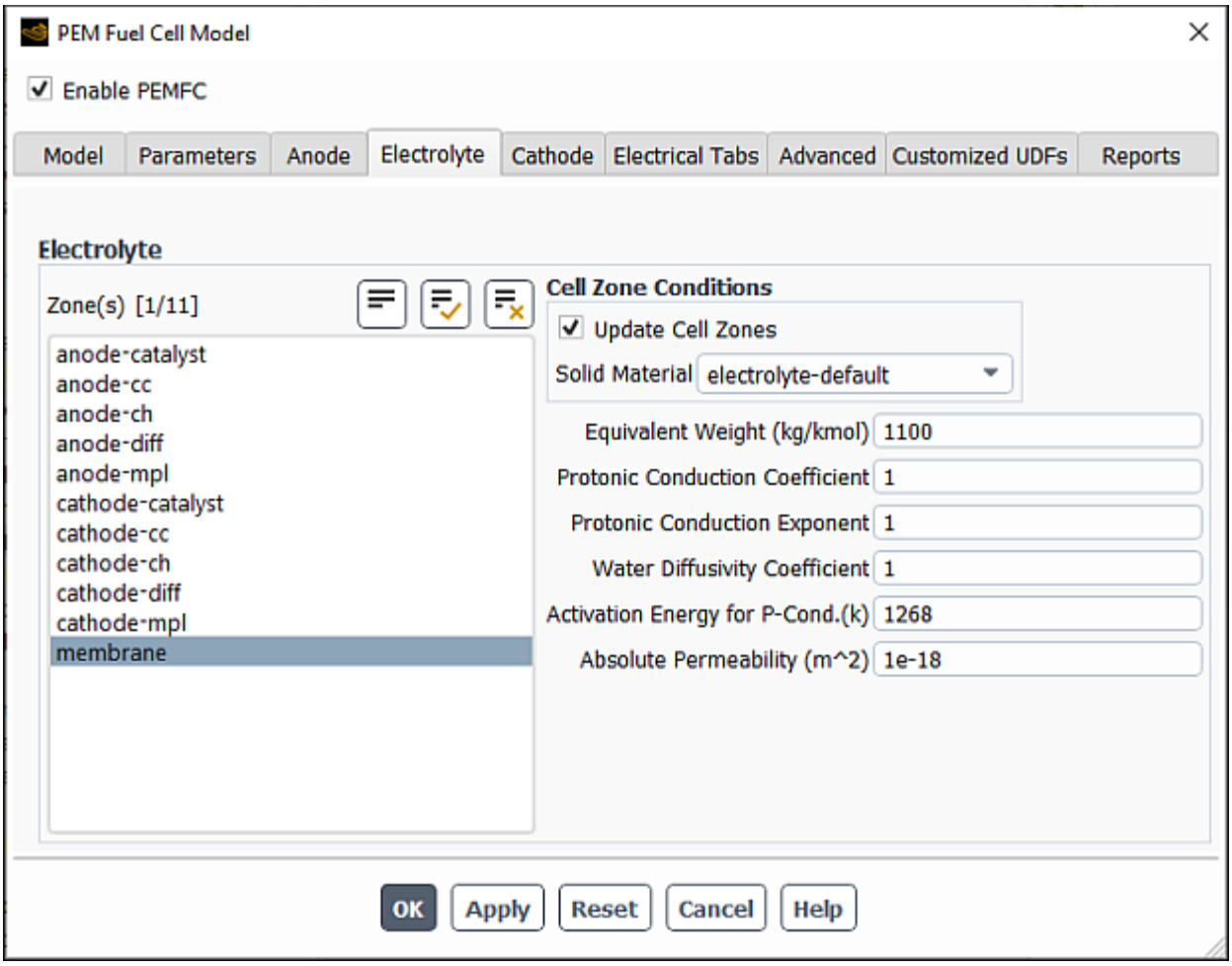
For each case of the anode's current collector, diffusion layer, micro porous layer, and catalyst layer, you assign a solid material and/or set the porosity and the viscous resistance. These settings represent setting a cell zone condition. With the **Update Cell Zones** option turned on (the default setting), this cell zone condition is applied to all selected zones in the **Zone(s)** list. If you want to set the cell zone conditions for each zone individually (using the **Cell Zone Conditions** task page), you should turn off the **Update Cell Zones** option.

32.1.6.4. Specifying Electrolyte/Membrane Properties (Electrolyte Tab)

You can use the **Electrolyte** tab of the **PEM Fuel Cell Model** dialog box to specify zones and properties of the electrolyte/membrane portion of the fuel cell.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 32.9: The Electrolyte Tab of the PEM Fuel Cell Model Dialog Box



1. From the **Zone(s)** selection list, select a corresponding zone. If you are modeling a fuel cell stack, then you must select *all* zones of a particular type as a group.

2. From the **Solid Material** drop-down list, select the appropriate material. You can use the **Create/Edit Materials** dialog box to customize solid materials.

3. Specify values for the following parameters:

   Equivalent Weight

   is the EW in Equation 20.26 and Equation 20.27.

   Protonic Conduction Coefficient

   is the β_{mem} in Equation 20.49.

   Protonic Conduction Exponent

   is the ω_i in Equation 20.48.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Water Diffusivity Coefficient

is the η_λ in Equation 20.50.

Activation Energy for P-Cond

is the E_i in Equation 20.48 (available only when the **Effective P-Conductivity in MEA** option is selected under the **Model** tab).

Absolute Permeability

is the K in Equation 20.30.

Note that the PEMFC model allows you to model the electrolyte/membrane as a solid zone only. However, it still allows for dissolved water, liquid or capillary pressure, and the ionic current to pass through.

32.1.6.4.1. Specifying Cell Zone Conditions for the Membrane

When you assign a solid material to the membrane, you are setting a cell zone condition. With the **Update Cell Zones** option turned on (the default setting), this cell zone condition is applied to all selected zones in the **Zone(s)** list. If you want to set the cell zone conditions for each zone individually (using the **Cell Zone Conditions** task page), you should turn off the **Update Cell Zones** option.

32.1.6.5. Specifying Cathode Properties (Cathode Tab)

You can use the **Cathode** tab of the **PEM Fuel Cell Model** dialog box to specify zones and properties of the current collector, the flow channel, the diffusion layer, the micro porous layer, and the catalyst layer for the cathode portion of the fuel cell.

32.1.6.5.1. Specifying Current Collector Properties for the Cathode

The procedure for specifying the current collector properties for the cathode is similar to that for the anode. For specific steps, refer to Specifying Current Collector Properties for the Anode (p. 3416) for details and then substitute “cathode” for “anode” where appropriate.

32.1.6.5.2. Specifying Flow Channel Properties for the Cathode

The procedure for specifying the flow channel properties for the cathode is similar to that for the anode. For specific steps, refer to Specifying Flow Channel Properties for the Anode (p. 3417) and then substitute “cathode” for “anode” where appropriate.

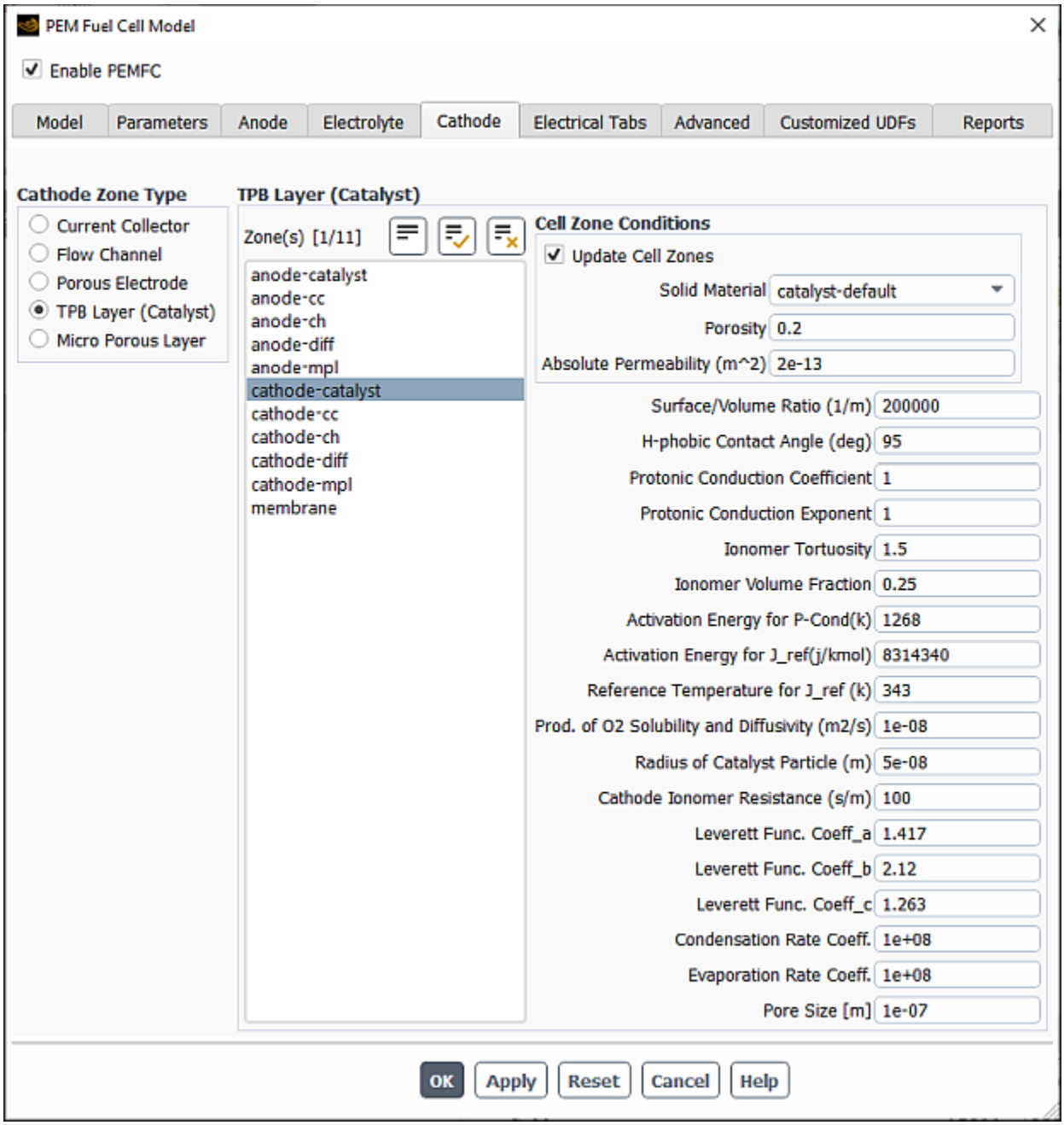
32.1.6.5.3. Specifying Porous Electrode Properties for the Cathode

The procedure for specifying the porous electrode properties for the cathode is similar to that for the anode. For specific steps, refer to Specifying Porous Electrode Properties for the Anode (p. 3418) and then substitute “cathode” for “anode” where appropriate.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.1.6.5.4. Specifying Catalyst Layer Properties for the Cathode

Figure 32.10: The Cathode Tab of the PEM Fuel Cell Model Dialog Box with TPB Layer (Catalyst) Selected



1. Under **Cathode Zone Type**, select **TPB Layer (Catalyst)**.

2. From the **Zone(s)** selection list, select a corresponding zone. If you are modeling a fuel cell stack, then you must select *all* zones of a particular type as a group.

3. From the **Solid Material** drop-down list, select the appropriate material. You can use the **Create/Edit Materials** dialog box to customize solid materials. Note that for the **Electrical Conductivity**, you can only select a constant value in the **Create/Edit Materials** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The solid electrical conductivity value is the diffusivity of the solid phase potential in the solid zones.

4. Specify values for the following parameters:

Porosity

is ε in Equation 20.45.

Absolute Permeability

is K in Equation 20.30.

Surface/Volume Ratio

is the specific active surface area ζ in Equation 20.6.

H-phobic Contact Angle

is the hydrophobic contact angle θ_c in Equation 20.36.

Protonic Conduction Coefficient

is the model constant β_{ca} in Equation 20.49 (available only when the **Effective P-Conductivity in MEA** option is selected under the **Model** tab).

Protonic Conduction Exponent

is the model constant ω_i in Equation 20.48 (available only when the **Effective P-Conductivity in MEA** option is selected under the **Model** tab).

Ionomer Tortuosity

is the τ_c in Equation 20.49 (available only when the **Effective P-Conductivity in MEA** option is selected under the **Model** tab).

Ionomer Volume Fraction

is the ς_c in Equation 20.49 (available only when the **Effective P-Conductivity in MEA** option is selected under the **Model** tab).

Activation Energy for P-Cond

is the E_i in Equation 20.48 (available only when the **Effective P-Conductivity in MEA** option is selected under the **Model** tab).

Activation Energy for J_ref

is the E_{cat} in Equation 20.10 (available only when **Temperature Dependent Jref** is enabled in the **Model** tab (**Options** group box)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reference Temperature for J_ref

is the T_{cat}^{ref} in Equation 20.10 (available only when **Temperature Dependent Jref** is enabled in the **Model** tab (**Options** group box)).

Besides the parameters for the catalyst layer listed above, you can also specify the following parameters for The Cathode Particle Model:

Prod. Of O2 Solubility and Diffusivity

is $K_w D_w$ in Equation 20.16.

Radius of Catalyst Particle

is r_p in Equation 20.16.

Cathode Ionomer Resistance

is $\mathfrak{R}_{ion}$ in Equation 20.15.

Leverett Func. Coeff_a

is the Leverett function coefficient a in Equation 20.37 (default = 1.417).

Leverett Func. Coeff_b

is the Leverett function coefficient b in Equation 20.37 (default = 2.12).

Leverett Func. Coeff_c

is the Leverett function coefficient c in Equation 20.37 (default = 1.263).

Pore Size

is the average pore size, r_e in Equation 20.105

32.1.6.5.5. Specifying Micro Porous Layer (Optional) Properties for the Cathode

The procedure for specifying the catalyst layer properties for the cathode is similar to that for the anode. For specific steps, refer to Specifying Micro Porous Layer (Optional) Properties for the Anode (p. 3423) and then substitute “cathode” for “anode” where appropriate.

32.1.6.5.6. Specifying Cell Zone Conditions for the Cathode

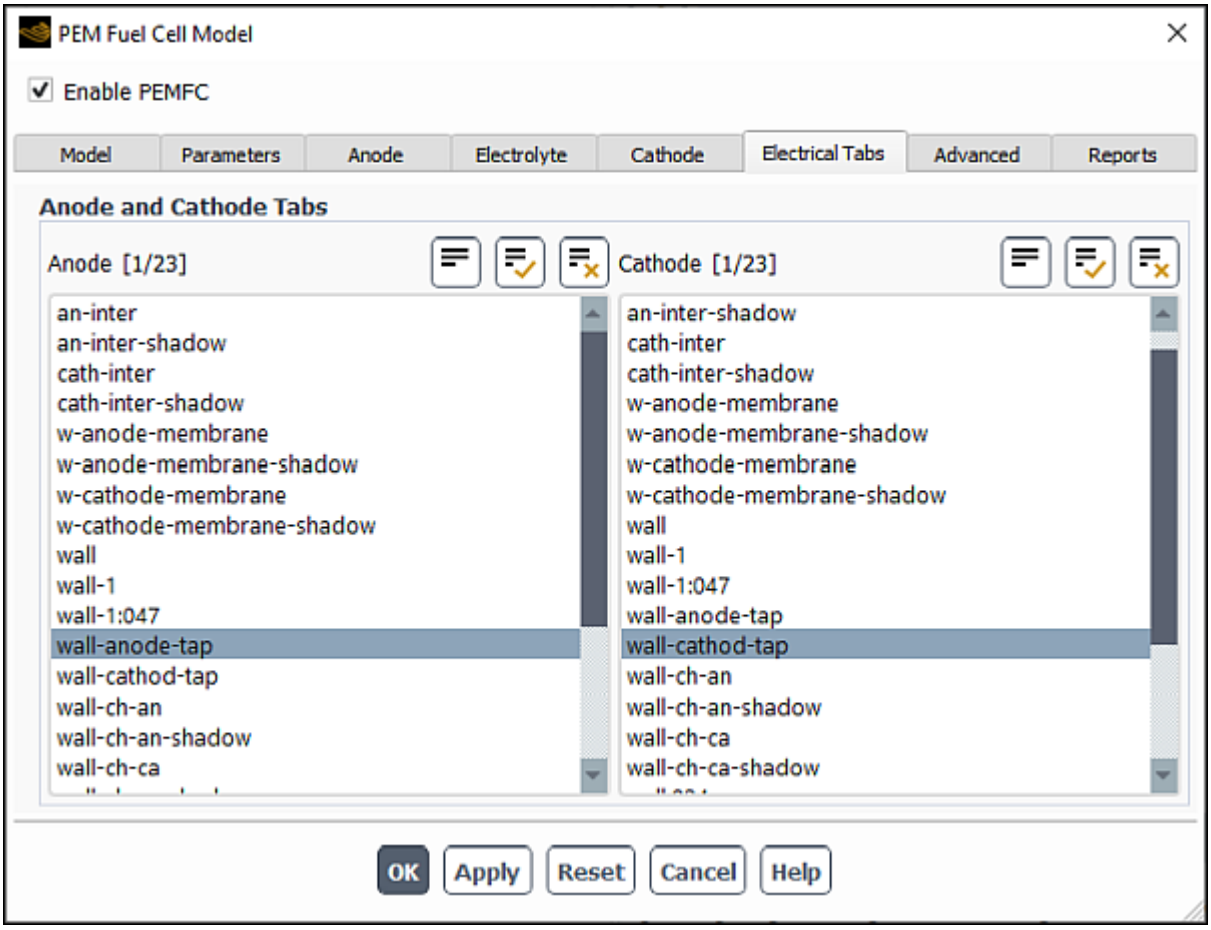
For each case of the cathode’s current collector, diffusion layer, micro porous layer, and catalyst layer, you assign a solid material and/or set the porosity and the viscous resistance. These settings represent setting a cell zone condition. With the **Update Cell Zones** option turned on (the default setting), this cell zone condition is applied to all selected zones in the **Zone(s)** list. If you want to set the cell zone conditions for each zone individually (using the **Cell Zone Conditions** task page), you should turn off the **Update Cell Zones** option.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.1.6.6. Setting the External Electrical Tabs (Electrical Tabs Tab)

You can use the **Electrical Tabs** tab of the **PEM Fuel Cell Model** dialog box to specify **Anode and Cathode Tabs**. The anode and cathode tabs are the external wall surfaces where electrical potential (UDS0) boundary conditions are applied using the method selected in the **Electrical System Setup** group box (**Model** tab).

Figure 32.11: The Electrical Tabs Tab of the PEM Fuel Cell Model Dialog Box



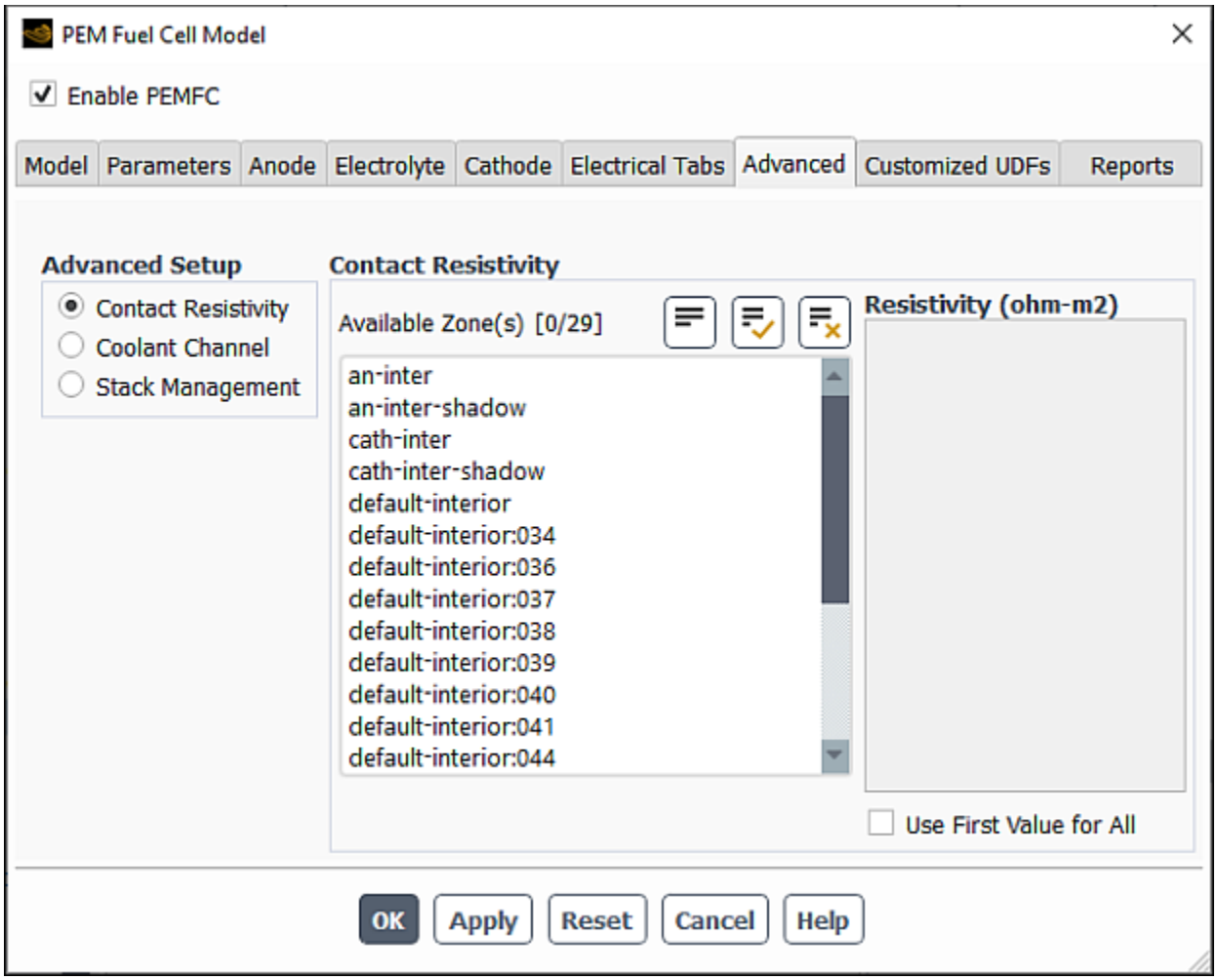
32.1.6.7. Setting Advanced Properties (Advanced Tab)

You can use the **Advanced** tab of the **PEM Fuel Cell Model** dialog box to specify the contact resistivity for any material interface in the geometry, set parameters for coolant channels, and define fuel stack units for managing stacks of fuel cells.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.1.6.7.1. Setting Contact Resistivities for the PEMFC Model

Figure 32.12: The Advanced Tab of the PEM Fuel Cell Model Dialog Box for Contact Resistivities



1. Under **Advanced Setup**, select **Contact Resistivity**.
2. From the **Available Zone(s)** selection list, select any number of corresponding interfaces. These zones are face zones over which a jump in electrical potential is caused by imperfect conduction.

   Note:

   The contact resistance will be applied only to `wall` and `porous jump` types of interfaces.

3. Specify a value for the **Resistivity** for each specified zone.

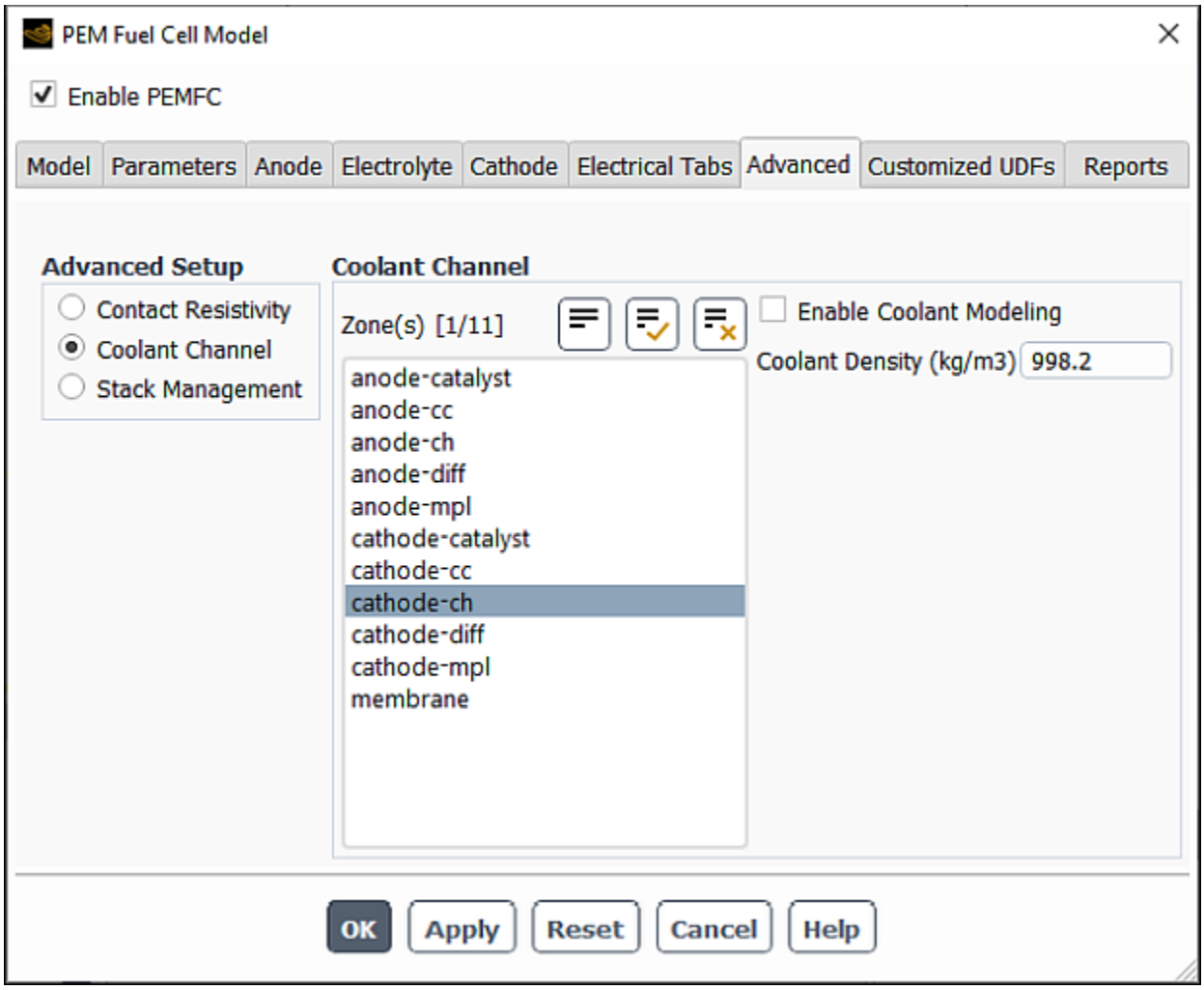
   Note that at porous jump and wall/wall-shadow interfaces, the contact resistivity is applied at each face. Therefore, you need to specify the resistivity only at one of the interface walls, otherwise the actual resistance will be doubled at the interface.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. To simplify the input, you can choose to use the resistivity value of the first selected zone for all others as well by selecting the **Use First Value for All** option.

32.1.6.7.2. Setting Coolant Channel Properties for the PEMFC Model (Optional)

Figure 32.13: The Advanced Tab of the PEM Fuel Cell Model Dialog Box for the Coolant Channel



1. Under **Advanced Setup**, select **Coolant Channel**.
2. From the **Zone(s)** selection list, select any number of corresponding zones.
3. Specify a value for the **Coolant Density**.
4. To enable the coolant channel, select the **Enable Coolant Modeling** option. Amongst other settings, this will automatically create a new mixture material `pem+cool-mixture` consisting of `hydrogen`, `oxygen`, `water-vapor`, `coolant-liquid`, and `nitrogen`.

   The coolant species mixture material exists only in the coolant channel where its mass fraction is 1.0 and will have no effect on the gas phase in all other fluid zones. This is reflected in the mixture composition and in the density specification method for the mixture. All other mixture properties in the gas channels are automatically set to **ideal-gas-mixing-law**.

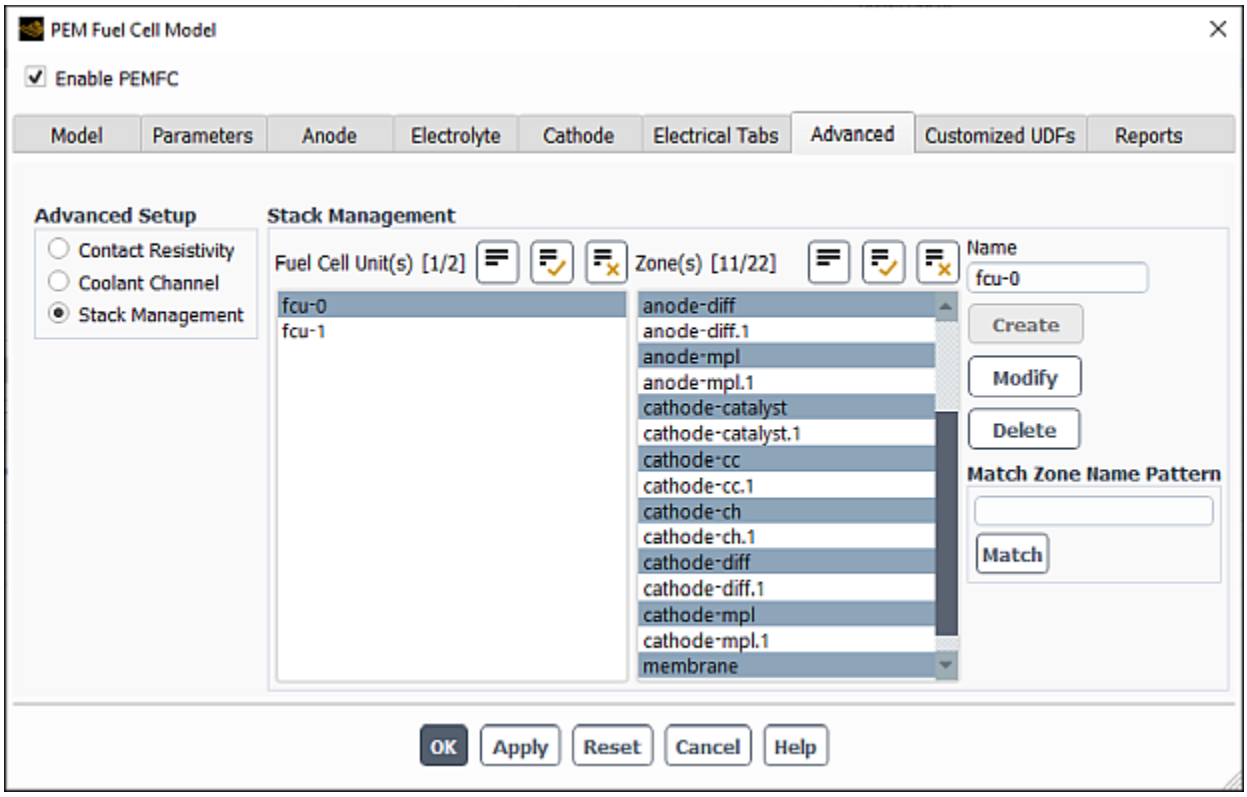
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Using the **Create/Edit Materials** dialog box, set all other material properties for the new fluid `coolant-liquid`.

6. Specify the inlet and outlet conditions for cooling channels using standard boundary conditions dialog boxes.

32.1.6.7.3. Managing Stacks for the PEMFC Model

Figure 32.14: The Advanced Tab of the PEM Fuel Cell Model Dialog Box for Stack Management



The Ansys Fluent PEMFC model allows you to model fuel cell stacks as well as individual fuel cells. In the **Advanced** tab of the **PEM Fuel Cell Model** dialog box, you can define *fuel cell units* for each fuel cell in a stack. A fuel cell unit consists of all zones of a single fuel cell in the stack.

Important:

If you are only modeling a single fuel cell, then you do not need to set anything for **Stack Management** in the **Advanced** tab of the **PEM Fuel Cell Model** dialog box.

---

1. Select the **Advanced** tab of the **PEM Fuel Cell Model** dialog box.

2. Select **Stack Management** under **Advanced Setup**.

3. Since a fuel cell unit consists of all zones of a single fuel cell in the stack, select the corresponding zones from the **Zone(s)** list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Create a new fuel cell unit by clicking the **Create** button. The new fuel cell is listed under **Fuel Cell Unit(s)** with a default name.

5. Edit a pre-existing fuel cell unit by selecting it in the **Fuel Cell Unit(s)** list. The zones in this fuel cell unit are automatically selected in the **Zone(s)** list. You can then modify the zones that make up the fuel cell unit and/or change its name in the **Name** field and click **Modify** to save the new settings.

6. Remove a pre-existing fuel cell unit by selecting it in the **Fuel Cell Unit(s)** list and clicking the **Delete** button.

   Note:

   When deleting a unit, make sure that the electrical connectivity remains intact.

7. If your model contains many zone names, you can use the **Match Zone Name Pattern** field to specify a pattern to look for in the names of zones. Type the pattern in the text field and click **Match** to select (or deselect) the zones in the **Zones** list with names that match the specified pattern. You can match additional characters using * and ?. For example, if you specify `wall*`, all surfaces whose names begin with **wall** (for example, **wall-1**, **wall-top**) will be selected automatically. If they are all selected already, they will be deselected. If you specify `wall?`, all surfaces whose names consist of **wall** followed by a single character will be selected (or deselected, if they are all selected already).

   For example, in a stack there are many fuel cells, say 10–100, each having at least 9 zones (current collector, gas channel, diffusion layer, and catalyst layer for both anode and cathode and a membrane). Additionally, there may be coolant channels, and it may be that for mesh construction reasons each of these physical zones is made up of more than one mesh zone. Even for small stacks, you can easily end up having hundreds of cell zones in an Ansys Fluent mesh. Therefore, you may want to consider numbering the fuel cells in a stack and to use the assigned fuel cell number in the names of the mesh zones. When you set up your stacked fuel cell case, you would use the **Match Zone Name Pattern** field to pick all the zones belonging to a single fuel cell in the stack, rather than scrolling through the potentially very long list and selecting them manually.

32.1.6.8. Customizing the PEM Fuel Cell Module

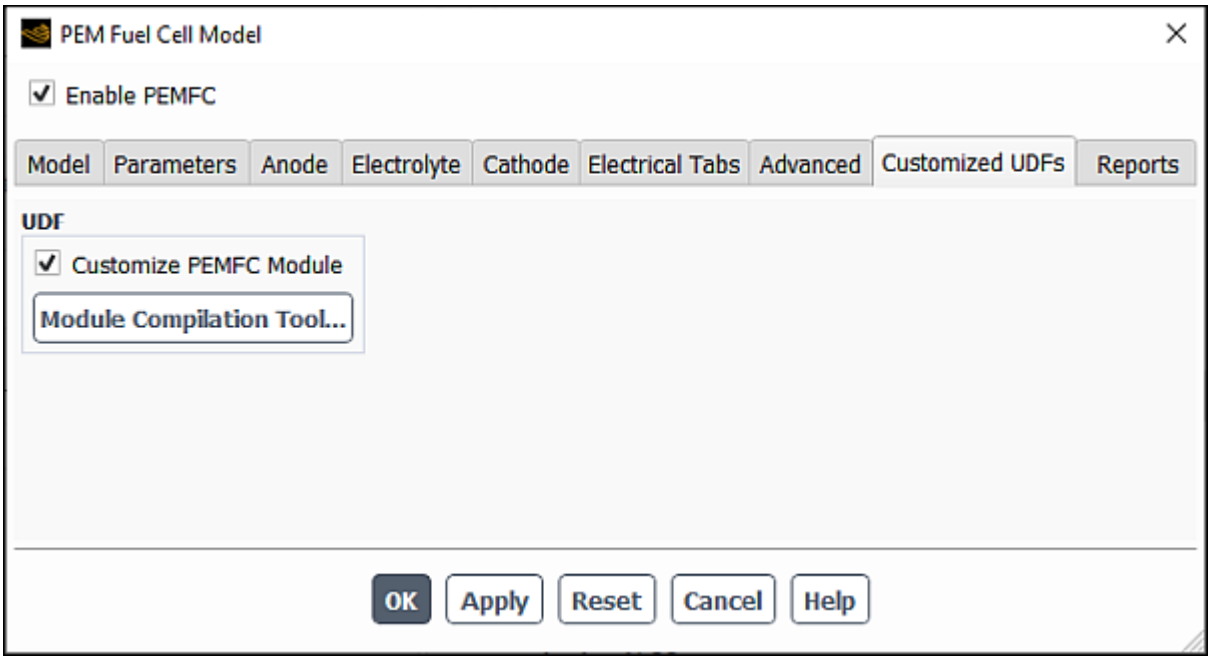
In the **Customized UDFs** tab, you can compile your user-defined function (UDF) in order to customize the PEM Fuel Cell Module. For information about compilers that can be used, see Compilers in the *Fluent Customization Manual*.

To compile your UDF:

1. In the **UDF** group box, enable **Customize PEMFC Module**.

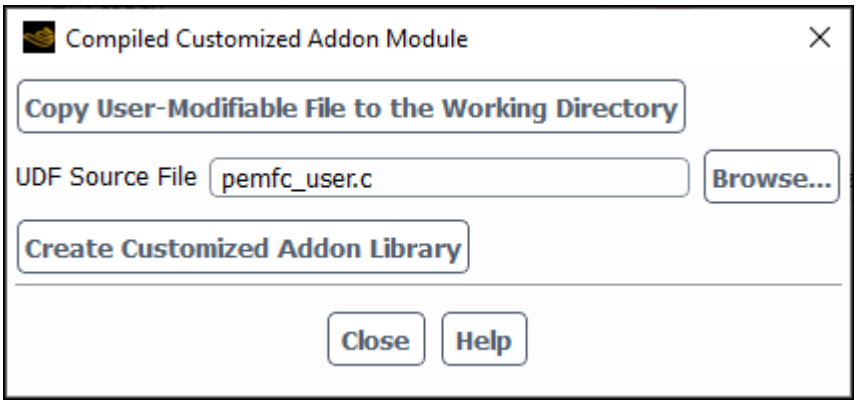
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 32.15: The Customized UDFs Tab in the PEM Fuel Cell Dialog Box



2. Click **Module Compilation Tool...**

Figure 32.16: The Compiled Customized Addon Module Dialog Box



3. In the **Compiled Customized Addon Module** dialog box that opens, do one of the following:

   - If you want to create your UDF from a source file provided by Ansys Fluent:

     a. Click **Copy User-Modifiable File to the Working Directory** to automatically copy the source code file from the Ansys Fluent installation directory to your working directory.

        Note that depending on the fuel cell module you are using, the copied file will be different:
        – `pemfc_user.c` for the PEMFC model
        – `pem_user.c` for the Fuel Cell and Electrolysis models
        – `constit.c` for the SOFC model

     b. Modify the copied source code file as desired.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you want to use your existing source file:

  a. Click **Browse...** next to **UDF Source File** to open the **Select File** dialog box.

  b. Navigate to the desired directory and select the source file.

4. Click **Create Customized Addon Library**.

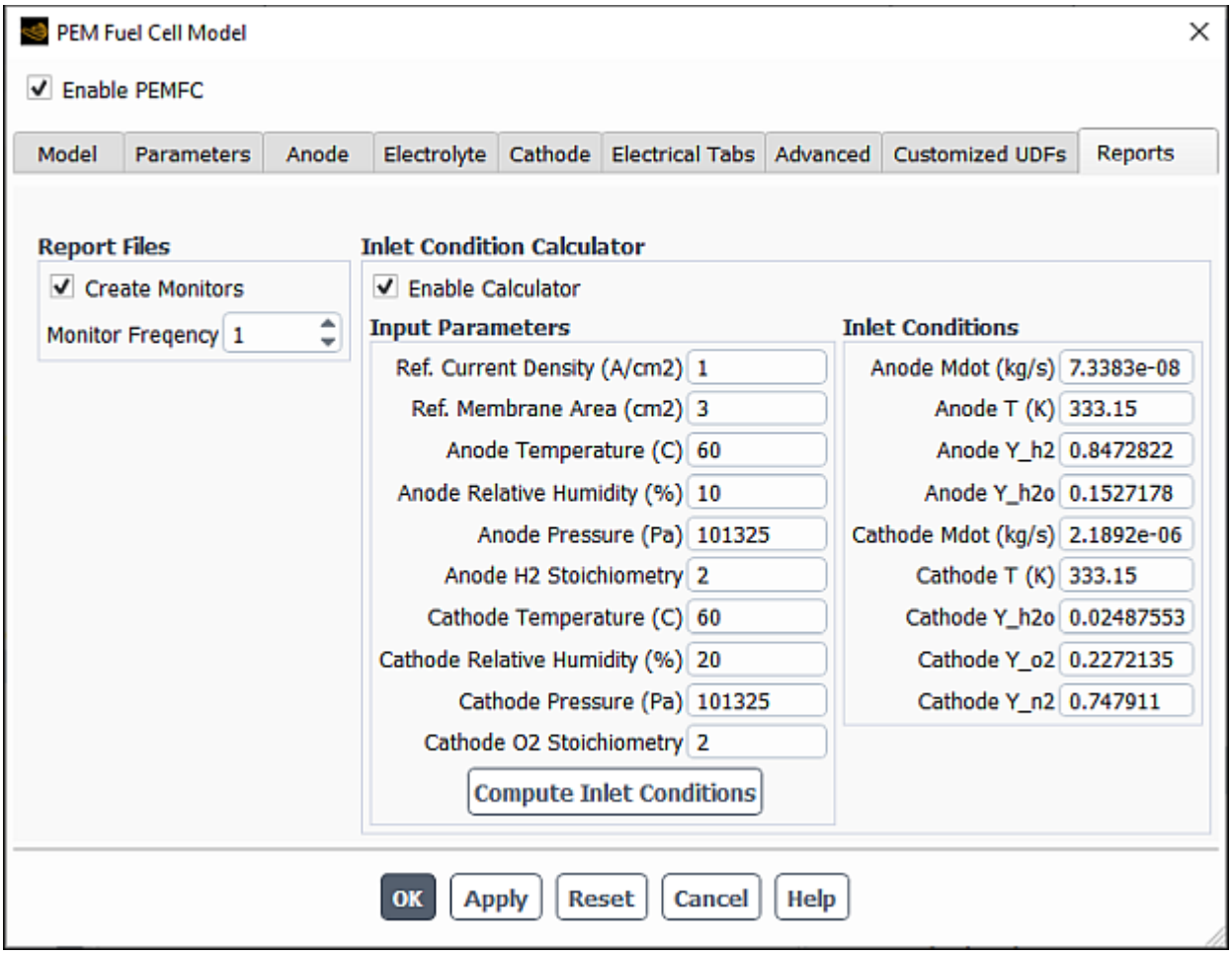
5. Click **Close**.

32.1.6.9. Reporting on the Solution (Reports Tab)

You can use the **Reports** tab of the **PEM Fuel Cell Model** dialog box to:

- set up parameters for reporting data relevant to the fuel cell
- use the calculator to compute the inlet boundary conditions that you can then enter in corresponding boundary conditions dialog boxes.

Figure 32.17: The Reports Tab of the PEM Fuel Cell Model Dialog Box



When you enable **Create Monitors**, then during the solution run, Ansys Fluent will write the following output files to monitor cell voltage, mean and minimum values of H_2 mole fractions in anode fluid zones, mean and minimum values of O_2 mole fractions in cathode fluid zones, and total current generated in anode and cathode catalyst zones:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- `cell-voltage.out`
- `mean_h2.out`
- `min_h2.out`
- `mean_o2.out`
- `min_o2.out`
- `total-current-anode.out`
- `total-current-cathode.out`

You can specify the **Monitor Frequency** at which the monitoring data will be written during the calculation. For example, if you enter 10 in the **Monitor Frequency** text box, the output files will be updated after every 10 iterations.

If you want to use the inlet condition calculator, follow the below steps:

1. Select **Enable Calculator** in the **Inlet Condition Calculator** group box.
2. Enter the following **Input Parameters**:
   - **Ref. Current Density**: the current density that is used as a reference or target
   - **Ref. Membrane Area**: the membrane area that is used to compute the total current
   - **Anode Temperature**: the temperature at anode flow inlet (Celsius)
   - **Anode Relative Humidity**: the relative humidity at anode flow inlet as a percentage value
   - **Anode Pressure**: the system pressure in the anode flow channel (Pascal)
   - **Anode H2 Stoichiometry**: the hydrogen stoichiometry based upon the total current
   - **Cathode Temperature**: the temperature at cathode flow inlet (Celsius)
   - **Cathode Relative Humidity**: the relative humidity at cathode flow inlet as a percentage value
   - **Cathode Pressure**: the system pressure in the cathode flow channel (Pascal)
   - **Cathode O2 Stoichiometry**: the oxygen stoichiometry based upon the total current
3. Click **Compute Inlet Conditions**.

   The resulting parameters are displayed in the **Inlet Conditions** group box:
   - **Anode Mdot**: the total mass flow rate at anode inlet(s) (kg/s)
   - **Anode T**: the temperature at anode flow inlet(s) (Kelvin)
   - **Anode Y_h2**: the mass fraction of hydrogen at anode flow inlet(s)
   - **Anode Y_h2o**: the mass fraction of water vapor at anode flow inlet(s)
   - **Cathode Mdot**: the total mass flow rate at cathode inlet(s) (kg/s)
   - **Anode T**: the temperature at cathode flow inlet(s) (Kelvin)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Anode Y_h2o**: the mass fraction of water vapor at cathode flow inlet(s)
- **Anode Y_o2:** the mass fraction of oxygen at cathode flow inlet(s)
- **Anode Y_n2**: the mass fraction of nitrogen at cathode flow inlet(s)

Note:

- The inlet condition calculator is used only to help you convert your input parameters to parameters that are recognized by Ansys Fluent. You still need to enter these values into relevant boundary conditions dialog boxes.
- The converted values are based upon the total reference current of the cell or stack you specified. If multiple inlets are present in a cell or stack, you should exercise care when using the computed inlet conditions. For example, if you have two identical anode inlets, you need to split the computed total mass flow rates equally at each inlet.

32.1.6.10. User-Defined Functions Hooked to the PEMFC Module

Once you click the **OK** or **Apply** button in the **PEM Fuel Cell Model** dialog box, Ansys Fluent automatically hooks several user-defined functions (UDFs) to the PEMFC solver. The UDFs either are hooked to corresponding PEMFC model parameters or perform specific operations, as shown in Table 32.1: User-Defined Functions for the PEMFC Module (p. 3438).

Note:

If `DEFINE_SOURCE` user defined functions are not hooked consistently with the PEMFC model settings, Ansys Fluent issues a warning message in the console. You can restore the default source term settings by closing and reopening the **PEM Fuel Cell Model** dialog box and then clicking **OK** or **Apply**.

Table 32.1: User-Defined Functions for the PEMFC Module

UDF Name	UDF Type	Description/Operation
density	DEFINE_PROPERTY	Density property for the **pem+cool-mixture** material. This UDF is used only if **Enable Coolant Modeling** is selected in the **Advanced** tab.
diff	DEFINE_DIFFUSIVITY	Mass diffusivity for the mixtures
cond	DEFINE_DIFFUSIVITY	UDS diffusivity for the mixtures
uds_flux	DEFINE_UDS_FLUX	Flux function for user-defined scalars (UDSs)
source_s	DEFINE_SOURCE	Source terms in the catalyst zones for electric potential (UDS-0)
source_m	DEFINE_SOURCE	Source terms in the catalyst zones for protonic potential (UDS-1)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UDF Name	UDF Type	Description/Operation
source_h2	DEFINE_SOURCE	Source terms in the anode catalyst zones for species H_2
source_o2	DEFINE_SOURCE	Source terms in the cathode catalyst zones for species O_2
source_h2o	DEFINE_SOURCE	Source terms in all fluid zones for species H_2O
source_mass	DEFINE_SOURCE	Source terms in all fluid zones for mass
source_lambda	DEFINE_SOURCE	Source terms in the catalyst layers and membrane for the water content
source_energy	DEFINE_SOURCE	Source terms in the fluid zones for the energy equation
source_liquid	DEFINE_SOURCE	Source terms in the porous layers and catalyst layers for capillary pressure (UDS-2)
source_chan_liq	DEFINE_SOURCE	Source terms in the flow channels for liquid saturation in the channel equation
source_momentum	DEFINE_SOURCE	Source terms in the flow channels for the momentum equations
initialize	DEFINE_INIT	Initializes solution
adjust	DEFINE_ADJUST	Adjust function (executed at every iteration)
unsteady	DEFINE_UDS_UNSTEADY	Unsteady function for UDSs (transient simulations only)
write_sec_hdf	DEFINE_RW_HDF_FILE	-
read_sec_hdf	DEFINE_RW_HDF_FILE	-
pem_lib_load	DEFINE_EXECUTE_ON_LOADING	-
exec_at_end	DEFINE_EXECUTE_AT_END	-
set_new_stack_voltage	DEFINE_ON_DEMAND	-
set_new_stack_current	DEFINE_ON_DEMAND	-
get_old_stack_voltage	DEFINE_ON_DEMAND	-
get_old_stack_current	DEFINE_ON_DEMAND	-
list_pemfc_udf	DEFINE_ON_DEMAND	Lists all used UDSs and UDMs
report_current	DEFINE_ON_DEMAND	Reports the total current
aniso_econd	DEFINE_ANISOTROPIC_DIFFUSIVITY	UDS diffusivity for electric potential. This UDF is used only when **Anisotropic E-cond. in GDL/MPL** is enabled in the **Model** tab.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.1.7. PEMFC Model Boundary Conditions

The following boundary conditions need to be defined for the PEMFC simulation based on your problem specification:

- Anode Inlet
  - Mass flow rate
  - Temperature
  - Direction specification method
  - Mass fractions (for example, **h2**, and **h2o**)
  - The coolant *must* be set to zero if coolant channels are enabled
  - UDS-4 (Water Saturation in Channels) must be set to zero
- Cathode Inlet
  - Mass flow rate
  - Temperature
  - Direction specification method
  - Mass fractions (for example **o2**, **h2o**, and **n2**)
  - The coolant *must* be set to zero if coolant channels are enabled
  - UDS-4 (Water Saturation in Channels) must be set to zero
- Coolant Inlet (if any)
  - Mass flow rate
  - Temperature
  - Direction specification method
  - Coolant mass fraction set to 1
  - UDS-4 (Water Saturation in Channels) must be set to zero
- Pressure Outlets (all)

  Realistic backflow conditions.
- Terminal Anode
  - Temperature (or heat flux if known)
  - (when **Set Up in B.C. Panels** is selected as the **Electrical System Setup** method in the **Model** tab) Electric potential

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the **UDS** tab of the **Wall** dialog box for the anode tab, set **Electric Potential** (UDS-0) to ground voltage (zero). Note that for the anode tab, the voltage must be anchored to zero, whether you want to specify the total voltage or current density condition for the cathode tab.

- Terminal Cathode
  - Temperature (or heat flux if known)
  - (when **Set Up in B.C. Panels** is selected as the **Electrical System Setup** method in the **Model** tab) Electric potential

    In the **UDS** tab of the **Wall** dialog box for the cathode tab, set **Electric Potential** (UDS-0) to:

    → voltage (V) of the cathode when using the **Specified Value** option, or

    → current density (A/m^2) when using the **Specified Flux** option.

    Note that the sign of the UDS-0 flux on the cathode side should be negative since the current (positive charge) is leaving the cathode electrical terminal.

32.1.8. Solution Guidelines for the PEMFC Model

For potentiostatic boundary conditions, after initialization, solutions are calculated easily for cell voltages close to the open-circuit voltage. The same can be said for galvanostatic boundary conditions and low electric current. By lowering the cell voltage or by raising the average electric current, you can calculate subsequent stationary solutions.

In the event of convergence problems, it is recommended that you change the multigrid cycle to **F-cycle** with **BCGSTAB** (bi-conjugate gradient stabilized method) selected as the stabilization method for the species and the two potential equations. For the species and the user-defined scalar equations, it may be necessary to reduce the termination (criteria) of the multigrid-cycles to 1×10^{-3}. For stack simulations, the termination criterion may be reduced to $\times10^{-7}$ for the two potential equations.

Also, it may be useful to turn off **Joule Heating** and **Reaction Heating** in the **PEM Fuel Cell Model** dialog box (in the **Model** tab) for the first few (approximately 5-10) iterations after initialization. This allows the two electric potentials to adjust from their initial values to more physical values, avoiding the possibility of extreme electrochemical reactions and electric currents that would in turn adversely impact the solution.

32.1.9. Postprocessing the PEMFC Model

You can perform postprocessing using standard Ansys Fluent quantities and by using user-defined scalars and user-defined memory allocations. By default, the Ansys Fluent PEMFC model defines several user-defined scalars and user-defined memory allocations, described in Table 32.2: User-Defined Scalar Allocations (p. 3441) and Table 32.3: User-Defined Memory Allocations (p. 3442).

Table 32.2: User-Defined Scalar Allocations

UDS Name	Description
UDS 0	Electric Potential (solid phase potential) (Volts)
UDS 1	Protonic Potential (membrane phase potential) (Volts)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UDS 2	Capillary Pressure
UDS 3	Water Content
UDS 4	Liquid Saturation in Channels

Table 32.3: User-Defined Memory Allocations

UDM Name	Description
UDM 0	X Current Flux Density (A/m^2)
UDM 1	Y Current Flux Density (A/m^2)
UDM 2	Z Current Flux Density (A/m^2)
UDM 3	Current Flux Density Magnitude (A/m^2)
UDM 4	Ohmic Heat Source (W/m^3)
UDM 5	Reaction Heat Source (W/m^3)
UDM 6	Overpotential (Volts)
UDM 7	Evaporation to Vapor Phase Change Source (kg/m^3-s) (S_{gl} in Equation 20.30 in the *Fluent Theory Guide*)
UDM 8	Osmotic Drag Coefficient
UDM 9	Water Activity
UDM 10	Equilibrium Water Content
UDM 11	Absorption from Vapor phase change source (kg/m^3-s) (S_{gd} in Equation 20.25 in the *Fluent Theory Guide*)
UDM 12	Absorption from Liquid Phase Change Source (kg/m^3-s) (S_{ld} in Equation 20.25 in the *Fluent Theory Guide*)
UDM 13	Transfer Current (A/m^3)
UDM 14	Liquid Saturation in Porous Media
UDM 15	GDL Liquid Removal (kg/m^3-s)
UDM 16	Osmotic Drag Source (kg/m^3-s)
UDM 17	Capillary Pressure (Pa)
UDM 18	Pressure Gradient Source (kg/m^3-s)

You can obtain this list by opening the **Execute On Demand** dialog box and pulling down the **Function** drop-down list.

User Defined → **User Defined** → **Execute on Demand...**

and access the execute-on-demand function called `list_pemfc_udf`.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Alternatively, you can view the listing that appears when you first load your PEMFC case, or you can type `list_pemfc_udf` in the text user interface and the listing will appear in the console window.

Note:

- For field variables that are stored in UDM, use the corresponding variables for postprocessing. Postprocessing the UDM itself is not recommended.
- For reviewing simulation results in CFD-Post for cases involving UDM or UDS, export the solution data as CFD-Post-compatible file (`.cdat`) in Fluent and then load this file into CFD-Post.

Important:

When you load older PEM Fuel Cell cases into Ansys Fluent, and you are monitoring a UDS using volume or surface monitors, make sure you re-visit the corresponding monitors dialog box (for example, the **Volume Monitor** or the **Surface Monitor** dialog box) to verify that the correct UDS name is used for the appropriate monitor.

In addition, you can generate graphical plots or alphanumeric reports of the following variable (available under the **Species...** category):

N2 crossover

is the mass transfer rate of N_2 due to the crossover effect (J_{N_2} in Equation 20.22 in the *Fluent Theory Guide*). This item is available only when the **N2 Crossover** option is selected in the **Model** tab (**Options** group).

Ice formation rate

is the formation rate of the ice phase (S_{ice} in Equation 20.42 in the *Fluent Theory Guide*). This item is available only turbulent simulations when the **Ice Phase** option is selected in the **Model** tab (**Options** group).

Ice formation rate

is the ice volume fraction (s_{ice} in Equation 20.41 in the *Fluent Theory Guide*). This item is available only turbulent simulations when the **Ice Phase** option is selected in the **Model** tab (**Options** group).

32.1.10. User-Accessible Functions

As noted in Properties, you can directly incorporate your own formulations and data for the properties of the fuel cell membrane using the `pemfc_user.c` source code file.

The following listing represents a description of the contents of the `pemfc_user.c` source code file:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

real Get_P_sat(real T, cell_t c, Thread *t)

Returns the value of the water vapor saturation pressure as a function of temperature (Equation 20.54) or other user-specified values.

real Water_Activity(real P, real T, cell_t c, Thread *t)

Returns the value of water activity (Equation 20.29).

real Osmotic_Drag_Coefficient(cell_t c, Thread *t)

Returns the value of the osmotic drag coefficient (Equation 20.53).

real Membrane_Conductivity(real lam, cell_t c, Thread *t)

Returns the value of the protonic conductivity in MEA (Equation 20.48).

real Water_Content_Diffusivity(real lam, real T, real mem_mol_density,cell_tc,Thread*t)

Returns the value of the water content diffusivity in the MEA (Equation 20.50).

real Gas_Diffusivity(cell_t c, Thread *t, int j_spe)

Returns the value of the gaseous species diffusivities in the channels, gas diffusion layers, micro porous layers, and catalysts (Equation 20.45).

real MCD_Gas_Diffusivity(cell_t c, Thread *t, int i)

Returns the tortuosity-corrected value of the gas species diffusion coefficients computed with the multicomponent diffusion option (Equation 20.45).

real Compute_Js(real s, Thread *t)

Computes the Leverett function; namely, the p_c-s relation (Equation 20.36).

Real Capillary_P_Diffusivity(real sat, real K_abs, cell_t c, Thread *t)

Returns the $\rho_l K K_r / \mu_l$ in Equation 20.30.

real Water_Content(cell_t c, Thread *t)

Computes the water content λ.

real Anode_AV_Ratio(cell_t c, Thread *t)

Returns the value of the specific active surface area (ζ_{an} in Equation 20.5) for the anode catalyst.

real Cathode_AV_Ratio(cell_t c, Thread *t)

Returns the value of the specific active surface area (ζ_{cat} in Equation 20.6) for the cathode catalyst.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

real Anode_J_TransCoef(cell_t c, Thread *t)

Returns the value of the anode reaction reference current density $\frac{\zeta_{an} j_{an}^{ref}}{\left([H_2]_{ref}\right)^{\gamma_{an}}}$ used in Equation 20.5.

real Cathode_J_TransCoef(cell_t c, Thread *t)

Returns the value of the cathode reaction reference current density $\frac{\zeta_{cat} j_{cat}^{ref}}{\left([O_2]_{ref}\right)^{\gamma_{cat}}}$ used in Equation 20.6.

real Open_Cell_Voltage(cell_t c, Thread *t)

Returns the value of the half-cell potential U^0 used in Equation 20.11 and Equation 20.12.

real Leakage_Current(cell_t c, Thread *t)

Returns the value of the total leakage current (I_l in Equation 20.55 through Equation 20.57).

real resistance_in_channel (real sat)

Returns momentum resistance as a function of liquid saturation in gas channels.

void Set_UDS_Names(char uds[n_uds_required][STRING_SIZE])

Used to rename user-defined scalars (UDSs). Note that the units of the user defined scalars cannot be changed.

```
void Set_UDS_Names(char uds[n_uds_required][STRING_SIZE])
 {
      strncpy(uds[0], "Electric Potential", STRING_SIZE-1);
      strncpy(uds[1], "Protonic Potential", STRING_SIZE-1);
      strncpy(uds[2], "Cap. Pressure", STRING_SIZE-1);
      strncpy(uds[3], "Water Content", STRING_SIZE-1);
      strncpy(uds[4], "Liq. Saturation in Channels",  STRING_SIZE-1);
 }
```

If you want to change the names of UDSs, change the second argument of the `strncpy` functions, recompile and link the module as with any modification to `pemfc_user.c`. Note that `STRING_SIZE` is fixed in `pemfc.h` and should not be changed.

Important:

When you load older PEM Fuel Cell cases into Ansys Fluent, and you are monitoring a UDS using volume or surface monitors, make sure you re-visit the corresponding monitors dialog box (for example, the **Volume Monitor** or the **Surface Monitor** dialog box) to make sure that the correct UDS name is used for the appropriate monitor.

void Set_UDM_Names(char udm[n_udm_required][STRING_SIZE])

Used to rename user defined memory (UDMs). Note that the units of user defined memory cannot be changed.

```
void Set_UDM_Names(char udm[n_udm_required][STRING_SIZE])
 {
```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
    strncpy(udm[0], "X Current Flux Density",  STRING_SIZE-1);
    strncpy(udm[1], "Y Current Flux Density",  STRING_SIZE-1);
    strncpy(udm[2], "Z Current Flux Density",  STRING_SIZE-1);
    strncpy(udm[3], "Current Flux Density Magnitude", STRING_SIZE-1);
    strncpy(udm[4], "Ohmic Heat Source", STRING_SIZE-1);
    strncpy(udm[5], "Reaction Heat Source",  STRING_SIZE-1);
    strncpy(udm[6], "Overpotential", STRING_SIZE-1);
    strncpy(udm[7], "Evaporation to Vapor", STRING_SIZE-1);
    strncpy(udm[8], "Osmotic Drag Coefficient", STRING_SIZE-1);
    strncpy(udm[9], "Water Activity", STRING_SIZE-1);
    strncpy(udm[10], "Equilibrium Water Content",  STRING_SIZE-1);
    strncpy(udm[11], "Absorption from Vapor", STRING_SIZE-1);
    strncpy(udm[12], "Absorption from Liquid", STRING_SIZE-1);
    strncpy(udm[13], "Transfer Current",  STRING_SIZE-1);
    strncpy(udm[14], "Liquid Saturation",  STRING_SIZE-1);
    strncpy(udm[15], "GDL Liquid Removal",  STRING_SIZE-1);
    strncpy(udm[16], "Osmotic Drag Source (PEM)", STRING_SIZE-1);
    strncpy(udm[17], "Cap. Pressure", STRING_SIZE-1);
    strncpy(udm[18], "Pressure Gradient Source", STRING_SIZE-1);
}
```

If you want to change the names of UDMs, change the second argument of the `strncpy` functions, recompile and link the module as with any modification to `pemfc_user.c`. Note that `STRING_SIZE` is fixed in `pemfc.h` and should not be changed.

Important:

When you load older PEM Fuel Cell cases into Ansys Fluent, and you are monitoring a UDM using volume or surface monitors, make sure you re-visit the corresponding monitors dialog box (for example, the **Volume Monitor** or the **Surface Monitor** dialog box) to make sure that the correct UDM name is used for the appropriate monitor.

`real electric_contact_resistance(face_t f, Thread *t, int ns)`

Returns the value for the electrical contact resistance.

`real Transfer_Current(real i_ref, real gamma, int species_i, real alpha_a, real alpha_c, real *dRade, real *dRcde, Thread *t, cell_t c, real cathode_volume)`

Computes the transfer current (A/m^3), corresponding to R_{an} in Equation 20.5 and R_{cat} in Equation 20.6.

Inputs for this function include:

`i_ref`	effective transfer current coefficient, computed by `Cathode_J_TransCoef(c,t)` or `Anode_J_TransCoef(c,t)`
`gamma`	cathode or anode concentration exponent
`species_i`	species index used in fuel cells (for example `i_o2`, `i_h2`, `i_h2o`)
`alpha_a`	product of anode exchange coefficient and $\frac{F}{RT}$
`alpha_c`	product of cathode exchange coefficient and $\frac{F}{RT}$
`t`	current thread
`c`	current cell

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

`cathode_volume`	total volume of cathode catalyst layer

Outputs for this function include:

`source`	anode or cathode volumetric transfer current (R_{an} in Equation 20.5 or R_{cat} in Equation 20.6)
`*dRade`	partial derivative of R_{an} with respect to activation loss
`*dRcde`	partial derivative of R_{cat} with respect to activation loss

`Thermal_ctk_pemfc(face_t f, Thread *t)`

Used to change the constant value of **Thermal Contact Resistance** set in the **Porous Jump** dialog box to a locally variable value.

`Phase_Change_const(cell_t c, Thread *t)`

Used to change the constant value of γ_{gl} used to compute the gas-liquid phase change rate.

`Equilibrium_Water_Content (cell_t c, Thread *t, real act, real sat)`

Used to change the equilibrium water content computation in Equation 20.28 in the *Fluent Theory Guide*.

Additional inputs for this function include:

`act`	water activity (a in Equation 20.28 in the *Fluent Theory Guide*)
`sat`	liquid saturation (s in Equation 20.28 in the *Fluent Theory Guide*)

32.2. Using the Fuel Cell and Electrolysis Model

The procedure for setting up and solving fuel cell problems using the Fuel Cell and Electrolysis Model is described in detail in this chapter. Refer to the following sections for more information:

32.2.1. Overview and Limitations

32.2.2. Geometry Definition for the Fuel Cell and Electrolysis Model

32.2.3. Installing the Fuel Cell and Electrolysis Model

32.2.4. Loading the Fuel Cell and Electrolysis Module

32.2.5. Workflow for Using the Fuel Cell and Electrolysis Module

32.2.6. Setting Up the Fuel Cell and Electrolysis Module

32.2.7. Modeling Current Collectors

32.2.8. Fuel Cell and Electrolysis Model Boundary Conditions

32.2.9. Solution Guidelines for the Fuel Cell and Electrolysis Model

32.2.10. Postprocessing the Fuel Cell and Electrolysis Model

32.2.11. User-Accessible Functions

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.2.1. Overview and Limitations

The Ansys Fluent Fuel Cell and Electrolysis Model are made up of several user-defined functions (UDFs) and a graphical user interface. The potential fields are solved as user-defined scalars. The liquid water saturation, s, and the water content, λ, are also solved as user-defined scalars. The electrochemical reactions occurring on the catalyst are modeled through various source terms while other model parameters are handled through the user interface. The Fuel Cell and Electrolysis Model can be used in parallel Ansys Fluent as well.

Note the following limitation when using the Fuel Cell and Electrolysis model:

- The anisotropic species diffusivity option is not compatible with the Fuel Cell and Electrolysis model.

32.2.2. Geometry Definition for the Fuel Cell and Electrolysis Model

Due to the fact that there is a number of different physical zones associated with the fuel cell, the following regions must be present in the fuel cell mesh:

- Anode flow channel
- Anode gas diffusion layer
- Anode catalyst layer
- Membrane layer
- Cathode catalyst layer
- Cathode gas diffusion layer
- Cathode flow channel

The following zones have to be identified, if present in the fuel cell mesh:

- Anode current collector
- Cathode current collector
- Coolant channel

32.2.3. Installing the Fuel Cell and Electrolysis Model

The Fuel Cell and Electrolysis Model is provided as an add-on module with the standard Ansys Fluent licensed software. The module is installed with the standard installation of Ansys Fluent in a directory called `addons/fuelcells` in your installation area. The Fuel Cell and Electrolysis Model consists of a UDF library and a pre-compiled Scheme library, which must be loaded and activated before calculations can be performed.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.2.4. Loading the Fuel Cell and Electrolysis Module

The Fuel Cell and Electrolysis module can be loaded only after a valid Ansys Fluent case file has been set or read.

There are two different ways in which you can load the Fuel Cell and Electrolysis module into Ansys Fluent:

- Through the ribbon

  In the **Physics** ribbon tab, click **More** (**Models** group), and select **Fuel Cell & Electrolysis Model...**.

  Physics → **Models** → **More** → **Fuel Cell & Electrolysis Model...**

  Once the Fuel Cell and Electrolysis module is loaded into Ansys Fluent, the **Fuel Cell and Electrolysis Models** dialog box opens where you can set an appropriate model as described in Specifying Cathode Properties (**Cathode** Tab) (p. 3462).

- Through the text user interface (TUI)

  The text command to load the add-on module is

  `define` → `models` → `addon-module`

  A list of Ansys Fluent add-on modules is displayed:

```
> /define/models/addon-module
Fluent  Addon Modules:
     0. None
     1. MHD Model
     2. Fiber Model
     3. Fuel Cell and Electrolysis Model
     4. SOFC Model with Unresolved Electrolyte
     5. Population Balance Model
     6. Adjoint Solver
     7. Single Potential Battery Module
     8. MSMD Battery Model
     9. PEM Fuel Cell Model
Enter Module Number: [0] 3
```

  Select the Fuel Cell and Electrolysis Model by entering the module number 3.

During the loading process, a Scheme library (containing the graphical and text user interface) and a UDF library (containing a set of user defined functions) are loaded into Ansys Fluent.

32.2.5. Workflow for Using the Fuel Cell and Electrolysis Module

The Fuel Cell and Electrolysis Model can be used to model polymer electrolyte membrane fuel cells (PEMFC), solid oxide fuel cells (SOFC), and the process of high-temperature electrolysis. The following describes an overview of the procedure required in order to use the Fuel Cell and Electrolysis Model in Ansys Fluent.

1. Start Ansys Fluent.

2. Read the case file.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Scale the grid, if necessary.
4. Use the **Fuel Cell and Electrolysis Models** dialog box to define the fuel cell model parameters.
5. Define material properties.
6. Set the operating conditions.
7. Set the boundary conditions.
8. Start the calculations.
9. Save the case and data files.
10. Process your results.

Important:

- The **Fuel Cell and Electrolysis Models** dialog box greatly simplifies the input of parameters and boundary conditions, but it does not replace the boundary conditions interface. Therefore, it is a good policy to start the setup with the **Fuel Cell and Electrolysis Models** dialog box and do the finishing steps for boundary conditions afterwards.
- Note that the majority of this chapter describes how to set up the Ansys Fluent Fuel Cell and Electrolysis Model using the graphical user interface. You can also perform various tasks using the text user interface. For more information, see Using the Fuel Cell and Electrolysis Text User Interface.

32.2.6. Setting Up the Fuel Cell and Electrolysis Module

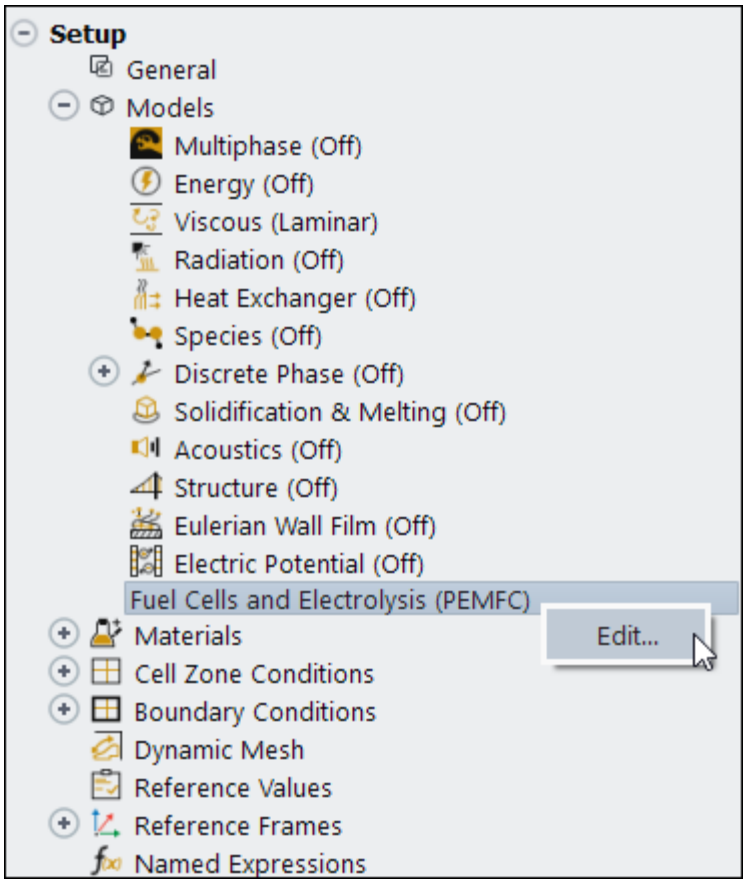
Once the module has been loaded, in order to set fuel cell model parameters and assign properties to the relevant regions in your fuel cell, you need to access the fuel cell graphical user interface (the **Fuel Cell and Electrolysis Models** dialog box).

The **Fuel Cell and Electrolysis Models** dialog box can be accessed either through the ribbon (as described in Loading the Fuel Cell and Electrolysis Module (p. 3449)) or from the **Setup/Models/Fuel Cells and Electrolysis** tree item.

Setup → Models → Fuel Cells and Electrolysis → Edit...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 32.18: The Fuel Cell and Electrolysis Option in the Tree



By default, the PEMFC model is already enabled, however, you can also choose the SOFC or the SOEC models.

Using the **Fuel Cell and Electrolysis Models** dialog box, you can identify the relevant zones for the current collectors, flow channels, gas diffusion layers, catalyst layers, and the membrane/electrolyte. You can specify the following inputs using the **Fuel Cell and Electrolysis Models** dialog box. Optional inputs are indicated as such.

1. Enable the appropriate type of fuel cell model, either **PEMFC**, **SOFC**, or **SOEC**.
2. Enable either the single-phase or the multi-phase fuel cell model (if **PEMFC** is selected).
3. Set the appropriate options for the fuel cell model (optional).
4. Set the various parameters for the fuel cell model.
5. Select the appropriate zones and specify the properties on the anode side.
6. Select the appropriate zones and specify the properties of the membrane/electrolyte.
7. Select the appropriate zones and specify the properties on the cathode side.
8. Provide input for advanced features such as contact resistivities, coolant channel properties, or stack management settings (optional).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9. Set solution controls such as under-relaxation factors (optional).

10. Provide input to assist reporting (optional).

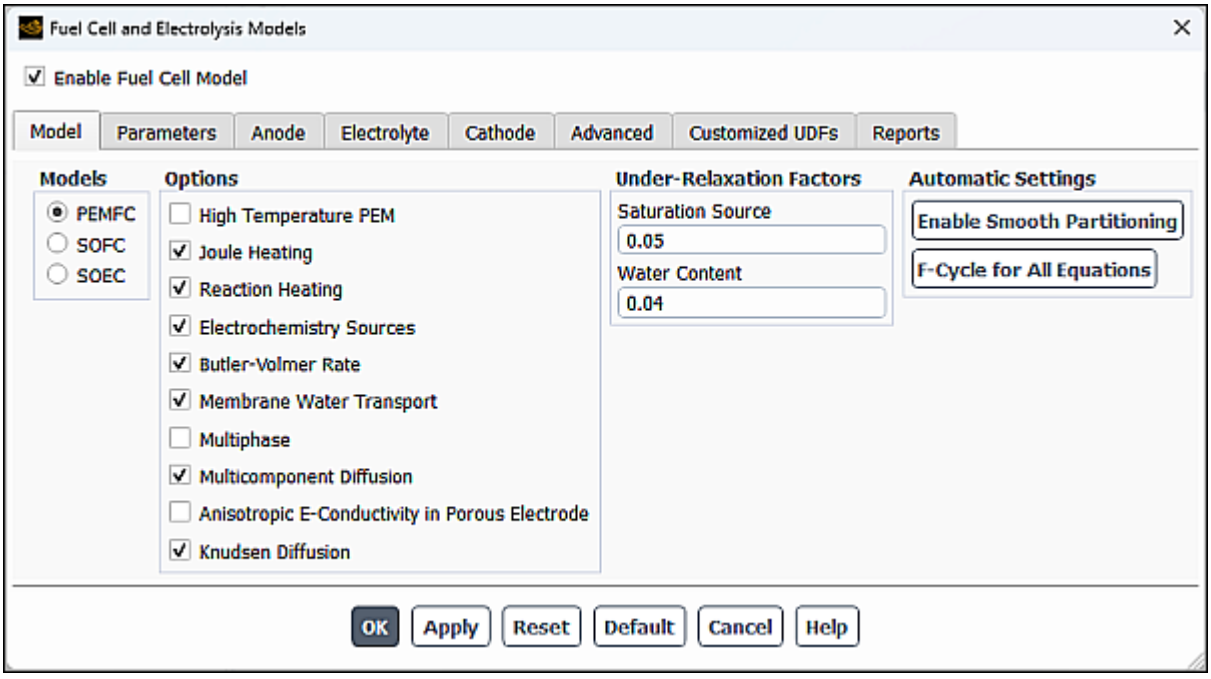
Refer to the following sections for more information:

32.2.6.1. Specifying Model Options (Model Tab)
32.2.6.2. Specifying Model Parameters (Parameters Tab)
32.2.6.3. Specifying Anode Properties (Anode Tab)
32.2.6.4. Specifying Electrolyte/Membrane Properties (Electrolyte Tab)
32.2.6.5. Specifying Cathode Properties (Cathode Tab)
32.2.6.6. Setting Advanced Properties (Advanced Tab)
32.2.6.7. Customizing the Fuel Cell and Electrolysis Module
32.2.6.8. Reporting on the Solution (Reports Tab)

32.2.6.1. Specifying Model Options (Model Tab)

The **Model** tab of the **Fuel Cell and Electrolysis Models** dialog box allows you to turn on or off various options when solving a fuel cell problem. To model polymer electrolyte membrane fuel cells, enable the **PEMFC** option in the **Model** tab. Likewise, to model solid oxide fuel cells, enable the **SOFC** option in the **Model** tab. Finally, to model electrolysis, enable the **SOEC** option in the **Model** tab.

Figure 32.19: The Model Options in the Fuel Cell and Electrolysis Models Dialog Box—PEMFC Enabled



Several fuel cell model options are available in the **Model** tab of the **Fuel Cell and Electrolysis Models** dialog box including:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The **High Temperature PEM** option enables the high-temperature PEMFC model (see High-Temperature PEMFC in the *Fluent Theory Guide* for more information). When this option is not selected, Ansys Fluent uses the low-temperature PEMFC model.

- The **Joule Heating** option takes into account ohmic heating. This option includes the I^2R term in the energy source term from Equation 20.82 in the calculations.

- The **Reaction Heating** option takes into account the heat generated by the electrochemical reactions, which includes the $h_{reaction}$ term, and the product of transfer current and the over-potentials in the energy source term from Equation 20.82 in the calculations.

- The **Electrochemistry Sources** option allows the Fuel Cell and Electrolysis Model to take electrochemistry effects into account. If you are only interested in the basic flow field throughout the fuel cell, you can turn off the **Electrochemistry Sources** option in order to suppress most effects of the Fuel Cell and Electrolysis Model. To turn off all effects of the Fuel Cell and Electrolysis Model, you should also turn off the **Membrane Water Transport** and **Multiphase** options.

- The **Butler-Volmer Rate** option (the default) is used to compute the transfer currents inside the catalyst layers. If this option is turned off, the Tafel approximation (Equation 20.69) is used.

- The **Membrane Water Transport** option takes into account the transport of water across the membrane. This option is only available for the low-temperature PEMFC model (that is, the **High Temperature PEM** option is not selected).

- The **Multiphase** option takes into account multiphase calculations. Use this option if you are solving for approximate liquid transport in the gas diffusion layer of the fuel cell (low-temperature PEMFC only).

- The **Multicomponent Diffusion** option is used to compute the gas species mass diffusivity using the full multicomponent diffusion method as described in Equation 20.89, as opposed to the default option that uses Equation 20.87.

- The **Anisotropic E-cond in GDL/MPL** option is used to model the typically non-isotropic electrical conductivity. It is applicable only for porous electrodes (gas diffusion layers).

  Due to the fibrous structure of the porous material that is used for the electrodes (or gas diffusion layer), the electrical conductivity is typically non-isotropical, with the cross-plane components being orders of magnitude smaller than the in-plane components. This can be modeled using the **Anisotropic E-cond in GDL/MPL** setting. When this option is enabled, the **Electrical Conductivity** for the solid material used in the electrolyte is no longer used. Instead, you need to specify, for this solid material, the electrical conductivity by choosing one of the three non-isotropical options for the UDS diffusivity (`UDS-0`). The three options are: **anisotropic**; **orthotropic**; and **cyl-orthotropic**. For more information about these UDS Diffusivity options, refer to the Ansys Fluent User's Guide (p. 1).

  For example, to use this feature, perform the following steps:

  – Select the **Anisotropic E-cond in GDL/MPL** option in the **Model** tab of the **Fuel Cell and Electrolysis Models** dialog box.

  – In the **Create/Edit Materials** dialog box, select `defined-per-uds` for **UDS Diffusivity** for the solid material that is to be used for the porous electrode.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– Select one of the three options for UDS-0: **anisotropic**; **orthotropic**; or **cyl-orthotropic** and set the appropriate values.

> **Important:**
>
> Note that, in this case, the **Electrical Conductivity** for *this* solid material is ignored.

- The **Knudsen Diffusion** option includes the effect of Knudsen diffusion in the calculation of the effective gas diffusivity using Equation 20.89 in the *Fluent Theory Guide*.
- For low-temperature PEMFC problems, you can use the **Under-Relaxation Factors** fields to influence the solution process.

  The saturation source term r_w in Equation 20.84 usually requires under-relaxation. You can change the default value for the under-relaxation factor by changing the value for **Saturation Source**.

  The water content, λ, in Equation 20.95 also may need under-relaxation. You can change the default value for the under-relaxation factor by changing the value for **Water Content**.

Under **Automatic Settings**, the following parallel multigrid solver control parameters are available:

- **Enable Smooth Partition**: Enables the smooth partitioning process when running in Parallel.

  When this option is enabled, the fuel cell parallel solver:

  – Uses the Metis method for partitioning the mesh

  – Enables Laplace Smoothing

  – Partitions and distributes the mesh data to all compute nodes

- **F-cycle for All Equations**: Sets the multigrid cycle type to F cycle for all equations that are being solved. This control overrides the equation cycle settings in the **Advanced Solution Controls** dialog box.

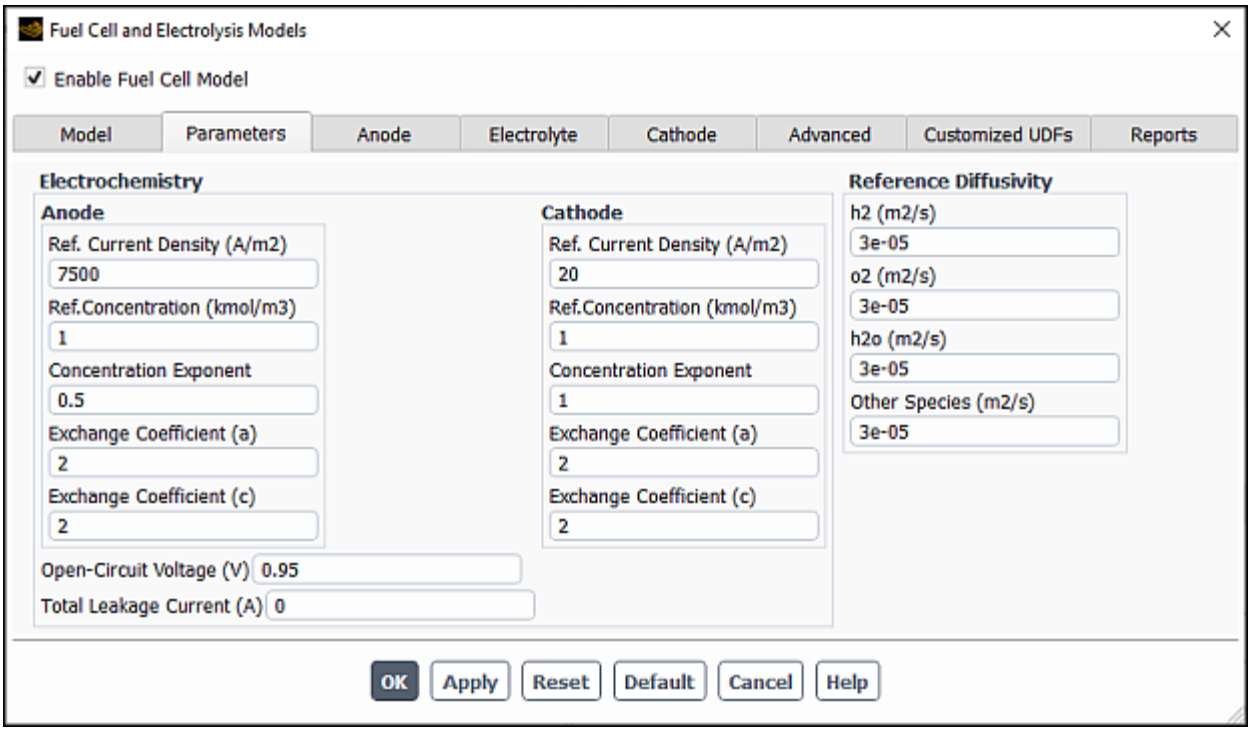
Nearly all options are turned on by default. You may want to override the default values, depending on the problem you want to model. For instance, if you are not concerned with the heat generated due to chemical reaction, then you may want to turn off the **Reaction Heating** option.

32.2.6.2. Specifying Model Parameters (Parameters Tab)

You can use the **Parameters** tab of the **Fuel Cell and Electrolysis Models** dialog box to specify the electrochemistry parameters for the Fuel Cell and Electrolysis Model, reference diffusivities for the reactants, among other model parameters.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 32.20: The Parameters Tab of the Fuel Cell and Electrolysis Models Dialog Box



There are various parameters under **Electrochemistry** in the **Fuel Cell and Electrolysis Models** dialog box. For both the anode and the cathode, you can also set the following parameters or leave the default values.

- The **Ref. Current Density** corresponds to j_{an}^{ref} and j_{cat}^{ref}, the reference exchange current density from Equation 20.66 and Equation 20.67.
- The **Ref. Concentration** corresponds to the reference concentration ($[H_2]_{ref}$ and $[O_2]_{ref}$) with units of 1 $kgmol/m^3$ (see Equation 20.66 and Equation 20.67).
- The **Concentration Exponent** corresponds to γ, the concentration dependence from Equation 20.66.
- The **Exchange Coefficient (a)** and **Exchange Coefficient (c)** correspond to the transfer coefficients, α_{an} and α_{cat}, from Equation 20.66 and Equation 20.67.
- The **Open-Circuit Voltage** corresponds to V_{oc} in Equation 20.71.
- The **Leakage Current** corresponds to I_{leak} in Equation 20.99 through Equation 20.104. This is the total amount of transfer current (A) due to fuel-oxidant cross-over (leakage through the electrolyte). When this happens, the fuel cell generates less current especially for cases with low values of fuel or air utilization. In addition to the constant value you can specify in the **Electrochemistry** tab, you can also specify the leakage current through the user-defined function `Leakge_Current()`. For more information, see User-Accessible Functions (p. 3477).

Moreover, the following parameters can also be set here:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The **Reference Diffusivities** correspond to D_i^0 from Equation 20.87, the species mass diffusivity. These are not be required if the **Multicomponent Diffusion** option is enabled in the **Model** tab.
- The **Pore Blockage for Gas Diffusion** corresponds to r_s from Equation 20.87 for multiphase PEMFC calculations.
- The **Pore Blockage for Transfer Current** is used to account for liquid blockage to the reaction surface by modifying R_{an} and R_{ca} in Equation 20.66 through Equation 20.69 as follows:

$$R_{an}^{new} = \left(1-s\right)^r R_{an}$$
$$R_{cat}^{new} = \left(1-s\right)^r R_{cat}$$

  where r is the pore blockage for transfer current.
- The **Exponent for relative permeability** corresponds to γ in Equation 20.85.

Note that, conventionally, the negative voltage is supplied to the cathode side in power consuming devices such as electrolyzers. Ansys Fluent adopts the inverse notation where the negative voltage is supplied to the anode side whilst cathode remains positively charged. The main reason for this discrepancy is that the same infrastructure is used for both the electrolysis and fuel cells models. Usage of the terms "anode" and "cathode" in this manual and in the user interface should be interpreted according to conventions for power-supplying devices.

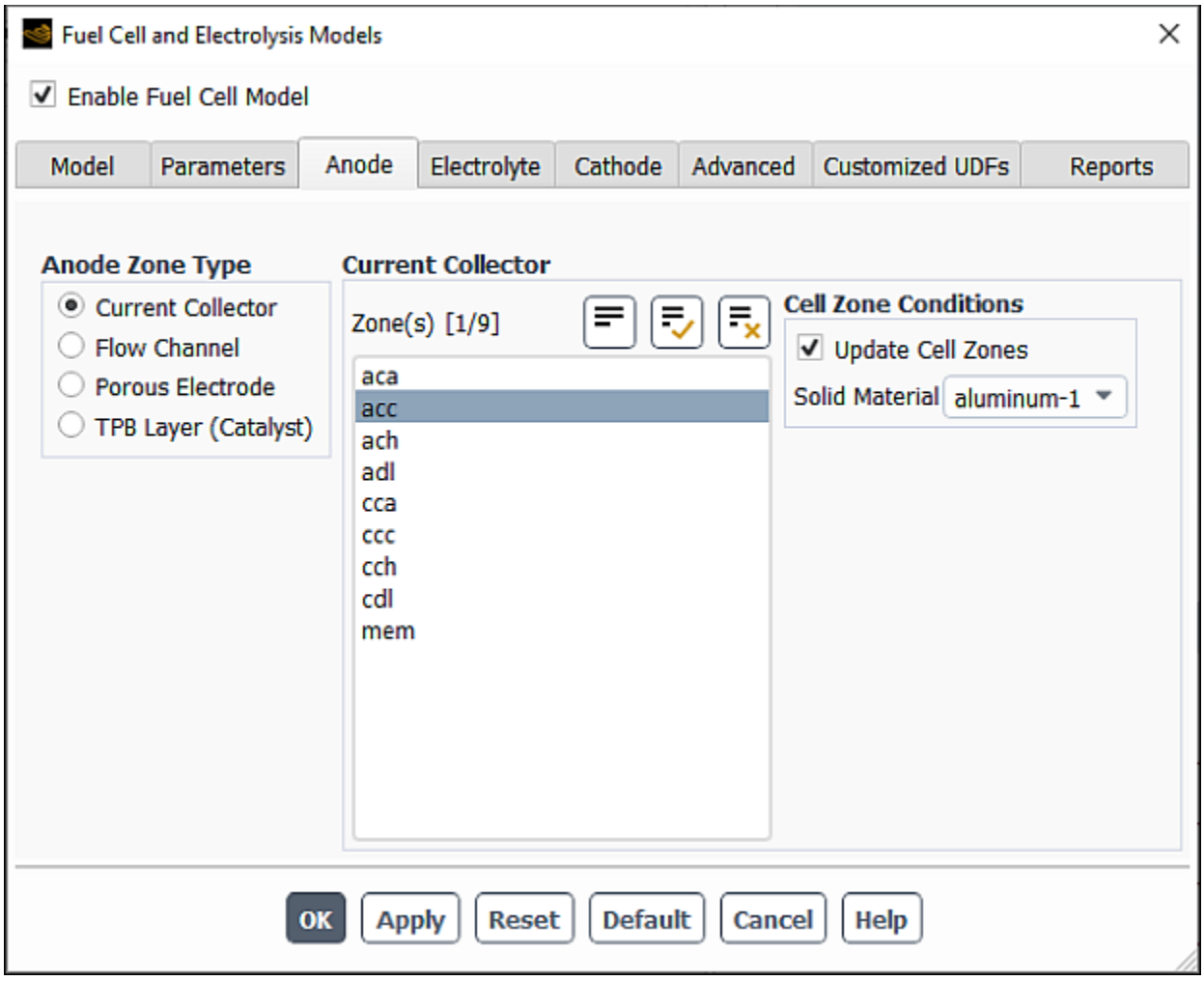
32.2.6.3. Specifying Anode Properties (Anode Tab)

You can use the **Anode** tab of the **Fuel Cell and Electrolysis Models** dialog box to specify zones and properties of the current collector, the flow channel, the diffusion layer, and the catalyst layer for the anode portion of the fuel cell.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.2.6.3.1. Specifying Current Collector Properties for the Anode

Figure 32.21: The Anode Tab of the Fuel Cell and Electrolysis Models Dialog Box With Current Collector Selected

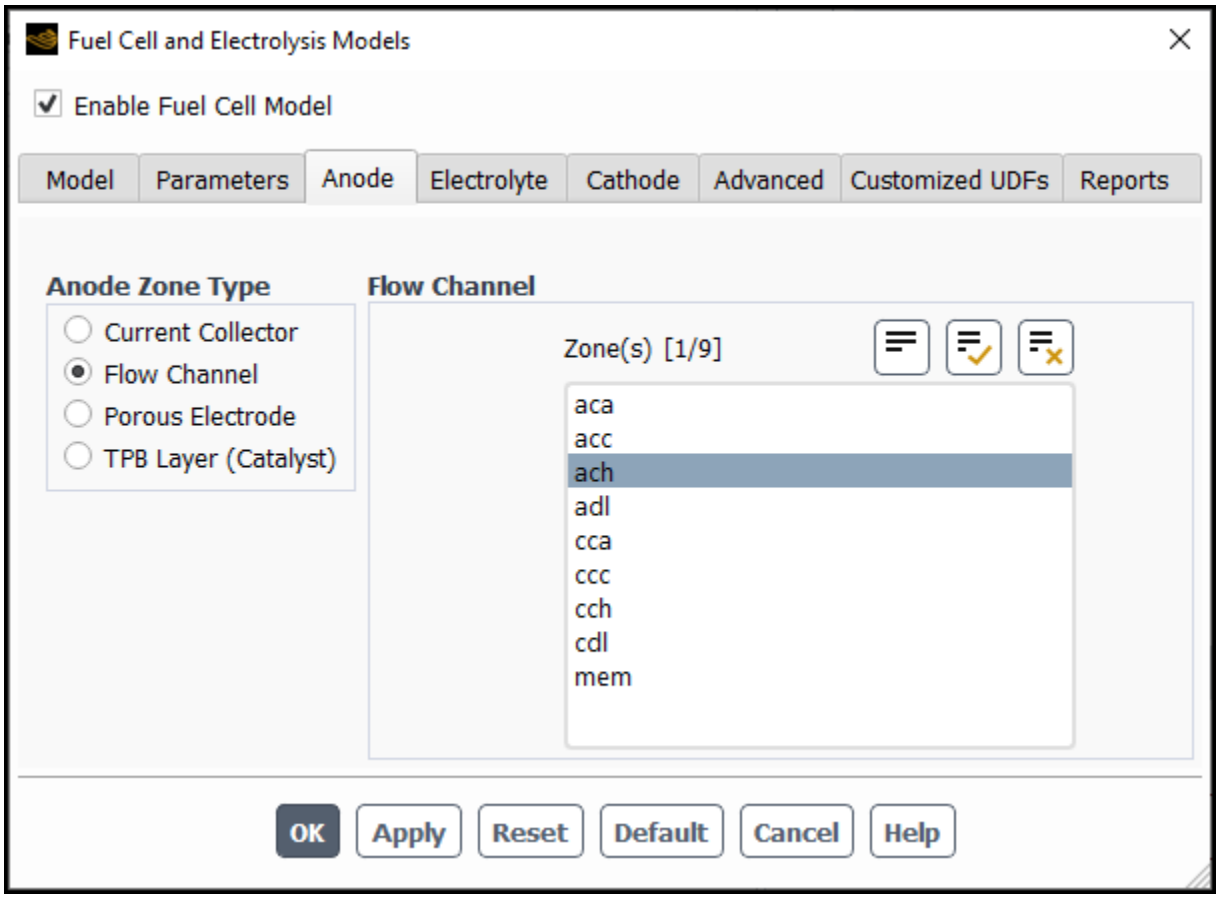


1. Select the **Anode** tab of the **Fuel Cell and Electrolysis Models** dialog box.
2. Select **Current Collector** under **Anode Zone Type**.
3. Select a corresponding zone from the **Zone(s)** list. If you are modeling a fuel cell stack, then you must pick *all* zones of a particular type as a group.
4. Select a **Solid Material** from the corresponding drop-down list. Solid materials can be customized using the **Create/Edit Materials** dialog box. Note that for the **Electrical Conductivity**, you can only choose a constant value in the **Create/Edit Materials** dialog box. The solid electrical conductivity value is the diffusivity of the solid phase potential in the solid zones.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.2.6.3.2. Specifying Flow Channel Properties for the Anode

Figure 32.22: The Anode Tab of the Fuel Cell and Electrolysis Models Dialog Box With Flow Channel Selected

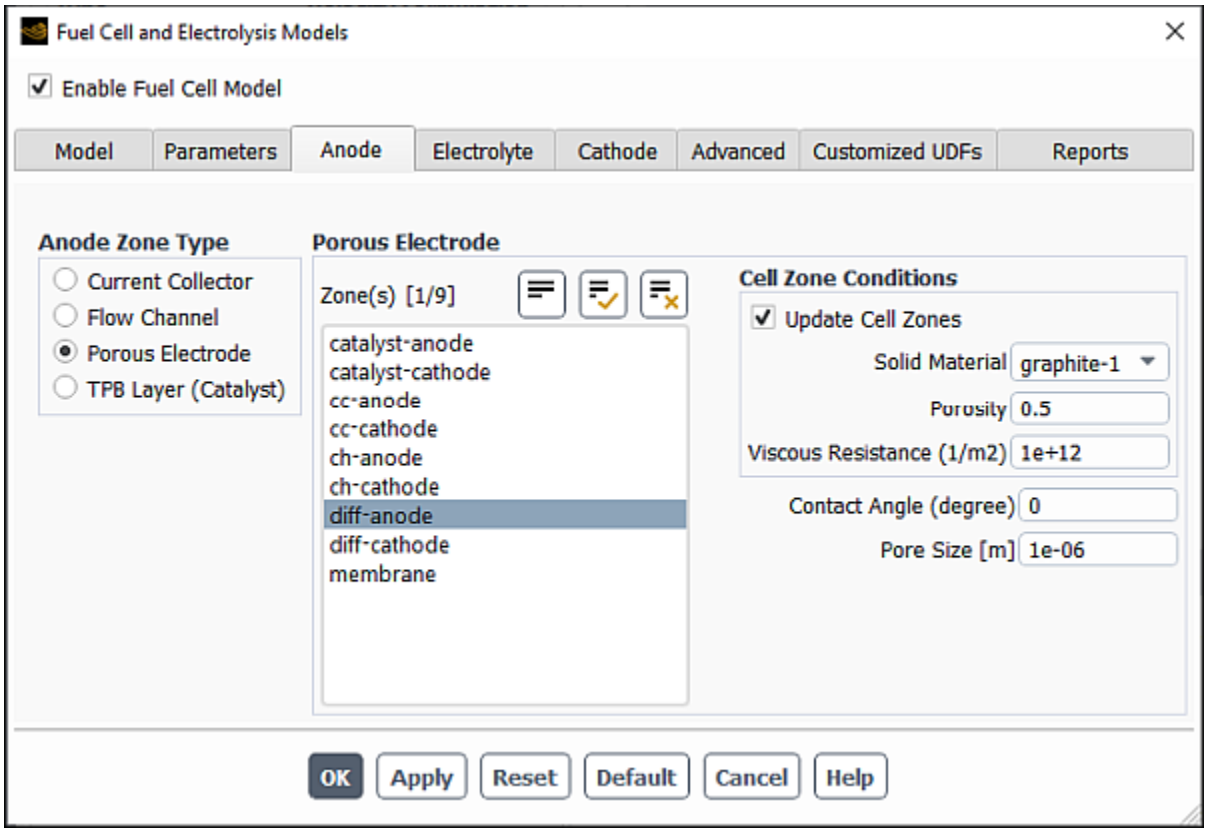


1. Select the **Anode** tab of the **Fuel Cell and Electrolysis Models** dialog box.
2. Select **Flow Channel** under **Anode Zone Type**.
3. Select a corresponding zone from the **Zone(s)** list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.2.6.3.3. Specifying Porous Electrode Properties for the Anode

Figure 32.23: The Anode Tab of the Fuel Cell and Electrolysis Models Dialog Box With Porous Electrode Selected

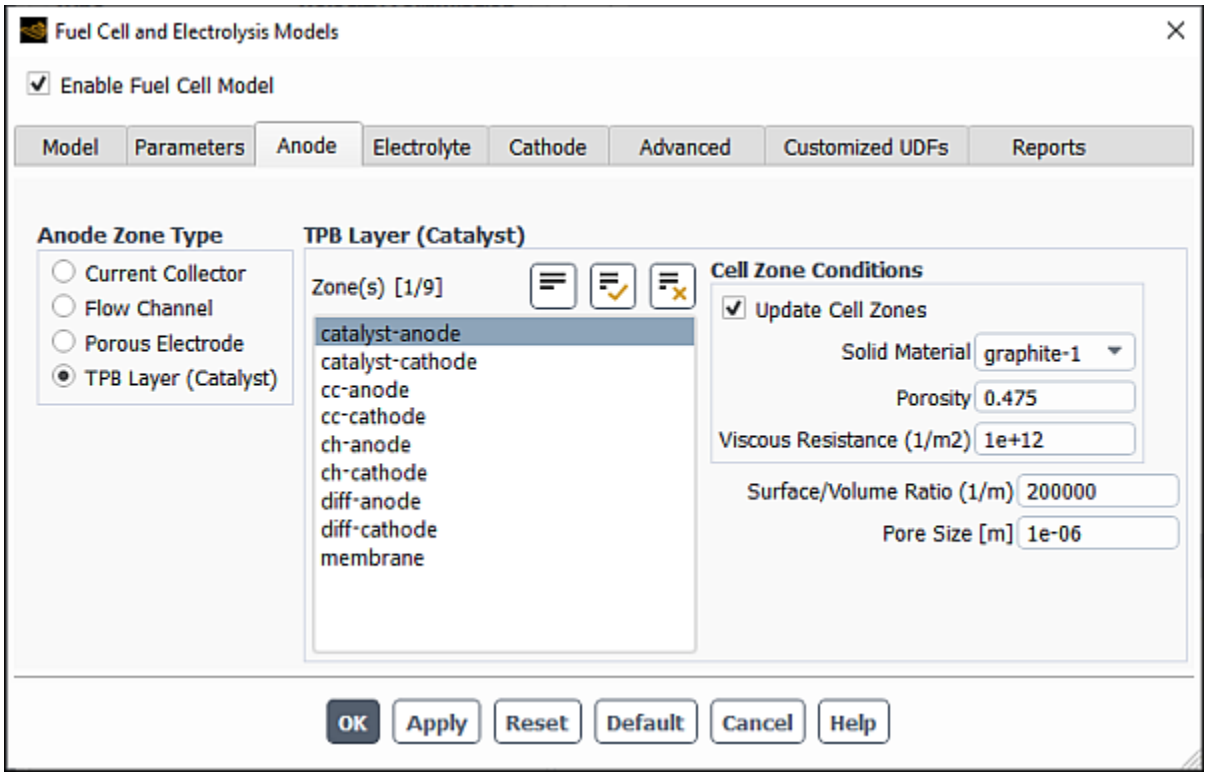


1. Select the **Anode** tab of the **Fuel Cell and Electrolysis Models** dialog box.
2. Select **Porous Electrode** under **Anode Zone Type**.
3. Select a corresponding zone from the **Zone(s)** list. If you are modeling a fuel cell stack, then you must pick *all* zones of a particular type as a group.
4. Select a **Solid Material** from the corresponding drop-down list. Solid materials can be customized using the **Create/Edit Materials** dialog box. Note that for the **Electrical Conductivity**, you can only choose a constant value in the **Create/Edit Materials** dialog box. The solid electrical conductivity value is the diffusivity of the solid phase potential in the solid zones.
5. Specify a value for the **Porosity**.
6. Specify a value for the **Viscous Resistance**.
7. Specify a value for the **Contact Angle** for multiphase fuel cell calculations (θ_c in Equation 20.86).
8. Specify the average **Pore Size** (r_e in Equation 20.105).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.2.6.3.4. Specifying Catalyst Layer Properties for the Anode

Figure 32.24: The Anode Tab of the Fuel Cell and Electrolysis Models Dialog Box With TPB Layer (Catalyst) Selected



1. Select the **Anode** tab of the **Fuel Cell and Electrolysis Models** dialog box.
2. Select **TPB Layer (Catalyst)** under **Anode Zone Type**.
3. Select a corresponding zone from the **Zone(s)** list. If you are modeling a fuel cell stack, then you must pick *all* zones of a particular type as a group.
4. Select a **Solid Material** from the corresponding drop-down list. Solid materials can be customized using the **Create/Edit Materials** dialog box. Note that for the **Electrical Conductivity**, you can only choose a constant value in the **Create/Edit Materials** dialog box. The solid electrical conductivity value is the diffusivity of the solid phase potential in the solid zones.
5. Specify a value for the **Porosity**.
6. Specify a value for the **Viscous Resistance**.
7. Specify a value for the **Surface/Volume Ratio** (the specific active surface area in Equation 20.66).
8. Specify a value for the **Contact Angle** for multiphase fuel cell calculations (θ_c in Equation 20.86).
9. Specify the average **Pore Size** (r_e in Equation 20.105).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

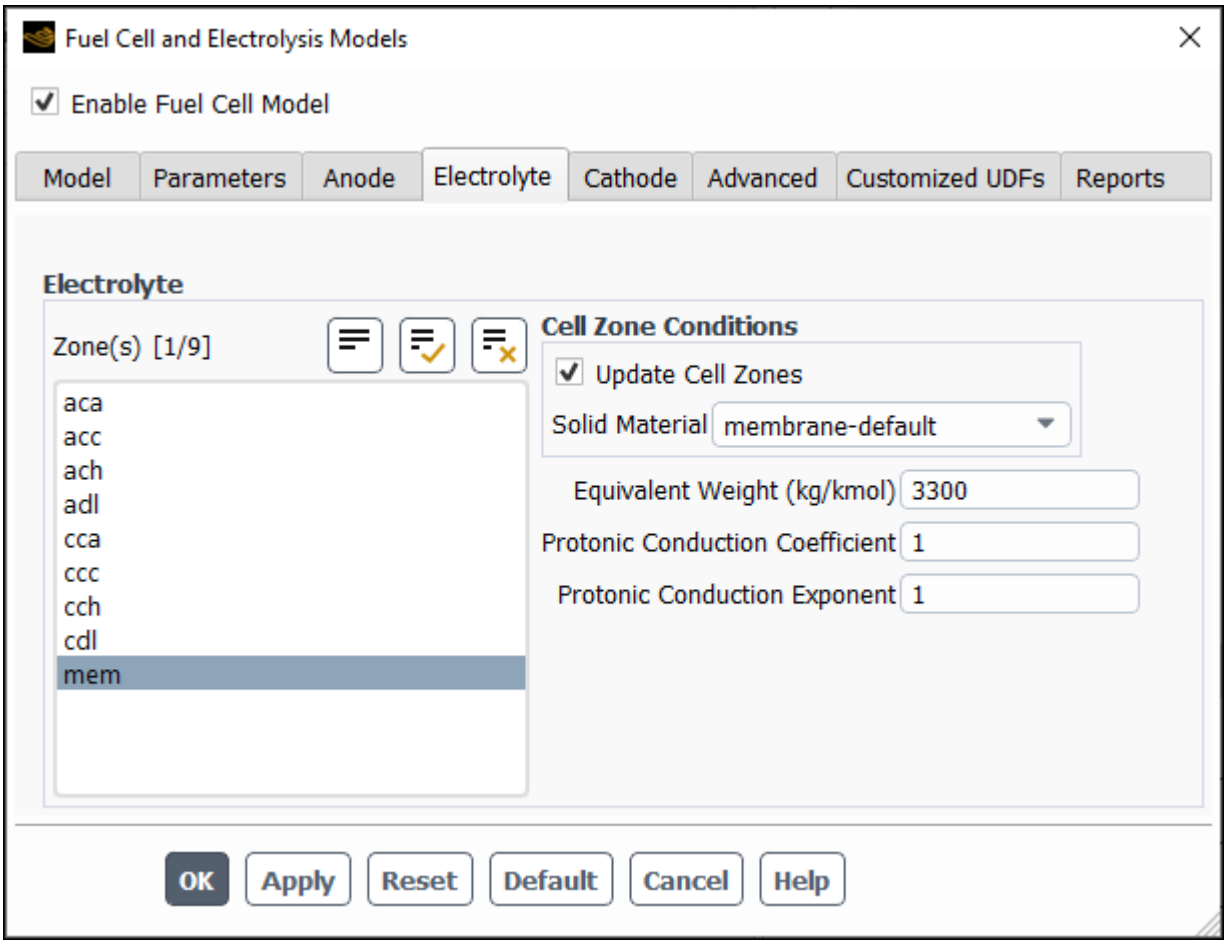
32.2.6.3.5. Specifying Cell Zone Conditions for the Anode

For each case of the anode's current collector, diffusion layer, and catalyst layer, you assign a solid material and/or set the porosity and the viscous resistance. These settings represent setting a cell zone condition. With the **Update Cell Zones** option turned on (the default setting), this cell zone condition is applied to all selected zones in the **Zone(s)** list. If you want to set the cell zone conditions for each zone individually (using the **Cell Zone Conditions** task page), you should turn off the **Update Cell Zones** option.

32.2.6.4. Specifying Electrolyte/Membrane Properties (Electrolyte Tab)

You can use the **Electrolyte** tab of the **Fuel Cell and Electrolysis Models** dialog box to specify zones and properties of the electrolyte/membrane portion of the fuel cell (for example, Figure 32.25: The Electrolyte Tab of the Fuel Cell and Electrolysis Models Dialog Box (p. 3461) for the low-temperature PEMFC).

Figure 32.25: The Electrolyte Tab of the Fuel Cell and Electrolysis Models Dialog Box



1. Select a corresponding zone from the **Zone(s)** list. If you are modeling a fuel cell stack, then you must pick *all* membrane zones as a group.

2. Select a **Solid Material** from the corresponding drop-down list. Solid materials can be customized using the **Create/Edit Materials** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. (low-temperature PEMFC only) Specify a value for the **Equivalent Weight** (M_m in Equation 20.93).

4. (low-temperature PEMFC only) Specify a value for the **Protonic Conduction Coefficient** (β in Equation 20.91). This is used to calculate the membrane phase electric conductivity.

5. (low-temperature PEMFC only) Specify a value for the **Protonic Conduction Exponent** (ω in Equation 20.91).

Note that the Fuel Cell and Electrolysis Model allows you to model the electrolyte/membrane as either a fluid zone or as a solid zone. For PEMFC, the Fuel Cell and Electrolysis Model still allows for water and the ionic current to pass through the electrolyte/membrane.

32.2.6.4.1. Specifying Cell Zone Conditions for the Membrane

When you assign a solid material to the membrane, you are setting a cell zone condition. With the **Update Cell Zones** option turned on (the default setting), this cell zone condition is applied to all selected zones in the **Zone(s)** list. If you want to set the cell zone conditions for each zone individually (using the **Cell Zone Conditions** task page), you should turn off the **Update Cell Zones** option.

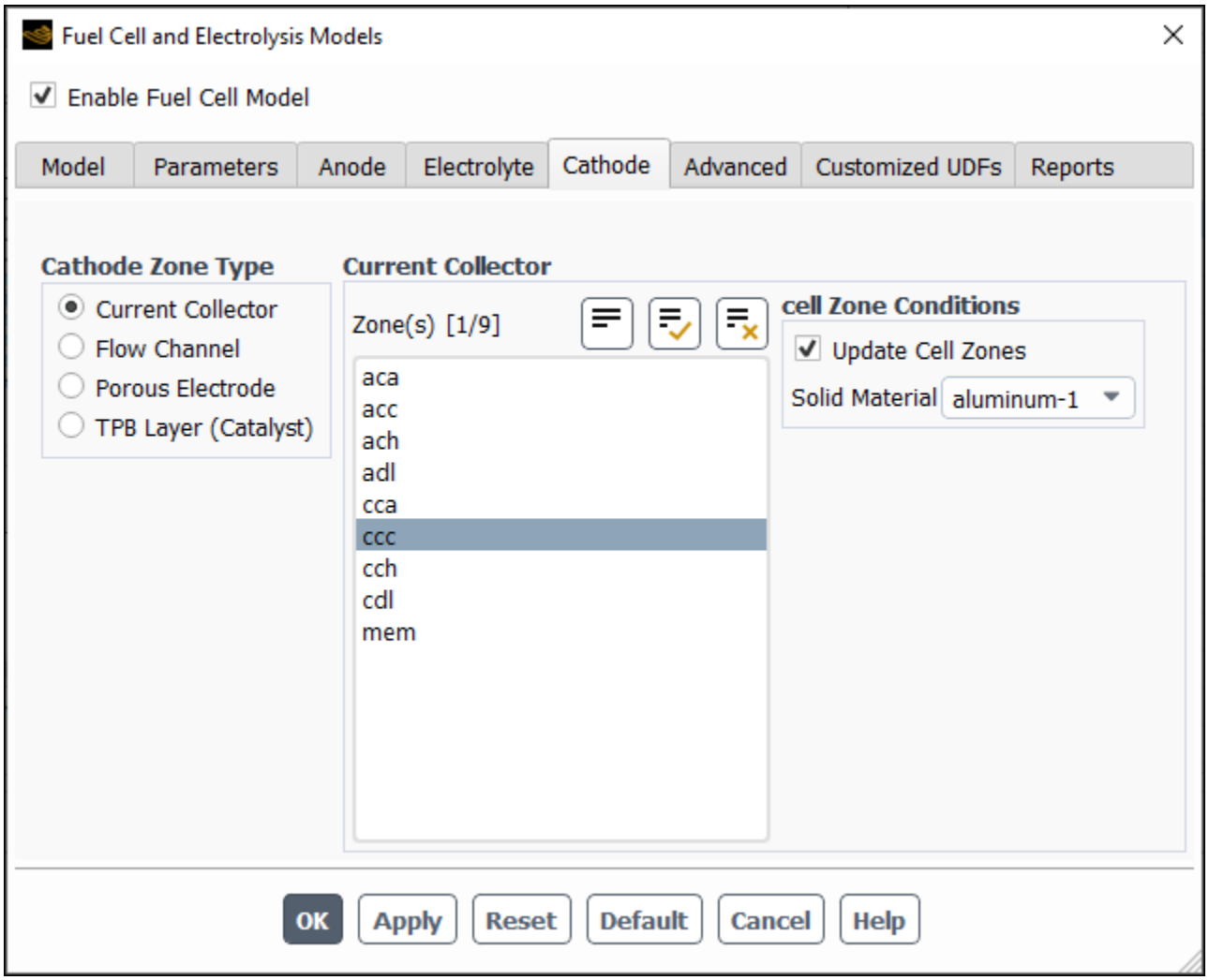
32.2.6.5. Specifying Cathode Properties (Cathode Tab)

You can use the **Cathode** tab of the **Fuel Cell and Electrolysis Models** dialog box to specify zones and properties of the current collector, the flow channel, the diffusion layer, and the catalyst layer for the cathode portion of the fuel cell.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.2.6.5.1. Specifying Current Collector Properties for the Cathode

Figure 32.26: The Cathode Tab of the Fuel Cell and Electrolysis Models Dialog Box With Current Collector Selected

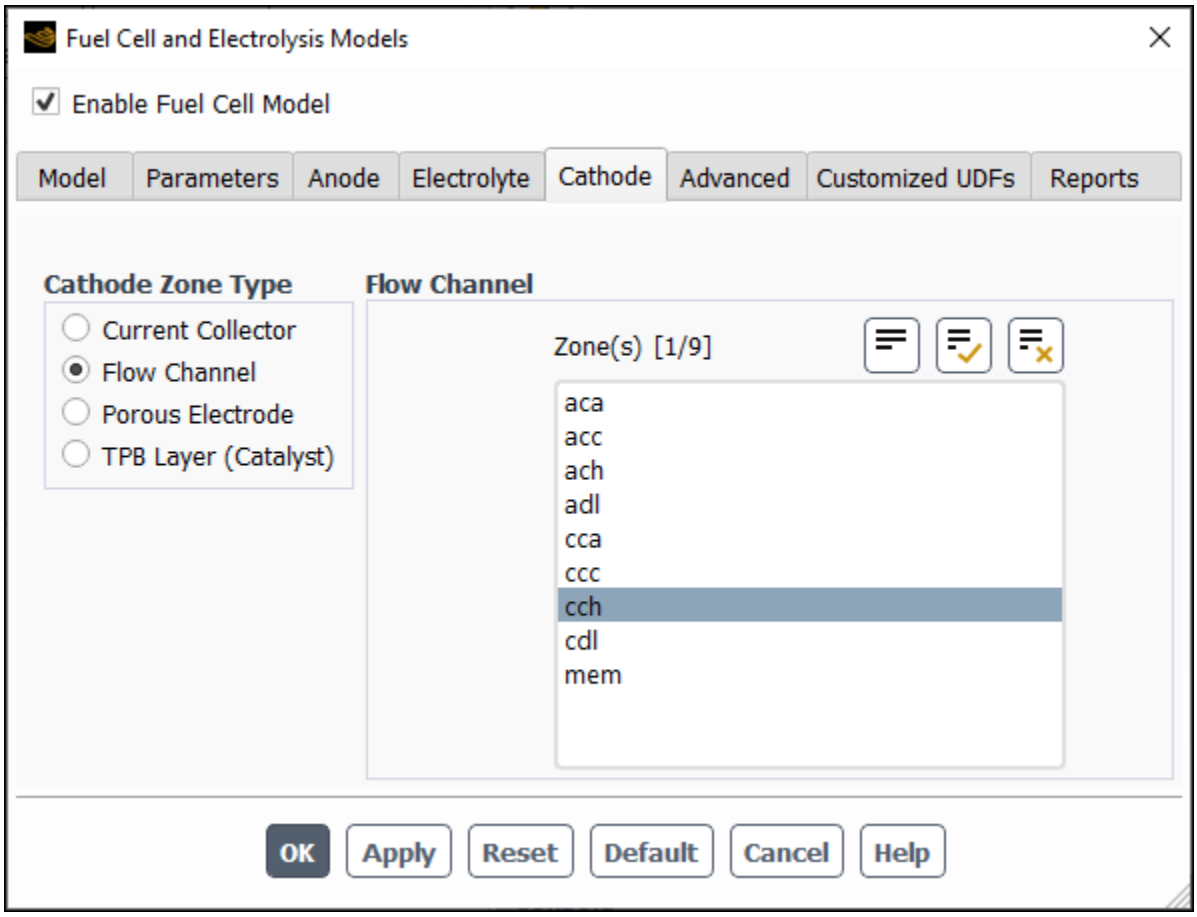


1. Select the **Cathode** tab of the **Fuel Cell and Electrolysis Models** dialog box.
2. Select **Current Collector** under **Cathode Zone Type**.
3. Select a corresponding zone from the **Zone(s)** list. If you are modeling a fuel cell stack, then you must pick *all* zones of a particular type as a group.
4. Select a **Solid Material** from the corresponding drop-down list. Solid materials can be customized using the **Create/Edit Materials** dialog box. Note that for the **Electrical Conductivity**, you can only choose a constant value in the **Create/Edit Materials** dialog box. The solid electrical conductivity value is the diffusivity of the solid phase potential in the solid zones.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.2.6.5.2. Specifying Flow Channel Properties for the Cathode

Figure 32.27: The Cathode Tab of the Fuel Cell and Electrolysis Models Dialog Box With Flow Channel Selected

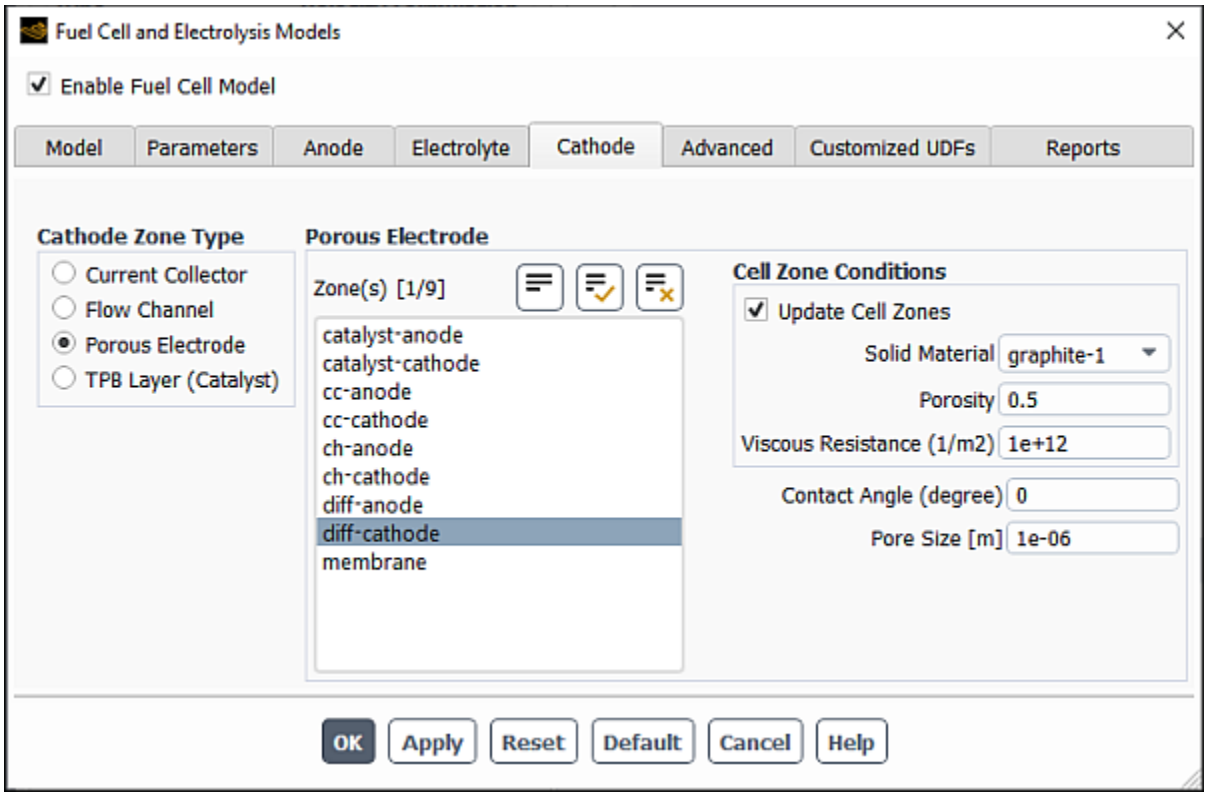


1. Select the **Cathode** tab of the **Fuel Cell and Electrolysis Models** dialog box.
2. Select **Flow Channel** under **Cathode Zone Type**.
3. Select a corresponding zone from the **Zone(s)** list. If you are modeling a fuel cell stack, then you must pick *all* zones of a particular type as a group.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.2.6.5.3. Specifying Porous Electrode Properties for the Cathode

Figure 32.28: The Cathode Tab of the Fuel Cell and Electrolysis Models Dialog Box With Porous Electrode Selected

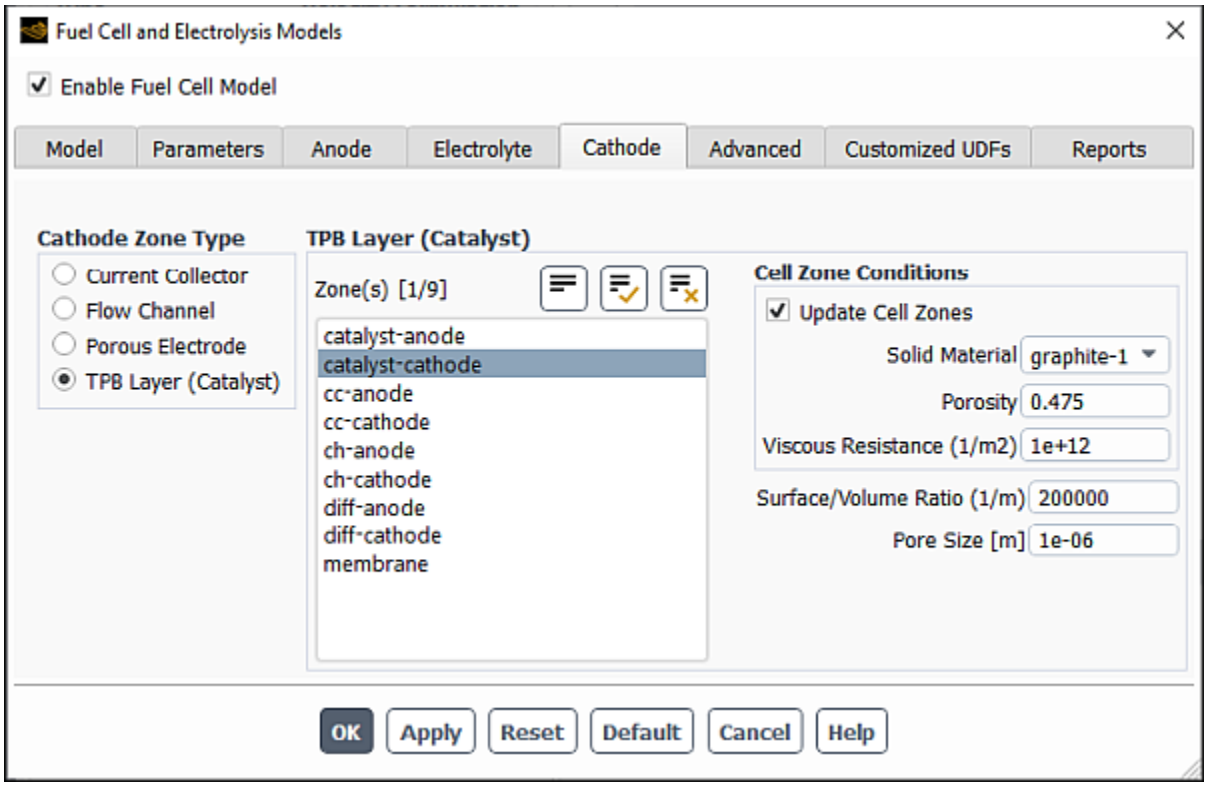


1. Select the **Cathode** tab of the **Fuel Cell and Electrolysis Models** dialog box.
2. Select **Porous Electrode** under **Cathode Zone Type**.
3. Select a corresponding zone from the **Zone(s)** list. If you are modeling a fuel cell stack, then you must pick *all* zones of a particular type as a group.
4. Select a **Solid Material** from the corresponding drop-down list. Solid materials can be customized using the **Create/Edit Materials** dialog box. Note that for the **Electrical Conductivity**, you can only choose a constant value in the **Create/Edit Materials** dialog box. The solid electrical conductivity value is the diffusivity of the solid phase potential in the solid zones.
5. Specify a value for the **Porosity**.
6. Specify a value for the **Viscous Resistance**.
7. Specify a value for the **Contact Angle** for multiphase fuel cell calculations (θ_c in Equation 20.86).
8. Specify the average **Pore Size** (r_e in Equation 20.105).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.2.6.5.4. Specifying Catalyst Layer Properties for the Cathode

Figure 32.29: The Cathode Tab of the Fuel Cell and Electrolysis Models Dialog Box With TPB Layer (Catalyst) Selected



1. Select the **Cathode** tab of the **Fuel Cell and Electrolysis Models** dialog box.
2. Select **TPB Layer(Catalyst)** under **Cathode Zone Type**.
3. Select a corresponding zone from the **Zone(s)** list. If you are modeling a fuel cell stack, then you must pick *all* zones of a particular type as a group.
4. Select a **Solid Material** from the corresponding drop-down list. Solid materials can be customized using the **Create/Edit Materials** dialog box. Note that for the **Electrical Conductivity**, you can only choose a constant value in the **Create/Edit Materials** dialog box. The solid electrical conductivity value is the diffusivity of the solid phase potential in the solid zones.
5. Specify a value for the **Porosity**.
6. Specify a value for the **Viscous Resistance**.
7. Specify a value for the **Surface/Volume Ratio** (the specific active surface area in Equation 20.67).
8. Specify a value for the **Contact Angle** for multiphase fuel cell calculations (θ_c in Equation 20.86).
9. Specify the average **Pore Size** (r_e in Equation 20.105).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.2.6.5.5. Specifying Cell Zone Conditions for the Cathode

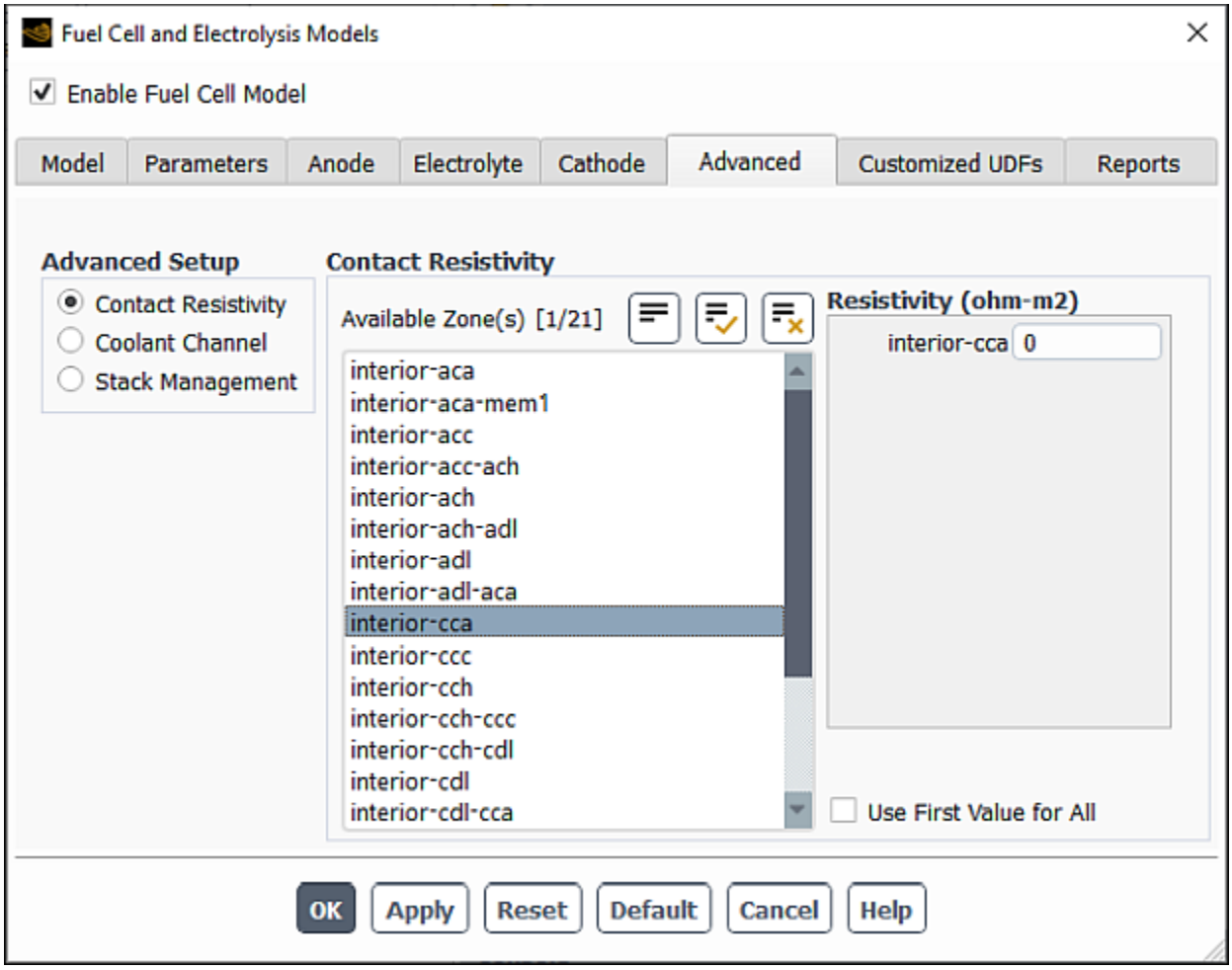
For each case of the cathode's current collector, diffusion layer, and catalyst layer, you assign a solid material and/or set the porosity and the viscous resistance. These settings represent setting a cell zone condition. With the **Update Cell Zones** option turned on (the default setting), this cell zone condition is applied to all selected zones in the **Zone(s)** list. If you want to set the cell zone conditions for each zone individually (using the **Cell Zone Conditions** task page), you should turn off the **Update Cell Zones** option.

32.2.6.6. Setting Advanced Properties (Advanced Tab)

You can use the **Advanced** tab of the **Fuel Cell and Electrolysis Models** dialog box to specify the contact resistivity for any material interface in the geometry, set parameters for coolant channels, and define fuel stack units for managing stacks of fuel cells.

32.2.6.6.1. Setting Contact Resistivities for the Fuel Cell and Electrolysis Model

Figure 32.30: The Advanced Tab of the Fuel Cell and Electrolysis Models Dialog Box for Contact Resistivities



1. Select the **Advanced** tab of the **Fuel Cell and Electrolysis Models** dialog box.
2. Select **Contact Resistivity** under **Advanced Setup**.

3. Select any number of corresponding interfaces from the **Available Zone(s)** list. These zones are face zones over which a jump in electrical potential is caused by imperfect conduction.

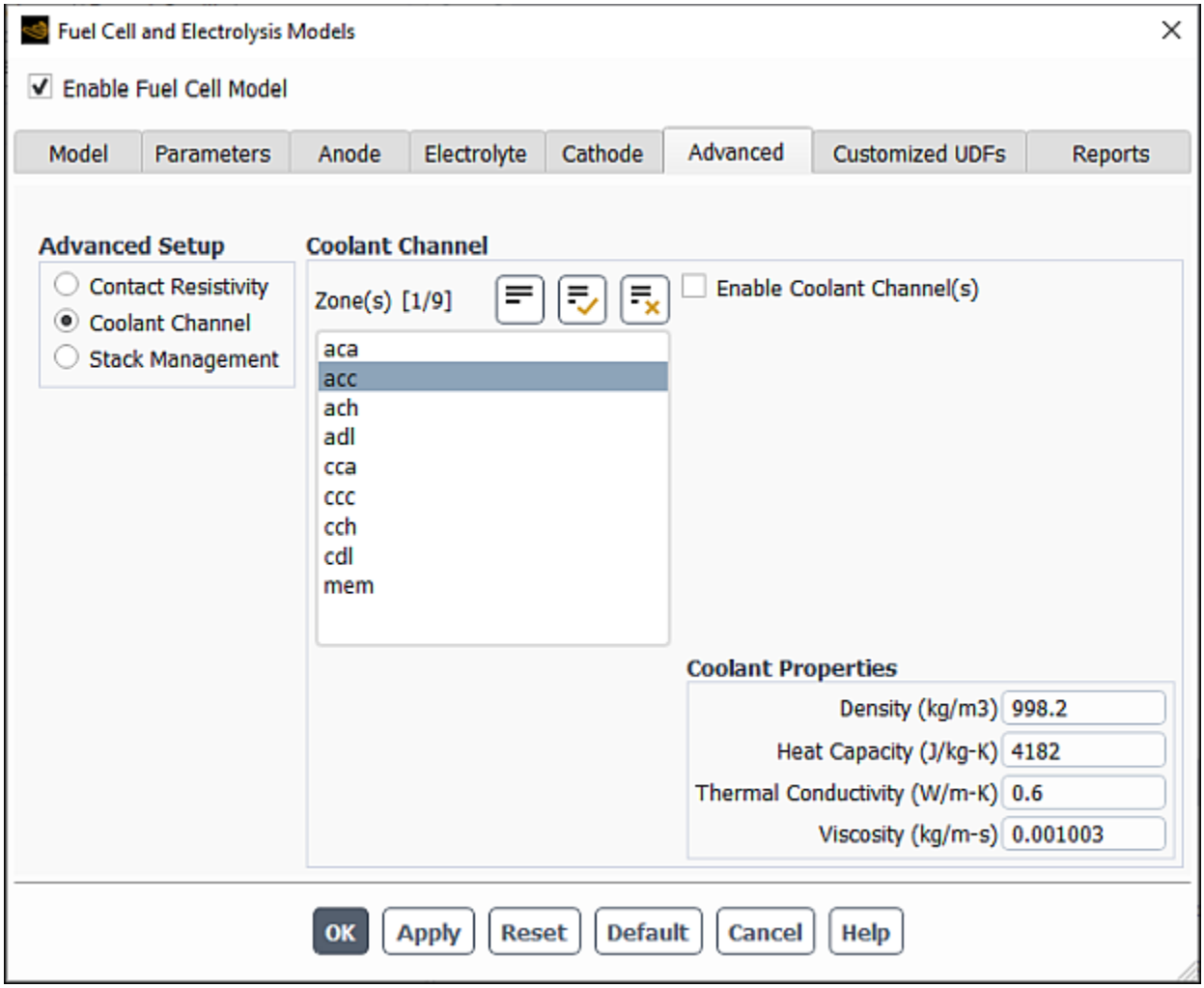
4. Specify a value for the **Resistivity** for each specified zone.

   Note that at porous jump and wall/wall-shadow interfaces, the contact resistivity is applied at each face. Therefore, you need to specify the resistivity only at one of the interface walls, otherwise the actual resistance will be doubled at the interface.

5. To simplify the input, you can choose to use the resistivity value of the first selected zone for all others as well by turning on the **Use First Value for All** option.

32.2.6.6.2. Setting Coolant Channel Properties for the Fuel Cell and Electrolysis Model

Figure 32.31: The Advanced Tab of the Fuel Cell and Electrolysis Models Dialog Box for the Coolant Channel



1. Select the **Advanced** tab of the **Fuel Cell and Electrolysis Models** dialog box.

2. Select **Coolant Channel** under **Advanced Setup**.

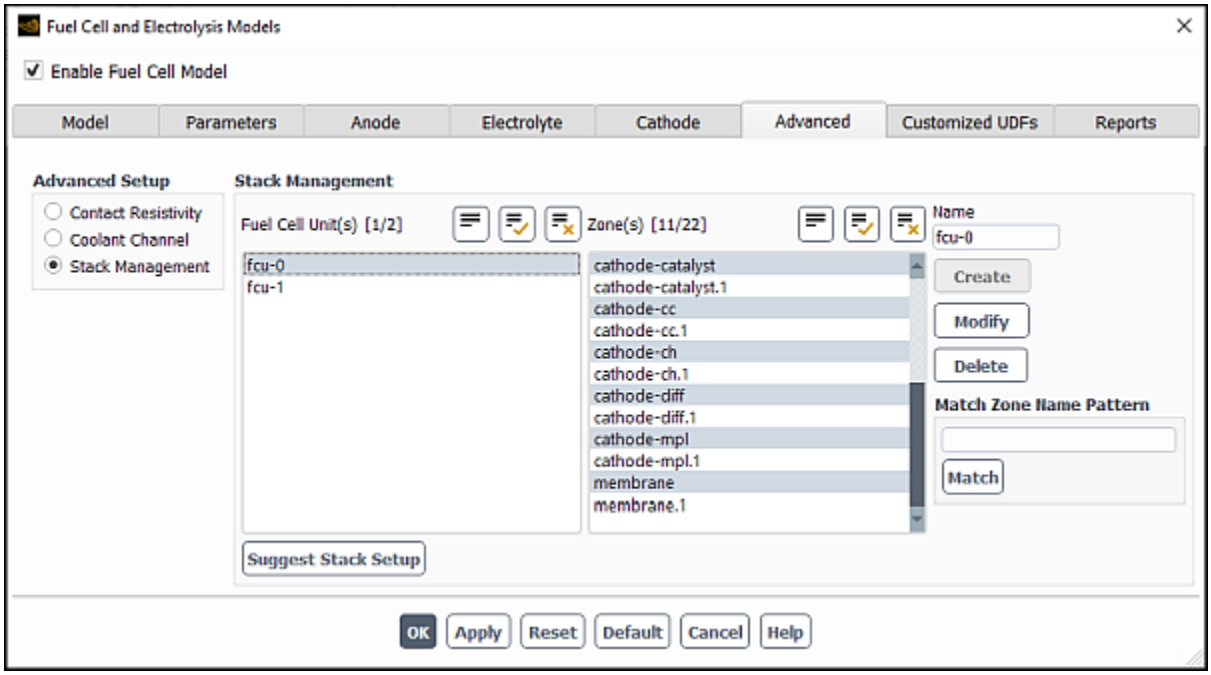
3. Select any number of corresponding zones from the **Zone(s)** list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Specify a value for the **Density**.
5. Specify a value for the **Heat Capacity**.
6. Specify a value for the **Thermal Conductivity**.
7. Specify a value for the **Viscosity**.
8. To enable the coolant channel, turn on the **Enable Coolant Channel(s)** option. Amongst other settings, this will change the mixture to include the coolant species, which is otherwise absent.

32.2.6.6.3. Managing Stacks for the Fuel Cell and Electrolysis Model

Figure 32.32: The Advanced Tab of the Fuel Cell and Electrolysis Models Dialog Box for Stack Management



The Ansys Fluent Fuel Cell and Electrolysis Model allows you to model fuel cell stacks as well as individual fuel cells. In the **Advanced** tab of the **Fuel Cell and Electrolysis Models** dialog box, you can define *fuel cell units* for each fuel cell in a stack. A fuel cell unit consists of all zones of a single fuel cell in the stack.

Important:

If you are only modeling a single fuel cell, then you do not need to set anything for **Stack Management** in the **Advanced** tab of the **Fuel Cell and Electrolysis Models** dialog box.

---

1. Select the **Advanced** tab of the **Fuel Cell and Electrolysis Models** dialog box.
2. Select **Stack Management** under **Advanced Setup**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Since a fuel cell unit consists of all zones of a single fuel cell in the stack, select the corresponding zones from the **Zone(s)** list.

4. Create a new fuel cell unit by clicking the **Create** button. The new fuel cell is listed under **Fuel Cell Unit(s)** with a default name.

5. Edit a pre-existing fuel cell unit by selecting it in the **Fuel Cell Unit(s)** list. The zones in this fuel cell unit are automatically selected in the **Zone(s)** list. You can then modify the zones that make up the fuel cell unit and/or change its name in the **Name** field and click **Modify** to save the new settings.

6. Remove a pre-existing fuel cell unit by selecting it in the **Fuel Cell Unit(s)** list and clicking the **Delete** button.

7. If your model contains many zone names, you can use the **Match Zone Name Pattern** field to specify a pattern to look for in the names of zones. Type the pattern in the text field and click **Match** to select (or deselect) the zones in the **Zones** list with names that match the specified pattern. You can match additional characters using `*` and `?`. For example, if you specify `wall*`, all surfaces whose names begin with **wall** (for example, **wall-1**, **wall-top**) will be selected automatically. If they are all selected already, they will be deselected. If you specify `wall?`, all surfaces whose names consist of **wall** followed by a single character will be selected (or deselected, if they are all selected already).

   For example, in a stack there are many fuel cells, say 10–100, each having at least 9 zones (current collector, gas channel, diffusion layer, and catalyst layer for both anode and cathode and a membrane). Additionally, there may be coolant channels, and it may be that for mesh construction reasons each of these physical zones is made up of more than one mesh zone. Even for small stacks, you can easily end up having hundreds of cell zones in an Ansys Fluent mesh. Therefore, you may want to consider numbering the fuel cells in a stack and to use the assigned fuel cell number in the names of the mesh zones. When you set up your stacked fuel cell case, you would use the **Match Zone Name Pattern** field to pick all the zones belonging to a single fuel cell in the stack, rather than scrolling through the potentially very long list and selecting them manually.

8. You can have Ansys Fluent attempt to automatically determine the zones that constitute a single fuel cell in a stack using the **Suggest Stack Setup** button. Manually performing this task is often time-consuming and error-prone. Using the **Suggest Stack Setup** button can save you from having to manually enter this information yourself for potentially hundreds of zones.

   When using the **Suggest Stack Setup** button, Ansys Fluent needs to correctly identify electrically conducting parts and their connectivity (anode, electrolyte, cathode, coolant channels, and external contacts) using zone information generally required by the fuel cell model anyway. Ansys Fluent requires zone information to have been specified in *all* of the following tabs in the **Fuel Cell and Electrolysis Models** dialog box:

   - In the **Anode** tab, specify zone information for the current collector, the porous electrode, and the TPB catalyst layer.

   - In the **Electrolyte** tab, specify zone information for the electrolyte/membrane.

   - In the **Cathode** tab, specify zone information for the current collector, the porous electrode, and the TPB catalyst layer.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- In the **Advanced** tab, specify zone information for the coolant channel.
- In the **Reports** tab, specify the external contact interface(s).
- In the **Advanced** tab, click the **Suggest Stack Setup** button.

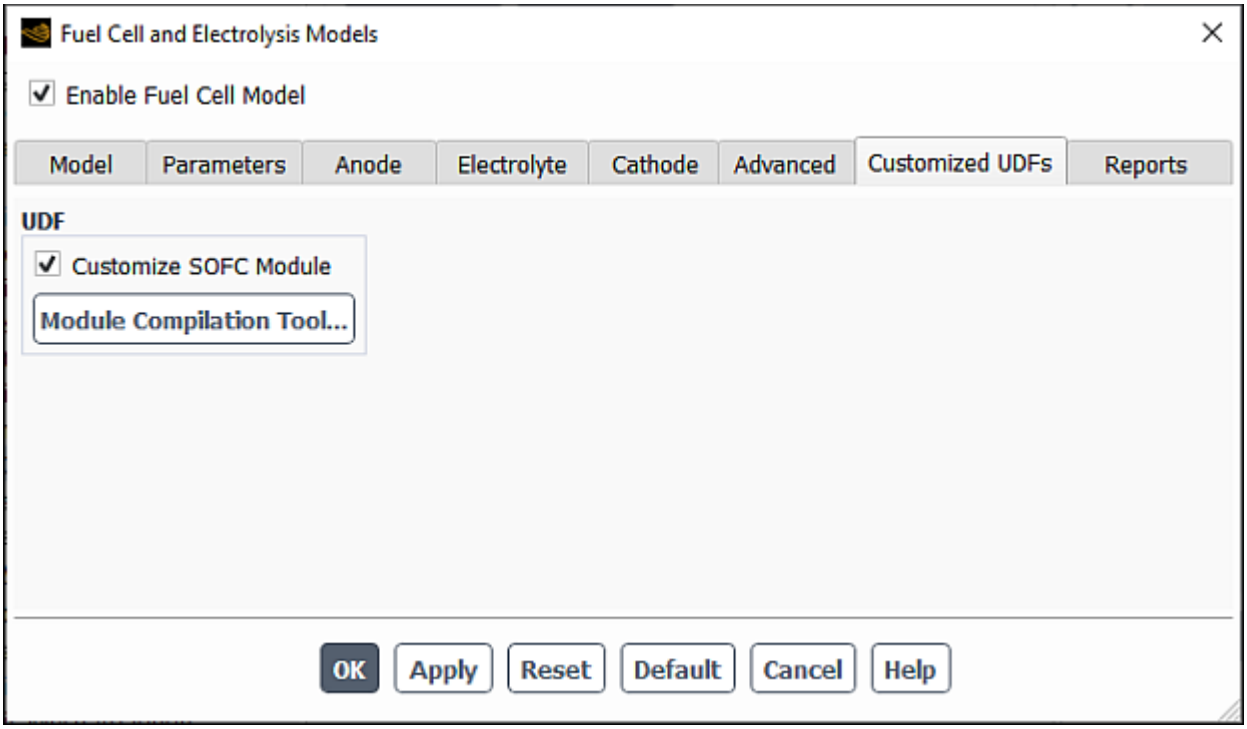
Important:

If your fuel cell model inputs are incorrect (or incomplete), Ansys Fluent cannot completely be sure that the resulting setup is correct. Also, even if your inputs are correct, an unconventional fuel cell design may cause Ansys Fluent to suggest an inaccurate stack setup. So, it is your responsibility to verify the stack setup prior to clicking either the **OK** or the **Apply** buttons.

32.2.6.7. Customizing the Fuel Cell and Electrolysis Module

In the **Customized UDFs** tab, you can compile your user-defined function (UDF) in order to customize the Fuel Cell and Electrolyses module.

Figure 32.33: The Customized UDFs Tab in the Fuel Cell and Electrolysis Models Dialog Box



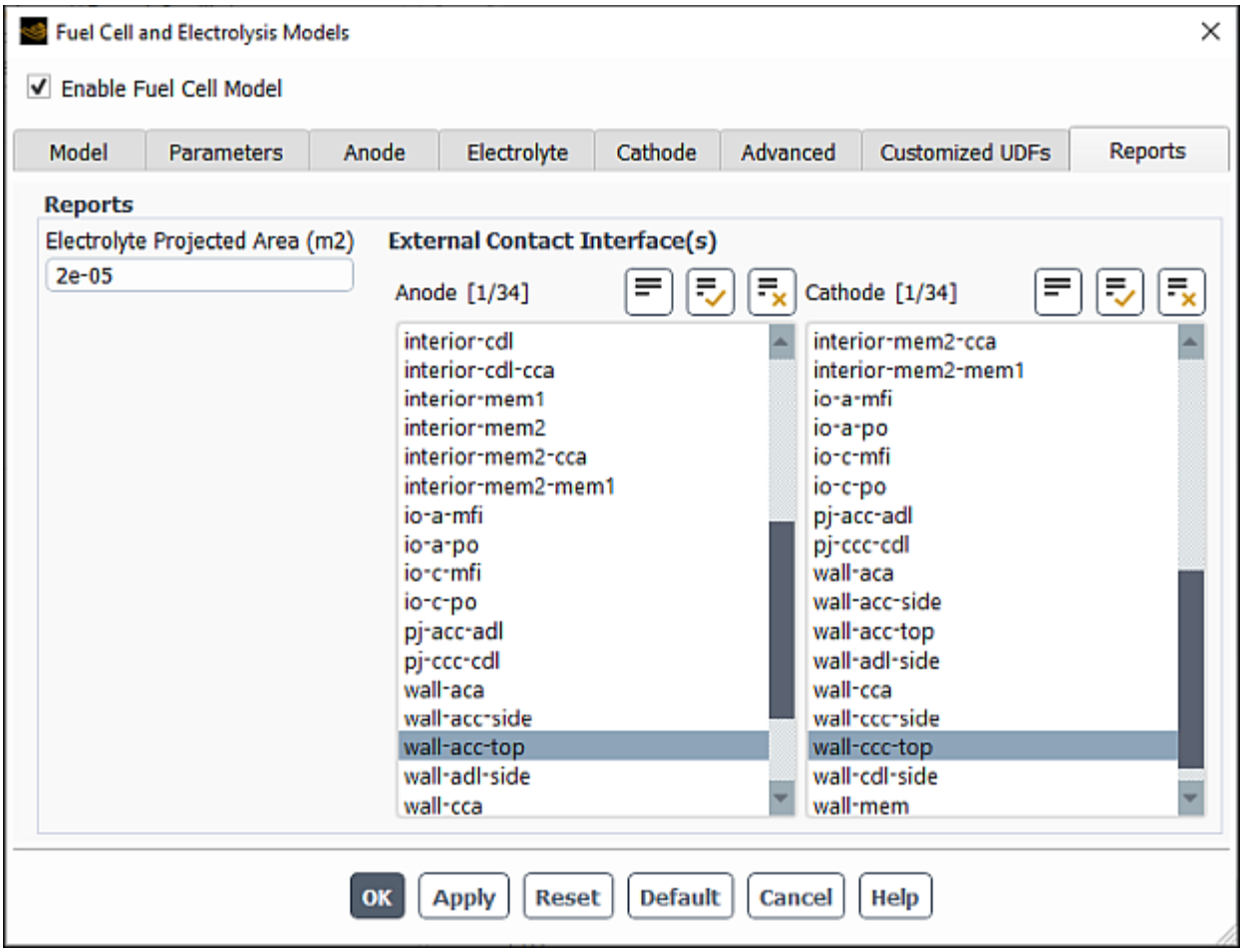
See Customizing the PEM Fuel Cell Module (p. 3434) for details on how to compile your UDF.

32.2.6.8. Reporting on the Solution (Reports Tab)

You can use the **Reports** tab of the **Fuel Cell and Electrolysis Models** dialog box to set up parameters that will be useful in reporting data relevant to the fuel cell.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 32.34: The Reports Tab of the Fuel Cell and Electrolysis Models Dialog Box



The **Electrolyte Projected Area** field requires the projected area of the Membrane Electrolyte Assembly (MEA) and is only used to calculate the average current density. The assembly consists of the membrane and the catalyst layers above and below the membrane. The value of the projected area can be computed from the **Projected Surface Areas** dialog box.

Results → Reports → Projected Areas Edit...

The **External Contact Interface(s)** fields requires the face zones that act as external contact surfaces for the anode and the cathode.

These inputs are used to report cell voltage. For potentiostatic boundary conditions, this is the difference between the provided values, but for galvanostatic boundary conditions, the cell voltage is part of the solution.

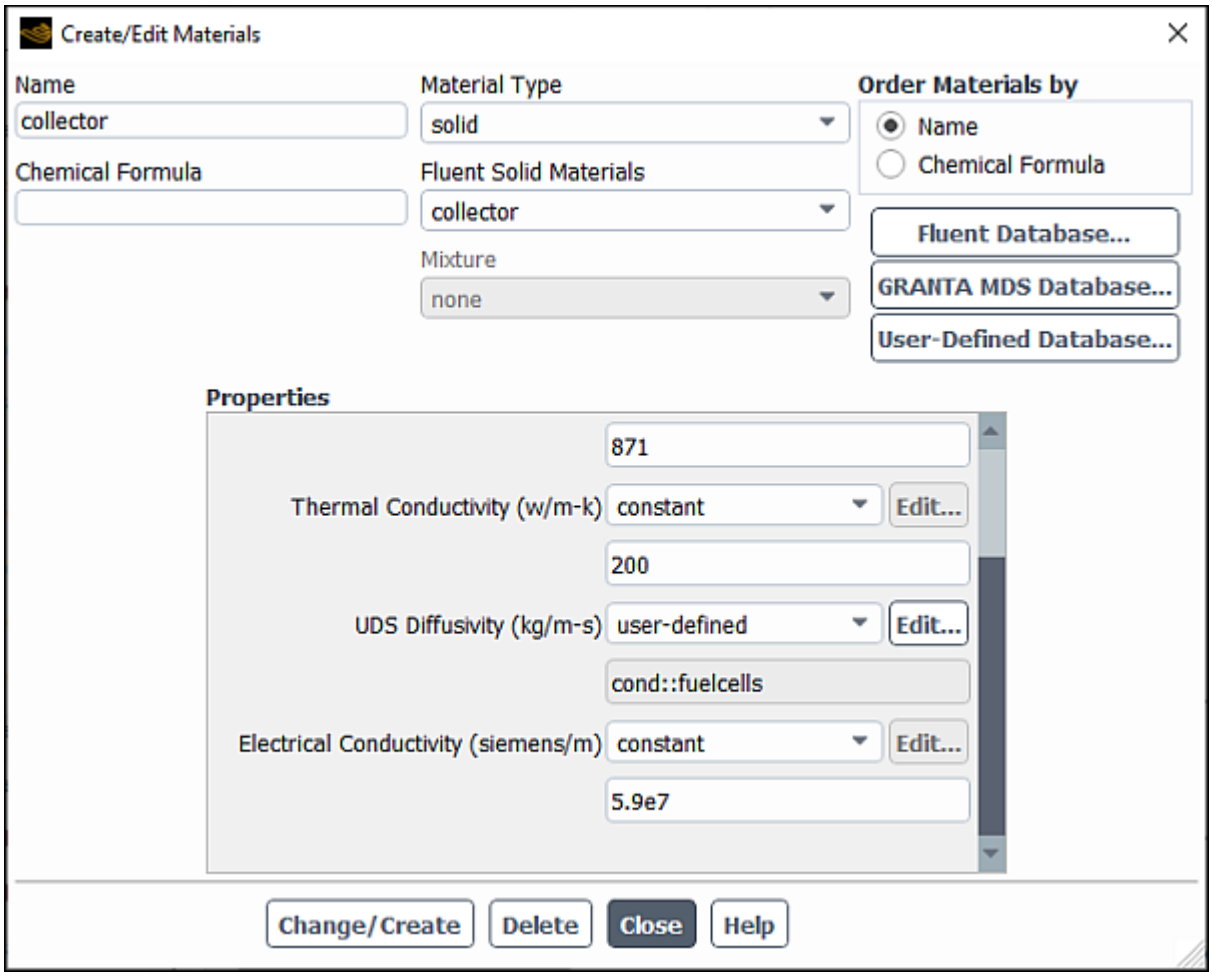
32.2.7. Modeling Current Collectors

In previous versions of Ansys Fluent, user-defined scalar (UDS) equations could only be solved in fluid zones. This restriction is now removed. As a result, the Fuel Cell and Electrolysis module allows you to model current collectors as solid, as well as fluid zones. One advantage of using solids as the current collector is that the convergence of the species equations are not hindered by the potentially skewed mesh inside the current collectors.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If fluid zones are used to model solid current collectors, Ansys Fluent automatically sets velocities to zero and cuts off species transport into these zones. If solid zones are used, however, you need to activate the solution of the electric potential (UDS-0) in these solid zones (see the separate Ansys Fluent User's Guide for details). The value of the **Electrical Conductivity** for the solid material must be assigned in the **Create/Edit Materials** dialog box.

Figure 32.35: The Electric Conductivity Field in the Create/Edit Materials Dialog Box



Important:

Note that the **UDS Diffusivity** should be set to `user-defined (cond::fuelcells)`. Do not use the `defined-per-uds` option.

For more information on the user-defined scalar diffusivity, see the separate Ansys Fluent User's Guide (p. 1).

Note that the Fuel Cell and Electrolysis Model allows you to model current collectors either as porous media zones (if you want to allow for mass and momentum transport within the collectors) or as solid zones.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.2.8. Fuel Cell and Electrolysis Model Boundary Conditions

The following boundary conditions need to be defined for the Fuel Cell and Electrolysis simulation based on your problem specification:

- Anode Inlet
  - Mass flow rate
  - Temperature
  - Direction specification method
  - Mass fractions (for example, **h2**, and **h2o**).
  - The coolant *must* be set to zero if coolant channels are enabled.
  - UDS-2 (Water Saturation) must be set to `0`
- Cathode Inlet
  - Mass flow rate
  - Temperature
  - Direction specification method
  - Mass fractions (for example **o2**, **h2o**, and **n2**).
  - The coolant *must* be set to zero if coolant channels are enabled.
  - UDS-2 (Water Saturation) must be set to `0`
- Coolant Inlet (if any)
  - Mass flow rate
  - Temperature
  - Direction specification method
  - Coolant mass fraction set to `1`
  - UDS-2 (Water Saturation) must be set to `0`
- Pressure Outlets (all)

  Realistic backflow conditions.
- Terminal Anode
  - Temperature (or flux if known)
  - UDS-0 (electric potential) set to ground voltage

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Terminal Cathode
  - Temperature (or flux if known)
  - UDS-0 (electric potential) is set to the voltage of the cathode (if solving to constant voltage), or the UDS-0 (electric potential) flux is set to the current density in A/m^2(SI units) (if solving for constant current). Note that the sign of the UDS-0 flux on the cathode side is negative.

32.2.9. Solution Guidelines for the Fuel Cell and Electrolysis Model

For potentiostatic boundary conditions, after initialization, steady-state solutions are calculated easily for cell voltages close to the open-circuit voltage. The same can be said for galvanostatic boundary conditions and low electric current. By lowering the cell voltage or by raising the average electric current, you can calculate subsequent stationary solutions.

In the event of convergence problems, it is recommended that you change the multigrid cycle to **F-cycle** with **BCGSTAB** (bi-conjugate gradient stabilized method) selected as the stabilization method for the species and the two potential equations. For the species and the user-defined scalar equations, it may be necessary to reduce the termination (criteria) of the multigrid-cycles to 1×10^{-3}. For stack simulations, the termination criterion may be reduced to $\times10^{-7}$ for the two potential equations.

Also, it may be useful to turn off **Joule Heating** and **Reaction Heating** in the **Fuel Cell and Electrolysis Models** dialog box (in the **Model** tab) for the first few (approximately 5-10) iterations after initialization. This allows the two electric potentials to adjust from their initial values to more physical values, avoiding the possibility of extreme electrochemical reactions and electric currents that would in turn adversely impact the solution.

32.2.10. Postprocessing the Fuel Cell and Electrolysis Model

You can perform postprocessing using standard Ansys Fluent quantities and by using user-defined scalars and user-defined memory allocations. By default, the Ansys Fluent Fuel Cell and Electrolysis Model defines several user-defined scalars and user-defined memory allocations, described in Table 32.4: User-Defined Scalar Allocations (p. 3475) and Table 32.5: User-Defined Memory Allocations (p. 3475).

Table 32.4: User-Defined Scalar Allocations

UDS Name	Description
UDS 0	Electric Potential (solid phase potential) (Volts)
UDS 1	Protonic Potential (membrane phase potential) (Volts)
UDS 2	Water Saturation (liquid saturation)
UDS 3	Water Content

Table 32.5: User-Defined Memory Allocations

UDM Name	Description
UDM 0	X Current Flux Density (A/m^2)
UDM 1	Y Current Flux Density (A/m^2)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UDM 2	Z Current Flux Density (A/m^2)
UDM 3	Current Flux Density Magnitude (A/m^2)
UDM 4	Ohmic Heat Source (W/m^3)
UDM 5	Reaction Heat Source (W/m^3)
UDM 6	Overpotential (Volts)
UDM 7	Phase Change Source (kg/m^3-s)
UDM 8	Osmotic Drag Coefficient
UDM 9	Liquid Water Activity
UDM 10	Membrane Water Content
UDM 11	Protonic Conductivity (1/ohm-m)
UDM 12	Back Diffusion Mass Source (kg/m^3-s)
UDM 13	Transfer Current (A/m^3)
UDM 14	Osmotic Drag Source (kg/m^3-s)

You can obtain this list by opening the **Execute On Demand** dialog box and pulling down the **Function** drop-down list.

User Defined → **User Defined** → **Execute on Demand...**

and access the execute-on-demand function called `list_pemfc_udf`.

Alternatively, you can view the listing that appears when you first load your Fuel Cell and Electrolysis case, or you can type `list_pemfc_udf` in the text user interface and the listing will appear in the console window.

Note:

- For field variables that are stored in UDM, use the corresponding variables for postprocessing. Postprocessing the UDM itself is not recommended.
- For reviewing simulation results in CFD-Post for cases involving UDM or UDS, export the solution data as CFD-Post-compatible file (`.cdat`) in Fluent and then load this file into CFD-Post.

Important:

When you load older Fuel Cell and Electrolysis cases into Ansys Fluent, and you are monitoring a UDS using volume or surface monitors, make sure you re-visit the corresponding monitors dialog box (for example, the **Volume Monitor** or the **Surface Monitor** dialog box) to verify that the correct UDS name is used for the appropriate monitor.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.2.11. User-Accessible Functions

As noted in Properties, you can directly incorporate your own formulations and data for the properties of the fuel cell membrane using the `pem_user.c` source code file.

The following listing represents a description of the contents of the `pem_user.c` source code file:

`real Get_P_sat(real T, cell_t c, Thread *t)`

Returns the value of the water vapor saturation pressure as a function of temperature (Equation 20.98) or other user-specified values.

`real Water_Activity(realP, realT, cell_t c, Thread *t)`

Returns the value of water activity (Equation 20.96).

`real Water_Content(real act)`

Returns the value of the membrane water content at the membrane catalyst interface (Equation 20.95).

`real Osmotic_Drag_Coefficient(real P, real T, cell_t c, Thread *t)`

Returns the value of the osmotic drag coefficient (Equation 20.92).

`real Membrane_Conductivity(real lam, cell_t c, Thread *t)`

Returns the value of the membrane's protonic conductivity (Equation 20.91).

`real Electrolyte_Conductivity(cell_t c, Thread *t)`

Returns the value of the ionic conductivity in the electrolyte (Equation 20.90). (SOFC and Electrolysis only)

`real Water_Content_Diffusivity(real lam, real T, real mem_mol_density,cell_tc,Thread*t)`

Returns the value of the water content diffusivity in the membrane (Equation 20.94).

`real Gas_Diffusivity(cell_tc, Thread *t, int j_spe)`

Returns the value of the gaseous species diffusivities in the channels, gas diffusion layers and catalysts (Equation 20.87).

`real MCD_Gas_Diffusivity(cell_t c, Thread *t, int i)`

Returns the tortuosity-corrected value of the gas species diffusion coefficients computed with the multicomponent diffusion option (Equation 20.89).

`real Saturation_Diffusivity(real sat, real cos_theta, real porosity, cell_t c, Thread *t)`

Returns the value of diffusivity of the liquid saturation. It computes the term $\rho_l \frac{Ks3}{\mu_l} \frac{dp_c}{ds}$ from Equation 20.83.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

real Anode_AV_Ratio(cell_t c, Thread *t)

Returns the value of the specific active surface area (ζ_{an} in Equation 20.66) for the anode catalyst.

real Cathode_AV_Ratio(cell_t c, Thread *t)

Returns the value of the specific active surface area (ζ_{cat} in Equation 20.67) for the cathode catalyst.

real Anode_J_TransCoef(cell_t c, Thread *t)

Returns the value of the anode reaction reference current density $\frac{\zeta_{an} j_{an}^{ref}}{\left([H_2]_{ref}\right)^{\gamma_{an}}}$ used in Equation 20.66.

real Cathode_J_TransCoef(cell_t c, Thread *t)

Returns the value of the cathode reaction reference current density $\frac{\zeta_{cat} j_{cat}^{ref}}{\left([O_2]_{ref}\right)^{\gamma_{cat}}}$ used in Equation 20.67.

real Open_Cell_Voltage(cell_t c, Thread *t)

Returns the value of the open-circuit voltage V_{oc} used in Equation 20.71.

real Leakage_Current(cell_t c, Thread *t)

Returns the value of the leakage current (I_{leak} in Equation 20.99 through Equation 20.104).

void Set_UDS_Names(char uds[n_uds_required][STRING_SIZE])

Used to rename user defined scalars (UDSs). Note that the units of the user defined scalars cannot be changed.

```
void Set_UDS_Names(char uds[n_uds_required][STRING_SIZE])
 {
      strncpy(uds[0], "Electric Potential", STRING_SIZE-1);
      strncpy(uds[1], "Protonic Potential", STRING_SIZE-1);
      strncpy(uds[2], "Water Saturation",  STRING_SIZE-1);
      strncpy(uds[3], "Water Content", STRING_SIZE-1);
 }
```

If you want to change the names of UDSs, change the second argument of the `strncpy` functions, recompile and link the module as with any modification to `pem_user.c`. Note that `STRING_SIZE` is fixed in `pem.h` and should not be changed.

Important:

When you load older Fuel Cell and Electrolysis cases into Ansys Fluent, and you are monitoring a UDS using volume or surface monitors, make sure you re-visit the corresponding monitors dialog box (for example, the **Volume Monitor** or the **Surface Monitor** dialog box) to make sure that the correct UDS name is used for the appropriate monitor.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

void Set_UDM_Names(char udm[n_udm_required][STRING_SIZE])

Used to rename user defined memory (UDMs). Note that the units of user defined memory cannot be changed.

```
void Set_UDM_Names(char udm[n_udm_required][STRING_SIZE])
 {
     strncpy(udm[0], "X Current Flux Density",  STRING_SIZE-1);
     strncpy(udm[1], "Y Current Flux Density",  STRING_SIZE-1);
     strncpy(udm[2], "Z Current Flux Density",  STRING_SIZE-1);
     strncpy(udm[3], "Current Flux Density Magnitude", STRING_SIZE-1);
     strncpy(udm[4], "Ohmic Heat Source", STRING_SIZE-1);
     strncpy(udm[5], "Reaction Heat Source",  STRING_SIZE-1);
     strncpy(udm[6], "Overpotential", STRING_SIZE-1);
     strncpy(udm[7], "Phase Change Source (PEM)", STRING_SIZE-1);
     strncpy(udm[8], "Osmotic Drag Coefficient (PEM)", STRING_SIZE-1);
     strncpy(udm[9], "Liquid Water Activity (PEM)", STRING_SIZE-1);
     strncpy(udm[10], "Membrane Water Content (PEM)",  STRING_SIZE-1);
     strncpy(udm[11], "Protonic Conductivity", STRING_SIZE-1);
     strncpy(udm[12], "Back Diffusion Source (PEM)", STRING_SIZE-1);
     strncpy(udm[13], "Transfer Current",  STRING_SIZE-1);
     strncpy(udm[14], "Osmotic Drag Source (PEM)", STRING_SIZE-1);
 }
```

If you want to change the names of UDMs, change the second argument of the `strncpy` functions, recompile and link the module as with any modification to `pem_user.c`. Note that `STRING_SIZE` is fixed in `pem.h` and should not be changed.

Important:

When you load older Fuel Cell and Electrolysis cases into Ansys Fluent, and you are monitoring a UDM using volume or surface monitors, make sure you re-visit the corresponding monitors dialog box (for example, the **Volume Monitor** or the **Surface Monitor** dialog box) to make sure that the correct UDM name is used for the appropriate monitor.

real electric_contact_resistance(face_t f, Thread *t, int ns)

Returns the value for the electrical contact resistance.

real Transfer_Current(real i_ref, real gamma, int species_i, real alpha_a, real alpha_c, real *dRade, real *dRcde, Thread *t, cell_t c)

Computes the transfer current (A/m^3), corresponding to R_{an} in Equation 20.66 and R_{cat} in Equation 20.67.

Inputs for this function include:

`i_ref`	effective transfer current coefficient, computed by `Cathode_J_TransCoef(c,t)` or `Anode_J_TransCoef(c,t)`
`gamma`	cathode or anode concentration exponent
`species_i`	species index used in fuel cells (for example `i_o2`, `i_h2`, `i_h2o`)
`alpha_a`	product of anode exchange coefficient and $\frac{F}{RT}$
`alpha_c`	product of cathode exchange coefficient and $\frac{F}{RT}$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

t	current thread
c	current cell

Outputs for this function include:

source	anode or cathode volumetric transfer current (R_{an} in Equation 20.66 or R_{cat} in Equation 20.67)
*dRade	partial derivative of R_{an} with respect to activation loss
*dRcde	partial derivative of R_{cat} with respect to activation loss

Thermal_ctk_pemfc(face_t f, Thread *t)

Used to change the constant value of **Thermal Contact Resistance** for the porous zone set in the **Porous Jump** dialog box to a locally variable value.

32.3. Using the Solid Oxide Fuel Cell With Unresolved Electrolyte Model

The procedure for setting up and solving solid oxide fuel cell (SOFC) problems (with unresolved electrolyte) is described in detail in this chapter. Refer to the following sections for more information:

32.3.1. Limitation on Modeling Solid Oxide Fuel Cells
32.3.2. Installing the Solid Oxide Fuel Cell With Unresolved Electrolyte Model
32.3.3. Loading the Solid Oxide Fuel Cell With Unresolved Electrolyte Module
32.3.4. Solid Oxide Fuel Cell With Unresolved Electrolyte Module Set Up Procedure
32.3.5. Setting the SOFC Model
32.3.6. User-Accessible Functions for the Solid Oxide Fuel Cell With Unresolved Electrolyte Model

32.3.1. Limitation on Modeling Solid Oxide Fuel Cells

The anisotropic species diffusivity option is not compatible with the Solid Oxide Fuel Cell model.

32.3.2. Installing the Solid Oxide Fuel Cell With Unresolved Electrolyte Model

The Solid Oxide Fuel Cell (SOFC) With Unresolved Electrolyte Model is provided as an addon module with the standard Ansys Fluent licensed software. The module is installed with the standard installation of Ansys Fluent in a directory called addons/sofc in your installation area. The SOFC With Unresolved Electrolyte Model consists of a UDF library and a pre-compiled Scheme library, that must be loaded and activated before calculations can be performed.

32.3.3. Loading the Solid Oxide Fuel Cell With Unresolved Electrolyte Module

The module can be loaded only after a valid Ansys Fluent mesh or case file has been set or read.

There are two different ways in which you can load the module into Ansys Fluent:

- Through the ribbon

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In the **Physics** ribbon tab, click **More** (**Models** group), and select **SOFC Model with Unresolved Electrolyte...**.

Physics → **Models** → **More** → **SOFC Model with Unresolved Electrolyte...**

Once the module is loaded into Ansys Fluent, the **SOFC Model** dialog box opens where you can enable and set the model as further described.

- Through the text user interface (TUI)

  The text command to load the addon module is

  `define` → `models` → `addon-module`

  A list of Ansys Fluent addon modules is displayed:

```
> /define/models/addon-module
Fluent  Addon Modules:
     0. None
     1. MHD Model
     2. Fiber Model
     3. Fuel Cell and Electrolysis Model
     4. SOFC Model with Unresolved Electrolyte
     5. Population Balance Model
     6. Adjoint Solver
     7. Single Potential Battery Module
     8. MSMD Battery Model
     9. PEM Fuel Cell Model
Enter Module Number: [0] 4
```

  Select the SOFC With Unresolved Electrolyte Model by entering the module number 4.

During the loading process, a Scheme library (containing the graphical and text user interface) and a UDF library (containing a set of user defined functions) are loaded into Ansys Fluent.

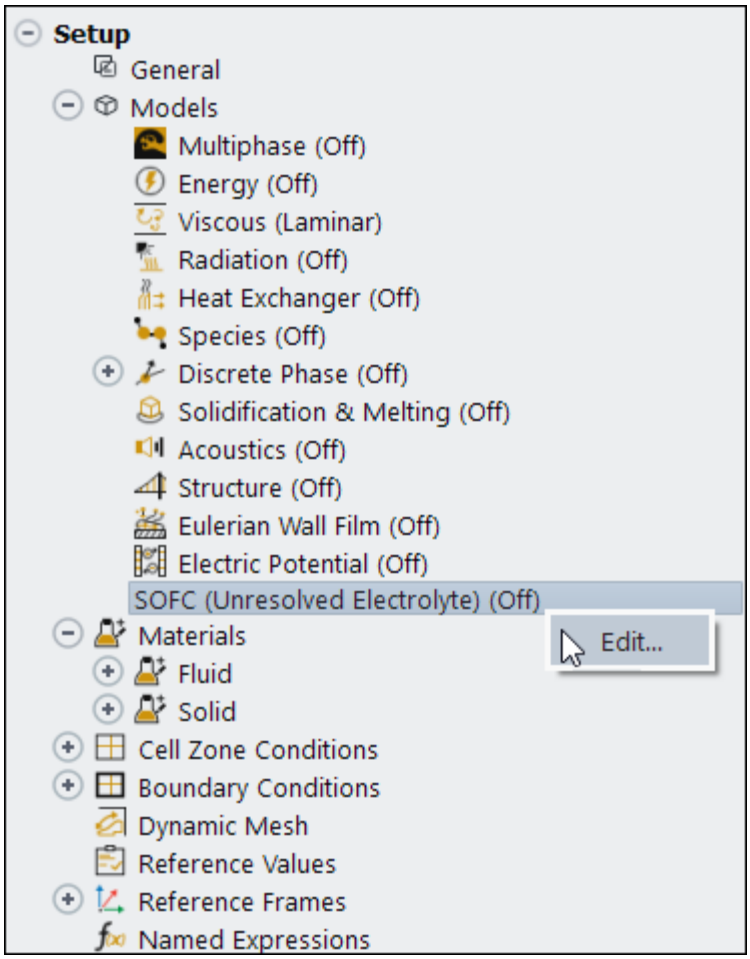
Once the module has been loaded, the **SOFC (Unresolved Electrolyte)** option appears in the tree under the **Models** branch and in the **Models** task page.

You can enable the SOFC model in the **SOFC Model** dialog which can be accessed either through the ribbon as described above or from the **Setup/Models/SOFC (Unresolved Electrolyte)** tree item.

Setup → **Models** → **SOFC (Unresolved Electrolyte)** **Edit...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 32.36: Opening the SOFC Model Dialog Box in the Outline View



When the **SOFC Model** dialog box opens, select the **Enable SOFC Model** option.

32.3.4. Solid Oxide Fuel Cell With Unresolved Electrolyte Module Set Up Procedure

The following describes an overview of the procedure required in order to use the SOFC With Unresolved Electrolyte Model in Ansys Fluent.

1. Start Ansys Fluent.

   You must start Ansys Fluent in 3D double-precision mode. Note that the SOFC With Unresolved Electrolyte Model is only available in 3D.

2. Read the case file.

   File → **Read** → **Case...**

3. Scale the mesh.

   Setup → **General** → **Scale...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Enable the **SOFC Model** and set its parameters as described in Setting the SOFC Model (p. 3485).

   Setup → Models → SOFC (Unresolved Electrolyte) Edit...

   When you initially enable the SOFC model, Ansys Fluent automatically applies the following options and settings:

   - **Pressure-Based Steady** solver with the **Absolute** velocity formulation
   - Energy equation
   - **Laminar** viscous model
   - **Species Transport** model with **Volumetric** reactions and the following settings:
     - **Inlet Diffusion**: disabled (see Diffusion at Inlets in the *Fluent Theory Guide*)
     - **Diffusion Energy Source**: enabled (see Treatment of Species Transport in the Energy Equation in the *Fluent Theory Guide*)
     - **Full Multicomponent Diffusion**: enabled (see Mass Diffusion in Laminar Flows in the *Fluent Theory Guide*)
     - **Thermal Diffusion**: enabled (see Mass Diffusion in Laminar Flows in the *Fluent Theory Guide*)
   - **F-cycle** for AMG solver for all equations
   - Metis method for parallel partition with Laplace smoothing

   Ansys Fluent also automatically creates the following materials (if they have not been defined in your case yet):

   - mixture material

     The new mixture material consists of the **h2o**, **o2**, **h2**, and **n2** species and uses the following definitions for the material properties:

     - **ideal-gas-mixing-law** for the **Thermal Conductivity** and the **Viscosity**
     - **diffusivity::sofc** user-defined function for **Mass Diffusivity**
     - **E_Conductivity::sofc** user-defined function for **UDS Diffusivity**
   - solid materials:
     - **anode-collector-default**
     - **anode-default**
     - **cathode-collector-default**
     - **collector-default**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can edit these material definitions or create new solid materials for your case.

Important:

Note that although the Ansys Fluent SOFC with Unresolved Electrolyte Model does support the shell conduction model that you use to take into account the transversal conductive heat inside the electrolyte material, it does not currently support a similarly transversal conduction of electric current inside the electrolyte.

You can adjust these settings as appropriate for your case. If you consider H_2/CO electrochemistry, ensure that your mixture material include CO and CO_2 as well.

5. Set the boundary conditions and operating conditions as appropriate for your case.

   Setup → **Boundary Conditions**

   Setup → **Boundary Conditions** → **Operating Conditions...**

6. Define the convergence criteria.

   Solution → **Monitors** → **Residuals** **Edit...**

   In the **Residual Monitors** dialog box, set the **Convergence Criterion** for all equations to `1e-08`.

7. Initialize the flow field.

   Solution → **Initialization** → **Initialize**

   Retain the default values for all parameters.

Descriptions of UDMs are listed in Table 32.6: User-Defined Memory Allocations (p. 3484).

The current density in `UDM-0` is a conservative flux of electricity (A/m^2). Performing an area integral over the electrolyte surface will sum to the total current (A). That value is computed at the electrolyte faces during the electric field solution. The values in `UDM-4` through `UDM-6` contain cell-centered values of the current density vector. These are not, and cannot be, conservative, so depending on the material conductivity and the geometric configuration, these can sometimes unavoidably produce values that do not match the UDM values. The same issues exist when interpolating velocity values to obtain the mass fluxes.

Table 32.6: User-Defined Memory Allocations

UDM Name	Description
`UDM-0`	Interface Current Density (A/m^2)
`UDM-1`	Nernst Potential (Volts)
`UDM-2`	Activation Overpotentail (Volts)
`UDM-3`	Volumetric Ohmic Source (W/m^3)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UDM-4	x Component of the Current Density (A/m^2)
UDM-5	y Component of the Current Density (A/m^2)
UDM-6	z Component of the Current Density (A/m^2)
UDM-7	Electrolyte Voltage Jump
UDM-8	Electrolyte Resistivity (Ohm-m)
UDM-9	Effective Electric Resistance
UDM-10	Anode Activation
UDM-11	Cathode Activation
UDM-12	Electrochemical Source (W/m^3)
UDM-13	Magnitude of Current Density (A/m^2)

Note that UDM-7 and UDM-8 are the linearized values that Ansys Fluent uses to solve the potential field and electrochemical coupling. They are necessary to the calculations but do not contain any real physical meaning.

Note that UDM-10 and UDM-11 contain the disaggregated activation polarizations at the anode and cathode.

By default, the Ansys Fluent SOFC With Unresolved Electrolyte Model defines a single user-defined scalar:

Table 32.7: User-Defined Scalar Allocations

UDS Name	Description
UDS-0	Electric Potential (Volts)

Note:

- For field variables that are stored in UDM, use the corresponding variables for post processing. Postprocessing the UDM itself is not recommended.
- For reviewing simulation results in CFD-Post for cases involving UDM or UDS, export the solution data as CFD-Post-compatible file (.cdat) in Fluent and then load this file into CFD-Post.

32.3.5. Setting the SOFC Model

The inputs for the SOFC model are entered in the **SOFC Model** dialog box using the following tabs:

- **Model Parameters**
- **Electrochemistry**
- **Electrolyte and Porous Zones**
- **Electric Field**
- **Customized UDFs**

A detailed description of the SOFC model inputs is given in the following sections:

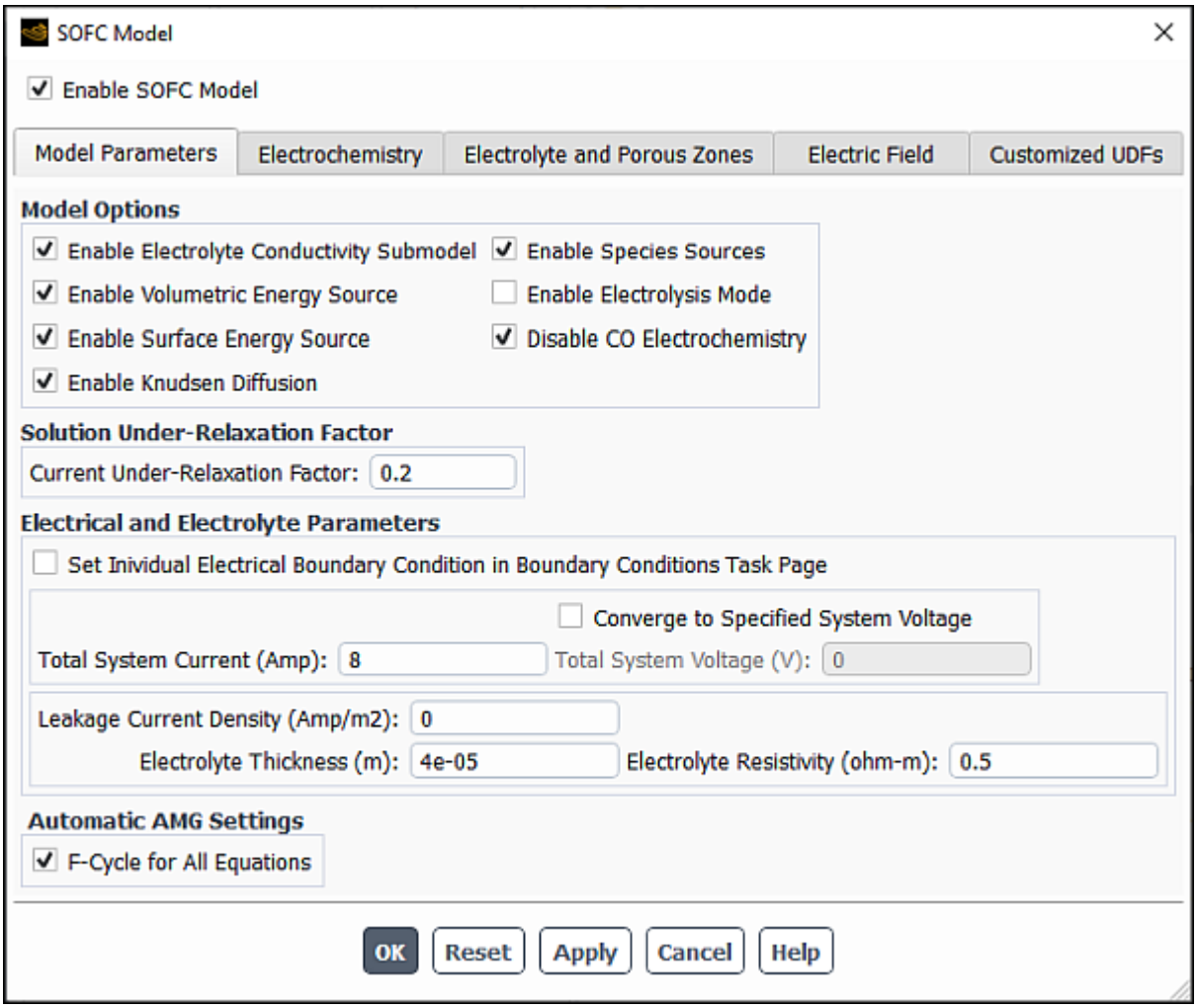
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

32.3.5.1. Setting the Parameters for the SOFC With Unresolved Electrolyte Model
32.3.5.2. Setting Up the Electrochemistry Parameters
32.3.5.3. Setting Up the Electrode-Electrolyte Interfaces
32.3.5.4. Setting Up the Electric Field Model Parameters
32.3.5.5. Customizing the SOFC Module

32.3.5.1. Setting the Parameters for the SOFC With Unresolved Electrolyte Model

In the **Model Parameters** tab, you can specify the general settings for the SOFC With Unresolved Electrolyte Model.

Figure 32.37: The Model Parameters Tab in the SOFC Model Dialog Box



From this tab, you can set the various parameters for the SOFC With Unresolved Electrolyte Model such as total system current, electrolyte thickness, electrolyte resistivity, and so on.

1. In the **Model Options** group box, select the appropriate SOFC model options. The following options are available:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Enable Electrolyte Conductivity Submodel

This option allows the ionic conductivity (or resistivity) of the electrolyte to change as a function of temperature. The correlation that provides ionic conductivity (or resistivity) of the electrolyte as function of temperature is expressed as:

$$resistivity = \frac{0.3685 + 0.002838 e^{\frac{10300}{T}}}{100} \tag{32.2}$$

Important:

Note that this is valid only for temperatures ranging from 1073 K to 1373 K.

This option should be selected if the electrolyte resistivity changes as a function of temperature.

Enable Volumetric Energy Source

This option includes the ohmic heating throughout the electrically conducting zones. You should keep this option turned off (to avoid slowing the convergence rates) until a certain rate of convergence for the potential field has been achieved, at which point, you should turn the option on manually. Note that this option is important so that the solution can account for the effects of the internal Ohmic heating.

Enable Surface Energy Source

When enabled, Ansys Fluent excludes the heat addition due to electrochemistry and all the reversible processes. This option must be turned on at all times.

Enable Knudsen Diffusion

When enabled, Ansys Fluent includes the effect of Knudsen diffusion in the calculation of the effective gas diffusivity. This option is useful when the Knudsen Number (the ratio of mean free path over average pore diameter, $D_{k,ij}$ in Equation 20.105 in the *Fluent Theory Guide*) in the porous media is much larger then 10. See Modeling Fluid Flow, Heat Transfer, and Mass Transfer in the *Fluent Theory Guide* for details.

Enable Species Sources

This option must be enabled on at all times.

Enable Electrolysis Mode

This option reverses the SOFC to produce hydrogen and oxygen from water vapor. Note the following:

- If you want to specify **Total System Current**, you must enter a negative value; however, if you specify a positive total current, Ansys Fluent will automatically change its sign internally.
- If you want to specify **Total System Voltage** (**Converge to Specified System Voltage** option), you must enter a value larger than the open-circuit voltage of the cell.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Disable CO Electrochemistry

This option must be enabled if there is carbon monoxide (CO) in the fuel line and if you do not want to include the CO in the electrochemistry. This option is unavailable with **Enable Electrolysis Mode**.

2. Set the **Current Under-Relaxation Factor** to a value of either `0.3` or `0.2`.

   These values are recommended for a more effective solution since the calculations are very sensitive to large current fluctuations early in the solution process.

3. Set the **Electrical and Electrolyte Parameters** according to your problem specification.

   The following parameters and options are available:

   - **Total System Current**
   - **Converge to Specified System Voltage**

     You can use this option when you want to specify a **Total System Voltage** instead of **Total System Current** as an input.

   - **Set Individual Electrical Boundary Condition from Boundary Conditions Task Page**

     When enabled, the option allows you to directly specify the current density or voltage for each individual current-collecting boundary using the Ansys Fluent **Boundary Conditions** task page. This allows you to apply multiple current types or multiple voltage types (either constant or variable) to your boundary conditions.

   - **Leakage Current Density**

     This is the total amount of current due to the leakage of oxidizer to the fuel side (through the electrolyte) and the electric current across the electrolyte due to any short circuit.

     If the leakage current density is temperature-dependent, you can specify your own temperature-dependent implementation of the leakage current density by performing the following steps:

     a. Make the required changes to `Leakage_Current_Density(real T)` which is a real function in the user-modifiable source code, `constit.c`. See User-Accessible Functions for the Solid Oxide Fuel Cell With Unresolved Electrolyte Model (p. 3494) for details.

     b. Compile `constit.c` and make your own `sofc` UDF library.

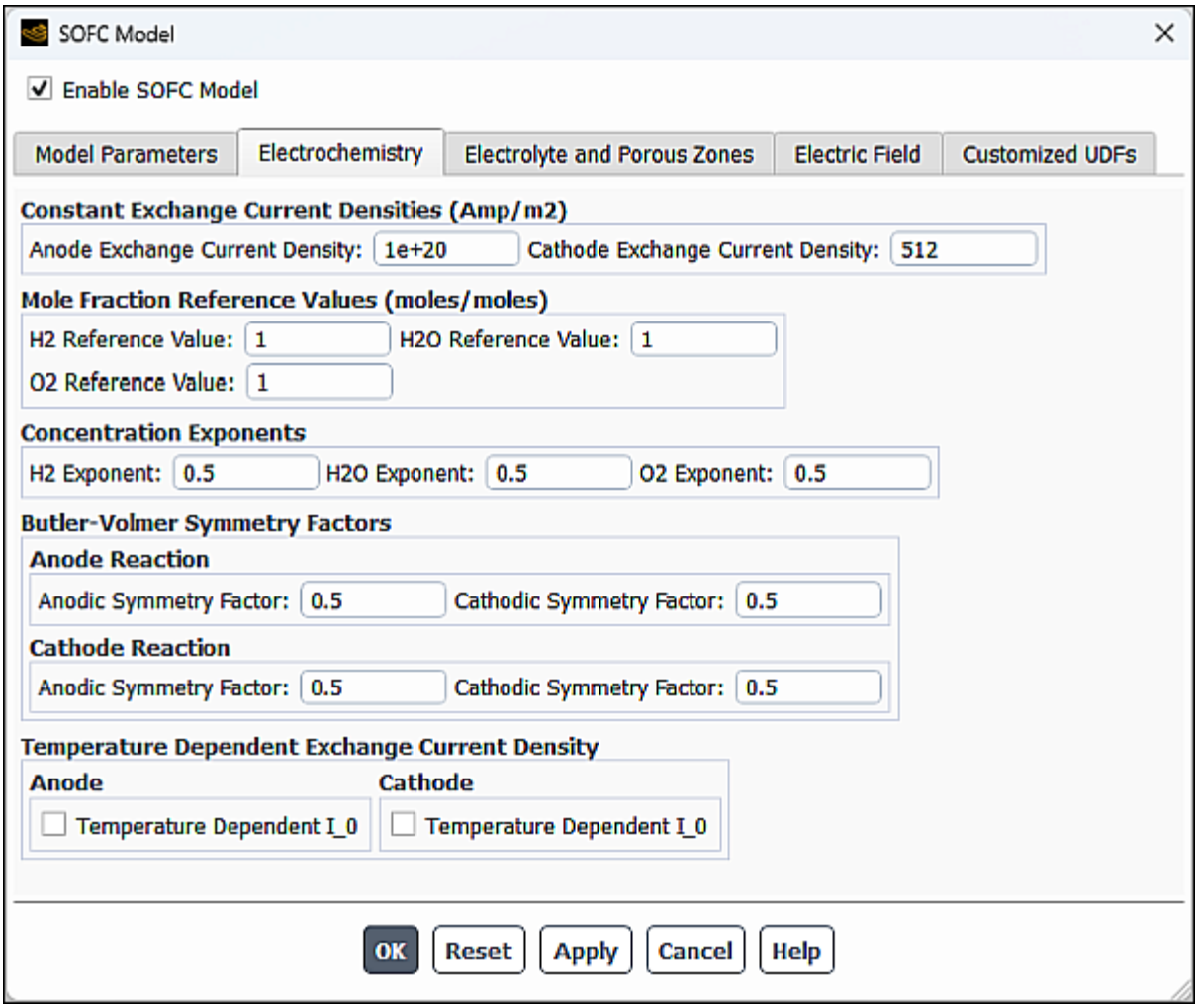
4. By default, the **F-Cycle for All Equations** option is enabled in the **Automatic AMG Settings** group box. Once you click **OK** or **Apply** in the **SOFC Model** dialog box, this option sets the multigrid cycle type to **F cycle** for all equations that are being solved. This is a recommended setting. However, you can adjust the equation cycle settings in the **Advanced Solution Controls** dialog box.

32.3.5.2. Setting Up the Electrochemistry Parameters

In the **Electrochemistry** tab, you can specify electrochemistry settings for the SOFC With Unresolved Electrolyte Model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 32.38: The Electrochemistry Tab in the SOFC Model Dialog Box



In the **Electrochemistry** tab, you can set the anode and cathode exchange current density, the anode and cathode mole fraction reference values, the concentration exponents, the Butler-Volmer symmetry factors, and the temperature-dependent exchange current density.

1. Enter values for the **Constant Exchange Current Densities** for the anode and the cathode. These are the i_0 values in the Butler-Volmer equation (Equation 20.118) for the anodic and cathodic reactions. By default, the **Anode Exchange Current Density** is set to `1e+10` Amps and the **Cathode Exchange Current Density** is set to `100` Amps.

2. Enter values for the **Mole Fraction Reference Values**.

   These are the species concentrations at which the exchange current densities were taken (Equation 20.119-Equation 20.121). These are used to adjust the i_0 values of reactant species that are depleted. By default, **H2 Reference Value**, **O2 Reference Value**, and **H2O Reference Value** are set to `1` moles/moles.

3. In the **Concentration Exponents** group box, enter values for the **H2 Exponent**, **H20 Exponent**, and **O2 Exponent** (defaulted to `0.5`). These are the stoichiometric factors in the electrochemical reaction equation. They are used as exponents as part of the i_0 scaling (Equation 20.120 and Equation 20.121.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Enter values for the **Butler-Volmer Symmetry Factors** by setting values for the **Anodic Transfer Coefficient** (α_a in Equation 20.118) and the **Cathode Transfer Coefficient** (α_c in Equation 20.118) for both the anode reaction and the cathode reaction. By default, these values are set to `0.5` because of the nearly universal assumption that there is a symmetric balance between the forward and backward reactions. In most cases, these default values will be sufficient. These are the alpha values in the Butler-Volmer equation (Equation 20.118). They represent the forward and backward rates of reaction at both the anode and cathode.

   If you find yourself changing the Butler-Volmer symmetry factors, or if you have some other rate-limiting reaction in your fuel cell simulation, you may also want to consider changing the exponents for the stoichiometric coefficients for the fuel cell reactants. These exponents can be specified in the **Concentration Exponents** group box.

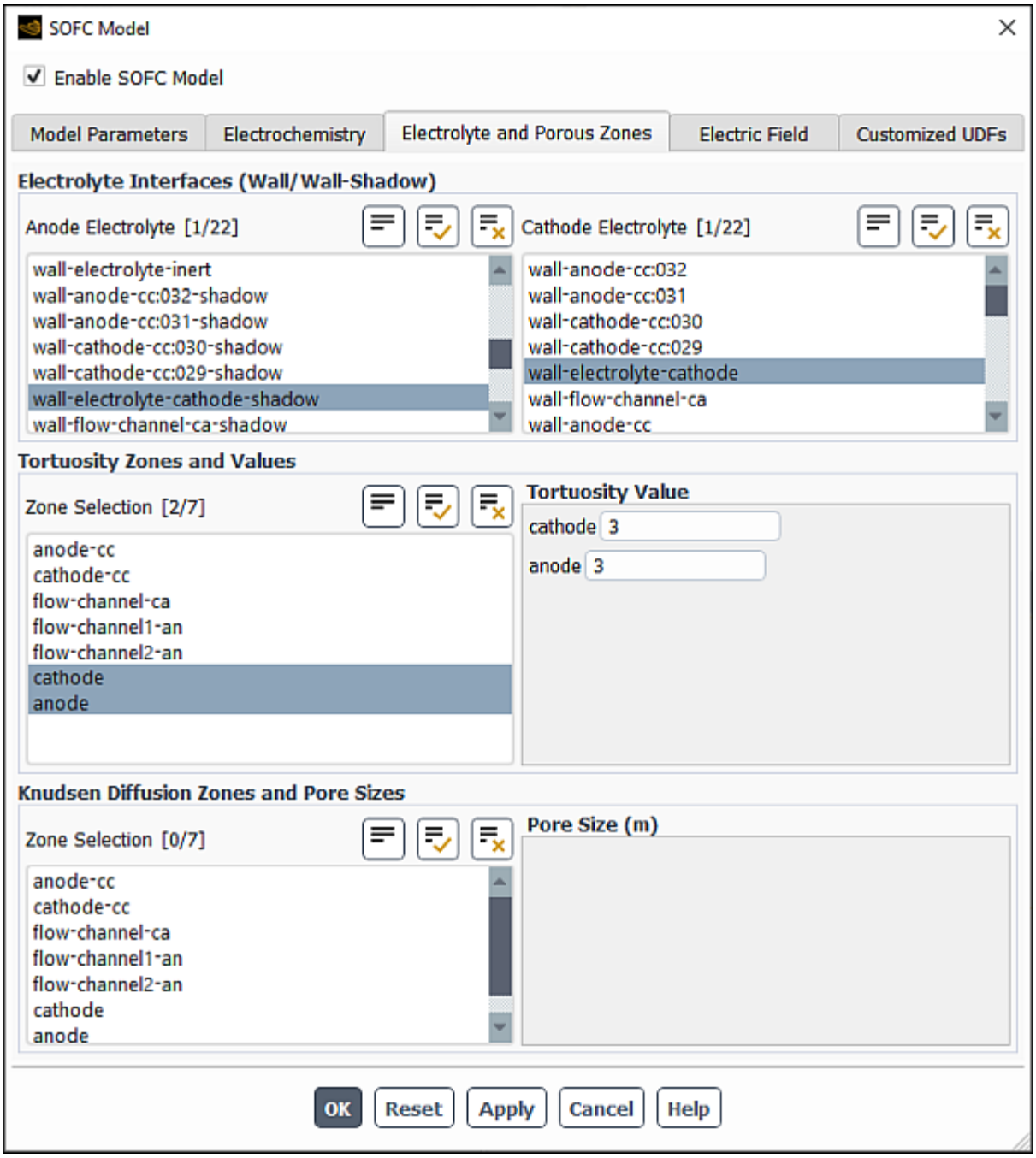
5. If you want to consider the temperature dependence of the exchange current density at the anode and/or cathode side, select **Temperature Dependent I_0** and specify **A** and **B** (rate coefficients in Equation 20.122 in the *Fluent Theory Guide*) at the anode and/or cathode (**Temperature Dependent Exchange Current Density** group box).

32.3.5.3. Setting Up the Electrode-Electrolyte Interfaces

In the **Electrode and Porous Zones** tab, you can set the anode and cathode interface components as well as the porous zones parameters for the SOFC With Unresolved Electrolyte Model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 32.39: The Electrolyte and Porous Zones Tab in the SOFC Model Dialog Box



1. In the **Anode Electrolyte** multiple-selection list, select zones that represent the anode electrode interface with the electrolyte.

2. In the **Cathode Electrolyte** multiple-selection list, select zones that represent the cathode electrode interface with the electrolyte.

3. Set up the tortuosity parameters.

   a. In the **Tortuosity Zones and Values** group box, select appropriate porous zones from the **Zone Selection** multiple-selection list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. As required, assign tortuosity values to any porous zones (under **Tortuosity Value**). In a porous zone, the mass diffusion coefficient is reduced as follows due to the porosity effect:

$$D_{eff} = \frac{porosity}{tortousity} D \tag{32.3}$$

There typically are no standard means of measuring tortuosity as it must be either measured experimentally or tuned to match other experimental data. Tortuosity value may typically be in the range of 2 to 4, although you can use much higher values.

4. When the **Enable Knudsen Diffusion** is selected in the **Model Parameters** tab of the **SOFC Model** dialog box, set up the Knudsen diffusion parameters.

   a. In the **Knudsen Diffusion Zones and Pore Sizes** group box, select appropriate porous zones from the **Zone Selection** multiple-selection list.

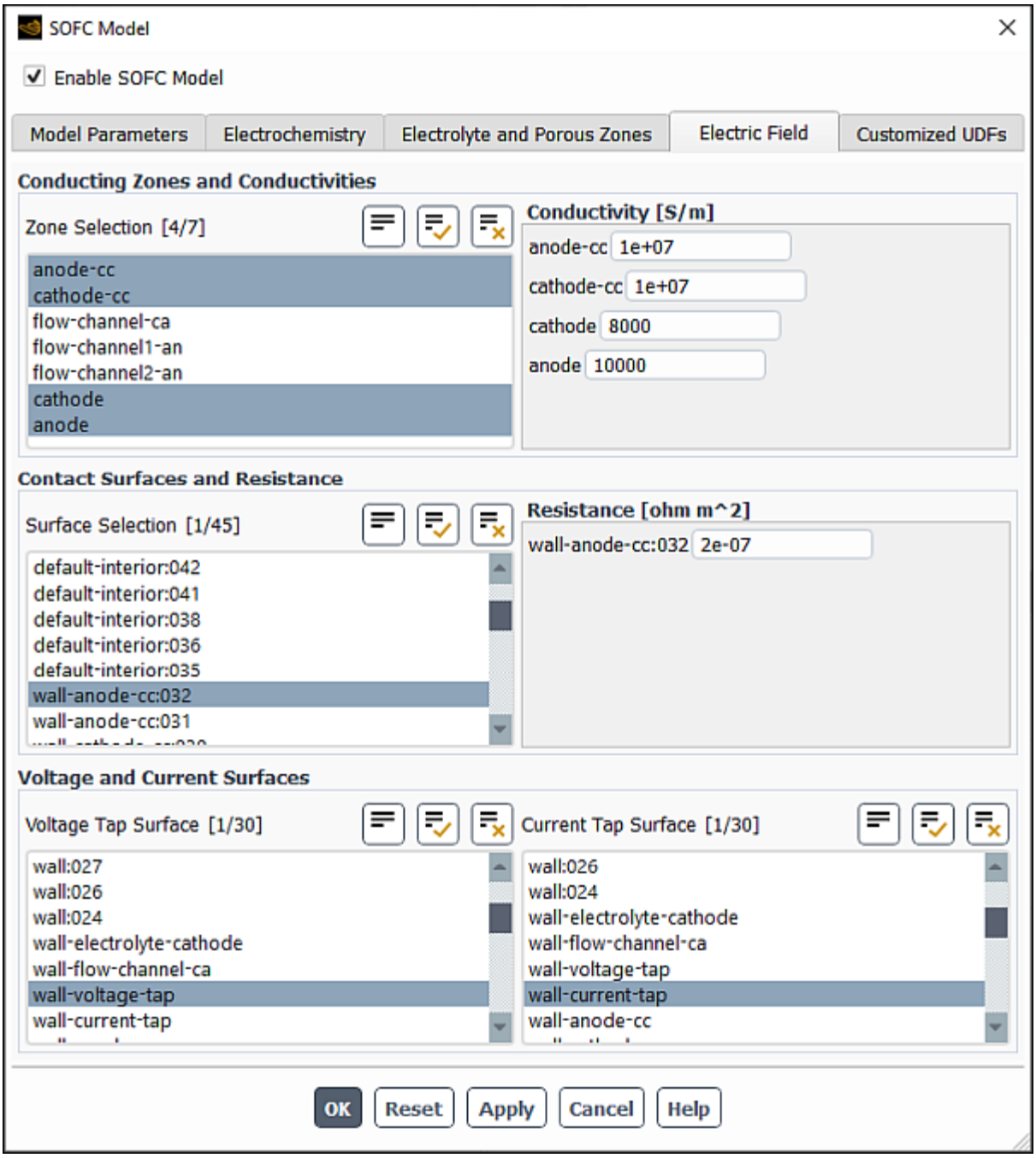
   b. Under **Pore Size** , assign an effective radius of the pore (r_e in Equation 20.105 in the *Fluent Theory Guide*) to each selected porous zone.

32.3.5.4. Setting Up the Electric Field Model Parameters

In the **Electric Field** tab, you can set the parameters for the electric field model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 32.40: The Electric Field Tab in the SOFC Model Dialog Box



Here, you can designate conductive regions and specify their conductivity, assign boundaries to be contact surfaces and specify the contact resistances, as well as specify which surfaces are grounded and which surfaces exhibit a current.

1. Select electrically conductive regions from the **Conducting Zones and Conductivities** multiple selection list, and for each region, set its **Conductivity**.

2. Select contact surfaces from the **Contact Surfaces and Resistance** multiple selection list and specify their **Resistance**.

3. Select a voltage tap surface.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Voltage Tap Surface** is the surface that is grounded (that is, $V=0$).

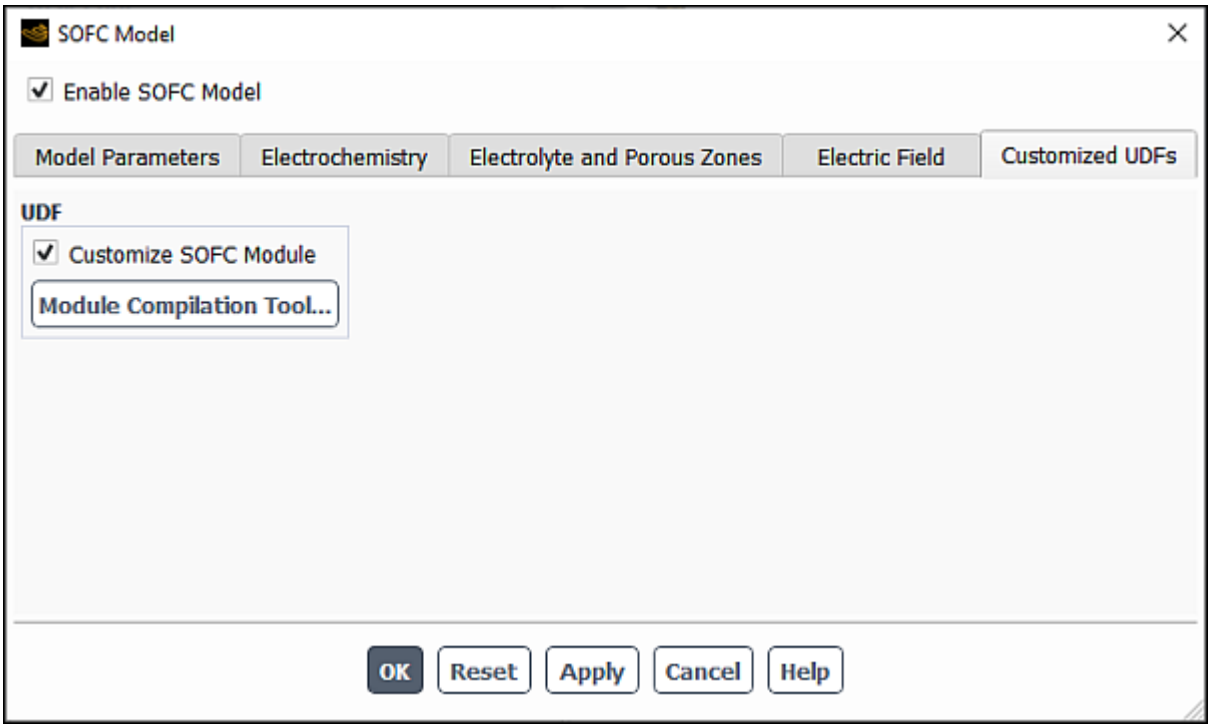
4. Select a current tap surface.

   The **Current Tap Surface** is the surface that current is being drawn from (that is, i is assigned a value).

32.3.5.5. Customizing the SOFC Module

In the **Customized UDFs** tab, you can select and compile your user-defined function (UDF) to customize the SOFC module.

Figure 32.41: The Customized UDFs Tab in the SOFC Model Dialog Box



See Customizing the PEM Fuel Cell Module (p. 3434) for details on how to compile your UDF.

32.3.6. User-Accessible Functions for the Solid Oxide Fuel Cell With Unresolved Electrolyte Model

You can directly incorporate your own formulations and data for the properties of the solid oxide fuel cell model using the `constit.c` source code file.

The following listing represents a description of the contents of the `constit.c` source code file:

```
real Nernst(real sp_a[], real sp_c[], real P_a, real P_c, real T_a, real
T_c, real *DNDY)
```

Returns the Nernst potential using Equation 20.114.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

real Activation(real Y[], real P, real T, real i, real i_0_ref, real alpha_a, real alpha_b, int species, real *dA_di)

Returns the activation overpotential by solving Equation 20.119.

real Resistivity(real T)

Returns the electrolyte resistance (either a constant value, or computed using Equation 32.2 (p. 3487)).

real Leakage_Current_Density(real T)

Returns the leakage current density specified in the **SOFC Model** dialog box and can be overwritten as a function of temperature.

real Solve_Butler_Volmer_NR(real i, i_0_ref, real A, real B, real *detadi);

Returns the activation overpotential by solving Equation 20.119.

real CONDUCTIVITY_CELL(cell_t c, Thread *t)

Returns the cell conductivity value used in the **Electric Field** tab of the **SOFC Model** dialog box. This function can be overwritten to look up the conductive zone group ID (1 through 5). A conditional statement allows you to perform various tasks for respective conductive zones. For non-conductive zones, this function returns a value of 0.

real CONTACT_RESIST_FACE(face_t f, Thread *t)

Returns the contact resistance value used in the **Electric Field** tab of the **SOFC Model** dialog box. This function can be overwritten to look up the contact resistance group ID (1 through 3). A conditional statement allows you to perform various tasks for respective contact resistance surfaces. For non-contact resistance surfaces, this function returns a value of 0.

real h2_co_split_func(cell_t c_an, Thread *t_an)

Returns the H_2/CO split factor defined in Equation 20.138. This can be overwritten as needed to define your own split factor.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 33: Modeling Magnetohydrodynamics

The Ansys Fluent Magnetohydrodynamics (MHD) Module Chapter tells you what you need to know to model magnetohydrodynamics with Ansys Fluent. Additional information about the model is provided in the following sections:

33.1. Introduction

33.2. Implementation

33.3. Using the Ansys Fluent MHD Module

33.4. Guidelines For Using the Ansys Fluent MHD Model

33.5. Definitions of the Magnetic Field

33.6. External Magnetic Field Data Format

33.1. Introduction

Magnetohydrodynamics refers to the interaction between an applied electromagnetic field and a flowing, electrically-conductive fluid. The Ansys Fluent MHD model enables you to analyze the behavior of electrically conducting fluid flow under the influence of constant (DC) or oscillating (AC) electromagnetic fields. The externally-imposed magnetic field may be generated either by selecting simple built-in functions or by importing a user-supplied data file. For multiphase flows, the MHD model is compatible with both the discrete phase model (DPM), the volume-of-fluid (VOF) and Eulerian Multiphase in Ansys Fluent, including the effects of a discrete phase on the electrical conductivity of the mixture.

This chapter describes the Ansys Fluent MHD model. Modeling Magnetohydrodynamics provides theoretical background information. Implementation (p. 3497) summarizes the UDF-based software implementation. Instructions for getting started with the model are provided in Using the Ansys Fluent MHD Module (p. 3499). Guidelines For Using the Ansys Fluent MHD Model (p. 3515) provides a condensed overview on how to use the MHD model, while Definitions of the Magnetic Field (p. 3518) contains definitions for the magnetic field, External Magnetic Field Data Format (p. 3518) describes the external magnetic field data format, and Magnetohydrodynamics Text Commands lists the text commands in the MHD model.

33.2. Implementation

The MHD model is implemented using user-defined functions (UDF) as an Ansys Fluent add-on module, which is loaded into Ansys Fluent at run time. The model is accessed through a number of UDF schemes. The magnetic induction equation given by Equation 21.14 or Equation 21.16 and the electric potential equation given by Equation 21.22 are solved through user-defined scalar (UDS) transport equations. Other model-related variables such as the external magnetic field data, current density, Lorentz force and Joule heat are stored as user-defined memory (UDM) variables. The MHD model setup and parameters are entered using the **MHD Model** graphical user interface (GUI) dialog box and a set of text user interface (TUI) commands described in Using the Ansys Fluent MHD Module (p. 3499). Detailed information can be found in the following sections:

33.2.1. Solving Magnetic Induction and Electric Potential Equations

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

33.2.2. Calculation of MHD Variables
33.2.3. MHD Interaction with Fluid Flows
33.2.4. MHD Interaction with Discrete Phase Model
33.2.5. General User-Defined Functions

33.2.1. Solving Magnetic Induction and Electric Potential Equations

The magnetic induction equation and the electric potential equations are solved through user-defined scalar transport equations. For the magnetic induction equation a set of 2 or 3 scalar equations are solved, each representing a Cartesian component of the induced magnetic field vector in a 2D or 3D case. For the electric potential equation a single scalar equation is solved.

The convection and the diffusion terms of the scalar equations are defined using user functions `DEFINE_UDS_FLUX(mhd_flux, ..., ns)` and `DEFINE_DIFFUSIVITY (mhd_ magnetic_diffusivity, ..., ns)` respectively. The user-defined scalar equation is identified by the scalar index `ns`.

The source terms to the induction equations and the potential equation are implemented using user function `DEFINE_SOURCE(mhd_mag_source, ..., eqn)` and `DEFINE_SOURCE (mhd_phi_source, ..., eqn)` respectively, where `eqn` identifies the scalar equations.

For transient cases, the additional unsteady source term is introduced through the user function `DEFINE_UDS_UNSTEADY(mhd_unsteady_source, ..., ns)`, where `ns` identifies the scalar being solved.

The induction and potential equations can also be solved in solid zones, in which case the fluid velocity terms in the equations are not considered. For multiphase flows, the MHD equations are solved in the mixture domain only.

The wall boundary conditions are implemented through user profile functions (`DEFINE_PROFILE(mhd_bc_...)`, and are applied to the Cartesian components of the induced magnetic field vector or to the electric potential. For external wall boundaries, three types of boundary conditions (electrically insulating, conducting, and "thin wall"), can be applied. The ‘thin wall’ type boundary refers to an external wall where a 1D magnetic or electric potential diffusion normal to the boundary is assumed, and the wall material and the thickness are specified for the boundary. For internal wall boundaries, that is the boundaries between fluid/solid or solid/solid zones, a coupled boundary condition is applied.

33.2.2. Calculation of MHD Variables

Apart from the Cartesian components of the magnetic field vectors and the electric potential function, which are stored as user-defined scalars, other MHD-related variables include the induced electric current density vector, induced electric field vector, the Lorentz force vector and Joule heat. These variables are stored in user-defined memory locations. Updating of MHD variables is accessed through the user function `DEFINE_ADJUST(mhd_adjust,...)`. The variables are updated at the start of each iteration using the solved induced magnetic field from the previous iteration.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

33.2.3. MHD Interaction with Fluid Flows

Additional source terms due to the magnetic induction are added to the flow momentum and energy equations as user defined source terms. For the momentum equation, user function `DEFINE_SOURCE(mhd_mom_source, ..., eqn)` is used to introduce the Lorentz force to the equation, where `eqn` identifies the Cartesian component of the fluid momentum. For the energy equation, the additional source due to Joule heating is added through user function `DEFINE_SOURCE(mhd_energy_source,..., eqn)`, where `eqn` is the energy equation index.

33.2.4. MHD Interaction with Discrete Phase Model

In discrete phase modeling, the Lorentz force acting on charged particles is introduced through the user function `DEFINE_DPM_BODY_FORCE(mhd_dpm_force, ...)`. User function `DEFINE_DPM_SOURCE(mhd_dpm_source, ...)` is used to update the volume fraction of the discrete phase inside a fluid cell and the volume-weighted electric conductivity of the discrete phase.

33.2.5. General User-Defined Functions

Several general UDFs are used as part of the MHD model implementation.

- `DEFINE_INIT(mhd_init,...)` is an initialization function called during the general case initialization to set up the external magnetic field and initialize MHD model parameters and variables.
- `DEFINE_ADJUST(mhd_adjust,...)` is called at the start of each iteration. It is used to adjust the magnetic boundary conditions and update MHD related variables and properties.

33.3. Using the Ansys Fluent MHD Module

This chapter provides basic instructions to install the magnetohydrodynamics (MHD) module and solve MHD problems in Ansys Fluent. It assumes that you are already familiar with standard Ansys Fluent features, including the user-defined function procedures described in the Fluent Customization Manual. Guidelines For Using the Ansys Fluent MHD Model (p. 3515) also outlines the general procedure for using the MHD model. This chapter describes the following:

33.3.1. MHD Module Installation
33.3.2. Loading the MHD Module
33.3.3. MHD Model Setup
33.3.4. MHD Solution and Postprocessing
33.3.5. Limitations

33.3.1. MHD Module Installation

The MHD module is provided as an add-on module with the standard Ansys Fluent licensed software. The module is installed in a directory called `addons/mhd` in your installation area. The MHD module consists of a UDF library and a pre-compiled Scheme library, which must be loaded and activated before calculations can be performed.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

33.3.2. Loading the MHD Module

The MHD module can be loaded into Ansys Fluent only when a valid Ansys Fluent case file has been set or read.

There are two different ways in which you can load the MHD module into Ansys Fluent:

- Through the ribbon

  In the **Physics** ribbon tab, click **More** (**Models** group), and select **MHD Model...**.

  Physics → **Models** → **More** → **MHD Model**

  Once the MHD module is loaded into Ansys Fluent, the **MHD Model** dialog box opens with the **Enable MHD** option automatically selected (see Figure 33.2: The MHD Model Dialog Box (p. 3503)).

- Through the text user interface (TUI)

  The text command to load the module is

  `define` → `models` → `addon-module`.

  A list of Ansys Fluent add-on modules is displayed:

```
> /define/models/addon-module
Fluent  Addon Modules:
     0. None
     1. MHD Model
     2. Fiber Model
     3. Fuel Cell and Electrolysis Model
     4. SOFC Model with Unresolved Electrolyte
     5. Population Balance Model
     6. Adjoint Solver
     7. Single Potential Battery Module
     8. MSMD Battery Model
     9. PEM Fuel Cell Model
Enter Module Number: [0] 1
```

  Select the MHD model by entering the module number 1.

  Once the MHD module is loaded into Ansys Fluent, you can enable the MHD model as described in Enabling the MHD Model (p. 3502).

During the loading process a Scheme library containing the graphical and text user interface, and a UDF library containing a set of user defined functions are loaded into Ansys Fluent. A message **Addon Module: mhd...loaded!** is displayed at the end of the loading process.

The basic set up of the MHD model is performed automatically when the MHD module is loaded successfully. The set up includes:

- Selecting the default MHD method
- Allocating the required number of user-defined scalars and memory locations naming of:
  - User-defined scalars and memory locations
  - All UDF Hooks for MHD initialization and adjustment

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - MHD equation flux and unsteady terms
  - Source terms for the MHD equations
  - Additional source terms for the fluid momentum and energy equations
  - Default MHD boundary conditions for external and internal boundaries
  - A default set of model parameters
- DPM related functions are also set, if the DPM option has been selected in the Ansys Fluent case set up.

The MHD module set up is saved with the Ansys Fluent case file. The module is loaded automatically when the case file is subsequently read into Ansys Fluent. Note that in the saved case file, the MHD module is saved with the absolute path. Therefore, if the locations of the MHD module installation or the saved case file are changed, Ansys Fluent will not be able to load the module when the case file is subsequently read.

To unload the previously saved MHD module library, open the **UDF Library Manager** dialog box

Parameters & Customization → **User Defined Functions** **Manage...**

and reload the module as described above. Note that the previously saved MHD model setup and parameters are preserved.

33.3.3. MHD Model Setup

The settings related to the MHD model are specified in the **MHD Model** dialog box, which can be accessed either via the tree:

Setup → **Models** → **MHD Model** **Edit...**

or using the text command:

`define` → `models` → `mhd-model`

Both the **MHD Model** dialog box and TUI commands are designed for the following tasks:

- Enable/disable the MHD model.
- Select the MHD method.
- Apply an external magnetic field.
- Set boundary conditions.
- Set solution control parameters.

Operations of these tasks through the **MHD Model** dialog box are described in the following sections. The set of MHD text commands are listed in Magnetohydrodynamics Text Commands .

For more information, see the following sections:

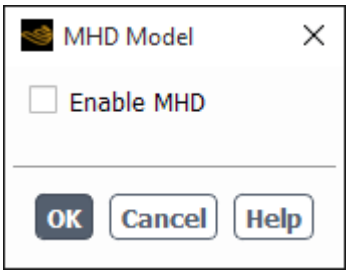
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

33.3.3.1. Enabling the MHD Model
33.3.3.2. Selecting an MHD Method
33.3.3.3. Applying an External Magnetic Field
33.3.3.4. Setting Up Boundary Conditions
33.3.3.5. Solution Controls

33.3.3.1. Enabling the MHD Model

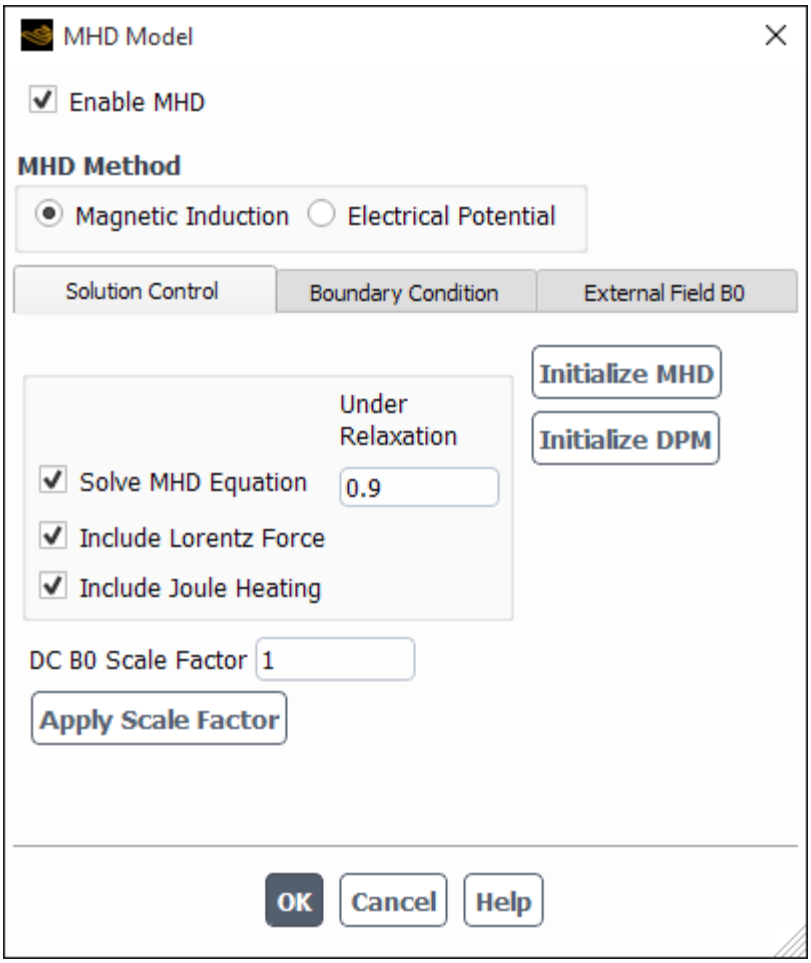
If the MHD model is not yet enabled after the MHD module is loaded, you can enable it by selecting **Enable MHD** in the **MHD Model** dialog box, shown in Figure 33.1: Enabling the MHD Model Dialog Box (p. 3502). The dialog box expands to its full size when the model is enabled, as shown in Figure 33.2: The MHD Model Dialog Box (p. 3503).

Figure 33.1: Enabling the MHD Model Dialog Box



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 33.2: The MHD Model Dialog Box



33.3.3.2. Selecting an MHD Method

The method used for MHD calculation can be selected under **MHD Method** in the **MHD Model** dialog box. The two methods, **Magnetic Induction** and **Electrical Potential**, are described in Magnetic Induction Method and Electric Potential Method,

For the Magnetic Induction method, 2 or 3 user-defined scalars are allocated for the solution of the induced magnetic field in 2D or 3D cases. The scalars are listed as B_x, B_y and B_z representing the Cartesian components of the induced magnetic field vector. The unit for the scalar is Tesla.

For the Electrical Potential method, 1 user-defined scalar is solved for the electric potential field. The scalar is listed as φ and has the unit of Volt.

Table 33.1: User-Defined Scalars in MHD Model (p. 3503) lists the user-defined scalars used by the two methods.

Table 33.1: User-Defined Scalars in MHD Model

Method	Scalar	Name	Unit	Description
Induction	Scalar-0	B_x	Tesla	X component of induced magnetic field (b_x)
	Scalar-1	B_y	Tesla	Y component of induced magnetic field (b_y)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

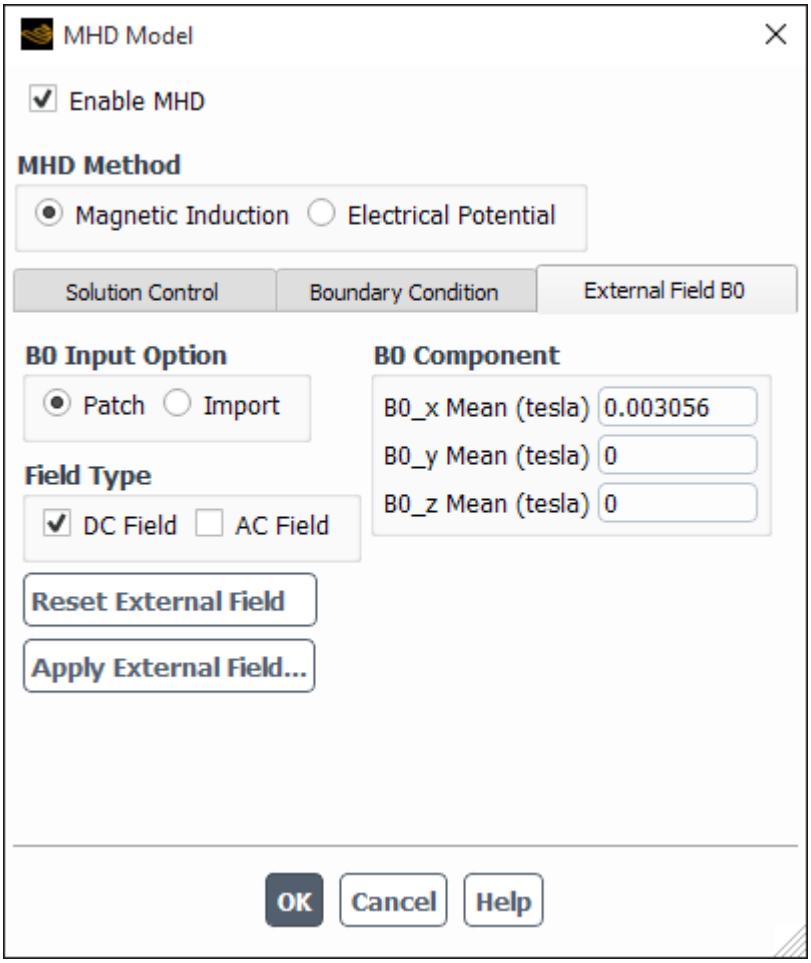
Method	Scalar	Name	Unit	Description
	Scalar-2 (3-D)	B_z	Tesla	Z component of induced magnetic field (b_z)
Potential	Scalar-0	Phi	Volt	Electric potential (φ)

33.3.3.3. Applying an External Magnetic Field

Application of an external magnetic field to the computation domain is done under the **External Field B0** tab in the **MHD Model** dialog box, as shown in Figure 33.3: The MHD Model Dialog Box for Patching an External Magnetic Field (p. 3504). Two **B0 Input Option**s are available for setting up the external magnetic field. One option is to **Patch** the computational domain with a constant (**DC Field**) and/or varying (**AC Field**) type. The other option is to **Import** the field data from a magnetic data file that you provide.

With the **Patch** option enabled, the AC field can be expressed as a function of time (specified by **Frequency**), and space (specified by wavelength, propagation direction and initial phase offset). The space components are set under the **B0 Component**, as in Figure 33.3: The MHD Model Dialog Box for Patching an External Magnetic Field (p. 3504).

Figure 33.3: The MHD Model Dialog Box for Patching an External Magnetic Field

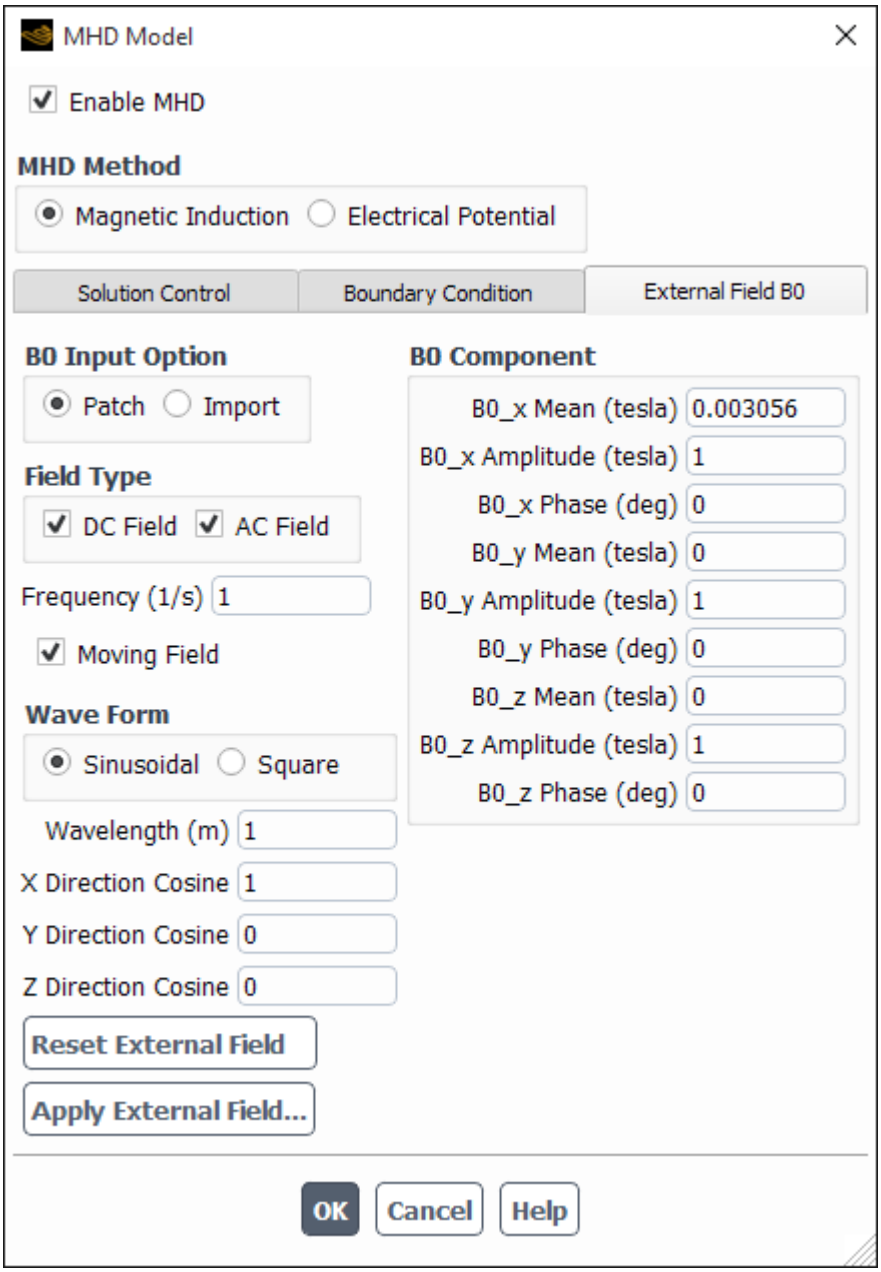


You can also specify a **Moving Field** with a wave form that is either a sinusoidal or a square wave function (Figure 33.4: The MHD Model Dialog Box for Specifying a Moving Field (p. 3505)). Definitions

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

for the sinusoidal and square wave forms of patched magnetic fields are provided in Definitions of the Magnetic Field (p. 3518).

Figure 33.4: The MHD Model Dialog Box for Specifying a Moving Field

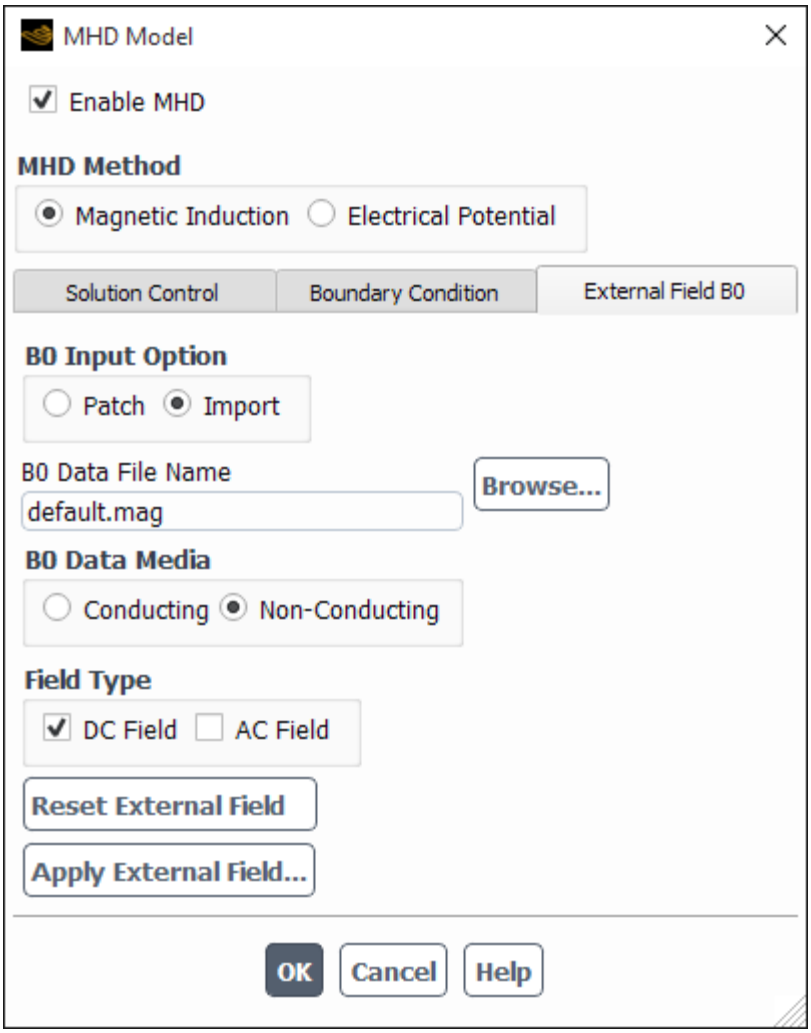


Selecting **Import** under the **B0 Input Option** in the **MHD Model** dialog box, as seen in Figure 33.5: The MHD Model Dialog Box for Importing an External Magnetic Field (p. 3506), will result in the import of magnetic field data. The data filename can be entered in the **B0 Data File Name** field, or selected from your computer file system using the **Browse...** button. Magnetic data can also be generated using a third-party program such as MAGNA. The required format of the magnetic field data file is given in External Magnetic Field Data Format (p. 3518).

When using the **Import** option, the **B0 Data Media** is either set to **Non-Conducting** or **Conducting**, depending on the assumptions used in generating the magnetic field data. (These choices correspond to "Case 1" and "Case 2", respectively, as discussed in Magnetic Induction Method.)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 33.5: The MHD Model Dialog Box for Importing an External Magnetic Field



The **Field Type** is determined by the field data from the data file. The choice of either the **DC Field** or the **AC Field** option in the dialog box is irrelevant if the import data is either DC or AC. However, selection of both options indicates that data of both field types are to be imported from the data file, and superimposed together to provide the final external field data. Make sure that the data file contains two sections for the required data. See External Magnetic Field Data Format (p. 3518) for details on data file with two data sections.

The **Apply External Field...** button opens the **Apply External B0 Field** dialog box as shown in Figure 33.6: Apply External B0 Field Dialog Box (p. 3507). To apply the external field data to zones or regions in the computational domain, select the zone names or register names of marked regions from the dialog box and click the **Apply** button.

The **Reset External Field** button sets the external magnetic field variable to zero.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 33.6: Apply External B0 Field Dialog Box

33.3.3.4. Setting Up Boundary Conditions

Boundary conditions related to MHD calculations are set under the **Boundary Condition** tab in the **MHD Model** dialog box. Boundary conditions can be set to cell zones and wall boundaries.

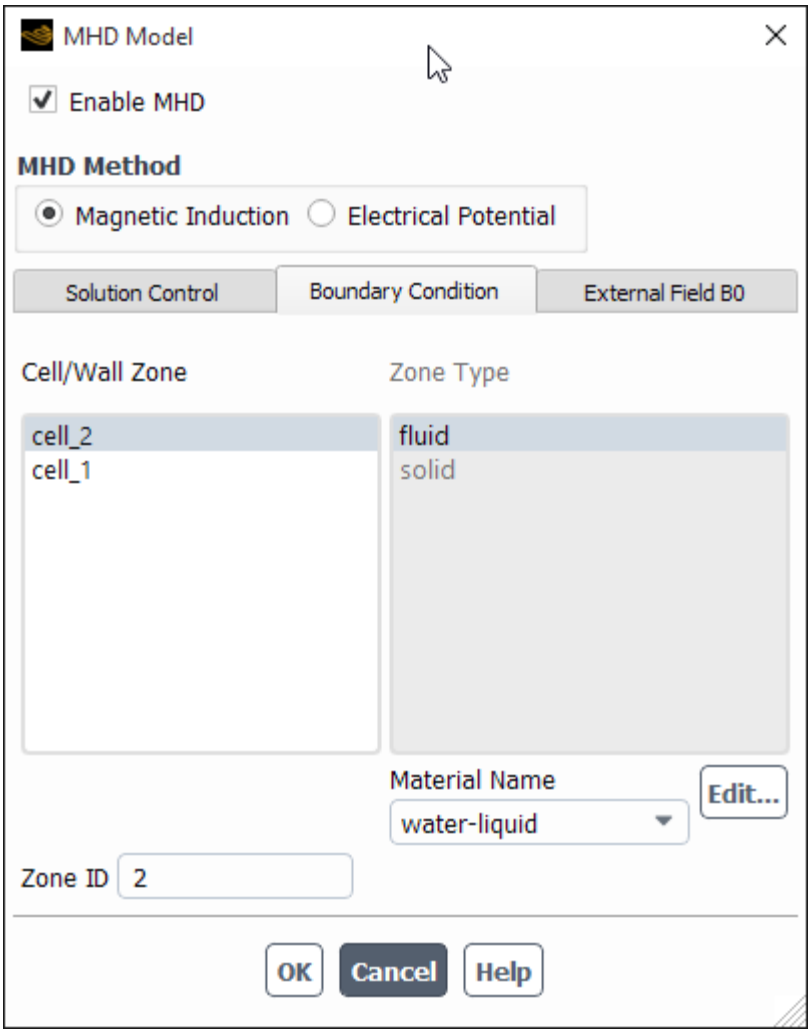
Note:

Note that all boundary conditions used by the MHD model must be set in the **Boundary Condition** tab of the **MHD Model** dialog box. Customized boundary conditions of any form, such as parameters and expressions, are not supported.

For cell zones, only the associated material can be changed and its properties modified. Figure 33.7: Cell Boundary Condition Setup (p. 3508) shows the dialog box for cell zone boundary condition setup. The cell zone material can be selected from the **Material Name** drop-down list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

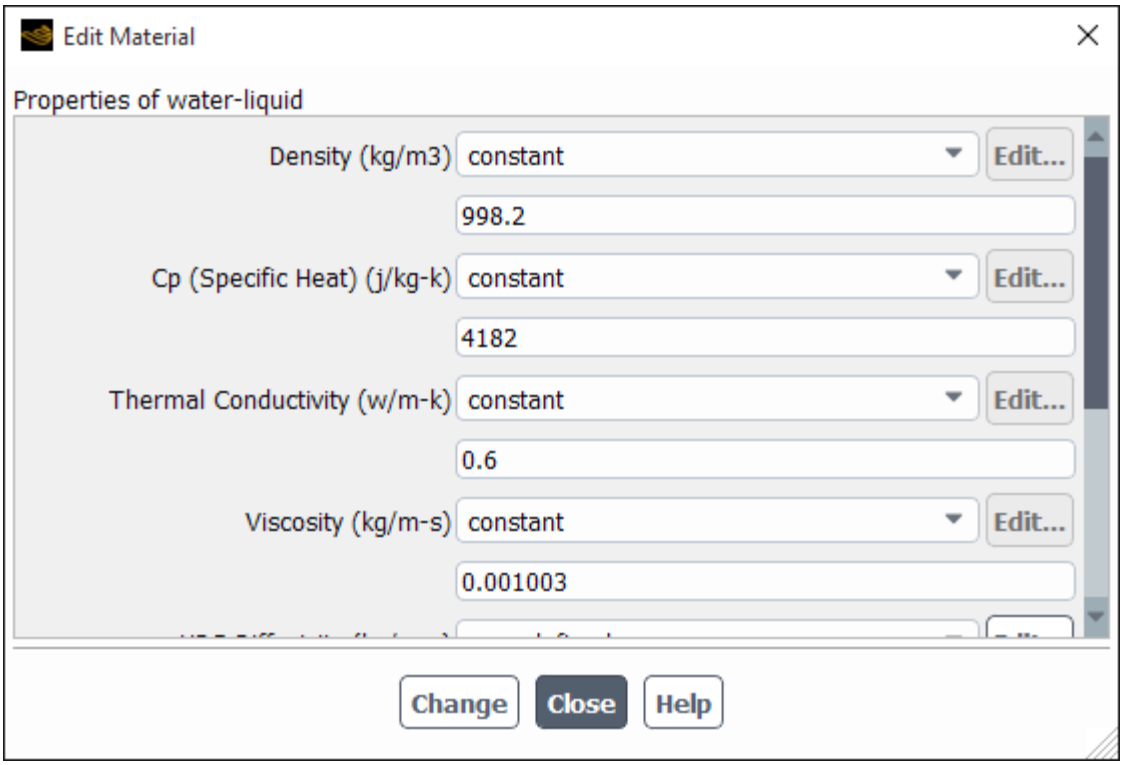
Figure 33.7: Cell Boundary Condition Setup



Note that the materials available in the list are set in the general Ansys Fluent case set up. Refer to the Ansys Fluent User Guide for details on adding materials to an Ansys Fluent case. The properties of the selected material can be modified in the **Boundary Condition** tab by clicking **Edit...** to the right of the material name. This opens the **Edit Material** dialog box, as shown in Figure 33.8: Editing Material Properties within Boundary Condition Setup (p. 3509). The material properties that may be modified include the electrical conductivity and magnetic permeability. The material electrical conductivity can be set as constant, a function of temperature in forms of piecewise-linear, piecewise-polynomial or polynomial, or as a user-defined function. The material magnetic permeability can only be set as a constant.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

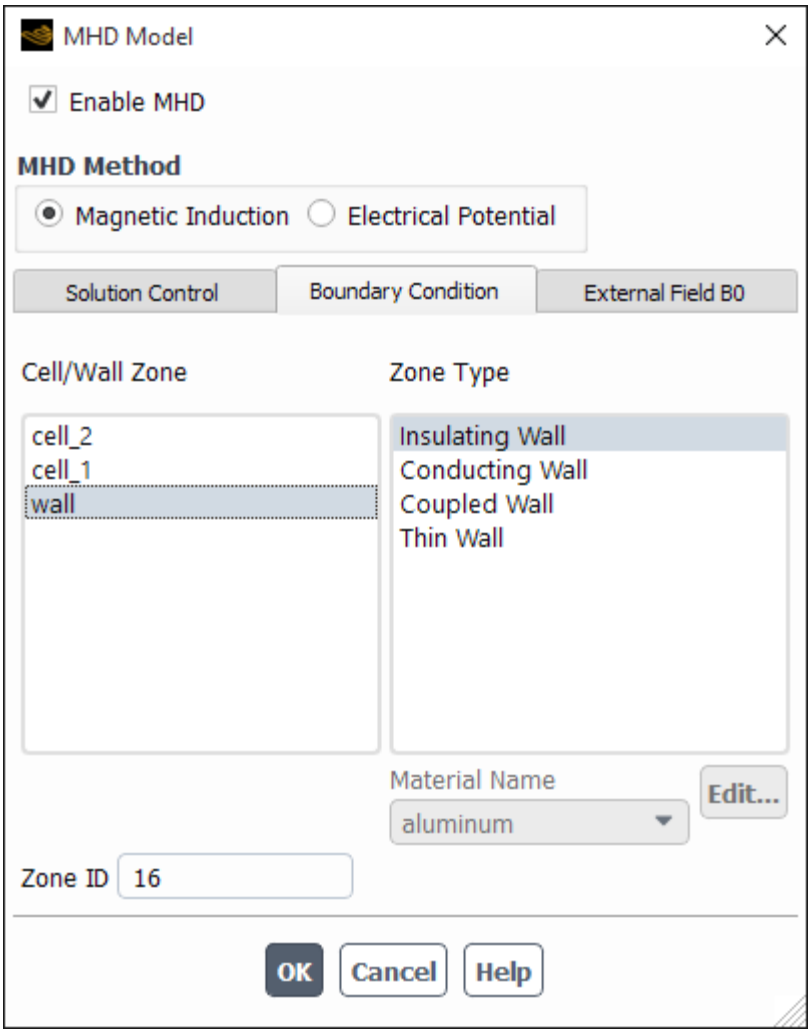
Figure 33.8: Editing Material Properties within Boundary Condition Setup



For wall boundaries, the boundary condition can be set as an **Insulating Wall**, **Conducting Wall**, **Coupled Wall** or **Thin Wall** (Magnetic Induction method only). The dialog box for wall boundary condition set up is shown in Figure 33.9: Wall Boundary Condition Setup (p. 3510).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 33.9: Wall Boundary Condition Setup



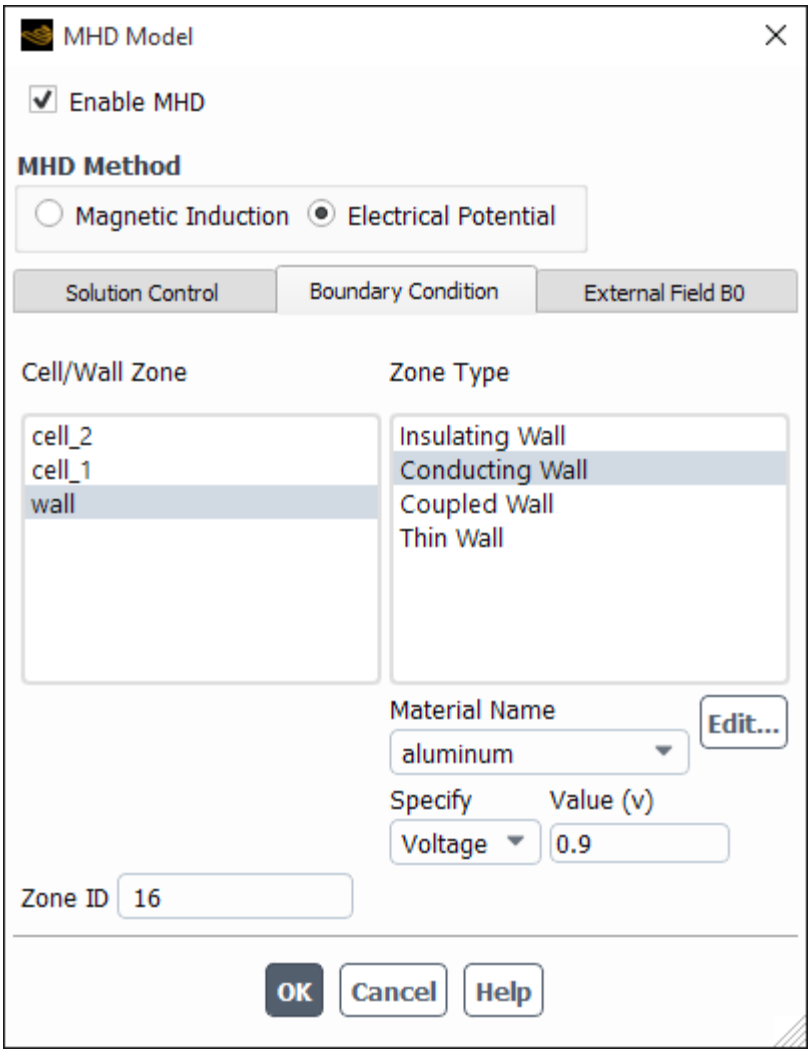
- The insulating wall is used for boundaries where there is no electric current going through the boundary.
- The conducting wall is used for boundaries that are perfect conductors.
- The coupled wall should be used for wall boundaries between solid/solid or solid/fluid zones where the MHD equations are solved.
- (Magnetic Induction method only) The thin wall type boundary can be used for external wall that has a finite electrical conductivity.

For conducting walls and thin wall boundaries, the wall material can be selected from the **Material Name** drop-down list, and its properties modified through the **Edit Material** dialog box. A wall thickness must be specified for thin wall type boundaries.

If the **Electric Potential** method is selected, the conducting wall boundary is specified by either of **Voltage** or **Current Density** at the boundary, as shown in Figure 33.10: Conducting Wall Boundary Conditions in Electrical Potential Method (p. 3511).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 33.10: Conducting Wall Boundary Conditions in Electrical Potential Method

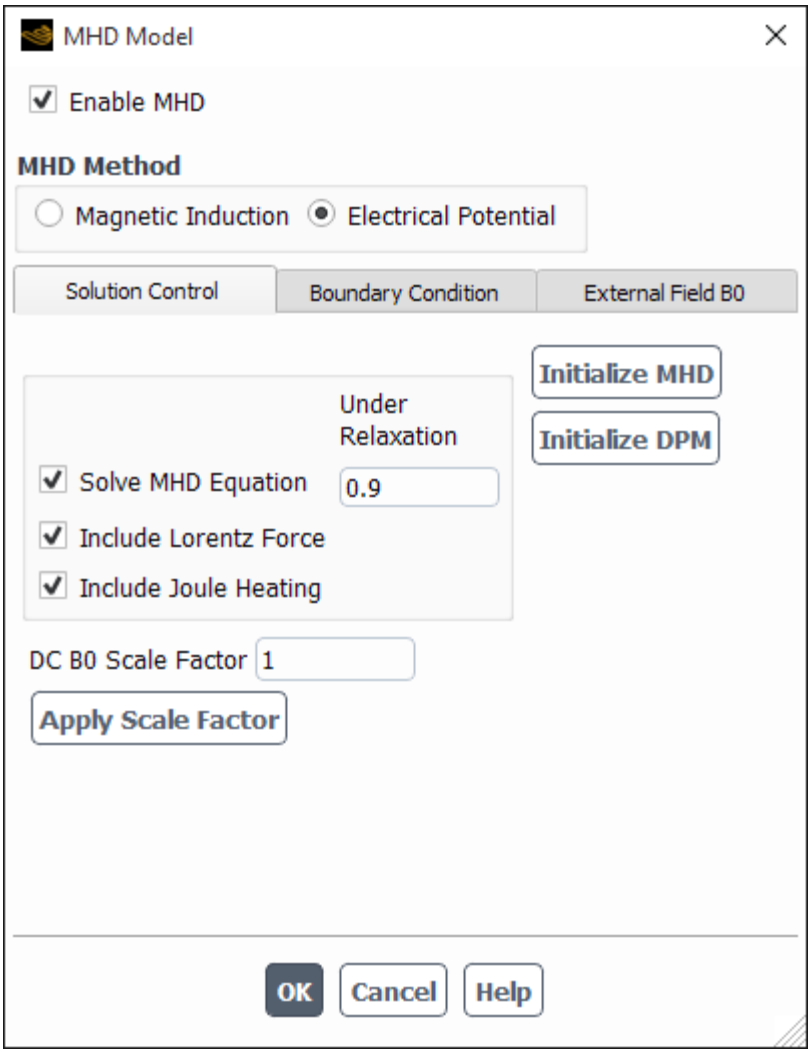


33.3.3.5. Solution Controls

Under the **Solution Control** tab in the **MHD Model** dialog box, Figure 33.11: Solution Control Tab in the MHD Model Dialog Box (p. 3512), a number of parameters can be set that control the solution process in an MHD calculation. The MHD model can be initialized using the **Initialize MHD** button. When the DPM model is enabled in the Ansys Fluent case set up, the related variables used in the MHD model can be initialized using the **Initialize DPM** button.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 33.11: Solution Control Tab in the MHD Model Dialog Box



You have the option to enable or disable the **Solve MHD Equation**. When the **Solve MHD Equation** is enabled, you have the choice to **Include Lorentz Force** and/or **Include Joule Heating** in the solution of flow momentum and energy equations. The under-relaxation factor for the MHD equations can also be set.

The strength of the imposed external magnetic field can be adjusted by specifying and applying scale factors to the external DC and/or AC magnetic field data.

33.3.4. MHD Solution and Postprocessing

The following sections describe the solution and postprocessing steps for the MHD model:

33.3.4.1. MHD Model Initialization

33.3.4.2. Iteration

33.3.4.3. Postprocessing

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

33.3.4.1. MHD Model Initialization

Initialization of the MHD model involves setting the externally-imposed magnetic field and initializing all MHD related user-defined scalars and memory variables.

When an Ansys Fluent case is initialized, all user-defined scalar and memory variables are set to zero. The external magnetic field data is set from the **External Field B0** tab in the **MHD Model** dialog box. The **Initialize MHD** button under the **Solution Control** tab can be used to initialize the model during an Ansys Fluent solution process. It is used when MHD effects are added to a fully or partially solved flow field, or when the model parameters are changed during an MHD calculation. It only clears the scalar variables and most of the memory locations used in the MHD model, the memory variables for the external magnetic field data are preserved.

33.3.4.2. Iteration

It is often an effective strategy to begin your MHD calculations using a previously-converged flow field solution. With this approach, the induction equations themselves are generally easy to converge. The under-relaxation factors for these equations can be set to 0.8 ~ 0.9, although for very strong magnetic fields, smaller values may be needed. For the electric potential equation, the convergence is generally slow. However, the under-relaxation value for this equation should not be set to 1. As additional source terms are added to the momentum and energy equations, the under-relaxation factors for these equations should generally be reduced to improve the rate of convergence. In case of convergence difficulties, another helpful strategy is to use the **B0 Scale Factor** in the **Solution Control** tab (Figure 33.2: The MHD Model Dialog Box (p. 3503)). This will gradually increase the MHD effect to its actual magnitude through a series of restarts. When the strength of the externally imposed magnetic field is strong, it is advisable to start the calculation with a reduced strength external field by applying a small scale factor. When the calculation is approaching convergence the scale factor can be increased gradually until the required external field strength is reached.

33.3.4.3. Postprocessing

You can use the standard postprocessing facilities of Ansys Fluent to display the MHD calculation results.

Contours of MHD variables can be displayed using

Results → **Graphics** → **Contours** **Edit...**

The MHD variables can be selected from the variable list.

Vectors of MHD variables, such as the magnetic field vector and current density vector, can be displayed using

Results → **Graphics** → **Vectors** **Edit...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The vector fields of the MHD variables are listed in the **Vectors of** drop-down list in the **Vectors** dialog box. Table 33.2: MHD Vectors (p. 3514) lists the MHD related vector fields.

Table 33.2: MHD Vectors

Name	Unit	Description
Induced- $\vec{B}$-Field	Tesla	Induced magnetic field vector
External- $\vec{B}$-Field	Tesla	Applied external magnetic field vector
Current-Density $\vec{J}$	$\mathrm{A/m^2}$	Induced current density field vector
Electric-Field $\vec{E}$	V/m	Electric field density vector
Lorentz-Force $\vec{F}$	$\mathrm{N/m^3}$	Lorentz force vector

Note:

- For field variables that are stored in UDM, use the corresponding variables for postprocessing. Postprocessing the UDM itself is not recommended.
- For reviewing simulation results in CFD-Post for cases involving UDM or UDS, export the solution data as CFD-Post-compatible file (`.cdat`) in Fluent and then load this file into CFD-Post.

33.3.5. Limitations

Many MHD applications involve the simultaneous use of other advanced Ansys Fluent capabilities such as solidification, free surface modeling with the volume of fluid (VOF) approach, DPM, Eulerian multiphase, and so on. You should consult the latest Ansys Fluent documentation for the limitations that apply to those features. In addition, you should be aware of the following limitations of the MHD capability.

- As explained in Modeling Magnetohydrodynamics, the MHD module assumes a sufficiently conductive fluid so that the charge density and displacement current terms in Maxwell's equations can be neglected. For marginally conductive fluids, this assumption may not be valid. More information about this simplification is available in the bibliography.
- For electromagnetic material properties, only constant isotropic models are available. Multiphase volume fractions are not dependent on temperature, species concentration, or field strength. However, sufficiently strong magnetic fields can cause the constant-permeability assumption to become invalid.
- You must specify the applied magnetic field directly. The alternative specification of an imposed electrical current is not supported.
- In the case of alternating-current (AC) magnetic fields, the capability has been designed for relatively low frequencies; explicit temporal resolution of each cycle is required. Although not a fundamental limitation, the computational expense of simulating high-frequency effects may become excessive due to small required time step size. Time-averaging methods to incorporate high-frequency MHD effects have not been implemented.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

33.4. Guidelines For Using the Ansys Fluent MHD Model

This section provides a basic outline for installing the magnetohydrodynamics (MHD) module and solving MHD problems in Ansys Fluent.

33.4.1. Installing the MHD Module

33.4.2. An Overview of Using the MHD Module

Important:

While Using the Ansys Fluent MHD Module (p. 3499) covers much of the same material in greater detail, this section presents a set of guidelines for solving typical MHD problems with Ansys Fluent, with occasional references to Using the Ansys Fluent MHD Module (p. 3499) where more information can be found.

33.4.1. Installing the MHD Module

Before using the MHD module, you first need to install the necessary files onto your computer. These files are provided with your standard installation of Ansys Fluent. They can be found in your installation area in a directory called `addons/mhd`. The MHD module is loaded into Ansys Fluent through the text user interface (TUI)

`define` → `models` → `addon-module`

only after a valid Ansys Fluent case file has been set or read.

Once the MHD model is installed, beneath the `mhd` directory there are two subdirectories: a `lib` directory, and a directory corresponding to your specific architecture, `ntx86` for example. The `lib` directory holds a Scheme code called `addon.bin` that contains the MHD module graphical interface. The specific architecture directory, `ntx86` for example, contains the following subdirectories that hold various Ansys Fluent files:

```
2d   2ddp   3d   3ddp
2d_host  2ddp_host  3d_host  3ddp_host
2d_node  2ddp_node  3d_node  3ddp_node
```

33.4.2. An Overview of Using the MHD Module

To use the MHD module in an Ansys Fluent simulation, follow the general guidelines:

1. Start Ansys Fluent.

   To begin modeling your MHD simulation, you need to start an appropriate Ansys Fluent session. Choose from either the 2D, 3D, **Double Precision**, or the parallel version of Ansys Fluent.

2. Read in a mesh file or a case file.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can have Ansys Fluent read in your mesh file, a previously saved non-MHD case file, or a previously saved MHD case file.

Important:

Note that if you read in a new mesh file, you need to perform the appropriate mesh check and mesh scale procedures.

3. Load the MHD module.

   The MHD module is loaded into Ansys Fluent using the text command

   `define` → `models` → `addon-module`

   and entering the corresponding module number (Loading the MHD Module (p. 3500)).

4. Set up the MHD model.

   The **MHD Model** dialog box is accessed using the graphical user interface (GUI):

   Setup → **Models** → **MHD Model** **Edit...**

   If the MHD model is not enabled after the MHD module is loaded for the first time, you can enable it by clicking the **Enable MHD** button, which will display the expanded dialog box (Enabling the MHD Model (p. 3502)).

5. Select an MHD method.

   The method used for the MHD calculation can be selected under **MHD Method**. The two methods are

   - **Magnetic Induction** (Magnetic Induction Method)
   - **Electrical Potential** (Electric Potential Method)

6. Apply an external magnetic field.

   This is done by entering values for the B0 components in the **External Field B0** tab. B0 input options can either be

   - **Patch**ed, or
   - **Import**ed

   The **Field Type** will either be the **DC Field** or the **AC Field**. The **Field Type** is determined by the field data from the data file. Refer to Applying an External Magnetic Field (p. 3504) for details on applying an external magnetic field.

7. Set up the boundary conditions.

   Under the **Boundary Condition** tab, cell zones and wall boundaries can be selected as well as the corresponding zone type.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cell zone materials are selected from the **Material Name** drop-down list. The properties of the selected material can be modified by clicking on the **Edit...** button to the right of the material name. Note that the materials available in the list are set in the general Ansys Fluent case setup.

Setup → **Materials**

The material properties that may be modified include the electrical conductivity and magnetic permeability.

Wall boundary conditions can be set as an **Insulating Wall**, **Conducting Wall**, **Coupled Wall** or **Thin Wall** (see Setting Up Boundary Conditions (p. 3507)).

8. Set solution controls.

   Under the **Solution Control** tab:

   - The MHD equation is enabled or disabled.
   - Lorentz force and Joule heat sources are enabled or disabled.
   - under-relaxation factors are set (reasonable under-relaxation factors for the MHD equations are 0.8 ~ 0.9).
   - Scale factors can be used to adjust the strength of the imposed external magnetic field. As the calculation approaches convergence, the scale factor in the **Solution Control** tab can be increased gradually until the required external field strength is reached (Solution Controls (p. 3511)).
   - The MHD model is initialized (MHD Model Initialization (p. 3513)).

9. Run the Ansys Fluent MHD simulation.

   Solution → **Run Calculation**

   Set the number of iterations. It is often an effective strategy to begin your MHD calculations using a previously-converged flow field solution. With this approach, the induction equations themselves are generally easy to converge. For more information, see Iteration (p. 3513).

10. Process the solution data.

    You can use the standard postprocessing facilities of Ansys Fluent to display the results of an MHD calculation. Contours of MHD variables can be displayed.

    Results → **Graphics** → **Contours** **Edit...**

    The MHD variables can be selected from the variable list. Vectors of MHD variables, such as the magnetic field vector and current density vector, can be displayed using custom vectors.

    Results → **Graphics** → **Vectors** **Edit...**

    For more information, see Postprocessing (p. 3513).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

33.5. Definitions of the Magnetic Field

The sinusoidal form of the magnetic field is defined as:

$$B_0=\overline{B}_0+A_0\cos\left(2\pi ft-K\cdot R+\phi\right)$$
$$K=\frac{1}{\lambda}\left\{\frac{1}{\cos\alpha}i+\frac{1}{\cos\beta}j+\frac{1}{\cos\gamma}k\right\} \quad (33.1)$$

where $\overline{B}_0$ is the mean vector, A_0 is the amplitude vector, K is defined as the propagation vector, R is the position vector of an arbitrary point. $\cos\alpha$, $\cos\beta$ and $\cos\gamma$ are the x, y and z direction cosines respectively. The quantities f, λ, and ϕ are the frequency, wavelength, and phase offset, respectively. For a non-moving field, the propagation vector is zero.

The square form of the magnetic field is defined as:

$$B_0=\overline{B}_0+A_0\frac{\cos\left(2\pi ft-K\cdot R+\phi\right)}{\left|\cos\left(2\pi ft-K\cdot R+\phi\right)\right|} \quad (33.2)$$

The definition of the propagation vector is the same as for the sinusoidal form.

33.6. External Magnetic Field Data Format

The external magnetic field data file is in text format and of the following structure:

$MAG\text{-}DATA$					
nX	nY	nZ			
$X1$	Xn				
$Y1$	Yn				
$Z1$	Zn				
nAC	$Freq$				
BX_{re-1}	BY_{re-1}	BZ_{re-1}	BX_{im-1}	BY_{im-1}	BZ_{im-1}
...					
BX_{re-n}	BY_{re-n}	BZ_{re-n}	BX_{im-n}	BY_{im-n}	BZ_{im-n}

The first line is an identification tag for the data file. The second line defines the number of data points in the x, y and z directions. The next three lines define the ranges in x, y and z directions. The data points are assumed to be evenly distributed along each direction. Line 6 defines the AC field flag and frequency. When `nAC = 0`, the magnetic field is static. For AC field, `nAC = 1` and `Freq` is the frequency in Hz.

The rest of the data file contains the magnetic field data points. Each line defines the components of the real and imaginary parts of the magnetic field vector on one data point. The data points are indexed as:

$$l=i+nX\left(\left(j-1\right)+nY\left(k-1\right)\right)$$
$$i=1,\ldots,nX;j=1,\ldots,nY;k=1,\ldots,nZ \quad (33.3)$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The data is listed in the ascending order from 1 to n, where n is the total number of data points given by $n = nXnYnZ$.

For magnetic fields composed of both DC and AC fields, the entire file structure described above is repeated for the DC and AC parts. These two sections within the same file will be imported into Ansys Fluent and stored separately. The order of the DC and AC sections of the file is not important.

The imported data is interpreted as a snapshot of the applied magnetic field at an instant in time. Complex form is used to accommodate oscillating/moving fields. Thus, using complex numbers, and with reference to the quantities defined in Definitions of the Magnetic Field (p. 3518),

$$\begin{aligned}\vec{B}_0 &\equiv \begin{Bmatrix} BX_{re} \\ BY_{re} \\ BZ_{re} \end{Bmatrix} + i \begin{Bmatrix} BX_{im} \\ BY_{im} \\ BZ_{im} \end{Bmatrix} \\ &= \bar{B}_0 + A_0 exp\left[i\left(2\pi f t + \phi\right)\right]\end{aligned} \tag{33.4}$$

For a DC field,

$$\bar{B}_{o,i} = B_{re,i} \qquad i = x, y, z \tag{33.5}$$

For an AC field,

$$A_{o,i} = \sqrt{B_{re,i}^{\ 2} + B_{im,i}^{\ 2}} \qquad i = x, y, z \tag{33.6}$$

and

$$\phi_i = tg^{-1}\left(\frac{B_{im,i}}{B_{re,i}}\right) \qquad i = x, y, z \tag{33.7}$$

Note that when the external magnetic field import option is used, the frequency, f, read from this file supersedes the value specified in the GUI.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 34: Modeling Continuous Fibers

The continuous fiber model is provided as an add-on module with the standard Ansys Fluent licensed software.

Several fiber spinning techniques exist in industrial fiber production. The most common types are melt spinning and dry spinning.

In melt spinning, the polymer is heated above its melting point and extruded in a liquid state through nozzles into a vertical spinning chamber. The molten polymer is processed in an inert gas environment, such as nitrogen, then extruded at high pressure and a constant rate into a cooler air stream, thus solidifying the fiber filaments.

In dry spinning, the liquefaction of the polymer is obtained by dissolving it in a suitable solvent. This technique often is applied to polymers that are destroyed thermally before reaching its melting point or if the production process leads to a solvent/polymer mixture. In the spinning chamber, the solvent vaporizes by drying with a hot air stream. The solidification ensures that the fiber is nearly free of solvent.

Ansys Fluent's continuous fiber model allows you to analyze the behavior of fiber flow, fiber properties, and coupling between fibers and the surrounding fluid due to the strong interaction that exists between the fibers and the surrounding gas.

This document describes the Ansys Fluent Continuous Fiber model. Modeling Continuous Fibers provides theoretical background information.

The procedure for setting up and solving fiber spinning flows is described in detail in this chapter. Only the steps related to fiber modeling are shown here.

Important:

Note that the continuous fiber model is available for the pressure-based solver, only.

For information about inputs related to other models used in conjunction with the fiber models, see the appropriate sections for those models in the Ansys Fluent User's Guide (p. 1).

34.1. Installing the Continuous Fiber Module

34.2. Loading the Continuous Fiber Module

34.3. Getting Started With the Continuous Fiber Module

34.4. Fiber Models and Options

34.5. Fiber Material Properties

34.6. Defining Fibers

34.7. User-Defined Functions (UDFs) for the Continuous Fiber Model

34.8. Fiber Model Solution Controls

34.9. Postprocessing for the Continuous Fibers

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

34.1. Installing the Continuous Fiber Module

The continuous fiber model is provided as an add-on module with the standard Ansys Fluent licensed software. The module is installed with the standard installation of Ansys Fluent in a directory called `addons/fiber` in your installation area. The continuous fiber module consists of a UDF library and a pre-compiled Scheme library, which must be loaded and activated before calculations can be performed.

34.2. Loading the Continuous Fiber Module

The continuous fiber module can be loaded only when a valid Ansys Fluent case file has been set or read.

There are two different ways in which you can load the continuous fiber module into Ansys Fluent:

- Through the ribbon

  In the **Physics** ribbon tab, click **More** (**Models** group), and select **Fiber Model...**.

  Physics → **Models** → **More** → **Fiber Model...**

  Once the continuous fiber module is loaded into Ansys Fluent, the **Fiber Model** dialog box opens where you can set the model as described in Modeling Macroscopic Particles (p. 2895).

- Through the text user interface (TUI)

  The text command to load the module is

  `define` → `models` → `addon-module`

  A list of Ansys Fluent add-on modules is displayed:

```
> /define/models/addon-module
Fluent  Addon Modules:
     0. None
     1. MHD Model
     2. Fiber Model
     3. Fuel Cell and Electrolysis Model
     4. SOFC Model with Unresolved Electrolyte
     5. Population Balance Model
     6. Adjoint Solver
     7. Single Potential Battery Module
     8. MSMD Battery Module
     9. PEM Fuel Cell Model
Enter Module Number: [0] 2
```

  Select the continuous fiber model by entering the module number 2. During the loading process a Scheme library containing the graphical and text user interface, and a UDF library containing a set of user-defined functions (UDFs) are loaded into Ansys Fluent.

  During this process, you will be asked the question

```
 Preset all fiber model specific UDF hooks? [no]
```

  If you answer `yes` the standard fiber source term UDFs will be assigned to *all* fluid zones in your case, and a message will be reported to the console window confirming this:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
 Assigning standard fiber source terms to all fluid zones.
```

If you answer no to presetting source term UDFs to all fluid zones in the domain, then three options will be available to you when setting up source terms for fluid zones in your fiber model: no source, constant source, or UDF source. Note that it is your responsibility to specify the rest of the settings for a proper fiber simulation. See Source Term UDF Setup (p. 3524) for details.

If you are loading an existing case file then you should answer the question with no. Otherwise, your saved source term settings will be replaced by a UDF.

If a mixture material has been defined, then you will be asked an additional question

```
Preset mass exchange source terms hooks? [no]
```

If you intend to conduct a dry spinning simulation, then you should reply yes.

If you did not preset the fiber model specific UDF hooks, you will need to check allocation of user-defined memory, hook an adjust function (fm_adjust) to Ansys Fluent, and set up the source terms on your own. This is explained is User-Defined Memory and the Adjust Function Setup (p. 3524).

During the loading process the UDF library for the continuous fiber module is loaded in Ansys Fluent. This is reported to the console (see below). The UDF library also becomes visible as a new entry in the **UDF Library Manager** dialog box. The basic setup of the continuous fiber model is performed automatically when the fiber module is successfully loaded.

```
Opening library "/.../addons/fiber"...
Library "/...addons/fiber/lnx86/3d/libudf.so" opened
fm_adjust
fm_src_mass
fm_src_x_mom
fm_src_y_mom
fm_src_z_mom
fm_src_enthalpy
fm_src_dom
fm_on_demand Done.

Addon Module: fiber...loaded!
```

Important:

Note that user-defined memory locations for the fiber model will not be allocated properly if you do not initialize the flow field. If you are setting up a fiber computation based on a converged case, you *must* re-load the Ansys Fluent data file after initializing the solution.

The continuous fiber module setup is saved with the Ansys Fluent case file. The module is loaded automatically when the case file is subsequently read into Ansys Fluent. Note that in the saved case file, the continuous fiber module is saved with the absolute path. Therefore, if the locations of the continuous fiber module installation or the saved case file are changed, then Ansys Fluent will not be able to load the module when the case file is subsequently read. In this situation, you will have to unload the UDF library using the **UDF Library Manager** dialog box after the case file is read, and then reload the continuous fiber module. To unload the UDF library go to the **UDF Library Manager** dialog box

User Defined → **User Defined** → **Functions** → **Manage...**

select the fiber library under **UDF Libraries**, and click **Unload**. Previously-saved continuous fiber model setup and parameters will be preserved in this process.

34.3. Getting Started With the Continuous Fiber Module

The continuous fiber model is implemented by user-defined functions (UDFs) and scheme routines in Ansys Fluent. A number of UDFs are used to solve the fiber equations. When you loaded the fiber module in the previous step (Loading the Continuous Fiber Module (p. 3522)), UDF and scheme libraries that are required by the continuous fiber model were *automatically* loaded. Before you can begin the process of defining your fiber model, however, you will need to perform some additional setup tasks that involve allocating user-defined memory for the UDFs and hooking an adjust UDF to Ansys Fluent. Follow the procedure below.

For more information, see the following sections:

34.3.1. User-Defined Memory and the Adjust Function Setup

34.3.2. Source Term UDF Setup

34.3.1. User-Defined Memory and the Adjust Function Setup

1. Allocate user-defined memory for the model by incrementing the **Number of User-Defined Memory Location** to 8 in the **User-Defined Memory** dialog box.

   User Defined → **User Defined** → **Memory...**

   Important:

   Note that you *must* initialize your solution (in the **Solution Initialization** task page) in order for user-defined memory to be allocated properly. If you are setting up a fiber simulation based on a converged case, then you will have to reload the Ansys Fluent data file after initializing the solution.

2. Hook the adjust function UDF to Ansys Fluent by choosing `fm_adjust::fiber` from the drop-down list for **Adjust** in the **User-Defined Function Hooks** dialog box.

   User Defined → **User Defined** → **Function Hooks...**

34.3.2. Source Term UDF Setup

When loading the continuous fiber module via TUI, if you answered `no` to presetting all fiber model-specific UDF hooks during the loading process (Loading the Continuous Fiber Module (p. 3522)) then you will need to set source terms, individually, for each fluid zone in your model. Alternatively, you can leave the default settings (`none` for no source term).

For each fluid zone in your model, specify `none`, `constant`, or UDF for all of the source terms by following the procedure below:

Setup → **Cell Zone Conditions**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. In the **Cell Zone Conditions** task page, select a fluid zone under **Zone** and click **edit...**. This opens the **Fluid** dialog box.

2. In the **Fluid** dialog box, check **Source Terms** and click the **Source Terms** tab.

3. For each of the source terms in the scroll list (**Mass, X Momentum**, and so on), click the **Edit...** button next to each source term to open the corresponding source dialog box. Leave the default `none`, or choose `constant` or UDF from the drop-down list. Choose the UDF in the drop-down list that corresponds to the particular source term. For example, `udf fm_src_mass` corresponds to the `Mass` source term. Use the table below (Table 34.1: Source Terms and Corresponding UDFs (p. 3525)) as a reference guide.

   Table 34.1: Source Terms and Corresponding UDFs

Mass	`udf fm_src_mass`
X Momentum or Axial Momentum	`udf fm_src_x_mom`
Y Momentum or Radial Momentum	`udf fm_src_y_mom`
Z Momentum	`udf fm_src_z_mom`
Energy	`udf fm_src_enthalpy`
discrete ordinates model	`udf fm_src_dom`

4. Click **OK** when all of the UDFs have been hooked.

5. Repeat this process for the remaining fluid zones in your Ansys Fluent model.

Important:

If you want to include radiative interaction of the fibers with the discrete ordinate (DO) radiation model, then the appropriate source term UDF (`udf fm_src_dom`) will be hooked *automatically* when you select **Fiber Radiation Interaction** in the **Fiber Model** dialog box. You *must* initialize the solution (which will allocate memory for the DO model) before the fiber model will be ready to accept the fiber radiation interaction data.

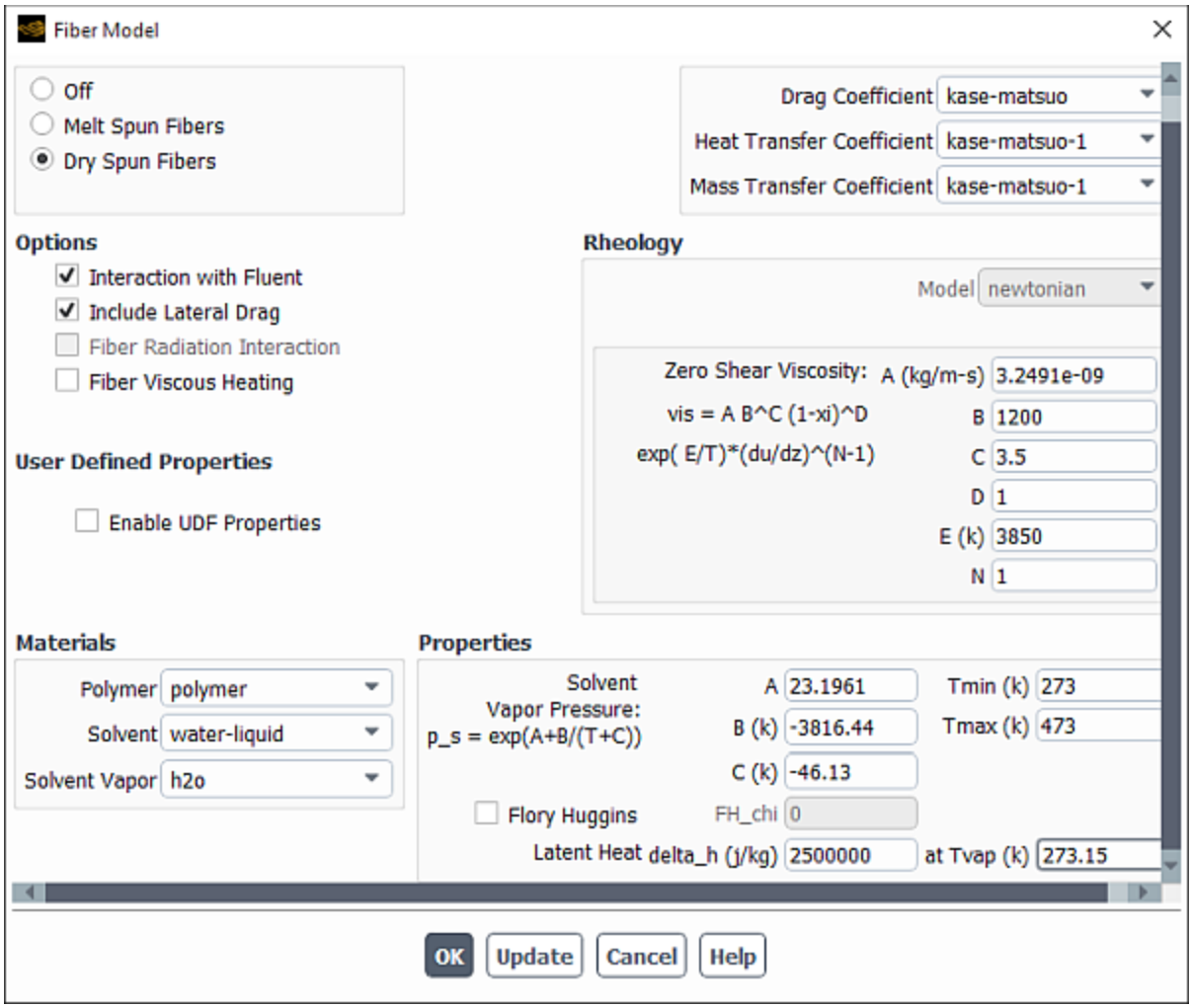
34.4. Fiber Models and Options

This section provides information on how to choose the type of fiber model you want to implement as well as select various model options that are available in Ansys Fluent's fiber model. The fiber **Model** and **Options** are selected in the **Fiber Model** dialog box (Figure 34.1: The Fiber Model Dialog Box (p. 3526)).

Setup → Models → Continuous Fiber Spinning → Edit...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 34.1: The Fiber Model Dialog Box



For more information, see the following sections:

34.4.1. Choosing a Fiber Model
34.4.2. Including Interaction With Surrounding Flow
34.4.3. Including Lateral Drag on Surrounding Flow
34.4.4. Including Fiber Radiation Interaction
34.4.5. Viscous Heating of Fibers
34.4.6. Drag, Heat and Mass Transfer Correlations

34.4.1. Choosing a Fiber Model

With Ansys Fluent's fiber model, you can model two types of fibers: melt spun and dry spun. The model you choose depends on the polymer that is used to draw the fibers. If the fiber polymer can be molten without being destroyed thermally, then a melt spun fiber process is typically used to produce the fibers. In such cases, select **Melt Spun Fibers** under **Model** in the **Fiber Model** dialog box (Figure 34.1: The Fiber Model Dialog Box (p. 3526)).

When the fiber polymer can be liquefied with a suitable solvent, or the fiber polymer's production process involves a solvent, the fibers are formed typically in a dry spinning process. In such cases,

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

select **Dry Spun Fibers** under **Model** in the **Fiber Model** dialog box (Figure 34.1: The Fiber Model Dialog Box (p. 3526)).

34.4.2. Including Interaction With Surrounding Flow

If the fibers in your simulation strongly influence the flow of the surrounding fluid and need to be considered, you must select **Interaction with Fluent** under **Options** in the **Fiber Model** dialog box (Figure 34.1: The Fiber Model Dialog Box (p. 3526)). When iterating a solution with Ansys Fluent, the fiber model equations are solved alternating with Ansys Fluent's flow equations. Source terms are also computed to couple the fiber equations with the fluid flow equations. The calculation of the source terms is performed only during the course of an Ansys Fluent computation. See Coupling Between Fibers and the Surrounding Fluid for a description of the source terms.

34.4.3. Including Lateral Drag on Surrounding Flow

In typical fiber simulations, the axial drag of the fibers is the most important force acting on the fibers as well as on the surrounding fluid. In some situations, the lateral or cross-flow drag can become important. This is the case when the fibers are mainly cooled or dried in a cross-flow situation. In such cases you can select **Include Lateral Drag** under **Options** in the **Fiber Model** dialog box (Figure 34.1: The Fiber Model Dialog Box (p. 3526)). The drag is estimated for a cylinder in cross-flow and is shown in Equation 22.30. Lateral drag is not considered by the fiber equations and therefore lateral bending of the fibers is not considered.

34.4.4. Including Fiber Radiation Interaction

In some situations radiative heat exchange is important. If you are using Ansys Fluent's P-1 radiation model or Ansys Fluent's discrete ordinates (DO) radiation model, you can consider the effects of irradiation on the cooling and heating of the fibers. When you are using the DO radiation model, the effects of the fibers upon the DO model can be considered. If the DO radiation model is enabled, you can select **Fiber Radiation Interaction** (**Options** group box) and enter the fiber's **Emissivity** in the **Properties** group box (see Figure 34.1: The Fiber Model Dialog Box (p. 3526)).

34.4.5. Viscous Heating of Fibers

When high elongational drawing rates are combined with large fiber viscosities, viscous heating of fibers may become important. To consider this effect in the fiber energy equation (Equation 22.12), you can select **Fiber Viscous Heating** under **Options** in the **Fiber Model** dialog box (Figure 34.1: The Fiber Model Dialog Box (p. 3526)).

34.4.6. Drag, Heat and Mass Transfer Correlations

The effects of the boundary layer of the fiber are modeled in terms of drag, heat transfer, and mass transfer coefficients. These parameters are specified under **Exchange** in the **Fiber Model** dialog box (Figure 34.1: The Fiber Model Dialog Box (p. 3526)).

- For the **Drag Coefficient**, you can choose between **const-drag**, **kase-matsuo**, **gampert** and **user-defined** from the drop-down list. If you choose **const-drag**, the constant you enter must be specified as a dimensionless value. See Drag Coefficient for a description of these options. User-defined functions (UDFs) are described in detail in User-Defined Functions (UDFs) for the Continuous Fiber Model (p. 3543).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For the **Heat Transfer Coefficient** you can choose between **const-htc**, **kase-matsuo-1**, **kase-matsuo-2**, **gampert**, and **user-defined** from the drop-down list. If you choose **const-htc**, the constant you enter must be specified SI units of W/(m^2K). See Heat Transfer Coefficient for a description of these options. User-defined functions (UDFs) are described in detail in User-Defined Functions (UDFs) for the Continuous Fiber Model (p. 3543).
- For the **Mass Transfer Coefficient** you can choose between **const-mtc**, **kase-matsuo-1**, **kase-matsuo-2**, **gampert**, and **user-defined**. If you choose **const-mtc**, you must enter the mass transfer rate in units of kg/(m^2s) instead of the mass transfer coefficient. See Mass Transfer Coefficient for a description of these options. User-defined functions (UDFs) are described in detail in User-Defined Functions (UDFs) for the Continuous Fiber Model (p. 3543).

34.5. Fiber Material Properties

For more information, see the following sections:

34.5.1. The Concept of Fiber Materials

34.5.2. Description of Fiber Properties

34.5.1. The Concept of Fiber Materials

The material properties you specify for the fibers are used for all fibers defined in your model. You cannot consider fibers consisting of different fiber materials in one simulation.

The continuous fiber model makes use of Ansys Fluent's material concept for the **Material Type** of the **fluid**. Because not all properties are available in Ansys Fluent's **Create/Edit Materials** dialog box for this material type, some additional property information can be provided through the **Fiber Model** dialog box (Figure 34.1: The Fiber Model Dialog Box (p. 3526)). The procedure to define the material properties for the fibers in your simulation is as follows:

1. In the **Create/Edit Materials** dialog box, set the **Material Type** as **fluid**. This fluid will be used as the polymer or solvent in the fiber.
2. Enter all data for this material.

   You can use all profiles available in the **Create/Edit Materials** dialog box to define the properties as functions of temperature. In order to invoke user-defined fiber properties, you need to use the UDF template file (see User-Defined Functions (UDFs) for the Continuous Fiber Model (p. 3543) for details). UDF access is available for viscosity, density, specific heat capacity, thermal conductivity and solvent liquid-vapor equilibrium pressure.
3. Select the **Polymer** in the **Materials** group box of the **Fiber Model** dialog box.
4. For dry spinning simulations, you also have to select **Solvent** and the gas phase species that represents the **Solvent Vapor**.
5. Enter any additional data needed for the fiber material in the **Fiber Model** dialog box.

34.5.2. Description of Fiber Properties

The properties that appear in the **Fiber Model** dialog box vary depending on the fiber model type.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following list describes the properties you may need for a fiber material. For every property listed, the dialog box name is provided where the property can be defined.

Blending Interval

(**Fiber Model** dialog box) is the temperature interval used to compute an average of the fiber viscosities in liquid and solid state of **Melt Spun Fibers**. This option is only available when the **Melt Spun Fibers** option is selected. See Fiber Viscosity for details about how the **Blending Interval** is applied to the fiber viscosity.

Cp

(**Create/Edit Materials** dialog box) is the specific heat, $C_{p,f}$, of the fiber in units of energy per mass and temperature. In the case of dry spun fibers, a mass average is computed based on the values entered for **Polymer** and its **Solvent**. You can use any of the functions available to define temperature dependency. If you want to use a user-defined function profile, you need to modify the UDF template provided by Ansys Fluent. See User-Defined Functions (UDFs) for the Continuous Fiber Model (p. 3543) for details.

Density

(**Create/Edit Materials** dialog box) is the density, ρ_f, of the fiber in units of mass per unit volume.

This density is the mass density and not the volume density. In the case of dry spun fibers, a mass average is computed based on the values entered for **Polymer** and its **Solvent**. You can use any of the functions available to define temperature dependency. If you want to use a user-defined function profile, you need to modify the UDF template provided by Ansys Fluent. See User-Defined Functions (UDFs) for the Continuous Fiber Model (p. 3543) for details.

Emissivity

(**Fiber Model** dialog box) is the emissivity of fibers in your model, ε_f, used to compute radiation heat transfer to the fibers (Equation 22.13, Equation 22.14, Equation 22.24 and Equation 22.25) when the P-1 or discrete ordinates radiation model is active. Note that you must enable radiation to fiber, using the **Fiber Radiation Interaction** option in the **Fiber Model** dialog box.

Flory Huggins

(**Fiber Model** dialog box) can be enabled to apply Equation 22.3 to compute the vapor-liquid equilibrium at the fiber surface. When it is enabled, you have to specify an appropriate value for the dimensionless Flory Huggins parameter, χ. This option is only visible when **Dry Spun Fibers** has been chosen.

Latent Heat

(**Fiber Model** dialog box) is the latent heat of vaporization of the **Solvent** when evaporated from a dry spun fiber. Note that you have to enter the vaporization or reference temperature, T_{vap}, where the specified value of the latent heat has been measured. This vaporization temperature is used to automatically consider the change of latent heat with temperature. See Equation 22.16 and Equation 22.39 for more information on how this is achieved. These options are only visible when **Dry Spun Fibers** has been chosen.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solidification Temperature

(**Fiber Model** dialog box) is the temperature below which the fiber polymer of a **Melt Spun Fiber** will solidify. It will be used when computing the viscosity of **Melt Spun Fibers**. This option is only visible when **Melt Spun Fibers** has been chosen.

Solvent Vapor Pressure

(**Fiber Model** dialog box) is the vapor pressure of the solvent evaporating from the fiber surface in dry spinning. You have to enter coefficients for an Antoine-type equation (Equation 22.38). Note that the coefficients must be entered in such units that the outcome of the Antoine-type equation is in Pascal. In addition to the coefficients of the Antoine equation, you have to enter the range of validity for the vapor pressure. Below the minimal temperature the vapor pressure at the minimal temperature will be used. Above the maximal temperature, the vapor pressure at the maximal temperature is used.

Thermal Conductivity

(**Create/Edit Materials** dialog box) is the thermal conductivity, λ_f, of the fiber in units of power per length and temperature. In the case of dry spun fibers, a mass average is computed based on the values entered for **Polymer** and its **Solvent**. You can use any of the functions available to define temperature dependency. If you want to use a user-defined function profile, you need to modify the UDF template provided by Ansys Fluent. See User-Defined Functions (UDFs) for the Continuous Fiber Model (p. 3543) for details.

Zero Shear Viscosity

(**Create/Edit Materials** dialog box and/or the **Fiber Model** dialog box, depending on the chosen fiber model) is the fiber viscosity, η_f, at zero shear rate. It is not the elongational or Trouton viscosity.

For **Melt Spun Fibers**, you have to enter the viscosity of the fiber in solid state in the **Create/Edit Materials** dialog box for the fluid you have selected as the fiber polymer. Typically this value will be very high compared to the liquid fiber viscosity to represent the fibers as solids. For the solid fiber viscosity you can make use of any temperature-dependent function available in the **Create/Edit Materials** dialog box. If you want to use a user-defined function profile, you need to modify the UDF template provided by Ansys Fluent. See User-Defined Functions (UDFs) for the Continuous Fiber Model (p. 3543) for details.

In the **Fiber Model** dialog box, you have to enter the coefficients for the fiber viscosity in liquid state (Equation 22.35). To define viscosity as a function of fiber velocity gradient, set N to a value different than 1. In the case of **Melt Spun Fibers**, you also have to enter data for the **Solidification Temperature** and the **Blending Interval**. The blending of the viscosities in liquid and solid state will be computed based on Equation 22.36.

For **Dry Spun Fibers** you only have to enter the coefficients for Equation 22.37 in the **Fiber Model** dialog box. To specify fiber viscosity as a function of fiber velocity gradient, set N to a

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

value different than 1. Any value entered in the **Create/Edit Materials** dialog box for viscosities of the fluids used as fiber polymer and fiber solvent will not be considered.

Note:

Note that depending on fiber velocity gradient, the numerical behavior of the Fiber Model equations may become unstable in combination with fiber viscosity.

34.6. Defining Fibers

For more information, see the following sections:

34.6.1. Overview
34.6.2. Fiber Injection Types
34.6.3. Working with Fiber Injections
34.6.4. Defining Fiber Injection Properties
34.6.5. Point Properties Specific to Single Fiber Injections
34.6.6. Point Properties Specific to Line Fiber Injections
34.6.7. Point Properties Specific to Matrix Fiber Injections
34.6.8. Define Fiber Grids

34.6.1. Overview

The primary inputs that you must provide for the continuous fiber model calculations in Ansys Fluent are the starting positions, mass flow rate, take up positions, and other parameters for each fiber. These provide the boundary conditions for all dependent variables to be solved in the continuous fiber model. The primary inputs are:

- Start point (x, y, z coordinates) of the fiber.
- Number of fibers in group. Each defined fiber can represent a group of fibers that will be used only to compute the appropriate source terms of a group of fibers.
- Diameter of the fiber nozzle, d_f.
- Mass flow rate per nozzle to compute the velocity of the fiber fluid in the nozzle. The velocity is used as boundary condition for the fiber momentum equation.
- Temperature of the fiber at the nozzle, T_f.
- Solvent mass fraction of the fiber fluid in the nozzle. This value is used as boundary condition for the solvent continuity equation.
- Take-up point (x, y, z coordinates) of the fiber.
- Velocity or force at take-up point to describe the second boundary condition needed for the fiber momentum equation (see Equation 22.4).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In addition to these parameters, you have to define parameters for the grid that is distributed between the start position and take-up point of the fibers. On this grid the dependent fiber variables are solved, by discretizing Equation 22.1, Equation 22.4, and Equation 22.11.

You can define any number of different sets of fibers provided that your computer has sufficient memory.

34.6.2. Fiber Injection Types

You will define boundary conditions and grids for a fiber by creating a fiber "injection" and assigning parameters to it. In the continuous fiber model the following types of injections are provided:

- **single**

  Use this option to define a single fiber.

- **line**

  Use this option when the fibers you want to define start from a line and the starting points are located at constant intervals on this line.

- **matrix** (only in 3D)

  Use this option when the fiber starting points are arranged in the shape of a rectangle.

- **file**

  Use an ASCII file for entering coordinates and material properties of individual fiber injections in a tabular format as shown below.

```
;  1st commentary line
;  2nd commentary line
;  m-th commentary line
x1   y1   z1   df1   m1   Tf1   Ys1
x2   y2   z2   df2   m2   Tf2   Ys2
... ...  ...  ...  ...  ...  ...
xn   yn   zn   dfn   mn   Tfn   Ysn
```

  where each injection is described by the following data fields, all on one line, separated by spaces:

  x, y, and z are the Cartesian coordinates of the injection point. The z coordinate must be included for both 3D and 2D solvers to maintain the sequence in which the data entries are interpreted, however, the z coordinate is not used in 2D.

  d_f is the fiber diameter

  m is the mass flow rate

  T_f is the temperature

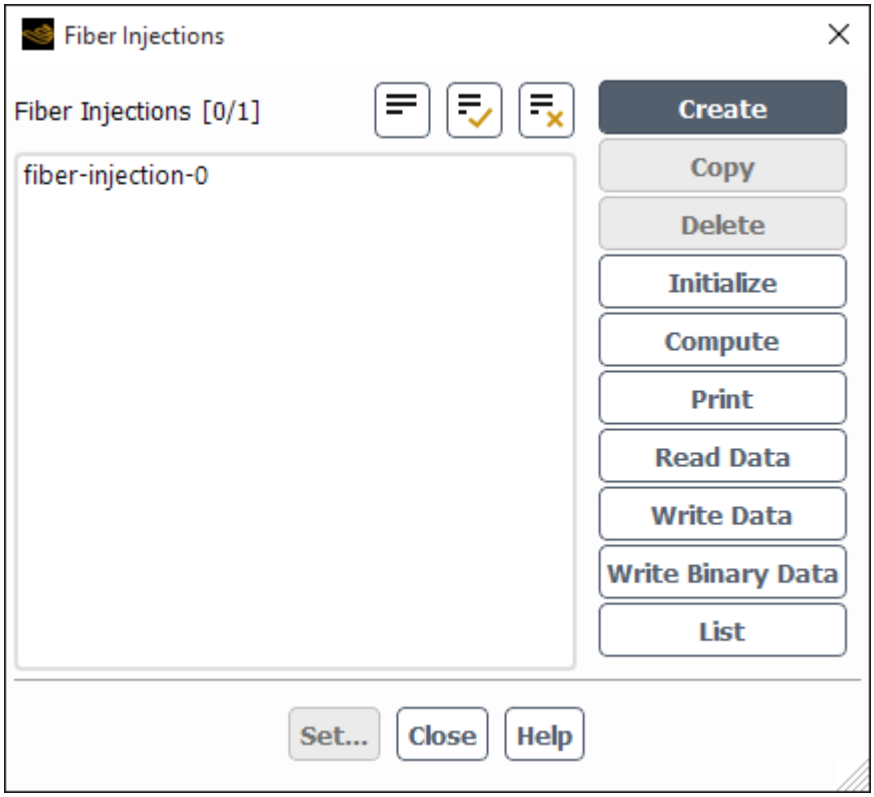
  Y_s is the solvent liquid mass fraction (ignored for melt spinning)

  You can include an arbitrary number of comments anywhere in the file. Commentary lines must begin with a semicolon (`;`).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

34.6.3. Working with Fiber Injections

Figure 34.2: The Fiber Injections Dialog Box



You will use the **Fiber Injections** dialog box (Figure 34.2: The Fiber Injections Dialog Box (p. 3533)) to create, modify, copy, delete, initialize, compute, print, read, write, and list fiber injections. To access the **Fiber Injections** dialog box, first make sure you enable a fiber model, then go to

Setup → Models → Fiber-Injections Edit...

34.6.3.1. Creating Fiber Injections

To create a fiber injection, click **Create**. A new fiber injection appears in the **Fiber Injections** list and the **Set Fiber Injection Properties** dialog box opens automatically to enable you to specify the fiber injection properties (as described in Point Properties Specific to Single Fiber Injections (p. 3540)).

34.6.3.2. Modifying Fiber Injections

To modify an existing fiber injection, select its name in the **Fiber Injections** list and click **Set...**. The **Set Fiber Injection Properties** dialog box opens, and you can modify the properties as needed.

34.6.3.3. Copying Fiber Injections

To copy an existing fiber injection to a new fiber injection, select the existing injection in the **Fiber Injections** list and click **Copy**. The **Set Fiber Injection Properties** dialog box will open with a new

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

fiber injection that has the same properties as the fiber injection you have selected. This is useful if you want to set another injection with similar properties.

34.6.3.4. Deleting Fiber Injections

You can delete a fiber injection by selecting its name in the **Fiber Injections** list and clicking **Delete**.

34.6.3.5. Initializing Fiber Injections

To initialize all solution variables of the fibers defined in a fiber injection, select its name in the **Fiber Injections** list and click **Initialize**. The solution variables will be set to the boundary condition values at the starting points of the fibers.

You can select several fiber injections when you want to initialize several fiber injections at one time.

Important:

If you do not select a fiber injection and click **Initialize**, all fiber injections will be initialized.

34.6.3.6. Computing Fiber Injections

To solve the fiber equations of a fiber injection for a number of iterations, select its name in the **Fiber Injections** list and click **Compute**. The solution variables of the fibers defined in this fiber injection will be updated for the number of iterations specified in Figure 34.13: Fiber Solution Controls Dialog Box (p. 3551).

You can select several fiber injections when you want to compute several fiber injections at one time.

Important:

If you do not select a fiber injection and click **Compute**, all fiber injections will be computed.

34.6.3.7. Print Fiber Injections

To print the fiber solution variables of a fiber injection into a file, select its name in the **Fiber Injections** list and click **Print**. A file name is generated automatically based on the name of the fiber injection and the number of the fiber. The solution variables of each fiber is stored in a separate file. You may use this file for an external postprocessing or analysis of fiber data.

You can select several fiber injections when you want to print several fiber injections at one time.

Important:

If you do not select a fiber injection and click **Print**, all fiber injections will be printed.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

34.6.3.8. Read Data of Fiber Injections

To read the data (properties and solution variables) of a fiber injection previously stored in a file, click **Read Data**. A file selection dialog box is opened where you can select the name of the file in a list or you can enter the file name directly. The names of all fiber injections included in the file are compared with the fiber injections already defined in the model. If the fiber injection exists already in your model, you are asked to overwrite it.

34.6.3.9. Write Data of Fiber Injections

While the settings of the continuous fiber model for numerics and models are stored in the Ansys Fluent case file, the data of the fibers and defined injections have to be stored in a separate file. To write the data of a fiber injection to a file, click **Write Data**. A file selection dialog box is opened where you can select the name of an existing file to overwrite it or you can enter the name of a new file.

You can select several fiber injections when you want to store several fiber injections in one file.

Important:

If you do not select a fiber injection and click **Write Data**, all fiber injections will be stored in the specified file.

34.6.3.10. Write Binary Data of Fiber Injections

To write the data of a fiber injection in binary format to a file, click **Write Binary Data**. A file selector dialog box is opened where you can select the name of an existing file to overwrite it or you can enter the name of a new file.

You can select several fiber injections when you want to store several fiber injections in binary format in one file.

Important:

If you do not select a fiber injection and click **Write Binary Data**, all fiber injections will be stored in binary format in the specified file.

34.6.3.11. List Fiber Injections

To list starting positions and boundary conditions of the fibers defined in a fiber injection, click **List**. Ansys Fluent displays a list in the console window. For each fiber you have defined, the list contains the following (in SI units):

- File number in the injection in the column headed (`NO`).
- x, y, and z position of the starting point in the columns headed (`X`), (`Y`), and (`Z`).
- Fiber velocity in the column headed (`U`).
- Temperature in the column headed (`T`).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Solvent mass fraction in the column headed (SOLVENT).
- Diameter in the column headed (DIAM).
- Mass flow rate in the column headed (MFLOW).
- The number of fibers represented by this fiber group in the column headed (FIBERS).
- The number of fiber grid cells defined for this fiber in the column headed (CELLS).
- A notification whether the starting point of the fiber is located inside or outside the domain in the column headed (IN DOMAIN?).

The boundary conditions at the take up point are also listed. This list consists of the following (in SI units):

- x, y, and z position of the take-up point in the columns headed (X), (Y), and (Z).
- Boundary condition type and its specified value (VELOCITY for given take-up velocity, FORCE for given force in the fiber) in the column headed (BOUNDARY CONDITION).

You can select several fiber injections when you want to list several fiber injections.

Important:

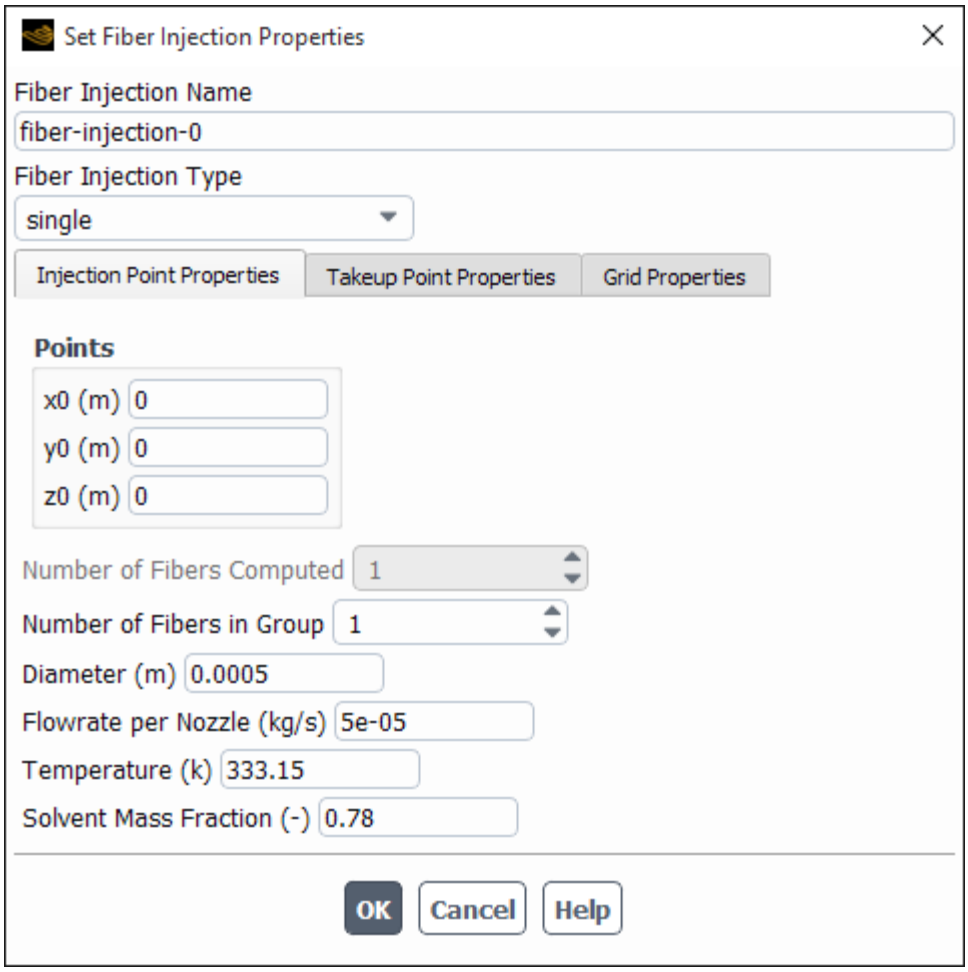
If you do not select a fiber injection and click **List**, all fiber injections will be listed.

34.6.4. Defining Fiber Injection Properties

Once you have created an injection (using the **Fiber Injections** dialog box, as described in Creating Fiber Injections (p. 3533)), you will use the **Set Fiber Injection Properties** dialog box (Figure 34.3: The Set Fiber Injection Properties Dialog Box (p. 3537)) to define the fiber injection properties. (Remember that this dialog box opens when you create a new fiber injection, or when you select an existing fiber injection and click **Set...** in the **Fiber Injections** dialog box.)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 34.3: The Set Fiber Injection Properties Dialog Box



The procedure for defining a fiber injection is as follows:

1. If you want to change the name of the fiber injection from its default name, enter a new one in the **Fiber Injection Name** field. This is recommended if you are defining a large number of injections so you can easily distinguish them.

2. Choose the type of fiber injection in the **Fiber Injection Type** drop-down list. The three choices (**single**, **line**, and **matrix**) are described in Fiber Injection Types (p. 3532).

3. Click the **Injection Point Properties** tab (the default), and specify the point coordinates according to the fiber injection type, as described in Point Properties Specific to Single Fiber Injections (p. 3540)–Point Properties Specific to Matrix Fiber Injections (p. 3540).

4. If each of the defined fibers is referring to a group of fibers, enter the number of fibers in **Number of Fibers in Group**. If your nozzle plate has 400 holes and you can simulate them as a line fiber injection with 5 groups, you have to enter a value of 80. This means that only 5 fibers are solved numerically, but each of these fibers stands for 80 fibers to be used to compute the source terms

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

for the surrounding fluid. This enables you to reduce computing efforts while achieving a proper coupling with the surrounding fluid.

Important:

Note that the **Number of Fibers in Group** is applied to all fibers, defined in your injection. If the number of fiber groups in your line or matrix injection is not the same for all fibers in this injection, you should split this injection into several fiber injections.

5. Specify the diameter of the nozzle in the **Diameter** field.
6. Enter the mass flow rate for a single nozzle in the **Flow rate per Nozzle** field. This will be used to compute the starting velocity of the fiber fluid.

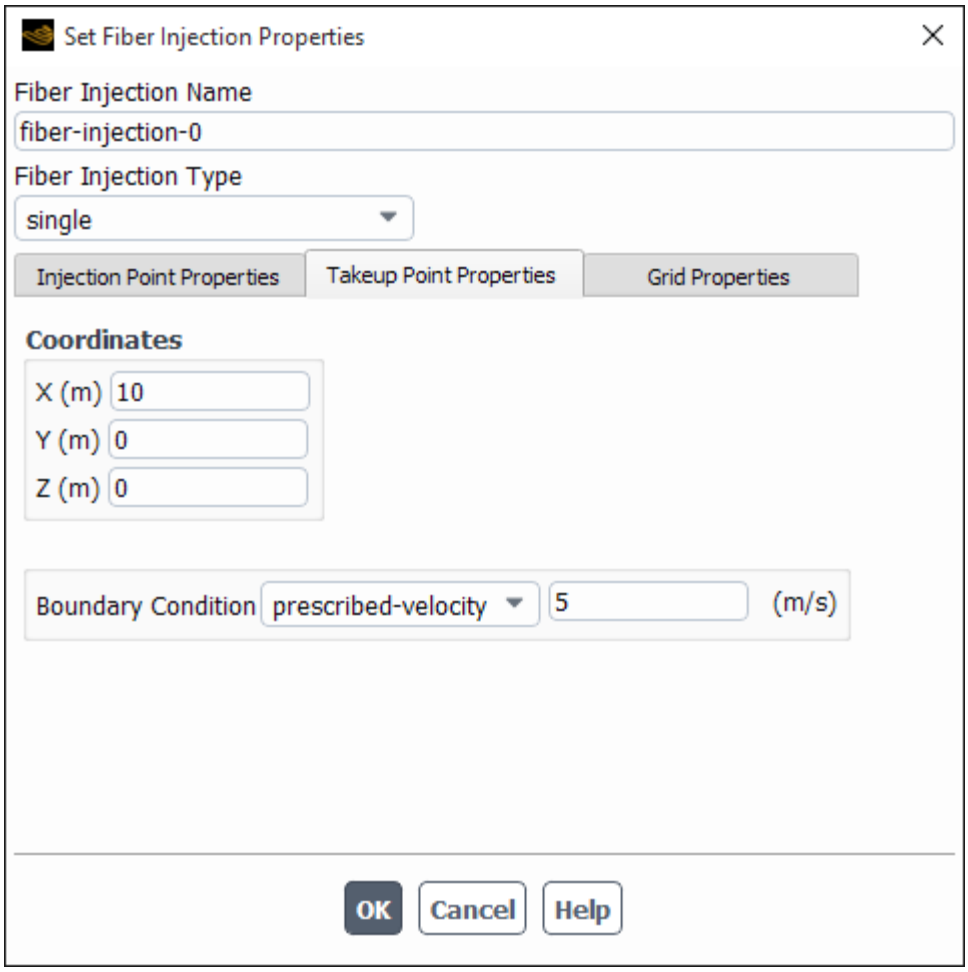
Important:

Note that the value specified refers to one single nozzle and not to the mass flow rate of all fibers defined in this fiber injection.

7. Specify the temperature of the fiber fluid leaving the nozzle in the **Temperature** field.
8. If you are modeling dry spun fibers you also have to enter the solvent's mass fraction at the nozzle in the **Solvent Mass Fraction** field.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 34.4: The Set Fiber Injection Properties Dialog Box With Take-Up Point Properties



9. Click the **Takeup Point Properties** tab and enter the coordinates of the take-up point (see Figure 34.4: The Set Fiber Injection Properties Dialog Box With Take-Up Point Properties (p. 3539)).

   Important:

   Note that all fibers defined in the fiber injection are collected at the same point. If the fibers of your line or matrix injection vary in this property, you have to define them using several fiber injections.

10. Select the appropriate boundary condition from the **Boundary Condition** drop-down list and specify the value for this boundary condition. Choose **prescribed-velocity** if you know the drawing velocity (also called take-up velocity). Choose **tensile-force** with a value of zero if you want to simulate free falling fibers at the take-up point.

11. Click the **Grid Properties** tab and enter all data needed to generate the fiber grid as described in Define Fiber Grids (p. 3541).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

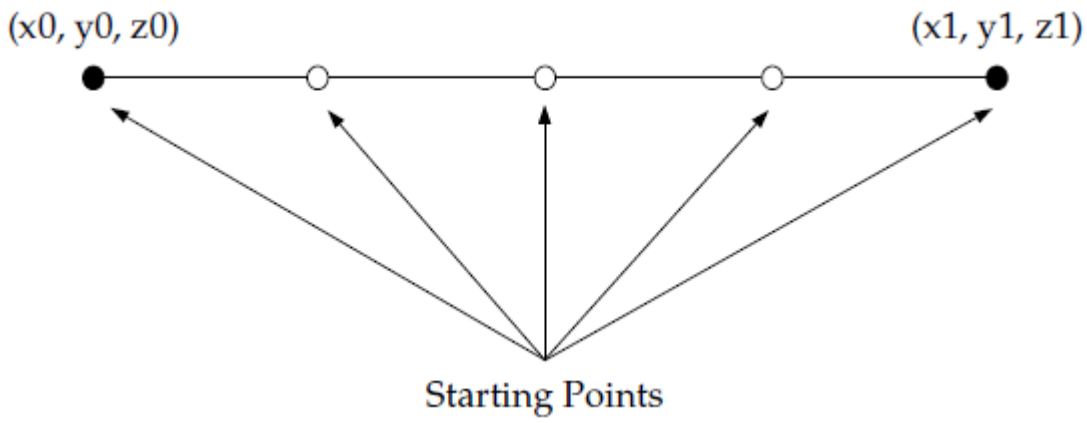
34.6.5. Point Properties Specific to Single Fiber Injections

For a single fiber injection, you have to specify the coordinates of the starting point of the fiber. Click the **Injection Point Properties** tab and set the x, y, and z coordinates in the **x0**, **y0**, and **z0** fields of the **Points** box. (**z0** will appear only for 3D problems.)

34.6.6. Point Properties Specific to Line Fiber Injections

In a line fiber injection the starting points of the fibers are arranged on a line at a constant distance (see Figure 34.5: Line Injections (p. 3540)). You have to specify the coordinates of the starting point and the end point of this line. Click the **Injection Point Properties** tab in the **Set Fiber Injection Properties** dialog box (Figure 34.3: The Set Fiber Injection Properties Dialog Box (p. 3537)). In the **Points** region, set the x, y, and z coordinates in the **x0**, **y0**, and **z0** fields for the starting point of the line and the x, y, and z coordinates in the **x1**, **y1**, and **z1** fields for the end point of the line. (**z0** and **z1** will appear only for 3D problems.)

Figure 34.5: Line Injections



In addition to the coordinates, you have to set the number of fibers defined in the line injection by entering the appropriate value in the **Point Density Edge1** field. See Figure 34.5: Line Injections (p. 3540) for an example of a line fiber injection with a **Point Density Edge1** of 5.

34.6.7. Point Properties Specific to Matrix Fiber Injections

In a matrix fiber injection the starting points of the fibers are arranged in several rows having the shape of a rectangle or a parallelogram. Each row has the same distance to the previous and is divided into equal sections (see Figure 34.6: Matrix Injections (p. 3541)).

You have to specify the coordinates of the starting point and the end of point of the first row and the coordinates where the last row should start. Click the **Injection Point Properties** tab in the **Set Fiber Injection Properties** dialog box (Figure 34.3: The Set Fiber Injection Properties Dialog Box (p. 3537)). In the **Points** region, set the x, y, and z coordinates in the **x0**, **y0**, and **z0** fields for the starting point and the x, y, and z coordinates in the **x1**, **y1**, and **z1** fields for the end point of the first row of fibers. The x, y, and z coordinates of the starting point of the last row have to be entered in the **x2**, **y2**, and **z2** fields.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

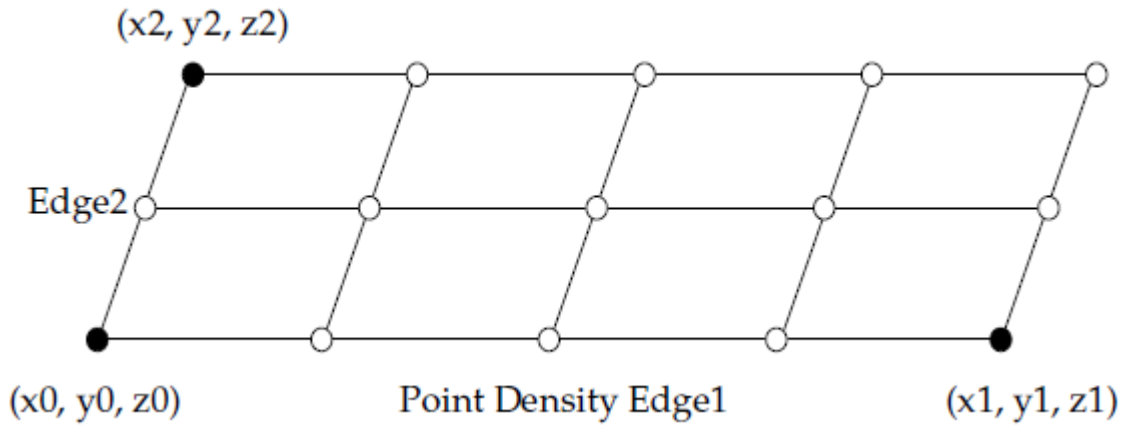
At each row, the number of fibers specified in the **Point Density Edge1** field are injected. The number of rows to be injected is specified in **Edge2**.

You can double check the number of fibers computed in this fiber injection if you inspect the **Numbers of Fibers Computed** field.

Important:

Note that this fiber injection type is only available for 3D problems.

Figure 34.6: Matrix Injections



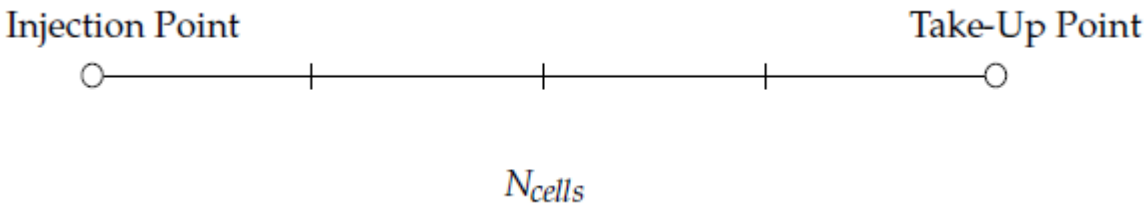
34.6.8. Define Fiber Grids

Each fiber consists of a straight line between the injection point and the take-up point. It is divided into a number of finite volume cells. Every fiber defined in a fiber injection has its own grid, which you can specify if you click the **Grid Properties** tab in the **Set Fiber Injection Properties** dialog box.

34.6.8.1. Equidistant Fiber Grids

To define an equidistant grid for the fibers select **equidistant** from the **Grid Type** drop-down list. Specify the **Number of Cells** into which every fiber of the fiber injection will be divided.

Figure 34.7: Equidistant Fiber Grid

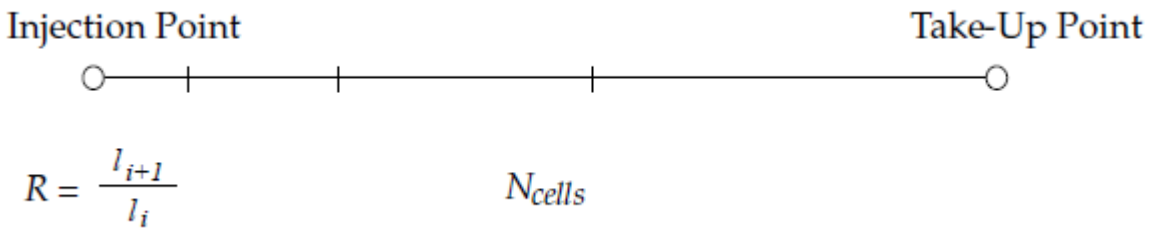


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

34.6.8.2. One-Sided Fiber Grids

A one-sided grid is graded near the injection point of the fiber. To define a one-sided grid for the fibers select **one-sided** from the **Grid Type** drop-down list. Specify the **Number of Cells** into which every fiber of the fiber injection will be divided. In addition, you have to specify the ratio, R, between each subsequent fiber cell in the **Grid Growth Factor at Injection Point** field. Values larger than 1 refine the grid, while values smaller than 1 coarsen the grid.

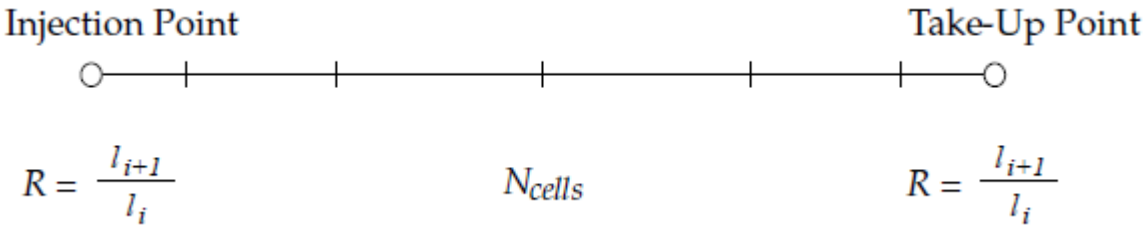
Figure 34.8: One-Sided Fiber Grid



34.6.8.3. Two-Sided Fiber Grids

A two-sided grid is graded near the injection point as well as at the take-up point of the fiber. To define a two-sided grid for the fibers, select **two-sided** from the **Grid Type** drop-down list. Specify the **Number of Cells** into which every fiber of the fiber injection will be divided. You also have to specify the ratio, R, between each subsequent fiber cell at the injection point in the **Grid Growth Factor at Injection Point** field and the ratio, R, near the take-up point in the **Grid Growth Factor at Takeup Point** field. Values larger than 1 refine the grid, while values smaller than 1 will coarsen it.

Figure 34.9: Two-Sided Fiber Grid



34.6.8.4. Three-Sided Fiber Grids

A three-sided grid consists of three sides where the fiber grid can be graded. The first side is near the injection point. The other two sides are around a refinement point within the fiber grid. Both sides at the refinement point are graded at the same ratio between the fiber grid cell lengths. To define a three-sided grid for the fibers, select **three-sided** from the **Grid Type** drop-down list (see Figure 34.11: Defining a Three-Sided Fiber Grid Using the Set Fiber Injection Properties Dialog Box (p. 3543)). Specify the **Number of Cells until Grid Refinement Point**, the **Grid Growth Factor at Injection Point**, and the **Grid Growth Factor at Local Grid Refinement Point**. You also have to specify the location of the grid refinement point in the **Location of Local Grid Refinement Point** field in a dimensionless way. The value you have to enter is relative to the fiber length and

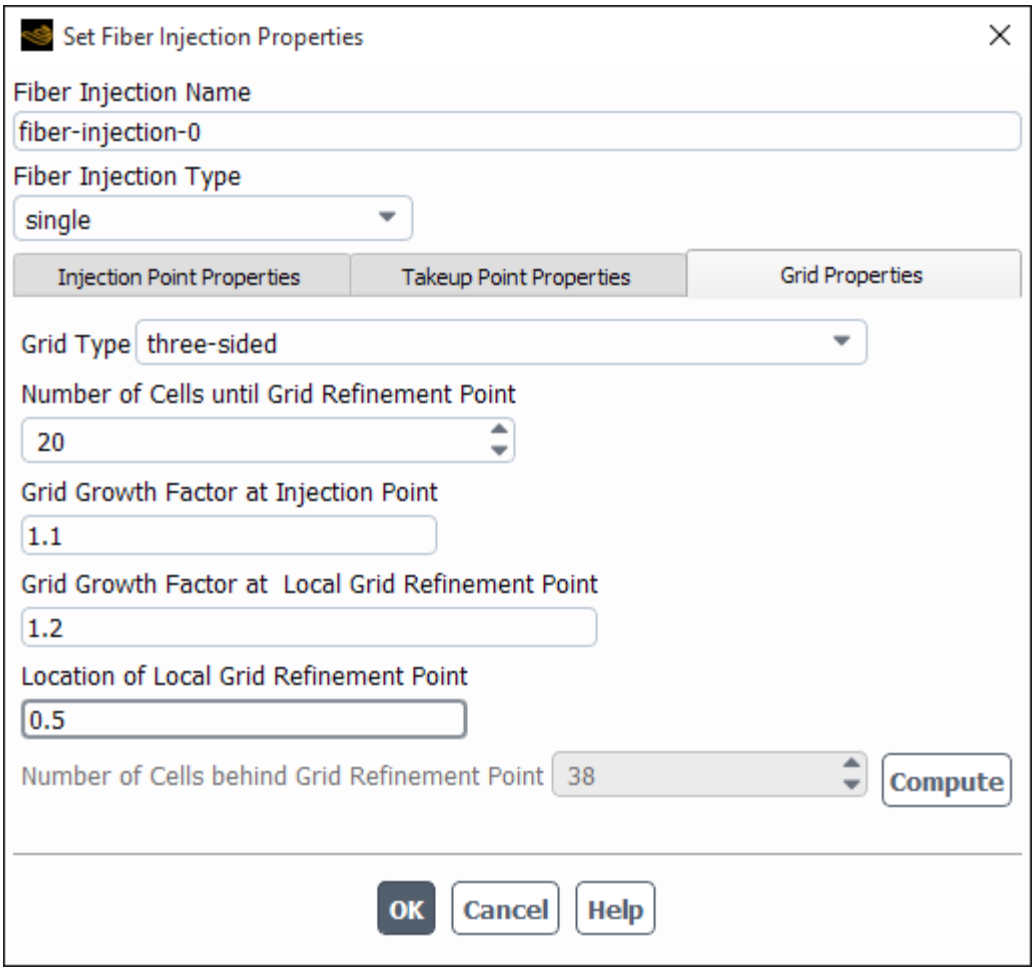
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

may be between 0 and 1. Values larger than 1 refine the grid, while values smaller than 1 coarsen the grid.

Click the **Compute** button to estimate the **Number of Cells behind Grid Refinement Point**.

Figure 34.10: Three-Sided Fiber Grid

Figure 34.11: Defining a Three-Sided Fiber Grid Using the Set Fiber Injection Properties Dialog Box



34.7. User-Defined Functions (UDFs) for the Continuous Fiber Model

The continuous fiber model enables you to apply custom correlations for drag, heat transfer, mass transfer coefficients, as well as material properties to the fibers by means of user-defined functions (UDFs). You are provided with a C template file named `fiber_fluent_interface.c` that contains

source code for the drag, heat transfer, mass transfer coefficient, and the fiber property UDFs. You will need to modify this UDF source file to suit your application, compile it, and hook the resulting UDF object(s) to the fiber model. This process is described below.

For more information, see the following sections:

34.7.1. UDF Setup

34.7.2. Customizing the fiber_fluent_interface.c File for Your Fiber Model Application

34.7.3. Compile Fiber Model UDFs

34.7.4. Hook UDFs to the Continuous Fiber Model

34.7.1. UDF Setup

Before you can edit the UDF template and create your custom UDF for drag, heat transfer or mass transfer coefficient, you must copy the `fiber` directory to your working directory. The `fiber` directory contains all of the libraries and support files that are required to compile UDFs for the fiber model and build shared libraries for the architecture that you specify.

34.7.1.1. Linux Systems

Make a local copy of the `fiber` directory by copying it from the path below to your working directory.

path`/ansys_inc/v242/fluent/fluent24.2.0/addons/fiber/`

where *path* is the directory in which you have installed Ansys Fluent.

34.7.1.2. Windows Systems

1. Open a Visual Studio `.NET` DOS prompt.
2. Make a local copy of the `fiber` folder (not a symbolic link) by copying it from the folder below to your working folder.

   path`\ANSYS Inc\v242\fluent\fluent24.2.0\addons\fiber\`

   where *path* is the folder in which you have installed Ansys Fluent (by default, the path is `C:\Program Files`).

34.7.2. Customizing the fiber_fluent_interface.c File for Your Fiber Model Application

Now that you have copied the `fiber` directory to your working directory, you can edit the UDF template file and customize it to fit your model needs.

1. In your working directory, change directories to `fiber/src`. The `/src` directory contains the UDF template source file `fiber_fluent_interface.c`.
2. In the `/src` directory, use any text editor and edit `fiber_fluent_interface.c`.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Scroll down to the bottom of the `fiber_fluent_interface.c` file to the section that contains concatenated functions for friction factor (drag coefficient), heat transfer coefficient, mass transfer coefficient, fiber viscosity, fiber density, fiber thermal conductivity, fiber specific heat, and vapor liquid equilibrium, respectively. These are the UDFs that you can modify and customize.

4. Edit the function(s) you require for your particular application. Save `fiber_fluent_interface.c` and overwrite the existing file.

Important:

Do not save the file with another name because it will not be recognized by the system.

Important:

The function names of the templates `user_friction_factor`, `user_heat_transfer_coefficient`, and `user_mass_transfer_coefficient` *must not* be altered because they are called by other routines in the continuous fiber model.

34.7.2.1. Example: Heat Transfer Coefficient UDF

Below is an example of a heat transfer coefficient UDF that is defined in `fiber_fluent_interface.c`. The function is taken from Kase and Matsuo 291 and is implemented as the `kase-matsuo-1` option in the fiber model. The function name `user_heat_transfer_coefficient` *must not*, under any circumstances, be altered because it is called by other functions in the continuous fiber model.

There are two arguments to the `user_heat_transfer_coefficient` UDF: `Fiber` and `Local_Fiber_Data_Type`. `Fiber *f` is a pointer to the fiber structure that contains information about the fiber and `Local_Fiber_Data_Type *fd` accesses temporary variables that are needed during the calculation of the fiber.

In the sample UDF below, a loop is performed over all fiber grid cells using the macro `FIB_N(f)` which represents the number of grid cells for the fiber `f`. The Reynolds number is computed based on the relative velocity, $u_f - u_{par}$ (`ABS(FIB_C_U(f,i)-fd->up[i])`); the fiber diameter, `FIB_C_D(f,i)`; the density of the surrounding fluid, `fd->rho[i]` and the viscosity of the surrounding fluid, `fd->vis[i]`. The heat transfer coefficient α is computed from the Nusselt number using the thermal conductivity of the surrounding fluid, λ, (`fd->k[i]`) and is stored in `fd->alpha[i]`.

```
void
user_heat_transfer_coefficient(Fiber *f, Local_Fiber_Data_Type *fd)
{
 int i;
 real Red, Nud;

 /* model from Kase/Matsuo (1967) */
 for (i=0; iFIB_N(f); i++)
 {
   /* compute Reynolds number based on relative velocity */
   Red = ABS(FIB_C_U(f,i)-fd->up[i])*FIB_C_D(f,i)*fd->rho[i]/fd->vis[i];
   Nud = 0.42*pow(Red, 0.334);
   /* store heat transfer coefficient for latter use */
```

```
    fd->alpha[i] = Nud*fd->k[i]/FIB_C_D(f,i);
   }
 }
```

All variables and macros that are used in `user_heat_transfer_coefficient` are defined in header files provided with the continuous fiber model. For example, you will find the type definition `Fiber` and the macros that are used to access variables of a single fiber in the header file `fiber.h`. Temporary variables used in the type definition of `Local_Fiber_Data_Type` can be found in the header file `fib-mem.h`.

Important:

Note that you must not modify the header files provided with the continuous fiber model. Otherwise the compiled library will not be compatible with Ansys Fluent and will show run time errors.

34.7.2.2. Example: Fiber Specific Heat Capacity UDF

You can also calculate fiber material properties using UDFs. Ansys Fluent provides a UDF template for every fiber property. You can modify the UDF templates for your specific case.

The template for the fiber density is as follows:

```
real User_Fiber_Specific_Heat(Fiber *f, real temp, real liq, int i)
{
  real yi[MAX_SPE_EQNS];
  real cp_p;
/* compute cp based on FLUENT's standard procedure selected in materials panel */
  cp_p = Specific_Heat(FIB_MATERIAL_POLYMER(f), temp, 0., yi);
  if (FIB_DRY_SPINNING(f))
  {
    real cp_s;
    cp_s = Specific_Heat(FIB_MATERIAL_SOLVENT(f), temp, 0., yi);
    return cp_p*(1.-liq)+cp_s*liq;
   }
  return cp_p;
}
```

Important:

The function names of the templates `User_Friction_Factor`, `User_Heat_Transfer_Coefficient`, `User_Mass_Transfer_Coefficient`, `User_Fiber_Viscosity`, `User_Fiber_Density`, `User_Fiber_Thermal_Conductivity`, `User_Fiber_Specific_Heat`, and `User_Fiber_Vle` must not be altered since they are called by other routines of the continuous fiber model.

All variables and macros defined in the header files are provided with the continuous fiber model. For example, you will find the type definition `Fiber` and the macros to access variables of a single

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

fiber in the header file `fiber.h`. The temporary variables used in the type definition of `Local_Fiber_Data_Type` can be found in the header file `fib-mem.h`.

Important:

You must not modify the header files provided with the continuous fiber model. Otherwise the compiled library will not be compatible with Ansys Fluent resulting in runtime errors (messages will be printed in the console).

34.7.3. Compile Fiber Model UDFs

Once you have customized the UDF template `fiber_fluent_interface.c` for your fiber model application, you are now ready to compile the source file using the text interface. Follow the procedure below for Linux Systems and Windows Systems, respectively.

34.7.3.1. Linux Systems

In the `/fiber` in your working directory, run the `make` command that will compile your fiber mode UDF and build a shared library for the architecture you are running. To do this, type the following command at the prompt:

```
make -f Makefile-client FLUENT_ARCH=your_arch
```

where `your_arch` is replaced by the architecture of the machine you are running (for example, lnx86).

For example, if your computer architecture is `lnx86` type the following command in a terminal session:

```
make -f Makefile-client FLUENT_ARCH=lnx86
```

To identify the architecture of the machine you are running on, scroll up the Ansys Fluent console window to the message that begins with "Starting".

When you run the `make` process, the source code (`fluent_fiber_interface.c`) will be compiled into object code and a shared library will be built for the computer architecture and version of Ansys Fluent you are running. Messages about the compile/build process will be displayed on the console window. You can view the compilation history in the log file that is saved in your working directory. Below is an example of console messages for a lnx86 architecture running a 2D version of Ansys Fluent.

```
Working...
for d in lnx86[23]*; do \
   ( \
      cd $d; \
      for f in ../../src*.[ch] ../../src/makefile; do \
         if [ ! -f 'basename $f' ]; then \
           echo "# linking to " $f "in" $d; \
           ln -s $f .; \
        fi; \
      done; \
      echo ""; \
      echo "# building library in" $d; \
      make -kmakelog 2&1; \
      cat makelog; \
```

```
  ) \
done
# linking to ...  myudf.c in lnx86/2d

# building library in lnx86/2d
make[1]: Entering directory ..../udf_names.c
# Generating udf_names
make[2]: Entering directory ..../fluent_fiber_interface.c
make libudf.so ...
# Compiling udf_names.o ...
# Compiling profile.o ...
# Linking libudf.so ...
make[2]: Leaving directory ..../udf_names.c
make[1]: Leaving directory ..../fluent_fiber_interface.c

You can also see the 'log'-file in
the working directory for compilation history

Done.
```

34.7.3.2. NT/Windows Systems

1. Make a local copy of the `fiber` folder. Do not create a shortcut.

   a. Create a directory for the custom `fiber` libraries (for example, `/home/custom-build`).

   b. Copy the `fiber` library from the installation of Ansys Fluent (`Ansys Inc\v242\fluent\fluent24.2.0\addons`) to your newly created directory.

2. Open the **Visual Studio .NET** DOS prompt (see Compiler Requirements for Windows Systems in the *Ansys, Inc. Installation Guides* for compilers that are compatible with Ansys Fluent).

3. Set the `FLUENT_INC` environment variable to the current Ansys Fluent installation directory, for example:

   `FLUENT_INC="<driveletter>\ANSYS Inc\v242\fluent"`

   where `<driveletter>` is the target drive letter including the colon (for example, `D:`).

4. Enter the `fiber\src` folder:

   ```
   cd fiber/src
   ```

5. Make changes to the `fiber_fluent_interface.c` file.

6. Return to the `fiber` folder:

   ```
   cd ..
   ```

7. Issue the following command in the command window:

   ```
   build_fiber_udf.bat
   ```

8. Define the `FLUENT_ADDONS` environment variable to correspond to your customized version of the continuous fiber module in one of following ways:

   - Set the path to `FLUENT_ADDONS` globally.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- In the **Environment** tab of Fluent Launcher, enter information for `FLUENT_ADDONS` in the **Other Environment Variables** field, for example:

  `FLUENT_ADDONS=<driveletter>\home\custom-build`

  where `<driveletter>` is the target drive letter including the colon (for example, `D:`). Note that there should be no spaces in the above path.

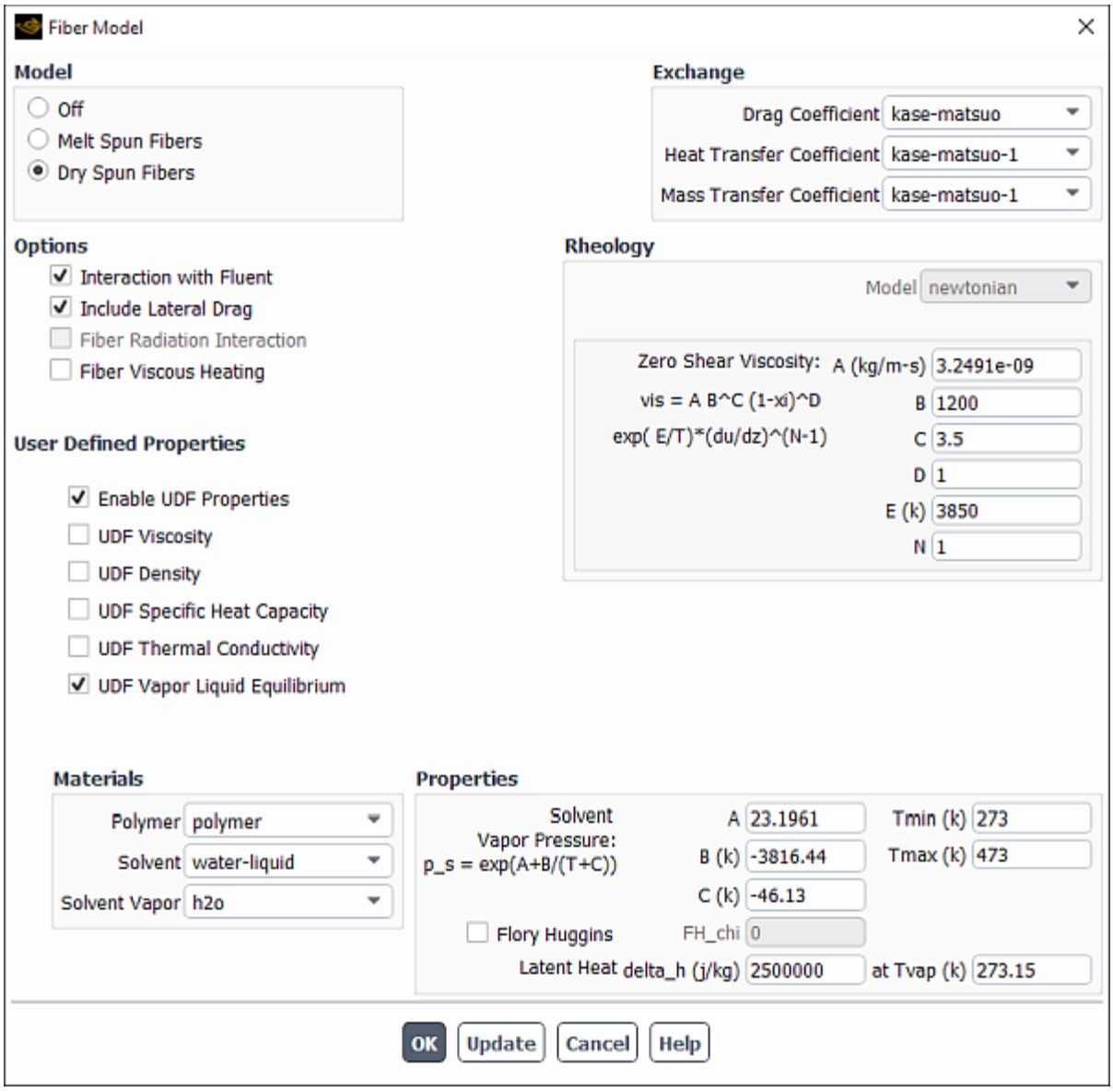
34.7.4. Hook UDFs to the Continuous Fiber Model

Once you have successfully compiled your continuous fiber model UDF(s), the `user-defined` option will appear in drop-down lists for parameters under **Exchange** in the **Fiber Model** dialog box (Figure 34.12: The Fiber Model Dialog Box (p. 3550)). To hook your UDF to a particular fiber model parameter, simply select `user-defined` from the drop-down list for **Drag Coefficient**, **Heat Transfer Coefficient**, or **Mass Transfer Coefficient** and click **OK**.

If you want to use UDFs to define fiber material properties, select the **Enable UDF Properties** check box, and then enable individual property UDFs under the **User Defined Properties** group box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 34.12: The Fiber Model Dialog Box



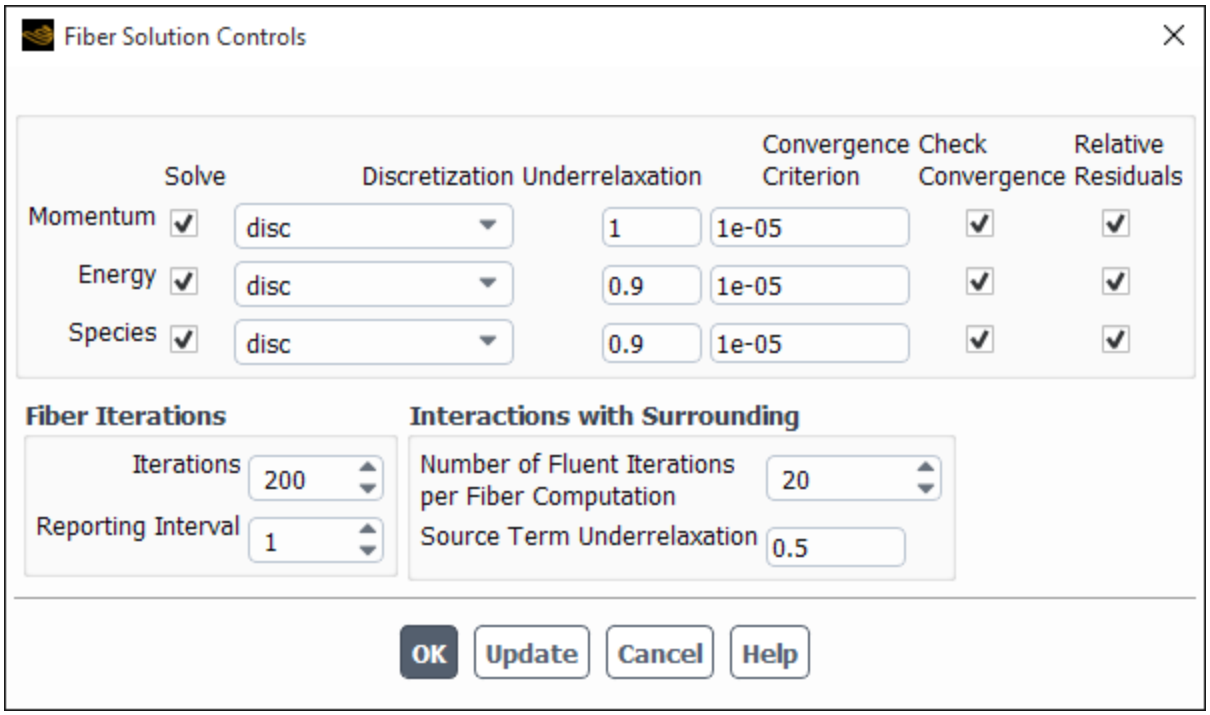
34.8. Fiber Model Solution Controls

To access the **Fiber Solution Controls** dialog box, go to

Setup → Models → Fiber-Controls Edit...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 34.13: Fiber Solution Controls Dialog Box



The **Fiber Solution Controls** dialog box enables you to set common solution parameters for the fiber equations and their coupling with the surrounding fluid.

Solve

is used to enable and disable the solution of the fiber equations for **Momentum** (Equation 22.4), **Energy** (Equation 22.11), and **Species** (Equation 22.1). When switching off the solution of one of these equations, it is not computed for any fibers defined in your model.

Discretization

provides a drop-down list where you can assign to each of the fiber equations one of three different discretization schemes explained in Discretization of the Fiber Equations.

Underrelaxation

contains all under-relaxation factors for all fiber equations that are being solved in the continuous fiber model. See Under-Relaxation for additional background information and see Solution Strategies for how to make use of under-relaxation factors in your solution strategy.

Convergence Criterion

is used to stop the fiber iterations when the residuals of all fiber equations are below the prescribed criteria. You can define a separate convergence criterion for every fiber equation.

Check Convergence

must be turned on if you want to compare the residuals of the fiber equations with the **Convergence Criterion**. If you turn this option off, the given number of fiber iterations will be computed.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Relative Residuals

are used to compute the change of the residual of two subsequent iterations relative to the residual of the last iteration by applying Equation 22.20. The result of this is compared to the **Convergence Criterion** to check whether convergence has been achieved.

Iterations

defines the number of **Fiber Iterations** performed every time the fiber equations are updated.

Reporting Interval

sets the number of fiber iterations that will pass before the residuals will be printed.

You can reduce the output to the last fiber iteration by specifying **Reporting Interval** as 0. This is recommended when performing a solution that is coupled with the surrounding flow.

Number of Fluent Iterations per Fiber Computation

sets the number of Fluent iterations before the fiber equations are updated in a solution that is coupled with the surrounding flow.

Source Term Underrelaxation

factor is used to under-relax the fiber source term exchange to the surrounding fluid. In a converged solution this value does not influence your predictions.

For additional information on how to set and choose values for the options in the **Fiber Solutions Control** dialog box, see Solution Strategies.

34.9. Postprocessing for the Continuous Fibers

After you have completed your inputs and performed any coupled calculations, you can display the location of the fibers and source terms of the fibers, and you can write fiber data to files for further analysis of fiber variables.

The following data can be displayed using graphical and alphanumeric reporting facilities:

- Graphical display of fiber locations
- Exchange terms with surrounding fluid
- Fiber solution variables.

For more information, see the following sections:

34.9.1. Display of Fiber Locations and Grid Points

34.9.2. Exchange Terms of Fibers

34.9.3. Analyzing Fiber Variables

34.9.4. Running the Fiber Module in Parallel

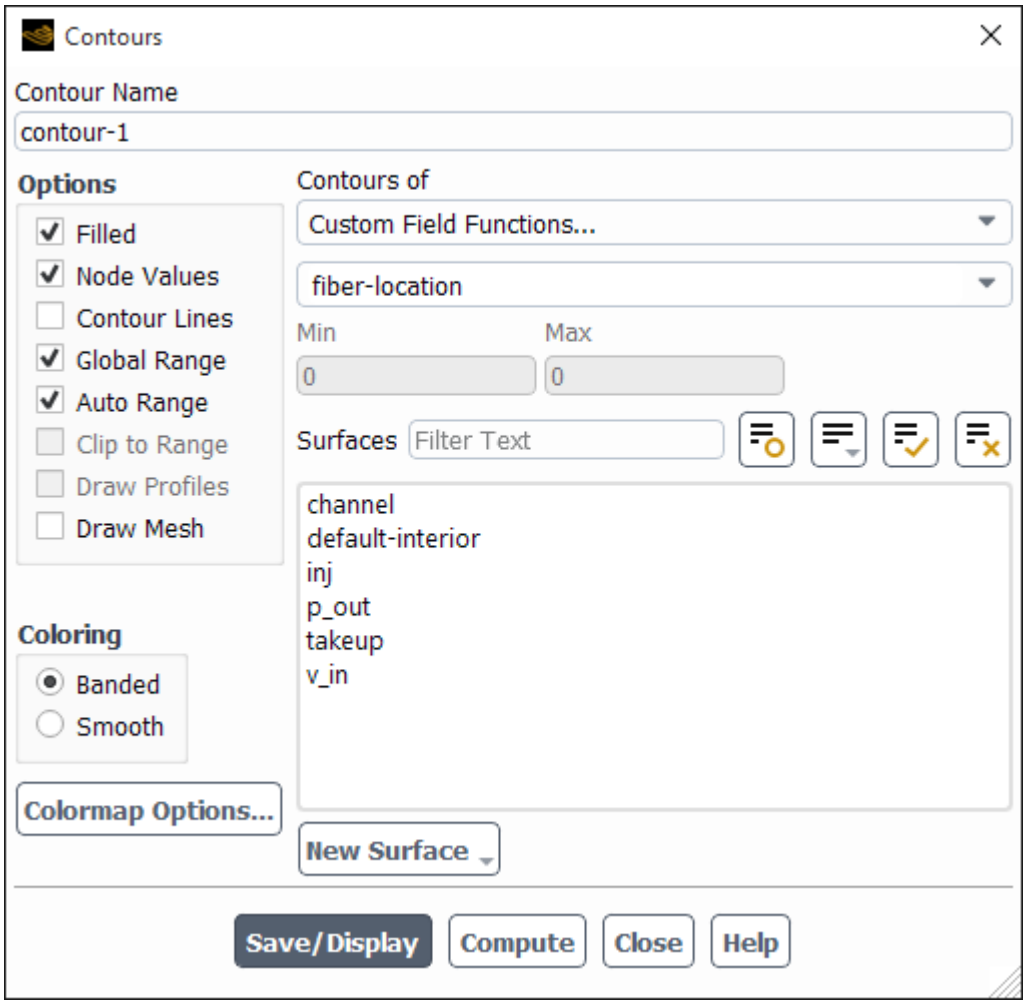
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

34.9.1. Display of Fiber Locations and Grid Points

When you have defined fiber injections, as described in Defining Fibers (p. 3531), you can display the location of the fibers using Ansys Fluent's **Contour** plot facility (Figure 34.14: Displaying Fiber Locations Using the Contours Dialog Box (p. 3553)).

Results → Graphics → Contours Edit...

Figure 34.14: Displaying Fiber Locations Using the Contours Dialog Box



In the **Contours of** drop-down list, select **Custom Field Functions...**, then select **fiber-location**. The values in this field are between zero and the number of fiber cells in an Ansys Fluent grid cell.

You may generate iso-surfaces of constant values of **fiber-location** to display the fibers in 3D problems.

You can also display the locations of the fibers and their grid discretization using the **Fiber Mesh Display** dialog box, which can be accessed from the **Results**/**Graphics**/**Fiber Mesh** tree item.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

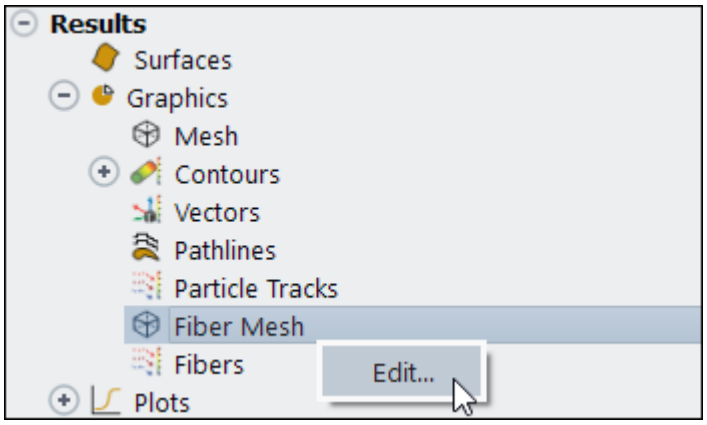
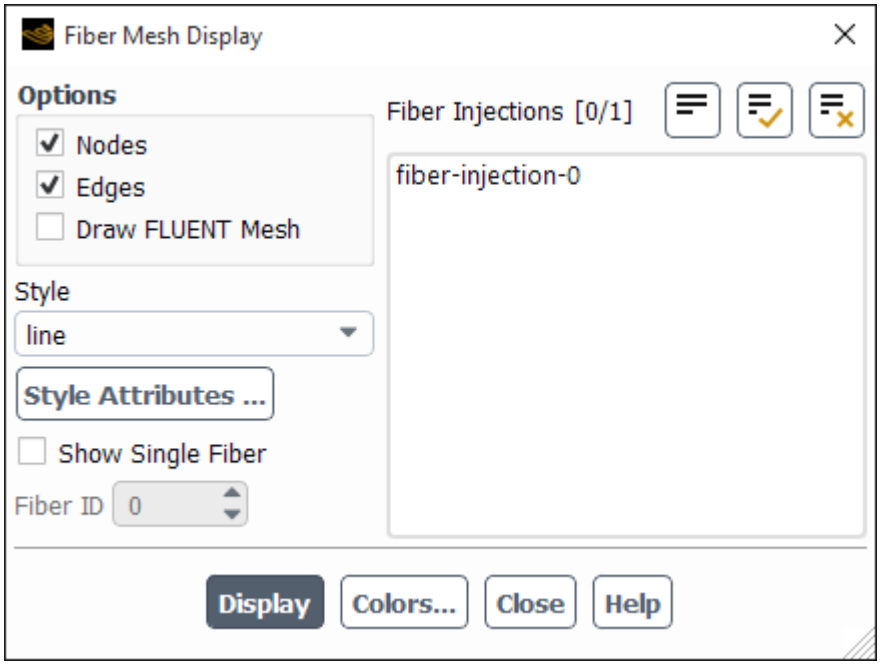


Figure 34.15: The Fiber Mesh Display Dialog Box



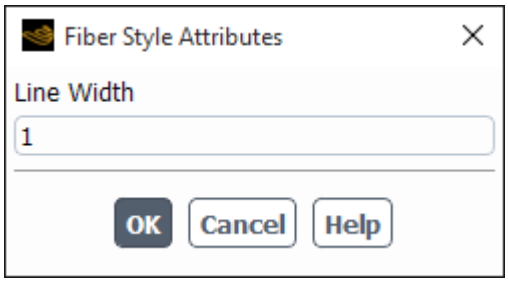
The following mesh discretization options are available:

- **Nodes**: displays the end points of fiber volume cells
- **Edges**: displays a line along the path of fibers in the domain
- **Draw Fluent mesh**: Enables the display of the CFD mesh geometry with the mesh display settings specified in the **Mesh Display** dialog box. The **Mesh Display** dialog box will appear automatically when you enable the **Draw Fluent mesh** option.

You can set the fiber **Line Width** in the **Fiber Style Attribute** dialog box accessed by clicking the **Style Attributes...** button.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 34.16: The Fiber Style Attributes Dialog Box



For the injections selected in the **Fiber Injections** selection list, you can display either all injection fibers or single fibers with a specified **Fiber ID** (when the **Show Single Fiber** option is selected).

34.9.2. Exchange Terms of Fibers

The continuous fiber model computes and stores the exchange of momentum, heat, mass, and radiation in each control volume in your Ansys Fluent model. You can display these variables graphically by drawing contours, profiles, and so on. They all are contained in the **Custom Field Functions...** category of the variable selection drop-down list that appears in postprocessing dialog boxes:

fiber-mass-source

defined by Equation 22.22.

fiber-x-momentum-source

is the x-component of the momentum exchange, defined by Equation 22.21.

fiber-y-momentum-source

is the y-component of the momentum exchange, defined by Equation 22.21.

fiber-z-momentum-source

is the z-component of the momentum exchange, defined by Equation 22.21.

fiber-energy-source

defined by Equation 22.23.

fiber-dom-absorption

defined by Equation 22.24.

fiber-dom-emission

defined by Equation 22.25.

Note that these exchange terms are updated and displayed only when coupled computations are performed.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

34.9.3. Analyzing Fiber Variables

You can use Ansys Fluent's **Plot** facilities to analyze fiber solution variables such as fiber velocity, temperature, diameter, and so on.

34.9.3.1. XY Plots

To investigate fiber variables you have to generate an xy-file for every variable using the following procedure from the text command interface (assuming that you have already defined a fiber injection):

1. Store an xy-file with the variable of interest.

   `continuous-fiber` → `print-xy`

2. Enter the fiber injection of interest when asked for

   `Fiber Injection name>`

3. Specify a variable for `x-column`. This will be used as abscissa in the file plot.

4. Specify another variable for `y-column`, which will be used as ordinate in the file plot.

5. Specify a file name when asked for `XY plot file name`.

6. Load the file with Ansys Fluent's xy plot facilities, described in the Ansys Fluent manual.

The following variables can be entered as `x-column` and `y-column`:

- axial drag
- conductivity
- curvature
- diameter
- enthalpy
- evaporated mass flux
- heat transfer coefficient
- lateral drag
- length
- mass flow
- mass fraction
- mass transfer coefficient
- Nusselt number

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

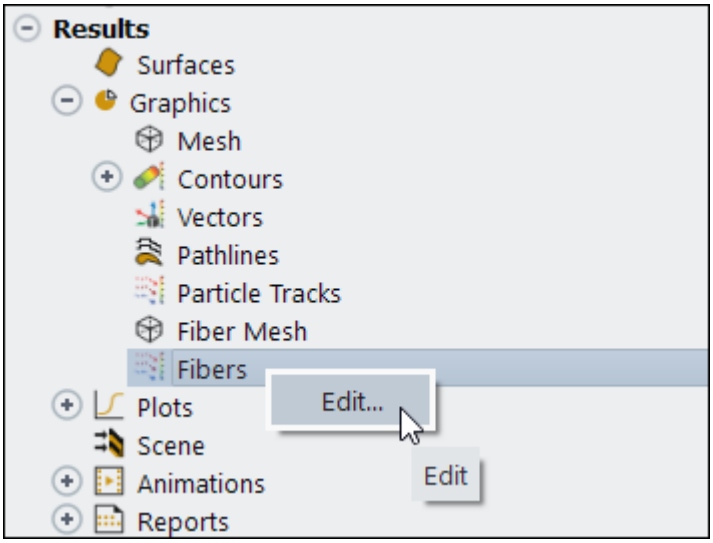
- Sherwood number
- Reynolds number
- specific heat
- surface mass fraction
- temperature
- tensile force
- user variable
- velocity
- velocity gradient
- viscosity

If a fiber injection consists of several fibers, the data of all fibers in this fiber injection will be stored in the xy-file.

In addition to this, you can store a fiber history file for a fiber injection as described in Print Fiber Injections (p. 3534). This can be used to analyze a single fiber using Ansys Fluent xy-plot facilities or by external postprocessing programs.

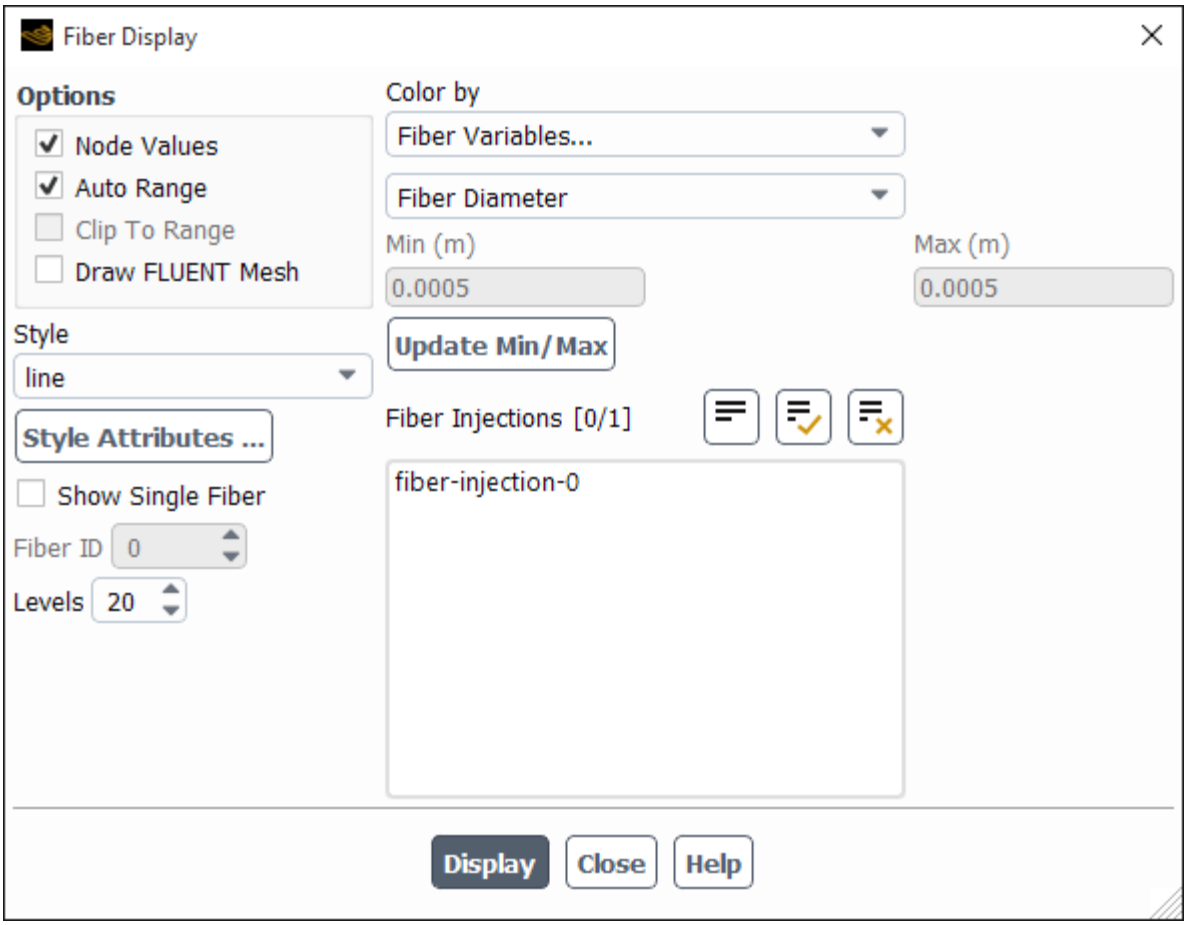
34.9.3.2. Fiber Display

To visualize injection fibers, you can use the **Fiber Display** dialog box, which can be accessed from the **Results**/**Graphics**/**Fibers** tree item.



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 34.17: The Fiber Display Dialog Box



The **Fiber Display** dialog box offers the same display options as those in the **Fiber Mesh Display** dialog box (Figure 34.15: The Fiber Mesh Display Dialog Box (p. 3554)). Similarly, for the injections selected in the **Fiber Injections** selection list, you can display either all injection fibers or single fibers with a specified **Fiber ID** (when the **Show Single Fiber** option is selected).

You can color the fibers by the fiber variables listed in XY Plots (p. 3556) or by the following additional fiber variables:

- **Fiber ID**
- **Fiber Density**
- **Fiber Tension**

You can control the smooth gradation of fiber colors by specifying the number of fiber **Levels**.

34.9.4. Running the Fiber Module in Parallel

The Fiber Module can be used in serial as well as in parallel calculations. Simulations in parallel require no additional input. The fiber iterations are performed entirely on the host process, whereas the flow field is parallelized. As a result, the overall performance may be limited by the host process if the fiber calculations are more time-consuming than the parallel flow-field iterations. Therefore, when simulating a large number of fibers, it is recommended to spawn the host process on a suitable ma-

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

chine. Refer to Parallel Processing in the *Fluent User's Guide* (p. 4293) for further information on how to set up parallel calculations.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 35: Creating Reduced Order Models (ROMs)

You can use a stand-alone Fluent session to define a reduced order model (ROM) for production in the Ansys Workbench environment (through Ansys DesignXplorer).

You can create a ROM from a series of 2D or 3D simulations. The ROM is a mathematical approximation of your full model, where you can change some inputs and get the solution immediately. These inputs are defined as input parameters in Ansys Fluent and can be material properties, inlet velocities, pressures and so on.

The process of creating and analyzing ROMs can be broken into two parts:

- ROM Production—You can setup a ROM case in stand alone Fluent and read this case into Fluent in Workbench, as described in Defining a ROM (p. 3561) and ROM Production in the *DesignXplorer User's Guide*.
- ROM Consumption—The produced ROM can be consumed by exporting the ROM as either an FMU 2.0 file or a ROMZ filein, see ROM Consumption in the *DesignXplorer User's Guide* for more detials.

To learn more about ROMs, refer to Using ROMs in the *DesignXplorer User's Guide*.

35.1. Defining a ROM

Important:

For release 2024 R2, ROMs are only applicable for steady-state, Newtonian fluids without shock waves (Ma<1) or radiation.

Building a ROM in Fluent:

1. Enable the ROM addon module by entering `define/models/addon-module/11` in the Fluent console.
2. Initialize the case or read a data file.

   Solution → **Initialization** → **Initialize**

   File → **Read** → **Data...**
3. Create input parameters for the quantities of interest for the ROM.

   You can choose any input parameters that you are interested in. Later, in the 3D ROM system, you can deselect input parameters if you decide that there are parameters that you no longer want to vary. Note that if you pick too many input parameters you may require additional snapshots to maintain accuracy

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

with the ROM. Refer to Creating a New Parameter (p. 1311) for additional information on creating input parameters.

Note:

General guidance on the number of input parameters defined:

- Low: 1 to 2
- Medium: 3 to 6
- High: more than 7
- Very high: more than 15

4. Create at least one output parameter.

   The output parameter has no impact on the ROM, but you can use it to monitor the results while the design points are being updated in Ansys DesignXplorer. Ideally, the output parameter should include a variable that is selected for creating the ROM. Refer to Creating Output Parameters (p. 4133) for additional information on output parameters.

   If you want to have an output parameter for a derived quantity, follow these steps:

   a. Create a custom field function using the desired variables (these should be ones that are selected when creating the ROM).

      User Defined → Field Functions → Custom...

   b. Create a report definition (surface or volume) that includes the custom field function defined in the previous step.

      Solution → Reports → Definition → New → Surface Report | Volume Report

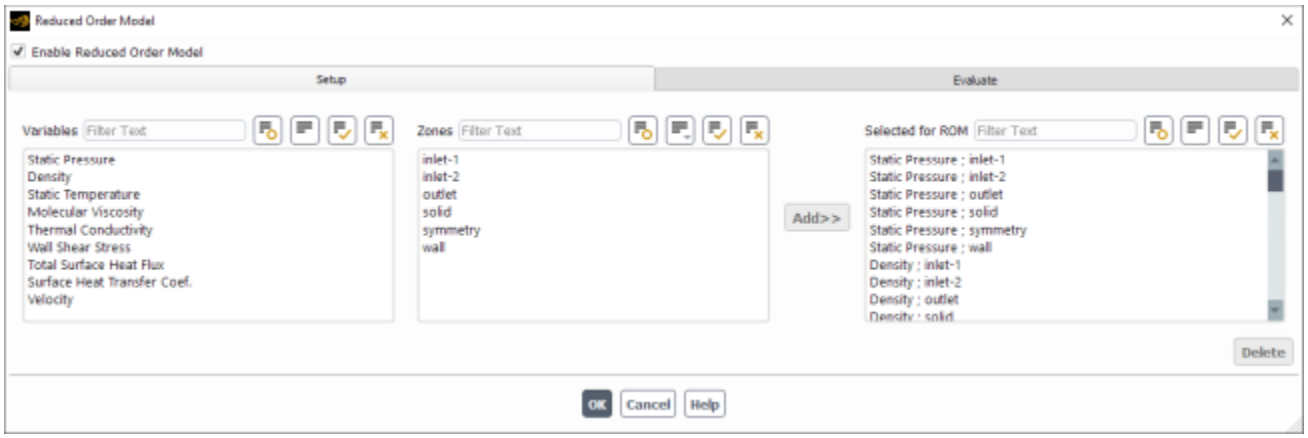
   c. Enable **Create Output Parameter** in the report definition dialog box and click **OK**.

5. Enable the **Reduced Order Model**.

   Setup → Models → Reduced Order Model → Edit...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 35.1: The Reduced Order Model Dialog Box



6. Select your desired **Variables** and **Zones** and click **Add>>**. *Any Custom Field Functions (p. 4285) that you have defined are available for selection.*

   *You can remove items from the **Selected for ROM** list by selecting them and clicking **Delete**.*

   Important:

   - You are free to select all of the variables and zones that you are interested in, however, be aware that if you select too many variables/zones, you will end up with bigger files. This is especially true if you have a larger model. For particularly large models, you may want to select smaller zones that adequately capture the physics of interest.
   - You must select the variables that correspond or are associated with the input and output parameters that you have defined. For example, if you have **Mass Flow Rate** selected as an output parameter, then you must have **Velocity** selected in the **Variable** list (because mass flow rate = density * area * velocity).

7. Review the selections you have made in the **Selected for ROM** list and click **OK** to confirm.

35.2. Reduced Order Model (ROM) Evaluation in Fluent

You may want to quickly visualize the results directly within Fluent, which you can do when you are running Fluent in Workbench.

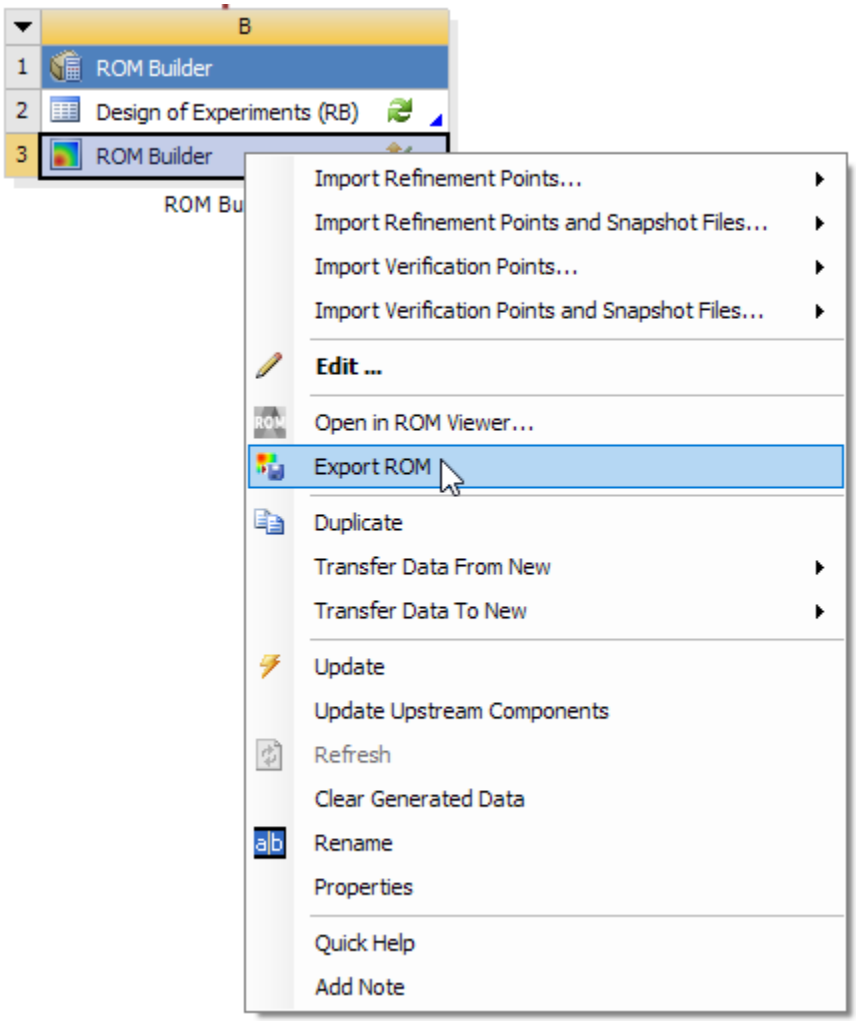
From within Fluent you can view:

- Contours
- Vectors
- Pathlines
- XY Plots
- Surface and volume integral reports

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To evaluate a ROM in Fluent:

1. In Workbench, generate the `romz` file by right-clicking the **ROM Builder** cell and selecting **Export ROM**. *Save the `romz` file to your desired location.*

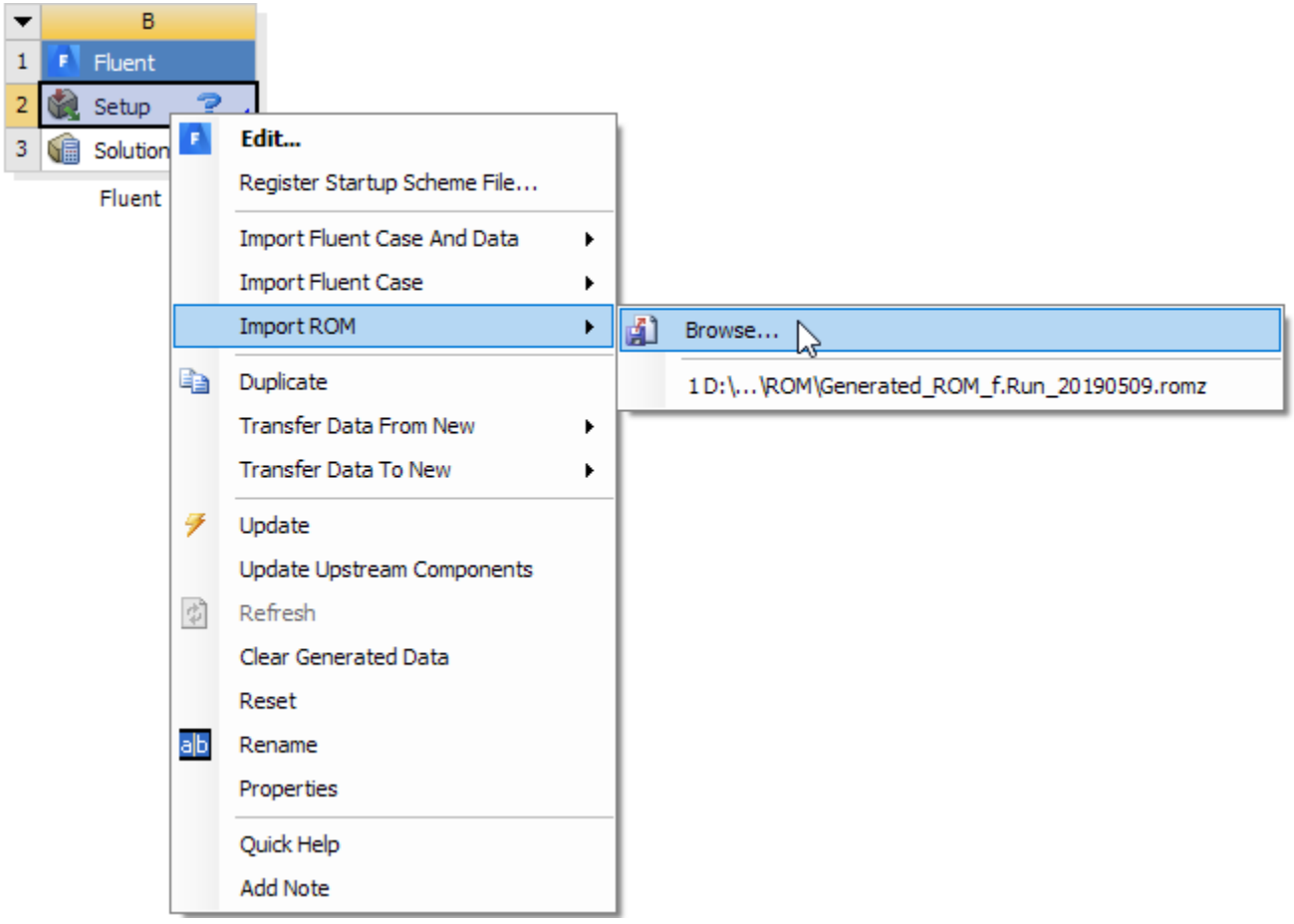


Note:

You cannot import a ROMZ file into the Fluent system used to produced the 3D ROM in the same instance of Workbench.

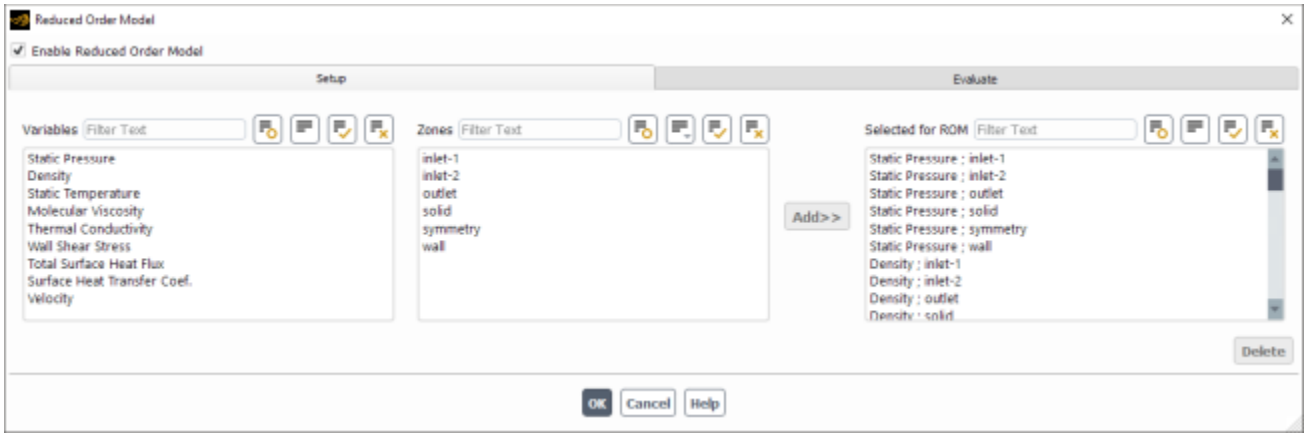
2. Right-click the Fluent **Setup** cell, and select **Import ROM** → **Browse**, to open the **Select File** dialog box and import the `romz` file.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



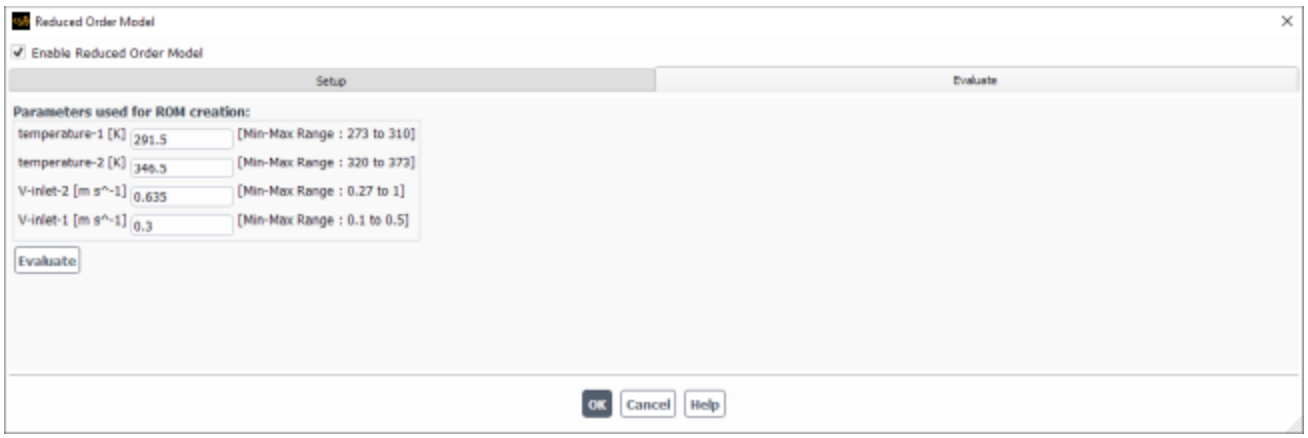
3. Generate the solution in Fluent by clicking **Update Project** in the toolbar.
4. Launch Fluent by double-clicking the **Solution** cell.
5. Open the **Reduced Order Model** dialog box.

→ Setup → Models → ROM → Edit...



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Click the **Evaluate** tab. *Here you can see all of the parameters used in creating the ROM. The range of acceptable inputs is listed to the right of each variable.*



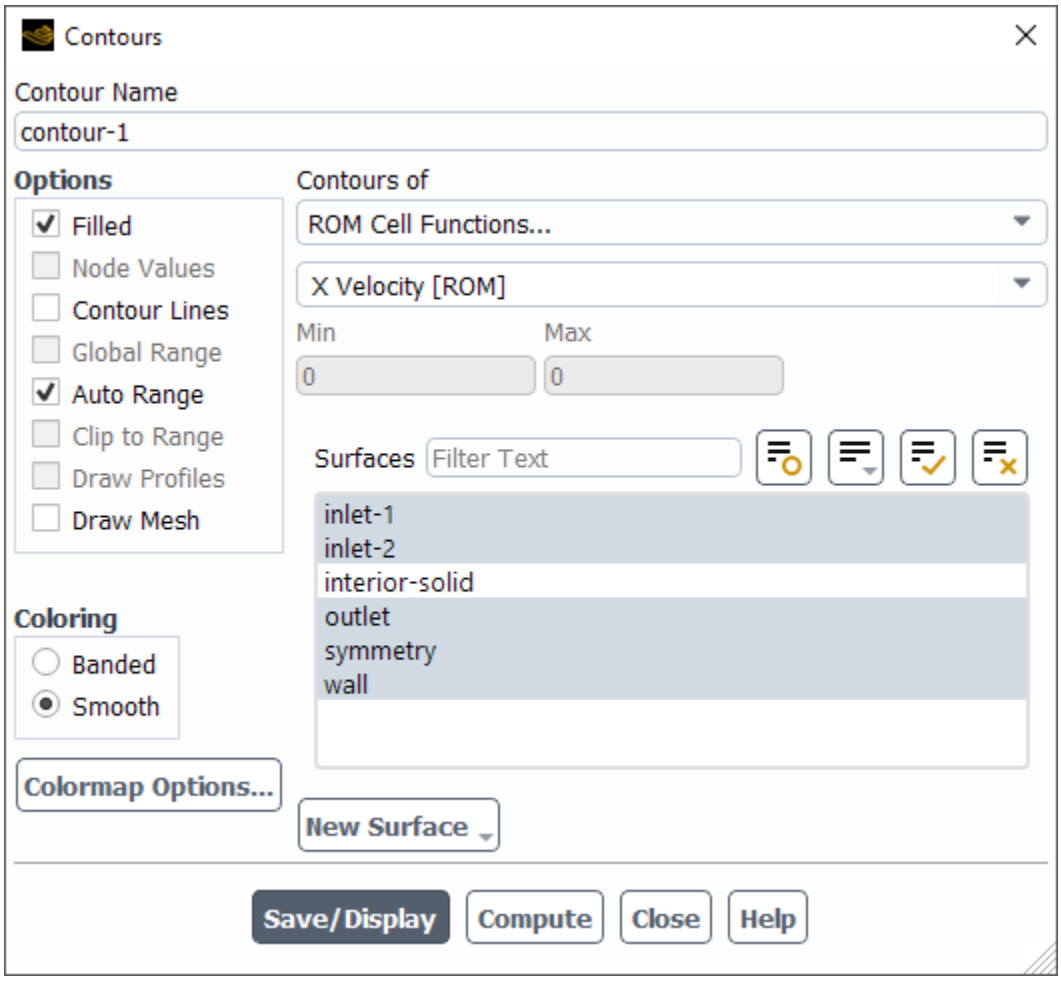
7. Enter a value for the variable you want evaluated (within the allowed range) and click **Evaluate**.

You can visually postprocess ROM variables using contour plots, vector plots, and so on.

1. Open the **Contours** dialog box.

   → **Results** → **Graphics** → **Contours** **New...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Ensure that **Node Values** are disabled. *ROMs are evaluated at cell centers.*
3. Select **ROM Cell Functions...** from the **Contours of** drop-down list.

> **Note:**
>
> The **ROM Cell Functions...** option is only available after you click **Evaluate** in the **Evaluate** tab of the **Reduced Order Model** dialog box.

4. Select the ROM variable that you want to visualize from the second **Contours of** drop-down list.
5. Select the surfaces where you want to visualize the specified ROM variable. *The list of available surfaces is filtered to show only the surfaces where the ROM is evaluated, as defined during ROM setup.*
6. Click **Save/Display**.

Limitations

The following limitations apply to postprocessing reduced order models in Fluent:

- You cannot continue with the ROM solution after evaluating it.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Exporting solution data/variables will not include ROM snapshot data.
- Flux and Force Reports and report definitions are not available for ROM evaluated variables using ROM data.
- Output parameters are not updated for Design of Experiments when the ROM is being evaluated in Fluent.

35.3. Exporting Reduced Order Model (ROM) Results from Fluent

After solving your reduced order model (ROM) and loading it into Fluent (see Reduced Order Model (ROM) Evaluation in Fluent (p. 3563) for details on loading your results into Fluent), you can export the ROM scalar values as Profiles (p. 1564) using the Write Profile Dialog Box (p. 4988).

To export ROM scalar outputs:

1. Open the **Write Profiles** dialog box.

   Fluent in Workbench:

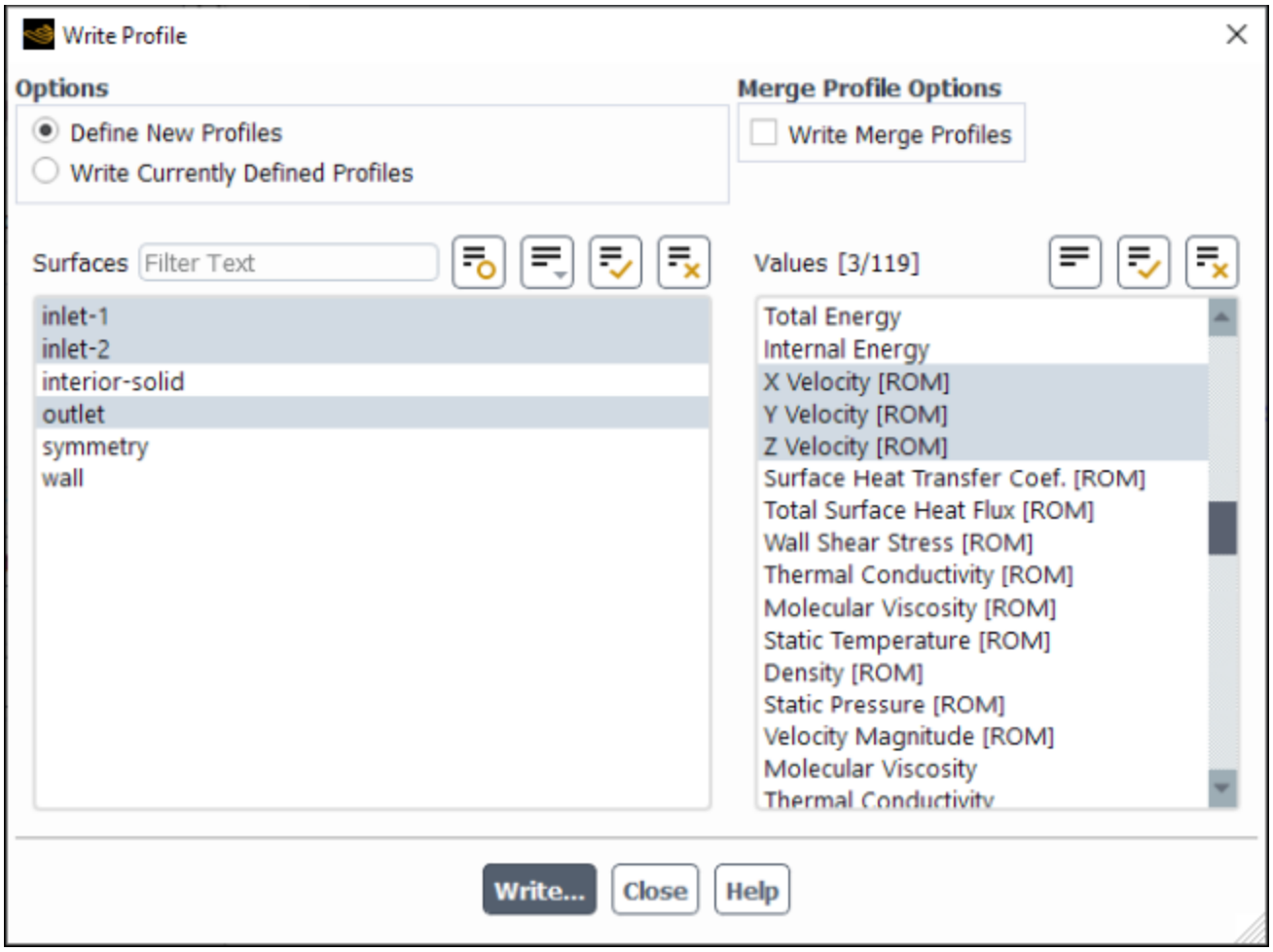
   File → **Export** → **Profile...**

   Stand-alone Fluent:

   File → **Write** → **Profile...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 35.2: The Write Profile Dialog Box



2. Select the ROM cell functions and the surfaces where you want them applied in the **Values** and **Surfaces** lists.

35.4. ROM Limitations

The following list of limitations should be viewed in conjunction with the list of limitations provided in the Ansys DesignXplorer documentation (ROM Limitations and Known Issues in the *DesignXplorer User's Guide*):

- You must have at least one input and one output parameter defined before you can enable the reduced order model in Fluent.
- Interface zones are not available for use with ROMs.
- The 3D ROM does not support geometry parameter updates.
- ROMs can only be created for steady-state cases.
- Combustion and radiation models are not supported for ROMs.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Parametric optimization using the ROM through Fluent is not supported.
- Moving and deforming meshes are not supported for ROMs.
- You must select the variables that correspond with the input and output parameters that you have defined.
- Report Definitions: Expression and User-Defined report definitions are not available for use with ROM variables. You can, however, create a named expression that includes a report definition for a ROM variable.
- Report Definitions: It is recommended that you do not use Mass-Weighted Average, Vertex Average, Vertex Minimum, or Vertex Maximum report definitions, as the ROM only has data available on cell face centers and cell centers. Instead, use Area-Weighted Average, Facet Average, Facet Minimum, or Facet Maximum report definitions.
- Output Parameters are not supported for the following report definition types:
  - User Defined
  - Expression
  - Flux Report
  - Surface Report—Flow Rate

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 36: Using the Solver

This chapter describes how to use the Ansys Fluent solver. For more information about the theory behind the Ansys Fluent solver, see Solver Theory in the Theory Guide. Choosing the Solver (p. 3573) provides an overview, and the remaining sections provide detailed instructions.

36.1. Overview of Using the Solver

36.2. Choosing the Spatial Discretization Scheme

36.3. Pressure-Based Solver Settings

36.4. Density-Based Solver Settings

36.5. Setting Algebraic Multigrid Parameters

36.6. Setting Solution Limits

36.7. Setting Multi-Stage Time-Stepping Parameters

36.8. Selecting Gradient Limiters

36.9. Initializing the Solution

36.10. Full Multigrid (FMG) Initialization

36.11. Hybrid Initialization

36.12. Performing Steady-State Calculations

36.13. Performing Time-Dependent Calculations

36.14. Performing Calculations with a Pseudo Time Method

36.15. Monitoring Solution Convergence

36.16. Convergence Conditions

36.17. Executing Commands During the Calculation

36.18. Automatic Initialization of the Solution and Case Modification

36.19. Animating the Solution

36.20. Checking Your Case Setup

36.21. Convergence and Stability

36.22. Solution Steering

36.1. Overview of Using the Solver

In Ansys Fluent, two solver technologies are available:

- pressure-based
- density-based

Both solvers can be used for a broad range of flows, but in some cases one formulation may perform better (that is, yield a solution more quickly or resolve certain flow features better) than the other. The

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

pressure-based and density-based approaches differ in the way that the continuity, momentum, and (where appropriate) energy and species equations are solved, as described in Overview of Flow Solvers in the Theory Guide.

The pressure-based solver traditionally has been used for incompressible and mildly compressible flows. The density-based approach, on the other hand, was originally designed for high-speed compressible flows. Both approaches are now applicable to a broad range of flows (from incompressible to highly compressible), but the origins of the density-based formulation may give it an accuracy (that is shock resolution) advantage over the pressure-based solver for high-speed compressible flows.

Two formulations exist under the density-based solver: implicit and explicit. The density-based explicit and implicit formulations solve the equations for additional scalars (for example, turbulence or radiation quantities) sequentially. The implicit and explicit density-based formulations differ in the way that they linearize the coupled equations. For more details about the solver formulations, see Overview of Flow Solvers in the Theory Guide.

Due to broader stability characteristics of the implicit formulation, a converged steady-state solution can be obtained much faster using the implicit formulation rather than the explicit formulation. However, the implicit formulation requires more memory than the explicit formulation.

Two algorithms also exist under the pressure-based solver in Ansys Fluent: a segregated algorithm and a coupled algorithm. In the segregated algorithm the governing equations are solved sequentially, segregated from one another, while in the coupled algorithm the momentum equations and the pressure-based continuity equation are solved in a coupled manner. In general, the coupled algorithm significantly improves the convergence speed over the segregated algorithm, however, the memory requirement for the coupled algorithm is more than the segregated algorithm.

When selecting a solver and an algorithm you must consider the following issues:

- The model availability for a given solver.
- Solver performance for the given flow conditions.
- The size of the mesh under consideration and the available memory on your machine. This issue could be an important factor in deciding whether to use an explicit or implicit formulation when the density-based solver is selected, or to use a segregated or coupled algorithm when the pressure-based solver is selected.

The following two lists highlight the model availability for each solver:

Important:

Note that the pressure-based solver provides several physical models or features that are not available with the density-based solver:

- Cavitation model
- Volume-of-fluid (VOF) model
- Multiphase mixture model
- Eulerian multiphase model

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Non-premixed combustion model
- Premixed combustion model
- Partially premixed combustion model
- Composition PDF transport model
- Soot model
- Rosseland radiation model
- Melting/solidification model
- Shell conduction model
- Floating operating pressure
- Fixed variable option
- Physical velocity formulation for porous media
- Specified mass flow rate for streamwise periodic flow

The following features are available with the density-based solver, but not with the pressure-based solver:

- Wet steam multiphase model

For additional information, see the following sections:

36.1.1. Choosing the Solver

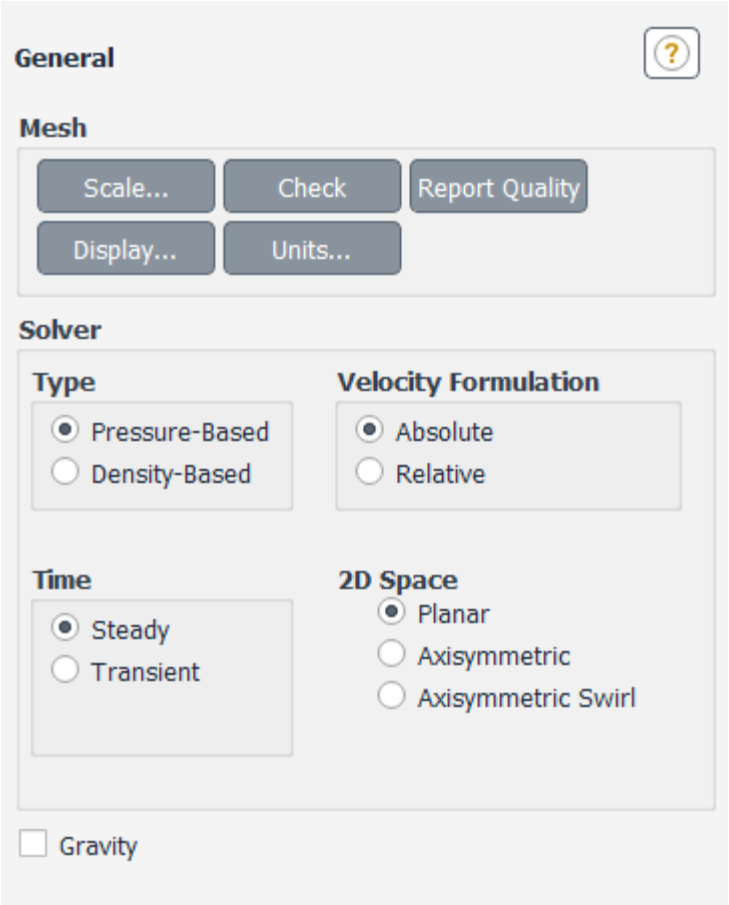
36.1.1. Choosing the Solver

To choose one of the solvers, you will use the General Task Page (p. 4684) (Figure 36.1: The General Task Page (p. 3574)).

Setup → **General**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 36.1: The General Task Page



To use the pressure-based solver, retain the default selection of **Pressure-Based** under **Solver**.

To use the density-based solver, select **Density-Based** under **Solver**.

After you have defined your model and specified which solver you want to use, you are ready to run the solver. The following steps outline a general procedure you can follow:

1. (pressure-based solver only) Select the pressure-velocity coupling method (see Choosing the Pressure-Velocity Coupling Method (p. 3585)).

2. Choose the spatial discretization scheme and, for the pressure-based solver, the pressure interpolation scheme (see Choosing the Spatial Discretization Scheme (p. 3575)).

3. (pressure-based solver only) Select the porous media velocity method (see Porous Media Conditions (p. 1333)).

4. Select how you want the derivatives to be evaluated by choosing a gradient option (see Evaluation of Gradients and Derivatives in the Theory Guide).

5. Set the under-relaxation factors (see Setting Under-Relaxation Factors (p. 3589)).

6. (density-based explicit formulation only) Set up the FAS multigrid (see Turning On FAS Multigrid (p. 3609)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. Make any additional modifications to the solver settings that are suggested in the chapters or sections that describe the models you are using.

8. Enable the appropriate solution monitors (see Monitoring Solution Convergence (p. 3684)).

9. Initialize the solution (see Initializing the Solution (p. 3627)).

10. Start calculating (see Performing Steady-State Calculations (p. 3639) for steady-state calculations, or Performing Time-Dependent Calculations (p. 3643) for time-dependent calculations).

11. If you have convergence trouble, try one of the methods discussed in Convergence and Stability (p. 3738).

The default settings for the first three items listed above are suitable for most problems and need not be changed. The following sections outline how these and other solution parameters can be changed, and when you may want to change them.

36.2. Choosing the Spatial Discretization Scheme

Gradients are needed not only for constructing values of a scalar at the cell faces, but also for computing secondary diffusion terms and velocity derivatives. For more information about the different gradients, see Evaluation of Gradients and Derivatives in the Theory Guide.

The three gradients that are available in Ansys Fluent are

- **Green-Gauss Cell Based**
- **Green-Gauss Node Based**
- **Least Squares Cell Based**

The gradient options are selectable from the **Gradient** drop-down list, in the **Solution Methods** task page.

Solution → **Methods**

In addition, Ansys Fluent allows you to choose the discretization scheme for the convection terms of each governing equation. (Second-order accuracy is automatically used for the viscous terms.) By default, single-phase problems using either the pressure-based or density-based solver are solved using second-order upwind discretization for the convection terms of the flow equations and all scalar equations except those for turbulence quantities, which are solved using first-order upwind discretization. For multiphase flows, the flow equations use first-order upwind discretization by default. For a complete description of the discretization schemes available in Ansys Fluent, see Discretization in the Theory Guide.

In addition, when you use the pressure-based solver, you can specify the pressure interpolation scheme. For a description of the pressure interpolation schemes available in Ansys Fluent, see Pressure Interpolation Schemes in the Theory Guide.

For additional information, see the following sections:

36.2.1. First-Order Accuracy vs. Second-Order Accuracy

36.2.2. Other Discretization Schemes

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.2.3. Choosing the Pressure Interpolation Scheme
36.2.4. Choosing the Density Interpolation Scheme
36.2.5. High Order Term Relaxation (HOTR)
36.2.6. User Inputs

36.2.1. First-Order Accuracy vs. Second-Order Accuracy

When the flow is aligned with the mesh (for example, laminar flow in a rectangular duct modeled with a quadrilateral or hexahedral mesh) the first-order upwind discretization may be acceptable. When the flow is not aligned with the mesh (that is, when it crosses the mesh lines obliquely), however, first-order convective discretization increases the numerical discretization error (numerical diffusion). For triangular and tetrahedral meshes, since the flow is never aligned with the mesh, you will generally obtain more accurate results by using the second-order discretization. For quad/hex meshes, you will also obtain better results using the second-order discretization, especially for complex flows.

In summary, while the first-order discretization generally yields better convergence than the second-order scheme, it generally will yield less accurate results, especially on tri/tet meshes. See Convergence and Stability (p. 3738) for information about controlling convergence.

For most cases, you will be able to use the second-order scheme from the start of the calculation. In some cases, however, you may need to start with the first-order scheme and then switch to the second-order scheme after a few iterations. For example, if you are running a high-Mach-number flow calculation that has an initial solution much different than the expected final solution, you will usually need to perform a few iterations with the first-order scheme and then turn on the second-order scheme and continue the calculation to convergence. Alternatively, full multigrid initialization is also available for some flow cases that allow you to proceed with the second-order scheme from the start.

For a simple flow that is aligned with the mesh (for example, laminar flow in a rectangular duct modeled with a quadrilateral or hexahedral mesh), the numerical diffusion will be naturally low, so you can generally use the first-order scheme instead of the second-order scheme without any significant loss of accuracy.

Finally, if you run into convergence difficulties with the second-order scheme, you should try the first-order scheme instead.

36.2.1.1. First- to Higher-Order Blending

While the higher-order scheme may result in greater accuracy, it can also result in convergence difficulties and instabilities at certain flow conditions. On the other hand, using a first-order scheme may not provide the desired accuracy. One approach to achieving improved accuracy while maintaining good stability is to use a discretization blending factor. This feature is available for both density-based and pressure-based solvers and can be invoked using the following text command:

`solve → set → numerics`

Enter a value between `0` and `1` when asked for the blending factor: `1st-order to higher-order blending factor [min=0.0 - max=1.0]`

A blending factor of `0` reduces the gradient reconstruction to a first-order discretization scheme, whereas `1` will recover high-order discretization. A blending factor of less than `1` (typically 0.75 or

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

0.5) will make the convective fluxes more diffusive, which in some flow conditions can stabilize a solution that is otherwise unstable when the full higher-order discretization scheme is employed.

Important:

Note that in order to use this feature effectively, make sure that one of the allowed higher-order discretization schemes is selected for the desired variables in the **Solution Methods** task page.

36.2.2. Other Discretization Schemes

The QUICK and third-order MUSCL discretization schemes may provide better accuracy than the second-order scheme for rotating or swirling flows. The QUICK scheme is applicable to quadrilateral or hexahedral meshes, while the MUSCL scheme is used on all types of meshes. In general, however, the second-order scheme is sufficient and the QUICK scheme will not provide significant improvements in accuracy.

Important:

If QUICK is used for hybrid meshes, it will be used only for quadrilateral and hexahedral cells. Second-order upwind discretization will be applied to all other cells.

The bounded central differencing and (for the pressure-based solver) central differencing schemes are available when you are using the LES, DES, SAS, SBES, and SDES turbulence models, and the central differencing scheme should be used only when the mesh spacing is fine enough so that the magnitude of the local Peclet number is less than 1. The Peclet number is defined as [114] (p. 5719), [115] (p. 5719):

$$Pe=\frac{\rho u L}{\Gamma} \tag{36.1}$$

where ρ is the fluid density, u is local cell velocity, L is a length scale (for example, mesh size), and Γ is the diffusion coefficient of the fluid.

If you select the bounded central differencing scheme, you have the option to specify the **BCD Scheme Boundedness** as a constant or expression. For more details, see Bounded Central Differencing Scheme in the *Fluent Theory Guide*.

A modified HRIC scheme (see Modified HRIC Scheme in the Theory Guide) is also available for VOF simulations using either the implicit or explicit formulation.

36.2.3. Choosing the Pressure Interpolation Scheme

As discussed in Pressure Interpolation Schemes in the Theory Guide, a number of pressure interpolation schemes are available when the pressure-based solver is used in Ansys Fluent. For most single-phase cases, the second-order scheme is acceptable, but some types of models may benefit from one of the other schemes:

- For flows with high swirl numbers, high-Rayleigh-number natural convection, high-speed rotating flows, and flows in strongly curved domains, the **PRESTO!** scheme is recommended.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For problems involving large body forces, the **Body Force Weighted** scheme is recommended.
- For Eulerian multiphase models, the default scheme is **PRESTO!**. However, for multi-fluid VOF problems that use the non-iterative solver, the **Second Order** interpolation scheme can be more robust for cases with skewed meshes, and the **Body Force Weighted** interpolation scheme can be more robust for cases with large body forces and skewed meshes.
- For VOF and Mixture multiphase models, either the **PRESTO!** (default) or **Modified Body Force Weighted** scheme should be used:
  - The **PRESTO!** scheme is more comprehensive than the other schemes.
  - The **Modified Body Force Weighted** scheme is a good alternative to both **PRESTO!** and **Body Force Weighted** for most problems. It exhibits less sensitivity to mesh and time-step sizes compared to the **PRESTO!** scheme.

    The **Modified Body Force Weighted** scheme is recommended for:

    → the non-iterative solver (as it offers better robustness)

    → cases where large body forces are present

    The scheme can be used for highly viscous and rotating flows where the **Body Force Weighted** scheme has severe limitations.

36.2.4. Choosing the Density Interpolation Scheme

As discussed in Density Interpolation Schemes in the Theory Guide, four density interpolation schemes are available when the pressure-based solver is used to solve a single-phase compressible flow.

The second-order upwind scheme (the default) provides reasonable stability for the discretization of the pressure-correction equation, and gives good results for most classes of flows. The first-order upwind scheme will provide greater stability, but may tend to smooth shocks in compressible flows. If you are calculating a compressible flow with shocks you should use the second-order-upwind or QUICK scheme. Using the QUICK scheme for all variables, including density, is highly recommended for compressible flows with shocks when using quadrilateral, hexahedral, or hybrid meshes. The third-order MUSCL scheme is applicable to arbitrary meshes and has the potential to improve spatial accuracy for all types of meshes by reducing numerical diffusion.

Important:

In the case of multiphase flows, the selected density scheme is applied to the compressible phase and arithmetic averaging is used for incompressible phases.

36.2.5. High Order Term Relaxation (HOTR)

The purpose of the relaxation of high order terms is to improve the startup and the general solution behavior of flow simulations when higher order spatial discretizations are used (higher than first). It has also shown to prevent convergence stalling in some cases. Such high-order terms can be of significant importance in certain cases and lead to numerical instabilities. This is particularly true at aggressive solution settings. In such cases, high order relaxation is a useful strategy to minimize your

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

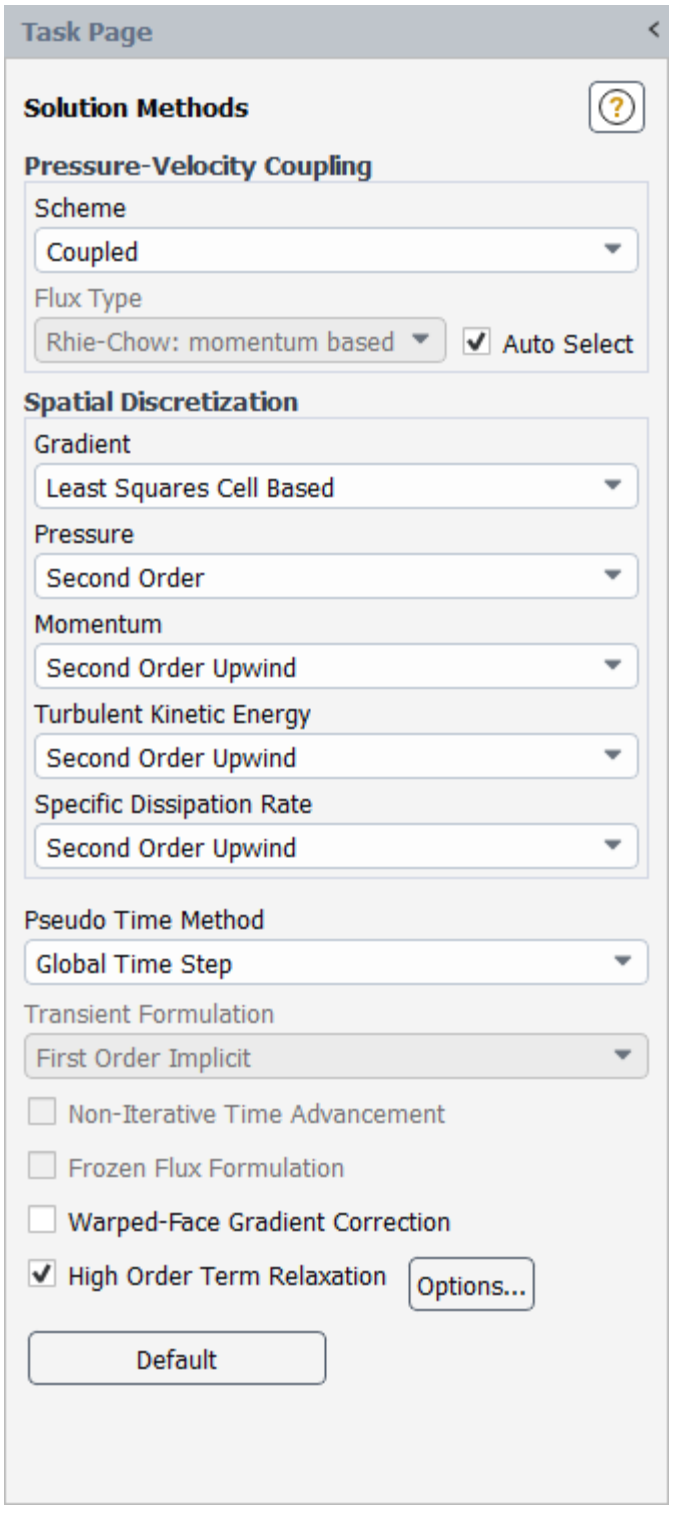
interaction during the solution. This can be an effective alternative to starting the solution first order, then switching to second order spatial discretization at a later stage.

The **High Order Term Relaxation** option can be enabled from the **Solution Methods** task page, as shown in Figure 36.2: The Solution Methods Task Page for the HOTR Option (p. 3580).

Solution → **Methods**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

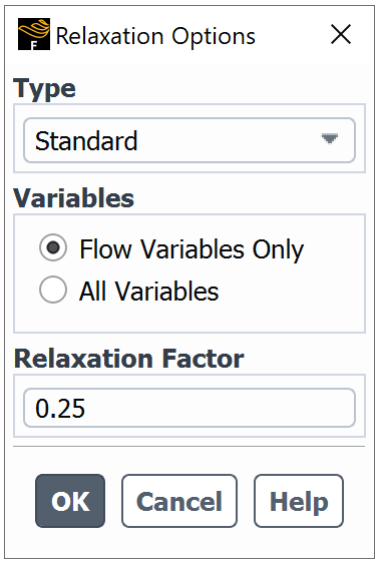
Figure 36.2: The Solution Methods Task Page for the HOTR Option



Further control of **High Order Term Relaxation** can be obtained after clicking **Options...** and making the necessary selections and settings in the **Relaxation Options** dialog box (Figure 36.3: The Relaxation Options Dialog Box with the Pressure-Based Solver (p. 3581)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 36.3: The Relaxation Options Dialog Box with the Pressure-Based Solver



When using the pressure-based solver, you can select the **Type** of HOTR that you would like to apply:

- **Standard**

  HOTR is applied to both convection and diffusion terms. This is the default selection.

- **Convection Only**

  HOTR is applied to exclusively to the convection terms. This has the potential to deepen and accelerate convergence for some cases, such as steady-state turbomachinery simulations that use the default SST k-ω turbulence model.

For both the pressure-based and the density-based solvers, you have the option of selecting **All Variables** to be under-relaxed instead of only the default flow variables (**Flow Variables Only**).

- If you select **Flow Variables Only**, then the following variables will be under-relaxed:
  - Velocity components
  - Pressure
  - Energy
  - Density
  - Turbulence quantities (excluding Reynolds stresses)
  - Volume fraction
- If you select **All Variables**, then relaxation is applied to each variable discretized with a higher order scheme.

The default values for the **Relaxation Factor** is 0.25 for steady-state cases and 0.75 for transient cases. The same factor is applied to all equations solved.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For theoretical information about high order term relaxation, see High Order Term Relaxation in the Theory Guide.

36.2.5.1. Limitations

The following limitations exist when using the **High Order Term Relaxation** option:

- The **High Order Term Relaxation** option is not available when the **Non-Iterative Time Advancement** option is enabled, since the simulation would not achieve the required level of high order spatial accuracy.
- In general, high order term relaxation is available for transient flows. Nevertheless, it should be used with care. To achieve high order accuracy at convergence for each time step, you must increase the number of iterations per time step to ensure that the original convergence criteria have been met.
- When the QUICK scheme is selected for specific transport equations, no under-relaxation is applied to this equation.

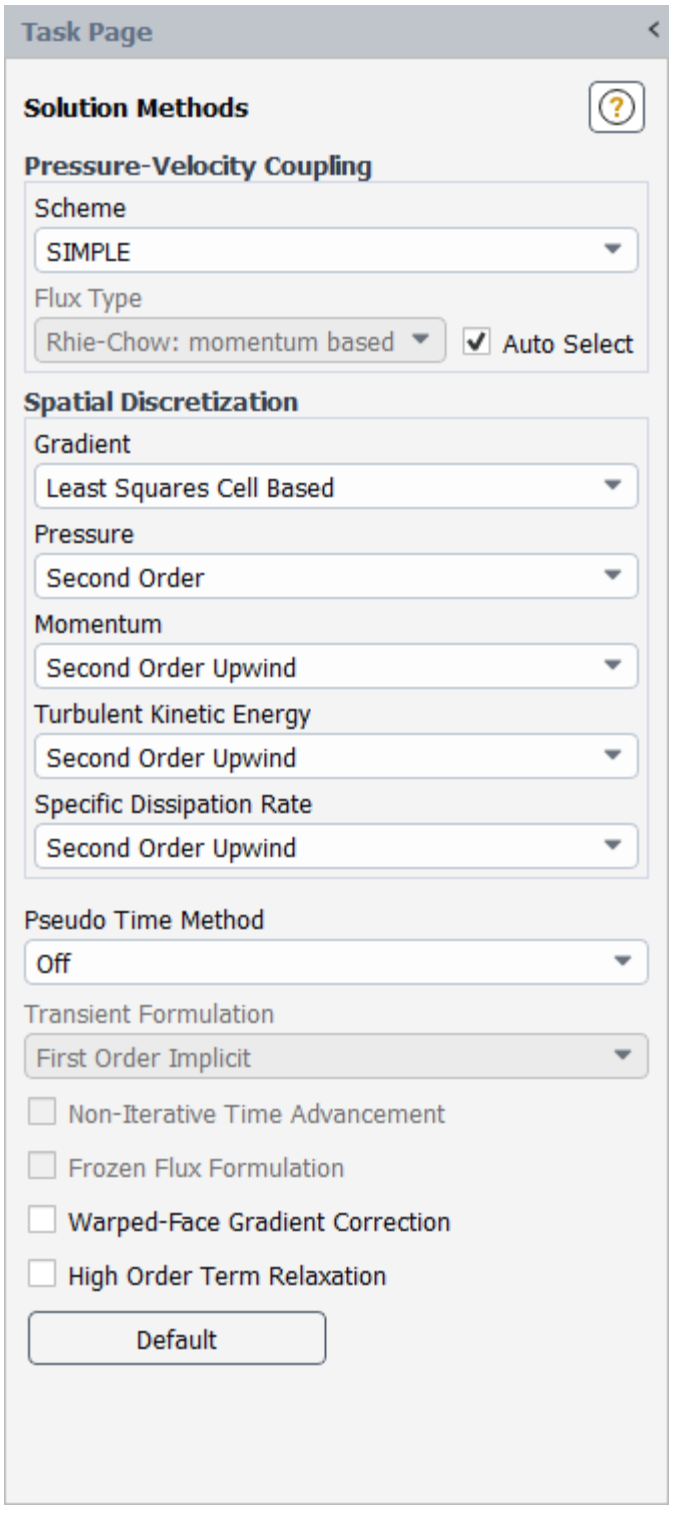
36.2.6. User Inputs

You can specify the discretization scheme and, for the pressure-based solver, the pressure interpolation scheme in the Solution Methods Task Page (p. 5157) (Figure 36.4: The Solution Methods Task Page for a Pressure-Based Segregated Algorithm (p. 3583)).

Solution → **Methods**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 36.4: The Solution Methods Task Page for a Pressure-Based Segregated Algorithm



For each scalar equation listed under **Spatial Discretization** (**Momentum**, **Energy**, **Turbulent Kinetic Energy**, and so on, for the pressure-based solver or **Turbulent Kinetic Energy**, **Turbulent Dissipation Rate**, and so on, for the density-based solver) you can choose **First Order Upwind**, **Second Order Upwind**, **QUICK**, **Third-Order MUSCL**, or (for **Momentum** when you are using the LES, DES, SAS, SDES, or SBES turbulence model) **Bounded Central Differencing** (the default) or **Central Differencing** in the adjacent drop-down list. For the density-based solver, you can choose either **First Order Upwind**,

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Second Order Upwind, **Third-Order MUSCL**, or (when you are using the LES, DES, SAS, SDES, or SBES turbulence model) **Bounded Central Differencing** for the **Flow** equations (which include momentum and energy). Note that the task page shown in Figure 36.4: The Solution Methods Task Page for a Pressure-Based Segregated Algorithm (p. 3583) is for the pressure-based solver.

If you are using the pressure-based solver, select the pressure interpolation scheme under **Spatial Discretization**, in the drop-down list next to **Pressure**. You can choose **Standard**, **PRESTO!**, **Linear**, **Second Order**, **Body Force Weighted**, or **Modified Body Force Weighted**.

Important:

The low-order formulation of PRESTO! can be applied by limiting the high-order terms for the PRESTO! scheme. This is done using the following text command:

`solve → set → numerics`

When asked `limit high-order terms for PRESTO! pressure scheme?`, enter `yes`.

This modification can be used to stabilize the solution process when the pressure-based coupled algorithm is used and when the original PRESTO! scheme either fails to converge or produces unphysical oscillations (wiggles) in the pressure field.

If you are using the pressure-based solver and your flow is compressible (that is, you are using the ideal gas law for density), select the density interpolation scheme under **Spatial Discretization**, in the drop-down list next to **Density**. You can choose **First Order Upwind**, **Second Order Upwind**, **QUICK** or **Third-Order MUSCL**. (Note that **Density** will not appear for incompressible flows.)

If you enable the VOF model while using the pressure-based solver, the volume fraction interpolation schemes that are available are **Geo-Reconstruct**, **CICSAM**, **Modified HRIC**, **Compressive**, and **QUICK**.

If your case involves species transport, you can set the scheme for the individual species as **First Order Upwind**, **Second Order Upwind**, **QUICK**, or **Third-Order MUSCL**. However, if you want all your species to use the same discretization scheme, then rather than setting each one individually, simply enable the **Set All Species Discretizations Together** option. Notice that you will no longer see your list of individual species, instead a **Species** field will appear with the scheme of your choice.

If you change the settings for the **Spatial Discretization**, but you then want to return to Ansys Fluent's default settings, you can click the **Default** button.

Important:

If your mesh topology has step-wise prism mesh near the walls, do not use node-based gradients with MUSCL.

36.3. Pressure-Based Solver Settings

For additional information, see the following sections:

36.3.1. Choosing the Pressure-Velocity Coupling Method

36.3.2. Mass Flux Types

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.3.3. Setting Under-Relaxation Factors
36.3.4. Setting Solution Controls for the Non-Iterative Solver
36.3.5. Equation Order
36.3.6. Using the Correction Form of Momentum Discretization

36.3.1. Choosing the Pressure-Velocity Coupling Method

The pressure-velocity coupling scheme controls the manner in which pressure and velocity are updated when the pressure-based solver is used. The scheme can be either segregated (pressure and velocity are updated sequentially) or coupled (pressure and velocity are updated simultaneously).

Ansys Fluent provides the following segregated types of algorithms:

- SIMPLE
- SIMPLEC
- PISO
- Fractional Step (FSM) (time-dependent flows with **Non-Iterative Time Advancement** (NITA) only)

In general, segregated methods are faster per iteration, while the coupled algorithm usually requires fewer iterations to converge. For this reason, the coupled solver is usually recommended for steady-state simulations. For transient simulations, the coupled solver has the best robustness properties, especially for large time step sizes, but SIMPLEC, PISO, or NITA may give faster overall solution times for small time step sizes.

Note:

Steady-state and transient simulations differ in their default selection for the pressure-velocity coupling scheme. If you select a scheme, it will not necessarily be retained if you then switch between a steady and transient calculation.

The following mechanisms are available to under-relax the equations:

- Pseudo time method (not available for the transient coupled solver)
- Courant number (coupled solver only)

The coupled solver with a pseudo time method selected is the default for cases with the following settings:

- Steady-state
- Single-phase
- No battery or fuel cell model
- No solidification and melting model

Limitations and controls for the pseudo time method are described in Performing Calculations with a Pseudo Time Method (p. 3673). The most important (and typically the only) control you need to adjust

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

in practice is the **Pseudo Time Step Size** for the coupled solver or the **Pseudo Time Courant Number** for the segregated solver, as described in Pseudo Time Settings for the Calculation (p. 3680).

Important:

Pressure-velocity coupling is relevant only for the pressure-based solver.

36.3.1.1. SIMPLE vs. SIMPLEC

In Ansys Fluent, both the standard SIMPLE algorithm and the SIMPLEC (SIMPLE-Consistent) algorithm are available. SIMPLE is the default for transient simulations, but many problems will benefit from using SIMPLEC, particularly because of the increased under-relaxation that can be applied, as described below.

For relatively uncomplicated problems (laminar flows with no additional models enabled) in which convergence is limited by the pressure-velocity coupling, you can often obtain a converged solution more quickly using SIMPLEC. With SIMPLEC, the pressure-correction under-relaxation factor is generally set to 1.0, which aids in convergence speed-up. In some problems, however, increasing the pressure-correction under-relaxation to 1.0 can lead to instability due to high mesh skewness. For such cases, you will need to use one or more skewness correction iterations, use a slightly more conservative under-relaxation value (up to 0.7), or use the SIMPLE algorithm. For complicated flows involving turbulence and/or additional physical models, SIMPLEC will improve convergence only if it is being limited by the pressure-velocity coupling. Often it will be one of the additional modeling parameters that limits convergence; in this case, SIMPLE and SIMPLEC will give similar convergence rates.

36.3.1.2. PISO

The PISO algorithm (see PISO in the Theory Guide) with neighbor correction is highly recommended for all transient flow calculations, especially when you want to use a large time step size. (For problems that use the LES turbulence model, which usually requires a small time step size, using PISO may result in an increased computational expense, so SIMPLE or SIMPLEC should be considered instead.) PISO can maintain a stable calculation with a larger time step size and an under-relaxation factor of 1.0 for both momentum and pressure. For steady-state problems, PISO with neighbor correction does not provide any noticeable advantage over SIMPLE or SIMPLEC with optimal under-relaxation factors.

PISO with skewness correction is recommended for both steady-state and transient calculations on meshes with a high degree of distortion.

When you use PISO neighbor correction, under-relaxation factors of 1.0 or near 1.0 are recommended for all equations. If you use just the PISO skewness correction for highly-distorted meshes (without neighbor correction), set the under-relaxation factors for momentum and pressure so that they sum to 1 (for example, 0.3 for pressure and 0.7 for momentum). If you use both PISO skewness correction and neighbor correction, follow the under-relaxation recommendations for PISO neighbor correction, above.

For most problems, it is not necessary to disable the default coupling between neighbor and skewness corrections. For highly distorted meshes, however, disabling the default coupling between neighbor and skewness corrections is recommended.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.3.1.3. Fractional Step Method

The Fractional Step method (FSM), described in Fractional-Step Method (FSM) in the Theory Guide, is available when you choose to use the NITA scheme (that is, the **Non-Iterative Time Advancement** option in the **Solution Methods** task page). With the NITA scheme, the FSM is slightly less computationally expensive compared to the PISO algorithm. Whether you select FSM or PISO depends on the application. For some problems (for example, simulations that use VOF), FSM could be less stable than PISO.

In most cases, the default values for the solution methods are enough to set a robust convergence of the internal pressure correction sub-iterations due to skewness. Only very complex problems (for example, moving deforming meshes, sliding interfaces, or the VOF model) could require a reduction of relaxation for pressure up to a value of 0.7 or 0.8.

36.3.1.4. Coupled

Selecting **Coupled** from the **Pressure-Velocity Coupling** drop-down list indicates that you are using the pressure-based coupled algorithm, described in Coupled Algorithm in the Theory Guide. This solver offers some advantages over the pressure-based segregated algorithm. The pressure-based coupled algorithm obtains a more robust and efficient single phase implementation for steady-state flows. It is not available for cases using the **Non-Iterative Time Advancement** option (NITA).

Note:

In some cases using porous jump boundary conditions, the **Coupled** scheme may suffer from convergence issues that do not respond to changes in the coupled solver settings. This behavior depends on the specific flow configuration and porous jump boundary condition values. If convergence instability is observed in cases using porous jump boundary conditions and the **Coupled** scheme, it is recommended that you change the pressure-velocity coupling to one of the segregated schemes.

36.3.1.5. User Inputs

You can specify the pressure-velocity coupling method in the Solution Methods Task Page (p. 5157) (Figure 36.4: The Solution Methods Task Page for a Pressure-Based Segregated Algorithm (p. 3583)).

Solution → **Methods**

Choose **SIMPLE**, **SIMPLEC**, **PISO**, **Fractional Step**, or **Coupled** in the **Pressure-Velocity Coupling** drop-down list.

If you choose **PISO**, the task page will expand to show the additional parameters for pressure-velocity coupling. By default, the number of iterations for **Skewness Correction** and **Neighbor Correction** are set to `1`. If you want to use only **Skewness Correction**, then set the number of iterations for **Neighbor Correction** to `0`. Likewise, if you want to use only **Neighbor Correction**, then set the number of iterations for **Skewness Correction** to `0`. For most problems, you do not need to change the default iteration values. By default, the **Skewness-Neighbor Coupling** option is enabled to allow for a more economical, but a less robust variation of the PISO algorithm. For further details, see

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Neighbor Correction, Skewness Correction, and Skewness - Neighbor Coupling in the *Fluent Theory Guide*.

If you choose **SIMPLEC** under **Pressure-Velocity Coupling**, you can also increase the number of iterations for the **Skewness Correction** from the default value of `0`. For further details, see Skewness Correction in the *Fluent Theory Guide*.

When a segregated solver is selected, you can use the following text command to enable an enhanced formulation for the skewness correction that computes the pressure-correction gradient. It is only used during the simulation when **SIMPLEC** or **PISO** is selected with one or more active **Skewness Correction** iterations. In a mesh with very skewed elements, the enhanced formulation should improve solver robustness—that is, avoid stalled convergence or divergence—when compared to the default skewness correction formulation. In the rare case where it decreases stability, you can attempt to remedy it by lowering the under-relaxation factor and (for **PISO**) disabling the **Skewness-Neighbor Coupling** option. Note that the use of either default or enhanced skewness correction formulation does not affect the accuracy of a converged simulation.

`solve` → `set` → `advanced` → `skewness-correction-enhanced`

If you choose **SIMPLE**, **SIMPLEC**, or **PISO** and select **Local Time Step** from the **Pseudo Time Method** drop-down list, you will have to specify the **Pseudo Time Courant Number** in the **Solution Controls** task page, which is set at `5` by default. For more information, refer to Local Time Step Method in the Theory Guide.

If you choose **Coupled** for a transient case or select **Off** from the **Pseudo Time Method** drop-down list for a steady case, you will have to specify the **Flow Courant Number** in the **Solution Controls** task page, which is set at `200` by default. You will also specify the **Explicit Relaxation Factors** for **Momentum** and **Pressure**, which are set at `0.5` by default. For more information about these options, refer to Pressure-Velocity Coupling and Steady-State Iterative Algorithm in the Theory Guide.

For cases with very skewed meshes, the run can be stabilized by further reduction of the explicit relaxation factor to `0.25`. If Ansys Fluent immediately diverges in the AMG solver, then the CFL number is too high and should be reduced. Reducing the CFL number below `10` is not recommended since it would be better to use the segregated algorithm for the pressure-velocity coupling.

In most transient cases, the CFL number should be set to a large value such as 10^7 and explicit relaxation factors to 1.0.

If you choose **Coupled** and have **Global Time Step** selected from the **Pseudo Time Method** drop-down list, you will set the **Pseudo Time Explicit Relaxation Factors** in the **Solution Controls** task page, as described in Global Time Step Method Settings (p. 3676).

36.3.2. Mass Flux Types

The method for calculating the mass flux is defined using the **Flux Type** drop-down list in the **Solution Methods** task page.

Solution → **Methods**

For the pressure-based solver, Fluent offers two formulations for the mass flux computation:

- **Rhie-Chow: momentum based**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This flux option follows a momentum-coefficient-weighted high-order velocity interpolation with a Rhie-Chow correction for the pressure gradient difference.

- **Rhie-Chow: distance based**

  This flux option applies a distance-weighted high-order velocity interpolation with a Rhie-Chow correction for the pressure gradient difference.

By default, the **Auto Select** option is enabled next to the **Flux Type** list so that Fluent automatically selects the most robust flux formulation based on the flow models and mesh attributes. The **Rhie-Chow: momentum based** option is selected for most cases. The **Rhie-Chow: distance based** option is automatically selected only for the following combination: a transient simulation that uses a Scale-Resolving Simulation (SRS) model (see Scale-Resolving Simulation (SRS) Models (p. 2073)), a compressible fluid, and an adapted mesh. In this context, a mesh is considered to be adapted if you have manually adapted the mesh (even if you subsequently coarsened the mesh), you have defined automatic adaption criteria prior to starting the calculation, and/or it is a hexcore, poly-hexcore, or CutCell mesh (see Generating the Hexcore Mesh (p. 795), Generating Poly-Hexcore Meshes (p. 808), or Generating the CutCell Mesh (p. 810), respectively) that has a hanging node as the result of hex refinement.

Note:

The following statements describe the behavior of the automatic selection of the mass flux formulation after a time step has been calculated:

- The automatic flux type selection *will not* change if you then adapt the mesh or define an automatic adaption criterion. In such circumstances it is generally preferable to continue with the same flux type rather than to risk a jump in the residuals.
- The automatic flux type selection *may* change if you make a change in the temporal discretization (steady vs. transient), material property (compressible vs. incompressible), and/or turbulence model (scale-resolving vs. non-scale-resolving); you will be notified of such a change by an **Information** dialog box and a note in the console.

If you want greater control over the choice of mass flux formulation, you can disable the **Auto Select** option and then manually select your preferred flux formulation from the **Flux Type** drop-down list.

For further details about the **Flux Type** options, see Discretization of the Continuity Equation in the *Fluent Theory Guide*.

Note:

For multiphase models other than the wet steam model, the **Flux Type** list is not available and the **Rhie-Chow: momentum based** flux option is used.

36.3.3. Setting Under-Relaxation Factors

The pressure-based solver uses under-relaxation of equations to control the update of computed variables at each iteration (as described in Under-Relaxation of Equations in the Theory Guide). This means that all equations solved using the pressure-based solver, *including the non-coupled equations*

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

solved by the density-based solver (turbulence and other scalars, as discussed in Density-Based Solver in the Theory Guide), will have under-relaxation factors associated with them.

In Ansys Fluent, the default under-relaxation parameters for all variables are set to values that are near optimal for the largest possible number of cases. These values are suitable for many problems, but for some particularly nonlinear problems (for example, some turbulent flows or high-Rayleigh-number natural-convection problems) it is prudent to reduce the under-relaxation factors initially.

It is good practice to begin a calculation using the default under-relaxation factors. If the residuals continue to increase after the first 4 or 5 iterations, you should reduce the under-relaxation factors.

Occasionally, you may make changes in the under-relaxation factors and resume your calculation, only to find that the residuals begin to increase. This often results from increasing the under-relaxation factors too much. A cautious approach is to save a data file before making any changes to the under-relaxation factors, and to give the solution algorithm a few iterations to adjust to the new parameters. Typically, an increase in the under-relaxation factors brings about a slight increase in the residuals, but these increases usually disappear as the solution progresses. If the residuals jump by a few orders of magnitude, you should consider halting the calculation and returning to the last good data file saved.

Note that viscosity and density are under-relaxed from iteration to iteration. Also, if the enthalpy equation is solved directly instead of the temperature equation (that is, for non-premixed combustion calculations), the update of temperature based on enthalpy will be under-relaxed. To see the default under-relaxation factors, you can click the **Default** button in the Solution Controls Task Page (p. 5163).

For most flows, the default under-relaxation factors do not usually require modification. If unstable or divergent behavior is observed, however, you need to reduce the under-relaxation factors for pressure, momentum, k, and ε from their default values to about 0.2, 0.5, 0.5, and 0.5. (It is usually not necessary to reduce the pressure under-relaxation for SIMPLEC.) In problems where density is strongly coupled with temperature, as in very-high-Rayleigh-number natural- or mixed-convection flows, it is wise to also under-relax the temperature equation and/or density (that is, use an under-relaxation factor less than 1.0). Conversely, when temperature is not coupled with the momentum equations (or when it is weakly coupled), as in flows with constant density, the under-relaxation factor for temperature can be set to 1.0.

For other scalar equations (for example, swirl, species, mixture fraction and variance) the default under-relaxation may be too aggressive for some problems, especially at the start of the calculation. You may want to reduce the factors to 0.8 to facilitate convergence.

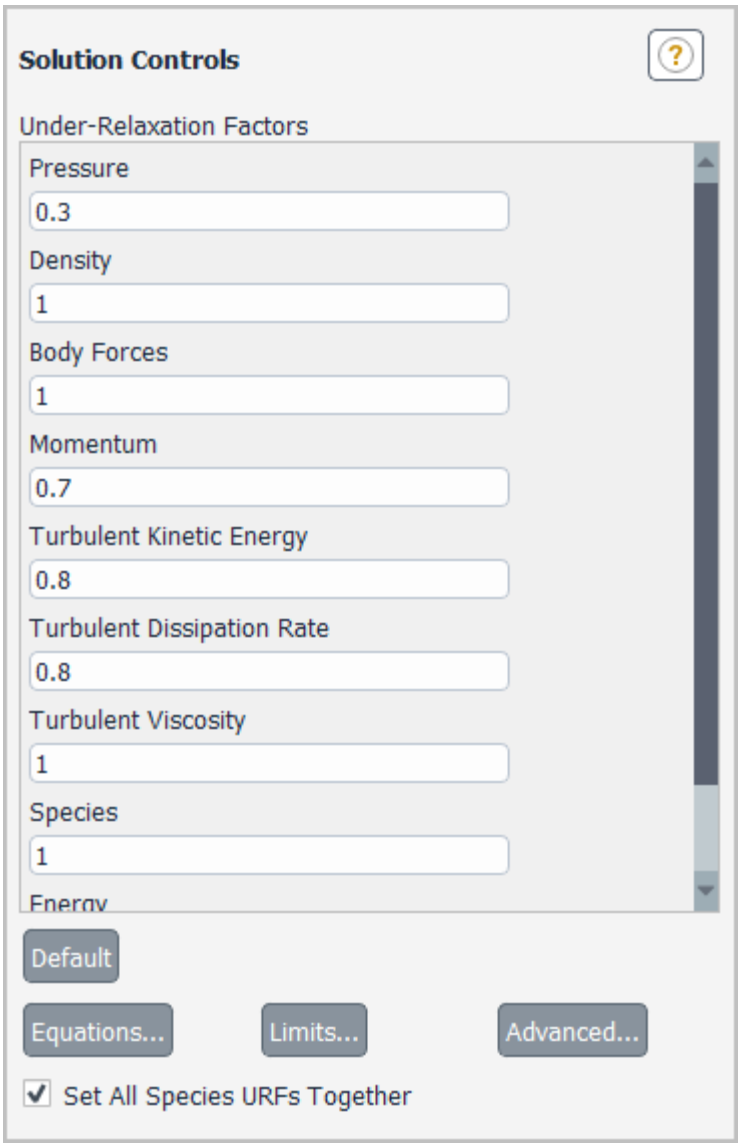
36.3.3.1. User Inputs

You can modify the under-relaxation factors in the Solution Controls Task Page (p. 5163) (Figure 36.5: The Solution Controls Task Page for the Pressure-Based Solver (p. 3591)).

Solution → **Controls**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 36.5: The Solution Controls Task Page for the Pressure-Based Solver



You can set the under-relaxation factor for each equation in the field next to its name under **Under-Relaxation Factors**.

Important:

If you are using the pressure-based solver, all equations will have an associated under-relaxation factor (see Under-Relaxation of Equations in the Theory Guide). If you are using the density-based solver, only those equations that are solved sequentially (see Density-Based Solver in the Theory Guide) will have under-relaxation factors.

If your case involves species transport, you can set the under-relaxation factors for each of the listed species. If you want all your species to use the same under-relaxation factors, simply enable the **Set All Species URFs Together** option. Notice that you will no longer see your list of individual species, instead a **Species** field will appear where you will specify the under-relaxation factor.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you change under-relaxation factors, but you then want to return to Ansys Fluent's default settings, you can click the **Default** button.

Note that with optimal settings, the convergence of the coupled pressure-velocity algorithm will be limited by the segregated solution of other scalar equations, for example, turbulence. For optimum solver performance, you will need to increase the relaxation factors for these equations to a value greater than the default values.

36.3.4. Setting Solution Controls for the Non-Iterative Solver

You can use the non-iterative solver (also referred to as "NITA"—see Time-Advancement Algorithm in the Theory Guide) for transient problems in order to increase the speed and efficiency of the calculations. You can also obtain a steady solution using the non-iterative multiphase solver for multiphase flows with quasi-steady conditions.

To control the change of computed variables at each iteration, you can specify explicit relaxation in the **Non-Iterative Solver Relaxation Factors** list in the Solution Controls Task Page (p. 5163). Additional information on relaxation factors can be found in Setting Under-Relaxation Factors (p. 3589).

Other solution controls are accessible via the **Expert** tab, in the **Advanced Solution Controls** dialog box (see Figure 36.6: The Advanced Solution Controls Dialog Box for the Pressure-Based Segregated Non-Iterative Solver (p. 3594)). The criteria for convergence include the **Correction Tolerance** (defined by the overall accuracy), **Residual Tolerance** (controlling the solution of the linear equations), and **Max. Corrections** (controlling the maximum number of sub-iterations for each individual equation). The default control settings are optimally designed in order to get a second-order accurate solution.

To use Ansys Fluent's non-iterative transient solver in order to boost the efficiency of transient simulations:

1. Go to the **Solution Methods** task page.

   Solution → **Methods**

2. Make sure that **Coupled** is not selected from the **Pressure-Velocity Coupling** drop-down list.

3. Enable **Non-Iterative Time Advancement**.

4. Under **Pressure-Velocity Coupling**, select a scheme from the drop-down list:

   - For single phase or VOF/Mixture multiphase flows: You can select either the **Fractional Step** or **PISO** scheme. For the **PISO** scheme, you can set the value for the **Neighbor Correction**; the value for skewness correction is determined automatically with the goal of convergence of the pressure equation (to print these "pressure correction equations" in the console during the solution, you can increase the multigrid verbosity using the `solve/set/multi-grid-amg` text command).

   - For Eulerian multiphase flows: Select the **Phase Coupled SIMPLE** scheme. This is the only pressure scheme for Eulerian multiphase simulations that allows the NITA option.

5. When using the Large Eddy Simulation (LES) turbulence model, you can enable the **Accelerated Time Marching** option in order to speed up the simulation. This option is intended for unreacting

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

flow simulations that use a constant-density fluid. It enables a modified NITA scheme and changes the following to a more aggressive setting:

- The maximum number of sub-iterations for the NITA solver equations (specified in the **Expert** tab of the **Advanced Solution Controls** dialog box) are revised: the **Max. Corrections** is set to `0` for **Pressure** and `1` for all other equations.
- The pressure-velocity coupling scheme (specified in the **Solution Methods** task page) is changed to the **Fractional Step** method.
- The multigrid solver settings (specified in the **Multigrid** tab of the **Advanced Solution Controls** dialog box) are revised: for the **Pressure** equation, the **Cycle Type** is changed to **F-Cycle** and the **Termination** criterion is set to `0.01`.

Note:

The following limitations / recommendations apply:

- The accelerated time marching option is not available with the Eulerian multiphase model.
- It is not recommended for compressible or variable-density fluids.
- When using this option, it is recommended that you use the **Bounded Central Differencing** scheme for the **Momentum** equation rather than the **Central Differencing** scheme.

36.3.4.1. User Inputs

You can modify the under-relaxation factors in the Solution Controls Task Page (p. 5163) (Figure 36.5: The Solution Controls Task Page for the Pressure-Based Solver (p. 3591)).

Solution → Controls

The relaxation factors define the explicit relaxation (see Under-Relaxation of Variables in the Theory Guide) of variables between sub-iterations. The relaxation factors can be used to prevent the solution from diverging.

If the solution is unstable:

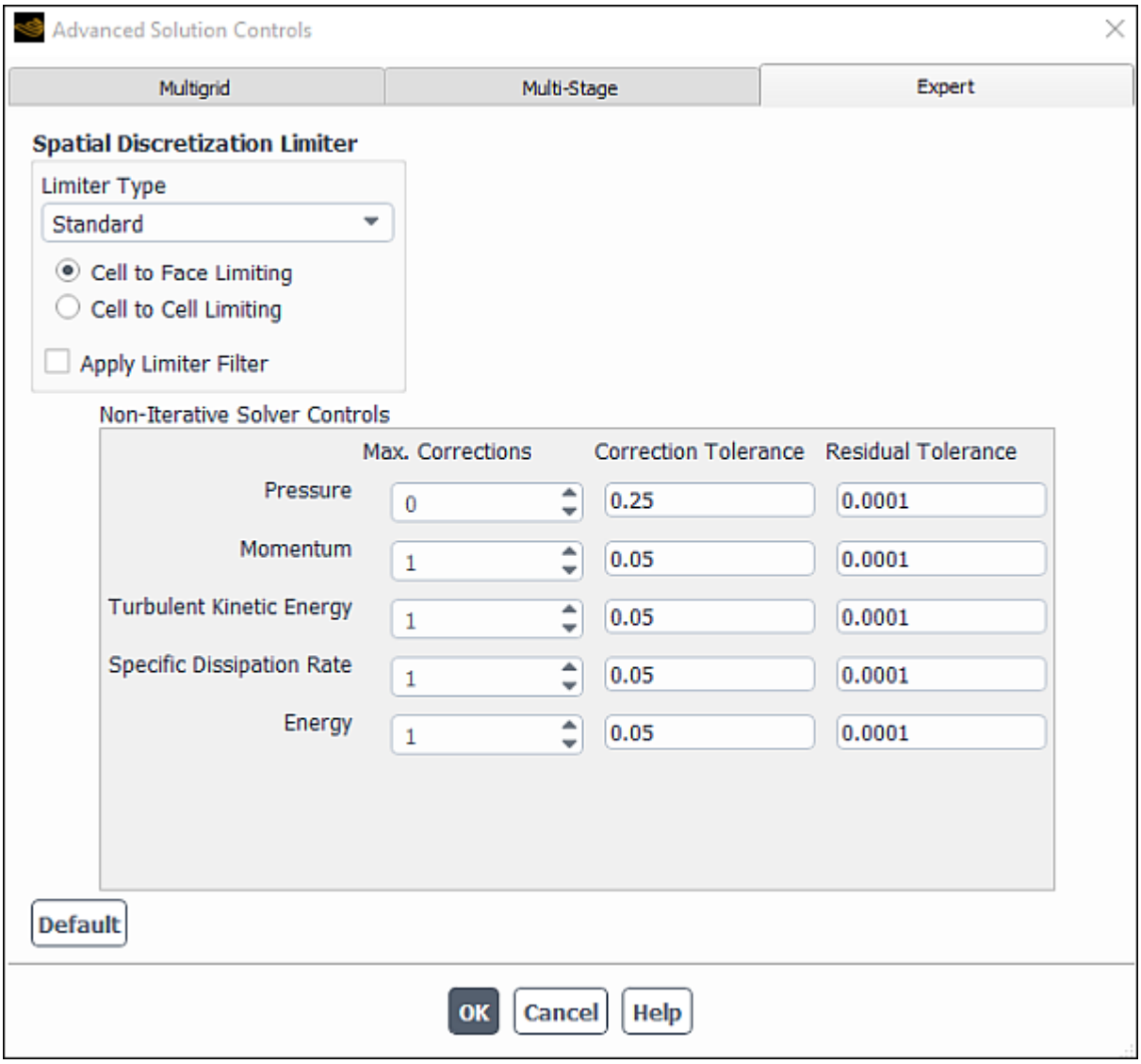
- For single phase flows: You should first try to stabilize the solution by lowering the relaxation factors for pressure to 0.7–0.8, and by reducing the time step size.
- For multiphase flows: Consider lowering the explicit pressure relaxation factor up to 0.5. This may help when meshes are poor, because a poor mesh will significantly affect the pressure gradient, which in turn may generate large source terms in the governing equations resulting in poor convergence. You can set the under-relaxation factors for the other flow variables to higher values (0.8-1.0). For compressible multiphase flow, you can use explicit relaxation of density in the range 0.5-0.8. For multiphase cases with mass transfer, you can use explicit relaxation of vaporization mass in the range 0.5-0.8.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can modify the non-iterative solution controls in the Advanced Solution Controls Dialog Box (p. 5168) (Figure 36.6: The Advanced Solution Controls Dialog Box for the Pressure-Based Segregated Non-Iterative Solver (p. 3594)).

Solution → **Controls** → **Advanced...**

Figure 36.6: The Advanced Solution Controls Dialog Box for the Pressure-Based Segregated Non-Iterative Solver



Under **Non-Iterative Solver Controls**, there are several parameters that control the sub-iterations for the individual equations.

The sub-iterations for an equation stop when the total number of sub-iterations exceeds the value specified for **Max. Corrections**, regardless of whether or not the convergence criteria (described below) are met.

The sub-iterations for an equation end when the ratio of the residuals at the current sub-iteration and the first sub-iteration is less than the value specified in the **Correction Tolerance** field. You can monitor the details of the sub-iteration convergence by looking at the AMG solver performance

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(that is, setting the **Verbosity** field in the **Multigrid** tab in the **Advanced Solution Controls** dialog box to **1**). Be sure to pay attention to the residuals for the current sub-iteration (that is, the residual for the 0-th AMG cycle at the current sub-iteration) and the initial residual of the time step (that is, the residual for the 0-th AMG cycle of the first sub-iteration). The ratio of these two residuals is what is controlled by the **Correction Tolerance** field. These two residuals are also the residuals plotted when using the **Residual Monitor** panel and reported in the Ansys Fluent console at the end of a time step. Note that the residuals reported at the end of a time step can be scaled or unscaled, depending on the settings in the **Residual Monitor** dialog box. The residuals reported when monitoring the AMG solver performance are always unscaled.

For each interim sub-iteration, the AMG cycles continue until the usual AMG termination criteria (0.1 by default, and set in the **Multigrid** tab) are met. However, for the last sub-iteration (that is, either when the maximum number of sub-iterations are reached or when the correction tolerance is satisfied), the AMG cycles continue until the ratio of the residual at the current cycle to the initial residual (the residual for the 0-th AMG cycle of the first sub-iteration of the time step) drops below the value specified for **Residual Tolerance**. You may want to adjust the **Residual Tolerance**, depending on the size of the time step selected. The default **Residual Tolerance** should be well suited for moderate-sized time steps (that is, for cell CFL numbers of 1 to 10). Note that you can display the cell CFL numbers for unsteady problems by selecting **Cell Courant Number** in the **Velocity...** category of all postprocessing dialog boxes. For very small time steps (cell CFL <<1), the diagonal dominance of the system is very high and the convergence should be driven further by reducing the **Residual Tolerance** value. For larger time steps (cell CFL >>1), it may be possible that the residual tolerance cannot be reached due to round-off errors, and unless the **Residual Tolerance** value is increased, AMG cycles can be wasted. Again, this can be monitored by monitoring the AMG solver performance.

36.3.4.2. Hybrid NITA for the VOF and Mixture Multiphase Models

For NITA cases that involve VOF or Mixture multiphase model, the **Hybrid NITA** option may help you improve solution stability and robustness, but at the cost of speed. Depending on your case, the hybrid NITA settings may optimize NITA expert controls, AMG settings, explicit relaxation factors, pressure interpolation scheme, neighbor corrections for PISO, and outer iterations.

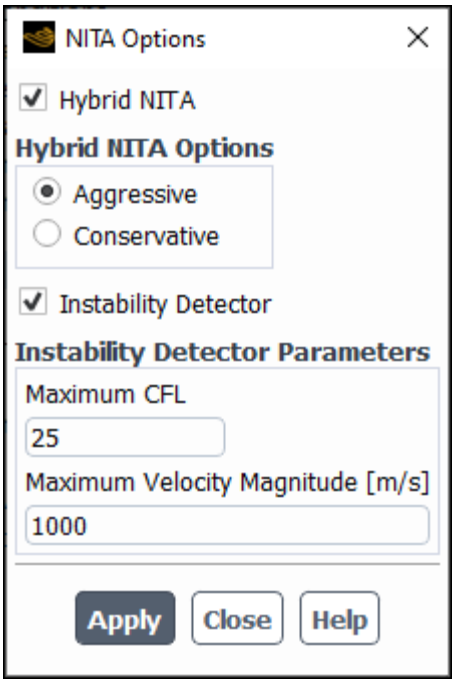
To enable and edit the Hybrid NITA settings:

1. In the **Solution Methods** task page, enable the **Non-Iterative Time Advancement** option and click the **Options** button next to it.

2. In the **NITA Options** dialog box that opens, enable **Hybrid NITA**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 36.7: The NITA Options Dialog Box



3. In the **Hybrid NITA Options** group box, select from of the following options:

   - **Aggressive** : (default) Uses two outer iterations per time-step
   - **Conservative** : Uses three outer iterations per time-step

   For each option, NITA settings are automatically optimized and, in general, do not require modification for VOF or Mixture multiphase cases.

4. In case of divergence when using NITA, you can enable **Instability Detector**.

   The hybrid NITA solver performs a fixed number of outer iterations. However, in case of local instabilities during the simulation arising due to meshing events or physical phenomena, large errors may accumulate due to insufficient convergence. The instability detector helps with possible instability problems.

   The instability detector traces local instabilities using the following **Instability Detector Parameters** :

   - **Maximum CFL** : Is the maximum of advective Courant number for the instability detection (default = 25)
   - **Maximum Velocity Magnitude** : Is the maximum of the velocity magnitude for the instability detection (default - 1000 m/s)

   The case when the local Courant number exceeds **Maximum CFL**, or the local velocity magnitude exceeds **Maximum Velocity Magnitude** will be considered as an unstable event. The solver will increase the number of outer iterations for that time step and will make some internal adjustments for stability.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

With the **Hybrid NITA** option enabled, you can use additional advanced hybrid NITA controls to improve stability. They can be accessed via the following text command:

```
solve/set/multiphase-numerics/advanced-stability-controls/hybrid-nita/
```

The following controls are available:

- CFL Type

  By default, the instability detector operates on the global advective CFL number calculated in the whole domain. You can change to the interfacial advective CFL definition using the following command:

  ```
  solve/set/multiphase-numerics/advanced-stability-controls/hybrid-nita/instability-detector/set-cfl-type
  Advective CFL type [0: Global, 1: Interfacial] (0 1) [0] 1
  ```

- Initial outer iterations

  In most transient applications, the initial conditions do not satisfy continuity. Therefore, being conservative at the start of simulation can help provide better solution stability. By default, Ansys Fluent uses 5 outer iterations for initial 5 time-steps. If necessary, you can change these settings using the following text command:

  ```
  solve/set/multiphase-numerics/advanced-stability-controls/hybrid-nita/initial-outer-iterations
  Number of initial time-steps [5]
  Number of initial outer iterations [5]
  ```

  Note:

  Initial outer iterations will overwrite outer-iterations for initial time-steps as specified, if initial outer iterations are larger than outer iterations.

- Outer iterations for an unstable event

  To increase the number of outer iterations for an unstable event, use the following text command:

  ```
  solve/set/multiphase-numerics/advanced-stability-controls/hybrid-nita/instability-detector/unstable
  -event-outer-iterations
  ```

  By default, the NITA solver uses 5 outer iterations.

- Outer iterations

  By default, the hybrid NITA scheme provides either 2 or 3 outer iterations for the `1 - aggressive` and `2 - conservative` options. You can increase the number of outer iterations for certain complex applications using the following text command:

  ```
  solve/set/multiphase-numerics/advanced-stability-controls/hybrid-nita/outer-iterations
  Number of outer iterations [2]
  ```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note, however, that this will increase the solution run time.

Note:

Currently, the residual reports display two residuals for each outer iteration.

Solution Strategies

It is generally not recommended to change the explicit relaxation factors and expert controls for NITA because improper convergence may lead to accumulation of errors and unphysical results. In case of divergence when using NITA, the following strategies may help:

- Enable the instability detector and optimize inputs for the **Maximum CFL** and **Maximum Velocity Magnitude**.
- Use the conservative option for Hybrid NITA.
- Reduce the pressure relaxation factor to up to 0.5.
- Reduce the time step size.
- For cases that involve strong inter-equation coupling (as in mass transfer, compressible flow, and highly viscous flow), using the **Conservative** option for Hybrid NITA is recommended.
- For the Mixture multiphase model, if you are solving for slip velocities, you can specify the explicit relaxation factor for **Slip Velocity** in the **Solution Controls** task page.

36.3.4.3. NITA Expert Options

- To obtain diagnostics on convergence of sub-iterations for individual equations, use the following text command:

  ```
  solve/set/nita-expert-controls/set-verbosity

  verbosity [0] 1
  ```

  The output information may help you to judge whether the specified number of sub-iterations is sufficient to reach the correction tolerance and also to diagnose the source of the solution divergence.

- For NITA cases with **PISO** selected as a **Pressure-Velocity Coupling** scheme, you can enable coupling of the neighbor and skewness corrections with the following TUI command:

  ```
  solve/set/nita-expert-controls/skewness-neighbor-coupling

  enable skewness neighbor coupling for nita [no] yes
  ```

  For more information about this option, see Skewness - Neighbor Coupling in the *Fluent Theory Guide*.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.3.4.4. Compatibility of the NITA Scheme with Other Ansys Fluent Models

The following is a list of models that are compatible with the non-iterative time advancement solver:

- Inviscid flow (excluding ideal gas)
- Laminar flow
- All models of turbulence (including LES and DES)
- S2S radiation model
- Heat transfer
- Non-reacting species transport
- General compressible flows (most subsonic and some transonic applications)
- VOF multiphase model (most applications)
- Mixture multiphase model
- Eulerian multiphase model
- Phase change (solidification and melting)
- Porous media model (isotropic resistance)
- Multiphase model with Multi-Fluid VOF
- Multiphase RSM turbulence model

The following is a list of models that are compatible with the non-iterative solver, but may result in some instabilities and inaccuracies for certain flow conditions:

- RSM turbulence model
- MDM
- Non-Newtonian fluids
- General compressible flows (aerospace supersonic applications)
- Reacting species and any type of combustion including PDF

The following is a list of models that are not compatible with the non-iterative solver:

- Radiation models (except S2S)
- DPM, spark, and crevice models
- UDS transport
- Porous jump

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Porous media model (anisotropic resistance)
- RSM turbulence model
- Floating operating pressure

Important:

- The PRESTO! pressure interpolation scheme, when used with the non-iterative time-advancement solver, is less stable than in the case of the iterative time-advancement solver. As a consequence, a smaller time step size may be required.
- As mentioned above, the default control settings are optimally designed to obtain a second-order solution. In order to save CPU time, in cases where transient accuracy is not a main concern (that is, first-order integration in time and space), or when NITA is used to converge toward a steady-state solution, you may want to set the **Max. Corrections** value to `1` in the **Advanced Solution Controls** dialog box (**Expert** tab) for all transport equations except pressure.

36.3.5. Equation Order

For transient simulations that use the pressure-based solver, the order in which the model equations are solved can affect the speed of convergence. You can specify the equation order using the following text command:

`solve` → `set` → `equation-ordering`

Note that the `standard` method is enabled by default and corresponds to the ordering shown in Figure 23.7: Overview of the Iterative Time Advancement Solution Method For the Segregate Solver and Figure 23.8: Overview of the Non-Iterative Time Advancement Solution Method in the Theory Guide ; alternatively, you can select the `optimized-for-volumetric-expansion` method, which is recommended for flows in which the density is strongly dependent on thermal effects, chemical composition, and so on (such as combustion simulations). This text command is not available when a multiphase model is enabled.

36.3.6. Using the Correction Form of Momentum Discretization

For simulations that use the pressure-based solver, a *correction* form of the discretized momentum equation (as described in Correction Form Discretization of the Momentum Equations in the Theory Guide) is used by default. The correction form includes the following advantages:

- Avoids floating point error accumulation while collecting source terms, such that equations with large source terms are numerically more robust.
- Allows for more simulations to be performed using single precision.
- Increases the robustness of the pressure-based coupled solver.
- Increases the speed of the pressure-based coupled solver.
- Becomes more aligned with the density-based coupled solver.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can disable the use of the correction form by using the following text command:

`solve → set → advanced → correction-form`

Note:

In a future release it will not be possible to disable the correction form, and the previous text command will be removed.

36.4. Density-Based Solver Settings

To use the density-based solver you must first select the solver type, and determine if the simulation is steady-state or transient from the **General** Task Page (see Choosing the Solver (p. 3573)). The density based solver settings are available mainly in two task pages: the Solution Methods Task Page (p. 5157) and the Solution Controls Task Page (p. 5163).

In the **Solution Methods** task page, you can select the following:

- Solution formulation type: Implicit or Explicit
- Flux Scheme type: Roe-FDS, AUSM, or Low Diffusion Roe-FDS
- Flow equation and model equation spatial discretization accuracy
- For the steady-state solution method, you can select additional solution options to accelerate convergence
  - Pseudo time method (Implicit formulation only)
  - Convergence acceleration for stretched meshes
- High-Speed Numerics

  For more details, see Enabling High-Speed Numerics (p. 3608).
- For the transient formulation you can select
  - first-order implicit
  - second-order implicit and for the explicit solver formulation, you can also select the explicit transient formulation

In the **Solution Controls** task page, you can select the following:

- For the density-based explicit solver, you can specify the Courant number, FAS multigrid level, and Residual smoothing
- For the density-based implicit solver, you can need to enter only the Courant number
- For both solver methods, you can enter the under-relaxation factors associated with other equations solved with the flow equations, such as equations of the turbulence model

The above options can be found in the following sections:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.4.1. Changing the Courant Number
36.4.2. Convective Flux Types
36.4.3. Convergence Acceleration for Stretched Meshes (CASM)
36.4.4. Enabling High-Speed Numerics
36.4.5. Preventing Divergence Using Local Under-Relaxation
36.4.6. Specifying the Explicit Relaxation
36.4.7. Turning On FAS Multigrid

36.4.1. Changing the Courant Number

For Ansys Fluent's density-based solver, the main control over the time-stepping scheme is the Courant number (CFL). The time step size is proportional to the CFL, as defined in Equation 23.89 in the Theory Guide.

Linear stability theory determines a range of permissible values for the CFL (that is, the range of values for which a given numerical scheme will remain stable). When you specify a permissible CFL value, Ansys Fluent will compute an appropriate time step size using Equation 23.89 in the Theory Guide. In general, taking a larger time step size leads to faster convergence, so it is advantageous to set the CFL as large as possible (within the permissible range).

The stability limits of the density-based implicit and explicit formulations are significantly different. The explicit formulation has a more limited range and requires lower CFL settings than does the density-based implicit formulation. Appropriate choices of CFL for the two formulations are discussed below.

36.4.1.1. Courant Numbers for the Density-Based Explicit Formulation

Linear stability analysis shows that the maximum allowable CFL for the multi-stage scheme used in the density-based explicit formulation will depend on the number of stages used and how often the dissipation and viscous terms are updated (see Changing the Multi-Stage Scheme (p. 3624)). But in general, you can assume that the multi-stage scheme is stable for Courant numbers up to 2.5. This stability limit is often lower in practice because of nonlinearities in the governing equations.

The default CFL for the density-based explicit formulation is 1.0, but you may be able to increase it for some 2D problems. You should generally not use a value higher than 2.0.

If your solution is diverging, that is, if residuals are rising very rapidly, and your problem is properly set up and initialized, this is usually a good sign that the Courant number must be lowered. Depending on the severity of the startup conditions, you may need to decrease the CFL to a value as low as 0.1 to 0.5 to get started. Once the startup transients are reduced you can start increasing the Courant number again.

36.4.1.2. Courant Numbers for the Density-Based Implicit Formulation

Linear stability theory shows that the density-based implicit formulation is unconditionally stable. However, as with the explicit formulation, nonlinearities in the governing equations will often limit stability.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The default CFL for the density-based implicit formulation is 5.0. It is often possible to increase the CFL to 10, 20, 100, or even higher, depending on the complexity of your problem. You may find that a lower CFL is required during startup (when changes in the solution are highly nonlinear), but it can be increased as the solution progresses.

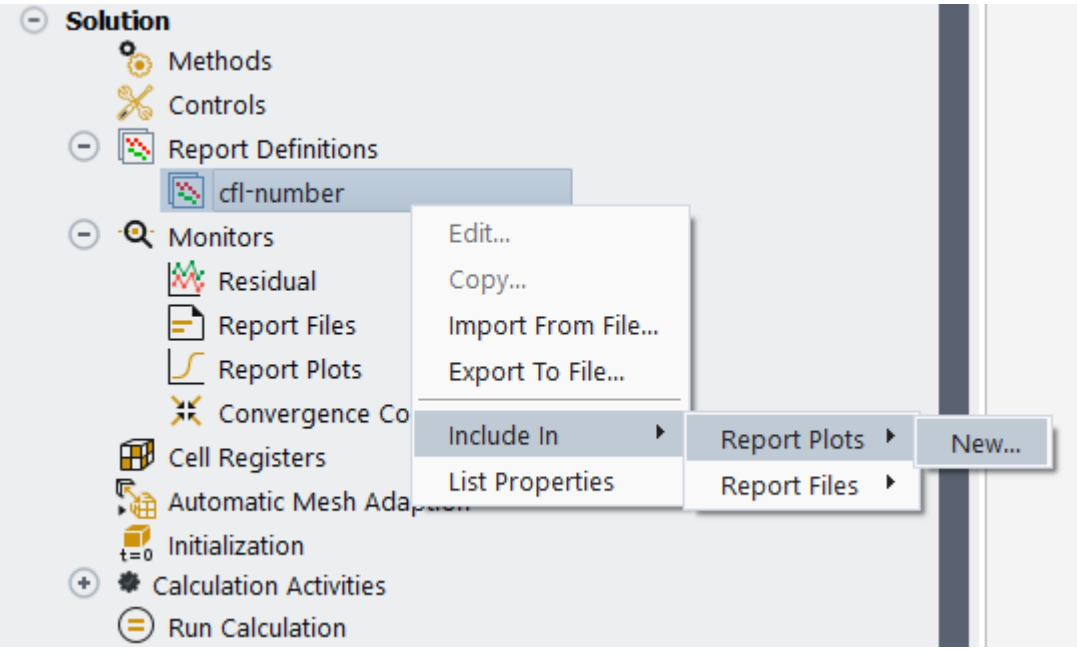
The coupled AMG solver has the capability to detect divergence of the multigrid cycles within a given iteration. If this happens, it will automatically reduce the CFL and perform the iteration again, and a message will be printed to the screen. Five attempts are made to complete the iteration successfully. Upon successful completion of the current iteration the CFL is returned to its original value and the iteration procedure proceeds as required.

36.4.1.3. Courant Number Monitor

A CFL report definition will be automatically created for the density-based implicit steady solver. This report definition can be utilized to track the CFL when solution steering (Solution Steering (p. 3746)) or divergence prevention (Preventing Divergence Using Local Under-Relaxation (p. 3608)) is activated.

You can monitor the CFL by creating a plot and/or saving it to a file by right-clicking the report definition and selecting **Include In** → **Report Plots**/**Report Files** → **New** as shown in Figure 36.8: Creating CFL Report Plots and Files (p. 3603). For more details on creating Report Files or Report Plots, see Report Files and Report Plots (p. 4123).

Figure 36.8: Creating CFL Report Plots and Files

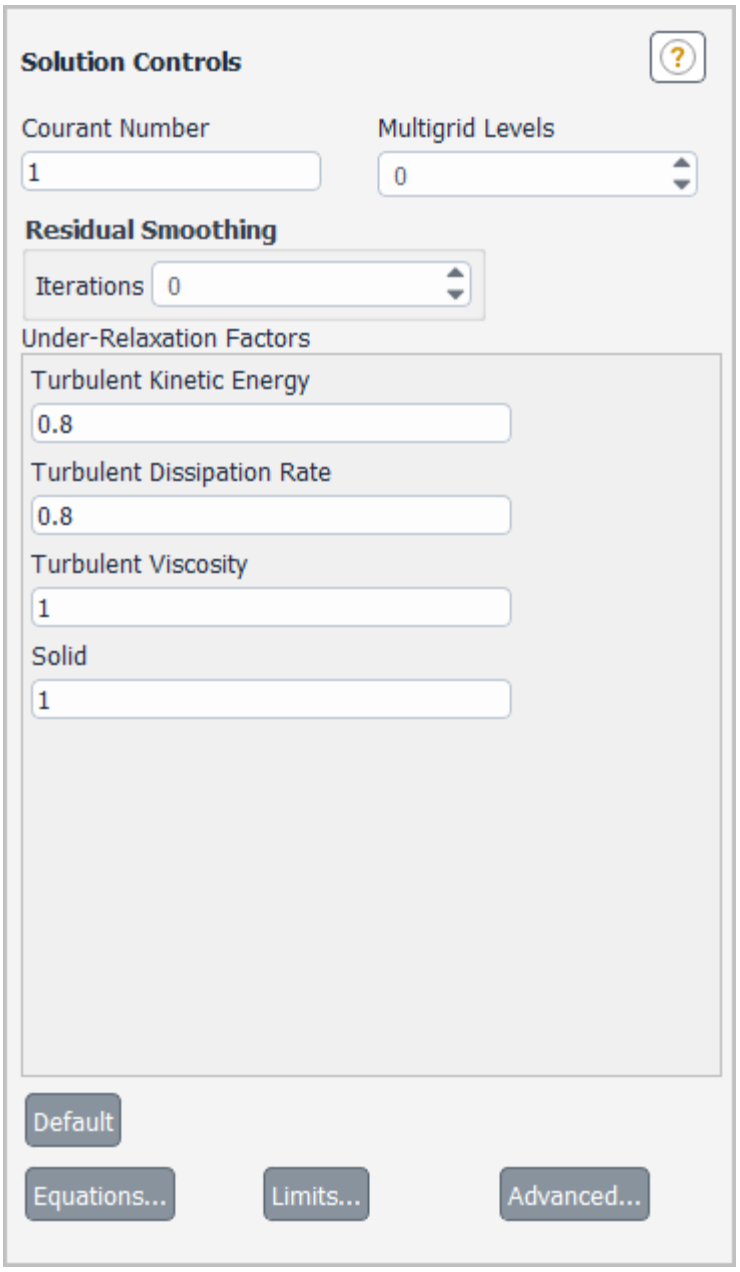


36.4.1.4. User Inputs

The Courant number is set in the Solution Controls Task Page (p. 5163) (Figure 36.9: The Solution Controls Task Page for the Density-Based Explicit Formulation (p. 3604)).

Solution → **Controls**

Figure 36.9: The Solution Controls Task Page for the Density-Based Explicit Formulation



Enter the value for **Courant Number**.

When you select **Explicit** from the **Formulation** drop-down list in the Solution Methods Task Page (p. 5157), Ansys Fluent will automatically set the **Courant Number** to 1; when you select **Implicit** from the **Formulation** drop-down list, the **Courant Number** will be changed to 5 automatically.

36.4.2. Convective Flux Types

The convective fluxes are selected from the **Flux Type** drop-down list in the **Solution Methods** task page.

Solution → **Methods**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can select from the following when using the density-based solver:

- Roe flux-difference splitting (**Roe-FDS**)

  Roe-FDS splits the fluxes in a manner that is consistent with their corresponding flux method eigenvalues. It is the default and is recommended for most cases.

- Advection Upstream Splitting Method (**AUSM**)

  AUSM provides exact resolution of contact and shock discontinuities and it is less susceptible to Carbuncle phenomena.

- Low diffusion Roe flux-difference splitting (**Low Diffusion Roe-FDS**)

  Low Diffusion Roe-FDS is available when a Scale-Resolving Simulation (SRS) turbulence model is enabled along with the time-implicit formulation. It reduces the dissipation in turbulence calculations. This should be used only for subsonic flows.

Note:

For cases that use an SRS turbulence model and/or include acoustic calculations, it is recommended that you use the **Roe-FDS** with the **Bounded Central Differencing** (BCD) discretization scheme selected for the flow equations, as this combination is more numerically robust than **Low Diffusion Roe-FDS** and just as accurate. The BCD scheme alters the standard Roe-FDS scheme such that the diffusion is reduced. Note that if you are not using an SRS turbulence model, to access the BCD scheme you will first need to enable the following text command: `solve/set/advanced/show-all-discretization-schemes`.

36.4.3. Convergence Acceleration for Stretched Meshes (CASM)

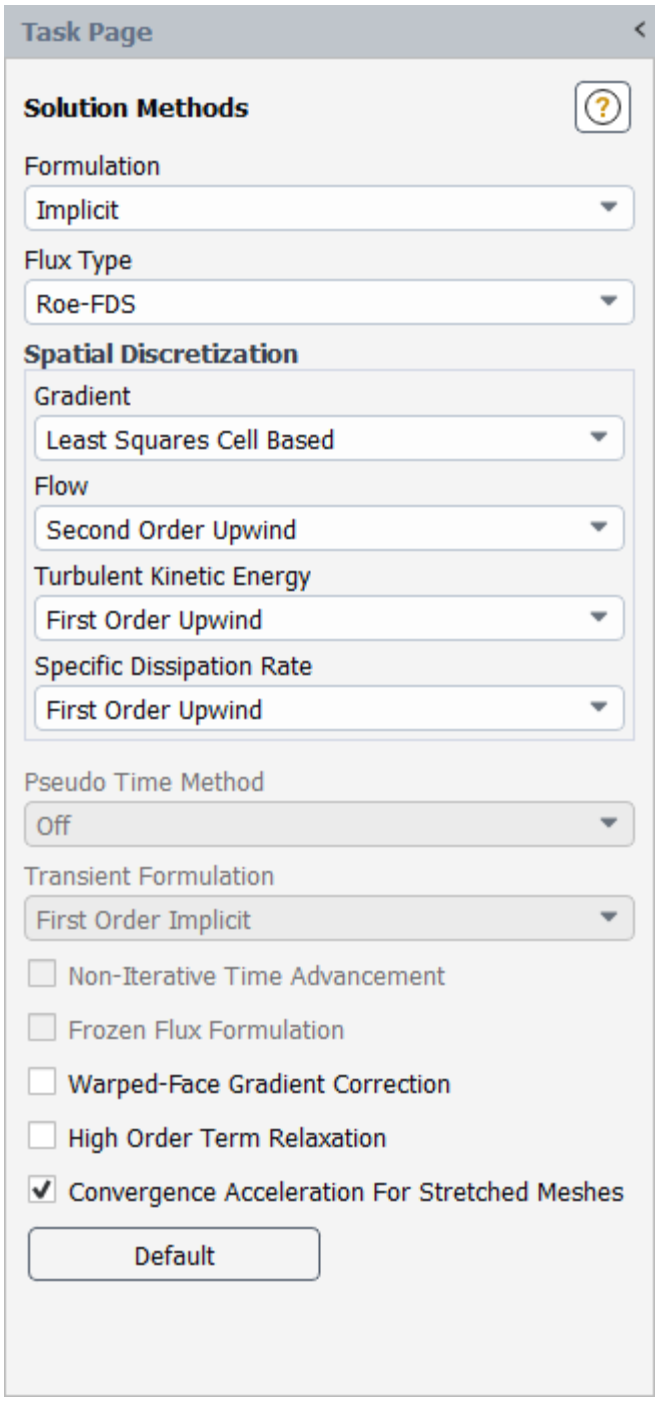
When using the density-based solver with the implicit solution formulation in steady-state you can accelerate the convergence of your solution on highly-stretched and anisotropic meshes (like the one used when modeling external aerodynamic problems) by selecting the **Convergence Acceleration For Stretched Meshes** in the **Solution Methods** task page. For further information and theoretical background on this solution acceleration option, see Convergence Acceleration for Stretched Meshes in the Theory Guide. The **Convergence Acceleration For Stretched Meshes** option provides an optimum solution convergence of the implicit solution method.

To apply convergence acceleration for stretched meshes, perform the following:

1. Specify the solver options by selecting **Density-Based** and **Steady** in the **General** task page.
2. Select **Implicit** from the **Formulation** drop-down list and enable **Convergence Acceleration For Stretched Meshes** in the **Solution Methods** task page (Figure 36.10: The Solution Methods Task Page for the Density-Based Implicit Formulation (p. 3606)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 36.10: The Solution Methods Task Page for the Density-Based Implicit Formulation



Extra settings for CASM can be set using the following text command:

`solve` → `set` → `convergence-acceleration-for-stretched-meshes/`

Enter `yes` in response to the `Use convergence acceleration for stretched meshes (CASM)?` question. You will also be asked for a cut-off on the CFL value multiplier. By default this value is set to 100. Typically, you do not need to adjust this value. But if convergence difficulties are encountered and reduction of the CFL value alone does not help improve convergence, then it is

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

advisable to reduce this CFL multiplier cut-off to a lower value (for example from 100 to 50, 20 or 10).

The use of CASM can typically give a much faster convergence over the standard solution method. In general, when using CASM, you do not need to specify a very large CFL value as you do with the standard solution method. A CFL value between 5 and 10 is typically used for converging most flow problems. When the **Convergence Acceleration For Stretched Meshes** option is selected, the solver will run when appropriate with a variable local CFL value proportional to the cell aspect ratios. Therefore, when the cell aspect ratio nears unity (typically far from walls), the local cell CFL value will be the same as the value that you supplied. However, as the cell is stretched and the cell aspect ratio increases (near walls), the local cell CFL value will be multiplied by the cell aspect ratio value. This is true until the cell stretching is beyond the multiplier cut-off value specified using the text command. The proportional change in CFL value on highly stretched cells helps accelerate the solution especially on highly packed and stretched meshes like the one used in modeling external flow problems.

When the cell aspect ratio is selected by default, the density-based implicit solver will operate with an explicit relaxation of 0.5 which can be adjusted by using the following text command: `solve/set/advanced/explicit-underrelaxation-value`.

The convergence of the solution is mainly controlled by adjusting the CFL value. Therefore, if convergence problems are encountered, lowering the CFL value will help improve the convergence. Additional solution parameters to be adjusted for more conservative solution settings are:

1. lowering the density-based implicit solver explicit relaxation (`solve/set/advanced/explicit-underrelaxation-value`)

2. adjusting the CFL multiplier cut-off value to a lower value (`solve/set/convergence-acceleration-for-stretched-meshes/`)

This solution convergence method is very aggressive. Therefore it is of paramount importance to start with a good initial guess especially if you start with second order spatial discretization. To get a good starting solution with the guess you provide, you are advised to use the full multi-grid initialization method (see Full Multigrid (FMG) Initialization (p. 3633)).

Note:

When the **Convergence Acceleration For Stretched Meshes** option is enabled, the **Pseudo Time Method** is turned off and is unavailable. You can either use **Convergence Acceleration For Stretched Meshes** or a **Pseudo Time Method**. These two cannot be used at the same time. Both methods help in obtaining faster convergence on anisotropic meshes. But one method requires that you enter a CFL value, while the other requires that you enter a pseudo time step value to march the solution to convergence. It is up to you to select the method with which you are most comfortable.

Convergence Acceleration For Stretched Meshes shows an advantage over the standard solution method, particularly with stretched meshes with low Y+ values (near unity). The use of **Convergence Acceleration For Stretched Meshes** results in an alteration of the numerical dissipation of the selected flux scheme. This change may slightly impact the monitored loading level if compared with the solution obtained without the use of this option.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.4.3.1. Enhanced Convergence Acceleration for Stretched Meshes

The density-based solver offers an enhanced implementation of the convergence-acceleration for stretched meshes (CASM). This alternative formulation is similar to the standard CASM, but employs a cell aspect ratio formula which improves robustness on very high aspect ratio meshes and is more sensitive to changing flow topologies. It is also independent of the initial solution field. The enhanced CASM formulation is engaged with the following text command:

```
> solve/set/convergence-acceleration-for-stretched-meshes
Use convergence acceleration for stretched meshes (CASM)? [yes] no
Use enhanced CASM formulation? [yes] yes
Engaging enhanced CASM with explicit-relaxation factor: 0.75
Enter CASM cut-off multiplier : [100] 100
.
--------------------------------------------------------------------------------
 Convergence Acceleration for Stretched Meshes (CASM) has been selected.
 - The use of FMG initialization is highly recommended to provide good initial
   solution field before the start of calculations with CASM option.
 - Maximum benefit of CASM option is realized when local flow is aligned
   with mesh stretching.
--------------------------------------------------------------------------------
```

Note that you must respond **no** to the first prompt and **yes** to the second prompt to engage the enhanced CASM.

The enhanced CASM formulation uses an explicit under-relaxation of 0.75 by default.

36.4.4. Enabling High-Speed Numerics

When using the density-based solver, built-in customized numeric settings are available that can help to stabilize and accelerate the convergence for high-speed flows, see High Speed Numerics (p. ?).

36.4.5. Preventing Divergence Using Local Under-Relaxation

It is a sign of local divergence when the temperature and/or pressure for individual cells approaches the minimum and/or maximum limits (set in the Solution Limits Dialog Box (p. 5166)), and this can precede the divergence of the global solution. If either of these quantities is being reset to a limiting value repeatedly (as indicated by the appropriate warning messages in the console), you should check the dimensions, boundary conditions, and properties to be sure that the problem is set up correctly, and try to determine the cause; you can create a field variable cell register to mark and display cells have a value equal to the limit (refer to Field Variable (p. 3855) for more information). If the setup appears appropriate but the density-based solver still diverges, it is recommended that you use divergence prevention, which has been shown to be helpful for challenging cases (such as those that undergo shocks or experience difficulties related to species calculations). With this option, Fluent will identify the locally diverging cells and automatically apply a treatment to them that freezes the pressure and/or temperature values (that is, sets the under-relaxation factor—which is similar to α in Equation 23.65 in the *Fluent Theory Guide* —to 0) and applies under-relaxation to the other variables. To enable divergence prevention, use the following text command:

`solve` → `set` → `divergence-prevention` → `enable?`

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When prompted, enter `yes`, and then define the under-relaxation factor for the other variables: the default value of 0.1 is recommended, though if this fails to prevent divergence you can try increasing this under-relaxation by reducing the factor to 0.

Note:

When you decide to enable divergence prevention, note the following: if the warning messages indicate that only a small percentage of cells are approaching the temperature and/or pressure limits, then you can just continue the calculation; if the percentage of cells is a higher value, it is recommended that you restart the calculation from initialization.

36.4.6. Specifying the Explicit Relaxation

To improve the convergence to steady-state for some flow cases when using the density-based implicit solver you can specify the explicit relaxation using the following text command:

`solve` → `set` → `advanced` → `explicit-underrelaxation-value`

Enter a value between `0` and `1`.

For more information about explicit relaxation, see Under-Relaxation of Variables in the Theory Guide.

36.4.7. Turning On FAS Multigrid

As discussed in Multigrid Method in the Theory Guide, FAS multigrid is an optional component of the density-based explicit formulation, while AMG multigrid is always on, by default for the density-based implicit formulation. Since nearly all density-based explicit calculations will benefit from the use of the FAS multigrid convergence accelerator, you should generally set a nonzero number of coarse grid levels before beginning the calculation. For most problems, this will be the only FAS multigrid parameter you will need to set. Should you encounter convergence difficulties, consider applying one of the methods discussed in Setting FAS Multigrid Parameters (p. 3618).

Important:

Note that you cannot use FAS multigrid with explicit time stepping (described in Temporal Discretization in the Theory Guide) because the coarse grid corrections will destroy the time accuracy of the fine grid solution.

36.4.7.1. Setting Coarse Grid Levels

As discussed in Full-Approximation Storage (FAS) Multigrid in the Theory Guide, FAS multigrid solves on successively coarser grids and then transfers corrections to the solution back up to the original fine grid, thereby increasing the propagation speed of the solution and speeding convergence. The most basic way you can control the multigrid solver is by specifying the number of coarse grid levels to be used.

As explained in Full-Approximation Storage (FAS) Multigrid in the Theory Guide, the coarse grid levels are formed by agglomerating a group of adjacent "fine" cells into a single "coarse" cell. The optimal number of grid levels is therefore problem-dependent. For most problems, you can start out with 4 or 5 levels. For large 3D problems, you may want to add more levels (although memory

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

restrictions may prevent you from using more levels, since each coarse grid level requires additional memory). If you believe that multigrid is causing convergence trouble, you can decrease the number of levels.

If Ansys Fluent reaches a coarse grid with one cell before creating as many levels as you requested, it will simply stop there. That is, if you request 5 levels, and level 4 has only 1 cell, Ansys Fluent will create only 4 levels, since levels 4 and 5 would be the same.

To specify the number of grid levels you want, set the number of **Multigrid Levels** in the Solution Controls Task Page (p. 5163) (Figure 36.9: The Solution Controls Task Page for the Density-Based Explicit Formulation (p. 3604)).

Solution → **Controls**

You can also set the **Max Coarse Levels** under **FAS Multigrid Controls** in the **Multigrid** tab in the Advanced Solution Controls Dialog Box (p. 5168).

Changing the number of coarse grid levels in the **Solution Controls** task page will automatically update the number shown in the **Multigrid** tab in the Advanced Solution Controls Dialog Box (p. 5168).

Coarse grid levels are created when you first begin iterating. If you want to check how many cells are in each level, request one iteration and then click **Info** and select **Size** in the **Domain** ribbon tab (**Mesh** group box) (as described in Mesh Size (p. 1250)) to list the size of each grid level. If you are satisfied, you can continue the calculation; if not, you can change the number of coarse grid levels and check again.

For most problems, you will not need to modify any additional multigrid parameters once you have settled on an appropriate number of coarse grid levels. You can simply continue your calculation until convergence.

36.4.7.2. Using Residual Smoothing to Increase the Courant Number

In the density-based explicit formulation, implicit residual smoothing (or averaging) is a technique that can be used to reduce the time step size restriction of the solver, thereby allowing the Courant number to be increased. The implicit smoothing is implemented with an iterative Jacobi method, as described in Implicit Residual Smoothing in the Theory Guide.

Solution → **Controls**

By default, the number of **Iterations** for **Residual Smoothing** is set to zero, indicating that residual smoothing is disabled. If you increase the **Iterations** counter to 1 or more, you can enter the **Smoothing Factor**. A smoothing factor of 0.5 with 2 passes of the Jacobi smoother is usually adequate to allow the Courant number to be doubled.

36.5. Setting Algebraic Multigrid Parameters

As mentioned earlier, in most cases the multigrid solver will not require any special attention from you. If, however, you have convergence difficulties or you want to minimize the overall solution time by using more aggressive settings, you can monitor the multigrid solver and modify the parameters to improve its performance. (The instructions below assume that you have already begun calculations, since there is no need to monitor the solver if you do not fit into one of the two categories above.)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To determine whether your convergence difficulties can be alleviated by modifying the multigrid settings, you will check if the requested residual reduction is obtained on each grid level. To minimize solution time, you will check to see if switching to a more powerful cycle will result in overall reduction of work by the solver.

By default, the flexible cycle is used for all equations except pressure correction, which uses a V cycle. Typically, for a flexible cycle only a few (5–10) relaxations will be performed at the finest level and no coarse levels will be visited. In some cases one or two coarse levels may be visited. If the maximum number of fine level relaxations is not sufficient, you may want to increase the maximum number (as described in Flexible Cycle Parameters (p. 3617)) or switch to a V cycle (as described in Specifying the Multigrid Cycle Type (p. 3613)).

In the pressure-based segregated algorithm, the pressure correction uses a V cycle by default. If the maximum number of cycles (30 by default) is not sufficient, you can switch to a W cycle (using the **Multigrid** tab in the Advanced Solution Controls Dialog Box (p. 5168), as described in Specifying the Multigrid Cycle Type (p. 3613)). Note that efficiency may deteriorate with a W cycle.

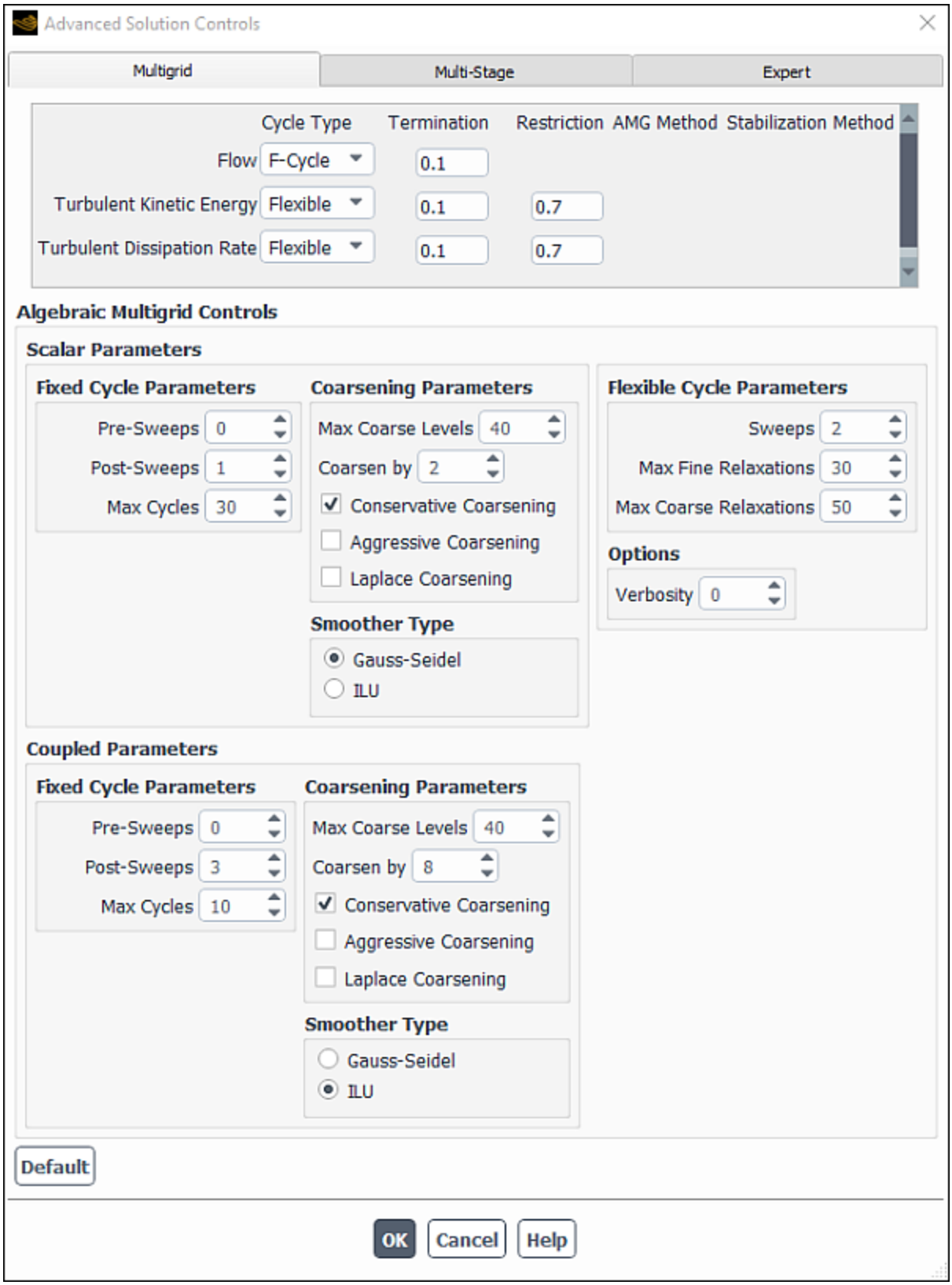
You can try increasing the maximum number of cycles by increasing the value of **Max Cycles** in the **Multigrid** tab, under **Fixed Cycle Parameters**.

In the pressure-based coupled algorithm and the density-based implicit formulation, there is no pressure correction. Instead, there is a flow correction, which by default uses the F cycle. The density-based explicit formulation uses the V cycle as the default flow correction.

Solution → **Controls** → **Advanced...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 36.11: The Multigrid Tab



For additional information, see the following sections:

36.5.1. Specifying the Multigrid Cycle Type

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.5.2. Setting the Termination and Residual Reduction Parameters
36.5.3. Setting the Stabilization Method
36.5.4. Additional Algebraic Multigrid Parameters
36.5.5. Setting FAS Multigrid Parameters

36.5.1. Specifying the Multigrid Cycle Type

By default, the V cycle is used for the pressure equation in the pressure-based segregated algorithm and the flexible cycle is used for all other equations with the exception that the F cycle is used for energy. In the pressure-based coupled algorithm, the F cycle is the default for the coupled flow equations and for the energy equation. All other scalar equations use the flexible cycle by default. In the density-based implicit formulation, the F cycle is default for the flow correction. The V cycle is default for the flow correction in the density-based explicit formulation. For both density-based formulations the flexible cycle is used for the scalar equations. (See Multigrid Cycles in the Theory Guide for a description of these cycles.) To change the cycle type for an equation, you will use the top portion of the **Multigrid** tab in the Advanced Solution Controls Dialog Box (p. 5168) (Figure 36.11: The Multigrid Tab (p. 3612)).

For each equation, you can choose **Flexible**, **V-Cycle**, **W-Cycle**, or **F-Cycle** in the adjacent drop-down list.

36.5.2. Setting the Termination and Residual Reduction Parameters

When you use the flexible cycle for an equation, you can control the multigrid performance by modifying the **Termination** and/or **Restriction** criteria for that equation at the top of the **Multigrid** tab in the Advanced Solution Controls Dialog Box (p. 5168) (Figure 36.11: The Multigrid Tab (p. 3612)).

Solution → **Controls** → **Advanced...**

The **Restriction** criterion is the residual reduction tolerance, β in Equation 23.131 in the Theory Guide. This parameter dictates when a coarser grid level must be visited (due to insufficient improvement in the solution on the current level). With a larger value of β, coarse levels will be visited less often (and vice versa). The **Termination** criterion, α in Equation 23.132 in the Theory Guide, governs when the solver should return to a finer grid level (that is, when the residuals have improved sufficiently on the current level).

For the V, W, or F cycle, the **Termination** criterion determines whether or not another cycle should be performed on the finest (original) level. If the current residual on the finest level does not satisfy Equation 23.132 in the Theory Guide, and the maximum number of cycles has not been performed, Ansys Fluent will perform another multigrid cycle. (The **Restriction** parameter is not used by the V, W, and F cycles.)

36.5.3. Setting the Stabilization Method

The AMG solver [171] (p. 5722) builds coarse levels by grouping fine level cells to make coarse level cells, and uses piecewise constant interpolation.

In the **Multigrid** tab (Figure 36.11: The Multigrid Tab (p. 3612)), you can choose another stabilization method rather than the AMG solver if you are using a fixed-type cycle (F, V, or W cycle); this is available

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

for all equations except the flow correction for the density-based solver. If desired, you can choose the bi-conjugate gradient stabilized method [16] (p. 5714) (**BCGSTAB**) option, the recursive projection method [142] (p. 5721) (**RPM**), or the generalized minimal residual method [134] (p. 5720) (**GMRES**) in order to improve the convergence of the linear solver. BCGSTAB and GMRES can be preconditioned, and all these methods can provide more stability and robustness than the AMG solver.

Ansys Fluent usually builds diagonally dominant matrices for the linear solver. However, this is not always possible. A linear system with highly dominant off-diagonal coefficients may occur during discretization of complex physical models such as multiphase cavitation. Using a stabilization method in such cases can be helpful. In addition, the AMG convergence in parallel can be improved using the **BCGSTAB** or **GMRES** option with AMG. It is recommended that you attempt a solution first with **BCGSTAB** ; the **GMRES** is usually more robust and more likely to converge, but it is more demanding in terms of memory usage and solver time.

If you are using the pressure-based segregated solver and the flow is incompressible, an additional stabilization method for the algebraic multigrid solver will appear in the **Stabilization Method** drop-down list. This is the conjugate gradient method, or **CG** [16] (p. 5714) and will be available only for the pressure equation. This method is typically used in conjunction with an AMG preconditioner (any available AMG method). The CG formulation requires a symmetric system matrix. Such matrices result from the finite volume discretization of steady or transient elliptic operators, such as the pressure correction equation in the incompressible case. The CG method provides a useful alternative to BCGSTAB, RPM, and GMRES, since it reduces the memory requirements and the number of floating point operations, especially when compared to BCGSTAB. In addition, in transient pressure-based segregated solvers, as typically used in LES and/or NITA simulations, the solution of the pressure correction equation constitutes a significant share of the overall computational effort.

Note that when divergence is detected for the AMG solver, the BCGSTAB method and/or the CG method, then an alternate method will be used for an iteration as a fallback, and you will be informed in the console that an equation is being stabilized to enhance linear solver robustness. If you see such console messages repeatedly for flow-related equations, it is an indication that you need to revise the settings in the **Solution Methods** task page and/or the **Algebraic Multigrid Controls** group box of the **Advanced Solution Controls** dialog box.

36.5.4. Additional Algebraic Multigrid Parameters

There are several additional parameters that control the algebraic multigrid solver, but there will usually be no need to modify them. These additional scalar and coupled parameters are all contained in the **Multigrid** tab in the Advanced Solution Controls Dialog Box (p. 5168) (Figure 36.11: The Multigrid Tab (p. 3612)).

Solution → **Controls** → **Advanced...**

Important:

When using the density-based explicit formulation or the pressure-based solver with any of the segregated algorithms, described in Pressure-Velocity Coupling in the Theory Guide and Choosing the Pressure-Velocity Coupling Method (p. 3585), only **Scalar Parameters** are set in the **Multigrid** tab. If you use the density-based implicit or the pressure-based coupled algorithm, described in Coupled Algorithm in the Theory Guide, then you can set the **Coupled Parameters**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.5.4.1. Fixed Cycle Parameters

For the fixed (V, W, and F) multigrid cycles, you can control the number of pre- and post-relaxations (β_1 and β_3 in Multigrid Cycles in the Theory Guide). **Pre-Sweeps** sets the number of relaxations to perform before moving to a coarser level. **Post-Sweeps** sets the number to be performed after coarser level corrections have been applied. Normally, under **Scalar Parameters**, one post-relaxation is performed and no pre-relaxations are done (that is, $\beta_3=1$ and $\beta_1=0$), but in rare cases, you may need to increase the value of β_1 to 1 or 2. Under **Coupled Parameters**, three post-relaxations are performed by default with no pre-relaxations. If you are using the pressure-based coupled solver for a steady simulation with a pseudo time method selected, three post-relaxations are performed under **Scalar Parameters** also.

Important:

- If you are using AMG with V-cycle to solve an energy equation with a solid conduction model presented with anisotropic or very high conductivity coefficient, there is a possibility of divergence with a default post-relaxation sweep of 1. In such cases you should increase the post-relaxation sweep (to say 2) in the AMG section for better convergence when using the pressure-based segregated algorithms.
- It is recommended that you use the Fixed F-cycle for the energy equation when running parallel Ansys Fluent.

36.5.4.2. Coarsening Parameters

For all multigrid cycle types, you can control the maximum number of coarse levels (**Max Coarse Levels** under **Scalar** or **Coupled Parameters**) that will be built by the multigrid solver.

Sets of coarser simultaneous equations are built until the maximum number of levels has been created, or the coarsest level has only 3 equations. Each level has about half as many unknowns as the previous level, so coarsening until there are only a few cells left will require about as much total coarse-level coefficient storage as was required on the fine mesh. Reducing the maximum coarse levels will reduce the memory requirements, but may require more iterations to achieve a converged solution. Setting **Max Coarse Levels** to 0 turns off the algebraic multigrid solver.

Another coarsening parameter you can control is the increase in coarseness on successive levels. The **Coarsen by** parameter specifies the number of fine grid cells that will be grouped together to create a coarse grid cell. The algorithm groups each cell with its strongest neighbor, then groups the cell and its strongest neighbor with the neighbor's strongest neighbor, continuing until the desired coarsening is achieved. Typical values for the scalar parameters are in the range from 2 to 10, with the default value of 2 for the Gauss-Seidel smoother giving the best performance, but also the greatest memory use. For coupled parameters and scalar parameters when using the pressure-based coupled solver with the pseudo method option enabled, the default values of 4 (for 2D) and

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8 (for 3D) for the ILU smoother give the best performance. You should not adjust this parameter unless you need to reduce the memory required to run a problem.

Important:

Depending on the smoother type, Gauss-Seidel or ILU, the **Coarsen by** and **Post-Sweeps** settings should be changed as follows when selecting the non-default smoother type:

ILU

: **Post-Sweeps** =3 and **Coarsen by** = 8

Gauss-Seidel

: **Post-Sweeps** =1 and **Coarsen by** = 2

For the scalar and coupled equations, you have the following AMG coarsening options. Note that these can be used singly or in combination with each other.

- **Conservative Coarsening**

  This option is enabled by default, and improves convergence for difficult problems by tuning multigrid coarsening based on coefficient strengths and, in parallel computations, the partitioning.

- **Aggressive Coarsening**

  This option specifies the use of a version of the AMG solver that is optimized for higher, more aggressive coarsening rates. It is recommended if the AMG solver diverges with the default settings, and is more likely to be beneficial in the following cases:

  – if the corresponding **Coarsen by** field is set to a value greater than 2 (which is the case by default if you are using the density-based implicit or the pressure-based coupled scheme)

  – if you are using higher core counts

- **Laplace Coarsening**

  This option is disabled by default, so that a cell's strongest neighbor is identified based on the magnitude of its interaction with the cell in question. Enabling it specifies that Laplace coefficients are used to evaluate neighbor strength when grouping cells for coarsening. This may improve stability in some cases because the coarser levels will not change as the solution evolves. It may also reduce computation time, particularly at high core counts, because the coarse levels don't need to be recreated every iteration.

36.5.4.3. Smoother Types

Two smoother types are available for scalar and coupled parameters. **Gauss-Seidel** is the simplest smoother type and is recommended when using the pressure-based segregated algorithm. **ILU** is more CPU intensive, but has better smoothing properties for block-coupled systems such as the pressure-based coupled solver and the density-based implicit formulation. The default scalar **Smoother Type** is **Gauss-Seidel**, while the coupled **Smoother Type** is **ILU**. The **ILU** smoother is also used for scalar equations when using the coupled solver with a pseudo time method selected.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For more information about the two smoother types, see The Coupled and Scalar AMG Solvers in the Theory Guide.

36.5.4.4. Flexible Cycle Parameters

To change the maximum number of relaxations, increase or decrease the value of **Max Fine Relaxations** or **Max Coarse Relaxations** in the **Multigrid** tab in the Advanced Solution Controls Dialog Box (p. 5168) (Figure 36.11: The Multigrid Tab (p. 3612)) under **Flexible Cycle Parameters**.

Solution → **Controls** → **Advanced...**

36.5.4.5. Setting the Verbosity

The steps for monitoring the solver are as follows:

1. Set multigrid **Verbosity** to 1 or 2 in the **Multigrid** tab in the Advanced Solution Controls Dialog Box (p. 5168).

   Solution → **Controls** → **Advanced...**

2. Request a single iteration using the Run Calculation Task Page (p. 5207).

   Solution → **Run Calculation**

If you set the verbosity to 2, the information printed in the Ansys Fluent console for each equation will include the following:

- equation name
- equation tolerance (computed by the solver using a normalization of the source vector)
- residual value after each fixed multigrid cycle or fine relaxation for the flexible cycle
- number of equations in each multigrid level, with the zeroth level being the original (finest-level) system of equations

Note that the residual printed at cycle or relaxation 0 is the initial residual before any multigrid cycles are performed.

When verbosity is set to 1, only the equation name, tolerance, and residuals are printed.

A portion of a sample printout is shown below:

```
pressure correction equation:
tol. 1.2668e-05
 0 2.5336e+00
 1 4.9778e-01
 2 2.5863e-01
 3 1.9387e-01
 multigrid levels:
 0 918
 1 426
 2 205
 3  97
 4  45
```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
5  21
6  10
7  4
```

36.5.4.6. Returning to the Default Multigrid Parameters

If you change the multigrid parameters, but you then want to return to Ansys Fluent's default settings, you can click the **Default** button in the **Multigrid** tab. Ansys Fluent will change all settings to the defaults, and the **Default** button will become the **Reset** button. To get your settings back again, you can click the **Reset** button.

36.5.5. Setting FAS Multigrid Parameters

For most calculations, you will not need to modify any FAS multigrid parameters once you have set the number of coarse grid levels. If, however, you encounter convergence difficulties, you may consider the following suggested procedures.

Important:

Recall that FAS multigrid is used only by the density-based explicit formulation.

36.5.5.1. Combating Convergence Trouble

Some problems may approach convergence steadily at first, but then the residuals will level off and the solution will "get stuck." In some cases (for example, long thin ducts), this convergence trouble may be due to multigrid's slow propagation of pressure information through the domain. In such cases, you should turn off multigrid by setting **Multigrid Levels** to 0 in the Solution Controls Task Page (p. 5163).

36.5.5.2. "Industrial-Strength" FAS Multigrid

In some cases, you may find that your problem is converging, but at an extremely slow rate. Such problems can often benefit from a more aggressive form of multigrid, which will speed up the propagation of the solution corrections. For such problems, you can try the "industrial-strength" multigrid settings.

Important:

These settings are very aggressive and assume that the solution information passed through the multigrid levels is somewhat accurate. For this reason, you should only attempt the procedure described here after you have performed enough iterations that the solution is off to a good start. Using "industrial-strength" multigrid too early in the calculation process—when the solution is far from correct—will not help convergence and may cause the calculation to become unstable, as very incorrect values are propagated quickly to the original grid. Note also that while these multigrid settings will usually reduce the total number of iterations required to reach convergence, they will greatly increase the computation time for each multigrid cycle. Thus the solver will be performing fewer but longer iterations.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The strategy employed is as follows:

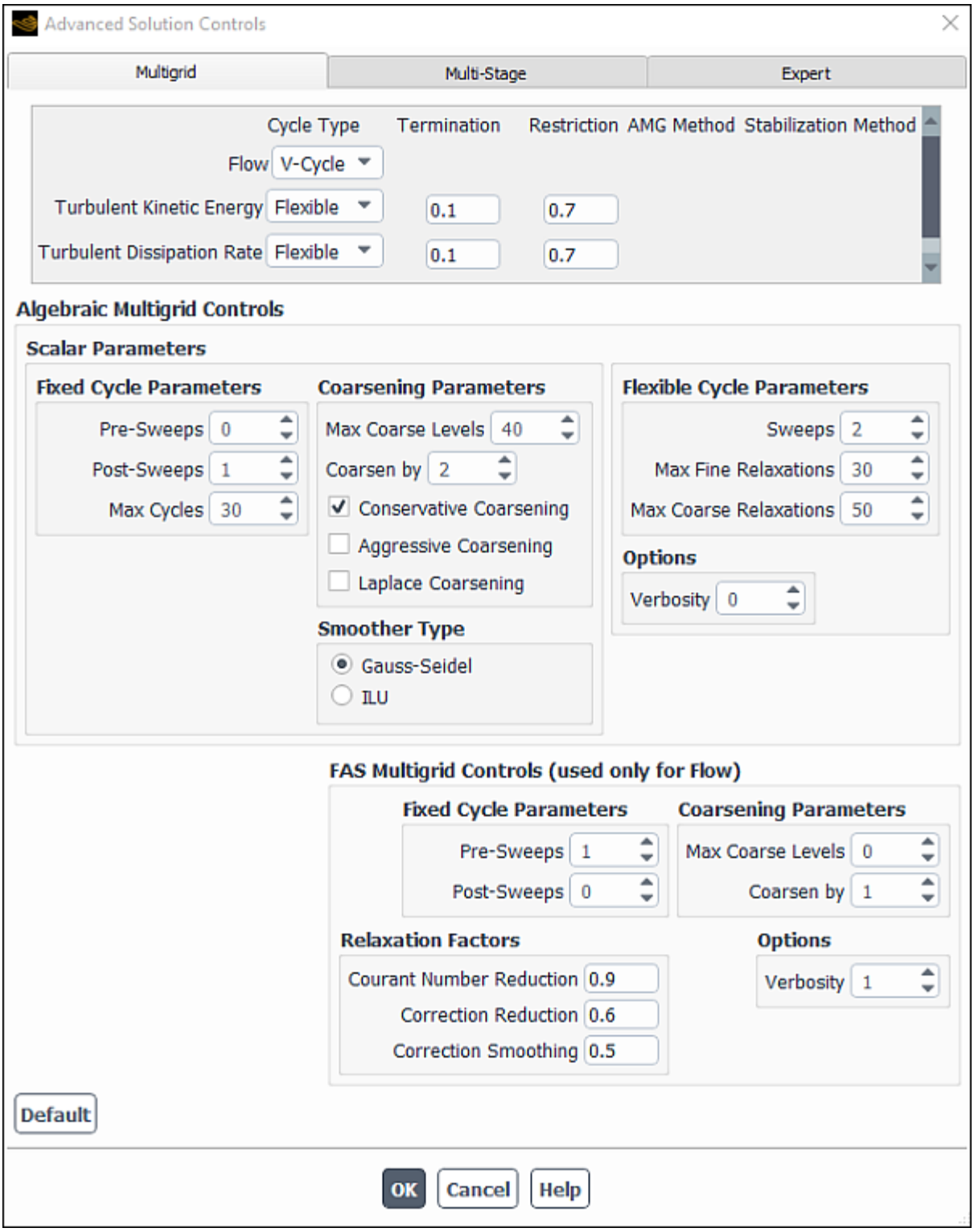
- Increase the number of iterations performed on each grid level before proceeding to a coarser level
- Increase the number of iterations performed on each grid level after returning from a coarser level
- Allow full correction transfer from one level to the next finer level, instead of transferring reduced values of the corrections
- Do not smooth the interpolated corrections when they are transferred from a coarser grid to a finer grid

You can set all of the parameters for this strategy under **FAS Multigrid Controls** in the **Multigrid** tab in the Advanced Solution Controls Dialog Box (p. 5168) (Figure 36.12: The Advanced Solution Controls Dialog Box (p. 3620)) and then continue the calculation.

Solution → **Controls** → **Advanced...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 36.12: The Advanced Solution Controls Dialog Box



Increasing the number of iterations performed on each grid level before proceeding to a coarser level (the value of β_1 described in Multigrid Cycles in the Theory Guide) will improve the solution passed from each finer grid level to the next coarser grid level. Try increasing the value of **Pre-Sweeps** (under **FAS Multigrid Controls**, *not* under **Algebraic Multigrid Controls**) to 10.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Increasing the number of iterations performed on each level after returning from a coarser level will improve the corrections passed from each coarser grid level to the next finer grid level. Errors introduced on the coarser grid levels can therefore be reduced before they are passed further up the grid hierarchy to the original grid. Try increasing the value of **Post-Sweeps** (under **FAS Multigrid Controls**, *not* under **Algebraic Multigrid Controls**) to 10.

By default, the full values of the multigrid corrections are not transferred from a coarser grid to a finer grid; only 60% of the value is transferred. This prevents large errors from transferring quickly up to the original grid and causing the calculation to become unstable. It also prevents a "good" solution from propagating quickly to the original grid. However, by increasing the **Correction Reduction** to 1, you can transfer the full values from coarser to finer grid levels, speeding the propagation of the solution and, usually, the convergence as well. The **Species Correction Reduction** sets the factor by which to reduce the magnitude of the species corrections to stabilize the multigrid calculation. This item appears only when species transport is being modeled.

When the corrections on a coarse grid are passed back to the next finer grid level, the values are, by default, interpolated and then smoothed. Disabling the smoothing so that the actual value in a coarse grid cell is assigned to the fine grid cells that make it up can also aid convergence. To disable smoothing, set the **Correction Smoothing** to 0. Large discontinuities between cells will be smoothed out implicitly as a result of the additional **Post-Sweeps** performed.

The **Courant Number Reduction** sets the factor by which to reduce the Courant number for coarse grid levels (that is, every level except the finest). Some reduction of the time step size (such as the default 0.9) is typically required because the stability limit cannot be determined as precisely on the irregularly shaped coarser grid cells.

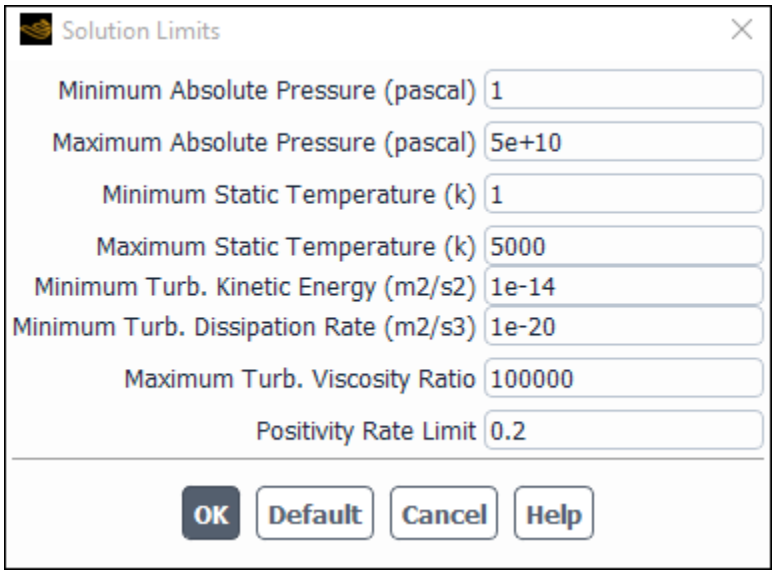
36.6. Setting Solution Limits

In order to keep the solution stable under extreme conditions, Ansys Fluent provides limits that keep the solution within an acceptable range. You can control these limits with the Solution Limits Dialog Box (p. 5166) (Figure 36.13: The Solution Limits Dialog Box (p. 3622)).

Solution → Controls → Limits...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 36.13: The Solution Limits Dialog Box



Fluent applies limiting values for pressure, static temperature, and turbulence quantities. Fluent also applies a minimum volume fraction for the matrix solution in Eulerian multiphase simulations run on a double-precision solver. The purpose of these limits is to keep the absolute pressure or static temperature from becoming zero, negative, or excessively large during the calculation, and to keep the turbulence quantities from becoming excessive. The purpose of the minimum volume fraction is to avoid singularity of the solution matrix when the volume fraction tends to zero.

Typically, you will not need to change the default solution limits. If pressure, temperature, or turbulence quantities are being reset to the limiting value repeatedly (as indicated by the appropriate warning messages in the console), you should check the dimensions, boundary conditions, and properties to be sure that the problem is set up correctly and try to determine why the variable in question is getting so close to zero or so large. You can create a field variable cell register to mark and display cells have a value equal to the limit (refer to Field Variable (p. 3855) for more information). In very rare cases, you may need to change the solution limits, but only do so if you are sure that you understand the reason for the solver's unusual behavior. For example, you may know that the temperature in your domain will exceed 5000 K. Be sure that any temperature-dependent properties are appropriately defined for high temperatures if you increase the maximum temperature limit. Another rare example is when the actual volume fraction in the domain needs to go below a safe limit of 1e-08, such as in nucleation and dispersion of very low amounts of contaminants in a system. By reducing this limit you may encounter instability problems.

Important:

The absolute pressure limit is enforced only for fluid materials that are modeled using the ideal-gas or one of the real-gas-* density methods.

For materials that use the compressible-liquid or a user-defined compressible fluid (liquid or gas) density method, a pressure limit is used to ensure that the density remains positive, but the actual pressure value itself is not limited.

Note that if you are using the density-based solver and the temperature and/or pressure values are approaching the minimum and/or maximum limits, you can enable divergence prevention to apply

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

under-relaxation to the variables in select cells rather than changing the limits. For details, see Preventing Divergence Using Local Under-Relaxation (p. 3608).

For additional information, see the following sections:

36.6.1. Limiting the Values of Solution Variables

36.6.2. Adjusting the Positivity Rate Limit

36.6.3. Resetting Solution Limits

36.6.1. Limiting the Values of Solution Variables

The limiting minimum and maximum values for absolute pressure are shown in the **Minimum** and **Maximum Absolute Pressure** fields. If the Ansys Fluent calculation predicts a value less than the **Minimum Absolute Pressure** or greater than the **Maximum Absolute Pressure**, the corresponding limiting value will be used instead. Similarly, the **Minimum** and **Maximum Temperature** are limiting values for energy calculations.

The **Minimum Turb. Kinetic Energy**, **Minimum Turb. Dissipation Rate**, and the **Maximum Turb. Viscosity Ratio** are limiting values for turbulent calculations. If the calculation predicts a value for k or ε that is less than the appropriate limiting value (that is, **Minimum Turb. Kinetic Energy** or **Minimum Turb. Dissipation Rate**, respectively), then the limiting value will be used instead. For the viscosity ratio limit, Ansys Fluent uses the limiting maximum value of turbulent viscosity ($C_\mu k^2 / \varepsilon$) in the flow field relative to the laminar viscosity. If the ratio calculated by Ansys Fluent exceeds the limiting value, the ratio is set to the limiting value by limiting ε to the necessary value.

For Eulerian and Mixture multiphase simulations run on a double-precision solver using the implicit volume fraction formulation, if the prediction of volume fraction falls below the value specified for **Minimum Vol. Frac. for Matrix Solution**, Ansys Fluent will use the limiting value in the matrix solution for phase-specific equations, depending on the model.

For the multiphase models supporting the explicit volume fraction formulation and for the VOF model, the **Volume Fraction Cutoff** limit specified in the **Multiphase Model** dialog box will serve as a limiting value when solving phase-specific equations, depending on the model.

Note:

The Eulerian multiphase model uses the specified solution limits for solving Momentum, Turbulence, Energy, Species, and UDS equations, whereas the VOF and Mixture multiphase models use the specified limits for solving Species and UDS equations.

36.6.2. Adjusting the Positivity Rate Limit

In Ansys Fluent's density-based solver, the rate of reduction of temperature is controlled by the **Positivity Rate Limit**. The default value of 0.2, for example, means that temperature is not allowed to decrease by more than 20% of its previous value from one iteration to the next. If the temperature change exceeds this limit, the time step in that cell is reduced to bring the change back into range and a "time step reduced" warning is printed. (This reduced time step will be used for the solution of all variables in the cell, not just for temperature.) Rapid reduction of temperature is an indication that the temperature may become negative. Repeated "time step reduced" warnings should alert you

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

that something is wrong in your problem setup. (If the warning messages stop appearing, the calculation may have "recovered" from the time-step reduction.)

Important:

For high-speed flow, if your solution is diverging particularly for the energy equation, then lowering this limit to 0.05 or 0.02 might help in overcoming divergence.

When High-Speed Numerics (as described in Enabling High-Speed Numerics (p. 3608)) is enabled, the positivity rate limit is also applied to the pressure update to ensure that the absolute pressure does not decrease by more than the prescribed positivity rate of reduction. A separate warning message for pressure indicating the number of limited cells is also printed.

36.6.3. Resetting Solution Limits

If you change and save the value of one of the solution limits, but you then want to return to the default limits set by Ansys Fluent, you can reopen the Solution Limits Dialog Box (p. 5166) and click the **Default** button. Ansys Fluent will change the values to the defaults and the **Default** button will become the **Reset** button. To get your values back again, you can click the **Reset** button.

36.7. Setting Multi-Stage Time-Stepping Parameters

The most common parameter you will change to control the convergence of the multi-stage time-stepping scheme is the Courant number. Instructions for modifying the Courant number are presented in Changing the Courant Number (p. 3602). The **Multi-Stage** tab is accessible from the **Advanced Solution Controls** dialog box when using the density-based explicit formulation.

For additional information, see the following section:

36.7.1. Changing the Multi-Stage Scheme

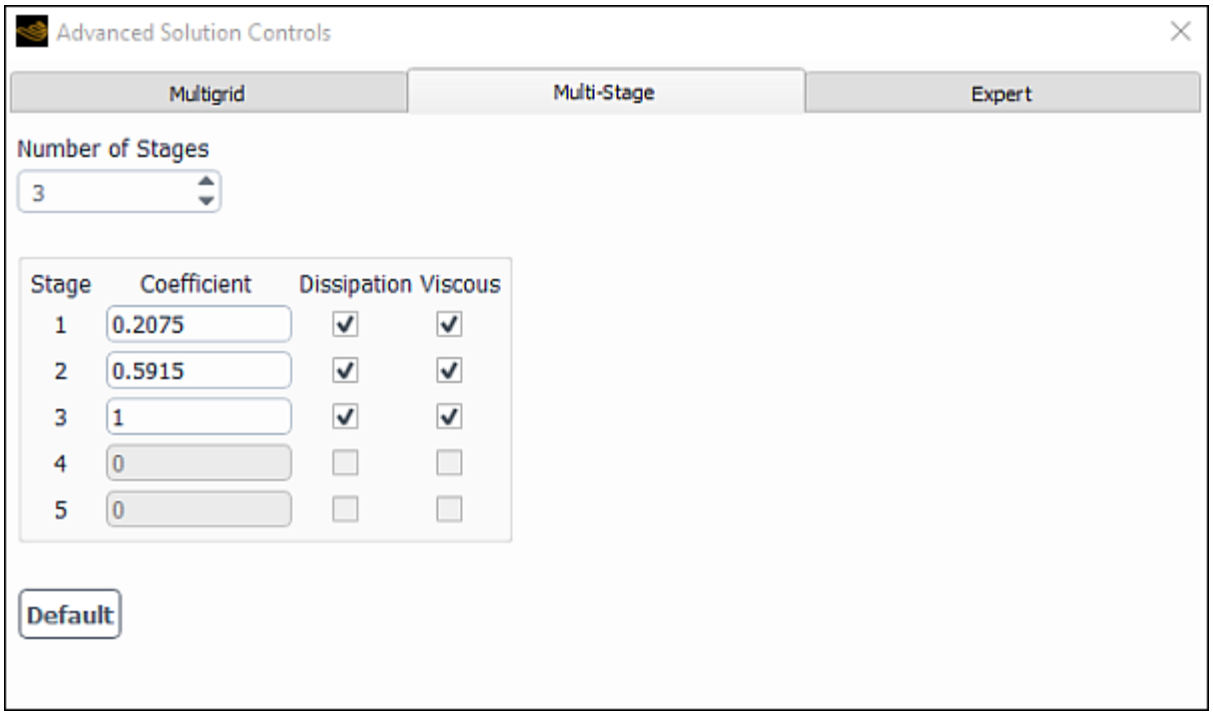
36.7.1. Changing the Multi-Stage Scheme

It is possible to make several changes to the multi-stage time-stepping scheme itself. You can change the number of stages and set a new multi-stage coefficient for each stage. You can also control whether or not dissipation and viscous stresses are updated at each stage. These changes are made in the **Multi-Stage** tab in the Advanced Solution Controls Dialog Box (p. 5168) (Figure 36.14: The Multi-Stage Tab (p. 3625)).

Solution → **Controls** **Advanced...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 36.14: The Multi-Stage Tab



Important:

You should not attempt to make changes to Ansys Fluent's multi-stage scheme unless you are very familiar with multi-stage schemes and are interested in trying a different scheme found in the literature.

36.7.1.1. Changing the Coefficients and Number of Stages

By default, the Ansys Fluent multi-stage scheme uses 3 stages for steady-state solutions with coefficients of 0.2075, 0.5915, and 1.0, and 4 stages for unsteady solutions with coefficients of 0.25, 0.3333, 0.5, and 1.0. You can decrease or increase the number of stages using the arrow buttons for **Number of Stages** in the **Multi-Stage** tab. (If you want to increase the number of stages beyond five, you will need to use the text-interface command `solve/set/multi-stage`.)

For each stage, you can modify the **Coefficient**. Coefficients must be greater than 0 and less than 1. The final stage should always have a coefficient of 1.

36.7.1.2. Controlling Updates to Dissipation and Viscous Stresses

For each stage, you can indicate whether or not artificial dissipation and viscous stresses are evaluated. If a **Dissipation** box is selected for a particular stage, artificial dissipation will be updated on that stage. If not selected, artificial dissipation will remain "frozen" at the value of the previous stage. If a **Viscous** box is selected for a particular stage, viscous stresses will be updated on that stage. If not selected, viscous stresses will remain "frozen" at the value of the previous stage. Viscous stresses should always be computed on the first stage, and successive evaluations will increase the "robustness" of the solution process, but will also increase the expense (that is, increase the CPU

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

time per iteration). For steady problems, the final solution is independent of the stages on which viscous stresses are updated.

36.7.1.3. Resetting the Multi-Stage Parameters

If you change the multi-stage parameters, but you then want to return to the default scheme set by Ansys Fluent, you can click the **Default** button in the **Multi-Stage** tab in the Advanced Solution Controls Dialog Box (p. 5168). Ansys Fluent will change the values to the defaults and the **Default** button will become the **Reset** button. To get your values back again, you can click the **Reset** button.

36.8. Selecting Gradient Limiters

The default gradient limiter in Ansys Fluent is the **Standard** limiter. Each of the limiters is described in detail in Gradient Limiters in the Theory Guide. The gradient limiters are accessible from the **Expert** tab in the **Advanced Solution Controls** dialog box.

Solution → Controls **Advanced...**

You can select **Standard**, **Multidimensional**, or **Differentiable** from the **Spatial Discretization Limiter Type** drop-down list.

Each of these options can also be accessed by typing the following text command:

`solve → set → slope-limiter-menu`

The type of slope limiter changed by selecting the menu item `type` and by choosing one of the following options: `default`, `multi-dimensional`, and `differentiable`. Note that the `default` slope limiter in the TUI is equivalent to the **Standard** option in the GUI.

For each of the gradient limiter methods, Ansys Fluent provides three limiting directions:

- **Cell to Face Limiting** - the limited value of the reconstruction gradient is determined at cell face centers. This is the default method.
- **Cell to Cell Limiting** - the limited value of the reconstruction gradient is determined along a scaled line between two adjacent cell centroids. On an orthogonal mesh (or when cell-to-cell direction is parallel to face area direction) this method becomes equivalent to the default cell to face method. For smooth field variation, cell to cell limiting may provide less numerical dissipation on meshes with skewed cells.
- **Cell to Node Limiting** - the limited value of the reconstruction gradient is determined at the vertices of a cell. This limiting direction is more conservative than the default cell to face strategy, leading to improved monotonicity and robustness when computing strong gradients on unstructured grids. It is particularly advantageous for meshes containing stretched, skewed, or high aspect-ratio cells.

Note:

The **Cell to Node Limiting** direction is not available for overset meshes, or when using the `multi-dimensional` limiter.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ansys Fluent also provides the option to apply a limiter filter to the **Standard** and **Differentiable** limiters. The purpose of the limiter filter is to maintain higher-order accuracy for the main flow variables. It suppresses limiter intervention arising from small numerical noise, while maintaining limiter control when there are actual discontinuities or large gradients in the solution (at shocks, boundary layers, and so on). The limiter filter can also help in improving the apparent convergence of the solution residual.

To use the limiter filter enable **Apply Limiter Filter**. When the limiter filter is turned on, then by default the filter mechanism is applied to the main flow and turbulent variables only. The filter mechanism is based on comparing local cell changes to average global domain changes of a particular flow variable. When the local changes are very small compared to the average global changes, then the limiter intervention will be suppressed.

Note:

The limiter filter is not available when using the **Multidimensional** limiter.

36.9. Initializing the Solution

Before starting your CFD simulation, you must provide Ansys Fluent with an initial "guess" for the solution flow field. In many cases, you must take extra care to provide an initial solution that will allow the desired final solution to be attained. A real-life supersonic wind tunnel, for example, will not "start" if the back pressure is simply lowered to its operating value; the flow will choke at the tunnel throat and will not transition to supersonic. The same holds true for a numerical simulation: the flow must be initialized to a supersonic flow or it will simply choke and remain subsonic.

There are two methods for initializing the solution:

- Initialize the entire flow field (in all cells). Three methods are available:
  - Standard initialization (see Initializing the Entire Flow Field Using Standard Initialization (p. 3628))
  - FMG initialization (see Full Multigrid (FMG) Initialization (p. 3633))
  - Hybrid initialization (see Hybrid Initialization (p. 3635))
- Patch values or functions for selected flow variables in selected cell zones or cell registers.

Important:

- Before patching initial values in selected cells, you must first initialize the entire flow field. You can then patch the new values over the initialized values for selected variables.
- If you are using one of the following in your analyses:
  - user-defined real gas model (UDRGM)
  - user-defined density (via UDF)
  - solution-dependent expressions

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

then before initializing the solution using either the **Hybrid Initialization** method or values from a specific zone selected in the **Compute from** drop-down list, you must first perform standard initialization with no zone selected from the **Compute from** drop-down list.

In the first step, the standard initialization will yield the preliminary solution values, which will be then used in the second step to obtain the desired initial values. The recommended workflow will help resolve potential circular dependencies (such as the dependence of solution initialization values on material properties or expressions and dependence of material properties or expressions on solution values).

For additional information, see the following sections:

36.9.1. Initializing the Entire Flow Field Using Standard Initialization

36.9.2. Patching Values in Selected Cells

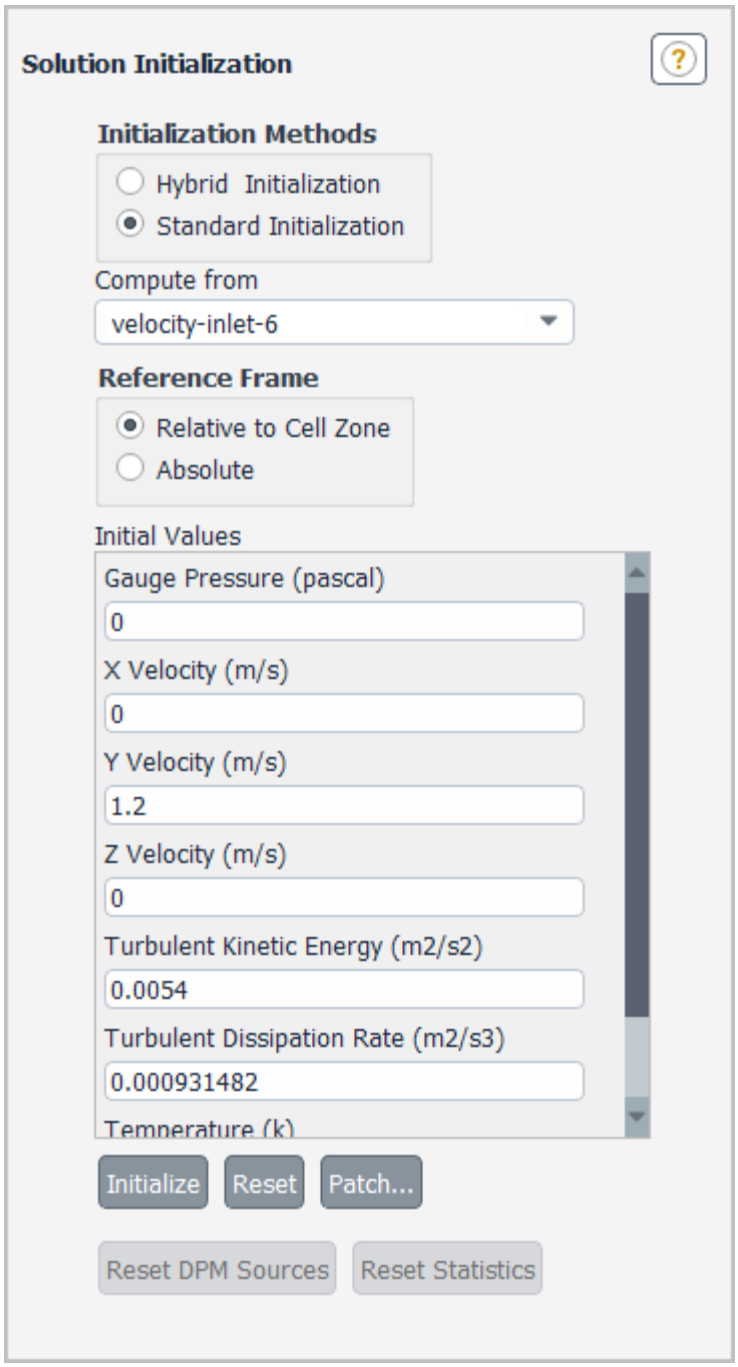
36.9.1. Initializing the Entire Flow Field Using Standard Initialization

Before you start your calculations *or* patch initial values for selected variables in selected cells (Patching Values in Selected Cells (p. 3630)) you must initialize the flow field in the entire domain. The Solution Initialization Task Page (p. 5179) (Figure 36.15: The Solution Initialization Task Page (p. 3629)) allows you to set initial values for the flow variables and initialize the solution using these values.

Solution → **Initialization**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 36.15: The Solution Initialization Task Page



You can compute the values from information in a specified zone, enter them manually, or have the solver compute average values based on all zones. You can also indicate whether the specified values for velocities are absolute or relative to the velocity in each cell zone. The steps for standard initialization are as follows:

1. Select **Standard Initialization** as the **Initialization Method**.

2. Set the initial values:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To initialize the flow field using the values set for a particular zone, select the zone name in the **Compute from** drop-down list. All values under the **Initial Values** heading will automatically be computed and updated based on the conditions defined at the selected zone.
- To initialize the flow field using computed average values, select **all-zones** in the **Compute from** drop-down list. Ansys Fluent will compute and update the **Initial Values** based on the conditions defined at all boundary zones.
- If you want to change one or more of the values, you can enter new values manually in the fields next to the appropriate variables. If you prefer to enter all values manually, you can do so without selecting a zone in the **Compute from** list. For species transport cases with a single phase flow, you can select boundary species to be displayed in the list using the **Select Boundary Species** dialog box (see Figure 19.3: The Select Boundary Species Dialog Box (p. 2367)) that can be accessed by clicking **Species** in the **Solution Initialization** task page.

3. If your problem involves moving reference frames or sliding meshes, indicate whether the initial velocities are absolute velocities or velocities relative to the motion of each cell zone by selecting **Absolute** or **Relative to Cell Zone** under **Reference Frame**. (If no zone motion occurs in the problem, the two options are equivalent.) The default reference frame for velocity initialization in Ansys Fluent is relative. If the solution in most of your domain is rotating, using the relative option may be better than using the absolute option.

4. After you are satisfied with the **Initial Values** displayed in the task page, you can click the **Initialize** button to initialize the flow field. If solution data already exist (that is, if you have already performed some calculations or initialized the solution), you must confirm that it is OK to overwrite those data.

36.9.1.1. Saving and Resetting Initial Values

When you initialize the solution by clicking on **Initialize**, the initial values will also be saved; should you need to reinitialize the solution later, you will find the correct values in the task page when you reopen it.

If you accidentally select the wrong zone from the **Compute from** list or manually set a value incorrectly, you can use the **Reset** button to reset all fields to their "saved" values.

36.9.2. Patching Values in Selected Cells

Once you have initialized (or calculated) the entire flow field, you may patch different values for particular variables into different cells. If you have multiple fluid zones, for example, you may want to patch a different temperature in each one. You can also choose to patch a custom field function (defined using the Custom Field Function Calculator Dialog Box (p. 5428)) instead of a constant value. If you are patching velocities, you can indicate whether the specified values are absolute velocities or velocities relative to the cell zone's velocity. You can also use expressions to patch cell zones and cell registers (refer to Fluent Expressions Language (p. 1037) for additional information on expressions). All patching operations are performed with the Patch Dialog Box (p. 5183) (Figure 36.16: The Patch Dialog Box (p. 3631)).

Solution → Initialization **Patch...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution → Initialization → Patch...

Figure 36.16: The Patch Dialog Box

1. Select the variable to be patched in the **Variable** list.

   Note:

   For species transport cases with a single phase flow, you can select boundary species to be displayed in the **Variable** list using the **Select Boundary Species** dialog box (see Figure 19.3: The Select Boundary Species Dialog Box (p. 2367)) that can be accessed by clicking the **Species** button in the **Solution Initialization** task page.

2. Choose the zone(s) and/or register(s) where you want to patch a value for the selected variable, in the **Zones to Patch** and/or **Registers to Patch** lists.

   Important:

   When a shell zone has been created (that is, by enabling shell conduction for a wall and running the calculation), the name of the shell zone will be listed in the **Zones to Patch** list as **shell :** < *wall-name* >, where < *wall-name* > is the name of the wall in which shell conduction has been enabled. Only temperature can be patched into the cells of a shell, and the same value or field function will be patched into every layer of the shell.

3. If you want to patch a constant value, simply enter that value in the **Value** field. If you want to specify a value that is more complex, you can either use an expression or a field function.

   - To use an expression, select **Expression** from the drop-down list to the right of the **Value** field, and click ( ) to open Figure 5.2: The Expression Editor Dialog Box (p. 1052).
   - To use a field function, enable the **Use Field Function** option and select the appropriate function in the **Field Function** list.

4. If you selected a velocity in the **Variable** list, and your problem involves moving reference frames or sliding meshes, indicate whether the patched velocities are absolute velocities or velocities relative to the motion of each cell zone by selecting **Absolute** or **Relative to Cell Zone** under

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reference Frame. (If no zone motion occurs in the problem, the two options are equivalent.) The default reference frame for velocity patching in Ansys Fluent is relative. If the solution in most of your domain is rotating, using the relative option may be better than using the absolute option.

5. Click the **Patch** button to update the flow-field data. (Note that patching will have no effect on the iteration or time-step count.)

Important:

If you apply a patch when setting up an Ansys Fluent simulation from Ansys Workbench, the patch will not be automatically performed during future automatic solution updates from Workbench. If you need to apply the patch each time the solution is updated from Workbench, you can do so by adding the text command `/solve/patch` (with appropriate arguments) to the **Original Settings** command list in the **Case Modification** tab of the **Automatic Solution Initialization and Case Modification** dialog box. See Automatic Initialization of the Solution and Case Modification (p. 3705) in this manual and Case Modification Strategies with Fluent and Workbench in the Fluent in Workbench User's Guide for more details on automatic case modification.

36.9.2.1. Using Registers

The ability to patch values in cell registers gives you the flexibility to patch different values within a single cell zone. For example, you may want to patch a certain value for temperature only in fluid cells with a particular range of concentrations for one species. You can create a cell register (basically a list of cells) using the cell registers feature (Using Cell Registers (p. 3851)). These functions allow you to mark cells based on physical location, cell volume, gradient or isovalue of a particular variable (using a field variable register), and other parameters. Once you have created a register, you can patch values in it as described above.

36.9.2.2. Using Field Functions

By defining your own field function using the Custom Field Function Calculator Dialog Box (p. 5428), you can patch a non-constant value in selected cells. For example, you may want to patch varying species mass fractions throughout a fluid region. To use this feature, simply create the function as described in Custom Field Functions (p. 4285), and then perform the function-patching operation in the Patch Dialog Box (p. 5183), as described above.

36.9.2.3. Using Patching Later in the Solution Process

Since patching affects only the variables for which you choose to change the value, leaving the rest of the flow field intact, you can use it later in the solution process without losing calculated data. (Initialization, on the other hand, resets all data to the initial values). For example, you might want to start a combustion calculation from a cold-flow solution. You can simply read in (or calculate) the cold-flow data, patch a high temperature in the appropriate cells, and continue the calculation.

Patching can also be useful when you are solving a problem using a step-by-step technique, as described in Step-by-Step Solution Processes (p. 3739).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.10. Full Multigrid (FMG) Initialization

For many complex flow problems such as those found in rotating machinery, or flows in expanding or spiral ducts, flow convergence can be accelerated if a better initial solution is used at the start of the calculation. The Full Multigrid initialization (FMG initialization) can provide this initial and approximate solution at a minimum cost to the overall computational expense.

For more information about FMG initialization, see Overview of FMG Initialization in the *Fluent Theory Guide*.

The limitations that apply to the FMG Initialization can be found in Limitations of FMG Initialization in the *Fluent Theory Guide*.

For additional information, see the following sections:

36.10.1. Steps in Using FMG Initialization

36.10.2. Convergence Strategies for FMG Initialization

36.10.3. Additional FMG Initialization Options with the Density-Based Solver

36.10.1. Steps in Using FMG Initialization

You can access and customize the FMG initialization procedure using the Solution Initialization task page (see Solution Initialization Task Page (p. 5179)) after the standard or hybrid flow initialization has been performed or if valid flow data is available (such as from reading a data file).

Alternatively, FMG initialization can be accessed from the text user interface (TUI). To customize the FMG initialization, use the following command after performing standard or hybrid initialization or reading data:

`solve` → `initialize` → `set-fmg-initialization`

You will be asked to enter:

- The number of multigrid levels for the FMG iteration (the default is 5).

  Important:

  For small cases (100, 000 cells or less), it is recommended that you lower the number of multigrid levels to 3 or 4.

- For each level of multigrid, you will be asked to enter the residual reduction (the default value is `0.001`), and the number of cycles per level (the defaults at each level are `100`, `200`, `400`, `800`, `800`, and `800`). In general, you should perform more iterations on coarse levels than fine levels. Level 0 is the finest level, which represents the original mesh.
- FMG iteration Courant-number (the default is `0.75`). This will be the CFL value that the FAS multigrid will use for the FMG initialization.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Enabling verbose mode (the default is `yes`). By enabling this option, you will be able to monitor the convergence at each level.

  Important:

  If you do not customize the FMG settings, then the default values will be used.

To perform the FMG initialization, type the following command after performing standard or hybrid initialization or reading data:

`solve` → `initialize` → `fmg-initialization`

When you are prompted to `Enable FMG initialization? [no]`, type `yes`.

When verbose mode is selected and the FMG initialization is being used, Ansys Fluent will first output the multigrid level information followed by convergence history for the FAS multigrid cycle on each level. The normalized residual value is printed after ten FAS cycles or when the number of FAS cycles is reached. The output will indicate when convergence is reached on each level and when the solution is being interpolated to the next level.

36.10.2. Convergence Strategies for FMG Initialization

When setting the FMG initialization parameters, you should consider performing more iterations on the coarse levels than on the fine levels. However, remember that the purpose of FMG initialization is to obtain a good initial solution at a low cost. You should try to avoid unreasonable convergence tolerance that will make the FMG initialization expensive.

Turn on the verbose mode to help you determine if the flow is converging as expected during the FMG iterations. If the solution is not converging to the desired tolerance, consider increasing the number of FAS multigrid cycles at each level. If the solution is diverging during the FAS cycles, then consider lowering the FMG iteration Courant number since the default value is probably too aggressive and is likely causing the solution to diverge.

For turbulent flows, it is very important to first perform standard initialization with proper and realistic values of the turbulence variables (for example k and ε). This can be done by computing the average values based on the conditions defined at the inflow boundary or at all boundary zones. Then, you can proceed with FMG initialization. Unrealistic initialization of turbulence variables may cause convergence difficulties during the first few iterations on the fine mesh, thereby nullifying the benefit of FMG initialization.

36.10.3. Additional FMG Initialization Options with the Density-Based Solver

Viscous FMG Initialization

By default, the FMG initialization procedure computes an initial solution based on the Euler equations of inviscid flows as described in Overview of FMG Initialization in the *Fluent Theory Guide*. The density-based solver offers the possibility of adding the viscous terms of the Navier-Stokes equation to the FMG initialization procedure. This allows boundary layers to develop as part of the initialization phase,

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

thus mitigating the solution transients encountered upon startup. The FMG viscous terms are engaged with the following text command:

```
> solve/initialize/set-fmg-options/viscous-terms?
Enable viscous terms during FMG initialization [no] yes
```

Note that although the turbulence model equations are not solved during the initialization, the FMG viscous terms consider the effective viscosity, which includes the contribution of the turbulent viscosity μ_t. Since the turbulent viscosity obtained from the freestream initialization might not be sufficient to develop adequate boundary-layers, a default turbulent viscosity ratio of 100 is used during FMG initialization. This value can be modified with the following text command:

```
> solve/initialize/set-fmg-options/set-turbulent-viscosity-ratio
Set turbulent viscosity ratio used during FMG initialization [100] 100
```

For conjugate heat-transfer cases, the FMG viscous terms also include heat conduction effects, which allow an initial temperature distribution to develop in solid zones.

Note:

The `solve/initialize/set-fmg-options/` text command menu is only available after performing standard or hybrid initialization or reading data.

Species-reaction Initialization

By default, the FMG initialization solves the transport species equations, but neglects volumetric reaction terms. The convergence of certain reactive flows can be improved by developing an initial solution that accounts for the production and destruction of species, and that is in thermo-chemical equilibrium. This is achieved by retaining the species reactions during the FMG initialization with the following text command, which is available after performing standard or hybrid initialization or reading data:

```
> solve/initialize/set-fmg-options/species-reaction?
Enable species volumetric reactions during FMG initialization [no] yes
```

FMG species reactions are currently restricted to the density-based solver. They are also not available with turbulence-chemistry interaction (TCI), Eddy-dissipation-concept (EDC) models, or with the CHEMKIN-CFD chemistry solver.

36.11. Hybrid Initialization

Hybrid initialization is yet another initialization method in Ansys Fluent. The other initialization methods are standard initialization and FMG initialization. Hybrid initialization is a collection of recipes and boundary interpolation methods. It solves Laplace's equation to determine the velocity and pressure fields. All other variables, such as temperature, turbulence, species fractions, and volume fractions, will be automatically patched based on domain averaged values or a particular interpolation recipe.

For more information about hybrid initialization, see Hybrid Initialization in the Theory Guide.

For additional information, see the following sections:

36.11.1. Steps in Using Hybrid Initialization

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.11.2. Solution Strategies for Hybrid Initialization

36.11.1. Steps in Using Hybrid Initialization

The default initialization method for single phase steady-state flows is the **Hybrid Initialization** method.

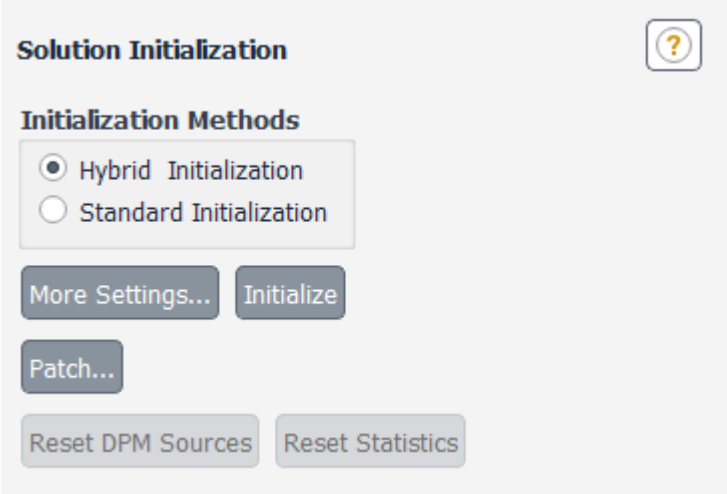
Note:

For other flow types, such as multiphase or unsteady simulations, the default initialization method is the **Standard Initialization** method. However both initialization methods are available for use in all flow conditions and types.

To use **Hybrid Initialization**, go to the Solution Initialization Task Page (p. 5179) (Figure 36.17: The Solution Initialization Task Page for Hybrid Initialization (p. 3636)) where you will select **Hybrid Initialization**.

Solution → **Initialization**

Figure 36.17: The Solution Initialization Task Page for Hybrid Initialization



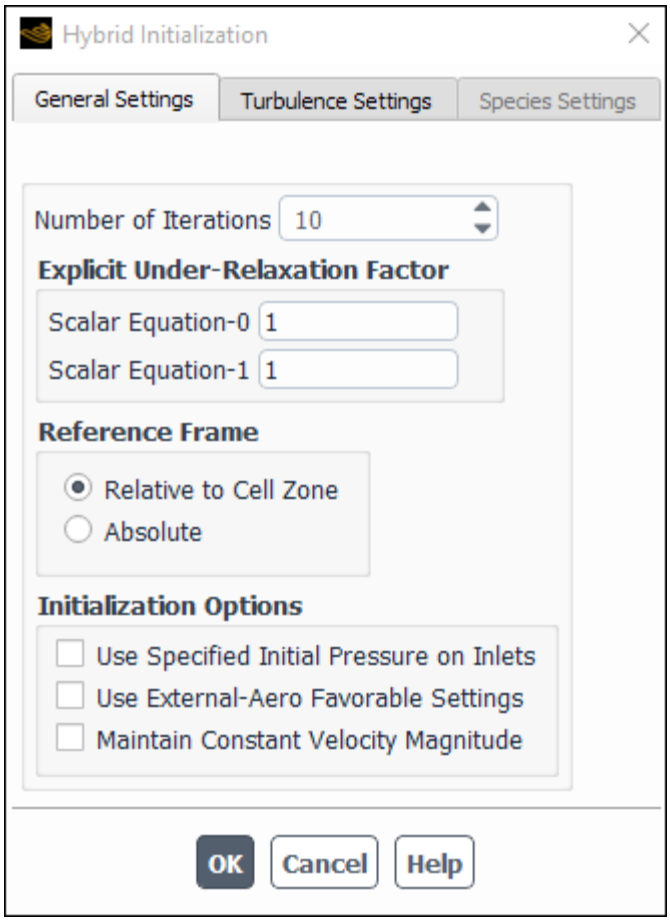
Note:

In most cases, you need not do anything more than click the **Initialize** button. However, should you decide to modify the default settings for the hybrid initialization method, click **More Settings...**.

If you click **More Settings...**, the **Hybrid Initialization** dialog box (Figure 36.18: The Hybrid Initialization Dialog Box (p. 3637)) will open. A host of settings that control the **Hybrid Initialization** strategy will be available for you to adjust.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 36.18: The Hybrid Initialization Dialog Box



You can make adjustments in three different areas:

- **General Settings** tab:
  - **Number of Iterations** uses a default value of 10. This is the number of iterations that will be performed while solving the Laplace equations to initialize the velocity and pressure. In general, you do not need to change the number of iterations. However, for complex and highly curved geometries, if the default number of iterations is not enough to reach the convergence tolerance of 1e-06 and the flow fields are not to your liking, then you may want to increase the number of iterations and re-initialize the flow.
  - **Explicit Under-Relaxation Factor** uses a default value of 1. This value will be used while solving the Laplace equation to initialize the velocity and pressure. In general, you do not need to change the explicit under-relaxation factor. However, for some cases, where the scalar residuals are oscillating and showing difficulty reaching the convergence tolerance of 1e-06, you may want to re-initialize the flow by reducing the under-relaxation factor. You may also want to increase the number of iterations to produce a smooth initialization field for the velocity and pressure.
  - **Reference Frame** is set to **Relative to Cell Zone** by default. If your problem involves moving reference frames or sliding meshes, indicate whether the initial velocities are absolute velocities or velocities relative to the motion of each cell zone by selecting **Absolute** or **Relative to Cell Zone**. If no zone motion occurs in the problem, the two options are equivalent. If

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the solution in most of your domain is rotating, using the relative option may be better than using the absolute option.

– **Initialization Options** allows you to include the following options:

→ **Use Specified Initial Pressure on Inlets** if you want the specified pressure for **Supersonic/Initialization Gauge Pressure** at the inlet boundaries to be used for solving the Laplace equation for the pressure. Otherwise, Ansys Fluent uses a predetermined recipe to determine the initial pressure field, as described in Hybrid Initialization of the Theory Guide.

→ **Use External-Aero Favorable Settings** if you want to have the velocity potential patched with a linear value to help accelerate convergence of **Scalar Equation–0** and to obtain a better guess of the velocity field for external-aero problems, such as flow over wings, airfoils, or automobiles.

→ **Maintain Constant Velocity Magnitude** if you want to use the flow direction obtained from solving the velocity potential (**Scalar Equation–0**), while maintaining a constant velocity magnitude throughout the computational domain. This option is helpful in some incompressible external flow problems, porous media problems, or if there are narrow channels where large undesirable velocities can be reached.

- **Turbulence Settings** tab uses by default the domain averaged values for the turbulence parameters. If you want to use variable turbulence parameters you can deselect the **Average Turbulent Parameters** check box. When this option is disabled, then it calculates the turbulent parameters, such as kinetic energy and dissipation energy, using local flow parameters.

- **Species Settings** tab will by default initialize secondary species with zero mass or mole fractions. If you want to specify the appropriate value for the species, you will need to enable **Specify Species Parameters**. Note that only the boundary species that you have selected in the **Select Boundary Species** dialog box appear in the list (see Figure 19.3: The Select Boundary Species Dialog Box (p. 2367)). You can add or remove a species using the **Select Boundary Species** that can be accessed by clicking **Species** in the **Solution Initialization** task page.

36.11.2. Solution Strategies for Hybrid Initialization

In general, you do not need to make any extra adjustments to the hybrid initialization default settings. However, if the hybrid initialization is not producing the initial field to your liking, then you can play with the various options available in the **Hybrid Initialization** dialog box (described in Steps in Using Hybrid Initialization (p. 3636)), or you can also use the patching option in addition to the **Hybrid Initialization**. For example, if you are solving a user-defined scalar, then hybrid initialization will initialize them with a value of zero. However, you can specify the value with which you want to patch.

Note:

You can also use a user-defined function to initialize the flow or certain flow variable in conjunction with hybrid initialization.

Standard Initialization is the recommended initialization method for porous media simulations. The default **Hybrid initialization** method does not account for the porous media properties, and depending on boundary conditions, may produce an unrealistic

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

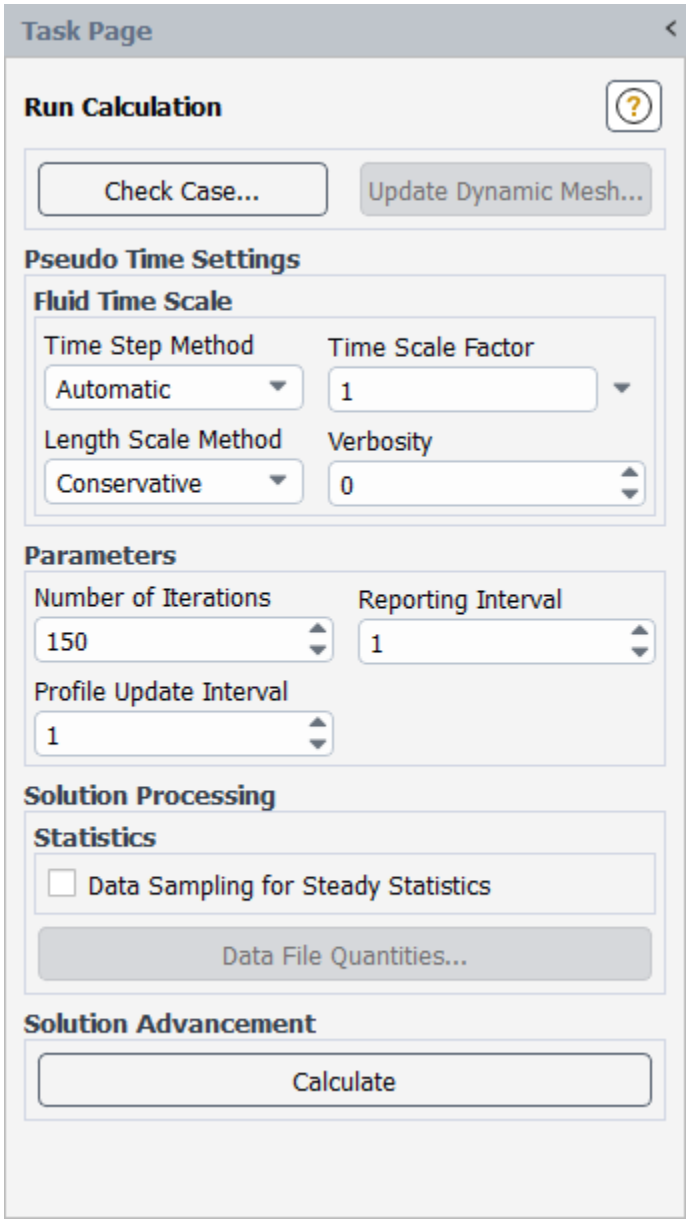
initial velocity field. For porous media simulations, the **Hybrid initialization** method can only be used with the **Maintain Constant Velocity Magnitude** option.

36.12. Performing Steady-State Calculations

For steady-state calculations, you will request the start of the solution process using the Run Calculation Task Page (p. 5207) (Figure 36.19: The Run Calculation Task Page (p. 3639)).

Solution → Run Calculation

Figure 36.19: The Run Calculation Task Page



If **Global Time Step** is selected from the **Pseudo Time Method** drop-down list in the **Solution Methods** task page with the pressure-based coupled solver or the density-based implicit solver, you must begin

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

by defining the settings in the **Pseudo Time Settings** group box, as described in Global Time Step Method Settings (p. 3676). Then for all cases, you must supply the number of additional iterations to be performed by defining the **Number of Iterations** field in the **Parameters** group box. (For unsteady calculation inputs, see Inputs for Time-Dependent Problems (p. 3645)). If no calculations have been performed yet, Ansys Fluent will begin calculations starting at iteration 1, using the initial solution. If you are starting from current solution data, Ansys Fluent will begin at the last iteration performed, using the current solution data as its starting point.

By default, Ansys Fluent will update the convergence monitors (described in Monitoring Solution Convergence (p. 3684)) after each iteration. If you increase the **Reporting Interval** from the default of 1 you can get reports less frequently. For example, if you set the **Reporting Interval** to 2, the monitors will print or plot reports at every other iteration. Note that the **Reporting Interval** also specifies how often Ansys Fluent should check if the solution is converged. For example, if your solution converges after 40 iterations, but your **Reporting Interval** is set to 50, Ansys Fluent will continue the calculation for an extra 10 iterations before checking for (and finding) convergence.

Important:

The **Reporting Interval** only affects convergence checking of residuals, not of report definitions.

When you click the **Calculate** button, Ansys Fluent will begin to calculate. While the calculation is in progress, a progress bar will appear at the bottom of the Fluent window. For steady-state simulations, clicking the **Stop** button next to the progress bar will interrupt the calculation at the earliest safe stopping point after the current iteration. Alternatively, you can type **Ctrl**+**c** in the console. Transient simulations have additional options for stopping, as described in Inputs for Time-Dependent Problems (p. 3645).

For additional information, see the following sections:

36.12.1. Updating UDF Profiles and Named Expressions

36.12.2. Resetting Data

36.12.3. Data Sampling for Steady Statistics

36.12.1. Updating UDF Profiles and Named Expressions

If you have used a user-defined function (UDF) or a named expression (Fluent Expressions Language (p. 1037)) to define any boundary conditions, you can control the frequency with which the function is updated by modifying the value of the **Profile Update Interval**. If **Profile Update Interval** is set to n, the function will be updated after every n iterations.

By default, the **Profile Update Interval** is set to 1. You might want to increase this value if your profile computation is expensive. See the Fluent Customization Manual for details about creating and using UDFs.

36.12.2. Resetting Data

After you have performed some iterations, if you decide to start over again from the first iteration (for example, after making some changes to the problem setup), you can reinitialize the solution using the Solution Initialization Task Page (p. 5179), as described in Initializing the Entire Flow Field Using Standard Initialization (p. 3628).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.12.3. Data Sampling for Steady Statistics

While notionally steady, some simulations tend to exhibit oscillatory or quasi-transient behavior. While in principle, these simulations could be solved as unsteady flow problems and actually resolve the transient behavior, this is often not practical and sometimes unnecessary. **Data Sampling for Steady Statistics** is useful under such circumstances and helps generate averaged results and trends that are better behaved and useful in flow predictions.

If you enable the **Data Sampling for Steady Statistics** option in the Run Calculation Task Page (p. 5207) (see Performing Steady-State Calculations (p. 3639) for details), Ansys Fluent will compute the iteration average (mean) of the instantaneous values, the root-mean-square, and the root-mean-square-errors of those quantities and custom field functions that are enabled/selected in the Sampling Options Dialog Box (p. 5232) and/or the Zone-Specific Sampling Options Dialog Box (p. 5233).

The **Sampled Iterations** field displays the iterations over which data has been sampled for the postprocessing of the mean, RMS, and RMSE values.

Important:

Statistics collected for the magnitude of a vector quantity (such as *mean-wall-shear-stress*) using the **Sampling Options** dialog box are computed differently than those specified using the **Zone-Specific Sampling Options** dialog box. The former are computed as the magnitude of the mean, whereas the latter are computed as mean of the magnitude. This can lead to differences in values between the two statistics collection methods.

When you are reviewing the available variables under the **Steady Statistics...** drop-down lists during postprocessing of graphics and plot objects (contour, vector, XY plot, and so on), statistics collected based on selections in the **Zone-Specific Sampling Options** dialog box contain the `-dataset` extension.

Sampling Options Dialog Box Method

Use the following steps to have Ansys Fluent gather data for steady statistics while calculating using the **Sampling Options** dialog box:

1. (Optional) Create a custom field function for each of the variables of interest for steady statistics postprocessing (for example, P* | V |) using the Custom Field Function Calculator Dialog Box (p. 5428). *Note that you do not need to create custom field functions for the flow shear stresses, flow heat fluxes, wall statistics, or discrete phase variables, as these variables are available for selection later.*

   User Defined → **Field Functions** → **Custom...**

2. Enable **Data Sampling for Steady Statistics** in the **Run Calculation** task page.

   Solution → **Run Calculation** → ☑ **Data Sampling for Steady Statistics**

   Enabling this option allows you to display and report the mean and the root-mean-square-error (RMSE) values.

   The mean, root-mean-square (RMS), and root-mean-square-error (RMSE) values for solution variables will be available in the **Steady Statistics...** category of the variable selection drop-down list that

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

appears in postprocessing dialog boxes. For example, in the **Contours** dialog box, you could select **Steady Statistics...** and **RMSE-uns-custom-function-0** for the **Contours of** drop-down lists to display the root-mean-square-errors of a custom field function named **uns-custom-function-0**.

3. Specify the **Sampling Interval**.

   The **Sampled Iterations** field displays the iterations over which the data has been sampled for the postprocessing of the mean and RMSE values.

4. Select the variables for statistics sampling through the Sampling Options Dialog Box (p. 5232).

   Solution → **Run Calculation** → **Sampling Options...**

5. Initialize the flow statistics.

   Solution → **Initialization** → **Reset Statistics**

   *Note that you can also reset the flow statistics after you have some data for the steady statistics. For example, if you perform 10 iterations with **Data Sampling for Steady Statistics** enabled, check the results, and continue with the calculation for another 10 iterations, the steady statistics will include the data gathered in the first 10 iterations, unless you reinitialize the flow statistics.*

Zone-Specific Sampling Options Dialog Box Method

Use the following steps to have Ansys Fluent gather data for steady statistics while calculating using the **Zone-Specific Sampling Options** dialog box:

1. Enable **Data Sampling for Steady Statistics** in the **Run Calculation** task page.

   Solution → **Run Calculation** → ☑ **Data Sampling for Steady Statistics**

   Enabling this option allows you to display and report the minimum, maximum, mean, root-mean-square-error (RMSE), and moving average values.

   Assuming all options are enabled, the minimum, maximum, mean, root-mean-square-error (RMSE), and moving average values for solution variables will be available in the **Steady Statistics...** category of the variable selection drop-down list that appears in postprocessing dialog boxes. For example, in the **Contours** dialog box, you could select **Steady Statistics...** and **Mean-dynamic-pressure-dataset** for the **Contours of** drop-down lists to display the average value of the dynamic pressure.

2. Specify the **Sampling Interval**.

   The **Sampled Iterations** field displays the iterations over which the data has been sampled for the postprocessing of the selected values.

3. Select the variables and desired statistics (minimum, maximum, root-mean-square-error, moving average) for statistics sampling through the Sampling Options Dialog Box (p. 5232).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution → **Run Calculation** → **Sampling Options (Zone Selection)...**

Note:

Statistics are not gathered or computed on shell surfaces.

4. Initialize the flow statistics.

   Solution → **Initialization** → **Reset Statistics**

   Note that you can also reset the flow statistics after you have some data for the steady statistics. For example, if you perform 10 iterations with ***Data Sampling for Steady Statistics*** *enabled, check the results, and continue with the calculation for another 10 iterations, the steady statistics will include the data gathered in the first 10 iterations, unless you reinitialize the flow statistics.*

36.13. Performing Time-Dependent Calculations

Ansys Fluent can solve the conservation equations in a time-dependent manner, to simulate a wide variety of time-dependent phenomena, such as

- vortex shedding and other time-periodic phenomena
- compressible filling and emptying problems
- transient heat conduction
- transient chemical mixing and reactions

Figure 36.20: Time-Dependent Calculation of Vortex Shedding (t=36.6 sec) (p. 3644) and Figure 36.21: Time-Dependent Calculation of Vortex Shedding (t=41.6 sec) (p. 3644) illustrate the time-dependent vortex shedding flow pattern in the wake of a cylinder.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 36.20: Time-Dependent Calculation of Vortex Shedding (t=36.6 sec)

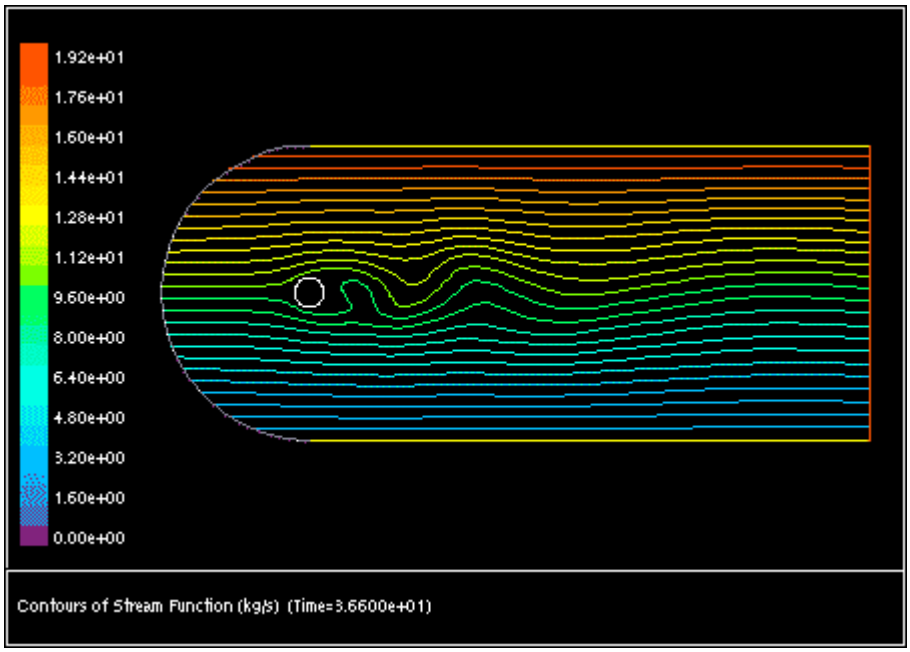
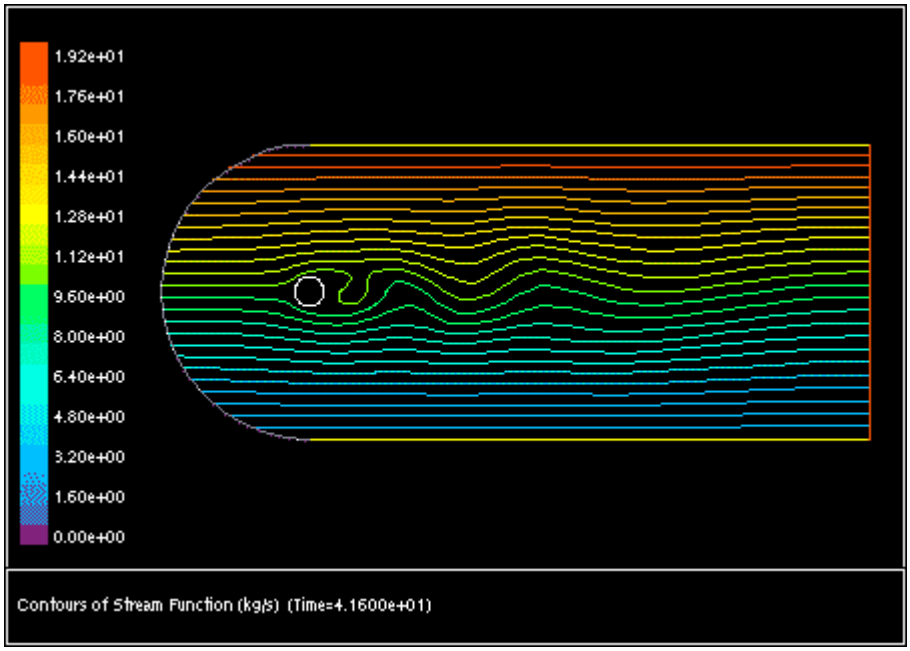


Figure 36.21: Time-Dependent Calculation of Vortex Shedding (t=41.6 sec)



Enabling time dependence is sometimes useful when attempting to solve steady-state problems that tend toward instability (for example, natural convection problems in which the Rayleigh number is close to the transition region). It is possible in many cases to reach a steady-state solution by integrating the time-dependent equations.

For details about temporal discretization, see Temporal Discretization in the Theory Guide.

For additional information, see the following sections:

36.13.1. Inputs for Time-Dependent Problems

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.13.2. CFL-Based Time Stepping
36.13.3. Error-Based Time Stepping
36.13.4. Multiphase-Specific Time Stepping
36.13.5. Postprocessing for Time-Dependent Problems
36.13.6. Runtime Discrete Fourier Transformation

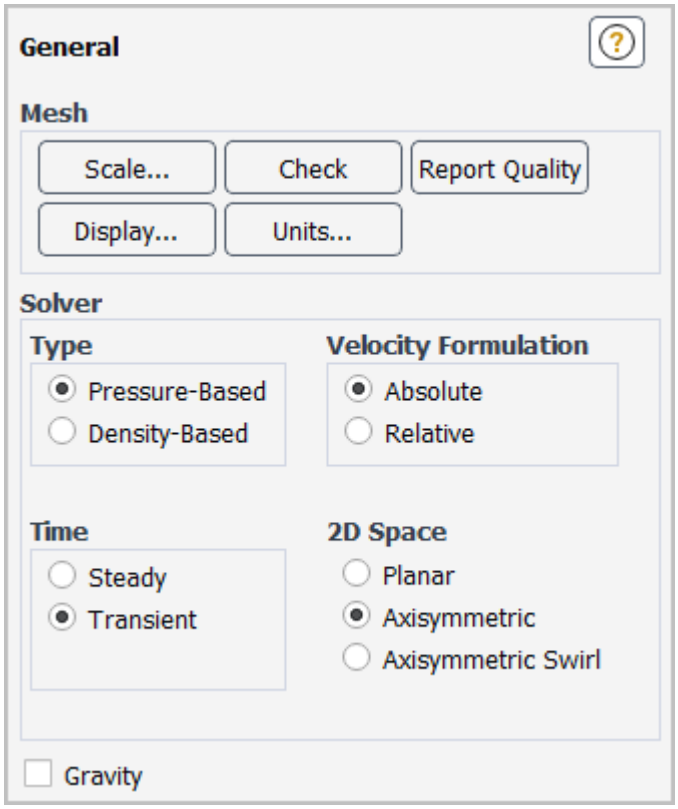
36.13.1. Inputs for Time-Dependent Problems

To solve a transient problem, you will follow the procedure outlined below:

1. Enable the **Transient** option in the **General** task page (Figure 36.22: The General Task Page for a Transient Calculation (p. 3645)).

   Setup → General

   Figure 36.22: The General Task Page for a Transient Calculation

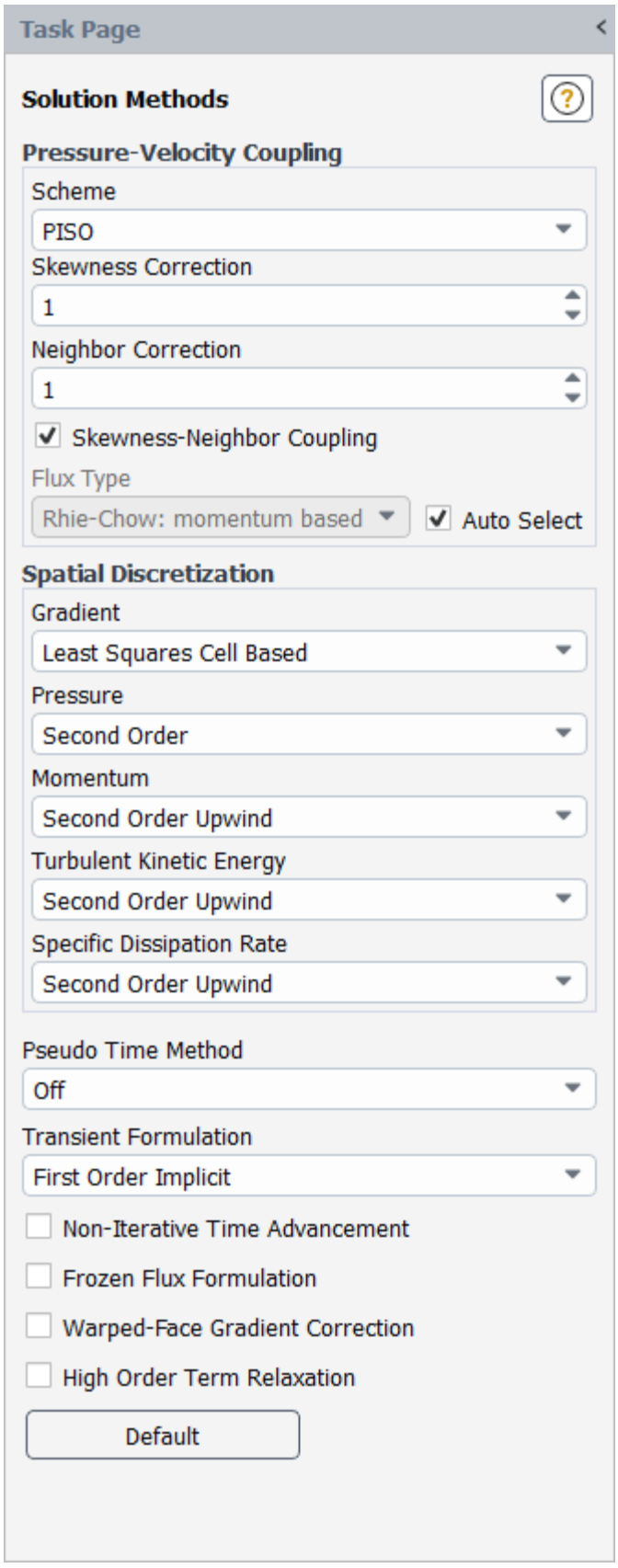
2. Define all relevant models and boundary conditions. Note that any boundary conditions specified using user-defined functions can be made to vary in time. See the Fluent Customization Manual for details.

3. Specify the desired parameters in the **Solution Methods** task page (Figure 36.23: The Solution Methods Task Page for a Transient Calculation (p. 3646)).

   Solution → Methods

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 36.23: The Solution Methods Task Page for a Transient Calculation



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. If you are using the pressure-based solver, select **PISO** from the **Scheme** drop-down list in the **Pressure-Velocity Coupling** group box. To increase the speed of the calculations, you may need to modify the parameters related to the PISO scheme from their default values. See PISO (p. 3586) for more information about the optimal use of the PISO algorithm.

Important:

- If you are using the LES turbulence model with a small time step size, the PISO scheme may be too computationally expensive. It is therefore recommended that you select **SIMPLE** or **SIMPLEC** instead of **PISO**.
- It is best to select the **Coupled** pressure-velocity coupling scheme if you are using a large time step size to solve your transient flow, or if you have a poor quality mesh.

---

b. Specify the desired **Transient Formulation**.

The **First Order Implicit** formulation is sufficient for most problems. If you need improved accuracy, you can use either **Second Order Implicit** or **Bounded Second Order Implicit**. The **Bounded Second Order Implicit** formulation would provide better stability, since time discretization would ensure global bounds for certain variables, if available. However, note that **Bounded Second Order Implicit** formulation cannot ensure fully bounded solution for all variables and may result in overshoots or undershoots, especially at larger time step sizes. For cases that involve the species transport model, the population balance model, or the energy equation, you can use the below text commands to allow the first order time formulation for energy, species transport, and/or population balance equation, while retaining the second order time formulation for all other equations:

```
solve/set/second-order-time-options
For transient discretization, use variable rather than fixed time-step size formulation?  [no]
Allow first order time formulation for energy, species and population balance? [no] yes
```

Important:

- Note that while the **Bounded Second Order Implicit** formulation provides the same accuracy as the **Second Order Implicit** formulation, it actually provides better stability.
- The **Bounded Second Order Implicit** formulation is available only for the pressure-based solver, and not for the density-based solver.
- When using **Second Order Implicit** or **Bounded Second Order Implicit** formulation for a dynamic mesh, do not run a series of simulations in which you vary the time step size. Doing so creates an error that reduces with a reduction of the time step jump.

---

c. (density-based solver only) You can use the **Explicit** formulation to capture the transient behavior of moving waves, such as shocks. For details, see Temporal Discretization in the Theory Guide.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

d. (pressure-based solver only) You have the option of using the following as part of your time-dependent flow calculations:

- **Non-Iterative Time Advancement** (see Time-Advancement Algorithm in the *Fluent Theory Guide*)
- **Frozen Flux Formulation** (see Steady-State Iterative Algorithm in the *Fluent Theory Guide*)

  This option is only available for single-phase transient problems that do not use a moving/deforming mesh model.
- **Pseudo Time Method** (see Performing Calculations with a Pseudo Time Method (p. 3673))

  For transient cases, this drop-down list is only available when you have selected a segregated **Scheme** for the **Pressure-Velocity Coupling** (**SIMPLE**, **SIMPLEC**, or **PISO**).

4. For transient cases that use the density-based explicit formulation, you can use the following text command to easily switch to a two-stage Runge-Kutta scheme for the time integration. This scheme results in faster calculations than the default three- or four-stage Runge-Kutta scheme (especially for laminar flows), though the reduced stability range may require changes to the setup as described below.

```
> solve/set/fast-transient-settings/rk2

Enable two-stage Runge-Kutta scheme? [no]
yes
```

   This text command will set the **Number of Stages** to 2 in the **Multi-Stage** tab of the Advanced Solution Controls Dialog Box (p. 5168), with coefficients of 0.5 and 1. For further details about these settings, see Setting Multi-Stage Time-Stepping Parameters (p. 3624). If your solution diverges with this scheme, you can attempt the calculation again with a lower Courant number (typically less than 1) and, for implicit time stepping, a higher value for the maximum number of iterations per time step; if necessary, you can use the `rk2` text command again to easily disable the two-stage Runge-Kutta scheme and revert back to the default multi-stage scheme for your time-stepping scheme (summarized in Summary of the Density-Based Solver in the *Fluent Theory Guide*).

5. (optional) If you are using the explicit transient formulation with specified Courant number *or* if you are using any of the adaptive time stepping methods (described in a later step) it is recommended that you enable the printing of the current time (for the explicit transient formulation) or the current time step size (for the adaptive time stepping method) at each iteration, using Figure 36.39: Report File for 'flow-time', 'delta-time', and 'iters-per-timestep' (p. 3696).

   Solution → **Monitors** → **Statistic** **Edit...**

   Make sure that the desired item is selected from the **Statistics** selection list (**time** for the current time or **delta_time** for the current time step size) and enable the **Print** option. When Ansys Fluent prints the residuals to the console at each iteration, it will include a column with the current time or the current time step size.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. (optional) Use the Drag Report Definition Dialog Box (p. 5446), the Lift Report Definition Dialog Box (p. 5503), the Moment Report Definition Dialog Box (p. 5541), or the Surface Report Definition Dialog Box (p. 5626) to monitor (and/or save to a file) time-varying force coefficient values or a report of a field variable or function on a surface as it changes with time. See Monitoring Solution Convergence (p. 3684) for details.

7. Set the initial conditions (at time $t=0$) using the **Solution Initialization** task page.

   Solution → **Initialization**

   You can also read in a steady-state data file to set the initial conditions.

   File → **Read** → **Data...**

8. Use the **Autosave** dialog box to specify the file name and frequency with which case and data files should be saved during the solution process. To open the **Autosave** dialog box, click the **Edit...** button next to **Autosave Every** in the **Calculation Activities** task page.

   Solution → **Calculation Activities** → **Autosave** **On**

   See Automatic Saving of Case and Data Files (p. 958) for details about automatic file saving.

   The **Calculation Activities** task page also allows you to export solution and particle history data during the transient calculation. See Exporting Data During a Transient Calculation (p. 1001) for details.

   If you want to create a graphical animation of the solution over time, you can use Figure 36.54: The Animation Definition Dialog Box (p. 3715) to set up the graphical displays that you want to use in the animation. See Animating the Solution (p. 3714) for details.

   You may also want to request automatic execution of other commands using the Execute Commands Dialog Box (p. 5201). See Executing Commands During the Calculation (p. 3700) for details.

9. Specify time-dependent solution parameters and start the calculation as described below for the implicit and explicit transient formulations:

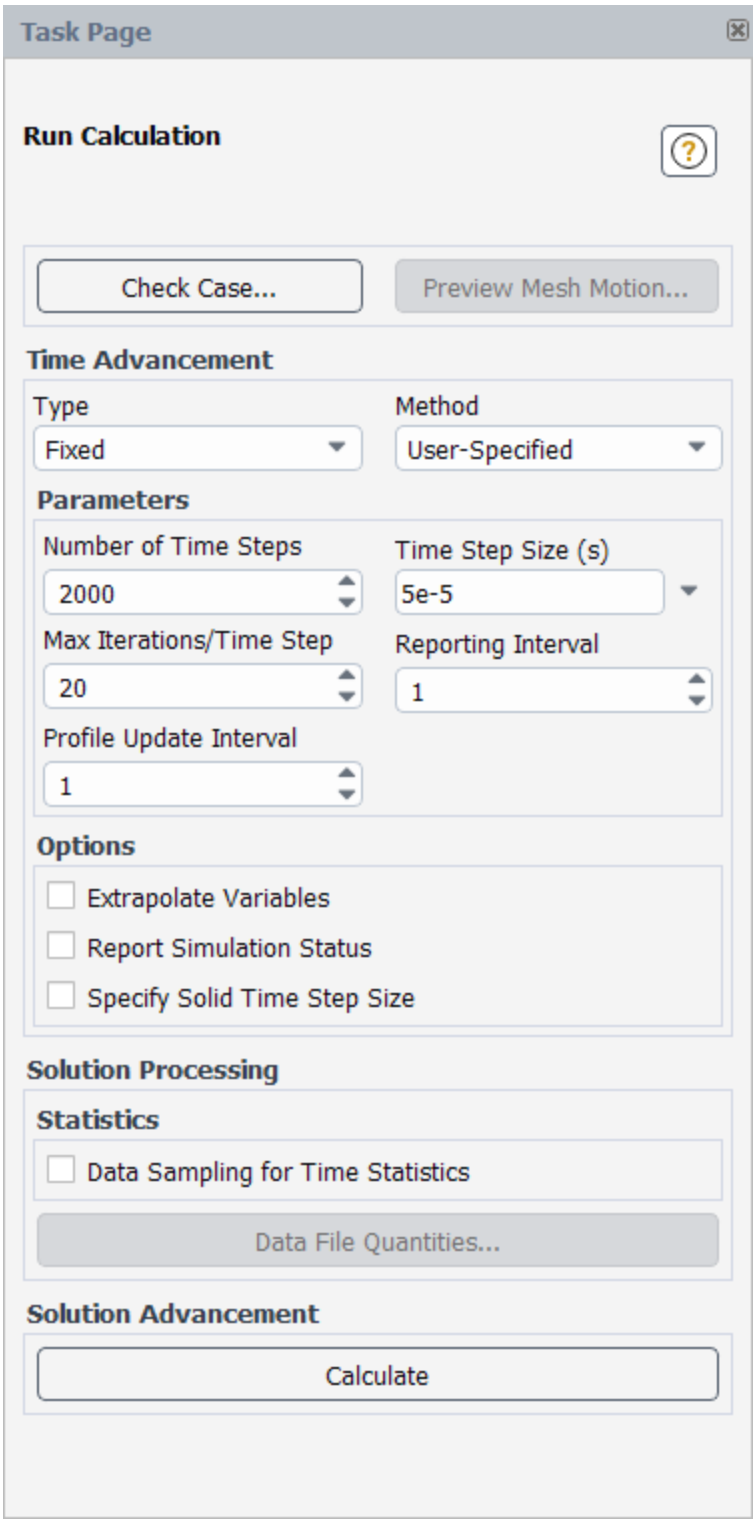
   - If you have chosen the **First Order Implicit**, **Second Order Implicit**, or **Bounded Second Order Implicit** formulation, the procedure is as follows:

     a. Open the Run Calculation Task Page (p. 5207) (see Figure 36.24: The Run Calculation Task Page for Implicit Transient Calculations (p. 3650)).

        Solution → **Run Calculation**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 36.24: The Run Calculation Task Page for Implicit Transient Calculations



b. If you want the time step size to remain fixed throughout the simulation, select **Fixed** from the **Type** drop-down list, and then select one of the following from the **Method** drop-down list:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– **User-Specified** : with this method, you will specify the desired **Number of Time Steps** and the **Time Step Size**.

– **Period-Based** or **Frequency-Based** : these methods are convenient when setting up unsteady turbomachinery cases or any case that exhibits transient periodic behavior. They allow you to define the setup in terms of the **Period** or **Frequency**, the **Time Steps per Period**, and the **Total Periods**, and then Fluent will automatically set an appropriate **Time Step Size** and **Number of Time Steps**.

When Ansys Fluent solves the time-dependent equations using the implicit formulation, multiple iterations may be necessary at each time step, so you can specify a value for the **Max Iterations/Time Step** you would like to allow. If the convergence criteria are met before this number of iterations is performed, the solution will advance to the next time step.

The time step size is the magnitude of Δt. Since the Ansys Fluent formulation is fully implicit, there is no stability criterion that must be met in determining Δt. However, to model transient phenomena properly, it is necessary to set Δt at least one order of magnitude smaller than the smallest time constant in the system being modeled. A good way to judge the choice of Δt is to observe the number of iterations Ansys Fluent needs to converge at each time step. The ideal number of iterations per time step is 5–10. If Ansys Fluent needs substantially more, the time step size is too large. If Ansys Fluent needs only a few iterations per time step, Δt should be increased. Frequently a time-dependent problem has a very fast "startup" transient that decays rapidly. Therefore, it is often wise to choose a conservatively small Δt for the first 5–10 time steps. Δt may then be gradually increased as the calculation proceeds.

For time-periodic calculations, you should choose the time step size based on the time scale of the periodicity. For a rotor/stator model, for example, you might want 20 time steps between each blade passing. For vortex shedding, you might want 20 steps per period.

To verify that your choice for Δt was proper after the calculation is complete, you can plot contours of the Courant number within the domain. To do so, select **Velocity...** and **Cell Convective Courant Number** from the **Contours of** drop-down lists in the **Contours** dialog box. For a stable, efficient calculation, the Courant number should not exceed a value of 20–40 in most sensitive transient regions of the domain.

Note:

The **Period-Based** and **Frequency-Based** methods are not available for in-cylinder dynamic mesh simulations; instead, the **User-Specified** method must be used, and the time step size will be computed based on the crank shaft speed and the crank angle step size.

---

c. If you want Ansys Fluent to modify the size of the time step as the calculation proceeds, select **Adaptive** from the **Type** drop-down list, and then select one of the following from the **Method** drop-down list:

– **CFL-Based**: with this method, Ansys Fluent will modify the size of the time step as the calculation proceeds such that a version of the Courant–Friedrichs–Lewy (CFL) condition is satisfied. See CFL-Based Time Stepping (p. 3660) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– **Error-Based**: with this method, the time step size is modified by Ansys Fluent based on the specified truncation **Error Tolerance**. See Error-Based Time Stepping (p. 3663) for details.

– **Multiphase-Specific**: with this method, the time step size is modified by Ansys Fluent based on the **Global Courant Number**. This method is available for all multiphase models using the implicit or explicit volume fraction formulation, except for the wet steam model. See Multiphase-Specific Time Stepping (p. 3665) for details. While this method and the **CFL-Based** approach both use a Courant number, they differ in that the **Multiphase-Specific** method considers the interface regions when computing the time step size, whereas the **CFL-Based** method is based on a global time marching approach.

Note:

The **Adaptive** type is not available for simulations that use the acoustic model and/or the structural model. It is also unavailable for in-cylinder dynamic mesh simulations; in this case, the fixed **User-Specified** method must be used, and the time step size will be computed based on the crank shaft speed and the crank angle step size.

Next, make a selection from the **Duration Specification Method** drop-down list in order to specify the way in which you will define the calculation duration. The duration can be defined by the total time, the total number of time steps, the incremental time, or the number of incremental time steps. In this context, "total" indicates that Fluent will consider the amount of time / steps that have already been solved and stop appropriately, whereas "incremental" indicates that the solution will proceed for a specified amount of time / steps regardless of what has previously been calculated.

It is a good idea to perform a few fixed-size time steps before switching to the adaptive time stepping. Sometimes spurious discretization errors can be associated with an impulsive start in time. These errors are dissipated during the first few time steps, but they can adversely affect the adaptive time stepping and result in extremely small time steps at the beginning of the calculation. You can specify the **Number of Fixed Time Steps** that should be performed before the size of the time step starts to change. The size of the fixed time step is the value specified for **Initial Time Step Size** ; this value must fall between the minimum and maximum time step sizes. For a better startup, it should be chosen such that the Courant number initially remains close to 1 or your specified value (whichever is lower). Note that if you initialize the solution for a previously run case with adaptive time stepping, then Ansys Fluent will use the time step size saved in the case.

Important:

When the solution tends to exhibit incomplete convergence for the **Error-Based** method, rather than increasing the time step size or keeping the same time step size in the next step, Ansys Fluent reduces the time step size for the next time step (making sure that the time step size does not go below the specified minimum time step size).

When Ansys Fluent solves the time-dependent equations using the implicit formulation, multiple iterations may be necessary at each time step, so you can specify a value for the **Max Iterations/Time Step** you would like to allow. If the convergence criteria are met before

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

this number of iterations is performed, the solution will advance to the next time step. You can also define the **Time Step Size Update Interval**, which specifies the number of time steps that will pass before the time step size is updated for adaptive time stepping methods.

Finally, click the **Settings...** button and complete the setup for the adaptive time stepping method in the Adaptive Time Stepping Dialog Box (p. 5220), setting the minimum / maximum time step sizes and change factors.

Tip:

If you need your simulation to end exactly at your specified total or incremental time, input smaller and smaller values for the minimum time step size until this condition is met.

Note:

For simulations that include automatic adaption (as described in Refining and Coarsening (p. 3777)), the changes to the mesh during the solution may necessitate that the adaptive time step size is recalculated. For **CFL-Based** and **Multiphase-Specific** methods, this recalculated time step size is used automatically (after the completion of any initial fixed time steps). For the **Error-Based** method or for any adaptive method in a case with a dynamic mesh and/or sliding mesh, the recalculated time step size will not be used but will be noted in the console as a warning; if there is a large discrepancy with the time step size that is used, you should consider revising the setup to ensure time accuracy. Note that the time step size is not recalculated if the **Extrapolate Variables** option is enabled in the **Run Calculation** task page and/or if the case combines an overset mesh with the Volume of Fluid (VOF) multiphase model.

---

d. If you want to define your own adaptive time stepping method rather than use one of the **Adaptive** types described previously, you can create a user-defined function for your method using the `DEFINE_DELTAT` macro. After you have compiled or interpreted it, you can select **User-Defined Function** from the **Type** drop-down list and then select it from the **User-Defined Time Step Size** drop-down list. You can then define the duration method and time / number of time steps, as described previously for the **Adaptive** types.

   See the Fluent Customization Manual for details about creating and using user-defined functions.

e. (optional) You can improve the convergence of the transient calculations by enabling the **Extrapolate Variables** option in the Run Calculation Task Page (p. 5207). This option instructs Ansys Fluent to predict the solution variable values for the next time step using a Taylor series expansion, and then inputs that predicted value as an initial guess for the inner iterations of the current time step. As a result, the absolute residual levels are lowered.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that the **Extrapolate Variables** option is not available if you are employing either the NITA scheme with the pressure-based solver or the explicit formulation with the density-based solver.

Important:

If you use the **Extrapolate Variables** option when modeling an incompressible flow with the density-based solver, it is recommended that you disable the extrapolation of pressure values. After you have enabled the **Extrapolate Variables** option, type the following text command in the console:

```
> solve/set/transient-controls/extrapolate-eqn-vars/pressure
Extrapolate Pressure? [yes]
no
```

f. To view details about the time advancement of the simulation, enable the **Report Simulation Status** option in the **Options** group box. While this option is enabled (before, after, or during a calculation), the Simulation Status Dialog Box (p. 5221) will be open and reporting information.

g. (optional) If you want Ansys Fluent to gather data for time statistics (that is, time-averaged, root-mean-square, and root-mean-square-error values for solution variables) during the calculation, you have two options for specifying the statistics that are collected during the simulation:

   – Use the Zone-Specific Sampling Options Dialog Box (p. 5233).

   – Use the Sampling Options Dialog Box (p. 5232).

Important:

Statistics collected for the magnitude of a vector quantity using the **Sampling Options** dialog box are computed differently than those specified using the **Zone-Specific Sampling Options** dialog box. The former are computed as the magnitude of the mean, whereas the latter are computed as mean of the magnitude. This can lead to differences in values between the two statistics collection methods.

When you are reviewing the available variables under the **Unsteady Statistics...** drop-down lists during postprocessing of graphics and plot objects (contour, vector, XY plot, and so on), statistics collected based on selections in the **Zone-Specific Sampling Options** dialog box contain the `-dataset` extension.

Unsteady Statistics using the Zone-Specific Sampling Options Dialog Box

i. Enable the **Data Sampling for Time Statistics** option in the Run Calculation Task Page (p. 5207).

Solution → **Run Calculation** → ☑ **Data Sampling for Time Statistics**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Enabling this option will allow you to display and report the minimum, maximum, mean, root-mean-square-error (RMSE), moving average, and Runtime Discrete Fourier Transformation (p. 3671) values, as described in Postprocessing for Time-Dependent Problems (p. 3669).

Enter a value for **Sampling Interval**, which is how often data samples are accumulated for computation of mean and RMS values. For example, a sampling interval of 5 means that only every 5th result is added to the statistical sampling.

The **Sampled Time** displays the time period over which data has been sampled for the postprocessing of the minimum, maximum, mean, root-mean-square-error (RMSE), and moving average values. For most quantities, as long as the time step size has been constant, dividing this by the time step size yields the number of data sets that have been collected. However, special consideration is required in the case of discrete phase quantities. As noted in Reporting of Unsteady DPM Statistics (p. 2871), sampling of DPM quantities occurs only when parcels pass through a cell, not necessarily at each specified sampling interval. Therefore, the number of samples of the DPM quantities taken during the sampled time period is stored in the separate postprocessing variable, **Accum DPM Parcels in Cell**. If the time step size is varied, every contribution of data sets sampled is automatically weighted by the current time step size.

Note:

– Input for **Sampling Interval** is not applied to the runtime DFT. Accumulation of the Fourier amplitudes is done after each timestep.

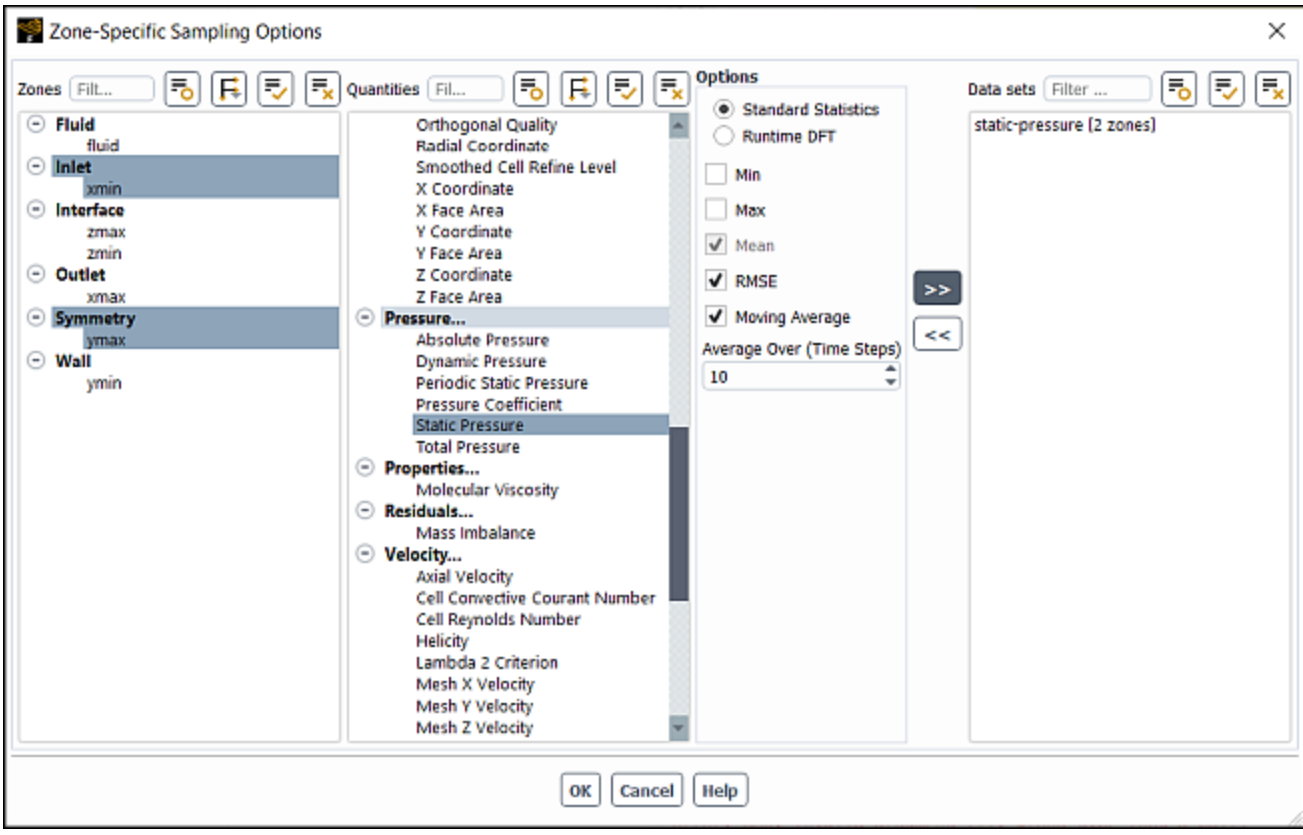
– Accumulation duration for the standard statistics options (minimum, maximum, mean, RMSE, moving average) may continue arbitrarily long until you stop the calculation. Differing from the standard statistics, accumulation duration for the DFT Fourier amplitudes is connected to the selected resolution of a spectrum and is fixed. Therefore, the displayed Sampling Time value may not be relevant to the runtime DFT. To estimate the DFT accumulation progress, see the value **Samples to Complete** for a particular DFT dataset in the Zone-Specific Sampling Options dialog box.

To select the quantities and locations where you want statistics collected, click the **Sampling Options (Zone Selection)...** button and make selections from the **Zone-Specific Sampling Options** dialog box that opens (Figure 36.25: The Zone-Specific Sampling Options Dialog Box for Standard Statistics (p. 3656)) and for Runtime DFT (Figure 36.28: Zone-Specific Sampling Options Dialog Box for Runtime DFT - Frequency Band (p. 3671) or Figure 36.29: Zone-Specific Sampling Options Dialog Box for Runtime DFT - Single Tone (p. 3672)).

Solution → **Run Calculation** → **Sampling Options (Zone Selection)...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 36.25: The Zone-Specific Sampling Options Dialog Box for Standard Statistics



Important:

– Note that gathering data for time statistics is not meaningful inside a moving cell zone (for example, a sliding zone in a sliding mesh problem, a moving zone in a dynamic mesh problem).

– Statistics are not gathered or computed on shell surfaces.

ii. Initialize the flow statistics.

Solution → Initialization → Reset Statistics

Note that you can also reset the flow statistics after you have gathered some data for time statistics. If you perform, say, 10 time steps with the **Data Sampling for Time Statistics** option enabled, check the results, and then continue the calculation for 10 more time steps, the time statistics will include the data gathered in the first 10 time steps unless you reinitialize the flow statistics.

Unsteady Statistics using the Sampling Options Dialog Box

i. Create a custom field function for the each of the variables for which you want to post-process unsteady statistics (for example, P*|V|), using the **Custom Field Function Calculator**. For detailed instructions, see Creating a Custom Field Function (p. 4286). Note that

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

you do not need to take the extra step of creating custom field functions for the flow shear stresses, flow heat fluxes, wall statistics, or discrete phase variables as there are options for selecting these variables directly in a later step.

Parameters & Customization → Custom Field Functions New...

Important:

The maximum number of custom field functions that can be calculated and postprocessed for unsteady statistics is 50.

ii. Enable the **Data Sampling for Time Statistics** option in the Run Calculation Task Page (p. 5207).

Solution → Run Calculation → Data Sampling for Time Statistics

Enabling this option will allow you to display and report the mean, the root-mean-square (RMS), and the root-mean-square-error (RMSE) values, as described in Postprocessing for Time-Dependent Problems (p. 3669).

To specify the sampling interval, enter a value for **Sampling Interval**.

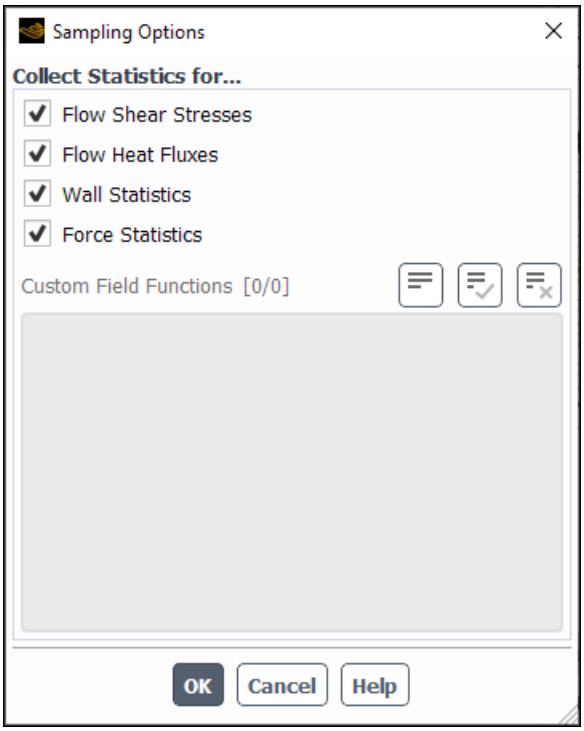
The **Sampled Time** displays the time period over which data has been sampled for the postprocessing of the mean, RMS, and RMSE values. For most quantities, as long as the time step size has been constant, dividing this by the time step size yields the number of data sets that have been collected. However, special consideration is required in the case of discrete phase quantities. As noted in Reporting of Unsteady DPM Statistics (p. 2871), sampling of DPM quantities occurs only when parcels pass through a cell, not necessarily at each specified sampling interval. Therefore, the number of samples of the DPM quantities taken during the sampled time period is stored in the separate postprocessing variable, **Accum DPM Parcels in Cell**. If the time step size is varied, every contribution of data sets sampled is automatically weighted by the current time step size.

To select the variables (which may be represented as custom field functions) for which you want to collect statistics, click the **Sampling Options...** button and make selections from the **Sampling Options** dialog box that opens (Figure 36.26: The Sampling Options Dialog Box (p. 3658)). Note that no custom field functions are selected by default.

Solution → Run Calculation → Sampling Options...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 36.26: The Sampling Options Dialog Box



Important:

Note that gathering data for time statistics is not meaningful inside a moving cell zone (for example, a sliding zone in a sliding mesh problem, a moving zone in a dynamic mesh problem).

iii. Initialize the flow statistics.

Solution → **Initialization** **Reset Statistics**

Note that you can also reset the flow statistics after you have gathered some data for time statistics. If you perform, say, 10 time steps with the **Data Sampling for Time Statistics** option enabled, check the results, and then continue the calculation for 10 more time steps, the time statistics will include the data gathered in the first 10 time steps unless you reinitialize the flow statistics.

h. Click the **Calculate** button. As it calculates a solution, Ansys Fluent will print the current time at the end of each time step.

- If you choose the **Explicit** transient formulation,you have two input options available for specifying the transient solution advancement: you can specify the transient advancement either directly by specifying a time step size ($\triangle t$) or indirectly via entering a Courant number value (CFL) in the **Solution Controls** task page. The steps for these two options are as follows:

  – If the default option **Specified Time Step** is enabled in the **Solution Controls** task page, you will be required to specify the time advancement settings in the **Run Calculation** task page:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

you need to define the time stepping as either a **Fixed** or **User-Defined Function** type (as described previously for the implicit formulations) and click the **Calculate** button.

Note:

The stability of an explicit formulation is limited by the Courant-Fredrichs-Lewy condition. Fluent checks the specified time step size against the stability limit. The stability limit is the maximum allowable time step size calculated based on CFL=1 in Equation 36.2 (p. 3659). The stability limit is also the minimum of all local time steps in the domain. If the specified time step size is larger than the maximum allowable time step, Fluent displays a warning during the calculation.

– If you disable the **Specified Time Step** option, you will be required to specify the **Courant Number** in the **Solution Controls** task page. Then you can define the **Number of Time Steps** in the **Run Calculation** task page and click **Calculate**. In this option the (CFL) value is used to calculate the time step size ($\triangle t$) using Equation 36.2 (p. 3659) :

$$\triangle t = \frac{2 * CFL * V}{\sum_f \lambda_f^{\max} * A_f} \tag{36.2}$$

where, V is the volume of the cell, A_f is the face area, and $\lambda_f^{\max}$ is the maximum local eigenvalue defined in Equation 23.84 in the *Fluent Theory Guide*.

When you click the **Calculate** button, Ansys Fluent will begin to calculate. While the calculation is in progress, a progress bar will appear at the bottom of the Fluent window. For transient simulations, clicking the buttons next to the progress bar will interrupt the calculation at the earliest safe stopping point: you can choose to stop the calculation at the end of either the current iteration or current time step. Alternatively, you can type **Ctrl**+**c** in the console: a single instance will stop the calculation at the end of the current time step, whereas typing it twice will stop it at the end of the current iteration.

You can access the information saved in a legacy data file (that is, a `.dat` file), which includes a standard set of quantities that were computed during the calculation, by clicking the **Data File Quantities...** button. More information about this feature is available in Setting Data File Quantities (p. 1026).

10. Save the final data file (and case file, if you have modified it) so that you can continue the transient calculation later, if desired.

File → **Write** → **Data...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.13.1.1. Additional Inputs

The procedures for setting the reporting interval, updating user-defined function profiles and Named Expressions, interrupting iterations, and resetting data are the same as those for steady-state calculations. See Performing Steady-State Calculations (p. 3639) for details.

Important:

If you are using a user-defined function and/or named expression in your time-dependent calculation, they will be updated after every n iterations, where n is the value of the **Profile Update Interval** (specified in the Run Calculation Task Page (p. 5207)).

36.13.2. CFL-Based Time Stepping

As mentioned in Inputs for Time-Dependent Problems (p. 3645), it is possible to have the size of the time step change as the calculation proceeds such that a version of the Courant–Friedrichs–Lewy (CFL) condition is satisfied. Note that the time step size will only vary if the solution of the flow equation is enabled, and it is a convection-dominant flow. This section provides brief descriptions of the algorithms that Ansys Fluent can use to compute the time step size, as well as an explanation of parameters that you can set to control the CFL-based time stepping.

Important:

CFL-based time stepping is available only with the pressure-based and density-based implicit formulations; it cannot be used with the density-based explicit formulation. Also, it cannot be used with multiphase flows, the in-cylinder dynamic mesh option, the acoustic model, and/or the structural model, nor is it available with the second-order implicit formulation if the fixed time step formulation is selected through the `solve/set/second-order-time-options` text command or through the enabling of a dynamic mesh.

Note:

If there are other time scales controlling the physics of problem (surface tension time scale, meshing-driven time scales for mesh deformation, and so on), then the CFL-based time stepping is not a good choice.

36.13.2.1. The CFL-Based Time Stepping Algorithm

The time step size is proportional to the defined Courant number, and can be computed considering the contribution of various time scales, such as acoustic, convective, and/or viscous time scales. By default, only the convective velocity is considered, which is suitable for low speed and incompressible flows. The time step size is calculated for each cell in the domain, and by default the minimum value is used (provided that it is within the minimum and maximum values that you have defined as part of the settings).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.13.2.2. Specifying Parameters for CFL-Based Time Stepping

For transient flows, when you select **Adaptive** for the **Type** and **CFL-Based** for the **Method** in the **Run Calculation** task page, you can then set the time stepping parameters as described in Inputs for Time-Dependent Problems (p. 3645). The following provides further details that are specific for the CFL-based time stepping.

In the **Run Calculation** task page:

Courant Number

is CFL in Equation 36.6 (p. 3662), which represents the ratio of the time step size to the characteristic time taken for the fluid to cross a control volume. For convection-dominant and/or wave propagation problems, it is recommended that you set this close to 1 in order to obtain more accurate results. Increasing it will result in an increase of the time step size and consequently a faster transient simulation; however, this may have a negative impact on the convergence, stability, and accuracy of your transient simulation.

Initial Time Step Size

is used for the first time step and then for as many subsequent steps as specified in the **Number of Fixed Time Steps** field. This value must fall between the minimum and maximum time step sizes. It should be chosen such that the **Courant Number** initially remains close to 1 or your specified value (whichever is lower) for a better startup. Note that if you initialize the solution for a previously run case with adaptive time stepping, then Ansys Fluent will use the time step size saved in the case file.

In the **Adaptive Time Stepping** dialog box (which is opened by clicking **Settings...** in the **Run Calculation** task page):

Minimum / Maximum Step Change Factor

limit the degree to which the time step size can change at each time step. Limiting the change results in a smoother calculation of the time step size, especially when high-frequency noise is present in the solution. The time step size change factor is computed as the ratio between the current time step and the previous time step.

When choosing the values, it is generally not recommended to allow large variation in time step size due to stability considerations. This is especially true when the time step size is ramping up. It is recommended that you use a minimum time step size factor in the range of 0.5 to 0.75, and a maximum time step size change factor in the range of 1.2 to 2. The default values of 0.5 and 2, respectively, provide a smooth transition of the time step size.

If necessary, you can use the `solve/set/transient-controls/cfl-based-time-stepping-advanced-options` text command to define the following advanced settings:

- You can specify whether to use the minimum CFL condition (that is, use the minimum time step calculated for all the cells in the domain) or the averaged CFL condition (in which the time step is the average value calculated for all the cells in the domain). By default the minimum CFL condition is used.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- You can select the CFL type, that is, the version of the Courant–Friedrichs–Lewy (CFL) condition that is satisfied. You have the following options, and for each one a local time step size is calculated for a cell based on the Courant number (CFL) that you have defined as part of the settings.

  – `[1]Convective`

    For this type, the time step size calculation is solely based on the convective velocity, according to the following formulation. This type may be more appropriate with low speed and incompressible flows, and is selected by default.

$$CFL=\frac{\Delta t\sum_{f}^{N_f}|\dot{m}_f|}{2\rho\Omega} \tag{36.3}$$

  – `[2]Convective-Diffusive`

    For this type, the time step size is calculated using the convective velocity contribution along with the viscous time scale, according to the following formulation. This type is recommended when viscous forces are dominant in the flow and the velocity is very small.

$$CFL=\frac{\Delta t\sum_{f}^{N_f}|\dot{m}_f|}{2\rho\Omega}+C\frac{\Delta t\left(\Lambda^{\text{vis-}x}+\Lambda^{\text{vis-}y}+\Lambda^{\text{vis-}z}\right)}{\Omega} \tag{36.4}$$

  – `[3]Acoustic`

    For this type, the time step size is calculated using the speed of sound rather than convective velocity according to the following formulation, and the contribution of viscous time scale is ignored. Thus, this type is recommended for supersonic compressible flows when viscous effects are not important.

$$CFL=\frac{\Delta t\sum_{f}^{N_f}\vec{u}_f\cdot\vec{A}_f}{2\Omega}+\frac{\Delta t\sum_{f}^{N_f}a_f\vec{A}_f}{2\Omega} \tag{36.5}$$

  – `[4]Flow`

    For this type, the time step size is calculated using a combination of acoustic, convective, and viscous time scales, according to the following formulation. It has been tested for various problems and is able to resolve most of the flow scales, though it may produce a less aggressive time step.

    The physical eigenvalues system is considered when performing calculations with the Courant number. Eigenvalues of the system are computed based on the cell center values.

$$CFL=\frac{\Delta t\left(\Lambda^{\text{inv-}x}+\Lambda^{\text{inv-}y}+\Lambda^{\text{inv-}z}\right)}{\Omega}+C\frac{\Delta t\left(\Lambda^{\text{vis-}x}+\Lambda^{\text{vis-}y}+\Lambda^{\text{vis-}z}\right)}{\Omega} \tag{36.6}$$

  In the previous equations

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\Lambda^{\text{inv}-x}=\lambda_{\max}\frac{1}{2}\sum_{f}^{N_f}|A_x|$$
$$\Lambda^{\text{inv}-y}=\lambda_{\max}\frac{1}{2}\sum_{f}^{N_f}|A_y| \tag{36.7}$$
$$\Lambda^{\text{inv}-z}=\lambda_{\max}\frac{1}{2}\sum_{f}^{N_f}|A_z|$$

$$\Lambda^{\text{vis}-x}=\varsigma_{\max}\frac{\left(\frac{1}{2}\sum_{f}^{N_f}|A_x|\right)^2}{\Omega}$$
$$\Lambda^{\text{vis}-y}=\varsigma_{\max}\frac{\left(\frac{1}{2}\sum_{f}^{N_f}|A_y|\right)^2}{\Omega} \tag{36.8}$$
$$\Lambda^{\text{vis}-z}=\varsigma_{\max}\frac{\left(\frac{1}{2}\sum_{f}^{N_f}|A_z|\right)^2}{\Omega}$$

and

Ω = Cell volume

C = Constant factor

$\vec{u}_f$ = Velocity vector for face f

a_f = Speed of sound for face f

$\lambda_{\max}$ = Inviscid spectral radii

$\varsigma_{\max}$ = Viscous spectral radii

$\vec{A}$ = Surface area vector

N_f = Number of cell faces

36.13.3. Error-Based Time Stepping

As mentioned in Inputs for Time-Dependent Problems (p. 3645), it is possible to have the size of the time step change as the calculation proceeds based on the truncation error tolerance. This section provides a brief description of the algorithm that Ansys Fluent uses to compute the time step size, as well as an explanation of parameters that you can set to control the error-based time stepping.

Important:

Error-based time stepping is available only with the pressure-based and density-based implicit formulations; it cannot be used with the density-based explicit formulation. It cannot be used with second-order time integration and/or the in-cylinder dynamic mesh option; it is also unavailable with Euler-Euler multiphase models (described in Approaches to Multiphase Modeling in the Theory Guide), with the exception of the Eulerian model when the implicit volume fraction formulation is enabled.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.13.3.1. The Error-Based Time Stepping Algorithm

The automatic determination of the time step size is based on the estimation of the truncation error associated with the time integration scheme. If the truncation error is smaller than a specified tolerance, the size of the time step is increased; if the truncation error is greater, the time step size is decreased.

An estimation of the truncation error can be obtained by using a predictor-corrector type of algorithm [56] (p. 5716) in association with the time integration scheme. At each time step, a predicted solution can be obtained using a computationally inexpensive explicit method: forward Euler for the first-order unsteady formulation. This predicted solution is used as an initial condition for the time step, and the correction is computed using the nonlinear iterations associated with the implicit (pressure-based or density-based) formulation. The norm of the difference between the predicted and corrected solutions is used as a measure of the truncation error. By comparing the truncation error with the desired level of accuracy (that is, the truncation error tolerance), Ansys Fluent is able to adjust the time step size by increasing it or decreasing it.

In cases where the truncation error remains above the specified tolerance, Ansys Fluent will try to meet the tolerance within 5 attempts. If this tolerance is met or the 5 attempts are completed, then the iteration moves on to the next time step. An explicit scheme is used to predict the solution at each time step, then the explicit prediction is corrected with an implicit scheme. The truncation error, which is a function of the difference between the predicted and corrected solutions at a specific time is used to calculate the next time step. However, if the calculated truncation error is greater than the tolerance limit, Ansys Fluent has the option of reverting from the currently performed iteration, which is moving from the n^{th} step to $n+1^{\text{th}}$ step, and performing the iteration with a smaller time step size. Since the truncation error is proportional to the time step size, decreasing the time step size reduces the truncation error. This can be done until the truncation error goes below the tolerance limit.

36.13.3.2. Specifying Parameters for Error-Based Time Stepping

For transient flows, when you select **Adaptive** for the **Type** and **Error-Based** for the **Method** in the **Run Calculation** task page, you can then set the time stepping parameters as described in Inputs for Time-Dependent Problems (p. 3645). The following provides further details that are specific for the error-based time stepping.

In the **Run Calculation** task page:

Error Tolerance

specifies the threshold value to which the computed truncation error is compared. Increasing this value will lead to an increase in the size of the time step and a reduction in the accuracy of the solution. Decreasing it will lead to a reduction in the size of the time step and an increase in the solution accuracy, although the calculation will require more computational time. For most cases, the default value of 0.01 is acceptable.

Initial Time Step Size

is used for the first time step and then for as many subsequent steps as specified in the **Number of Fixed Time Steps** field. This value must fall between the minimum and maximum time step sizes. It should be chosen such that the Courant number initially remains close to 1 for a better

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

startup. Note that if you initialize the solution for a previously run case with adaptive time stepping, then Ansys Fluent will use the time step size saved in the case file.

In the **Adaptive Time Stepping** dialog box:

Minimum/Maximum Step Change Factor

limit the degree to which the time step size can change at each time step. Limiting the change results in a smoother calculation of the time step size, especially when high-frequency noise is present in the solution. If the time step size change factor, f, is computed as the ratio between the specified truncation error tolerance and the computed truncation error, the size of time step Δt_n is computed as follows:

- If $1<f<f_{max}$, Δt_n is increased to meet the desired tolerance.
- If $1<f_{max}<f$, Δt_n is increased, but its maximum possible value is $f_{max}\Delta t_{n-1}$.
- If $f_{min}<f<1$, Δt_n is unchanged.
- If $f<f_{min}<1$, Δt_n is decreased.

36.13.4. Multiphase-Specific Time Stepping

Multiphase-specific time stepping is supported for all multiphase models using implicit or explicit volume fraction formulation, except for the wet steam model. It is not supported with the in-cylinder dynamic mesh option, the acoustic model, and/or the structural model.

By default, multiphase-specific time stepping is based on the convective time scale (Courant based), in which the time-step-size calculation depends on the mesh density and velocity in interfacial cells.

Note:

If there are other time scales controlling the physics of problem (surface tension time scale, meshing-driven time scales for mesh deformation, and so on), then the Courant-based time scale is not a good choice.

36.13.4.1. The Multiphase-Specific Time Stepping Algorithm

Ansys Fluent multiphase-specific time stepping algorithm uses different time scales depending on whether physics based and moving mesh constraints are enabled or not.

Cases Without Physics Based and Moving Mesh Constraints:

Convective time scale (flux based):

$$\Delta t_{conv,\,flux}=\frac{Vol}{\sum_f F_{conv}} \tag{36.9}$$

The global time step size Δt_{global} is calculated by:

$$\Delta t_{global} = C_{global}\Delta t_{conv,\,flux} \tag{36.10}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cases With Physics Based and/or Moving Mesh Constraints

Based on various scales, time step sizes are defined as follows:

- Convective time scale (flux based):

$$\Delta t_{conv,\,flux} = \frac{Vol}{\sum_f F_{conv}} \tag{36.11}$$

- Convective time scale (velocity based):

$$\Delta t_{conv,vel} = \frac{\Delta s_n}{V_f} \tag{36.12}$$

- Viscous time scale:

$$\Delta t_{visc} = \frac{0.5}{\upsilon_{max}\left(\frac{1}{\Delta s_{avg}^2}\right)} \tag{36.13}$$

- Gravitational time scale:

$$\Delta t_{grav} = \sqrt{\frac{g}{\Delta s_g}} \tag{36.14}$$

- Surface tension driven time scale:

$$\Delta t_{surf} = C_{surf}\sqrt{\frac{\rho_{avg}\Delta s_{avg3}}{2\pi\sigma}} \tag{36.15}$$

- Acoustic time scale:

$$\Delta t_{acoustic} = \frac{\Delta s_{avg}}{c} \tag{36.16}$$

- Mesh motion driven time scale (flux based):

$$\Delta t_{mesh} = \frac{Vol}{\sum_f F_{mesh}} \tag{36.17}$$

- Moving Mesh driven time scale (velocity based):

$$\Delta t_{mesh,vel} = \frac{\Delta s_v}{V_{mesh}} \tag{36.18}$$

- Convective time scale (aggregate):

  The aggregate convective time scale is estimated by blending of flux-averaged and velocity-based time scales:

$$\Delta t_{conv,\,aggregate} = 0.5\left(\Delta t_{conv,\,flux_avg} + \Delta t_{conv,vel}\right) \tag{36.19}$$

  where the flux-averaged time scale $\Delta t_{conv,\,flux_avg}$ is estimated by volumetric averaging of the flux-based quantities over the cell and face connected neighbors.

- Moving mesh driven time scale (aggregate):

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The aggregate moving mesh time scale is estimated by blending of flux-averaged and velocity-based time scales:

$$\Delta t_{mesh,aggregate} = 0.5\left(\Delta t_{mesh,flux_avg} + \Delta t_{mesh,vel}\right) \tag{36.20}$$

where flux-averaged time scale $\Delta_{mesh,flux_avg}$ is calculated by volumetric averaging of the flux-based quantities over the cell and face connected neighbors with non-zero mesh velocity.

The effective physics based time scale is calculated as:

$$\Delta t_{physical} = C_{global} \min\left(\Delta t_{conv,\ aggregate}, \Delta t_{visc}, \Delta t_{grav}, \Delta t_{acousutic}, \Delta t_{surf}\right) \tag{36.21}$$

The effective moving-mesh-driven time scale is calculated as:

$$\Delta t_{mesh} = C_{mesh}\,\Delta t_{mesh,\ aggregate} \tag{36.22}$$

Global time scale Δt_{global} is calculated as the minimum of physics-based and moving-mesh-driven time scales for each cell:

$$\Delta t_{global} = min\left(\Delta t_{physical},\ \Delta t_{mesh}\right) \tag{36.23}$$

The above equations use the following nomenclature:

Vol = volume

F_{conv} = outgoing relative flux due to the flow

F_{mesh} = outgoing flux due to mesh motion

g = gravity magnitude

ρ_{avg} = Arithmetic average of phase densities

σ = surface tension coefficient

c = sound speed

υ_{max} = maximum dynamic viscosity

Δs_g = cell size in the direction of gravity

Δs_n = cell size in interface normal direction

Δs_v = cell size in the direction of mesh velocity

Δs_{avg} = averaged cell size calculated from volume

C_{surf} = Calibration factor = 2

C_{mesh} - moving mesh Courant number

C_{global} - global Courant number

Note:

During the simulation, the **Effective Global Courant Number [Multiphase Specific Time Stepping Criteria]** is printed in the console. It is calculated as:

$$C_{global,\ eff} = C_{global}\frac{\Delta t_{global,\ eff}}{\Delta t_{global}}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where C_{global} is the global Courant number, Δt_{global} is the global time step, and $\Delta t_{global,eff}$ is the effective global time step.

The effective global time step $\Delta t_{global,eff}$ is initiated by the global time step Δt_{global} and is constrained in the following ways:

- $\Delta t_{old} f_{min} < \Delta t_{global,eff} < \Delta t_{old} f_{max}$

  Here, Δt_{old} is the previous time step, and f_{min} and f_{max} are the minimum and maximum step change factors, respectively.

- $\Delta t_{min} < \Delta t_{global,eff} < \Delta t_{max}$

  Here, Δt_{min} and Δt_{max} are the minimum and maximum time step sizes, respectively.

36.13.4.2. Specifying Parameters for Multiphase-Specific Time Stepping

For transient flows, when you select **Adaptive** for the **Type** and **Multiphase-Specific** for the **Method** in the **Run Calculation** task page, you can then set the time stepping parameters as described in Inputs for Time-Dependent Problems (p. 3645). The following provides further details that are specific for the multiphase-specific time stepping.

In the **Run Calculation** task page:

Global Courant Number

is the ratio of the time step size to the characteristic time taken for the fluid to cross a control volume. To calculate the Courant number, Ansys Fluent uses a flux-based definition where, in the region near the fluid interface, Ansys Fluent divides the volume of each cell by the sum of the outgoing fluxes. The resulting time represents the time it would take for the fluid to empty out of the cell. The smallest such time is used as the characteristic time of transit for a fluid element across a control volume. The default value for **Global Courant Number** is 2, which means that the estimated time step size is twice the characteristic time of transit.

The optimal value for **Global Courant Number** is dependent on a specific case. The general recommendations on choosing an appropriate value for Global Courant Number are summarized below.

- Explicit volume fraction formulation
  - For accuracy of an instantaneous-time solution, the default value of 2 is recommended.
  - For a time-averaged solution, you can use higher values, but not larger than 5.
- Implicit volume fraction formulation
  - For an instantaneous-time solution, you can set **Global Courant Number** to 2 (recommended) with the first order time formulation and to a higher value (up to 5) with the second order time formulation.
  - For a time-averaged solution, you can use even higher values (up to 20).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Initial Time Step Size

is used for the first time step and then for as many subsequent steps as specified in the **Number of Fixed Time Steps** field. This value must fall between the minimum and maximum time step sizes. It should be chosen such that the **Global Courant Number** initially remains close to 1 or your specified value (whichever is lower) for a better startup. Note that if you initialize the solution for a previously run case with adaptive time stepping, then Ansys Fluent will use the time step size saved in the case file.

In the **Adaptive Time Stepping** dialog box (which is opened by clicking **Settings...** in the **Run Calculation** task page):

Minimum / Maximum Step Change Factor

limit the degree to which the time step size can change at each time step. Limiting the change results in a smoother calculation of the time step size, especially when high-frequency noise is present in the solution. The time step size change factor is computed as the ratio between the current time step size and the previous time step size.

When choosing the values, it is generally not recommended to allow large variation in time step size due to stability considerations. This is especially true when time step size is ramping up. It is recommended that you use a minimum time step size factor in the range of 0.5 to 0.8 and a maximum time step size change factor in the range of 1.2 to 1.5. The default values of 0.8 and 1.2, respectively, provide a smooth transition of the time step size.

Moving Mesh CFL Constraint

(VOF model only) includes moving mesh CFL constraint in the time-step-size estimation. When this option is enabled, you can adjust **Moving Mesh Courant Number**, which has a default value of 1. For applications with a pure rigid body motion, **Moving Mesh Courant Number** may not be a significant parameter.

This item is available only for applications that involve either Frame Motion, or Mesh Motion, or Dynamic Mesh. See The Multiphase-Specific Time Stepping Algorithm (p. 3665) for details.

Physics Based Constraint

(VOF model only) provides an option to include physics-based constraints depending on a specific type of physics involved (such as viscous, acoustic, gravitational, surface-tension-driven, and so on) in the time-step-size estimation. See The Multiphase-Specific Time Stepping Algorithm (p. 3665) for details.

36.13.5. Postprocessing for Time-Dependent Problems

The postprocessing of time-dependent data is similar to that for steady-state data, with all graphical and alphanumeric commands available. You can read a data file that was saved at any point in the calculation (by you or with the autosave option) to restore the data at any of the time levels that were saved.

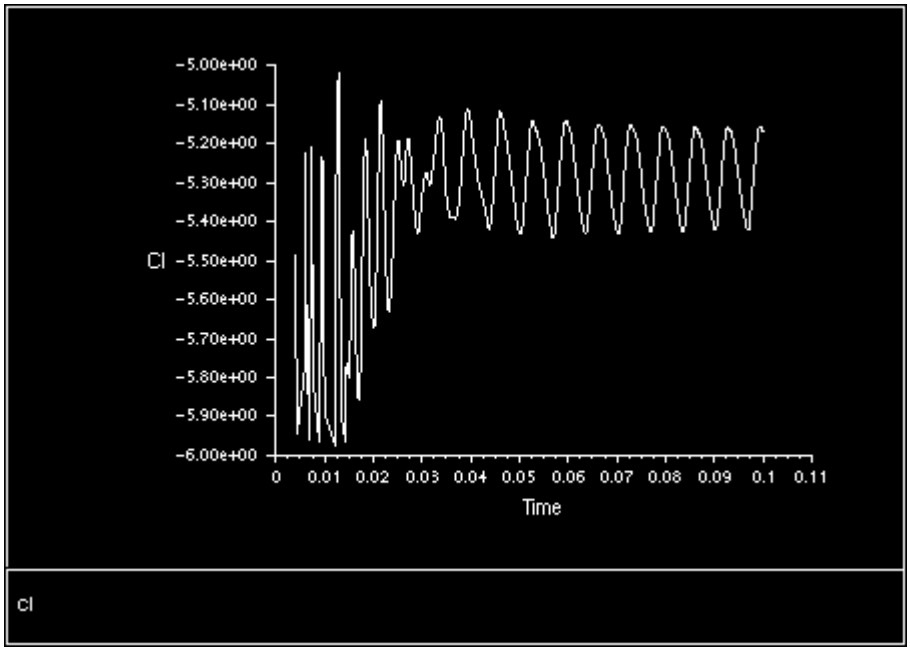
File → Read → Data...

Ansys Fluent will label any subsequent graphical or alphanumeric output with the time value of the current data set.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you save data from the force or surface report definitions to files (see step 5 in Inputs for Time-Dependent Problems (p. 3645)), you can read these files back in and plot them to see a time history of the monitored quantity. Figure 36.27: Lift Coefficient Plot for a Time-Periodic Solution (p. 3670) shows a sample plot generated in this way.

Figure 36.27: Lift Coefficient Plot for a Time-Periodic Solution



If you enable the **Data Sampling for Time Statistics** option in the Run Calculation Task Page (p. 5207) (see Performing Time-Dependent Calculations (p. 3643) for details), Ansys Fluent will compute the time average (mean) of the instantaneous values, the root-mean-square, and the root-mean-square-errors of those quantities and custom field functions that are enabled/selected in the **Sampling Options** and **Zone-Specific Sampling Options** dialog boxes. Additionally, minimum, maximum, and runtime discrete Fourier transform (DFT) values are available if the corresponding options are enabled in the **Zone-Specific Sampling Options** dialog box.

The **Sampled Time** displays the time over which data has been sampled for the postprocessing of the mean, RMS, and RMSE values. For most quantities, as long as the time step size has been constant, dividing this by the time step size yields the number of data sets that have been collected. However, special consideration is required In the case of discrete phase quantities. As noted in Reporting of Unsteady DPM Statistics (p. 2871), sampling of DPM quantities occurs only when parcels pass through a cell, not necessarily at each specified sampling interval. Therefore, the number of samples of the DPM quantities taken during the sampled time period is stored in the separate postprocessing variable, **Accum DPM Parcels in Cell**. If the time step size is varied, every contribution of data sets sampled is automatically weighted by the current time step size.

The mean, root-mean-square(RMS), and root-mean-square-error (RMSE) values for solution variables will be available in the **Unsteady Statistics...** category (or the **Unsteady DPM Statistics...** category for discrete phase quantities) of the variable selection drop-down list that appears in postprocessing dialog boxes. For example, in the **Contours** dialog box, you could select **Unsteady Statistics...** and **RMSE-uns-custom-functon-0** for the **Contours of** drop-down lists in order to display the root-mean-square-errors of a custom field function named **uns-custom-functon-0**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.13.6. Runtime Discrete Fourier Transformation

The purpose of the runtime discrete Fourier transformation (DFT) in Ansys Fluent is to:

- Provide spatial fields of the Fourier amplitudes and phases for postprocessing.
- Create animations of the frequency-filtered transient solutions by using the Inverse DFT. Inverse DFT is a beta feature in the current version 2024 R2 and is documented in the beta features manual (Creating Animations Using Inverse DFT).

Runtime DFT is available for transient simulations and can be applied to any existing field variable. Runtime DFT is implemented as a part of the zone-specific data sampling for time statistics.

Solution → Run Calculation → Sampling Options (Zone Selection)

This opens the **Zone-Specific Sampling Options** dialog box, see Figure 36.28: Zone-Specific Sampling Options Dialog Box for Runtime DFT - Frequency Band (p. 3671) and Figure 36.29: Zone-Specific Sampling Options Dialog Box for Runtime DFT - Single Tone (p. 3672). Similar to the standard zone-specific statistics, sampled DFT data are organized into datasets. Each DFT dataset contains DFT data for one quantity in one set of selected zones, for either one frequency band or one tone. If many quantities are selected at once with the same set of DFT options, then many datasets will be created, one dataset for each quantity.

Figure 36.28: Zone-Specific Sampling Options Dialog Box for Runtime DFT - Frequency Band

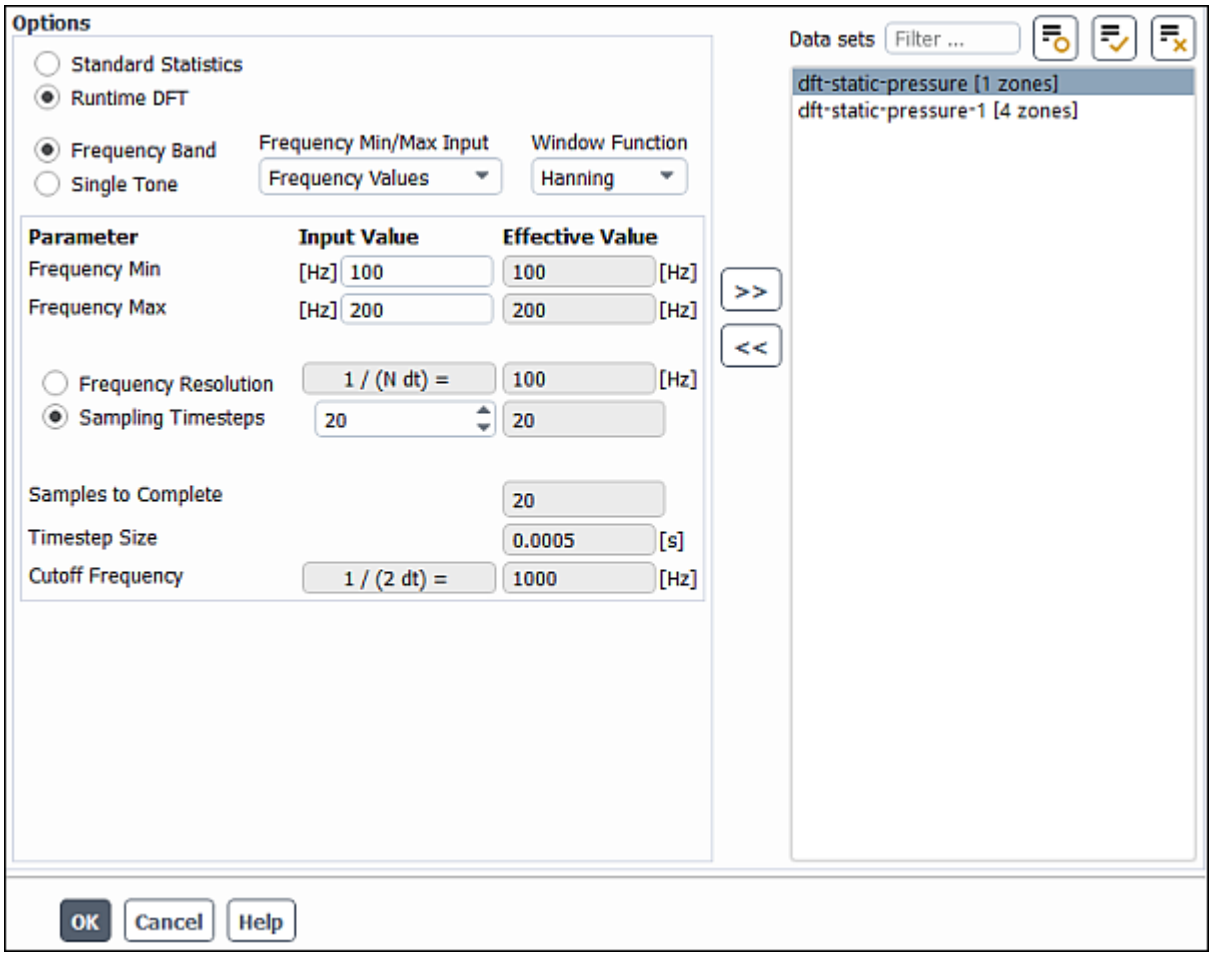
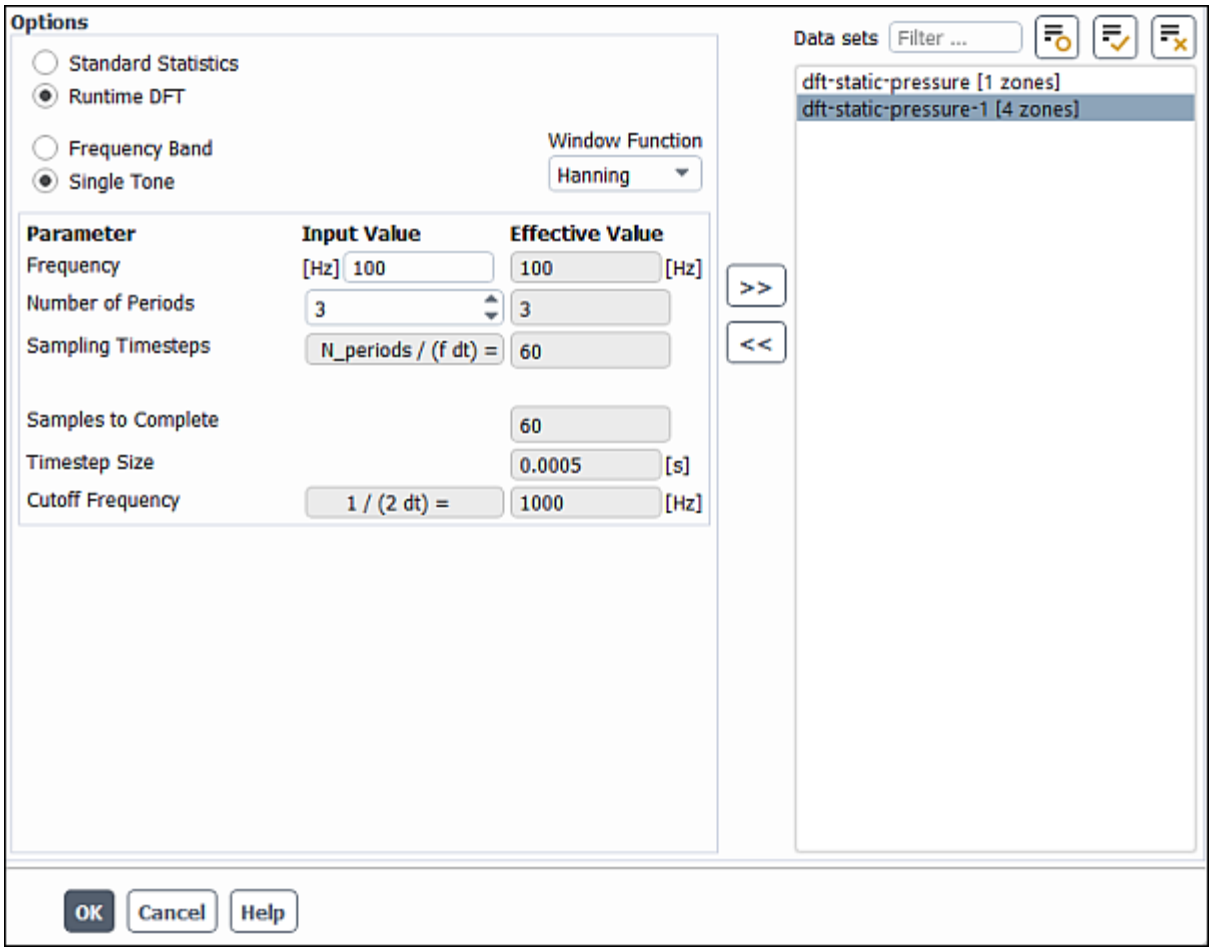


Figure 36.29: Zone-Specific Sampling Options Dialog Box for Runtime DFT - Single Tone



The following are limitations for the Runtime DFT:

- Runtime DFT is not available for the Density Based solver with the **Explicit** transient formulation.
- Runtime DFT requires constant time stepping. The DFT datasets must be created after specifying the timestep size, because the DFT parameters are timestep-dependent. If the timestep size is changed after the dataset definition, the dataset will be removed during the calculations.

36.13.6.1. Runtime DFT Formulation

The basic mathematical formulation for the runtime DFT is provided by Equation 40.8 (p. 4080) and Equation 40.9 (p. 4080) in Fast Fourier Transform (FFT) (p. 4080). To compute DFT for a non-periodic signal, the standard Hanning window function is made available:

$$W_j = 0.5\left[1-\cos\left(\frac{2\pi j}{N}\right)\right],\ 1 \le j \le N \tag{36.24}$$

Note the difference between this standard Hanning window and its "tapered" modification, which is used in Ansys Fluent's FFT tool, see Equation 40.5 (p. 4079). When the Hanning window function is used for the runtime DFT, a correction factor is applied to the accumulated Fourier amplitudes:

- For a single tone dataset, an amplitude-preserving correction factor of 2 is used.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For a frequency band, an energy-preserving correction factor of $\sqrt{8/3}$ is used.

36.13.6.2. Input Parameters for the Runtime DFT

For a frequency band, you have a choice to specify one of two values: either the band **Frequency Resolution** in Hz, or the number of the **Sampling Timesteps**. The other value is computed automatically, and your input is adjusted to the closest realizable values according to the timestep size and the frequency resolution to satisfy the following constraints:

- The sampling duration is the inverse frequency step. Since the sampling duration must be a multiple of the timestep size, the user-specified frequency step is automatically adjusted to the closest realizable value.
- The minimum and the maximum frequency of a band must be multiples of the frequency step. They are automatically adjusted to the closest multiples.

For a single tone, you provide the tone **Frequency** and the desired **Number of Periods** for sampling. The sampling duration is then computed using the known period size (inverse of the tone frequency). This duration is then adjusted to the closest multiple of the timestep size, and the tone frequency is adjusted accordingly, keeping the user-specified number of sampling periods. This procedure means, that with a higher number of sampling periods the needed adjustment of the user-specified tone frequency is smaller.

36.13.6.3. DFT Field Variables for Postprocessing

The following field variables are created for a DFT dataset in the **Runtime DFT…** category after the accumulation for a dataset is completed:

- For a single tone dataset, the magnitude and the phase field variables are created. The phase is provided in radians in the range $[-\pi, \pi]$.
- For a frequency band dataset, the magnitude field variable is created. This magnitude is computed as the square root of the band power, which is the sum of the squared magnitudes of all constituent harmonics within a band.

36.14. Performing Calculations with a Pseudo Time Method

For all calculations when using a pressure-based segregated solver (SIMPLE, SIMPLEC, or PISO) or for steady-state calculations when using the pressure-based coupled solver or the density-based implicit solver, you have the option of solving your flow with a pseudo time method. The pseudo time method is a form of implicit under-relaxation, described in Pseudo Time Method Under-Relaxation in the Theory Guide.

To apply pseudo time method under-relaxation, perform the following:

1. Select the **Density-Based** solver or the **Pressure-Based** solver. Except when you are using a pressure-based segregated solver, you must also select **Steady** for the time dependence.

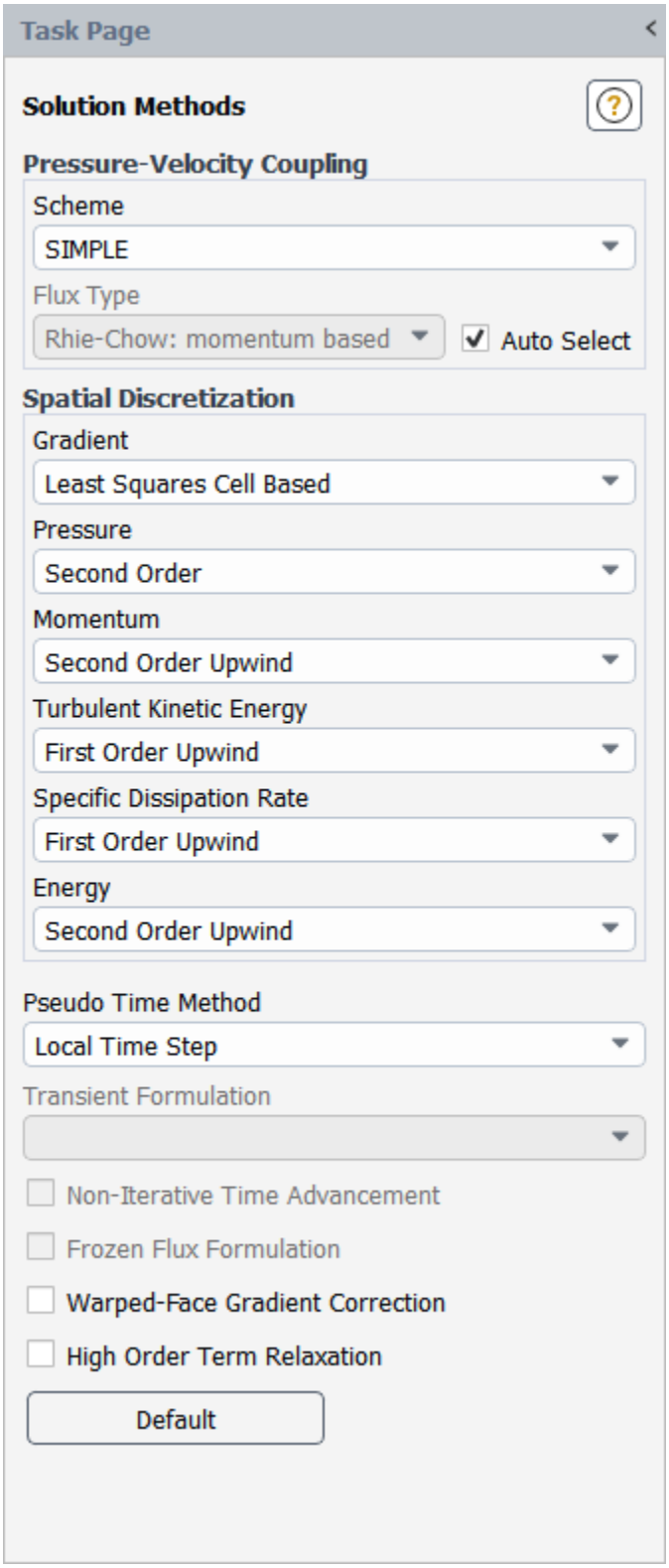
   Setup → **General**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Go to the **Solution Methods** task page (Figure 36.30: The Solution Methods Task Page (p. 3674)).

Solution → **Methods**

Figure 36.30: The Solution Methods Task Page



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. If you are using the pressure-based solver, the **Coupled** scheme under **Pressure-Velocity Coupling** is the default. You can also select one of the other segregated schemes (**SIMPLE**, **SIMPLEC**, or **PISO**).

b. If you are using the density-based solver, select the **Implicit** scheme under **Formulation**.

c. Select one of the following from the **Pseudo Time Method** drop-down list:

   - The **Local Time Step** method is available for a pressure-based segregated solver. Complete the setup for this method, as described in Local Time Step Method Setting (p. 3675).
   - The **Global Time Step** method is available for the pressure-based coupled solver or the density-based implicit solver. Complete the setup for this method, as described in Global Time Step Method Settings (p. 3676).

Note:

- The **Global Time Step** method is selected by default when using the pressure-based coupled solver for single-phase flows that do not include the battery, fuel cell, and/or solidification and melting models.
- When using a pressure-based segregated solver, the **Local Time Step** method is not available with the following:
  - the structural model
  - multiphase models
  - the **G Equation** model (as part of the premixed or partially premixed combustion model)
  - the crevice model
  - Non-Iterative Time Advancement (NITA)
- When using a pressure-based segregated solver, the **Local Time Step** method is not supported with the species transport model when the energy equation is disabled.

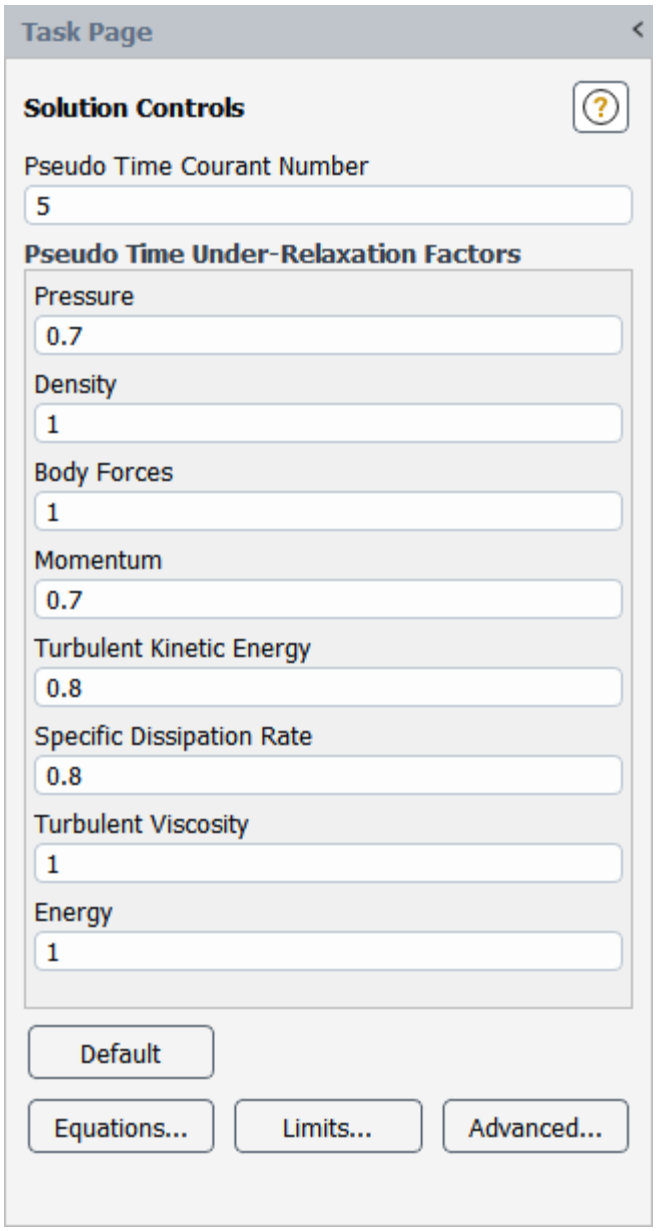
36.14.1. Local Time Step Method Setting

When **Local Time Step** is selected from the **Pseudo Time Method** list (for a case with the pressure-based segregated solver), the local pseudo time step size for each cell in the domain is based on your specified **Pseudo Time Courant Number** (see Local Time Step Method in the Theory Guide), which is available in the Solution Controls Task Page (p. 5163) (Figure 36.31: The Solution Controls Task Page for the Local Time Step Method (p. 3676)). You can also set the under-relaxation factor for each equation in the field next to its name under **Pseudo Time Under-Relaxation Factors**.

Solution → **Controls**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 36.31: The Solution Controls Task Page for the Local Time Step Method



Note that the **Local Time Step** method is recommended for better convergence behavior. While using a smaller value for the **Pseudo Time Courant Number** can improve the convergence behavior, it can also slow down the convergence rate.

36.14.2. Global Time Step Method Settings

When **Global Time Step** is selected from the **Pseudo Time Method** list (for a steady-state case with the pressure-based coupled solver or density-based implicit solver), pseudo time method under-relaxation is applied. You can also specify an explicit under-relaxation of the equation to control the update of computed variables at each iteration (see Pseudo Time Method Under-Relaxation in the Theory Guide). The default values of under-relaxation parameters for all variables are set to values that work well for most of the cases. It is good practice to start a calculation with the default under-relaxation parameters. If your case exhibits divergence or the residuals continue to increase after a few iterations, then you should reduce the under-relaxation factors.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To have further control over the parameters for each individual equation, you can specify advanced solution controls. Note that all equations can be modified, except for flow equations (that is, pressure and momentum). Generally, you will not need to enter equation-specific solution parameters. However, it may help in cases where a particular equation is giving convergence problems.

Finally, you must define settings for the running of the calculation: the pseudo time step size or time scale factor for the fluid and solid zones.

For details on defining these global time step method settings, see the following sections:

36.14.2.1. Pseudo Time Explicit Relaxation Factors

36.14.2.2. Advanced Solution Controls for the Pseudo Time Method

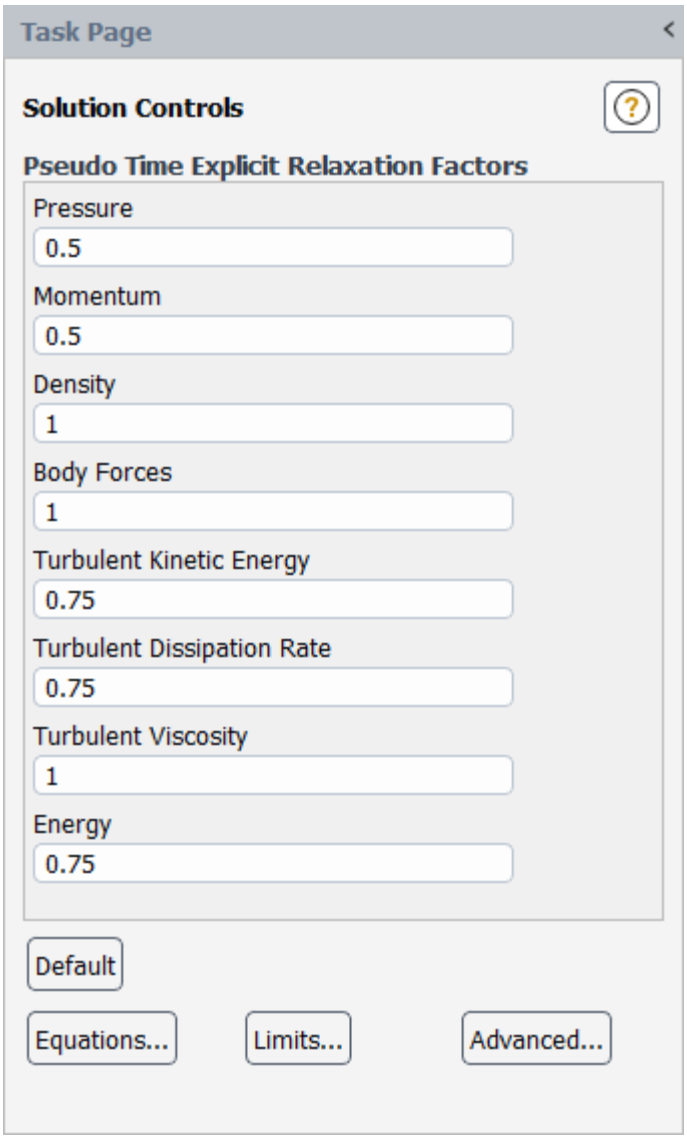
36.14.2.3. Pseudo Time Settings for the Calculation

36.14.2.1. Pseudo Time Explicit Relaxation Factors

When **Global Time Step** is selected from the **Pseudo Time Method** list in the **Solution Methods** task page, you can modify the pseudo time explicit relaxation factors in the Solution Controls Task Page (p. 5163) (Figure 36.32: The Solution Controls Task Page for the Global Time Step Method (p. 3678)).

Solution → **Controls**

Figure 36.32: The Solution Controls Task Page for the Global Time Step Method



You can set the under-relaxation factor for each equation in the field next to its name under **Pseudo Time Explicit Relaxation Factors**.

Important:

If you are using the pressure-based coupled solver, all equations will have an associated under-relaxation factor (see Under-Relaxation of Equations in the Theory Guide). If you are using the density-based implicit solver, only those equations that are solved sequentially (see Density-Based Solver in the Theory Guide) will have under-relaxation factors.

If you change under-relaxation factors, but you then want to return to Ansys Fluent's default settings, you can click the **Default** button.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.14.2.2. Advanced Solution Controls for the Pseudo Time Method

When **Global Time Step** is selected from the **Pseudo Time Method** list in the **Solution Methods** task page, Ansys Fluent provides two options to improve convergence in the **Expert** tab of in the **Advanced Solution Controls** dialog box (Figure 36.33: The Advanced Solution Controls Dialog Box for the Pseudo Time Method (p. 3680)):

1. Specify a time scale factor for the equation-specific time step size in lieu of using a uniform global pseudo time step size. This scaling factor scales the pseudo time step size employed for the flow equations specified in the **Run Calculation** task page (see Pseudo Time Settings for the Calculation (p. 3680)).

2. Use the standard steady-state method by turning off the pseudo time method usage for that particular equation. Here, the corresponding under-relaxation factor to be employed with that equation may be specified. The default values of under-relaxation parameters for all variables are set to values that work well for most of the cases.

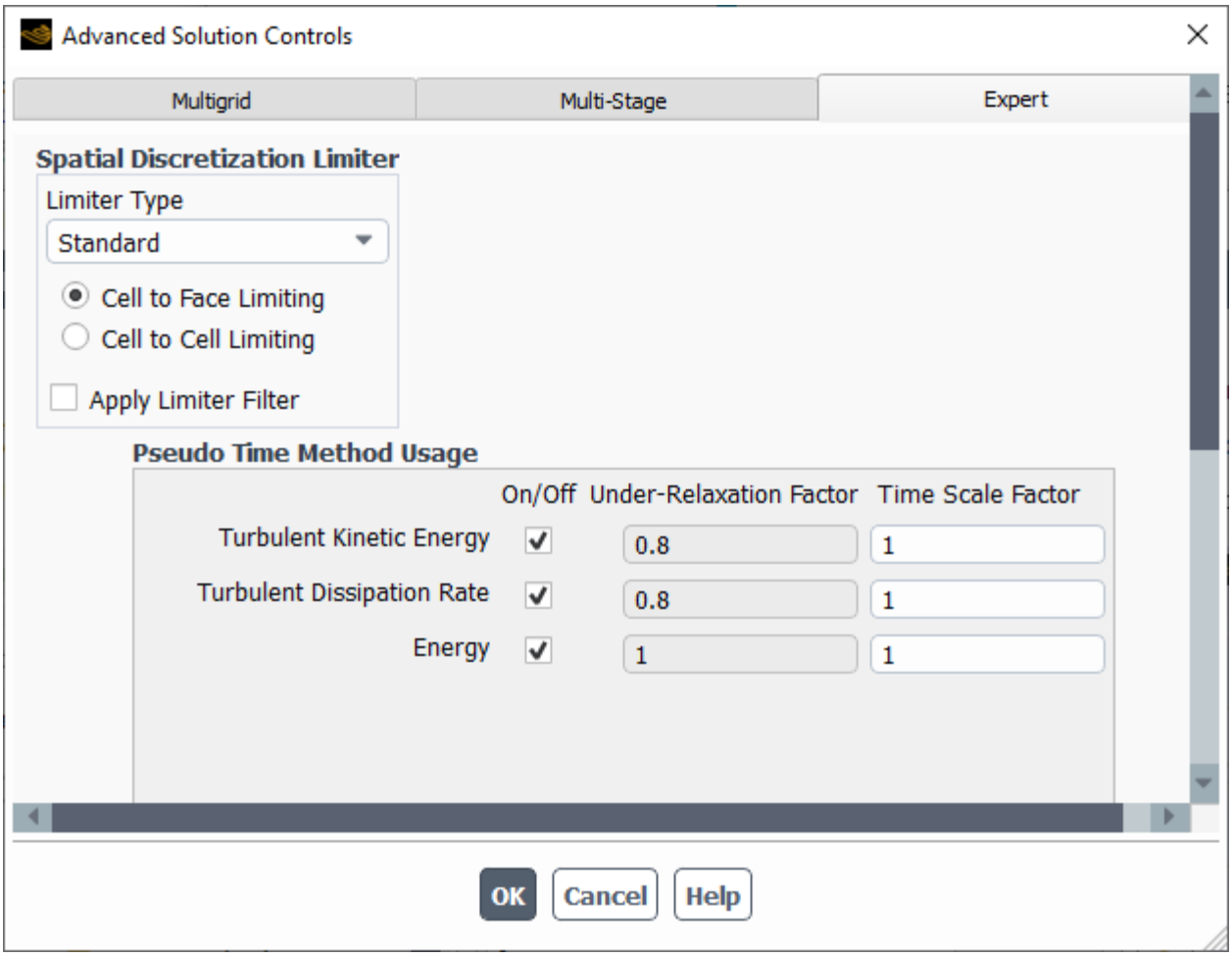
The default setting will have the pseudo time method usage turned on for all equations with the corresponding time scale factor set to unity, except when one of the combustion models is used. When a pseudo time method is selected in combustion cases, the species, enthalpy, and combustion variable equations will only use the pseudo time method if it is manually enabled in the **Expert** tab. Also, when using the premixed or partially-premixed combustion models, the pseudo time method can be manually enabled for the energy variable in the **Expert** tab.

By default, all user-defined scalars (UDS) have the pseudo time method usage disabled. This is because the physical parameters of the UDS and the appropriate time scale are not known to the solver before starting the calculation. You can manually enable the pseudo time method usage for UDS under the **Expert** tab in the Advanced Solution Controls Dialog Box (p. 5168).

Solution → **Controls** **Advanced...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 36.33: The Advanced Solution Controls Dialog Box for the Pseudo Time Method



Specify the equation-specific steady-state solution method for a particular equation (if needed) by enabling or disabling the pseudo time method usage using the **On/Off** check box next to the equation. The dialog box then allows either specification of a pseudo **Time Scale Factor** or **Under-Relaxation Factor** for that particular equation based on the check box setting.

Note:

For multiphase flows, the pseudo time method expert options for the volume fraction equation are available only when it is solved in a segregated fashion. It is not available when the **Coupled with Volume Fractions** option is enabled in the **Solution Methods** task page (see Selecting the Pressure-Velocity Coupling Method (p. 3187) for information about this setting).

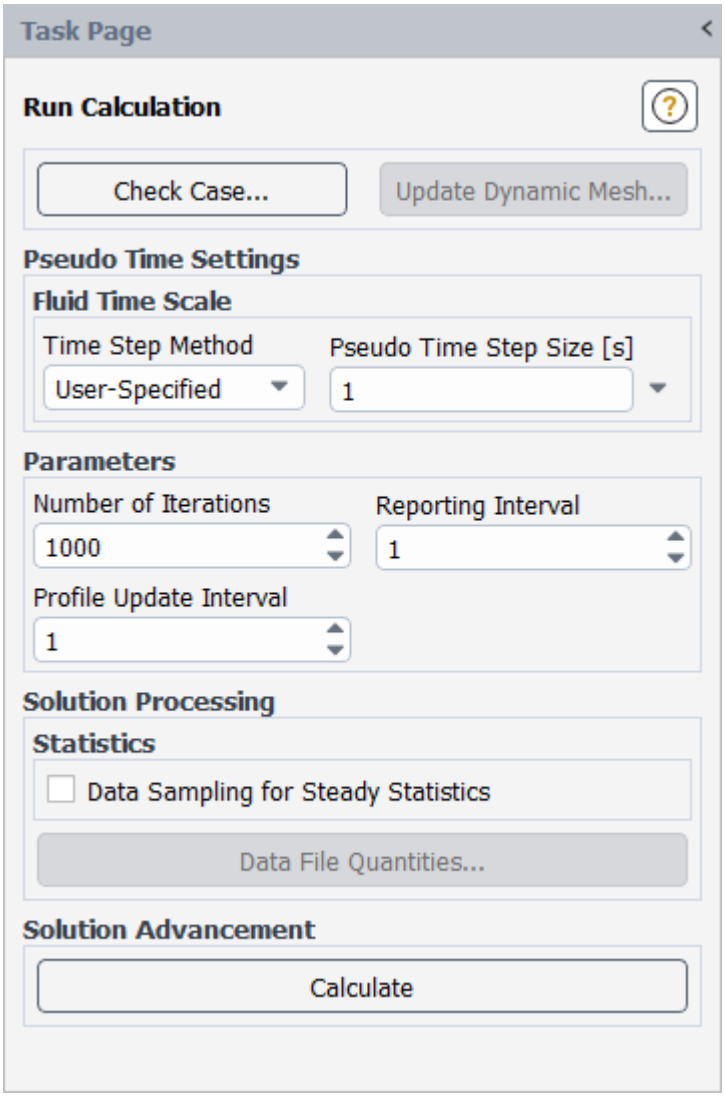
36.14.2.3. Pseudo Time Settings for the Calculation

When **Global Time Step** is selected from the **Pseudo Time Method** list in the **Solution Methods** task page, you can specify the time step size for the **Fluid Zone** and/or the **Solid Zone** under **Pseudo Time Settings** in the **Run Calculation** task page (Figure 36.34: The Run Calculation Task Page for the User-Specified Pseudo Time Step Method (p. 3681)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution → Run Calculation

Figure 36.34: The Run Calculation Task Page for the User-Specified Pseudo Time Step Method



1. Select the **Time Step Method** for the **Fluid Time Scale**.

   a. If you choose **User-Specified**, you will enter the **Pseudo Time Step Size**, which is used for every equation unless the equation specific time step is used for a particular equation, as mentioned in Advanced Solution Controls for the Pseudo Time Method (p. 3679). Typically, the time scale size should be relevant to the global time scale of the flow, for example:

   $$0.3 * \text{Global length} / \text{average velocity}$$

   where Global length is a representative length scale for the geometry (such as chord length).

   b. If you use the default **Automatic** method (see Figure 36.35: The Run Calculation Task Page for the Automatic Pseudo Time Step Method (p. 3683)), the pseudo time step size is calculated internally and used for each equation listed, unless the equation has a time step size specified as noted in Advanced Solution Controls for the Pseudo Time Method (p. 3679). The automatic

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

time scale is calculated from time scales estimated for various physics active in the simulation. For more information, refer to Global Time Step Method in the Theory Guide.

You can control the convergence process by adjusting the automatic pseudo time step size value using the **Time Scale Factor** field, which simply scales the calculated time step size by the specified value. The automatic time scale calculation is often somewhat conservative, and increasing the **Time Scale Factor** to **3** or **10** can often increase the convergence rate. Less commonly, you may need to reduce its value to **0.3** or **0.1** for better convergence.

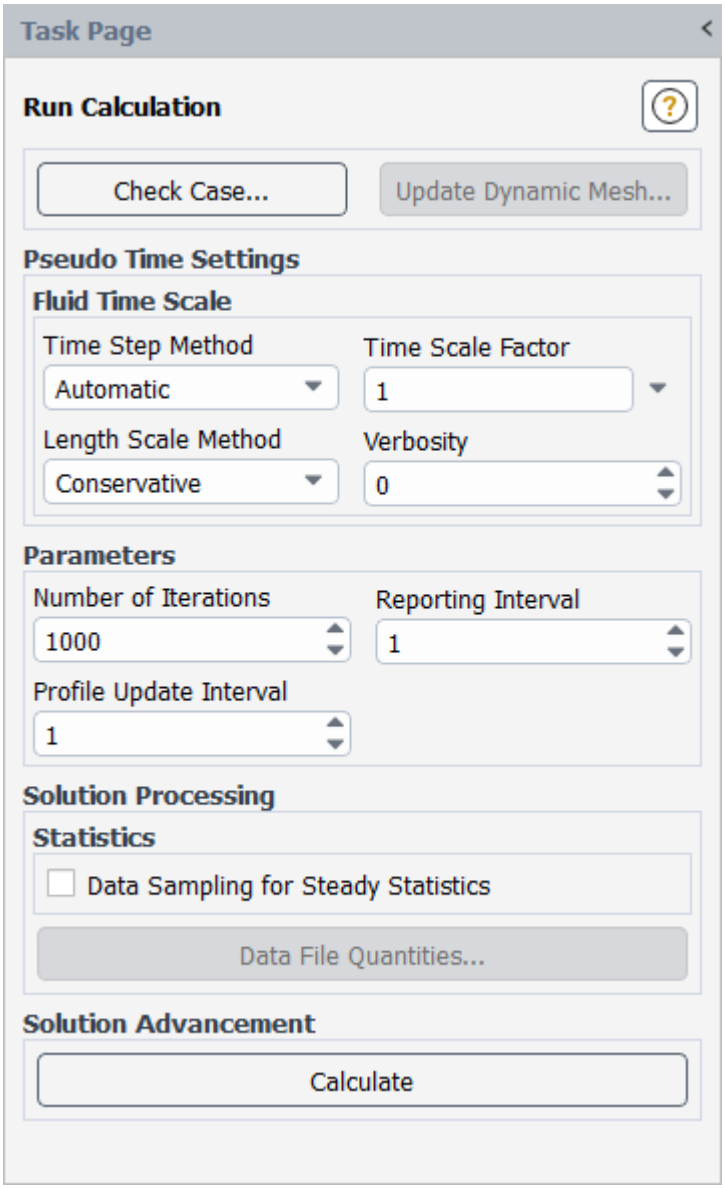
Alternatively, you can also fine-tune the automatic pseudo time step size by choosing the **Length Scale Method** used to calculate its value. The following options are available in the drop-down list:

- **Conservative** : (default) uses the cube root of the mesh volume for 3D cases and square root of area for 2D cases.
- **Aggressive** : uses the maximum geometry extent, leading to a larger time step size than **Conservative**.
- **User-Specified** allows you to directly control the **Length Scale**. This method is particularly useful in problems that have a specific length scale that is difficult to determine from the overall geometry of the problem. For example, in the case of flow over an airfoil, the appropriate length scale would be the length of the airfoil instead of the length scale calculated based on the geometry of the entire domain.

You can specify the **Verbosity**. This is an integer value of 0, 1, or 2. The default value is 0. A value of 1 prints the pseudo time step size. A value of 2 prints additional details about the calculation.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 36.35: The Run Calculation Task Page for the Automatic Pseudo Time Step Method



2. Select the **Time Step Method** for the **Solid Time Scale**.

Note:

The **Solid Time Scale** group box is only available when the **Energy** equation is enabled with a solid zone in the domain, the **Porous Zone** option enabled, or the **Solidification & Melting** model enabled.

---

a. If you choose **User-Specified**, you will enter the **Pseudo Time Step Size**, which is used where the **Solid Time Scale** is needed (that is, solid zones, porous zones, and/or solidification/melting model).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. If you choose **Automatic**, which is the default method, the pseudo time step size is calculated internally as described in Global Time Step Method in the Theory Guide. Specify a **Time Scale Factor** to control/adjust the pseudo time step size obtained from the automatically calculated **Solid Time Scale**.

3. Continue setting up your solution as you would a steady-state run, as described in Performing Steady-State Calculations (p. 3639).

36.15. Monitoring Solution Convergence

During the solution process you can monitor the convergence dynamically by checking residuals, statistics, force values, surface integrals, and volume integrals. You can print reports of or display plots of lift, drag, and moment coefficients, surface integrations, and residuals for the solution variables. For unsteady flows, you can also monitor elapsed time. Creating report definitions for monitoring is described in Creating Report Definitions (p. 4098). See Embedded Graphics Window Dashboards (p. 3941) for additional techniques that can be used to monitor the solution as it evolves.

For additional information, see the following sections:

36.15.1. Monitoring Residuals

36.15.2. Monitoring Statistics

36.15.3. Monitoring Solution Quantities

36.15.1. Monitoring Residuals

At the end of each solver iteration, the residual sum for each of the conserved variables is computed and stored, thereby recording the convergence history. This history is also saved in the data file. The residual sum is defined below.

On a computer with infinite precision, these residuals will go to zero as the solution converges. On an actual computer, the residuals decay to some small value ("round-off") and then stop changing ("level out"). For single-precision computations (the default for workstations and most computers), residuals can drop as many as six orders of magnitude before hitting round-off. Double-precision residuals can drop up to twelve orders of magnitude. Guidelines for judging convergence can be found in Judging Convergence (p. 3738).

36.15.1.1. Definition of Residuals for the Pressure-Based Solver

After discretization, the conservation equation for a general variable ϕ at a cell P can be written as

$$a_P \phi_P = \sum_{nb} a_{nb} \phi_{nb} + b \tag{36.25}$$

Here a_P is the center coefficient, a_{nb} are the influence coefficients for the neighboring cells, and b is the contribution of the constant part of the source term S_c in $S = S_c + S_P \phi$ and of the boundary conditions. In Equation 36.25 (p. 3684),

$$a_P = \sum_{nb} a_{nb} - S_P \tag{36.26}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The residual R^{ϕ} computed by Ansys Fluent's pressure-based solver is the imbalance in Equation 36.25 (p. 3684) summed over all the computational cells P. This is referred to as the unscaled residual. It may be written as

$$R^{\phi} = \sum_{cells\ P} \left| \sum_{nb} a_{nb}\phi_{nb} + b - a_P\phi_P \right| \tag{36.27}$$

In general, it is difficult to judge convergence by examining the residuals defined by Equation 36.27 (p. 3685) since no scaling is employed. This is especially true in enclosed flows such as natural convection in a room where there is no inlet flow rate of ϕ with which to compare the residual.

Ansys Fluent scales the residual using two kinds of scaling factors, representative of the flow rate of ϕ through the domain. The factors are termed *global scaling* and *local scaling*. The type of scaling can be selected from the **Residual Monitors** dialog box. The "globally scaled" residual is defined as

$$R^{\phi} = \frac{\sum_{cells\ P} \left| \sum_{nb} a_{nb}\phi_{nb} + b - a_P\phi_P \right|}{\sum_{cells\ P} \left| a_P\phi_P \right|} \tag{36.28}$$

There are two instances where ϕ_P is modified in the denominator term $a_P\phi_P$: for momentum equations, it is replaced with the magnitude of the velocity at cell P ; and for Reynolds stress equations, it is replaced with the trace of the Reynolds stress tensor at cell P divided by the number of dimensions of the case (2D or 3D).

The "locally scaled" residual is defined as

$$R_{\phi} = \frac{\sqrt{\sum_{cells}^{n} \left(\frac{1}{n}\right) \left(\frac{\sum_{nb} a_{nb}\phi_{nb} + b - a_P\phi_p}{a_p}\right)^2}}{\left(\phi_{max} - \phi_{min}\right)_{domain}} \tag{36.29}$$

Note that for the momentum and Reynolds stress equations, ϕ_{max} and ϕ_{min} are modified in the same manner as for global scaling.

The scaled residual is a more appropriate indicator of convergence for most problems. The selection of scaling and convergence criteria for different types of scaling are discussed in Judging Convergence (p. 3738). The default residuals displayed by Ansys Fluent are global scaling.

For the continuity equation, the unscaled residual for the pressure-based solver is defined as

$$R^{c} = \sum_{\text{cells } P} \left| \text{rate of mass creation in cell P} \right| \tag{36.30}$$

The local scaling is the same for all equations, except for the continuity residual, which by default uses the following enhanced formulation:

$$R_{\phi} = \sqrt{\frac{\sum_{cells}^{n} \left(\frac{R^{c}}{\sum_{f}^{nfaces} |\dot{m}_f|} \right)^2}{n}} \tag{36.31}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where R^c is the unscaled continuity residual defined in Equation 36.30 (p. 3685) for a cell P, $\dot{m}_f$ is the mass flow rate at face f of the cell, $nfaces$ is the total number of faces for the cell, and n is the total number of cells in the domain. This enhanced formulation is recommended, as the use of the absolute mass flow rate at each cell can produce behavior that is more consistent with the other residuals. Note that after you have enabled the **Compute Local Scale** option in the **Residual Monitors** dialog box (as described in a later section), you have the option of disabling this enhanced formulation for the continuity residual by using the following text command, so that it is instead calculated according to Equation 36.29 (p. 3685) like the other residuals: `solve/monitors/residual/enhanced-continuity-residual?`.

The global scaling also treats continuity in a unique way, and it is defined as

$$\frac{R^c_{\text{iteration } N}}{R^c_{\text{iteration } 5}} \tag{36.32}$$

The denominator is the largest absolute value of the continuity residual in the first five iterations.

The scaled residuals described above are useful indicators of solution convergence. Guidelines for their use are given in Judging Convergence (p. 3738). It is sometimes useful to determine how much a residual has decreased during calculations as an additional measure of convergence. For this purpose, Ansys Fluent allows you to normalize the residual (either scaled or unscaled) by dividing by the maximum residual value after M iterations, where M is set by you in the Residual Monitors Dialog Box (p. 5599) in the **Iterations** field under **Residual Values**.

$$\overline{R}^{\phi} = \frac{R^{\phi}_{\text{iteration } N}}{R^{\phi}_{\text{iteration } M}} \tag{36.33}$$

Normalization in this manner ensures that the initial residuals for all equations are of $O(1)$ and is sometimes useful in judging overall convergence.

By default, $M=5$. You can also specify the normalization factor (the denominator in Equation 36.33 (p. 3686)) manually in the Residual Monitors Dialog Box (p. 5599).

36.15.1.2. Definition of Residuals for the Density-Based Solver

A residual for the density-based solver is simply the time rate of change of the conserved variable (W). The RMS residual is the square root of the average of the squares of the residuals in each cell of the domain:

$$R(W) = \sqrt{\sum \left(\frac{\partial W}{\partial t} \right)^2} \tag{36.34}$$

Equation 36.34 (p. 3686) is the unscaled residual sum reported for all the coupled equations solved by Ansys Fluent's density-based solver.

Important:

The residuals for the equations that are solved sequentially by the density-based solver (turbulence and other scalars, as discussed in Density-Based Solver in the Theory Guide) are the same as those described above for the pressure-based solver.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In general, it is difficult to judge convergence by examining the residuals defined by Equation 36.34 (p. 3686) since no scaling is employed. This is especially true in enclosed flows such as natural convection in a room where there is no inlet flow rate of ϕ with which to compare the residual. As with the pressure-based solver, Ansys Fluent uses two types of scaling for the density-based solver. The globally scaled residual is defined as

$$\frac{R(\mathrm{W})_{\text{iteration } N}}{R(\mathrm{W})_{\text{iteration } 5}} \tag{36.35}$$

The denominator is the largest absolute value of the residual in the first five iterations.

The locally scaled residual is calculated from the local flux imbalance in the cell. It is calculated using Equation 36.36 (p. 3687) :

$$R_{\phi}=\frac{\sqrt{\sum_{n}^{cells}\left(\frac{1}{n}\right)\left(\frac{\partial w}{\partial t}V\right)^{2}}}{\left(w_{max}-w_{min}\right)_{domain}} \tag{36.36}$$

w is the conservative variable and V is the cell volume. The unscaled residual is always calculated from Equation 36.34 (p. 3686).

The scaled residuals described above are useful indicators of solution convergence. Guidelines for their use are given in Judging Convergence (p. 3738). It is sometimes useful to determine how much a residual has decreased during calculations as an additional measure of convergence. For this purpose, Ansys Fluent allows you to normalize the residual (either scaled or unscaled) by dividing by the maximum residual value after M iterations, where M is set by you in the Residual Monitors Dialog Box (p. 5599) in the **Iterations** field under **Residual Values**.

Normalization of the residual sum is accomplished by dividing by the maximum residual value after M iterations, where M is set by you in the Residual Monitors Dialog Box (p. 5599) in the **Iterations** field under **Residual Values** :

$$\overline{R}(\mathrm{W})=\frac{R(\mathrm{W})_{\text{iteration } N}}{R(\mathrm{W})_{\text{iteration } M}} \tag{36.37}$$

Normalization in this manner ensures that the initial residuals for all equations are of $O(1)$ and is sometimes useful in judging overall convergence.

By default, $M=5$, making the normalized residual equivalent to the scaled residual. You can also specify the normalization factor (the denominator in Equation 36.37 (p. 3687)) manually in the Residual Monitors Dialog Box (p. 5599).

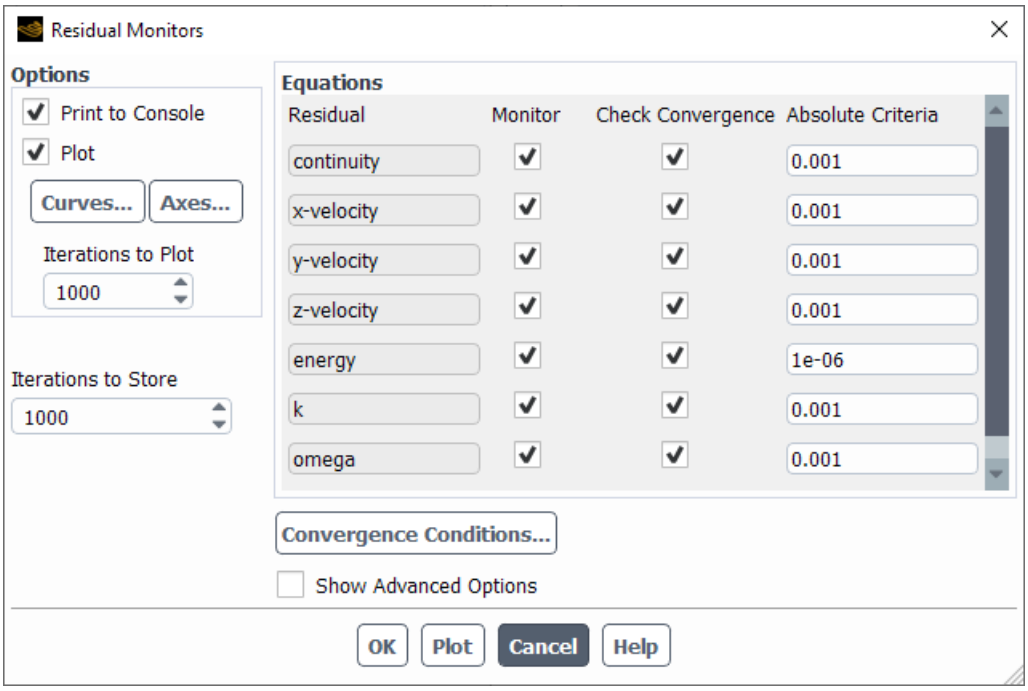
36.15.1.3. Overview of Using the Residual Monitors Dialog Box

All inputs controlling the monitoring of residuals are entered using the Residual Monitors Dialog Box (p. 5599) (Figure 36.36: The Residual Monitors Dialog Box (p. 3688)).

Solution → **Reports** → **Residuals...**

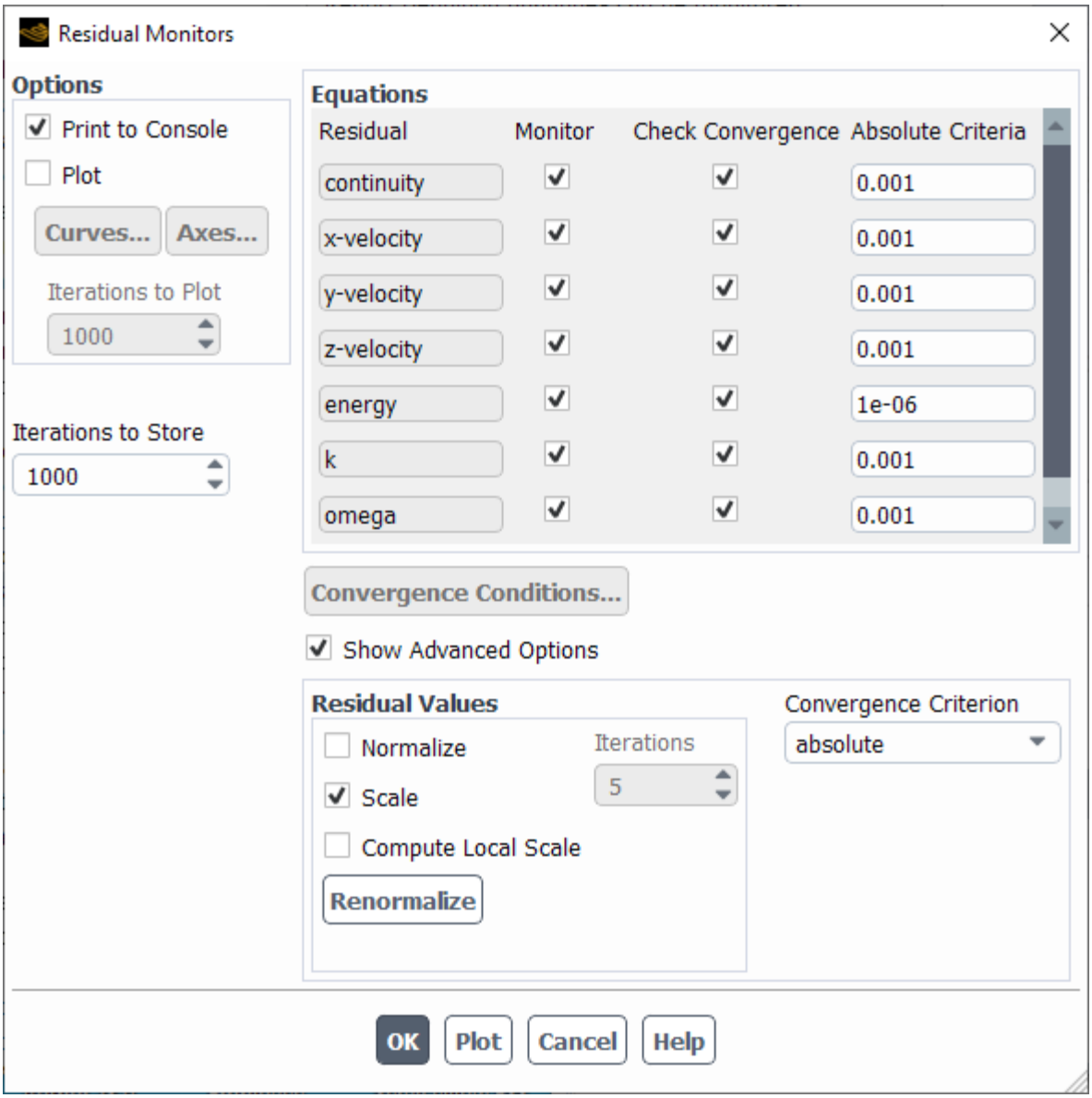
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 36.36: The Residual Monitors Dialog Box



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 36.37: The Residual Monitors Dialog Box with Advanced Options Shown



In general, you will only need to enable residual plotting and modify the convergence criteria using this dialog box. Additional controls are available for disabling monitoring of particular residuals, and modifying normalization and plot parameters.

36.15.1.4. Printing and Plotting Residuals

By default, residual values for all relevant variables are printed in the console after each iteration. If you want to disable this printout, turn off **Print to Console** under **Options**. To enable the plotting of residuals after each iteration, turn on **Plot** under **Options**. Residuals will be plotted in the graphics window during the calculation.

If you want to display a plot of the current residual history, simply click the **Plot** push button.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.15.1.5. Storing Residual History Points

Residual histories for each variable are automatically saved in the data file, regardless of whether they are being monitored. You can control the number of history points to be stored by changing the **Iterations to Store** entry. By default, up to 1000 points will be stored. If more than 1000 iterations are performed (that is, the limit is reached), every other point will be discarded—leaving 500 history points—and the next 500 points will be stored. When the total hits 1000 again, every other point will again be discarded, and so on. If you are performing a large number of iterations, you will lose a great deal of residual history information at the beginning of the calculation. In such cases, you should increase the **Iterations to Store** value to a more appropriate value. Of course, the larger this number is, the more memory you will need, the longer the plotting will take, and the more disk space you will need to store the data file.

36.15.1.6. Controlling Normalization and Scaling

Options for controlling normalization and scaling are found by enabling **Show Advanced Options** in the **Residual Monitors** dialog box (Figure 36.37: The Residual Monitors Dialog Box with Advanced Options Shown (p. 3689)).

By default, scaling of residuals (see Equation 36.28 (p. 3685), Equation 36.32 (p. 3686), or Equation 36.35 (p. 3687)) is enabled and the default convergence criterion is 10^{-6} for energy, DO, and P-1 equations and 10^{-3} for all other equations. When the **Scale** option is enabled, global scaling will be applied by default. You can then enable the **Compute Local Scale** option and select the **Reporting Option** from the drop-down list. You have a choice to plot or print to the console the **local scaling** or the **global scaling** of residuals. By default, the **global scaling** of residuals will be plotted. If **local scaling** is selected, the default convergence criterion for every equation is changed to 10^{-5} for steady cases or 10^{-4} for transient cases.

Note:

- When **Compute Local Scale** is enabled, Ansys Fluent computes and stores both the locally and globally scaled residuals from subsequent iterations, for the purpose of reporting. The scaled residuals are stored in the data file.
- You can enable **Local Residual Scaling** in **Preferences** (**Simulation** branch) to have local residuals plotted by default.

Important:

Once the **Compute Local Scale** option is enabled and you disable the **Scale** option, the **Compute Local Scale** option will not automatically be disabled. Instead, it will compute both the locally and globally scaled residuals, but only print or plot the unscaled residual.

Residual normalization (that is, dividing the residuals by the largest value during the first few iterations) is also available but disabled by default.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Normalization can be used with both scaled and unscaled residuals. Note that if normalization is enabled, the convergence criterion may need to be adjusted appropriately. See Judging Convergence (p. 3738) for information about judging convergence based on the different types of residual reports. (Both the raw residuals and scaling factors are stored in the data file, so you can switch between scaled and unscaled residuals.)

To report unscaled residuals, simply disable the **Scale** option under **Residual Values**.

Important:

If you switch from scaled to unscaled residuals (or vice versa) and you are normalizing the residuals (as described below), you must click the **Renormalize** button to recompute the normalization factors.

If you want to normalize the residuals (see Equation 36.33 (p. 3686) or Equation 36.37 (p. 3687)), enable the **Normalize** option under **Residual Values**. The **Normalization Factor** column will be added to the dialog box at this time. Ansys Fluent will normalize the printed or plotted residual for each variable by the value indicated as the **Normalization Factor** for that variable. The default **Normalization Factor** is the maximum residual value after the first 5 iterations. To use the maximum residual value after a different number of iterations (that is, specify a different value for M in Equation 36.33 (p. 3686) or Equation 36.37 (p. 3687)), you can modify the **Iterations** entry under **Residual Values**.

In some cases, the maximum residual may occur sometime after the iteration specified in the **Iterations** field. If this should occur, you can click the **Renormalize** button to set the normalization factors for all variables to the maximum values in the residual histories. Subsequent plots and printed reports will use the new normalization factor.

You can also specify the normalization factor (the denominator in Equation 36.33 (p. 3686) or Equation 36.37 (p. 3687)) explicitly. To modify the normalization factor for a particular variable, enter a new value in the corresponding **Normalization Factor** field in the Residual Monitors Dialog Box (p. 5599).

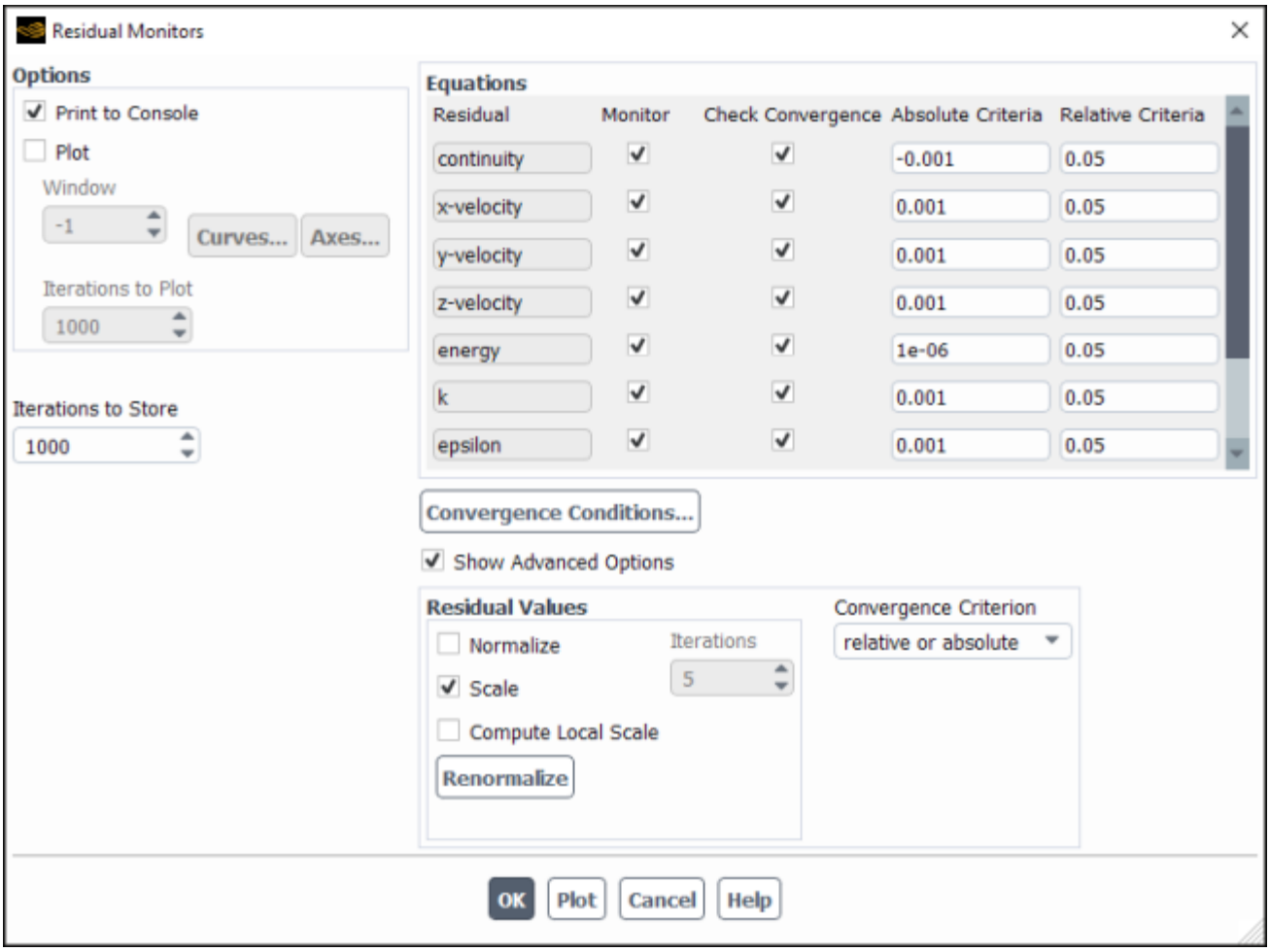
If you want to report unnormalized, unscaled residuals (Equation 36.27 (p. 3685) or Equation 36.34 (p. 3686)), disable the **Normalize** and **Scale** options under **Residual Values** in the Residual Monitors Dialog Box (p. 5599). Note that unnormalized, unscaled residuals are stored in the data file regardless of whether the reported residuals are normalized or scaled.

36.15.1.7. Choosing a Convergence Criterion

With **Show Advanced Options** enabled, you have the ability to choose certain convergence criteria, which provides you with alternative ways to check convergence when using the iterative transient solver. The various convergence criteria can be selected in the **Residual Monitors** dialog box from the **Convergence Criterion** drop-down list (Figure 36.38: The Residual Monitors Dialog Box Displaying Relative or Absolute Convergence (p. 3692)).

Solution → **Reports** → **Residuals...**

Figure 36.38: The Residual Monitors Dialog Box Displaying Relative or Absolute Convergence



Four options are available for checking an equation for convergence:

absolute

This is the default. For steady-state cases, **absolute** and **none** are the only options available for selection. The residual (scaled and/or normalized) of an equation at an iteration is compared with a user-specified value. If the residual is less than the user-specified value, that equation is deemed to have converged for a time step.

relative

The residual of an equation at an iteration of a time step is compared with the residual at the start of the time step. If the ratio of the two residuals is less than a user-specified value, that equation is deemed to have converged for a time step.

relative or absolute

If either the **absolute** convergence criterion or the **relative** convergence criterion is met, the equation is considered converged.

The **Relative Criteria** can be set when **relative** or **relative or absolute** is selected.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

none

Convergence checking is disabled.

In many situations, the **absolute** convergence criterion could be too stringent for transient flows causing a large number of iterations per time step. For example, the scaling of the continuity equation is based on the value of the continuity residual in the first five iterations. The scaling factor could be low if the initial continuity residual is small and therefore the scaled residual could fail to meet the **absolute** convergence criterion. With the **relative** convergence criterion, convergence is checked by comparing the residual at an iteration of a time step with the residual at the beginning of the time step and hence this problem is alleviated. The **relative or absolute** convergence criterion is useful in situations where the residuals of some of the equations are already very low at the start of a time step (for example, when a particular variable has reached steady-state), and the order of magnitude reduction in residuals is not possible. The **none** option allows you to disable convergence checking by selecting the option in the **Convergence Criterion** drop-down list.

Important:

relative and **relative or absolute** convergence criteria are available only with the unsteady pressure-based solver and unsteady density-based solver.

The text command used to access the convergence criterion is

`solve` → `monitors` → `residual` → `criterion-type`

When `criterion-type` is entered, you will have the following choices:

Criterion	Type
0	absolute
1	relative
2	relative or absolute
3	none

For `criterion-type` 1 or 2, the text command `relative-conv-criteria` will appear under the `residual` text menu, where the various `relative-conv-criteria` can be set.

Important:

If the NITA solver is enabled, no convergence criteria are available for selection.

36.15.1.8. Modifying Convergence Criteria

Depending on the **Convergence Criterion** you choose, Ansys Fluent will check for convergence. If convergence is being monitored, the solution will stop automatically when each variable meets its specified convergence criterion. Convergence checks can only be performed on variables whose residuals are being monitored (variables with the **Monitor** option enabled).

You can choose whether or not you want to check the convergence for each variable by enabling or disabling the **Check Convergence** option for it in the Residual Monitors Dialog Box (p. 5599). To

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

modify the convergence criterion for a particular variable, enter a new value in the corresponding convergence criterion field.

36.15.1.9. Disabling Monitoring

If your problem requires the solution of many equations (for example, turbulence quantities and multiple species), a plot that includes all residuals may be difficult to read. In such cases, you may choose to monitor only a subset of the residuals, perhaps those that affect convergence the most. You can indicate whether or not you want to monitor residuals for each variable by enabling or disabling the relevant check box in the **Monitor** list of the Residual Monitors Dialog Box (p. 5599).

36.15.1.10. Plot Parameters

If you choose to plot the residual values (either interactively during the solution or using the **Plot** button after calculations are complete), there are several display parameters you can modify.

In the **Window** field under **Options**, you can specify the ID of the graphics window in which the plot will be drawn. When Ansys Fluent is iterating, the active graphics window is temporarily set to this window to update the residual plot, and then returned to its previous value. Thus, the residual plot can be maintained in a separate window that does not interfere with other graphical postprocessing.

You can modify the number of residual history points to be displayed in the plot by changing the **Iterations to Plot** entry under **Options**. If you specify n points, Ansys Fluent will display the last n history points. Since the y axis is scaled by the minimum and maximum values of all points in the plot, you can zoom in on the end of the residual history by setting **Iterations to Plot** to a value smaller than the number of iterations performed. If, for example, the residuals jumped early in the calculation when you turned on turbulence, that peak broadens the overall range in residual values, making the smaller fluctuations later on almost indistinguishable. By setting the value of **Iterations to Plot** so that the plot does not include that early peak, your y -axis range is better suited to the values that you are interested in seeing. For more information on residual history points, refer to the discussion of storing residual history points, described earlier in this section.

You can also modify the attributes of the plot axes and the residual curves. Click the **Axes...** or **Curves...** button to open the Axes Dialog Box (p. 5313) or Curves Dialog Box (p. 5316). See Axes Dialog Box (p. 5313) and Curves Dialog Box (p. 5316) for details.

Important:

Note that entering a value for **Iterations to Plot** does not necessarily mean *solved* iterations but rather *stored* (or sampled) data points. Note also that the frequency of the data storage will diminish towards the start of the solution as the number of solved iterations increases. Due to this, whenever the *stored* iterations is greater than the *solved* iterations, if you plot *n* iterations, you actually see a history that goes back further than *n* solved iterations.

36.15.1.11. Postprocessing Residual Values

If you are having solution convergence difficulties, it is often useful to plot the residual value fields (for example, using contour plots) to determine where the high residual values are located. By default,

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

only the time step (for the density-based solver) and the mass imbalance (for the pressure-based solver) in each cell is available.

If you want to plot residual value fields for additional residual value fields, you will need to do the following:

1. Read in the case and data files of interest (if they are not already in the current session).
2. Use the following text command to enable the saving of residual values.

   `solve` → `set` → `advanced` → `retain-cell-residuals`

3. Perform at least one iteration.

The solution variables for which residual values are available will appear in the **Residuals...** category in the postprocessing dialog boxes. Note that residual values are *not* available for the radiative transport equations solved by the discrete ordinates radiation model.

Important:

The residual values that are made available for postprocessing are the unscaled values.

36.15.2. Monitoring Statistics

If you are solving a fully-developed periodic flow, you may want to monitor the pressure gradient or the bulk temperature ratio, as discussed in Periodic Flows (p. 1747).

If you are solving an unsteady flow (especially if you are using the explicit time stepping option), you may want to monitor the "time" that has elapsed during the calculation. The physical time of the flow field starts at zero when you initialize the flow. (See Performing Time-Dependent Calculations (p. 3643) for details about modeling unsteady flows.)

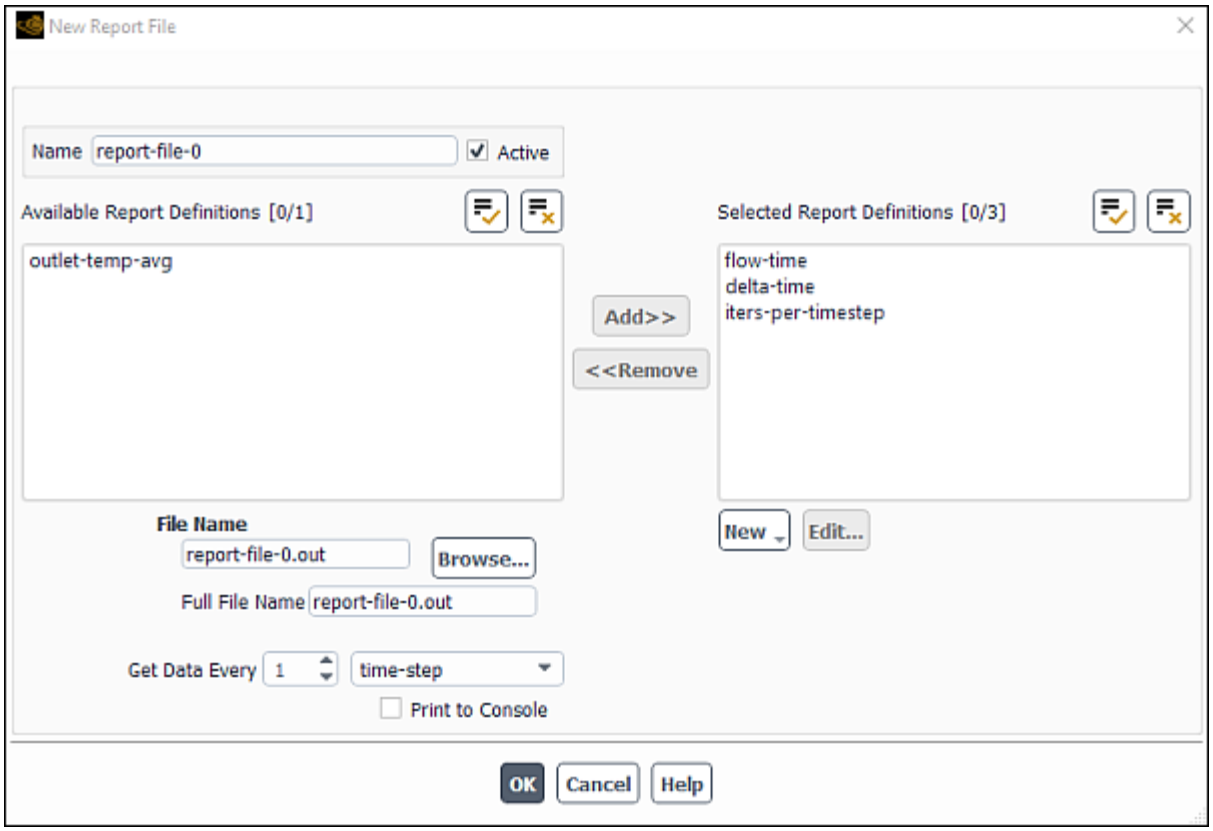
If you are using an adaptive time stepping method (described in Inputs for Time-Dependent Problems (p. 3645)), you may want to monitor the size of the time step, Δt.

You can create report files and plots for each of these quantities: flow-time, delta-time, iterations-per-timestep, periodic-pressure-gradient, and periodic-bulk-temperature-ratio. All can be included in a single report file; flow-time and delta-time can be included in the same report plot, as can periodic-pressure-gradient and periodic-bulk-temperature-ratio.

For information on setting up report files and plots, see Report Files and Report Plots (p. 4123).

Solution → **Monitors** → **Report File** **New...**

Figure 36.39: Report File for 'flow-time', 'delta-time', and 'iters-per-timestep'



36.15.3. Monitoring Solution Quantities

Solution quantities for which Report Definitions have been created can be monitored throughout the solution to aid in judging convergence. You can setup your case file so that force and moment coefficients and surface and volume integrals are computed and stored at the end of every iteration (for steady-state solutions) or time step (for transient solutions), and thereby create a convergence history. You can also choose to compute the value averaged over several iterations/time steps. Running averages can also help you determine if you have reached convergence for a solution that is oscillating or irregular. You can print and plot and save this convergence data to an external file using Report Files and Report Plots. The external file is written in the report file format described in Report Files and Report Plots (p. 4123).

Report Files and Report Plots are accessible under the **Monitors** node in the tree.

Solution → Monitors → Report Files or Report Plots

For additional information on setting up report definitions and report files and plots, see Monitoring and Reporting Solution Data (p. 4098) and Creating Report Definitions (p. 4098).

36.16. Convergence Conditions

The convergence conditions facility allows you to set convergence conditions on the solution that are based on the values from report definitions (surface, volume, lift, drag, and so on). There are two choices that you must make for declaring convergence using report definitions:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Do you want to check for convergence at every iteration or at every timestep? Make your choice in the **Check For** group box.

  – **Solution Convergence**

    Fluent checks for solution convergence at every time step.

  – **Time Step Convergence**

    Fluent checks for solution convergence at every iteration.

- Do you consider the solution converged if *all* of the active report definitions' criteria and enabled residuals are satisfied or *any* of the active report definitions' criteria or enabled residuals is satisfied? Make your choice in the **Choose Condition** group box.

  – **All Conditions are Met**

    The solution is considered converged when all of the active report definitions' criteria and enabled residuals are satisfied.

  – **Any Condition is Met**

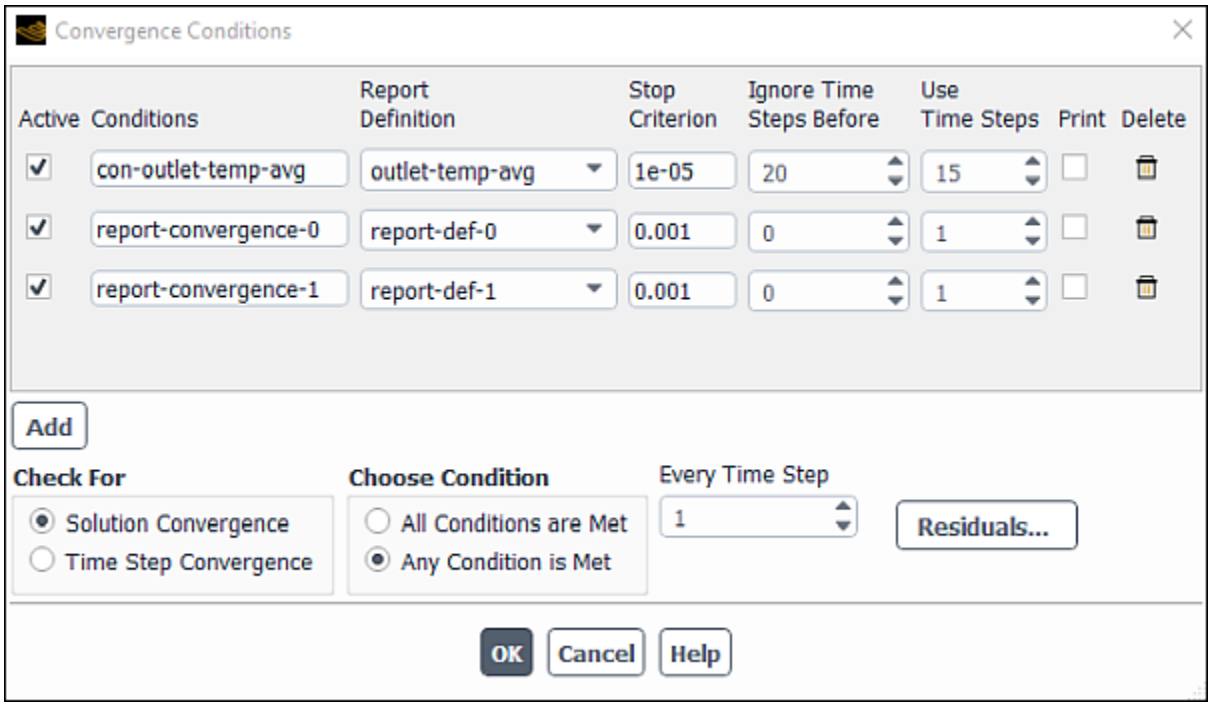
    The solution is considered converged when any of the active report definitions criteria and enabled residuals is satisfied.

By default, the solution convergence will happen if either report definition convergence criteria AND residuals convergence criteria is satisfied. If you want to rely only on the report definition values to decide convergence, clear the **Check Convergence** check boxes in the **Residual Monitors** dialog box.

To open the **Convergence Conditions** dialog box, click the **Convergence...** button in the **Solution** ribbon tab (**Reports** group box).

Solution → **Reports** → **Convergence...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Active** column of check boxes (on the left) allows you to deactivate/activate convergence conditions.

36.16.1. Setting Up the Convergence Conditions Dialog Box

Create new convergence conditions by clicking **Add** and selecting a **Report Definition** from the drop-down list.

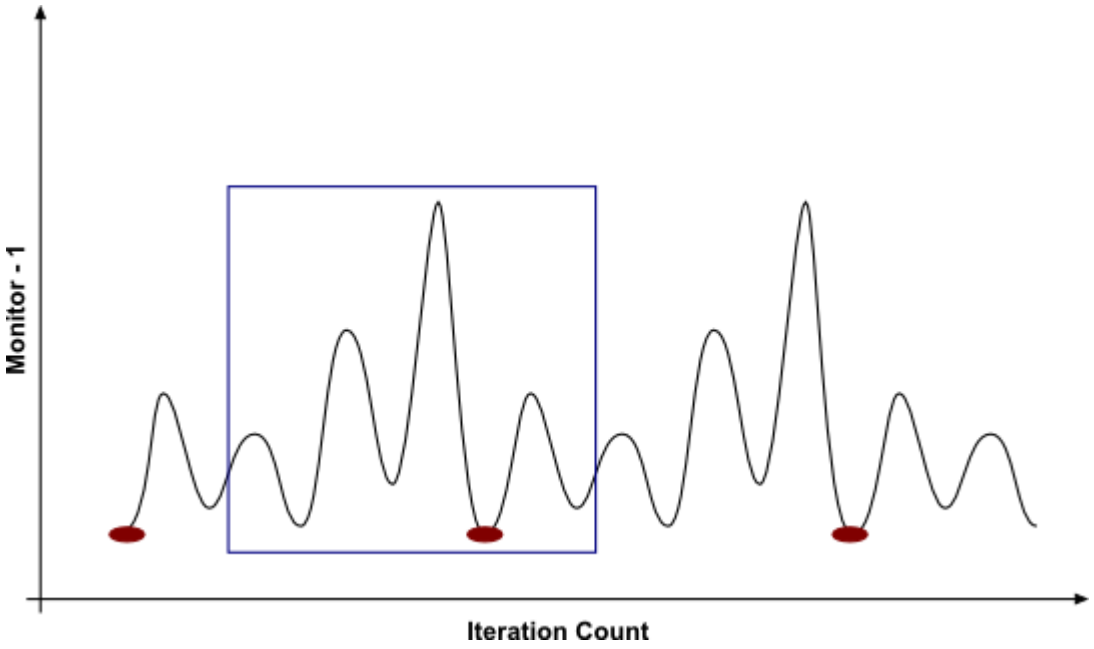
To create meaningful conditions for determining convergence from report definitions, set the levels of the convergence condition variables according to the following directions.

Enter a value in the **Ignore Iterations Before | Ignore Time Steps Before** column if you expect your solution to fluctuate in the first few iterations/time steps. Enter a value that represents the number of iterations/time steps you anticipate the fluctuations to continue. The calculation will begin after this number of iterations/time steps has finished.

Use the **Use Iterations | Use Time Steps** setting to select the number of previous iterations/time steps to be included in the monitor convergence check. For fluctuating simulations like Figure 36.40: Fluctuating Simulation Example (p. 3699), this number should be high enough to counteract the effect of the fluctuations.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 36.40: Fluctuating Simulation Example



The **Stop Criterion** indicates the criterion below which the solution is considered to be converged. The value of **Stop Criterion** is calculated as follows:

Res-m(1) = [abs (m(n) - m(n-1))]/m(n)

Res-m(2) = [abs (m(n) - m(n-2))]/m(n)

Res-m(3) = [abs (m(n) - m(n-3))]/m(n)

.

.

.

Res-m(Np) = [abs (m(n) - m(n-Np))]/m(n)

Where:

Res-m is the report definition residual

n is the iteration/time step number

m(n) is the value of the report definition at the n th iteration/time step

Np is the number of previous iterations/time steps to consider (**Use Iterations** / **Use Time Steps**)

The report definition residual is the absolute variance of the monitored quantity over the last Np iterations/time steps, divided by the current value of the monitored quantity.

If the maximum value of all of the Res-m values is less than the **Stop Criterion** number, the solution is considered to be converged at the n th iteration/time step. If the maximum value of all of the Res-

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

m values is greater than or equal to the **Stop Criterion** number, the calculation will proceed to the next iteration/time step.

Select the **Print** check box to print report definition residuals in the Ansys Fluent console. These residuals are calculated as follows with respect to their immediate previous value only.

Pres-m(n) = [abs (m(n) - m(n-1))]/m(n)

Where:

Pres- m(n) is the printed residual value for a report definition

n is the iteration that is currently printed in the console

m(n) is the value of the monitor at the n th iteration/time step

Therefore:

Pres-m(n +1) = [abs (m(n+1) - m(n))]/m(n+1)

The printed value at the (n+1)th iteration/time step is calculated with respect to the (n)th iteration/time step and the printed value at the (n)th iteration/time step is calculated with respect to the (n-1)th iteration/time step.

36.17. Executing Commands During the Calculation

As described in Monitoring Solution Convergence (p. 3684) and Animating the Solution (p. 3714), respectively, you can report and monitor various quantities (for example, residuals, force coefficients) and create animations of the solution while the solver is performing calculations. Ansys Fluent also includes a feature that allows you to define your own command(s) to be executed during the calculation at specified intervals. For example, you can ask Ansys Fluent to adapt the mesh based on gradient field value cell registers after a set number of iterations. You will specify a series of text commands or use the GUI to define the steps to be performed.

Important:

Note that the **Calculation Activities** task page provides options to perform the following during the calculation:

- save case and data files
- export transient solution files
- export transient particle history data files

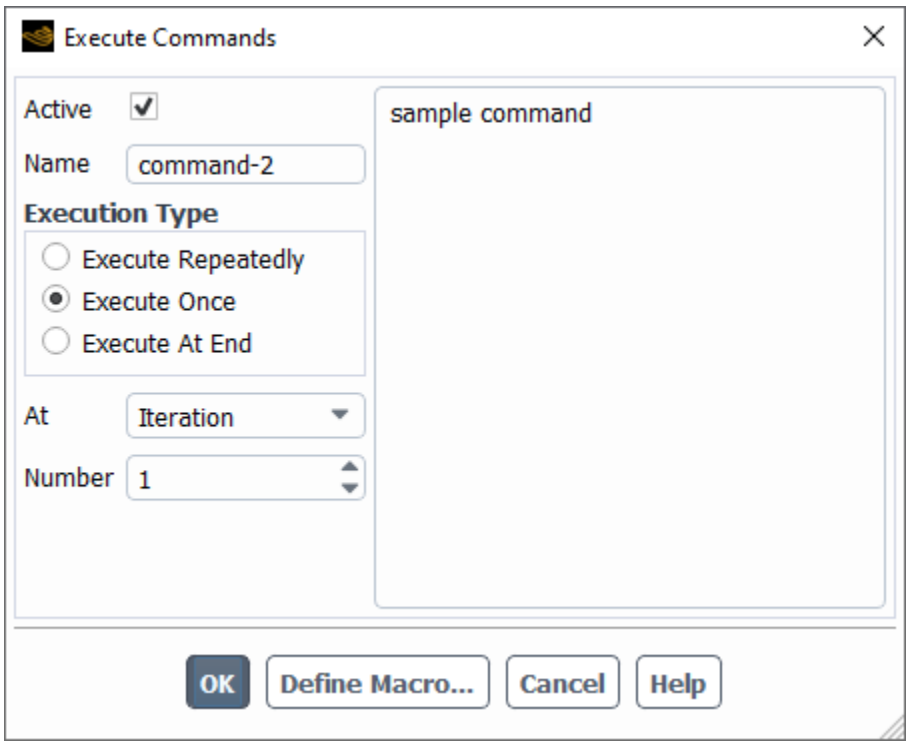
Each of these options has their own dialog box, which should be used rather than executing a command to perform them. See Automatic Saving of Case and Data Files (p. 958) and Exporting Data During a Transient Calculation (p. 1001) for details.

You will indicate the command(s) that you want the solver to execute at specified intervals during the calculation using the Execute Commands Dialog Box (p. 5201) (Figure 36.41: The Execute Commands Dialog Box (p. 3701)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution → Calculation Activities → Execute Commands New...

Figure 36.41: The Execute Commands Dialog Box



The procedure is as follows:

1. Enable the **Active** check button if you want it to be executed during the calculation. You may define multiple commands and choose to use only a subset of them by turning off the check button for those that you do not want to use.

2. Enter a **Name** for the command.

3. Select **Execute Repeatedly** if the command(s) are to be repeated as part of a sequence.

   Indicate how often you want the command to be executed by setting the interval under **Every** and selecting **Iteration**, **Time Step**, **Flow Time**, **Adjoint Iteration**, or **Design Iteration** in the drop-down list below **When**. (**Time Step** and **Flow Time** are only valid choices if you are calculating unsteady flow, and **Adjoint Iteration** and **Design Iteration** are only available if you have opened the **Execute Commands** dialog box from the **Gradient-Based Optimizer** or **Run Adjoint Calculation** dialog box.) For example, to execute the command every 10 iterations, you would enter `10` under **Every** and select **Iteration** under **When**.

   Important:

   If you specify an interval in iterations, be sure to keep the **Reporting Interval** in the Run Calculation Task Page (p. 5207) at its default value of 1.

4. Select **Execute Once** if the command(s) are only to be executed a single time.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Indicate how often the command is to be executed. You can enter the interval under **Number** and select **Iteration**, **Time Step**, **Flow Time**, **Adjoint Iteration**, or **Design Iteration** under **At**. (**Time Step** and **Flow Time** are only valid choices if you are calculating unsteady flow, and **Adjoint Iteration** and **Design Iteration** are only available if you have opened the **Execute Commands** dialog box from the **Gradient-Based Optimizer** or **Run Adjoint Calculation** dialog box.)

5. Select **Execute At End** if the command(s) are only to be executed at the end of the simulation.

6. Define the command by entering a series of text commands in the command text field on the right (using multiple lines if required), or by entering the name of a command macro you have defined (or will define) as described in Defining Macros (p. 3703).

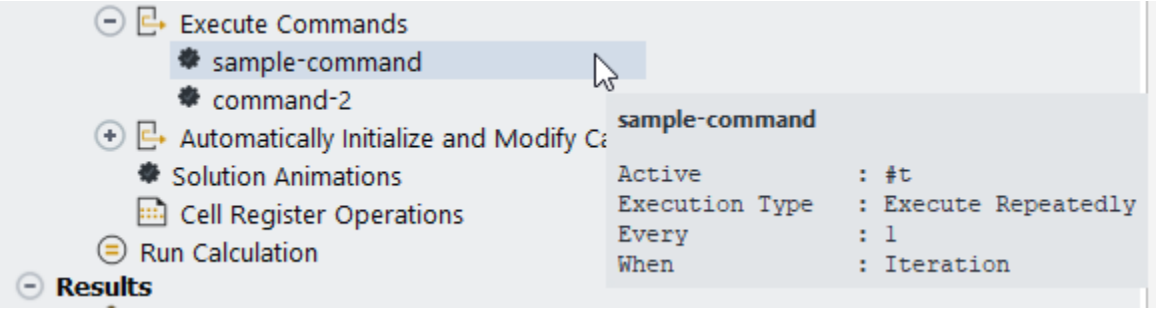
Important:

- If the command to be executed involves saving a file, see Saving Files During the Calculation (p. 3705) for important information.
- If the command enables or disables equations, you must disable the convergence check for these equations.

Note:

For any commands that are already defined in the Outline View, you can hover over them individually and Fluent will display a preview of the command properties.

Figure 36.42: Hovering Over Your Commands in the Outline View

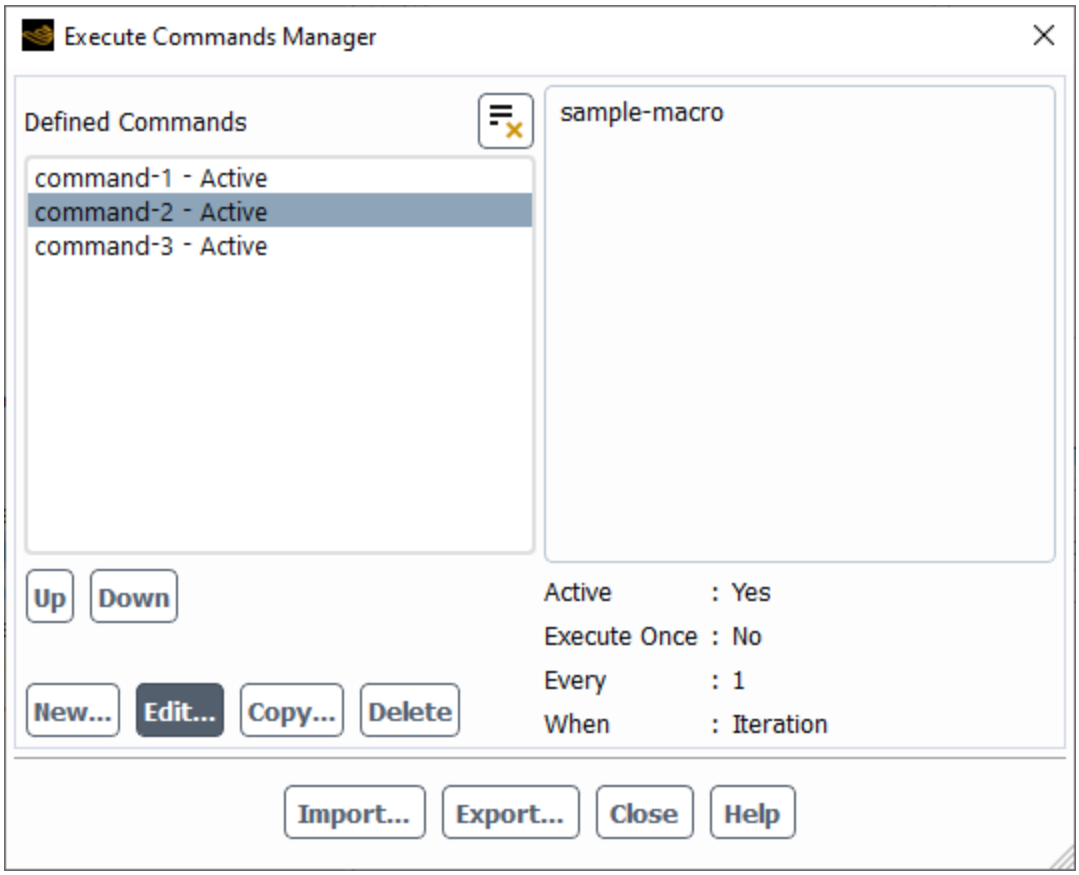


You can keep track of your defined commands using the Execute Commands Manager Dialog Box (p. 5200) (Figure 36.43: The Execute Commands Manager Dialog Box (p. 3703)) dialog box.

Solution → **Calculation Activities** → **Execute Commands** **Manage...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 36.43: The Execute Commands Manager Dialog Box



For additional information, see the following sections:

36.17.1. Defining Macros

36.17.2. Saving Files During the Calculation

36.17.1. Defining Macros

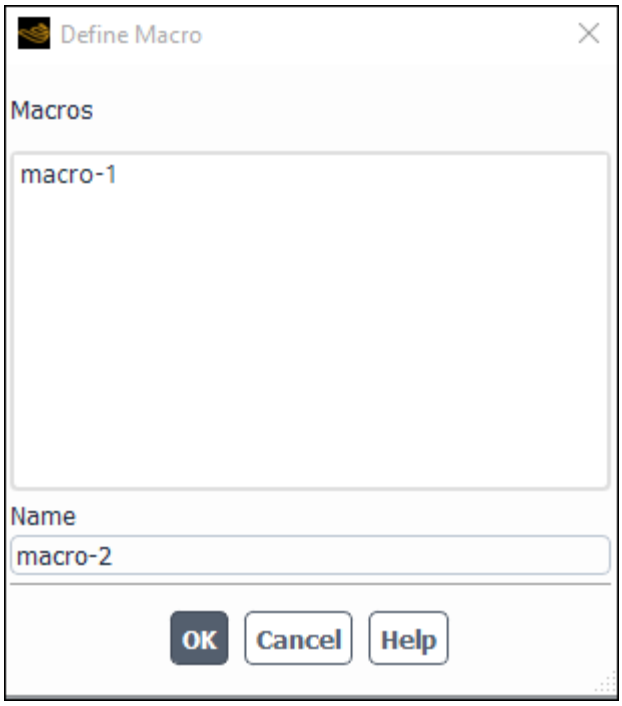
You can also use the macros that you define for automatic execution during the calculation, interactively during the problem setup or postprocessing. For example, if you define a macro that adapts the mesh after each iteration, you can also use the macro to perform this adaption interactively.

Definition of a macro is accomplished as follows:

1. In the Execute Commands Dialog Box (p. 5201), click the **Define Macro...** button to open the Define Macro Dialog Box (p. 5203) (Figure 36.44: The Define Macro Dialog Box (p. 3704)). Since this is a "modal" dialog box, the solver will not allow you to do anything else until you perform step 2, below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 36.44: The Define Macro Dialog Box



2. In the **Define Macro** dialog box, specify a **Name** for the macro (for example, `adapt1`) and click **OK**. (The **Define Macro...** button in the **Execute Commands** dialog box will become the **End Macro** button.)

3. Perform the steps that you want the macro to perform. For example, if you want the macro to perform adaption, open the **Manual Mesh Adaption** dialog box, specify the appropriate coarsening, refinement, and other parameters, and click **Adapt** to perform the adaption.

   Important:

   If the command to be executed involves saving a file, see Saving Files During the Calculation (p. 3705) for important information.

4. When you have completed the steps you want the macro to perform, click the **End Macro** button in the **Execute Commands** dialog box.

As noted above, once you have defined a macro for execution during the calculation, you can use it at any time. If you defined the macro called `adapt1` to adapt based on pressure gradient, you can simply type `adapt1` in the console to perform this adaption. This macro is independent of any text menus, so you need not move to a different text menu to use it. Macros can be saved to and read

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

from files. To save all macros that are currently defined, use the `file/write-macros` text command. To read all the macros in a macro file, use the `file/read-macros` text command.

Important:

A macro, like a journal file, is a simple record/playback function. It will therefore know nothing about the state in which it was recorded or the state in which it is being played back. You must be careful not to change directories while defining a macro. Also, you must be careful that all surfaces, variables, and so on that are used by the macro have been properly defined when you (or Ansys Fluent) invoke the macro.

36.17.2. Saving Files During the Calculation

If the command to be executed during the calculation involves saving a file, you should include a special character in the file name when you enter it in The Select File Dialog Box (p. 931) so that the solver will know to assign a new name to each file it saves. See Automatic Numbering of Files (p. 952) for details about these special characters for filenames.

Important:

Note that the **Calculation Activities** task page provides options to perform the following during the calculation:

- save case and data files
- export transient solution files
- export transient particle history data files

Each of these options has their own dialog box, which should be used rather than the **Execute Commands** dialog box. See Automatic Saving of Case and Data Files (p. 958) and Exporting Data During a Transient Calculation (p. 1001) for details.

36.18. Automatic Initialization of the Solution and Case Modification

While running a case manually, you can perform certain activities that may facilitate convergence. These actions may take place before initialization, after initialization and/or at other points during the calculation. The process described in this section allows you to enter text commands at each of these times when a case is run from the **Run Calculation** task page, Workbench, or in batch. In the **Calculation Activities** task page, enable **Automatically Initialize and Modify Case** to automatically modify the case.

Solution → **Calculation Activities** → ☑ **Automatically Initialize and Modify Case** → **Edit...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 36.45: The Automatically Initialize and Modify Case Option

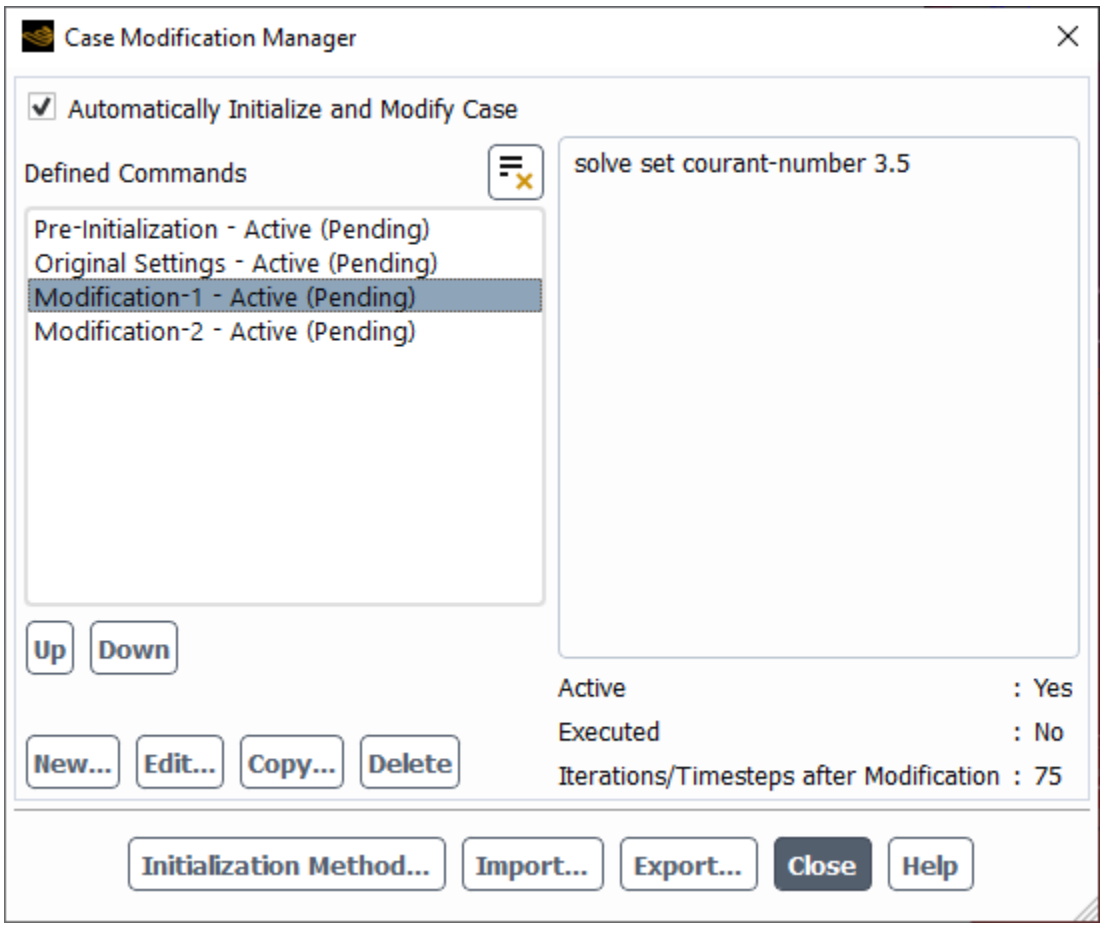
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When using this option, you can edit the calculation settings. Note that the original settings always exist and cannot be deleted. Immediately after enabling **Automatically Initialize and Modify Case**, you will notice that the **Calculation Activities** task page reports that the case's pre-initialization in not active and the original settings are active and pending.

You can now control the number of iterations or time steps for the calculation. When **Automatically Initialize Solution and Modify Case** option is disabled, you will have to specify the iterations or time steps using the **Run Calculation** task page.

For an uninitialized case, clicking the **Edit...** button will display the **Case Modification Manager** dialog box, which allows you to specify the initialization method and to modify the case.

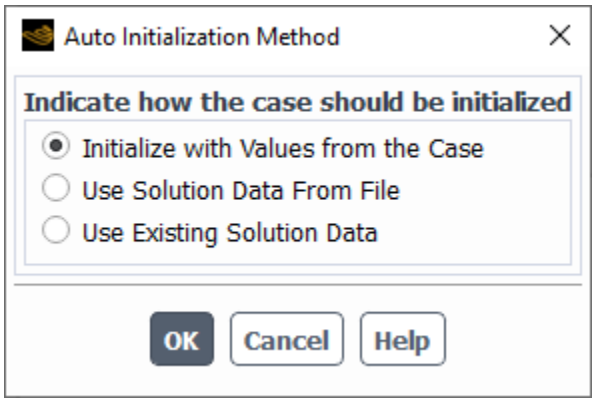
Figure 36.46: The Case Modification Manager Dialog Box With Pending Modifications



Click the **Initialization Method...** button to display the **Auto Initialization Method** dialog.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 36.47: The Auto Initialization Method Dialog Box



In the **Auto Initialization Method** dialog box, you can specify three different initialization methods:

Initialize with Values from the Case

uses the values set in the **Solution Initialization** task page.

> **Note:**
>
> Hybrid initialization is not performed for this option; even if you have selected **Hybrid Initialization** under the **Solution Initialization** task page.

Use Solution Data from File

requires you to read in a data file containing the desired initialization for this case.

Use Existing Solution Data

is analogous to changing the values in a case and continuing the calculation. However, the iteration counter will be reset to 0 so that the modifications can be applied. Use this method when no solution data exists, similar to the first run.

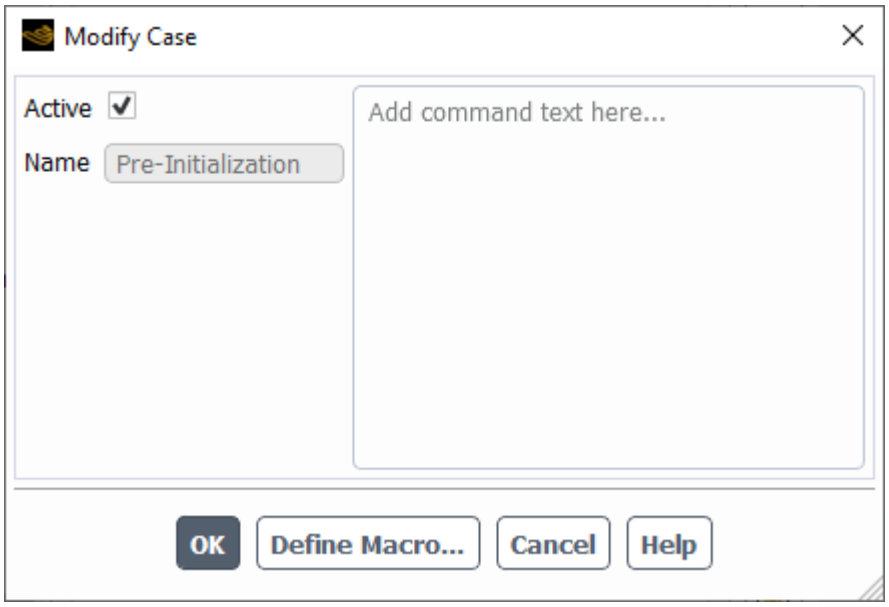
> **Important:**
>
> Whenever the case is initialized, the iteration count is set to 0.

In the **Case Modification Manager** dialog box, you can specify settings that you would like to apply before initialization, or you can change your original settings, or you create new settings.

- Select **Pre-Initialization** and click **Edit...** to open the **Modify Case** dialog box where you can make any pre-initialization modifications to the case settings. If **Pre-Initialization** is deactivated, you can enable it by selecting the **Active** checkbox and then you can enter the text commands accordingly in the field to the right.

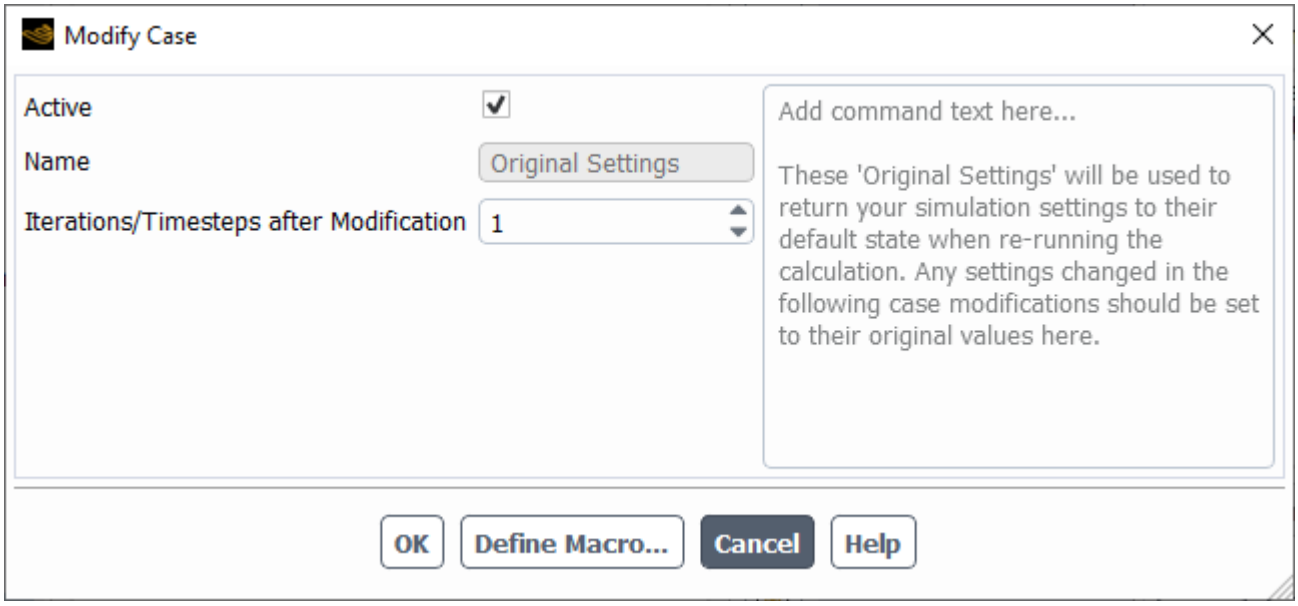
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 36.48: The Modify Case Dialog Box for Pre-Initialization



- Select **Original Settings** and click **Edit...** to open the **Modify Case** dialog box where you can make any modifications to the original case settings. If **Original Settings** is activated, you can enter the text commands in the field to the right, and/or specify the **Iterations/Time Steps after Modification**.

Figure 36.49: The Modify Case Dialog Box for Original Settings



- Click **New...** to open the **Modify Case** dialog box where you can make any modifications to the case settings.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 36.50: The Modify Case Dialog Box

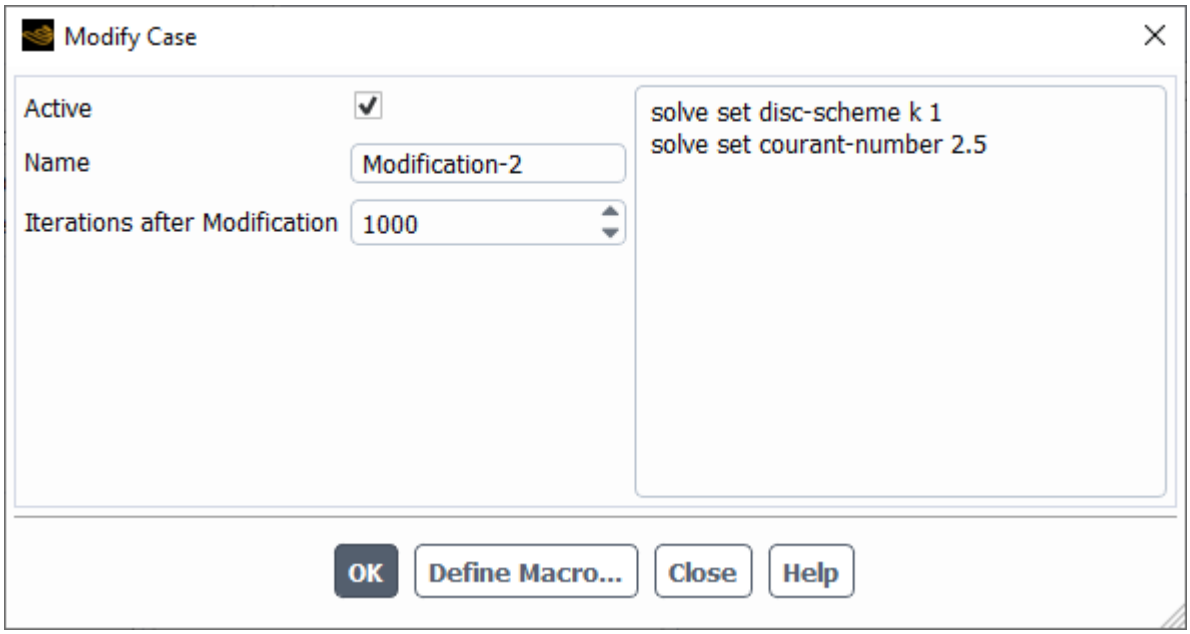
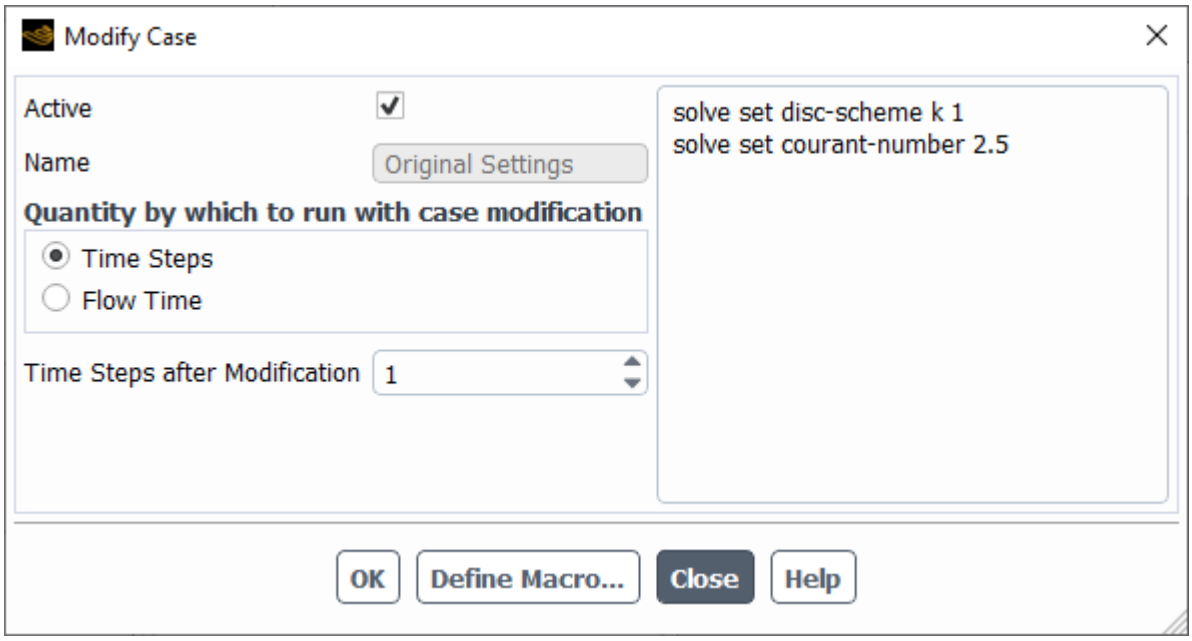


Figure 36.51: The Modify Case Dialog Box (Transient Cases)



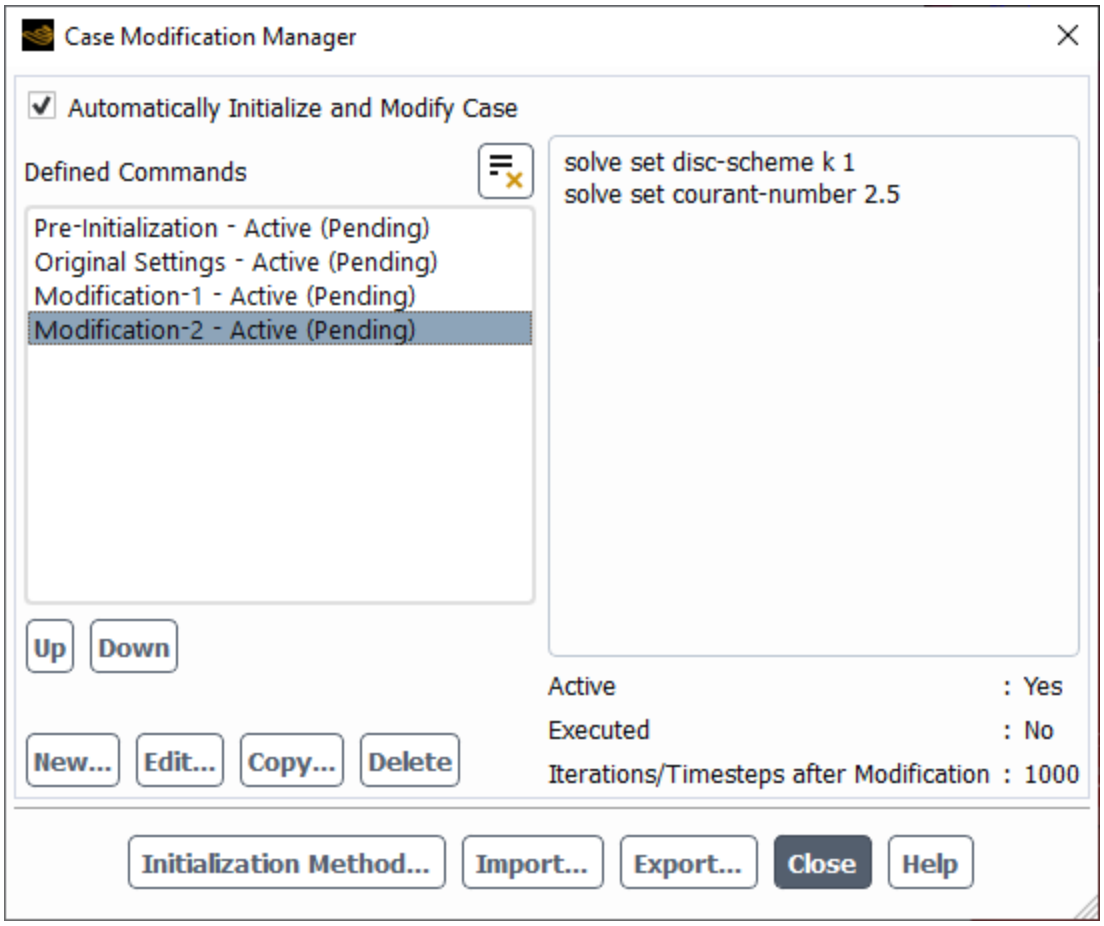
Important:

Note that some text commands require arguments (for example, numbers, file names, yes/no responses), which are requested through follow-up prompts in the console. When entering such text commands in the **Case Modification** dialog box, be sure to include your responses to the prompts. If you do not include these responses, the associated command will not be executed (unless it is a yes/no question, in which case Ansys Fluent will use `yes` by default). Also, when entering more than one text command, you can begin each text command with

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a /, or you can start a new line, as shown in Figure 36.52: The Case Modification Manager Dialog Box With Pending Modifications (p. 3711).

Figure 36.52: The Case Modification Manager Dialog Box With Pending Modifications



The settings shown in Figure 36.52: The Case Modification Manager Dialog Box With Pending Modifications (p. 3711) result in the following actions:

1. The default initial values for initialization are computed from the velocity-inlet zone **Velocity-inlet-5**.
2. The iteration count is set to 0 (in all situations) and the solution is initialized.
3. The calculation is run for 100 iterations or until convergence.
4. The Courant number is set to 3.5.
5. The calculation continues for 75 iterations or until convergence.
6. The discretization scheme for turbulent kinetic energy is set to second order upwind, and the Courant number is set to 2.5.
7. The calculation continues for 1000 iterations or until convergence.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can make the above changes sequentially and run your case, or you can specify them all at once. Change the order in which the commands are performed by selecting a command in the middle of the **Defined Commands** list and use the **Up** or **Down** buttons accordingly. Commands can also be copied and removed from the list using the **Copy** and **Delete** buttons. Use the **Export...** button to generate a tab-separated value file (`.tsv`) that contains your commands in their specified order. Pre-existing `.tsv` files can then later be imported using the **Import...** button.

When you have completed making the modifications, click **Close**. A **Question** dialog box may appear, prompting you to take specific actions. For example, if the **Original Settings** field is empty, then you may be notified that the original settings will be lost if the case is saved after the modifications are applied. It will prompt you for a response when asked if you would like to add commands that specify the original settings.

Important:

If you specify commands for the **Original Settings**, they will be applied to the case before the first iteration/time step.

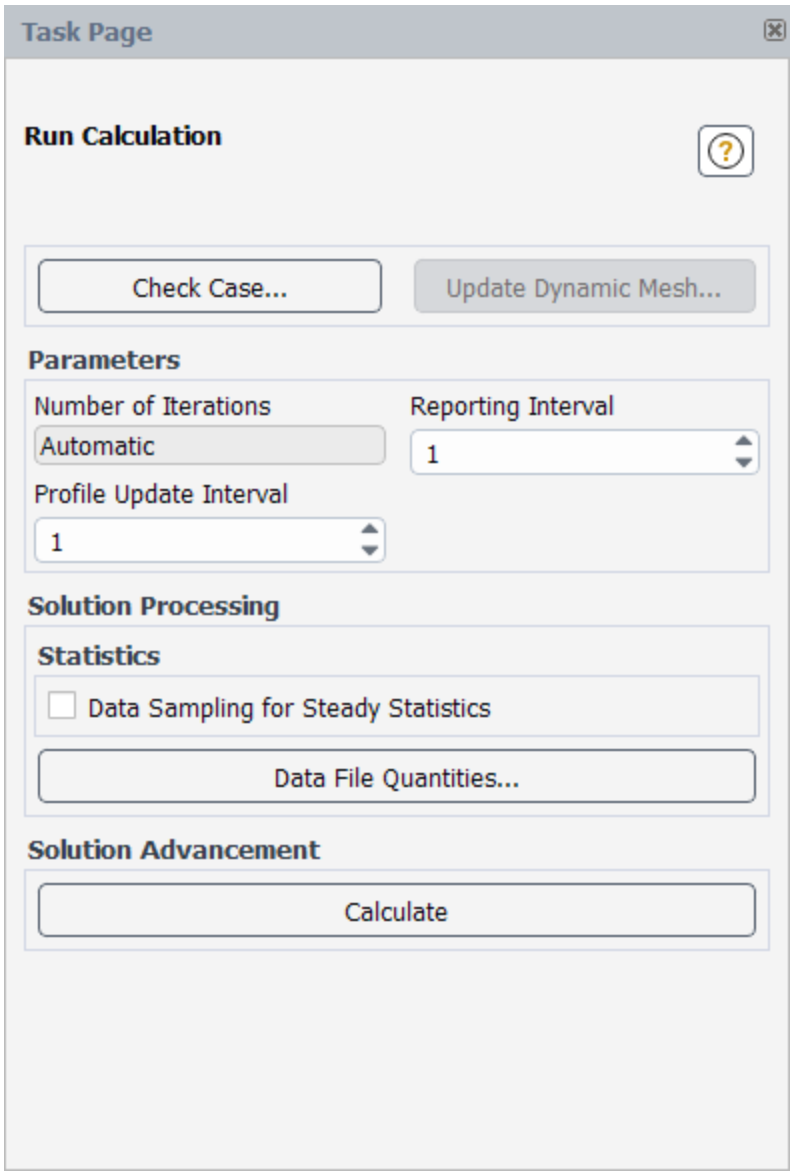
Your actions will be summarized in the **Calculation Activities** task page, as shown in Figure 36.45: The Automatically Initialize and Modify Case Option (p. 3706).

If you disable **Automatically Initialize and Modify Case**, the settings will be disabled and retained, but will not be applied to the case.

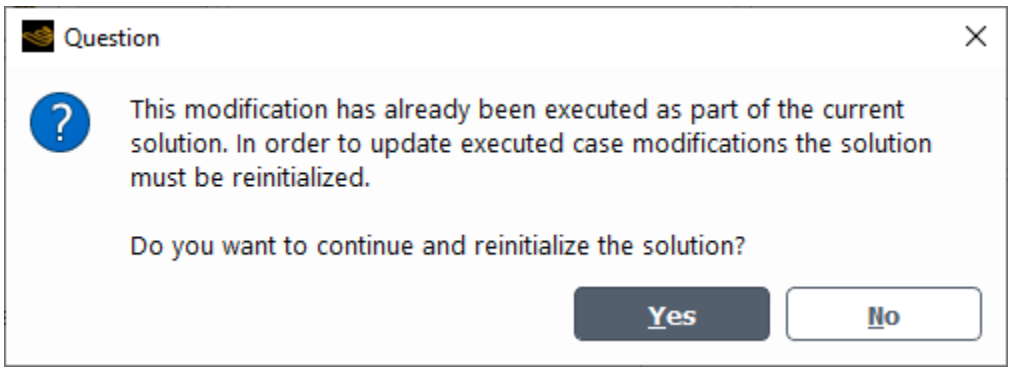
When **Automatically Initialize and Modify the Case** is enabled, settings defined in the **Run Calculation** task page will be ignored. Instead, the **Number of Iterations** will be defined as **Automatic**, as shown in Figure 36.53: The Run Calculation Task Page (p. 3713).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 36.53: The Run Calculation Task Page



If you decide to edit the solution initialization and case modification settings in the **Modify Case** dialog box, and one or more iterations have been calculated, then upon clicking **OK** in the dialog , Fluent will prompt you with a question about having already executed part of the solution and whether you want to continue.



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For additional information, see the following sections:

36.19. Animating the Solution

During the calculation, you can have Fluent create an animation of mesh (primarily for dynamic mesh cases), contours, vectors, pathlines, particle tracks, scenes, XY plots, report plots, residuals, or dashboards. Before you begin the calculation, you will specify and display the variables and types of plots you want to animate, and how often you want plots to be saved. At the specified intervals, Fluent will display the requested plots, and store each one. When the calculation is complete, you can play back the animation sequence, modify the view (for mesh, contour, vector, pathline, and particle tracks plots), if desired, and save the animation to a series of picture files or a video file (MP4).

Instructions for defining a solution animation sequence are provided in Creating an Animation Definition (p. 3714). Playing an Animation Sequence (p. 3717) describes how to play back and save the animations.

For additional information, see the following sections:

36.19.1. Creating an Animation Definition

36.19.2. Playing an Animation Sequence

36.19.3. Saving an Animation Sequence

36.19.4. Reading an Animation Sequence

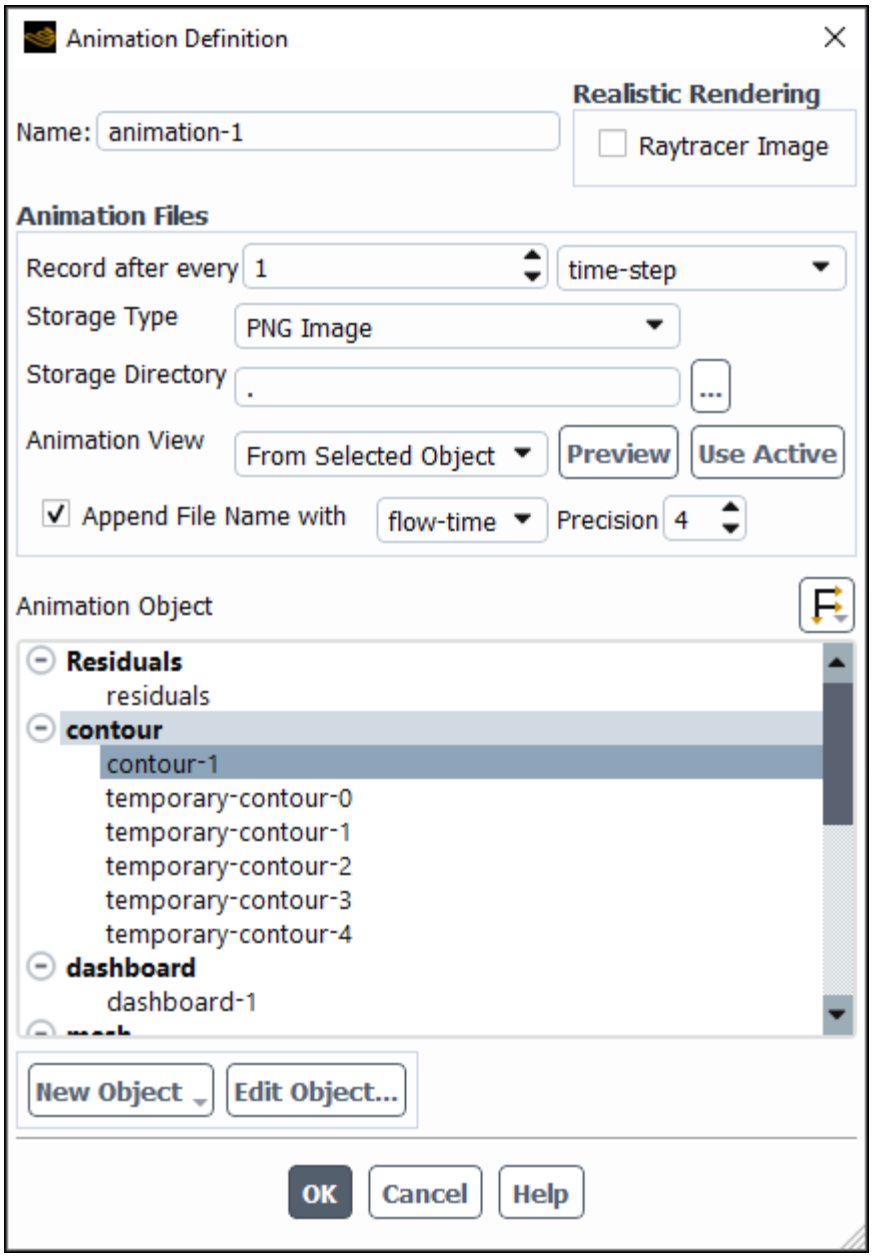
36.19.1. Creating an Animation Definition

You can use Figure 36.54: The Animation Definition Dialog Box (p. 3715) to specify the animation object and the frequency that it will be captured for animating.

Solution → **Activities** → **Create** → **Solution Animations...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 36.54: The Animation Definition Dialog Box



The procedure is as follows:

1. (Optional) Enter a **Name** for the animation definition.

2. (Optional) Enable **Raytracer Image** in the **Realistic Rendering** group box if you want a raytracer to render the image prior to saving. Refer to Realistic Rendering Using Raytracing (p. 3995) for additional information.

3. Indicate how often you want to create a new frame in the animation by specifying a frequency in **Record after every** and selecting **Iteration**, **Time Step**, or **Flow Time**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Specify how you want Fluent to save the animation frames by selecting **In Memory**, **PPM Image** (2D image file), **HSF File** (3D image file), **PNG Image** (2D image file), **JPEG Image** (2D image file), or **TIFF Image** (2D image file) from the **Storage Type** drop-down.

   Important:

   The advantage to saving the animation sequence using the **HSF File** option is that these files are highly compressible and can be viewed and interacted with using a "HOOPS Viewer" application on iOS and Android equipped devices. The "HOOPS Viewer" application can be downloaded to iOS and Android equipped devices from the "App Store" and "Google Play" respectively.

   An advantage to saving the animation sequence using the **PPM Image** option is that you can use the separate pixmap image files for the creation of a single GIF file. GIF file creation can be done quickly with graphics tools provided by other third-party graphics packages such as ImageMagick, that is, `animate` or `convert`. For example, if you save the PPM files starting with the string `sequence-2`, and you are using the ImageMagick software, you can use the `convert` command with the `-adjoin` option to create a single GIF file out of the sequence using the following command.

   ```
   convert -adjoin sequence-2_00*.ppm sequence2.gif
   ```

5. Choose the **Storage Directory** where Fluent will save the animation frames.

   Note:

   Animation files are written to the same directory as the case file, by default. This is stored as a relative path, represented by a dot (.), so that when a case file is shared, animation files are written to an appropriate location.

   If you manually specify a path, then what you are specifying is an absolute path that will not automatically be updated if the case file is shared or run from another directory.

6. (Optional) Specify the **Animation View**, which determines the orientation of the object for the animation frames captured during the calculation.

   *You can only specify the view once an **Animation Object** is selected*. Clicking **Preview** displays the selected object in a reserved graphics window, with the view shown in the drop-down. If your desired view is not available in the drop-down, you can preview the object, change its orientation in the graphics window, then click **Use Active** to save the current graphics window view and use it for this animation definition.

7. (Optional) Enable **Append File Name With** and select **time-step**, **iteration**, or **flow-time** to append a more meaningful number to the animation image names. *For **flow-time** you can control the **Precision**, which determines how many significant digits are included in the appended number.*

8. Select the graphics object you want to animate from the **Animation Object** list. If you want to animate an object that is not listed, you can create a **New Object**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Mesh...** opens the Mesh Display Dialog Box (p. 4688).
- **Contours...** opens the Contours Dialog Box (p. 5412).
- **Vectors...** opens the Vectors Dialog Box (p. 5655).
- **Pathlines...** opens the Pathlines Dialog Box (p. 5572).
- **Particle Tracks...** opens the Particle Tracks Dialog Box (p. 5558).
- **Scene...** opens Figure 40.50: The Scene Dialog Box (p. 3936).
- **XY Plot...** opens the Solution XY Plot Dialog Box (p. 5290).
- **Report Plot...** opens the New Report Plot Dialog Box (p. 5552).

Note:

If you select **residuals**, ensure that **Plot** is enabled in the **Residual Monitors** dialog box.

---

9. Click **OK** to create the animation definition.

36.19.1.1. Guidelines for Creating an Animation Definition

If you are defining an animation sequence containing mesh, contour, or vector displays, note the following when you are defining the display:

- If you want to include lighting effects in the animation frames, be sure to define the lights before you begin the calculation. See Adding Lights (p. 3984) for information about adding lights to the display.
- If you want to maintain a constant range of colors in a contour or vector display, you can specify a range explicitly by turning off the **Auto Range** option in the **Contours** or **Vectors** dialog box. See Specifying the Range of Magnitudes Displayed (p. 3902) or Specifying the Range of Magnitudes Displayed (p. 3917) for details.
- Advanced scene manipulations that are specified using the **Scene Description** dialog box will *not* be included in the animation sequence frames. View modifications such as mirroring across a symmetry plan *will* be included.
- For animations that include titles (enabled by clicking in the toolbar), ensure titles are displayed in the graphics window prior to beginning the computation, which will help the animation frames appear correctly.

36.19.2. Playing an Animation Sequence

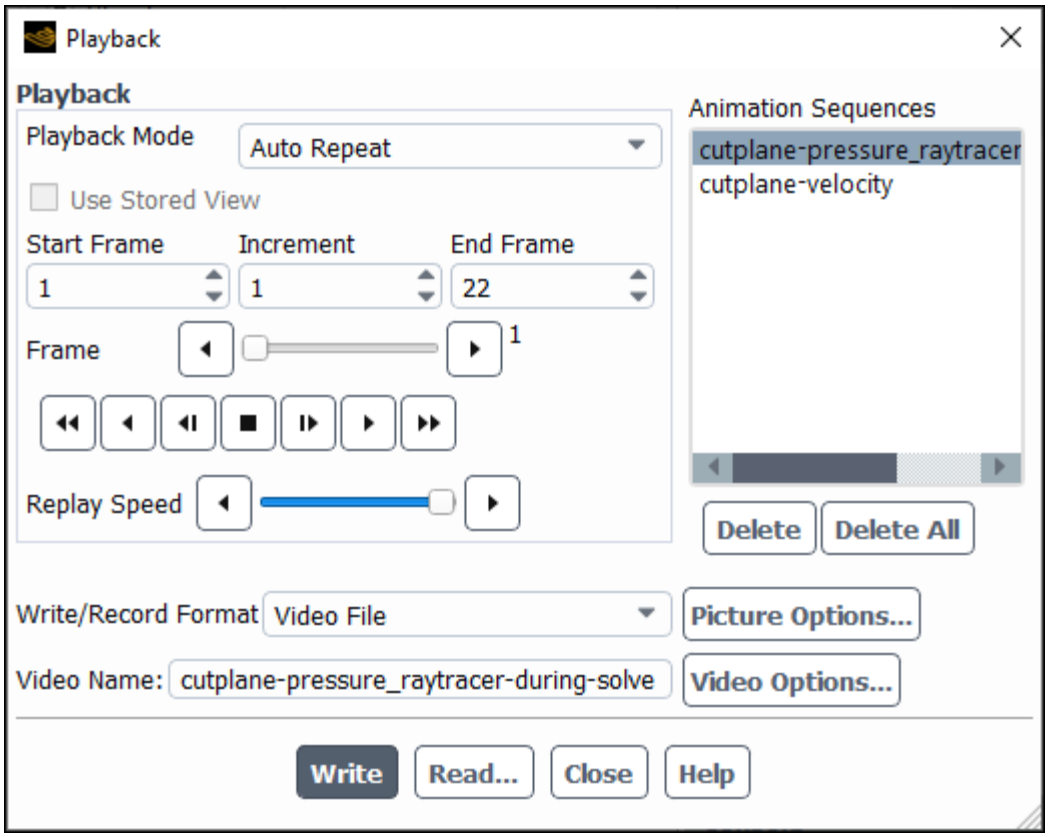
Once you have defined a sequence (as described in Creating an Animation Definition (p. 3714)) and performed a calculation, or read in a previously created animation sequence (as described in Reading

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

an Animation Sequence (p. 3725)), you can play back the sequence using the Playback Dialog Box (p. 5257) (Figure 36.55: The Playback Dialog Box (p. 3718)).

Results → Animations → Solution Animation Playback Edit...

Figure 36.55: The Playback Dialog Box



Select the animation you want to play in the **Animation Sequences** list. To play the animation once through from start to finish, click the "play" button under the **Playback** heading. (The buttons function in a way similar to those on a standard video player. "Play" is the second button from the right—a single triangle pointing to the right.) To play the animation backwards once, click the "play reverse" button (the second from the left—a single triangle point to the left). As the animation plays, the **Frame** scale shows the number of the frame that is currently displayed, as well as its relative position in the entire animation. If, instead of playing the complete animation sequence, you want to jump

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

to a particular frame, move the **Frame** slider bar to the desired frame number, and the frame corresponding to the new frame number will be displayed in the graphics window.

Important:

For smoother animations, enable the **Double Buffering** option in the Display Options Dialog Box (p. 5262) (see Modifying the Rendering Options (p. 3987)). This will reduce screen flicker during graphics updates.

Note:

For **HSF File** and **In Memory** 3D animations, you can disable **Use Stored View** to manipulate the view during playback.

Additional options for playing back animations are described below.

36.19.2.1. Modifying the View

If you want to replay the animation sequence with a different view of the scene, you can use your mouse to modify (for example, translate, rotate, zoom) it in the graphics window where the animation is displayed. Note that any changes you make to the view for an animation sequence will be lost when you select a new sequence (or reselect the current sequence) in the **Sequences** list.

36.19.2.2. Modifying the Playback Speed

Different computers will play the animation sequence at different speeds, depending on the complexity of the scene and the type of hardware used for graphics. You may want to slow down the playback speed for optimal viewing. Move the **Replay Speed** slider bar to the left to reduce the playback speed (and to the right to increase it).

36.19.2.3. Playing Back an Excerpt

You may sometimes want to play only one portion of a long animation sequence. To do this, you can modify the **Start Frame** and the **End Frame** under the **Playback** heading. For example, if your animation contains 50 frames, but you want to play only frames 20 to 35, you can set **Start Frame** to 20 and **End Frame** to 35. When you play the animation, it will start at frame 20 and finish at frame 35.

36.19.2.4. "Fast-Forwarding" the Animation

You can "fast-forward" or "fast-reverse" the animation by skipping some of the frames during playback. To fast-forward the animation, you will need to set the **Increment** and click the fast-forward button (the last button on the right—two triangles pointing to the right). If, for example, your **Start Frame** is 1, your **End Frame** is 15, and your **Increment** is 2, when you click the fast-forward button, the animation will show frames 1, 3, 5, 7, 9, 11, 13 and 15. Clicking on the fast-reverse button (the first button on the left—two triangles pointing to the left) will show frames 15, 13, 11,...1.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.19.2.5. Continuous Animation

If you want the playback of the animation to repeat continuously, there are two options available. To continuously play the animation from beginning to end (or from end to beginning, if you use one of the reverse play buttons), select **Auto Repeat** in the **Playback Mode** drop-down list. To play the animation back and forth continuously, reversing the playback direction each time, select **Auto Reverse** in the **Playback Mode** drop-down list.

To turn off the continuous playback, select **Play Once** in the **Playback Mode** list. This is the default setting.

36.19.2.6. Stopping the Animation

To stop the animation during playback, click the "stop" button (the square in the middle of the playback control buttons). If your animation contains very complicated scenes, there may be a slight delay before the animation stops.

36.19.2.7. Advancing the Animation Frame by Frame

To advance the animation manually frame by frame, use the third button from the right (a vertical bar with a triangle pointing to the right). Each time you click this button, the next frame will be displayed in the graphics window. To reverse the animation frame by frame, use the third button from the left (a left-pointing triangle with a vertical bar). Frame-by-frame playback allows you to freeze the animation at points that are of particular interest.

36.19.2.8. Deleting an Animation Sequence

If you want to remove one of the sequences that you have created or read in, select it in the **Sequences** list and click the **Delete** button. If you want to delete all sequences, click the **Delete All** button.

Important:

- Note that if you delete a sequence that has not yet been saved to disk (that is, if you selected **In Memory** under **Storage Type** in the **Animation Sequence** dialog box), it will be removed from memory permanently. If you want to keep any animation sequences that are stored only in memory, you should be sure to save them (as described in Saving an Animation Sequence (p. 3720)) before you delete them from the **Sequences** list or exit Ansys Fluent.
- Animations sequences with a corresponding animation definition must be deleted by deleting the corresponding animation definition object—they cannot be removed by clicking **Delete**. *Right-click an animation definition in the Outline View tree to see the **Delete** option (**Solution**>**Calculation Activities**>**Solution Animations**).*

36.19.3. Saving an Animation Sequence

Once you have created an animation sequence, you can save it in any of the following formats:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Solution animation file containing the Ansys Fluent metafiles
- Picture files, each containing a frame of the animation sequence
- Video file containing each frame of the animation sequence. Available video formats include MP4, AVI, FLV, MOV, and MPEG.

  Note that MPEG is not supported for saving on Ubuntu operating systems.

Note that, if you are saving picture files or a video file, you can modify the view (for example, translate, rotate, zoom) in the graphics window where the animation is displayed, and save the modified view instead of the original view.

36.19.3.1. Solution Animation File

If you selected **HSF File** or **PPM Image** for **Storage Type** in Figure 36.54: The Animation Definition Dialog Box (p. 3715), then Fluent will save the solution animation file for you automatically. It will be saved in the specified **Storage Directory**, and its name will be the **Name** you specified for the sequence, with a `.cxa` extension (for example, `pressure-contour.cxa`). In addition to the `.cxa` file, Fluent will also save image files with `.hsf` or `.ppm` extensions for each frame (for example, `pressure-contour_0002.hsf`). The `.cxa` file contains a list of the associated `.hsf` or `.ppm` files, and tells Fluent the order in which to display them.

If you selected **In Memory** under **Storage Type**, then the solution animation file (`.cxa`) and the associated files (`.hmf`) will be lost when you exit from Ansys Fluent, unless you save them as described below.

You can save the animation sequence to a file that can be read back into Ansys Fluent (see Reading an Animation Sequence (p. 3725)) when you want to replay the animation. As noted in Reading an Animation Sequence (p. 3725), the solution animation file can be used for playback in Fluent independent of the case and data files that were used to generate it.

To save a solution animation file (and the associated files), select **Animation Frames** in the **Write/Record Format** drop-down list in the **Playback** dialog box, and click the **Write** button. Ansys Fluent will save a `.cxa` file, as well as a `.hmf` file for each frame of the animation sequence. The filename for the `.cxa` file will be the specified sequence **Name** (for example, `pressure-contour.cxa`), and the file names for the metafiles will consist of the specified sequence **Name** followed by a frame number (for example, `pressure-contour_0002.hmf`). All of the files (`.cxa` and `.hmf`) will be saved in the current working directory.

36.19.3.2. Picture File

You can also generate a picture file for each frame in the animation sequence. This feature allows you to save your sequence frames to picture files used by an external animation program such as ImageMagick. As noted above, you can modify the view in the graphics window before you save the picture files.

To save the animation as a series of picture files, follow these steps:

1. Select **Picture Files** in the **Write/Record Format** drop-down list in the **Playback** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. If necessary, click the **Picture Options...** button to open the Save Picture Dialog Box (p. 5254) and set the appropriate parameters for saving the picture files. (If you are saving picture files for use with ImageMagick, for example, you may want to select the window dump format. See Window Dumps (Linux Systems Only) (p. 1025) **Apply** in the **Save Picture** dialog box to save your modified settings.

   Important:

   - Do not click the **Save...** button in the **Save Picture** dialog box. You will save the picture files from the **Playback** dialog box in the next step.
   - The **AVZ** and **VRML** formats are not supported for writing solution animation picture files.

3. In the **Playback** dialog box, click the **Write** button. Ansys Fluent will replay the animation, saving each frame to a separate file. The filenames will consist of the specified sequence **Name** followed by an animation sequence and a frame number (for example, `pressure-contour_1_0002.ps`), and they will all be saved in the current working directory.

36.19.3.3. Video File

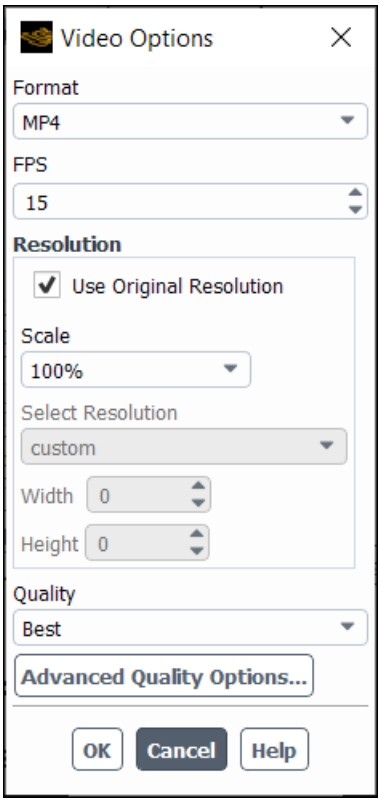
It is also possible to save all of the frames of the animation sequence in a video file. There are several video formats available including: MP4, AVI, FLV, MOV, MPEG.

Use the **Video Options** dialog box to control how the video file is written.

1. Select **Video File** in the **Write/Record Format** drop-down list in the **Playback** dialog box.
2. Provide a name for the saved video file in the **Video Name** field.
3. Click **Video Options...**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 36.56: The Video Options Dialog Box



4. Select the format you want the video written in from the **Format** drop-down list.

   - MP4—most common video file format.
   - AVI—developed by Microsoft, this format is common.
   - FLV—flash video format.
   - MOV—developed by Apple, this format can be played on both Macintosh and Windows machines.
   - MPEG—can be viewed using an MPEG decoder such as **mpeg_play**. Saving the entire animation to an MPEG file will require less disk space than storing individual window dump files (using the picture method), but the MPEG file will yield lower-quality images.

   Note:

   Video Options are not available with an academic license.

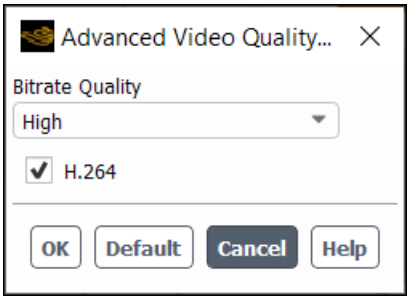
5. Specify the frames per second (**FPS**) for the video file.
6. Set the **Resolution** for how the video file will be written. Your options include:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Original resolution—the same resolution as the image files that make up the animation. If you want this resolution, ensure **Use Original Resolution** is enabled. You have the option of scaling the original resolution using the **Scale** drop-down list.
- 4K—3840x2160pixels
- 2K—2560x1440pixels
- WUXGA—1920x1200pixels
- 1080p FHD—1920x1080pixels
- 720p HD—1280x720pixels
- 480p SD—845x480pixels
- custom—enter whatever resolution you want by specifying the **Width** and **Height**.

7. Select the quality of the video file from the **Quality** drop-down list.
8. (Optional) Click **Advanced Quality Options...** to access the **Advanced Video Quality Options** dialog box.

Figure 36.57: The Advanced Video Quality Options Dialog Box



   a. Select the **Bitrate Quality** from the drop-down list.
   b. Enable **H.264** to use this method for video compression.
   c. Click **OK** to save the advanced quality options.

9. Click **OK** to save the video options.
10. Click **Write**.

Note:

You can specify your preferred video settings in the **Save Video Settings** node of the **Graphics** branch in **Preferences** (accessed via **File/Preferences...**).

These preferences are not applicable for academic licenses.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.19.4. Reading an Animation Sequence

If you have saved an animation sequence to a solution animation file (as described in Saving an Animation Sequence (p. 3720)), you can read that file back in at a later time (or in a different session) and play the animation. Note that you can read a solution animation file into any Ansys Fluent session; you do not need to read in the corresponding case and data files. In fact, you do not need to read in any case and data files at all before you read a solution animation file into Ansys Fluent.

To read a solution animation file, click the **Read...** button in the Playback Dialog Box (p. 5257). In The Select File Dialog Box (p. 931), specify the name of the file to be read.

36.20. Checking Your Case Setup

After you have set up your case, and prior to solving it, you can check your case setup using the Case Check Dialog Box (p. 5219) (Figure 36.58: The Case Check Dialog Box (p. 3725)). This function provides you with guidance and best practices when choosing case parameters and models. Your case will be checked for compliance in the mesh, models, boundary and cell zone conditions, material properties, and solver categories. Established rules will be available for each category, with recommended changes to your current settings. At your discretion, you may elect to apply the recommendations, or keep your current settings.

To access the Case Check Dialog Box (p. 5219) (Figure 36.58: The Case Check Dialog Box (p. 3725)), go to

Solution → **Run Calculation** **Check Case...**

If there are no problems with your case setup, then an information dialog box (Figure 36.59: The Information Dialog Box (p. 3726)) will appear stating that no recommendations need to be made at this time, otherwise, the Case Check Dialog Box (p. 5219) will open.

Figure 36.58: The Case Check Dialog Box

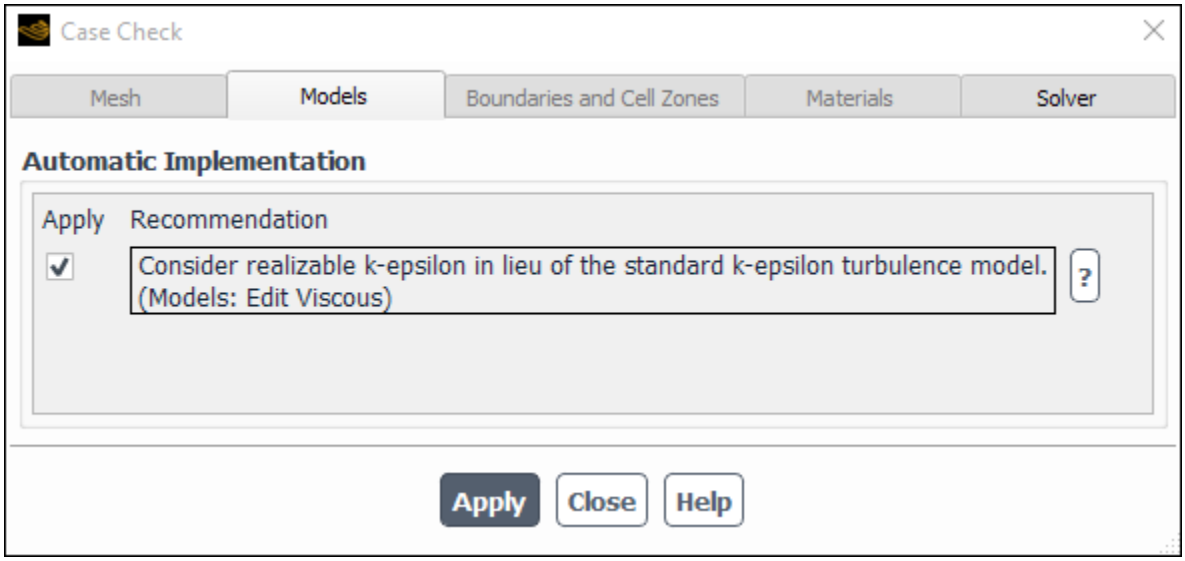
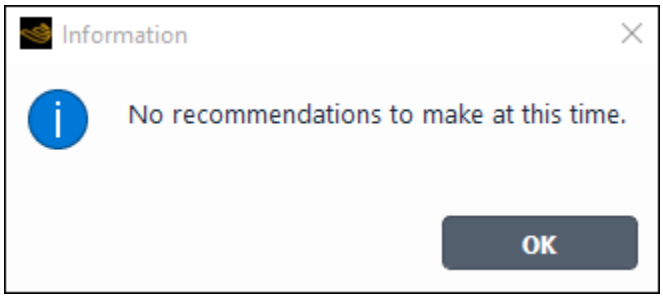


Figure 36.59: The Information Dialog Box



In the **Case Check** dialog box, each of the tabs **Mesh**, **Models**, **Boundaries and Cell Zones**, **Materials**, and **Solver** may contain recommendations. For each of the tabs that are enabled, best practices will be listed.

In some cases, the dialog box will be split based on the method that the recommendation is applied. There are two ways you can apply the listed recommendations:

Automatic Implementation

Ansys Fluent applies the change for you.

Manual Implementation

You will manually change your case settings.

For additional information, see the following sections:

36.20.1. Automatic Implementation

36.20.2. Manual Implementation

36.20.1. Automatic Implementation

To the left of each of the recommendations listed under **Automatic Implementation** (for example, Figure 36.61: The Models Tab in the Case Check Dialog Box (p. 3729)), there is an enabled **Apply** check box. An enabled check box will result in Ansys Fluent applying the change to your case automatically. If there are some recommendations that you do not want Ansys Fluent to implement automatically, then click the **Apply** check box to toggle off and disable the implementation of a particular recommendation. After going through all the tabs and determining which rules you want applied automatically, click the **Apply** button at the bottom of the dialog box. Changes to your settings will be applied to all recommendations throughout the dialog box with an enabled **Apply** check box. Ansys Fluent will print a message in the console notifying you that the applied recommendation has been implemented.

Ansys Fluent will ask you if you want to save the case before proceeding to the next step. If you choose **Yes**, The Select File Dialog Box (p. 931) will open allowing you to save your case with the new settings. If you select **No**, all the changes made to the case file will be lost once you exit Ansys Fluent.

36.20.2. Manual Implementation

For recommendations that are listed under **Manual Implementation**, Ansys Fluent cannot apply the changes for you. Therefore, if you opt to make a change to your current settings, based on the listed

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

recommendations, then you will need to manually make the changes by opening the affected dialog boxes or task pages and applying what was recommended.

To the right of the recommendations is a **?**, which essentially acts as a help button, leading you to related documentation on the specific topic.

At the bottom of each recommendation, there is a path that will guide you to the dialog box or task page where you can make the changes. For example, in the **Mesh** tab, you will see the following recommendation:

```
Check your mesh.
(General: Click [Check])
```

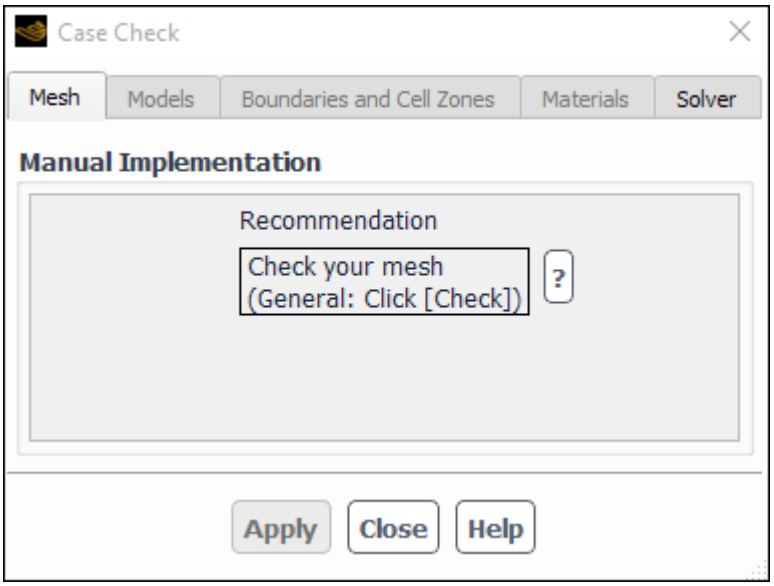
To perform the action, highlight **General** in the tree, then click the **Check** button in the **Mesh** group box. You will see a path for each recommendation, in each of the tabs.

Each of the case check rules are described in the following sections:

36.20.2.1. Checking the Mesh
36.20.2.2. Checking Model Selections
36.20.2.3. Checking Boundary and Cell Zone Conditions
36.20.2.4. Checking Material Properties
36.20.2.5. Checking the Solver Settings

36.20.2.1. Checking the Mesh

Figure 36.60: The Mesh Tab in the Case Check Dialog Box



The following recommendations appear under the **Mesh** tab (Figure 36.60: The Mesh Tab in the Case Check Dialog Box (p. 3727)):

- **Check your mesh.**

  If you have not already checked your mesh, it is best practice that you do so immediately after reading in your mesh, or after any mesh modification. To check your mesh go to

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setup → General → Check

Checking the mesh will help you detect any mesh trouble before you get started with your problem setup. You can learn more about the information obtained when checking your mesh, by going to Checking the Mesh (p. 1245).

- **Minimum Orthogonal Quality is less than 0.01. Consider improving the mesh quality before proceeding with your simulation.**

  Check the quality of your mesh immediately after reading in your mesh, or after any mesh modification. The quality of the mesh plays a significant role in the accuracy and stability of the numerical computation. You can learn more about the quality of your mesh by going to Mesh Quality (p. 1146).

  Setup → General → Report Quality

- **Preview zone motion and mesh motion before beginning the simulation.**

  After setting up your case using the **Dynamic Mesh** model, it is worth while to preview your mesh prior to running your simulation. You can preview **Zone Motion** by going to

  Setup → Dynamic Mesh → Dynamic Mesh → Display Zone Motion...

  To preview the **Mesh Motion**, go to

  Setup → Dynamic Mesh → Dynamic Mesh → Preview Mesh Motion...

  or

  Solution → Run Calculation → Preview Mesh Motion...

  or

  Solution → Run Calculation → Preview Mesh Motion...

  It is important that you preview zone motion first and then mesh motion. Zone motion shows the motion of all dynamic zones with the prescribed rigid body motion, using the graphics library. It is a very fast process and does not alter the mesh. Previewing the zone motion will show you if the motion is set up properly (for example zones moving in the wrong direction or rotating about the wrong center). It is much more difficult to detect these problems with mesh motion because small time steps are performed and no continuous animation is shown.

  Mesh motion should always be done for dynamic mesh cases with prescribed motion. Mesh motion will only show the validity of the mesh during the simulation. Mesh deformation and dynamic zones without rigid body motion will be considered during a mesh motion preview.

  Both the **Mesh Motion** and **Zone Motion** dialog boxes will have a **Preview** button that will allow you to view the mesh or zone motion prior to running your case. You can obtain more information on mesh motion and zone motion by going to Previewing the Dynamic Mesh (p. 1922).

- **Translate the mesh for axisymmetric geometry containing nodes below the x-axis.**

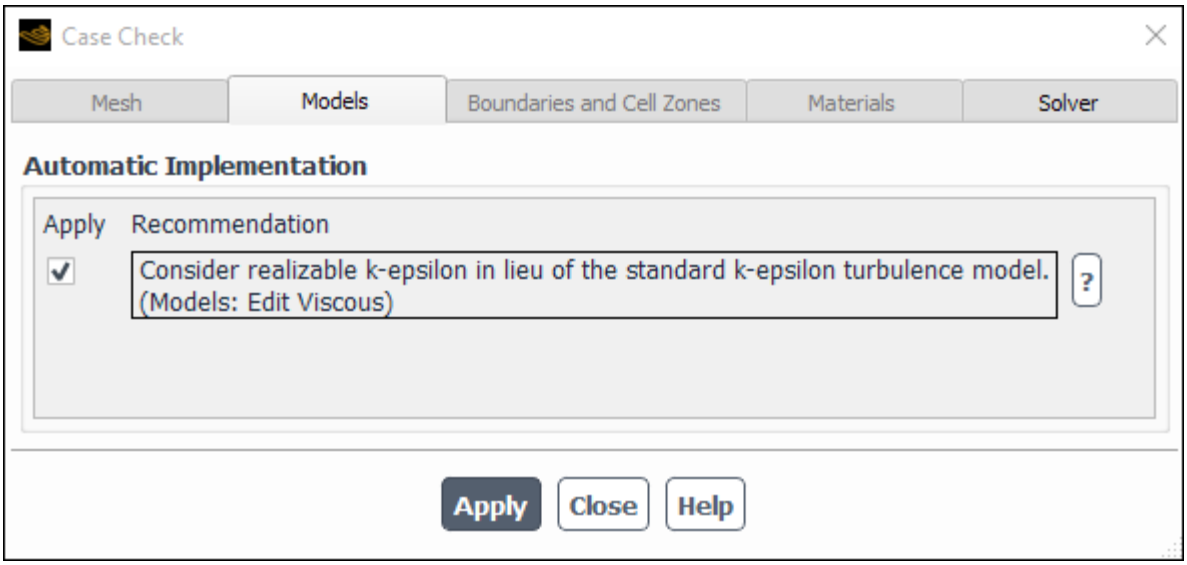
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If either **Axisymmetric** or **Axisymmetric Swirl** is specified in the **General** task page and there are mesh nodes that fall below the x-axis, then it is recommended that you translate the mesh. Nodes below the x axis are forbidden for axisymmetric cases, since the axisymmetric cell volumes are created by rotating the 2D cell volume about the x axis; therefore nodes below the x axis would create negative volumes. To find out if there are any nodes that lie below the x-axis, perform a mesh check (Checking the Mesh (p. 1245)). For information on translating the mesh, see Translating the Mesh (p. 1288). To access the **Mesh Translate** dialog box, go to

Domain → **Mesh** → **Transform** → **Translate...**

36.20.2.2. Checking Model Selections

Figure 36.61: The Models Tab in the Case Check Dialog Box



The following recommendations appear under the **Models** tab (Figure 36.61: The Models Tab in the Case Check Dialog Box (p. 3729)):

- **Consider realizable k-epsilon in lieu of the standard k-epsilon turbulence model.**

  The realizable k - ε model is a more recent development of the standard k - ε model and differs from it in that the realizable k - ε model contains a new formulation for the turbulent viscosity, as well as a new transport equation for the dissipation rate, ε, derived from an exact equation for the transport of the mean-square vorticity fluctuation.

  realizable k - ε model means that the model satisfies certain mathematical constraints on the Reynolds stresses, consistent with the physics of turbulent flows. For more information on the standard k - ε model and the realizable k - ε model, visit Standard k-ε Model and Realizable k-ε Model (in the Theory Guide), respectively.

  Setup → **Models** → **Viscous** → **Edit...**

  For information on all k - ε model options, go to Standard, RNG, and Realizable k-ε Models in the Theory Guide.

- **Disable DO/Energy coupling if the optical thickness is less than 10.**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DO/Energy coupling should only be used when the optical thickness is greater than 10. Refer to Energy Coupling and the DO Model in the Theory Guide for more information.

Setup → Models → Radiation → Edit...

- **Verify that the temperature specified for boundary zones that do not participate in the view factor calculation is appropriate.**

  When using the S2S radiation model, make sure that you set the temperature for boundaries that do not participate in the view factor calculation to an appropriate value. In most cases the appropriate value is the ambient temperature, which by default is assumed to be 300 K. See Specifying Boundary Zone Participation (p. 2206) for more information.

  Setup → Models → Radiation → Edit...

- **Change the under-relaxation factor for the mixing plane model to 1.0.**

  If you have created a mixing plane, set the **Under-Relaxation** in the **Mixing Plane** dialog box to 1. Look under Global Parameters in Setting Up the Legacy Mixing Plane Model (p. 1967) for information about the mixing plane under-relaxation.

  Domain → Mesh Models → Mixing Planes...

- **Enable the smoothing option for dynamic mesh simulations when remeshing.**

  When your case involves the use of dynamic meshes and remeshing is enabled, then it is recommended that you also perform smoothing on the mesh. For a complete discussion of smoothing and remeshing, see Setting Dynamic Mesh Modeling Parameters (p. 1814).

  Setup → Dynamic Mesh → Dynamic Mesh

- **Include turbulence interaction for the NOx model.**

  When running thermal NOx simulations and your flow is turbulent, then be sure to set the NOx **Turbulence Interaction Mode**.

  Setup → Models → NOx → Edit...

  In turbulent combustion calculations, Ansys Fluent solves the density-weighted time-averaged Navier-Stokes equations for temperature, velocity, and species concentrations or mean mixture fraction and variance. Methods of modeling the mean turbulent reaction rate can be based on either moment methods or probability density function (PDF) techniques. Ansys Fluent uses the PDF approach.

  To learn about how this feature is set up, go to Setting Turbulence Parameters (p. 2598).

- **Consider using the default Schnerr-Sauer or the Zwart-Gerber-Belamri cavitation model.**

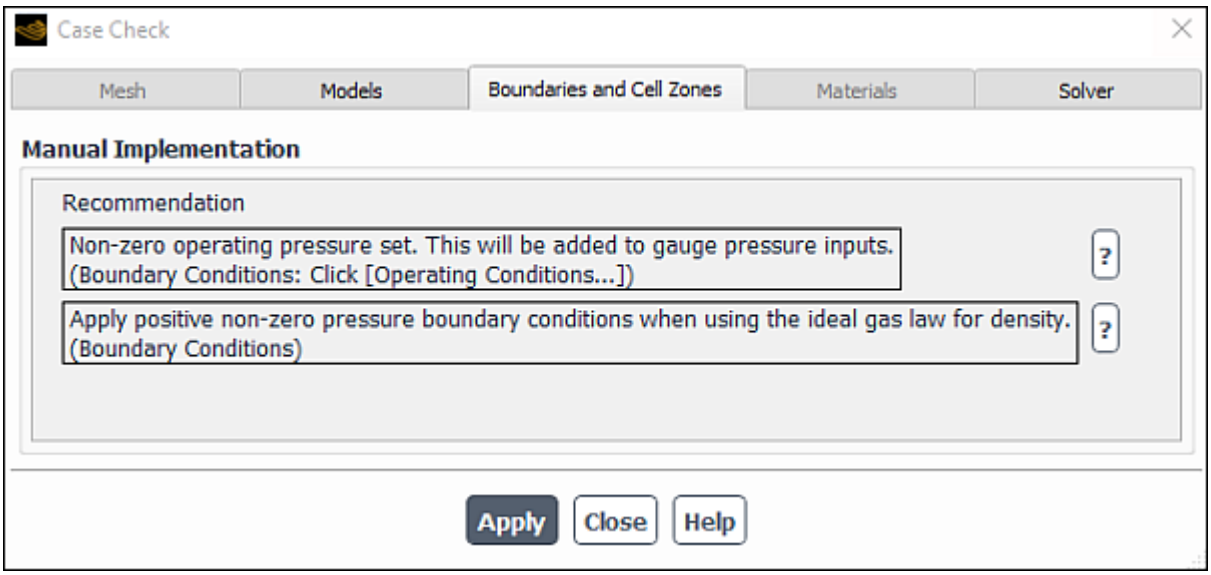
  When using the mixture multiphase model with the Singhal et al. cavitation model enabled, consider changing it to either the Schnerr-Sauer or the Zwart-Gerber-Belamri cavitation model. Refer to Cavitation Models in the Theory Guide for more information.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setup → Models → Phases → Interaction...

36.20.2.3. Checking Boundary and Cell Zone Conditions

Figure 36.62: The Boundaries and Cell Zones Tab in the Case Check Dialog Box



The following recommendations appear under the **Boundaries and Cell Zones** tab (Figure 36.62: The Boundaries and Cell Zones Tab in the Case Check Dialog Box (p. 3731)):

- **Apply an axis boundary on the centerline (x-axis).**

  For geometry that is axisymmetric or axisymmetric swirl (as set in the **General** task page), the centerline (x-axis) boundary type should be set to **axis**. See Axis Boundary Conditions (p. 1504).

  Boundary Conditions

- **Change outlet boundary conditions. A combination of pressure and outflow boundaries is not compatible.**

  Outflow boundary conditions in Ansys Fluent are used to model flow exits where the details of the flow velocity and pressure are not known prior to solution of the flow problem. One of the limitations when using outflow boundary conditions is that outflow boundary conditions are not compatible with pressure inlets. Therefore, it is recommended that you use velocity or mass-flow inlets instead of pressure inlets when used in combination with outflow boundaries. See Outflow Boundary Conditions (p. 1448) for a list of limitations that exist with outflow boundaries.

  Setup → Boundary Conditions

- **Change outlet boundary conditions. Outflow boundary conditions are not compatible with the ideal gas law for density.**

  Outflow boundaries cannot be used if you are modeling unsteady flows with varying density, even if the flow is incompressible. See Outflow Boundary Conditions (p. 1448) for more limitations that exist with outflow boundaries.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setup → **Boundary Conditions**

- **Non-zero operating pressure set. This will be added to gauge pressure inputs.**

  For cases that have density specified as the ideal gas law, and the operating pressure is greater than zero, the operating pressure will be added to the gauge pressure to yield the absolute pressure. For more information, see Density Inputs for the Ideal Gas Law for Compressible Flows (p. 1632) and Operating Pressure, Gauge Pressure, and Absolute Pressure (p. 1678).

Setup → **Boundary Conditions** → **Operating Conditions...**

- **Apply positive non-zero pressure boundary conditions when using the ideal gas law for density.**

  In compressible flows , isentropic relations for an ideal gas are applied to relate total pressure, static pressure, and velocity at a pressure inlet boundary. Your input of total pressure, p_0', at the inlet and the static pressure, p_s', in the adjacent fluid cell are related, as described in Equation 7.93 (p. 1410) Equation 7.94 (p. 1410) of Calculation Procedure at Pressure Inlet Boundaries (p. 1410). It is recommended that pressure boundary conditions are not set to zero for compressible flows that use the ideal gas law.

Setup → **Boundary Conditions**

- **Review turbulence specifications at flow boundaries. Default values detected.**

  If your case setup has any of the turbulence models enabled, be sure to review the default parameters for **K and Epsilon** or **K and Omega** for the **Turbulence Specification Method** in the outlet and inlet boundary conditions. Ansys Fluent's default parameters for the (**Backflow**) **Turbulent Kinetic Energy**, (**Backflow**) **Turbulent Dissipation Rate**, and (**Backflow**) **Specific Dissipation Rate** are 1. You can either adjust the values, or select a different **Turbulence Specification Method**. For general information turbulence parameters, see Determining Turbulence Parameters (p. 1395).

Setup → **Boundary Conditions**

- **Assign non-zero layer thicknesses for wall boundaries with shell conduction.**

  When the **Shell Conduction** option is enabled in the **Wall** boundary condition dialog box, Ansys Fluent will compute heat conduction for the wall not only in the normal direction, but also in the planar directions. To enable such computations, you must specify a nonzero **Thickness** for each layer in the **Shell Conduction Layers** dialog box. See Shell Conduction (p. 1475) for information on shell conduction in thin walls.

Setup → **Boundary Conditions**

- **Assign a value of 0 or 1 for VOF at the inlet or outlet boundary conditions.**

  When enabling the VOF model, the **Volume Fraction** in the inlet and outlet boundary conditions for each phase should be set either to `0` or `1`. No intermediate values are permitted. For general

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

information on boundary condition setup, see Defining Multiphase Cell Zone and Boundary Conditions (p. 2957).

Setup → **Boundary Conditions**

- **Change the outlet boundary condition. Outflow boundary condition is not compatible with current multiphase settings.**

  You cannot assign an outflow boundary condition when using the mixture and Eulerian multiphase models. Note the limitations of this boundary condition in Outflow Boundary Conditions (p. 1448). Ansys Fluent can model the effects of open channel flow using the VOF formulation. In such a case, outflow boundary conditions can be used at the outlet of open channel flows, to model flow exits where the details of the flow velocity and pressure are not known prior to solving the flow problem. See Open Channel Flow in the Theory Guide, under the heading Outflow Boundary, for more information.

Setup → **Boundary Conditions**

- **Review wall motion. Stationary wall motion relative to adjacent cell zone detected.**

  In cases where the fluid zone motion type is specified as **Moving Mesh** or **Moving Reference Frame**, all wall zones should be set to **Moving Wall** in the **Momentum** tab in the **Wall** boundary conditions dialog box. The wall motion should be defined **Relative to Adjacent Cell Zone**. The exception to this is if the walls are stationary in the absolute frame. To define wall motion, see Inputs at Wall Boundaries (p. 1458).

Setup → **Boundary Conditions**

- **Assign non-zero velocities when specifying a moving fluid zone.**

  If selecting either **Moving Mesh** or **Moving Reference Frame** in the **Fluid** dialog box, be sure to set nonzero values for the rotational and translational velocities. Refer to Defining Zone Motion (p. 1323) for user inputs.

Setup → **Cell Zone Conditions**

- **Review flow specifications at inlet boundaries. Default values detected.**

  For **mass-flow-inlet** and **velocity-inlet** boundary conditions, the default values in Ansys Fluent are $1kg/s$ and $0m/s$, respectively. Review the settings and adjust accordingly. See Default Settings at Velocity Inlet Boundaries (p. 1416) and Default Settings at Mass-Flow Inlet Boundaries (p. 1424) for default parameters of velocity inlets and mass-flow inlets, respectively.

Setup → **Boundary Conditions**

- **Define the porous zone when using the heat exchanger model.**

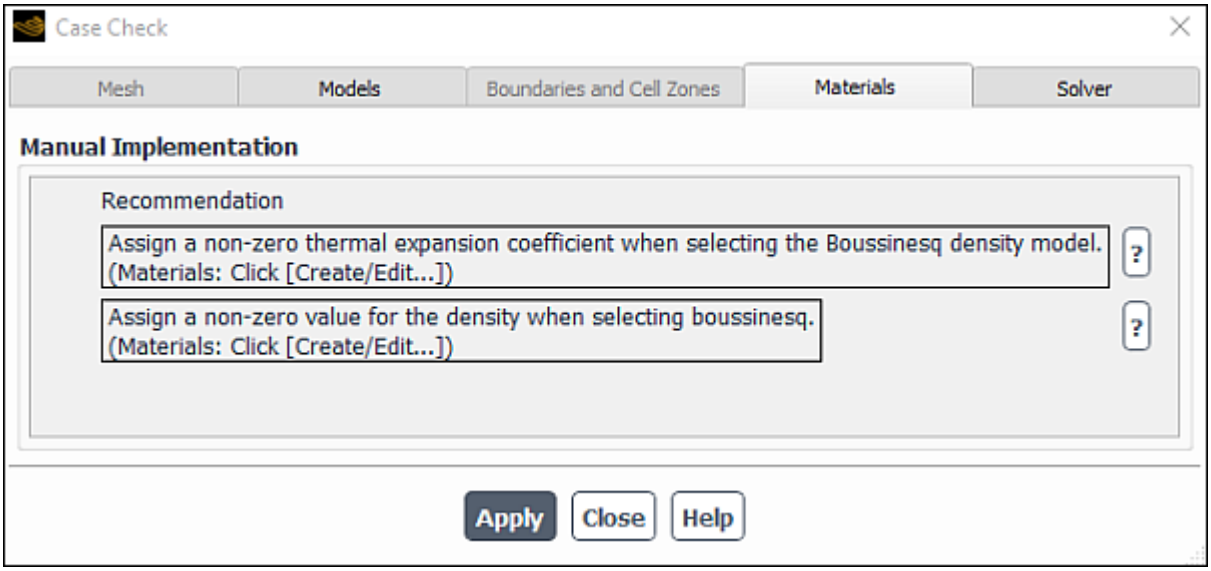
  Heat exchanger models always require the definition of the porous media zone on the primary side for the macro model and for both primary and auxiliary sides for the dual cell model. See Streamwise Pressure Drop in the Theory Guide for more information.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setup → **Cell Zone Conditions**

36.20.2.4. Checking Material Properties

Figure 36.63: The Materials Tab in the Case Check Dialog Box



The following recommendations appear under the **Materials** tab (Figure 36.63: The Materials Tab in the Case Check Dialog Box (p. 3734)):

- **Assign individual fluid Cps to polynomial functions of temperature.**

  For cases with species transport and volumetric reactions, it is best practice to specify the specific heat capacity C_p as a polynomial that is a function of temperature. See Defining Properties for the Mixture and Its Constituent Species (p. 2374) and Specific Heat Capacity as a Function of Temperature (p. 1663) for information on defining material properties for the species in the mixture.

  Setup → **Materials**

- **Assign a non-zero value for the density when selecting boussinesq.**

  The Boussinesq model is used for natural convection problems involving small changes in temperature. To enable the Boussinesq approximation for density, choose **boussinesq** from the **Density** drop-down list in the **Create/Edit Materials** dialog box and specify a constant value for **Density**. See Inputs for the Boussinesq Approximation (p. 1626).

  Setup → **Materials**

- **Review the absorption coefficient. Default value detected.**

  If any of the radiation models are enabled. Enter an absorption coefficient for the material listed (Radiation Properties (p. 1665)).

  Setup → **Materials**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

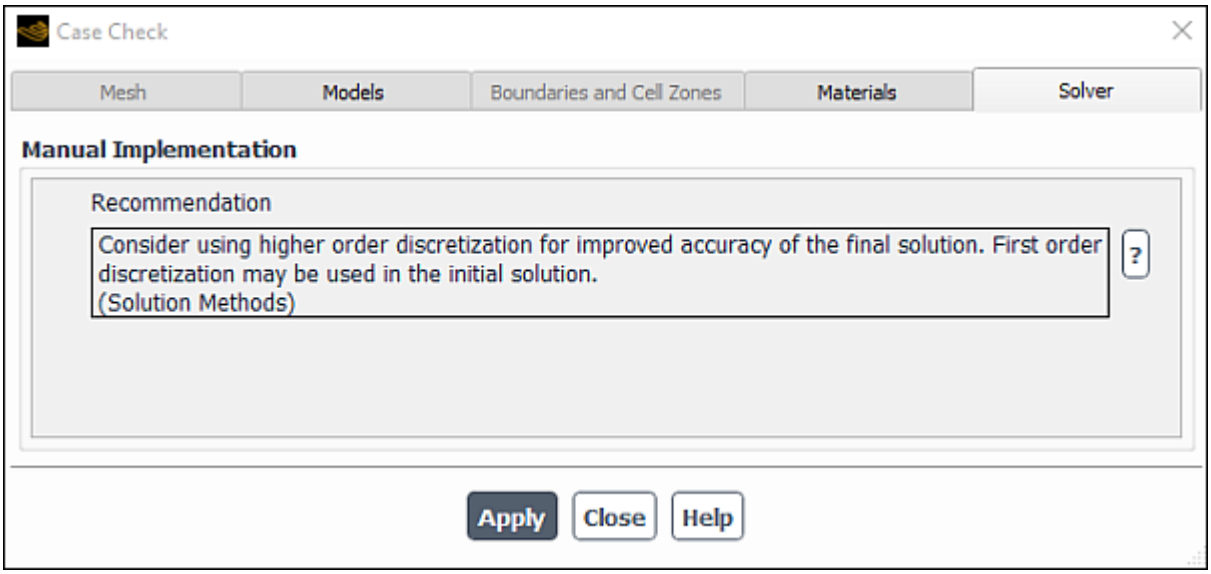
- **Assign a non-zero thermal expansion coefficient when selecting the Boussinesq density model.**

  When selecting **boussinesq** to describe the density of your material, be sure to enter a valid thermal expansion coefficient for your material. For detailed information on the Boussinesq model, see The Boussinesq Model (p. 1388).

  Setup → **Materials**

36.20.2.5. Checking the Solver Settings

Figure 36.64: The Solver Tab in the Case Check Dialog Box



The following recommendations appear under the **Solver** tab (Figure 36.64: The Solver Tab in the Case Check Dialog Box (p. 3735)):

- **Enable the unsteady solver option when selecting moving mesh for the fluid boundary.**

  If the motion type of the fluid boundary condition is specified as **Moving Mesh**, then your case should be specified as **Transient** in the **General** task page. Visit Setting Up the Sliding Mesh Problem (p. 1805) for steps on setting up moving mesh problem.

  Setup → **General**

- **Assign LSQ cell-based gradient reconstruction.**

  The least squares cell-based averaging scheme is known to be as accurate as the node-based gradient for irregular unstructured meshes, but less expensive to compute than the node-based gradient. Therefore, it is recommended that least squares cell-based gradient reconstruction is used. See Evaluation of Gradients and Derivatives in the Theory Guide for more information on gradient options.

  Solution → **Methods**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Change the under-relaxation factor for the energy equation to at least 0.90.**

  You should set the energy under-relaxation factor between 0.90 and 1.0. If you decide to apply this recommendation, then Ansys Fluent will automatically set the energy under-relaxation factor to `0.90`. If you want to increase this value, you can manually make the change by going to the **Solution Controls** task page. See Solution Strategies for Heat Transfer Modeling (p. 2167) for the under-relaxation of the energy equation.

  Solution → **Controls**

- **Increase the NOx under-relaxation factor to at least 0.90.**

  If the NOx model is enabled, set the NOx under-relaxation factor to a value of at least 0.90 to fully converge the solution. Note that the under-relaxation factor could be lower at the start of the run, but can then be increased after an initial solution is obtained. If you decide to apply this recommendation, then Ansys Fluent will automatically set the NOx under-relaxation factor to `0.90`. If you want to increase this value, you can manually make the change by going to the **Solution Controls** task page. See Using the NOx Model (p. 2585).

  Solution → **Controls**

- **Increase the Discrete Ordinates under-relaxation factor to at least 0.90.**

  If the Discrete Ordinates (DO) radiation model is enabled, set the radiation under-relaxation factor to a value of at least 0.90 to fully converge the solution. Note that the under-relaxation factor could be lower at the start of the run, but can then be increased after an initial solution is obtained. If you decide to apply this recommendation, then Ansys Fluent will automatically set the radiation under-relaxation factor to `0.90`. If you want to increase this value, you can manually make the change by going to the **Solution Controls** task page. See DO Solution Parameters (p. 2239).

  Solution → **Controls**

- **Increase the P1 under-relaxation factor to 1.0.**

  If the P1 radiation model is enabled, set the radiation under-relaxation factor to 1.0 to fully converge the solution. Note that the under-relaxation factor could be lower at the start of the run, but can then be increased after an initial solution is obtained. If you decide to apply this recommendation, then Ansys Fluent will automatically set the radiation under-relaxation factor to `1.0`. See P-1 Model Solution Parameters (p. 2236).

  Solution → **Controls**

- **Increase the species and energy under-relaxation factors to at least 0.90.**

  For a case with species transport and energy defined, set the species and energy under-relaxation factors to a value of at least 0.90. If you decide to apply this recommendation, then Ansys Fluent will automatically set the species and energy under-relaxation factors to `0.90`. If you want to increase this value, you can manually make the change by going to the **Solution Controls** task page. See Solution Procedures for Chemical Mixing and Finite-Rate Chemistry (p. 2395).

  Solution → **Controls**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Assign a value of 1 for the under-relaxation factor for unsteady DPM with 1 DPM update per time step.**

  It is recommended that the DPM under-relaxation factor be set to 1 for unsteady DPM with 1 DPM update per time step.

  Solution → **Controls**

- **Increase the mean mixture fraction under-relaxation factor to at least 0.90.**

  If the non-premixed or partially premixed combustion models are enabled, then it is best to set the mean mixture under-relaxation factor to a value of at least 0.90 to ensure full convergence. If you decide to apply this recommendation, then Ansys Fluent will automatically set the mean mixture under-relaxation factor to `0.90`. If you want to increase this value, you can manually make the change by going to the **Solution Controls** task page. See Solving the Flow Problem (p. 2530).

  Solution → **Controls**

- **Consider using higher order discretization for improved accuracy of the final solution. First-order discretization may be used in the initial solution.**

  It is generally advisable to obtain an initial solution using first-order accurate discretization, however, second order discretization is recommended for improved accuracy of the final solution. See Choosing the Spatial Discretization Scheme (p. 3575) for more information on discretization schemes.

  Solution → **Methods**

- **Select the absolute reference frame for initializing cases when using the MRF model.**

  When using the MRF model, always use the absolute reference frame while initializing the solution. Select **Absolute** under **Reference Frame** in the **Solution Initialization** task page. If the **Relative to Cell Zone** option is selected, which is the default option, the initial flow field can contain discontinuities, which can cause convergence problems in the first few iterations. Refer to Initializing the Entire Flow Field Using Standard Initialization (p. 3628) for more information.

  Solution → **Initialization** **Initialize**

- **Choose PRESTO! for the pressure discretization scheme.**

  When using the VOF model, it is recommended that you use **PRESTO!** as the pressure discretization scheme. This scheme is recommended for flows with high swirl numbers, a high-Rayleigh-number natural convection, high-speed rotating flows, flows involving porous media, and flows in strongly curved domains. See Choosing the Pressure Interpolation Scheme (p. 3577) for more information.

  Solution → **Methods**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.21. Convergence and Stability

Convergence can be hindered by a number of factors. Large numbers of computational cells, overly conservative under-relaxation factors, and complex flow physics are often the main causes. Sometimes it is difficult to know whether you have a converged solution. In the following sections, some of the numerical controls and modeling techniques that can be exercised to enhance convergence and maintain stability are examined.

36.21.1. Judging Convergence

36.21.2. Step-by-Step Solution Processes

36.21.3. Modifying Algebraic Multigrid Parameters

36.21.4. Modifying the Multi-Stage Parameters

36.21.5. Robustness with Meshes of Poor Quality

36.21.6. Warped-Face Gradient Correction

36.21.7. Numerical Noise Filter for the Energy Equation

You should also refer to Choosing the Spatial Discretization Scheme (p. 3575) and Choosing the Pressure-Velocity Coupling Method (p. 3585) for information about how the choice of discretization scheme or (for the pressure-based solver) pressure-velocity coupling scheme can affect convergence. Manipulation of under-relaxation parameters and multigrid settings to enhance convergence is discussed in Setting Under-Relaxation Factors (p. 3589) and Modifying Algebraic Multigrid Parameters (p. 3741).

36.21.1. Judging Convergence

There are no universal metrics for judging convergence. Residual definitions that are useful for one class of problem are sometimes misleading for other classes of problems. Therefore it is a good idea to judge convergence not only by examining residual levels, but also by monitoring relevant integrated quantities such as drag or heat transfer coefficient.

For many problems, the default convergence criterion in Ansys Fluent is sufficient. This criterion requires that the globally scaled residuals, defined by Equation 36.28 (p. 3685), Equation 36.32 (p. 3686), or Equation 36.35 (p. 3687) decrease to 10^{-3} for all equations except the energy and P-1 equations, for which the criterion is 10^{-6}.

Sometimes, however, this criterion may not be appropriate. For example:

- Because the continuity residual is scaled by its behavior during the first five iterations, the continuity residual level is initial guess-dependent and startup-dependent. A better initial guess may lead to a higher continuity residual, which is counterintuitive for judging convergence.
- Because the global scale factors depend on the solution level, sometimes the residuals are significantly below the default convergence target even when the solution is still changing. This behavior occurs most frequently with the energy equation.
- For some equations, such as for turbulence quantities, a poor initial guess may result in high scale factors. In such cases, scaled residuals will start low, increase as nonlinear sources build up, and eventually decrease. It is therefore good practice to judge convergence not just from the value of the residual itself, but from its behavior. You should ensure that the residual continues to decrease (or remain low) for several iterations (say 50 or more) before concluding that the solution has converged.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When issues like the above arise, you should consider using locally scaled residuals defined by Equation 36.29 (p. 3685), Equation 36.31 (p. 3685), or Equation 36.36 (p. 3687), which have a default convergence criterion for all equations of 10^{-5} and 10^{-4} for steady-state flows and transient flows, respectively. Experience suggests that locally scaled residuals may be more universal for the purposes of judging convergence than globally scaled residuals. Note that you may want to adjust the convergence criterion from the default value for the locally scaled continuity residual based on your specific application.

Another popular approach to judging convergence is to require that the unscaled residuals drop by three orders of magnitude. Ansys Fluent provides residual normalization for this purpose, as discussed in Definition of Residuals for the Pressure-Based Solver (p. 3684), where residuals are defined for both the pressure-based solver and the density-based solver. In this approach the convergence criterion is that the normalized unscaled residuals should drop to 10^{-3}. However, this requirement may not be appropriate in many cases:

- If you have provided a very good initial guess, the residuals may not drop three orders of magnitude. In a nearly-isothermal flow, for example, energy residuals may not drop three orders if the initial guess of temperature is very close to the final solution.
- If the governing equation contains nonlinear source terms which are zero at the beginning of the calculation and build up slowly during computation, the residuals may not drop three orders of magnitude. In the case of natural convection in an enclosure, for example, initial momentum residuals may be very close to zero because the initial uniform temperature guess does not generate buoyancy. In such a case, the initial nearly-zero residual is not a good scale for the residual.
- If the variable of interest is nearly zero everywhere, the residuals may not drop three orders of magnitude. In fully-developed flow in a pipe, for example, the cross-sectional velocities are zero. If these velocities have been initialized to zero, initial (and final) residuals are both close to zero, and a three-order drop cannot be expected.

Regardless of the residual scaling method, it is wise to monitor important solution quantities, such as drag or overall heat transfer coefficient, before concluding that the solution has converged.

Conversely, it is possible that if the initial guess is very bad, the initial residuals are so large that a three-order drop in residual does not guarantee convergence. This is specially true for k and ε equations where good initial guesses are difficult. Here again it is useful to examine overall integrated quantities that you are particularly interested in. If the solution is unconverged, you may drop the convergence tolerance, as described in Modifying Convergence Criteria (p. 3693).

36.21.2. Step-by-Step Solution Processes

One important technique for speeding convergence for complex problems is to tackle the problem one step at a time. When modeling a problem with heat transfer, you can begin with the calculation of the isothermal flow. To solve turbulent flow, you might start with the calculation of laminar flow. When modeling a reacting flow, you can begin by computing a partially converged solution to the non-reacting flow, possibly including the species mixing. When modeling a discrete phase, such as fuel evaporating from droplets, it is a good idea to solve the gas-phase flow field first. Such solutions generally serve as a good starting point for the calculation of the more complex problems. These step-by-step techniques involve using the Solution Controls Task Page (p. 5163) to turn equations on and off in the **Equations** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.21.2.1. Selecting a Subset of the Solution Equations

Ansys Fluent automatically solves each equation that is turned on using the **Models** family of dialog boxes. If you specify in the Viscous Model Dialog Box (p. 4715) that the flow is turbulent, equations for conservation of turbulence quantities are turned on. If you specify in the Energy Dialog Box (p. 4714) that Ansys Fluent should enable energy, the energy equation is enabled. Convergence can be sped up by focusing the computational effort on the equations of primary importance. The **Equations** list in the Equations Dialog Box (p. 5166) allows you to turn individual equations on or off temporarily.

Solution → **Controls** **Equations...**

A typical example is the computation of a flow with heat transfer. Initially, you will define the full problem scope, including the thermal boundary conditions and temperature-dependent flow properties. Following the problem setup, you will use the **Equations** dialog box to temporarily turn off the energy equation. You can then compute an isothermal flow field, remembering to set a reasonable initial value for the temperature of the fluid.

Important:

This is possible only for the pressure-based solver; the density-based solver solves the energy equation together with the flow equations in a coupled manner, so you cannot turn off the energy equation as described above.

When the isothermal flow is reasonably well converged, you can turn the energy equation back on. You can actually turn off the momentum and continuity equations while the initial energy field is being computed. When the energy field begins to converge well, you can turn the momentum and continuity equations back on so that the flow pattern can adjust to the new temperature field. The temperature will couple back into the flow solution by its impact on fluid properties such as density and viscosity. The temperature field will have no effect on the flow field if the fluid properties (for example, density, viscosity) do not vary with temperature. In such cases, you can compute the energy field without turning the flow equations back on again.

Important:

If you have specified temperature-dependent flow properties, you should be sure that a realistic value has been set for temperature throughout the domain before disabling calculation of the energy equation. If an unrealistic temperature value is used, the flow properties dependent on temperature will also be unrealistic, and the flow field will be adversely affected. Instructions for initializing the temperature field or patching a temperature field onto an existing solution are provided in Initializing the Solution (p. 3627).

36.21.2.2. Turning Reactions On and Off

To solve a species mixing problem prior to solving a reacting flow, you should set up the problem including all of the reaction information, and save the complete case file. To turn off the reaction so that only the species mixing problem can be solved, you can use the Species Model Dialog Box (p. 4768) to turn off the **Volumetric** option under **Reactions**.

Setup → **Models** → **Species** → **Edit...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Once the species mixing problem has partially converged, you can return to the **Species Model** dialog box and turn the **Volumetric Reactions** option on again. You can then resume the calculation starting from the partially converged data.

For combustion problems you may want to patch a hot temperature in the vicinity of the anticipated reactions before you restart the calculation. See Patching Values in Selected Cells (p. 3630) for information about patching an initial value for a flow variable.

36.21.3. Modifying Algebraic Multigrid Parameters

The default algebraic multigrid settings are appropriate for nearly all problems, but in rare cases you may need to make minor adjustments. Setting Algebraic Multigrid Parameters (p. 3610) describes how to analyze the multigrid solver's performance to determine which parameters.

36.21.4. Modifying the Multi-Stage Parameters

It is possible to make several changes to the multi-stage time-stepping scheme itself. See Changing the Multi-Stage Scheme (p. 3624) for detailed information.

36.21.5. Robustness with Meshes of Poor Quality

Poor quality meshes are meshes containing highly skewed cells, highly non-orthogonal cells, non-convex cells, or cells with left-handed faces. Such mesh elements tend to decrease the numerical stability of traditional CFD discretization algorithms. These mesh elements require special treatment, namely, a numerical correction of the transport equation discretization, which is intended to improve the numerical properties of the solution algorithms at mesh cells of poor quality. This correction is referred to as "poor mesh numerics", and it is enabled by default so that it is applied to cells with an orthogonal quality of 0.

Note:

If your mesh exhibits any type of poor quality except left-handed faces, you can maintain second order accuracy by using the warped-face gradient correction and disabling poor mesh numerics. To disable poor mesh numerics, set `solve/set/poor-mesh-numerics/enable?` to `no`. For additional information on this method, see Warped-Face Gradient Correction (p. 3745).

If you read in a poor quality mesh containing cells and faces with corrupt metrics, you will see a warning in the console of the following form:

```
Info: The mesh contains elements that are invalid or of poor quality.
      A different numerical scheme will be applied to these elements,
      which may affect the quality of the solution. It is recommended
      that you consider removing the invalid and poor quality elements
      in the mesh.
      For more information on the invalid and poor quality elements,
      please use the following TUI commands:
      /mesh/check
      /mesh/repair-improve/report-poor-elements.
```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For details on the tools available for investigating corrupt cells and faces and repairing them, see Repairing Meshes (p. 1247). For the poor quality cells that cannot be repaired, you can rely on the application of poor mesh numerics to allow you to proceed.

The local solution correction applied by poor mesh numerics can be of 0th, 1st, or 2nd order:

- The 0th order scheme applies an algorithm that computes the solution variable for the transport equation in the bad cells by assembling the solution directly from the surrounding solution in the better quality cells.
- The 1st order scheme applies locally low order discretization methods and neglects some non-orthogonal contributions to the gradients when computing the diffusive fluxes. This scheme is selected by default.
- The 2nd order scheme only modifies the numerics in the bad cells by assembling the gradient vector for the given solution variable from the gradients in the surrounding better quality cells.

In other words, the discretization error will be independent of the mesh size in this region if 0th order is used. It will decrease linearly with subsequent grid refinement when 1st order is used, and quadratically when the 2nd order option is selected.

It is strongly recommended that you use the default 1st order solution correction, which provides a reasonable compromise between accuracy and stability gain. When only a few cells are invalid, you can attempt to use the other schemes: for meshes of better (and yet still low) quality, you can try the 2nd order option, which will preserve the mesh convergence behavior provided by the convection term discretization schemes; if no convergence is obtained with any of the aforementioned schemes, you can try the 0th order option, which will provide the highest stability and at the same time the lowest accuracy. Note that in regions with highly nonlinear flow physics, the 0th order scheme can yield highly nonphysical results. To change the order scheme, use the prompt in the following text command: `solve/set/poor-mesh-numerics/enable?`.

The three solution correction schemes should be applied to only a small percentage of the mesh cells, in order to strike a balance between the robustness of the solution and solution efficiency. The computational time (or cost or run time requirements) for all three order schemes will increase linearly as the number of cells identified as poor quality cells increases. Provided that there is reasonable convergence behavior with all three schemes, the 1st order option will be the fastest; 0th and 2nd order schemes will have similar run times to each other. If a large percentage of the cells are marked for correction due to poor quality metrics, then it is highly recommended that you abandon the mesh for one with better quality metrics.

If the default poor mesh numerics settings are not sufficient to provide stability for your mesh, you can enable the following text command so that the cell centroids of poor cells are relocated with the goal of enhancing the orthogonality metrics: `solve/set/poor-mesh-numerics/orthogonality-enhancing-cell-centroids?`. This can be used with all mesh types, and is applied to cells when the criterion value is equal to or less than a threshold value defined as part of the text command. Increasing the threshold for the orthogonal quality can increase solver robustness, but at the cost of reduced solution accuracy. Note that the enhanced metrics are only apparent when reporting the quality in the solution mode of Fluent, and not in the meshing mode.

If you continue to have stability problems, you can increase the range of cells to which the poor mesh numerics correction is applied. As stated previously, when the `solve/set/poor-mesh-numer-`

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ics/enable? text command is enabled, only cells with an orthogonal quality of 0 are corrected by default; this is a conservative setting, and can be changed by enabling the following text commands:

solve/set/poor-mesh-numerics/cell-quality-based?

This ensures that the correction is applied to cells with an orthogonal quality of 0.05 or less, and you can further adjust this threshold by using the following text command: solve/set/poor-mesh-numerics/set-quality-threshold.

solve/set/poor-mesh-numerics/gradient-quality-based?

This ensures that poor cells are detected and treated using a criterion based on the cell gradient quality. Poor mesh numerics are applied when the criterion value is equal to or less than a threshold value defined as part of this text command. This text command is only available with the pressure-based solver, and is not supported for cases that have periodic boundaries. Note that after initialization, you can postprocess this criterion using the **Gradient Quality Measure** field variable (in the **Mesh...** category); poor cells have a value closer to 0, whereas good cells have a value closer to 1. When displaying contours, it is recommended that you enable the **Filled** option and disable the **Node Values** option, since this measure is calculated at the cell centers.

solve/set/poor-mesh-numerics/solution-and-quality-based?

This ensures that poor cells are detected and treated using a criterion based on the solution and cell quality. Poor mesh numerics are applied when the criterion value is equal to or less than a threshold value defined as part of this text command, and at a specified frequency (of iterations or time steps). This text command is only available with the pressure-based solver. Note that after you have run an iteration, you can postprocess the cells marked by this criterion using the **Solution and Cell Quality Measure** field variable (in the **Mesh...** category); cells with a criterion value below the threshold (that is, poor cells) and their neighboring cells are assigned a value of 1, whereas all other cells (that is, good cells) are assigned a value of 0. When displaying contours, it is recommended that you enable the **Filled** option and disable the **Node Values** option, since this measure is calculated at the cell centers.

solve/set/poor-mesh-numerics/register-based/

This menu provides text commands that allow you to specify that poor mesh numerics are applied to cells in specified cell registers. The cell registers can be marked manually at the start of the calculation and/or automatically at a specified interval during the calculation. The automatic marking can be useful if the mesh will be changing during the solution (for example, due to mesh manipulation or adaption). For manual or automatic marking, you must enter new in this menu, and after you provide a name for the definition you can define the following settings:

- register

  This allows you to specify the name of the cell register used to mark the cells for poor mesh numerics treatment. For more information about cell registers, see Using Cell Registers (p. 3851).

- frequency/

  Using the text commands in this menu, you can define when the cells of the register are marked. Using the option text command, you can specify that the cells are marked either manually at the start of the calculation (single-execution), or automatically at an interval of iterations or (for transient simulations) time-steps. For automatic marking, after you set the

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

option you must then define the interval value using the iterations or time-steps text command. By default, the marking occurs automatically at every iteration.

- active?

  This allows you to activate or deactivate the marking of cells from the register for poor mesh numerics treatment. It is active by default.

- verbosity

  This determines the level of detail about cell marking for register-based poor mesh numerics printed in the console during the calculation. The default value of 0 prints no information, whereas entering 1 results in information being printed about the number of cells that are marked.

After you have set up the register-based definition you can use the new text command again to create additional definitions, and/or use the following from the solve/set/poor-mesh-numerics/register-based/ text command menu:

- edit

  This allows you to edit a register-based definition for poor mesh numerics, using the same text commands available from the new text command.

- delete

  This allows you to delete a register-based definition for poor mesh numerics.

- list

  This prints the names of the register-based definitions for poor mesh numerics in the console.

- list-properties

  This allows you to print the settings for a register-based definition for poor mesh numerics in the console.

Important:

When manually marking cells in a register for poor mesh numerics treatment (using the single-execution option), it is your responsibility to maintain the register-based cells. If you change the mesh, (for example, using mesh manipulation or adaption), then you must use the active? text command repeatedly to update the marking: enter no to deactivate it, and then enter yes to activate it and mark the cells in the updated register.

When you initialize or read data for a case with non-premixed or partially premixed combustion, a register-based definition for poor mesh numerics named pmn_pdf_stability_marker is automatically created. This definition is deactivated by default; if activated, it may increase the robustness of combustion simulations that use a PDF table. For further details, see Enabling Robust Numerics for Combustion with a PDF Table (p. 2532).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that when using register-based definitions for poor mesh numerics, the reporting of poor mesh element statistics (using the `mesh/repair-improve/report-poor-elements` text command) will also include the cells added from the registers, as well as their neighboring cells, resulting in an overall larger number of cells being identified. For example:

```
Poor Mesh Element Statistics:
Identified    0 faces with too small area.
Identified    0 faces adjacent to negative volume cells.
Identified    0 faces adjacent to bad quality cells.
Identified   43 cells from register-based poor mesh numerics and neighbors.
```

There are additional text commands that can be useful when applying poor mesh numerics, which are defined as follows:

`solve/set/poor-mesh-numerics/print-poor-elements-count`

Print out a listing of the number of poor cells marked for each criterion: default, cell quality, and register-based, and (if enabled) cell gradient quality and solution and cell quality.

```
> solve/set/poor-mesh-numerics/print-poor-elements-count
- Number of marked cells based on default criterion  = 5
- Number of marked cells based on cell quality criterion  = 11
- Number of marked cells based on register-based criterion and neighbors  = 0
- Number of marked cells based on cell gradient quality criterion  = 0
- Number of marked cells based on solution and cell quality criterion  = 0
```

`solve/set/poor-mesh-numerics/reset-poor-elements?`

Reset the poor cells included by the default, cell quality, register-based, cell gradient quality, and solution and cell quality criteria. The poor cells identified from the internal mesh check will be retained.

`solve/set/poor-mesh-numerics/turbulence-production-term?`

Enables a specific poor mesh numerics treatment for turbulence models. The shear production term is approximated by the destruction term in order to avoid zero turbulence shear production and thus improve the behavior on poor mesh cells. This treatment is available for the following two-equation turbulence models:

- Standard, RNG, and Realizable k-ε models (also DES with Realizable k-ε)
- Standard k-ω, BSL, SST, and GEKO (also in combination with SAS, DES, SBES, and also algebraic and one-equation transition models if available)
- Transition SST

36.21.6. Warped-Face Gradient Correction

The warped-face gradient correction (WFGC) is designed to improve gradient accuracy for all gradient methods—cell based, node based and least squares based gradients. WFGC is recommended for 3D simulations on polyhedral, hexahedral, and hybrid meshes (meshes containing a mix of hexahedral and polyhedral cells), and it is enabled by default when you read a mesh that contains polyhedra (note that if you append a mesh that contains polyhedra to a mesh that doesn't, you would have to enable WFGC manually in the **Solution Methods** task page if you want to use it). Note that these types of mesh cells may not have perfectly planar faces, which can affect the overall accuracy of the

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Green Gauss surface integral. WFGC is also recommended when you encounter numerical difficulties in simulations that involve a mesh that has large differences in the volumes of neighboring cells, such as hex cells with hanging node interfaces.

WFGC computations aim to resolve gradient accuracy degradation due to very high aspect ratio, cells with non-flat faces in the boundary layer, and highly deformed cells where the formal cell centroid lies outside of the control volume.

WFGC has two modes, fast mode and memory saving mode. Fast mode does not increase gradient computation time for static meshes (except for cases that use the **Green-Gauss Node Based** gradient method with a turbulence model), but it does require additional memory. There is roughly a 22% increase in memory for a basic flow solution, but this additional overhead does not change as more models (turbulence, species transport, radiation, and so on) are enabled. Memory saving mode does not increase memory consumption, but it slows down iteration execution time for static and dynamic meshes.

Important:

Enabling **Warped-Face Gradient Correction** through the **Solution Methods** task page enables fast mode. To switch to memory saving mode, you must use the TUI command `solve/set/warped-face-gradient-correction/enable?` and enter `no` in response to the `use fast mode?` prompt.

WFGC does *not* correct for cells with left-handed faces. If your mesh contains left-handed faces, you must use poor mesh numerics to obtain a stable solution.

36.21.7. Numerical Noise Filter for the Energy Equation

Numerical noise can arise in the energy solution field when there are large variations in properties at certain locations, such as the heat capacity at coupled walls or the combustion variables in cases with phase change. With the use of high-order discretization, the numerical noise remains in the solution fields, which contaminates the solution and may lead to divergence. An energy equation numerical noise filter is available for the pressure-based solver, which aims to remove the numerical noise from the domain by applying a mathematical extrema condition in the numerical noise region. This process slightly reduces accuracy in those regions, but increases the robustness of the calculation. The numerical noise filter is not applied after the numerical noise is removed from the region, in order to regain high-order accuracy. To enable / disable the energy equation numerical noise filter, use the following text command:

`solve` → `set` → `advanced` → `energy-numerical-noise-filter`

Note that the energy equation numerical noise filter is not enabled by default. For problematic cases where the energy equation diverges or generates non-physical results, it is recommended that you use this filter from the start of the simulation, rather than from intermediate data where the solution already has symptoms of divergence.

36.22. Solution Steering

For additional information, see the following sections:

36.22.1. Overview of Solution Steering

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

36.22.2. Solution Steering Strategy
36.22.3. Using Solution Steering

36.22.1. Overview of Solution Steering

Solution steering in the density-based implicit solver provides you with an expert system that will help navigate the flow solution from a starting initial guess to a converged solution with minimum user interaction. When you apply solution steering, you will be required to select the type of flow that best characterizes the solution domain and the maximum desired accuracy, and then allow the solver to take the solution to convergence. As the solver proceeds with the solution iteration, certain solver parameters will be adjusted behind the scenes to insure that a converged solution to steady-state is possible.

Important:

Solution steering is available only for steady-state flows in the density-based implicit solver.

36.22.2. Solution Steering Strategy

The convergence to steady-state solution is achieved in two stages. The parameters that are used in these stages are determined and set based on user input for the type of flow that can best characterize the solution domain. The type of flows available for selection are classified based on flow compressibility as well as the dominant flow Mach number in the solution domain.

The following flow types are available:

- Incompressible (if the flow is incompressible, that is density is constant)
- Subsonic (if the flow is compressible and M<0.75)
- Transonic (if the flow is compressible and 0.65<M<1.2)
- Supersonic (if the flow is compressible and 1.10< M<2.5)
- Hypersonic (if the flow is compressible and 2.0< M)

Important:

There is no exact Mach number cut-off for these regions, therefore, the above Mach number ranges are just a simple guideline to help you select a flow type.

Solution steering will typically perform full multigrid (FMG) initialization followed by two iterative stages. The purpose of each stage is described below.

36.22.2.1. Initialization

Immediately before the start of the iteration, solution steering will perform full multigrid initialization to obtain the best possible initial starting solution.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Stage 1:

The purpose of Stage 1 is to navigate the solution from the difficult initial phase of the solution toward convergence by insuring maximum stability. During this stage, the solution is advanced gradually from 1st-order accuracy to maximum accuracy (user specified and typically 2nd-order) at a constant low CFL value.

Stage 2:

In this stage the solution is driven hard towards convergence by regular adjustments of the CFL value to insure fast convergence as well as to prevent possible divergence.

In stage 2, the residual history is monitored and analyzed through regular intervals to determine if an increase or decrease in CFL value is needed to obtain fast convergence or to prevent divergence.

36.22.3. Using Solution Steering

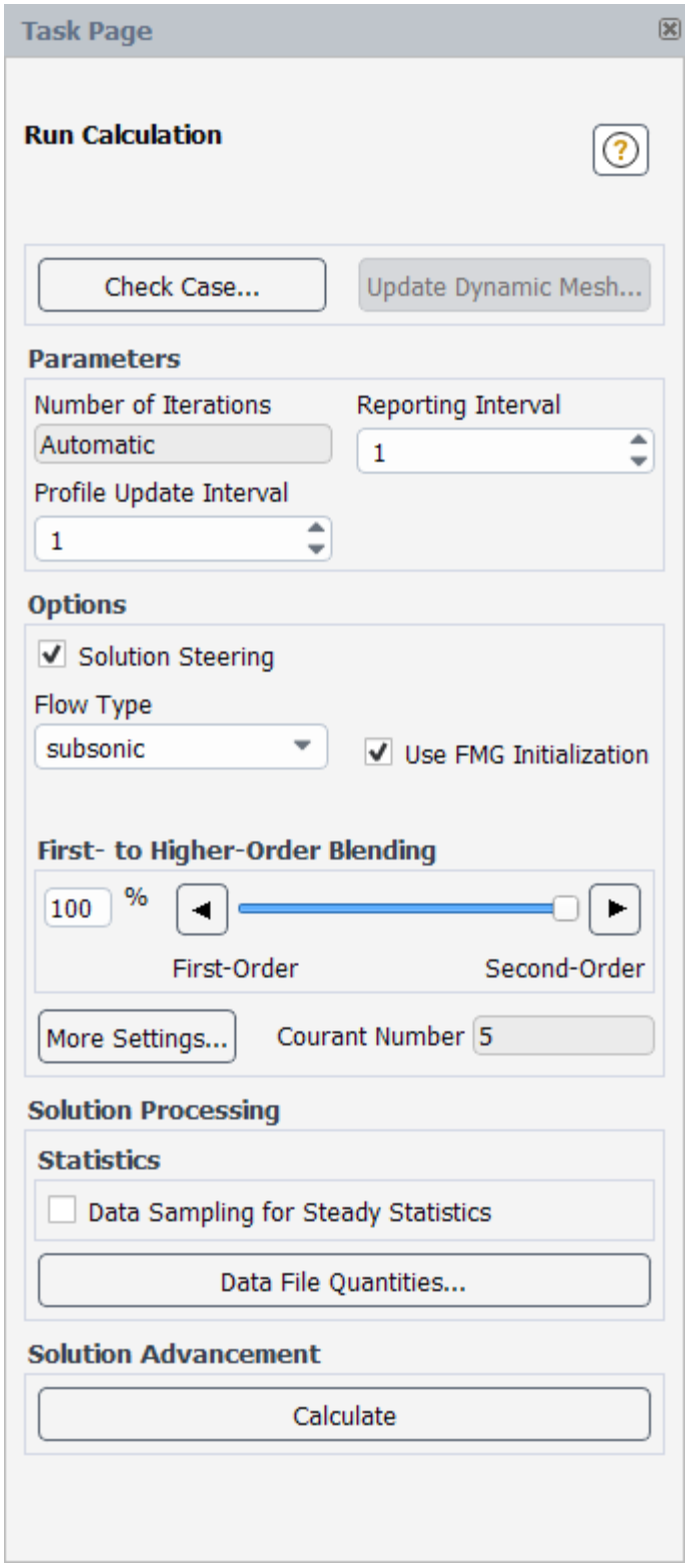
Solution steering is disabled by default. However, when the following criteria are met, the solution steering feature will become available for selection:

Feature	Setting
Solver Type	Density Based
Solver Formulation	Implicit
Time Formulation	Steady
Data is valid (either data file has been read or flow has been initialized)	

To turn on solution steering, enable the **Solution Steering** option as shown in Figure 36.65: The Run Calculation Task Page with Solution Steering Enabled (p. 3749).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 36.65: The Run Calculation Task Page with Solution Steering Enabled



The **Run Calculation** task page will then expand to display the solution steering main controls (see Figure 36.65: The Run Calculation Task Page with Solution Steering Enabled (p. 3749)). To obtain a flow solution using solution steering, you will need to perform the following:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the type of flow.
2. Select the maximum accuracy desired (first- to second-order blending).
3. Click **Calculate**.

You can also adjust the number of iterations, or customize the parameters of the solution steering if the default setting is not sufficient for the type of flow problem being solved.

Before using solution steering, you will need to prepare and set up the case as usual as described in the Getting Started (p. 1) part of this manual.

After **Solution Steering** is enabled, specify the following:

Flow Type

allows you to select the flow type that best describes the flow in the solution domain. Five choices are available: **incompressible**, **subsonic**, **transonic**, **supersonic**, and **hypersonic**.

Use FMG Initialization

when enabled allows for full multigrid initialization before starting stages 1 and 2. FMG initialization is enabled by default.

First- to Higher-Order Blending

allows you to reduce the desired solution accuracy by selecting a blending factor less than 100%. The default setting is 100%. See First- to Higher-Order Blending in the Theory Guide for more information. The blending factor will be grayed out if **Second Order Upwind** discretization for the **Flow** equations is not selected in the **Solution Methods** task page. The solution accuracy may be reduced (typical values are 75% or 50%) if it is not possible to obtain a converged solution with the maximum second-order accuracy (that is blending = 100%).

Courant Number

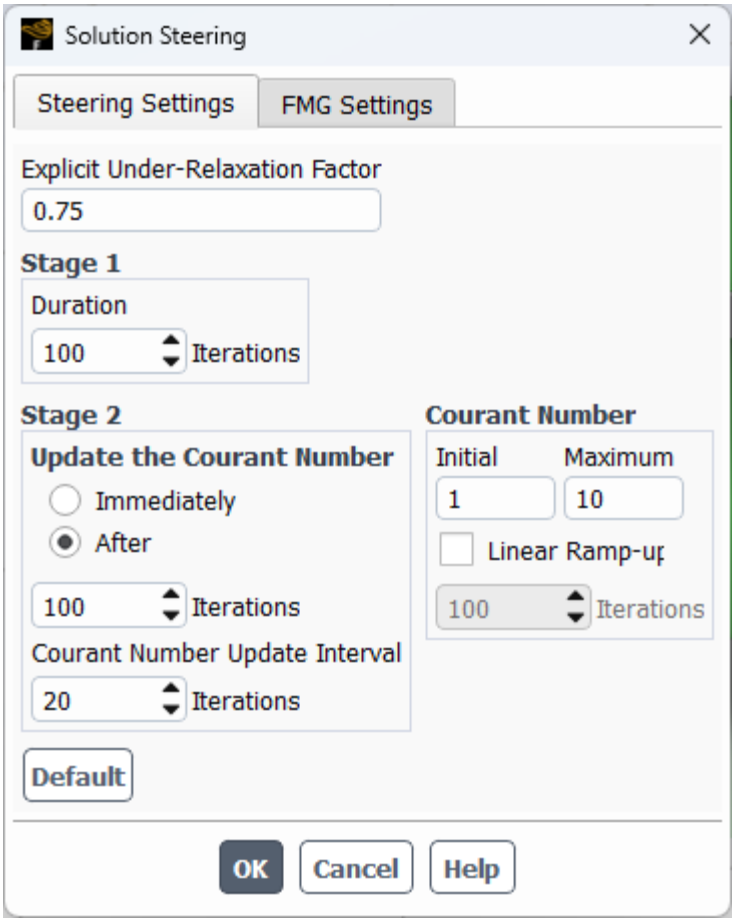
in the **Run Calculation** task page is a non-adjustable field displaying the current CFL number, which allows you to view it during the calculation.

More Settings...

opens the **Solution Steering** dialog box, providing a host of settings that control the solution steering strategy, as shown in Figure 36.66: The Solution Steering Dialog Box (p. 3751).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 36.66: The Solution Steering Dialog Box



The **Solution Steering** dialog box, shown in Figure 36.66: The Solution Steering Dialog Box (p. 3751), contains two tabs. The **Steering Settings** tab sets the solution steering parameters and the **FMG Settings** sets the full multigrid initialization parameters.

In the **Steering Setting** tab, you can modify the parameters used in Stages 1 and 2.

Explicit Under-Relaxation Factor

allows the solution to be under-relaxed to improve convergence. The under-relaxation value is determined by the **Flow Type** that you selected in the **Run Calculation** task page, when **Solution Steering** was enabled. In general, you do not need to alter the default value set in this field. Refer to Under-Relaxation of Variables in the Theory Guide for more information about explicit relaxation.

Stage 1

Duration is the number of iterations in stage 1. The CFL number used during these iterations is set in the **Initial** field, in the **Courant Number** group box.

Stage 2

The Courant number update in stage 2 can start immediately after the end of stage 1, or after a certain designated number of iterations. If the Courant number update is to start immediately after stage 1 then **Immediately** should be selected (this is the default option). If the Courant number update is desired after some lagged period of iterations, then **After** should be selected

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and the lag in the number of iterations should be entered in the field below it. The frequency at which the Courant number is updated is defined in **Courant Number Update Interval** field.

Courant Number

Initial is the starting Courant number and **Maximum** is the maximum allowed Courant number. The solution steering algorithm will not allow the solver to exceed the maximum Courant number, but will allow the solver to use a Courant number less than the initial Courant number if divergence in the solution has occurred.

If **Linear Ramp-up** is selected, the active Courant number steering in stage 2 is replaced by a linear ramp-up of the CFL. The Courant number is increased from the **Initial** to **Maximum** value over a set number of iterations, determined by the **Iterations** field. The Courant number may be automatically reduced if divergence is detected.

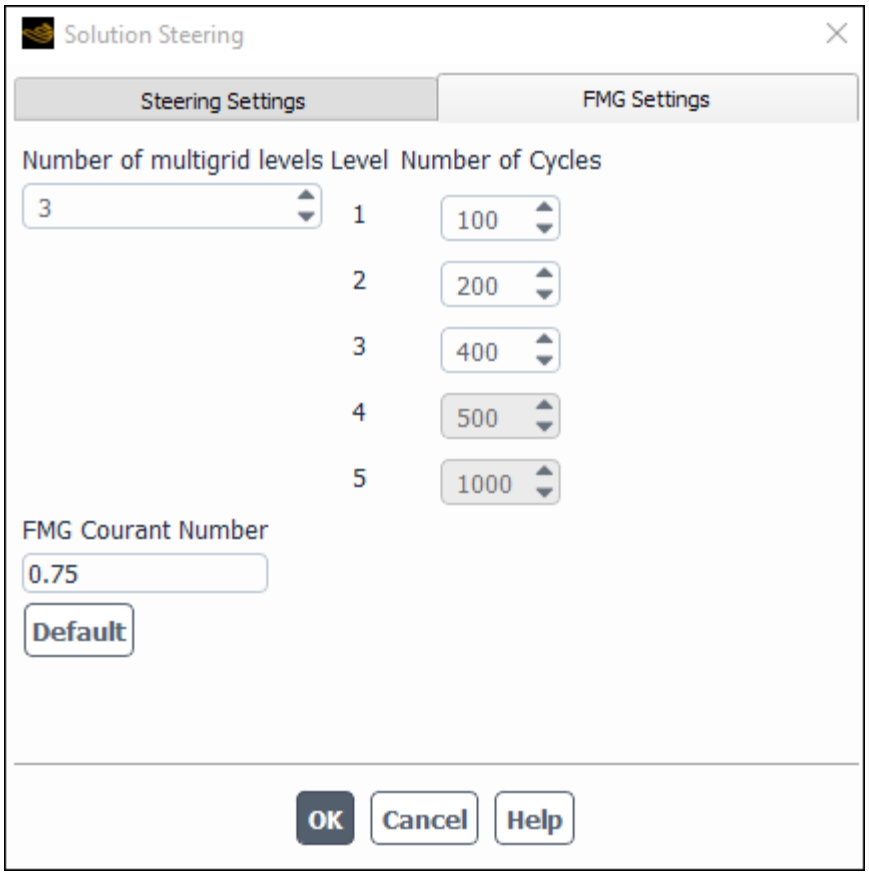
Default

is available in the **Steering Settings** tab to reset any changes made to the parameters to their original default values.

In the **FMG Settings** tab (Figure 36.67: The FMG Settings Tab in the Solution Steering Dialog Box (p. 3753)), the **Number of multigrid levels** and **Number of Cycles** in each **Level**, as well as the **FMG Courant Number** used in the FMG initialization can be adjusted. The default values used in the multigrid settings are determined from the type of flow that you selected, the size of the mesh, and the flow dimensionality. The **Default** button is used to reset any changes to the original default values. For more information about FMG initialization, refer to Full Multigrid (FMG) Initialization (p. 3633).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 36.67: The FMG Settings Tab in the Solution Steering Dialog Box



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 37: Using the Fluent Native GPU Solver

The following sections describe the capabilities of the GPU-powered solver in Ansys Fluent.

37.1. Introduction to the Fluent GPU Solver

37.2. Supported GPUs and Drivers

37.3. Basic Steps for CFD Analysis Using the Fluent GPU Solver

37.4. Starting the Fluent GPU Solver

37.5. Exiting the Fluent GPU Solver

37.6. Graphical User Interface (GUI)

37.7. Using CPU Processes for Setup and Postprocessing

37.8. Reading Fluent Case Files Into the Fluent GPU Solver

37.9. Features Supported by the Fluent GPU Solver

37.10. Fluent GPU Solver Limitations

37.11. Transitioning from a Steady-State Solution to a Transient Calculation

37.12. Troubleshooting Cases

37.13. Resolving GPU Solver Performance Issues

37.14. GPU Memory Usage

37.1. Introduction to the Fluent GPU Solver

The Fluent GPU solver is a native GPU-powered Ansys Fluent solver, which uses the high performance of graphics processing units (GPUs) to facilitate the computation of complex simulations. The Fluent GPU solver can be used with one or multiple GPUs which is specified in the Fluent Launcher or the command line. Unlike CPUs, GPUs are optimized for parallel processing and have high bandwith memory, making them ideal for calculating computationally intensive flow solutions. This increase in performance of the GPU-powered solver has shown to increase the speed of flow calculations by up to 5 times that of traditional multi-core CPU-based computations.

37.2. Supported GPUs and Drivers

The Fluent GPU Solver is enabled by CUDA for NVIDIA GPUs and AMD ROCm for AMD GPUs, both of which are high performance computing platforms developed by NVIDIA and AMD, respectively.

To run the GPU solver on one or multiple NVIDIA GPUs, your driver(s) must be compatible with CUDA version 11.8 or newer. If your driver is not compatible with at least CUDA version 11.8, then you must update your driver. You can check to see if your driver is compatible by entering `nvidia-smi` in the command prompt window, as described in Starting the Fluent GPU Solver from the Command Line (p. 3758). To update your GPU driver with the latest version of CUDA see https://developer.nvidia.com/cuda-downloads.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To run the GPU solver on one or multiple AMD GPUs, you must have AMD ROCm-6.0 or newer installed. To update your AMD ROCm software to the latest version, see https://www.amd.com/en/developer/resources/rocm-hub/hip-sdk.html#sdk.

37.3. Basic Steps for CFD Analysis Using the Fluent GPU Solver

Before you begin your CFD analysis using the Fluent GPU Solver, careful consideration of the following issues will contribute significantly to the success of your modeling effort. Additionally, when planning a CFD project, be sure to take advantage of the customer support available to all Ansys Fluent users.

Many of the features available for the CPU-driven solution mode of Ansys Fluent outlined in the Fluent User's Guide are not yet supported by the Fluent GPU Solver. Your CFD analysis using the Fluent GPU Solver must adhere to the supported features and limitations as outlined in Features Supported by the Fluent GPU Solver (p. 3760) and Fluent GPU Solver Limitations (p. 3767).

After you have determined the important features of the problem you want to solve perform the following basic procedural steps:

1. Define the modeling goals.
2. Specify one or multiple GPUs to be used with the GPU solver within the Fluent Launcher or within the command prompt, as outlined in Starting the Fluent GPU Solver (p. 3756).
3. Read in a 3D mesh or case file (with or without a corresponding data file) to represent the fluid domain. For information on reading a Fluent case file into the Fluent GPU Solver, see Reading Fluent Case Files Into the Fluent GPU Solver (p. 3760).
4. Set up the models, materials, cell zones, boundary properties, and report definitions.

   The settings defined for your CFD analysis using the Fluent GPU Solver must adhere to the GPU solver supported features and limitations.
5. Set up calculation activities.
6. Compute the solution. For information on computing steady-state solutions and transient solutions see Performing Steady-State Calculations (p. 3639) and Performing Time-Dependent Calculations (p. 3643), respectively.
7. Examine and save the results.
8. Consider revisions to the setup, if necessary.

37.4. Starting the Fluent GPU Solver

The following sections describe how to start the GPU solver in Fluent solution mode:

37.4.1. Starting the Fluent GPU Solver Using the Fluent Launcher
37.4.2. Starting the Fluent GPU Solver from the Command Line

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.4.1. Starting the Fluent GPU Solver Using the Fluent Launcher

To start the Fluent GPU Solver from the Fluent Launcher:

1. Open the Fluent Launcher. *Refer to Starting Ansys Fluent Using Fluent Launcher in the Fluent User's Guide (p. 180) for additional information on opening the Fluent Launcher.*

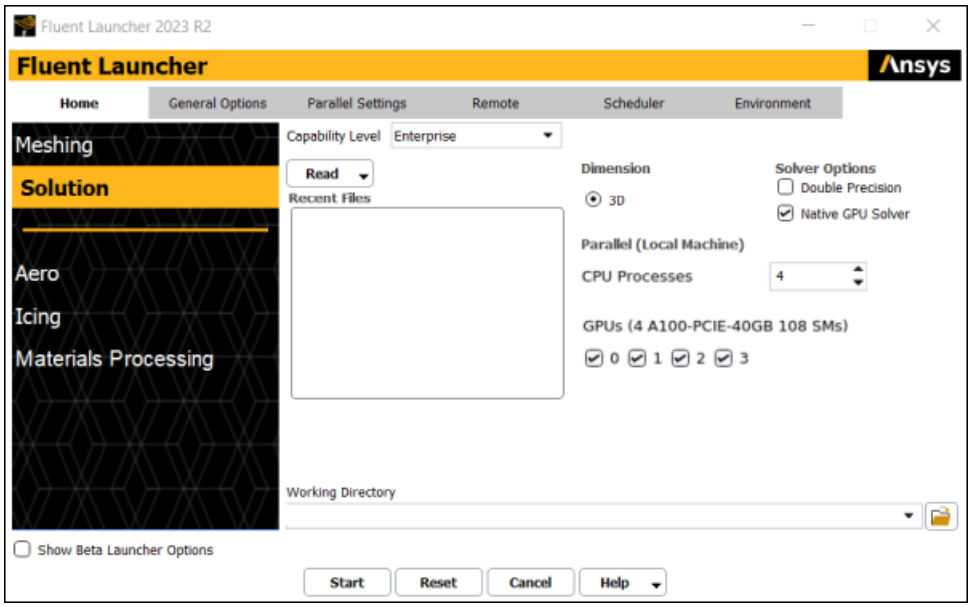
2. Select **Enterprise** from the **Capability Level** drop-down list.

   Important:

   The Fluent GPU Solver is only available at the **Enterprise** licensing level.

3. Select **Solution** in the top-left selection list.

4. Under **Solver Options**, enable the **Native GPU Solver** option.



5. If you have multiple GPUs installed on your machine and intend to use the Fluent GPU Solver with multiple GPUs, enable the IDs of the GPUs you would like to use. To check which IDs are associated with the GPUs on your machine, you can open a command prompt window and enter the `nvidia-smi` command. Note that for each additional GPU used by the GPU solver, an additional CPU process is required and will be added under **Parallel (Local Machine)**.

6. (Optional) Under **Parallel (Local Machine)**, you can increase the number of **CPU Processes** to use multiple CPU compute nodes in parallel during case setup and postprocessing. Note that the number of **CPU Processes** specified must be greater than or equal to the number of GPUs specified.

   CPU Processes *are only used for case setup and postprocessing and are not involved with the GPU solver calculation. For more information on using the GPU solver with multiple CPU processes see Using CPU Processes for Setup and Postprocessing (p. 3759)*

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

7. (Optional) Select a Case | Case and Data | Mesh | Journal to start. Selecting any of these starting files and launching the Fluent GPU Solver automatically loads the selected file. *The **Start** button also becomes **Start With Selected Options**.*

8. Click **Start**.

37.4.2. Starting the Fluent GPU Solver from the Command Line

For Windows, prior to starting the Fluent GPU Solver from the command line, ensure that the path to your home directory is in your command search path environment variable by executing the `setenv.exe` program located in the Ansys Fluent directory (for example, `C:\Program Files\ANSYS Inc\v232\fluent\ntbin\win64`).

Note:

For Linux, the best graphics driver is selected automatically, and the `X11` driver is used by default when the required graphics support is not detected. If you feel it is necessary to use an alternate driver, you can include the `-setenv="HOOPS_PICTURE=` *driver_name* `"` command line option, where *driver_name* is the preferred graphics driver.

Important:

Using the Fluent GPU Solver is only available at the **Enterprise** licensing level.

Once your working directory path is specified, you can execute the `nvidia-smi` command in the command prompt window. This will display the GPUs available on your machine as well as their current usage. Each GPU on your machine will have an ID of `0` to `7` after executing `nvidia-smi`.

```
C:\Program Files\ANSYS Inc\v231\fluent\ntbin\win64> nvidia-smi
+-----------------------------------------------------------------------------+
| NVIDIA-SMI 496.13       Driver Version: 496.13       CUDA Version: 11.5     |
|-------------------------------+----------------------+----------------------+
| GPU  Name            TCC/WDDM | Bus-Id        Disp.A | Volatile Uncorr. ECC |
| Fan  Temp  Perf  Pwr:Usage/Cap|         Memory-Usage | GPU-Util  Compute M. |
|                               |                      |               MIG M. |
|===============================+======================+======================|
|   0  Quadro P4000       WDDM  | 00000000:D5:00.0  On |                  N/A |
| 46%   35C    P8    11W / 105W |   3463MiB /  8192MiB |      1%      Default |
+-------------------------------+----------------------+----------------------+

+-----------------------------------------------------------------------------+
| Processes:                                                                  |
|  GPU   GI   CI        PID   Type   Process name                  GPU Memory |
|        ID   ID                                                   Usage      |
|=============================================================================|
|  No running processes found                                                 |
+-----------------------------------------------------------------------------+
```

The command to launch the GPU solver (with single precision) within the solution mode of Ansys Fluent resembles the following format:

```
fluent 3d -tn -gpu
```

Alternatively, you can replace `3d` with `3ddp` to use the GPU solver with double precision.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Depending on the number of GPUs available on your machine, n may represent either the number of CPU processes only, or both the number of CPU processes and the number of GPUs. Note that the GPU solver uses multiple CPU processes for case setup and postprocessing only (see Using CPU Processes for Setup and Postprocessing (p. 3759) for details). The value of n will specify the CPU and GPU configuration for the GPU solver as follows:

- If only one GPU is available on your machine, the GPU solver will use one GPU and the value of n will only specify the number of parallel CPU processes.
- If multiple GPUs are available on your machine and the value of n is less than the number of available GPUs, the above command will specify the GPU solver to use n CPU processes and n GPUs. By default, the GPU solver will use n GPUs in the order they were listed by the `nvidia-smi` command.
- If multiple GPUs are available on your machine and the value of n is greater than or equal to the number of available GPUs, the above command will specify the GPU solver to use n CPU processes and all of the GPUs available on your machine.

You can manually specify the GPU solver to use specific GPUs available on your machine by setting `-gpu` equal to the IDs of the GPUs you want to use. For example, entering `fluent 3d -t3 -gpu=1,2,4` will launch the GPU solver using the second, third, and fifth GPUs on your machine, which have IDs of 1, 2, and 4, respectively.

37.5. Exiting the Fluent GPU Solver

You can exit the Fluent GPU Solver by selecting **Exit** in the **File** ribbon tab or by clicking the ✕ button in the top right corner of the application. For the latter, a **Question** dialog box will open to confirm if you want to proceed.

37.6. Graphical User Interface (GUI)

The graphical user interface (GUI) of the Fluent GPU Solver is similar to the GUI of Fluent Solution Mode, with the only difference being that certain Fluent features that are not supported by the GPU solver are hidden in the GUI. For details on using the GUI in Fluent solution mode (such as setting preferences or writing and reading journals, case, data, and mesh files) see Graphical User Interface (GUI) (p. 901).

37.7. Using CPU Processes for Setup and Postprocessing

For large cases, such as cases that contain a large number of cell zones, you can utilize multiple CPU processes in parallel to reduce both the setup and postprocessing time for your simulaton. Multiple CPU processes for setup and postprocessing can be specified in the Fluent Launcher or through the command prompt, as outlined in Starting the Fluent GPU Solver (p. 3756).

After reading in a mesh or Fluent case file, you must partition or subdivide the mesh into groups of cells that will be solved on each CPU processor. For detailed information on partitioning the mesh see Mesh Partitioning and Load Balancing (p. 4315). Upon starting the calculation, the partitions created for setting up the case will be combined into a single partition for each GPU being used by the Fluent GPU Solver.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.8. Reading Fluent Case Files Into the Fluent GPU Solver

Case files that have been created within Fluent solution mode can be read into the Fluent GPU Solver. When reading in a Fluent case as outlined in Reading and Writing Case Files (p. 956), Fluent will attempt to convert any features that are not supported by the GPU solver. Additionally, unsupported features that can not be converted will be disabled by default. After reading in a case, a table similar to the following will be printed in the console that shows any converted or disabled features:

```
GPU solver defaults are activated for the following unsupported settings (5)
================================================================================================
Area                  Settings                          From                To
------------------------------------------------------------------------------------------------
methods               density                           second-order upwind  first-order upwind
solver                rhie chow                         distance-weighted    momentum-weighted
solver                warped face gradient correction   on                   off
solver                residual scaling                  global               local
hybrid initialization averaged turbulent parameters     enabled              disabled
```

Although Fluent will attempt to convert or disable most unsupported features after reading a case, there may be some settings that have not been disabled or converted, and are not printed in the console. Upon starting the calculation, if there are any unsupported features still defined, a message similar to the following will be printed in the console:

```
Following report-files are not valid:
  voltage_vp-rfile:
    Contains following invalid reports:
      voltage_vp:
        Field is not available

Run can not be started until validation issues are resolved.
Deactivate invalid report clients to continue?
```

You should ensure that the Fluent case file only contains features supported by the GPU solver as outlined in Features Supported by the Fluent GPU Solver (p. 3760), and make any necessary adjustments to the case.

37.9. Features Supported by the Fluent GPU Solver

This section describes the features of the CPU-driven solution mode of Fluent that can be used with the Fluent GPU Solver. The features outlined in this section can be defined after you have read in a previously defined Fluent case file (see Reading Fluent Case Files Into the Fluent GPU Solver (p. 3760)) or after reading in a mesh file (see Reading Mesh Files (p. 953)).

This section only covers those settings relevant to setting up your case and does not include information on calculating or postprocessing the GPU solver solution. The settings and procedure for defining calculation activities, calculating the solution, and postprocessing the solution within the Fluent GPU Solver is the same as that of the CPU-driven Fluent solver. Note that the supported features outlined in this section may contain limitations, which are detailed in Fluent GPU Solver Limitations (p. 3767).

Prior to setting up your case in the Fluent GPU Solver, note the following general requirements and capabilities:

- The Fluent GPU Solver can only be used with either a single NVIDIA GPU or multiple NVIDIA GPUs with shared/distributed memory. For details on setting specifying GPUs to be used see Starting the

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fluent GPU Solver Using the Fluent Launcher (p. 3757) or Starting the Fluent GPU Solver from the Command Line (p. 3758).

- The Fluent GPU Solver can be used in single precision or double precision.
- Multiple CPU processes can be specified for setting up your case and postprocessing the GPU solver solution. See Using CPU Processes for Setup and Postprocessing (p. 3759) for details.
- Only **3D** geometries are supported.
- Mesh topologies with polyhedral, hexahedral, tetrahedral, pyramid, and prism cells, as well as meshes with hanging nodes are supported.

The settings that can be defined when setting up your case within the Fluent GPU Solver are detailed in the following sections:

37.9.1. Models
37.9.2. Material Properties
37.9.3. Solver Settings
37.9.4. Parametric Studies
37.9.5. Cell Zone and Boundary Conditions
37.9.6. Mesh Interfaces
37.9.7. Solution Monitors and Report Definitions

37.9.1. Models

The following physics modeling capabilities are supported with the Fluent GPU Solver:

- Conjugate Heat Transfer (CHT) including for anisotropic solids as described in Conjugate Heat Transfer (p. 2169) and Settings for Anisotropic Solid Zones (p. 2179).
- Discrete Ordinates (DO) radiation model. See Setting Up the DO Model (p. 2211) for details.
- Species transport with **Volumetric** reactions or without reactions as outlined in Volumetric Reactions (p. 2356) and Species Transport Without Reactions (p. 2427), respectively. Species transport models can be set up with the following settings:
  - **Finite-Rate/No TCI** and **Eddy-Dissipation** turbulence-chemistry interaction models.
  - **Stiff Chemistry Solver** and **None - Direct Source** chemistry solvers.
- Compressible flows (see Compressible Flows (p. 1759) for details).
- The following turbulence models are supported:
  - **Laminar**
  - **Standard** and **Realizable k-epsilon**
  - **k-omega**, **GEKO**, and **k-omega SST**
  - **Large Eddy Simulation (LES)**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For the **LES** turbulence model, the following sub-grid scale models are supported:

→ **Smagorinsky-Lilly**

→ **WALE**

For the **LES** turbulence model, the following additional model options are available:

→ **Near-Wall-RANS-Layer (WMLES)**

When enabled, this option activated the Generic WMLES model (Generic WMLES Formulation in the *Fluent Beta Features Manual*), which uses LES for the bulk of the domain and the algebraic RANS model for the thin near-wall section of the boundary layer.

→ **Optimized LES Numerics**

When enabled, this option will automatically apply optimal solution controls to enhance stability of the solution.

– **Stress-Blended Eddy Simulation (SBES)**

Only supported with any of the **k-omega** models.

For detailed information on setting up the turbulence models listed above see Modeling Turbulence (p. 2061).

- Sponge layers acoustics model (see Sponge Layers (p. 2667) for details).

  Note that multiple sponge layers are not supported and only one sponge layer can be activated within the **Manage Sponge Layers** dialog box. If multiple sponge layers are activated, the GPU Solver will only simulate the first sponge layer listed and all other sponge layers will be ignored.

37.9.2. Material Properties

The Fluent GPU Solver supports the following material properties:

- **Density**
  - **piecewise-linear**
  - **piecewise-polynomial**
  - **polynomial**
  - **ideal-gas**
  - **incompressible-ideal-gas**
  - **boussinesq**
  - **constant**
- **Cp**:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - **constant**
  - **piecewise-linear**
  - **piecewise-polynomial**
  - **polynomial**
  - **kinetic-theory**
- **Thermal Conductivity**
  - **constant**
  - **piecewise-linear**
  - **piecewise-polynomial**
  - **polynomial**
  - **kinetic-theory**
  - **cyl-orthotropic**
  - **orthotropic**
  - **anisotropic**

  The **cyl-orthotropic**, **orthotropic**, and **anisotropic** options listed above are available when simulating conjugate heat transfer with anisotropic/orthotropic conductivity for solids. For details see Anisotropic Thermal Conductivity for Solids (p. 1648).
- **Viscosity**
  - **constant**
  - **piecewise-linear**
  - **piecewise-polynomial**
  - **polynomial**
  - **kinetic-theory**
  - **sutherland**
- **Molecular Weight**, **Standard State Enthalpy**, **Standard State Entropy**, and **Reference Temperature** can be defined as:
  - **constant**

For more details on setting up material properties for your simulation see Physical Properties (p. 1597).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.9.3. Solver Settings

The Fluent GPU solver supports the pressure based solver with absolute velocity formulation for both transient and steady-state calculations. Pressure-velocity coupling can be specified as **Coupled** (steady state only), or segregated with the **SIMPLE** and **SIMPLEC** schemes. The GPU solver supports the **Least Squares Cell Based** gradient and both second-order and first-order discretization schemes for the flow equations. Additionally, the **Bounded Central Differencing** scheme is available when the **Large Eddy Simulation (LES)** or **Stress-Blended Eddy Simulation (SBES)** turbulence models are enabled. For incompressible time-dependant calculations, both first order and second order implicit are supported for the **Transient Formulation** (see Inputs for Time-Dependent Problems (p. 3645)). Additionally, the **Flow Courant Number** is supported for the **Coupled** solver when **Pseudo Time Method** is set to **Off**.

For poor quality meshes, you can incorporate poor mesh numerics by entering the following TUI command, which will delete any cells with left-handed faces:

```
solve/set/poor-mesh-removal/enable? yes
```

For steady-state cases, **Data sampling for Steady Statistics** is supported within the **Run Calculation** task page.

For additional information on specifying the above solver settings, see Choosing the Spatial Discretization Scheme (p. 3575).

37.9.4. Parametric Studies

The Fluent GPU solver supports updating parametric studies sequentially for variations in supported parameters. For more information on performing a sequential parametric study see Performing Parametric Studies (p. 4387).

37.9.5. Cell Zone and Boundary Conditions

This section describes the cell zone and boundary conditions that can be defined when using the Fluent GPU Solver.

The boundary conditions listed below can be defined as constant or as input parameters. Additionally, certain boundary condition settings can also be defined using a steady-state profile, as described in Profiles (p. 1564). Note that input parameters and steady-state profiles are not supported for **Mass Flow Rate** and **Speed** for rotational wall motion.

The following boundary conditions are supported:

- **Wall**
  - **Stationary Wall** and **Moving Wall**, including **Rotational** wall motion. See Wall Boundary Conditions (p. 1457) for details.
  - **Thermal** conditions. See Thermal Boundary Conditions at Walls (p. 1467) for details.
- **Symmetry**. See Symmetry Boundary Conditions (p. 1499) for details.
- **Rotational Periodic** boundary conditions. See Periodic Boundary Conditions (p. 1501) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Translational Periodic** boundary condition with or without a non-conformal mesh interface. See Periodic Boundary Conditions (p. 1501) for details.
- **Inlet**
  - **Velocity-inlet**. See Defining the Velocity (p. 1413) for details.
  - **Pressure-inlet**. See Pressure Inlet Boundary Conditions (p. 1401) for details.
  - **Mass-flow-inlet**. See Mass-Flow Inlet Boundary Conditions (p. 1417) for details.
- **Intake Fan**. See Intake Fan Boundary Conditions (p. 1432) for details.
- **Outlet**
  - **Pressure-outlet**. See Pressure Outlet Boundary Conditions (p. 1434) for details.
  - **Mass-flow-outlet**. See Mass-Flow Outlet Boundary Conditions (p. 1425) for details.
- **Pressure-far-field**
- **Outlet Vent**. See Outlet Vent Boundary Conditions (p. 1451) for details.
- **Porous Jump**. See Porous Jump Boundary Conditions (p. 1519) for details.

The following boundary conditions can be defined for both incompressible and compressible flows:

- **Velocity-inlet**
- **Mass-flow-inlet**
- **Mass-flow-outlet**

The following settings can be specified for fluid cell zone conditions:

- **Frame Motion** for Moving reference frames (MRF). See Modeling Flows with Moving Reference Frames (p. 1769) for details.
- **Mesh Motion**. See Defining Zone Motion (p. 1323) for details.
- **Porous Zone**. See Porous Media Conditions (p. 1333) for details.
- **Source Terms** for heat, momentum, and species are supported, as outlined in Defining Mass, Momentum, Energy, and Other Sources (p. 1382), and can be defined as constant or using a steady-state profile.
- **Reaction** mechanisms for species transport with reactions. See Specifying a Reaction Mechanism (p. 1323) for details.

37.9.6. Mesh Interfaces

The Fluent GPU Solver supports sliding meshes with both conformal and non-conformal mesh interfaces as outlined in Using Sliding Meshes (p. 1804). For details on defining non-conformal mesh interfaces see Using a Non-Conformal Mesh in Ansys Fluent (p. 1185).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

37.9.7. Solution Monitors and Report Definitions

This section describes the solution monitors that are supported by the Fluent GPU Solver.

Note:

Solution monitors can not be created on the following user defined surfaces:

- Iso-surfaces
- Iso-clips
- Zone surfaces
- Quadric-surfaces

The report definitions listed below can be created on an existing surface (boundary condition) or on a **Point...**, **Plane...**, or **Line...** surface.

- **Force**

  For details on defining the following force reports see Force and Moment Report Definitions (p. 4106).

  - **Drag...**
  - **Lift...**
  - **Moment...**
  - **Force...**

- **Flux**

  For details on defining the following flux reports see Flux Report Definition (p. 4111).

  - **Mass Flow Rate...**
  - **Total Heat Transfer Rate...**

- **Surface**

  For details on defining the following surface reports see Surface Report Definitions (p. 4103).

  - **Area...**
  - **Area-Weighted Average...**
  - **Facet Average...**
  - **Facet Maximum...**
  - **Facet Minimum...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - **Integral...**
  - **Mass Flow Rate...**
  - **Volume Flow Rate...**
  - **Mass-Weighted Average...**
  - **Sum...**
  - **Standard Deviation...**
  - **Uniformity Index - Mass Weighted...**
  - **Uniformity Index - Area Weighted...**

- **Volume**

  For details on defining the following volume reports see Volume Report Definitions (p. 4104).

  - **Mass-Average...**
  - **Mass Integral...**
  - **Mass...**
  - **Max...**
  - **Min...**
  - **Volume...**
  - **Volume-Average...**
  - **Volume Integral...**
  - **Sum...**

37.10. Fluent GPU Solver Limitations

This section outlines the current limitations of the Fluent GPU Solver. Fluent features that are not supported by the Fluent GPU Solver are not contained in this section. For information on Fluent features supported by the GPU solver see Features Supported by the Fluent GPU Solver (p. 3760).

The following list contains the current limitations of the Fluent GPU Solver:

- The following limitations apply to using the DO radiation model with the GPU solver:
  - Solar Load modelling is not supported.
  - DO/Energy Coupling is not supported.
  - Postprocessing of Wall Temperature (Thin) is not supported.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - Inputting radiation properties via expression or UDF is not supported.
  - Shell Conduction is not supported.
  - Directions indexing for the DO model is not consistent between the GPU and CPU Solver. Postprocessing directional variables may lead to differences observed when comparing GPU to CPU Solver.

- The GPU solver is not available in the Ansys Workbench environment.
- Larger reporting intervals for residuals and monitors is recommended for comparing the performance of the GPU solver to the CPU-driven Fluent solution. A reporting interval of `20` is recommended and can be specified with the following TUI command:

  ```
  /solve set report-interval 20
  ```

- Profiles in cylindrical coordinate systems, which includes those used for swirl inlets, are not supported.
- Monitors and report definitions must be defined prior to solution initialization or before calculating the solution. Additionally, if the solver settings are changed from steady to transient, it may be necessary to redefine report definitions.
- When monitoring mass flow rate at a non-conformal mesh interface where the fluxes are collected from intersected faces to the parent faces, the printed result will have the wrong sign (e.g -1 instead of 1).
- When monitoring **Dynamic Pressure** (**Pressure...** category) at a stationary wall boundary, the resulting value will be zero as expected. However, results computed using a report (as outlined in Creating Output Parameters (p. 4133)) will show an incorrect non-zero value.
- Static expressions are supported by the GPU solver but are not recommended.
- The **Turbulent Viscosity Ratio** is clipped to `10e5` in the same manner as the CPU-driven Fluent solver, however the GPU solver will not provide feedback to the transcript or console.
- When thermal effects are defined for your case as outlined in Modeling Thermal Energy (p. 2163), using a steady-state solution to initialize a transient case does not work correctly when solving the energy equation. This limitation can be resolved by performing the procedure outlined in Transitioning from a Steady-State Solution to a Transient Calculation (p. 3769).
- With the CPU-based Fluent solver, when modeling supersonic flow at a pressure inlet or velocity inlet all flow characteristics enter into the flow domain and the flow state must be fully specified. Similarly, when there is supersonic flow at a pressure outlet, all flow characteristics exit the domain and the specified pressure will be ignored by the solver. However, when modeling supersonic flow with the GPU solver, the flow will not be discretized correctly at pressure inlets, velocity inlets, and pressure outlets and the flow will be treated as subsonic at these boundary types. Supersonic static pressure at inlets will therefore be ignored, leading to an underspecified state which may produce an unreliable solution. Additionally, the specified static pressure on pressure outlets will still be enforced, which may lead to inconsistencies in the solution directly adjacent to the boundary.
- When modeling incompressible or incompressible ideal gas materials with the energy equation enabled, the temperature field calculated by the CPU-based Fluent solver and the GPU Solver may be different. This is due to different energy formulations (total energy and thermal energy) being used by the two solvers when computing incompressible flows. The CPU-based Fluent solver will use the thermal energy

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

model when incompressible or incompressible ideal gas materials are defined and viscous heating is not enabled. However, for all other physics models the CPU-based Fluent solver will use the total energy model. Additionally, the GPU Solver will always use the total energy model for all physics models. While the thermal energy model is more likely to satisfy temperature boundedness conditions, the thermal energy model may yield a more robust solution than the total energy model in some cases.

- When restarting a simulation using second-order transient, the solver uses the first-order Backward Euler scheme on the first timestep after restarting.
- Restarting a GPU Solver solution on the CPU-driven Fluent solver will not restart correctly if the energy equation with a compressible material is calculated in the GPU Solver solution.

37.11. Transitioning from a Steady-State Solution to a Transient Calculation

When calculating the solution for a transient case, it is common to first calculate a steady-state solution prior to running the transient calculation. To transition to a transient calculation from a steady-state solution, the following procedure is recommended:

1. Set up your case as steady.
2. Calculate the steady-state solution.
3. Change the case to transient within the general task page. See General Task Page (p. 4684) for details.
4. Calculate the transient solution.

37.12. Troubleshooting Cases

The following steps are recommended for resolving diverging cases within the Fluent GPU Solver:

1. Ensure that any enabled models, boundary conditions, or solver settings are supported by the GPU solver. If performing the calculation after reading in a Fluent case file, ensure that all unsupported features have been properly converted (see Reading Fluent Case Files Into the Fluent GPU Solver (p. 3760)).
2. If the flow solver is having difficulty when calculating a linear solution, you may encounter the following messages in the console:

   ```
   AMG (energy/solver/solve) It 5, Diverged.
   ```

   or

   ```
   AMG (energy/solver/solve) It 8, Stalled.
   ```

   Depending on the severity, you can increase the verbosity of the AMG solver within the **Advanced Solution Controls** dialog box as described in Setting Algebraic Multigrid Parameters (p. 3610) and restart the calculation. During the calculation, information similar to the following will be printed in the console:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
AMG (energy/solver/solve) residual history:
  It 0: Residual [rel,abs] = 1.0000e+00,6.4496e+04
  It 1: Residual [rel,abs] = 6.4405e-01,4.1539e+04
  It 2: Residual [rel,abs] = 7.8587e-01,5.0686e+04
  It 3: Residual [rel,abs] = 1.1034e+00,7.1168e+04
  It 4: Residual [rel,abs] = 1.6327e+00,1.0530e+05
  It 5: Residual [rel,abs] = 2.4890e+00,1.6053e+05
AMG (energy/solver/solve) It 5, Diverged.
```

If the residual values decrease at the beginning of the linear iterations but then stall, the warning printed in the console may not be indicating an issue with the solver settings. However, if the residual values increase as the solution progresses, this may indicate that the nonlinear residual convergence is poor and the solution may not be reliable. Performing either of the following steps may resolve this issue :

- If you are using the coupled solver and encounter the `Diverged.` or `Stalled.` messages in the console, you can activate the **BCGSTAB** option for the **Stabilization Method** within the **Advanced Solution Controls** dialog box for the coupled flow system, as described in Setting Algebraic Multigrid Parameters (p. 3610).

3. Decrease the **Under-Relaxation Factor** values within the **Solution Controls** task page. If using the **SIMPLE** scheme for a steady-state flow, the following under-relaxation factor adjustments may be beneficial:

   - 0.2 for **Pressure**
   - 0.4 for **Momentum**
   - 0.4 for all turbulence parameters

   Note that the above uner-relaxation factor adjustments may lead to slower convergence in some cases. For more information on adjusting solution controls settings for turbulent flows see Solution Strategies for Turbulent Flow Simulations (p. 2147).

4. Change the **Spatial Discretization Schemes** from second order to first order (see Choosing the Spatial Discretization Scheme (p. 3575)). Switching to first order schemes will affect the solution accuracy. However, the decrease in accuracy is negligible when modeling turbulence.

5. Check the mesh quality and improve the mesh quality if the quality is poor or if there are any left-handed faces. The mesh can be improved either by regenerating the mesh with higher quality or by using the Fluent mesh improvement capability with the following text command:

   ```
   /mesh/repair-improve/improve-quality
   ```

37.13. Resolving GPU Solver Performance Issues

When experiencing GPU performance issues while using the Fluent GPU Solver with a single GPU, the following procedure is recommended to resolve any issues:

- Check to see if the GPU is in a free mode by executing `nvidia-smi` in the command prompt window as outlined in Starting the Fluent GPU Solver from the Command Line (p. 3758).
- Increase the reporting interval for residuals and monitors with the following tui command.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
/solve set report-interval 20
```

A reporting interval of 20 is recommended.

- Check if the convergence is as expected, specifically when the residuals plot appears irregular. You can turn on the AMG verbosity within the **Advanced Solution Controls** dialog box (see Setting the Verbosity in the *Fluent User's Guide* (p. 3617)) to see if the linear solver is converging normally and if the solver is calculating through all 50 maximum cycles.

When experiencing GPU performance issues while using the Fluent GPU Solver with multiple GPUs, first perform the procedure above for resolving single GPU performance issues, then perform the following additional steps:

- Calculating the solution for too small of a case will cause performance issues when solving on multiple GPUs. For multiple GPU calculations it is recommended to have no less than two million cells per GPU.

- Check if the load balance is satisfactory by first reading in your case and then printing the load balance using the following TUI command:

  ```
  /parallel/partition/print-active
  ```

  If the load balance is not good, it is recommended to use the stored cell partition scheme by entering the following TUI commands:

  ```
  /parallel/partition/method metis
  ```

  ```
  /parallel/partition/use-stored-partitions
  ```

  For more information on checking the load balance and interpreting the partition statistics see Interpreting Partition Statistics (p. 4339).

- Check if the communication between GPUs is satisfactory using the following TUI commands:

  ```
  /parallel/bandwidth
  ```

  ```
  /parallel/latency
  ```

  For more information on interpreting the bandwith and latency data see Checking Latency and Bandwidth (p. 4349).

37.14. GPU Memory Usage

GPU memory consumption when using the Fluent GPU Solver is dependant on the size of the case, mesh type, enabled models, precision, and certain solver settings. The following information provides an understanding of the GPU memory usage required for your specific case:

- When modeling turbulence with a hexahedral mesh, single precision, and one million cells, about 1 GB of GPU memory is required.

- Enabling double precision will require 50 percent more GPU memory.

- A polyhedra mesh will require 20 to 40 percent more GPU memory.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Using maximal independent set for the AMG solver aggregation type will reduce GPU memory consuption by 20 to 25 percent. Maximal independent set can be enabled for the AMG solver by entering the following rpvar command in the console:

  ```
  (rpsetvar 'gpuapp/aggregation-method "mis")
  ```

  Note:

  Maximal independent set will be slightly less stable than the default AMG solver aggregation type but will increase computational speed and decrease GPU memory consumption.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 38: Adapting the Mesh

The solution-adaptive mesh refinement feature of Ansys Fluent allows you to refine and/or coarsen your mesh based on geometric and numerical solution data. In addition, Ansys Fluent provides tools for creating and viewing adaption fields customized to particular applications. For information about the theory behind mesh adaption in Ansys Fluent, see Adapting the Mesh in the Theory Guide. Information about using the adaption process in Ansys Fluent is described in detail in the following sections.

38.1. Using Adaption

38.2. Refining and Coarsening

38.3. Adaption Examples

38.4. Geometry-Based Adaption with the Hanging Node Method

38.1. Using Adaption

Two significant advantages of the unstructured mesh capability in Ansys Fluent are:

- reduced setup time compared to structured meshes
- the ability to incorporate solution-adaptive refinement of the mesh

By using solution-adaptive refinement, you can add cells where they are needed in the mesh, thus enabling the features of the flow field to be better resolved. When adaption is used properly, the resulting mesh is optimal for the flow solution because the solution is used to determine where more cells need to be added. Thus, computational resources are not wasted by the inclusion of unnecessary cells, as occurs in the structured mesh approach. Also, the effect of mesh refinement on the solution can be studied without completely regenerating the mesh.

Note:

When you are running in parallel, Fluent performs load balancing, which redistributes cells across compute nodes. Refer to Partitioning the Mesh Automatically (p. 4316) and Load Balancing (p. 4329) for additional information on load balancing. After the mesh is adapted, Fluent automatically performs a load balancing step, by default.

For more information about the adaption process, see Adaption Process in the Theory Guide.

See the following sections for information on performing mesh adaption:

38.1.1. Adaption Example

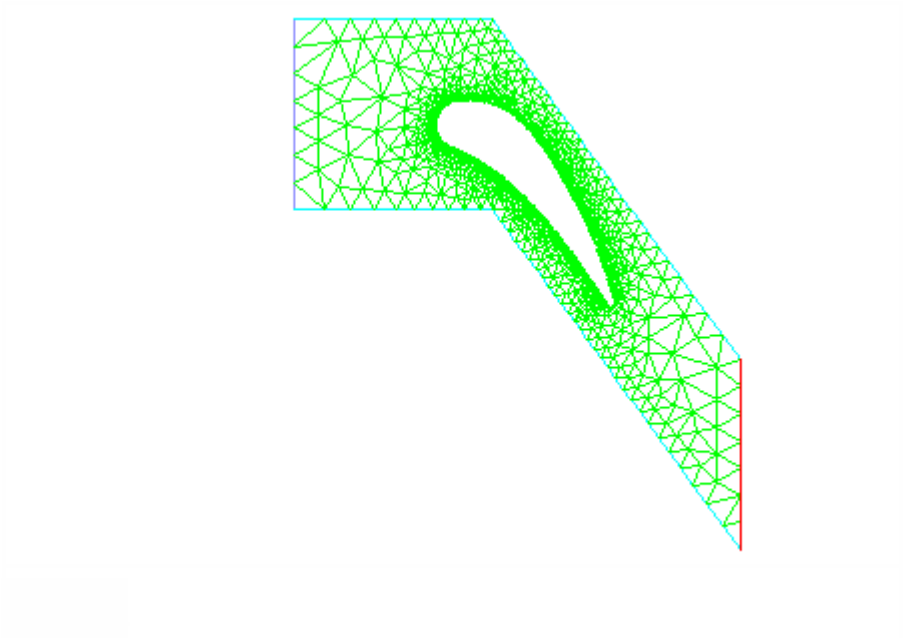
38.1.2. Adaption Guidelines

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

38.1.1. Adaption Example

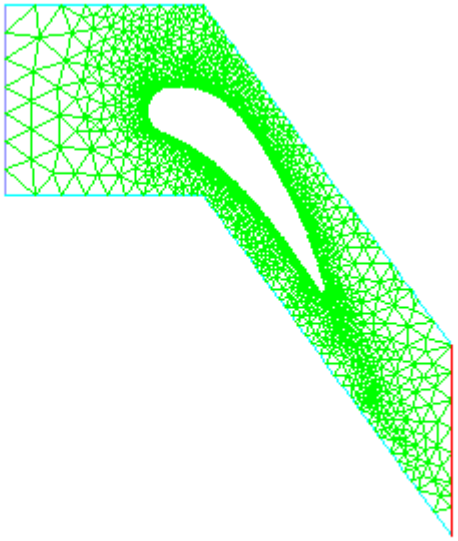
An example of the effective use of adaption is in the solution of the compressible, turbulent flow through a 2D turbine cascade. The initial mesh around the blade is fine, as shown in Figure 38.1: Turbine Cascade Mesh Before Adaption (p. 3774). The surface node distribution thus provides adequate definition of the blade geometry, and enables the turbulent boundary layer to be properly resolved without further adaption. On the other hand, the mesh on the inlet, outlet, and periodic boundaries is comparatively coarse. To ensure that the flow in the blade passage is appropriately resolved, solution-adaptive refinement was used to create the mesh shown in Figure 38.2: Turbine Cascade Mesh after Adaption (p. 3775).

Figure 38.1: Turbine Cascade Mesh Before Adaption



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 38.2: Turbine Cascade Mesh after Adaption



While the procedure for solution adaption will vary according to the flow being solved, it is instructive to examine the adaption process used for the turbine cascade shown in the previous figure. Though this example involves compressible flow, the general procedure is applicable for incompressible flows as well.

1. Display contours of pressure to determine a suitable refinement threshold.
2. Mark the cells within the refinement threshold, creating a refinement register (see Field Variable (p. 3855)).
3. Repeat the process described in steps 1 and 2, using gradients of Mach number as a refinement criterion.
4. To refine in the wake region, use isovalues of total pressure as a criterion (see Field Variable (p. 3855)). This causes cells within the boundary layer and the wake to be marked, since these are both regions of high total-pressure loss.
5. Refine the cells contained in the register (see Refining and Coarsening (p. 3777)).

The effect of refining on gradients is evident in the finer mesh ahead of the leading edge of the blade and within the blade passage (Figure 38.2: Turbine Cascade Mesh after Adaption (p. 3775)). The finer mesh in the wake region is due to the adaption using isovalues of total pressure.

38.1.2. Adaption Guidelines

The advantages of solution-adaptive refinement, when used properly (as in the turbine cascade example in Adaption Example (p. 3774)), are significant. However, this capability must be used carefully to avoid certain pitfalls. Some guidelines for proper usage of solution-adaptive refinement are as follows:

- The surface mesh must be fine enough to adequately represent the important features of the geometry.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For example, it would be bad practice to place too few nodes on the surface of a highly-curved airfoil, and then use solution refinement to add nodes on the surface. The surface will always contain the facets contained in the initial mesh, regardless of the additional nodes introduced by refinement.

- The initial mesh should contain sufficient cells to capture the essential features of the flow field.

  Consider the following example, in which you want to predict the shock forming around a bluff body in supersonic flow. To obtain a reasonable first solution, the initial mesh should contain enough cells and also have sufficient resolution to represent the shape of the body. Subsequent gradient adaption can be used to sharpen the shock and to establish a mesh-independent solution.

- Polyhedral cells are only eligible for adaption with the default polyhedral unstructured mesh adaption (PUMA) method.

- For the hanging node adaption method, the presence of polyhedral cells in a mesh may or may not limit the eligibility of other cells for adaption, depending on the manner in which the polyhedral cells were created:

  – If the domain was converted to polyhedra (see Converting the Domain to a Polyhedra (p. 1253)), then no part of the mesh can be adapted (even if hexahedral cells are present in the mesh after conversion).

  – If the polyhedra are a result of converting skewed tetrahedral cells (see Converting Skewed Cells to Polyhedra (p. 1258)) or converting cells adjacent to hanging nodes / edges (see Converting Cells with Hanging Nodes / Edges to Polyhedra (p. 1259)), then the non-polyhedral cells may be adapted. The polyhedral cells, however, will be automatically unmarked from the register when adaption is initiated and will remain unchanged.

- Obtain a reasonably well-converged solution before performing an adaption. If you adapt to an incorrect solution, cells will be added in the wrong region of the flow.

  Use careful judgment in deciding how well to converge the solution before adapting, because there is a trade-off between adapting too early to an unconverged solution and wasting time by continuing to iterate when the solution is not changing significantly. This does not directly apply to automatic adaption, because here the solution is adapted at regular intervals of iterations or time steps, depending on which solver is being used.

- Write a case and data file before starting the adaption process. If you generate an undesirable mesh, you can restart the process with the saved files. This does not directly apply to automatic adaption, because here the solution is adapted at regular intervals of iterations or time steps, depending on which solver is being used.

- Select suitable variables when performing gradient adaption. For some flows, the choice is clear. For instance, adapting on gradients of pressure is a good criterion for refining in the region of shock waves. In most incompressible flows, however, it makes little sense to refine on pressure gradients. A more suitable parameter in an incompressible flow might be mean velocity gradients. If the flow feature of interest is a turbulent shear flow, it will be important to resolve the gradients of turbulent kinetic energy and turbulent energy dissipation, so these might be appropriate refinement variables. In reacting flows, temperature or concentration (or mole or mass fraction) of reacting species might be appropriate.

- Do not over-refine a particular region of the solution domain. It causes very large gradients in cell volume. Such poor adaption practice can adversely affect the accuracy of the solution.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

38.2. Refining and Coarsening

You can refine or coarsen your mesh based on cell registers (Using Cell Registers (p. 3851)), named expressions (Named Expressions (p. 1054)), or expressions using the **Manual Mesh Adaption** and/or **Automatic Mesh Adaption** dialog box, as described in the following steps. For information about adapting overset meshes, see Overset Mesh Adaption (p. 1219).

To set up mesh adaption:

1. For 3D meshes, you have the choice of refining the cells using either the default polyhedral unstructured mesh adaption (PUMA) method, the 2.5D PUMA method (which is suitable for meshes that are one cell thick), or the hanging node method. For details and limitations on these methods, see Adaption Process in the Theory Guide. To change the adaption method, use the following text command:

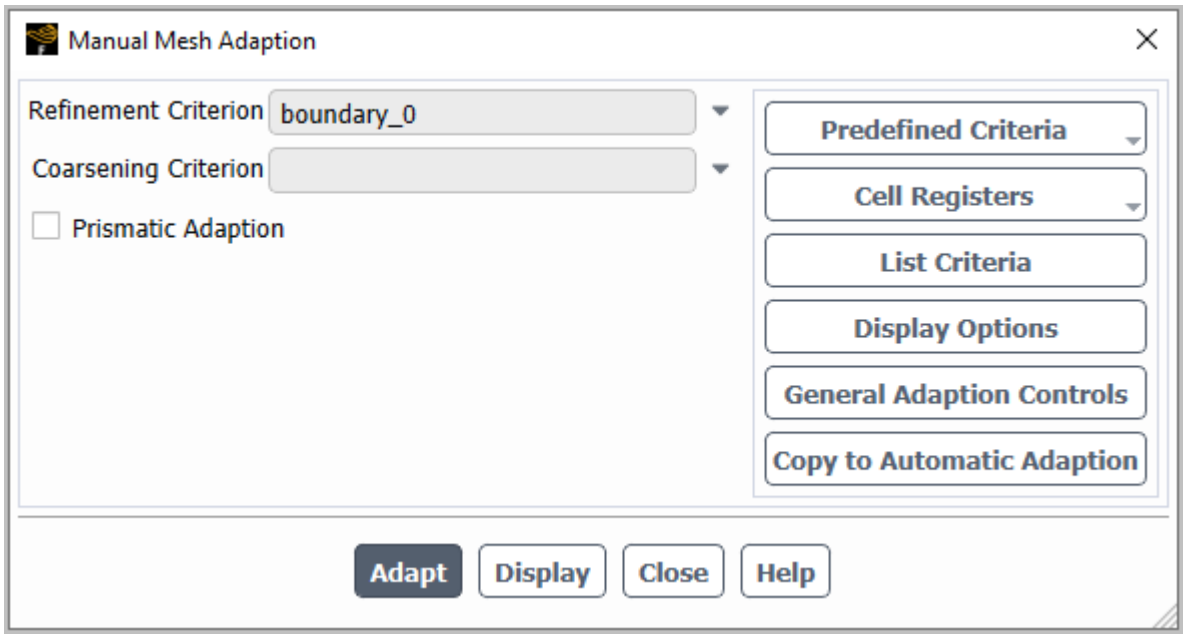
   `mesh → adapt → set → method`

2. Begin the adaption setup by opening the appropriate dialog box:

   - To perform manual adaption, open the **Manual Mesh Adaption** dialog box.

     Domain → Adapt → Manual...

     Figure 38.3: The Manual Mesh Adaption Dialog Box

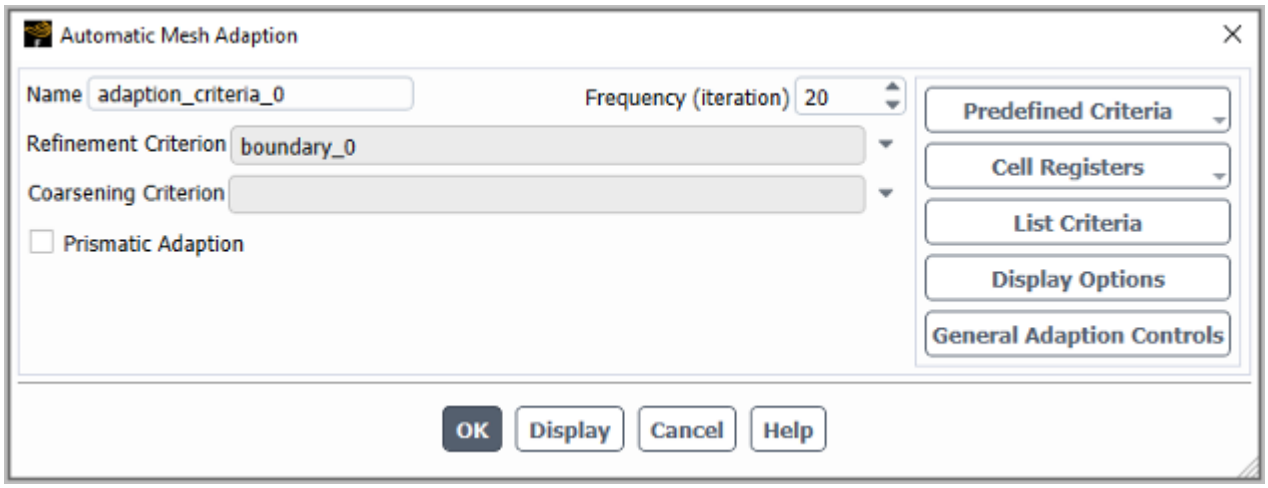
     

   - To have the mesh automatically adapted throughout the calculation, open the **Automatic Mesh Adaption** dialog box.

     Domain → Adapt → Automatic...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 38.4: The Automatic Mesh Adaption Dialog Box



3. For automatic mesh adaption, enter a **Name** for the adaption criteria, and specify how frequently Fluent adapts the mesh by setting the **Frequency** to the number of iterations or time steps that must pass between two consecutive automatic mesh adaptions.

4. (Optional) You can make a selection from the **Predefined Criteria** drop-down list to easily set up the adaption with commonly used criteria; this can be done instead of or in combination with the steps that follow. The available criteria are useful for:

   - meshes that require boundary layer refinement based on a cell's proximity to one or more boundaries
   - aerodynamic flow simulations that have a shock or significant variations for an error indicator
   - flows that use a combustion model
   - flows that use the Volume of Fluid (VOF) multiphase model
   - overset mesh simulations that are adversely affected by orphans and/or donor-receptor cell size differences

   For further details, see Predefined Criteria for Adaption (p. 3784).

5. (Optional) Create cell registers for the types of cells that you want to adapt by using the **Cell Registers** drop-down list. For further details, see Using Cell Registers (p. 3851).

6. Specify the **Refinement Criterion** you want Fluent to use to determine when the mesh should be refined. You can select an existing cell register or named expression, or provide an expression.

7. Specify the **Coarsening Criterion** you want Fluent to use to determine when the mesh should be coarsened. You can select an existing cell register or named expression, or provide an expression.

8. (Optional) Enable the **Prismatic Adaption** option if you want the cells to be split anisotropically, that is, only along certain directions. This is typically used with meshes that contain boundary layers (which can be marked using the boundary layer predefined criterion, a boundary register, or a Yplus/Ystar register). The following limitations are associated with this prismatic adaption:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- It is for 3D meshes only.
- You must use the PUMA adaption method.
- Only prismatic cells (that is, hexahedral cells or wedge or polyhedral cells that have the same number of nodes on the top and bottom faces relative to the splitting direction) in layers that are adjacent to a boundary will be refined anisotropically; all other cells will be refined isotropically.

Note:

This option corresponds to the `mesh/adapt/set/prismatic-adaption?` text command (for manual adaption) and the `prismatic?` setting that is available when using the `mesh/adapt/manage-criteria/add` text command (for automatic adaption).

When using the **Prismatic Adaption** option, there are additional global settings that are applied to all adaption criteria, which you can modify if necessary:

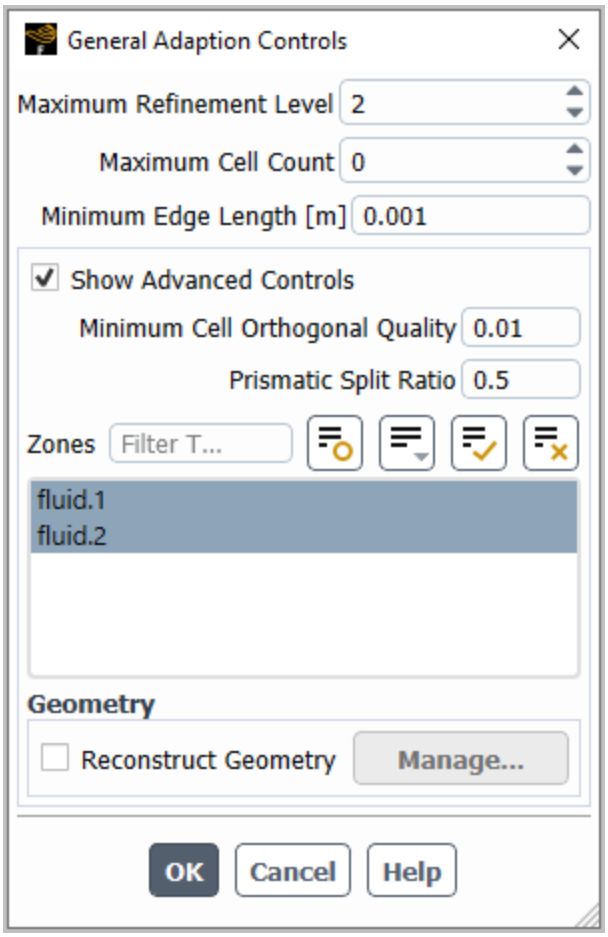
- By default, the cells that are split anisotropically are divided into two equal halves (that is, with a split ratio of .5). You can change this split ratio using the **General Adaption Controls** dialog box, as described in a step that follows.
- The splitting direction is defined by default as being perpendicular to the boundary zone adjacent to the layers of prismatic cells that are eligible for adaption. Depending on the mesh, some cells may have multiple splitting directions defined by default. You can reduce the number of splitting directions by using the following text command to specify which boundary zone or zones you want used for anisotropic refinement:

  `mesh` → `adapt` → `set` → `prismatic-boundary-zones`

9. Click the **General Adaption Controls** button and define the general adaption settings in the **General Adaption Controls** dialog box that opens. Note that these settings will be applied to all adaption criteria.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 38.5: The General Adaption Controls Dialog Box



a. Specify the **Maximum Refinement Level**, which controls the number of levels of refinement used to split cells during the adaption. A value of zero places no limits on the number of levels. *The default of 2 is a good start for most problems. This field essentially keeps Fluent from infinitely refining the mesh. Specifying too large of a value can quickly lead to a very large cell count.*

b. Specify the **Maximum Cell Count**, which sets an approximate limit to the total cell count of the mesh. Fluent uses this value to determine when to stop marking cells for refinement. A value of zero places no limits on the number of cells. This field is only available with the PUMA adaption method.

c. Set the **Minimum Edge Length**. This field restricts the size of the cells that Fluent considers for refinement. Even if a cell is marked for refinement, it will not be refined if (for 3D) its volume is less than the cube of this field or (for 2D) its area is less than the square of this field. The default value of zero places no limits on the size of cells that are refined.

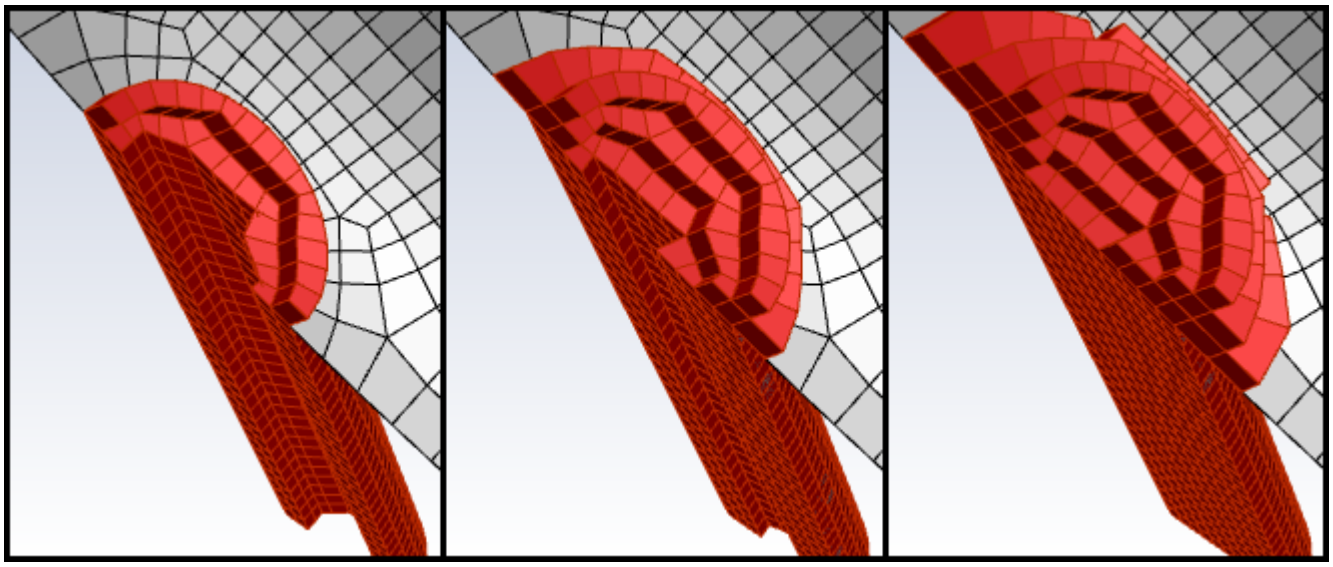
d. (Optional) Enable **Show Advanced Controls** to reveal the following advanced controls:

   - For transient cases, you can specify the number of **Additional Refinement Layers** that can be refined as part of automatic adaption. Note that the newly marked cells will respect the limits for **Maximum Refinement Level** and **Minimum Edge Length** defined in the **General Adaption Controls** dialog box. When set in the **General Adaption Controls** dialog box, the **Additional Refinement Layers** value is a global setting; to customize it for an individual automatic adaption criterion, you can use the `mesh/adapt/manage-criteria/edit` text

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

command. The effect of this setting is illustrated in Figure 38.6: Additional Refinement Layers: 1, 2, 3 (p. 3781).

Figure 38.6: Additional Refinement Layers: 1, 2, 3



- When you have enabled the **Prismatic Adaption** option in the **Manual Mesh Adaption** or **Automatic Mesh Adaption** dialog box (as described in a previous step), you may want to revise the **Prismatic Split Ratio** that is applied to the cells, in order to control exactly where the cells are split relative to boundary zones that define the direction. The default value for this ratio is 0.5, and it is defined as follows:

$$\text{split ratio}=\frac{\text{height (from the base face) of the splitting point on the edge}}{\text{original height of the edge}}$$

- You can specify the **Minimum Cell Orthogonal Quality** that you would like Fluent to attempt to maintain in the cells that result from adaption. Note that this value is not guaranteed to be maintained, as to do so would negatively affect the solution speed. The worse quality cells will have an orthogonal quality closer to 0 and better quality cells will have an orthogonal quality closer to 1. For more information about cell quality, see Mesh Quality (p. 1146).

  Note:

  The **Minimum Cell Orthogonal Quality** setting is not available for 2D cases or if the hanging node adaption method is selected.

- For cases that have multiple cell zones, you can limit the adaption to a particular cell zone or zones by selecting them in the **Zones** list. An example of when you might want to do this is when you want to exclude certain cell zones from being adapted, because they already possess sufficient mesh resolution for an accurate result. By default, adaption will be performed in all cell zones.

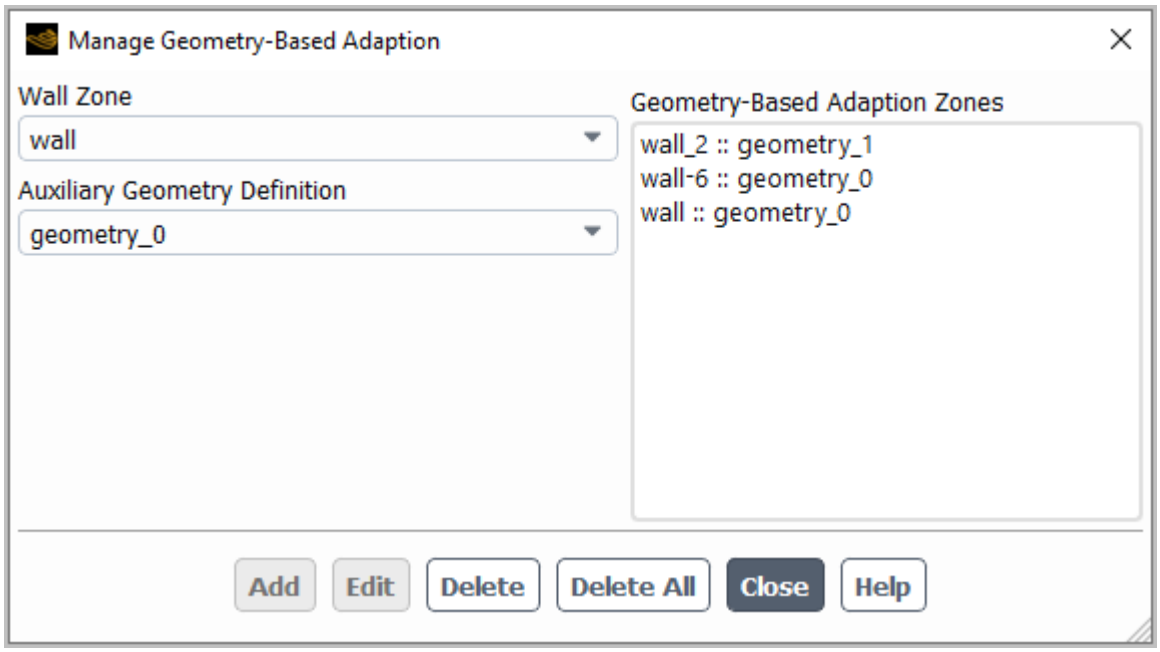
- If you have a coarse mesh on a wall zone that has curved profiles and/or sharp corners, you may want to use geometry-based adaption, so that the new nodes created during adaption

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

are projected onto a specified auxiliary geometry definition. In this way, you can recover finer details of the wall zone shape during refinement. For further details about the concepts involved, see Geometry-Based Adaption in the Theory Guide.

To perform geometry-based adaption using the PUMA adaption method, you must first define one or more auxiliary geometry definitions, as described in Managing Auxiliary Geometry Definitions (p. 1791). Then you can enable the **Reconstruct Geometry** option under the **Show Advanced Controls** option, and click the **Manage...** button to open the **Manage Geometry-Based Adaption** dialog box.

Figure 38.7: The Manage Geometry-Based Adaption Dialog Box



In the **Manage Geometry-Based Adaption** dialog box, select a **Wall Zone** and the **Auxiliary Geometry Definition** to which you want the zone to conform, and click the **Add** button (note that you must ensure that the existing nodes on the wall zone already conform to this auxiliary geometry definition). The pair will be listed in the **Geometry-Based Adaption Zones** list. At any point you can select that **Wall Zone** again and select a different **Auxiliary Geometry Definition** and click **Edit** to edit the definition, or you can select it in the **Geometry-Based Adaption Zones** list and click **Delete** to delete it. You can repeat this step with different zones to add additional definitions; note that a zone cannot be paired with more than one geometry definition, but a geometry definition can be paired with multiple zones. Click **Close** when your geometry-based adaption is fully set up.

By default, the nodes that move to conform to the geometry definition will only be those on the wall zone surface. If you would like the boundary displacement to also be applied to some degree to the prismatic boundary layers adjacent to the surface, you can use the following text command to edit the relevant geometry-based adaption zone(s) and enter a positive, nonzero value for the `exponent`. Lower values for the `exponent` cause the displacement to propagate further from the surface, and a value of 0 disables such propagation altogether. It is recommended that you start with a value of 0.01 and adjust as needed.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

`mesh` → `adapt` → `geometry` → `manage` → `edit`

Note:

If you want to perform geometry-based adaption when using the hanging node adaption method, you must use the Geometry Based Adaption Dialog Box (p. 5491), as described in Geometry-Based Adaption with the Hanging Node Method (p. 3810).

e. Click **OK** to close the **General Adaption Controls** dialog box, and continue setting up the **Manual Mesh Adaption** or **Automatic Mesh Adaption** dialog box.

10. (Optional) Click **List Criteria** to print the number of cells that Fluent has identified for being refined, coarsened, or both, to the console.

11. (Optional) Click **Display Options** to open the Display Options - Adaption Dialog Box (p. 5435), which allows you to customize how the cells marked for adaption are displayed.

12. (Optional) Click **Display** to review the cells that are marked for adaption.

13. For manual adaption, click **Adapt** to adapt the mesh. Note that if at any point you want to switch to using automatic adaption, you can click the **Copy to Automatic Adaption** to copy the settings to the **Automatic Mesh Adaption** dialog box.

14. For automatic adaption:

    a. Click **OK** to save the adaption criterion.

    b. Create additional adaption criteria as needed, as described in the previous steps.

    Domain → **Adapt** → **Automatic...**

    c. (Optional) You can specify that one or more of the initial instances of adaption are skipped for a given automatic adaption criterion. This can be done using the following text command:

    `mesh` → `adapt` → `manage-criteria` → `edit`

    After entering the name of the criterion to edit, enter `skip-until`, followed by a positive, nonzero integer that defines the iteration / time step when automatic adaption is allowed to begin. By default this field is set to `1`, so that adaption is applied during the first iteration / time step and is never skipped.

    The following is an example to illustrate how it works: you could set `skip-until` to 40 iterations, and if the frequency is set to 20, adaption will be skipped on the first and twentieth iterations, and will not begin until the fortieth iteration. This is true even if you were to stop the calculation at the thirtieth iteration and then restart (that is, it will skip adaption until the tenth iteration after the restart, because it is based on the iteration count as printed in the console).

Note:

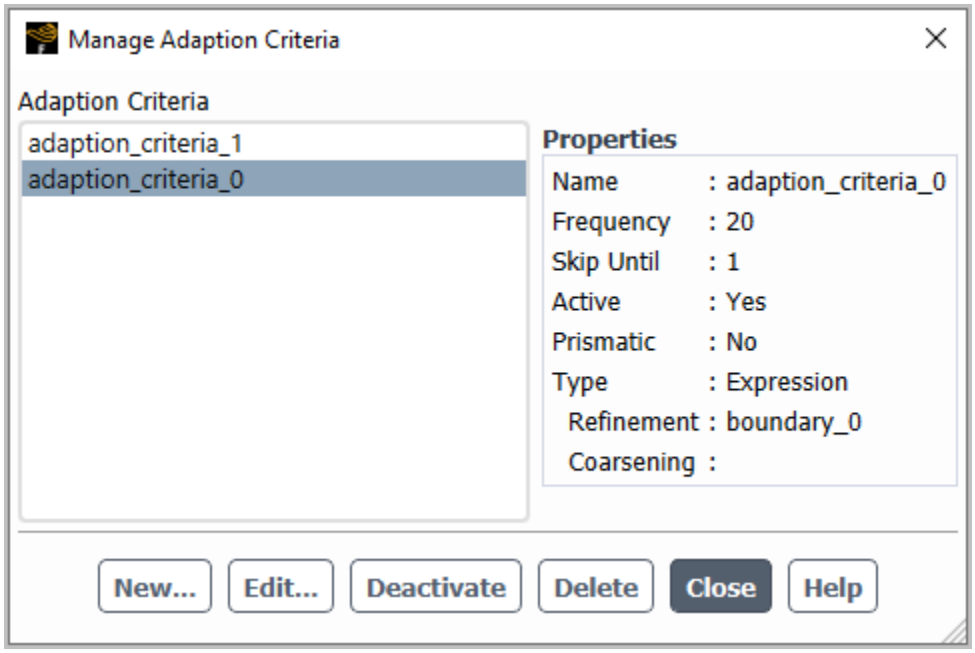
- The `skip-until` setting can also be defined when adding a new automatic adaption criterion through the `mesh/adapt/manage-criteria/add` text command.

- The `skip-until` setting is automatically set to 2 for the **Pressure Hessian Indicator**, **Mach Hessian Indicator**, and **Combined Hessian Indicator** predefined criteria, as typically the calculation of these indicators do not yield useful results in the first iteration / time step. For further details on these criteria, see Aerodynamics Adaption (p. 3787).

d. Review your adaption criteria using the **Manage Adaption Criteria** dialog box.

Domain → Adapt → Manage...

Figure 38.8: The Manage Adaption Criteria Dialog Box



If you make a selection from the **Adaption Criteria** list, information about that criterion is displayed under **Properties**. You can then: click the **Edit...** button to revise the settings; click **Deactivate** if you do not want to use the criterion for the time being (the button will change to **Activate** so that it can be used later); and click **Delete** to delete the criterion entirely.

e. Run your calculation, and your adaption will be automatically applied at regular intervals.

38.2.1. Predefined Criteria for Adaption

A **Predefined Criteria** drop-down list is available in the **Manual Mesh Adaption** and **Automatic Mesh Adaption** dialog boxes to increase the ease of setup. It provides a list of commonly used criteria for adapting the mesh; if one is selected, Fluent will generate the necessary cell registers and/or named expressions for refinement and coarsening, and will define the associated settings. Note that your predefined criteria selection may require you to specify certain mandatory settings (such as the **Minimum Cell Length Scale**), based on the details of your case.

For details, see the following:

38.2.1.1. Boundary Layer Adaption Based on Cell Distance

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

38.2.1.2. Aerodynamics Adaption
38.2.1.3. Combustion Adaption
38.2.1.4. VOF Adaption
38.2.1.5. Overset Adaption

38.2.1.1. Boundary Layer Adaption Based on Cell Distance

You can use predefined criteria to easily adapt the mesh manually for boundary layers, based on a cell's proximity to one or more boundaries. Fluent will automatically generate the necessary cell register for refinement and, where possible, will enable prismatic adaption so that the cells are only split along certain directions.

To use predefined criteria for adapting a boundary layer, perform the following steps:

1. Read a mesh. If you want prismatic adaption, it must be a 3D mesh that contains prismatic cells adjacent to the relevant boundary zones. Prismatic cells are hexahedral cells or wedge or polyhedral cells that have the same number of nodes on the top and bottom faces relative to the splitting direction.

2. If you want prismatic adaption, ensure that the PUMA method is selected by using the following text command:

   `mesh` → `adapt` → `set` → `method`

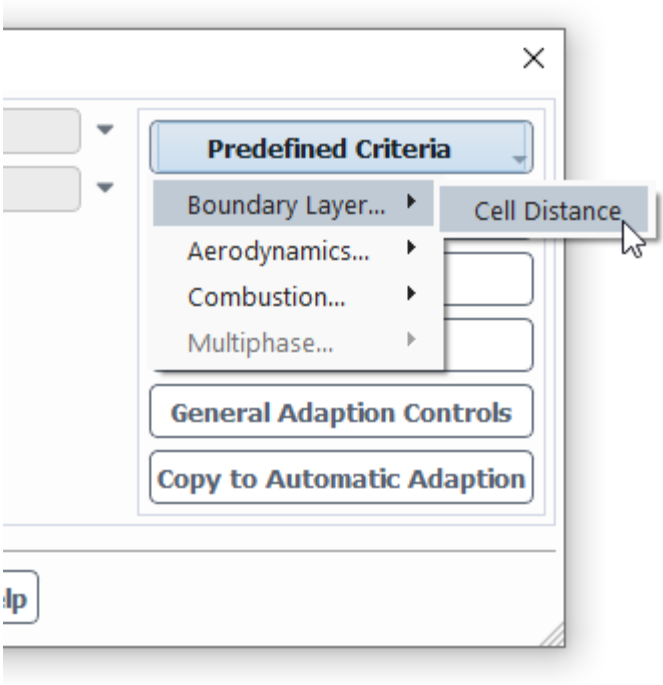
3. Open the **Manual Mesh Adaption** dialog box.

   Domain → **Adapt** → **Manual...**

   a. Select **Boundary Layer...** from the **Predefined Criteria** drop-down list, and then select **Cell Distance** from the menu that opens.

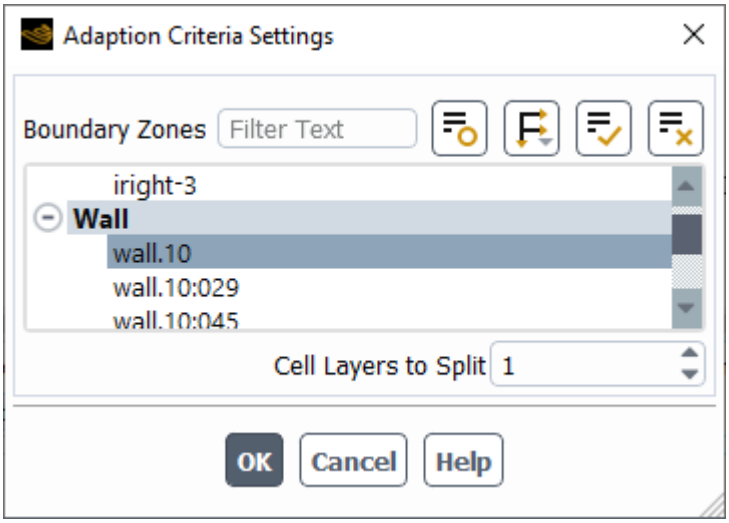
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 38.9: The Predefined Criteria Drop-Down List



b. Complete the setup in the **Adaption Criteria Settings** dialog box that opens. You must select the **Boundary Zones** where you want adaption, define the number of **Cell Layers to Split**, and click **OK**.

Figure 38.10: The Adaption Criteria Settings Dialog Box for the Cell Distance Criterion



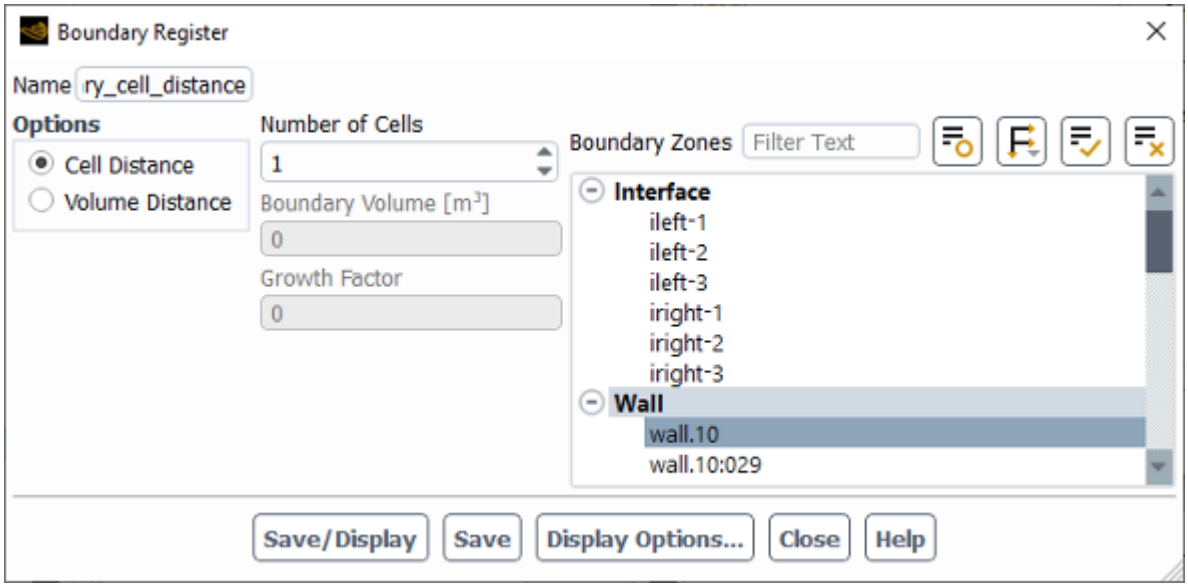
c. When using the PUMA adaption method, by default the **Prismatic Adaption** option will be enabled, so that the cells are refined anisotropically where possible; such cells will only be split along the directions that are perpendicular to the zones selected in the **Adaption Criteria Settings** dialog box. Note that you can control exactly where the cells are split by revising the value for the **Prismatic Split Ratio** field in the **General Adaption Controls** dialog box, as described in Refining and Coarsening (p. 3777).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

d. A cell register named **boundary_cell_distance** is created as part of the predefined criteria and selected in the **Refinement Criterion** field. Review it in the **Boundary Register** dialog box to verify that it is appropriate:

Solution → **Cell Registers** → **boundary_cell_distance** **Edit...**

Figure 38.11: The Boundary Register Dialog Box for the Refinement Criterion



Note that the **Boundary Register** dialog box allows you to switch to the **Volume Distance** method when the **Cell Distance** method is not preferred for your case.

e. Click **Adapt** in the **Manual Mesh Adaption** dialog box to perform the adaption.

38.2.1.2. Aerodynamics Adaption

When simulating aerodynamic flows, you can use predefined criteria to set up the mesh adaption. With this predefined criteria, Fluent will automatically generate the necessary cell registers for refinement and coarsening, based on the gradient of the density (in order to account for shocks), an indicator for error in the pressure calculation (in order to account for significant pressure variations), an error indicator that considers variations in the Mach number, an indicator for error in an observable calculation due to mesh refinement, or an error indicator that considers variations in several flow fields.

To use predefined criteria for adapting a mesh for aerodynamic simulations, perform the following steps:

1. Set up your aerodynamic simulation, and initialize and/or run the calculation to generate data.
2. When setting up the mesh adaption for an adjoint observable:
   a. Define the observable to be used for mesh adaption. For details on defining observables see Defining Observables (p. 4515).
   b. Initialize and compute the adjoint solution.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. After calculating the adjoint solution, the **Goal based error indicator (Raw)** field variable can be displayed (under the **Sensitivities...** category) to identify regions where mesh adaption will have a larger or smaller effect on the observable. High value regions will have a larger effect on the observable and will benefit from cell refinement, whereas low value regions will have a smaller effect on the observable and may benefit from cell coarsening.

3. Open the **Manual Mesh Adaption** or **Automatic Mesh Adaption** dialog box.

Domain → **Adapt** → **Manual...**

or

Domain → **Adapt** → **Automatic...**

a. Select **Aerodynamics...** from the **Predefined Criteria** drop-down list, and then select one of the following from the menus that open:

- **Shock Indicator** and **Density-based**

  This is useful if you want to adapt the mesh to properly account for shocks, based on the gradient of the density. This criterion is always available if you have selected the **Density-Based** solver in the **General** task page, but is only available with the **Pressure-Based** solver if you have also selected a real-gas or ideal-gas model from the **Density** drop-down list in the Create/Edit Materials Dialog Box (p. 4885) for the fluid.

- **Error-based** and **Pressure Hessian Indicator**

  This is useful if you want to adapt the mesh to properly account for significant pressure variations. It is based on the solution error, as approximated using an indicator that includes the Hessian (the matrix of second derivatives) of the pressure distribution. It targets key areas for adaption in cases with a variety of different scales and flow phenomena: for example, you can capture a shock and wake with different strengths simultaneously, or anisotropic features like errors in stretched prism layer cells. For further details on how the indicator is calculated, see the description of the Hessian approach in Approaches For Deriving Field Values (p. 3855).

  Note:

  This criterion is not recommended for cases with a very smooth solution, as it may target widespread areas and lead to excessive refinement.

- **Error-based** and **Mach Hessian Indicator**

  This is useful if you want to adapt the mesh based on an error indicator that considers variations in the Mach number. This indicator includes the Hessian (the matrix of second derivatives) of the Mach number. It targets adaption for a wider range of Mach number flows compared to the **Pressure Hessian Indicator** while being based on only a single scalar value for the input field, in contrast to the multiple flow fields used in the **Com-**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

bined Hessian Indicator. For further details on how the indicator is calculated, see the description of the Hessian approach in Approaches For Deriving Field Values (p. 3855).

Note:

The **Mach Hessian Indicator** criterion is only available if the density of the fluid material is defined using the ideal gas law or a real gas model.

- **Error-based** and **Combined Hessian Indicator**

  This is useful if you want to adapt the mesh based on an error indicator that considers variations in several flow fields (pressure, temperature, velocity, density, and turbulence, when each is applicable). This indicator includes the Hessian (the matrix of second derivatives) of these field variables. It is suitable for use across a range of Mach numbers, and can target key areas for adaption in cases with a variety of different scales and flow phenomena. For further details on how the indicator is calculated, see the description of the Hessian approach in Approaches For Deriving Field Values (p. 3855).

- **Error-based** and **Goal-Based Error Indicator**

  This indicator is used to adapt the mesh to reduce the influence of mesh refinement on an observable as part of an adjoint calculation. Selecting **Goal-Based Error Indicator** will open the **Adaption Criteria Settings** dialog box, where you can customize the adaption settings for your case. For details on defining these settings see Adaption Criteria Settings Dialog Box (p. 5386).

  After setting up the adaption criteria and before computing the flow solution, you can set up solution monitors for the observable and the cell count to observe the change in the observable during mesh adaption and the total cell count, respectively.

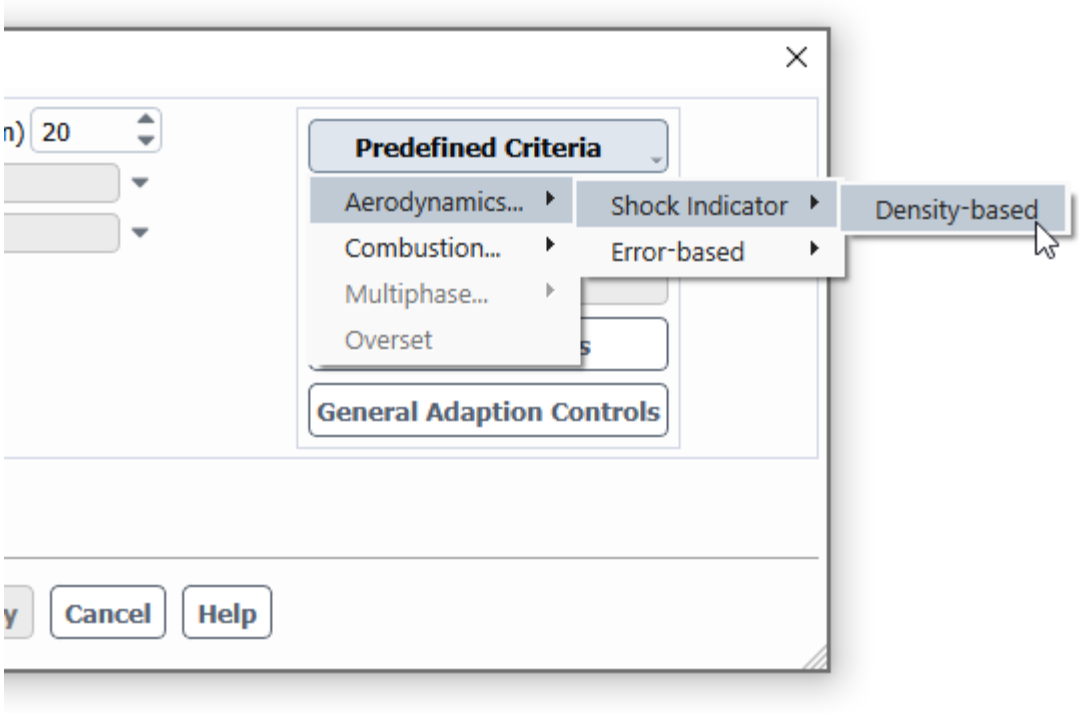
  Once the necessary adaption and refinement/coarsening criteria have been defined for your case, you can calculate the conventional flow solution to begin the mesh adaption.

  Note:

  Typically the calculation of the **Pressure Hessian Indicator**, **Mach Hessian Indicator**, or the **Combined Hessian Indicator** does not yield useful results immediately after initialization or during the first iteration / time step. For this reason, the `skip-until` setting is automatically set to 2 for any automatic adaption that uses these predefined criteria; for further details on the `skip-until` setting, see Refining and Coarsening (p. 3777).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 38.12: The Predefined Criteria Drop-Down List



b. For an automatic adaption criterion, by default the cells will be refined at a **Frequency** based on the criterion:

   - **Density-based**: 2 time steps / 20 iterations
   - **Pressure Hessian Indicator**: 20 time steps / 20 iterations
   - **Mach Hessian Indicator**: 100 time steps / 100 iterations
   - **Combined Hessian Indicator**: 100 time steps / 100 iterations
   - **Goal-Based Error Indicator**: 100 time steps / 20 iterations

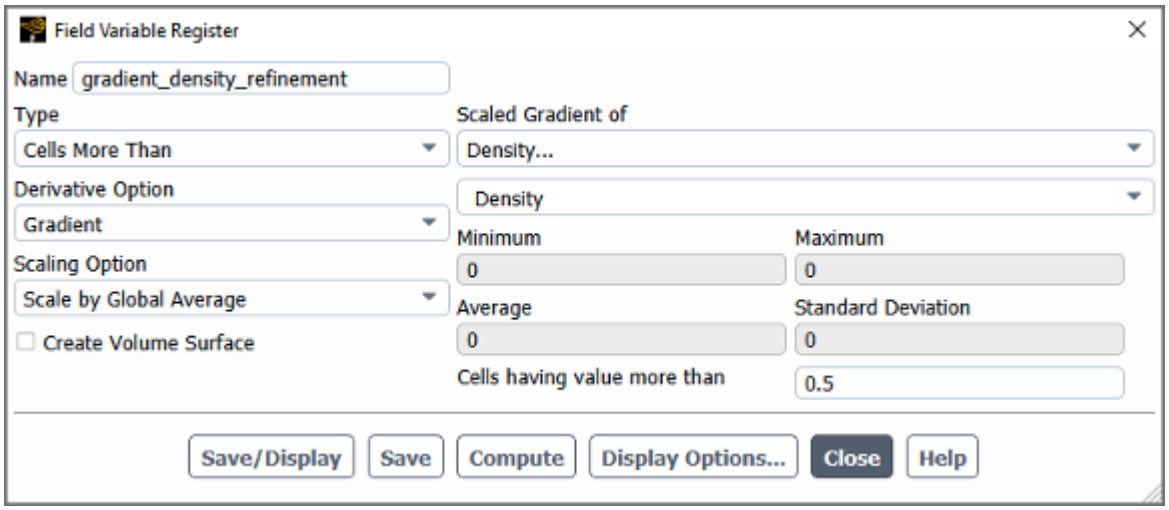
   You can adjust this setting as necessary: for example to allow more convergence between steps to avoid oscillating refinement / coarsening of a region. Note that for the **Mach Hessian Indicator** or **Combined Hessian Indicator** criterion, a longer frequency should be used compared to gradient registers, as these indicators are more sensitive to small variations and so may otherwise apply unneeded adaptations due to local fluctuations in the solution.

c. Cell registers are created as part of the predefined criteria. Note their names in the **Refinement Criterion** and **Coarsening Criterion** fields or in the console, and review them in the **Field Variable Register** dialog box to verify that they are appropriate. For example:

   Solution → Cell Registers → gradient_density_refinement → Edit...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 38.13: The Field Variable Register Dialog Box for a Refinement Criterion



Note that for the **Mach Hessian Indicator** or the **Combined Hessian Indicator**, the resulting cell registers use the **Hessian** derivative option, which is described in Field Variable (p. 3855). If you change the setup, existing indicators of these types may become invalid and may need to be adjusted or regenerated.

d. The indicator used in the **Pressure Hessian Indicator** criterion can be postprocessed by selecting the **Pressure Hessian Indicator** field variable (under the **Derivatives...** category). Note that this field variable is scaled by the global average in the cell registers created for the adaption predefined criterion, as it is a relative value.

38.2.1.3. Combustion Adaption

Predefined criteria is available for adapting the mesh of a combustion simulation, so that the mesh is refined along a progressing flame front using various criteria like temperature, vorticity, species, and DPM concentration (depending on which models are used). There is also an option for refining a spherical spark region prior to a transient run, which after a specified time is then coarsened back to the original mesh. To use this predefined criteria, perform the following steps:

1. Enable a combustion model.

   Setup → Models → Species → Edit...

2. Ensure that the energy equation is enabled.

   Setup → Models → Energy → On

3. Set up your simulation, and initialize and/or run the calculation to generate data.

4. Open the **Manual Mesh Adaption** or **Automatic Mesh Adaption** dialog box.

   Domain → Adapt → Manual...

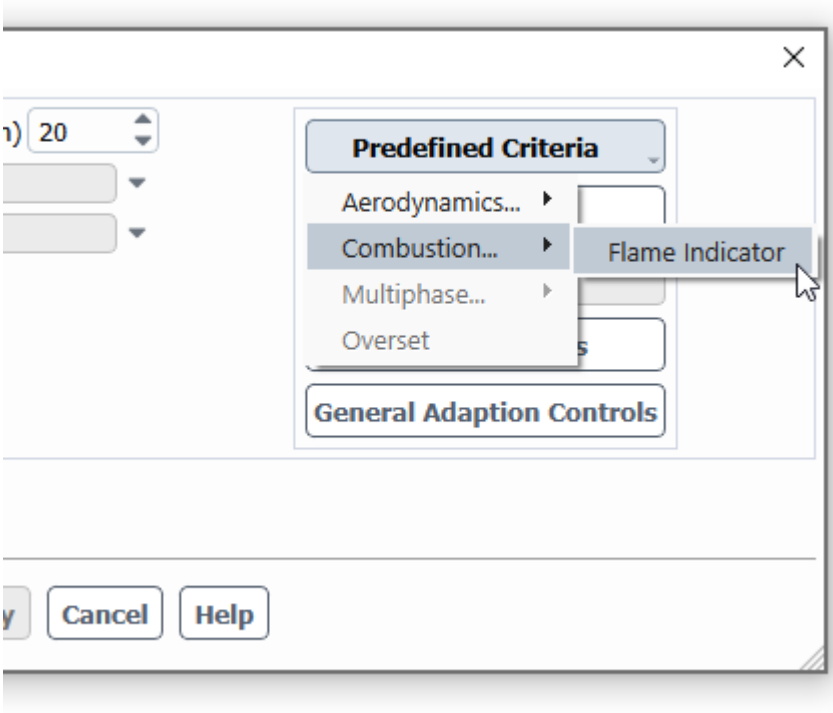
   or

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Domain → Adapt → Automatic...

a. Select **Combustion...** from the **Predefined Criteria** drop-down list, and then select **Flame Indicator** from the menu that opens.

Figure 38.14: The Predefined Criteria Drop-Down List



b. Complete the setup in the **Adaption Criteria Settings** dialog box that opens:

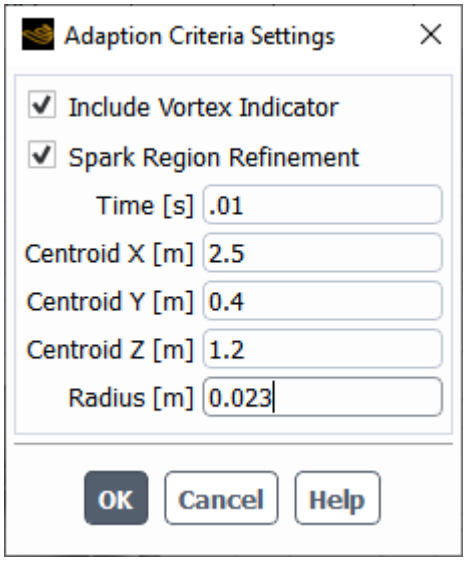
1. The **Include Vortex Indicator** option (which is enabled by default) specifies that the magnitude of vorticity is included as part of the criteria for refinement and coarsening.

2. Enable the **Spark Region Refinement** option to refine a temporary spherical spark region at the start of the simulation, which will then be coarsened back to the original mesh. The following settings define the spark region:

   - **Time** sets the maximum time during which refinement is applied to the spark region.
   - The **Centroid X**, **Y**, and **Z** fields set the coordinates for the center of the spark region.
   - **Radius** sets the radius of the spherical spark region.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 38.15: The Adaption Criteria Settings Dialog Box for the Flame Indicator Criterion



c. For an automatic adaption criterion, by default the cells will be refined at a **Frequency** of 2 iterations or time steps. You can adjust this setting as necessary.

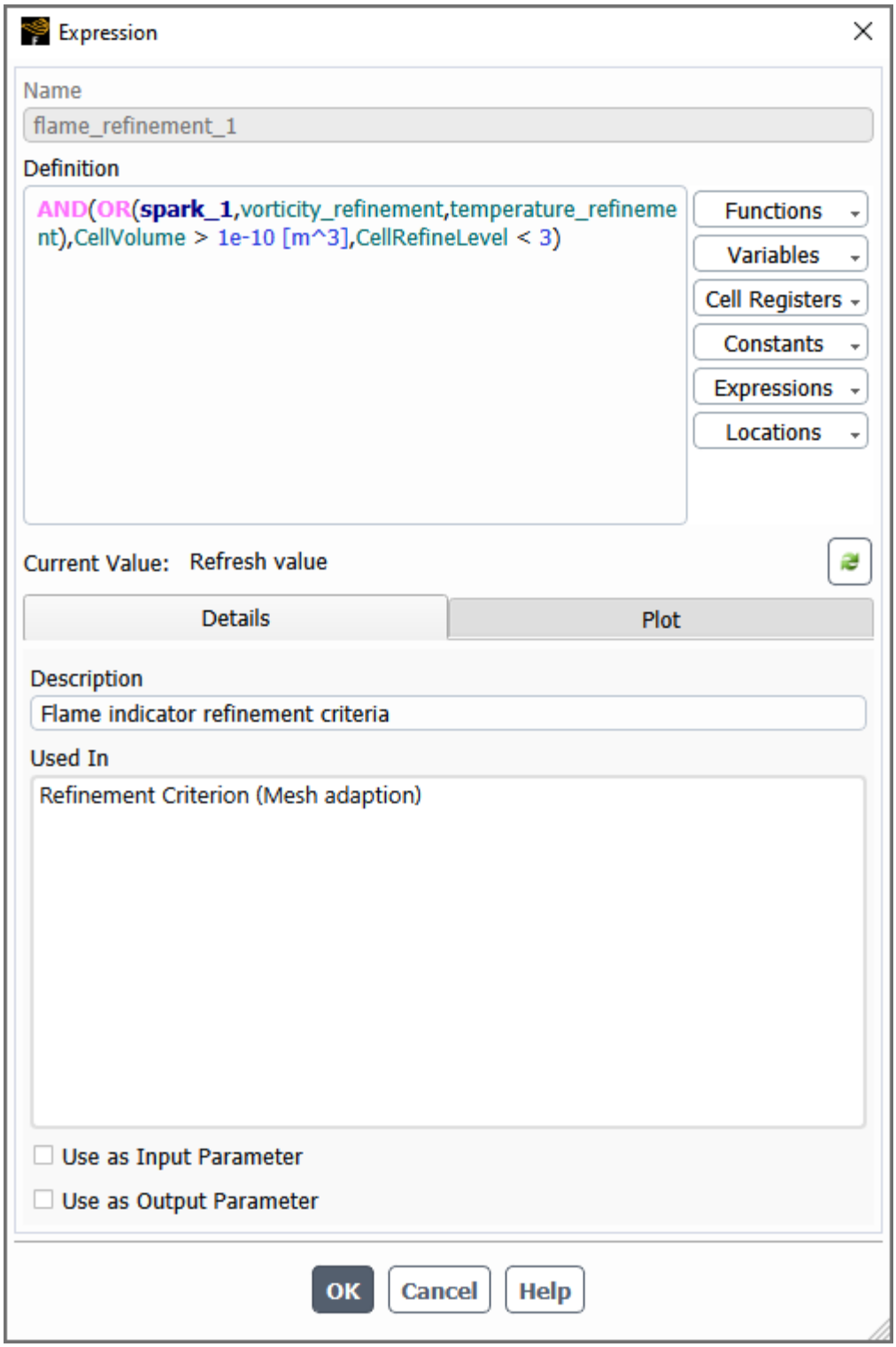
d. Other adaption settings are defined as part of the predefined criteria. Click the **General Adaption Controls** button and review them in the **General Adaption Controls** dialog box to verify that they are appropriate.

e. Named expressions are created as part of the predefined criteria. Note their names in the **Refinement Criterion** and **Coarsening Criterion** fields or in the console, and review them in the **Expression** dialog box to verify that they are appropriate. For example:

Setup → Named Expressions → flame_refinement_1 Edit...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 38.16: The Expression Dialog Box for the Refinement Criterion

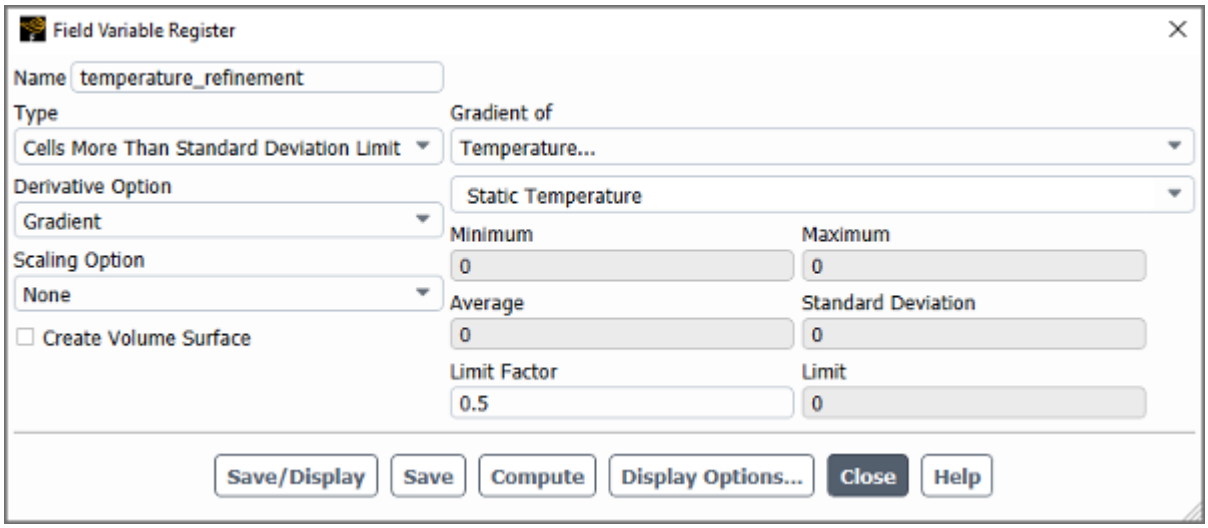


The named expressions may use cell registers that are also created as part of the predefined criteria. You can review them in the **Field Variable Register** dialog box. For example:

Solution → Cell Registers → temperature_refinement → Edit...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 38.17: The Field Variable Register Dialog Box for the Refinement Expression



38.2.1.4. VOF Adaption

When using the Volume of Fluid (VOF) multiphase model (with or without the VOF-to-DPM model transition mechanism), you can use predefined criteria to set up the mesh adaption. With this predefined criteria, Fluent will automatically generate the necessary cell registers (based on field variables) and/or named expressions for refinement and coarsening, based on the volume fraction of the secondary phase.

To use predefined criteria for adapting the mesh of a VOF simulation, perform the following steps:

1. Enable the **Volume of Fluid** multiphase model.

   Setup → **Models** → **Multiphase** → **Volume of Fluid**

2. Set up your VOF simulation, and initialize and/or run the calculation to generate data.

3. Open the **Manual Mesh Adaption** or **Automatic Mesh Adaption** dialog box.

   Domain → **Adapt** → **Manual...**

   or

   Domain → **Adapt** → **Automatic...**

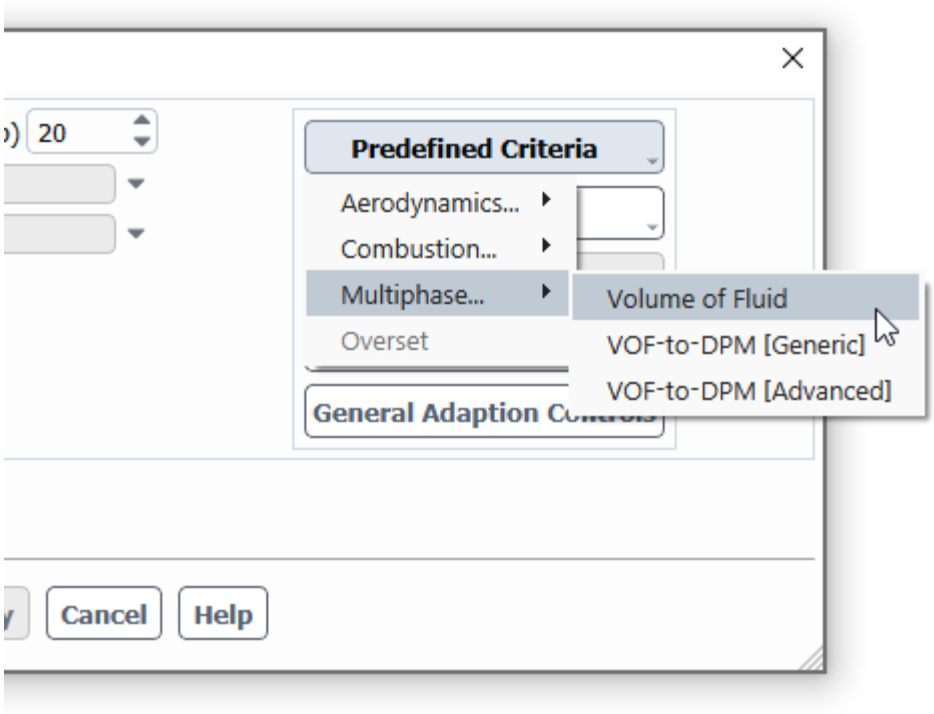
   a. Select **Multiphase...** from the **Predefined Criteria** drop-down list, and then select one of the following from the menu that opens:

      - **Volume of Fluid**: This is suitable for standard VOF simulations.
      - **VOF-to-DPM [Generic]**: This is suitable when using the VOF-to-DPM model transition mechanism; the resulting adaption criterion will be defined by fairly straightforward cell registers.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

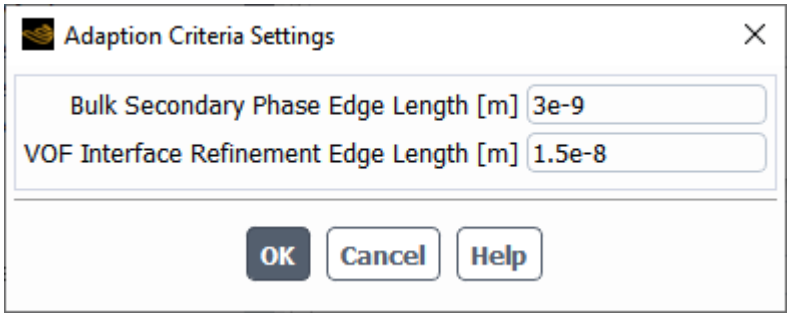
- **VOF-to-DPM [Advanced]**: This is suitable when using the VOF-to-DPM model transition mechanism; the resulting adaption criterion will be defined by more complex expressions that draw upon cell registers as well as parameters that you will need to define.

Figure 38.18: The Predefined Criteria Drop-Down List



b. An **Information** dialog box will open to notify you that you must define additional settings or parameters in the refinement / coarsening criterion expression; follow the instructions to complete the setup in the **Adaption Criteria Settings** dialog box that also opens.

Figure 38.19: The Adaption Criteria Settings Dialog Box for the VOF-to-DPM [Advanced] Criterion



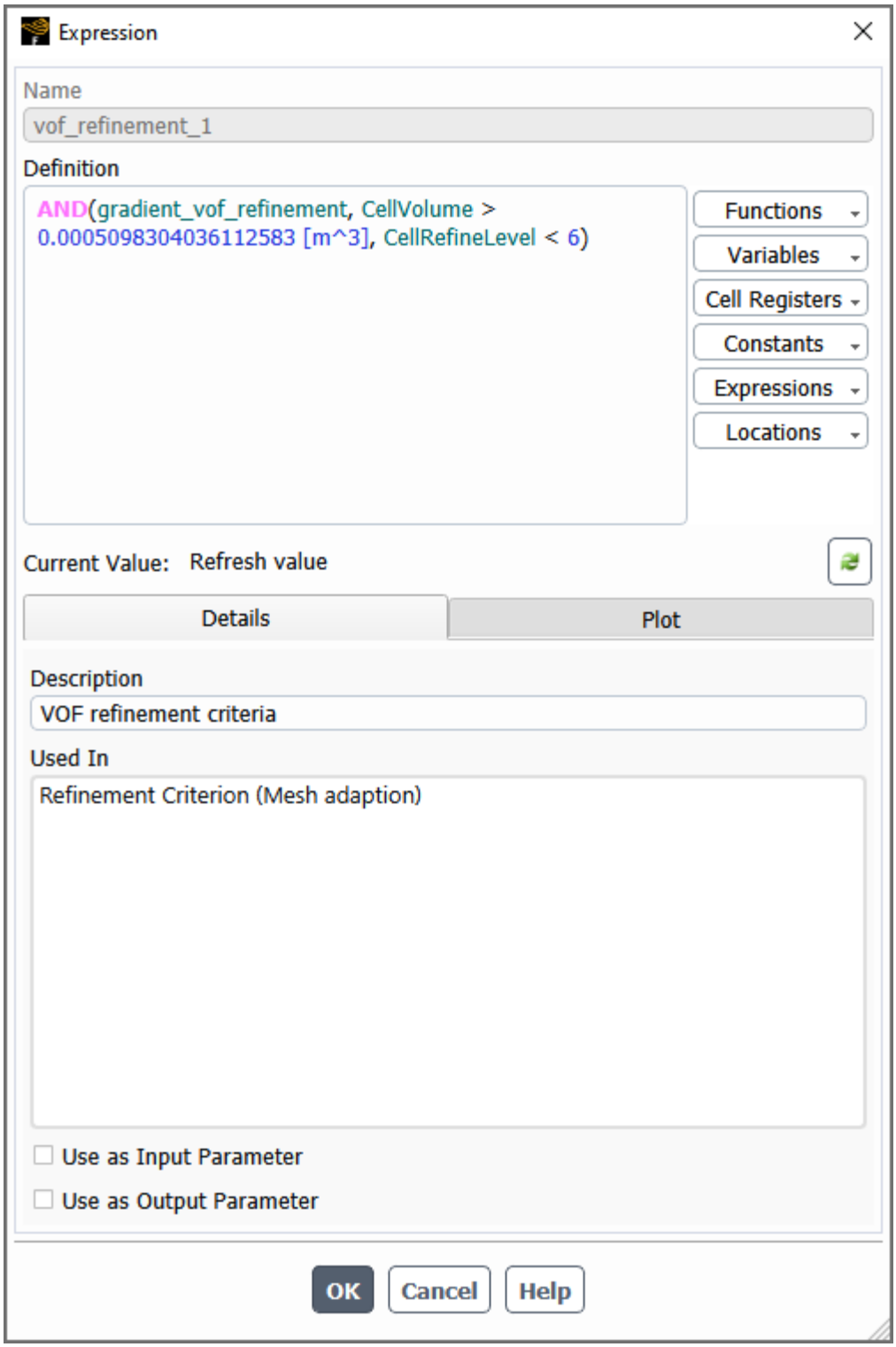
c. For an automatic adaption criterion, by default the cells will be refined at a **Frequency** of 2 iterations or time steps. You can adjust this setting as necessary.

d. Other adaption settings are defined as part of the predefined criteria. Click the **General Adaption Controls** button and review them in the **General Adaption Controls** dialog box to verify that they are appropriate.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

e. Named expressions and cell registers will be created as part of the predefined criteria. Note their names in the **Refinement Criterion** and **Coarsening Criterion** fields or in the console, and review them in the **Expressions** and **Field Variable Register** dialog boxes to verify that they are appropriate. For example:

Figure 38.20: The Expression Dialog Box for the Refinement Criterion

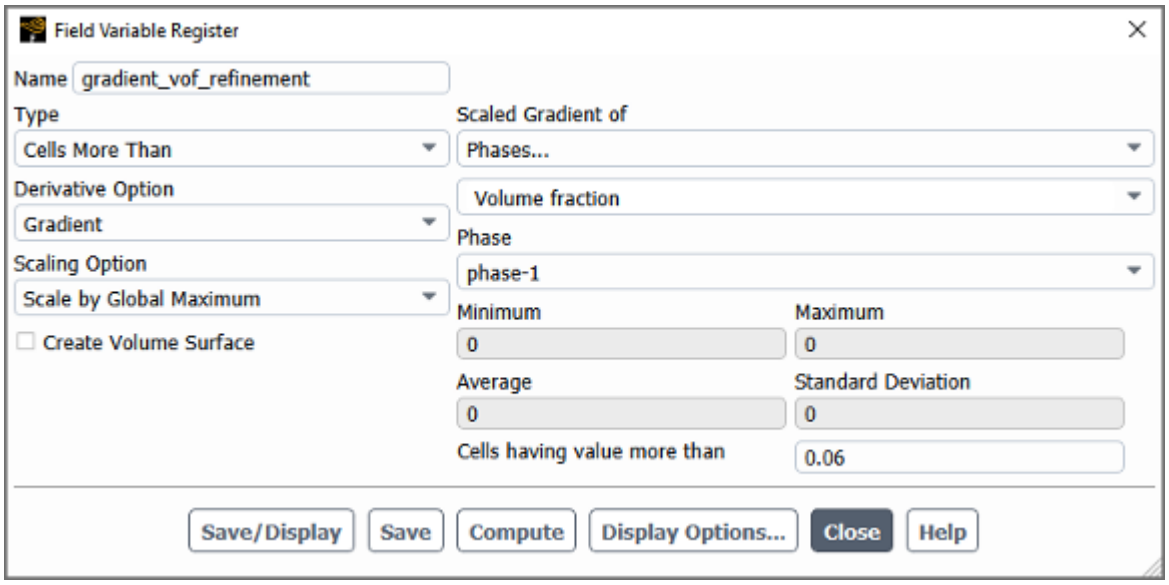


The previous expression uses the following cell register:

Solution → Cell Registers → gradient_vof_refinement → Edit...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 38.21: The Field Variable Register Dialog Box for the Refinement Criterion



38.2.1.5. Overset Adaption

You can use predefined criteria to easily set up automatic adaption for an overset mesh, with the goal of removing orphan cells, reducing size mismatches between donor and receptor cells, and/or increasing the mesh resolution in gaps as needed (in order to prevent the creation of orphan cells). Typically the overset meshes that benefit from automatic adaption involve moving and dynamic meshes, where it can be difficult to create an initial mesh that meets all of the needs throughout the simulation while also being computationally efficient. Unlike the other predefined criteria, cell registers are not used for the overset predefined criterion, but instead overset-specific adaption tools analyze the initialized overset interface and perform marking and adaption of cells in a single step. Note that by default, 3D prismatic cells that are deemed suitable will be refined anisotropically as part of overset orphan adaption and/or overset gap adaption, in order to reduce the resulting cell count.

To use predefined criteria for automatically adapting an overset mesh, perform the following steps:

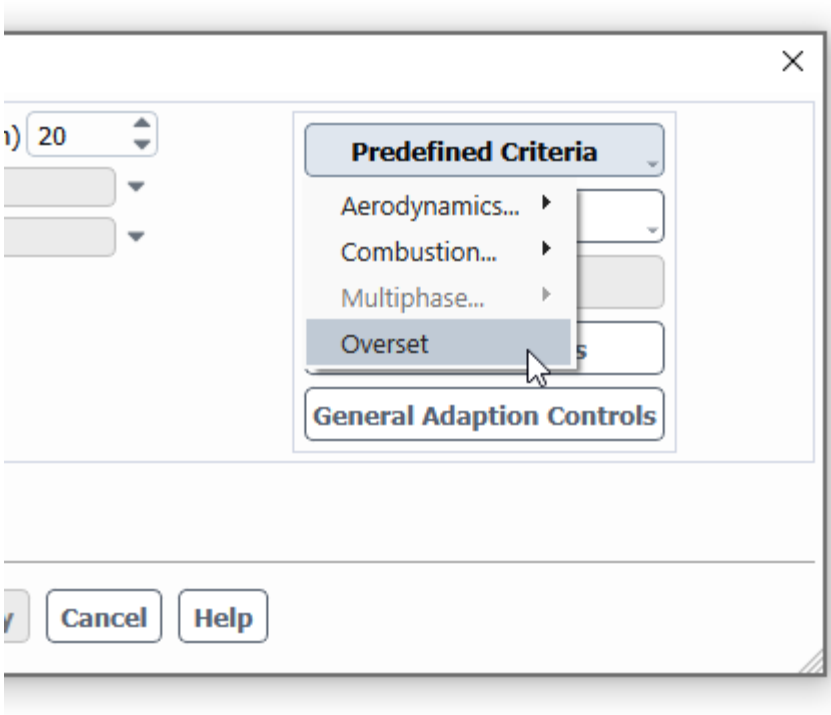
1. Set up your overset mesh simulation, being sure to define an overset interface.
2. Open the **Automatic Mesh Adaption** dialog box.

   Domain → **Adapt** → **Automatic...**

   a. Select **Overset** from the **Predefined Criteria** drop-down list.

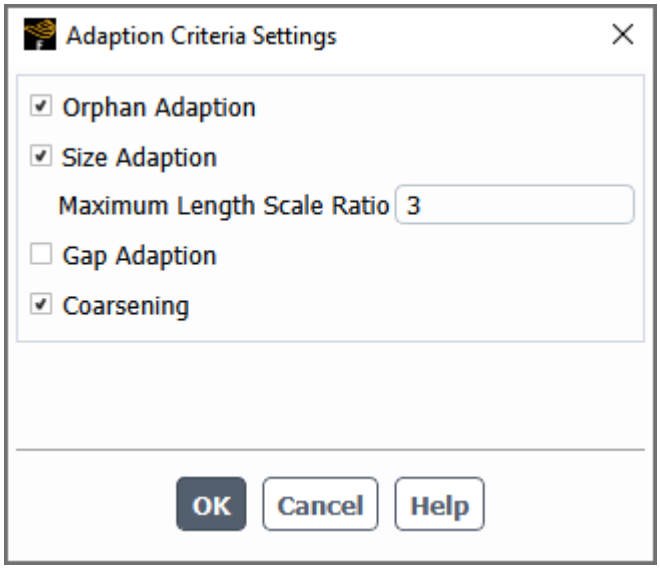
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 38.22: The Predefined Criteria Drop-Down List



b. Complete the setup in the **Adaption Criteria Settings** dialog box that opens:

Figure 38.23: The Adaption Criteria Settings Dialog Box for the Overset Criterion



i. You can enable the **Orphan Adaption** option to attempt to remove orphan cells. For details, see Marking for Orphan Adaption (p. 1219).

ii. You can enable the **Size Adaption** option to reduce size mismatches between donor and receptor cells. For details, see Marking for Size Adaption (p. 1219). As part of this option, you can define the **Maximum Length Scale Ratio** threshold (R_l) used to de-

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

termine which cells are marked for adaption based on donor-receptor cell size differences; this is set to 3 by default.

iii. You can enable the **Gap Adaption** option to increase the mesh resolution in gaps as needed (in order to prevent the creation of orphan cells). For details, see Marking for Gap Adaption (p. 1219). As part of this option, you can define the target (minimum) **Gap Resolution** used when marking cells for gap adaption; this is set to 4 by default.

iv. You can enable the **Coarsening** option to coarsen the mesh if mesh refinement is no longer needed.

v. Click **OK** to save the settings.

3. You can review and adjust additional automatic overset adaption settings in the `define/overset-interfaces/adapt/set` text command menu, as described in Overset Adaption Controls (p. 1221).

38.3. Adaption Examples

The following are examples of adapting the mesh based on different types of cell registers and one using just expressions.

38.3.1. Boundary Cell Register

38.3.2. Region Cell Register

38.3.3. Field Variable Cell Registers (Gradients, Scaling, and So On)

38.3.4. Expression Adaption Refinement

38.3.1. Boundary Cell Register

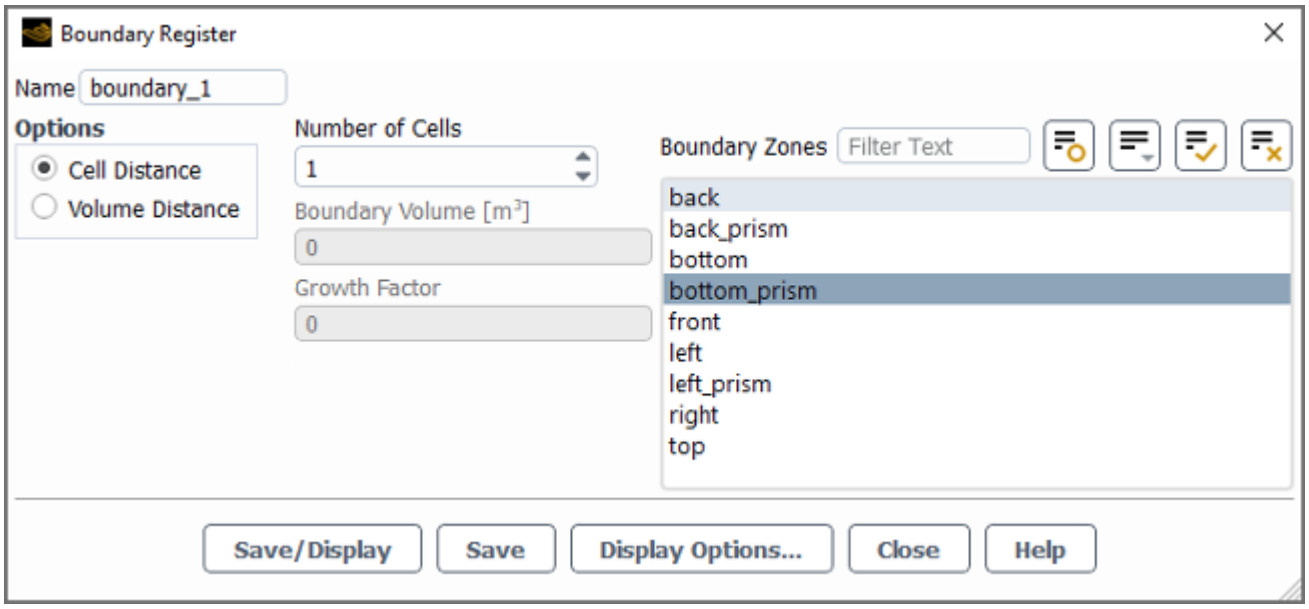
If more cells are required on a boundary, they can be added using boundary adaption, which allows you to mark or refine cells in the proximity of the selected boundary zones. The ability to refine the mesh near one or more boundary zones is provided because important fluid interactions often occur in these regions (for example, the development of strong velocity gradients in the boundary layer near a wall).

An example of a mesh that can be improved with boundary adaption is shown in Figure 38.25: Mesh Before Adaption (p. 3802).

To mark cells for refinement, you can create a boundary cell register by using either the **Predefined Criteria** button in the **Manual Mesh Adaption** dialog box (as described in Boundary Layer Adaption Based on Cell Distance (p. 3785)) or the **Boundary Register** dialog box (as described in Boundary (p. 3854) and shown in Figure 38.24: Marking Boundary Cells (p. 3802)).

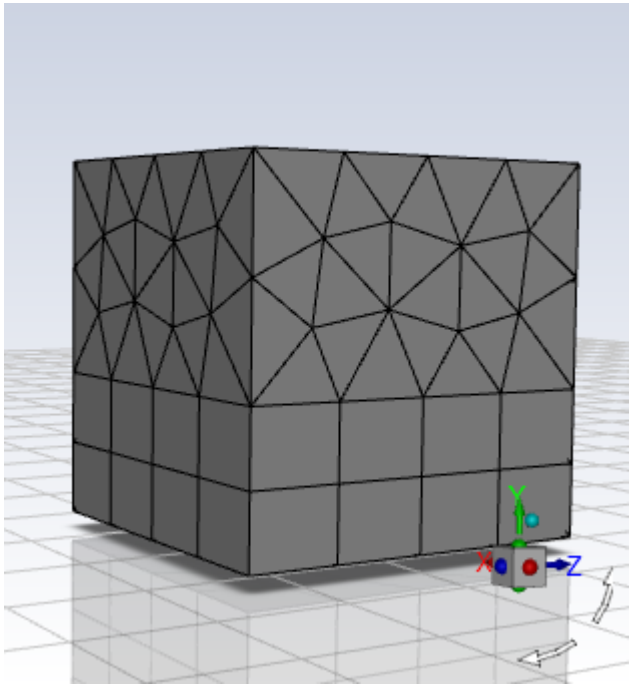
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 38.24: Marking Boundary Cells



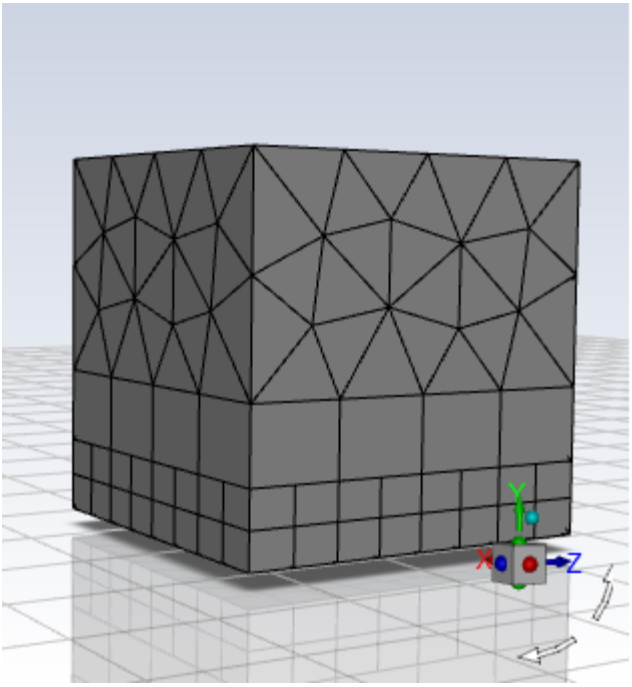
Boundary adaption can be used to increase the number of cells, as shown in Figure 38.26: Mesh after Boundary Adaption (p. 3803). This procedure cannot increase the resolution of a curved surface. Therefore, if more cells are required on a curved surface where the shape of the surface is important, create the mesh with sufficient surface nodes before reading it into the solver.

Figure 38.25: Mesh Before Adaption



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 38.26: Mesh after Boundary Adaption

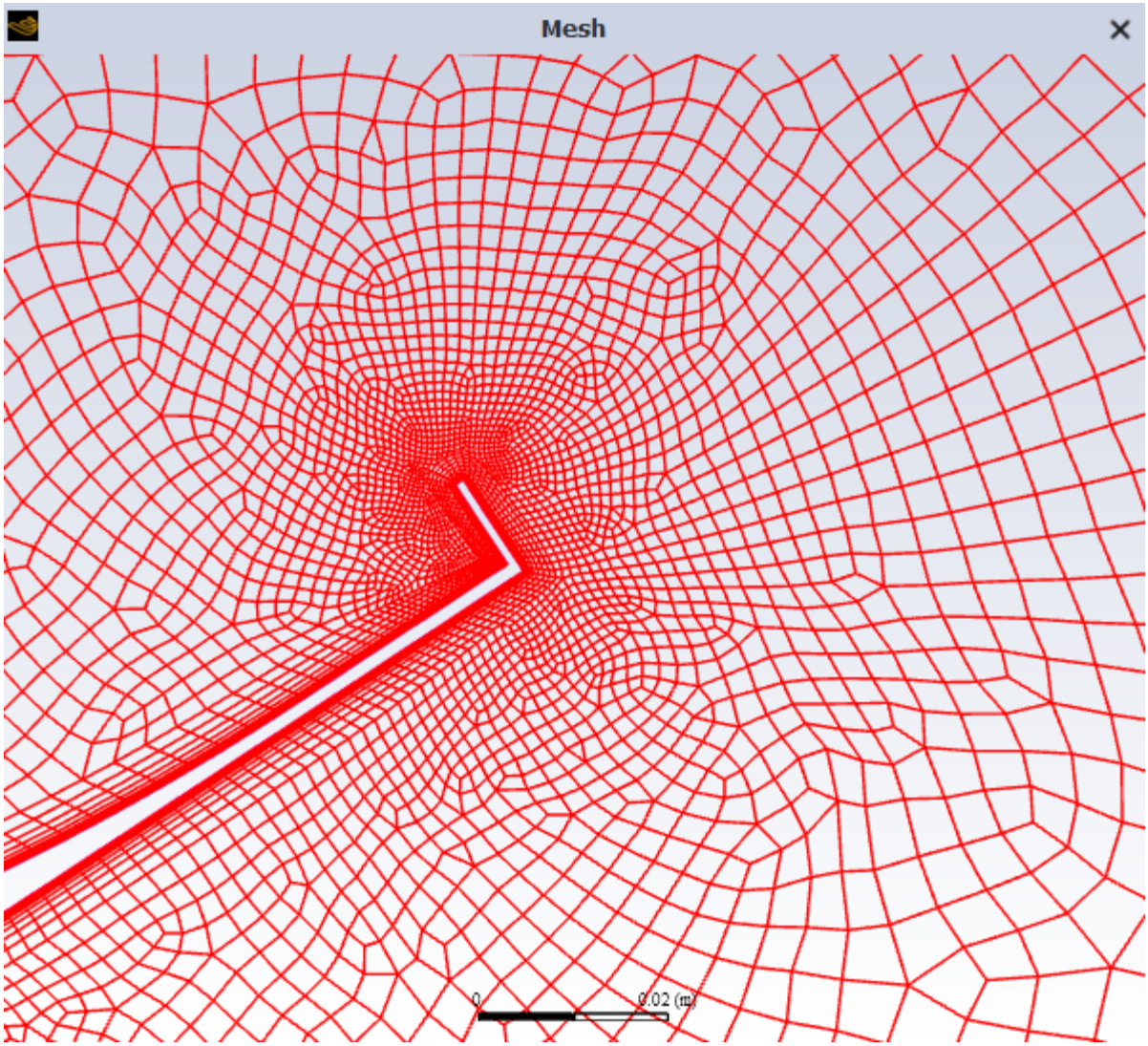


38.3.2. Region Cell Register

Figure 38.27: Wing Mesh Before Adaption (p. 3804) shows a mesh that was created for solving the flow around an airfoil wing. The mesh is fine near the surface of the wing. To capture the flow separation that occurs on the surface of the wing lip, the mesh is refined after marking cells within a circular region (selected by mouse probe) surrounding the lip of the wing. Figure 38.28: Marking Cells Based on Region (p. 3805) shows the settings that are used to mark the cells in the wake region.

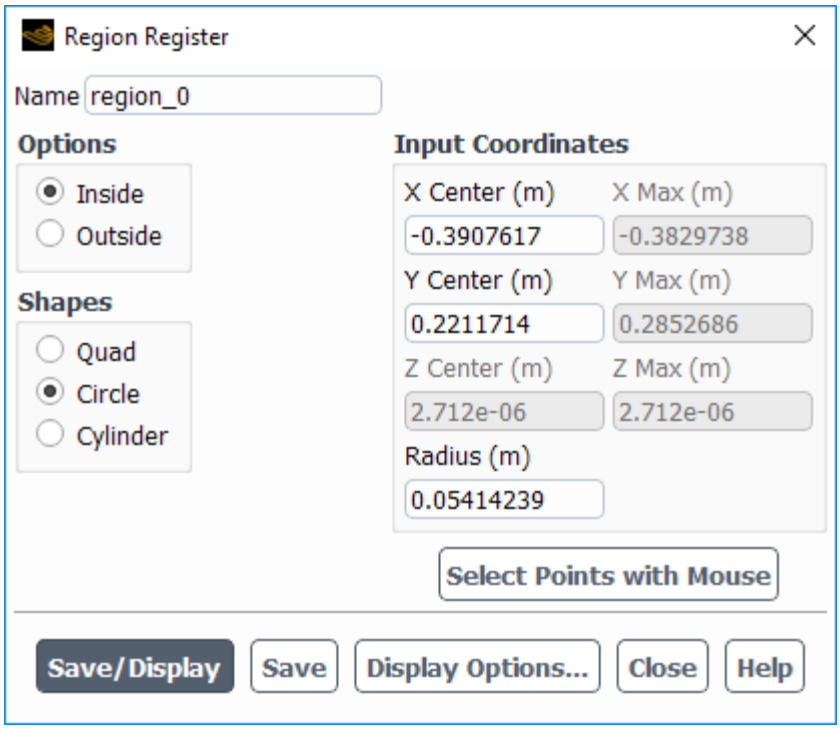
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 38.27: Wing Mesh Before Adaption



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

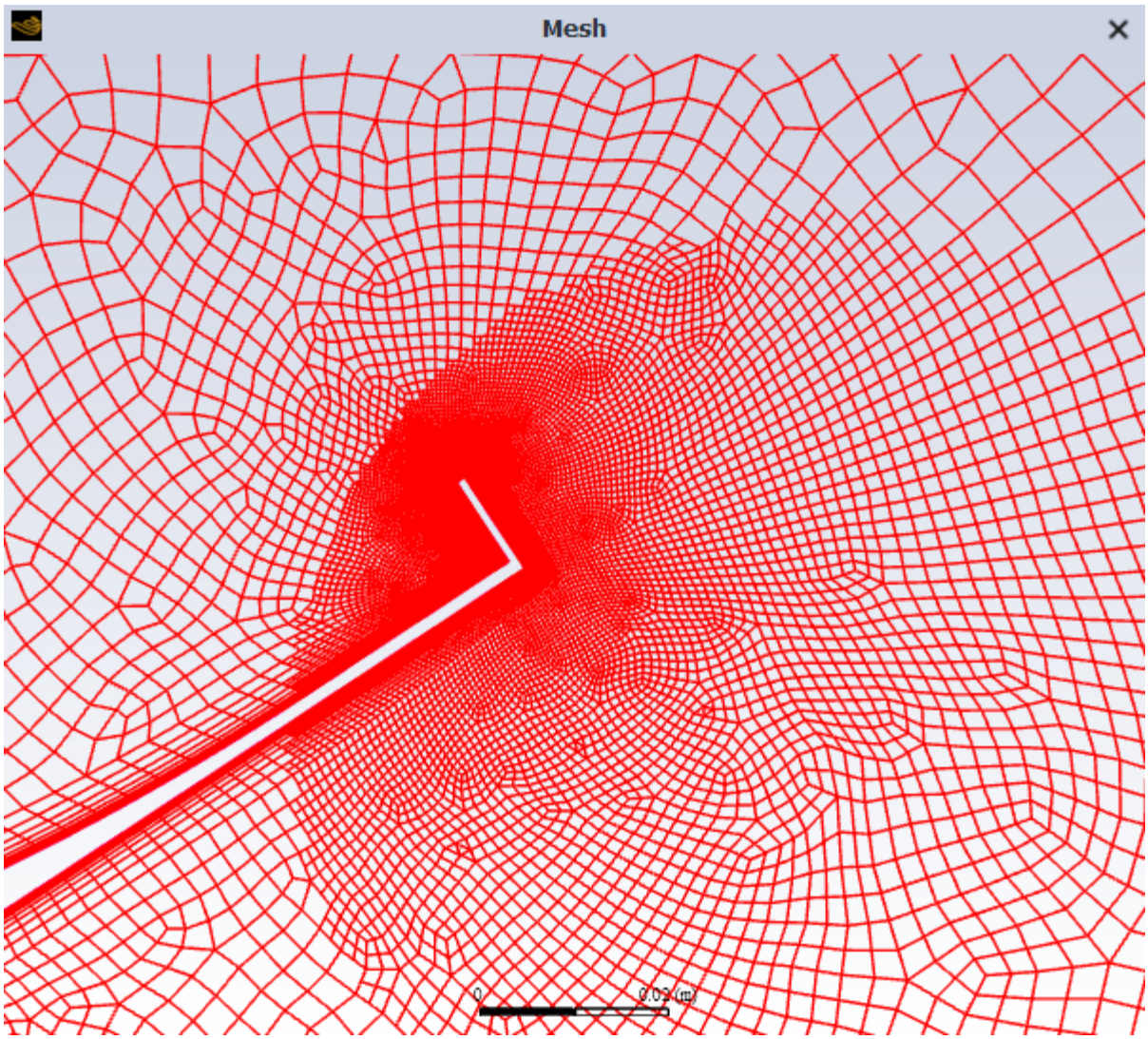
Figure 38.28: Marking Cells Based on Region



The resulting adapted mesh is shown in Figure 38.29: Wing Mesh After Region Cell Register-Based Adaption (p. 3806).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 38.29: Wing Mesh After Region Cell Register-Based Adaption



38.3.3. Field Variable Cell Registers (Gradients, Scaling, and So On)

Direct error estimation for point-insertion adaption schemes is difficult because of the complexity of accurately estimating and modeling the error in the adapted meshes. A comprehensive mathematically rigorous theory for error estimation and convergence is not yet available for CFD simulations. Assuming that maximum error occurs in high-gradient regions, the readily available physical features of the evolving flow field may be used to drive the mesh adaption process.

- When using the **Scaling Option**, suitable first-cut values for the **Cells More Than** and the **Cells Less Than** are 0.3 to 0.5, and 0.7 to 0.9, respectively. Smaller values will result in larger adapted regions.

- **Scale by Global Maximum**, scales the values of e_{i1}, e_{i2}, or e_{i3} by their maximum value in the domain, therefore always returning a problem-independent range of [0, 1] for any variable used for adaption, that is:

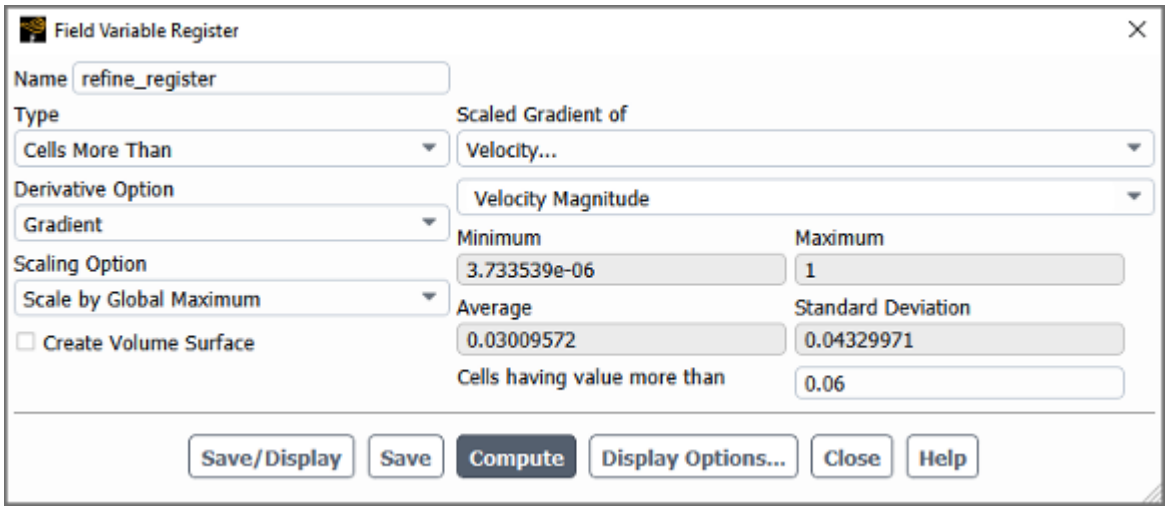
$$\frac{|e_i|}{\max|e_i|} \tag{38.1}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When using the **Scale by Global Maximum** option, suitable first-cut values for the Coarsening threshold and the refinement threshold are 0.2 to 0.4, and 0.5 to 0.9, respectively. Smaller values will result in larger adapted regions.

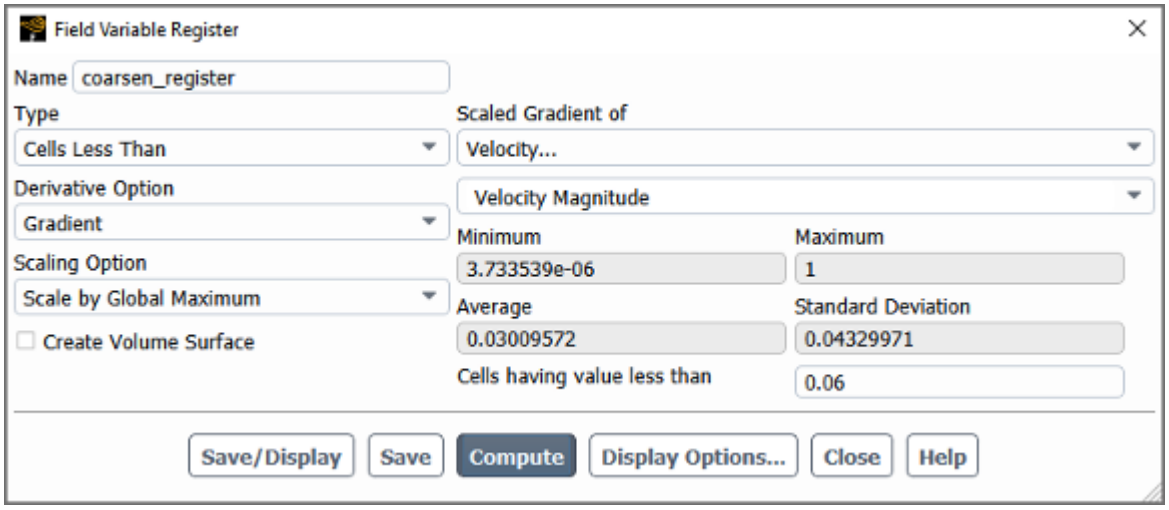
When you are using the derivative and scaling options, you can create a refinement register, like the one shown in Figure 38.30: Field Variable Refinement Register (p. 3807).

Figure 38.30: Field Variable Refinement Register



Then, using the **Cells Less Than** type of field variable register, you can create a coarsening cell register, as shown in Figure 38.31: Field Variable Coarsening Register (p. 3807).

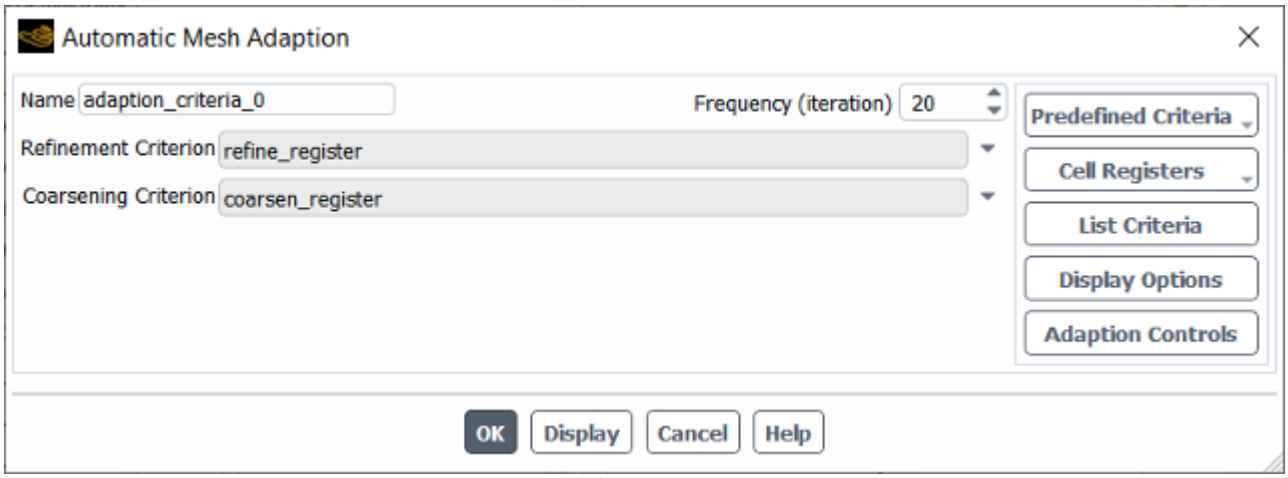
Figure 38.31: Field Variable Coarsening Register



These cell registers can then be combined in the **Manual Mesh Adaption** or **Automatic Mesh Adaption** dialog box to define parameters for adaption, as shown in Figure 38.32: Controls for Refining and Coarsening the Mesh (p. 3808).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

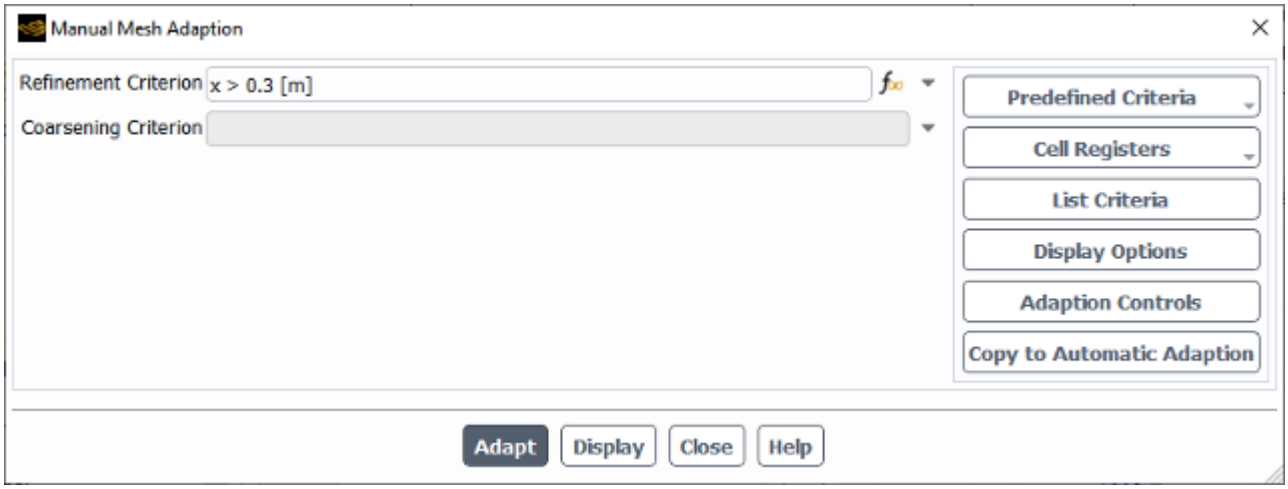
Figure 38.32: Controls for Refining and Coarsening the Mesh



38.3.4. Expression Adaption Refinement

There are several ways to use expressions to perform adaption. Expressions can be used with or without cell registers to define refinement and coarsening criteria, as shown in Figure 38.33: Simple Expression Refinement Setting (p. 3808) and Figure 38.35: Advanced Expression Refinement Setting (p. 3809).

Figure 38.33: Simple Expression Refinement Setting



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 38.34: Cells Marked for Simple Expression Refinement

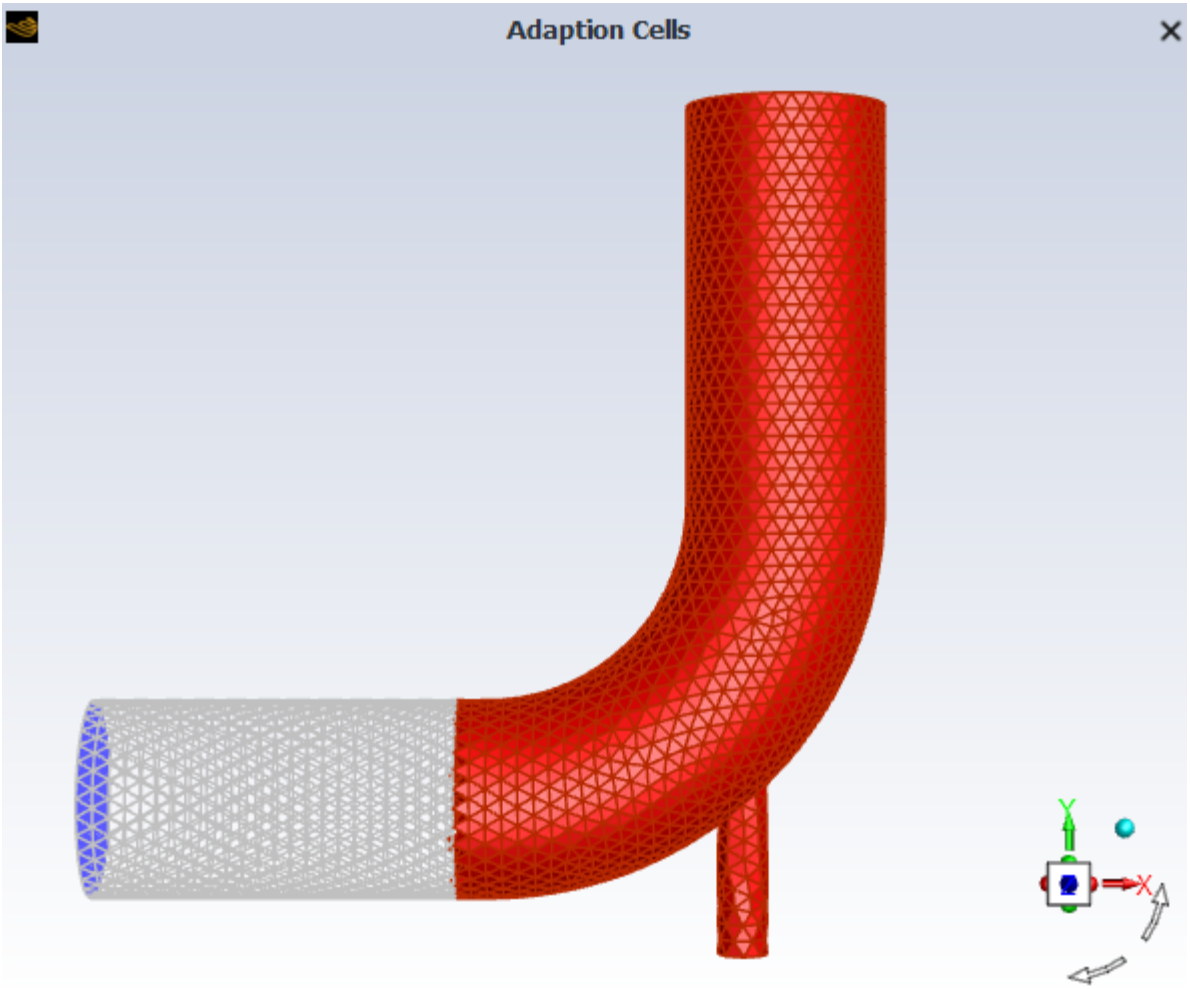
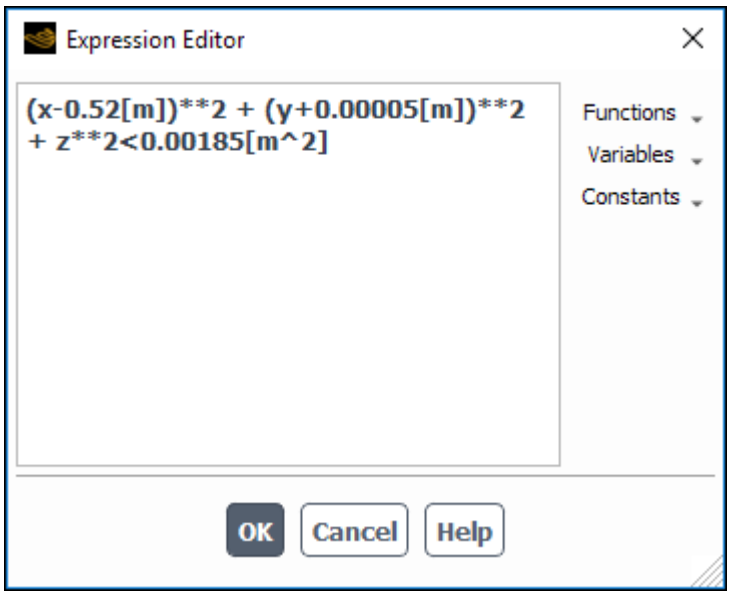
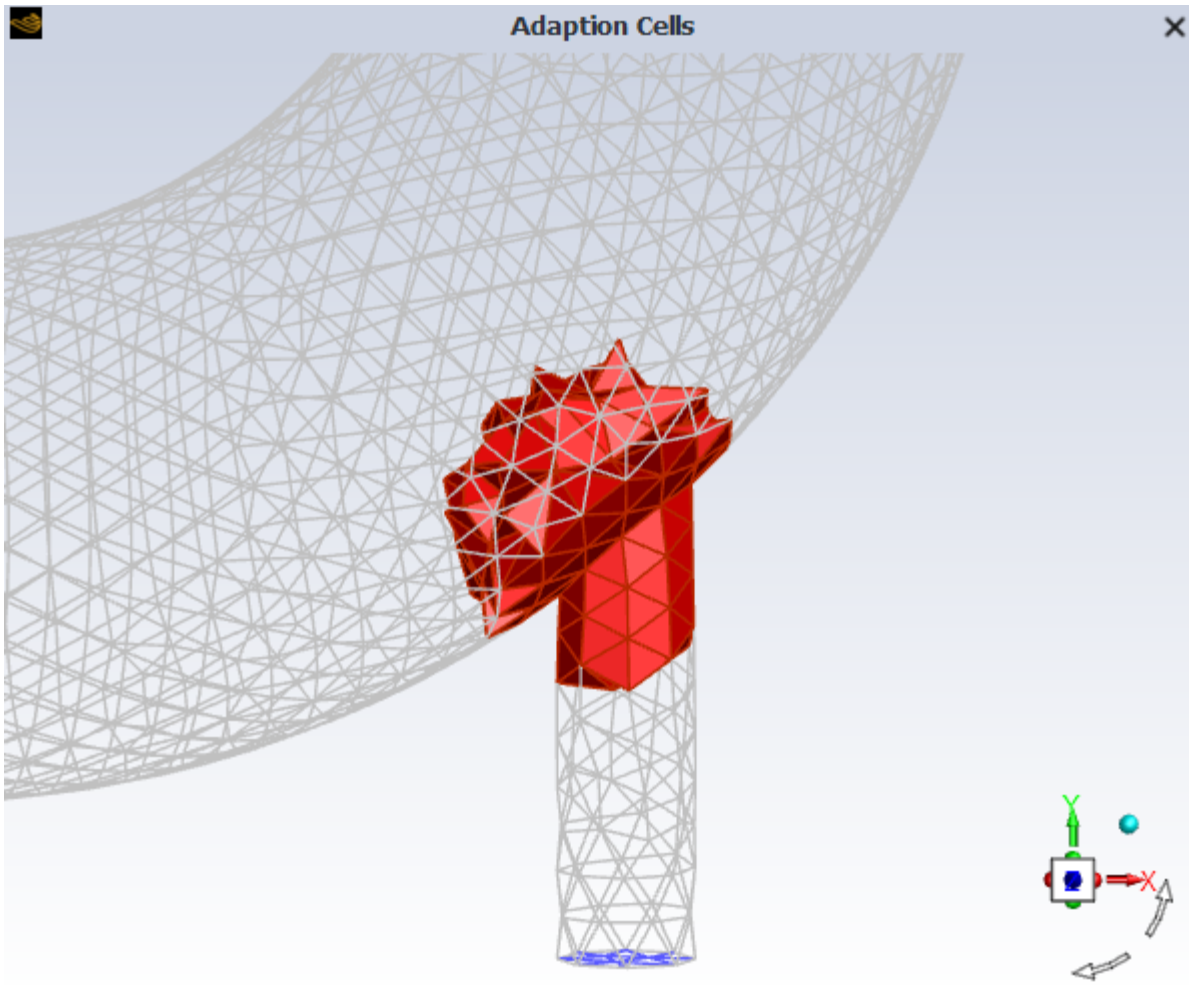


Figure 38.35: Advanced Expression Refinement Setting



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 38.36: Cells Marked for Advanced Expression Refinement



Combined Boolean Operation and Variables

You can combine booleans with various postprocessing variables. For example, `AND(OR(Static-Pressure<0.5[Pa], CellRefineLevel<3), CellVolume>1e-06[m^3])`

Combined Boolean Operation and Cell Register

If you want to refine cells near an outlet where the pressure is less than `p0`, you can use an expression: `AND(Pressure<p0 [Pa], near_outlet_cells)`. Here, `p0` is a constant real value and `near_outlet_cells` is a boundary cell register with specified values (refer to Boundary (p. 3854) for additional information on boundary cell registers).

38.4. Geometry-Based Adaption with the Hanging Node Method

This section describes how to perform geometry-based adaption with the hanging node method. For more information about the concept of geometry-based adaption, see Geometry-Based Adaption in the Theory Guide.

For additional information, see the following section:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

38.4.1. Performing Geometry-Based Adaption with the Hanging Node Method

38.4.1. Performing Geometry-Based Adaption with the Hanging Node Method

The Geometry Based Adaption Dialog Box (p. 5491) (Figure 38.37: The Geometry Based Adaption Dialog Box (p. 3811)) allows you to reconstruct the geometry while performing hanging node adaption on boundaries.

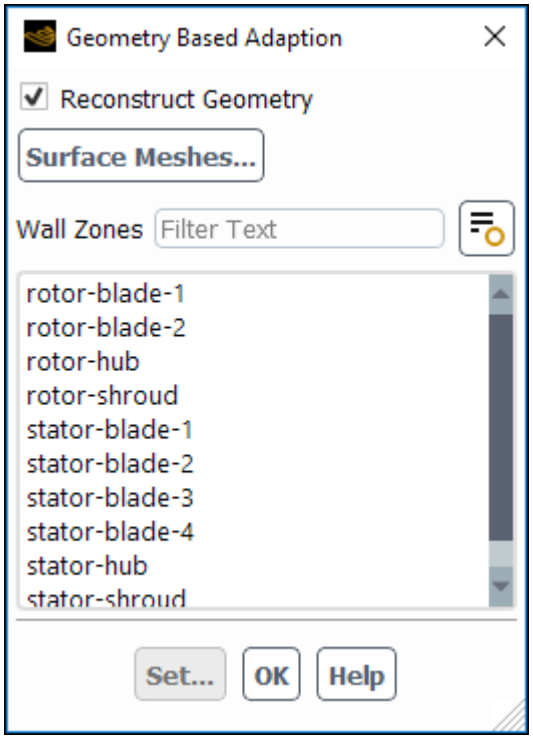
The procedure for performing geometry-based hanging node adaption is as follows:

1. Enable hanging node adaption using the following text command: `mesh/adapt/set/method hanging-node`.

2. Open the Geometry Based Adaption Dialog Box (p. 5491).

   Domain → Adapt → More → Geometry...

   Figure 38.37: The Geometry Based Adaption Dialog Box

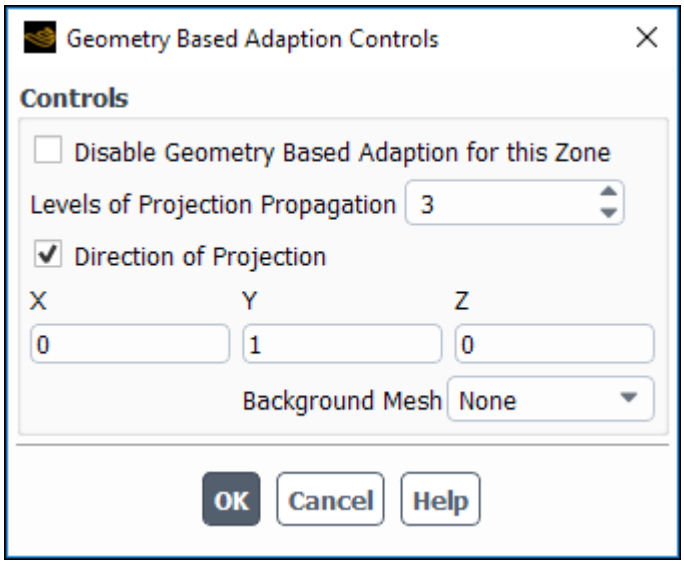
   a. Enable the **Reconstruct Geometry** option.

   b. (optional) If you have a fine surface mesh you want to use for the geometry, click the **Surface Meshes...** button and read a surface mesh using the **Surface Meshes** dialog box (Figure 6.24: The Surface Meshes Dialog Box (p. 1160)), as described in Reading Surface Mesh Files (p. 1159). Be sure to select the units that the **Mesh was created in** when using the **Select File** dialog box.

   c. Under **Wall Zones**, select the zone you want to adapt and click **Set...**. The Geometry Based Adaption Controls Dialog Box (p. 5490) will open.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 38.38: The Geometry Based Adaption Controls Dialog Box



In the Geometry Based Adaption Controls Dialog Box (p. 5490), set the following parameters:

- Specify **Levels of Projection Propagation** to indicate the number of layers of the nodes you want to project.
- Enable **Direction of Projection** and specify the directions in which you want to project the nodes.

  This will make the **X** *,* **Y** *, and* **Z** *fields available. If you want node projection in the X direction, enter* 1 *for* **X** *. If you do not enable this option, the node projection will take place at the nearest point.*
- (optional) If you read a fine surface mesh for the geometry in an earlier step, you can use the **Background Mesh** option to load the surface mesh as a background mesh. This will project the nodes based on the background mesh and reconstruct the geometry more accurately.
- To disable the geometry reconstruction for any zone in the domain, enable the **Disable Geometry Based Adaption for this Zone** option.

d. To disable geometry-based adaption for the whole domain, disable **Reconstruct Geometry**.

e. After setting the parameters for geometry-based adaption, proceed to perform mesh adaption as described in Refining and Coarsening (p. 3777).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 39: Creating Surfaces and Cell Registers for Displaying and Reporting Data

Ansys Fluent enables you to select portions of the domain to be used for visualizing the flow field. The domain portions are called *surfaces*, and there are many ways to create them. Surfaces are required for graphical analysis of 3D problems because you cannot display vectors, contours, and so on, or create an XY plot for the entire domain at once. In 2D you can usually visualize the flow field on the entire domain, but to create an XY plot of a variable in a portion of the interior of the domain, you must generate a surface. In addition, in both 2D and 3D, you will need one or more surfaces if you want to generate a surface-integral report. Note that Ansys Fluent will automatically create a surface for each boundary zone in the domain. Surface information is stored in the case file.

While you can create surfaces to visualize portions of the domain, you can create cell registers to visualize and perform operations on cell groupings. You can group cells by:

- Region
- Boundary
- Variable Limiter
- Field Variable
- Residuals
- Volume
- Yplus/Ystar

Cell registers are saved with the case file and can be used to display marked cells, adapt the mesh, apply poor mesh numerics, delete cells, and report on the cells.

Important:

All cell register and surface names must begin with either a letter or an underscore (_) and they cannot contain any slashes.

The following sections explain creating surfaces and cell registers:

39.1. Using Surfaces

39.2. Using Cell Registers

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

39.1. Using Surfaces

In order to visualize the internal flow of a 3D problem or create XY plots of solution variables for 3D results, you must select portions of the domain (surfaces) on which the data is to be displayed. Surfaces can also be used for visualizing or plotting data for 2D problems, and for generating surface-integral reports.

Ansys Fluent allows you to create several kinds of surfaces, and stores all surfaces in the case file. You can create the surfaces through the ribbon and the outline view and you can review and modify the existing surfaces in the outline view under the **Results** branch. These surfaces and their uses are described briefly below:

Zone Surfaces:

If you want to create a surface that will contain the same cells/faces as an existing cell/face zone, you can generate a zone surface. This kind of surface is useful for displaying results on boundaries.

Partition Surfaces:

When you are using the parallel version of Ansys Fluent, you may find it useful to create surfaces that are defined by the boundaries between mesh partitions. You can then display data on each side of a partition boundary.

See Parallel Processing (p. 4293) for more information about running the parallel solver.

Point Surfaces:

To monitor the value of some variable or function at a particular location in the domain, you can create a surface consisting of a single point.

Structural Point Surfaces:

To monitor the value of some variable or function at a point in a particular cell of a solid zone in a simulation that uses the structural model, you can create a surface consisting of a single point. For example, you could monitor the displacement of a solid cell as it moves / deforms as part of a two-way intrinsic fluid-structure interaction (FSI) simulation.

Line and Rake Surfaces:

To generate and display pathlines, you must specify a surface from which the particles are released. Line and rake surfaces are well-suited for this purpose and for obtaining data for comparison with wind tunnel data. A rake surface consists of a specified number of points equally spaced between two specified endpoints. A line surface is simply a line that includes the specified endpoints and extends through the domain; data points will be at the centers of the cells through which the line passes, and consequently will not be equally spaced.

Plane Surfaces:

If you want to display flow-field data on a specific plane in the domain, you can create a plane surface. A plane surface is simply a plane that passes through three specified points. *Note that you can create multiple plane surfaces at once using Figure 39.12: The Plane Surface Dialog Box (p. 3831).*

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Quadric Surfaces:

To display data on a line (2D), plane (3D), circle (2D), sphere (3D), or quadric surface you can specify the surface by entering the coefficients of the quadric function that defines it. This feature provides you with an explicit method for defining surfaces.

Iso-surfaces:

You can use an iso-surface to display results on cells that have a constant value for a specified variable. Generating an iso-surface based on x, y, or z coordinate, for example, will give you an x, y, or z cross-section of your domain. Generating an iso-surface based on pressure will enable you to display data for another variable on a surface of constant pressure. *Note that you can create multiple iso-surfaces at once using Figure 39.18: The Iso-Surface Dialog Box (p. 3839).*

Note:

- In some instances, newly created surfaces (except zone/line/point/rake) may contain erroneous lines. This can happen for surfaces created in any type of cell zone, but it is more likely in zones containing polyhedral mesh elements. Displaying a contour plot on these surfaces or displaying the surfaces directly may show these line edges. To reduce their appearance you can change the lighting method to **Flat**.
- 2D planar surfaces, such as planes and iso-surfaces of the mesh, may seem to disappear when viewed from an angle that is parallel to the line of sight. This occurs because the 2D surface does not have a thickness. If you enable **Edges** in the **Mesh Display** dialog box before displaying, then the surface will appear as a line instead of disappearing.

39.1.1. Zone Surfaces

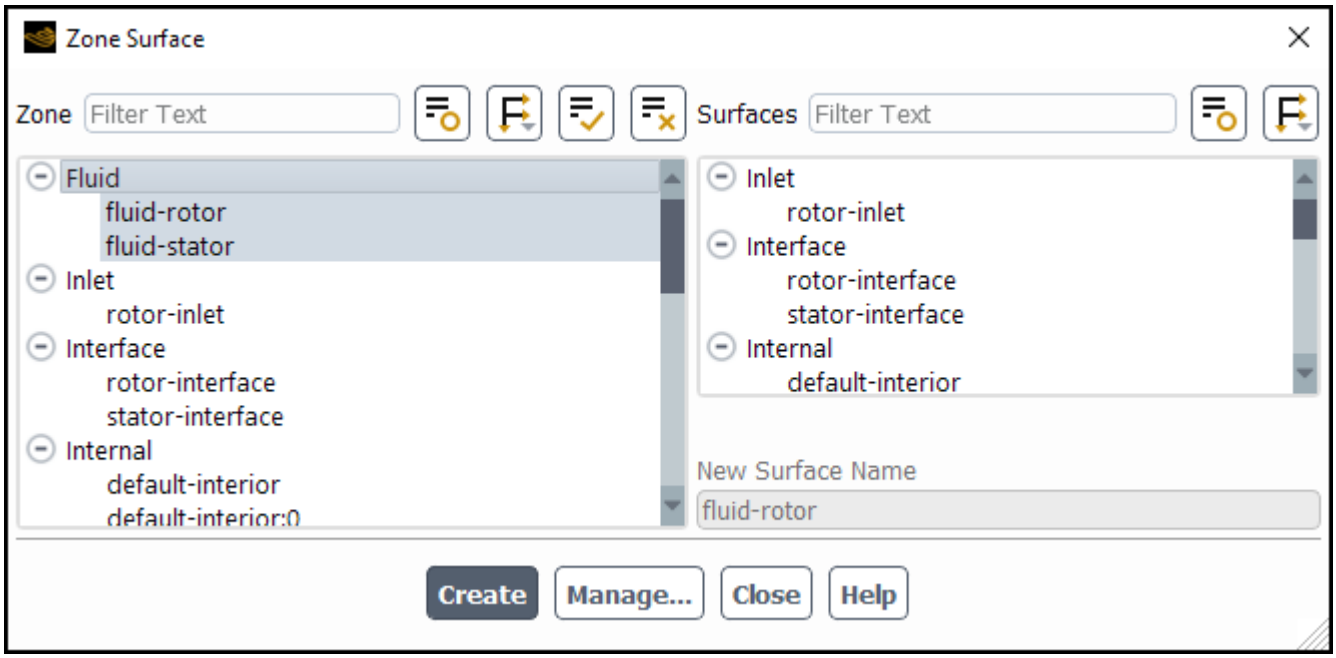
Zone surfaces are useful for displaying results on boundaries. For example, you may want to plot contours of velocity magnitude at the inlet and outlet of the problem domain, or temperature contours on the domain's walls. To do so, you need to have a surface that contains the same faces (or cells) as an existing face (or cell) zone. Zone surfaces are created automatically for all boundary face zones in the domain, so you will generally not need to create any zone surfaces unless you accidentally delete one.

Use the text command: `surface/reset-zone-surfaces` to recreate any missing surface zones by resetting the case surface list.

To create a zone surface, you will use the Zone Surface Dialog Box (p. 5663) (Figure 39.1: The Zone Surface Dialog Box (p. 3816)).

Domain → Surface → Create → Zone...

Figure 39.1: The Zone Surface Dialog Box



The steps for creating the zone surface are as follows:

1. Select the zone(s) in the **Zone** list.

2. Enter a name in **New Surface Name** if you do not want Ansys Fluent to assign a default name to the newly created surface. By default, Fluent assigns each surface the same name as the zone that it corresponds to. If a surface already exists with that name, Fluent assigns the surface a unique name using the `zone-surface` prefix (for example, `zone-surface-6`).

   If you do not want to use the default name, enter a new name in **New Surface Name**. If the **New Surface Name** that you enter is the same as a surface that already exists, Fluent assigns the new surface a unique name using the `zone-surface` prefix (for example, `zone-surface-7`).

   Important:

   - The surface name that you enter must begin with an alphabetical letter. If your surface name begins with any other character or number, Fluent rejects the entry.
   - If you are selecting multiple zones, you do not have the option to choose the surface names. Ansys Fluent automatically give the surfaces the same names as their corresponding zone.

   ---

3. Click **Create**. The new surface name is added to the **Surfaces** list in the dialog box.

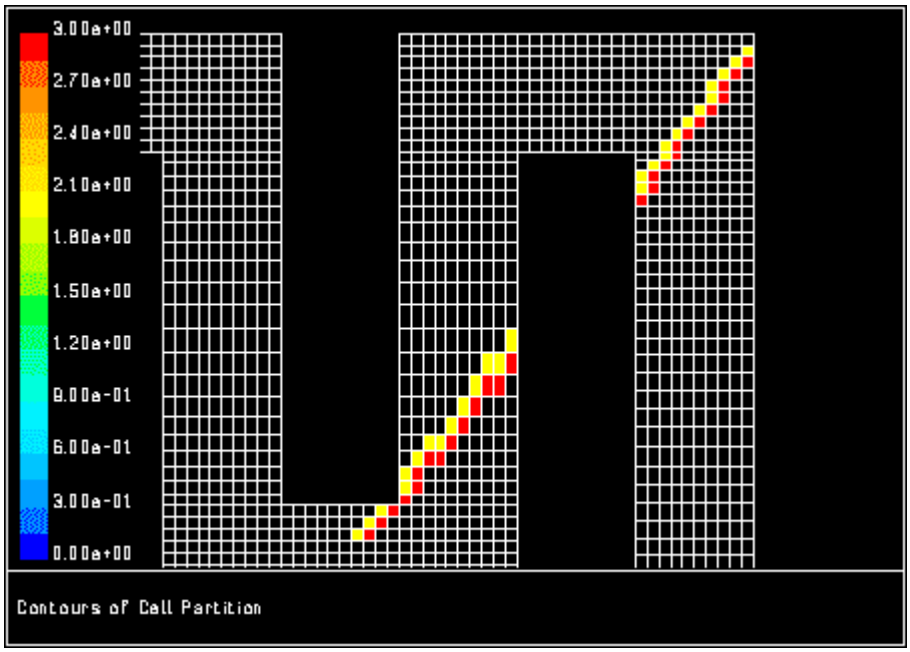
If you want to delete or otherwise manipulate any surfaces, click **Manage...** to open the Surfaces Dialog Box (p. 5629). For details, see Grouping, Prioritizing, Editing, Renaming, and Deleting Surfaces (p. 3847).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

39.1.2. Partition Surfaces

If you are using the parallel version of Ansys Fluent (see Parallel Processing (p. 4293)), you may find it useful to create data surfaces defined by the boundaries of mesh partitions. As described in Mesh Partitioning and Load Balancing (p. 4315), partitioning the mesh divides it into groups of cells that can be solved on separate processors when you use a parallel solver. A partition surface will contain faces or cells on the boundary of two mesh partitions. For example, you can plot solution values on the partition surface to determine how the solution is changing across a partition interface, as shown in the following figure:

Figure 39.2: Contours of Cell Partitions on Partition Surface Overlaid on Mesh

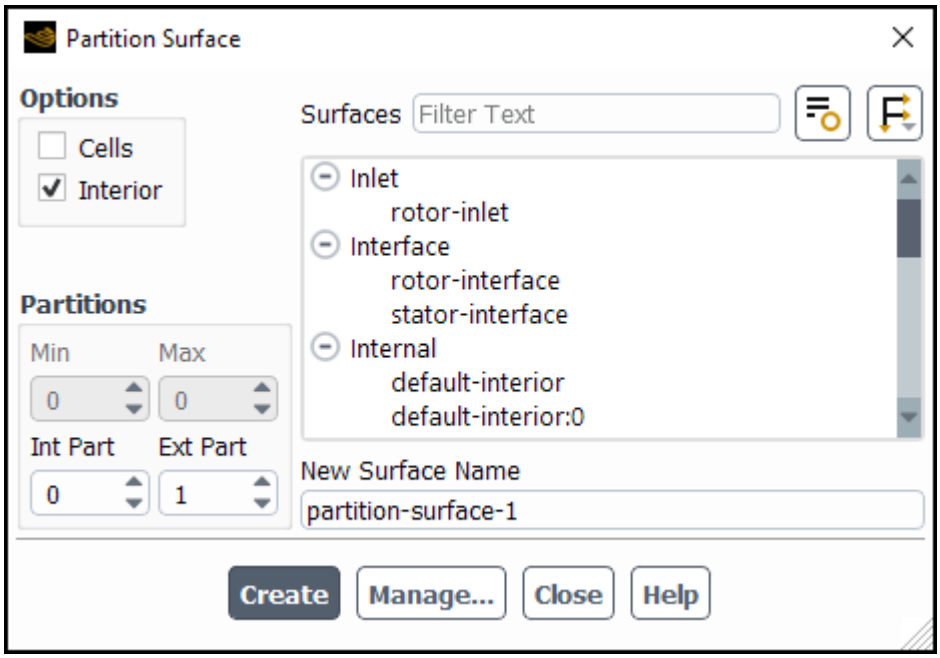


To create a partition surface, you will use the Partition Surface Dialog Box (p. 5565) (Figure 39.3: The Partition Surface Dialog Box (p. 3818)).

Domain → Surface → Create → Partition...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 39.3: The Partition Surface Dialog Box



The procedure for creating the partition surface are as follows:

1. Specify the partition boundary in which you are interested by indicating the two bordering partitions under the **Partitions** heading. The boundary that defines the partition surface is the boundary between the "interior partition" and the "exterior partition". **Int Part** indicates the ID number of the interior partition (that is, the partition under consideration), and **Ext Part** indicates the ID number of the bordering (exterior) partition. The **Min** and **Max** fields will indicate the minimum and maximum ID numbers of the mesh partitions. The minimum is always zero, and the maximum is one less than the number of processors. If there are more than two mesh partitions, each interior partition will share boundaries with several exterior partitions. By setting the appropriate values for **Int Part** and **Ext Part**, you can create surfaces for any of these boundaries.

2. Choose interior or exterior faces or cells to be contained in the partition surface by selecting or clearing **Cells** and **Interior** under **Options**. To obtain a surface consisting of cells that are on the "interior" side of the partition boundary, select both **Cells** and **Interior**. To create one consisting of cells that are on the "exterior" side, select **Cells** and clear **Interior**. If you want the surface to contain the faces on the boundary instead of the cells, clear the **Cells** option. To have the faces reflect data values for the interior cells, select the **Interior** check box, and to have them reflect values for the exterior cells, clear it.

3. If you do not want to use the default name assigned to the surface, enter a new name under **New Surface Name**. The default name is the concatenation of the surface type and an integer that is the new surface ID (for example, `partition-surface-6`). (If the **New Surface Name**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

you enter is the same as the name of a surface that already exists, Ansys Fluent will automatically assign the default name to the new surface when it is created.)

Important:

The surface name that you enter must begin with an alphabetical letter. If the name of the surface begins with any other character or number, Ansys Fluent rejects the entry.

4. Click **Create**. The new surface name is added to the **Surfaces** list in the dialog box.

If you want to delete or otherwise manipulate any surfaces, click **Manage...** to open the Surfaces Dialog Box (p. 5629). For details, see Grouping, Prioritizing, Editing, Renaming, and Deleting Surfaces (p. 3847).

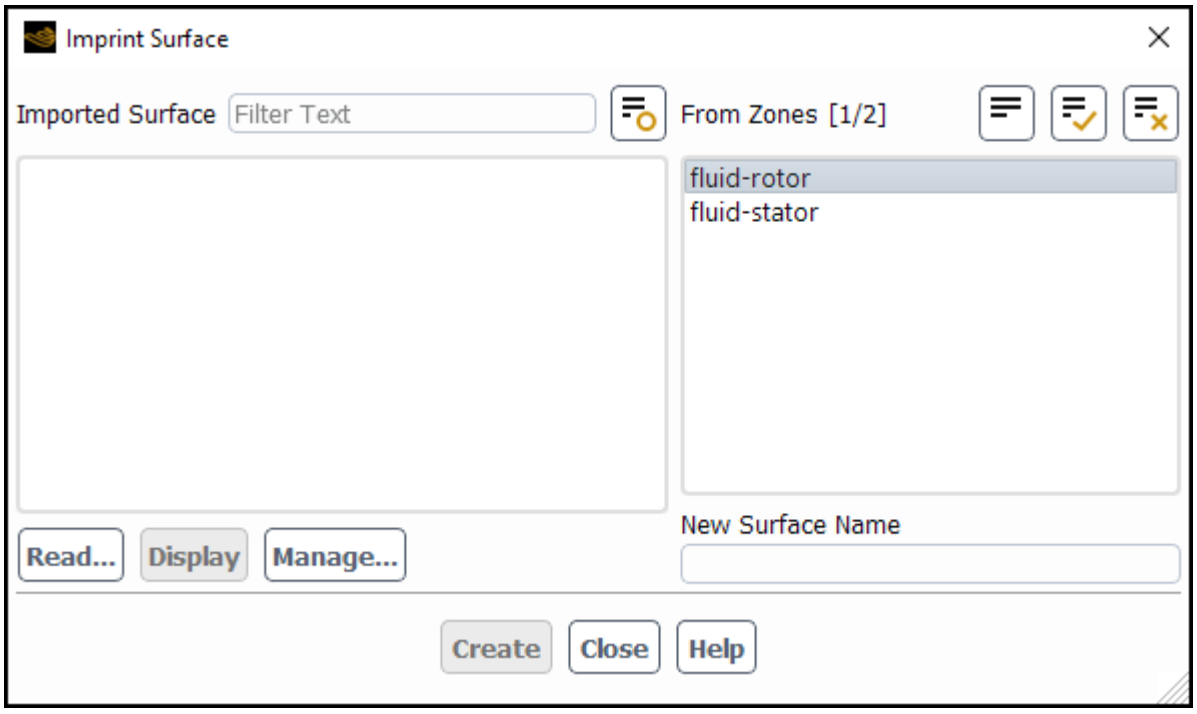
39.1.3. Imprint Surfaces

If you want to display results on surfaces that are more complicated than a simple point or plane, then you can import a custom surface representing the region of interest (`.stl`, `.msh`, or `.cas` format) and "imprint" this surface into the current mesh. For example, you may want to visualize only a smaller custom portion of a much larger (complex) geometry.

To create an imprinted surface, use the Imprint Surface Dialog Box (p. 5494) (Figure 39.4: The Imprint Surface Dialog Box (p. 3819)).

Domain → Surface → Create → Imprint...

Figure 39.4: The Imprint Surface Dialog Box



Create an imprinted surface as follows:

1. Click **Read** and select your geometry file (`.stl`, `.msh`, or `.cas` format) using the **Select File** dialog box.

   Specify the units the mesh was created in (default is meters).

2. Click **OK** to upload your file.

   The name of your geometry file appears in the **Imported Surface** *dialog box.*

3. Select the zone where your geometry is located in the **From Zones** group box.

   Note:

   If you do not specify a zone, Fluent uses the entire domain for imprinting. For a large mesh this could be time consuming.

4. Enter a name for your new surface in the **New Surface Name** dialog box.

   The default name is `<orig-name>(imprint)<id>`.

   Note:

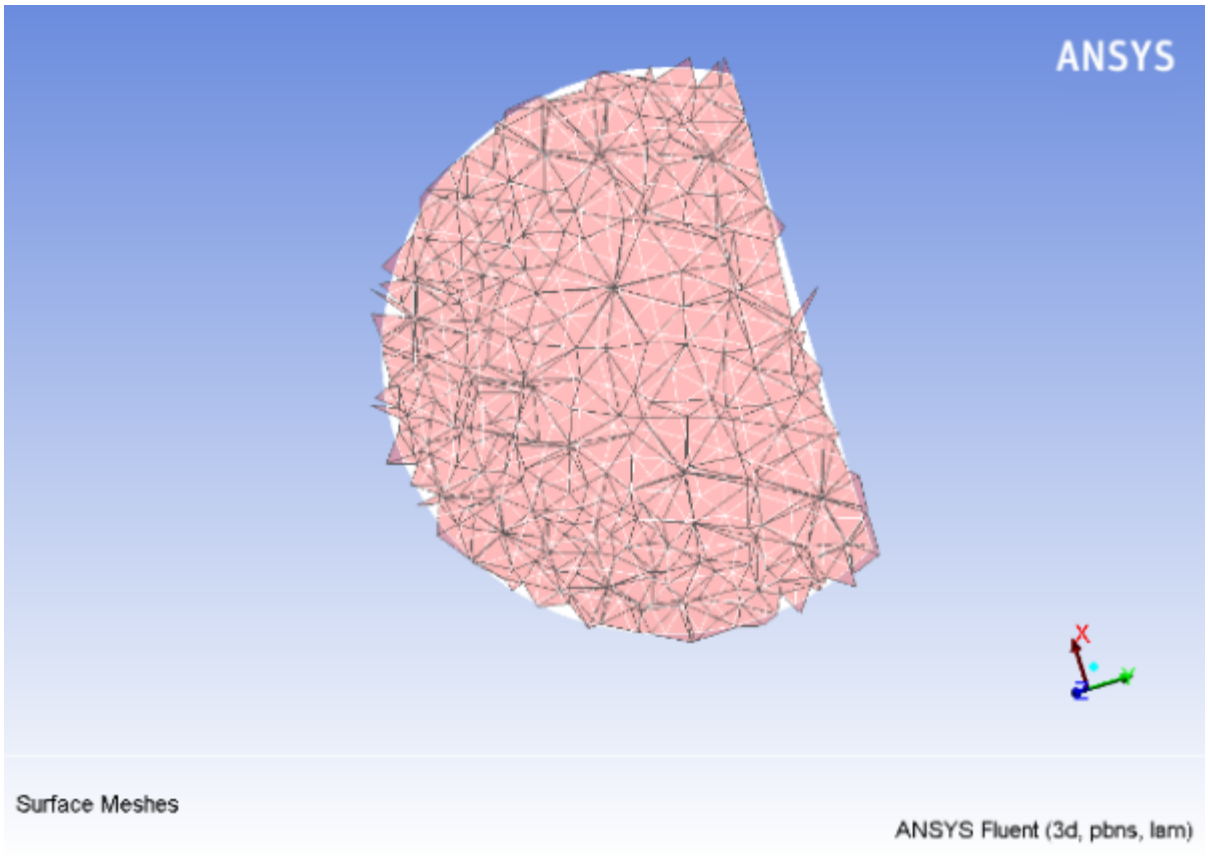
   When importing multiple surfaces, the names shown in the **Imported Surface** list may not match the name of the surface defined in the file. Confirm that imported surfaces are named and located as intended after completing the import of all surfaces. A workaround is to use binary STL files or remove the surface name from within the STL file.

5. Click **Create** to create the new surface.

When an imported surface is imprinted, a postprocessing surface that coincides with the imported surface is created. The imprinted surface will not have facets for the portions of the original surface that lie outside the domain. Different postprocessing operations such as contour and vector plots, and surface integrals can be performed on this surface. However, the edges of the imprinted surface may not coincide with those of the original surface, if the edges of the original surface do not coincide with mesh edges. An example is shown in the following picture.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 39.5: Imprinted Surface (pink) Superimposed Over Imported Surface (white)



As you can see in Figure 39.5: Imprinted Surface (pink) Superimposed Over Imported Surface (white) (p. 3821), there are deformities around the edges of the imprinted surface.

Important:

Ensure the units and scaling are consistent between the imprint surfaces and the case file.

If you want to create an imprint surface from multiple disconnected source surfaces, it is better to create multiple imprint surfaces (one per source surface) and, if required, group them together after they are created. For information on grouping surfaces, see Grouping, Prioritizing, Editing, Renaming, and Deleting Surfaces (p. 3847).

39.1.4. Point Surfaces

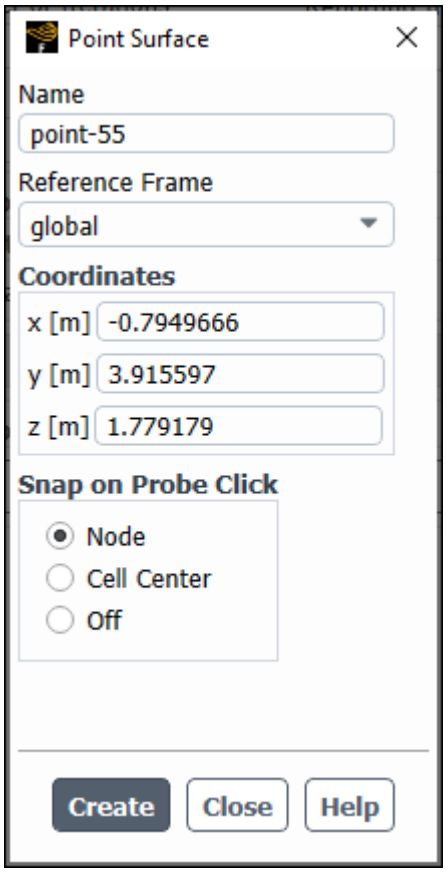
You may often be interested in displaying results at a single point in the domain. For example, you may want to monitor the value of some variable or function at a particular location. To do this, you must first create a "point" surface, which consists of a single point. When you display node-value data on a point surface, the value displayed is a linear average of the neighboring node values. If you display cell-value data, the value at the cell in which the point lies is displayed.

To create a point surface, use the Point Surface Dialog Box (p. 5582) (Figure 39.6: The Point Surface Dialog Box (p. 3822)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Domain → Surface → Create → Point...

Figure 39.6: The Point Surface Dialog Box



Create a point surface as follows:

1. (Optional) Provide a name for the point surface if you do not want to use the default name.

   Important:

   The surface name that you provide must begin with an alphabetical letter. If the name of the surface begins with any other character or number, Ansys Fluent rejects the entry.

2. Specify the location of the point. There are three different ways to do this:

   - Right-click in the domain to specify the point using the mouse probe tool (you can also left-click if you click the probe button in the toolbar—). The point tool automatically snaps

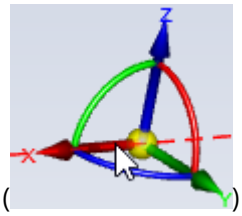
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

to the nearest **Node** or **Cell Center**, depending on the option selected in the **Snap on Probe Click** group box.

Note:

– You can make it so points are snapped to their closest valid location on the domain by enabling the `surface/closest-point-search` text command in the Console. This setting avoids point surface creation failures when the point surface is created from the text user interface or graphics user interface.

– In some instances, Fluent may snap to the center of a neighboring cell, when **Snap on Probe** is set to **Cell Center**.

- Move the point tool () to the location in the domain where you want the point surface created. Refer to Using the Point Tool (p. 3823) for additional information.
- Enter the coordinates (**x, y, z**) under **Coordinates**.

3. Click **Create** to create the new surface.

If you want to check that your new surface has been added to the list of all defined surfaces, or you want to delete or otherwise manipulate any surfaces, expand the **Surfaces** branch in the Outline View tree (under **Results**). For details, see Grouping, Prioritizing, Editing, Renaming, and Deleting Surfaces (p. 3847).

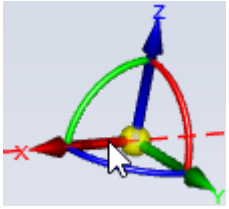
Note:

If you have trouble visualizing a point surface while running Fluent in parallel, contact your support representative for assistance with resolving the issue.

39.1.4.1. Using the Point Tool

The point tool allows you to define the location of a point surface directly in the graphics window. You can drag the point tool to the desired location on the model.

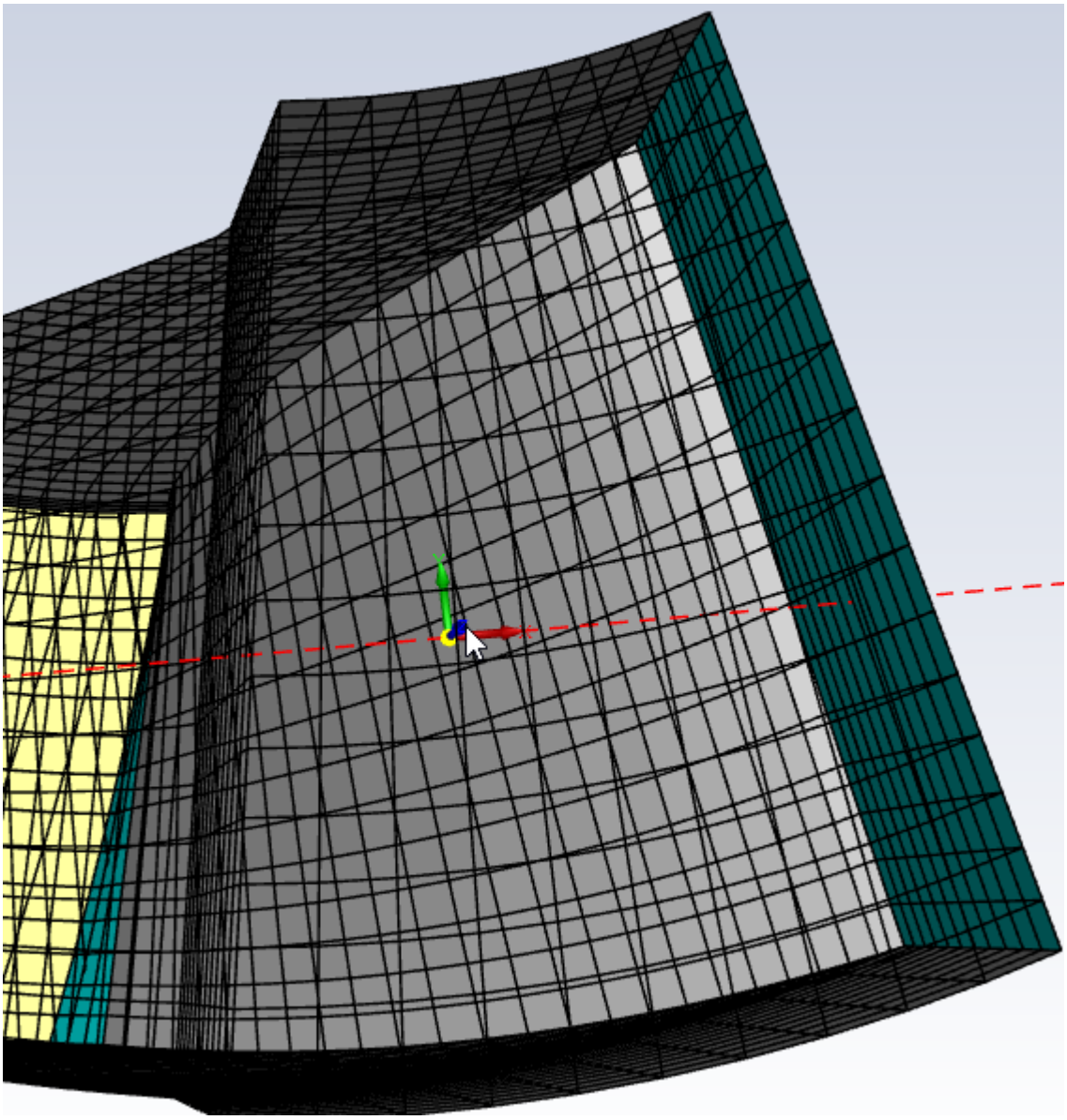
Figure 39.7: The Point Tool



Move the tool by left-clicking and holding any of the axes to move in an axial direction, or by clicking the ball in the middle of the point tool to move it freely. (You may find it helpful to display mesh faces to ensure that the point tool is correctly positioned inside the domain.)

(Optional) Click the **Front Faces Transparent** icon in the toolbar ( ) to make it easier to see the point tool once it is inside the model.

Figure 39.8: Using the Point Tool with Front Faces Transparent



Note:

- For 2D cases, snapping with the point tool requires a cell zone displayed with faces enabled in the active window. Displaying only the "default-interior" does not provide Fluent with the information needed to snap to the selected location.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- You can control the size of the point tool triad using the **Graphics triad tool size** field in the **Graphics** branch of **Preferences** (**File**>**Preferences...**). *Note that the updated tool size only appears after restarting the Fluent application.*
- If you manually input a value for the x, y, or z-coordinate that is outside the domain, the point tool will move to that location, but it will lose its interactivity.

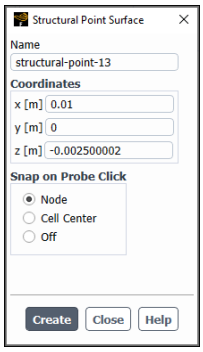
39.1.5. Structural Point Surfaces

You may be interested in reporting data at a point in a particular cell of a solid zone in a simulation that uses the structural model. For example, you may want to monitor the displacement of a solid cell as it moves / deforms as part of a two-way intrinsic fluid-structure interaction (FSI) simulation. To do this, you must first create a structural "point" surface, which consists of a single point. When you display node-value data on a structural point surface, the value displayed is a linear average of the neighboring node values. If you display cell-value data, the value at the cell in which the point lies is displayed.

To create a structural point surface, use the Structural Point Surface Dialog Box (p. 5623) (Figure 39.9: The Structural Point Surface Dialog Box (p. 3825)).

Domain → Surface → Create → Structural Point...

Figure 39.9: The Structural Point Surface Dialog Box



Create a structural point surface as follows:

1. (Optional) Provide a name for the structural point surface if you do not want to use the default name.

   Important:

   The surface name that you provide must begin with an alphabetical letter. If the name of the surface begins with any other character or number, Ansys Fluent rejects the entry.

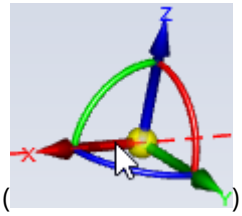
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Specify the location of the point. There are three different ways to do this:

   Note:

   The point must be in a solid cell zone. The structural model must also be enabled for the **Structural Point Surface** dialog box to be available (refer to Modeling Fluid-Structure Interaction (FSI) Within Fluent (p. 3227) for additional information).

   - Right-click in the domain to specify the point using the mouse probe tool (you can also left-click if you click the probe button in the toolbar—). The point tool automatically snaps to the nearest **Node** or **Cell Center**, depending on the option selected in the **Snap on Probe Click** group box.

   - Move the point tool () to the location in the domain where you want the point surface created. Refer to Using the Point Tool (p. 3823) for additional information.

   - Enter the coordinates (**x, y, z**) under **Coordinates**.

3. Click **Create** to create the new surface.

   Note:

   Clicking **Center** moves the point tool and coordinates to the center of the current graphics window view.

If you want to check that your new surface has been added to the list of all defined surfaces, or you want to delete or otherwise manipulate any surfaces, expand the **Surfaces** branch in the Outline View tree (under **Results**). For details, see Grouping, Prioritizing, Editing, Renaming, and Deleting Surfaces (p. 3847).

39.1.6. Line and Rake Surfaces

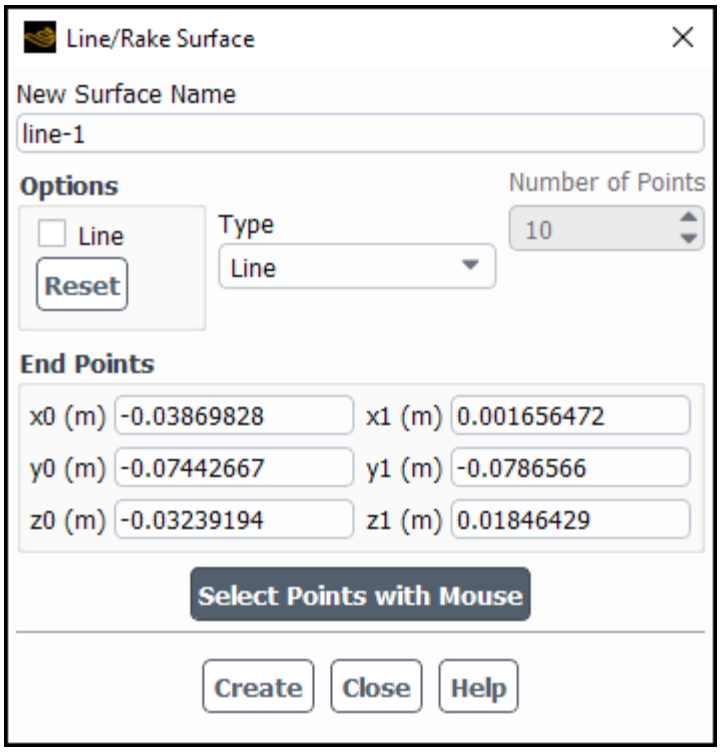
You can create lines and rakes in the domain for releasing particles, obtaining data for comparison with tunnel data, and so on. A rake consists of a specified number of points equally spaced between two specified endpoints. A line is simply a line that extends up to and includes the specified endpoints; data points will be located where the line intersects the faces of the cell, and consequently may not be equally spaced.

To create a line or rake surface, you will use the Line/Rake Surface Dialog Box (p. 5514) (Figure 39.10: The Line/Rake Surface Dialog Box (p. 3827)).

Domain → Surface → Create → Line/Rake...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 39.10: The Line/Rake Surface Dialog Box



The steps for creating the line or rake surface are as follows:

1. Indicate whether you are creating a **Line** surface or a **Rake** surface by selecting the appropriate item in the **Type** drop-down list.

2. If you are creating a rake surface, specify the **Number of Points** to be equally spaced between the two endpoints.

3. Specify the location of the line or rake surface. There are three different ways to define the location:

   - Enter the coordinates of the first point (**x0, y0, z0**) and the last point (**x1, y1, z1**) under **End Points**.

   - Click **Select Points With Mouse** and then select the endpoints by clicking on locations in the active graphics window with the mouse-probe button. (See Controlling the Mouse Button Functions (p. 4020) for information about setting mouse button functions.)

   - Use the **Line Tool** option to interactively position a line in the graphics window. You can set the initial location of this line using one of the two methods described above for specifying endpoints (or you can start from the position defined by the default **End Points**). See Using the Line Tool (p. 3828) for information about using the line tool.

   Note that when you use the second or third method described above, the coordinates of the **End Points** are updated automatically.

4. If you do not want to use the default name assigned to the surface, enter a new name under **New Surface Name**. The default name is the concatenation of the surface type and an integer that is the new surface ID (for example, `line-5` or `rake-6`). If the **New Surface Name** you

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

enter is the same as the name of a surface that already exists, Ansys Fluent will automatically assign the default name to the new surface when it is created.

Important:

The surface name that you enter must begin with an alphabetical letter. If the name of the surface begins with any other character or number, Ansys Fluent rejects the entry.

5. Click **Create** to create the new surface.

If you want to check that the new surface has been added to the list of all defined surfaces, or you want to delete or otherwise manipulate any surfaces, click **Manage...** to open the Surfaces Dialog Box (p. 5629). See Grouping, Prioritizing, Editing, Renaming, and Deleting Surfaces (p. 3847) for details.

39.1.6.1. Using the Line Tool

The line tool enables you to interactively fine-tune the definition of a line or rake using graphics. Starting from an initial line, you can translate, rotate, and resize the line until its position, orientation, and length are as desired. For example, if you need to position a rake surface just inside the inlet to a duct, you can start with the line tool near the desired location (such as on the inlet), and translate, rotate, and resize it until you are satisfied. You may find it helpful to display mesh faces to ensure that the line tool is correctly positioned inside the domain.

Important:

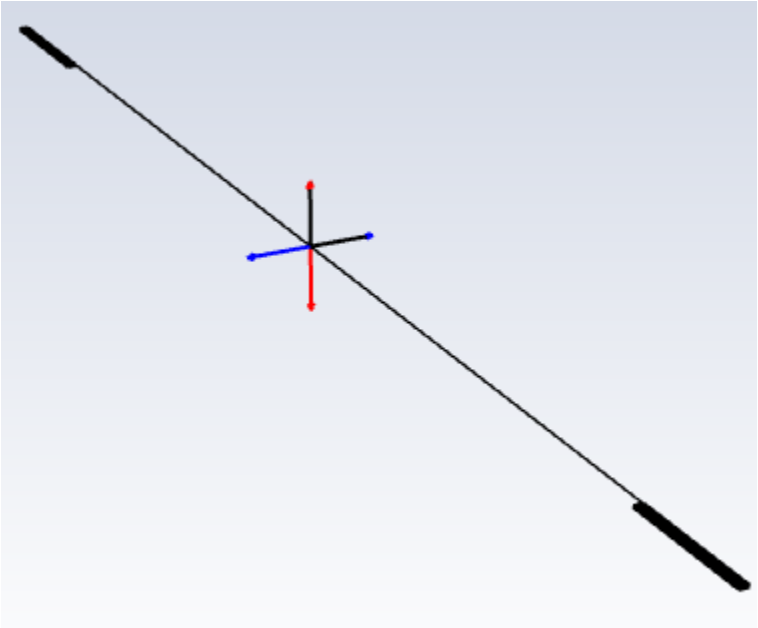
You must update the assignment of **Probe** in **Preferences** for the line tool to work. Select **Probe** for **Shift RMB Drag** (or similar drag + key combination) in the **Navigation** branch in **Preferences** (**File** > **Preferences...**).

39.1.6.1.1. Initializing the Line Tool

Before enabling the **Line Tool** option, set the **End Points** to suitable starting values. You can enter values manually, or use the **Select Points With Mouse** button. Often it is convenient to display the mesh for an inlet or iso-surface on or near where you want to place the line or rake surface and then select two points on that mesh to specify the initial position of the line tool. Once you have specified the appropriate **End Points**, enable the **Line Tool** option. The line tool appears in the graphics window, as shown in Figure 39.11: The Line Tool (p. 3829).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 39.11: The Line Tool



You can then translate, rotate, and/or resize the line tool as described in the following sections.

39.1.6.1.2. Translating the Line Tool

To translate the line tool in the direction along the red axis, click the mouse-probe button (the right button by default) anywhere on the "line" part of the tool and drag the mouse until the tool reaches the desired location. Green arrows will show the direction of motion For information about changing the mouse functions see Controlling the Mouse Button Functions (p. 4020).

Important:

Do not click the axes of the line tool that have arrows on the ends. These axes control rotation of the tool. Click only on the portion of the tool that represents the prospective line surface. This portion is designated by the rectangles attached to each end.

To translate the tool in the transverse directions (that is, along either of the axes within the plane perpendicular to the red axis), press **Shift**, click the mouse-probe button anywhere on the "line" part of the tool, and drag the mouse until the tool reaches the desired location. Two sets of green arrows will show the possible directions of motion. In 2D, there will be only one set of green arrows because there is only one other direction for translation. If you find the perspective distracting when performing this type of translation, you can disable it in the Camera Parameters Dialog Box (p. 5277) (opened from the Views Dialog Box (p. 5272)), as described in Controlling Perspective and Camera Parameters (p. 4033).

39.1.6.1.3. Rotating the Line Tool

To rotate the line tool, you click the mouse-probe button on one of the white axes with arrows. When you click one of these axes, a green ribbon will encircle the other arrowed axis, designating

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

it as the axis of rotation. As you drag the mouse along the circle to rotate the tool, the green circle will become yellow.

Important:

Do not click the red axis to rotate the line tool.

39.1.6.1.4. Resizing the Line Tool

If you plan to generate a rake surface, you can resize the line tool to define the length of the rake. Click the mouse-probe button in one of the white rectangles at the ends of the "line" part of the tool (shown in black in Figure 39.11: The Line Tool (p. 3829)) and drag the mouse to lengthen or shorten the tool. Green arrows will show the direction of stretching/shrinking.

39.1.6.1.5. Resetting the Line Tool

If you "lose" the line tool, or want to reset it for any other reason, you can either click **Reset** to return the line tool to the default position and start from there, or turn the tool off and re-initialize it as described above. In the default position, the line tool will lie midway along the x and y lengths of the domain, spanning the z domain extent.

39.1.7. Plane Surfaces

To display flow-field data on a specific plane in the domain, you will use a plane surface. You can create surfaces that cut through the solution domain along arbitrary planes (3D cases only) and you have the option to create more than one plane at once.

There are three types of plane surfaces that you can create:

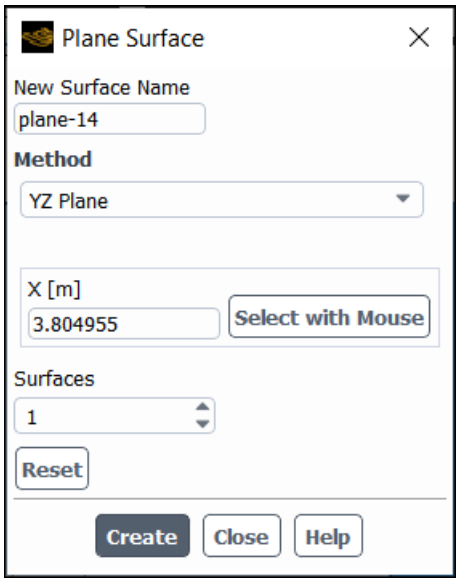
- Coordinate system-based—the plane is created in the YX, ZX, or XY directions, bounded by the extents of the domain. You can move the plane to the desired location in the domain using the plane tool. For example, if you are using the **YZ Plane** method, you can drag the plane in the (+) or (-) X direction.
- Point and normal—the plane orientation is determined by selecting a point and specifying a direction normal to that point. The extents of the plane are the edges of the domain. You have the option to control the orientation of the plane using the plane tool or you can compute the normal from a surface.
- Three points—the plane orientation and extents are bounded by three points that you can select. You also have the option to manipulate the points and orientation of the plane directly in the graphics window using the plane tool.

To create a plane surface, you will use the Plane Surface Dialog Box (p. 5580) (Figure 39.12: The Plane Surface Dialog Box (p. 3831)).

Domain → **Surface** → **Create** → **Plane...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 39.12: The Plane Surface Dialog Box



The procedure for creating the plane surface vary depending on the method you want to use.

YZ Plane, ZX Plane, or XY Plane

1. (Optional) Provide a name for the plane.

   Important:

   The surface name that you enter must begin with an alphabetical letter. If the name of the surface begins with any other character or number, Ansys Fluent rejects the entry.

   If you are creating more than one plane at once, the **Name** field is updated to **Name Format** with contents that look like `plane-x={x:+.6f}`. This naming format results in plane names that reflect the coordinates of the plane. For example, if you create three YZ planes, with a spacing of 0.05m, starting from X=2.325m, the planes will be named `plane-x=+2.325000`, `plane-x=+2.375000`, and `plane-x=+2.425000`. You can control how many significant digits are included in the plane name by varying the value represented by 6 in the example name format. The `f` in the example name indicates that the output is a float. Changing the `f` to an `e` results in the coordinate being represented as an exponential.

2. Drag the plane tool to the desired location, as shown in Figure 39.13: The Plane Tool for the YZ Plane Method (p. 3834).

   Alternatively, you can enter the coordinate that determines the location of the plane, for example the **X** coordinate in a **YZ** plane. You can click **Select with Mouse** to use the probe functionality (right-click) to select the third point in the graphics window.

3. (Optional) Increase the number of **Surfaces** to create more than one surface at once.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Specify the **Spacing** between the planes. You can either directly enter the value, or you can use the plane tool to move the planes closer or further apart.

4. Click **Create**.

Point and Normal

1. (Optional) Provide a name for the plane.

   Important:

   The surface name that you enter must begin with an alphabetical letter. If the name of the surface begins with any other character or number, Ansys Fluent rejects the entry.

   If you are creating more than one plane at once, Ansys Fluent appends an incremented value of the plane to the plane surface name. For example, if your **New Surface Name** is `plane-12` and you are creating 3 planes with a spacing of 0.01m, the names of the planes will be `plane-12=0`, `plane-12=0.01`, and `plane-12=0.02`.

2. Drag the plane tool to the desired location, as shown in Figure 39.14: The Plane Tool for the Point and Normal Method (p. 3835).

   Alternatively, you can specify a **Point** (either by providing the coordinates or by clicking **Select with Mouse** and selecting the point using the probe function—typically a right-click) then specify the **Normal**. For the **Normal** you can either provide the components or compute it from a surface or the view plane.

3. (Optional) Increase the number of **Surfaces** to create more than one surface at once.

   - Specify the **Spacing** between the planes. You can either directly enter the value, or you can use the plane tool to move the planes closer or further apart.

4. Click **Create**.

Three Points

1. (Optional) Provide a name for the plane.

   Important:

   The surface name that you enter must begin with an alphabetical letter. If the name of the surface begins with any other character or number, Ansys Fluent rejects the entry.

   If you are creating more than one plane at once, Ansys Fluent appends an incremented value of the plane to the plane surface name. For example, if your **New Surface Name** is `plane-12` and you are creating 3 planes with a spacing of 0.01m, the names of the planes will be `plane-12=0`, `plane-12=0.01`, and `plane-12=0.02`.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Drag the plane tool to the desired location, as shown in Figure 39.15: The Plane Tool for the Three Points Method (p. 3835).

   Alternatively, you can specify **Point 1**, **Point 2**, and **Point 3** (either by providing the coordinates or by clicking **Select with Mouse** and selecting the points using the probe function—typically a right-click).

3. (Optional) You can further constrain the plane by enabling the **Bounded** and **Sample Points** options, as well as by specifying a **Sample Density**.

   - **Bounded**—the plane will be a bounded parallelogram where 3 of the 4 points are defined by the three points specified during creation of the plane.
   - **Sample Points**—allows you to specify a uniform distribution of data points on the plane.
   - **Sample Density**—specifies the density of points when the **Sample Points** option is enabled.

     Edge 1, **Edge2**—sets the point density along the two directions of the plane. (Edge1 extends from point 0 to point 1, and edge 2 extends from point 1 to point 2.)

4. (Optional) Increase the number of **Surfaces** to create more than one surface at once.

   - Specify the **Spacing** between the planes. You can either directly enter the value, or you can use the plane tool to move the planes closer or further apart.

5. Click **Create**.

New surfaces are added to the **Surfaces** branch in the Outline View Tree.

Results → Surfaces

Refer to Grouping, Prioritizing, Editing, Renaming, and Deleting Surfaces (p. 3847) for information on modifying existing surfaces.

39.1.7.1. Using the Plane Tool

The plane tool enables you to interactively fine-tune the definition of a plane using graphics. Starting from an initial plane, you can translate, rotate, and resize the plane until its position, orientation, and size are as desired. For example, if you need to position a plane surface at a cross-section of an irregularly-shaped, curved duct, you can start with the plane tool near the desired location, resize it, translate it until it is within the duct walls, and rotate it to the proper orientation. You may find it helpful to display mesh faces to ensure that the plane tool is correctly positioned inside the domain.

Additionally, if you increase the number of surfaces being created to more than 1, the corresponding number of planes will appear in the graphics window (as shown in Figure 39.16: The Plane Tool for Creating Multiple Planes (p. 3836)).

The exact appearance of the plane tool depends on the **Method** that you select in Figure 39.12: The Plane Surface Dialog Box (p. 3831). The plane tool appears in the graphics window as soon as you open the **Plane Surface** dialog box.

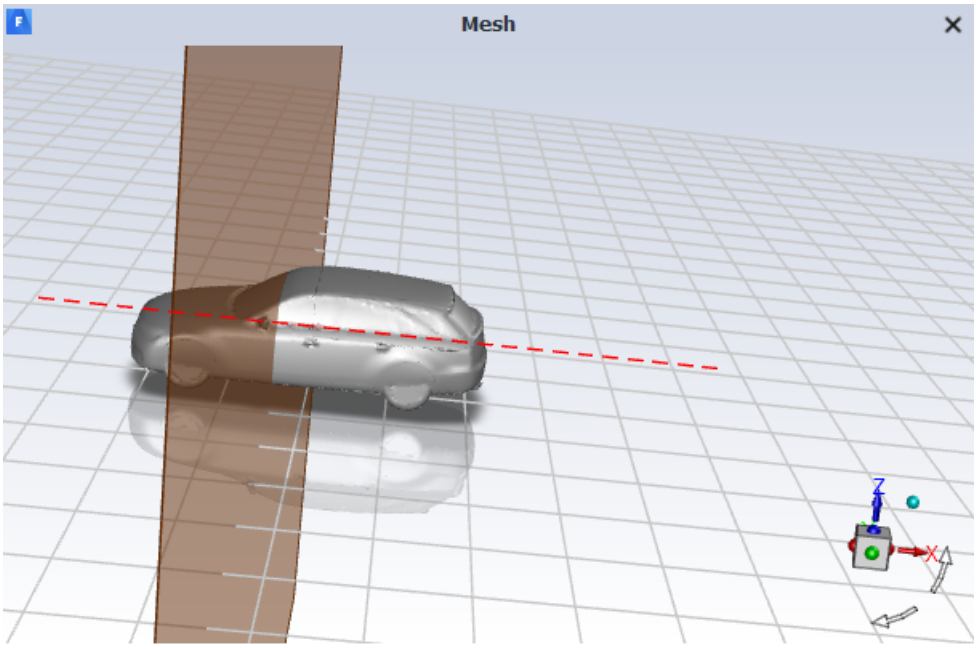
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The plane tool for the **YZ Plane**, **ZX Plane**, and **XY Plane** allows you to drag the plane along its perpendicular axis, as shown in Figure 39.13: The Plane Tool for the YZ Plane Method (p. 3834).

Note:

You can control the size of the plane tool triad using the **Graphics triad tool size** field in the **Graphics** branch of **Preferences** (**File**>**Preferences...**). *Note that the updated tool size only appears after restarting the Fluent application.*

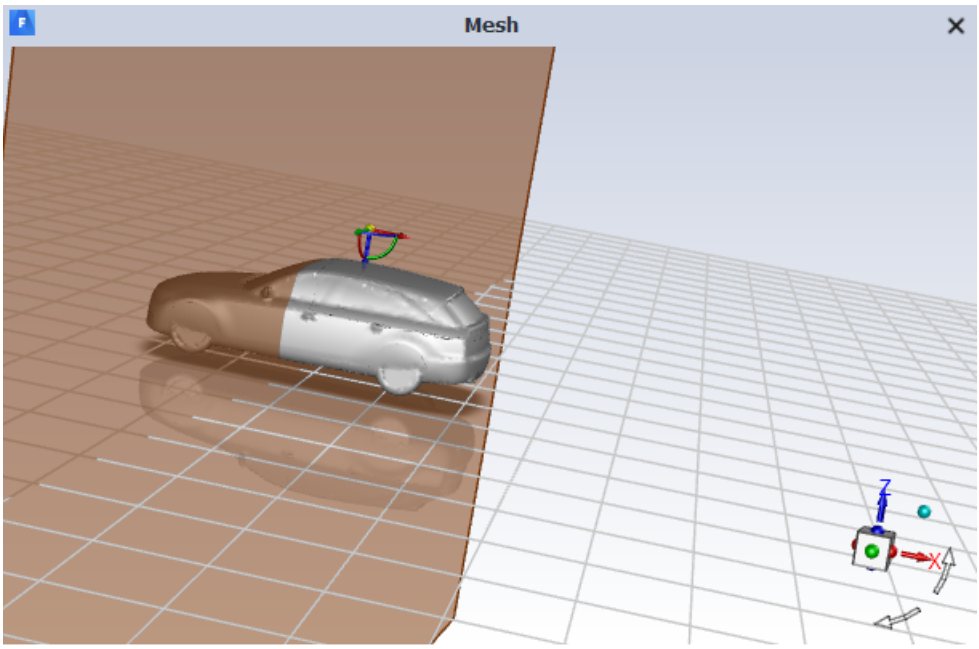
Figure 39.13: The Plane Tool for the YZ Plane Method



The plane tool for the **Point and Normal** method allows you to move the plane by manipulating the axis triad located in the middle of the plane, as shown in Figure 39.14: The Plane Tool for the Point and Normal Method (p. 3835). You can rotate on an axis, move the plane within a single axis, or "grab" the center of the axis and relocate the point (and entire plane) anywhere within the domain.

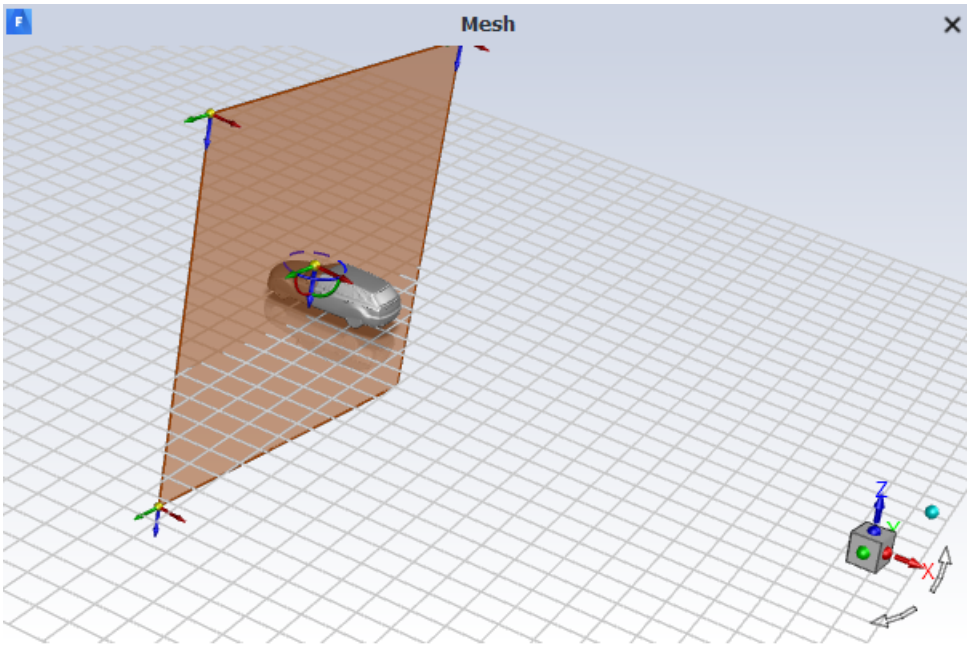
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 39.14: The Plane Tool for the Point and Normal Method



The plane tool for the **Three Points** method allows you to manipulate the location, orientation, and extents of the plane by manipulating the center axis triad and the axis triads located at three of the corners, as shown in Figure 39.15: The Plane Tool for the Three Points Method (p. 3835). Each of the axis triads can be moved along within a single plane, or moved freely by "grabbing" the center of the triad. In addition, the center axis triad can also be rotated, which moves the other three axis triads.

Figure 39.15: The Plane Tool for the Three Points Method

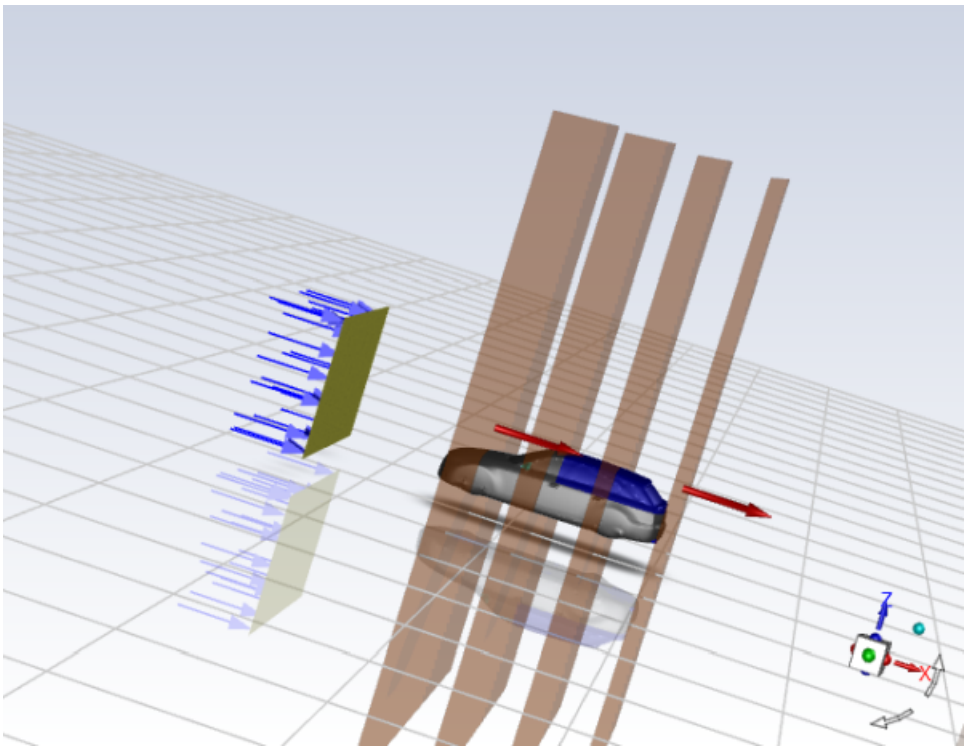


Increasing the number of **Surfaces** in the **Plane Surface** dialog box adds the corresponding number of planes to the graphics window. Using the tool, you can move the location of the first and last

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

planes, with the first plane moving the location of all the planes and the final plane changing the spacing between them.

Figure 39.16: The Plane Tool for Creating Multiple Planes



39.1.8. Quadric Surfaces

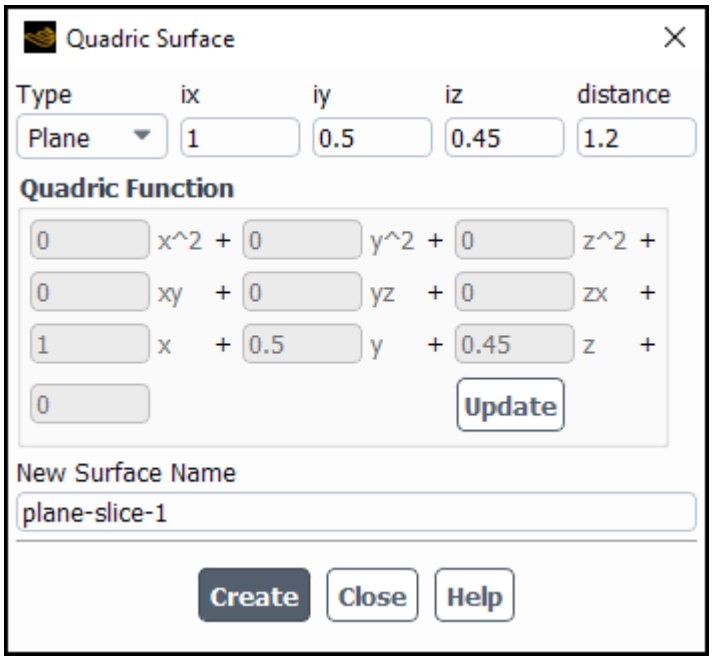
If you want to display data on a line (2D), plane (3D), circle (2D), sphere (3D), or general quadric surface, you can specify the surface by entering the coefficients of the quadric function that defines it. This feature provides you with an explicit method for defining surfaces. See Line and Rake Surfaces (p. 3826) and Plane Surfaces (p. 3830) for additional methods for creating line and plane surfaces.

To create a quadric surface, you will use the Quadric Surface Dialog Box (p. 5584) (Figure 39.17: The Quadric Surface Dialog Box (p. 3837)).

Domain → Surface → Create → Quadric...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 39.17: The Quadric Surface Dialog Box



The steps for creating the quadric surface are as follows:

1. Decide which type of quadric surface you want to create. In 3D, choose **Plane**, **Sphere**, or (general) **Quadric** in the **Type** drop-down list. In 2D, choose **Line**, **Circle**, or **Quadric**.

2. Specify the defining equation for the surface *in SI units*.

   - Line or plane surface: If you have selected **Line** (in 2D) or **Plane** (in 3D) as the surface type, the surface will consist of all points on the domain that satisfy the equation $ix*x+iy*y+iz*z=\text{distance}$. You will specify **ix** (the coefficient of x), **iy** (the coefficient of y), **iz** (the coefficient of z), and **distance** (the distance of the line or plane from the origin) in the fields to the right of the **Type** drop-down list. When you click **Update** under the **Quadric Function** heading, the display of the quadric function coefficients will change to reflect your inputs.

   - Circle or sphere surface: If you have selected **Circle** (in 2D) or **Sphere** (in 3D) as the surface type, the surface will consist of all points on the domain that satisfy the equation $(x-x0)^2+(y-y0)^2+(z-z0)^2=r^2$. You will specify **x0**, **y0**, **z0** (the x, y, and z coordinates of the sphere or circle's center) and **r** (the radius) in the fields to the right of the **Type** drop-down list. When you click **Update** under the **Quadric Function** heading, the display of the quadric function coefficients will change to reflect your inputs.

   - Quadric surface: If you have selected **Quadric** as the surface type, the surface will consist of all points in the domain that satisfy the general quadric function $Q=\text{value}$. You will specify the coefficients of the quadric function Q (the coefficients of the terms x^2, y^2, z^2, xy, yz, zx, x, y, z and the constant term) directly in the **Quadric Function** box, and you will set **value** to the right of the **Type** drop-down list. Note that the **Update** button will be disabled when you choose this type of surface.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. If you do not want to use the default name assigned to the surface, enter a new name under **New Surface Name**. The default name is the concatenation of the surface type and an integer that is the new surface ID (for example, `sphere-slice-7` or `quadric-slice-10`). If the **New Surface Name** you enter is the same as the name of a surface that already exists, Ansys Fluent will automatically assign the default name to the new surface when it is created.

   Important:

   The surface name that you enter must begin with an alphabetical letter. If the name of the surface begins with any other character or number, Ansys Fluent rejects the entry.

4. Click **Create** to create the new surface.

If you want to check that your new surface has been added to the list of all defined surfaces, or you want to delete or otherwise manipulate any surfaces, click **Manage...** to open the Surfaces Dialog Box (p. 5629). See Grouping, Prioritizing, Editing, Renaming, and Deleting Surfaces (p. 3847) for details.

39.1.9. Iso-surfaces

If you want to display results on cells that have a constant value for a specified variable, you will need to create an iso-surface of that variable. Generating an iso-surface based on x, y, or z coordinate, for example, will give you an x, y, or z cross-section of your domain; generating an iso-surface based on pressure will enable you to display data for another variable on a surface of constant pressure. You can create an iso-surface from an existing surface or from the entire domain. Furthermore, you can restrict any iso-surface to a specified cell zone. You also have the option of creating multiple iso-surfaces at once that are separated by a specified value.

Important:

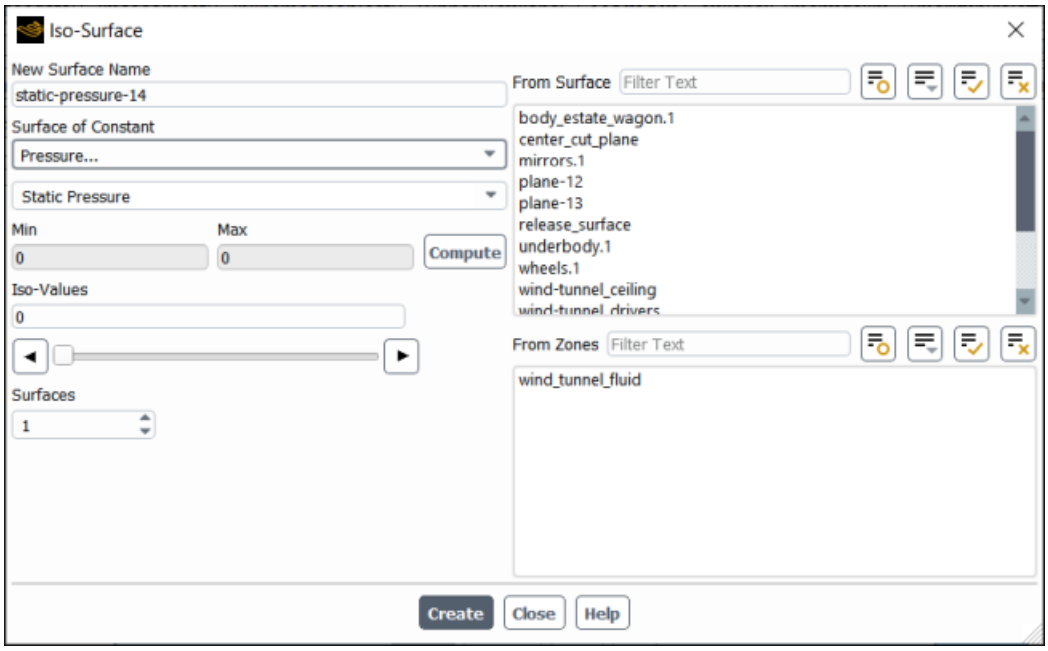
- Note that you cannot create an iso-surface until you have initialized the solution, performed calculations, or read a data file.
- While an iso-surface can be created and the domain patched after the case is initialized, the patched iso-surface cannot be displayed correctly until you advance the solution for at least one iteration (alternatively, you can reopen the **Iso-surface** dialog box and click **Save**).

To create an iso-surface, you will use the Iso-Surface Dialog Box (p. 5501) (Figure 39.18: The Iso-Surface Dialog Box (p. 3839)).

Domain → **Surface** → **Create** → **Iso-Surface...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 39.18: The Iso-Surface Dialog Box



The steps for creating one or more iso-surfaces are as follows:

1. Choose the scalar variable to be used for isosurfacing in the **Surface of Constant** drop-down list. First, select the desired category in the upper list. You can then select from related quantities from the lower list. (See Field Function Definitions (p. 4167) for an explanation of the variables in the list.)

2. If you want to create an iso-surface from an existing surface (that is, generate a new surface of constant x, y, temperature, pressure, and so on that is a subset of another surface), choose that surface in the **From Surface** list. You can specify the cell zone on which you want to create an iso-surface by selecting the zone in the **From Zones** list.

   - If you do not select a surface from the list, the isosurfacing will be performed on the entire domain.
   - If you do not select a zone from the list, then the isosurfacing will not be restricted to any cell zone and will run through the entire domain.

3. Click **Compute** to calculate the minimum and maximum values of the selected scalar field in the domain or on the selected surface (in the **From Surface** list). The minimum and maximum values will be displayed in the **Min** and **Max** fields.

4. Set the isovalue using one of the following methods. (Note that the second method will enable you to define multiple isovalues in a single iso-surface.)

   - You can set an iso-value interactively by moving the slider with the left mouse button. The value in the **Iso-Values** field will be updated automatically. This method will also create a

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

temporary iso-surface in the graphics window. Using the slider enables you to preview an iso-surface before creating it.

Important:

Even though the iso-surface is displayed, it is only a temporary surface. To create an iso-surface, use the **Create** button after deciding on a particular isovalue.

- You can type in isovalues in the **Iso-Values** field directly, separating multiple values by white space. Multiple isovalues will be contained in a single iso-surface; that is, you cannot select subsurfaces within the resulting iso-surface.

5. (Optional) Increase the number of **Surfaces** to create more than one surface at once.
   - Specify the **Spacing** between the iso-surfaces.

6. If you do not want to use the default name assigned to the surface, enter a new name under **New Surface Name**. The default name is the concatenation of the surface type and an integer that is the new surface ID (for example, `z-coordinate-6`). (If the **New Surface Name** you enter is the same as the name of a surface that already exists, Ansys Fluent will automatically assign the default name to the new surface when it is created.)

   Important:

   The surface name that you enter must begin with an alphabetical letter. If the name of the surface begins with any other character or number, Ansys Fluent rejects the entry.

   If you are creating multiple iso-surfaces, the listed name is appended with the iso-value incremented by the specified spacing.

7. Click **Create**. The new surface name will be added to the **From Surface** list in the dialog box.

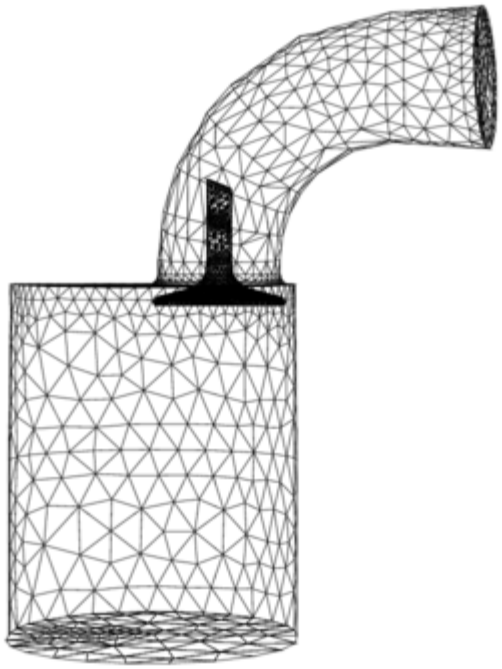
If you want to delete or otherwise manipulate any surfaces, click **Manage...** to open the Surfaces Dialog Box (p. 5629). See Grouping, Prioritizing, Editing, Renaming, and Deleting Surfaces (p. 3847) for details.

39.1.10. Clipping Surfaces

If you have created a surface, but you do not want to use the whole surface to display data, you can clip the surface between two isovalues to create a new surface that spans a specified subrange of a specified scalar quantity. The clipped surface consists of those points on the selected surface where the scalar field values are within the specified range. For example, in Figure 39.19: External Wall Surface Isoclipped to Values of x Coordinate (p. 3841) the external wall has been clipped to values of x coordinate less than 0 to show only the back half of the wall, enabling you to see the valve inside the intake port.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 39.19: External Wall Surface Isoclipped to Values of x Coordinate



Important:

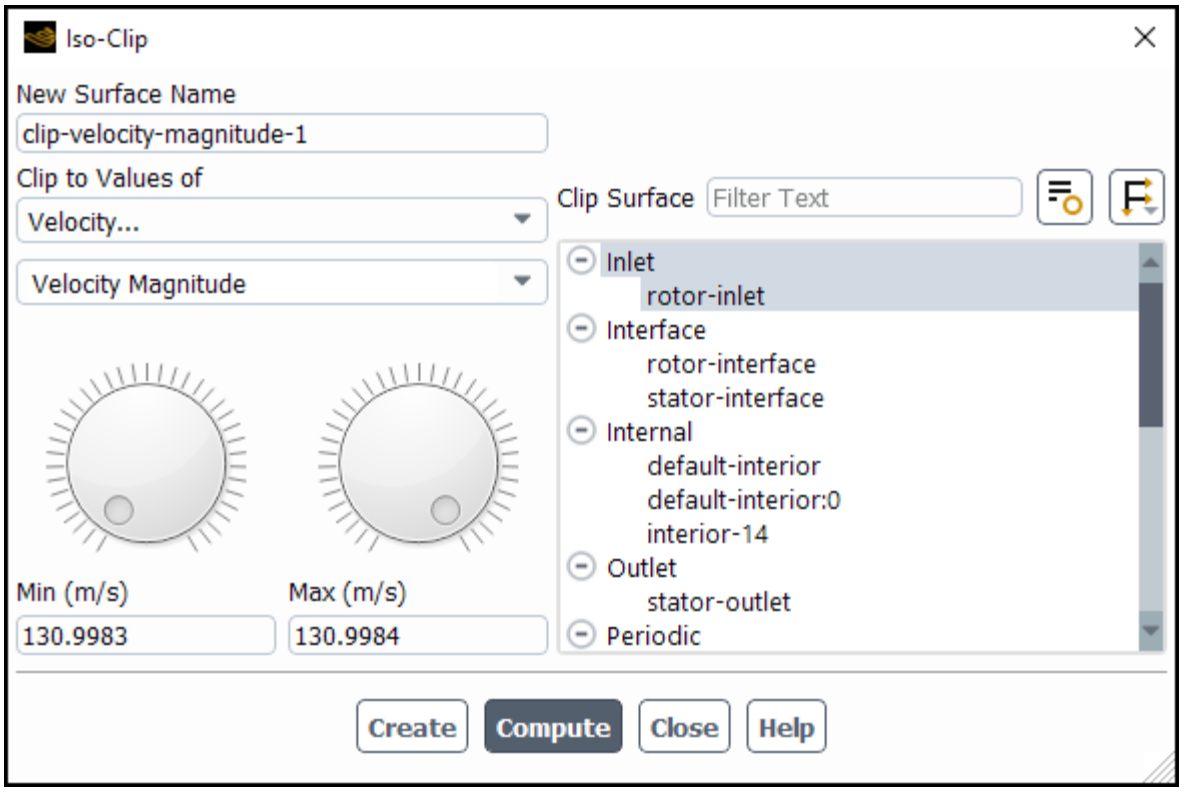
- Note that you cannot create an iso-clip until you have initialized the solution, performed calculations, or read a data file.
- While an iso-clip can be created and the domain patched after the case is initialized, the patched iso-clip cannot be displayed correctly until you advance the solution for at least one iteration (alternatively, you can reopen the **Iso-Clip** dialog box and click **Save**).

To clip an existing surface, you will use the Iso-Clip Dialog Box (p. 5500) (Figure 39.20: The Iso-Clip Dialog Box (p. 3842)).

Domain → Surface → Create → Iso-Clip...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 39.20: The Iso-Clip Dialog Box



The steps for clipping a surface are as follows:

1. Choose the scalar variable on which the clipping will be based in the **Clip To Values Of** drop-down list. First, select the desired category in the upper list. You can then select from related quantities from the lower list. See Field Function Definitions (p. 4167) for an explanation of the variables in the list.

2. Select the surface to be clipped in the **Clip Surface** list.

3. Click **Compute** to calculate the minimum and maximum values of the selected scalar field on the selected surface. The minimum and maximum values are displayed in the **Min** and **Max** fields.

4. Define the clipping range using one of the following methods.

   - You can set the upper and lower limits of the clipping range interactively by moving the indicator in each dial (that is, the dial above the **Min** or **Max** field) with the left mouse button. The value in the corresponding **Min** or **Max** field is updated automatically. This method will also create a temporary surface in the graphics window. Using the dials enables you to preview a clipped surface before creating it.

     Important:

     Even though the clipped surface is displayed, it is only a temporary surface. To create the new surface, use the **Clip** button after deciding on the clipping range.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- You can type the minimum and maximum values in the clipping range directly in the **Min** and **Max** fields.

5. If you do not want to use the default name assigned to the surface, enter a new name under **New Surface Name**. The default name is the concatenation of the surface type and an integer that is the new surface ID (for example, `clip-density-8`). (If the **New Surface Name** you enter is the same as the name of a surface that already exists, Ansys Fluent will automatically assign the default name to the new surface when it is created.)

   Important:

   The surface name that you enter must begin with an alphabetical letter. If the name of the surface begins with any other character or number, Ansys Fluent rejects the entry.

6. Click **Clip**. The new surface name is added to the **Clip Surface** list in the dialog box. (The original surface will remain unchanged.)

If you want to delete or otherwise manipulate any surfaces, click **Manage...** to open the Surfaces Dialog Box (p. 5629). See Grouping, Prioritizing, Editing, Renaming, and Deleting Surfaces (p. 3847) for details.

39.1.11. Transforming Surfaces

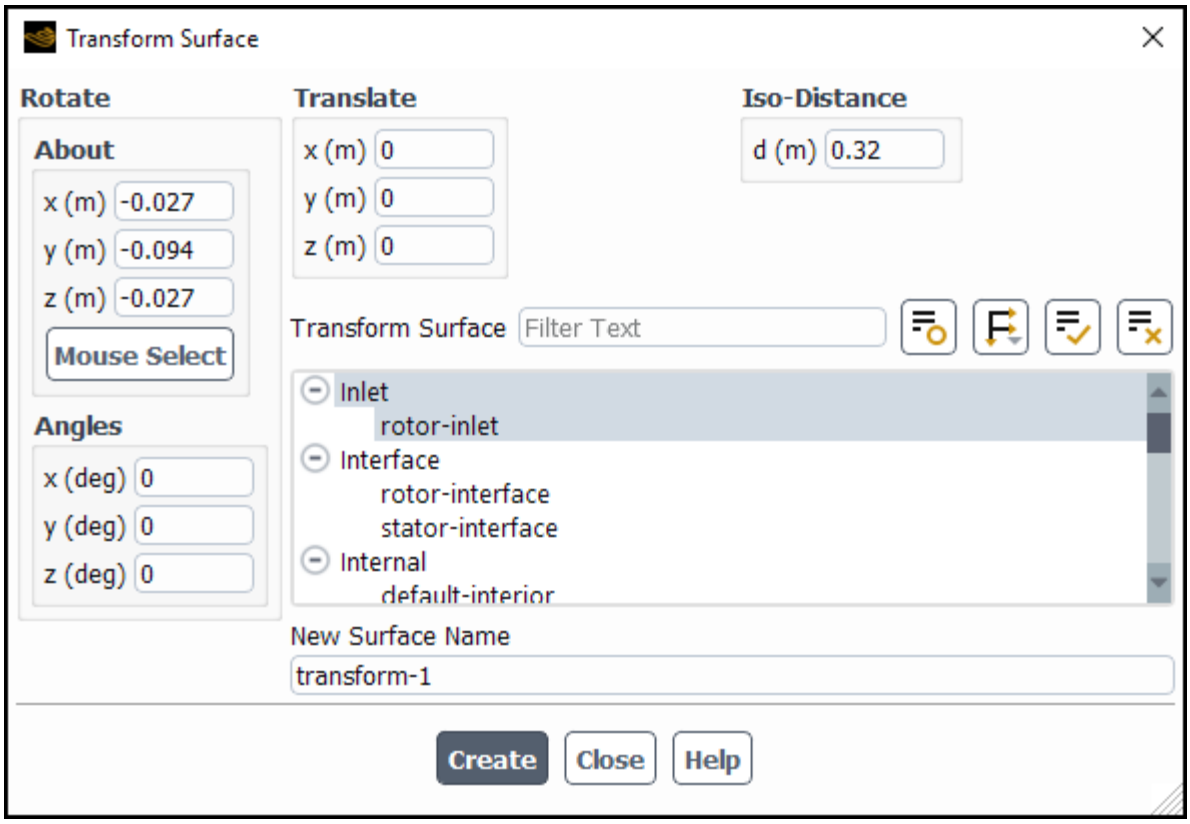
You can create a new data surface from an existing surface by rotating and/or translating the original surface. For example, you can rotate the surface of a complicated turbomachinery blade to plot data in the region between blades. You can also create a new surface at a constant normal distance from the original surface.

To transform an existing surface to create a new one, you will use the Transform Surface Dialog Box (p. 5633) (Figure 39.21: The Transform Surface Dialog Box (p. 3844)).

Domain → **Surface** → **Create** → **Transform...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 39.21: The Transform Surface Dialog Box



The steps for transforming a surface are as follows:

1. Select the surface to be transformed in the **Transform Surface** list.

   Important:

   The transform operation is not available for user-created surfaces such as lines, points, iso-surfaces, and so on. To create a transformed line, point, iso-surface, or other user-created surface, you must manually translate the input point(s) and create a new line/point/iso-surface in the respective dialog box (**Line/Rake**, **Point Surface**, **Iso-Surface**, and so on).

2. Set the appropriate transformation parameters, as described below. You can perform any combination of translation, rotation, and "isodistancing" on the surface.

   - Rotation: To rotate a surface, specify the origin about which the surface is rotated, and the angle by which the surface is rotated.

     In the **About** box under **Rotate**, specify a point, and the origin of the coordinate system for the rotation will be set to that point. (The x, y, and z directions will be the same as for the global coordinate system.) For example, if you specified the point (1,5,3) in 3D, rotation would be about the x, y, and z axes anchored at (1,5,3). You can either enter the point's coordinates in the **x**, **y**, and **z** fields or click **Mouse Select** and select a point in the graphics window using

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the mouse-probe button. See Controlling the Mouse Button Functions (p. 4020) for information about mouse button functions.

In the **Angles** box under **Rotate**, specify the angles about the x, y, and z axes (that is, the axes of the coordinate system with the origin defined under **About**) by which the surface is rotated. For 2D problems, you can specify rotation about the z axis only.

- Translation: To translate a surface, define the distance by which the surface is translated in each direction. Set the **x**, **y**, and **z** translation distances under **Translate**.
- Isodistancing: To create a surface positioned at a constant normal distance around the original surface, set the value for **d** under **Iso-Distance**.

3. If you do not want to use the default name assigned to the surface, enter a new name under **New Surface Name**. The default name is the concatenation of the surface type and an integer that is the new surface ID (for example, `transform-9`). If the **New Surface Name** you enter is the same as the name of a surface that already exists, Ansys Fluent will automatically assign the default name to the new surface when it is created.

   Important:

   The surface name that you enter must begin with an alphabetical letter. If the name of the surface begins with any other character or number, Ansys Fluent rejects the entry.

4. Click **Create**. The new surface name is added to the **Transform Surface** list in the dialog box. The original surface will remain unchanged.

If you want to delete or otherwise manipulate any surfaces, click **Manage...** to open the Surfaces Dialog Box (p. 5629). See Grouping, Prioritizing, Editing, Renaming, and Deleting Surfaces (p. 3847) for details.

39.1.12. Expression Volumes

Expression volumes allow you to visualize a subset of the domain constrained according to your provided criteria. They can also be used to define a surface or volume that can be used in computing surface and volume reports and report definitions. *Expression volumes can behave as either a surface or a volume, depending on the context. For example, an expression volume used in a contour plot is represented as a surface. That same volume used in a volume report definition is treated as a volume.*

There are two primary methods for creating volume surfaces:

- Using the relevant cell register dialog box (Using Cell Registers (p. 3851))

  Note:

  Expression volumes created using the **Create Volume Surface** option in a cell register dialog box are equivalent to those created by opening Figure 39.22: The Expression Volume Dialog Box (p. 3846), selecting a cell register from the drop-down list and clicking

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Create. *You can confirm this by reviewing the definition of a volume surface created using the **Create Volume Surface** option (listed under **Surfaces** in the **Outline View** tree).*

- Using the Expression Volumes dialog box (Fluent Expressions Language (p. 1037))

To create a volume surface:

1. Read a data file or initialize the solution.

   File → Read → Data...

   or

   Solution → Initialization → Initialize

2. (Cell register-based volume) Create a cell register that defines the desired volume:

   a. Right-click **Cell Registers** in the **Outline View** tree and create a new cell register of the desired type.

   Solution → Cell Registers → New → <cell register type...>

   b. Enable the **Create Volume Surface** option prior to saving the register.

3. Open the **Expression Volume** dialog box.

   Results → Surfaces → New → Expression-Volume...

   Figure 39.22: The Expression Volume Dialog Box

4. Enter the desired expression definition. *For example,* `StaticPressure<200[Pa]`.

5. Click **Create** to create the volume.

6. You can display volume you just created by right-clicking it in the **Outline View** tree and selecting **Display**. It is also available for selection in the **Surfaces** list of various postprocessing graphics objects, such as mesh, contour and vector plots.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Results → Surfaces → <volume name> **Display**

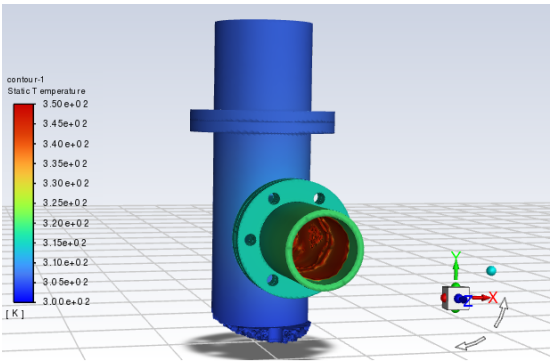
Important:

Visualizing a volume surface with the same variable used to mark the cells for creating the surface may result in values being displayed that are outside of the range used to create the surface.

For example, an expression volume for temperature iso-values in the range of 305K to 345K can result in the image as shown in Figure 39.23: Example Temperature Contour on Volume Surface (p. 3847). *The temperature contour plot shows a range of 300K to 350K.*

There are two separate but related entities involved in this example. There is the expression volume, which is created using cell-center values. Then, there is the expression volume surface, which is made available for visualization purposes. The values on this surface are interpolated from the cell-center values, leading to the differences seen in the example.

Figure 39.23: Example Temperature Contour on Volume Surface

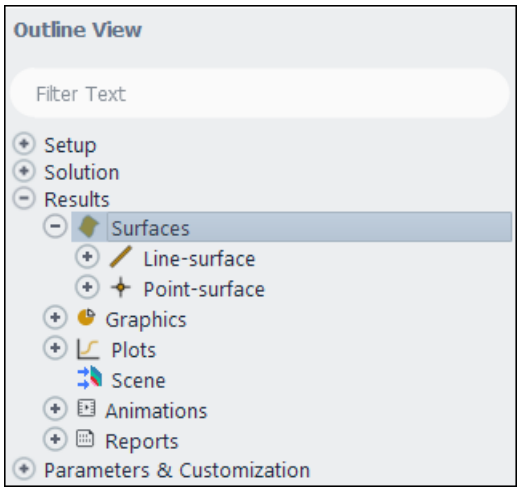


39.1.13. Grouping, Prioritizing, Editing, Renaming, and Deleting Surfaces

Once you have created a number of surfaces, you can edit, rename, delete, and group surfaces and obtain information about their components. Grouping surfaces is useful if you want to perform postprocessing on a number of surfaces at a time. For example, you may want to group several wall surfaces together to generate a contour plot of temperature on all walls. To postprocess results on each wall surface individually, "ungroup" the surfaces.

You can manipulate existing surfaces and create new ones through the **Surfaces** branch of the **Outline View**.

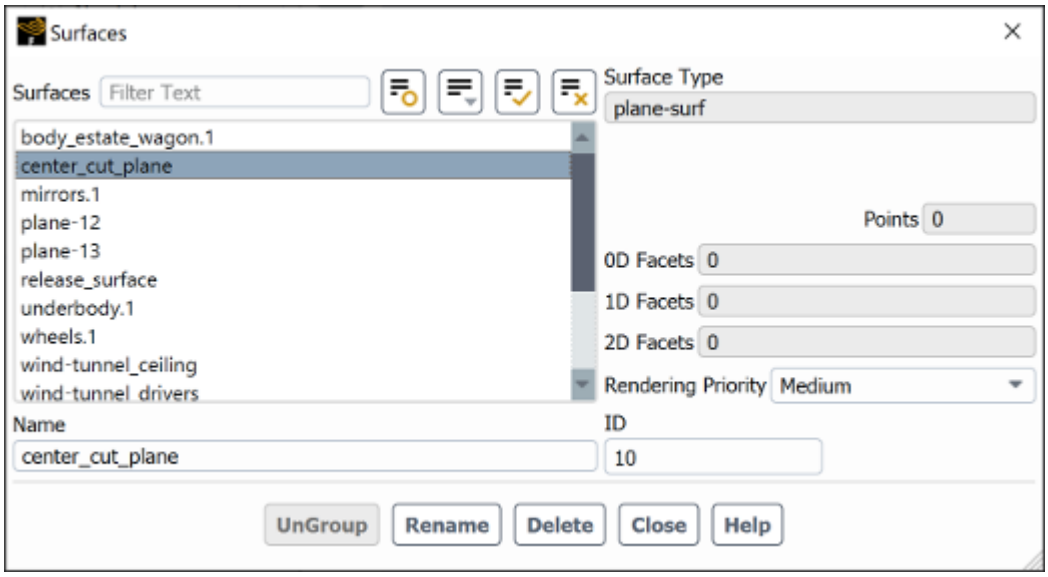
Results → Surfaces



Additionally, you can manipulate existing surfaces within the Surfaces Dialog Box (p. 5629) (Figure 39.24: The Surfaces Dialog Box (p. 3848)).

Domain → Surface → Manage...

Figure 39.24: The Surfaces Dialog Box



You can also open this dialog box by clicking **Manage...** in one of the surface creation dialog boxes described in the previous sections or by double-clicking the **Surfaces** branch in the **Outline View**.

For additional information, see the following sections:

- 39.1.13.1. Grouping Surfaces
- 39.1.13.2. Editing and Renaming Surfaces
- 39.1.13.3. Setting Surface Rendering Priority
- 39.1.13.4. Deleting Surfaces
- 39.1.13.5. Surface Statistics

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

39.1.13.1. Grouping Surfaces

As mentioned above, you may want to group several surfaces together in order to perform post-processing on all of them at once. To create a surface group, select the surfaces to be grouped in the **Surfaces** list. You can define a new name for the group in the **Name** field, or you can use the default name, which is the name of the first surface you selected in the **Surfaces** list. Then click **Group**. The selected surfaces disappear from the **Surfaces** list, and the name of the surface group is added to the list.

Important:

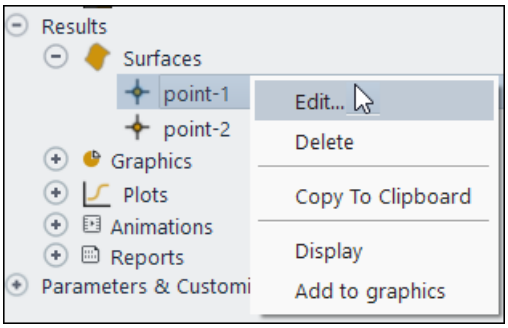
The **Group** button will not appear until you have selected at least two surfaces. As soon as you choose a second surface in the **Surfaces** list, the **Rename** button will change to the **Group** button.

To ungroup the surfaces, simply select the surface group in the **Surfaces** list and click **UnGroup**. The group name will disappear from the list and the names of the original surfaces in the group will reappear in the list.

39.1.13.2. Editing and Renaming Surfaces

You can edit surfaces either by double-clicking the surface in the **Surfaces** branch of the **Outline View** or by right-clicking it and selecting **Edit...**. *This opens the respective surface dialog box where you can modify settings and rename the surface.*

Results → Surfaces **<surface-name> → Edit...**



To change the name of an existing surface, select the surface in the **Surfaces** list of the **Surfaces** dialog box, enter a new name in the **Name** field, and then click **Rename**. The new name replaces the old name in the **Surfaces** list and the surface is otherwise unchanged.

If you select more than one surface, the **Rename** button will not appear in the dialog box. When more than one surface is selected, the **Rename** button is replaced by the **Group** button.

Important:

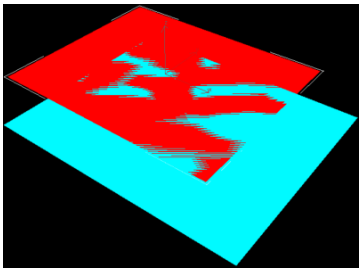
The surface name that you enter must begin with an alphabetical letter. If the name of the surface begins with any other character or number, Ansys Fluent rejects the entry.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

39.1.13.3. Setting Surface Rendering Priority

In some instances you may observe a graphical anomaly when viewing one or more postprocessing objects such as mesh or contour plots in the graphics window. It could appear as a "checkerboard" like effect or as missing pixels, or similar. This type of display issue is often referred to as "z-fighting", "depth buffer", or "plane fighting" and it occurs when Fluent is trying to display multiple things at two or more coincident surfaces, such as shown in Figure 39.25: Example of Z-fighting (p. 3850). *Note that mesh objects take a lower priority to all other graphics objects.*

Figure 39.25: Example of Z-fighting



To specify a surface rendering priority:

1. Select one or more **Surfaces**.
2. Select the desired priority from the **Rendering Priority** drop-down list.

   The updated priority is saved as soon as you complete the selection.

Note:

Surface rendering priority is not applied to the global mesh display (which is accessed via the mesh display toolbar button ; it is also used for the default mesh display if **Display Mesh After Reading** is enabled when you read your case).

39.1.13.4. Deleting Surfaces

If you find that a surface is no longer useful, you may want to delete it to prevent the list of surfaces from becoming too cluttered. Select the surface or surfaces to be deleted in the **Surfaces** list, and then click **Delete**. The delete operation is not reversible, so if you want to get a deleted surface back again you will need to recreate it using one of the surface-creation dialog boxes described in the previous sections.

39.1.13.5. Surface Statistics

You can also use the Surfaces Dialog Box (p. 5629) to retrieve topological information about surfaces. **Points** is the total number of nodes in a surface. **0D Facets** is the number of isolated nodes in a surface (that is, nodes that have no connectivity, such as point surfaces or nodes in a rake), **1D Facets** is the number of linear faces (consisting of two connected nodes) in a surface in a 2D problem, and **2D Facets** is the number of 2D faces (triangular or quadrilateral) in a surface in a 3D

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

problem. Note that an **interior** zone surface in a 3D problem consists of 2D facets, and similarly an **interior** zone surface in a 2D problem consists of 1D facets.

These statistics are listed for the surface(s) selected in the **Surfaces** list. If more than one surface is selected, the sum over all selected surfaces is displayed for each quantity.

Note that if you want to check these statistics for a surface that was read from a case file, you will need to first display it.

39.2. Using Cell Registers

Cell registers are a means for marking cells based on certain criteria, such as iso-values or proximity to a boundary. You can then use these marked cell registers to perform operations like refining or coarsening the mesh (Adapting the Mesh (p. 3773)), patching the solution (Patching Values in Selected Cells (p. 3630)), or deleting cells (Deleting Cells (p. 1298)). You can also create a volume from the cell register that you can then use for visualization in applications such as postprocessing (Expression Volumes (p. 3845)).

The following sections describe how to create and use cell registers:

39.2.1. Region
39.2.2. Boundary
39.2.3. Variable Limiter
39.2.4. Field Variable
39.2.5. Residuals
39.2.6. Volume
39.2.7. Yplus/Ystar
39.2.8. Manage Cell Registers
39.2.9. Cell Register Operations
39.2.10. Copying and Renaming Cell Registers

Note:

- If you have multiple cell registers that you want to display within the same graphics window, you can enable the **Overlays** option in the **Scene Description** dialog box (accessed by clicking **Compose...** in the **View** ribbon tab). With this option enabled, each subsequent cell register you display is shown on top of the existing graphics window display.
- (Parallel only) The exact number of cells in a register may vary between sessions running on different core counts for the same case. However, any cell count variations are not to a degree that they could impact the results.
- You can only define a maximum of 16 cell registers.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

39.2.1. Region

Region registers allow you to mark cells inside or outside of a given region that you can specify using text or mouse inputs. Using text input, you can specify the region as a hexahedron (quadrilateral in 2D), a sphere (circle in 2D), or a cylinder.

39.2.1.1. Defining a Region

The basic approach to creating a region cell register is to first define a region:

- The hexahedron (quadrilateral) is defined by entering the coordinates of two points defining the diagonal.
- The sphere (circle) is defined by entering the coordinates of the center of the sphere and its radius.
- To define a cylinder, specify the coordinates of the points defining the cylinder axis, and the radius. In 3D this will define a cylinder. In 2D, you will have an arbitrarily oriented rectangle with length equal to the cylinder axis length and width equal to the radius.

  A rectangle defined using the cylinder option differs from one defined with the quadrilateral option in that the former can be arbitrarily oriented in the domain, while the latter must be aligned with the coordinate axes.

  You can either enter the exact coordinates into the appropriate real entry fields or select the locations with the mouse on displays of the mesh or solution field. After the region is defined, each cell that has a centroid inside/outside the specified region is marked.

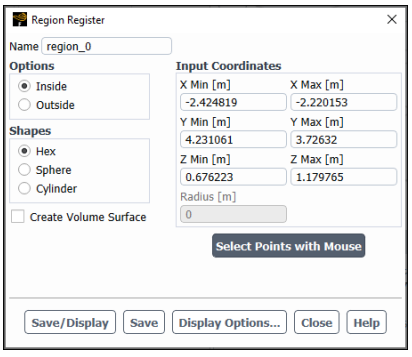
39.2.1.2. Setting Up a Region Cell Register

To create a region cell register:

1. Open the Figure 39.26: Region Register Dialog Box (p. 3852).

   Solution → Cell Registers → New → Region...

   Figure 39.26: Region Register Dialog Box



2. Specify whether the register includes cells **Inside** or **Outside** of the defined region.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you choose **Inside**, cells with centroids within the specified region will be marked.
- If you choose **Outside**, cells with centroids outside the specified region will be marked.

3. Choose the shape for the region register.

   In 2D, you may specify a **Quad** (that is, a quadrilateral), **Circle**, or **Cylinder** by making a selection from the **Shapes** group box. In 3D, your options include **Hex** (that is, a hexahedron), **Sphere**, or **Cylinder**.

4. Define the region by entering values into the dialog box or by using the mouse.

   In the dialog box, the inputs are as follows:

   - To define a hexahedron or quadrilateral, enter the coordinates of two points defining the diagonal of the box.

     For a hexahedron, define **X Min**, **Y Min**, and **Z Min**, as well as **X Max**, **Y Max**, and **Z Max**. For a quadrilateral, define **X Min** and **Y Min**, as well as **X Max** and **Y Max**.

   - To define a sphere or circle, enter the values for the **Radius** and the coordinates of its center: **X Center**, **Y Center**, and **Z Center** for a sphere, or **X Center** and **Y Center** for a circle.

   - To define a cylinder, enter the value for the **Radius** and the minimum and maximum coordinates defining the cylinder axis: **X-Axis Min**, **Y-Axis Min**, and **Z-Axis Min**, as well as **X-Axis Max**, **Y-Axis Max**, and **Z-Axis Max** for 3D, or **X-Axis Min** and **Y-Axis Min**, as well as **X-Axis Max** and **Y-Axis Max** for 2D. In 2D, this will be the width of the resulting rectangle.

   To define the region using the mouse, click the **Select Points with Mouse** button. Using the right mouse button, select the input coordinates from a display of the mesh or solution field. After selecting the points, the values will be loaded automatically into the appropriate fields in the dialog box. See Controlling the Mouse Button Functions (p. 4020) for details about mouse button functions.

   You have the option of editing these values before marking .

   - To define a hexahedron or quadrilateral, select the two points of the diagonal in any order.
   - To define a sphere or circle, first select the location of the centroid and then select a point that lies on the sphere/circle (that is, a point that is one radius away from the centroid).
   - To define a cylinder, first select the two points that define the cylinder axis and then select a point that is one radius away from the axis.

5. (Optional) Enable **Create Volume Surface** to create a volume "surface" that can be selected for visualization in graphics objects such as mesh and contour plots.

6. Click **Save** to create the cell register.

Note:

You can change the way the register is displayed by using the Cell Register Display Options Dialog Box (p. 5407) dialog box that opens by clicking **Display Options...**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

39.2.2. Boundary

Boundary registers allow you to mark cells within the proximity of a boundary. The proximity to the boundary can be based on:

- cell distance—the distance of a cell from the boundary is measured by the number of cells.
- volume distance—the distance is based on a target boundary volume and growth factor.

  Cells are marked based on the following equation:

$$V_n > V_{boundary} e^{\alpha d} \tag{39.1}$$

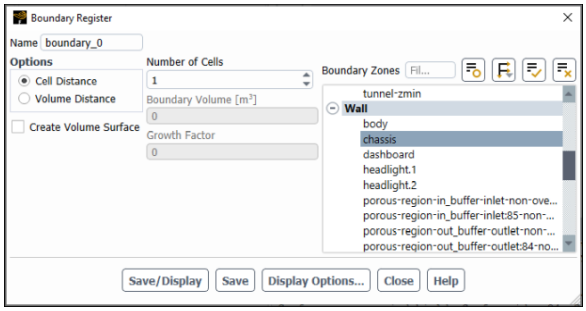
  where V_n is the cell volume, $V_{boundary}$ is the specified boundary volume (**Boundary Volume**), α is the exponential growth factor (**Growth Factor**), and d is the normal distance of the cell centroid from the selected boundaries. $V_{boundary} e^{\alpha d}$ is the target volume for a cell.

To create a boundary cell register:

1. Open the Figure 39.27: Boundary Register Dialog Box (p. 3854).

   Solution → **Cell Registers** → **New** → **Boundary...**

   Figure 39.27: Boundary Register Dialog Box



2. Choose whether the boundary register is based on **Cell Distance** or **Volume Distance**.
3. Specify the **Number of Cells** or **Boundary Volume** and **Growth Factor**, depending on which distance method you selected in the previous step.
4. Select the boundary zone(s) for the cell register.
5. (Optional) Enable **Create Volume Surface** to create a volume "surface" that can be selected for visualization in graphics objects such as mesh and contour plots.
6. Click **Save** to create the cell register.

Note:

You can change the way the register is displayed by using the Cell Register Display Options Dialog Box (p. 5407) dialog box that opens by clicking **Display Options...**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

39.2.3. Variable Limiter

Limit registers allow you to mark the cells that have reached a solution limit while solving. These limits can be viewed and modified in Figure 36.13: The Solution Limits Dialog Box (p. 3622). For additional information on solution limits, see Setting Solution Limits (p. 3621).

To create a limit cell register:

1. Open the Figure 39.28: Limit Register Dialog Box (p. 3855).

   Solution → Cell Registers → New → Variable Limiter...

   Figure 39.28: Limit Register Dialog Box

2. Select the variable for creating the cell register.

3. Click **Save** to create the cell register.

Note:

You can change the way the register is displayed by using the Cell Register Display Options Dialog Box (p. 5407) dialog box that opens by clicking **Display Options...**.

39.2.4. Field Variable

Field variable registers allow you to mark cells based on the value of a field variable. Additionally, you can specify if these field variable values are derived, and if so, whether it is via gradient, curvature, or Hessian-based error indicator. You can also scale the values by the zonal or global average or maximum. You can mark cells based on the minimum/maximum value of a field variable, with values below or above a certain field variable value, within a certain percentage of the minimum/maximum value of a field variable, inside or outside of a certain range of field variable values, inside or outside of a certain range defined in terms of the standard deviation, or with values below or above a limit defined in terms of the standard deviation.

39.2.4.1. Approaches For Deriving Field Values

There are four approaches for using the available physical features of the flow field to mark cells available in Ansys Fluent:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Gradient approach**: In this approach, Ansys Fluent multiplies the Euclidean norm of the gradient of the selected solution variable by a characteristic length scale [36] (p. 5715). For example, the gradient function in two dimensions has the following form:

$$|e_{i1}| = \left(A_{cell}\right)^{\frac{r}{2}} |\nabla f| \tag{39.2}$$

  where e_{i1} is the error indicator, A_{cell} is the cell area, r is the gradient volume weight, and ∇f is the Euclidean norm of the gradient of the desired field variable, f.

  The default value of the gradient volume weight is unity, which corresponds to full volume weighting. A value of zero will eliminate the volume weighting, and values between 0 and 1 will use proportional weighting of the volume.

  This approach is recommended for problems with strong shocks, for example, supersonic inviscid flows.

- **Curvature approach**: This is an equidistribution marking technique that multiplies the undivided Laplacian of the selected solution variable by a characteristic length scale [168] (p. 5722).

  For example, the gradient function in two dimensions has the following form:

$$|e_{i2}| = \left(A_{cell}\right)^{\frac{r}{2}} |\nabla^2 f| \tag{39.3}$$

  where e_{i2} is the error indicator, A_{cell} is the cell area, r is the gradient volume weight, and $\nabla^2 f$ is the undivided Laplacian of the desired field variable (f).

  The default value of the gradient volume weight is unity, which corresponds to full volume weighting. A value of zero will eliminate the volume weighting, and values between 0 and 1 will use proportional weighting of the volume.

  This approach is recommended for problems with smooth solutions.

- **Hessian approach**: This calculates a Hessian-based error indicator for a field variable, which can be useful when creating a cell register to use as part of mesh adaption for an external aerodynamic simulation (as described in Aerodynamics Adaption (p. 3787)). The indicator is based on the relationship between the second derivatives of the solution and the local interpolation error expanded around a point [59] (p. 5716).

$$e(x) = |u(x) - u_h(x)| \approx |x^T H x|, \qquad H = \begin{bmatrix} \partial_{xx} u & \partial_{xy} u \\ \partial_{xy} u & \partial_{yy} u \end{bmatrix}, \tag{39.4}$$

  where $u(x)$ is the exact solution, $u_h(x)$ is the approximate solution and H is the Hessian matrix composed of second derivatives. This key idea has seen widespread use due to the capability of detecting a variety of flow features at various scales and strengths [7] (p. 5713), [8] (p. 5713), [126] (p. 5720). This allows adaption to effectively target accuracy improvements at reduced computational cost.

- **Isovalue approach**: This approach is not based on derivatives. Instead, the isovalues of the required field variable f, are used to control the marking. Therefore, the function is of the form:

$$e_{i3} = f \tag{39.5}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where e_{i3} is the error indicator. This approach is recommended for problems where derivatives are not helpful. For example, if you want to mark the mesh where the reaction is taking place, you can use the isovalues of the reaction rate and mark at high reaction rates. This approach also allows you to customize the criteria for controlling the marking using custom field functions, user-defined scalars, and so on

Any of the field variables available for contours display can be used in the marking cell gradients. These scalar functions include both geometric and physical features of the numerical solution. Therefore, in addition to traditional marking based on physical features, such as the velocity, you may choose to mark to the cell volume field to reduce rapid variations in cell volume.

In addition to the standard (no normalization) approach available in Ansys Fluent, four options are available for normalization [49] (p. 5715):

- **Scale by Global Average**, scales the values of e_{i1}, e_{i2}, or e_{i3} by their average value in the domain, that is:

$$\frac{|e_i|}{\overline{|e_i|}} \tag{39.6}$$

- **Scale by Zone Average** scales the values of e_{i1}, e_{i2}, or e_{i3} by their average value in each zone.

- **Scale by Global Maximum**, scales the values of e_{i1}, e_{i2}, or e_{i3} by their maximum value in the domain, therefore always returning a problem-independent range of [0, 1] for any variable used for marking, that is:

$$\frac{|e_i|}{\max|e_i|} \tag{39.7}$$

- **Scale by Zone Maximum**, scales the values of e_{i1}, e_{i2}, or e_{i3} by their maximum value in each zone, therefore always returning a problem-independent range of [0, 1] for any variable used for marking.

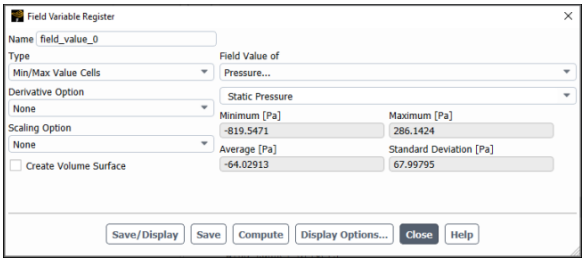
39.2.4.2. Setting Up a Field Variable Cell Register

To create a field variable cell register:

1. Open the Figure 39.29: Field Variable Register Dialog Box (p. 3857).

   Solution → Cell Registers → New → Field Variable...

   Figure 39.29: Field Variable Register Dialog Box



2. Enter an appropriate **Name** for the cell register.
3. Specify how the field variable values are derived in the **Derivative Option** drop-down. You can choose from:
   - **None**
   - **Gradient**
   - **Curvature**
   - **Hessian**

   Note:

   When using the **Hessian** derivative option, it is recommended that you also select **Scale by Global Average** from the **Scaling Option** drop-down list, as they are relative values that are scaled based on the local mesh size.

4. Specify how the field variable values are scaled in the **Scaling Option** drop-down. You can choose from:
   - **None**
   - **Scale by Global Average**
   - **Scale by Zone Average**
   - **Scale by Global Maximum**
   - **Scale by Zone Maximum**
5. Select the field variable for creating the cell register from the **Field Values of** drop-down lists. *The name of this field will change based on the selections you make for derivative and scaling options.*
6. Choose the **Type** of field variable register and define the related fields. Note that you can use **Compute** to calculate the minimum value, maximum value, average value, and standard deviation from the average value of the selected field variable, as well as any resulting range minimum, range maximum, and limit. Your options include:
   - **Min/Max Value Cells**

     Cells containing the minimum and maximum values of the specified field variable will be marked.
   - **Cells Less Than**

     Cells with values that are less than the specified threshold value will be marked.
   - **Cells More Than**

     Cells with values that are greater than the specified threshold value will be marked.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Top Value Cells**

  Cells with values within the top 'n' percent will be marked.

- **Low Value Cells**

  Cells with values within the lowest 'n' percent will be marked.

- **Cells Inside Range**

  Cells with isovalues / gradient values / curvature values / Hessian values between your specified minimum and maximum will be marked.

- **Cells Outside Range**

  Cells with isovalues / gradient values / curvature values / Hessian values outside of your specified minimum and maximum will be marked.

- **Cells Inside Standard Deviation Range**

  Cells with values within a range defined in terms of the standard deviation will be marked. You define the **Lower Limit Factor** and **Upper Limit Factor**. These factors are then multiplied with the computed **Standard Deviation**, and then that product is added to the computed **Average** to define the **Range Minimum** and **Range Maximum**, respectively.

- **Cells Outside Standard Deviation Range**

  Cells with values outside of a range defined in terms of the standard deviation will be marked. You define the **Lower Limit Factor** and **Upper Limit Factor**. These factors are then multiplied with the computed **Standard Deviation**, and then that product is added to the computed **Average** to define the **Range Minimum** and **Range Maximum**, respectively.

- **Cells Less Than Standard Deviation Limit**

  Cells with values less than a limit defined in terms of the standard deviation will be marked. You define the **Limit Factor**. This factor is then multiplied with the computed **Standard Deviation**, and then that product is added to the computed **Average** to define the **Limit**.

- **Cells More Than Standard Deviation Limit**

  Cells with values greater than a limit defined in terms of the standard deviation will be marked. You define the **Limit Factor**. This factor is then multiplied with the computed **Standard Deviation**, and then that product is added to the computed **Average** to define the **Limit**.

7. (Optional) Enable **Create Volume Surface** to create a volume "surface" that can be selected for visualization in graphics objects such as mesh and contour plots.

8. Click **Save** to create the cell register.

Note:

You can change the way the register is displayed by using the Cell Register Display Options Dialog Box (p. 5407) dialog box that opens by clicking **Display Options...**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

39.2.5. Residuals

Residual registers allow you to mark cells based on their residuals values. You can visualize the cells with the highest residual values and know their location so that you can modify them as necessary.

Note:

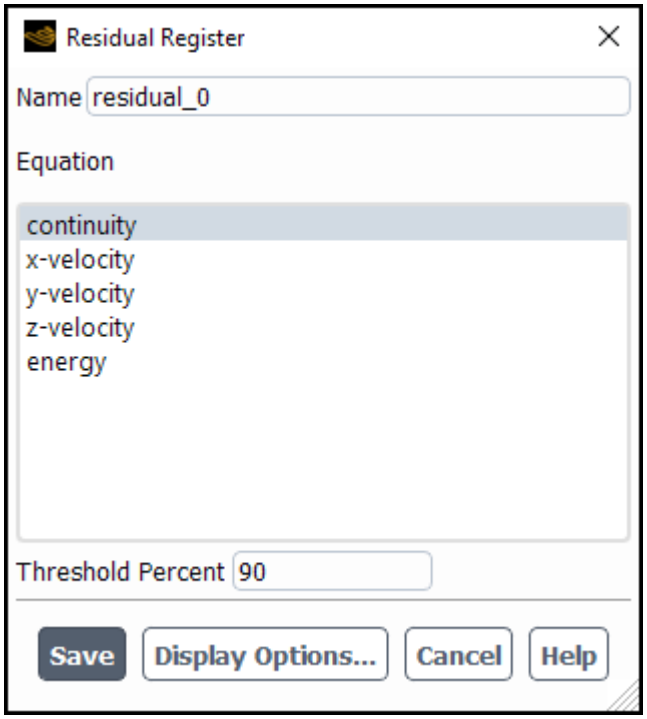
Residual registers are only available for the pressure-based numerical solver.

To create a residual cell register:

1. Open the Figure 39.30: Residual Register Dialog Box (p. 3860).

   Solution → **Cell Registers** **New** → **Residuals...**

   Figure 39.30: Residual Register Dialog Box



2. Select the **Equation**(s) for creating the residual cell register.
3. Specify the **Threshold Percent** for creating the residual cell register. Cells with residual values above this threshold for the selected equations are marked.
4. Click **Save** to create the cell register.

Note:

You can change the way the register is displayed by using the Cell Register Display Options Dialog Box (p. 5407) dialog box that opens by clicking **Display Options...**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

39.2.6. Volume

Volume registers allow you to mark cells based on the volume of the cells, either the magnitude or the change in volume.

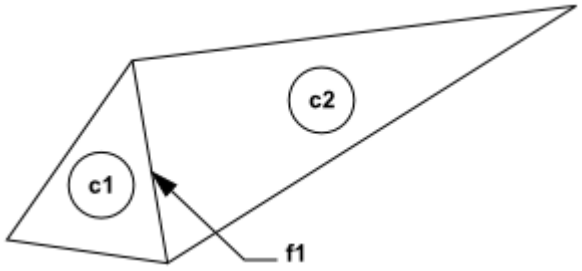
39.2.6.1. Volume Cell Register Approach

Marking the mesh based on volume magnitude is often used to identify large cells. The procedure is to mark cells with a volume greater than the specified threshold volume.

Marking the mesh based on the change in cell volume is used to identify areas where you may want to improve the smoothness of the mesh. The procedure is to mark any cell that has a volume change greater than the specified threshold value. The volume change is computed by looping over the faces and comparing the ratio of the cell neighbors to the face.

For example, in Figure 39.31: Volume Change—Ratio of the Volumes of the Cells (p. 3861) the ratio of V1/V2 and the ratio of V2/V1 is compared to the threshold value. If V2/V1 is greater than the threshold, then C2 is marked.

Figure 39.31: Volume Change—Ratio of the Volumes of the Cells



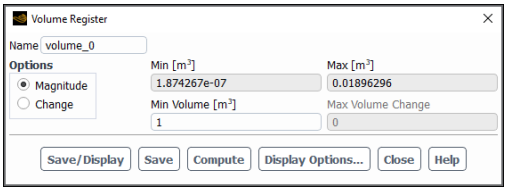
39.2.6.2. Setting Up a Volume Cell Register

To create a volume cell register:

1. Open the Figure 39.32: Volume Register Dialog Box (p. 3861).

   Solution → Cell Registers → New → Volume...

   Figure 39.32: Volume Register Dialog Box



2. Choose whether the cell register is based on the minimum size of the cells (**Magnitude**) or the **Change** in volume of the cells.

3. Specify the **Min Volume** or **Max Volume Change**. Note that you can click **Compute** to compute the minimum and maximum cell volumes.

4. Click **Save** to create the cell register.

Note:

You can change the way the register is displayed by using the Cell Register Display Options Dialog Box (p. 5407) dialog box that opens by clicking **Display Options...**.

39.2.7. Yplus/Ystar

Yplus/Ystar registers allow you to mark cells based on their Yplus/Ystar values.

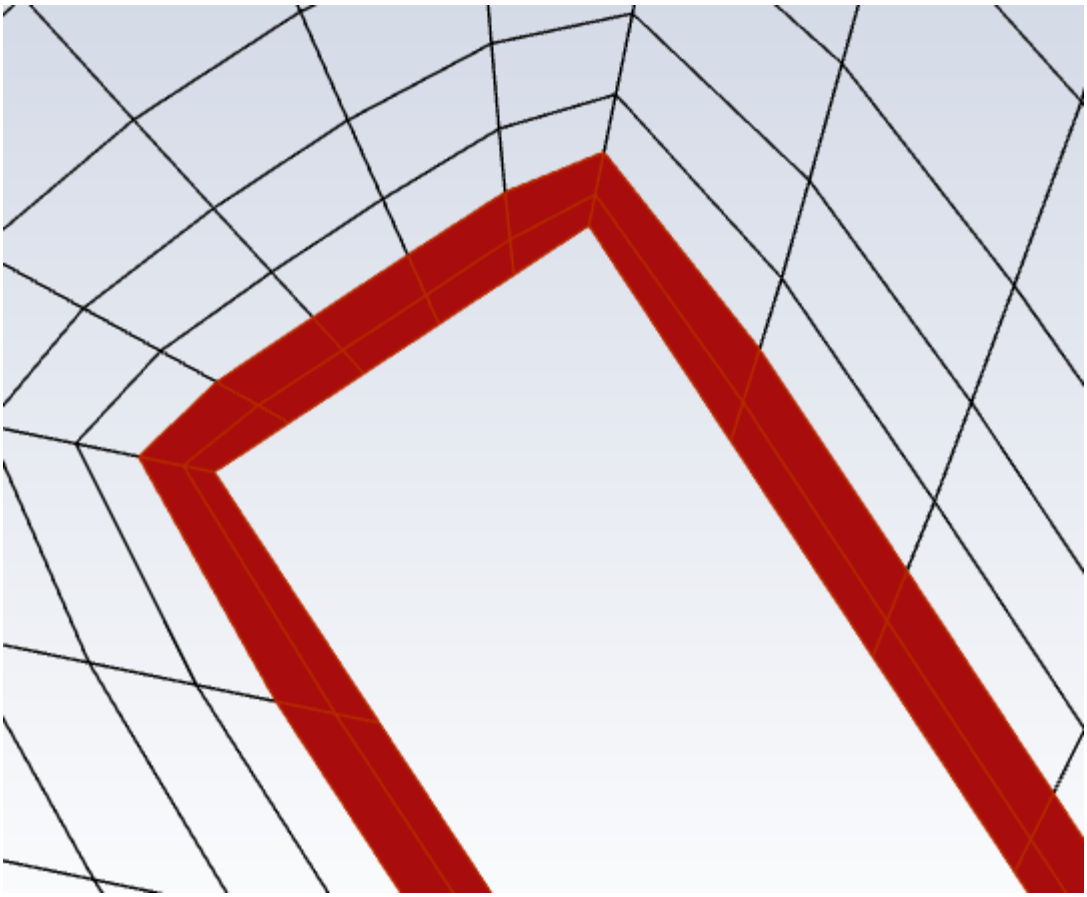
39.2.7.1. Yplus/Ystar Approach

The approach is to compute y^+ or y^* for boundary cells on the specified viscous wall zones, define the minimum and maximum allowable y^+ or y^*, and mark the appropriate cells. Cells with y^+ or y^* values below the minimum allowable threshold will be marked, as will cells with y^+ or y^* values above the maximum allowable threshold.

Figure 39.33: Airfoil Wall Cells Marked by Y+ Values (p. 3863) shows a mesh created to solve the flow over an airfoil, where the bottom boundary of the mesh that is shown is the top surface of the airfoil. After an initial solution, it was determined that y^+ values of some of the cells on the top surface were too large, and y^+ was used to mark them.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 39.33: Airfoil Wall Cells Marked by Y+ Values



39.2.7.2. Setting up a Yplus/Ystar Cell Register

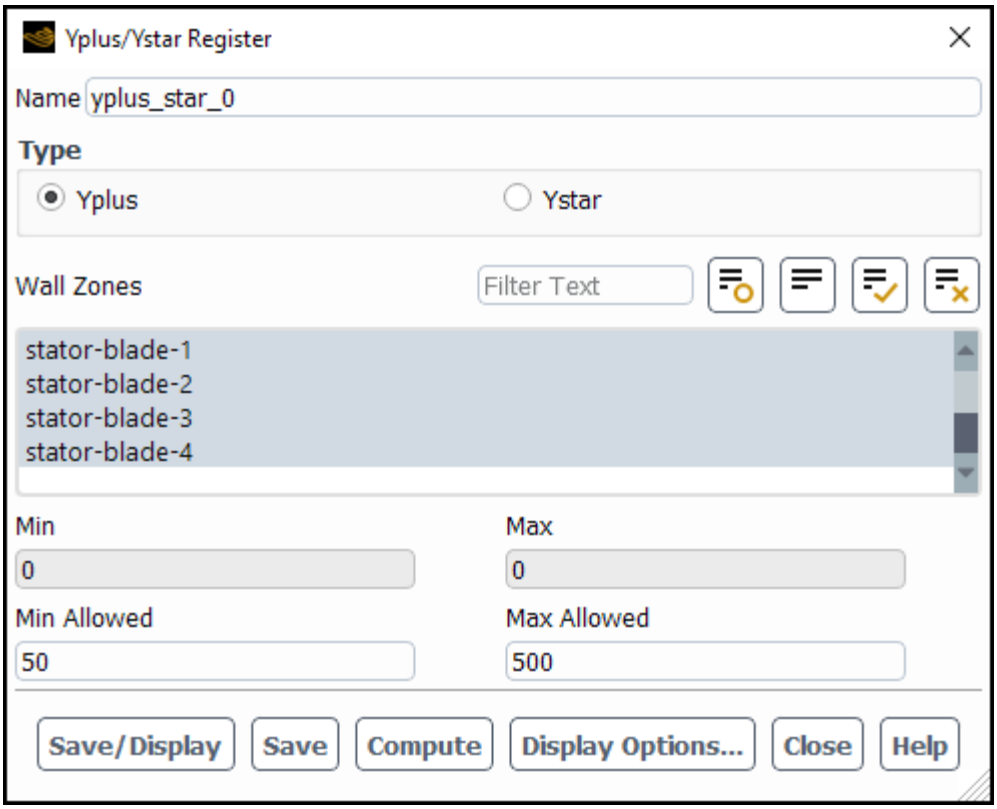
To create a Yplus/Ystar cell register:

1. Open the Figure 39.34: Yplus/Ystar Register Dialog Box (p. 3864).

   Solution → **Cell Registers** → **New** → **Yplus/Ystar...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 39.34: Yplus/Ystar Register Dialog Box



2. Choose whether the cell register is based on **Yplus** or **Ystar** values.
3. Select the **Wall Zones** for the Yplus or Ystar calculations.
4. Specify the minimum and maximum Yplus/Ystar values allowed. Note that clicking **Compute** calculates the minimum and maximum Yplus/Ystar values.
5. Click **Save** to create the cell register.

Note:

You can change the way the register is displayed by using the Cell Register Display Options Dialog Box (p. 5407) dialog box that opens by clicking **Display Options...**.

39.2.8. Manage Cell Registers

The Figure 39.35: Manage Cell Registers Dialog Box (p. 3865) allows you to review the properties of the currently defined cell registers. You can also use it to display and delete the selected cell register, as well as to create new cell registers.

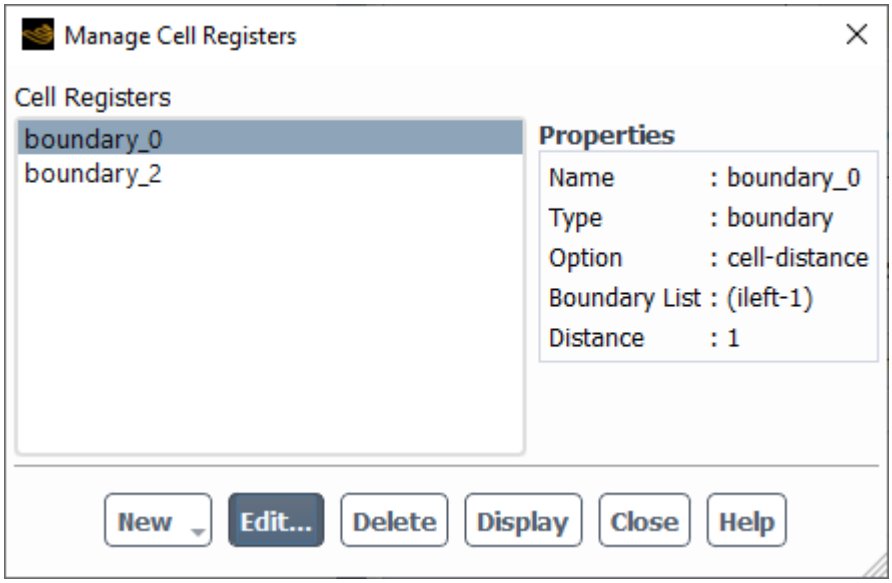
To manage cell registers:

1. Open the Figure 39.35: Manage Cell Registers Dialog Box (p. 3865).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution → Cell Registers Edit...

Figure 39.35: Manage Cell Registers Dialog Box



2. Select an existing cell register to view its properties, display it, edit it, or delete it; or create a new cell register by clicking the **New** button and making a selection from the menu that opens.

39.2.9. Cell Register Operations

Once a cell register is defined, you can create a cell register operation to report the residual values of the selected cell registers at the interval you specify.

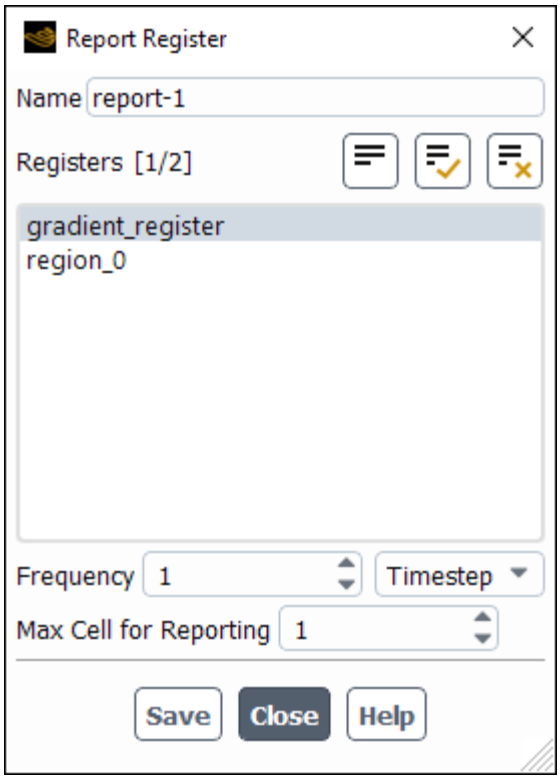
To create a register report:

1. Open the Figure 39.36: Report Register Dialog Box (p. 3866).

Solution → Calculation Activities → Cell Register Operations New Report...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 39.36: Report Register Dialog Box



Note that this dialog box is only available after you have defined a cell register.

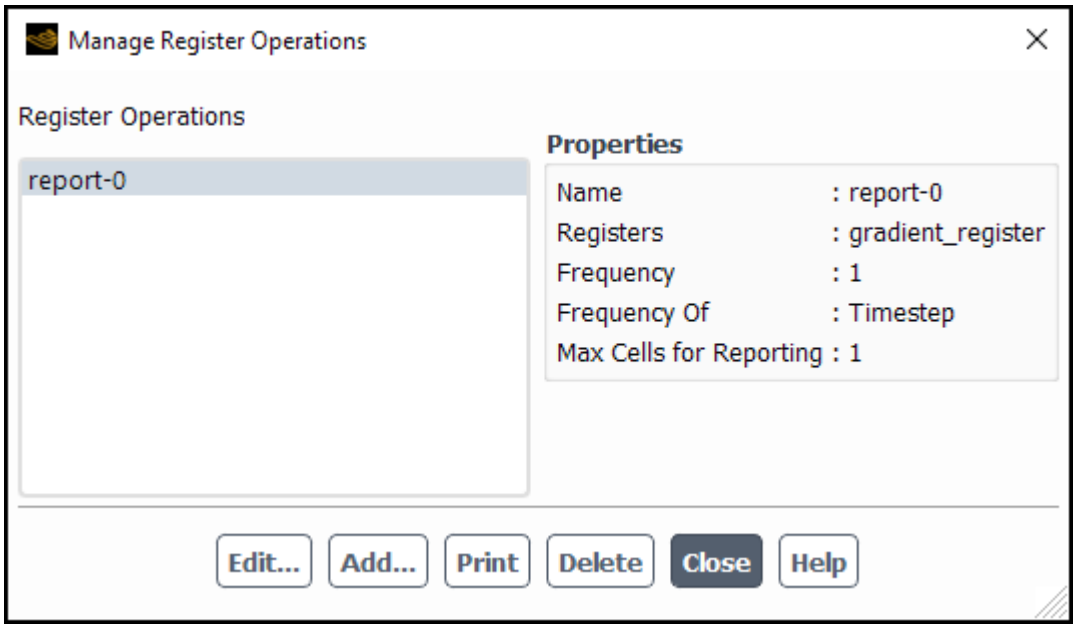
2. Select the cell register(s) you want included in the report.
3. Specify the reporting **Frequency**.
4. Specify the maximum number of cells to include in the report.
5. Click **Save** to create the cell register report.

The Figure 39.37: Manage Register Operations Dialog Box (p. 3867) allows you to view properties, create, modify, and delete report registers.

Solution → Calculation Activities → Cell Register Operations Edit...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 39.37: Manage Register Operations Dialog Box



39.2.10. Copying and Renaming Cell Registers

The procedures for copying a cell register is as follows:

1. In the **Outline View**, right-click the name of a cell register that you want to copy (under **Setup/Solution/Cell Registers**) and select **Copy...** to open the corresponding register dialog box.
2. Enter the new name in the **Name** field and click **Save/Display**.

   A new register will be added under the **Cell Registers** branch in the **Outline View**.

This is useful if you want to create a cell register with similar properties.

To change the name of a register:

1. In the **Outline View**, select the cell register you want to rename. It becomes highlighted.
2. Click the highlighted name again or press **F2** to enter the edit mode.
3. Type the new name and click outside of the name field to save it.

*Alternatively, you can open the cell register dialog box, change the name, and click **Save**.*

This procedure is similar to changing file names in Windows Explorer.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 40: Displaying Graphics

Graphics tools available in Ansys Fluent enable you to process the information contained in your CFD solution and easily view the results. The following sections explain how to use these tools to examine your solution. The procedure for saving picture files of graphics displays is described in Saving Picture Files (p. 1019).

40.1. Basic Graphics Generation

40.2. Customizing the Graphics Display

40.3. Enhanced Graphics Visual Effects

40.4. Realistic Rendering Using Raytracing

40.5. Controlling the Mouse Button Functions

40.6. Viewing the Application Window

40.7. Measuring Distance and Angle

40.8. Controlling the Display State and Modifying the View

40.9. Advanced Scene Composition

40.10. Animating Graphics

40.11. Histogram and XY Plots

40.12. Fast Fourier Transform (FFT) Postprocessing

40.13. Cumulative Force, Moment, and Coefficients Plots

40.1. Basic Graphics Generation

In Ansys Fluent, you can generate graphics displays showing meshes, contours, profiles, vectors, pathlines, and scenes. Some graphics are generated using variables that are plotted directly from the Ansys Fluent data file once the file has been read. The variables listed in the data file depend on the models active at the time the file is written. Variables that are required by the solver, based on the current model settings, but are missing from the data file are set to their default values. For those missing variables, one iteration should be performed in order to obtain the required values for generating the plot. A complete list of variables stored in the data file is available in (`xfile.h`) and can be accessed

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

as stated in Data Sections (p. 5691). The following sections describe how to create these plots. (Generation of histogram and XY plots is discussed in Histogram and XY Plots (p. 4057).)

Important:

If your model includes a discrete phase, you can also display the particle trajectories, as described in Displaying of Trajectories (p. 2834).

Note:

There are two kinds of graphics objects—those whose settings are saved with the case file for reuse during the simulation, and those that are not. Persistent graphics objects (saved with the case file) are accessed by clicking **New...**.

Results → **Graphics** → **Contours** → **New...**

Non-persistent objects (typically created by clicking **Edit...**) are deprecated and hidden from the user interface by default. To bring back this functionality, enable **Expose legacy non-persistent graphics** in the **Graphics** branch of **Preferences** (accessed via **File**>**Preferences...**).

Remote Display:

If graphics displays appear slow, it may be related to the graphics driver. Ensure Display Power Management Signaling (DPMS) is enabled by entering the `xset -display :0 dpms force on` command in the Xterm window.

If glxgears is not working correctly (at least a few thousand frames per second), then you may require assistance from your IT team.

This section discusses the following topics:

40.1.1. Graphics Performance
40.1.2. Displaying the Mesh
40.1.3. Displaying Contours and Profiles
40.1.4. Displaying Vectors
40.1.5. Displaying Pathlines
40.1.6. Displaying Line Integral Convolutions (LICs)
40.1.7. Displaying a Scene
40.1.8. Displaying Results on a Sweep Surface
40.1.9. Hiding the Graphics Window Display

40.1.1. Graphics Performance

A number of factors can influence the graphics performance of your machine, including the model size and mesh type, the quality of your graphics card, what you are displaying (type of graphics object and number of surfaces being displayed), and your Fluent Preferences.

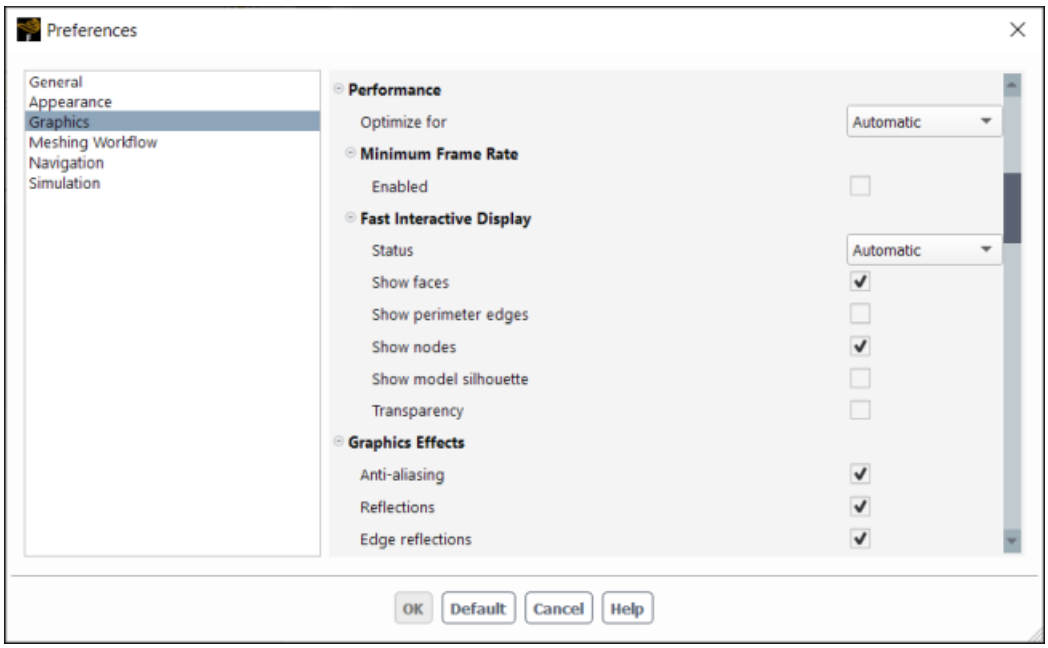
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Using a modern graphics driver, such as OpenGL2 (Linux, Windows) or DX11 (Windows) will let you take full advantage of Fluent's graphical capabilities.

You can set preferences to automatically use the best graphics options for your case, or you can set them to optimize for appearance or performance. Refer to Optimizing Graphical Priorities (p. 3990) for additional information.

File → Preferences...

Figure 40.1: Performance Preferences



The **Status** of **Fast Interactive Display** indicates whether the strategically reduced display is invoked automatically based on the size of your model display (number of faces being displayed) and your machine's ability to display it, or if it is always on or always off.

You can specify what will be shown when the fast interactive reductions are invoked by enabling the desired options under **Fast Interactive Display**.

Note:

- The first time your model is displayed with **Show perimeter edges** enabled, you may experience a delay in the interaction while Fluent generates the reduced display.
- Graphics objects may partially or completely disappear during graphics window mouse interactions (such as zooming or panning). Disable **Fast Interactive Display** in **Preferences** to avoid this behavior.

Important:

Fast Interactive Display options are not compatible with **Minimum Frame Rate**.

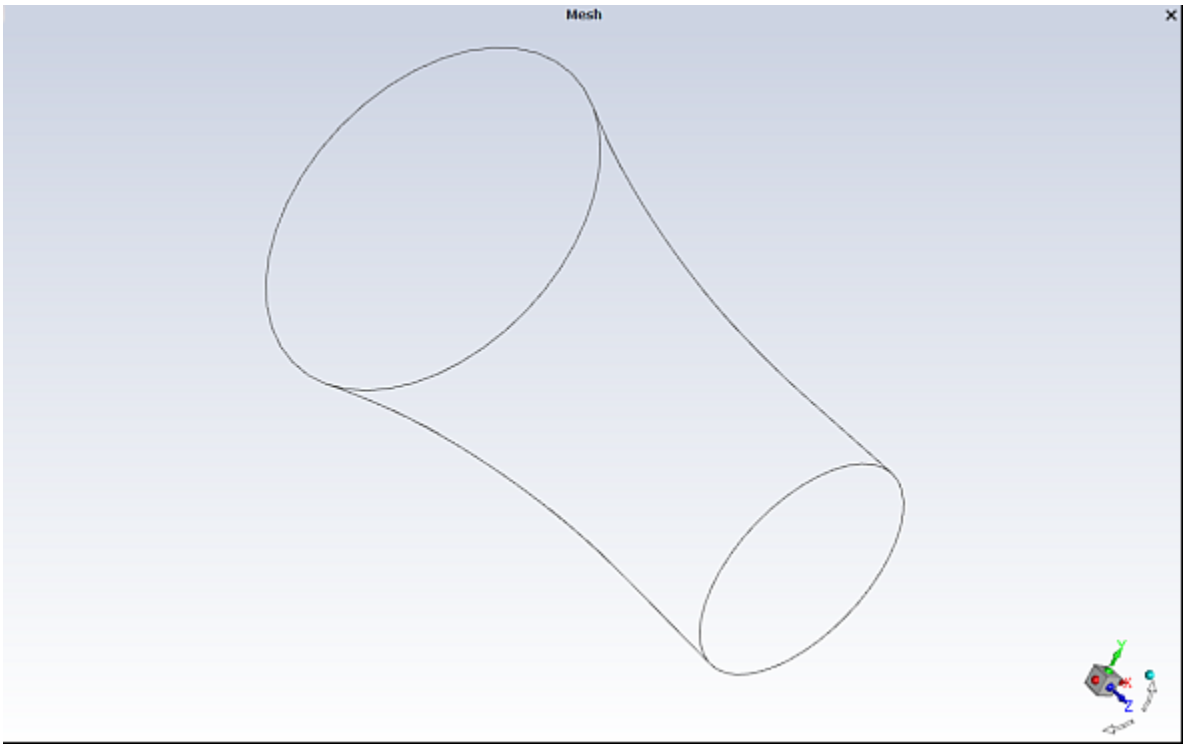
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

40.1.2. Displaying the Mesh

During the problem setup or when you are examining your solution, you may want to look at the mesh associated with certain surfaces. You can do the following:

- Display the outline of all or part of the domain, as shown in Figure 40.2: Outline Display (p. 3872).

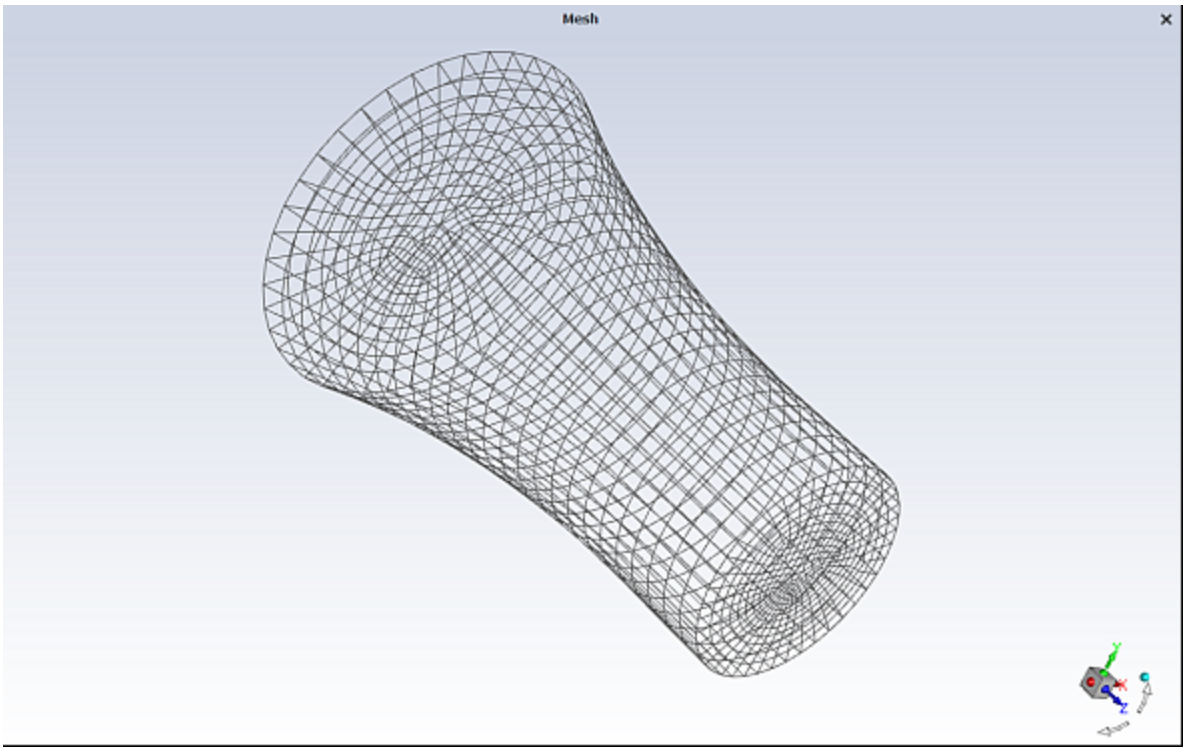
  Figure 40.2: Outline Display



- Draw the mesh lines (edges), as shown in Figure 40.3: Mesh Edge Display (p. 3873).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

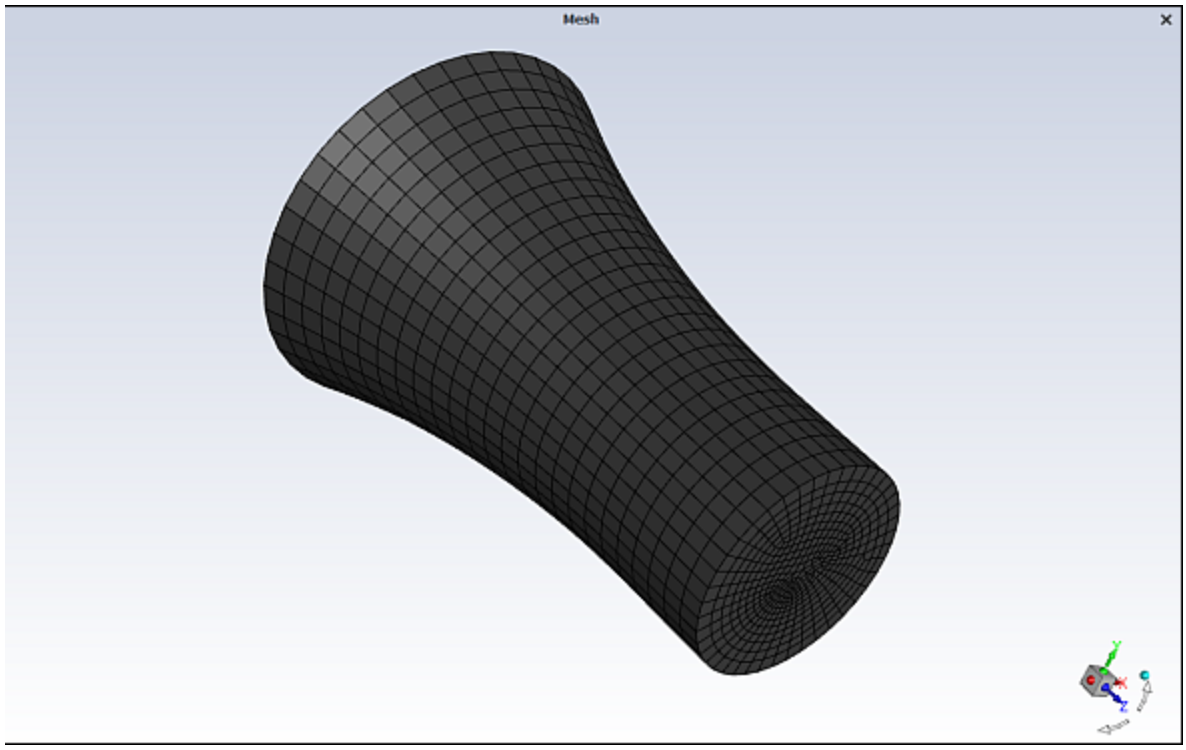
Figure 40.3: Mesh Edge Display



- Draw the solid surfaces (filled meshes) for a 3D domain, as shown in Figure 40.4: Mesh Face (Filled Mesh) Display (p. 3874).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.4: Mesh Face (Filled Mesh) Display



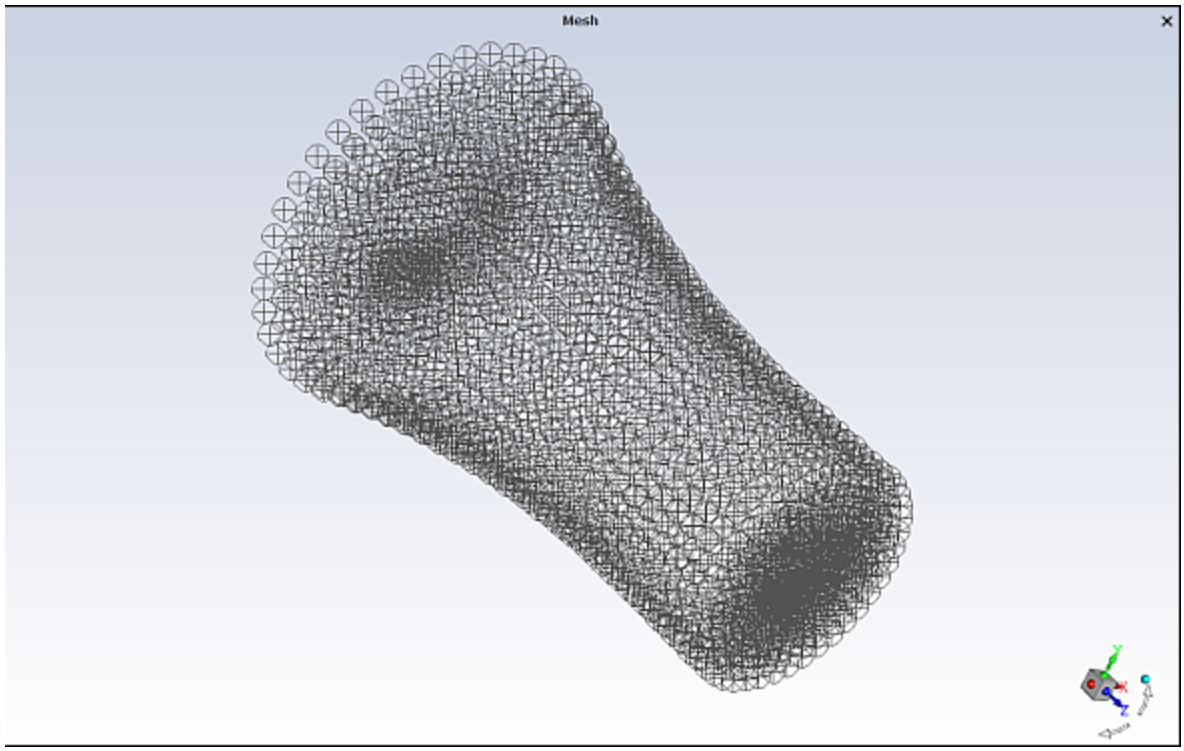
Note:

You can control how reflective the surfaces appear using the **Surface specularity for other surfaces** field in the **Appearance** branch of **Preferences**. Values closer to 1 are more reflective, while those closer to 0 are less reflective.

- Draw the nodes on the domain surfaces, as shown in Figure 40.5: Node Display (p. 3875).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.5: Node Display



Note:

(Parallel Fluent sessions) Mesh displays with **Edges** enabled and either **Feature** or **Outline** selected will not show any partition lines by default. This default behavior is controlled by the **Remove partition lines** option in the **Graphics** branch of **Preferences** (accessed via **File**>**Preferences...**).

If it appears that your model is de-featured or faces are collapsed, try reducing the **Partition line removal tolerance** setting in **Preferences**. Conversely, if you are still seeing fragments of partition lines, try increasing the **Partition line tolerance**.

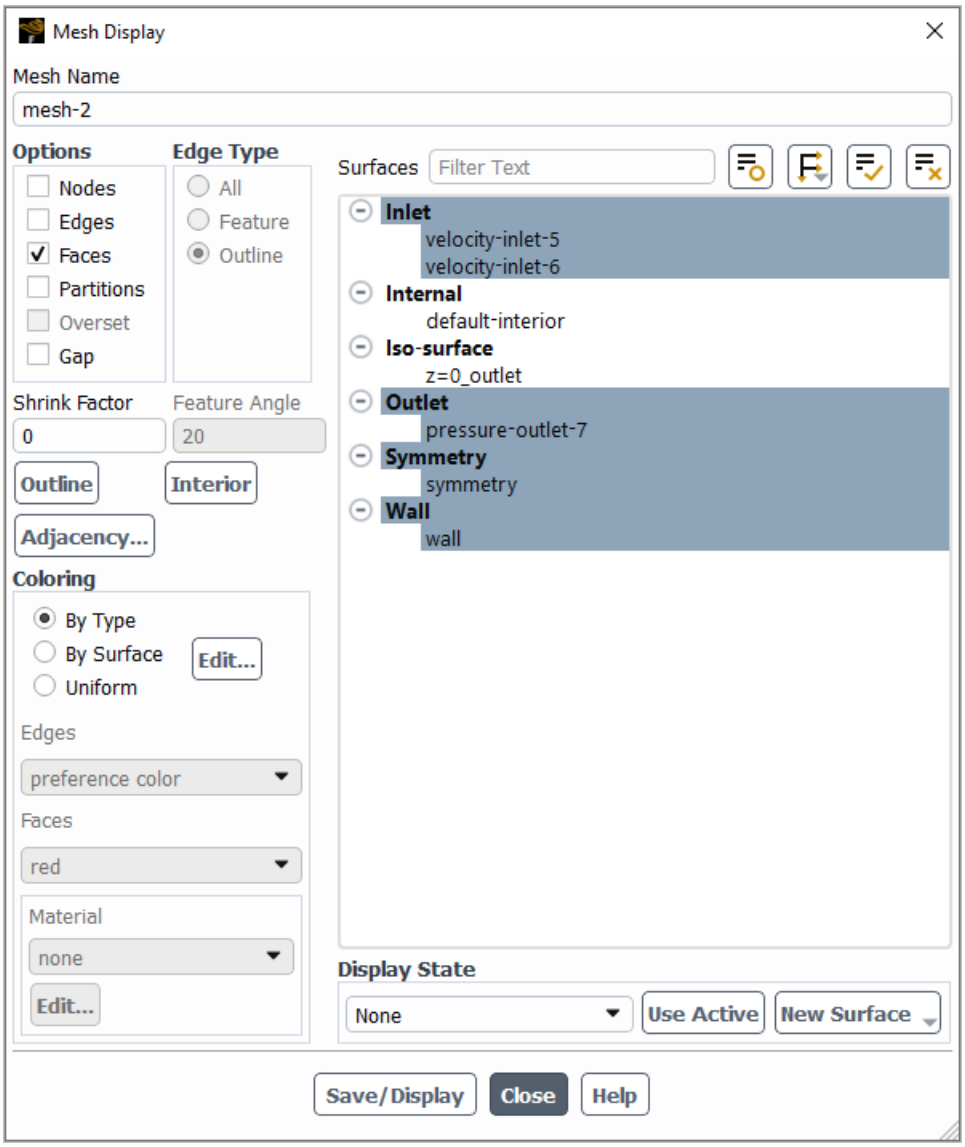
For information about displaying the mesh on a surface that sweeps through the domain, see Displaying Results on a Sweep Surface (p. 3937).

40.1.2.1. Generating Mesh or Outline Plots

You can draw the mesh or outline for all or part of your domain using the Mesh Display Dialog Box (p. 4688) (Figure 40.6: The Mesh Display Dialog Box (p. 3876)).

Results → **Graphics** → **Mesh** **New...**

Figure 40.6: The Mesh Display Dialog Box



The basic steps for generating a mesh or outline plot are as follows:

1. Choose the surfaces for which you want to display the mesh or outline in the **Surfaces** list.

   If you want to select several surfaces of the same type, click and select **Surface Type** under **Group By**, which organizes the surfaces in a tree view that is grouped by surface type.

   Another shortcut is to use the **Filter Text** entry box to filter the **Surfaces** list to show only the surfaces that match the pattern you enter. For additional information on using the **Filter Text** entry box, see Filter Text Entry Boxes (p. 926).

2. Depending on what you want to draw, do one or more of the following:

   - To draw an outline of the selected surfaces (as in Figure 40.2: Outline Display (p. 3872)), select **Edges** under **Options** and **Outline** under **Edge Type**. If you need more detail in the outline display of a complex geometry, see the description of the **Feature** option, below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To draw the mesh edges (as in Figure 40.3: Mesh Edge Display (p. 3873)), select **Edges** under **Options** and **All** under **Edge Type**.

  Note:

  Mesh edges are disabled by default. You can enable **Show model edges** in the **Appearance** branch of Preferences (accessed via the **File** ribbon tab) to change the default behavior so that all new mesh objects you create will have mesh edges enabled.

- To generate a filled-mesh display (as in Figure 40.4: Mesh Face (Filled Mesh) Display (p. 3874)), select **Faces** under **Options**.
- To draw the nodes on the selected surfaces (as in Figure 40.5: Node Display (p. 3875)), select **Nodes** under **Options**.

3. Set any of the mesh and outline display options described in the following section.
4. Click **Save/Display** to draw the specified mesh or outline in the active graphics window.

To display filled meshes, with smoothly shaded display, enable lighting and select a lighting interpolation method other than **Flat** in the Display Options Dialog Box (p. 5262) or the Lights Dialog Box (p. 5279).

If you display nodes, and you want to change the symbol representing the nodes, you can change the **Point Symbol** in the Display Options Dialog Box (p. 5262). See Modifying the Rendering Options (p. 3987) for details.

40.1.2.2. Modifying the Mesh Colors

Ansys Fluent enables you to control the colors that are used to render the meshes for each zone type or surface. This capability can help you to understand mesh plots quickly and easily. Ansys Fluent offers two distinct color palettes that you can use to color the mesh: **Classic** and **Pastel** (Figure 40.12: Mesh Display with Pastel Color Scheme (p. 3881)). The default color palette is controlled by the **Model Color Scheme** field in the **Appearance** branch of **Preferences** (**File**>**Preferences...**).

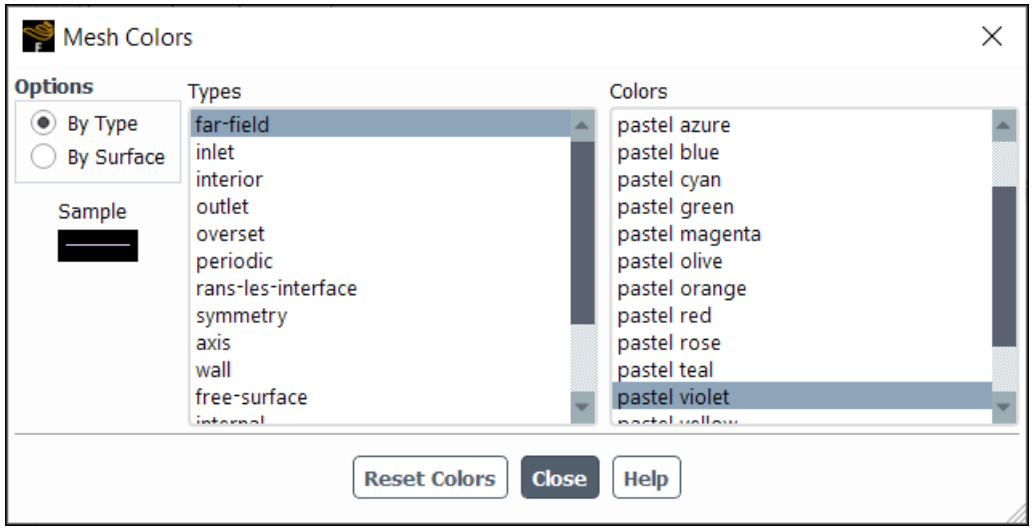
How you modify the mesh colors depends on the type of mesh display:

- If you are displaying the mesh using the non-persistent version of the **Mesh Display** dialog box, accessed by clicking the mesh display toolbar button ( ), then you can control the colors via the Mesh Colors Dialog Box (p. 4695) (the available colors vary depending on the selected palette and will look like Figure 40.7: The Mesh Colors Dialog Box (Classic Color Scheme) (p. 3878) or Figure 40.8: The Mesh Colors Dialog Box (Pastel Color Scheme) (p. 3878)) by clicking **Colors...** in the Mesh Display Dialog Box (p. 4688).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.7: The Mesh Colors Dialog Box (Classic Color Scheme)

Figure 40.8: The Mesh Colors Dialog Box (Pastel Color Scheme)



- If you are displaying the mesh using the persistent version of the **Mesh Display** dialog box accessed from the Outline View tree or **Results** ribbon tab, there are three approaches you can use for controlling the mesh colors—**By Type**, **By Surface**, **Uniform**.

- If you are displaying the mesh using the persistent version of the **Mesh Display** dialog box accessed from the Outline View tree or **Results** ribbon tab, then you can control the colors via the Zone Type Color and Material Assignment Dialog Box (p. 5664) (the available colors vary depending on the selected palette and will look like Figure 40.9: The Zone Type Color and Material Assignment Dialog Box (Classic Color Scheme) (p. 3879) or Figure 40.10: Zone Type Color and Material Assignment Dialog Box (Pastel Color Scheme) (p. 3879)) by clicking **Edit...** in the Mesh Display Dialog Box (p. 4688).

  – **By Type**—Coloring **By Type** is selected by default in the **Mesh Display** dialog box. Opening the **Zone Type Color and Material Assignment** dialog box (by clicking **Edit...** when **By Type** is selected) lets you assign colors based on zone type. To change the color used to draw the mesh for a particular zone type, select the zone type in the **Types** list and then select the new color in the **Colors** list. You will see the effect of your change when you next display the mesh.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that the **surface** type in the **Types** list applies to all surface meshes (that is, meshes that are drawn for surfaces created using the dialog boxes opened from the **Create** button in the **Surface** group box of the **Domain** ribbon tab) except zone surfaces.

Figure 40.9: The Zone Type Color and Material Assignment Dialog Box (Classic Color Scheme)

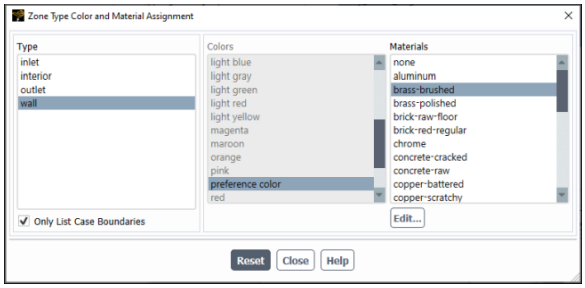
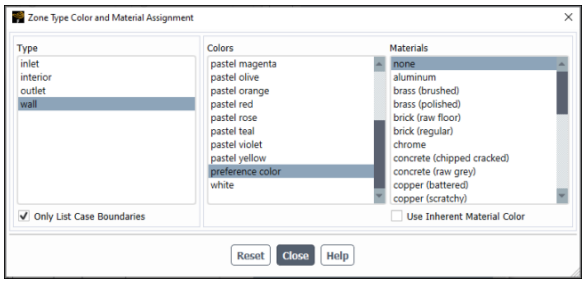


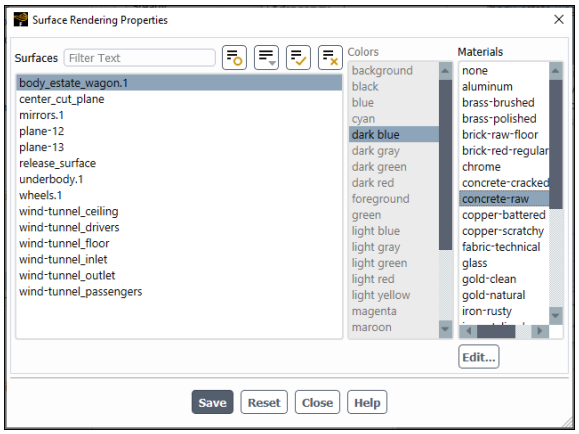
Figure 40.10: Zone Type Color and Material Assignment Dialog Box (Pastel Color Scheme)



– **By Surface**—Initially, each surface is assigned a random color based on its ID. However, clicking **Edit...** when **By Surface** is selected opens the **Surface Rendering Properties** dialog box. Here you can specify a color and/or material for each surface. These settings are preserved, so any subsequent mesh object that you display with **By Surface** selected will show the same colors/materials that you specify in the **Surface Rendering Properties** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.11: Surface Rendering Properties Dialog Box (Classic Color Scheme)



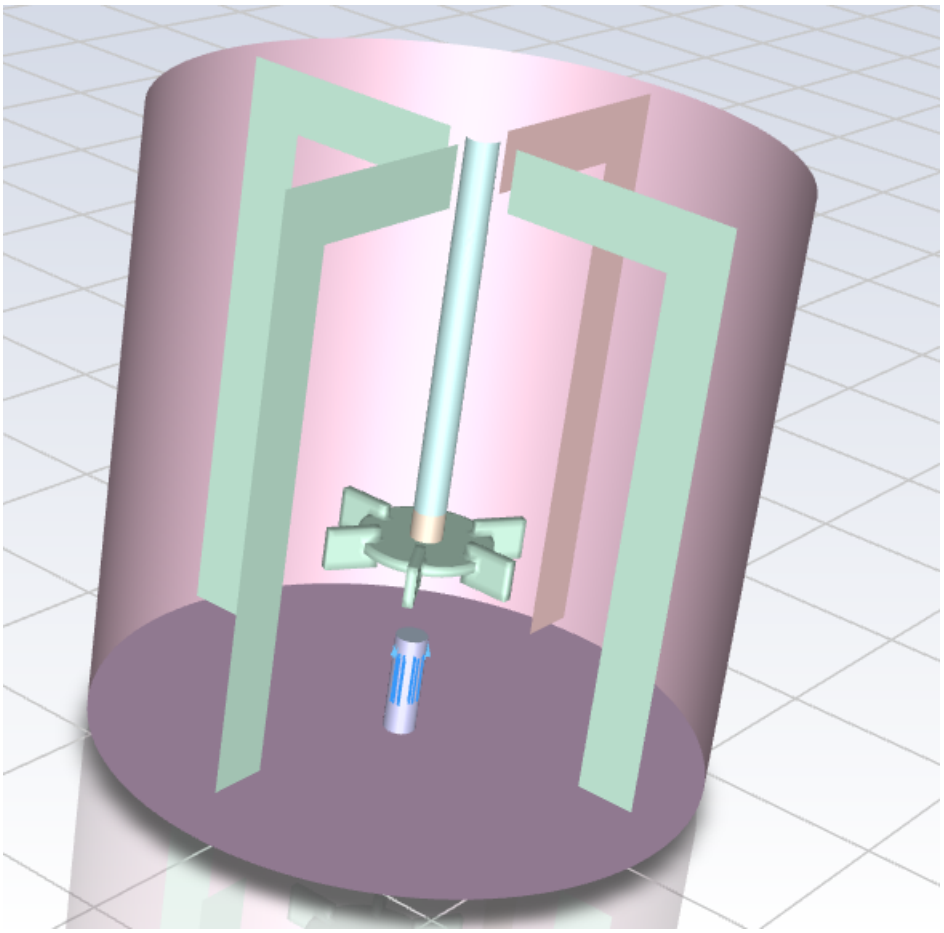
Note:

While color and material assignments are local to the selected surfaces when you click **Save**, the **Use Inherent Material Color** option is not. That is, if you change it for one surface, it is changed for all surface color/material assignments.

– **Uniform**—All the selected surfaces are colored according to the specified color and/or material selected in the **Faces** and **Material** drop-down lists in the **Coloring** group box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.12: Mesh Display with Pastel Color Scheme



You can set colors individually for the meshes displayed on each surface, by selecting the surface(s) in the graphics window, right-clicking and selecting **Color by** > **Color** > **<color name>** from the context menu. Once you have assigned the desired colors, you can right-click and select **Save as Scene** to create a **Scene** graphics object containing separate mesh objects for each color you specify.

If you prefer to use the colors Ansys Fluent assigns by zone ID, then you can display the mesh using the **By Surface** coloring option.

Note:

By default, walls are colored by **preference color**, which indicates that the color of the walls will change based on the **Graphics Color Theme** specified in **Preferences** (available from the **File** ribbon tab). For example, if you change the color theme from **Gray Gradient** to **Black**, the wall color will automatically change from black to white to maintain visibility (when **preference color** is selected for walls).

Randomizing the Mesh Colors

You can use **Ctrl**+**Shift**+**O** to randomize the mesh colors. Ensure the graphics window is in focus by clicking in it before using the hotkey combination (otherwise it won't work).

There are a couple of limitations associated with the color randomization:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Randomizing the colors won't work if the **Coloring** for the displayed mesh object is set to **By Type** in the **Mesh Display** dialog box. *It only works for **By Surface** coloring.*
- The wall color will not be randomized if the coloring option is set to **Color by Type** in the **Mesh Colors** dialog box (in this case, the wall color is taken from **Preferences**). *Note that the colors of all other surface types will still be randomized.*

40.1.2.3. Realistic Rendering of Materials

The following sections cover additional details on aspects of realistic material rendering:

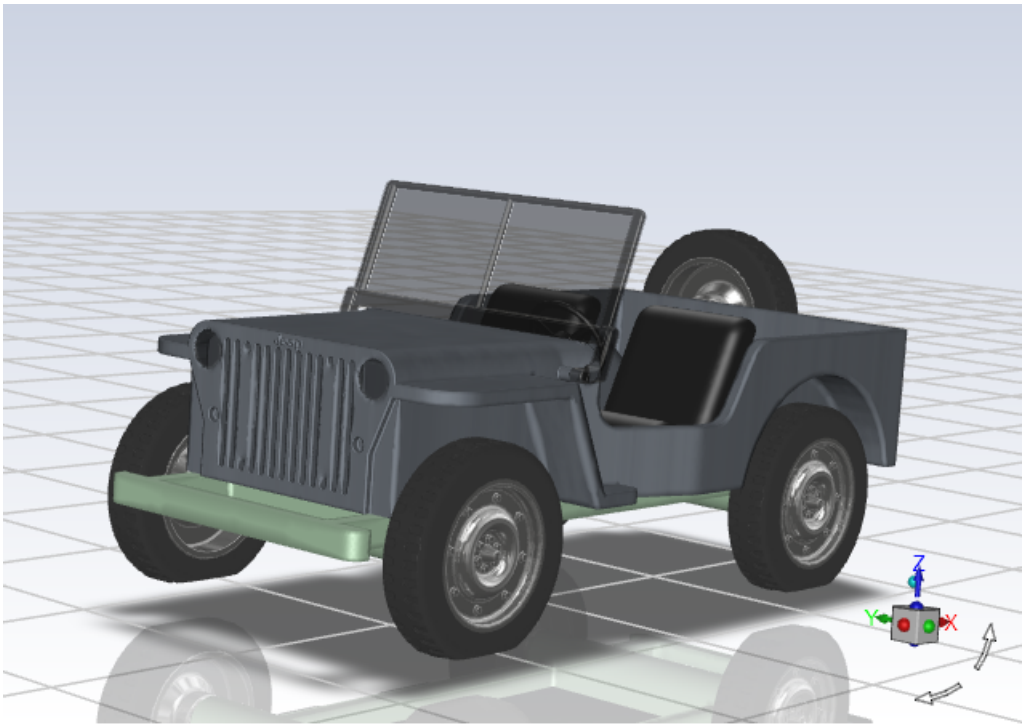
40.1.2.3.1. List of Default Realistic Materials

40.1.2.3.2. Creating Custom Realistic Materials

40.1.2.3.3. Limitations in Realistic Material Rendering

You can assign materials to mesh objects for realistic rendering, as shown in Figure 40.13: Jeep with Multiple Materials Assigned (p. 3882).

Figure 40.13: Jeep with Multiple Materials Assigned



You can assign realistic materials to a mesh object as follows:

- Using the **Mesh Display** dialog box.

  There are three approaches for coloring surfaces with realistic materials: using the **By Type** option, using the **By Surface** option, or using the **Uniform** option, discussed in By Surface Mesh Coloring (p. 3883) and Uniform Mesh Coloring (p. 3886), respectively.

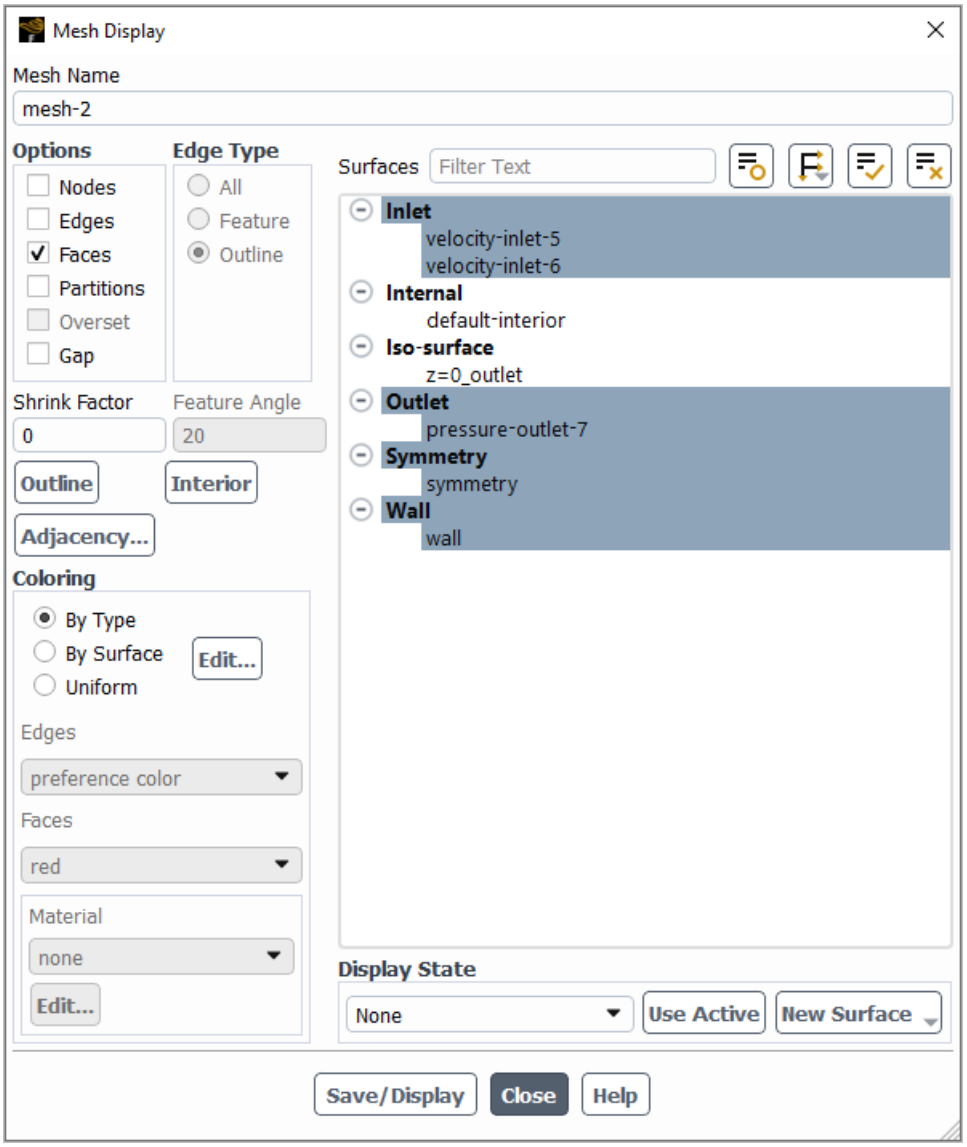
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

By Surface Mesh Coloring

Coloring **By Surface** allows you to set a color and/or material for all surfaces at once. Future mesh displays using the **By Surface** option will be colored using the assignments made in the **Surface Rendering Properties** dialog box.

1. Open the **Mesh Display** dialog box to display a persistent mesh object (this version of the dialog box contains a **Mesh Name** field and is saved with the case file).

   Results → Graphics → Mesh → New...



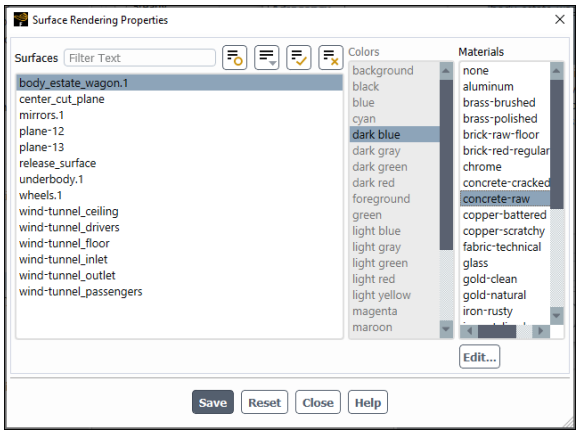
2. Ensure **Faces** is enabled in the **Options** group box.

3. Select **By Surface** in the **Coloring** dialog box.

4. Click **Edit...** to open the Figure 40.14: Surface Rendering Properties Dialog Box (p. 3884).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.14: Surface Rendering Properties Dialog Box



5. Set the desired colors or materials for the surfaces you plan to visualize and click **Save** to confirm the assignments. Click **Close** when you are done assigning colors and materials to surfaces.

6. Click **Save/Display**.

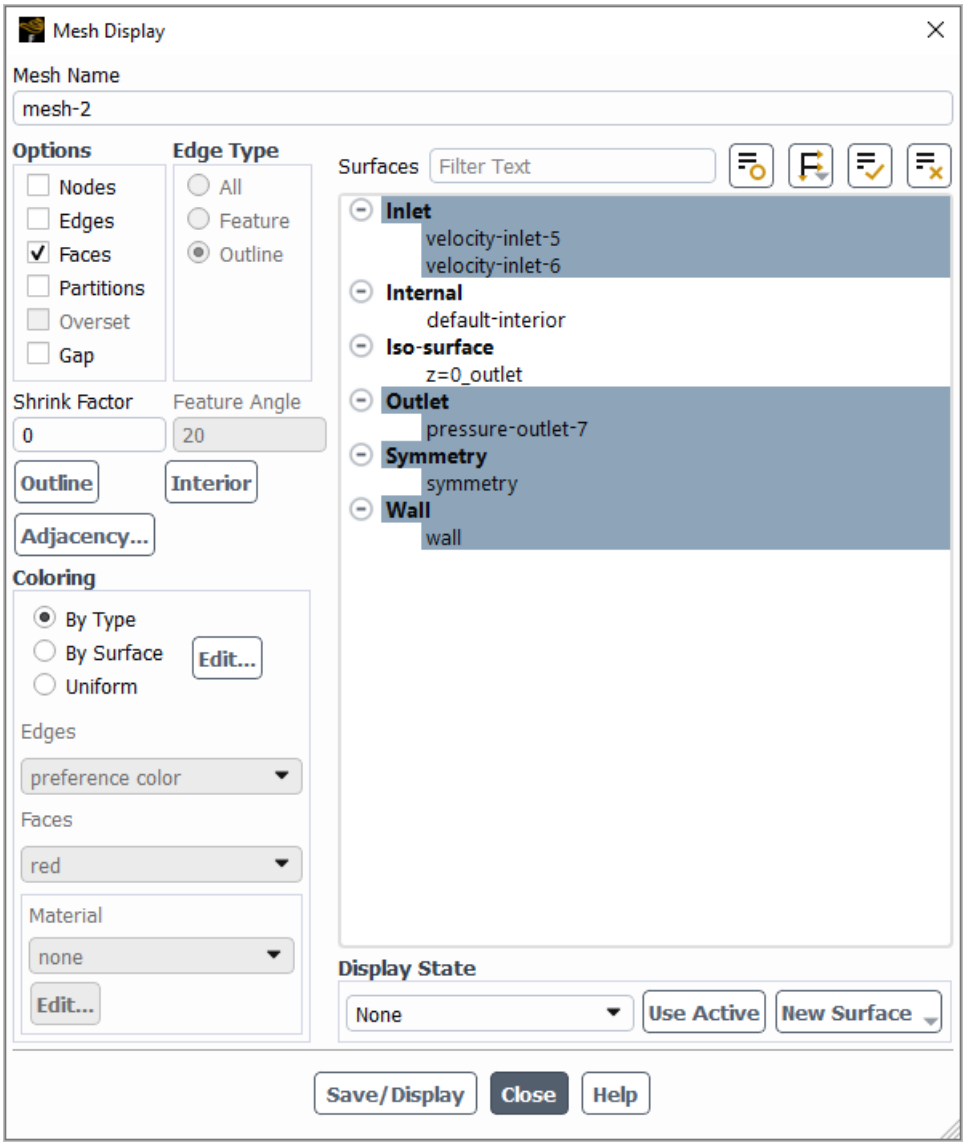
By Type Mesh Coloring

Coloring **By Type** allows you to set a color or material for each type of surface (walls, inlets, outlets, and do on). Future mesh displays using the **By Type** option will be colored using the assignments made in the **Zone Type Color and Material Assignments** dialog box.

1. Open the **Mesh Display** dialog box to display a persistent mesh object (this version of the dialog box contains a **Mesh Name** field and is saved with the case file).

   Results → Graphics → Mesh → New...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

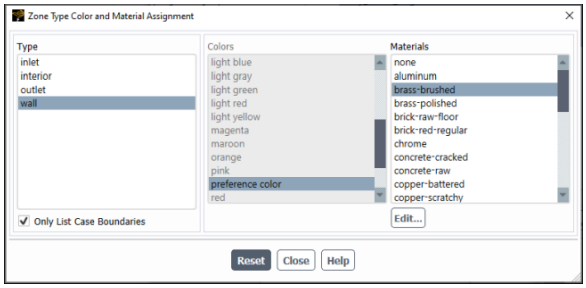


2. Ensure **Faces** is enabled in the **Options** group box.

3. Select **By Type** in the **Coloring** dialog box.

4. Click **Edit...** to open the Figure 40.15: Zone Type Color and Material Assignment Dialog Box (p. 3885).

Figure 40.15: Zone Type Color and Material Assignment Dialog Box



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Set the desired colors or materials for the surface types you plan to visualize and click **Save** to confirm the assignments. Click **Close** when you are done assigning colors and materials to surface types.

6. Click **Save/Display**.

Uniform Mesh Coloring

Unlike coloring **By Surface**, color and material assignments made using this method are only applied for this mesh object and they will not affect any other displays, except scene objects containing this mesh object.

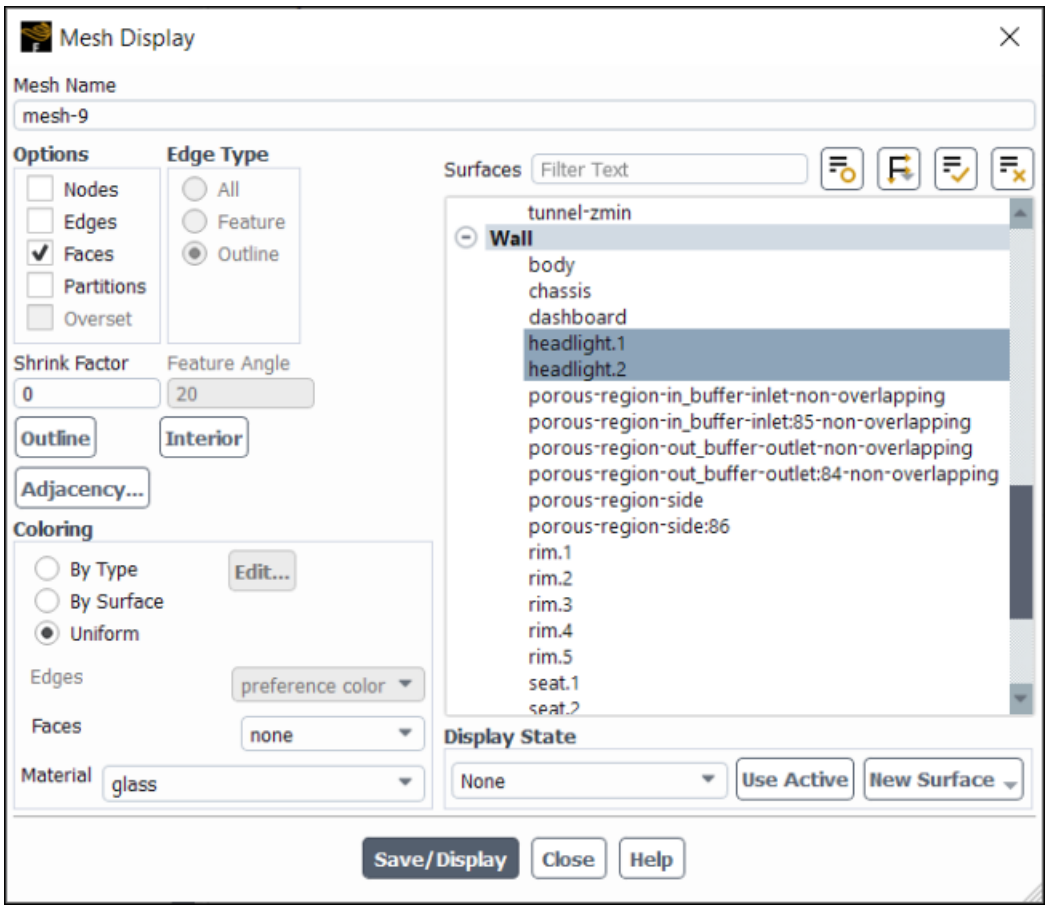
Note:

Using **Uniform** mesh coloring, you can only assign a color or material. If you want to change the color of a material, such as spraypaint, you must create a new material with the color you want. See Creating Custom Realistic Materials (p. 3889) for additional information.

---

1. Open the **Mesh Display** dialog box to display a persistent mesh object (this version of the dialog box contains a **Mesh Name** field and is saved with the case file).

   Results → Graphics → Mesh → New...

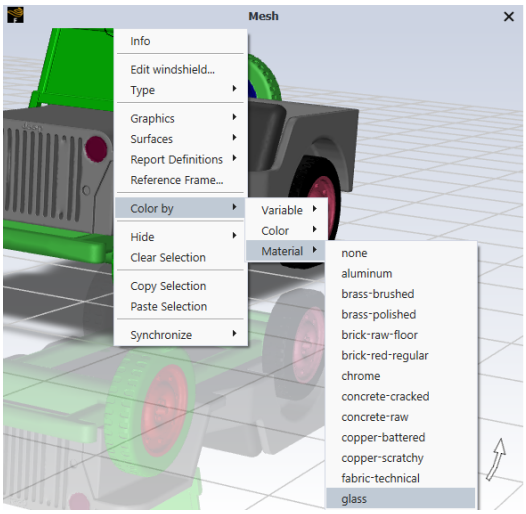


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Ensure **Faces** is enabled in the **Options** group box.
3. Select **Uniform** in the **Coloring** dialog box.
4. Select the desired realistic material from the **Material** drop-down list.
5. Select the surfaces you want colored with this material in the **Surfaces** list.
6. Click **Save/Display**.

- Using right-mouse-button clicks within the graphics window.

1. (Assuming the mesh is not already displayed) Right-click in the graphics window and select **Display**>**Mesh...**.
2. Ensure **Faces** is enabled under **Options**.
3. Select the mesh surfaces you want displayed in the **Surfaces** list. click **Display** and close the **Mesh Display** dialog box.
4. Select the surface(s) you want colored with a realistic material in the graphics window (left-mouse-button-click + **Ctrl** to select multiple).
5. Right-click within the graphics window and select **Color by**>**Material**>**material name**.



40.1.2.3.1. List of Default Realistic Materials

The available materials include:

- **aluminum**
- **brass-brushed**
- **brass-polished**
- **brick-raw-floor**
- **brick-red-regular**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **chrome**
- **concrete-cracked**
- **concrete-raw**
- **copper-battered**
- **copper-scratchy**
- **fabric-technical**
- **glass**
- **glass-diamond**
- **gold-clean**
- **gold-natural**
- **iron-rusty**
- **iron-stylized**
- **oil**
- **paint**
- **plastic**
- **rubber**
- **silver-mirror-polished**
- **spray-paint**
- **titanium-brushed**
- **titanium-linear-brushed**
- **water**
- **wood-beech**
- **wood-beech-natural**
- **wood-cedar**

Note:

You may observe that some of the realistic materials have the same texture, such as brass-brushed and brass-polished. These materials have different bump maps that control the differences in their appearance.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

40.1.2.3.2. Creating Custom Realistic Materials

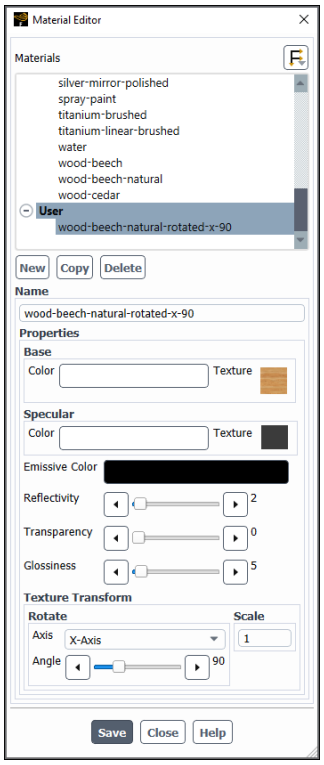
You can use the **Material Editor** dialog box to create custom materials for realistic rendering.

To create a custom material:

1. Open the **Material Editor** dialog box (Figure 40.16: Material Editor Dialog Box (p. 3889)) by clicking **Material Editor...** in the **View** ribbon tab (**Graphics** group box).

   View → Graphics → Material Editor...

   Figure 40.16: Material Editor Dialog Box



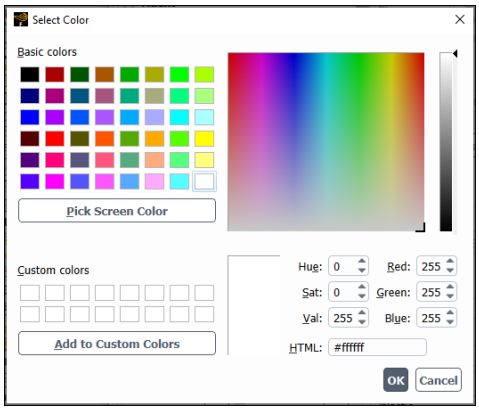
2. Select the material whose texture most closely matches that of the material you want to create and click **Copy**. *If you do not want to copy an existing material, you can click **New**.*

3. Enter a **Name** for the new material.

4. Set the desired material properties.

   a. Select a **Base Color** by clicking in the colored square to open the **Select Color** dialog box. Pick your color and click **OK**. *The base color specifies the primary color of the material.*

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



b. Select a **Specular Color** by clicking in the colored square to open the **Select Color** dialog box. Pick your color and click **OK**. *The specular color specifies the highlight color seen due to lighting effects.*

c. Select a **Emissive Color** by clicking in the colored square to open the **Select Color** dialog box. Pick your color and click **OK**. *The emissive color specifies the color emitted from the material.*

d. Use the slider to set the **Reflectivity**. *Reflectivity specifies how reflective the material appears.*

e. Use the slider to set the **Transparency**. *Transparency specifies how opaque or clear the material appears.*

f. Use the slider to set the **Glossiness**. *Glossiness specifies how glossy the material appears.*

g. Use the **Axis** drop-down and the **Angle** slider to rotate the texture by degrees. *The **Axis** drop-down controls which axis the texture is rotated about.*

h. Use the **Scale** input field to set the desired scaling of the texture. *Valid inputs are 0.0004 to 4, with a larger number increasing the size of the texture.*

5. Click **Save** to create the new material.

Once a new material is created, it is available for selection in the relevant dialog boxes (Surface Rendering Properties Dialog Box (p. 5625), Zone Type Color and Material Assignment Dialog Box (p. 5664), and Mesh Display Dialog Box (p. 4688)). Refer to Realistic Rendering of Materials (p. 3882) for guidance on how to use these dialog boxes to achieve your rendering goals.

Note:

If you delete a custom material that is being used to color a surface of a currently-displayed graphics object, that surface will become black. Re-display the graphics object to have the black surfaces re-rendered with the default color.

40.1.2.3.3. Limitations in Realistic Material Rendering

The following limitations exist for the rendering of realistic materials in Fluent:

- It is only supported for modern graphics drivers: OpenGL2 (Windows, Linux), DX11 (Windows).

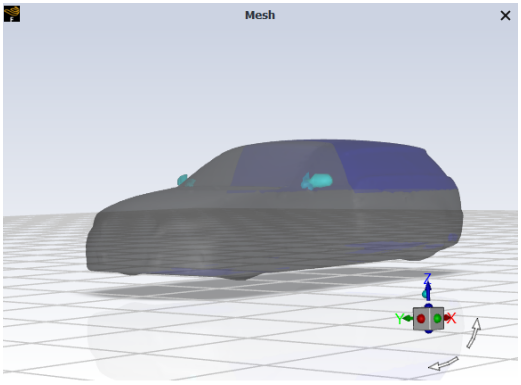
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- It is not supported for Ansys Fluent in 2D.
- It is not supported for the Fluent Remote Visualization Client.
- It is not supported from the global mesh display dialog box (accessed by clicking).
- Realistic rendering on large cases may reduce graphics performance.
- Boundary markers are not compatible with realistic rendering.

40.1.2.4. Controlling Mesh Transparency

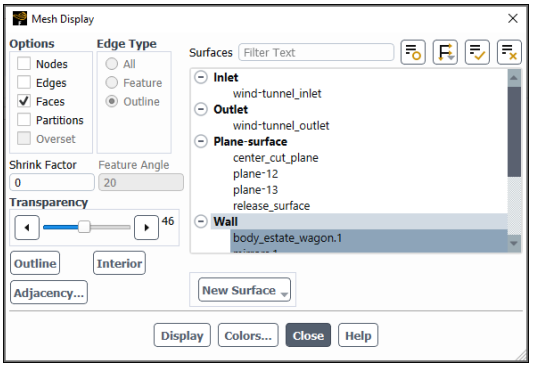
There are multiple ways that you can make the mesh display transparent, as shown in Figure 40.17: Transparent Mesh Display (p. 3891).

Figure 40.17: Transparent Mesh Display



- **Transparency** slider in the Figure 40.18: Global Mesh Display Dialog Box (p. 3891).

  Figure 40.18: Global Mesh Display Dialog Box



  The global version of the **Mesh Display** dialog box can by accessed by:

  – Clicking the mesh display toolbar button ( )

  – Clicking **Display...** in the **General** task page (**Mesh** group box)

Setup → **General** → **Mesh** → **Display...**

– Clicking **Display...** in the **Domain** ribbon tab (**Mesh** group box)

Domain → **Mesh** → **Display...**

- Clicking the transparency toolbar button ( ).
- Creating a scene containing a mesh graphics object (Displaying a Scene (p. 3934))

Important:

By default, you can only see through 2 layers of displayed transparent surfaces before the remaining transparent surfaces are not rendered. The next surface shown is the next non-transparent (opaque) surface. Refer to Figure 40.19: Mesh Display with Transparency: 2 Layers (p. 3892) and Figure 40.20: Mesh Display with Transparency: 3 Layers (p. 3892) to see an example.

You can control the transparency technique and the number of transparent layers using the **Transparency Algorithm for Modern Graphics Drivers** and **Depth Peeling Layers for Transparency** settings under **Transparency** in the **Graphics** branch of **Preferences** (**File**>**Preferences...**).

Figure 40.19: Mesh Display with Transparency: 2 Layers

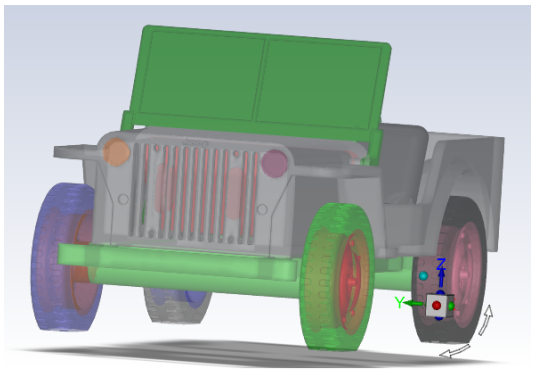
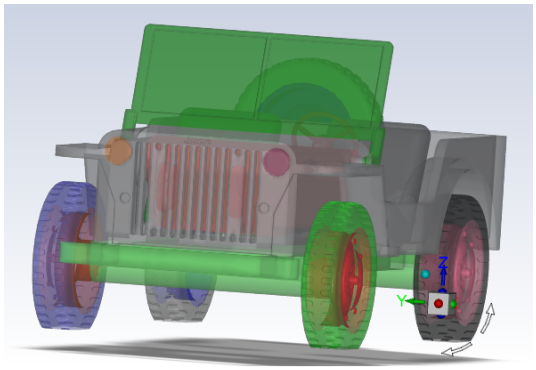


Figure 40.20: Mesh Display with Transparency: 3 Layers



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

40.1.2.5. Mesh and Outline Display Options

The options mentioned in the procedure in the previous section include modifying the mesh colors, adding the outline of important features to an outline display, drawing partition boundaries, and shrinking the faces and/or cells in the display.

40.1.2.5.1. Adding Features to an Outline Display

For closed 3D geometries such as cylinders, the standard outline display often will not show enough detail to accurately depict the shape. This is because for each boundary, only those edges on the "outside" of the geometry (that is, those that are used by only one face on the boundary) are drawn. In Figure 40.21: Standard Outline of Complex Duct (p. 3893), which shows the outline display for a complicated duct geometry, only the inlet and outlet are visible.

Figure 40.21: Standard Outline of Complex Duct

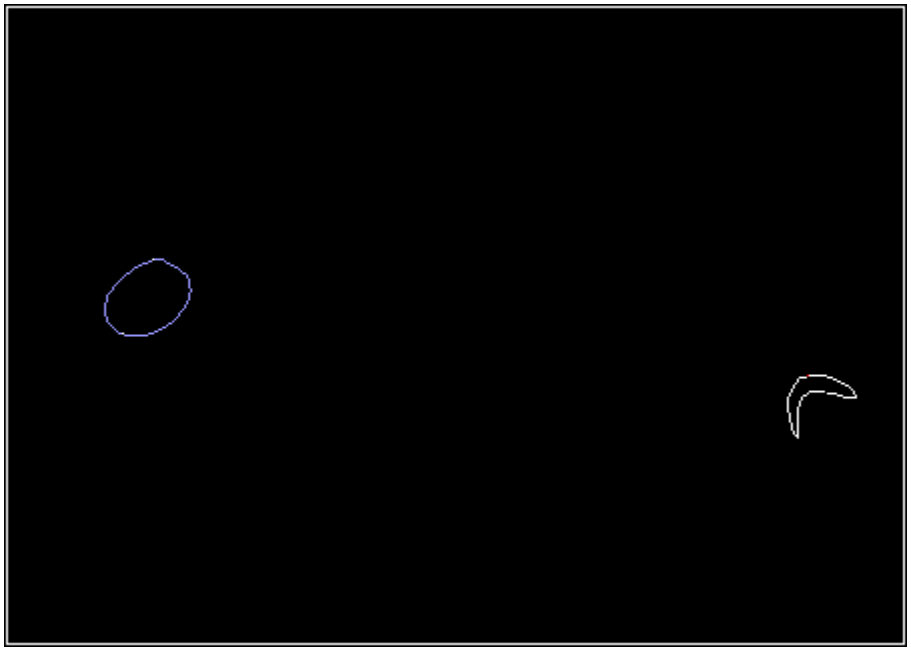
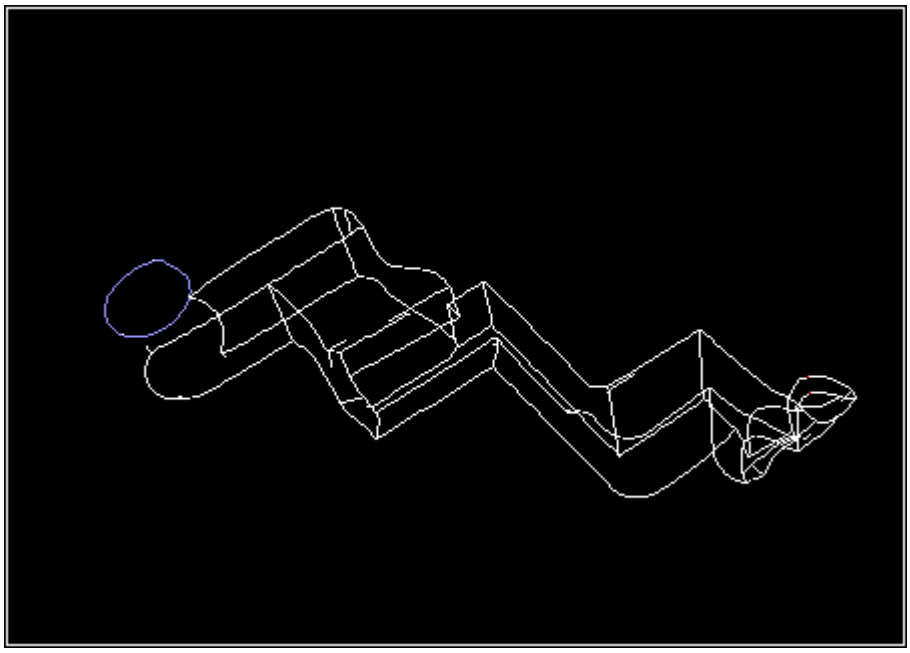


Figure 40.22: Feature Outline of Complex Duct



You can capture additional features using the **Feature** option in the Mesh Display Dialog Box (p. 4688). (See Figure 40.22: Feature Outline of Complex Duct (p. 3894).) Enable **Feature** under **Edge Type**, and then set the **Feature Angle**. With the default **Feature Angle** of 20, if the difference between the normal directions of two adjacent faces is more than 20°, the edge between those faces will be drawn. Decreasing the **Feature Angle** will result in more edge lines (that is, more detail) being added to the outline display. The appropriate angle for your geometry will depend on its curvature and complexity. You can modify the **Feature Angle** until you find the value that yields the best outline display.

40.1.2.5.2. Drawing Partition Boundaries

If you have partitioned your mesh for parallel processing, you can add the display of partition boundaries to the mesh display by turning on the **Partitions** option in the Mesh Display Dialog Box (p. 4688).

40.1.2.5.3. Shrinking Faces and Cells in the Display

To distinguish individual faces or cells in the display, enlarge the space between adjacent faces or cells by increasing the **Shrink Factor** in the Mesh Display Dialog Box (p. 4688). The default value of zero produces a display in which the edges of adjacent faces or cells overlap. A value of 1 creates the opposite extreme: each face or cell is represented by a point and there is considerable space between each one. A small value such as 0.01 may be large enough to enable you to distinguish one face or cell from its neighbor. Displays with different **Shrink Factor** values are shown in Figure 40.23: Mesh Display with Shrink Factor = 0 (p. 3895) and Figure 40.24: Mesh Display with Shrink Factor = 0.01 (p. 3895). Click **Display** to see the effect of the change in **Shrink Factor**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.23: Mesh Display with Shrink Factor = 0

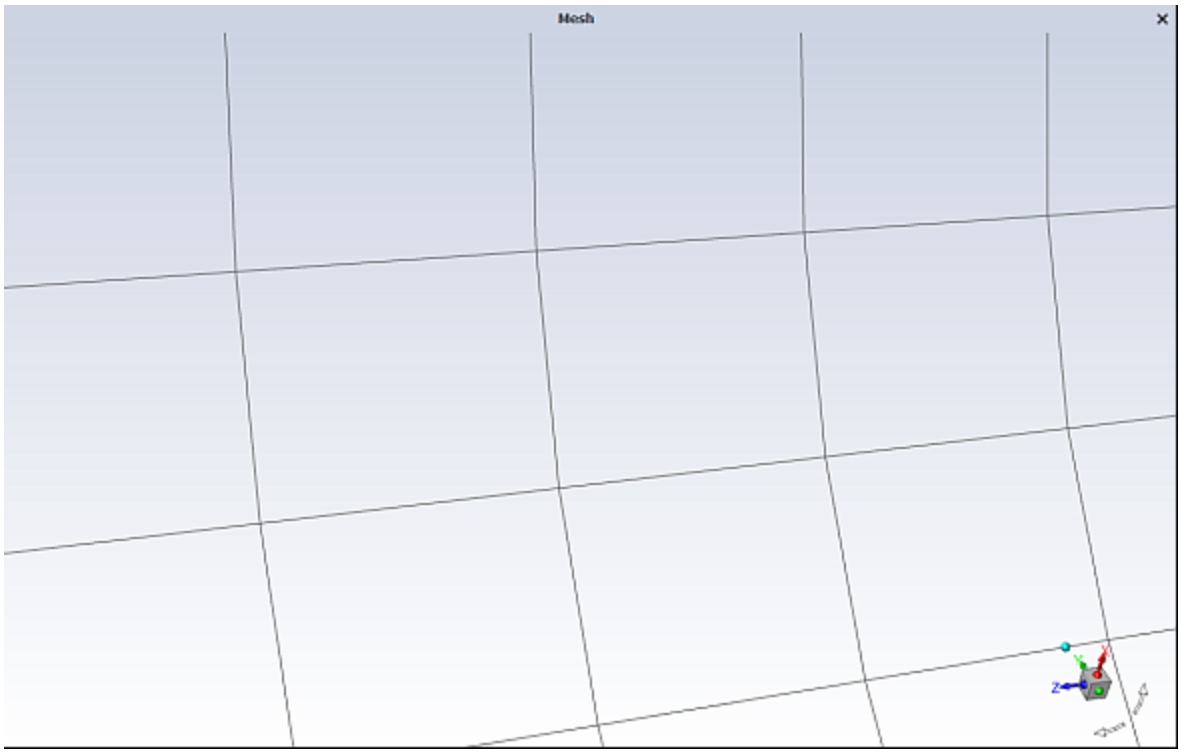
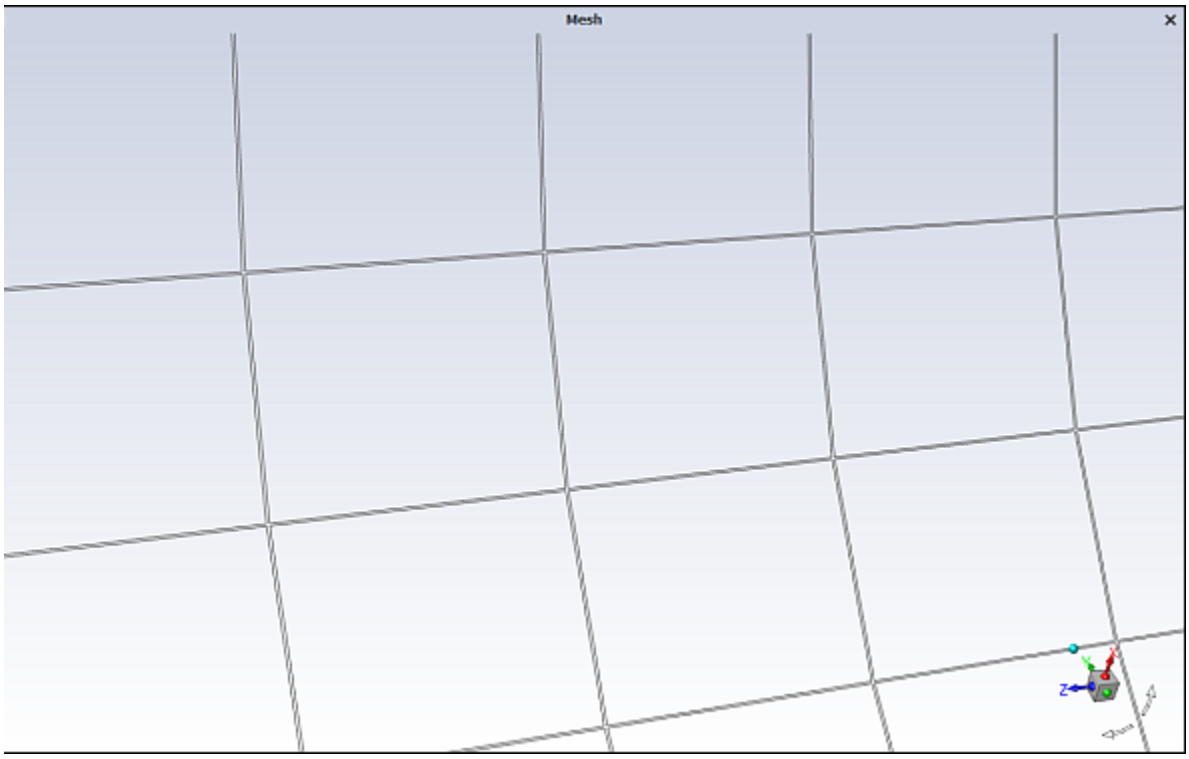


Figure 40.24: Mesh Display with Shrink Factor = 0.01



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

40.1.2.6. Creating and Using Mesh Plot Definitions

You can create named mesh plot definitions and save them for later use. See Creating and Using Contour Plot Definitions (p. 3910) for additional information on graphics object definitions.

The steps for creating and using mesh plot definitions are similar to the steps for Generating Mesh or Outline Plots (p. 3875), with the addition of a field for naming the mesh plot definition.

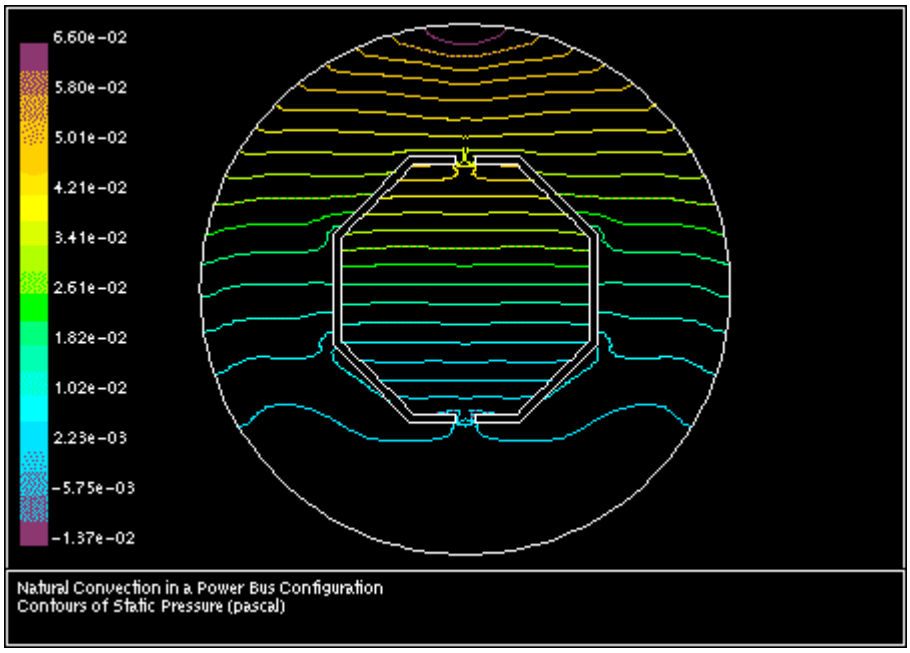
40.1.3. Displaying Contours and Profiles

Ansys Fluent enables you to plot contour lines or profiles superimposed on the physical domain. Contour lines are lines of constant magnitude for a selected variable (isotherms, isobars, and so on). A profile plot draws these contours projected off the surface along a reference vector by an amount proportional to the value of the plotted variable at each point on the surface. Sample plots are shown in Figure 40.25: Contours of Static Pressure (p. 3896) and Figure 40.26: Profile Plot of y Velocity (p. 3897).

For information about displaying contours or profiles on a surface that sweeps through the domain, see Displaying Results on a Sweep Surface (p. 3937).

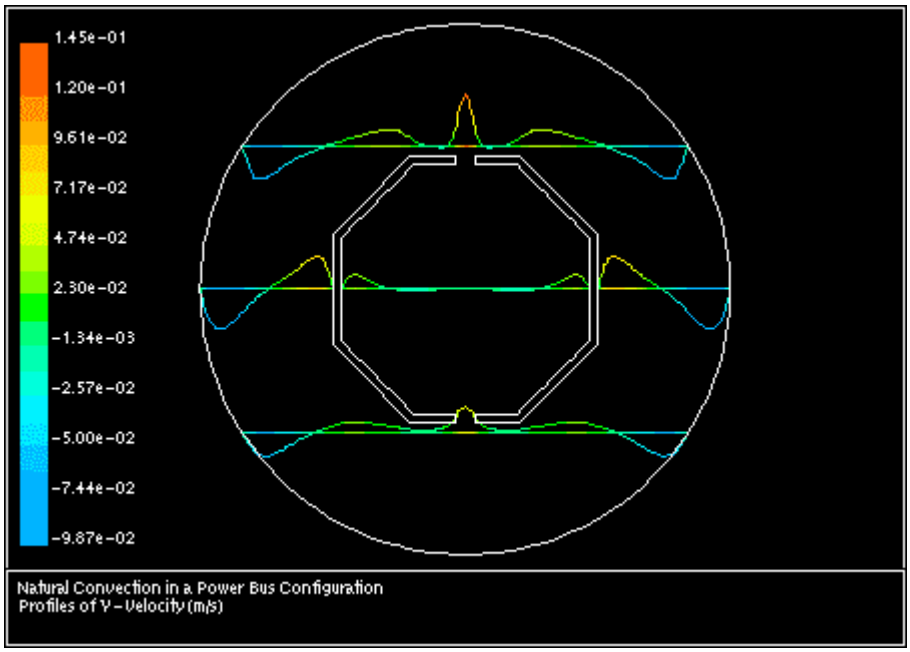
Contours are only displayed in regions where the variable is defined and relevant, which primarily applies to fluid-structure interaction cases (FSI). For example, deflection is not displayed in fluid cell zones, as this variable has no meaning in a fluid. *This only applies to predefined fields—expressions and custom field functions will display values in all cell zones.*

Figure 40.25: Contours of Static Pressure



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.26: Profile Plot of y Velocity



Note:

By default, the surfaces of contour plots do not have any reflective properties. However, you can make the surfaces appear more reflective and shiny by increasing the **Surface specularity for Contours** field under the **Appearance** branch in **Preferences**.

40.1.3.1. Quickly Coloring Surfaces by Field Variable Value

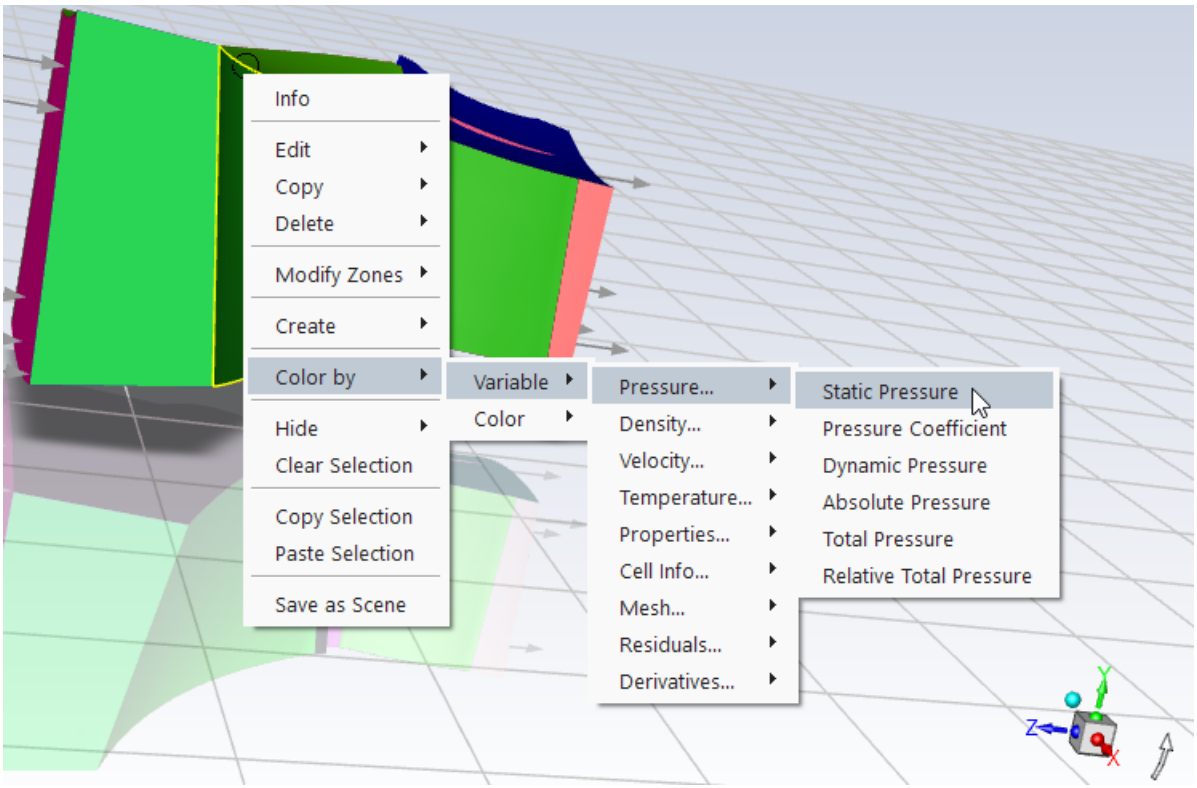
Once there is data associated with your case, you can quickly visualize the results with a contour plot in the graphics window.

To create a plot directly in the graphics window:

1. Select the desired surfaces for visualizing the results using **Ctrl** + left-click.
2. Right-click in the graphics window and select **Color by**>**Variable | Color**> desired color or variable. See Figure 40.27: Coloring Surfaces by Static Pressure (p. 3898) for an example.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.27: Coloring Surfaces by Static Pressure



3. You can color different surfaces by different colors or variables and they will be added to the same graphics window, which is similar to creating a scene (Displaying a Scene (p. 3934)).

4. You can choose to save the current graphics window display for later use by select the **Save as Scene** right-click menu option. This creates graphics objects for the items displayed in the graphics window (for example, mesh and contours) and adds them to a scene graphics object. Refer to Basic Graphics Generation (p. 3869) and Generating a Scene (p. 3935) for additional information on creating and manipulating graphics objects.

   Note that saving the current graphics window as a scene also saves the display state, which preserves the orientation, graphics effects, and other graphics settings. Refer to Controlling the Display State and Modifying the View (p. 4025) for additional information.

40.1.3.2. Generating Contour and Profile Plots

You can plot contours or profiles using the Contours Dialog Box (p. 5412) (Figure 40.28: The Contours Dialog Box (p. 3899)).

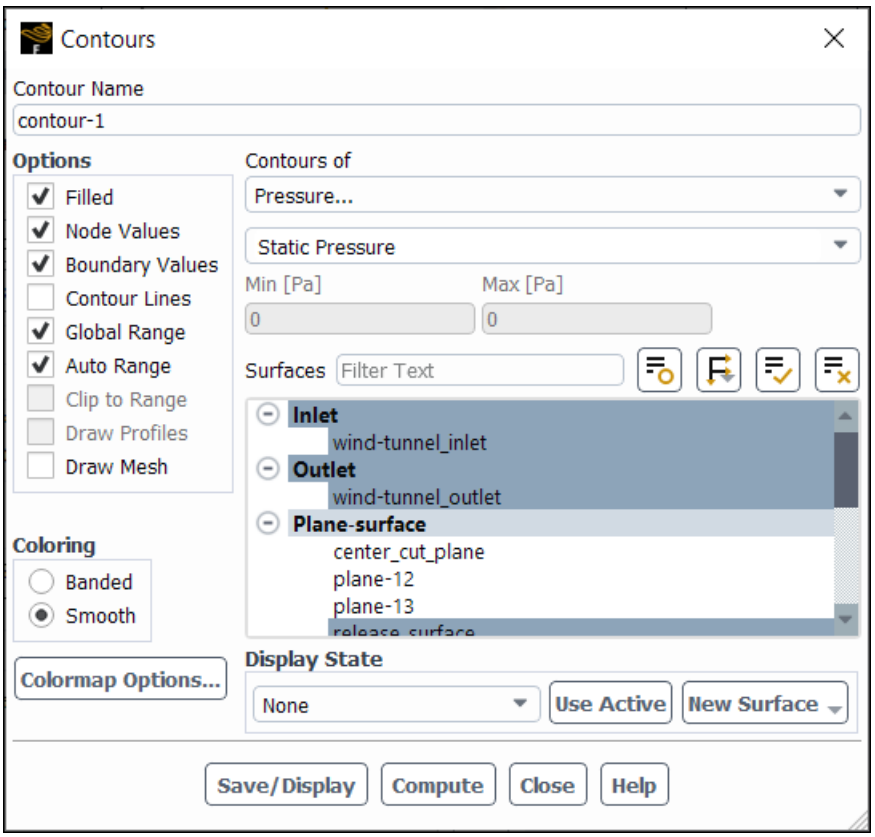
Important:

If you plan to plot profiles, you must enable the **Expose non-persistent graphics** option in the **Graphics** branch of **Preferences** (accessed by **File**>**Preferences...**), because profiles can only be displayed using the non-persistent contours object, accessed via the **Edit...** option.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Results → Graphics → Contours New...

Figure 40.28: The Contours Dialog Box



The basic steps for generating a contour or profile plot are as follows:

1. Select the variable or function to be contoured or profiled in the **Contours of** drop-down list. First select the desired category in the upper list; you may then select a related quantity in the lower list. (See Field Function Definitions (p. 4167) for an explanation of the variables in the list.)

2. Choose the surface or surfaces on which to draw the contours or profiles in the **Surfaces** list. For 2D cases, if no surface is selected, contouring or profiling is done on the entire domain. For 3D cases, you must always select at least one surface.

   If you want to select several surfaces of the same type, click and select **Surface Type** under **Group By**, which organizes the surfaces in a tree view that is grouped by surface type.

   Another shortcut is to use the **Filter Text** entry box to filter the **Surfaces** list to show only the surfaces that match the pattern you enter. For additional information on using the **Filter Text** entry box, see Filter Text Entry Boxes (p. 926).

3. Specify the number of contours or profiles in the **Levels** field. The maximum number of levels allowed is 100.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

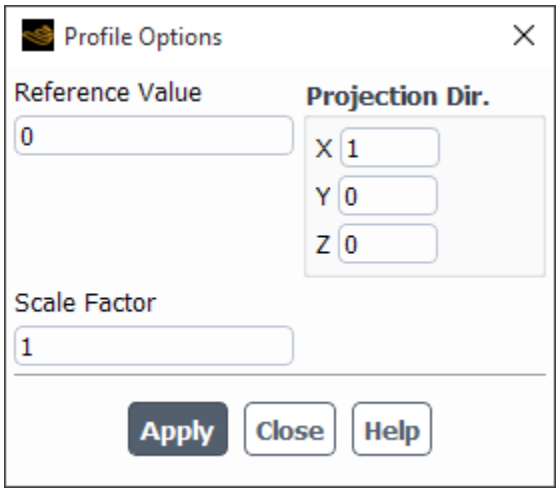
4. If you are generating a profile plot, enable the **Draw Profiles** option. In the resulting Profile Options Dialog Box (p. 5238) (Figure 40.29: The Profile Options Dialog Box (p. 3900)) you will define the profiles.

Note:

Draw Profiles is only available for the non-persistent/"global" version of the **Contours** dialog box that does not have a **Name** field. This version of the dialog box must first made available by enabling the **Expose non-persistent graphics** option in the **Graphics** branch of **Preferences** (accessed by **File**>**Preferences...**). It is then opened by:

- Right-clicking **Contours** in the Outline View tree and selecting **Edit...** (located under the **Results** branch).
- Clicking **Contours** and selecting **Edit...** under **Graphics** in the **Results** ribbon tab.

Figure 40.29: The Profile Options Dialog Box



a. Set the "zero height" reference value for the profile (**Reference Value**) and the length scale factor for projection (**Scale Factor**). Any point on the profile with a value equal to the **Reference Value** will be plotted exactly on the defining surface. Values greater than the **Reference Value** will be projected ahead of the surface (in the direction of **Projection Dir.**) and scaled by **Scale Factor**), and values less than the **Reference Value** will be projected behind the surface and scaled.

These parameters can be used to create fuller profiles when you need to display the variation in a variable that is small compared to the absolute value of the variable. Consider, for example, the display of temperature profiles when the temperature range in the domain is from 300 K to 310 K. The 10 K range in the temperature will be hard to detect when profiles are drawn using the default scaling (which will be based on the absolute magnitude of 310 K). To create a fuller profile, you can set the **Reference Value** to 300 and the profile **Scaling Factor** to 5 (for example) to magnify the display of the remaining 10 K range. In subsequent display of the profiles, the reference value of 300 will be effectively subtracted from the data before display so that the temperatures of 300 K will not be offset from the baselines. The profiles will then reflect only the variation of temperature from 300 K.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. Set the direction in which profiles are projected (**Projection Dir.**). In 2D, for example, a contour plot of pressure on the entire domain can be projected in the z direction to form a carpet plot, or a contour plot of y velocity on a sequence of y-coordinate slice lines can be projected in the y direction to form a series of velocity profiles (as shown in Figure 40.26: Profile Plot of y Velocity (p. 3897)).

c. Click **Apply** and close the Profile Options Dialog Box (p. 5238).

5. Set any of the contour and profile plot options described below.

6. Click **Save/Display** to draw the specified contours or profiles in the active graphics window.

7. (Optional) Associate a display state with this contour plot for a consistent appearance each time the contour plot is redisplayed.

   a. Set the display state. *There are two approaches*:

      - Setup the graphics window with your preferred model orientation, zoom level, graphics effects, ruler, and so on, and click **Use Active** to create a display state with these properties.

      - Select a display state from the **Display State** drop-down list (if you already created a display state that you would like to reuse for other graphics objects).

   b. Click **Save/Display** to link the selected display state with this contour plot.

   Refer to Controlling the Display State and Modifying the View (p. 4025) for additional information.

The resulting display will include the specified number of contours or profiles of the selected variable, with the magnitude on each one determined by equally incrementing between the values shown in the **Min** and **Max** fields.

40.1.3.3. Contour and Profile Plot Options

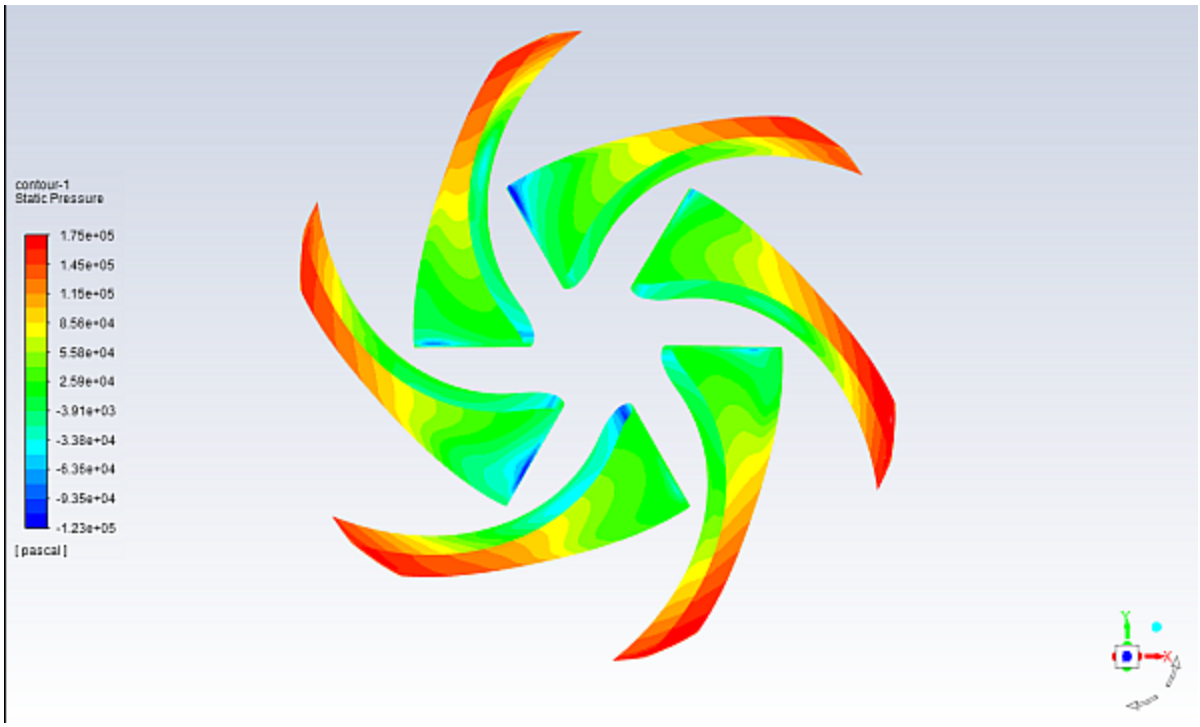
The options mentioned in the procedure above include drawing color-filled contours/profiles (instead of the default line contours/profiles), specifying a range of values to be contoured or profiled, including portions of the mesh in the contour or profile display, choosing node or cell values for display, and storing the contour or profile plot settings.

40.1.3.3.1. Drawing Filled Contours or Profiles

Color-filled contour or profile plots show a contour or profile display containing a continuous color display (see Figure 40.30: Filled Contours of Static Pressure (p. 3902)), instead of just drawing lines representing specific values. Note that a color-filled profile display is often referred to as a "carpet plot". To generate a filled contour or profile plot, enable the **Filled** option in the Contours Dialog Box (p. 5412) during step 5 in the previous section.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.30: Filled Contours of Static Pressure



To display smoothly shaded filled contours, ensure the lighting interpolation method is not set to **Flat** in the Display Options Dialog Box (p. 5262) or the Lights Dialog Box (p. 5279). You will not get smooth shading of filled contours if the **Clip to Range** option is turned on. Smooth shading of filled profiles is not available.

Note:

The **Draw Profile** option settings are not saved with the contour plot definition. Once the dialog box is closed these options will revert to being disabled.

40.1.3.3.2. Specifying the Range of Magnitudes Displayed

By default, the minimum and maximum values contoured or profiled are set based on the range of values in the entire domain. This means that the color scale will start at the smallest value in the domain (shown in the **Min** field) and end at the largest value (shown in the **Max** field). If you are plotting contours or profiles on a subset of the domain (that is, on a surface), your plot may cover only the midrange of the color scale. For example, if blue corresponds to 0 and red corresponds to 10, and the values on your surface range only from 4 to 6, your plot will contain mostly green contours or profiles, since green is the color at the middle of the default color scale.

- To focus in on a smaller range of values, so that blue corresponds to 4 and red to 6, manually reset the range to be displayed. You can also use the minimum and maximum values on the selected surfaces—rather than in the entire domain—to determine the range. Another reason to manually set the range is if you are interested only in certain values. For example, if you want to determine the region where pressure exceeds a certain value, you can increase the minimum value for display so that the lower pressure values are not displayed.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To manually set the contour/profile range, turn off the **Auto Range** option in the Contours Dialog Box (p. 5412). The **Min** and **Max** fields will become editable, and you can enter the new range of values to be displayed.

  – To show the default range at any time, click **Compute** and the **Min** and **Max** fields will be updated.

  – If you are drawing filled contours or profiles you can control whether or not values outside the prescribed **Min**/ **Max** range are displayed.

- To leave areas in which the value is outside the specified range empty (that is, draw no contours or profiles), enable the **Clip to Range** option. This is the default setting. If you turn **Clip to Range** off, values below the **Min** value will be colored with the lowest color on the color scale, and values above the **Max** value will be colored with the highest color on the color scale. Figure 40.31: Filled Contours with Clip to Range On (p. 3903) and Figure 40.32: Filled Contours with Clip to Range Off (p. 3904) show the results of enabling/disabling the **Clip to Range** option.

- To base the minimum and maximum values on the range of values on the selected surfaces, rather than in the entire domain, turn off the **Global Range** option in the Contours Dialog Box (p. 5412). The **Min** and **Max** values will be updated when you next click **Compute** or **Display**.

Figure 40.31: Filled Contours with Clip to Range On

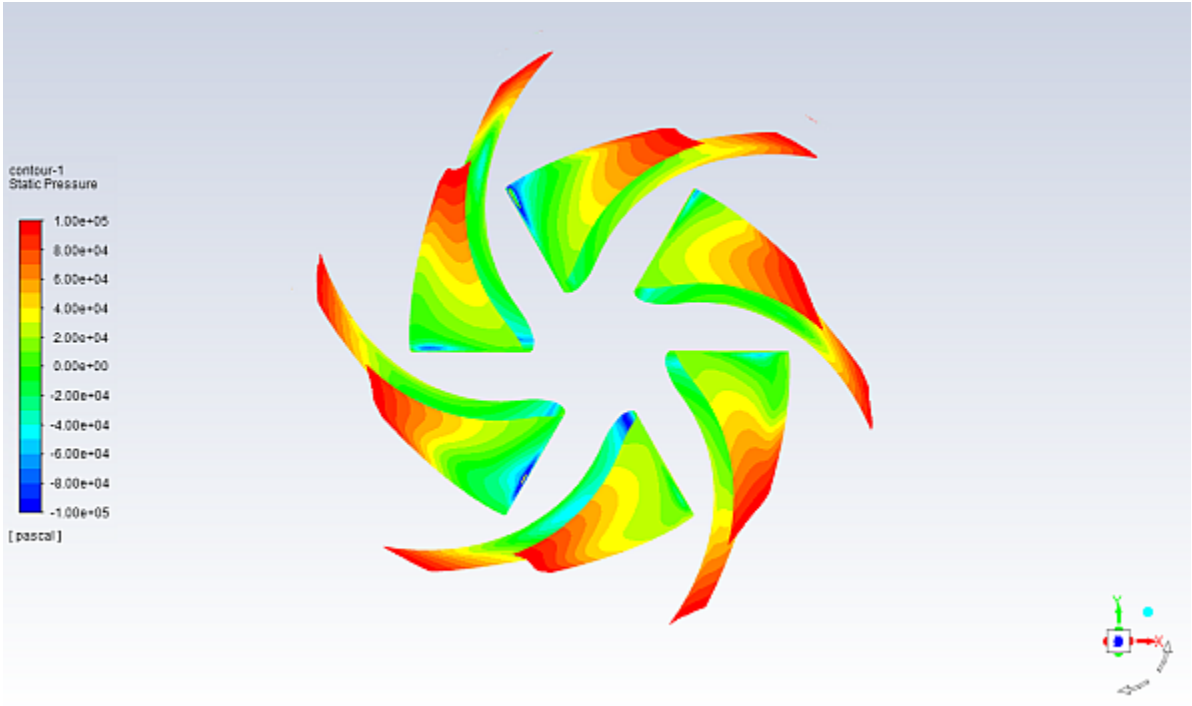
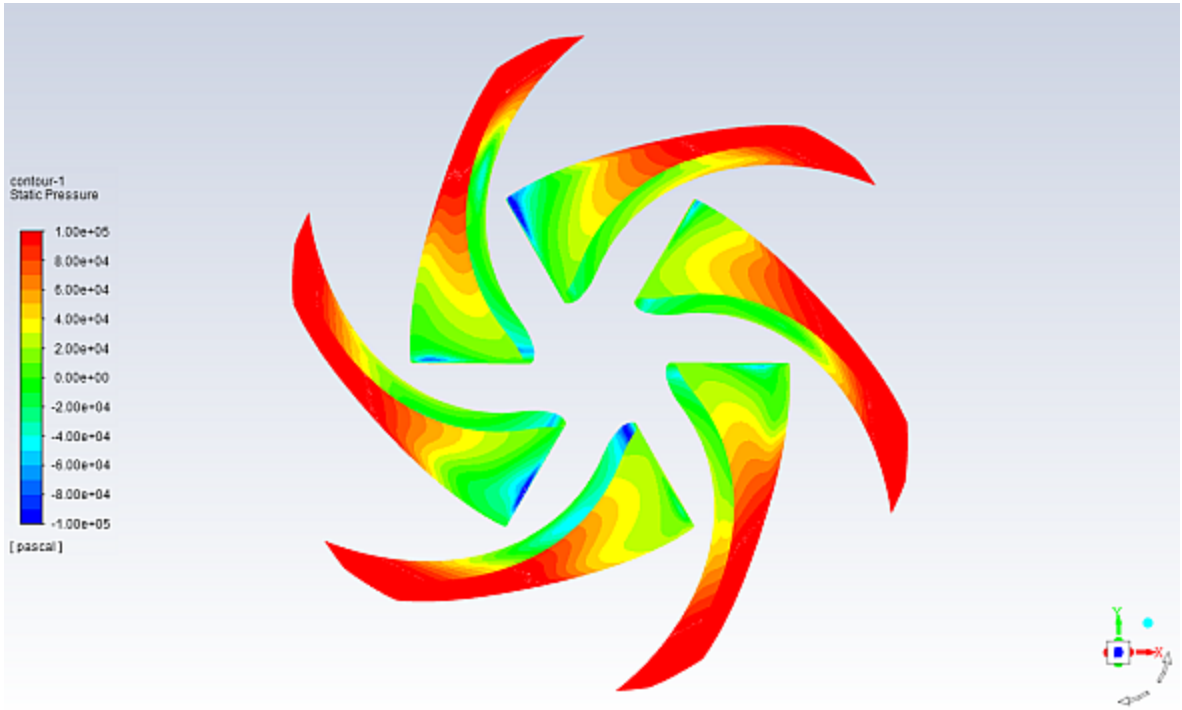


Figure 40.32: Filled Contours with Clip to Range Off



Note:

When **Global Range** is enabled, Fluent only computes and populates the range when:

1. You click **Compute**.
2. **Auto Range** is enabled.

*If **Auto Range** is disabled, the range will remain the same until you click **Compute**, even if you select a new variable for display.*

40.1.3.3.3. Including the Mesh in the Contour Plot

For some problems, especially complex 3D geometries, you may want to include portions of the mesh in your contour or profile plot as spatial reference points. For example, you may want to show the location of an inlet and an outlet along with the contours. This is accomplished by turning on the **Draw Mesh** option in the Contours Dialog Box (p. 5412). The Mesh Display Dialog Box (p. 4688) will appear automatically when you enable the **Draw Mesh** option, and you can set the mesh display parameters there. When you click **Display** in the Contours Dialog Box (p. 5412), the mesh display, as defined in the Mesh Display Dialog Box (p. 4688), will be included in the contour or profile plot.

Note:

The **Draw Mesh** option settings are not saved with the contour plot definition. Once the dialog box is closed these options will revert to being disabled.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To include the mesh with a contour plot that will be persisted with the case file, you can create a mesh plot definition and include both the mesh and contour plots in a scene. See Displaying a Scene (p. 3934) for more details.

40.1.3.3.4. Choosing Node or Cell Values and Node or Boundary Values

In Ansys Fluent you can choose to display the computed cell-center values or values that have been interpolated to the nodes. By default, the **Node Values** option is turned on, and the interpolated values are displayed. For line contours or profiles, node values are always used. To display filled contours or profiles and to display the cell values, turn off the **Node Values** option. Filled contours/profiles of node values will show a smooth gradation of color, while filled contours/profiles of cell values may show sharp changes in color from one cell to the next.

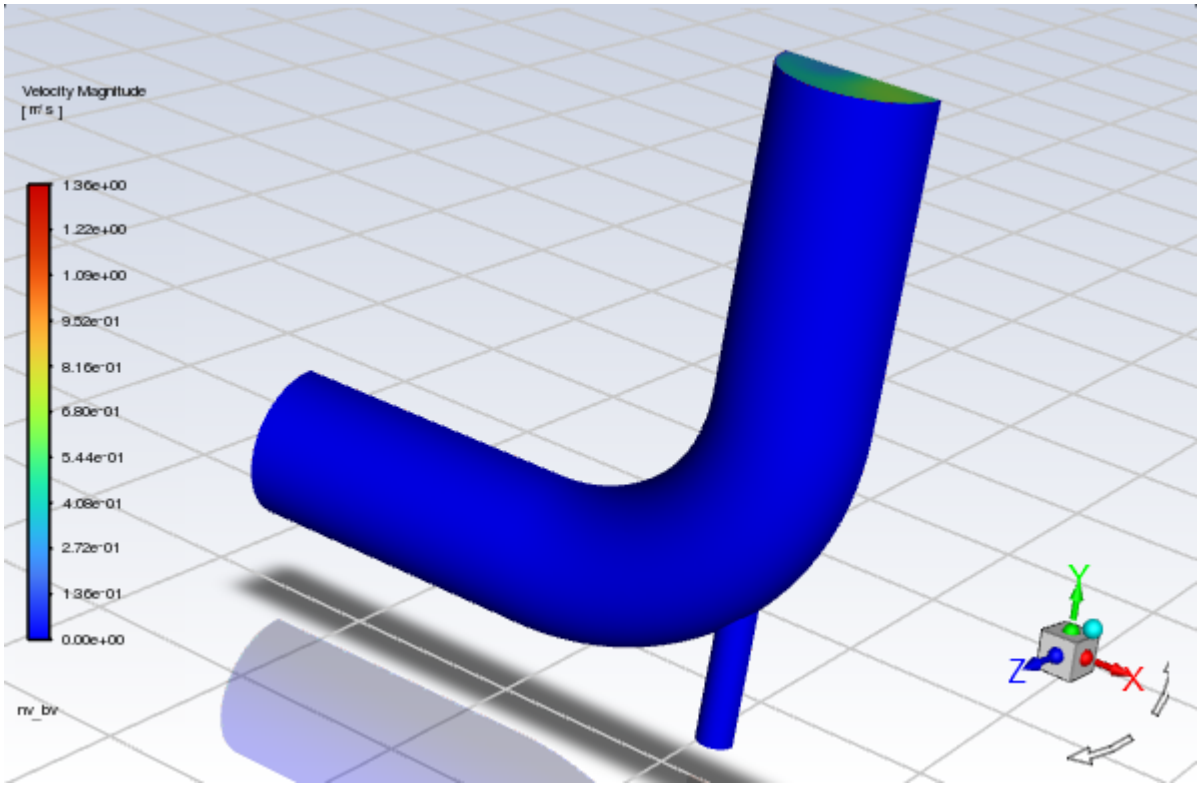
Enabling **Boundary Values** in addition to **Node Values**, displays interpolated boundary face values, when available. If **Boundary Values** are enabled with **Node Values** disabled, then the displayed values are boundary face values (not interpolated), if available. *Variables that have explicit node values available at boundaries are indicated by "bnv" in Table 42.2:**Pressure** and **Density** Categories (p. 4175) – Table 42.21:**Acoustics** Category (p. 4203)*. Note that when **Boundary Values** is enabled and velocity contours are selected for postprocessing, Fluent reports the user-specified wall velocity values for wall zones.

The displayed values are derived as follows:

- **Node Values** and **Boundary Values** *enabled* means that node values are displayed that are interpolated from boundary face values, as seen in Figure 40.33: Contours of Velocity Magnitude on a Wall + Outlet: Node Values + Boundary Values Enabled (p. 3906).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

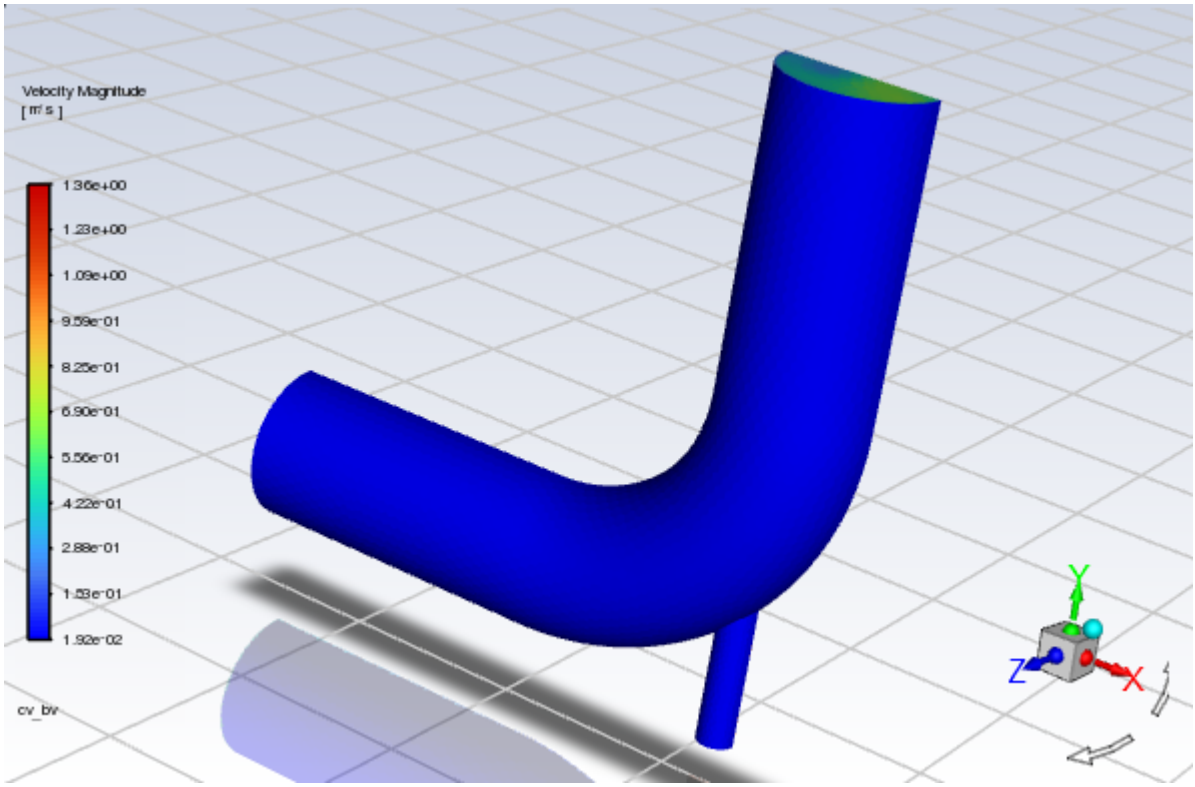
Figure 40.33: Contours of Velocity Magnitude on a Wall + Outlet: Node Values + Boundary Values Enabled



- **Node Values** *disabled* and **Boundary Values** *enabled* means that boundary face values are displayed, as seen in Figure 40.34: Contours of Velocity Magnitude: Node Values Disabled + Boundary Values Enabled (p. 3907).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

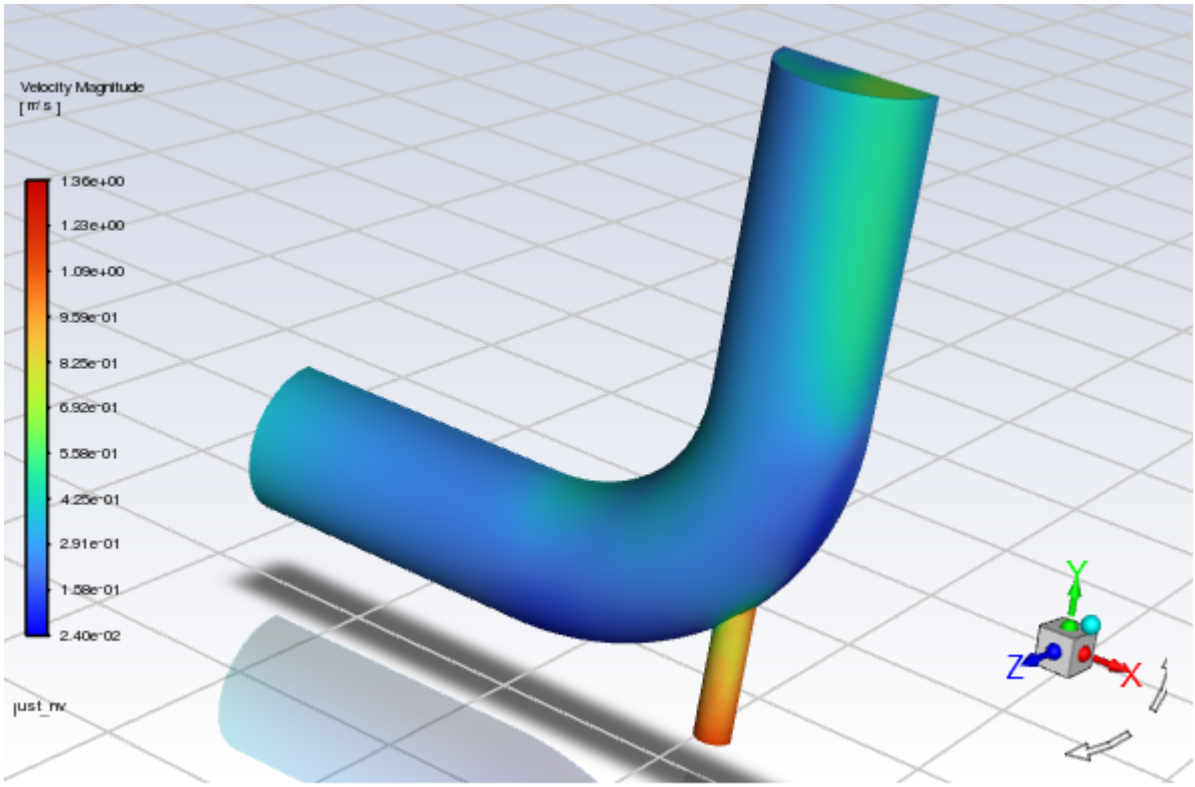
Figure 40.34: Contours of Velocity Magnitude: Node Values Disabled + Boundary Values Enabled



- **Node Values** *enabled* and **Boundary Values** *disabled* means that node values are displayed that interpolated from adjacent cell values, as seen in Figure 40.35: Contours of Velocity Magnitude: Node Values Enabled + Boundary Values Disabled (p. 3908).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

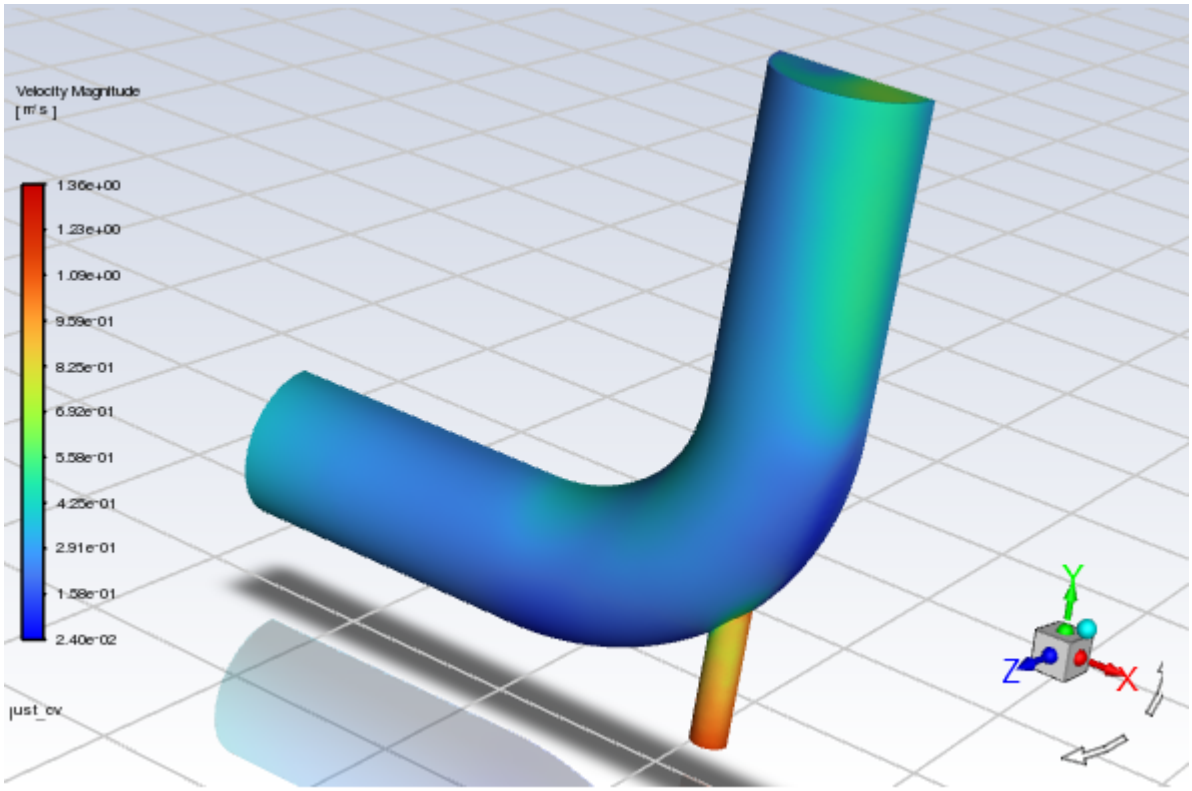
Figure 40.35: Contours of Velocity Magnitude: Node Values Enabled + Boundary Values Disabled



- **Node Values** *disabled* and **Boundary Values** *disabled* means that adjacent cell values are displayed, as seen in XREF.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.36: Contours of Velocity Magnitude: Node Values Disabled + Boundary Values Disabled



For face-only functions (for example, **Wall Shear Stress**), the cell values that are displayed for boundary zone surfaces will actually be the face values. This is only true in the case of boundary zone surfaces created for postprocessing, where the actual cell values are used for the part of the surface that lies in the interior. These face values are more accurate, as face-only functions are computed on the faces and not on the cells. For more information about cell values, see Cell Values (p. 4167).

If you are plotting contours to show the effect of a porous medium or 2D fan boundary, to depict a shock wave, or to show any other discontinuities or jumps in the plotted variable, you should use cell values; if you use node values in such cases, the discontinuity will be smeared by the node averaging for graphics and will not be shown clearly in the plot.

Important:

- If you are comparing contours at boundaries with different postprocessing methods such as reports or XY plots, then you must ensure that the **Boundary Values** option is enabled in the **Contours** dialog box to see consistent, comparable results.
- Results may differ for coupled walls that are intersecting boundaries. That is, you may see different results on a wall as compared to its wall-shadow depending on whether **Node Values** are enabled/disabled in this scenario.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

40.1.3.3.5. Storing Contour Plot Settings

For frequently used combinations of contour variables and options, you can store the information needed to generate the contour plot as follows:

1. In the Contours Dialog Box (p. 5412), specify a **Setup** number.
2. Set up the desired information in the Contours Dialog Box (p. 5412).
3. Click **Display**.

   The settings for **Options**, **Contours of**, **Min**, **Max**, and **Surfaces** will be saved.

You can then change the **Setup** number to an unused value (that is, an ID for which no information has been saved) and generate a different contour plot.

To generate a plot using the saved setup information:

1. Change the **Setup** number back to the value for which you saved contour information.
2. Click **Display**.

You can save up to 10 different setups.

Important:

Note that the number of contour **Levels**, the surfaces selected for display in the Mesh Display Dialog Box (p. 4688) (when the **Draw Mesh** option is activated), and the settings for profiles in the Profile Options Dialog Box (p. 5238) (when the **Draw Profiles** option is activated) will *not* be saved in the **Setup**, nor will the **Setup** be saved in the case file.

40.1.3.4. Creating and Using Contour Plot Definitions

You can create named contour plot definitions and save them for later use. This could be useful for demonstration purposes, scene creation, or for postprocessing the results of your simulation when varying the settings between runs (to name a few examples).

The number of contour plot definitions is not limited.

To create a contour plot definition:

Results → Graphics → Contours → New...

1. In the Contours Dialog Box (p. 5412), in the **Contour Name** text entry box, enter the name for your contour plot definition (or accept the default name `contour-`*`id`*).

Important:

Contour and vector plot definitions must have unique names.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

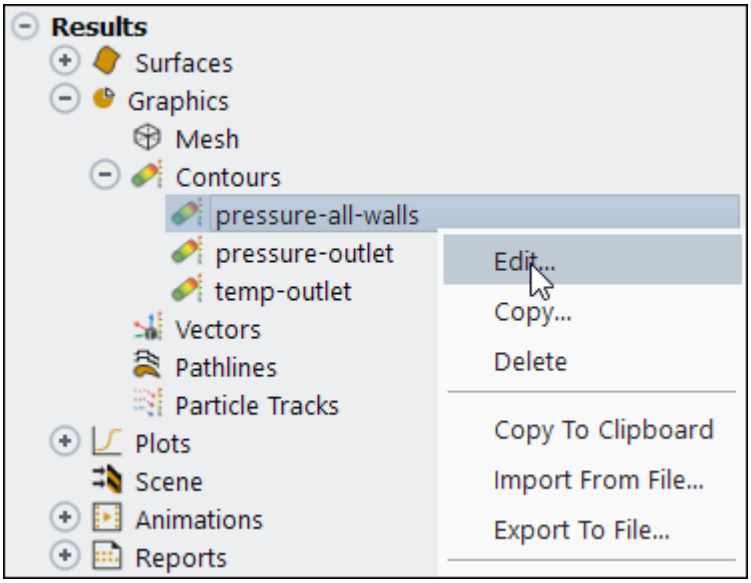
2. Set contour parameters and options.

   The **Draw Mesh** *and* **Draw Profile** *option settings are not saved with the contour plot definition. Once the dialog box is closed these options will revert to being disabled.*

3. Click **Save/Display**.

   The specified contour will be displayed in the active graphics window, and the new contour plot definition with the name you have entered will appear in the tree under **Contours**.

Once the contour plot definition is created, you can perform the following actions using the right-click menu commands:



- Edit the existing contour plot definition.
- Copy the contour plot definition.

  When creating a copy of the existing contour, you can modify contour settings in the **Contours** dialog box that will open.

- Delete the selected contour plot definition.
- Display the contour in the graphics window.
- Add the selected contour to the plot in the graphics window.

  Note that while **Display** replaces the current contents of the graphics window, the **Add to** menu option preserves the current graphics window contents.

Note:

- You can move the colormap of contour plot definitions by left-clicking and dragging it to the desired location. The colormap can also be resized by dragging the corners of the box that appears when you hover over the colormap. *The colormap location is not saved with the contour plot definition.*

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For 2D cases, Fluent automatically creates a new surface when you click **Save/Display** in the **Contours** dialog box, if you have not selected any surfaces for display. New surfaces are only created for fluid zones that do not already have a surface defined.

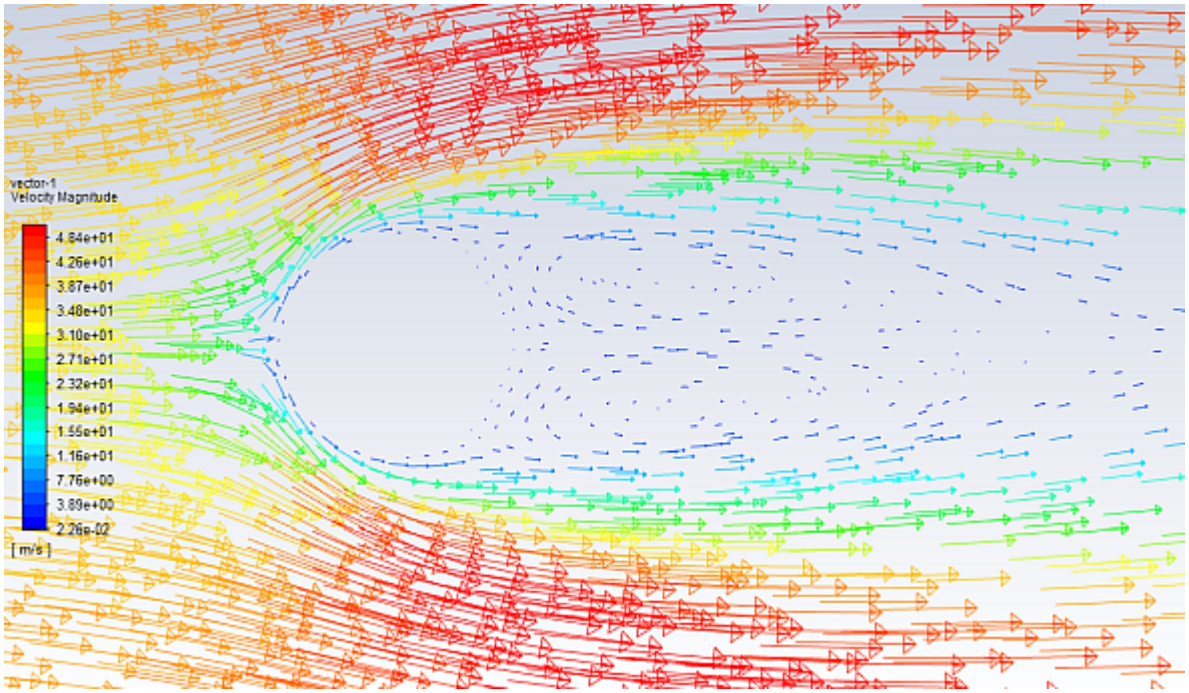
These commands are also available through the text user interface (`display/objects`).

40.1.4. Displaying Vectors

You can draw vectors in the entire domain, or on selected surfaces. By default, one vector is drawn at the center of each cell (or at the center of each facet of a data surface), with the length and color of the arrows representing the velocity magnitude (Figure 40.37: Velocity Vector Plot (p. 3912)). The spacing, size, and coloring of the arrows can be modified, along with several other vector plot settings. Velocity vectors are the default, but you can also plot vector quantities other than velocity. Note that cell-center values are always used for vector plots; you cannot plot node-averaged values.

For information about displaying vectors on a surface that sweeps through the domain, see Displaying Results on a Sweep Surface (p. 3937).

Figure 40.37: Velocity Vector Plot



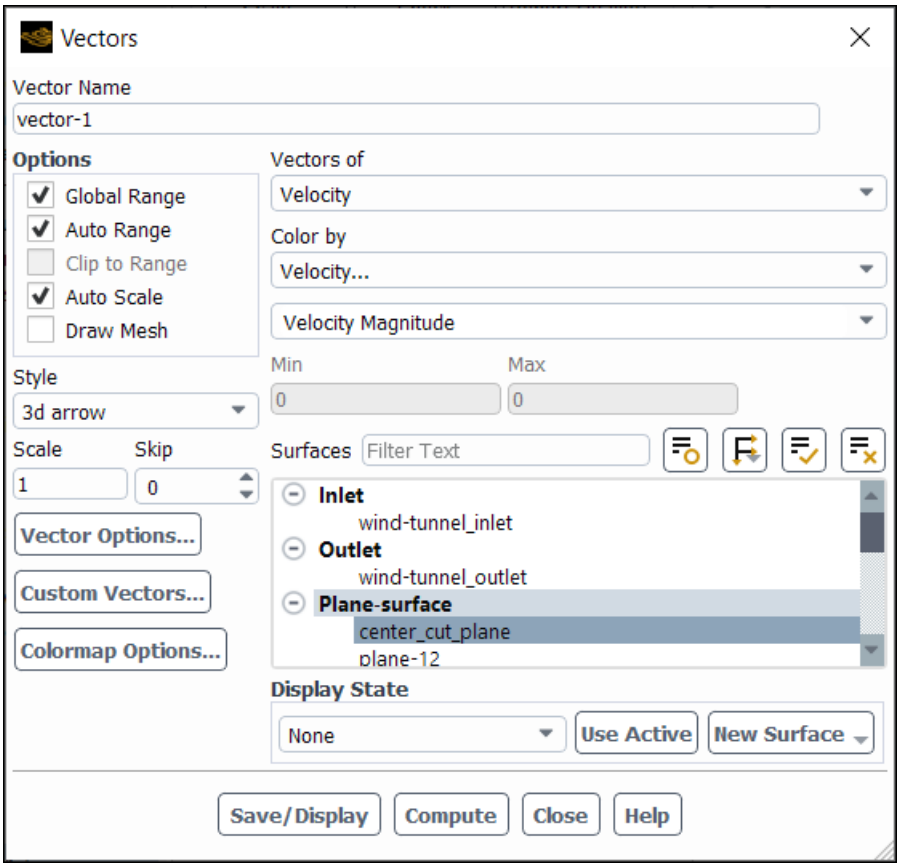
40.1.4.1. Generating Vector Plots

You can plot vectors using the Vectors Dialog Box (p. 5655) (Figure 40.38: The Vectors Dialog Box (p. 3913)).

Results → Graphics → Vectors → New...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.38: The Vectors Dialog Box



The procedure for generating a vector plot is as follows:

1. In the **Vectors of** drop-down list, select the vector quantity to be plotted. By default, only velocity and relative velocity are available, but you can create your own custom vectors as described in Creating and Managing Custom Vectors (p. 3919). Additional vector options are available for adjoint solver results, as described in Field Data (p. 4543).

2. In the **Surfaces** list, choose the surface(s) on which you want to display vectors. If you want to display vectors on the entire domain, select none of the surfaces in the list.

   If you want to select several surfaces of the same type, click and select **Surface Type** under **Group By**, which organizes the surfaces in a tree view that is grouped by surface type.

   Another shortcut is to use the **Filter Text** entry box to filter the **Surfaces** list to show only the surfaces that match the pattern you enter. For additional information on using the **Filter Text** entry box, see Filter Text Entry Boxes (p. 926).

3. Set any of the vector plot options as described in Vector Plot Options (p. 3914).

4. Click **Save/Display** to draw the vectors in the active graphics window.

5. (Optional) Associate a display state with this vector plot for a consistent appearance each time the vector plot is redisplayed.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. Set the display state. *There are two approaches*:

   - Setup the graphics window with your preferred model orientation, zoom level, graphics effects, ruler, and so on, and click **Use Active** to create a display state with these properties.
   - Select a display state from the **Display State** drop-down list (if you already created a display state that you would like to reuse for other graphics objects).

b. Click **Save/Display** to link the selected display state with this vector plot.

Refer to Controlling the Display State and Modifying the View (p. 4025) for additional information.

40.1.4.2. Displaying Relative Velocity Vectors

If you are solving your problem using one or more moving reference frames or moving meshes, you will have the option to display either the absolute vectors or the relative vectors. If you select **Velocity** (the default) in the **Vectors of** list, the vectors will be drawn based on the absolute, stationary reference frame. If you select **Relative Velocity**, the vectors will be drawn based on the reference frame of the **Reference Zone** in the Reference Values Task Page (p. 5154). See Setting the Reference Zone (p. 4161) for details. (If you are modeling a single moving reference frame, you need not specify the **Reference Zone**; the vectors will be drawn based on the moving reference frame.)

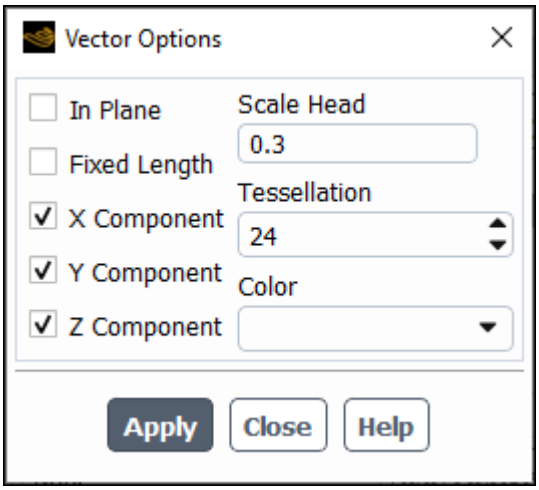
40.1.4.3. Vector Plot Options

The options mentioned in the procedure above include scaling the vector arrows, skipping the display of some vectors, displaying vectors in the plane of the data surface, displaying fixed-length or fixed-color vectors, displaying directional components of the vectors, specifying a range of values to be displayed, adjusting the tessellation of 3D vectors, coloring the vectors by a different scalar field, including portions of the mesh in the vector display, and changing the style of the arrows or the scale of the arrowheads.

The most common options are set in the Vectors Dialog Box (p. 5655), and others are set in the Vector Options Dialog Box (p. 5239) (Figure 40.39: The Vector Options Dialog Box (p. 3915)), which you can open by clicking **Vector Options...** in the Vectors Dialog Box (p. 5655).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.39: The Vector Options Dialog Box



40.1.4.3.1. Scaling the Vectors

By default, vectors are scaled automatically so that the arrows overlap minimally when no vectors are skipped. For instructions on thinning the vector display, see "Thinning" Pathlines (p. 3926). With the **Auto Scale** option, you can modify the **Scale** factor (which is set to 1 by default) to increase or decrease the vector scale from the default "auto scale". The main advantage of autoscaling is that the vector display with a scale factor of 1 will always be appropriate, regardless of the size of the domain, giving you a better starting point for fine-tuning the vector scale.

If you turn off the **Auto Scale** option, the vectors will be drawn at their actual sizes scaled by the scale factor (**Scale**, which is set to 1 by default). The "actual" size of a vector is the magnitude of the vector variable (velocity, by default) at the point where it is drawn. A vector drawn at a point where the velocity magnitude is 100 m/s is drawn 100 m long, whether the domain is 0.1 m or 1000 m. You can modify the vector scale by changing the value of **Scale** in the Vectors Dialog Box (p. 5655) until the size of the vectors (that is, the actual size multiplied by **Scale**) is satisfactory.

Note:

Vectors may appear larger than they did for the same values in previous releases. You can control the size of vectors using the **Scale** setting in the **Vectors** dialog box.

40.1.4.3.2. Skipping Vectors

If your vector display is difficult to understand because there are too many arrows displayed, you can "thin out" the vectors by changing the **Skip** value in the Vectors Dialog Box (p. 5655). By default, **Skip** is set to 0, indicating that a vector will be drawn for each cell in the domain or for each face on the selected surface (for example, n vectors). If you increase **Skip** to 1, every other vector will be displayed, yielding $n/2$ vectors. If you increase **Skip** to 2, every third vector will be displayed, yielding $n/3$ vectors, and so on. The order of faces on the selected surface (or cells in the domain) will determine which vectors are skipped or drawn; therefore adaption will change the appearance of the vector display when a nonzero **Skip** value is used.

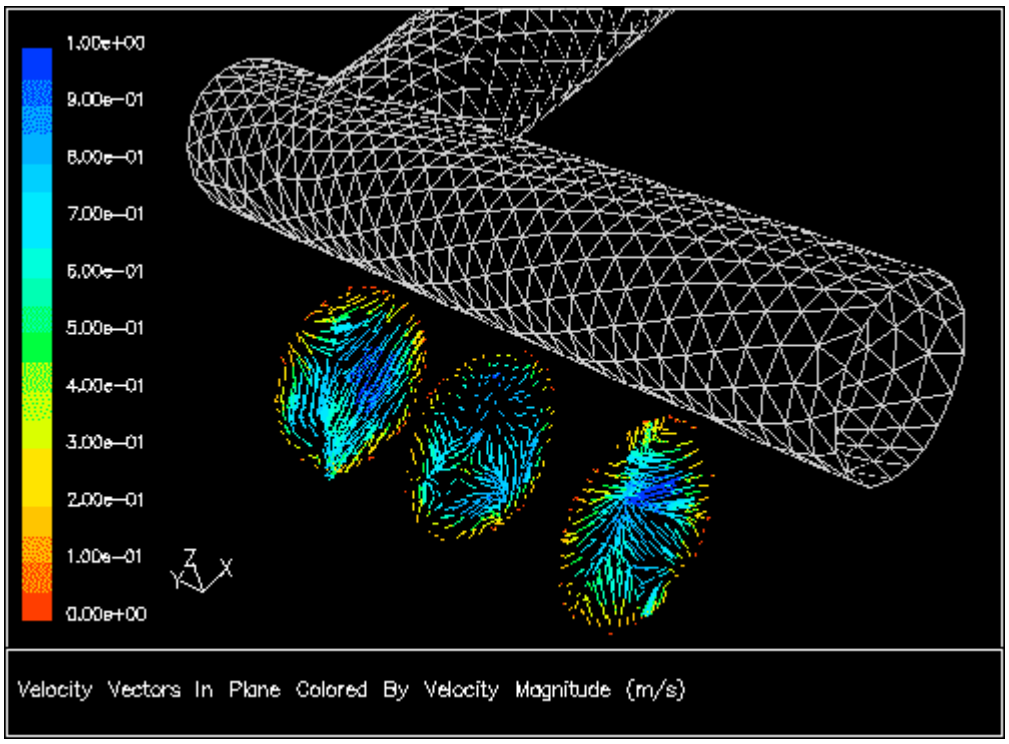
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

40.1.4.3.3. Drawing Vectors in the Plane of the Surface

For some problems, you may be interested in visualizing velocity (or other vector) components that are normal to the flow. These "secondary flow" components are usually much smaller than the components in the flow direction and are difficult to see when the flow direction components are also visible. To easily view the normal flow components, you can enable the **In Plane** option in the Vector Options Dialog Box (p. 5239). When this option is on, Ansys Fluent will display only the vector components in the plane of the surface selected for display. If the selected surface is a cross-section of the flow domain, you will be displaying the components normal to the flow.

Figure 40.40: Velocity Vectors Generated Using the In Plane Option (p. 3916) shows velocity vectors generated using the **In Plane** option. Note that these vectors have been translated outside the domain, as described in Transforming Geometric Objects in a Scene (p. 4046), so that they can be seen more easily.

Figure 40.40: Velocity Vectors Generated Using the In Plane Option



40.1.4.3.4. Displaying Fixed-Length Vectors

By default, the length of a vector is proportional to its velocity magnitude. If you want all of the vectors to be displayed with the same length, you can enable the **Fixed Length** option in the Vector Options Dialog Box (p. 5239). To modify the vector length, adjust the value of the **Scale** factor in the Vectors Dialog Box (p. 5655).

40.1.4.3.5. Displaying Vector Components

All Cartesian components of the vectors are drawn by default, so that the arrow points along the resultant vector in physical space. However, sometimes one of the components, say, the x component, is relatively large. In such cases, you may want to suppress the x component and scale up the vectors, in order to visualize the smaller y and z components. To suppress one or more

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

of the vector components, turn off the appropriate button(s) (**X**, **Y**, or **Z Component**) in the Vector Options Dialog Box (p. 5239).

40.1.4.3.6. Specifying the Range of Magnitudes Displayed

By default, the minimum and maximum vectors included in the vector display are set based on the range of vector-variable (velocity, by default) magnitudes in the entire domain. If you want to focus in on a smaller range of values, you can restrict the range to be displayed. The color scale for the vector display will change to reflect the new range of values. (You can also use the minimum and maximum values on the selected surfaces—rather than on the entire domain—to determine the range, or change the scalar field by which the vectors are colored from velocity magnitude to any other scalar, as described below.)

To manually set the range of velocity magnitudes (or the range of whatever scalar field is selected in the **Color by** drop-down list), turn off the **Auto Range** option in the Vectors Dialog Box (p. 5655). The **Min** and **Max** fields will become editable, and you can enter the new range of values to be displayed. For example, if you want to display velocity vectors only in regions where the velocity magnitude exceeds 150 m/s but is less than 300 m/s, you will change the value of **Min** to 150 and the value of **Max** to 300. Similarly, if you are coloring the vectors by static pressure, you can choose to display velocity vectors only in regions where the pressure is within a specified range. To show the default range at any time, click **Compute** and the **Min** and **Max** fields will be updated.

When you restrict the range of vectors displayed, you can also control whether or not values outside the prescribed **Min**/ **Max** range are displayed. To leave areas in which the value is outside the specified range empty (that is, draw no vectors), enable the **Clip to Range** option. This is the default setting. If you turn **Clip to Range** off, values below the **Min** value will be colored with the lowest color on the color scale, and values above the **Max** value will be colored with the highest color on the color scale. This feature is the same as the one available for displaying filled contours (see Figure 40.31: Filled Contours with Clip to Range On (p. 3903) and Figure 40.32: Filled Contours with Clip to Range Off (p. 3904)).

You can also choose to base the minimum and maximum values on the range of values on the selected surfaces, rather than the entire domain. To do this, turn off the **Global Range** option in the Vectors Dialog Box (p. 5655). The **Min** and **Max** values will be updated when you next click **Compute** or **Display**.

40.1.4.3.7. Changing the Scalar Field Used for Coloring the Vectors

If you want to color the vectors by a scalar field other than velocity magnitude (the default), you can select a different variable or function in the **Color by** drop-down list. Select the desired category in the upper list, and then choose one of the related quantities from the lower list. If you choose static pressure, for example, the length of the vectors will still correspond to the velocity magnitude, but the color of the vectors will correspond to the value of pressure at each point where a vector is drawn.

40.1.4.3.8. Controlling 3D Vector Tessellation for Performance and Appearance

You can adjust the tessellation of the **3d arrow** and **3d arrowhead** vector plot **Styles**. This control has a valid range of `0-74`.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tessellation is accessed by clicking **Vector Options...** in the **Vectors** dialog box. Tessellation affects how round the edges of the cylinder and arrowhead appear. Lower values appear less round, which comes with some performance benefit in rendering the vector plot.

Figure 40.41: 3D Arrow Vectors with Tessellation of 24

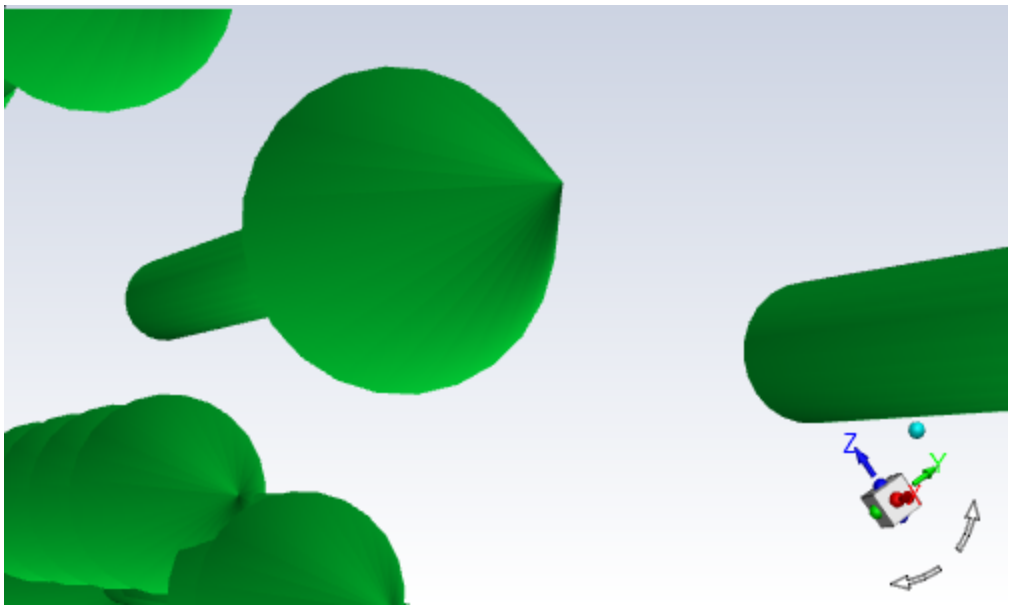
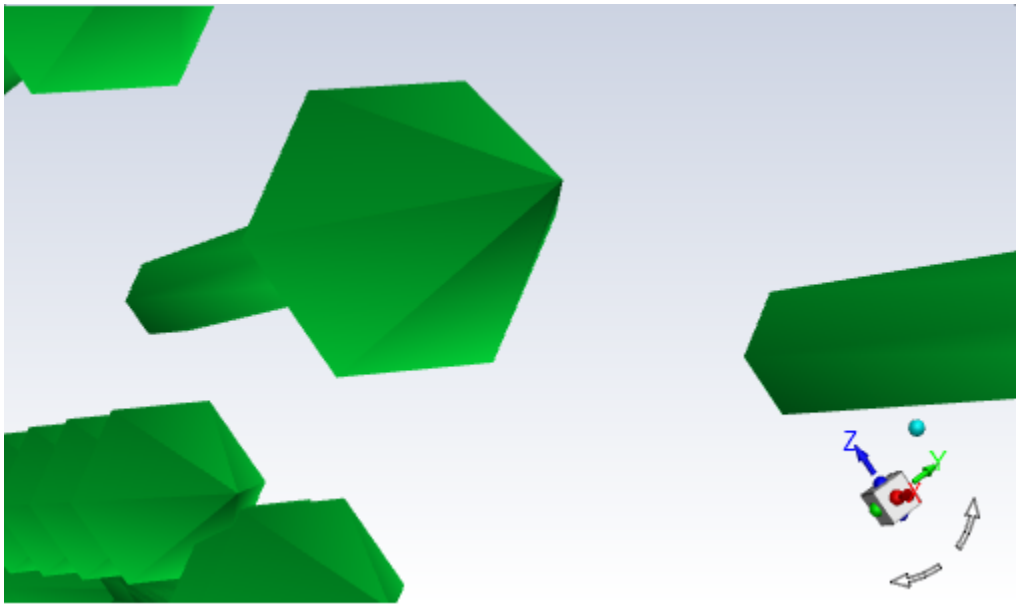


Figure 40.42: 3D Arrow Vectors with Tessellation of 6



40.1.4.3.9. Displaying Vectors Using a Single Color

If you want all of the vectors to be the same color, you can select the color to be used in the **Color** drop-down list in the Vector Options Dialog Box (p. 5239). If no color is selected (that is, if you choose the empty space at the top of the drop-down list—the default selection), the vector color will be determined by the **Color by** field specified in the Vectors Dialog Box (p. 5655). Single color vectors are useful in displays that overlay contours and vectors.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

40.1.4.3.10. Including the Mesh in the Vector Plot

For some problems, especially complex 3D geometries, you may want to include portions of the mesh in your vector plot as spatial reference points. For example, you may want to show the location of an inlet and an outlet along with the vectors. This is accomplished by turning on the **Draw Mesh** option in the Vectors Dialog Box (p. 5655). The Mesh Display Dialog Box (p. 4688) will appear automatically when you enable the **Draw Mesh** option, and you can set the mesh display parameters there. When you click **Display** in the Vectors Dialog Box (p. 5655), the mesh display, as defined in the Mesh Display Dialog Box (p. 4688), will be included in the vector plot.

Note:

The **Draw Mesh** option settings are not saved with the vector plot definition. Once the dialog box is closed these settings will revert to being disabled.

To include the mesh with a vector plot that will be persisted with the case file, you can create a mesh plot definition and include both the mesh and vector plots in a scene. See Displaying a Scene (p. 3934) for more details.

40.1.4.3.11. Changing the Arrow Characteristics

There are seven different vector arrow styles available. Choose **3d arrow**, **3d arrowhead**, **cone**, **filled-arrow**, **arrow**, **harpoon**, or **headless** in the **Style** drop-down list in the Vectors Dialog Box (p. 5655).

If you choose a vector arrow style that includes heads, you can control the size of the arrowhead by modifying the **Scale Head** value in the Vector Options Dialog Box (p. 5239).

40.1.4.4. Creating and Managing Custom Vectors

In addition to the velocity vector quantity provided by Ansys Fluent, you can also define your own custom vectors to be plotted. This capability is available with the Custom Vectors Dialog Box (p. 5240).

Any custom vectors that you define will be saved in the case file the next time that you save it. You can also save your custom vectors to a separate file, so that they can be used with a different case file.

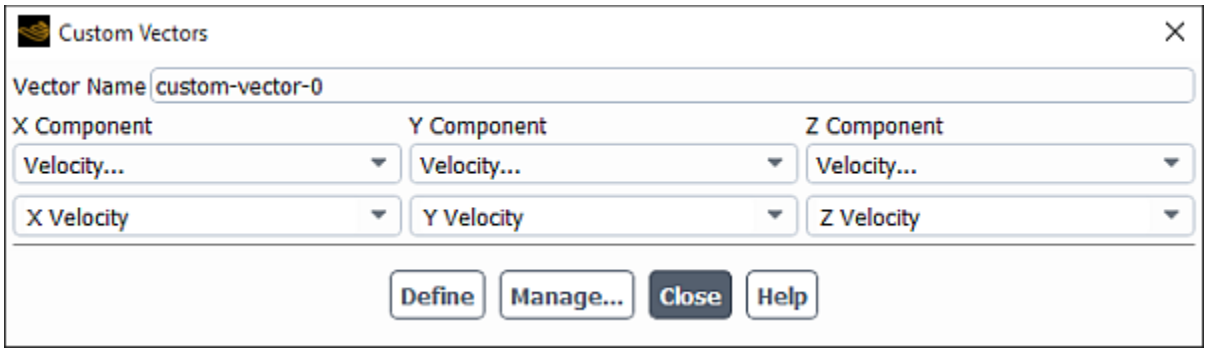
40.1.4.4.1. Creating Custom Vectors

To create your own custom vector, you will use the Custom Vectors Dialog Box (p. 5240) (Figure 40.43: The Custom Vectors Dialog Box (p. 3920)). This dialog box enables you to define custom vectors based on existing quantities. Any vectors that you define will be added to the **Vectors of** list in the Vectors Dialog Box (p. 5655).

To open the Custom Vectors Dialog Box (p. 5240), click **Custom Vectors...** in the Vectors Dialog Box (p. 5655).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.43: The Custom Vectors Dialog Box



The steps for creating a custom vector are as follows:

1. Specify the name of the custom vector in the **Vector Name** field.

   Important:

   Do not specify a name that is already used for a standard vector (for example, `velocity` or `relative-velocity`).

2. Select the variable or function for the *x* component of the vector in the **X Component** drop-down list. First select the desired category in the upper list; you may then select a related quantity in the lower list. For an explanation of the variables in the list, see Field Function Definitions (p. 4167).

3. Repeat the step above to select the variable or function for the y component (and, in 3D, the z component) of the custom vector.

   Important:

   You can use the **Custom Vectors** option to plot vectors in solid cell zones. The scalars that are selected in the *x*, *y* (and, in 3D, the *z*) components, and that are valid in solid regions, will have vector plots displayed in the solid cell zones. Note that if a vector has no valid components in the solid region, then that vector will not be plotted in the solid region. However, if at least one component of the vector is valid in the solid region, then only that component of the vector will be plotted.

4. Click **Define**.

40.1.4.4.2. Manipulating, Saving, and Loading Custom Vectors

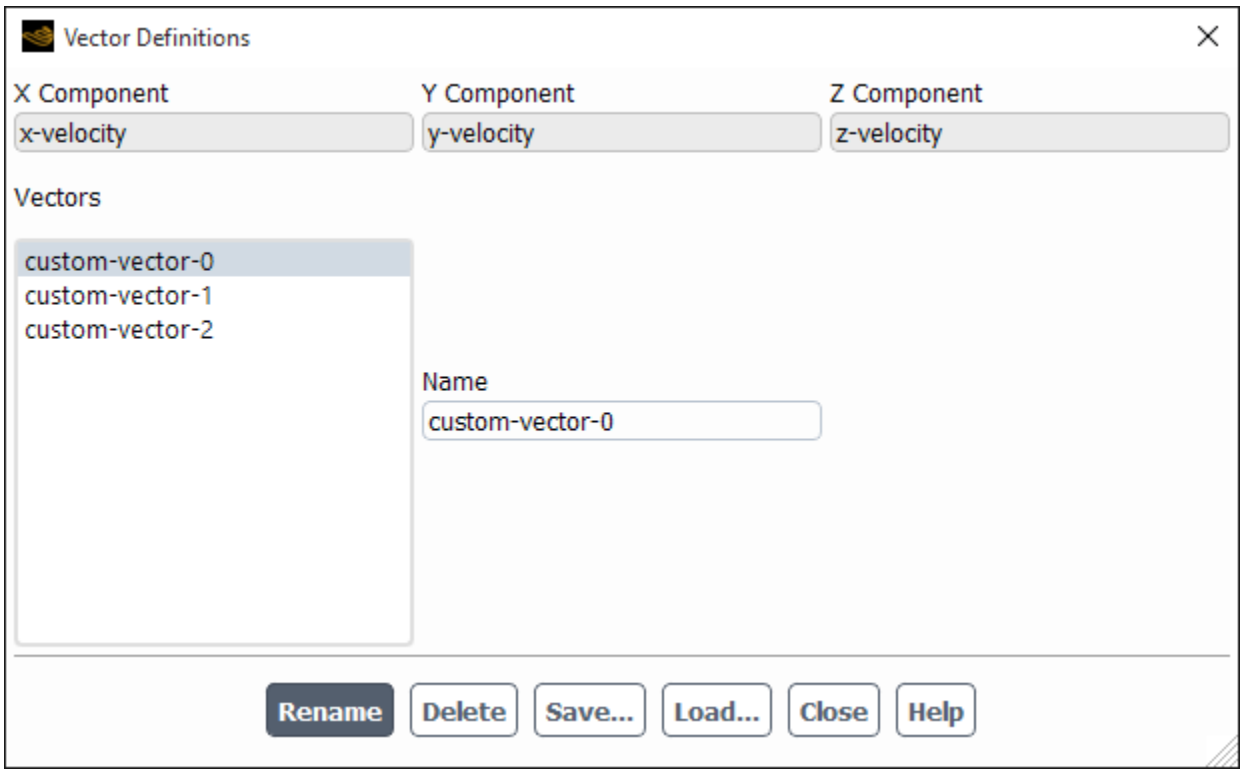
Once you have defined your vectors, you can manipulate them using the Vector Definitions Dialog Box (p. 5241) (Figure 40.44: The Vector Definitions Dialog Box (p. 3921)). You can display a vector definition to be sure that it is correct, delete the vector if you decide that it is incorrect and must be redefined, or give the vector a new name. You can also save custom vectors to a file or read

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

them from a file. The custom vector file enables you to transfer custom vectors between case files.

To open the Vector Definitions Dialog Box (p. 5241), click **Manage...** in the Custom Vectors Dialog Box (p. 5240).

Figure 40.44: The Vector Definitions Dialog Box



The following actions can be performed in the Vector Definitions Dialog Box (p. 5241):

- To check the definition of a vector, select it in the **Vectors** list. Its definition will be displayed in the **X Component**, **Y Component**, and **Z Component** fields at the top of the dialog box. This display is for informational purposes only; you cannot edit it. If you want to change a vector definition, you must delete the vector and define it again in the Custom Vectors Dialog Box (p. 5240).
- To delete a vector, select it in the **Vectors** list and click **Delete**.
- To rename a vector, select it in the **Vectors** list, enter a new name in the **Name** field, and click **Rename**.

  Important:

  Do not specify a name that is already used for a standard vector (for example, `velocity` or `relative-velocity`).

- To save all the vectors in the **Vectors** list to a file, click **Save...** and specify the file name in The Select File Dialog Box (p. 931).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To read custom vectors from a file that you saved as described above, click **Load** and specify the file name in The Select File Dialog Box (p. 931). (Custom vectors are valid Scheme functions, and can also be loaded with the **File/Read/Scheme...** ribbon tab item, as described in Reading Scheme Source Files (p. 972).)

40.1.4.5. Creating and Using Vector Plot Definitions

You can create named vector plot definitions and save them for later use. See Creating and Using Contour Plot Definitions (p. 3910) for additional information on plot definitions.

The steps for creating and using vector plot definitions are similar to the steps for Generating Vector Plots (p. 3912), with the addition of a field for naming the vector plot definition.

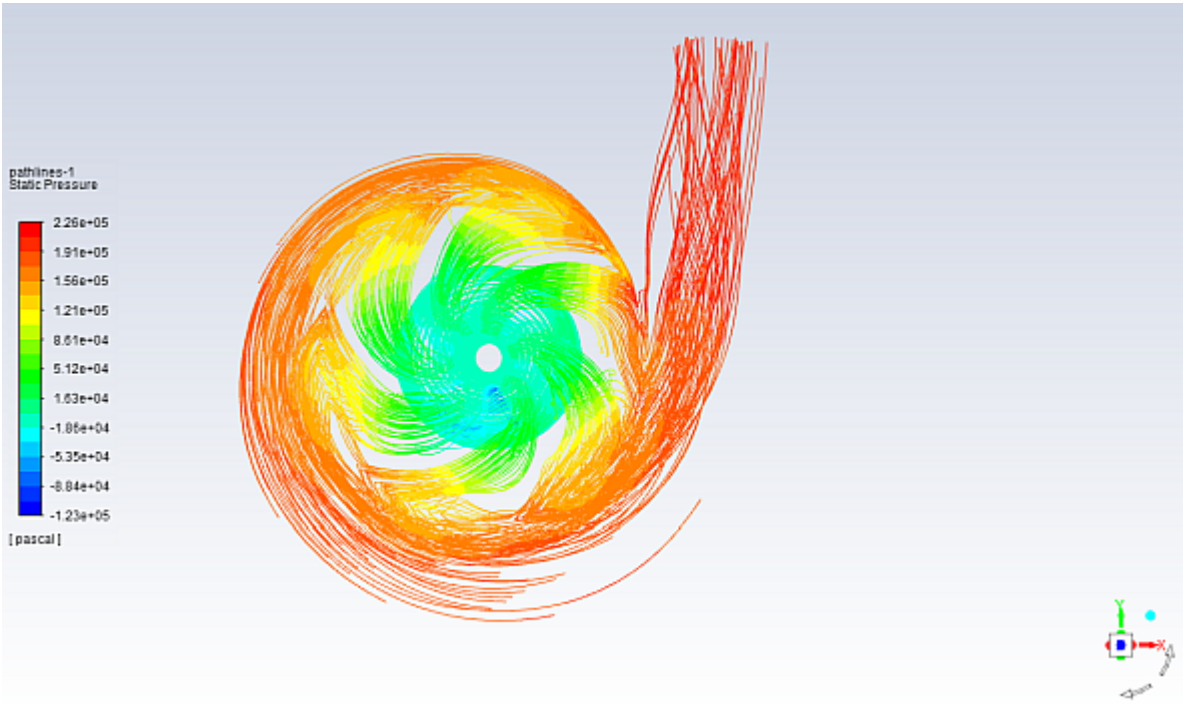
Note:

You can move the colormap of a vector plot definition by left-clicking and dragging it to the desired location. The colormap can also be resized by dragging the corners of the box that appears when you hover over the colormap. *The colormap location is not saved with the vector plot definition.*

40.1.5. Displaying Pathlines

Pathlines are used to visualize the flow of massless particles in the problem domain. The particles are released from one or more surfaces that you have created by clicking **Create** in the **Surface** group box of the **Domain** ribbon tab (see Creating Surfaces and Cell Registers for Displaying and Reporting Data (p. 3813)). A **line** or **rake** surface (see Line and Rake Surfaces (p. 3826)) is most commonly used. Figure 40.45: Pathline Plot (p. 3922) shows a sample plot of pathlines.

Figure 40.45: Pathline Plot



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

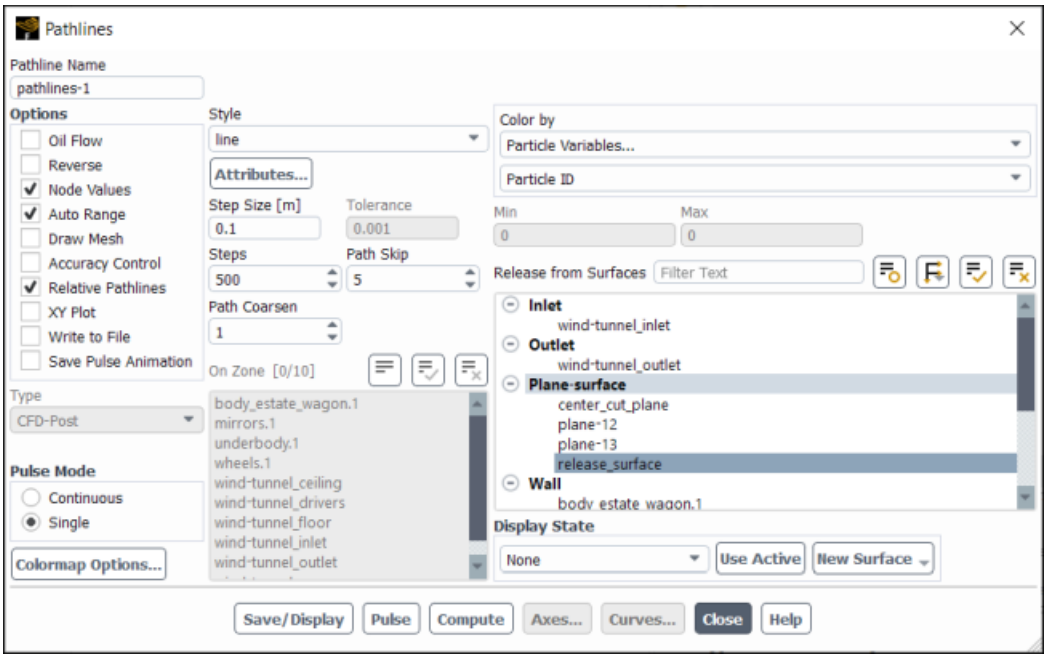
Note that the display of discrete-phase particle trajectories is discussed in Displaying of Trajectories (p. 2834).

40.1.5.1. Steps for Generating Pathlines

You can plot pathlines using the Pathlines Dialog Box (p. 5572) (Figure 40.46: The Pathlines Dialog Box (p. 3923)).

Results → Graphics → Pathlines New...

Figure 40.46: The Pathlines Dialog Box



The basic steps for generating pathlines are as follows:

1. Select the surface(s) from which to release the particles in the **Release From Surfaces** list.

2. (Eulerian Multiphase model only) From the **Track in Phase** drop-down list, select the phase in which pathlines will be tracked. If you want to track pathlines in the mixture of all phases, select **all-phase**. By default, particles are tracked in the primary phase.

3. Set the step size and the maximum number of steps. The **Step Size** sets the length interval used for computing the next position of a particle. Note that particle positions are always computed when particles enter/leave a cell; even if you specify a very large step size, the particle positions at the entry/exit of each cell will still be computed and displayed. The value of **Steps** sets the maximum number of steps a particle can advance. A particle will stop when it has traveled this number of steps or when it leaves the domain. One simple rule of thumb to follow when setting these two parameters is that if you want the particles to advance through a domain of length L, the **Step Size** times the number of **Steps** should be approximately equal to L.

4. Set any of the pathline plot options described in Options for Pathline Plots (p. 3924).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Click **Save/Display** to draw the pathlines, or click **Pulse** to animate the particle positions. The **Pulse** button will become the **Stop !** button during the animation, and you must click **Stop !** to stop the pulsing.

   Note:

   Enable **Save Pulse Animation** to save a video or picture files of the pulse animation. When this option is enabled, an additional **Write/Record Format** drop-down list appears that controls what type of file is written when you click **Save/Write...**.

   A **Continuous** pulse video file will be 5 seconds long, while a **Single** pulse video will be one full cycle.

6. (Optional) Associate a display state with this pathline plot for a consistent appearance each time the pathline plot is redisplayed.

   a. Set the display state. *There are two approaches*:

      - Setup the graphics window with your preferred model orientation, zoom level, graphics effects, ruler, and so on, and click **Use Active** to create a display state with these properties.
      - Select a display state from the **Display State** drop-down list (if you already created a display state that you would like to reuse for other graphics objects).

   b. Click **Save/Display** to link the selected display state with this pathline plot.

   Refer to Controlling the Display State and Modifying the View (p. 4025) for additional information.

40.1.5.2. Options for Pathline Plots

You can include the mesh in the pathline display, control the style of the pathlines (including the twisting of ribbon-style pathlines), and color them by different scalar fields and control the color scale. You can also "thin" the pathline display, trace the particle positions in reverse, and draw "oil-flow" pathlines. If you are "pulsing" the pathlines, you can control the pulse mode and save an animation of the pulse. If you are using larger time step size for calculations then you can control the accuracy of the pathline by specifying tolerance. In addition to the regular pathline display, you can also generate an XY plot of a specified quantity along the pathline trajectories. Finally, you can choose node or cell values for display (or plotting).

40.1.5.2.1. Including the Mesh in the Pathline Display

For some problems, especially complex 3D geometries, you may want to include portions of the mesh in your pathline display as spatial reference points. For example, you may want to show the location of an inlet and an outlet along with the pathlines (as in Figure 40.45: Pathline Plot (p. 3922)). This is accomplished by turning on the **Draw Mesh** option in the Pathlines Dialog Box (p. 5572). The Mesh Display Dialog Box (p. 4688) will appear when you enable the **Draw Mesh** option, where you can set the mesh display parameters. When you click **Display** in the Pathlines

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Dialog Box (p. 5572), the mesh display, as defined in the Mesh Display Dialog Box (p. 4688), will be included in the plot of pathlines.

Note:

The **Draw Mesh** option settings are not saved with the pathline definition. Once the dialog box is closed these settings will revert to being disabled. If you want these settings persisted within the current session, you can use the non-persistent **Pathlines** dialog box.

Results → **Graphics** → **Pathlines** → **Edit...**

40.1.5.2.2. Controlling the Pathline Style

Pathlines can be displayed as lines (with or without arrows), ribbons, cylinders (coarse, medium, or fine), triangles, spheres, or a set of points. You can choose **line**, **line-arrows**, **point**, **sphere**, **refined-sphere**, **ribbon**, **triangle**, **coarse-cylinder**, **medium-cylinder**, or **fine-cylinder** in the **Style** drop-down list in the Pathlines Dialog Box (p. 5572). Pulsing can be done only on **point**, **sphere**, **refined-sphere**, or **line** styles.

Once you have selected the pathline style, click **Style Attributes...** to set the pathline thickness and other parameters related to the selected **Style**:

- If you are using the **line** or **line-arrows** style, set the **Line Width** in the Path Style Attributes Dialog Box (p. 5243) that appears when you click **Style Attributes...**. For **line-arrows** you will also set the **Spacing Factor**, which controls the spacing between the lines. The size of the arrow heads can be adjusted by entering a value in the **Scale** text-entry box.
- If you are using the **point** style, you will set the **Marker Size** in the Path Style Attributes Dialog Box (p. 5243). The thickness of the pathline will be the thickness of the marker.
- If you are using the **sphere** style, you will set the **Diameter** and the **Detail** in the Path Style Attributes Dialog Box (p. 5243). Note that for the **refined-sphere**, you can only set the **Diameter**.

  Important:

  – The main difference between the **sphere**, and the **refined-sphere** option is that the **refined-sphere** is a higher quality sphere that has better performance (frames per second). The performance decreases when the diameter of the sphere is too large (for example, 10 or more).

  – Reflections and shadows are not shown for the **refined-sphere** style.

  The best diameter to use will depend on the dimensions of the domain, the view, and the particle density. However, an adequate starting point would be a diameter on the order of 1/4 of the average cell size or 1/4 step size. Units for the **Diameter** field correspond to the mesh dimensional units.

  The level of detail applied to the graphical rendering of the spheres can be controlled using the **Detail** field in the Path Style Attributes Dialog Box (p. 5243). The level of detail uses integer values ranging from 4 to 50. Note that the performance of the graphical rendering will be

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

better when using a small level of detail, that is, very coarse spheres, such as 6 or 8. The rendering performance significantly decreases with higher levels of detail. You should gradually increase the detail to determine the best-case scenario between performance and quality.

Also note that to take full advantage of spherical rendering, lighting should be turned on in the view. The Gouraud setting provides much smoother looking spheres than the Flat setting and better performance than the Phong setting. For more information on lighting, see Adding Lights (p. 3984).

- If you are using the **triangle** or any of the **cylinder** styles, you will set the **Width** in the Path Style Attributes Dialog Box (p. 5243). For triangles, the specified value will be half the width of the triangle's base, and for cylinders, the value will be the cylinder's radius.
- If you are using the **ribbon** style, clicking **Style Attributes...** opens the Ribbon Attributes Dialog Box (p. 5243), in which you can set the ribbon's **Width**. You can also specify parameters for twisting the ribbon pathlines. In the **Twist By** drop-down list, you can select a scalar field on which the pathline twisting is based (for example, helicity). Select the desired category in the upper list and then select a related quantity in the lower list. The twisting may not be displayed smoothly because the scalar field by which you are twisting the pathline is calculated at cell centers only (and not interpolated to a particle's position). The **Twist Scale** sets the amount of twist for the selected scalar field. To magnify the twist for a field with very little change, increase this factor; to display less twist for a field with dramatic changes, decrease this factor.

  When you click **Compute**, the **Min** and **Max** fields will be updated to show the range of the **Twist By** scalar field.

40.1.5.2.3. Controlling Pathline Colors

By default, the pathlines are colored by the particle ID number. That is, each particle's path will be a different color. You can also choose the color based on the surface from where the pathlines were released from using the surface ID as the particle variable. You can choose to color the pathlines by any of the scalar fields in the **Color by** drop-down list. Select the desired category in the upper list and then select a related quantity in the lower list. If you color the pathlines by velocity magnitude, for example, each particle's path will be colored depending on the speed of the particle at each point in the path.

The range of values of the selected scalar field will, by default, be the upper and lower limits of that field in the entire domain. The color scale will map to these values accordingly. If you prefer to restrict the range of the scalar field, turn off the **Auto Range** option (under **Options**) and set the **Min** and **Max** values manually beneath the **Color by** list. If you color the pathlines by velocity, and you limit the range to values between 30 and 60 m/s, for example, the "lowest" color will be used when the particle speed falls below 30 m/s and the "highest" color will be used when the particle speed exceeds 60 m/s. To show the default range at any time, click **Compute** and the **Min** and **Max** fields will be updated.

40.1.5.2.4. "Thinning" Pathlines

If your pathline plot is difficult to understand because there are too many paths displayed, you can "thin out" the pathlines by changing the **Path Skip** value in the Pathlines Dialog Box (p. 5572). By default, **Path Skip** is set to 0, indicating that a pathline will be drawn from each face on the selected surface (for example, n pathlines). If you increase **Path Skip** to 1, every other pathline will be displayed, yielding $n/2$ pathlines. If you increase **Path Skip** to 2, every third pathline will

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

be displayed, yielding $n/3$, and so on. The order of faces on the selected surface will determine which pathlines are skipped or drawn; therefore adaption will change the appearance of the pathline display when a nonzero **Path Skip** value is used.

40.1.5.2.5. Coarsening Pathlines

To further simplify pathline plots, and reduce plotting time, a coarsening factor can be used to reduce the number of points that are plotted. Providing a coarsening factor of n, will result in each n^{th} point being plotted for a given pathline in any cell. This coarsening factor is specified in the Pathlines Dialog Box (p. 5572), in the **Path Coarsen** field. For example, if the coarsening factor is set to 2, then Ansys Fluent will plot alternate points.

Important:

Note that if any particle or pathline enters a new cell, this point will always be plotted.

40.1.5.2.6. Reversing the Pathlines

If you are interested in determining the source of a particle for which you know the final destination (for example, a particle that leaves the domain through an exit boundary), you can reverse the pathlines and follow them from their destination back to their source. To do this, enable the **Reverse** option in the Pathlines Dialog Box (p. 5572). All other inputs for defining the pathlines will be exactly the same as for forward pathlines; the only difference is that the surface(s) selected in the **Release From Surfaces** list will be the final destination of the particles instead of their source.

40.1.5.2.7. Plotting Oil-Flow Pathlines

If you want to display "oil-flow" pathlines (that is, pathlines that are constrained to lie on a particular boundary), enable the **Oil Flow** option in the Pathlines Dialog Box (p. 5572). You will then need to select one or more boundary zones in the **On Zone** list. The selected zone(s) are the boundaries on which the oil-flow pathlines will lie.

40.1.5.2.8. Controlling the Pulse Mode

If you are going to use the **Pulse** button in the Pathlines Dialog Box (p. 5572) to animate the pathlines, you can choose one of two pulse modes for the release of particles that follow the pathlines. To release a single wave of particles, select the **Single** option under **Pulse Mode**. To release particles continuously from the initial positions, select the **Continuous** option.

You can save an animation of the pathline pulse by enabling **Save Pulse Animation** under **Options**. This creates an additional **Write/Record Format** drop-down to appear below the display state. You can specify whether you write a video file or picture files. Click **Video Options...** to open the Video Options Dialog Box (p. 5260), which controls additional settings for the **Video File** option. Click **Picture Options...** to open the Save Picture Dialog Box (p. 5254), which controls additional settings for the **Picture Files** option.

A **Continuous** pulse video file will be 5 seconds long, while a **Single** pulse video will be one full cycle.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

40.1.5.2.9. Controlling the Accuracy

If you are using large time step size for the calculation, there might be significant error introduced while calculating the pathlines. To control this error, select **Accuracy Control** and specify the value of **Tolerance**. The tolerance value will be taken in to consideration while calculating the pathlines for each time step.

40.1.5.2.10. Plotting Relative Pathlines

If you want to display the pathlines relative to the moving reference frame, enable the **Relative Pathlines** option in the Pathlines Dialog Box (p. 5572). You will then need to select the surfaces from the **Release from Surfaces** list.

40.1.5.2.11. Generating an XY Plot Along Pathline Trajectories

If you want to generate an XY plot along the trajectories of the pathlines you have defined, enable the **XY Plot** option in the Pathlines Dialog Box (p. 5572). The **Color by** drop-down list will be replaced by **Y Axis Function** and **X Axis Function** lists. Select the variable to be plotted on the y-axis in the **Y Axis Function** list, and specify whether you want to plot this quantity as a function of the **Time** elapsed along the trajectory, or the **Path Length** along the trajectory by selecting the appropriate item in the **X Axis Function** drop-down list. Specify the **Step Size**, number of **Steps**, and other parameters as usual for a standard pathline display. Then click **Plot** to display the XY plot.

Once you have generated an XY plot, you may want to save the plot data to a file. You can read this file into Ansys Fluent at a later time and plot it alone using the Plot Data Sources Dialog Box (p. 5297), as described in Creating an XY Plot From Multiple Data Sources (Including Files) (p. 4064), or add it to a plot of solution data, as described in Including External Data in the Solution XY Plot (p. 4063).

To save the plot data to a file, enable the **Write to File** option in the Pathlines Dialog Box (p. 5572). The **Plot** button changes to a **Write...** button. Clicking **Write...** opens The Select File Dialog Box (p. 931), in which you can specify a name and save a file containing the plot data. The format of this file is described in XY Plot File Format (p. 4071).

40.1.5.2.12. Saving Pathline Data

To save pathline data to a file, perform the following steps:

1. Enable the **Write to File** option in the Pathlines Dialog Box (p. 5572) (Figure 40.46: The Pathlines Dialog Box (p. 3923)).

2. In the **Type** drop-down list, select one of the following types of files:

   - **Standard** for `FIELDVIEW` (`.fvp`) format
   - **Geometry** for `.ibl` format (which can be read by GAMBIT)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **EnSight** format

  Important:

  If you plan to write the pathline data in **EnSight** format, you should first verify that you have already written the files associated with the **EnSight Case Gold** file type by using the **File/Export...** ribbon tab option (see EnSight Case Gold Files (p. 992)).

  For further information about the files that are written for any of these types, refer to the appropriate section following these steps.

3. Choose to color the pathlines by any of the scalar fields in the **Color by** drop-down lists.

4. Select the surface(s) from which to release the particles in the **Release From Surfaces** list.

5. If you selected **EnSight** under **Type**, you will need to specify the **EnSight Encas File Name**. Use **Browse...** to select the `.encas` file that was created when you exported the file with the **File/Export...** ribbon tab option. If you do not make a selection, then you will need to create an appropriate `.encas` file manually.

   You can also select the number of **Time Steps For EnSight Export**. This number directly determines how many time levels will be available for animation in EnSight.

6. Click **Write...** to open The Select File Dialog Box (p. 931), in which you can specify a name and save a file containing the pathline data.

To initiate saving pathline data through the text command interface enter the following TUI command:

```
display/path-lines/write-to-files
```

In addition to pathline data, you can also export particle data in either **Standard**, **EnSight** or **Geometry** type. For information on exporting particle data in **FIELDVIEW** (standard), **EnSight** or `.ibl` (geometry) format, refer to Exporting Steady-State Particle History Data (p. 999).

40.1.5.2.12.1. Standard Type

If **Standard** is selected under **Type**, Ansys Fluent will write the file in **FIELDVIEW** format, which can be exported and read into **FIELDVIEW**. The **FIELDVIEW** ASCII Particle Path Format is licensed from Intelligent Light, proprietor of an independent visualization software package (http://www.ilight.com). The file name that you use for saving the data must have a `.fvp` extension. You also have the ability to retrieve and display the particle and pathline trajectories from the file.

If the case is steady-state, the particle path information will be written in ASCII format. For transient or unsteady-state cases, the BINARY format must be used. The **FIELDVIEW** file contains a set of paths, where each path consists of a series of points. At every point the spatial location and selected variables are defined. A full description of the ASCII and BINARY formats can be found in Appendix K - Particle Path Formats of FIELDVIEW's Reference Manual [2] (p. 5713), available to licensed **FIELDVIEW** users.

The following is an example of the **FIELDVIEW** format for a steady-state case:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
FVPARTICLES 2 1
Tag Names
0
Variable Names
2
time
particle_id
3
0.2 0.8 1.3 0.2 0
0.3 0.9 1.3 0.4 0
0.5 1.1 1.3 0.6 0
```

The beginning of the file displays header information. `Tag Names` cannot be specified when the file is exported from Ansys Fluent, and hence will always be `0`. Ansys Fluent enables you to export two variables, which are listed under `Variable Names`: the first is determined by the scalar fields selected in the **Color by** drop-down lists (`time` in the example above); the second is always `particle_id`.

The rest of the file contains information about each path. A path section begins by listing the total number of points for the path. Then a line of data is presented for each point, with the X, Y, and Z locations listed in the first three columns and the variable information in the fourth and fifth columns. The example above presents a single pathline consisting of three points; the time ranges from `.2` to `.6`, and the ID of the particle is `0`.

40.1.5.2.12.2. Geometry Type

If **Geometry** is selected under **Type**, the file will be written in `.ibl` format. The resulting file contains particle paths in the form of a curve that can be read in GAMBIT. The following is an example of a **Geometry** file format that contains multiple curves:

```
Closed Index Arclength
Begin section ! 1
 Begin curve ! 1
 1                  185.61      0                 23.26
   2       88.90000000000001      0                -89.67

   Begin curve ! 2
   1       88.89999999999569      0      -89.6699999999997
   2       76.90221619148909      0     -101.2290490001453
   3       62.92208239159677      0     -110.2907424975297
   4       47.47166726362848      0     -116.5231659809653
   5       31.11689338997181      0     -119.6980363161113
   6       14.45680848476821      0     -119.6990633707006
   7      -1.898356710978934      0     -116.5262095254603
   8      -17.34954014966171      0     -110.2956910520416
   9      -31.33079110697006      0     -101.2357213074894
   10       -43.330000000007      0     -89.67815166483965

   Begin curve ! 3
   1                  -43.33      0     -89.67815166485001
   2                 -175.56      0         64.69066040289
```

The above example demonstrates how multiple curves can be imported; single curves may also be imported. After importing this file into GAMBIT, the file is read by first looking for a `Begin curve` string and then looking for the X, Y, and Z coordinates under the `Begin curve` line.

40.1.5.2.12.3. EnSight Type

By selecting **EnSight** under **Type**, you can generate files with the following extensions:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- `.mpg`
- `.mscl`
- `.encas`

An `.mpg` file will be written for every time step specified in the **Time Steps For EnSight Export** field. A sequential number will be appended to the `.mpg` extension to indicate the time step. Each file contains a header that lists the time at which the data was exported, as well as three columns listing the X, Y, and Z coordinates for every particle at that particular time step.

The following is an example of a file called `particle.mpg0003`, which contains data for nine particles at the third time step:

```
File is written from fluent in ensight measured particle format for
t = 2.42813e-04
particle coordinates
  9
  1-7.27734e-05 1.91710e-03 4.69093e-03
  2-1.75772e-04 1.97040e-03 3.92842e-03
  3-2.26051e-04 2.10134e-03 5.63228e-03
  4-1.16390e-04 2.32442e-03 5.23423e-03
  5-6.32735e-04 2.53326e-03 5.70791e-03
  6-9.69431e-04 2.37006e-03 5.27602e-03
  7-6.77868e-04 2.92054e-03 4.11570e-03
  8-9.78029e-04 2.75717e-03 4.13314e-03
  9-8.54859e-04 3.73727e-03 2.23796e-03
```

An `.mscl` file will be written for every time step specified in the **Time Steps For EnSight Export** field. A sequential number will be appended to the `.mscl` extension to indicate the time step. Each file contains the scalar information (specified under **Color By**) for every particle at a particular time step.

The following is an example of a file called `particle.mscl0006`, which captures **Particle ID** data for nine particles at the sixth time step:

```
particle id
 0.00000e+00 6.00000e+00 1.20000e+01 1.80000e+01 2.40000e+01 3.00000e+01
 3.60000e+01 4.20000e+01 4.80000e+01
```

A new `.encas` file will be written if a selection is made under **EnSight Encas File Name**. This new file is a modified version of the `.encas` file selected with the **Browse...** button, and contains information about all of the related files (including geometry, velocity, scalar and coordinate files). The name of the new file will be the root of the original file with `.new` appended to it (for example if `test.encas` is selected, a file named `test.new.encas` will be written). It is this new file that should be read into **EnSight**.

The following is an example of a file called `spray2-unsteady.new.encas`, that refers to the files generated when the data was originally exported as an **EnSight Case Gold** file type (`.geo`, `.vel`, `.sc11`, and `.sc12`) and the files created during the pathline data export (`.mpg` and `.mscl`):

```
FORMAT
type: ensight gold
GEOMETRY
model: spray2-unsteady.geo
measured: 1 particle.mpg****
VARIABLE
```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
scalar per measured node: 1 particle-id particle.mscl****
scalar per node: pressure                    spray2-unsteady.scl1
scalar per node: pressure-coefficient        spray2-unsteady.scl2
vector per node: velocity                    spray2-unsteady.vel
TIME
time set: 1 Model
number of steps: 10
filename start number:   1
filename increment:      1
time values: 0.00000e+00 1.21406e-04 2.42813e-04 3.64219e-04 4.85626e-04
6.07032e-04 7.28438e-04 8.49845e-04 9.71251e-04 1.09266e-03
```

40.1.5.2.13. Choosing Node or Cell Values

In Ansys Fluent you can determine the scalar field value at a particle location using the computed cell-center values or values that have been interpolated to the nodes. By default, the **Node Values** option is turned on, and the interpolated values are used. If you prefer to use the cell values, turn the **Node Values** option off. Note that for face-only functions like **Wall Shear Stress**, the cell value is the area-weighted average from the face values that define that cell as `c0`.

If you are plotting pathlines to show the effect of a porous medium or 2D fan boundary, to depict a shock wave, or to show any other discontinuities or jumps in the plotted variable, you should use cell values; if you use node values in such cases, the discontinuity will be smeared by the node averaging for graphics and will not be shown clearly in the plot.

40.1.5.3. Creating and Using Pathline Definitions

You can create named pathline definitions and save them for later use. See Creating and Using Contour Plot Definitions (p. 3910) for additional information on graphics object definitions.

The steps for creating and using pathline definitions are similar to the Steps for Generating Pathlines (p. 3923), with the addition of a field for naming the pathline definition.

Pathline definitions can be included in scenes as long as the **XY Plot** option is not enabled.

Note:

You can move the colormap of a pathline definition by left-clicking and dragging it to the desired location. The colormap can also be resized by dragging the corners of the box that appears when you hover over the colormap. *The colormap location is not saved with the pathline definition.*

40.1.6. Displaying Line Integral Convolutions (LICs)

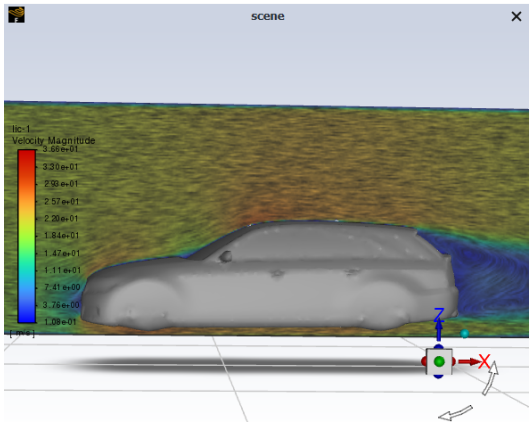
Line Integral Convolutions (LIC) is a technique of using a flow field to blur or smear (convolve) a texture image mapped to a surface in the flow domain to provide a visual representation of the vector field on that surface (like shown in Figure 40.47: LICs Plot Colored by Velocity (p. 3933)). This results in an effect that is similar to a static picture of the flow field for moving smoke or fluid injected into the fluid. Generally, the integration is done using the flow velocity field, but you can use any other vector field.

Standard LIC uses an isotropic convolution and has a dense texture that is analogous to higher concentrations of smoke in the flow. It can show flow lines but not the specific orientation. Left to right

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

flow will look the same as right to left flow. It can still be useful when the flow direction can easily be determined by geometric context.

Figure 40.47: LICs Plot Colored by Velocity



Limitations for LICs

- The mouse-probe functionality (accessed by selecting the probe toolbar icon) is not available for LICs plots.

40.1.6.1. Generating an LIC Plot

Note:

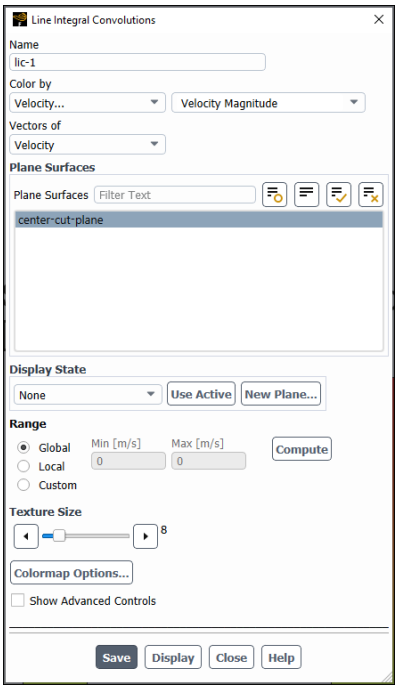
Before creating an LIC plot, you must create a plane surface for the LIC plot. Refer to Plane Surfaces (p. 3830) for additional information on creating plane surfaces.

To create an LIC plot:

1. Right-click **LICs** in the Outline View tree and select **New...**.

   Results → Graphics → LICs → New...

Figure 40.48: The Line Integral Convolutions Dialog Box



2. (Optional) Enter a **Name** for the LIC object.
3. Specify the field variable you want to use to color the LIC results by selecting the category and variable from the **Color by** drop-down lists.
4. Specify whether the vectors represent velocity or relative velocity using the **Vectors of** drop-down list.
5. Select the planar surface(s) where you want the LIC results displayed in the **Plane Surfaces** list.

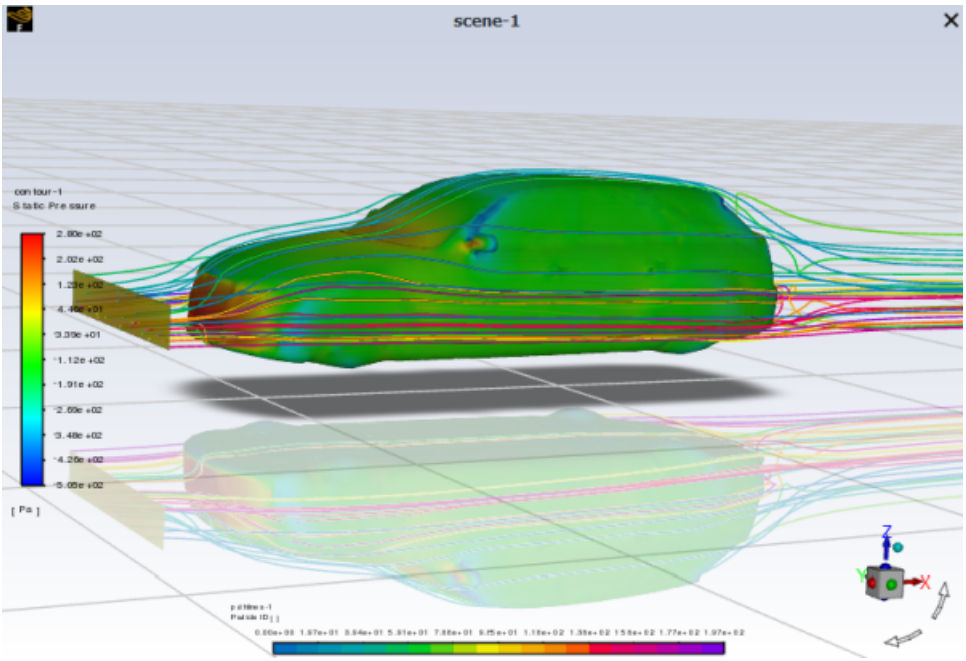
   *You can click **New Plane...** to open the Plane Surface Dialog Box in the Fluent User's Guide (p. 5580).*
6. (Optional) Specify the display state that you want associated with this object. *Refer to Controlling the Display State and Modifying the View (p. 4025) for additional information.*
7. Specify whether you want the range to be **Global**, **Local** to the selected surface(s), or **Custom**.
8. Set the **Texture Size** slider to control the size (in 100's of pixels) of the texture image used in the convolution.
9. Click **Display**.
10. Click **Save** if you are happy with the plot and want it to persist with the case file.

40.1.7. Displaying a Scene

Scenes can be used to display multiple graphics plots within a single window. For example, you could overlay contours of pressure across a valve with velocity vectors and the mesh at the same location. Scenes allow you to modify the transparency of each plot so that you can emphasize a particular plot or view.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.49: Scene Consisting of Contours and Pathlines



Note:

You can use the "make selected faces transparent" toolbar button ( ) to quickly and temporarily make surfaces transparent. However, if you want the applied transparency to persist with the case file, then you should use the scene object described in this section.

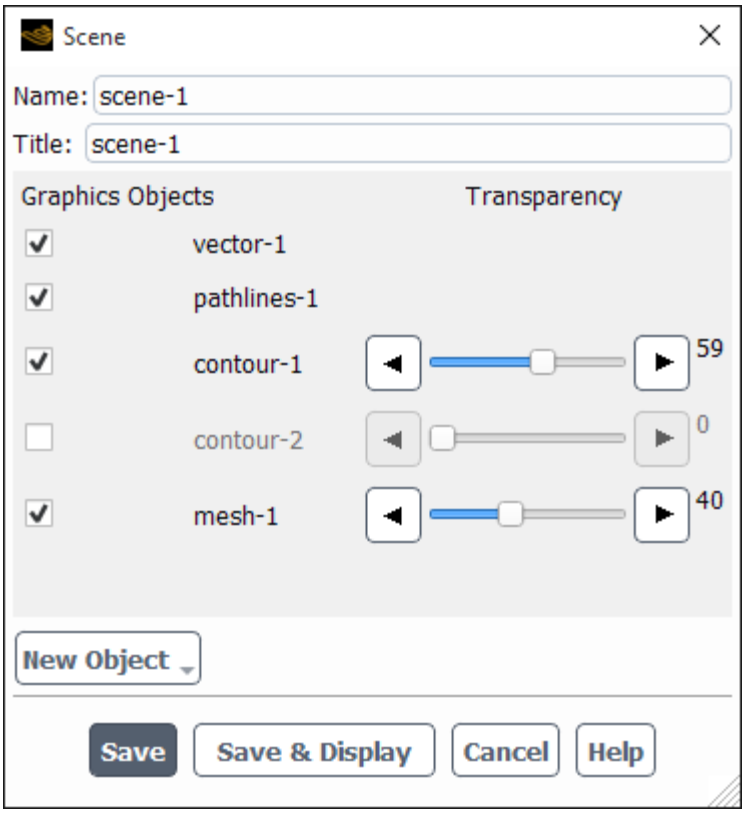
40.1.7.1. Generating a Scene

You can plot a scene made up of multiple graphics objects using the Figure 40.50: The Scene Dialog Box (p. 3936).

Results → Scene New...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.50: The Scene Dialog Box



The basic steps for creating a scene:

1. (Optional) Enter a **Name** for the scene.
2. (Optional) Enter a **Title** for the scene, which will appear in the graphics window tab where the scene is displayed.
3. Select the **Graphics Objects** that you want to include in the scene. *If you want to include additional mesh, contour, vector, or pathline plots in the scene, create them from the **New Object** drop-down list.*

   *Note that pathlines with the **XY Plot** option enabled are not compatible with scenes.*
4. (Optional) You can modify the transparency of the selected graphics objects using the **Transparency** slider. *If the slider is completely to the right the object will be fully transparent.*

Note:

The transparency slider does not work on mesh surfaces rendered with realistic materials (List of Default Realistic Materials (p. 3887)). If you want a surface colored by a material to appear more transparent, you can create a custom material with increased transparency using the **Material Editor** dialog box. *Refer to Creating Custom Realistic Materials (p. 3889) for additional information.*

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. Click **Save** or **Save & Display**.

Note:

You can also save a scene from the current graphics window display by right-click > **Save as Scene** in the graphics window, as discussed in Quickly Coloring Surfaces by Field Variable Value (p. 3897).

Scene Colormaps

Once the scene is displayed, you can move the colormaps by left-click and drag. Colormaps can also be resized by dragging the corners of the box that appears when you hover over the colormap.

Tip:

You can specify a different colormap scale for each graphic object to make it easier to determine the values that correspond to the displayed gradients (to do this, click **Colormap Options...** in the corresponding graphics object dialog box).

40.1.8. Displaying Results on a Sweep Surface

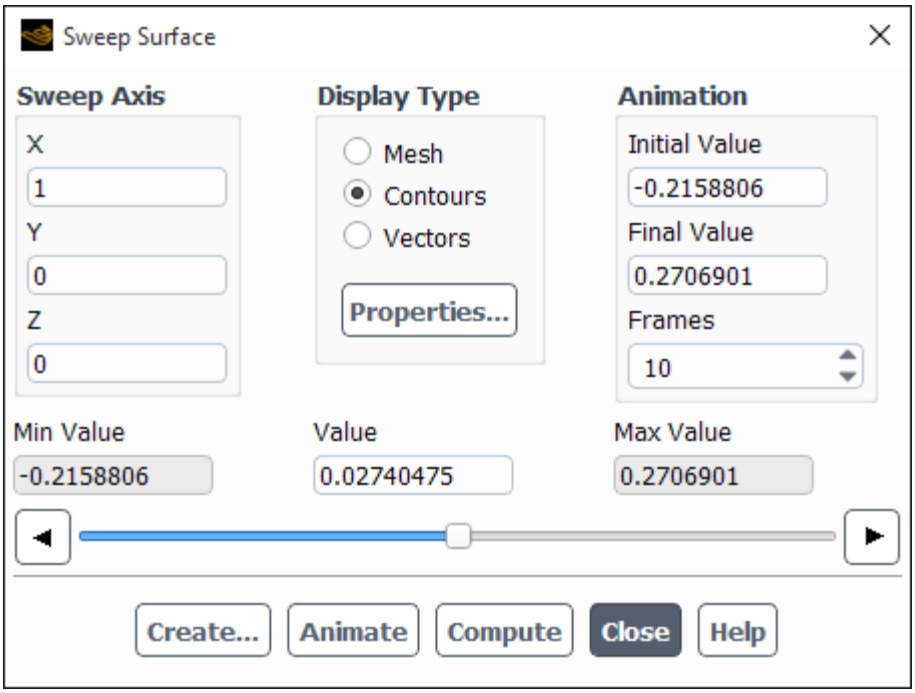
Sweep surfaces can be used when you want to examine the mesh, contours, or vectors on various sections of the domain without explicitly creating the corresponding surfaces. For example, if you want to display solution results for a 3D combustion chamber, instead of creating numerous surfaces at different cross-sections of the domain, you can use a sweep surface to view the variation of the flow and temperature throughout the chamber.

40.1.8.1. Steps for Generating a Plot Using a Sweep Surface

You can plot meshes, contours, or vectors on a sweep surface using the Sweep Surface Dialog Box (p. 5250) (Figure 40.51: The Sweep Surface Dialog Box (p. 3938)).

Results → Animations → Sweep Surface Edit...

Figure 40.51: The Sweep Surface Dialog Box



The basic steps for generating a mesh, contour, or vector plot using a sweep surface are as follows:

1. Under **Sweep Axis**, specify the (**X**, **Y**, **Z**) vector representing the axis along which the surface should be swept.

2. Click **Compute** to update the **Min Value** and **Max Value** to reflect the extents of the domain along the specified axis.

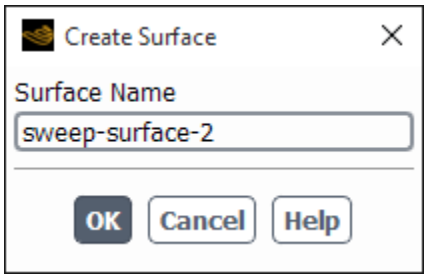
3. Under **Display Type**, specify the type of display you want to see: **Mesh**, **Contours**, or **Vectors**. The first time that you select **Contours** or **Vectors**, Ansys Fluent will open the Contours Dialog Box (p. 5412) or the Vectors Dialog Box (p. 5655) so you can modify the settings for the display. To make subsequent modifications to the display settings, click **Properties** to open the Contours Dialog Box (p. 5412) or Vectors Dialog Box (p. 5655).

4. Move the slider under **Value** (which indicates the value of *x*, *y*, or *z*) to move the sweep surface through the domain along the specified **Sweep Axis**. Ansys Fluent will update the mesh, contour, or vector display when you release the slider. You can also enter a position in the **Value** field and press **Enter** to update the display.

5. If you want to save the currently displayed sweep surface so that you can use it for a different type of plot (for example, a pathlines plot or an XY plot) or combine it with displays on other surfaces, click **Create...** to open the Create Surface Dialog Box (p. 5251) (Figure 40.52: The Create Surface Dialog Box (p. 3939)). Enter the **Surface Name** and click **OK**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.52: The Create Surface Dialog Box



The surface that is created is an isosurface based on the mesh coordinates; the contour or vector settings are not stored in the surface.

You can also animate the sweep surface display, as described below, rather than moving the slide bar yourself.

40.1.8.2. Animating a Sweep Surface Display

The steps for animating a sweep surface display are as follows:

1. Specify the **Sweep Axis** and **Display Type** as described above.
2. Under **Animation**, enter the **Initial Value** and **Final Value** for the animation. These values correspond to the minimum and maximum values along the **Sweep Axis** for which you want to animate the display.
3. Specify the number of **Frames** you want to see in the animation.
4. Click **Animate**.

40.1.9. Hiding the Graphics Window Display

There may be situations where displaying graphics on a local machine is not practical. Therefore, you may decide to hide (or disable) the graphics display window.

To disable the graphics display window when starting Ansys Fluent from the command line, you can specify the driver as null:

```
fluent -driver null
```

For an Ansys Fluent session that is already in progress, the graphics window display can be disabled using the following text command:

`display` → `set` → `rendering-options` → `driver` → `null`

Important:

All graphics windows must be closed prior to invoking the previous text command.

If the graphics window display is disabled, you can continue to save graphics using the **Save Picture** option, as described in Saving Picture Files (p. 1019). Note that the speed at which the graphics are

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

saved may be significantly slower with the driver set to null, and the saved graphics files may not be identical to those saved when the graphics window display is enabled.

For an Ansys Fluent session that is already in progress, perform the following steps to re-enable a graphics window display that had been previously disabled:

1. Enter the following text command:

   `display` → `set` → `rendering-options` → `driver`

2. Press the **Enter** key at the `driver>` prompt to display the available drivers.

3. Enter an appropriate driver, such as `opengl`.

If any graphics windows are open (which are not visible to you), Ansys Fluent will prompt you to close all open windows. You can close them using the following Scheme command:

```
(close-all-open-windows)
```

and then retype the text command to enable the graphics windows.

Important:

If you happen to be logged on to a machine remotely, then some drivers may not work on your system. Use `x11` (for Linux) or `msw` (for Windows) instead to enable your graphics windows.

40.2. Customizing the Graphics Display

There are a number of ways in which you can alter the graphical display once you have generated the basic elements in it (contours, meshes, and so on). For example, you can overlay multiple graphics, add descriptive text or lighting to the plot, and modify the captions or legend layout. These and other customizations are described in this section.

40.2.1. Embedded Graphics Window Dashboards
40.2.2. Advanced Graphics Overlays
40.2.3. Managing Multiple Graphics Windows
40.2.4. Showing Boundary Markers
40.2.5. Changing the Legend Display
40.2.6. Adding Text to the Graphics Window
40.2.7. Changing the Colormap and Range
40.2.8. Adding Lights
40.2.9. Modifying the Rendering Options

Important:

It is possible that the graphics window may become **[Out of Date]** (which would be indicated at the top of the graphics window), if you make changes to items that are already displayed.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To resolve this out of date state, right-click in the graphics window and click **Refresh Display** in the context menu that appears.

*If a context menu does not appear on a right-click of the graphics window, ensure that the right-mouse button is set to an action other than **Zoom** or **mouse-probe** and **long description** (**View** ribbon tab, **Mouse** group box).*

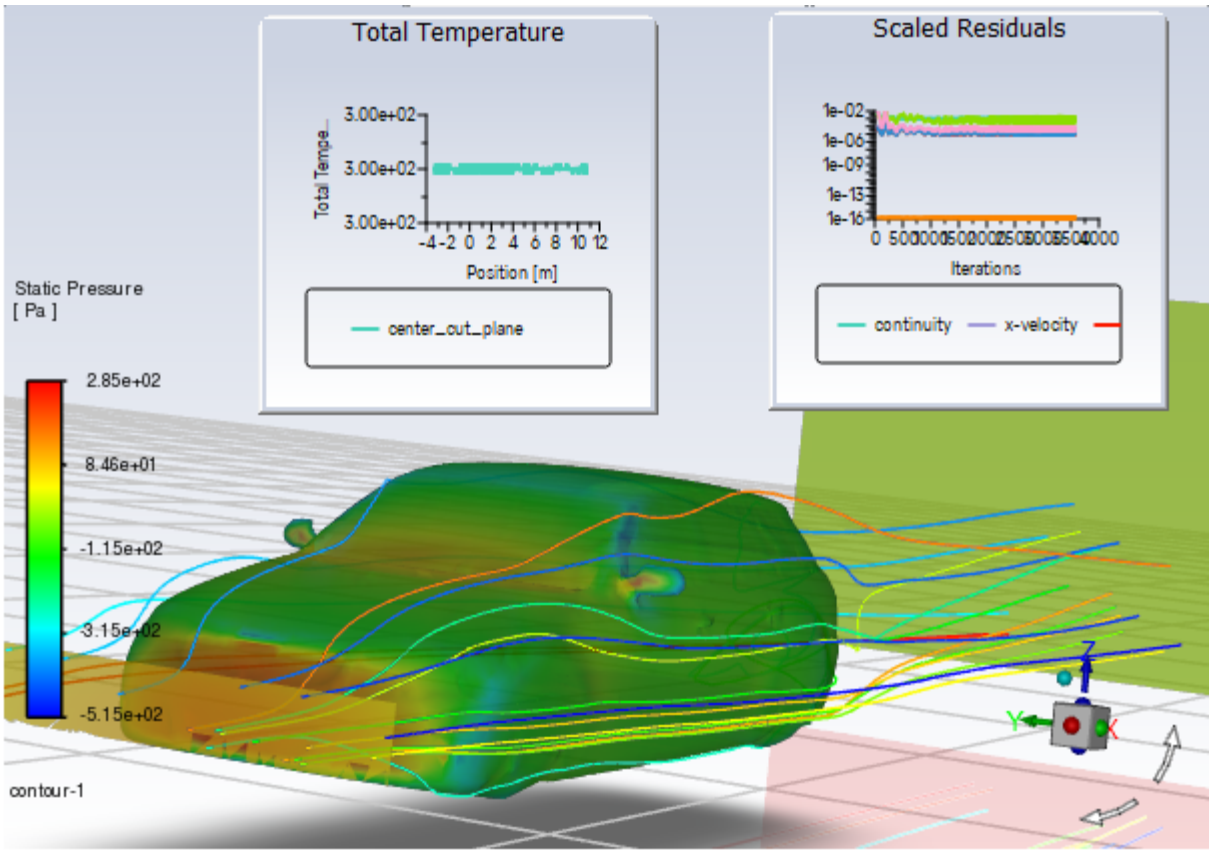
*For the refresh of a mesh display, Fluent displays the surfaces that are currently selected in the global **Mesh Display** dialog box (accessed by clicking* *in the toolbar). In some instances this may result in the refreshed display not matching what was originally displayed.*

40.2.1. Embedded Graphics Window Dashboards

You can embed multiple graphics windows into a separate window (also know as "picture in picture"), like in Figure 40.53: Example of Embedded Windows (p. 3942), which shows residuals and a report plot embedded within the Solution View. The feature lets you create a dashboard-like display containing both 2D and 3D plots, and the Solution View window (see Automatically Embedding Windows in the Solution View (p. 3942)) automatically updates the parent window at a set interval, allowing you to see the solution evolve on the displayed quantities. This can be a helpful technique for monitoring solution progress, postprocessing the results, or disseminating your conclusions as a part of a presentation. It is also possible to create an animation of this dashboard, which can then be saved in MP4 and other video formats.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.53: Example of Embedded Windows



Tip:

Display your dashboard(s) prior to beginning your solver run to ensure they appear as desired.

These sections describe the methods for creating embedded window dashboards

40.2.1.1. Automatically Embedding Windows in the Solution View

40.2.1.2. Embedding Windows with the Dashboard Definition Dialog Box

40.2.1.3. Manually Embedding Windows

40.2.1.4. Animating Embedded Window Dashboards

40.2.1.5. Limitations for Embedded Windows

40.2.1.1. Automatically Embedding Windows in the Solution View

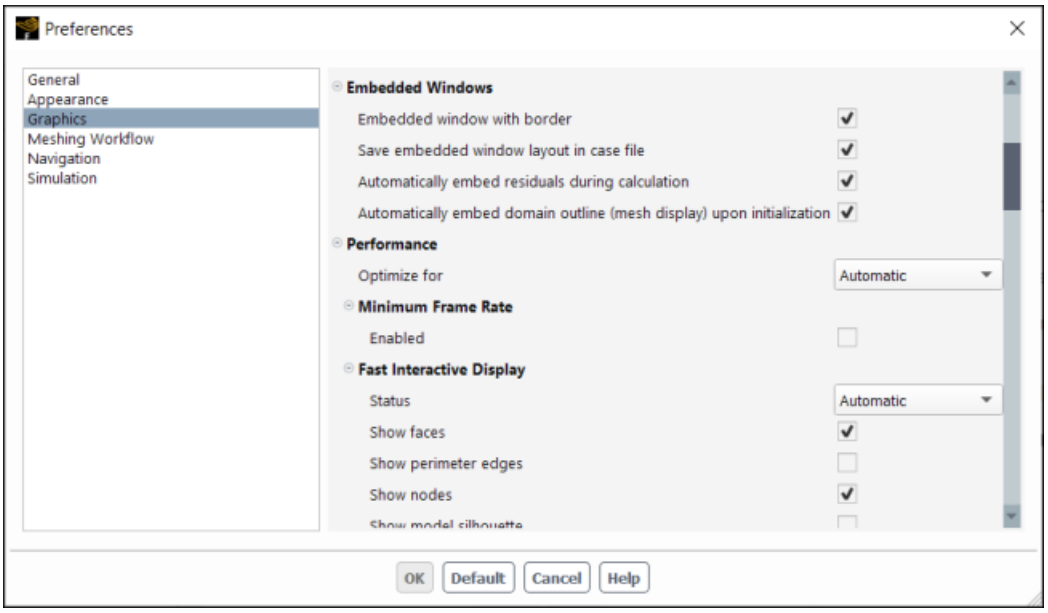
Fluent can automatically create an embedded window dashboard for you as soon as you begin calculating a solution. By default (with options enabled in Preferences), the parent window will be a scene graphics object, and the mesh and residuals will be embedded within that parent window (like Figure 40.53: Example of Embedded Windows (p. 3942)). If you do not have any graphics objects defined, Fluent will create one for you (3D simulations only).

There are two Preferences options for automatically embedding graphics windows: **Automatically embed residuals during calculation** and **Automatically embed domain outline (mesh display)**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

upon initialization. These provide you the choice of what will be done automatically when you initialize the solution and begin calculating. To control these options, open them in Preferences.

File → Preferences...



- **Automatically embed residuals during calculation**—the **Residuals** graphics window is automatically embedded within the **Solution View** graphics window as soon as the solver begins calculating. Once embedded, the residuals may be resized and relocated as desired.

  You can right-click the embedded residuals windows and select **Move Out Embedded Window** to make it so the residuals are no longer embedded. To re-embed residuals, right-click the **Residuals** tab and select **Embed In** > **target-window**.

  Note:

  Adjoint residuals are not automatically embedded in the **Solution View** graphics window. However, you may include them in your embedded window dashboard by right-clicking the **Adjoint Residuals** tab and selecting **Embed In** and picking the solution view window as the destination.

- **Automatically embed domain outline (mesh display) up on initialization**—the graphics window containing the global mesh display, that is, mesh either displayed automatically after reading the mesh/case or by clicking the **Mesh Display** button in the toolbar ( ), is automatically embedded within the **Solution View** graphics window as soon as you initialize the solution. Once embedded, the mesh may be resized and relocated as desired. You can also double-click the embedded mesh window and it will swap with the content displayed in the current parent window.

  You can change the contents of the parent window to include any type of graphics display, such as a contour plot or an XY plot. Just right-click the defined object in the **Outline View** tree and select **Display**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Both of the options for automatic embedding create a **Solution View** graphics window that will contain the parent and embedded windows. Fluent automatically selects the parent window in the following order: scene (Displaying a Scene (p. 3934)), contour (Displaying Contours and Profiles (p. 3896)), vector (Displaying Vectors (p. 3912)), pathline (Displaying Pathlines (p. 3922)), particle tracks (Displaying of Trajectories (p. 2834)). (3D cases only) If there are not any graphics objects associated with the case file, Fluent automatically creates a plane surface and contour graphics object for the solution view.

Simultaneously with the creation of the solution view window, Fluent creates a temporary **Dashboard** object that appears in the Outline View tree under **Results**. *This object is deleted if you close the dashboard.*

The exact contents of the **Solution View** window are not saved in the case file, but the layout of the embedded windows is preserved (assuming that **Save embedded layout in case file** is enabled in Preferences).

Important:

Only the object included in the **solution-view** animation definition is updated at a specified interval while Fluent is calculating. Objects in other windows (for example a mesh plot) embedded on the solution view window are not updated as the solution progresses.

40.2.1.2. Embedding Windows with the Dashboard Definition Dialog Box

You can create dashboard objects that are added to the tree and stored with the case file using the **Dashboard Definition**. These dashboards are functionally the same as dashboards created using other means, but their advantage comes from the ability have the dashboard retained with the case file and available in the Outline View tree. Further, these dashboards are easily edited and can be quickly redisplayed.

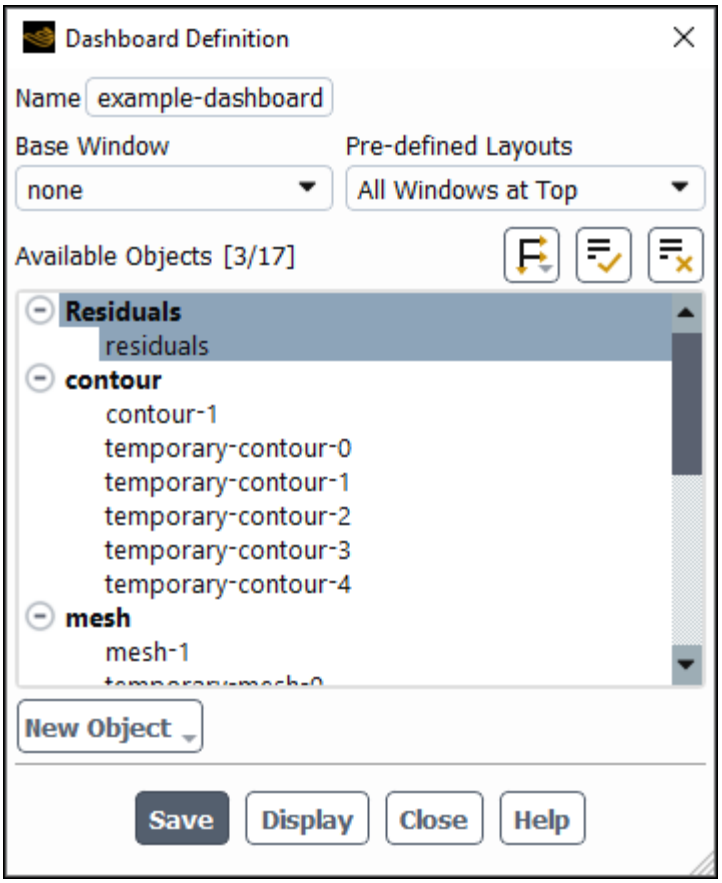
To create a dashboard using the **Dashboard Definition** dialog box:

1. Open the **Dashboard Definition** dialog box.

   Results → **Graphics** → **Dashboard...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.54: Dashboard Definition

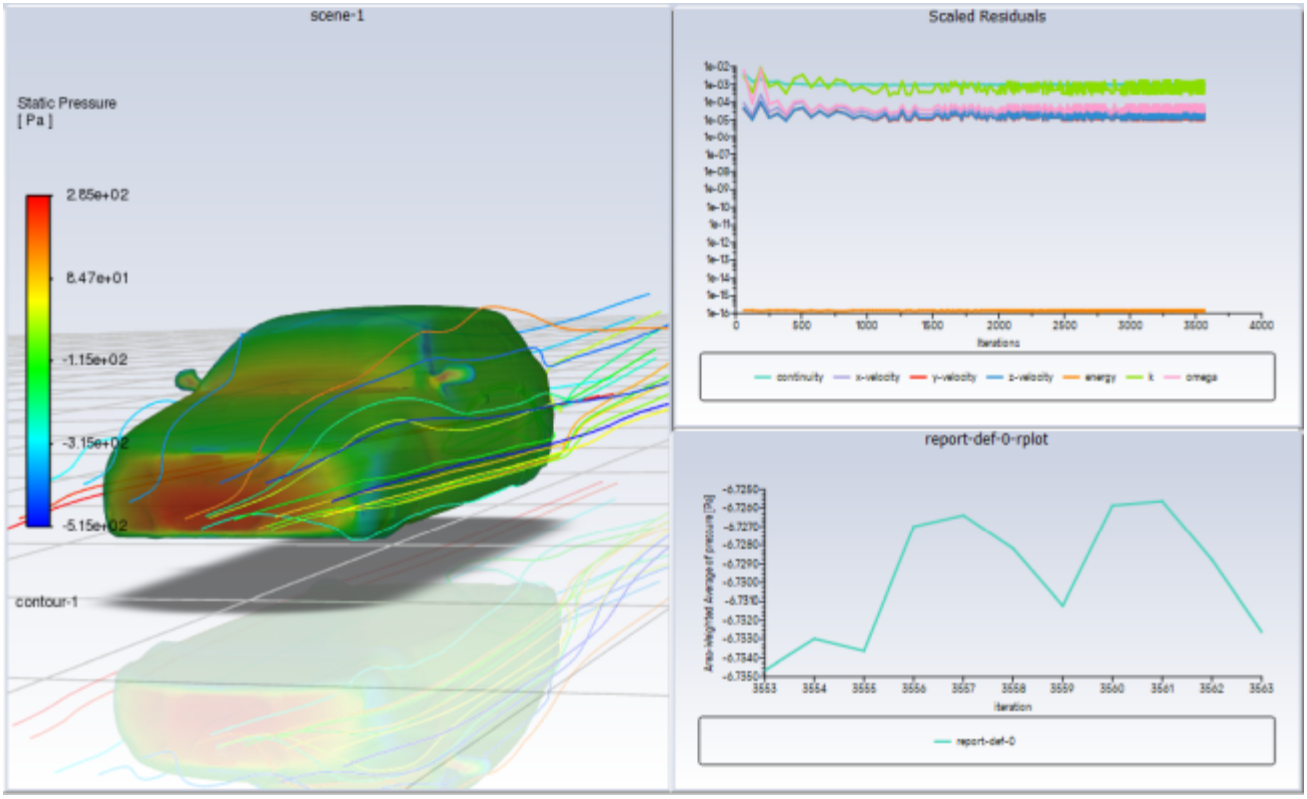


2. Enter a **Name** for the dashboard you are creating.

3. Use the **Base Window** drop-down list to specify which object will be shown in the large background graphics window that all other graphics objects will be embedded into. *Note that you can also leave this option set to **none**, so that all child windows are embedded into an empty graphics window. This allows you to tile the child windows such that there is no overlap. See Figure 40.55: Dashboard with "none" for Base Window (p. 3946) for an example.*

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.55: Dashboard with "none" for Base Window



4. Specify where you want the embedded windows to appear when they are first displayed from the **Pre-defined Layout** drop-down list.

Important:

- *Once the dashboard is displayed, you can drag the embedded windows around as desired. Fluent saves the updated positioning of the windows when you write the case file.*
- If you change the pre-defined layout after displaying a dashboard, you must close the displayed dashboard and re-display before you will see the updated layout.

---

5. Select the windows you want embedded into the **Base Window** from the **Child Windows** list.
6. Click **Display** to display the dashboard. Click **Save** to save the dashboard with the case file..

Note:

If a dashboard is visible in the graphics window, displaying another object (such as a contour or XY Plot) will be displayed in the active dashboard window, which automatically updates the dashboard definition.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

*Right-clicking an object in the Outline View tree and selecting **Display in New Window** avoids modifying the dashboard. Similarly, you can right-click the top of the displayed graphics window tab and select **New Window** to open an empty window for display.*

40.2.1.3. Manually Embedding Windows

To setup embedded windows:

1. Ensure graphics objects are available by either initializing the solution or loading a results (data) file.

2. Create an animation definition containing the desired graphics object(s). *Refer to Creating an Animation Definition (p. 3714) for additional information.*

   Note:

   Embedding into an animation graphics window allows you to see both the 2D plots (such as residuals and report plots) as well as the monitored quantity in the animation definition (whatever is specified within the selected graphics object) updated as the solution progresses.

   Additionally, layouts embedded into an animation graphics window are saved with the case and data file and are redisplayed automatically when you begin calculating in another session.

3. Determine which graphics window you want as the baseline/parent and note its name. *You must know the name so that you know which window to select when moving the other window(s).*

   Important:

   If you want to save the embedded window layout as an animation, you must use an animation definition as the baseline window. Further, this animation definition must be saved in one of the 2D image formats (that is, PPM, TIFF, PNG, or JPEG). Refer to Creating an Animation Definition (p. 3714) for additional information on creating animation definitions.

   Once other windows are embedded in the animation definition window, saved animation images will include the additional windows. These animations can then be saved in standard video formats, as described in Saving an Animation Sequence (p. 3720).

   *Note that the border of embedded windows will not be included in images saved using the **Save Picture** dialog box (Saving Picture Files (p. 1019)) or animations.*

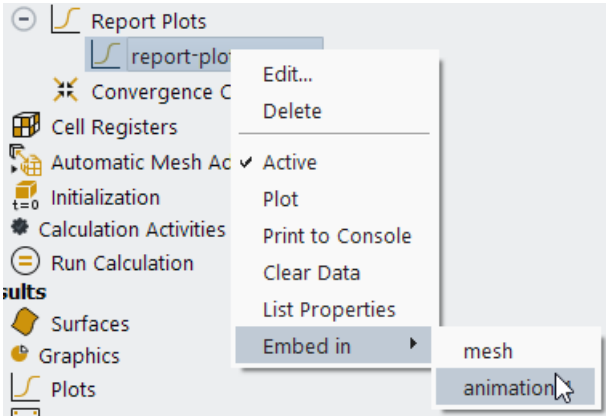
4. Add the windows you want embedded either by:

   - Right-clicking the desired graphics object in the Outline View tree and select **Embed in** > **<target window name>** from the context menu. This option is available for all viable object

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

types, including reserved windows such as report plots and animation definitions, even before data is generated by beginning the calculation. See Figure 40.56: Embedding a Window - Outline View Context Menu (p. 3948)for an example of this technique.

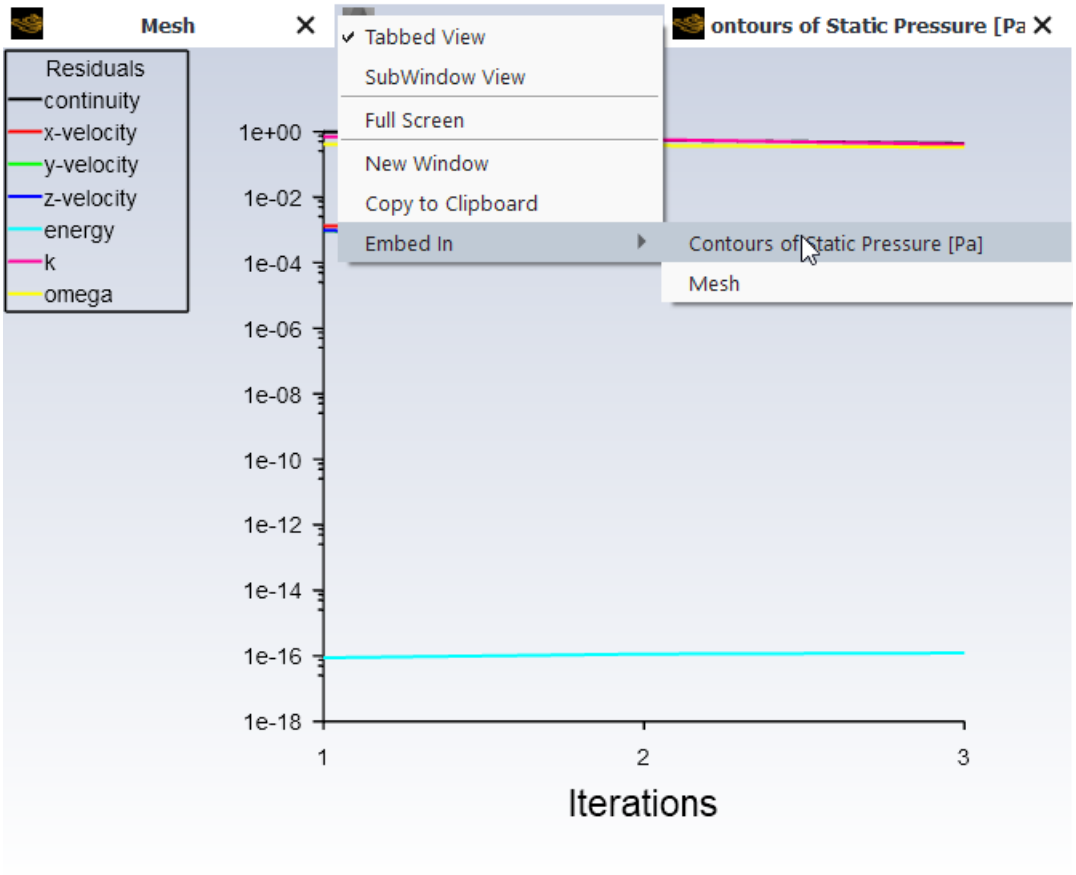
Figure 40.56: Embedding a Window - Outline View Context Menu



- Right-clicking the tab (or top of the window if you are viewing graphics windows as subwindows) of a window you want to embed and select **Embed In** → **<target window name>**. See Figure 40.57: Embedding a Window - Tab Context Menu (p. 3949) for an example of how this technique. *Note that you must begin iterating by starting the calculation, for reserved windows, such as residuals and report plots, to open automatically.*

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.57: Embedding a Window - Tab Context Menu



Note:

Once you have embedded one window into another, a **dashboard-n** definition is created and added to the Outline View tree for your embedded window dashboard.

5. Repeat the previous step for all windows that you want to embed.

6. Arrange the embedded windows in the best layout for your purposes. You can resize embedded windows by dragging at the edges of the window. You can move embedded windows (within the borders of the baseline window) by left-clicking and dragging at the top of the window.

7. Save your case and data to save the layout and contents of your embedded window dashboard.

   File → Write → Case & Data...

8. The next time you open your case and data, display the baseline/parent window and Fluent will automatically embed the child windows when you begin calculating. *Alternatively, you can right-click **dashboard-n** in the Outline View tree and click **Display** to display the embedded windows dashboard.*

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

40.2.1.4. Animating Embedded Window Dashboards

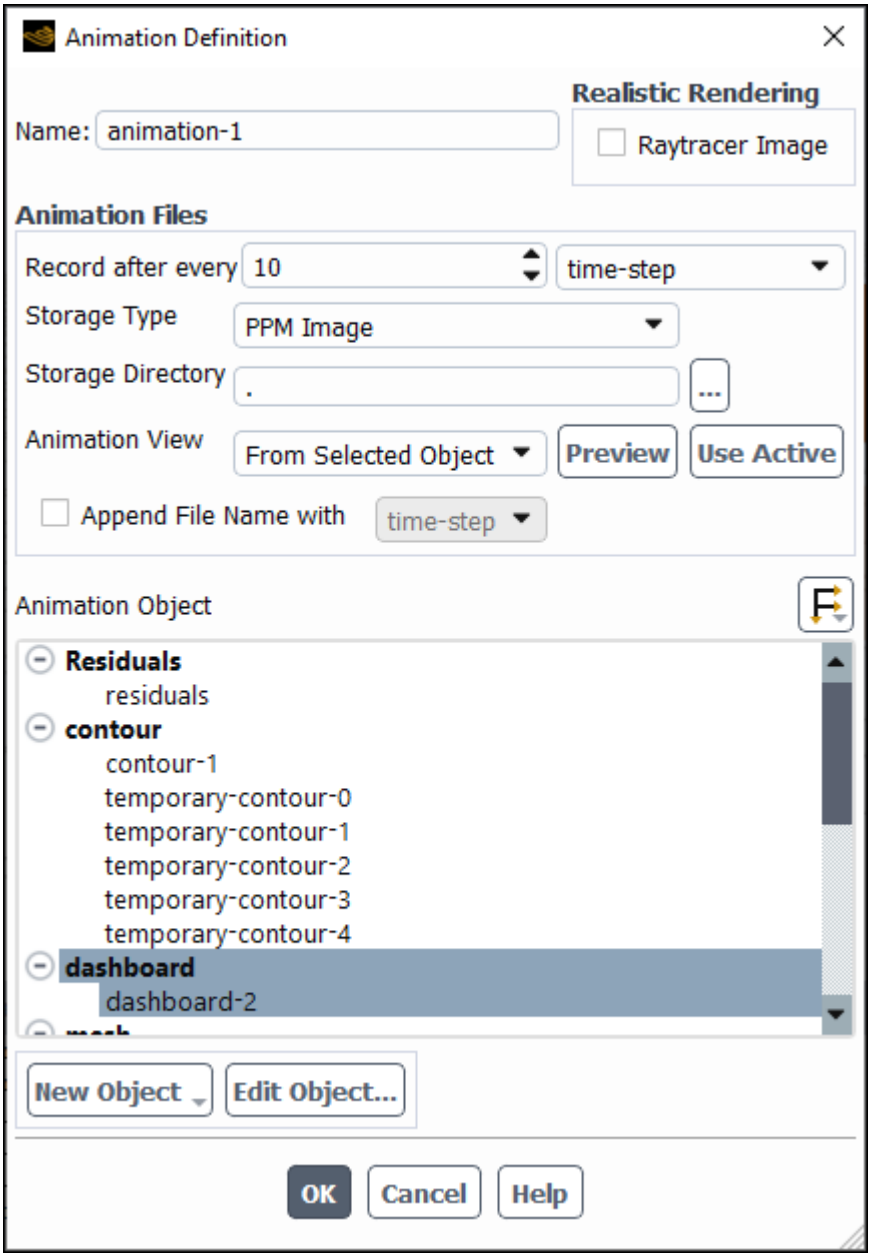
Embedded window dashboards can be animated, allowing you to playback whatever it is that you choose to include in your dashboard. This could include monitors such as report definitions and residuals for checking convergence, or it could be assorted graphics objects focusing on different areas of your model, showing how quantities evolve as the solution progresses.

There are two approaches you can take to create your dashboard animation:

- You create your dashboard manually as described in Manually Embedding Windows (p. 3947).

  Then you select the defined dashboard as the animation object in the **Animation Definition** dialog box (see Figure 40.58: Selecting a Dashboard to Animate (p. 3950) for an example). Preview the dashboard prior to closing the dialog box, so you can ensure the layout is as desired.

Figure 40.58: Selecting a Dashboard to Animate



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that the border of embedded windows will not be included in images saved using the ***Save Picture*** *dialog box (Saving Picture Files (p. 1019)) or animations.*

Important:

Before beginning the computation, close all windows associated with the dashboard you are animating (that includes any of the objects contained in the dashboard, and the dashboard itself). Click **Preview** in the **Animation Definition** dialog box and adjust the dashboard layout as desired, then start calculating. *This ensures the resulting animation appears as expected.*

Note:

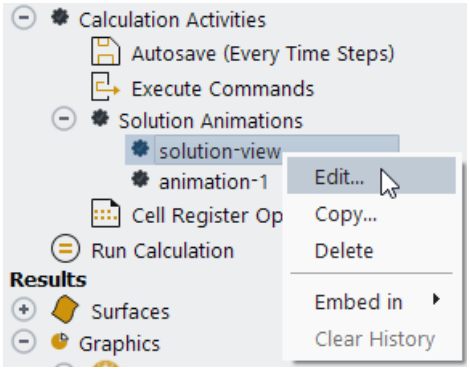
When defining the dashboard you want to animate, only include one type of graphics object (contours, vectors, and so on) as the embedded window, or the dashboard may not behave as expected. You can include as many plots (report plots and residuals) as desired.

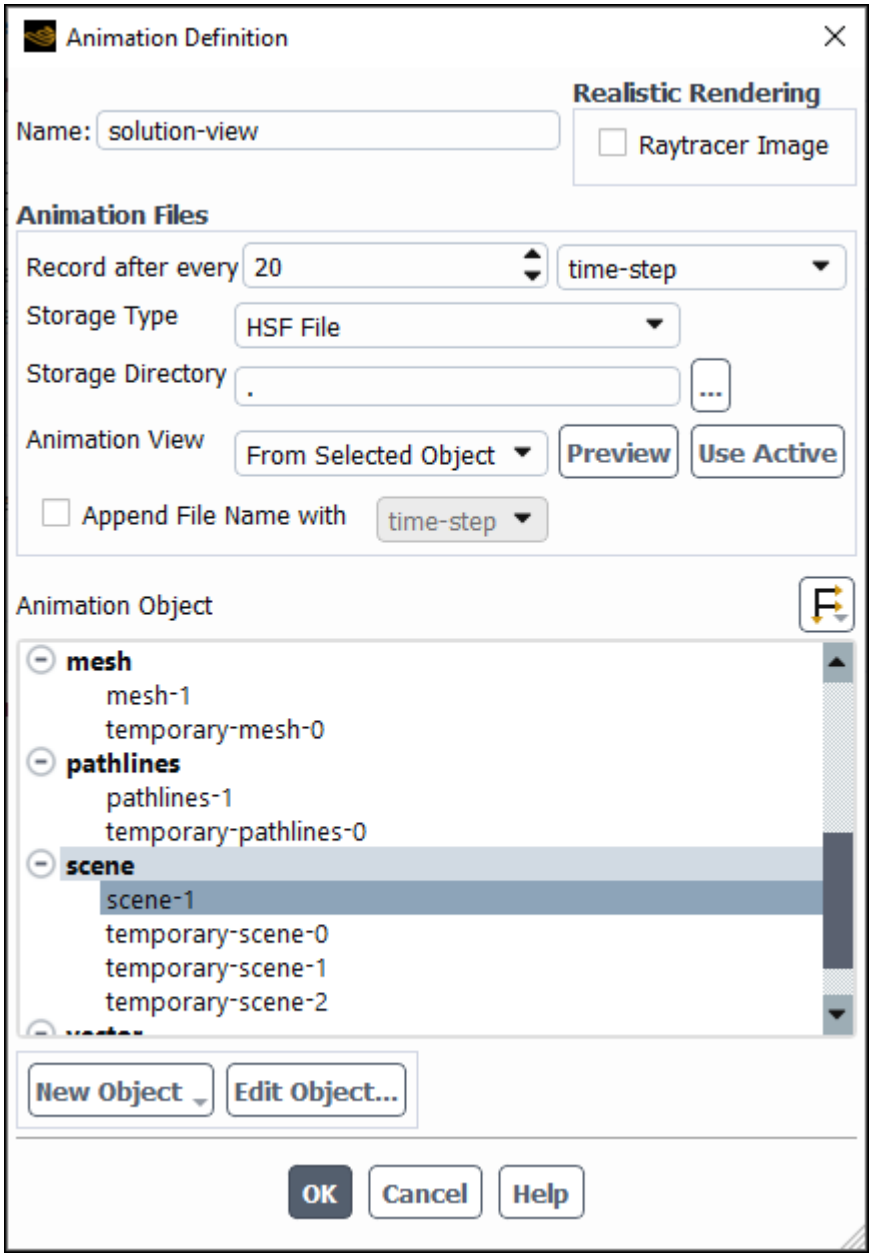
- Fluent automatically generates your dashboard, at least initially, as described in Automatically Embedding Windows in the Solution View (p. 3942). With the **Solution View** window created, you can control which object is represented as the parent window:

  1. Make **Solution View** the active graphics window by clicking the tab or sub-window.
  2. Click inside the parent window to make it active. *This ensures that the graphics object that you display in the next step replaces the current parent window, rather than replacing one of the embedded windows.*
  3. Right-click the desired graphics object in the Outline View tree and select **Display** in the context menu that appears.
  4. Specify how frequently you want the parent dashboard window to be updated and whether or not you want the images saved (by default the **Solution View** animation definition's storage type is set to **none**, so the parent window image is updated as the solution progresses, but the images are not retained for future use).

     a. Right-click **solution-view** under **Solution Animations** in the Outline View tree and click **Edit...** to open the **Animation Definition** dialog box.

     Solution → Calculation Activities → Solution Animation → solution view → Edit...





b. Set how frequently you want the parent dashboard window updated in the **Record After Every** field.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

c. Update the **Storage Type** to one of the available image types available in the drop-down list, if you plan to playback the animation or save it once the calculation is complete.

d. Confirm the remaining settings match your expectations (including the **Animation View**) and click **OK** when you are done.

Important:

Only the object included in the **solution-view** animation definition is updated at a specified interval while Fluent is calculating. Objects in other windows (for example a mesh plot) embedded on the solution view window are not updated as the solution progresses.

---

5. Proceed with the remainder of the solution setup. Once the computation is complete, the animation can then be saved in standard video formats, as described in Saving an Animation Sequence (p. 3720).

40.2.1.5. Limitations for Embedded Windows

The following limitations exist for embedded windows:

- **Save embedded layout in case file** option in **Preferences**:
  - This option works best (that is, your layout and the parent and child windows will be the same) when you manually create your embedded window dashboard, as described in Manually Embedding Windows (p. 3947). To see the dashboard in a new session, ensure the baselines/parent window is displayed before beginning the calculation.
  - The layout may not be preserved if you change the layout of the Fluent UI elements (such as collapsing the **Outline View** or **Task Page** or by one of the options made available via the **Arrange the workspace** drop-down list ( ).
  - (Fluent in Workbench) To preserve the **Solution View** layout, you must select the **Use settings changes for current and future calculations.** option in the **Settings have changed!** dialog box that appears when you close your Fluent session.
- Context menu options are limited in the embedded windows to the most relevant options. You can swap an embedded window with the primary window (either by double-clicking the embedded window or using the context menu) to access the full context menu options.
- **Embed in >** context menu options (accessed by right-clicking an object in the Outline View tree) are grayed-out when this object is already included in a saved embedded window layout from a previous session.
- When saving pictures by using the **Save Picture** dialog box:
  - Ensure that the parent window is in focus (and not an embedded window) if you want the entire embedded window dashboard captured.
  - The border and the window titles of embedded windows will not be included in the saved images.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - You can only save 2D images of the embedded window dashboard (that is, JPEG, PPM, TIFF, PNG).

- The layout of the embedded window may not remain exactly the same when you swap it with the parent window (by double-clicking or using the context menu) and then swap back.

40.2.2. Advanced Graphics Overlays

Normally, you can see only one picture at a time in the graphics window, unless you are Displaying a Scene (p. 3934). Sometimes, however, you may want to create an "exploded" view, where results translated or rotated out for the physical domain for enhanced display (see Figure 40.59: Exploded Scene Display of Temperature and Velocity (p. 3955)). You can do this by turning on the **Overlays** option (and clicking **Apply**) and using the transform operation in the Scene Description Dialog Box (p. 5265).

View → **Graphics** → **Compose...**

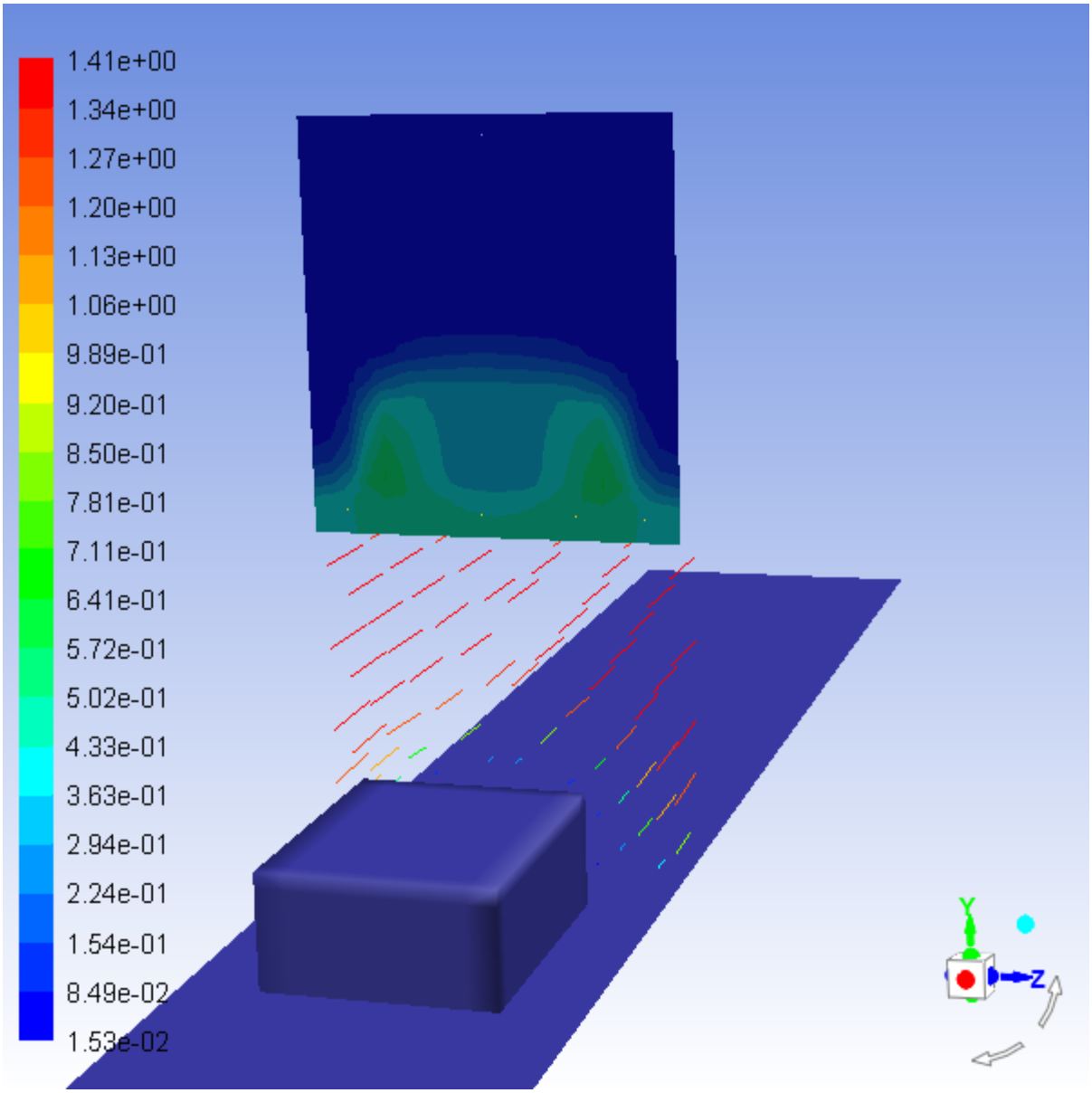
Once overlaying is enabled, subsequent graphics that you generate will be displayed on top of the existing display in the active graphics window. To generate a plot without overlays, you must turn off the **Overlays** option in the Scene Description Dialog Box (p. 5265) (and remember to click **Apply**).

When you are overlaying multiple graphics, the captions and color scale that will appear in the latest display are those that correspond to the most recently drawn graphic.

Note that when overlaying is enabled, it will apply to all graphics windows, including those that are not yet open. Turning overlays on and off does so for all graphics windows, not just for the active window. That is, if you enable overlays, open a new graphics window (as described in Managing Multiple Graphics Windows (p. 3955)), and then generate two or more graphics in that window, they will be overlaid.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.59: Exploded Scene Display of Temperature and Velocity



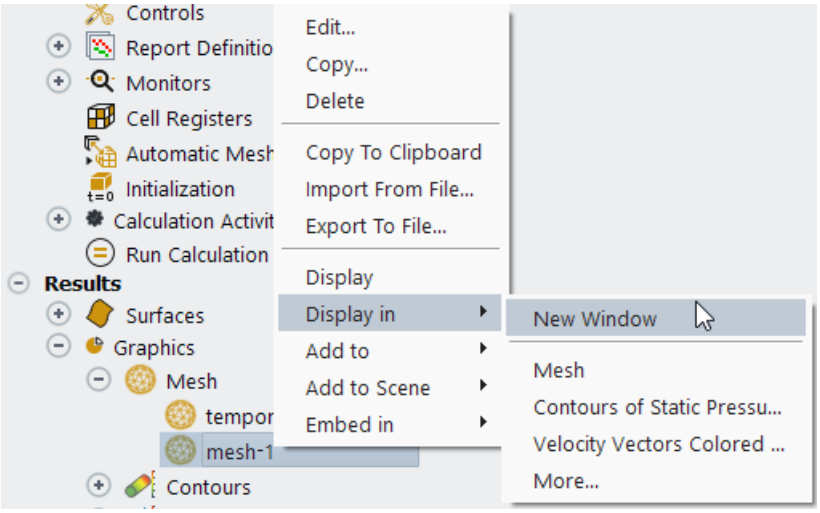
40.2.3. Managing Multiple Graphics Windows

During your Ansys Fluent session, you can open up to 50 graphics windows at one time and they may be viewed within the application window or in separate windows (sub-window view). Ansys Fluent can separately open up to an additional 50 windows, referred to as 'Reserved', which contain displays that are generated while the solver is running. These reserved windows include residuals (Printing and Plotting Residuals (p. 3689)), report plots (Report Files and Report Plots (p. 4123)), and animation definitions (Creating an Animation Definition (p. 3714)). Reserved windows are indicated by a grayed-out Fluent icon ( ) and they do not offer any context menu options. User-specified windows are indicated by a Fluent icon ( ) and they provide additional context menu options via right-click (see examples in Graphics Windows (p. 909)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

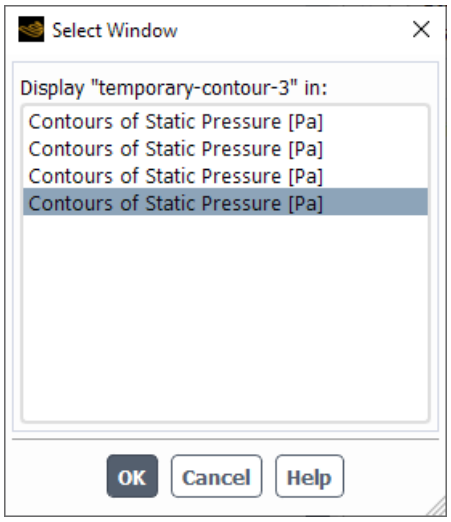
You can open new graphics windows by right-clicking the top of the graphics window tab (or the top of the window if you are in sub-window view) and select **New Window**. Alternatively, you can display boundaries and graphics objects in a new window by right-clicking them in the Outline View tree and clicking **Display in** → **New Window** (as shown in Figure 40.60: Outline View 'Display in' Example (p. 3956). Refer to The Outline View (p. 903) for additional information on the context menu options.

Figure 40.60: Outline View 'Display in' Example



If there are a lot of graphics windows open or if you want to be certain that you are selecting the intended window, you can select the **More...** context menu option (**Display in** → **More...**), which opens the **Select Window** dialog box (Figure 40.61: The Select Window Dialog Box (p. 3956)).

Figure 40.61: The Select Window Dialog Box



As soon as you select a window from the list, that window is made active. Clicking **OK** displays the selected object in the specified graphics window.

When you are viewing graphics windows in sub-window view (rather than tabbed) or if you are using the embedded windows feature (Embedded Graphics Window Dashboards (p. 3941)), you may want to synchronize the views so that manipulations performed in one window, such as zooming or panning, are also performed in the other open windows. Refer to Synchronizing Window Views (p. 3957) for additional information.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

40.2.3.1. Setting the Active Window

When you have more than one graphics window open, you must identify the active window so that Ansys Fluent will know which one to draw the plot in. Identify the active window by clicking any mouse button in the desired window. This window will remain active until you set a new active window.

40.2.3.2. Synchronizing Window Views

You can synchronize two or more open graphics windows so that their views are linked and graphical manipulations (such as panning or zooming) performed in one window are done in the other synchronized windows simultaneously.

There are two ways that you can start and stop the synchronization process:

- Click the Synchronize Views toolbar button ( ). This synchronizes the views of all open graphics windows to the current view of the active graphics window.
- Right-click in the graphics window with the view that you want other graphics windows to match and select either **Synchronize** > **Start** or **Synchronize** > **Synchronize all windows** from the context menu.

  You can then synchronize other graphics windows to the first window by right-clicking in the window and selecting **Synchronize** > **Add this window** (assuming you did not select the **Synchronize all windows** option).

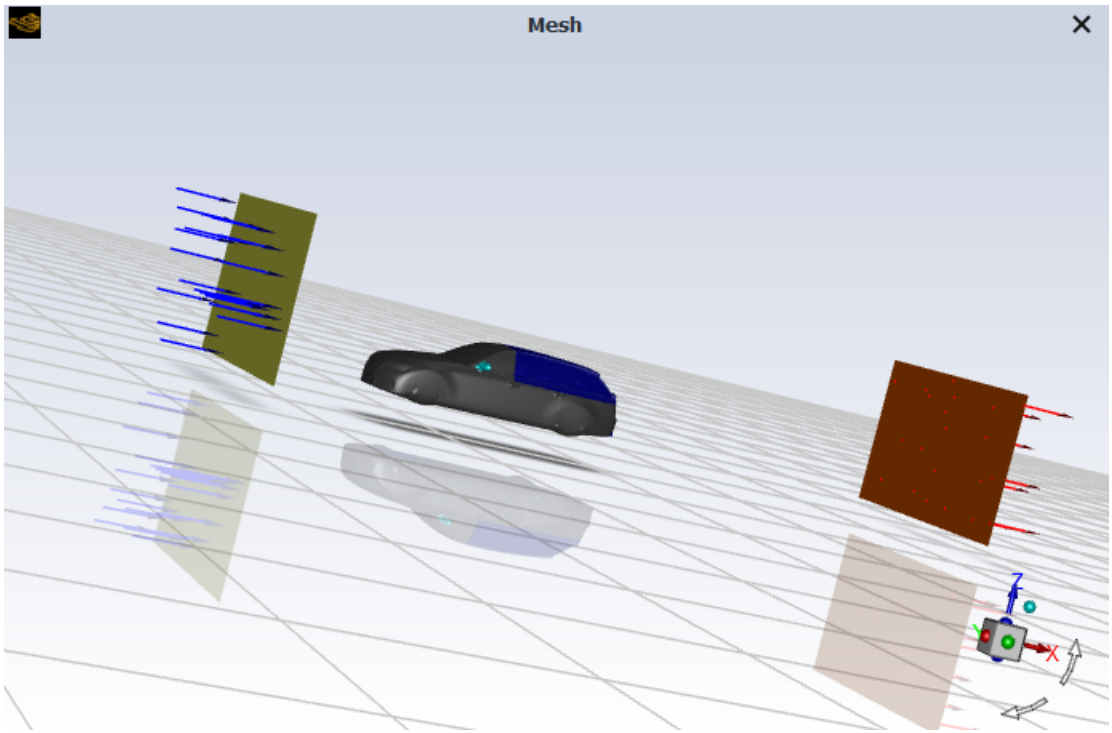
You can disable window synchronization either by clicking the unsynchronize button in the toolbar ( ), which stops the process for all windows, or by right-clicking the graphics window and selecting **Synchronize** > **Remove this window**, which only removes the selected window from the synchronization process.

40.2.4. Showing Boundary Markers

Enabled by default when the mesh is displayed, boundary markers allow you to quickly see the flow direction for inlets and outlets in the graphics window. Boundary markers are 2D arrows that point into the model for inlets and out of the model for outlets, as shown in Figure 40.62: Boundary Markers on an Inlet and Outlet (p. 3958).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.62: Boundary Markers on an Inlet and Outlet



The boundary markers button in the graphics window toolbar ( ) allows you to turn the boundary markers on and off.

You can have greater control over the appearance of the boundary markers in **Preferences** (accessed by selecting **Preferences...** from the **File** ribbon tab).

In **Preferences**, you can:

- Toggle whether boundary markers are enabled for inlets and outlets separately.
- Control the relative size of the boundary markers.
- Choose the color of the boundary markers.
- Specify the maximum number of boundary markers that can be displayed on a surface. (The actual number displayed will be based on this limit and the specified density.)
- Control the density of boundary markers on a surface (with values of 0.1 to 1). The larger the value for density, more tightly the arrows can be packed on the surface. *A value of 1 means that one boundary marker would be shown per surface facet (assuming the **Limit** is higher than the number of facets).*

Limitations

- Boundary markers only work on the mesh display if **Outline** is selected under **Edge Type** in the **Mesh Display** dialog box.
- Boundary Markers are not shown for the **Pressure Far-Field** inlet type.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For 2D cases, boundary markers may be unreliable and could show incorrect directions due to issues with how settings are stored and managed in global and saved graphics objects.

40.2.5. Changing the Legend Display

Ansys Fluent graphics include, by default, a caption or legend block that consists of fields of text describing the contents of the graphic, the Ansys Fluent product identification, an axis triad indicating the orientation of the displayed object, a color key defining the correspondence between each color and the magnitude of the plotted variable, and the ANSYS logo. You can turn off the display of the legend and color scale, and/or the axis triad. You can also hide the ANSYS logo. You can also display the colormap on any side of the display window as per convenience. In addition you can also edit the captions directly in the graphics window.

40.2.5.1. Controlling the Titles, Axes, Ruler, Logo, and Colormap

You can enable/disable the display of the **Titles**, **Axes**, and **Ruler** directly in the graphics window by toggling the appropriate icon (refer to Additional Display Options (p. 920) for more information). The **Logo** and the **Colormap** are controlled under **Layout** in the Display Options Dialog Box (p. 5262) (Display Options Dialog Box (p. 5262)).

Results → Graphics → Options...

Use the text interface to enable/disable the captions and color scale individually, and to change the size and position of the captions and color scale.

`display → set → windows → text →`

`display → set → windows → scale →`

See the separate Text Command List for details.

Controlling Titles

You can add text, such as your company name, to the title box in multiple locations using the `display/set/titles/` text command locations. Once you have provided text for one of these locations, you must ensure that `display/set/windows/text/company?` is set to yes, then re-render the display by displaying a graphics object (for example, mesh, contour, vector, and so on) or by entering `display/re-render` in the console.

Alternatively, if you always want your company name or other text displayed in the title box, you can set the `FLUENT_COMPANY` environment variable in the Fluent Launcher equal to your company name or the desired text, for example, `FLUENT_COMPANY=My Example Company Name`. *Note that **Titles** is disabled by default when you launch Fluent, so you will have to enable **Titles** before it will be displayed in the graphics window.*

40.2.5.2. Editing the Legend

When captions are displayed in the graphics window, you may choose to modify, delete, or add to the text that appears in the caption box. To do so, click the left mouse button in the desired location. A cursor will appear, and you can then type new text or delete the text that was originally there (using the backspace or delete key). Note that changes to existing text in the caption block will be

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

removed when you draw new graphics in the window (unless you are overlaying multiple graphics in the same window), but text that you add on a previously empty line in the caption block will not be removed until the default caption text makes use of that line.

40.2.5.3. Adding a Title to the Caption

You can define a title for your problem using the `title` text command:

`display` → `set` → `title`

The title you define will appear on the top line of the caption, at the far left, in all subsequent plots. It will also be saved in the case file.

Important:

You will need to enclose your title in quotation marks (for example, " `my  title` ").

40.2.5.4. Enabling/Disabling the Axes

You can disable the display of the axis triad by turning off the **Axes Visibility** button ( ) in the graphics window.

40.2.5.5. Enabling/Disabling the Ruler

You can disable the display of the ruler by turning off the **Ruler Visibility** button ( ) in the graphics window.

Note that if your project is 3D, then enabling the ruler automatically changes the view to orthographic so that the ruler markings are relevant for the model.

40.2.5.6. Modifying and Displaying/Hiding the Logo

The graphics window displays a white ANSYS logo in the upper right corner. You may choose between a white (default) or black logo by selecting **White** or **Black** from the **Color** drop-down list under **Layout** in the Display Options Dialog Box (p. 5262) (Display Options Dialog Box (p. 5262)).

You can prevent the ANSYS logo from being displayed in the graphics window by disabling the **Logo** option under **Layout**.

40.2.5.7. Colormap Alignment

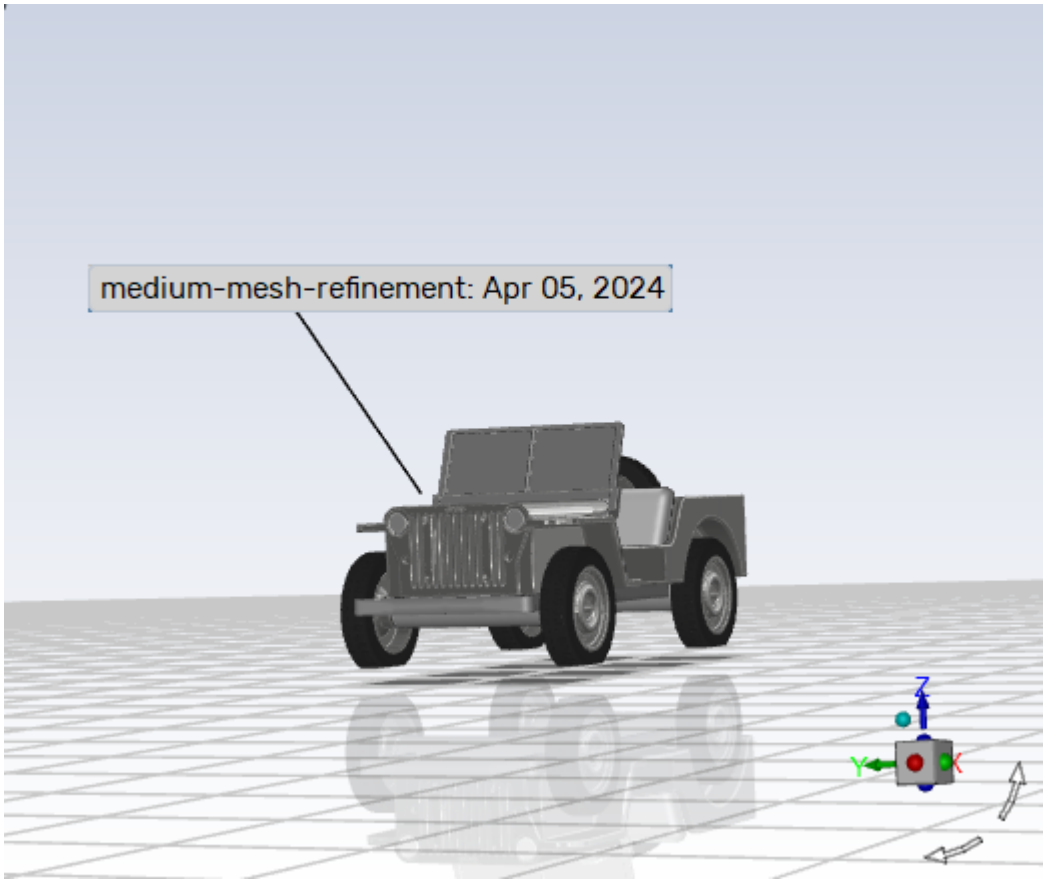
You can set the position of the colormap on any side (left, top, bottom, or right) of the display window. Default alignment for the colormap is set to the **Left**. If you want to change the alignment, select the required direction in the **Colormap Alignment** drop-down list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

40.2.6. Adding Text to the Graphics Window

You can use the Annotate Dialog Box (p. 5286) to add text to the graphics window with optional attachment lines. Figure 40.63: Graphics Window with Text Annotation (p. 3961) shows an example of a graphics display with annotated text in it.

Figure 40.63: Graphics Window with Text Annotation

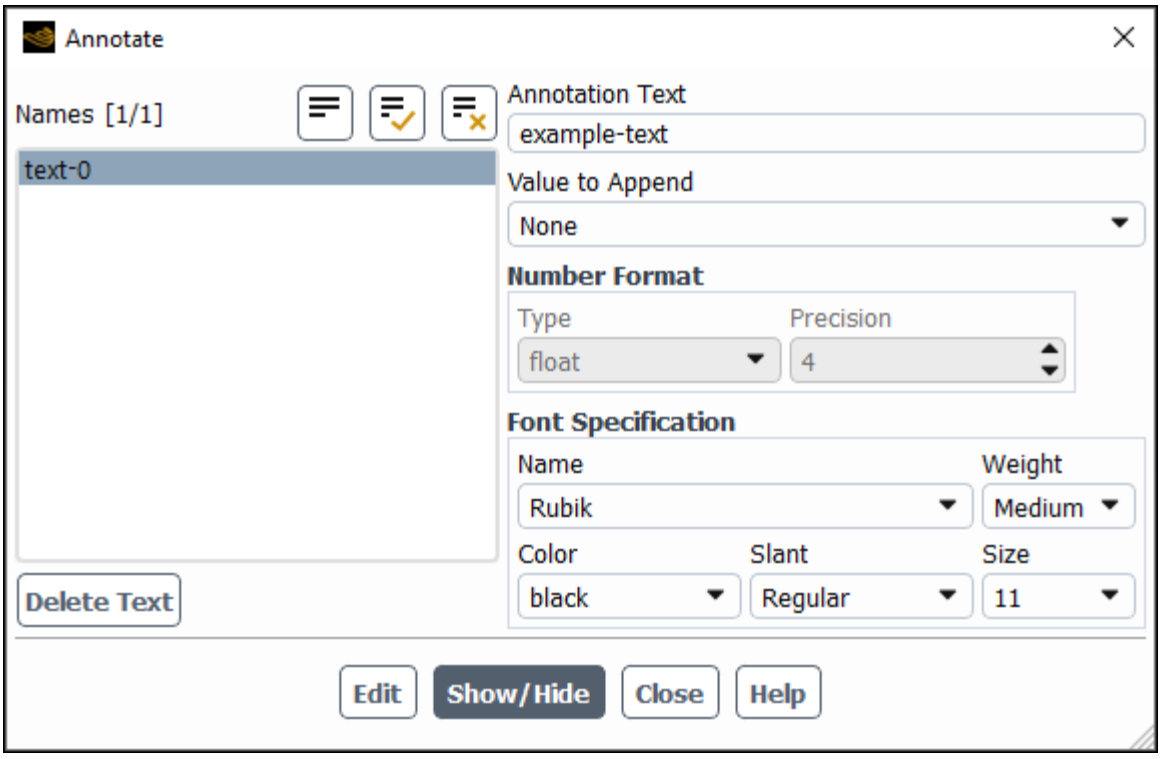


40.2.6.1. Adding Text Using the Annotate Dialog Box

Adding text to the graphics window using the Annotate Dialog Box (p. 5286) (Figure 40.64: The Annotate Dialog Box (p. 3962)) allows you to control the font and color of the text.

Results → Graphics Annotate...

Figure 40.64: The Annotate Dialog Box



The steps for adding text are as follows:

1. Enter the text that you want to add in the **Annotation Text** field.
2. (Optional) Select a **Value to Append** from the drop-down list.
3. (Flow-time and expressions only) Specify the desired formatting of the flow-time output. You can control the **Type** and **Precision** in the **Number Format** group box.
4. Under **Font Specification**, select the font type in the **Name** drop-down list, the font weight (**Medium** or **Bold**) in the **Weight** drop-down list, the size (in points) in the **Size** drop-down list, the color in the **Color** drop-down list, and the slant (**Regular** or **Italic**) in the **Slant** drop-down list.
5. Click **Add**. You will be asked to pick the location in the graphics window where you want to place the text, using the left mouse button.

   If you click the mouse button once in the desired location, the text will be placed at that point. Dragging the mouse with the left-mouse-button depressed will draw an attachment line from the point where the mouse was first clicked to the point where it was released. The annotation text will be placed at the point where the mouse button was released.

Important:

Annotations are added to a specific graphics window and they are not persisted with the displayed object. If you open a new graphics window, you will not see any annotations listed in the **Annotate** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For animations where you want an annotation to update, you must first **Preview** the animation object, then add the annotation. *When playing back the animation, first close the original graphics window where the animation definition was previewed, since the annotation will still be there and it may give the impression that the annotation is not updating as the animation plays.*

40.2.6.2. Editing Existing Annotation Text

Once you have added text to the graphics display, you may change the font characteristics of one or more text items, or delete individual text items.

To modify or delete existing text, follow these steps:

1. Select the appropriate item in the **Names** list in the Annotate Dialog Box (p. 5286) (Figure 40.64: The Annotate Dialog Box (p. 3962)). When you select a name, the associated text will be displayed in the **Annotation Text** field, and the **Add** button becomes the **Edit** button.

2. Modify the **Font Specification** entries as desired, and click **Edit** to modify the text, or simply click **Delete Text** below the **Names** list to delete the selected text.

Note that if you want to make changes to all current annotation text, you can select all of the **Names** instead of just one in step 1.

You can move the text by clicking (left mouse button) and dragging the annotation text in the graphics window. If a line is attached, it will move as needed to continue pointing at the original location and text.

40.2.6.3. Clearing Annotation Text

Annotation text is associated with the active graphics window and is removed only when the annotations are explicitly cleared. To remove the annotations from the graphics window, you must click **Clear** in the Annotate Dialog Box (p. 5286). If you draw new graphics in the window without clearing the annotations, they will remain visible in the new display.

40.2.7. Changing the Colormap and Range

The default colormap used by Ansys Fluent to display graphical data (for example, vectors) ranges from blue (minimum value) to red (maximum value). Additional predefined colormaps are available, and you can also create custom colormaps. To make any changes to the colormap, you will use the Colormap Dialog Box (p. 5280) (Figure 40.66: The Colormap Dialog Box (p. 3965)).

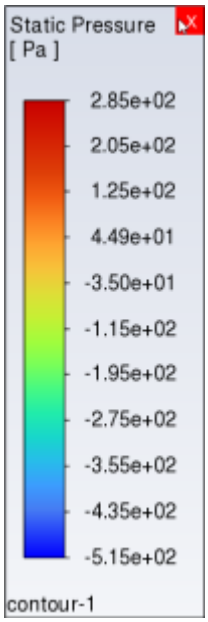
If you have already displayed a graphics object, such as a contour or vector plot, you can double-click the colormap in the graphics window to open Figure 40.67: The Colormap Quick-Edit Panel (p. 3966). This panel lets you quickly change colormap properties directly in the graphics window and the changes are saved as soon as you leave the field you are editing. These properties include the range of values the colormap is covering.

If you want to temporarily hide a colormap from the graphics window, for example, when capturing a picture (Saving Picture Files (p. 1019)), you can click the 'X' that appears when hovering over the upper-right corner of the colormap (as shown in Figure 40.65: Temporarily Hiding the Colormap (p. 3964)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

There is a **Show Colormap** context menu option (accessed by right-clicking in the graphics window) allowing you to bring back any hidden colormaps.

Figure 40.65: Temporarily Hiding the Colormap



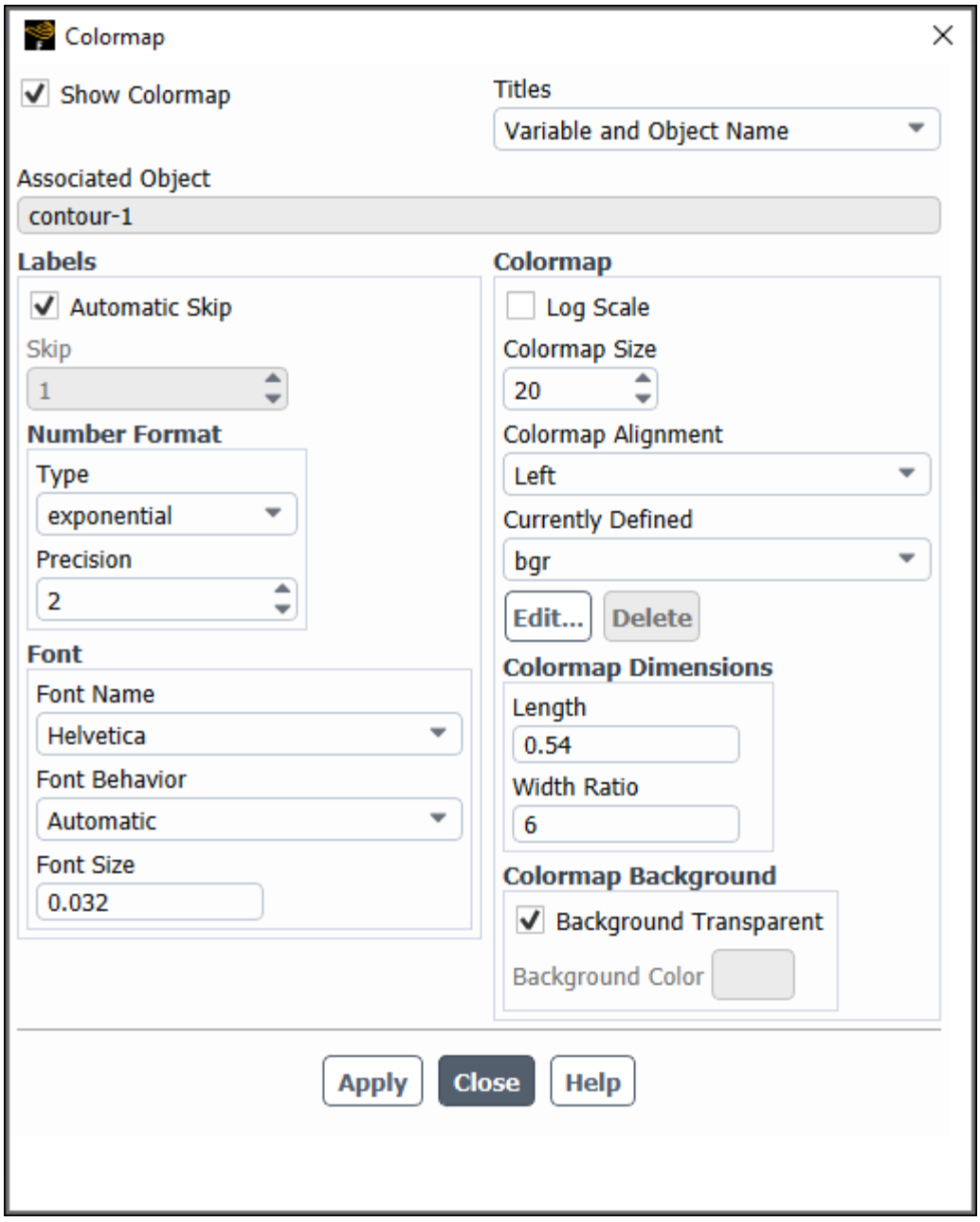
Note:

- You can control many colormap default settings using preferences, including the colormap font, font size, colormap size, location, physical size in the graphics window, number format type and precision, and more. All of the available settings are listed under **Colormap Settings** in the **Graphics** branch of Preferences (accessed via **File** → **Preferences...**).
- Opening the **Colormap** dialog box from within a graphics object (such as the **Contours** or **Vectors** dialog box) provides more options than opening it from the **Views** ribbon tab or a right-click of the **Graphics** branch in the Outline View tree).

Results → **Graphics** **Colormap...**

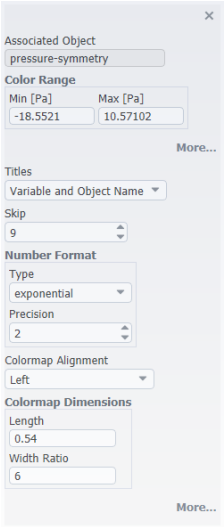
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.66: The Colormap Dialog Box



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.67: The Colormap Quick-Edit Panel



When you plot contours, you can temporarily modify the number of colors in the colormap by changing the number of contour levels in the Contours Dialog Box (p. 5412); you will only need to use the Colormap Dialog Box (p. 5280) if you want to change other characteristics of the colormap.

Important:

Note that if you are using a gray-scale colormap and you want to save a gray-scale picture, you should save a color picture. When you save a gray-scale picture, Ansys Fluent uses an internal gray scale, not the gray scale specified by the colormap. If you save a color picture, the colormap you selected (that is, your gray scale) will be used.

40.2.7.1. Colormap Nomenclature

The following are a list of common terms and definitions used in colormap labels:

- **Diverging**—when converted to greyscale (for example, when printing) these are bright in the center and darker at the ends.
- **Field**—using scenes (Generating a Scene (p. 3935)) you can visualize multiple fields within a single plot, such as pressure and temperature. To clearly differentiate the field variables, it is recommended that you use different colormaps for each variable. The field names included in the name of a predefined colormap are meant to indicate the recommended colormap for visualizing a particular variable.
- **Flow-feature-N-bright-peaks**—these are intended to help you identify relevant flow features. *N* indicates the number of bright peaks in the colormap.
- **Fringes**—brightness alternates multiple times.
- **Highlights**—used as a suffix. The colormap has 9 highlights, dividing it into 10 levels.
- **Sequential**—when converted to greyscale, these colormaps are dark at the bottom and bright at the top.

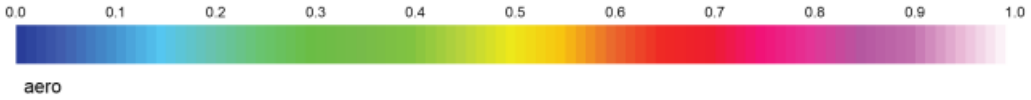
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Split**—two or more different colormaps in one. When there are two, on eis above the midpoint and one below.
- **Stacked**—three different colormaps stacked. The center one is greyscale. There is a line in a contrasting color at the midpoint.
- **Striped**—multiple contrasting stripes to better indicate quantitative values.
- **Symmetric**—symmetric around the midpoint.
- **Value**—colorful for the indicated value ranges, monochromatic outside the ranges.

40.2.7.2. Predefined Colormaps

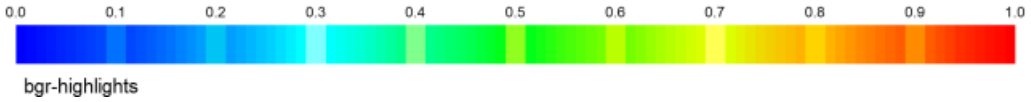
The following colormaps are automatically available in Ansys Fluent:

aero:

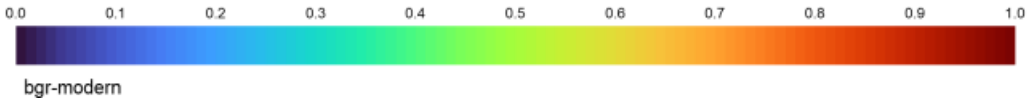


bgr:

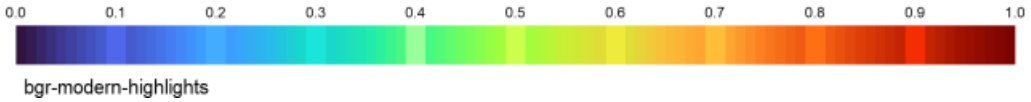
bgr-highlights:



bgr-modern:



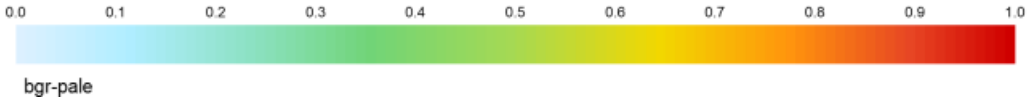
bgr-modern-highlights:



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

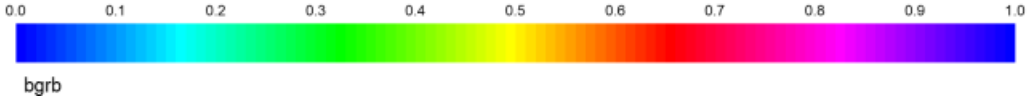
bgr-more-red:

bgr-pale:

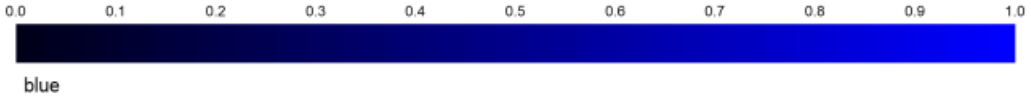


bgr-pink:

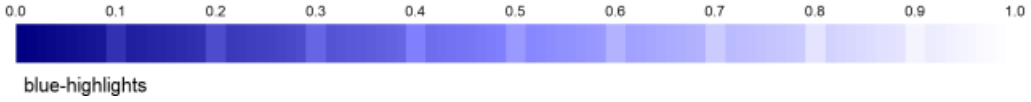
bgrb:



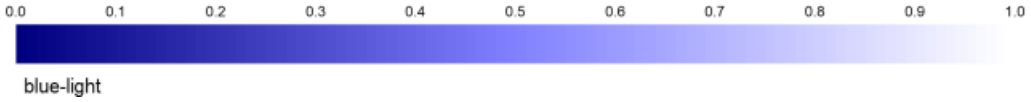
blue:



blue-highlights:

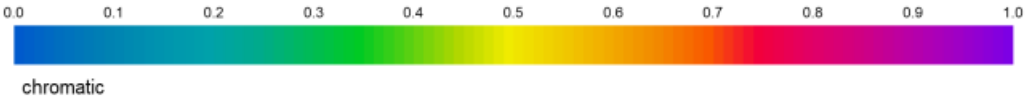


blue-light:

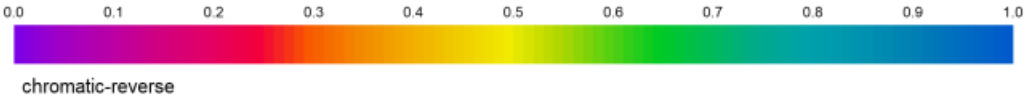


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

chromatic:

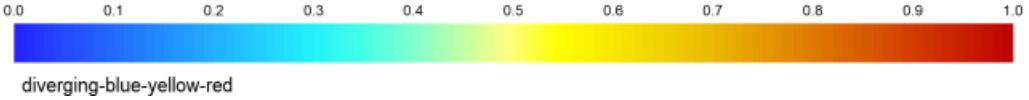


chromatic-reverse:

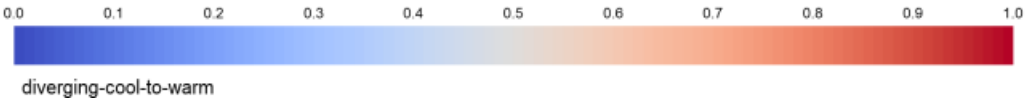


cyan-yellow:

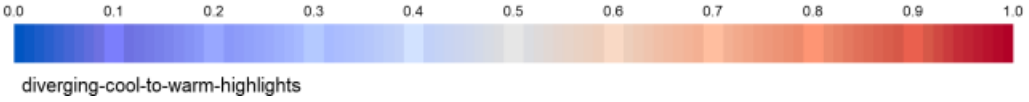
diverging-blue-yellow-red:



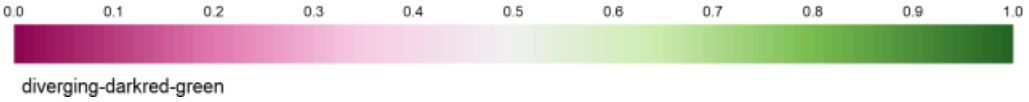
diverging-cool-to-warm:



diverging-cool-to-warm-highlights:

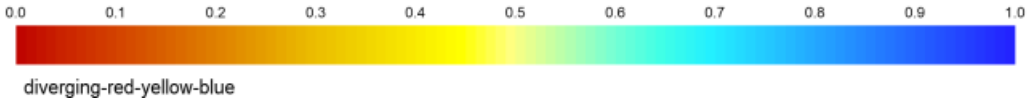


diverging-darkred-green:



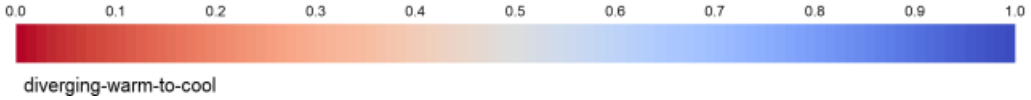
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

diverging-red-yellow-blue:

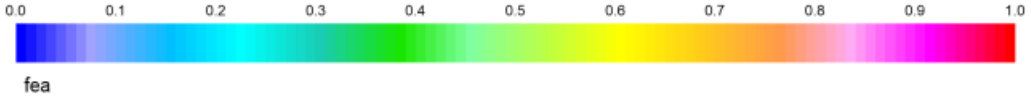


diverging-spectral:

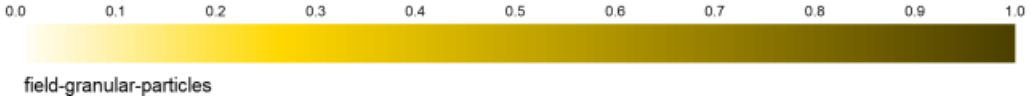
diverging-warm-to-cool:



fea:

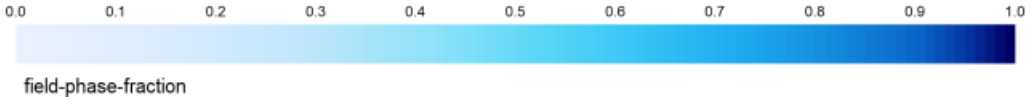


field-granular-particles:



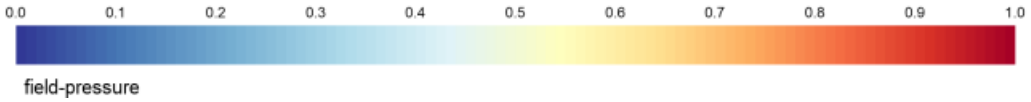
field-liquid:

field-phase-fraction:

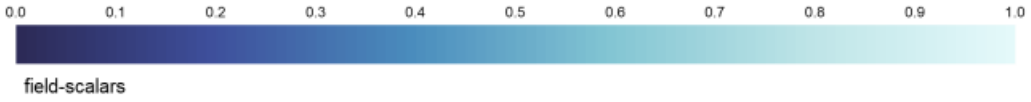


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

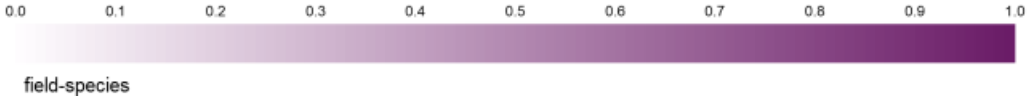
field-pressure:



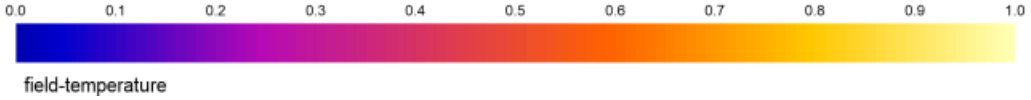
field-scalars:



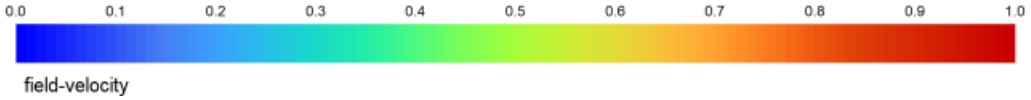
field-species:



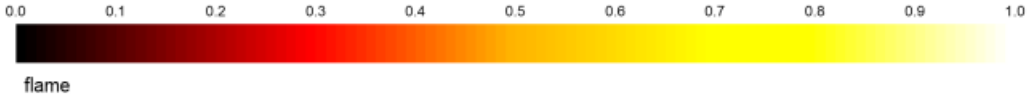
field-temperature:



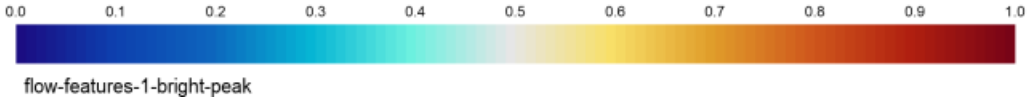
field-velocity:



flame:

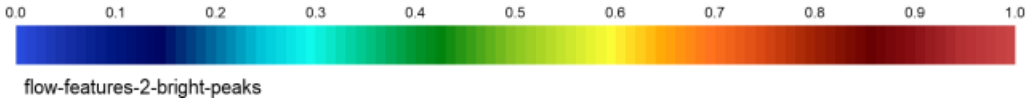


flow-features-1-bright-peak:

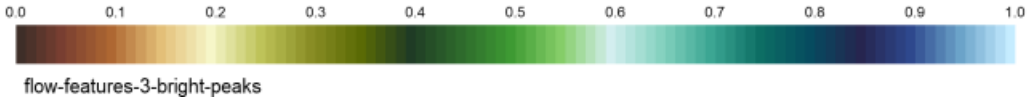


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

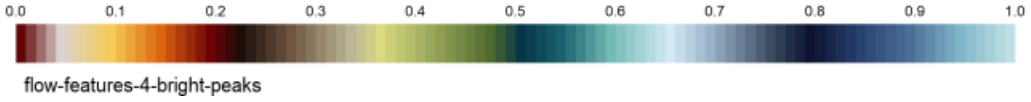
flow-features-2-bright-peaks:



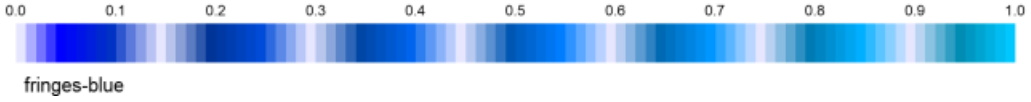
flow-features-3-bright-peaks:



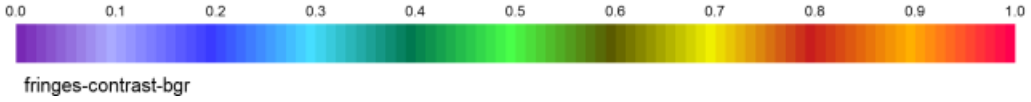
flow-features-4-bright-peaks:



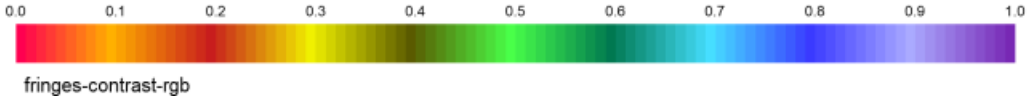
fringes-blue:



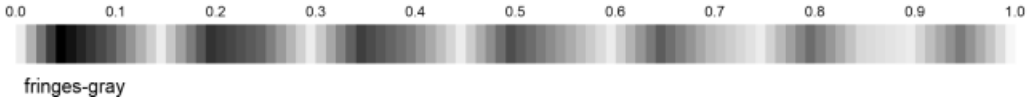
fringes-contrast-bgr:



fringes-contrast-rgb:

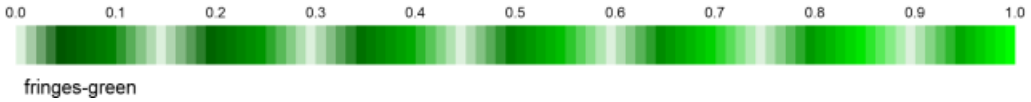


fringes-gray:

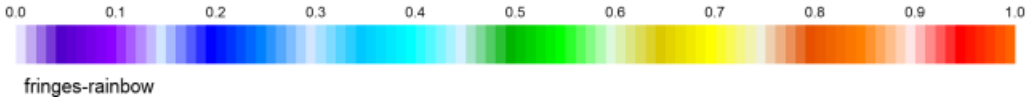


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

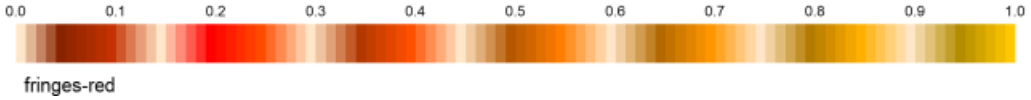
fringes-green:



fringes-rainbow:



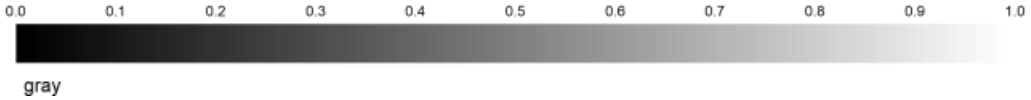
fringes-red:



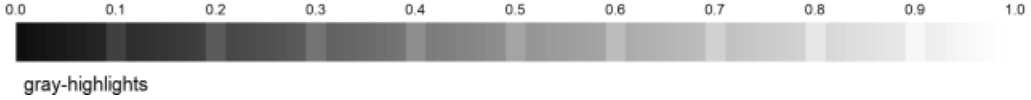
gambit:

gambit-reverse:

gray:

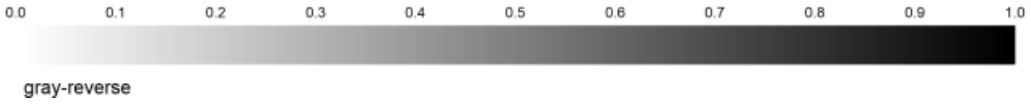


gray-highlights:

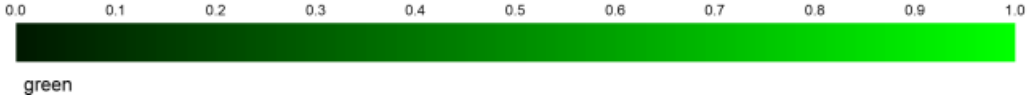


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

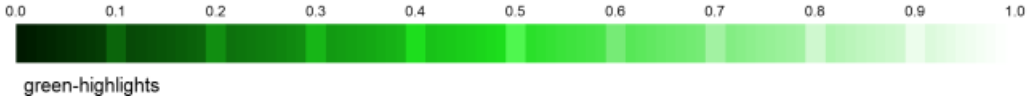
gray-reverse:



green:

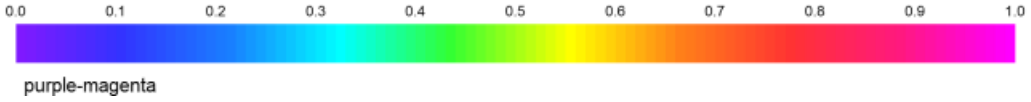


green-highlights:

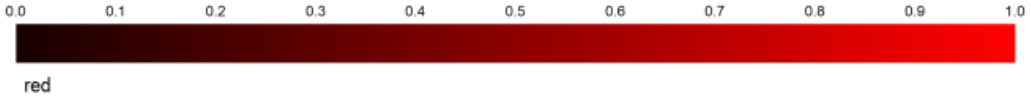


green-light:

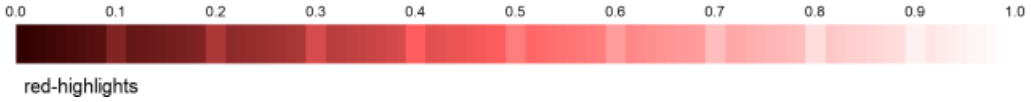
purple-magenta:



red:



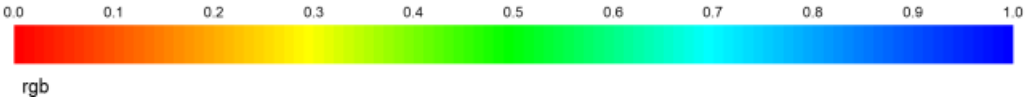
red-highlights:



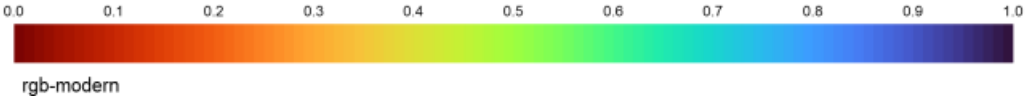
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

red-light:

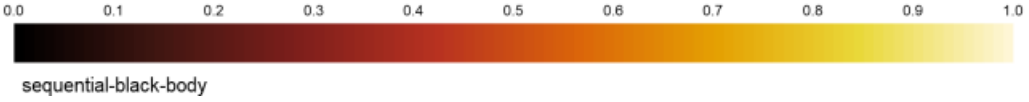
rgb:



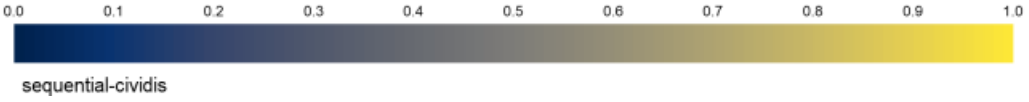
rgb-modern:



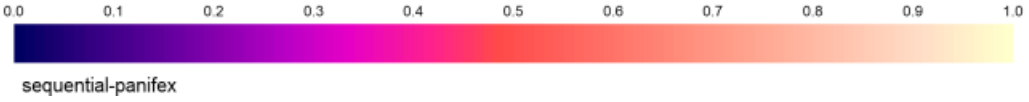
sequential-black-body:



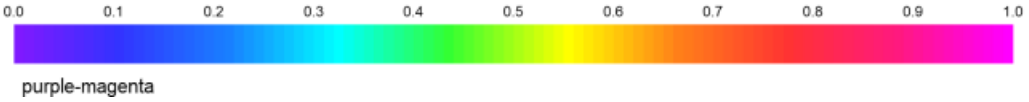
sequential-cividis:



sequential-inferno:



sequential-panifex:

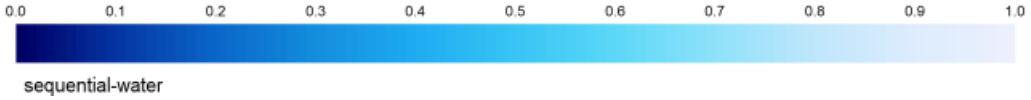


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

sequential-plasma:

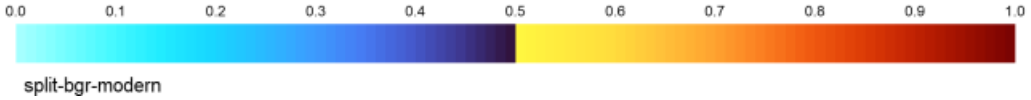
sequential-viridis:

sequential-water:



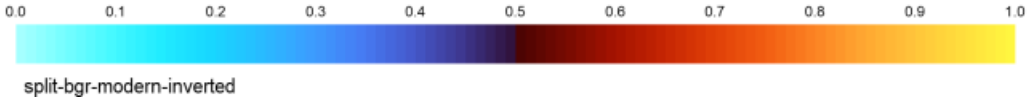
split-bgr-gray:

split-bgr-modern:



split-bgr-modern-black:

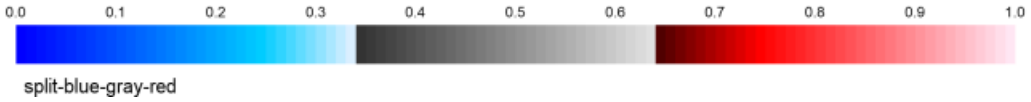
split-bgr-modern-inverted:



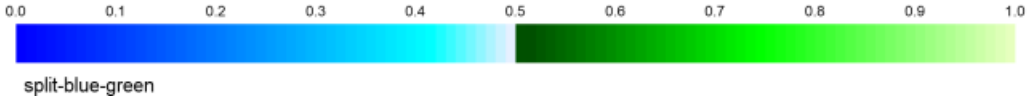
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

split-bgr-modern-white:

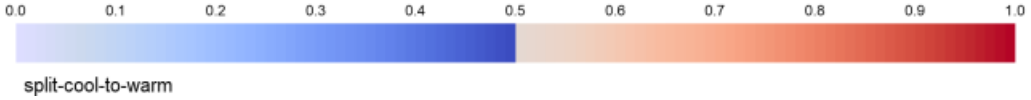
split-blue-gray-red:



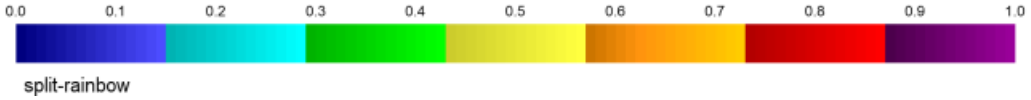
split-blue-green:



split-cool-to-warm:

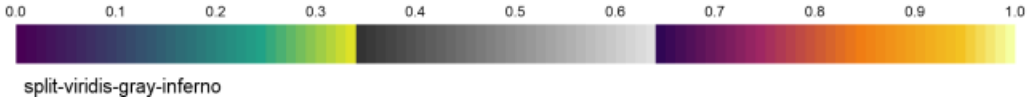


split-rainbow:



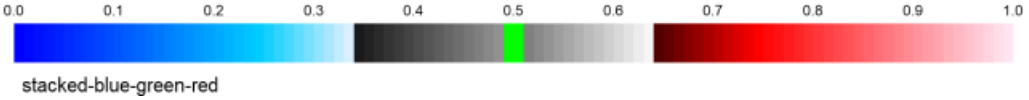
split-shaded-bgr:

split-viridis-gray-inferno:

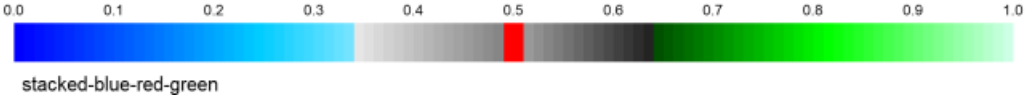


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

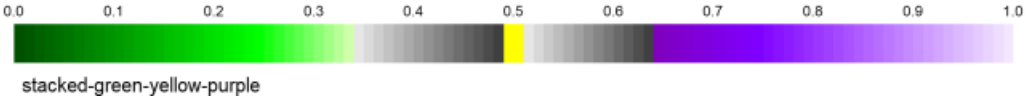
stacked-blue-green-red:



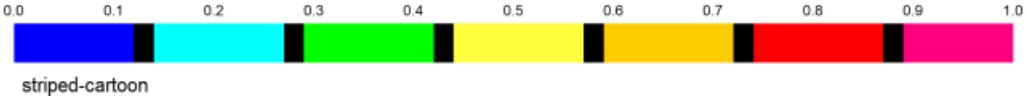
stacked-blue-red-green:



stacked-green-yellow-purple:

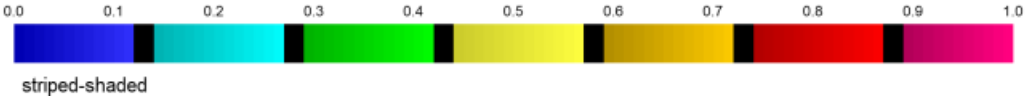


striped-cartoon:

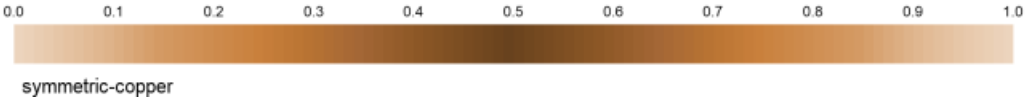


striped-gray:

striped-shaded:



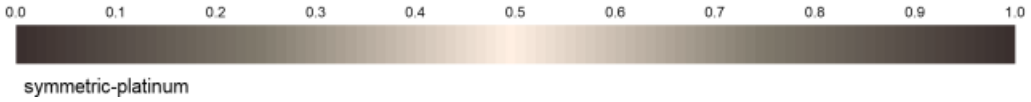
striped-wide-white:



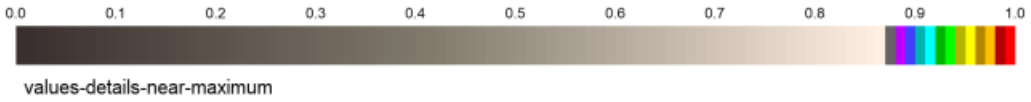
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

symmetric-copper:

symmetric-platinum:



values-details-near-maximum:

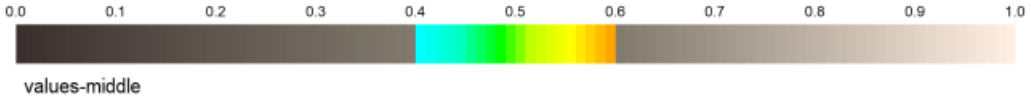


values-high:

values-high-and-low:

values-low:

values-middle:



The number of colors interpolated between the colors in the scale name (for example, between purple and magenta) will depend on the size of the colormap.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

40.2.7.3. Selecting a Colormap

The procedure for selecting a new colormap to be used in graphics displays is as follows:

1. In the Colormap Dialog Box (p. 5280) (Figure 40.66: The Colormap Dialog Box (p. 3965)), select the desired colormap in the **Currently Defined** drop-down list. This list will contain all of the colormaps predefined by Ansys Fluent as well as any custom colormaps that you have created as described in Creating a Customized Colormap (p. 3982).

2. Set the colormap size and scale as described in Specifying the Colormap Size and Scale (p. 3980).

3. Click **Apply** to update the current graphics display with the new colormap. All future displays will use the newly selected colormap and options.

40.2.7.3.1. Specifying the Colormap Size and Scale

Once you have selected the desired colormap from the **Currently Defined** list, you may modify the **Colormap Size** (recommended that you use multiples of 10). This value is the number of distinct colors in the color scale.

You can choose to use a logarithmic scale instead of a decimal scale by turning on the **Log Scale** option. With a log scale, the color used in the graphics display will represent the log of the value at that location in the domain. The values represented by the colors will, therefore, increase exponentially.

You can control the appearance of the colormap in the graphics window by modifying the **Length** and **Width Ratio** fields under **Colormap Dimensions** in the Colormap Dialog Box (p. 5280).

Note:

If you are looking to control the "physical" size of the colormap, that is, where and how large it appears in the graphics window, then you can use the **Alignment**, **Length** and **Width Ratio** controls (as well as others) under **Colormap Settings** the **Appearance** branch of Preferences (**File**>**Preferences...**).

40.2.7.3.2. Changing the Number Format

You can change the format of the labels that define the color divisions at the left of the graphics window using the controls under the **Number Format** heading in the Colormap Dialog Box (p. 5280).

- To display the real value with an integral and fractional part (for example, 1.0000), select **float** in the **Type** drop-down list. You can set the number of digits in the fractional part by changing the value of **Precision**.

- To display the real value with a mantissa and exponent (for example, 1.0e-02), select **exponential** in the **Type** drop-down list. You can define the number of digits in the fractional part of the mantissa in the **Precision** field.

- To display the real value with either float or exponential form, depending on the size of the number and the defined **Precision**, choose **general** in the **Type** drop-down list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

40.2.7.4. Displaying Colormap Labels and Titles

You can control which titles are included with the colormap as well as customize the number of values displayed on the colormap. The default titles include the variable name, units, and the name of the associated graphics object. The default number of labels that appear alongside the colormap is determined automatically based on the screen resolution.

Titles are controlled via the **Titles** drop-down list, which contains the following options:

- **Variable and Object Name**—both the variable name and the associated graphics object are listed with the colormap, along with the units.
- **Variable Only**—only the variable name and the units are listed with the colormap.
- **None**—only the units are listed with the colormap.

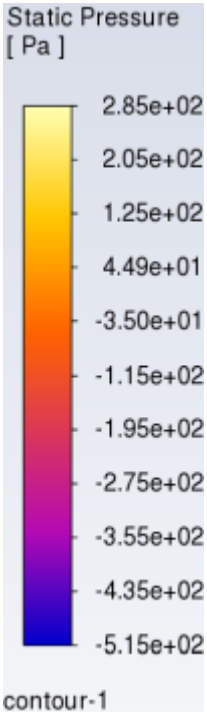
Controlling the number of lables:

- To change the number of labels that appear alongside the colormap, disable **Automatic Skip** in the Colormap Dialog Box (p. 5280) and specify the **Skip**. The total number of labels displayed will then be determined by the **Skip** and **Colormap Size** fields.

  To demonstrate what effect this command has on the display, enter a value of `10` for **Skip** (note that the value entered must be an integer). This will result in 9 intermediate labels being skipped (assuming you maintain the default colormap size) with the first and the last colormap values always being displayed (Figure 40.68: The Colormap with All Titles and Skipped Labels (p. 3981)).

- To reset the original colormap display, simply select **Automatic Skip**.

Figure 40.68: The Colormap with All Titles and Skipped Labels

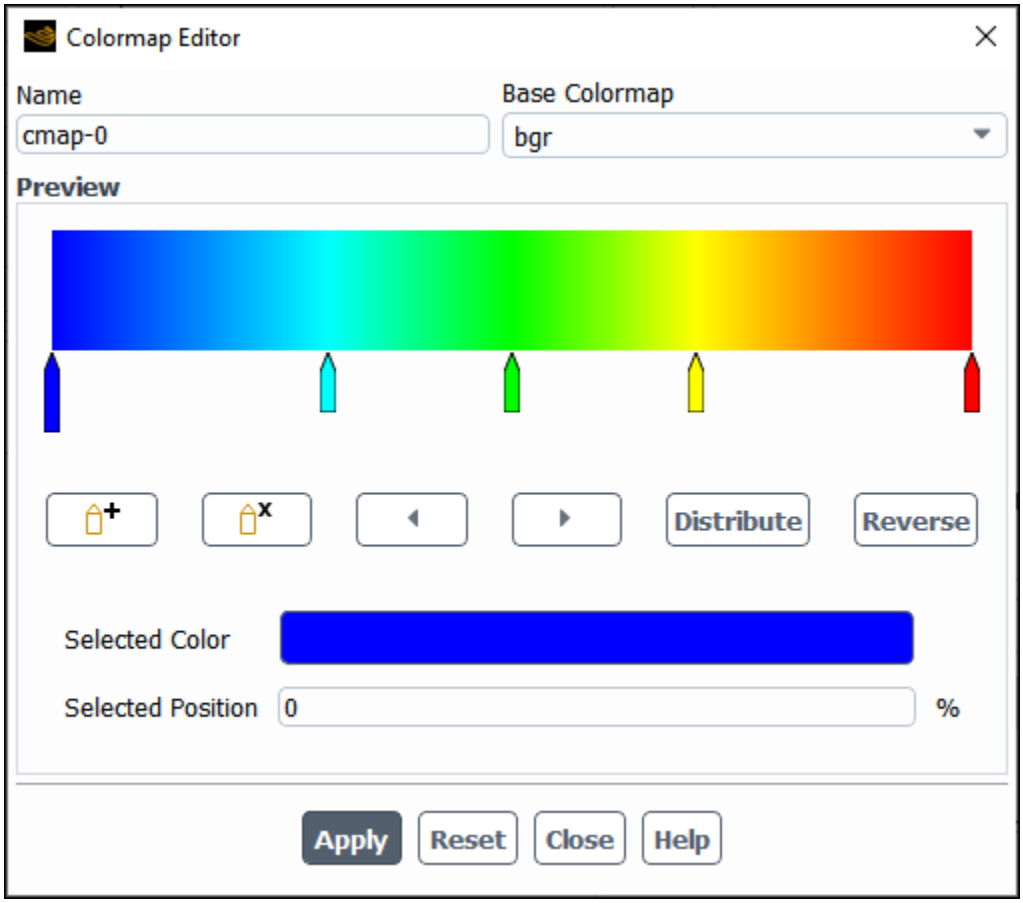


40.2.7.5. Creating a Customized Colormap

You can create your own colormap by manipulating the existing "color stops" and by adding more of them. A color scale is created by linear interpolation between the color stops. The color, number, and position of the anchor colors will therefore control the description of the colormap. By increasing the number of color stops, you can increase the total number of colors and obtain a color scale that changes more gradually. The procedure is as follows:

1. Click **Edit...** in the Colormap Dialog Box (p. 5280) to open the Colormap Editor Dialog Box (p. 5284) (Figure 40.69: The Colormap Editor Dialog Box (p. 3982)).

   Figure 40.69: The Colormap Editor Dialog Box



2. Provide a name for the custom colormap you are creating.

3. Select the colormap you want to use as a starting point for your custom colormap from the **Base Colormap** drop-down list.

4. Drag the colormap stops to customize the color distribution of the colormap.

   - Click "insert stop" ( ) to add a new stop next to the selected stop.

     Alternatively, you can click in the colormap to add new stops; the color of the new stop will be the same as the location you click.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.70: Double-Click to Add Color Stops

- Click "remove stop" ( ) to remove the selected color stop.
- You can either select color stops directly or you can switch which one is selected using the left and right arrows (and).
- You can change the color of the selected stop by clicking in the **Selected Color** field and choosing an alternate color from the **Select Color** dialog box.

  Alternatively, you can double-click a color stop to open the **Select Color** dialog box to change the color for that color stop.
- Instead of dragging the color stops, you can enter the location of the stop on the colormap in the **Selected Position** field.

5. (Optional) Click **Distribute** to evenly distribute the color stops on the colormap.

Important:

The total number of colors must not be less than the number of anchor points.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. (Optional) Click **Reverse** to reverse the order of the colors in the colormap.
7. Click **Apply** to save the custom colormap.
8. (Optional) Click **Reset** to reset the current display to the **Base Colormap**.

Custom colormap definitions will be saved in the case file.

40.2.7.6. Colormap References

The following are a list of resources used in the implementation of colormaps in Ansys Fluent.

Bibliography

[1] "Color Map Advice for Scientific Visualization". Kenneth Moreland. 2015,. https://www.kennethmoreland.com/color-advice/.

[2] Farge, Marie. "Wavelets and Turbulence". http://wavelets.ens.fr/.

[3] Farge, Marie. "Choice of Representation Modes and Color Scales for Visualization in CFD". *Science and Art Symposium 2000*. Kluwer. 91–100. 2000.

[4] Zeller, Stephanie and Rogers, David. "Visualizing Science: How Color Determines What We See". *Eos, Science News by AGU*. May 21, 2020.

[5] Mikhailov, Anton. "CFD Post Processing". Google AI Blog. August 20, 2019. https://ai.googleblog.com/2019/08/turbo-improved-rainbow-colormap-for.html.

[6] Toet, Willem. "Turbo, An Improved Rainbow Colormap for Visualization". Race Tech Magazine. February 5, 2019. https://www.racetechmag.com/2019/02/willem-toet-explains-cfd-post-processing/.

40.2.8. Adding Lights

In Ansys Fluent you can add lights with a specified color and direction to your display. These lights can enhance the appearance of the display when it contains 3D geometries. By default one light is defined. You can enable the effect of the existing light(s) using the Display Options Dialog Box (p. 5262) or the Lights Dialog Box (p. 5279), and you can add new lights using the Lights Dialog Box (p. 5279).

You can control default lighting behavior using the settings listed under **Lighting** in the **Graphics** branch of **Preferences** (Setting User Preferences/Options (p. 937)). Controls are available for:

- Lighting method
- Headlight behavior and intensity
- Ambient light intensity

Important:

While the best lighting method for most situations is handled automatically, if you are displaying mesh faces only, edges may appear rounded due to a third-party issue. To resolve this issue, you can either enable **Edges** in the Mesh Display Dialog Box (p. 4688), or you can

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

select the **Flat** lighting method, either from the drop-down list in the **View** ribbon tab or in the **Lighting Method** drop-down list under **Lighting** in the **Graphics** branch of **Preferences**.

40.2.8.1. Controlling Lighting Effects with the Display Options Dialog Box

Lighting is handled automatically by default, however you can turn the lights off and control the lighting method manually if desired using the Display Options Dialog Box (p. 5262).

Results → Graphics → Options...

The **Lights On** option is enabled with the method set to **Automatic** by default, but you can disable **Lights On** and change the method from the **Lighting** drop-down. Once you have changed the method you must click **Apply** for the change to take effect.

The lighting interpolation methods include: **Flat**, **Gouraud** and **Phong**. Flat is the most basic method: there is no interpolation within the individual polygonal facets. Gouraud and Phong have smoother gradations of color because they interpolate on each facet.

40.2.8.2. Controlling Lighting Effects with the Lights Dialog Box

You can also control lighting effects using the Lights Dialog Box (p. 5279) (Figure 40.71: The Lights Dialog Box (p. 3986)).

Results → Graphics → Lights...

The headlight provides constant lighting effects in the direction of view and it is controlled automatically by default to optimize the appearance for the type of graphics object being displayed. This option has the effect of a light source directly in front of the model, no matter what orientation the model is viewed in. You can control the headlight manually be setting **Headlight On** to **On** or **Off** using the drop-down in the Lights Dialog Box (p. 5279).

Fluent controls the lighting method automatically by default, optimizing the appearance of the graphics for the object being displayed. You have the option to control the method manually by selecting **Flat**, **Gouraud**, or **Phong** from the **Lighting Method** drop-down list. These methods are described in the previous section. Click **Apply** to see any changes made to the lighting effects in the active graphics window.

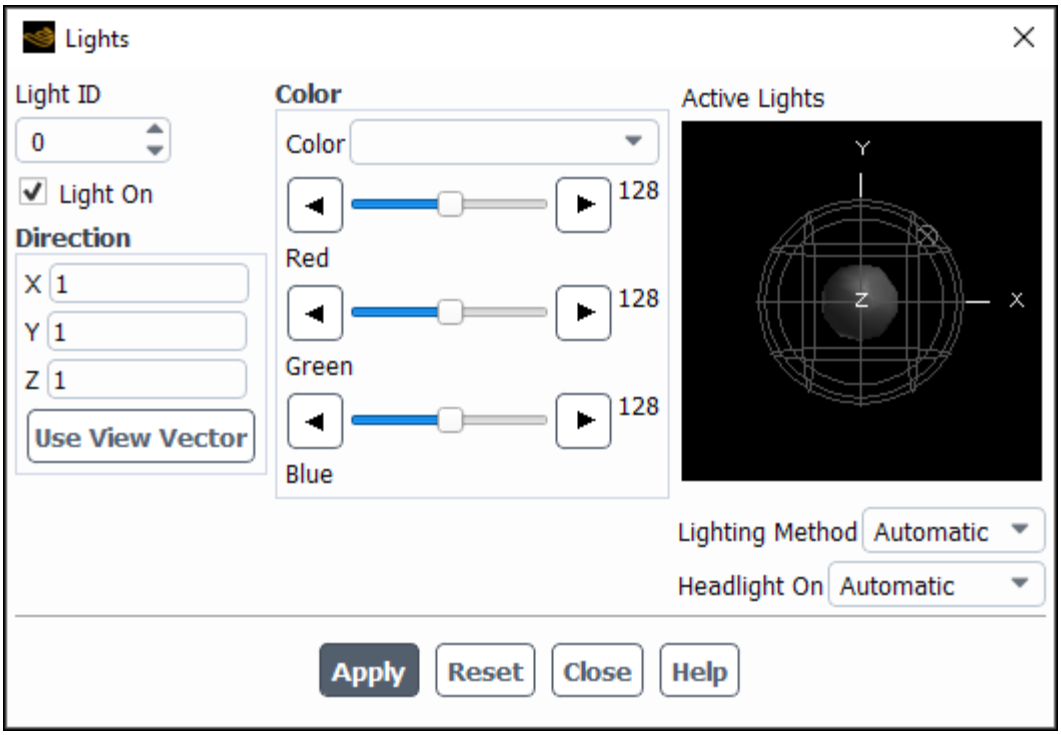
40.2.8.3. Defining Light Sources

You can control individual lights in the Lights Dialog Box (p. 5279) (Figure 40.71: The Lights Dialog Box (p. 3986)). The Lights Dialog Box (p. 5279) enables you to create a light and then turn it off without deleting it. In this way, you can retain lights that you have defined previously but do not want to use at present.

Results → Graphics → Lights...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.71: The Lights Dialog Box



(You can also open the Lights Dialog Box (p. 5279) by clicking **Lights...** in the Display Options Dialog Box (p. 5262).)

By default, light 0 is defined to be dark gray with a direction of (1,1,1). A light source is a distant light, similar to the sun. The direction (1,1,1) means that the rays from the light will be parallel to the vector from (1,1,1) to the origin. To create an additional light (for example, light 1), follow the steps listed below.

1. Increase **Light ID** to a new value (for example, 1).
2. Enable the **Light On** check box.
3. Define the light color by entering a descriptive string (for example, `lavender`) in the **Color** field, or by moving the **Red**, **Green**, and **Blue** sliders to obtain the desired color. The default color for all lights is dark gray.
4. Specify the light direction by doing one of the following:
   - Enter the (**X**, **Y**, **Z**) Cartesian components under **Direction**.
   - Click the middle mouse button in the desired location on the sphere under **Active Lights**. (You can also move the light along the circles on the surface of the sphere by dragging the mouse while holding down the middle button.) You can rotate the sphere by pressing the left mouse button and moving the mouse (like a trackball).
   - Use your mouse to change the view in the graphics window so that your position in reference to the geometry is the position from which you would like a light to shine. Then click **Use View Vector** to update the **X,Y,Z** fields with the appropriate values for your current position

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and update the graphics display with the new light direction. This method is convenient if you know where you want a light to be, but you are not sure of the exact direction vector.

5. Repeat steps 1–4 to add more lights.

6. When you have defined all the lights you want, click **Apply** to save their definitions.

40.2.8.3.1. Removing a Light

To remove a light, enter the ID number of the light to be removed in the **Light ID** field and then clear the **Light On** check box. When a light is turned off, its definition is retained, so you can easily add it to the display again at a later time by selecting the **Light On** check box. For example, you may want to define three different lights to be used in different scenes. You can define each of them, and then enable only one or two at a time, using the **Light ID** field and the **Light On** check box. Once you have made all the desired modifications to the lights, remember to click **Apply** to save the changes.

40.2.8.3.2. Resetting the Light Definitions

If you have made changes to the light definitions, but you have not yet clicked **Apply**, you can reset the lights by clicking **Reset**. All lighting characteristics will revert to the last saved state (that is, the lighting that was in effect the last time you opened the dialog box or clicked on **Apply**).

40.2.9. Modifying the Rendering Options

Depending on the objects in your display window and what kind of graphics hardware and software you are using, you may want to modify some of the rendering parameters listed below. All are listed under the **Rendering** heading in the Display Options Dialog Box (p. 5262) (Display Options Dialog Box (p. 5262)).

Results → Graphics → Options...

After making a change to any of these rendering parameters, click **Apply** to update how new graphics objects will appear. Note that some changes, such as those to line width and the logo are updated in all applicable graphics windows once you click **Apply**.

Line Width:

By default, all lines drawn in the display have a thickness of 1 pixel. If you want to increase the thickness of the lines, increase the value of **Line Width**.

Point Symbol:

By default, nodes displayed on surfaces and data points on line or rake surfaces are represented in the display by a + sign inside a circle. If you want to modify this representation (for example, to make the nodes easier to see), you can select a different symbol in the **Point Symbol** drop-down list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Animation Options:

There are two animation options that you can choose from. They are as follows:

All

uses a solid-tone shading representation of all geometry during mouse manipulation.

Wireframe

uses a wireframe representation of all geometry during mouse manipulation. If your computer has a graphics accelerator, you may not want to use this option; otherwise, the mouse manipulation may be very slow.

Double Buffering:

Enabling the **Double Buffering** option can dramatically reduce screen flicker during graphics updates. Note, however, that if your display hardware does not support double buffering and you turn this option on, double buffering will be done in software. Software double buffering uses extra memory.

Front Faces Transparent:

This option enables you to turn off the display of outward pointing faces of shells or meshes. This is sometimes useful for displaying both sides of a slit wall. By default, when you display a slit wall, one side will "bleed" through to the other. When you enable the **Front Faces Transparent** option, the display of a slit wall will show each side distinctly as you rotate the display. This option can also be useful for displaying two-sided walls (that is, walls with fluid or solid cells on both sides). Please note, however, that enabling it can also hides some unintended surfaces on certain viewing angles, e.g. while displaying Contours, it might be visible only from one side of the surface and not from the other

Hidden Surface Removal:

If you do not use hidden surface removal, Ansys Fluent will not try to determine which surfaces in the display are behind others; it will display all of them, and a cluttered display will result for most 3D mesh displays. For most 3D problems, therefore, you should enable the **Hidden Surface Removal** option. You should turn this option off (for optimal performance) if you are working with a 2D problem or with geometries that do not overlap.

You can choose one of the following methods for performing hidden surface removal in the **Hidden Surface Method** drop-down list. These options vary in speed and quality, depending on the device you are using.

Hardware Z-buffer

is the fastest method if your hardware supports it. The accuracy and speed of this method is hardware-dependent. Note that if this method is not available on your computer, selecting it will cause the **Software Z-buffer** method to be used.

Painters

will show fewer edge-aliasing effects than **Hardware-Z-buffer**. This method is often used instead of **Software-Z-buffer** when memory is limited.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Software Z-buffer

is the fastest of the accurate software methods available (especially for complex scenes), but it is memory-intensive.

Z-sort only

is a fast software method, but it is not as accurate as **Software-Z-buffer**.

40.2.9.1. Graphics Device Information

If you need to know which graphics driver you are using and what graphics hardware it recognizes, you can click **Info** in the Display Options Dialog Box (p. 5262). The graphics device information will be printed in the text (console) window.

Important:

Ensure your graphics driver is up-to-date to avoid issues with graphics displays (such as the display coming upside-down).

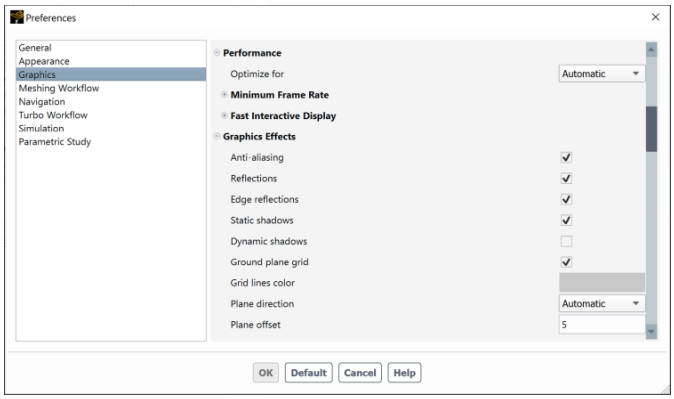
40.3. Enhanced Graphics Visual Effects

You can optimize the display of your model and results in the graphics window using preferences. Preference options allow you to choose whether you want better graphics or better performance. These pre-defined selections enable/disable various visual effects, letting you quickly change Fluent's graphics behavior. You can also elect to enable/disable the different graphics effects individually. Figure 40.76: Model with All Graphics Effects Enabled (p. 3995) demonstrates all of the available graphics effects.

To enable enhanced visual effects:

1. Open the **Preferences** dialog box by selecting **Preferences...** in the **File** drop-down menu.

   File → Preferences...



2. Enable the desired visual effects for **Graphics**.

Note:

It is important to verify that you have a sufficient graphics card and driver for the effects to appear as intended. For Linux, this means running with `OpenGL2`. On Windows, this means running with `DX11` or `OpenGL2`.

40.3.1. Optimizing Graphical Priorities

You can select from predefined selections of graphics effects to optimize performance for your situation.

The following selections are available under the **Performance** branch:

- **Automatic**—balances graphics performance with computer performance and adjusts the enabled graphics effects based on the current graphics driver.
- **Enhanced Graphics**—emphasizes graphics performance over computer performance, enabling all of the options under **Graphics Effects**.
- **Enhanced Performance**—emphasizes computer performance over graphics performance, disabling all of the options under **Graphics Effects**. **Fast Interactive Display** is also enabled and set to automatically start working for heavy geometries.
- **Custom**—indicates that you have selected the **Graphics Effects** that best suit your requirements.

You can learn more about the individual graphics effects in Graphics Effects Options (p. 3990).

40.3.2. Graphics Effects Options

As an alternative to selecting one of the predefined groupings of graphics effects, you can customize the selection of graphics effects to best suit your needs.

The following graphics effects are available:

Note:

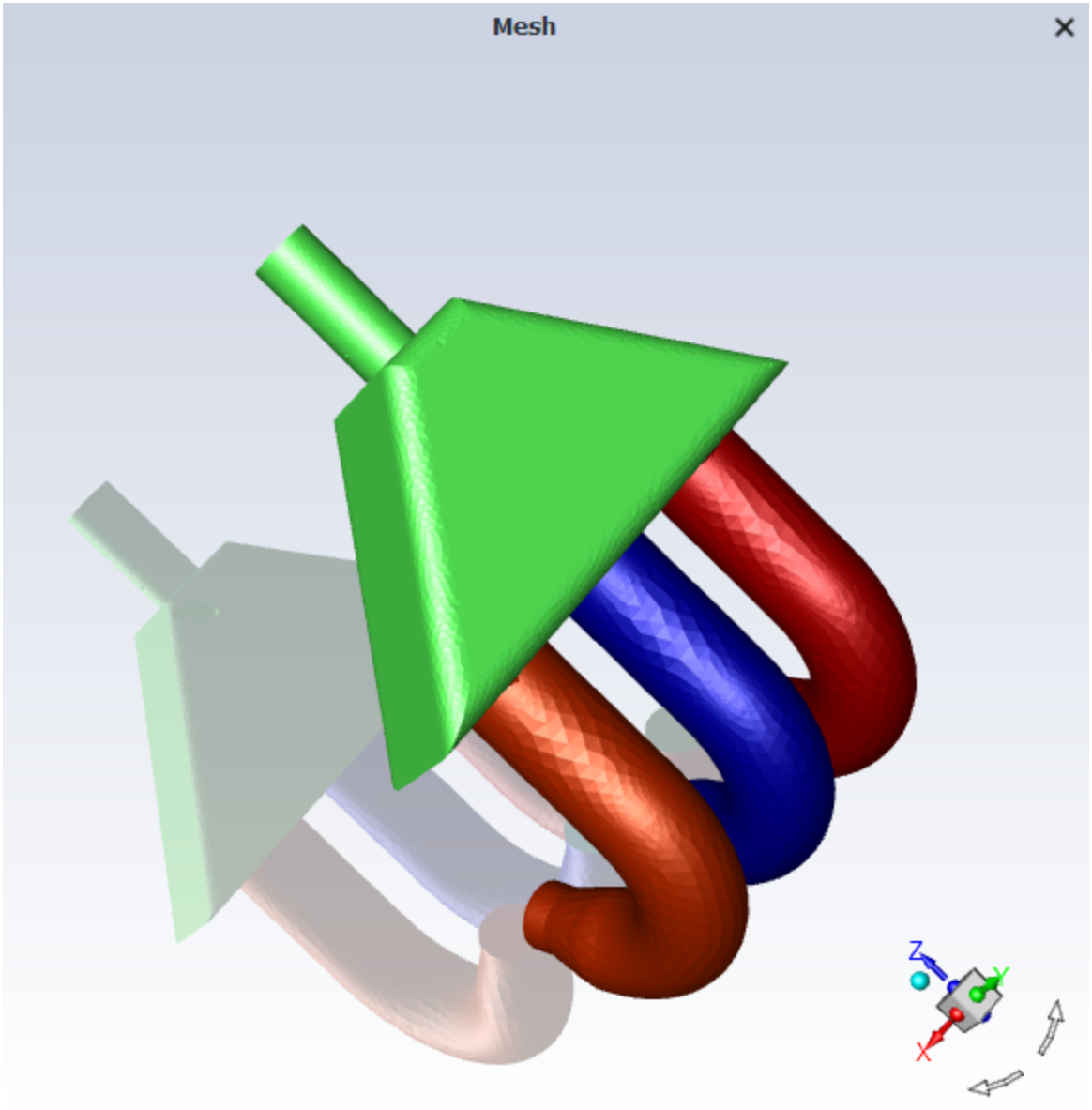
Enabling the **Reflections**, **Static shadows**, or **Ground plane grid** graphics effects adds two new fields to the **Graphics Effects** branch:

- **Plane direction**—controls which plane the shadow/reflection/ground plane grid is displayed in.
- **Plane offset**—specifies the distance between the model and the shadow/reflection/ground plane grid.

- **Anti-aliasing**—smooths model edges and lines as well as text.
- **Reflections**—creates a mirrored reflection of the model off of the specified plane.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

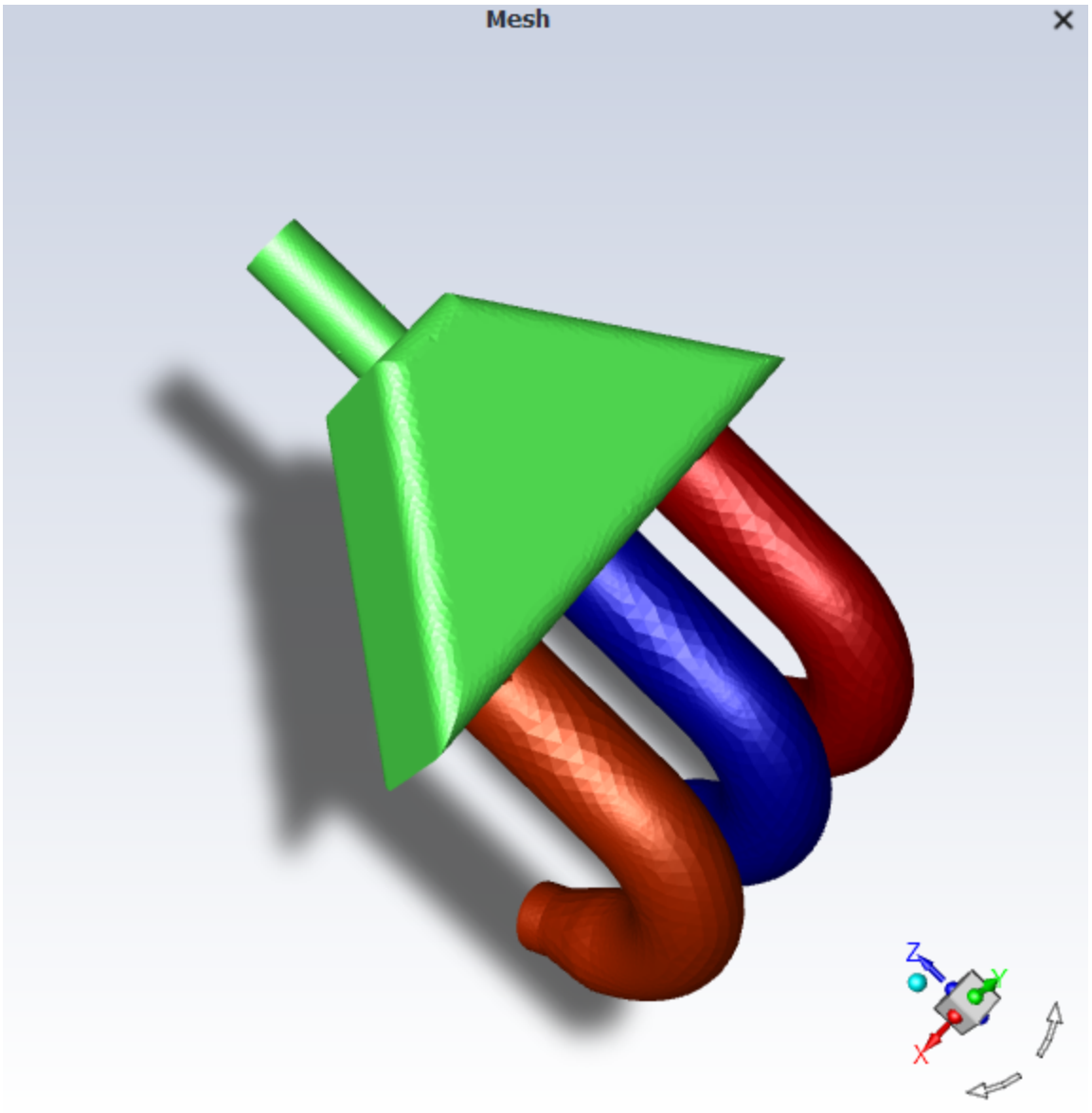
Figure 40.72: Model with Reflections Enabled



- **Static shadows**—shows a static shadow of the model on the specified base plane. This shadow is not affected by the current lighting conditions. *This option does not have much effect on the performance because it can be rendered quickly by the graphics card*.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

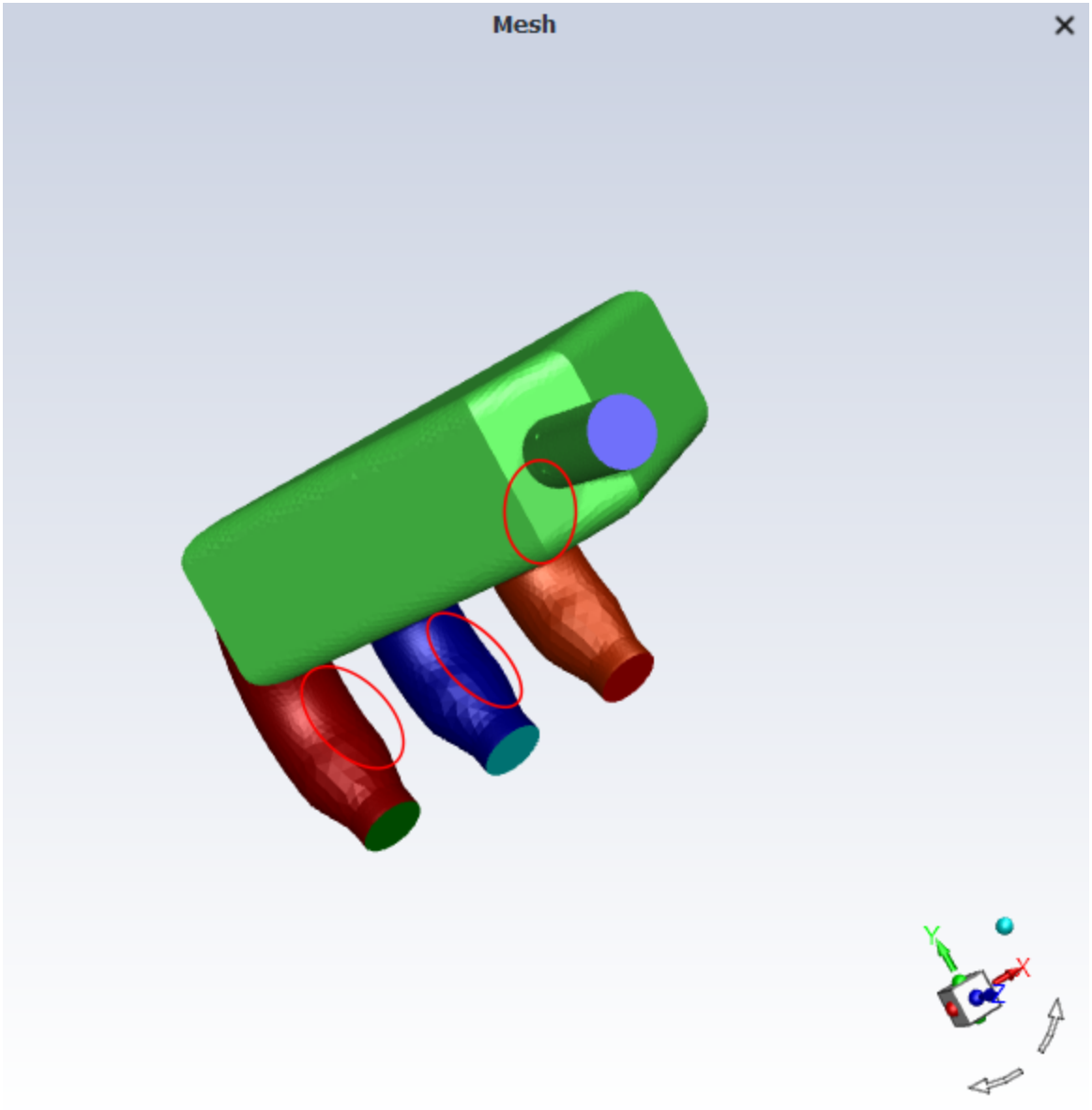
Figure 40.73: Model with Static Shadows Enabled



- **Dynamic shadows**—shows shadows of every part of the model on the appropriate surfaces, depending on the current lighting conditions. This option is more computationally expensive than static shadows because it is recalculated as you move the model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

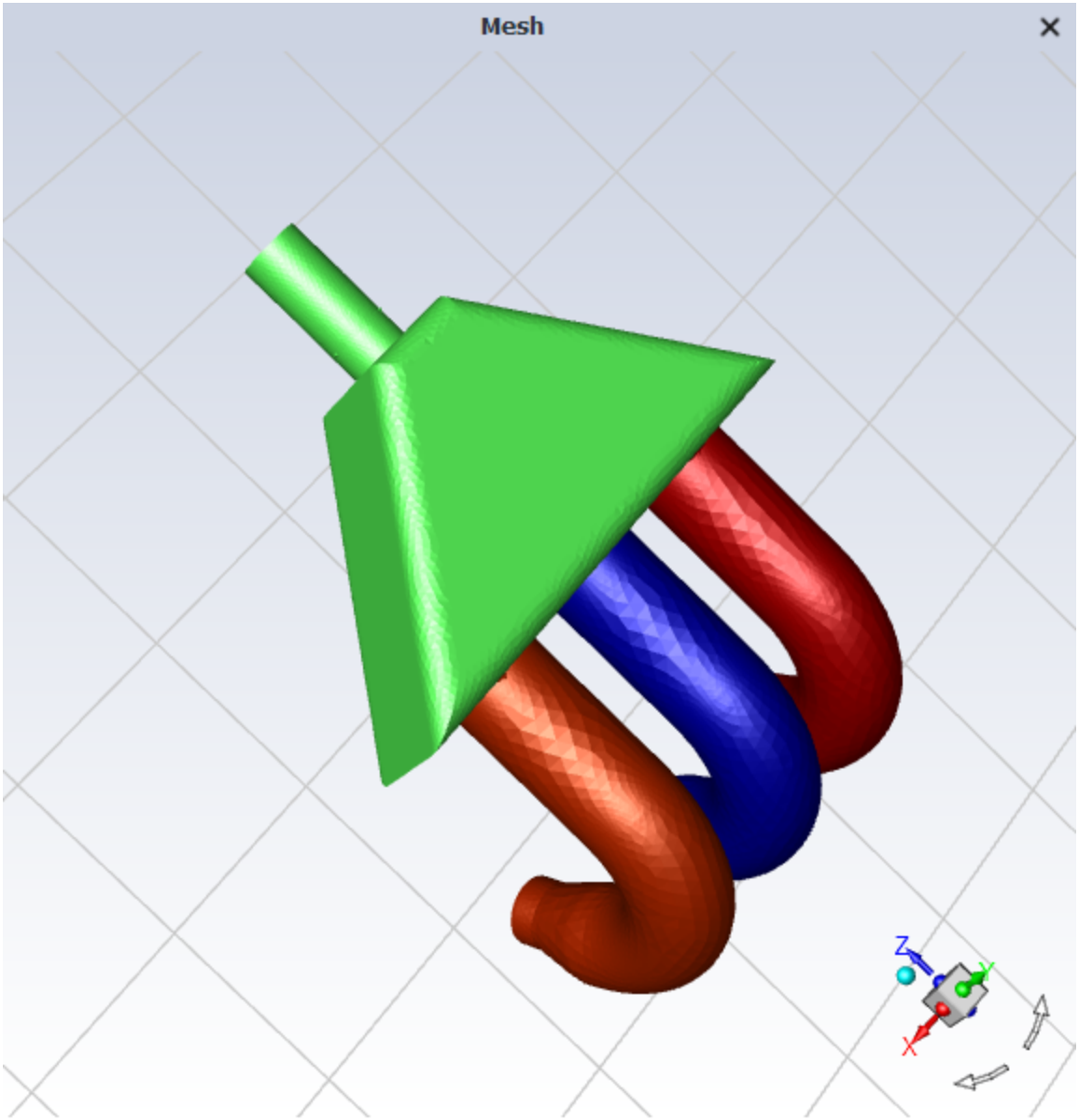
Figure 40.74: Model with Dynamic Shadows Enabled



- **Ground plane grid**—displays a finite grid at the base plane specified by the **Plane direction** field. This option also adds an additional field under **Graphics Effects**, called **Grid Lines Color**, which allows you to specify the color of the mesh lines added to the display.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

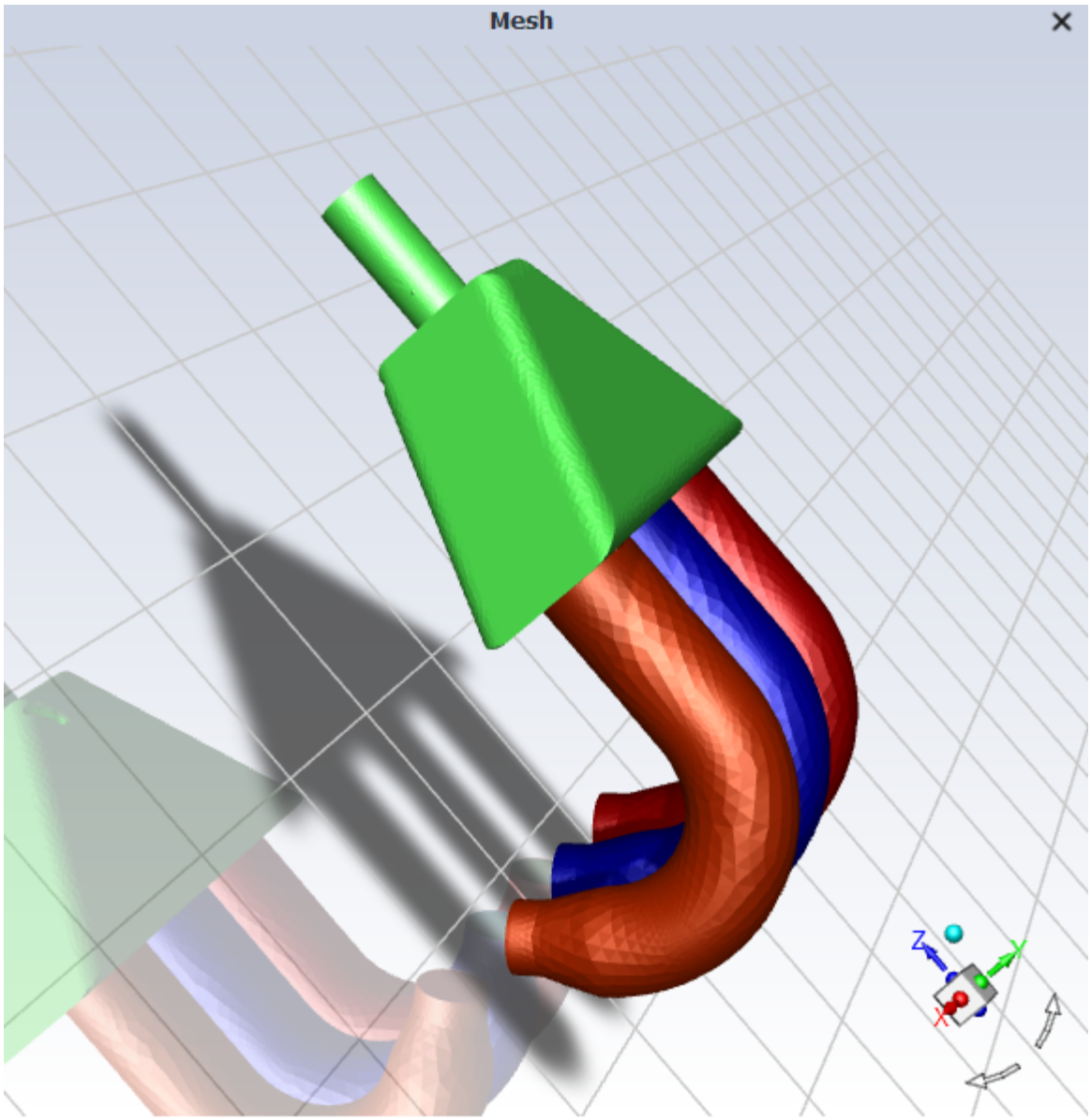
Figure 40.75: Model with Ground Plane Grid Displayed



- **Update after mouse interactions**—redisplays the graphics effects after each time the graphics window view changes.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.76: Model with All Graphics Effects Enabled



Note:

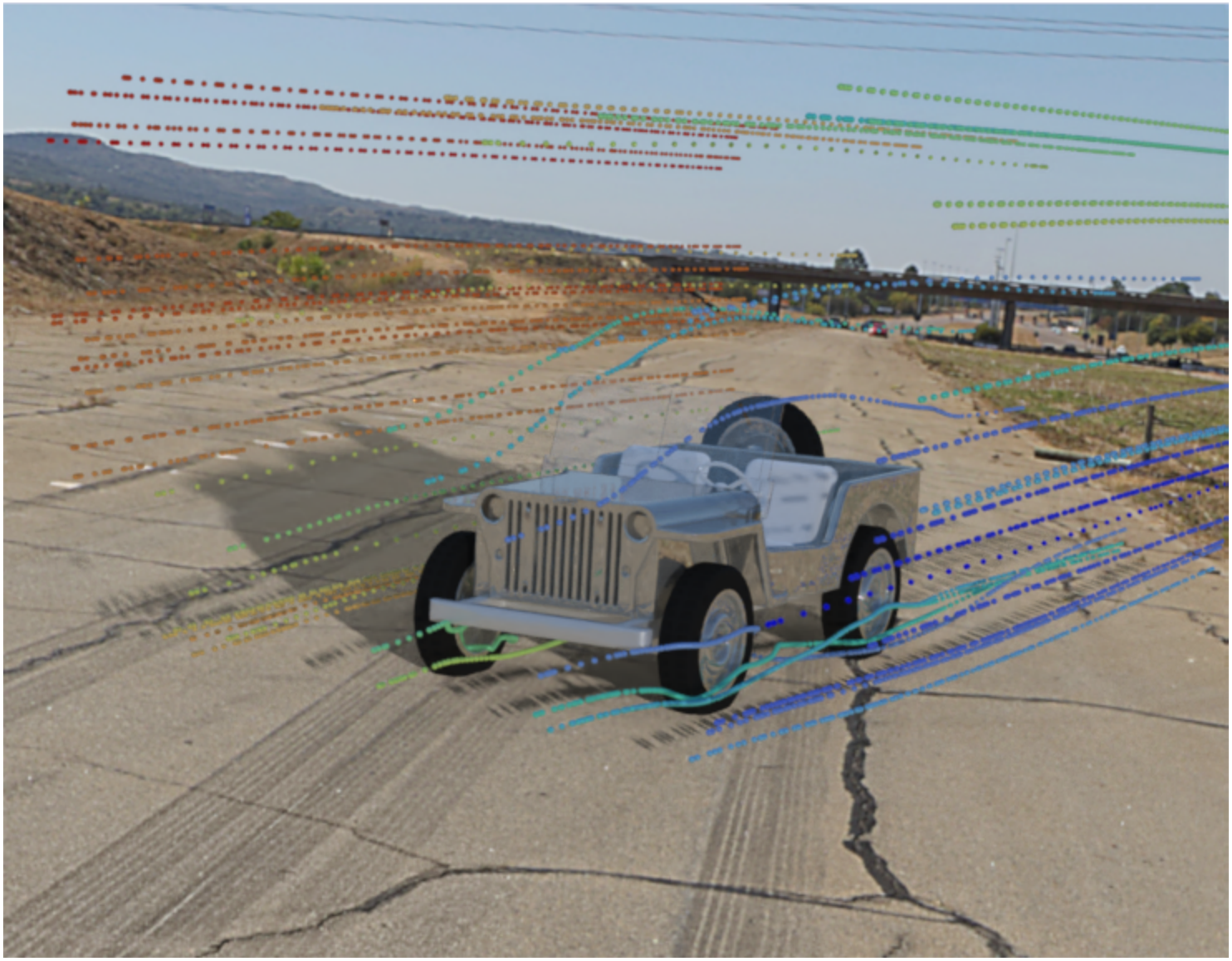
The Reflections, Static shadows, and Dynamic shadows options do not apply to particle tracking.

40.4. Realistic Rendering Using Raytracing

You can perform realistic rendering of your model using raytracing, as shown in Figure 40.77: Example of Realistic Rendering on a Highway Midday (p. 3996).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.77: Example of Realistic Rendering on a Highway Midday



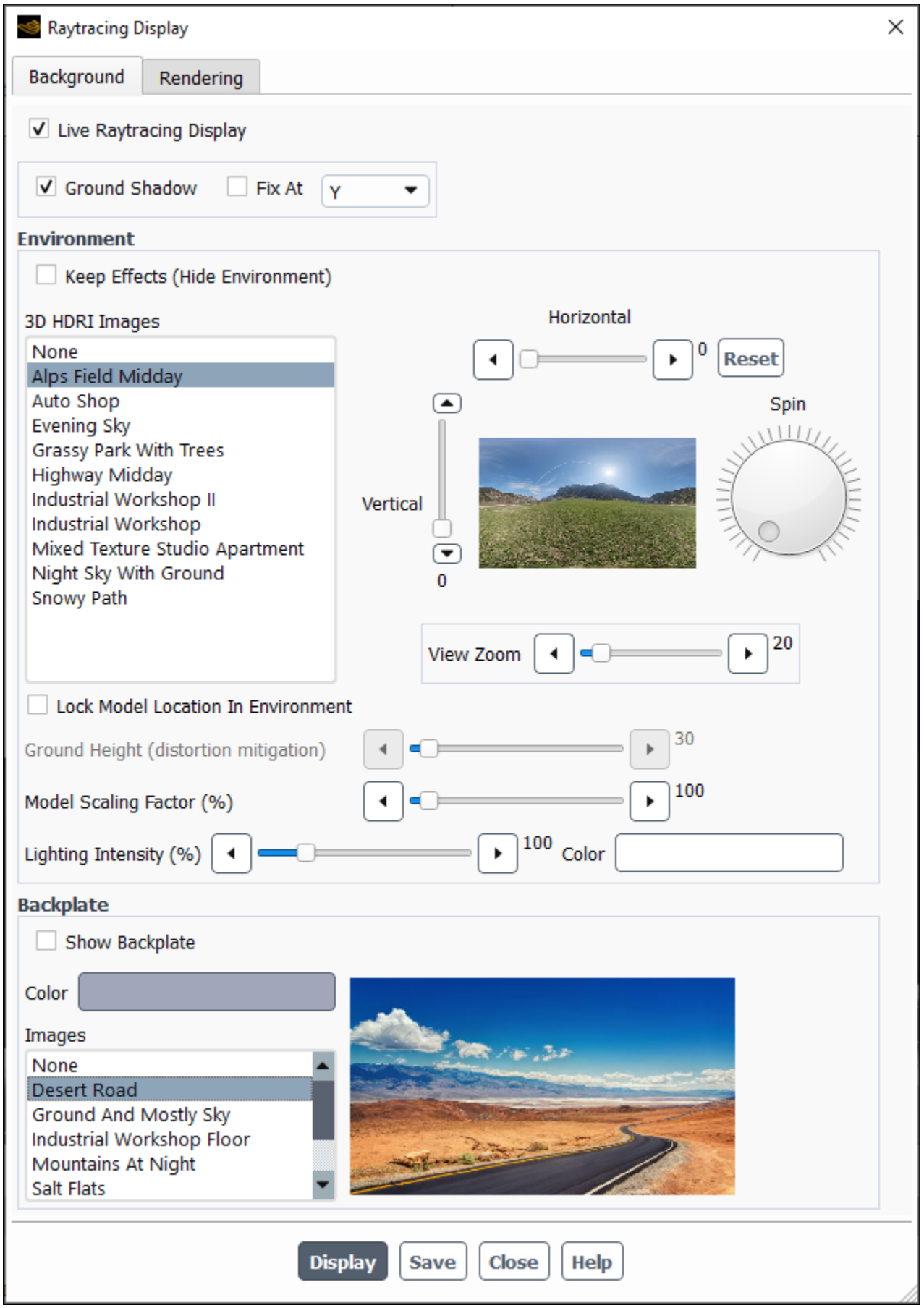
To create a realistic raytracing display:

1. Display the graphics object that you want to render with raytracing. *Mesh objects allow you to color your model with realistic materials that look especially good in the raytracing display. See* *Realistic Rendering of Materials (p. 3882)* *for additional information.*

2. Open the **Raytracing Display** dialog box.

   View → **Graphics** → **Raytracing Display...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. (Optional) Enable **Live Raytracing Display** to make it so that changes made to controls in the **Background** tab are immediately reflected in the raytracing window (which is only seen after you click **Display**).

4. (Optional) Enable **Ground Shadow** to have the displayed object cast shadows on the environment based on the defined lights and environment lighting.

   With the **Ground Shadow** enabled, you can also choose to lock the location of that shadow onto a plane by enabling **Fix At** and selecting the desired axis from the drop-down list. For example, if you have a model that is defined such that +Z represents the "up" direction, you would want to lock the shadow at **Z**.

   Note:

   You may want to define additional lights (Defining Light Sources (p. 3985)) to make the shadow(s) more apparent.

5. (Optional) Enable **Keep Effects (Hide Environment)** to keep the effects of the selected environment (such as lighting and reflections) without displaying the environment.

6. Select the environment you want displayed with your model from the **3D HDRI Images** list. Refer to Listing of Realistic Environments and Backplates (p. 4010) for more information on the available environments.

   Determine if you want to display your model in a 3D environment or if you just want a backplate as the background behind your model. If you want lighting effects on your model, you must pick an environment. You can then decide whether you also want one of the available backplates behind your model. Options for backplates include colors as well as a few different images.

7. Modify the environment properties to suit your requirements:

   - Adjust the orientation and appearance of the environment with respect to your model, using the **Vertical**, **Horizontal**, and **Spin** controls. *These controls operate based on the point of view of the graphics window. Since these controls are based on the current view, rather than a fixed XYZ origin, they may not ensure a consistent appearance even when the same values are used.*

   - Adjust the field of view using the **View Zoom** slider, which adjusts how much of the 3D environment is shown in the graphics window. You can resize the model as needed to match the environment using the **Model Scaling Factor** slider.

   - Fine tune the relationship between the displayed object and the environment. Enable **Lock Model Location in Environment** to "fix" the location of the displayed object so that as the model is rotated, its position does not change within the environment. Locking the model in the environment may result in some distortion, which can be minimized by adjusting the height of the environment ground using the **Ground Height (distortion mitigation)** slider.

   - Adjust the lighting **Intensity** and **Color** using the slider control and color picker.

8. (Optional) Enable **Show Backplate** to use a color or static image as the background to the raytracing display.

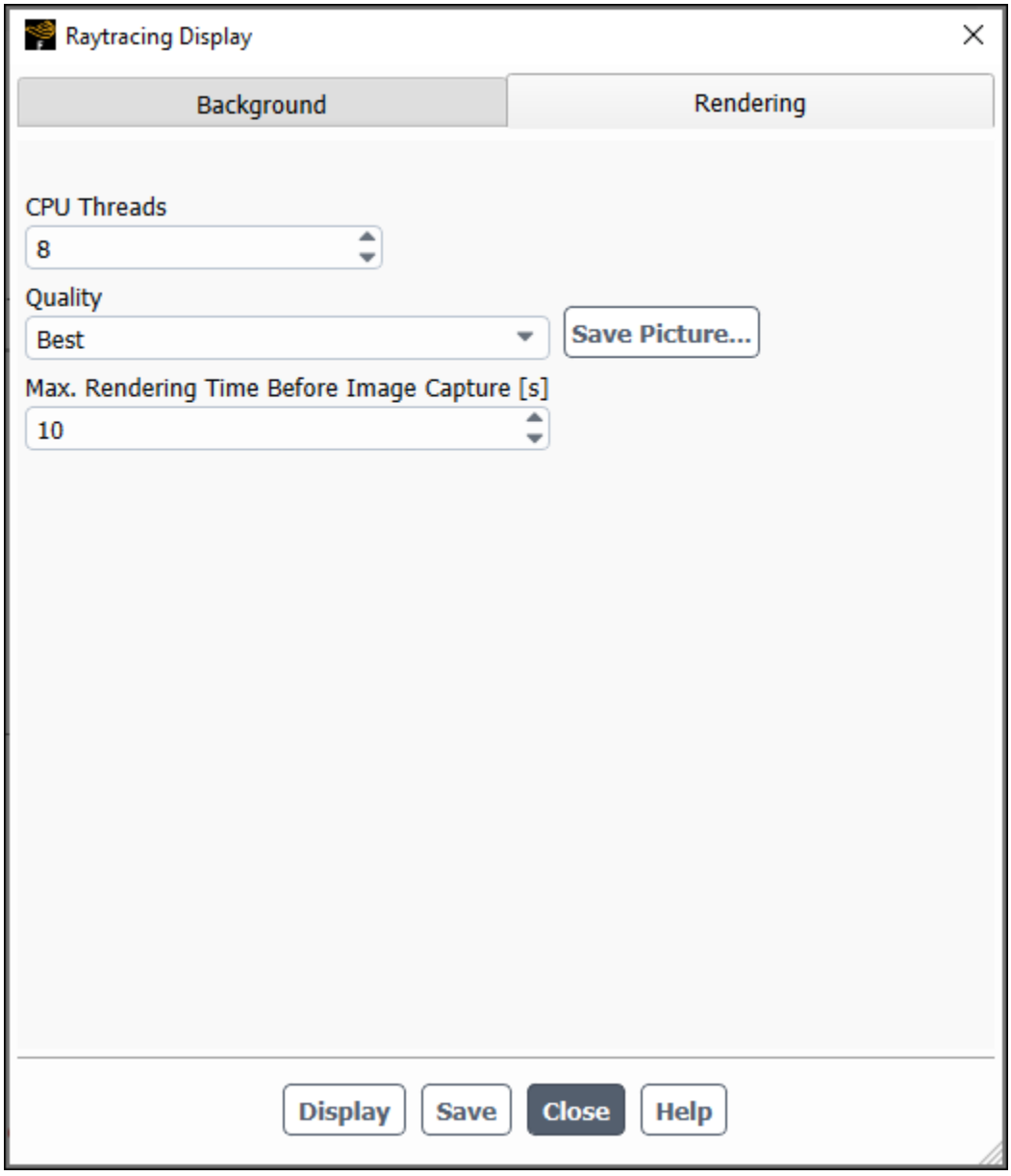
   a. Select either a color or an **Image** for the backplate.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

9. Click **Display** to render the raytracer image. *Clicking **Save** retains the settings shown in the **Raytracing Display** dialog box without the need to display the image. This allows you to control the settings for saving raytracer images without displaying first.*

(Optional) Specify the rendering options:

1. Click **Rendering** to open the **Rendering** tab, which contains options for controlling the quality of the raytracer image.



2. (Advanced) Set the number of **CPU Threads** that you want used for the raytracing computations. Note that if specify a number of threads that is greater than the number of cores available on your machine, the raytracer will use the maximum number of available cores.

3. Specify the **Quality** of image you want displayed and/or saved when you save the display using the **Save Picture** dialog box. *Higher quality displays take longer to render.*

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Specify how long you want to spend rendering a raytracing image before Fluent captures it and saves it to a file.

5. Click **Display** or **Save**.

Important:

The **Raytracing** window is not interactive, so the orientation, zoom, and location of the model (and 3D environment, if applicable) is controlled in the user-window that was active when you clicked **Display** in the **Raytracing Display** dialog box.

40.4.1. Animating with Realistic Raytracer Images

There are two approaches for creating realistic solution animations, as discussed in the following sections:

40.4.1.1. Realistic Raytracer Images Captured at Runtime

40.4.1.2. Raytracer Image Conversion Using the Playback Dialog Box

In either case, the animation definition must be created prior to calculating the solution for any results you want included in the animation.

40.4.1.1. Realistic Raytracer Images Captured at Runtime

While you are setting up your animation definition (as described in Animating the Solution (p. 3714)) you have the option save raytracer images as the solution is iterating.

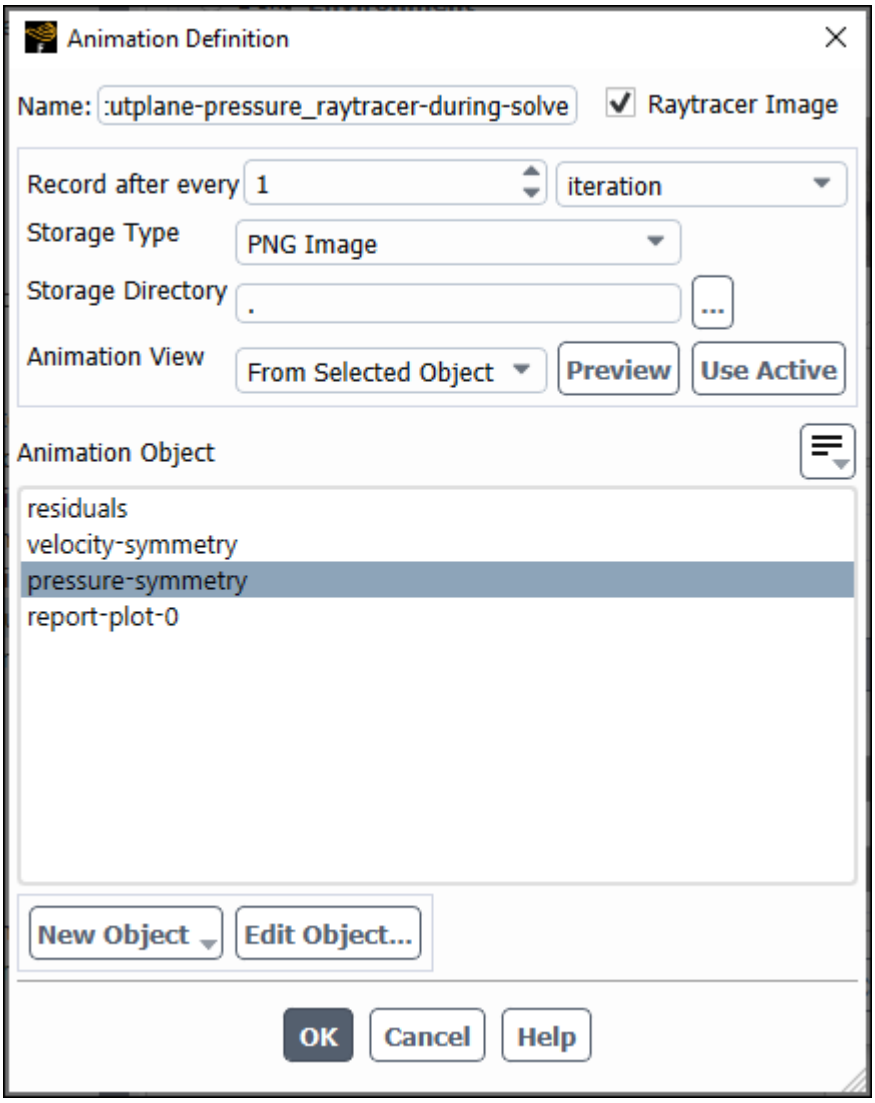
To directly save raytracer images:

1. Create and display the desired graphics object (*contours, vectors, pathlines, particle tracks, or scenes*).

2. Create an animation definition for this graphics object that includes enabling the **Raytracer Image** option. *Note that raytracer images can only be saved in 2D image formats, such as PNG or JPEG.* Refer to Animating the Solution (p. 3714) for additional information on creating animation definitions.

   Solution → Calculation Activities

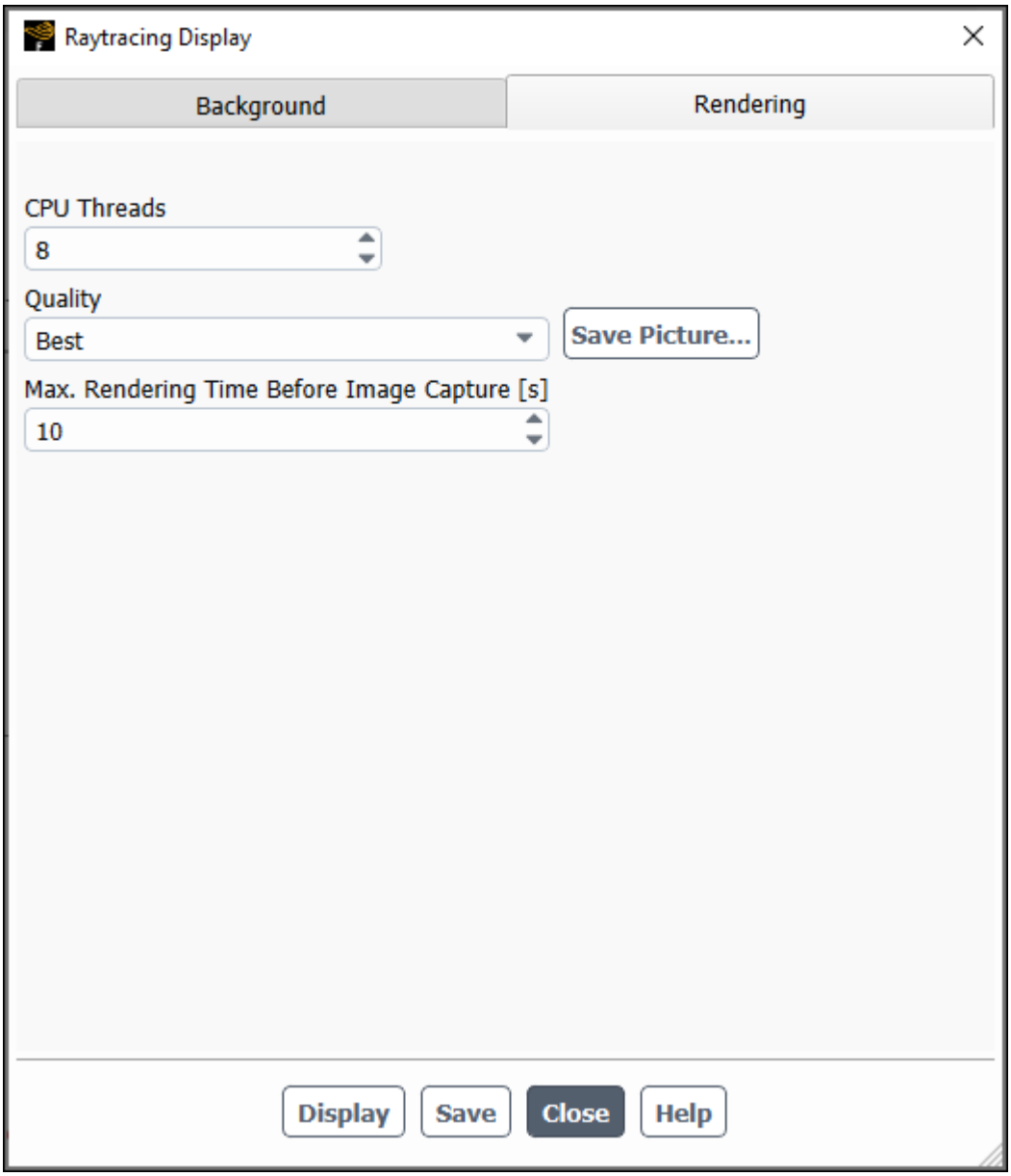
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



3. Open the **Raytracing Display** dialog box and select the desired environment and/or backplate and update the display to the desired appearance.

   View → **Graphics** → **Raytracing Display...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



4. Click **Save Picture...** in the **Rendering** tab to open the **Save Picture** dialog box.

5. Ensure settings like **Resolution** match your expectations, because this will control the size of the images saved for creating the animation. Click **Apply** to retain the desired settings.

40.4.1.2. Raytracer Image Conversion Using the Playback Dialog Box

Instead of saving raytracer images directly while the solver is iterating (as described in Realistic Raytracer Images Captured at Runtime (p. 4000)), you can save HSF animation files during the solver run, and convert them to raytracer images afterwards, using the **Playback** dialog box. The appeal of this option is that it is more expensive to save raytracer images, so if they are being saved during the solver run, it can take longer (clock time) for the solver to converge. Converting your animation to use better, raytracer images after the solver run gives you the flexibility to use alternative computing resources to attain your desired video quality and appearance.

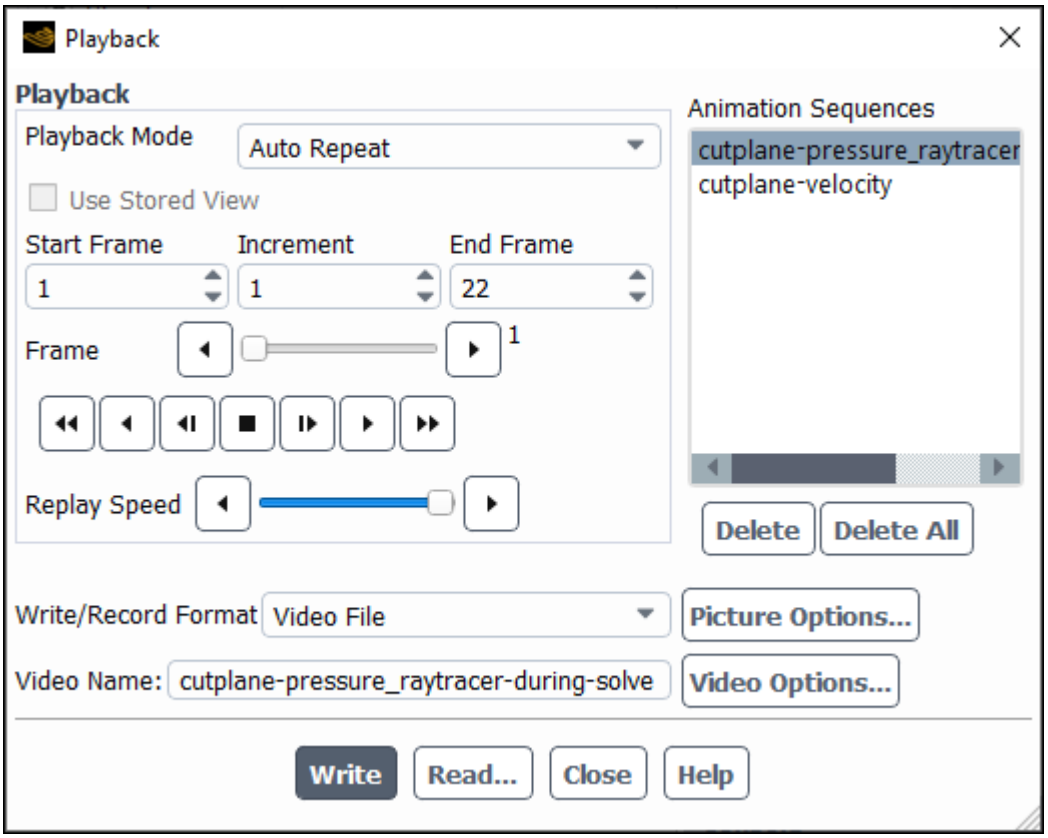
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To create raytracing animations during postprocessing:

1. Ensure your animation images are saved in HSF format by confirming that **HSF File** is selected for **Storage Type** in the **Animation Definition** dialog box.

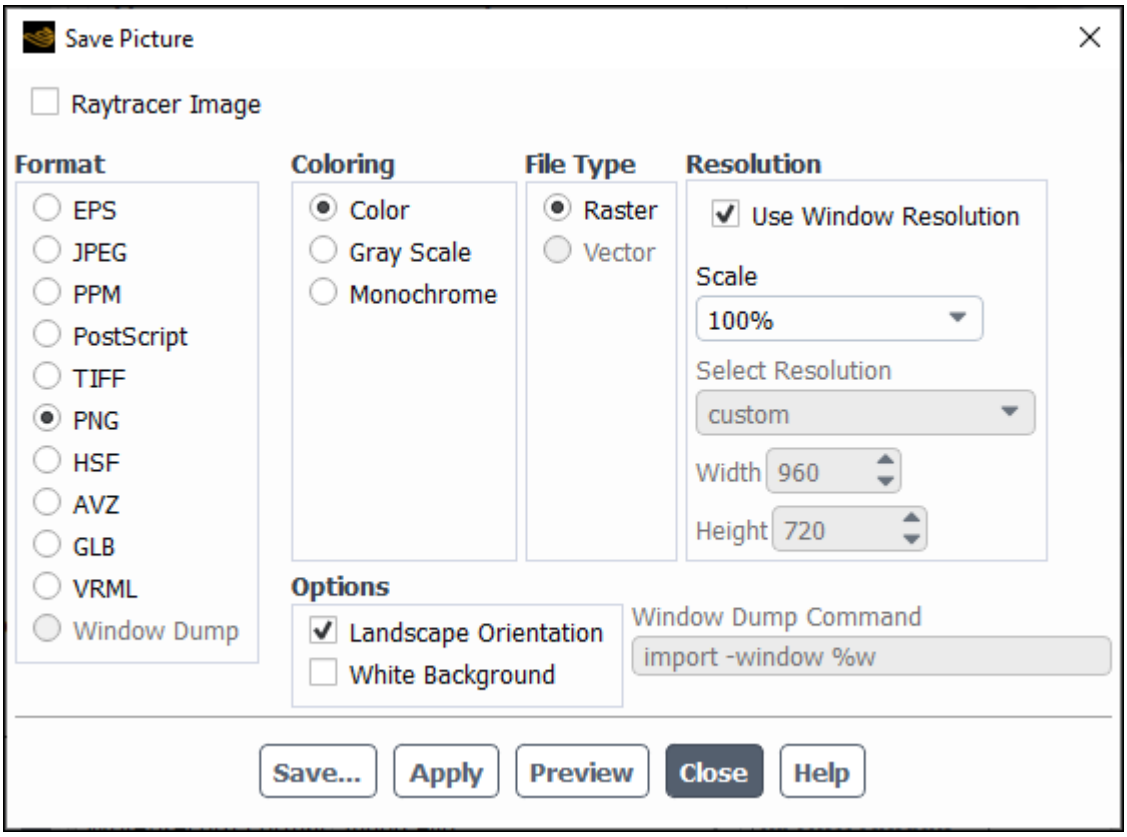
2. Open the **Playback** dialog box and read or select the desired **Animation Sequence**.

   Results → Animations → Playback Edit



3. Click **Picture Options...** to open the **Save Picture** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



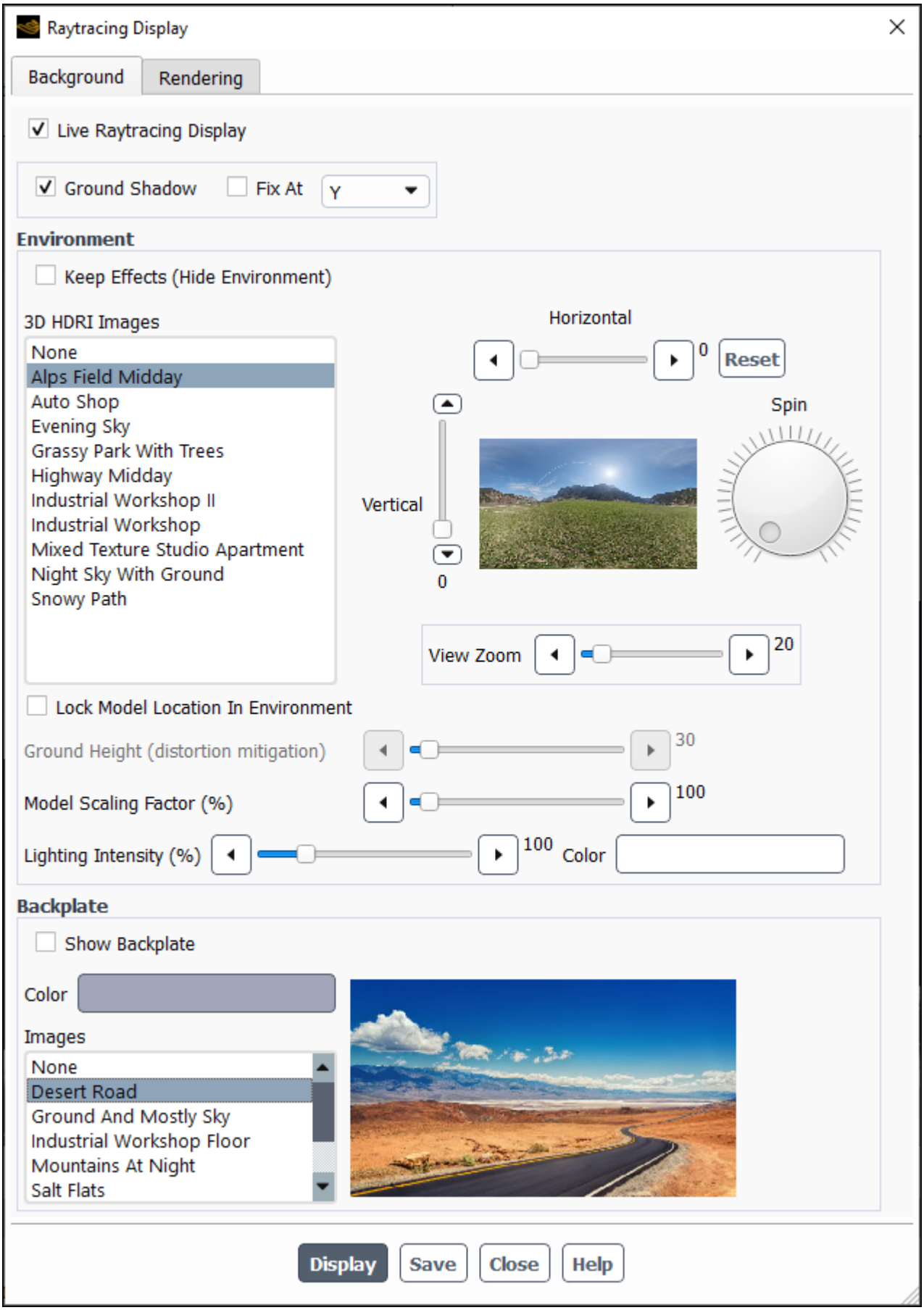
a. Enable **Raytracer Image**.

b. Set the desired **Format** and **Resolution** and click **Apply**, then **Close** the **Save Picture** dialog box.

4. Open the **Raytracing Display** dialog box.

   View → **Graphics** → **Raytracing Display...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. Specify the desired settings, including environment, backplate, and rendering options.

b. Click **Display**, then **Close**. *Alternatively, if you know your desired settings and do not need to make further changes after seeing the raytracing display, you can just click **Save** to avoid rendering the display, saving time.*

5. Provide a **Video Name** and click **Write** to save the raytracer animation. *Clicking **Video Options...** allows you to review additional video settings, such as frames per second (fps) and the video file format.*

Important:

When converting an HSF animation to a raytracer animation, you can change:

- Lighting, including adding and removing lights (Adding Lights (p. 3984))
- Camera orientation (Controlling the Display State and Modifying the View (p. 4025))
- Any settings available in the **Raytracing Display** dialog box

You cannot change the material (Realistic Rendering of Materials (p. 3882)) used in rendering your model.

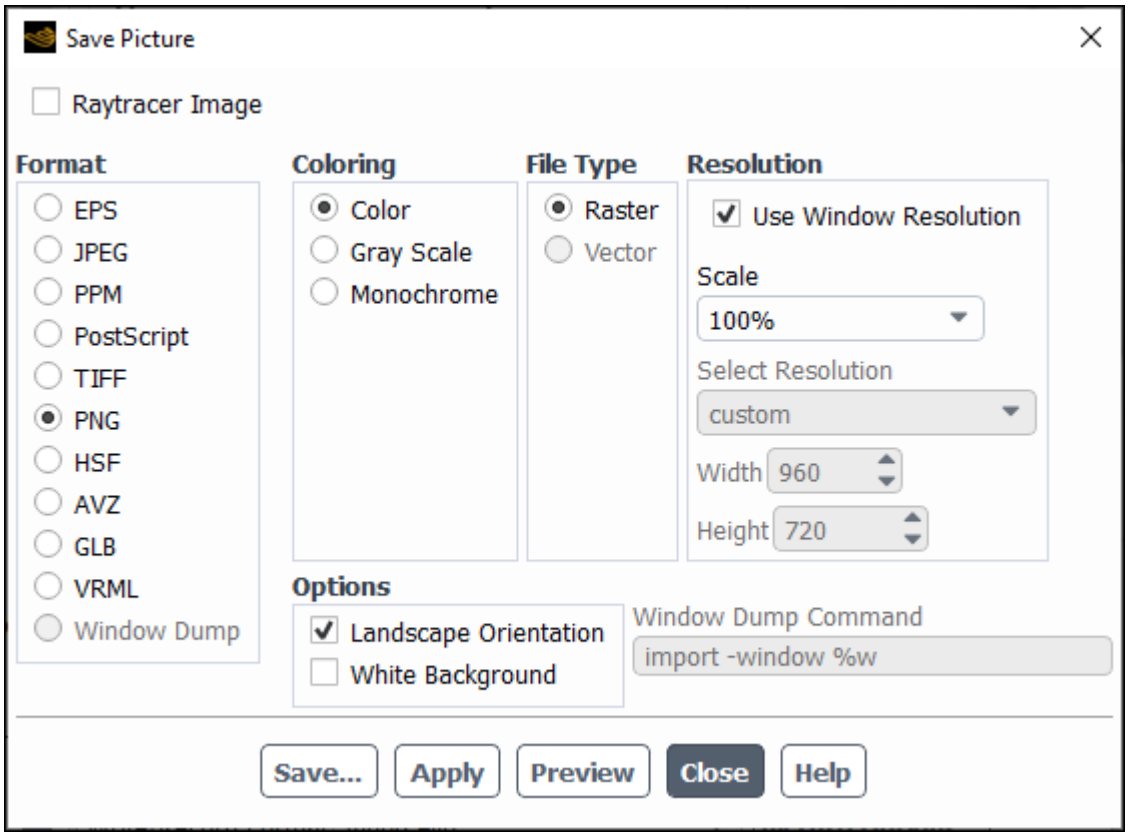
40.4.2. Saving Pictures with Realistic Raytracer Images

There are two ways that you can save raytracer images using the **Save Picture** dialog box. The type of image you want determines the approach you use.

Open the **Save Picture** dialog box.

File → Save Picture...

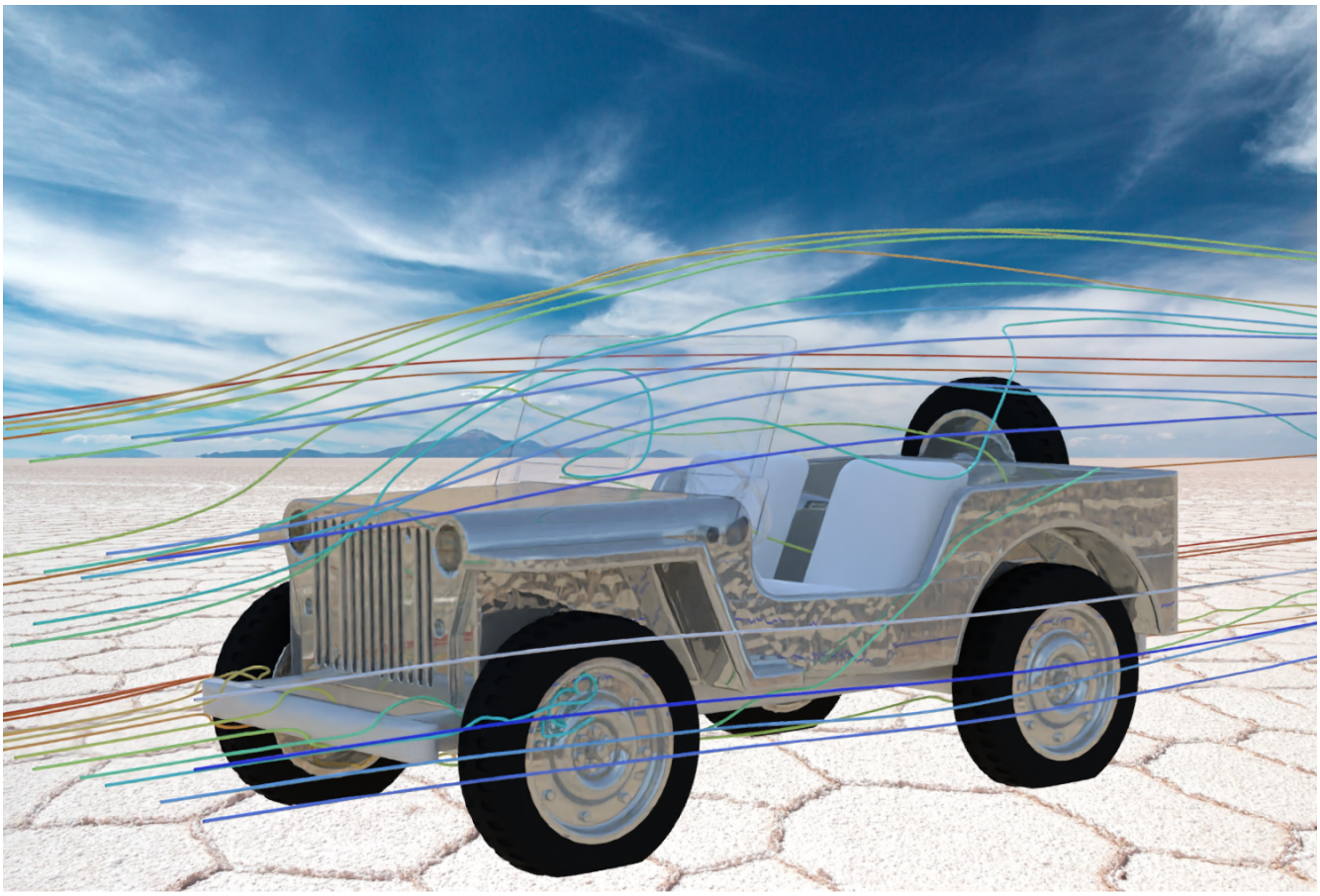
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



- Enable the **Raytracer Image** option if you want to save an image likeFigure 40.78: Raytracer Scene Display with Backplate (p. 4008).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

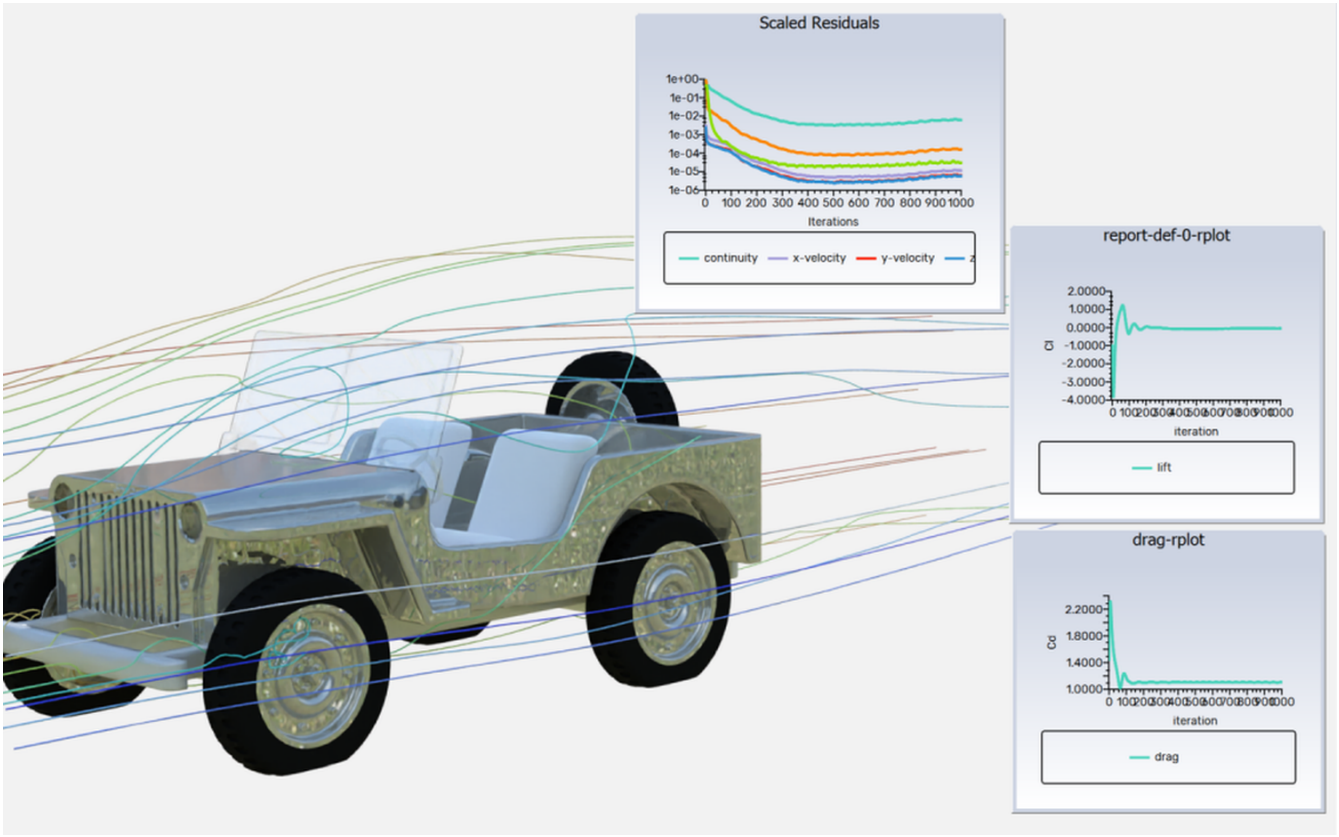
Figure 40.78: Raytracer Scene Display with Backplate



- Disable the **Raytracer Image** option if you want to save an image like Figure 40.79: Raytracer Scene Display with Embedded Plots (p. 4009), which is setup as an embedded window dashboard (Embedded Graphics Window Dashboards (p. 3941)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.79: Raytracer Scene Display with Embedded Plots



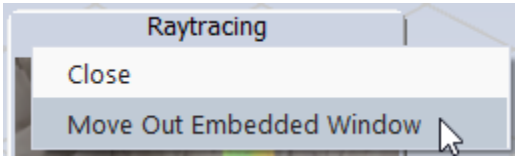
To create and capture this type of image:

1. Display the desired object that you want raytraced. *This can be a mesh, contours, vectors, pathlines, particle tracks, LIC, or scene plot.*

2. Open the **Raytracing Display** dialog box, specify the desired settings, and click **Display**.

3. Right-click the title bar of the raytracing window and select **Move Out Embedded Window**.

   Figure 40.80: Move Out Embedded Window

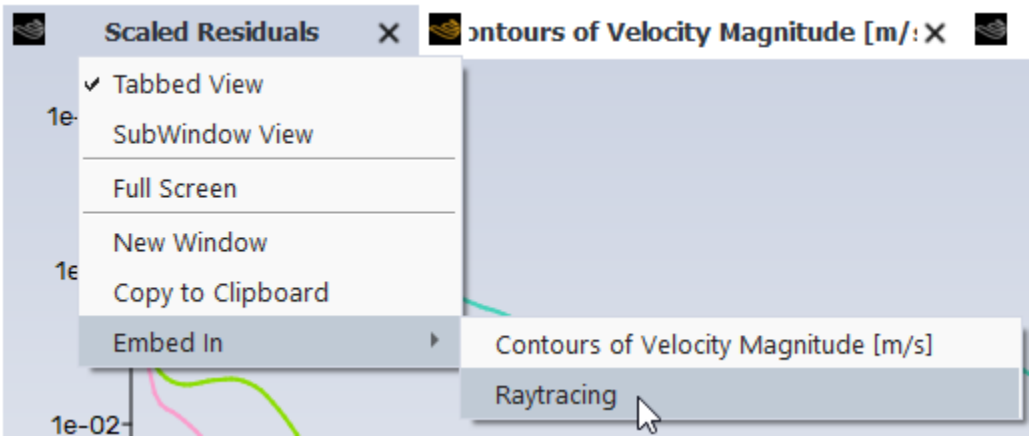
   

4. Display the 2D plots (or any other graphics display such as contours or vectors) you want to embed in the raytracing window. *Ensure each plot or object is opened in a new window.*

5. Right-click each 2D plot title bar and pick **Embed in** > **Raytracing**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.81: Embed in Raytracing Window



6. Click on the raytracing window to make it active, then open the **Save Picture** dialog box.
7. Ensure the **Raytracer Image** option is disabled. Specify the other desired settings and click **Save...**.

Important:

The **Raytracing** window will be saved at the same dimensions as the current window resolution. *Disabling **Use Window Resolution** and specifying an alternative resolution will only change the scaling of the other embedded windows (the plot windows).*

40.4.3. Listing of Realistic Environments and Backplates

The following is a listing of the available realistic environments.

- **Alps Field Midday**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Auto Shop**

- **Evening Sky**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Grassy park with Trees**

- **Highway Midday**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Industrial Workshop II**

- **Industrial Workshop**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Mixed Texture Studio Apartment**

- **Night Sky with Ground**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Snowy Path**

The following is a list of the available backplate images:

- **Desert Road**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Ground and Mostly Sky**

- **Industrial Workshop Floor**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Mountains at Night**

- **Salt Flats**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Space**

- **Warehouse with Skylights**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Water and Mostly Sky**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

40.4.4. Limitations with Realistic Raytracing

The limitations associated with realistic raytracing are listed below:

- Embedded window dashboard objects containing a realistic raytracing display are not saved with the case file.
- Realistic raytracing displays cannot be displayed in orthographic view, due to the nature of raytracing computations.
- Realistic raytracing displays are incompatible with the **Monochrome** option in the **Save Picture** dialog box. If a raytracing display window is embedded in a regular graphics display (such as a mesh object), the Fluent window will be monochrome, but the raytracing window will be in color.
- Realistic raytracing is only supported for graphics windows containing 3D surfaces. Windows only containing 2D displays, such as plots or animations (in 2D formats like PNG or JPG), cannot be raytraced.
- Scene animations, also referred to as keyframe animations, are not compatible with raytracing displays.

40.5. Controlling the Mouse Button Functions

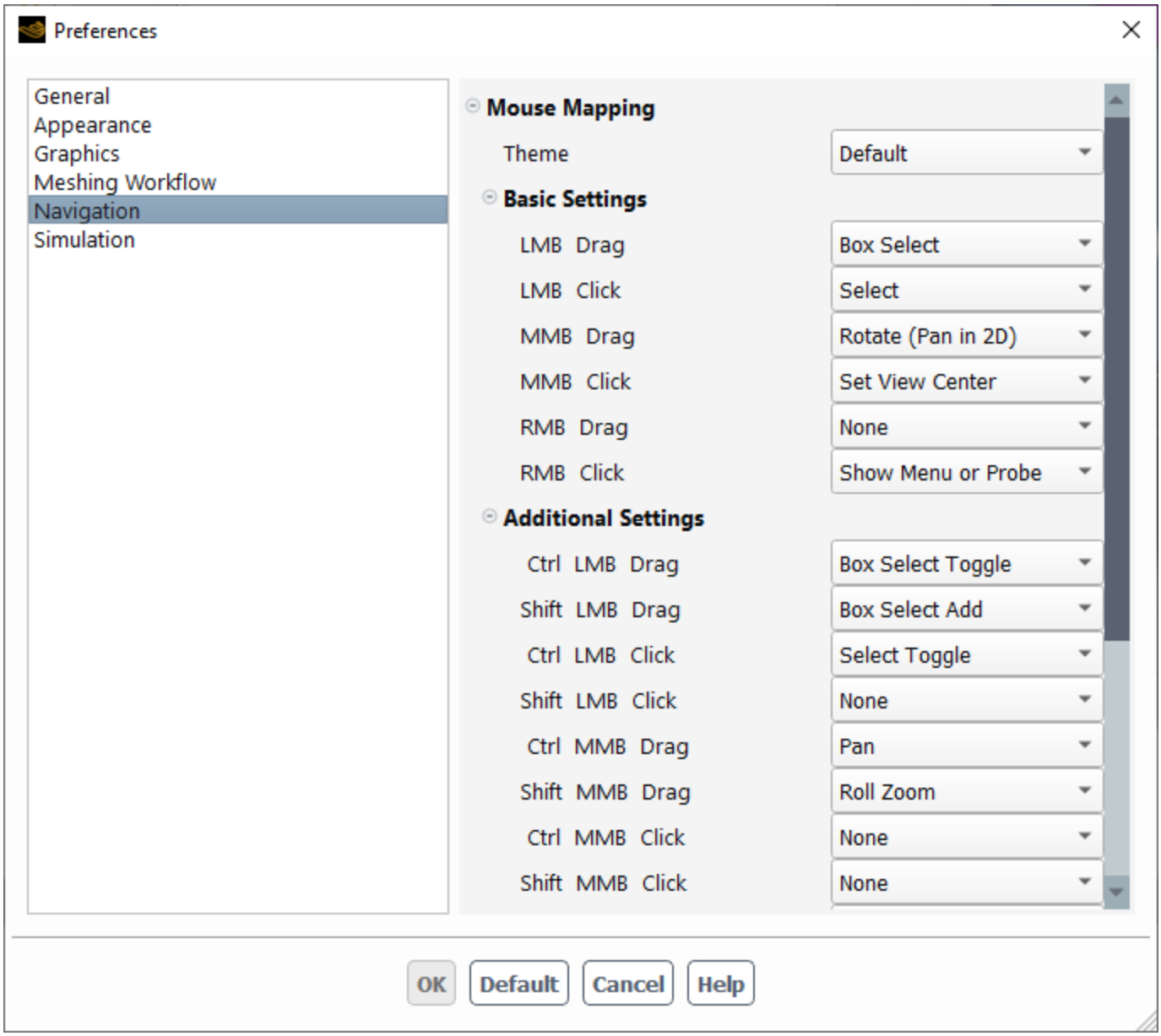
A convenient feature of Ansys Fluent is that it lets you assign a specific function to each of the mouse buttons. According to your specifications, clicking a mouse button in the graphics window will cause the appropriate action to be taken. These functions apply only to the graphics windows; they behave differently when an XY plot or histogram is displayed. For information about the use of mouse buttons in these plots, see Plot Types (p. 4057). Clicking any mouse button in a graphics window will make that window the active window.

You can control mouse button mappings using the **Navigation** branch in Preferences, accessed by clicking **Preferences...** under the **File** ribbon tab.

File → **Preferences...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.82: The Navigation Branch of Preferences



The **Theme** category allows you to choose from multiple predefined mouse assignments. It will change to the **Custom** theme once you make changes to the mouse actions.

The available button functions available are listed below:

Box Select

Dragging from left to right selects the surfaces and volumes that are completely enclosed within the box. Dragging from right to left selects any surface or volume that is in contact with the box.

Box Select Add

Dragging from left to right selects the surfaces and volumes that are completely enclosed within the box. Dragging from right to left selects any surface or volume that is in contact with the box. Each selection is added to the previous, not replaced.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Box Select Toggle

Dragging from left to right selects the surfaces and volumes that are completely enclosed within the box. Dragging from right to left selects any surface or volume that is in contact with the box. If you box select over a surface that is already selected, it will be de-selected.

Pan

Enables you to translate the view by dragging the mouse while holding down the button. The function completes when the mouse button is released or the cursor leaves the graphics window.

Probe

Enables you to select items from the graphics windows and request information about displayed scenes. If you have selected **short description** from the **Probe** drop-down list and then click the mouse-probe button in the graphics window, only the identity of the item you clicked will be printed out in the console; if you have selected **long description** instead, more detailed information about a selected item will be displayed.

Note:

If you want probe values to appear in the graphics window when you hover over graphics displays like contour plots, then you must click the probe button in the pointer toolbar ( ). *The **Hover-Over Probe Values** option must be enabled in **Preferences** (located in the **Appearance** branch under **Selections**) to see values appear in the graphics window when you hover over the displayed graphics.*

Important:

The values printed into the console when you probe may not match those that appear in the graphics window when you are hovering over graphics displays while probing using the toolbar button ( ). The hover-over probe values displayed are interpolated from the results, while the values printed to the console when you click to probe are based on the colormap. *Discrepancies between hover probe values and click-probe values printed in the console are likely to be apparent when you specify the range rather than having Fluent compute it automatically.*

Roll-Zoom

Enables you to rotate or zoom the view, depending on which direction you drag the mouse. If you drag the mouse horizontally, the display will rotate about the axis normal to the screen. If you drag it vertically, the display will be magnified (if you drag it down) or shrunk (if you drag it up). The function completes when the mouse button is released or the cursor leaves the graphics window.

Rotate (Pan in 2D)

Enables you to rotate the view by dragging the mouse across the screen. Dragging horizontally rotates the object about the screen's y-axis; vertical mouse movement rotates the object about the screen's

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

x-axis. The function completes when the mouse button is released or the cursor leaves the graphics window.

Set View Center

Moves the selected point to the center of the graphics window.

Zoom

Enables you to draw a zoom box, anchored at the point at which the button was pressed, by dragging the mouse with the button held down. When you release the button, if the dragging was from left to right, a magnified view of the area within the zoom box will fill the window. If the dragging was from right to left, the area of the window is shrunk to fit into the zoom box, resulting in a "zoomed out" view.

Note:

When **Zoom** is selected for the **Mouse wheel**, it behaves as a roll zoom, rather than a box zoom.

Zooming using the mouse wheel does not work while the solver is iterating. Assign **Roll-Zoom** (for similar behavior to zoom with the mouse wheel) or **Zoom** to another key combination to allow for zooming during solver computation.

Note:

Use **Esc** to deselect all surfaces in the graphics window.

Important:

3Dconnexion Space products (Mouse, Pilot, and Navigator) are not supported with Ansys Fluent.

40.6. Viewing the Application Window

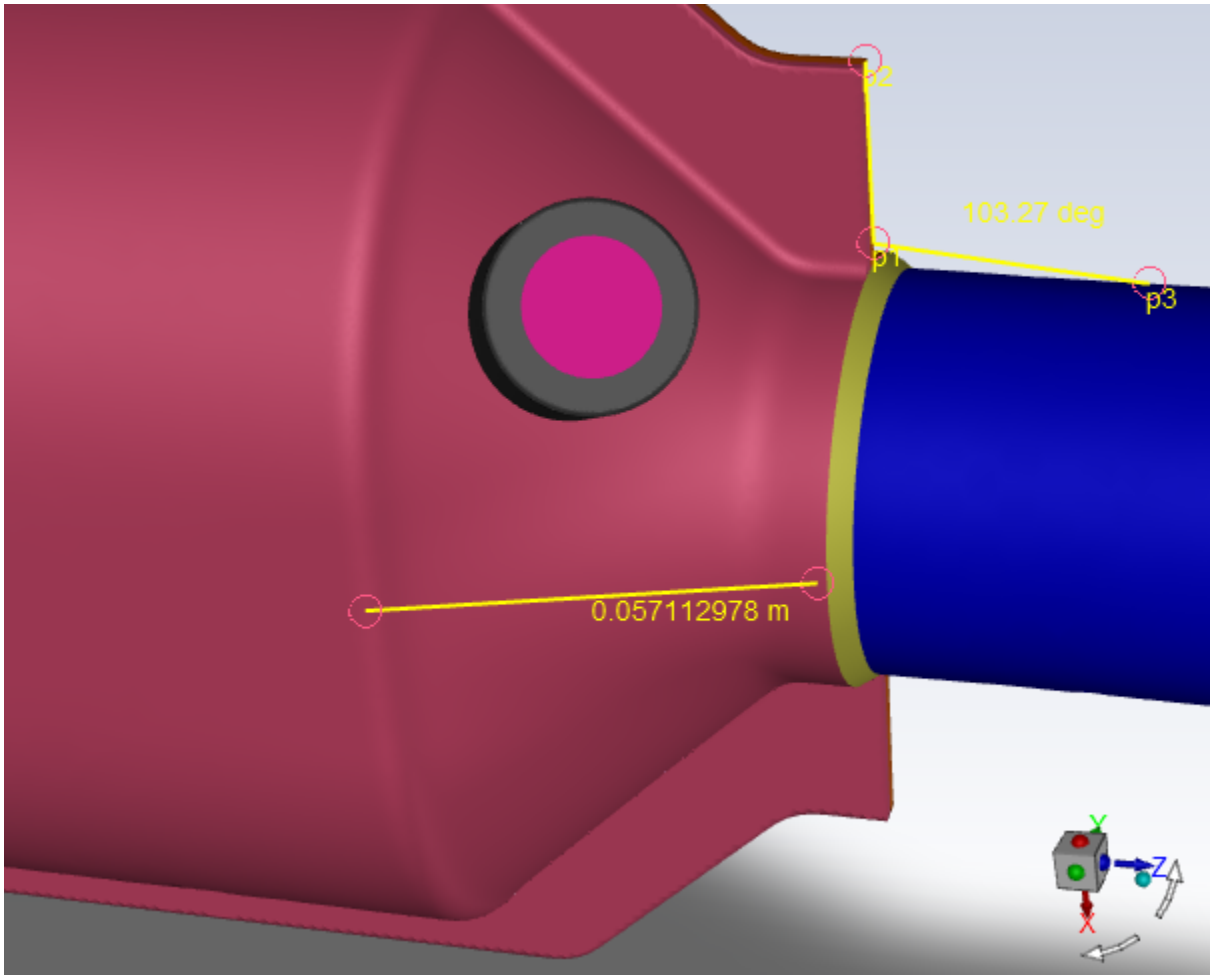
In Ansys Fluent, the application window houses the ribbon and console, as well as multiple graphics windows, task pages, and the tree. By default, all components are displayed and one graphics window is visible. You can toggle the visibility of the toolbars, task page, console, and graphics window using in the standard toolbar. The ribbon and the tree are never fully hidden—you can hide the ribbon contents using (also in the standard toolbar), but the tab names are always visible. You also have the option of viewing separate graphics windows as described in Managing Multiple Graphics Windows (p. 3955). For additional information about the various elements of the application window, see Graphical User Interface (GUI) (p. 901).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

40.7. Measuring Distance and Angle

You can interactively measure distances and angles between two or three points respectively on objects displayed in the graphics window, as demonstrated in Figure 40.83: Example Distance and Angle Measurement (p. 4024).

Figure 40.83: Example Distance and Angle Measurement



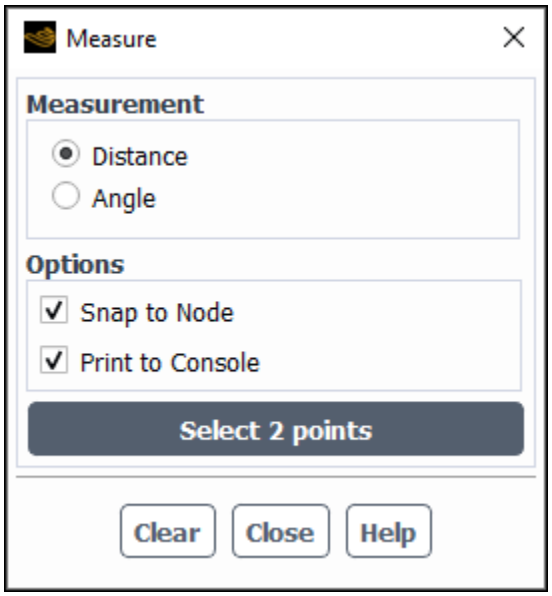
To measure a distance or angle:

1. Open the **Measure** dialog box by clicking **Measure...** in the **View** ribbon tab (**Display** group box).

   View → **Display** → **Measure...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.84: The Measure Dialog Box



2. Specify whether you are measuring a **Distance** or an **Angle** by selecting the appropriate option.
3. Specify the available **Options**:
   - Enable **Snap to Node** if you want your click location on the displayed object to be moved to the nearest node.
   - Enable **Print to Console** if you want the measurement details to be printed to the Console in addition to appearing in the graphics window.
4. Click **Select 2 Points** or **Select 3 Points** to initiate the measuring tool.
5. Select 2 points in the graphics window to measure distance or select 3 points if you are measuring an angle. You can use the left or right mouse buttons to make the selections. *For angle measurements, the first point you select is the vertex of the angle.*
6. Click **Close** in the **Measure** dialog box when you are done measuring.

Note:

You can clear measurements from the screen by clicking **Clear** in the **Measure** dialog box, pressing the **Esc** key, or clicking the **Clear selection** button ( ) in the toolbar.

40.8. Controlling the Display State and Modifying the View

Ansys Fluent allows you to control the appearance of the scene that is displayed in the graphics window. You can control the display state which consists of visual properties including the view, graphics effects and more. The view can be manipulated by scaling, centering, rotating, translating, or zooming the display. Display states can be saved, restored, copied and deleted (and the same applies to views). These operations are performed in the Display States Dialog Box (p. 5437) (Figure 40.85: The Display States Dialog

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Box (p. 4027)). For views, these operations are performed in the Views Dialog Box (p. 5272) (Figure 40.87: The Views Dialog Box (p. 4029)) or with the mouse, or simply using the shortcut in the graphics toolbar, as described in Selecting a View (p. 4027).

Important:

You can revert to the previous view by pressing **Ctrl**+**L** while the graphics window has the main focus. You can also use the text command `view last` from the top level of the text command tree. You can use the command to revert to any of the past 20 views.

For additional information, see the following sections:

40.8.1. Specifying a Display State

40.8.2. Selecting a View

40.8.3. Manipulating the Display

40.8.4. Controlling Perspective and Camera Parameters

40.8.5. Saving and Restoring Views

40.8.6. Mirroring and Periodic Repeats

40.8.1. Specifying a Display State

When you display a graphics object, you have the option of associating a display state with that object. The display state allows you to recreate the same display, both between multiple objects and across different sessions. You are not required to associate a state with an object, nor do you have to preserve every aspect of a particular display. The following fields can be included in a display state:

- View
- Front Faces Transparent
- Projection
- Axes
- Ruler
- Boundary Markers
- Titles
- Light ID

Note:

When you click **Use Active**, either in the **Display States** dialog box or from within a graphics object dialog box, the behavior depends on how many light IDs are active:

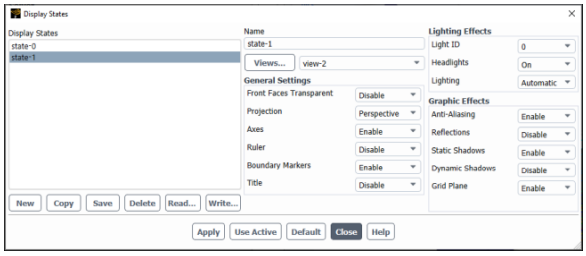
– If only one light ID is active then that light ID is populated in the **Light ID** field.

– If more than one light ID is active then the **Light ID** field is set to **Don't Save**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Headlight
- Lighting
- Anti-Aliasing
- Reflections
- Static Shadows
- Dynamic Shadows
- Grid Plane

Figure 40.85: The Display States Dialog Box



In many cases, you should not have to use the **Display States** dialog box, because you can setup the window and object(s) as desired, then right-click in the graphics window and select **Save State**. If you still have the dialog box for the graphics object open, for example the **Contours** dialog box, you can click **Use Active** to create a display state that matches that of the active graphics window and click **Save/Display** to associate that display state with the object. Alternatively, you can select a pre-existing display state from the drop-down list, which can allow you to create multiple objects that will display with a consistent orientation and appearance.

Important:

Display state settings (except the view) are applied to all user (non-2D plot) windows.

For example, if you display a contour plot with a display state that includes the ruler, then display a vector plot in a new window with a display state where the ruler is disabled, the ruler will be disabled in both plot windows.

Note:

Graphics object that are displayed with **None** selected as the **Display State** will be shown with the same display state settings as the last displayed object.

40.8.2. Selecting a View

You can use the Views Dialog Box (p. 5272) (Figure 40.87: The Views Dialog Box (p. 4029)) to select the orientation of your display or use the drop-down available in the graphics toolbar for 3D simulations. Furthermore, you can simply click the interactive triad (Figure 40.86: Using the Triad to Change

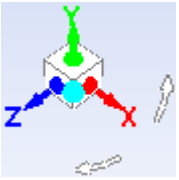
the Orientation of the Object (p. 4028)), displayed in the graphics window. The Table 40.1: Standard Views (p. 4028) relates the **Views** listed in the Views Dialog Box (p. 5272) to the views available in the graphics toolbar (see Pointer Tools (p. 915)).

Results → Graphics Views...

Table 40.1: Standard Views

View Listed in Dialog Box	View Shortcut in Graphics Toolbar
back	
bottom	
front	
isometric	
left	
right	
top	

Figure 40.86: Using the Triad to Change the Orientation of the Object



To change the orientation of the object in the graphics window using the triad (Figure 40.86: Using the Triad to Change the Orientation of the Object (p. 4028)), you can

- Click an axis/semi-sphere to point it in the positive/negative direction.
- Right-click an axis/semi-sphere to point it in the negative/positive direction.
- Click the cyan iso-ball to set the isometric view.
- Click the white rotational arrows at the bottom of the triad to perform in-plane clockwise or counterclockwise 90 degree rotations.
- Left-click and hold—in the vicinity of the triad—and use the mouse to perform free-rotations in any direction. Release left mouse button to stop rotating.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that clicking on the X-axis will result in the +X-axis pointing towards you.

Note:

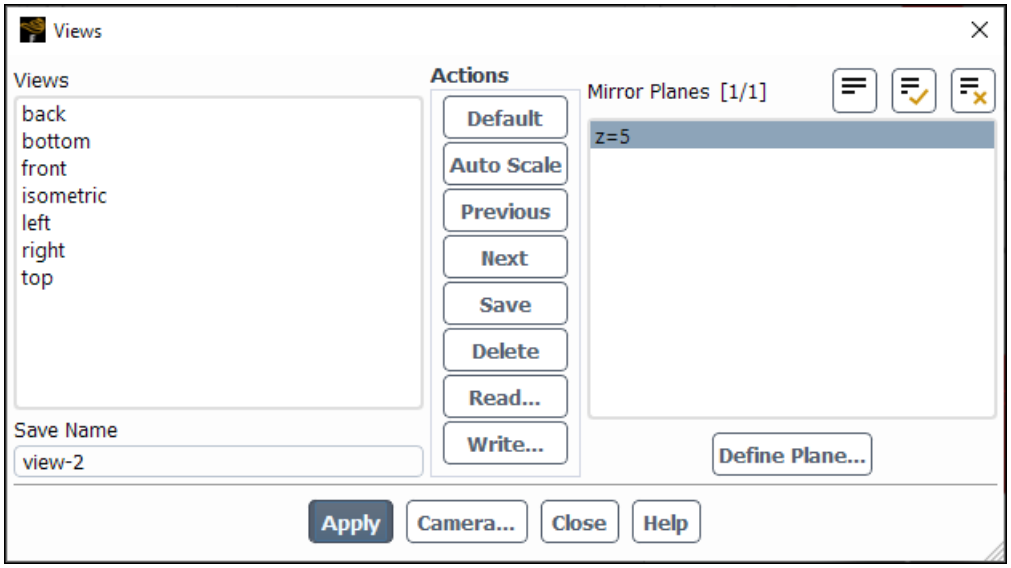
For 2D cases, the drop-down and the interactive triad are not available; in a front view, the X-axis is horizontal with the positive direction to the right, the Y-axis is vertical with the positive direction pointing up, and the Z-axis is determined by the right-hand rule.

40.8.3. Manipulating the Display

Most of the manipulating activities (like scaling and centering the display) are accomplished using the Views Dialog Box (p. 5272) (Figure 40.87: The Views Dialog Box (p. 4029)).

Results → Graphics → Views...

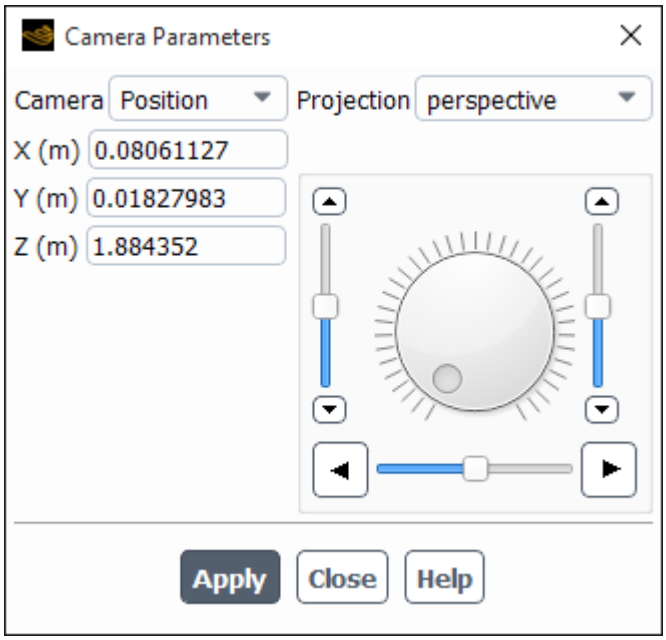
Figure 40.87: The Views Dialog Box



You can rotate, translate, and zoom the graphics display using either the mouse or the Camera Parameters Dialog Box (p. 5277) (Figure 40.88: The Camera Parameters Dialog Box (p. 4030)), which is opened from the Views Dialog Box (p. 5272).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.88: The Camera Parameters Dialog Box



40.8.3.1. Scaling and Centering

You can scale and center the current display without changing its orientation by clicking **Auto Scale** in the Views Dialog Box (p. 5272).

40.8.3.2. Rotating the Display

Use either the mouse or the Camera Parameters Dialog Box (p. 5277) to:

- Rotate the display in any direction (in 3D)
- Rotate the display about the axis normal to the screen (in 2D)

To rotate a display with the mouse, use the button with the **Rotate (Pan in 2D)** function (the left button, by default). For information about changing the mouse functions, see Controlling the Mouse Button Functions (p. 4020).

- Click and drag the left mouse button in the graphics window to rotate the geometry in the display.
- You can also click and drag the left mouse button on the (x, y, z) graphics triad in the lower left corner to rotate the display.
- If you press the **Shift** key when you first click the mouse button to begin the rotation, the rotation will be constrained to a single direction (for example, you can rotate about the screen's horizontal axis without changing the position relative to the vertical axis).
- If you want to constrain the rotation of a display to be about the axis normal to the screen, you can also use the **Roll-Zoom** function.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Click the appropriate mouse button and drag the mouse to the left for clockwise rotation, or to the right for counterclockwise rotation.

To rotate a 3D display using the Camera Parameters Dialog Box (p. 5277) (Figure 40.88: The Camera Parameters Dialog Box (p. 4030)) use the dial and the slider of the scales.

- To rotate about the horizontal axis at the center of the screen, move the slider on the scale to the left of the dial up or down (see Scales (p. 929) or instructions on using the scale).
- To rotate about the vertical axis at the center of the screen, move the slider on the scale below the dial to the left or right.
- To rotate about the axis at the center of and perpendicular to the screen, click the left mouse button on the indicator in the dial and drag it around the dial.

Important:

The position of the slider or the dial indicator does not reflect the cumulative rotation about the axis as the slider/indicator will return to its original position when you release the mouse button.

40.8.3.2.1. Spinning the Display with the Mouse

When you use the mouse for rotation, you have the option to "push" the display into a continuous spin. This feature can be used in conjunction with video recording, or simply for interactive viewing of the domain from different angles. To use this option, use the `auto-spin?` text command:

`display` → `set` → `rendering-options` → `auto-spin?`

Then display the graphics (or, if the graphics are already displayed, you can click **Apply** in the Display Options Dialog Box (p. 5262)). The **Rotate (Pan in 2D)** button will then have two uses:

- To perform the standard rotation, stop dragging the mouse before you release the **Rotate (Pan in 2D)** button.
- To start the continuous spin, release the **Rotate (Pan in 2D)** button while you are still dragging the mouse. The display will continue to rotate on its own until you click any mouse button in the graphics window again. The speed of the rotation will depend on how fast you are dragging the mouse when you release the button.

For smoother rotation, enable the **Double Buffering** option in the Display Options Dialog Box (p. 5262) (see Modifying the Rendering Options (p. 3987)). This will reduce screen flicker during graphics updates.

40.8.3.3. Translating the Display

By default the left mouse button is set to **Pan** in 2D. For information about changing the mouse functions, see Controlling the Mouse Button Functions (p. 4020). Click and drag the left mouse button in the graphics window to translate the geometry in the display.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

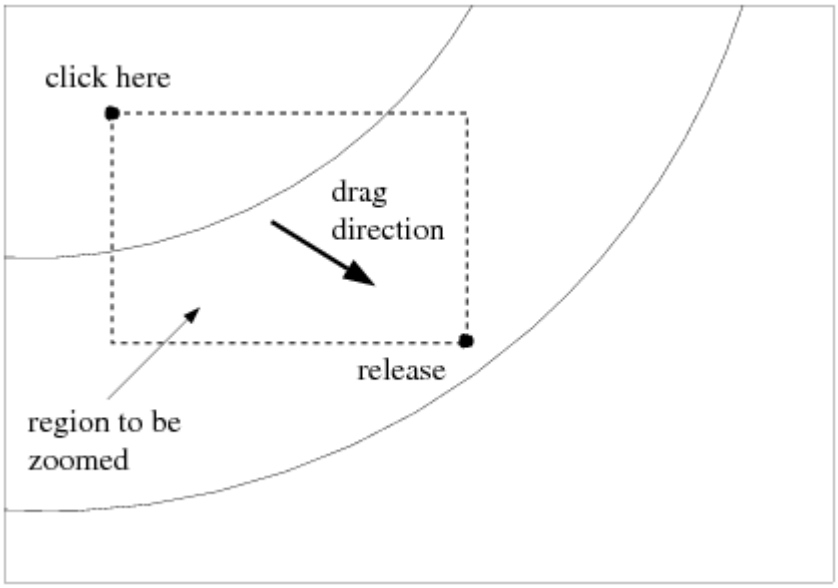
In 3D, you can either change one of the button functions to **Pan** and follow the instructions above for 2D, or use the **Zoom** button (the middle button by default). Click the middle button once on the point in the display that you want moved to the center of the screen. Ansys Fluent will redisplay the graphic with that point in the center of the window. This method can also be used in 2D.

40.8.3.4. Zooming the Display

In both 2D and 3D you will use the mouse button with the **Zoom** function (the middle button by default) or the **Roll-Zoom** function (see Controlling the Mouse Button Functions (p. 4020) for information about enabling this optional function), or the Camera Parameters Dialog Box (p. 5277) to magnify and shrink the display.

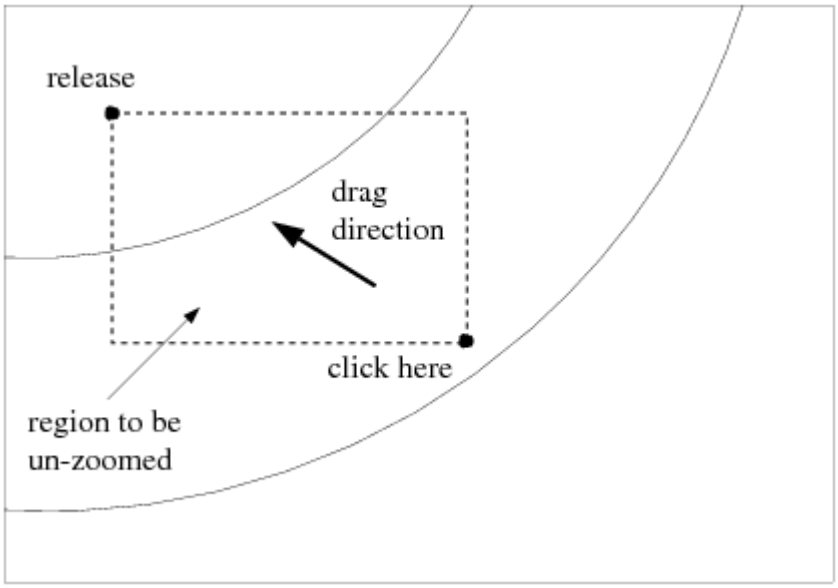
With the **Zoom** function, click the middle mouse button and drag it from left to right (creating a "zoom box") to magnify the display. Figure 40.89: Zooming In (Magnifying the Display) (p. 4032) displays the correct dragging of the mouse, from upper left to lower right on the display, in order to zoom. You can also drag from lower left to upper right. After you release the mouse button, Ansys Fluent will redisplay the graphic, filling the graphics window with the portion of the display that previously occupied the zoom box.

Figure 40.89: Zooming In (Magnifying the Display)



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.90: Zooming Out (Shrinking the Display)



Click the middle mouse button and drag it from right to left to shrink the display. Figure 40.90: Zooming Out (Shrinking the Display) (p. 4033) displays the correct dragging of the mouse, from lower right to upper left on the display, in order to "zoom out". You can also drag from upper right to lower left. After you release the mouse button, Ansys Fluent will redisplay the graphic, shrinking the graphical display by the ratio of sizes of the zoom box you created and the previous display.

With the **Roll-Zoom** function, click the appropriate mouse button and drag the mouse down to zoom in continuously, or up to zoom out. In the Camera Parameters Dialog Box (p. 5277) (Figure 40.88: The Camera Parameters Dialog Box (p. 4030)), use the scale to the right of the dial to zoom the display. Move the slider bar up to zoom in and down to zoom out.

Important:

If you want to zoom in or out while the solver is iterating, click to assign **Zoom In/Out** to the left-mouse-button prior to beginning the calculation. *Roll zoom, using the middle scroll wheel, does not work while the solver is running.*

40.8.4. Controlling Perspective and Camera Parameters

Perspective and other camera parameters are defined in the Views Dialog Box (p. 5272), which you can open by clicking **Camera...** in the Camera Parameters Dialog Box (p. 5277) (Figure 40.87: The Views Dialog Box (p. 4029)).

Results → Graphics → Views...

40.8.4.1. Perspective and Orthographic Views

You may choose to display either an orthographic view or a perspective view of your graphics. To show a perspective view (the default), select **Perspective** in the **Projection** drop-down list in the

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

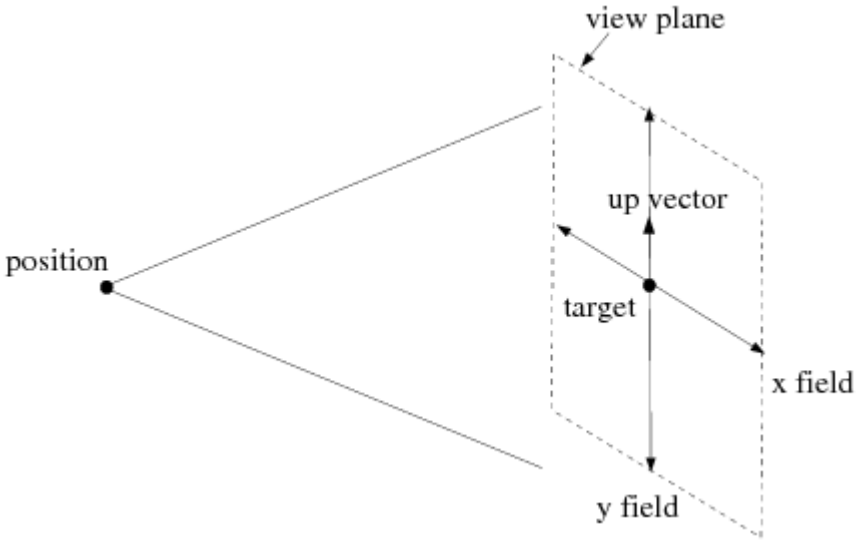
Views Dialog Box (p. 5272) (Figure 40.88: The Camera Parameters Dialog Box (p. 4030)). To turn off perspective, select **Orthographic** in the **Projection** drop-down list.

40.8.4.2. Modifying Camera Parameters

Instead of translating, rotating, and zooming the display as described in Manipulating the Display (p. 4029), you may sometimes want to modify the "camera" through which you are viewing the graphics display.

The camera is defined by four parameters: position, target, up vector, and field, as illustrated in Figure 40.91: Camera Definition (p. 4034). "Position" is the camera's location. "Target" is the location of the point the camera is looking at, and "up vector" indicates to the camera which way is up. "Field" indicates the field of view (width and height) of the display.

Figure 40.91: Camera Definition



To modify the camera's position, select **Position** in the **Camera** drop-down list and specify the **X**, **Y**, and **Z** coordinates of the desired point. To modify the target location, select **Target** in the **Camera** drop-down list and specify the coordinates of the desired point. Select **Up Vector** to change the up direction. The up vector is the vector from (0,0,0) to the specified (**X,Y,Z**) point. Finally, to change the field of view, select **Field** in the **Camera** list and enter the **X** (horizontal) and **Y** (vertical) field distances.

Important:

Click **Apply** after you change each camera parameter (**Position**, **Target**, **Up Vector**, and **Field**).

40.8.5. Saving and Restoring Views

After you make changes to the view shown in your graphics window, you may want to save the view so that you can return to it later. Several default views are predefined for you, and can be easily restored. All saving and restoring functions are performed with the Views Dialog Box (p. 5272) (Figure 40.87: The Views Dialog Box (p. 4029)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Results → Graphics → Views...

Important:

Note that settings for mirroring and periodic repeats are not saved in a view.

40.8.5.1. Restoring the Default View

When experimenting with different view manipulation techniques, you may accidentally "lose" your geometry in the display. You can easily return to the default (front) view by clicking **Default** in the Views Dialog Box (p. 5272).

40.8.5.2. Returning to Previous Views

After manipulating the display and viewing it from different angles, you can return to previous displays by clicking **Previous** in the Views Dialog Box (p. 5272).

40.8.5.3. Saving Views

Once you have created a new view that you want to save for future use, enter a name for it in the **Save Name** field in the Views Dialog Box (p. 5272) and click **Save**. Your new view will be added to the list of **Views**, and you can restore it later as described below.

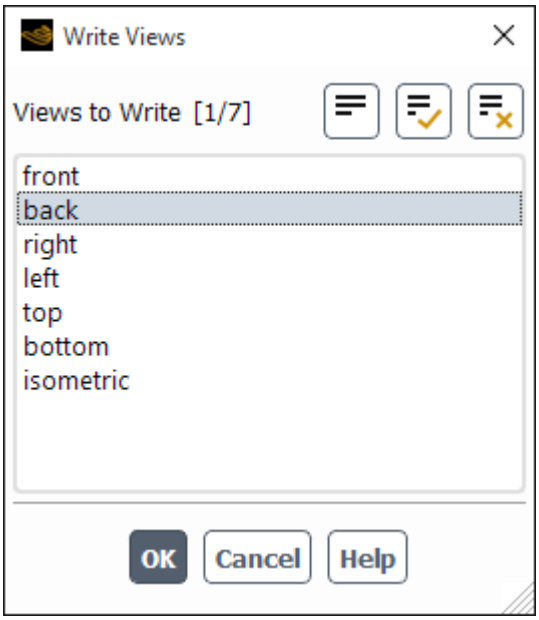
If a view with the same name already exists, you will be asked in a Question Dialog Box (p. 931) if it is OK to overwrite the existing view. If you overwrite one of the default views (top, left, right, front, and so on), be sure to save it in a view file if you want to use it in a later session. Although all views are saved to the case file, the default views are recomputed automatically when a case file is read in. Any custom view with the same name as a default view will be overwritten at that time.

As mentioned previously, all defined views will be saved in the case file when you write one. If you plan to use your views with another case file, you can write a “view file” containing just the views. You can read this view file into another Fluent session involving a different case file and restore any of the defined views, as described below.

To save a view file, click **Write...** in the Views Dialog Box (p. 5272). In the resulting Write Views Dialog Box (p. 5274) (Figure 40.92: The Write Views Dialog Box (p. 4036)), select the views you want to save in the **Views to Write** list and click **OK**. You will then use the **Select File** dialog box to specify the file name and save the view file. See The Select File Dialog Box (p. 931) for more details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.92: The Write Views Dialog Box



40.8.5.4. Reading View Files

If you have saved views to a view file (as described above), you can read them into your current Fluent session by clicking **Read...** in the Views Dialog Box (p. 5272), and indicating the name of the view file in The Select File Dialog Box (p. 931). If a view that you read has the same name as a view that already exists, you will be asked in a Question Dialog Box (p. 931) if it is OK to overwrite (that is, replace) the existing view.

40.8.5.5. Deleting Views

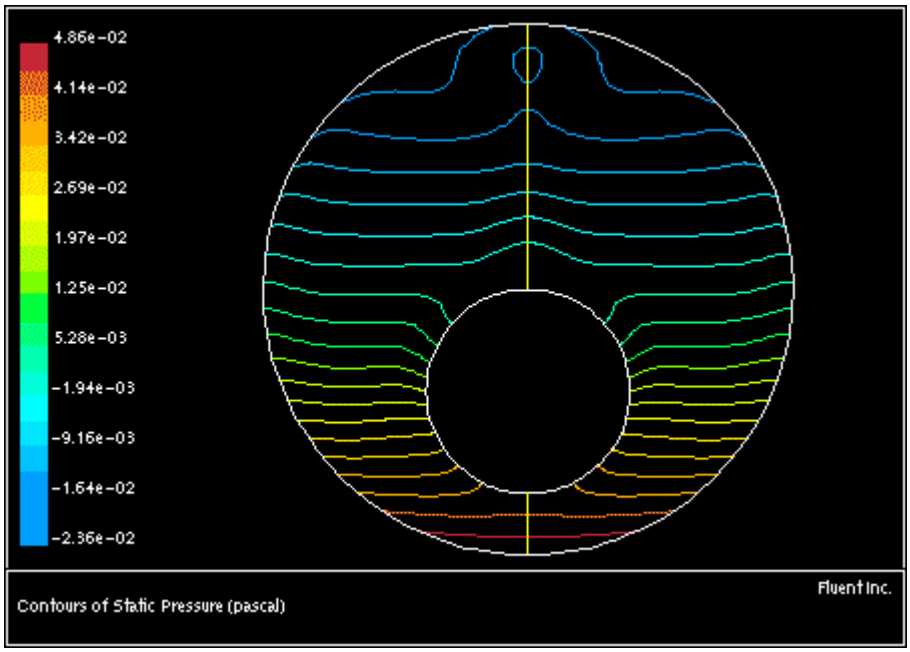
If you decide that you no longer want to keep a particular view, you can delete it by selecting it in the **Views** list and clicking on **Delete**. Use this option carefully, so that you do not accidentally delete one of the predefined views.

40.8.6. Mirroring and Periodic Repeats

If you model the problem domain as a subset of the complete geometry using symmetry or periodic boundaries, you can display results on the complete geometry by mirroring or repeating the domain. For example, only one half of the annulus shown in Figure 40.93: Mirroring Across a Symmetry Boundary (p. 4037) was modeled, but the graphics are displayed on both halves. You can also define mirror planes or periodic repeats just for graphical display, even if you did not model your problem using symmetry or periodic boundaries.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

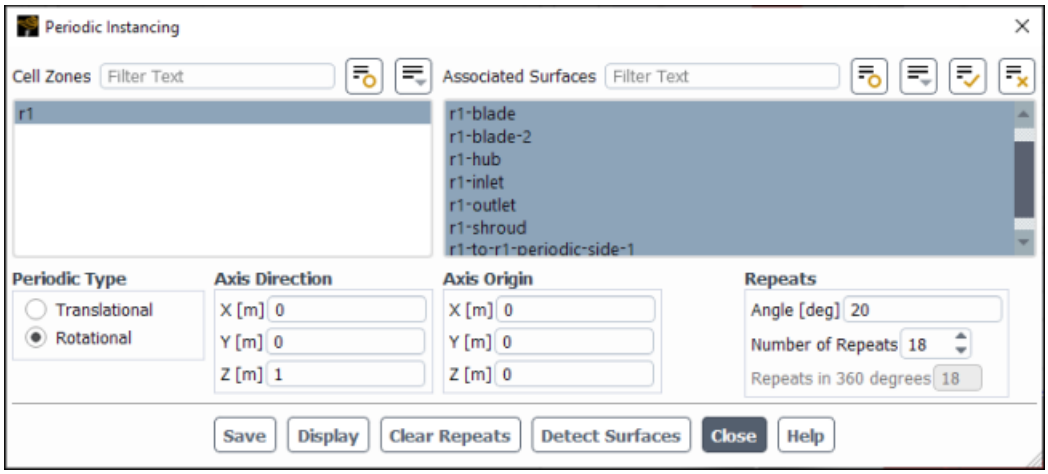
Figure 40.93: Mirroring Across a Symmetry Boundary



Display of periodic repeats is accessed directly in the **View** ribbon tab, while display of symmetries is accessed from the Views Dialog Box (p. 5272) (Figure 40.95: The Views Dialog Box (p. 4038)), which is also available in the **View** ribbon tab.

View → Display → Periodic Instancing...

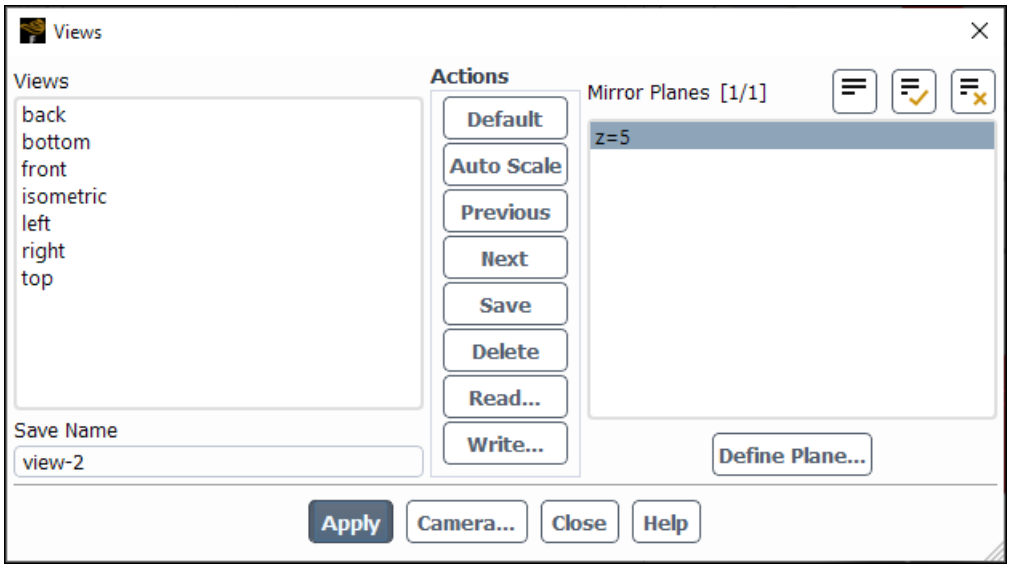
Figure 40.94: The Periodic Instancing Dialog Box



View → Display → Views...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.95: The Views Dialog Box



For a symmetric domain, all symmetry boundaries are listed in the **Mirror Planes** list. Select one or more of these boundaries as the plane(s) about which to mirror the display.

For a periodic domain, click **Periodic Instancing...** to open the Periodic Instancing Dialog Box (p. 5276), to access the periodicity parameters. Specify the number of times to repeat the modeled portion by increasing the value of **Number of Repeats**. If, for example, you modeled a 90° sector of a duct and you wanted to display results on the entire duct, you would set **Number of Repeats** to 4.

In some cases, there may be multiple zones with different periodicity in the domain. For example, in turbomachinery problems with multiple blade rows using the mixing plane model, the periodic angles are different for each blade row. One blade may contain 20 blades (18° periodic angle) and other may contain 15 blades (24° periodic angle). In such cases select the required cell zone and specify the number of repeats for that particular cell zone.

When you click **Display** in the Periodic Instancing Dialog Box (p. 5276) the graphics display will be immediately updated to show the requested periodic repeats (assuming the periodic zones are displayed, either as mesh or as another representative graphics object).

Figure 40.96: Before Applying Periodicity (p. 4039) and Figure 40.97: After Applying Periodicity (p. 4039) shows the display for the sample geometry before and after applying the periodic repeats respectively. In this case the value of **Number of Repeats** is set to 6 for the 60° sector (outer part) and to a value of 4 is set for the 90° sector (inner part) of the geometry.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.96: Before Applying Periodicity

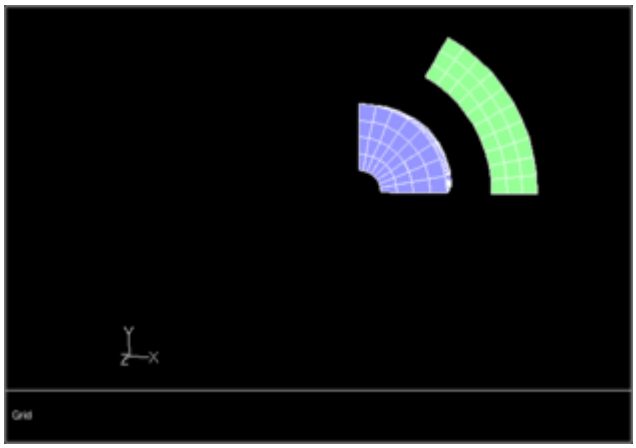
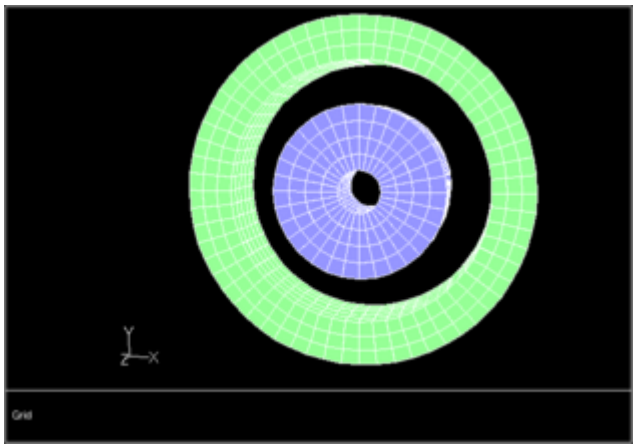


Figure 40.97: After Applying Periodicity



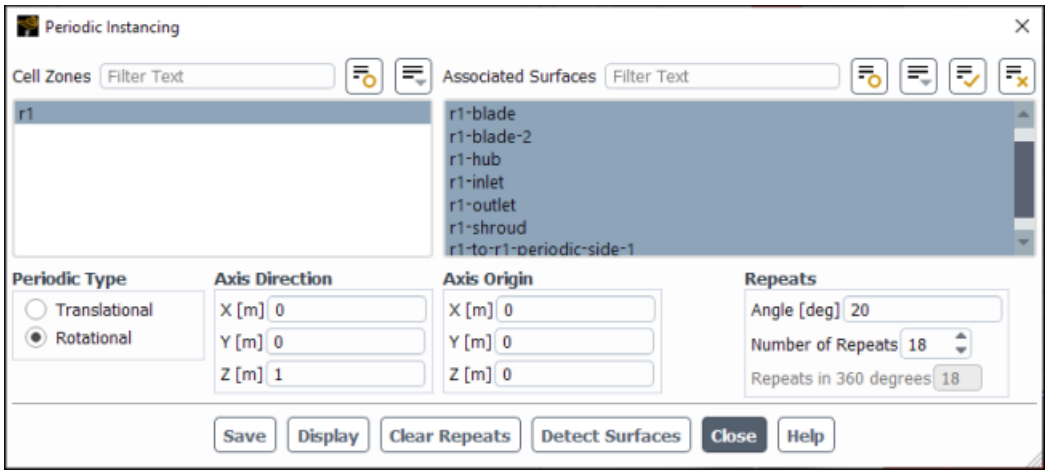
40.8.6.1. Periodic Repeats for Graphics

To define graphical periodicity for a non-periodic domain, do the following:

1. Click **Periodic Instancing...** in the **View** ribbon tab (**Display** group box) to open Figure 40.98: The Periodic Instancing Dialog Box (p. 4040).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.98: The Periodic Instancing Dialog Box



2. (Optional) Click **Detect Surfaces** to have Fluent automatically assess the case file and select what it thinks are the appropriate cell zones, surfaces, and other settings (such as angle and number of repeats) for this problem, and saves them.

   *Assuming Fluent correctly identifies the expected periodics, you can proceed with clicking **Display**, since the periodic instancing setup is now complete (assuming the periodic zones are displayed, either as mesh or as another representative graphics object).*

3. Select the **Cell Zone** you want repeated.

4. Select **Associated Surfaces** that you want repeated.

5. Specify **Rotational** or **Translational** as the **Periodic Type**.

6. For translational periodicity, specify the **Translation** distance of the repeated domain in the **X**, **Y**, and **Z** directions. For rotational periodicity, specify the axis about which the periodicity is defined and the **Angle** by which the domain is rotated to create the periodic repeat. For 3D problems, the axis of rotation is the vector passing through the specified **Axis Origin** and parallel to the vector from (0,0,0) to the (**X,Y,Z**) point specified under **Axis Direction**. For 2D problems, you will specify only the **Axis Origin**; the axis of rotation is the z-direction vector passing through the specified point.

7. Specify the **Number of Repeats** for the selected cell zone. Note that specifying a negative number will repeat the selected zone in the opposite direction.

8. Click **Save**.

9. Follow the same procedure for other cell zones.

10. Click **Display** in the Views Dialog Box (p. 5272) to visualize the modified display.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can clear any periodicities you have defined for the graphics by clicking **Clear Repeats**.

Note:

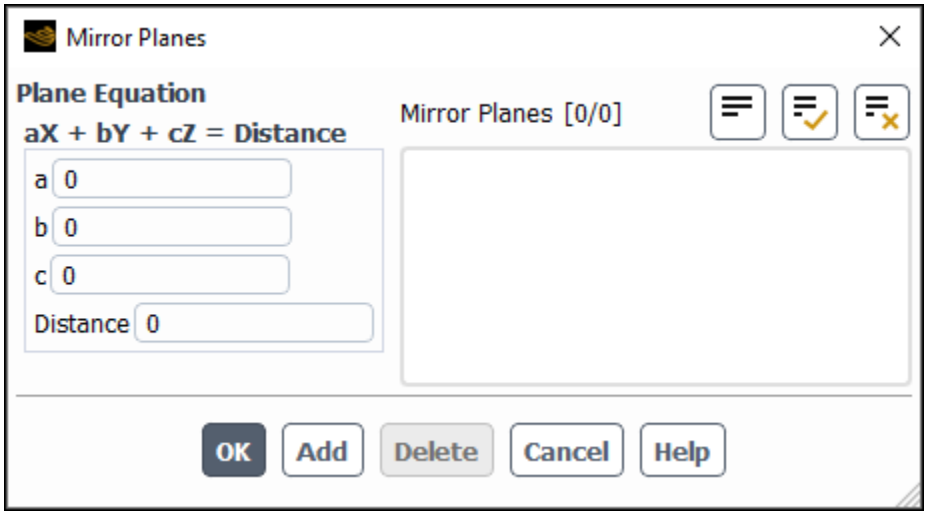
- For the 3D domain with multiple periodic zones having different periodicity, Ansys Fluent can repeat only mesh, contour and vector plots, and not the pathlines and particle tracks. Also if such domain contains, iso-surfaces and clip-surfaces, that are associated with a particular cell zone, they are repeated using the same periodicity that is defined for that cell zone. However, if the surface is not associated with any cell zone, you cannot specify the periodicity for that surface.
- If you have an iso-surface that traverses multiple fluid zones, then this iso-surface will not be included in the surface detection (initiated via **Detect Surfaces**). The proper way of taking advantage of the periodic instancing feature is to create an independent iso-surface cut within each fluid zone.
- Transparencies assigned to the graphics display using the **Scene** dialog box are not applied to the repeat surfaces.

40.8.6.2. Mirroring for Graphics

To define a mirror plane for a non-symmetric domain, do the following:

1. Click **Define Plane...** under **Mirror Planes** in the Views Dialog Box (p. 5272).

   Figure 40.99: The Mirror Planes Dialog Box



2. In the resulting Mirror Planes Dialog Box (p. 5275) (Figure 40.99: The Mirror Planes Dialog Box (p. 4041)), set the coefficients **a**, **b**, **c**, and the **Distance** (of the plane from the origin) in the following equation for the mirror plane:

$$Ax+By+Cz=\text{distance} \tag{40.1}$$

3. Click **Add** to add the defined plane to the **Mirror Planes** list. When you are done creating mirror planes, click **OK**. The newly defined plane(s) will now appear in the **Mirror Planes** list

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

in the Views Dialog Box (p. 5272). To include the mirroring in the display, select the plane(s) and click **Apply**, as described above.

If you want to delete a mirror plane that you have defined, select it in the **Mirror Planes** list in the Mirror Planes Dialog Box (p. 5275) and click **Delete**. When you click **OK** in this dialog box, the deleted plane will be removed permanently from the **Mirror Planes** list in the Views Dialog Box (p. 5272).

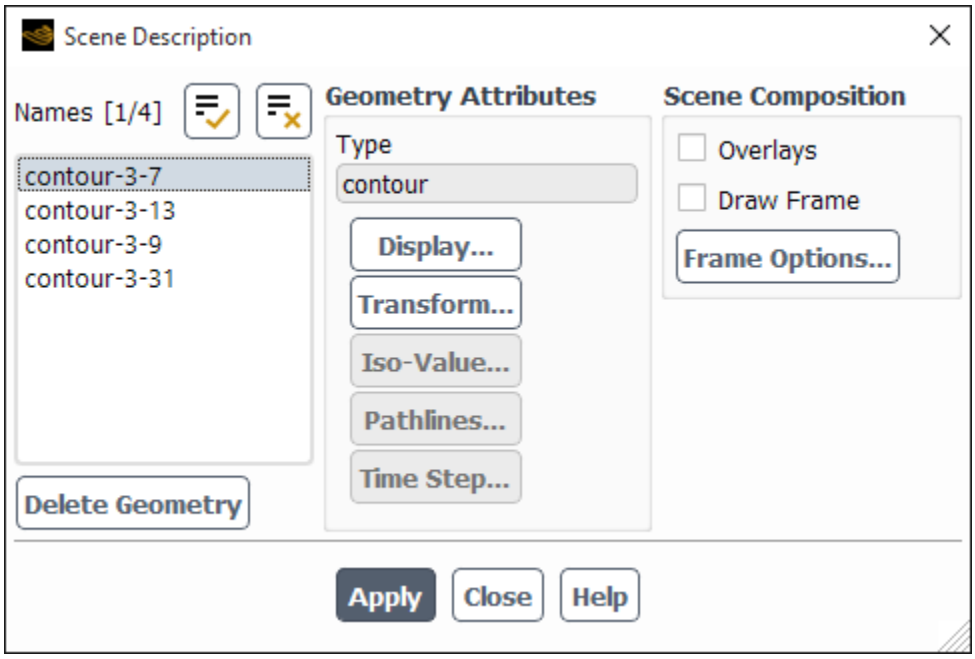
40.9. Advanced Scene Composition

Figure 40.100: The Scene Description Dialog Box (p. 4042) allows you to create a more complex scene than what you can create using Figure 40.50: The Scene Dialog Box (p. 3936).

Once you have displayed some geometric objects (meshes, surfaces, contours, vectors, and so on) in your graphics window, you may want to move them around and change their characteristics to increase the effectiveness of the scene displayed. You can use the Scene Description Dialog Box (p. 5265) (Figure 40.100: The Scene Description Dialog Box (p. 4042)) and the Display Properties Dialog Box (p. 5267) (Figure 40.101: The Display Properties Dialog Box (p. 4044)) and Transformations Dialog Box (p. 5269) (Figure 40.103: The Transformations Dialog Box (p. 4047)), which are opened from within it, to rotate, translate, and scale each object individually, as well as change the color and visibility of each object.

View → Graphics → Compose...

Figure 40.100: The Scene Description Dialog Box



The Iso-Value Dialog Box (p. 5270) (Figure 40.104: The Iso-Value Dialog Box (p. 4048)), which is also opened from within the Scene Description Dialog Box (p. 5265), enables you to change the isovalue of a selected isosurface. The Pathline Attributes Dialog Box (p. 5271) (Figure 40.105: The Pathline Attributes Dialog Box (p. 4049)) lets you set some pathline attributes. The ability to make geometric objects visible and invisible is especially useful when you are creating an animation (see Animating Graphics (p. 4051)) because it enables you to add or delete objects from the scene one at a time. The ability to change the color and position of an object independently of the others in the scene is also useful for setting up animations, as is the ability to change isosurface isovalues. You will find the features in the Scene Description Dialog

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Box (p. 5265) useful even when you are not generating animations because they enable you to manage your graphics window efficiently. The procedure for overlaying graphics, which uses the **Scene Description** dialog box, is described in Advanced Graphics Overlays (p. 3954). (Note that you cannot use the Scene Description Dialog Box (p. 5265) to control XY plot and histogram displays.)

For additional information, see the following sections:

40.9.1. Selecting the Object(s) to be Manipulated
40.9.2. Changing an Object's Display Properties
40.9.3. Transforming Geometric Objects in a Scene
40.9.4. Modifying Iso-Values
40.9.5. Modifying Pathline Attributes
40.9.6. Deleting an Object from the Scene
40.9.7. Adding a Bounding Frame

40.9.1. Selecting the Object(s) to be Manipulated

In order to manipulate the objects in the scene, you will begin by selecting the object or objects of interest in the **Names** list in the Scene Description Dialog Box (p. 5265) (Figure 40.100: The Scene Description Dialog Box (p. 4042)). The **Names** list is a list of the geometric objects that currently exist in the scene (including those that are presently invisible). If you select more than one object at a time, any operation (transformation, color specification, and so on) will apply to all the selected objects. You can also select objects by clicking on them in the graphics display using the mouse-probe button, which is, by default, the right mouse button. (See Controlling the Mouse Button Functions (p. 4020) for information about mouse button functions.) To deselect a selected object, simply click its name in the **Names** list.

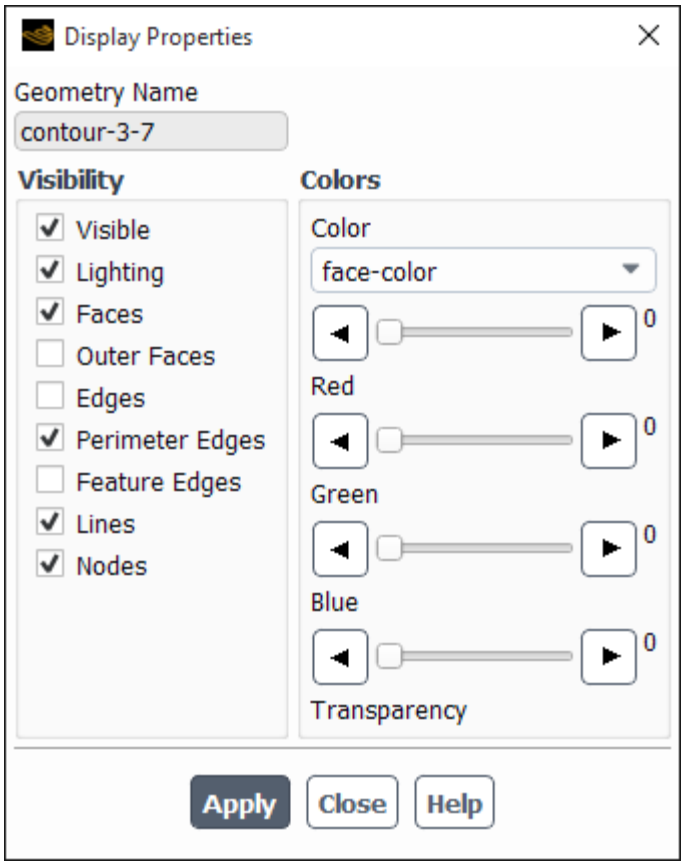
When you select one or more objects (either in the **Names** list or in the display), the **Type** field will report the type of the selected object(s). Possible types for a single object include `mesh`, `surface`, `contour`, `vector`, `path`, and `text` (that is, annotation text). This information is especially helpful when you need to distinguish two or more objects with the same name. When more than one object is selected, the type displayed is `Group`.

40.9.2. Changing an Object's Display Properties

To enhance the scene in the graphics window, you can change the color, visibility, and other display properties of each geometric object in the scene. You can specify different colors for displaying the edges and faces of a mesh object to show the underlying mesh (edges) when the faces of the mesh are filled and shaded. You can also make a selected object temporarily invisible. If, for example, you are displaying the entire mesh for a complicated problem, you can make objects visible or invisible to display only certain boundary zones of the mesh without regenerating the mesh display using the Mesh Display Dialog Box (p. 4688). You can also use the visibility controls to manipulate geometric objects for efficient graphics display or for the creation of animations. These features, plus several others, are available in the Display Properties Dialog Box (p. 5267) (Figure 40.101: The Display Properties Dialog Box (p. 4044)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.101: The Display Properties Dialog Box



To set the display properties described above, select one or more objects in the **Names** list in the Scene Description Dialog Box (p. 5265) and then click **Display...** to open the Display Properties Dialog Box (p. 5267) for that object or group of objects.

40.9.2.1. Controlling Visibility

There are several ways for you to control the visibility of an object. All visibility options are listed under the **Visibility** heading in the Display Properties Dialog Box (p. 5267).

- To make the selected object(s) invisible, turn off the **Visible** option. To "undo" invisibility, enable the **Visible** option again.
- To turn the effect of lighting for the selected object(s) on or off, use the **Lighting** check box. You can choose to have lighting affect only certain objects instead of all of them. Note that if **Lighting** is turned on for an object such as a contour or vector plot, the colors in the plot will not be exactly the same as those in the colormap at the left of the display.
- To toggle the filled display of faces for the selected mesh or surface object(s), use the **Faces** option. Turning **Faces** on here has the same effect as turning it on for the entire mesh in the Mesh Display Dialog Box (p. 4688).
- To turn the display of outer edges on or off, use the **Outer Faces** option. This option is useful for displaying both sides of a slit wall. By default, when you display a slit wall, one side will "bleed" through to the other. When you turn off the **Outer Faces** option, the display of a slit wall will

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

show each side distinctly as you rotate the display. This option can also be useful for displaying two-sided walls (that is, walls with fluid or solid cells on both sides).

- To turn the display of interior and exterior edges of the geometric object(s) on or off, use the **Edges** option.
- To turn the display of the outline of the geometric object(s) on or off, use the **Perimeter Edges** check box.
- To toggle the display of feature lines (described in Adding Features to an Outline Display (p. 3893)), if any, for the selected object(s), use the **Feature Edges** option.
- To toggle the display of the lines (if any) in the geometric object(s), use the **Lines** check box. Pathlines, line contours, and vectors are "lines".
- To toggle the display of nodes (if any) in the geometric object(s), use the **Nodes** check box.

Once you have set the appropriate display parameters, click **Apply** to update the graphics display.

40.9.2.2. Controlling Object Color and Transparency

The Display Properties Dialog Box (p. 5267) also lets you control an object's color and how transparent it is. All color and transparency options are listed under the **Colors** heading.

- To modify the color of faces, edges, or lines in the selected object(s), choose **face-color**, **edge-color**, **line-color**, or **node-color** in the **Color** drop-down list. The **Red**, **Green**, and **Blue** color scales will show the RGB components of the face, edge or line color, which you can modify by moving the sliders on the color scales. When you are satisfied with the color specification, click **Apply** to save it and update the display. The ability to set the colors for faces and edges can be useful when you want to have a filled display for the mesh or surface, but you also want to be able to see the mesh lines. You can achieve this effect by specifying different colors for the faces and the edges.
- To set the relative transparency of an object, select **face-color** in the **Color** drop-down list. Move the slider on the **Transparency** scale and click **Apply** to update the graphics display. An object with a transparency of 0 is opaque, and an object with a transparency of 100 is transparent. By specifying a high transparency value for the walls of a pipe, for example, you will be able to see contours that you have displayed on cross-sections inside the pipe. This feature is available on all platforms when the software z buffer is used for hidden surface removal, but if your display hardware supports transparency, it will be more efficient to use the hardware z buffer as the hidden surface method instead. You can select these methods in the Display Options Dialog Box (p. 5262), as described in Modifying the Rendering Options (p. 3987).

Important:

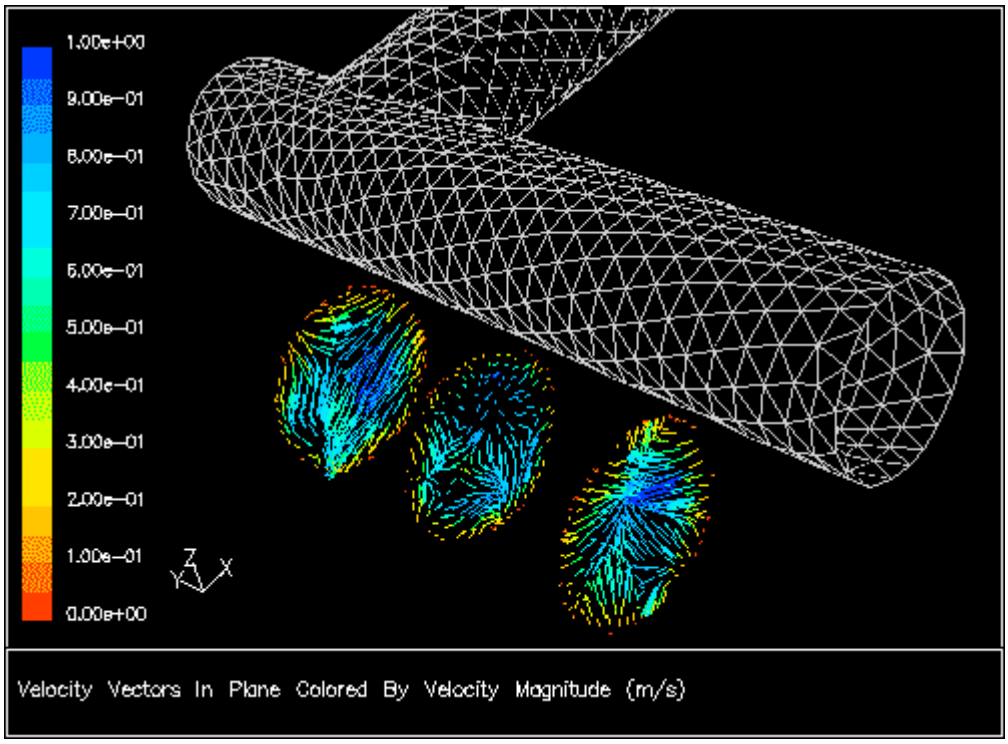
If you save a picture of a display with transparent surfaces, you should *not* set the **File Type** in the Save Picture Dialog Box (p. 5254) to **Vector**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

40.9.3. Transforming Geometric Objects in a Scene

When you are composing a scene in your graphics window, you might find it helpful to move a particular object from its original position or to increase or decrease its size. For example, if you have displayed contours or vectors on cross-sections of an internal flow domain (such as a pipe), you might want to translate these cross-sections so that they will appear outside of the pipe, where they can be seen and interpreted more easily. Figure 40.102: Velocity Vectors Translated Outside the Domain for Better Viewing (p. 4046) shows such an example.

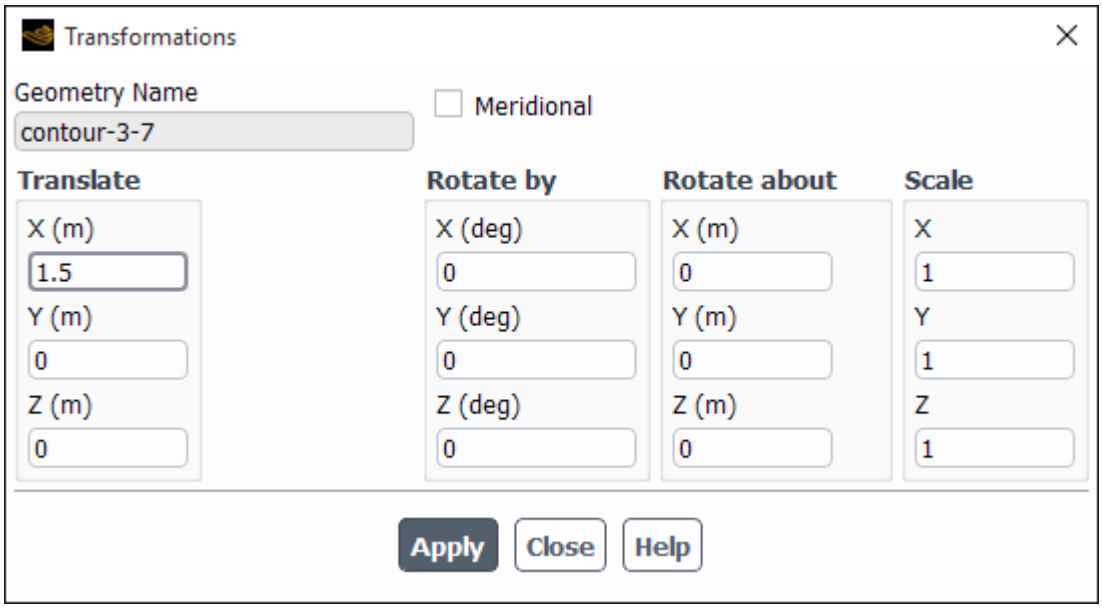
Figure 40.102: Velocity Vectors Translated Outside the Domain for Better Viewing



You can also move an object by rotating it about the x-, y-, or z-axis. If you want to display one object more prominently than the others, you can scale its size. If your geometry is rotating or has rotational symmetry, you can display the meridional view. All of these capabilities are available in the Transformations Dialog Box (p. 5269) (Figure 40.103: The Transformations Dialog Box (p. 4047)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.103: The Transformations Dialog Box



To perform the transformations described above, select one or more objects in the **Names** list in the Scene Description Dialog Box (p. 5265) and then click **Transform...** to open the Transformations Dialog Box (p. 5269) for that object or group of objects.

40.9.3.1. Translating Objects

To translate the selected object(s), enter the translation distance in each direction in the **X**, **Y**, and **Z** real number fields under **Translate**. Note that you can check the domain extents in the Scale Mesh Dialog Box (p. 4686) or the Iso-Surface Dialog Box (p. 5501). Translations are not cumulative, so you can easily return to a known state. To return to the original position, simply enter 0 in all three real number fields.

40.9.3.2. Rotating Objects

To rotate the selected object(s), enter the number of degrees by which to rotate about each axis in the **X**, **Y**, and **Z** integer number fields under **Rotate By**. You can enter any value between -360 and 360. By default, the rotation origin will be (0,0,0). If you want to spin an object about its own origin, or about some other point, specify the **X**, **Y**, and **Z** coordinates of that point under **Rotate About**.

Rotations are not cumulative, so you can easily return to a known state. To return to the original position, simply enter 0 in all three integer number fields under **Rotate By**.

40.9.3.3. Scaling Objects

To scale the selected object(s), enter the amount by which to scale in each direction in the **X**, **Y**, and **Z** real number fields under **Scale**. To avoid distortion of the object's shape, be sure to specify the same value for all three entries. Scaling is not cumulative, so you can easily return to a known state. To return the object to its original size, simply enter 1 in all three real number fields.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

40.9.3.4. Displaying the Meridional View

To display the meridional view of the selected object(s), enable the **Meridional** option. This option is available only for 3D models. It is applicable to cases with a defined axis of rotation and is especially useful in turbomachinery applications.

The meridional transformation projects the selected entities onto a surface of constant angular coordinate, θ. The resultant projection thus lies in an (r, ζ) plane where ζ is in the direction of the rotation axis and r is normal to it. The value of θ used for the projection is taken as that corresponding to the minimum (r, ζ) point of the entity.

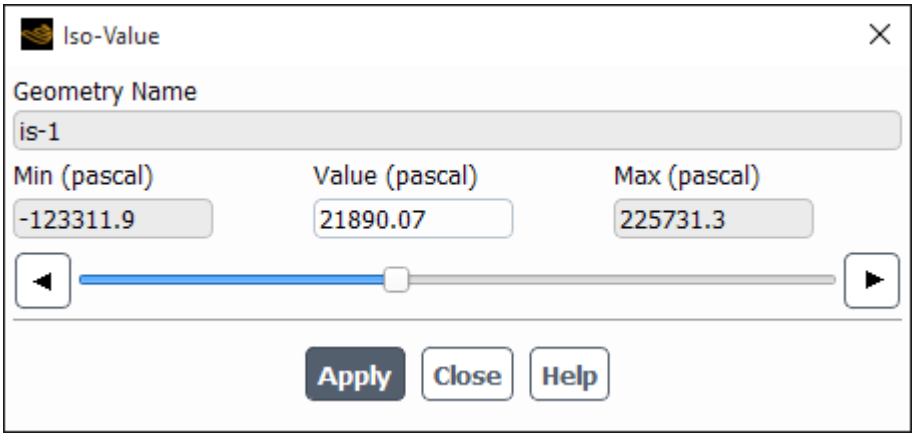
40.9.4. Modifying Iso-Values

You can use the ability to generate surfaces with intermediate values between two isosurfaces (with different isovalues) to create snapshots of the results. If the surfaces have contours, vectors, or pathlines displayed on them, Ansys Fluent will generate and display contours, vectors, or pathlines on the intermediate surfaces that it creates.

40.9.4.1. Steps for Modifying Iso-Values

You can modify an isosurface's isovalue directly by selecting it in the **Names** list in the Scene Description Dialog Box (p. 5265) or indirectly by selecting an object displayed on the isosurface. Then click **Iso-Value...** to open the Iso-Value Dialog Box (p. 5270) (Figure 40.104: The Iso-Value Dialog Box (p. 4048)) for the selected object. Note that this button is available only if the geometric object selected in the **Names** list is an isosurface or an object on an isosurface (contour on an isosurface, for example); otherwise it is grayed out.

Figure 40.104: The Iso-Value Dialog Box



In the Iso-Value Dialog Box (p. 5270), set the new isovalue in the **Value** field, and click **Apply**. Contours, vectors, or pathlines that were displayed on the original isosurface will be displayed for the new isovalue.

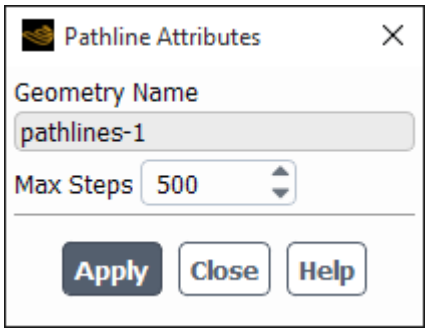
40.9.5. Modifying Pathline Attributes

If you are creating animations of existing pathlines, you may want to change the number of steps used in the computation of the pathlines. This enables you to animate pathlines advancing through the domain. To do so, select the pathlines in the **Names** list in the Scene Description Dialog Box (p. 5265)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and then click **Pathlines...** to open the Pathline Attributes Dialog Box (p. 5271) (Figure 40.105: The Pathline Attributes Dialog Box (p. 4049)).

Figure 40.105: The Pathline Attributes Dialog Box



In the Pathline Attributes Dialog Box (p. 5271), set the new maximum number of steps for pathline computation (**Max Steps**). After you change the value and click **Apply**, the selected pathline will be recomputed and redrawn.

40.9.6. Deleting an Object from the Scene

If you are composing a complex scene with overlays and find that you no longer want to keep one of the objects, it is possible to delete it without affecting any of the other objects in the scene. The ability to delete individual objects is especially useful if you have overlays on and you generate an unwanted object (for example, if you generate contours of the wrong variable). You can simply delete the unwanted object and continue your scene composition, instead of starting over from the beginning. Note that it is also possible to make objects temporarily invisible, as described in Controlling Visibility (p. 4044).

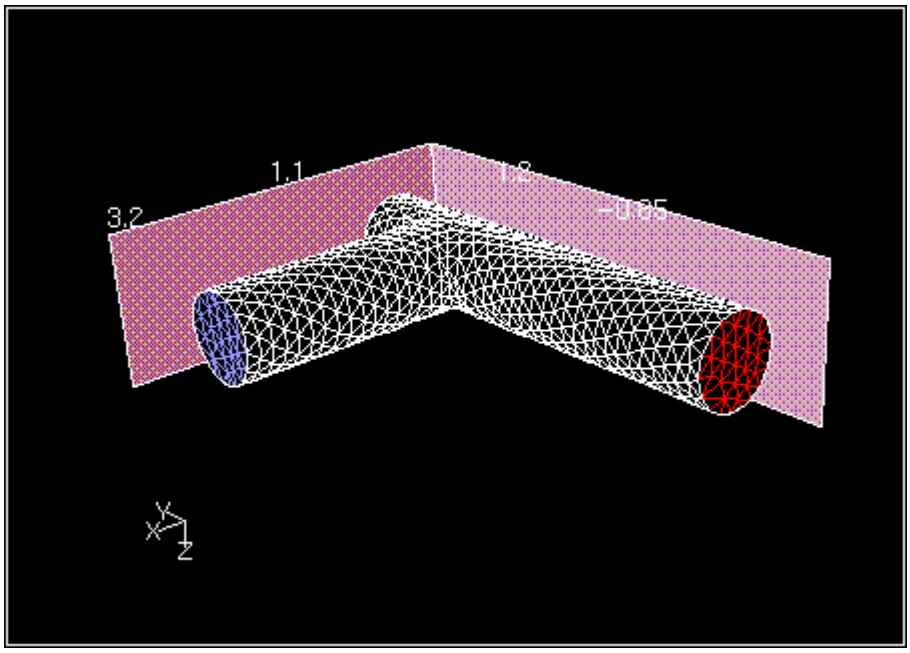
Object deletion is performed in the Scene Description Dialog Box (p. 5265) (Figure 40.100: The Scene Description Dialog Box (p. 4042)). To delete an object from the scene, select it in the **Names** list and then click **Delete Geometry**. The selected name will disappear from the **Names** list, and the display will be updated immediately.

40.9.7. Adding a Bounding Frame

Ansys Fluent enables you to add a bounding frame around your displayed domain. You may also include measure markings on the bounding frame to indicate the length, height, and/or width of the domain, as shown in Figure 40.106: Graphics Display with Bounding Frame (p. 4050).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

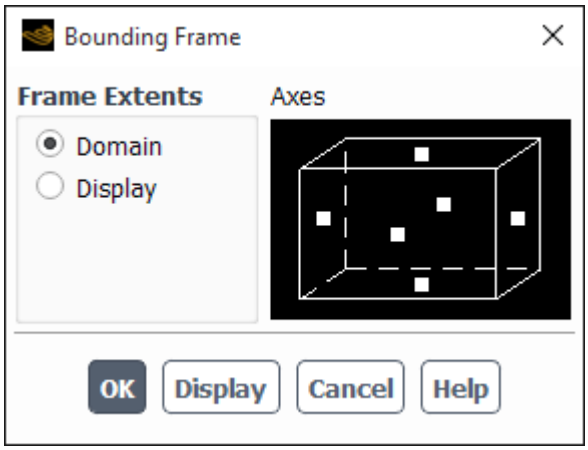
Figure 40.106: Graphics Display with Bounding Frame



To add a bounding frame to your display, you will follow the procedure below:

1. Click **Frame Options...** in the Scene Description Dialog Box (p. 5265) (Figure 40.100: The Scene Description Dialog Box (p. 4042)) to open the Bounding Frame Dialog Box (p. 5271) (Figure 40.107: The Bounding Frame Dialog Box (p. 4050)).

   Figure 40.107: The Bounding Frame Dialog Box

2. Under **Frame Extents** in the Bounding Frame Dialog Box (p. 5271), select **Domain** or **Display** to indicate whether the bounding frame should encompass the domain extents or only the portion of the domain that is shown in the display.

3. In the **Axes** portion of the Bounding Frame Dialog Box (p. 5271), specify the frame boundaries and measurements to be shown in the display:

   - Indicate the bounding plane(s) (for example, the *x*-*z* and *y*-*z* planes shown in Figure 40.106: Graphics Display with Bounding Frame (p. 4050)) to be displayed by clicking on the

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

white square on the appropriate plane of the box shown under the **Axes** heading. You can use any of the mouse buttons. The square will turn red to indicate that the associated bounding plane will be displayed in the graphics window.

- Specify where you would like to see the measurement annotations by clicking on the appropriate edge of the box. The edge will turn red to indicate that the markings will be displayed along that edge of the displayed geometry.

Important:

If you have trouble determining which square or edge corresponds to which location in your domain, you can easily find out by displaying one or two bounding planes to get your bearings. You can then select the appropriate objects to obtain the final display.

---

4. Click **Display** to update the display with the current settings. If you are not satisfied with the frame, repeat steps 2 and/or 3 and click **Display** again.

5. Once you are satisfied with the bounding frame that is displayed, click **OK** to close the Bounding Frame Dialog Box (p. 5271) and save the frame settings for future displays.

6. If you want to include the bounding frame in all subsequent displays, enable the **Draw Frame** option in the Scene Description Dialog Box (p. 5265) and click **Apply**. If this option is not enabled, the bounding box will appear only in the current display; it will not be redisplayed when you generate a new display (unless you have overlays enabled).

The bounding planes and axis annotations will appear in the **Names** list of the Scene Description Dialog Box (p. 5265), and you can manipulate them in the same way as any other geometric object in the display. For example, you can use the Display Properties Dialog Box (p. 5267) to change the face color of a bounding plane or to make it transparent (see Changing an Object's Display Properties (p. 4043)).

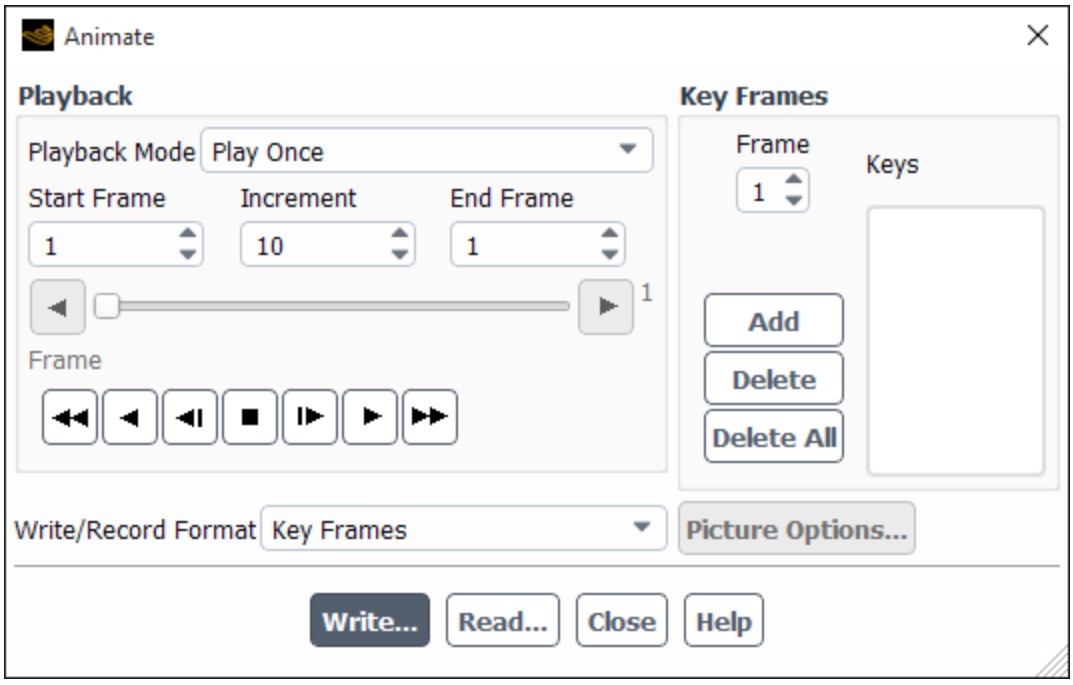
40.10. Animating Graphics

To generate animations that progress from one static view of the graphics display to the next, you can set up "key frames" (individual static images) using the Animate Dialog Box (p. 5251) (Figure 40.108: The Animate Dialog Box (p. 4052)).

Results → Animations → Scene Animation → Edit...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.108: The Animate Dialog Box



You can compose a scene in the graphics window and define it as a single key frame. Then, modify the scene by moving or scaling objects, making some objects invisible or visible, changing colors, changing the view, or making other changes, and define the new scene as another key frame. Ansys Fluent can then interpolate smoothly between the two frames that you defined, creating a specified number of intermediate frames.

If you want to create a graphical animation of the solution over time, you can use Figure 36.54: The Animation Definition Dialog Box (p. 3715) to set up the graphical displays that you want to use in the animation. You can choose the type of display you want to animate by selecting it from the **Animation Object** list or by creating a new animation object. For details on animating the solution, see Animating the Solution (p. 3714). For more information on generating, displaying, and saving pathlines and particle tracks, refer to Displaying Pathlines (p. 3922). See Displaying Results on a Sweep Surface (p. 3937) for information about displaying the mesh, contours, or vectors on a surface that sweeps through the domain.

Note:

- "Key frame" animations created through the **Animate** dialog box are not compatible with iso-surfaces.
- "Key frame" animations of pathlines are only supported for the non-persistent/"global" version of the **Pathlines** dialog box that does not have a **Name** field. This version of the **Pathlines** dialog box is accessed by right-clicking **Pathlines** in the Outline View tree and selecting **Edit...**.

  You must enable **Expose legacy non-persistent graphics** in the **Graphics** branch of **Preferences** (**File** > **Preferences...**) to see the **Edit...** option for pathlines as described above.

For additional information, see the following sections:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

40.10.1. Creating an Animation
40.10.2. Playing an Animation
40.10.3. Saving an Animation
40.10.4. Reading an Animation File
40.10.5. Notes on Animation

40.10.1. Creating an Animation

You can define any number of key frames (up to 3000) to create your animation. By assigning the appropriate numbers to the key frames, you provide the information Ansys Fluent needs to create the correct number of intermediate frames. For example, to create a simple animation that begins with a front view of an object, moves to a side view, and ends with a rear view of the object, you would follow the procedure outlined below:

1. Determine the number of frames that you want in the animation. For this example, consider the animation to be 31 frames.

2. Determine the number of key frames that you need to specify. In this example, you will specify three: one showing the front view, one showing the side view, and one showing the rear view.

3. Determine the appropriate key frame numbers to assign to the 3 specified frames. Here, the front view will be specified as key frame 1, the side view will be key frame 16, and the rear view will be key frame 31.

4. Compose the scenes for each view to be used as a key frame. You can use the Scene Description Dialog Box (p. 5265) (see Advanced Scene Composition (p. 4042)) and the Views Dialog Box (p. 5272) (see Controlling the Display State and Modifying the View (p. 4025)) to modify the display, and any other dialog boxes or commands to create contours, vectors, pathlines, and so on to be included in each scene. After you complete each scene, create the appropriate key frame by setting the **Frame** number and clicking **Add** under **Key Frames** in the Animate Dialog Box (p. 5251). For special considerations related to key frame definition, see Notes on Animation (p. 4056).

   Important:

   Be sure to change the **Frame** number before clicking **Add**, or you will overwrite the last key frame that you created.

   You can check any of the key frames that you have created by selecting it in the **Keys** list. The selected key frame will be displayed in the graphics window.

5. When you complete the animation, you can play it back as described in Playing an Animation (p. 4054) and/or save it as described in Saving an Animation (p. 4055).

40.10.1.1. Deleting Key Frames

If, during the creation of your animation, you want to delete one of the key frames that you have defined, select the key frame in the **Keys** list and click **Delete**. If you want to delete all key frames and start over again, click **Delete All**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

40.10.2. Playing an Animation

Once you have defined the key frames (as described in Creating an Animation (p. 4053)) or read in a previously created animation file (as described in Reading an Animation File (p. 4056)), you can play back the animation and Ansys Fluent will interpolate between the frames that you specified to complete the animation.

To play the animation once through from start to finish, click the "play" button under the **Playback** heading in the Animate Dialog Box (p. 5251). (The buttons function in a way similar to those on a standard video cassette player. "Play" is the second button from the right—a single triangle pointing to the right.) To play the animation backwards once, click the "play reverse" button (the second from the left—a single triangle point to the left). As the animation plays, the **Frame** scale shows the number of the frame that is currently displayed, as well as its relative position in the entire animation. If, instead of playing the complete animation, you want to jump to a particular key frame, move the **Frame** slider bar to the desired frame number, and the frame corresponding to the new frame number will be displayed in the graphics window.

Additional options for playing back animations are described below. Be sure to check Notes on Animation (p. 4056) as well for important notes about playing back animations.

40.10.2.1. Playing Back an Excerpt

You may sometimes want to play only one portion of a long animation. To do this, you can modify the **Start Frame** and the **End Frame** under the **Playback** heading in the Animate Dialog Box (p. 5251). For example, if your animation contains 50 frames, but you want to play only frames 20 to 35, you can set **Start Frame** to 20 and **End Frame** to 35. When you play the animation, it will start at frame 20 and finish at frame 35.

40.10.2.2. "Fast-Forwarding" the Animation

You can "fast-forward" or "fast-reverse" the animation by skipping some of the frames during playback. To fast-forward the animation, set the **Increment** and click the fast-forward button. If, for example, your **Start Frame** is 1, your **End Frame** is 15, and your **Increment** is 2, when you click the fast-forward button (the last button on the right—two triangles pointing to the right), the animation will show frames 1, 3, 5, 7, 9, 11, 13 and 15. Clicking on the fast-reverse button (the first button on the left—two triangles pointing to the left) will show frames 15, 13, 11,...1.

40.10.2.3. Continuous Animation

If you want the playback of the animation to repeat continuously, there are two options available.

- To continuously play the animation from beginning to end (or from end to beginning, if you use one of the reverse play buttons), select **Auto Repeat** in the **Playback Mode** drop-down list in the Animate Dialog Box (p. 5251).
- To play the animation back and forth continuously, reversing the playback direction each time, select **Auto Reverse** in the **Playback Mode** drop-down list.

To turn off the continuous playback, select **Play Once** in the **Playback Mode** list. This is the default setting.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

40.10.2.4. Stopping the Animation

To stop the animation during playback, click the "stop" button (the square in the middle of the playback control buttons). If your animation contains very complicated scenes, there may be a slight delay before the animation stops.

40.10.2.5. Advancing the Animation Frame by Frame

To advance the animation manually frame by frame, use the third button from the right (a vertical bar with a triangle pointing to the right). Each time you click this button, the next frame will be displayed in the graphics window. To reverse the animation frame by frame, use the third button from the left (a left-pointing triangle with a vertical bar). Frame-by-frame playback enables you to freeze the animation at points that are of particular interest.

40.10.3. Saving an Animation

Once you have created your animation, you can save it in any of the following formats:

- Animation file containing the key frame descriptions
- Picture files, each containing a frame of the animation
- MPEG file containing each frame of the animation

  Note that MPEG is not supported for saving on Ubuntu operating systems.

40.10.3.1. Animation File

You can save the key frame definitions to a file that can be read back into Ansys Fluent (see Reading an Animation File (p. 4056)) when you want to replay the animation. Since the animation file will contain only the key frame definitions, you must be sure that you have a case and data file containing the necessary surfaces and other information referred to by the key frame descriptions.

To write an animation file, select **Key Frames** in the **Write/Record Format** drop-down list in the Animate Dialog Box (p. 5251), and click **Write...**. In The Select File Dialog Box (p. 931), specify the name of the file and save it.

40.10.3.2. Picture File

You can also generate a picture file for each frame in the animation. This feature enables you to save your animation frames to picture files used by an external animation program such as ImageMagick. To save the animation as a picture file, follow these steps:

1. Select **Picture Files** in the **Write/Record Format** drop-down list in the Animate Dialog Box (p. 5251).

2. If necessary, click **Picture Options...** to open the Save Picture Dialog Box (p. 5254) and set the appropriate parameters for saving the picture files. (If you are saving picture files for use with ImageMagick, for example, you may want to select the window dump format. See Window

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Dumps (Linux Systems Only) (p. 1025) for details.) Click **Apply** in the Save Picture Dialog Box (p. 5254) to save your modified settings.

Important:

Do not click **Save...** in the Save Picture Dialog Box (p. 5254). You will save the picture files from the Animate Dialog Box (p. 5251) in the next step.

3. In the Animate Dialog Box (p. 5251), click **Write...**. In The Select File Dialog Box (p. 931), specify the filename and click **OK** to save the files. (See Window Dumps (Linux Systems Only) (p. 1025) for information about specifying filenames that increment automatically as additional pictures are saved.) Ansys Fluent will replay the animation, saving each frame to a separate file.

40.10.3.3. MPEG File

It is also possible to save all of the frames of the animation in an MPEG file, which can be viewed using an MPEG decoder such as **mpeg_play**. Saving the entire animation to an MPEG file will require less disk space than storing the individual window dump files (using the picture method), but the MPEG file will yield lower-quality images. To save the animation to an MPEG file, follow these steps:

1. Select **MPEG** in the **Write/Record Format** drop-down list in the Animate Dialog Box (p. 5251).

2. In the Animate Dialog Box (p. 5251), click **Write...**. In The Select File Dialog Box (p. 931), specify the filename, and click **OK** to save the files.

   Ansys Fluent replays the animation and saves each frame to a separate scratch file; it then combines all of the files into a single MPEG file.

40.10.4. Reading an Animation File

If you have saved the key frames defining an animation to an animation file (as described in Saving an Animation (p. 4055)), you can read that file back in at a later time (or in different session) and play the animation. Before reading in an animation file, be sure that the current case and data contain the surfaces and any other information that the key frame description refers to.

To read an animation file, click **Read...** in the Animate Dialog Box (p. 5251). In The Select File Dialog Box (p. 931), specify the name of the file to be read.

40.10.5. Notes on Animation

When you are creating and playing back animations, note the following:

- For smoother animations, enable the **Double Buffering** option in the Display Options Dialog Box (p. 5262) (see Modifying the Rendering Options (p. 3987)). This will reduce screen flicker during graphics updates.

- When you are defining key frames, you must create all geometric objects that will be used in the animation before you create any key frames. You cannot create a key frame using one set of geometric objects and then generate a new geometry (such as a vector plot) and include that in an-

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

other key frame. Create all geometric objects first, and then use the Display Properties Dialog Box (p. 5267) to control the visibility of the objects in each key frame (see Controlling Visibility (p. 4044)).

- A single animation sequence can contain up to 3000 key frames.
- When you play back an animation, the colormap used will be the one that is currently active, *not* the one that was active during "recording."

40.11. Histogram and XY Plots

In addition to the many graphics tools already discussed, Ansys Fluent also provides tools that enable you to generate XY plots and histograms of solution, file, profile, and residual data. You can modify the colors, titles, legend, and axis and curve attributes to customize your plots. The following sections describe the XY and histogram plotting features in Ansys Fluent.

40.11.1. Plot Types
40.11.2. XY Plots of Solution Data
40.11.3. Creating an XY Plot From Multiple Data Sources (Including Files)
40.11.4. XY Plots of Profiles
40.11.5. XY Plots of Circumferential Averages
40.11.6. XY Plot File Format
40.11.7. Residual Plots
40.11.8. Histograms
40.11.9. Modifying Axis Attributes
40.11.10. Modifying Curve Attributes

40.11.1. Plot Types

Data can be plotted in XY (abscissa/ordinate) form or histogram form. Each form is described below.

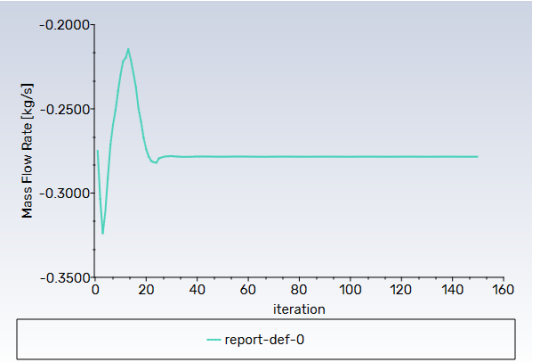
40.11.1.1. XY Plots

An XY (abscissa/ordinate) plot is a line and/or symbol chart of data. Virtually any defined variable or function is accessible for this type of plot. Furthermore, you may read in an externally-generated data file in order to compare your results with experimental data. You can also use the XY-plot facility to plot out profile data, the residual histories of variables, or the time histories if you have a transient problem.

Ansys Fluent provides tools for controlling many aspects of the XY plot, including background color, legend, and axis and curve attributes. Figure 40.109: Sample XY Plot (p. 4058) shows a sample XY plot.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.109: Sample XY Plot



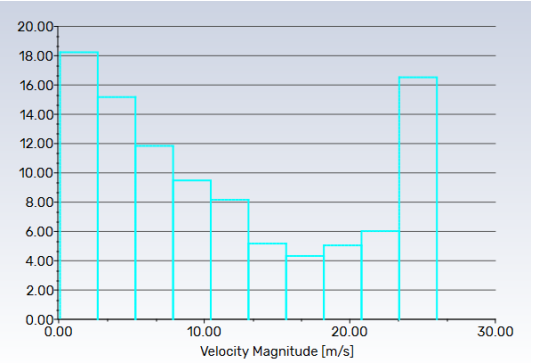
To differentiate the data being displayed, you can customize the pattern, color and weight of the data lines and the shape, color, and size of the data markers.

If you want to move or resize the legend, double-click it, then you can move and resize it using the left-mouse-button.

40.11.1.2. Histograms

A histogram plot is a bar chart of data. It is a representation of a frequency of distribution by means of rectangles of widths representing class intervals and with areas proportional to the corresponding frequencies. Figure 40.110: Sample Histogram (p. 4058) shows a sample histogram.

Figure 40.110: Sample Histogram



See Histogram Reports (p. 4158) for information about printing histogram reports. For more information on histogram plots, see Histograms (p. 4072).

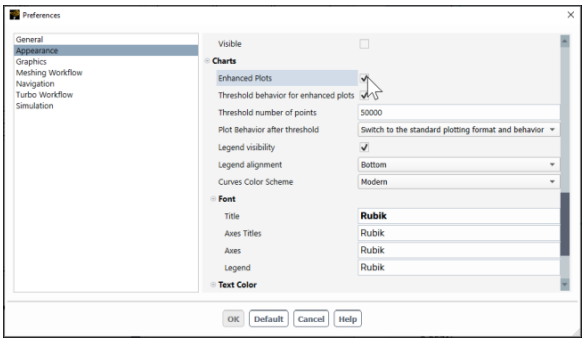
40.11.1.3. Enhanced Interactive Plots

Enhanced plotting, enabled by default, allows you to zoom in/out on 2D plots, enable/disable curves, quickly query values at various points, and so on. The status of this plotting method is the controlled using the **Enhanced Plots** option in Preferences.

File → Preferences...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.111: Enabling Enhanced Plots



Enhanced Plots provide the following interactions for 2D plots in the graphics window:

- Enable/disable curves by clicking the associated colored square in the legend.
- See values at any location along a plotted curve by hovering over the point of interest.

> **Note:**
>
> The values shown on mouse-hover of Residual and Report Plot curves are not interpolated.

- Zoom in/out on the plot using the mouse wheel or by using a box zoom with the left mouse button (drag from upper left to lower-right to zoom in and the opposite motion to zoom out).
- Pan the plot using **Ctrl** + **MMB** (middle mouse button) + drag.
- Fit the plot to the size of the graphics window by clicking the Fit to Window button ( ) or by **Ctrl** + **A**.
- Edit the properties of a curve by double-clicking it in the plot window. Here you can edit properties like line thickness, marker style and so on. *You can control the available colors using the **Curves Color Scheme** field in **Preferences** (**Appearance** branch).*

> **Note:**
>
> If you change the color scheme, the updated colors will only appear for new plots, so you will have to re-plot any existing plots to see the new color scheme applied.

- Resize and scroll through the legend when the number of entries exceeds the size of the plot, allowing you to review all of the entries.
- Edit axis properties by double-clicking the axis.
- Control the color and fonts of the plot title, axis title, axes, and legend. If you want to move or resize the legend, double-click it, then you can move and resize it using the left-mouse-button.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can control the default legend behavior, such as whether a legend is displayed and its location, using the ***Legend visibility*** *and* ***Legend Alignment*** *fields in* ***Preferences*** *(****Appearance*** *branch).*

- Specify a threshold for the total number of plot points, after which, the plot can be reverted to the standard plot behavior to preserve performance. *This is controlled by the* ***Threshold behavior for enhanced plots****,* ***Threshold number of points****, and* ***Plot behavior after threshold*** *fields in* ***Preferences*** *(****Appearance*** *branch).*

Limitations

- Annotations are not compatible with enhanced plots.
- Solution animations of 2D enhanced plots are not compatible with the HSF image file format.
- The **Writing System**s available within the **Fonts** dialog box in **Preferences** are not supported with enhanced plots.

40.11.2. XY Plots of Solution Data

You can produce a very sophisticated XY plot by using data from several zones, surfaces, or files and modifying the axis and curve attributes. Using the capability for loading external data files, you can create plots that compare your Ansys Fluent results with data from other sources. To get further information about the solution, you can investigate the frequency of distribution of the data using a histogram (see Histograms (p. 4072)).

40.11.2.1. Steps for Generating Solution XY Plots

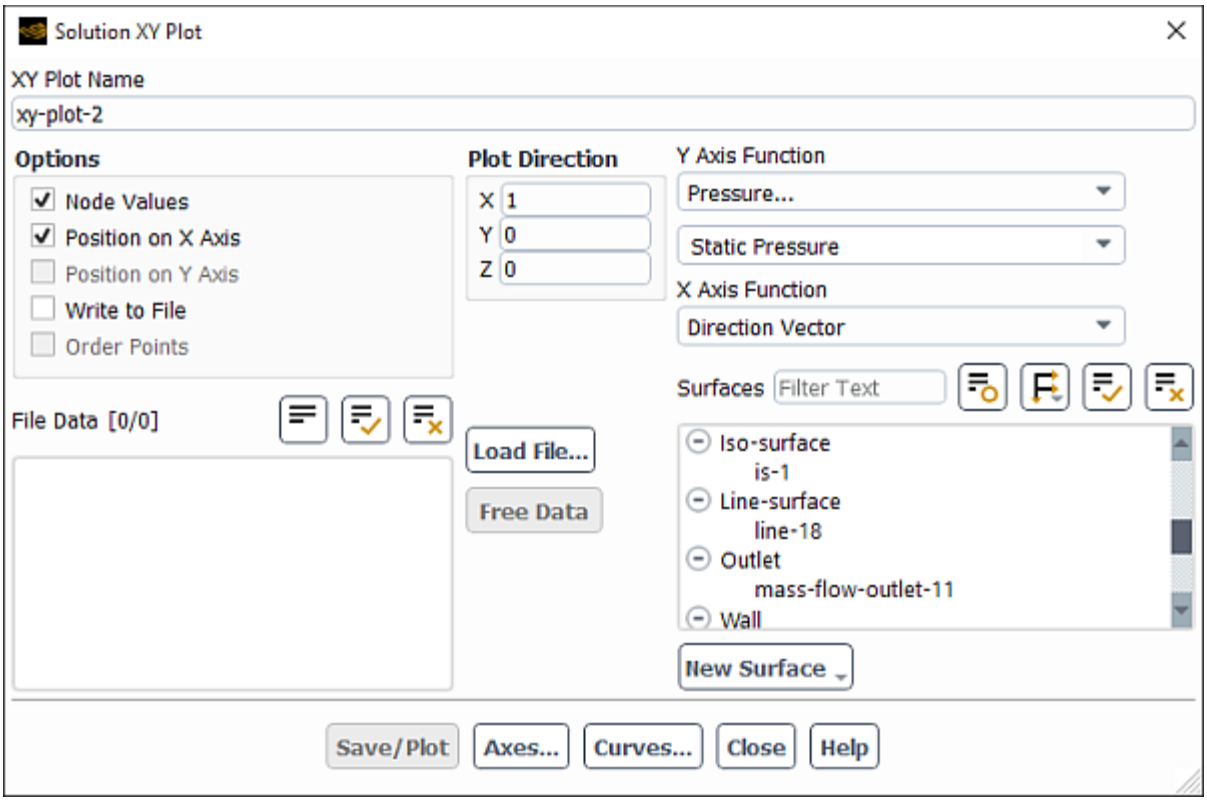
You can create an XY plot of solution data using the Solution XY Plot Dialog Box (p. 5290) (Figure 40.112: The Solution XY Plot Dialog Box (p. 4061)).

Named XY plot definitions can be saved for later use. See Creating and Using Contour Plot Definitions (p. 3910) for additional information on graphics object definitions.

Results → Plots → XY Plot → New...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.112: The Solution XY Plot Dialog Box



The basic steps for generating a solution XY plot are as follows:

1. Specify the variable(s) you are plotting:

   - To plot a variable on the *y*-axis as a function of position on the *x*-axis, enable the **Position on X Axis** option and choose the variable to be plotted on the *y*-axis in the **Y Axis Function** drop-down list. Select a category from the upper list and then choose the desired quantity in the lower list. For an explanation of the variables in the list, see Field Function Definitions (p. 4167).

   - To plot a variable on the *x*-axis as a function of position on the *y*-axis, enable the **Position on Y Axis** option and choose the variable to be plotted on the *x*-axis in the **X Axis Function** drop-down list.

   - To plot one variable as a function of another, turn off both the **Position on X Axis** and **Position on Y Axis** options and select the variables to be plotted in the **X Axis Function** and **Y Axis Function** drop-down lists.

2. Specify the plot direction:

   - To plot a variable as a function of position along a specified direction vector, select **Direction Vector** in the **X Axis Function** or **Y Axis Function** drop-down list (whichever is the position axis), and specify the components of the direction vector for plotting under **Plot Direction**. The position axis of the plot is indicated by the selection of **Position on X Axis** or **Position on Y Axis**. The positions plotted will have coordinate values that correspond to the dot product of the data coordinate vector with the plot direction vector. For example, if you are

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

plotting a variable at the pressure outlet of the geometry shown in Figure 40.113: Geometry Used for XY Plot (p. 4063), you would specify the **Plot Direction** vector (1,0,0) since you are interested in how the variable changes as a function of *x*. Figure 40.114: Data Plotted at Outlet Using a Plot Direction of (1,0,0) (p. 4063) shows the resulting XY plot. (If you specified (0,1,0) as the plot direction, all variable values would be plotted at the same position (see Figure 40.115: Data Plotted at Outlet Using a Plot Direction of (0,1,0) (p. 4063)), because the *y* value is the same at every point on the pressure outlet.)

- It is also possible to plot a variable as a function of position along the length of a specified curvilinear surface. The curvilinear surface must be piecewise linear and it cannot contain more than one closed curve, such as a complete circle. To plot a variable in this way, select **Curve Length** in the **X Axis Function** or **Y Axis Function** drop-down list (whichever is the position axis). Then specify the plot direction along the surface: to plot the variable along the direction of increasing curve length, select **Default** under **Plot Direction**; to plot the variable in the direction of decreasing surface length, select **Reverse**. To check the direction in which the variable will be plotted along a surface, select the surface in the **Surfaces** list and click **Show** under **Plot Direction**. Ansys Fluent will display the selected surface in the graphics window, marking the start of the surface with a blue dot and the end of the surface with a red dot. Ansys Fluent will also display arrows on the surface showing the direction in which the variable will be plotted.

  Note:

  When running the parallel version of Ansys Fluent, **Curve Length** is only supported when the curvilinear surface is contained within a single partition.

3. Choose the surface(s) on which to plot data in the **Surfaces** list. Note that if you are plotting a variable as a function of position along the length of a curvilinear surface, you can select only one surface in the **Surfaces** list.

4. Set any of the options described below, or modify the attributes of the axes or curves as described in Modifying Axis Attributes (p. 4074) and Modifying Curve Attributes (p. 4076).

5. Click **Plot** to generate the XY plot in the active graphics window.

If you want to modify XY plots with annotations (see Adding Text to the Graphics Window (p. 3961)) you must first disable enhanced plotting (Enhanced Interactive Plots (p. 4058)).

Note:

Once created, you can re-display solution XY plots by right-clicking the desired plot in the **Outline View** tree (**Results**>**Plots**>**XY Plot**><**plot-name**>) and selecting **Plot**, **Plot in** (to specify which window), or **Add to** (to add this plot to another 2D plot). *Note that if you are using the **Add to** option, it is your responsibility to make a meaningful plot by ensuring the plots have consistent units.*

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.113: Geometry Used for XY Plot

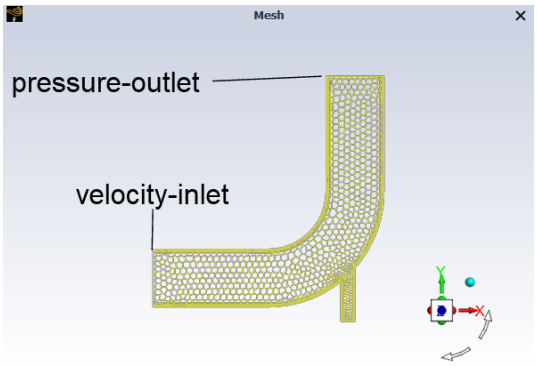


Figure 40.114: Data Plotted at Outlet Using a Plot Direction of (1,0,0)

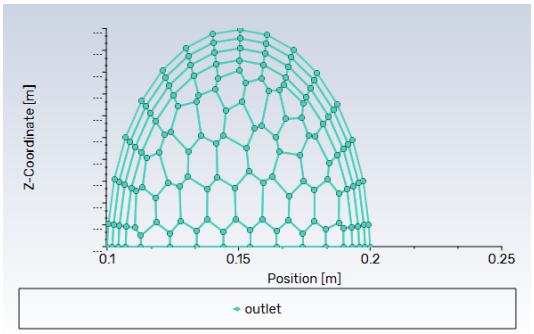
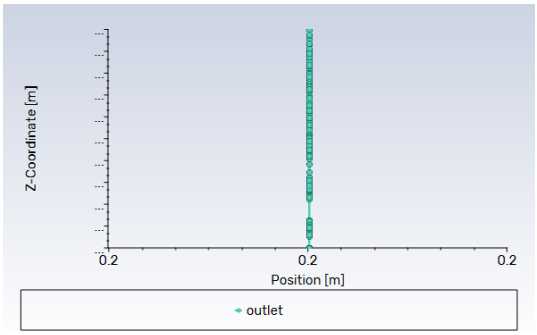


Figure 40.115: Data Plotted at Outlet Using a Plot Direction of (0,1,0)



40.11.2.2. Options for Solution XY Plots

The options mentioned in Steps for Generating Solution XY Plots (p. 4060) include the following.

- You can include data from an external file in the solution XY plot to compare your results with experimental data.
- You can also choose node or cell values to be plotted, and save the plot data to a file.

40.11.2.2.1. Including External Data in the Solution XY Plot

To add external data to your XY plot for comparison with your results, you must first ensure that any external data files are in the format described in XY Plot File Format (p. 4071). You can then load the file(s) by clicking on the **Load File...** button and specifying the file(s) to be read in The

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select File Dialog Box (p. 931). Once a file has been loaded, its title will appear in the **File Data** list. You can choose the data file(s) to be included in your plot from the titles in this list.

To remove a file from the **File Data** list, select it and then click **Free Data**.

40.11.2.2.2. Choosing Node or Cell Values

In Ansys Fluent you can choose to display the computed cell-center values or values that have been interpolated to the nodes, with boundary face values being displayed at boundaries in either case, when available. By default, the **Node Values** option is enabled and the interpolated values are displayed. If you prefer to display cell-centered values, disable the **Node Values** option. Node-averaged data curves may be somewhat smoother than curves for cell values. *If you want to display cell-center values instead of boundary face values, then you must set the* `plot/set-boundary-val-off` *to* `yes`*, and disable* ***Node Values****.*

For face-only functions (for example, **Wall Shear Stress**), the cell values that are displayed for boundary zone surfaces will actually be the face values. These face values are more accurate, as face-only functions are computed on the faces and not on the cells. For these face-only functions, the cell values on postprocessing surfaces will display the values in the cell. For more information about cell values, see Cell Values (p. 4167).

If you are displaying the XY plot to show the effect of a porous medium or 2D fan boundary, to depict a shock wave, or to show any other discontinuities or jumps in the plotted variable, you should use cell values; if you use node values in such cases, the discontinuity will be smeared by the node averaging for graphics and will not be shown clearly in the plot.

40.11.2.2.3. Saving the Plot Data to a File

Once you have generated an XY plot, you may want to save the plot data to a file. You can read this file into Ansys Fluent at a later time and plot it alone using the Plot Data Sources Dialog Box (p. 5297), as described in Creating an XY Plot From Multiple Data Sources (Including Files) (p. 4064), or add it to a plot of solution data, as described above.

To save the plot data to a file, enable the **Write to File** option in the Solution XY Plot Dialog Box (p. 5290). The **Plot** button will change to the **Write...** button. Clicking on the **Write...** button will invoke The Select File Dialog Box (p. 931), in which you can specify a name and save a file containing the plot data. The format of this file is described in XY Plot File Format (p. 4071).

To sort the saved plot data in order of ascending *x*-axis value, enable the **Order Points** option in the Solution XY Plot Dialog Box (p. 5290) before you click **Write...**. This option is available only when the **Write to File** option is enabled.

40.11.3. Creating an XY Plot From Multiple Data Sources (Including Files)

You can produce XY plots using data contained in external files and combine them with data from in-session XY plots and report plots. The Plot Data Sources Dialog Box (p. 5297) enables you to display data read from external files in an abscissa/ordinate plot form. The format of the plot file is described in XY Plot File Format (p. 4071).

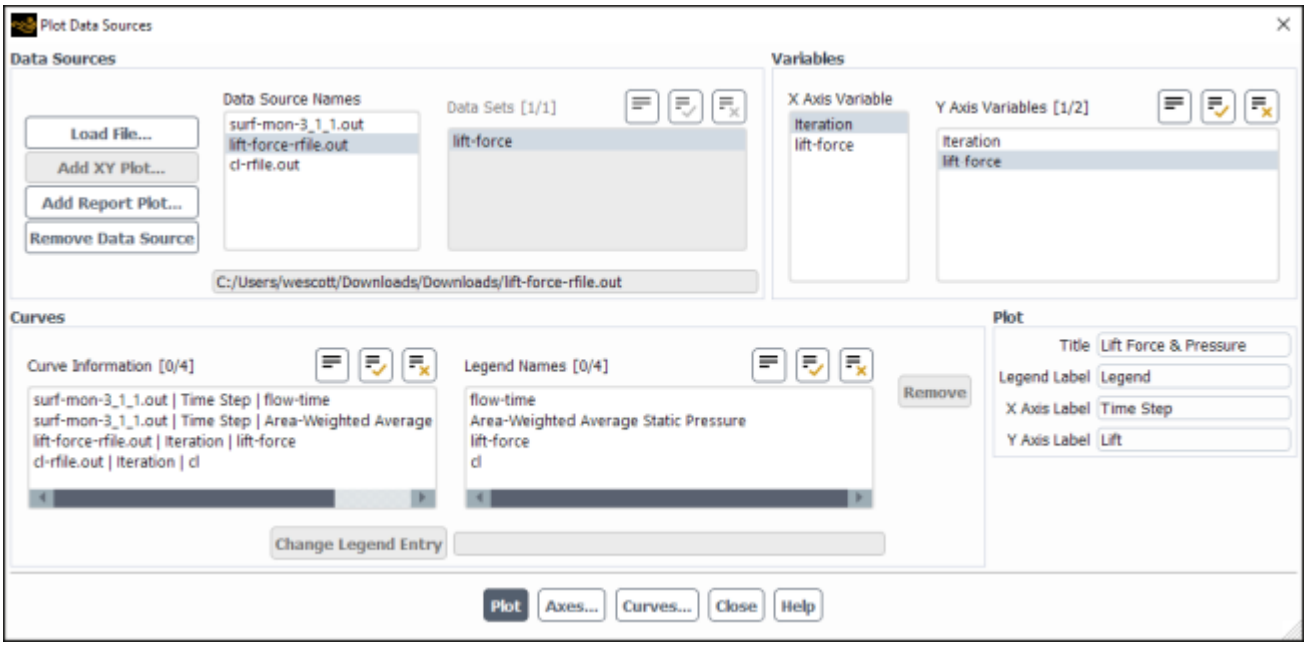
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

40.11.3.1. Steps for Generating XY Plots of Data from Multiple Sources

You can create an XY plot combining data from multiple sources, including in-session report plots and data sourced from one or more external files, using the Plot Data Sources Dialog Box (p. 5297) (Figure 40.116: The Plot Data Sources Dialog Box (p. 4065)).

Results → Plots → Data Sources

Figure 40.116: The Plot Data Sources Dialog Box



The steps for generating a file XY plot are as follows:

1. Select the files and/or plots that you want to display:

   - Click **Load...** to select the file(s) you want to plot. Note that report files and `.xy` files are supported (see XY Plot File Format (p. 4071) for information on the `.xy` file format).
   - Click **Add XY Plot...** to select one or more XY Plot objects from the list of available objects.
   - Click **Add Report Plot...** to select on or more report plots from the list of available plots.

   As soon as the files are loaded, the curves (plots) are all added to the **Curve Information** list.

   For each data source, you can control which data set(s) are included from that source and what variables should be assigned to which axis.

2. Review the **Curve Information** list to ensure that the desired data sets are used and verify that variables are assigned to the axes you intend.

   Selecting/de-selecting Y variables and Data Sets adds or removes them from the **Curve Information** list.

   Click **Remove** (to the right of **Legend Names**) to remove the selected curve(s) from the **Curve Information** list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Modify legend names as needed. To modify a legend name:

    1. Select a name in the **Legend Names** list.

    2. Enter the new name in the text entry box to the right of the **Change Legend Entry** button.

    3. Click **Change Legend Entry**.

4. You can control the appearance of the axes and curves by clicking **Axes...** or **Curves...**.

5. Provide names for the **Title**, **Legend Label**, **X Axis Label** and **Y Axis Label** in the **Plots** group box.

6. Click **Plot** to generate an XY plot of the data listed in the **Curve Information** list.

7. (Optional) Click **Remove Data Source** if you accidentally selected a plot that you did not intend or if you are done with a data set.

Note:

Plots created using the **Plot Data Sources** dialog box will always be displayed in SI units.

40.11.4. XY Plots of Profiles

Ansys Fluent enables two options for generating XY plots of data related to boundary profiles. Using the Plot Profile Data Dialog Box (p. 5299), you can plot the original data points from the profile file you have read into Ansys Fluent. Alternatively, you can plot the values assigned to the cell faces on the boundary after the profile file has been interpolated, by using the Plot Interpolated Data Dialog Box (p. 5300).

Important:

Note that you must have valid data when trying to use the profile plotting options.

For more information about boundary profiles, see Profiles (p. 1564).

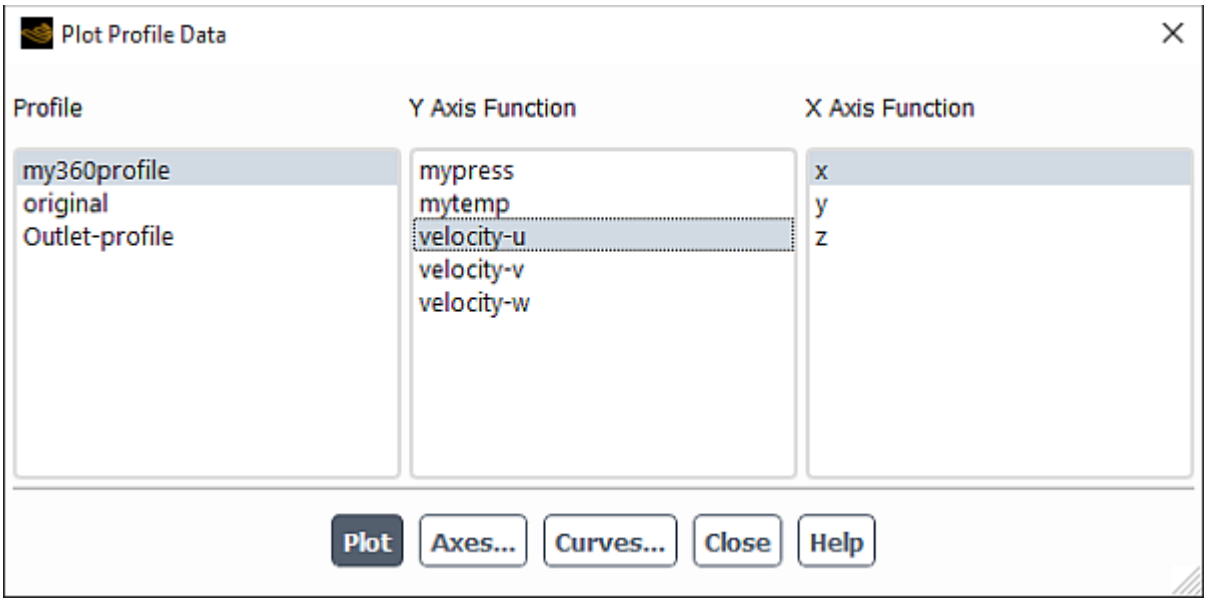
40.11.4.1. Steps for Generating Plots of Profile Data

Once you have read a profile file, it is available for plotting by using the Plot Profile Data Dialog Box (p. 5299).

Results → **Plots** → **Profile Data** → **Set Up...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.117: The Plot Profile Data Dialog Box



The procedure for generating an XY plot of the original profile data is as follows:

1. Select one of the profiles you have read from the **Profile** selection list.
2. Select a field of the profile from the **Y Axis Function** selection list.
3. Choose a variable against which you want to plot the field data, and select it from the **X Axis Function** selection list. The available variables will vary depending on the profile, and include **x**, **y**, **z**, **r**, and **time**.
4. Modify the attributes of the axes or curves as described in Modifying Axis Attributes (p. 4074) and Modifying Curve Attributes (p. 4076).
5. Click **Plot** to generate an XY plot of the profile field data.

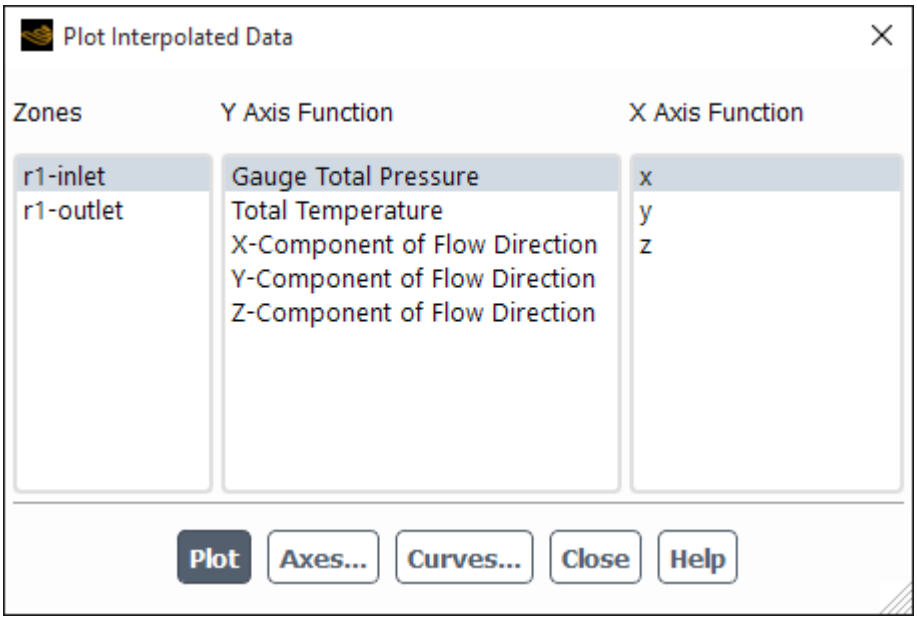
40.11.4.2. Steps for Generating Plots of Interpolated Profile Data

To interpolate a profile you must first read a profile file for the case, and select a profile field in a boundary conditions dialog box (for example, the Velocity Inlet Dialog Box (p. 5072)). After the flow solution has been initialized, the cell face values of the boundary zone can be plotted by using the Plot Interpolated Data Dialog Box (p. 5300).

Results → **Plots** → **Interpolated Data** → **Set Up...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.118: The Plot Interpolated Data Dialog Box



The procedure for generating an XY plot of the interpolated data is as follows:

1. Select a zone from the **Zones** selection list. Only the zones for which you have set a profile field as one or more of the parameters will be available in this list.

2. Select a profile-related parameter of the zone from the **Y Axis Function** selection list. The name of the parameter will be the same as that of the drop-down list in the boundary condition dialog box from which the profile field was selected.

3. Choose a variable against which you want to plot the field data, and select it from the **X Axis Function** selection list. The available variables are **x**, **y**, and (for 3D cases) **z**.

4. Modify the attributes of the axes or curves as described in Modifying Axis Attributes (p. 4074) and Modifying Curve Attributes (p. 4076).

5. Click **Plot** to generate an XY plot of the cell face values on the boundary.

40.11.5. XY Plots of Circumferential Averages

You can also generate a plot of circumferential averages in Ansys Fluent. This enables you to find the average value of a quantity at several different radial or axial positions in your model. Ansys Fluent computes the average of the quantity over a specified circumferential area using either area-weighted or mass-weighted averaging, and then plots the average against the radial or axial coordinate.

40.11.5.1. Steps for Generating an XY Plot of Circumferential Averages

You can generate an XY plot of circumferential averages in the radial direction using the `circum-avg-radial` text command:

`plot` → `circum-avg-radial`

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

or you can use the `circum-avg-axial` text command to generate an average in the axial direction:

`plot` → `circum-avg-axial`

The procedure for generating an XY plot of circumferential averages is as follows:

1. Specify the variable to be averaged by typing its name when Ansys Fluent prompts you for `averages of`. You can press **Enter** to see a list of available variables.

2. Choose the surface on which to plot data by typing its name when Ansys Fluent prompts you for `on surface`.

   Important:

   - Use the Mesh Display Dialog Box (p. 4688) to see a list of surfaces on which you can plot data. Pressing **Enter** will not show a list of available surfaces.
   - In 2D, use the interior volume to create the axial and radial bands and not an external zone surface. While this may have little impact on the results for area-weighted averaging, mass-weighted averaging would be 0 due to the absence of a flux on a constructed surface around the volume.

   ---

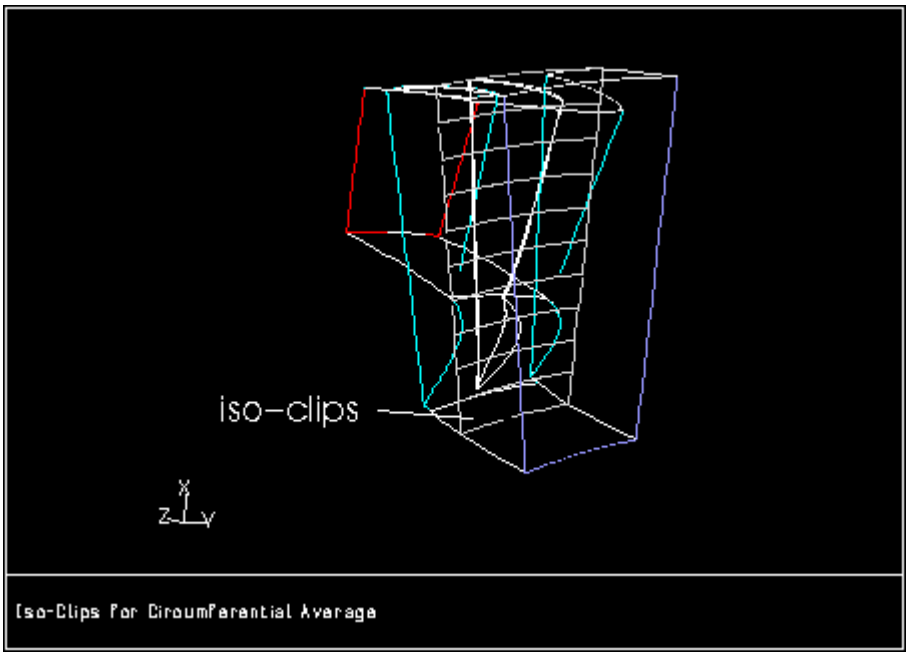
3. Specify the `number of bands` to be created. The default number of bands is 5.

   Ansys Fluent will create circumferential bands by isoclipping the specified surface into equal bands of radial or axial coordinate. An example of the iso-clips created is shown in Figure 40.119: Iso-Clips Created For Circumferential Averaging (p. 4070). The radial or axial coordinate is derived from the rotation axis of the **Reference Zone** specified in the Reference Values Task Page (p. 5154).

4. Specify whether Ansys Fluent uses area-weighted averaging (default) or mass-weighted averaging.

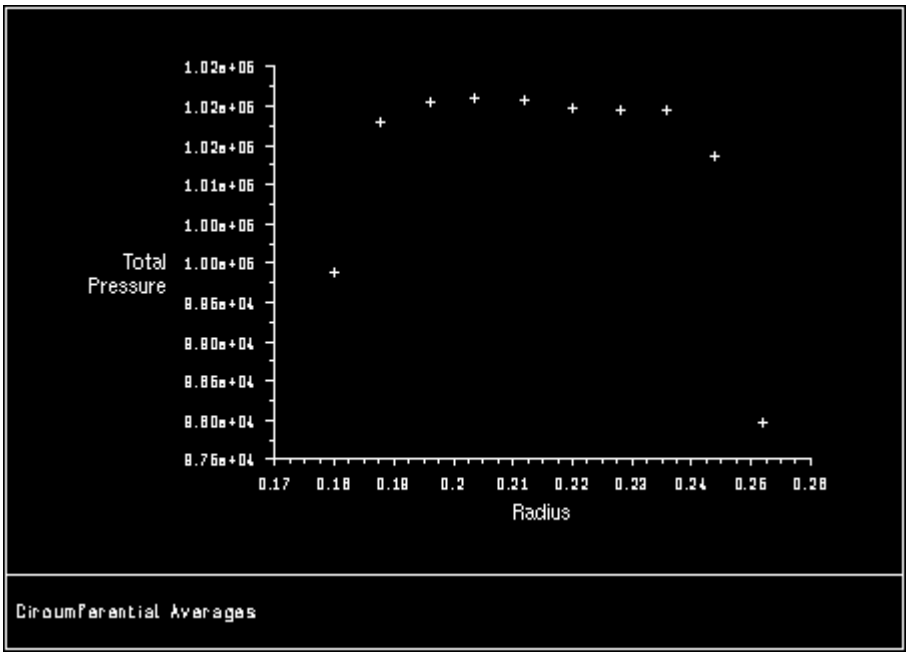
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.119: Iso-Clips Created For Circumferential Averaging



Ansys Fluent then computes the average of the variable for each band using the specified averaging method, as described in Computing Surface Integrals of the Theory Guide. Finally, it plots the average of the variable as a function of radial or axial coordinate. Figure 40.120: XY Plot of Circumferential Averages (p. 4070) shows an example of an XY plot of circumferential averages using radial coordinates.

Figure 40.120: XY Plot of Circumferential Averages



When the circumferential average plot is generated, Ansys Fluent also creates a new surface called `radial-bands` or `axial-bands`, which contains the iso-clips described above (see Figure 40.119: Iso-Clips Created For Circumferential Averaging (p. 4070)). You can use this surface to

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

generate other XY plots. For more information on the creation and manipulation of surfaces, see Creating Surfaces and Cell Registers for Displaying and Reporting Data (p. 3813).

40.11.5.2. Customizing the Appearance of the Plot

If you want to customize the appearance of the axes or curves in a circumferential average plot, you can save the plot data to a file (using the `plot-to-file` text command, as described in XY Plot File Format (p. 4071)), read the file into Ansys Fluent and plot it again (using the Plot Data Sources Dialog Box (p. 5297), as described in Creating an XY Plot From Multiple Data Sources (Including Files) (p. 4064)), and then use the Axes Dialog Box (p. 5313) and Curves Dialog Box (p. 5316) (as described in Modifying Axis Attributes (p. 4074) and Modifying Curve Attributes (p. 4076)) to modify the appearance of the plot.

To save the plot data to a file, first use the `plot-to-file` text command to specify the name of the file.

`plot` → `file-set` → `plot-to-file`

Then generate the circumferential average XY plot as described above. Ansys Fluent will display the plot in the graphics window, and also save the plot data to the specified file.

40.11.6. XY Plot File Format

The XY file format read or written by Ansys Fluent includes the following information:

- The title of the plot
- The label for the abscissa and the ordinate
- Cortex variables and pairs of abscissa/ordinate data for each curve in the plot

The following sample file illustrates the XY file format:

```
(title "Velocity Magnitude")
(labels "Position" "Velocity Magnitude")

((xy/key/label "pressure-inlet-8")
  (xy/key/visible? #t)
  (xy/line/pattern "--")
  0.0000 230.097 0.
  0625 160.551
  0.1250 149.205
  ...
  0.5000 183.007
)
```

Similar to the case file format, parentheses bound the various pieces of information in the formatted, ASCII file. The title (`title " "`) and labels (`labels " "`) must be first in the file, then each curve has information in the form (`(cxvar value) x y x y x y...`), where there may be zero or more Cortex variables defined for each curve.

You do not have to include Cortex variables to import your XY data. For example, you may want to import experimental data to compare with the Ansys Fluent solution. The following example would use the default Cortex variables in the code to define the data. After you import the file into Ansys

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fluent, you could then use the Axes Dialog Box (p. 5313) and the Curves Dialog Box (p. 5316) to customize the XY plot, as described in Modifying Axis Attributes (p. 4074) and Modifying Curve Attributes (p. 4076).

```
(title "Experiment, Run 11")
(labels "X, m" "Cp")
( 0 1.5
  1.5 1.3
  3.2 1.5
  5.1 1.2
)
```

40.11.7. Residual Plots

Residual history can be displayed using an XY plot. The abscissa of the plot corresponds to the number of iterations and the ordinate corresponds to the log-scaled residual values.

To plot the current residual history, click **Plot** in the Residual Monitors Dialog Box (p. 5599).

Solution → **Monitors** → **Residuals** **Edit...**

For additional information about using the Residual Monitors Dialog Box (p. 5599) to plot residuals, see Printing and Plotting Residuals (p. 3689).

40.11.8. Histograms

Histograms can be displayed in a graphics window using a bar chart (or printed in the console window, as described in Histogram Reports (p. 4158)). The abscissa of the chart is the desired solution quantity and the ordinate is the percentage of the total number of cells.

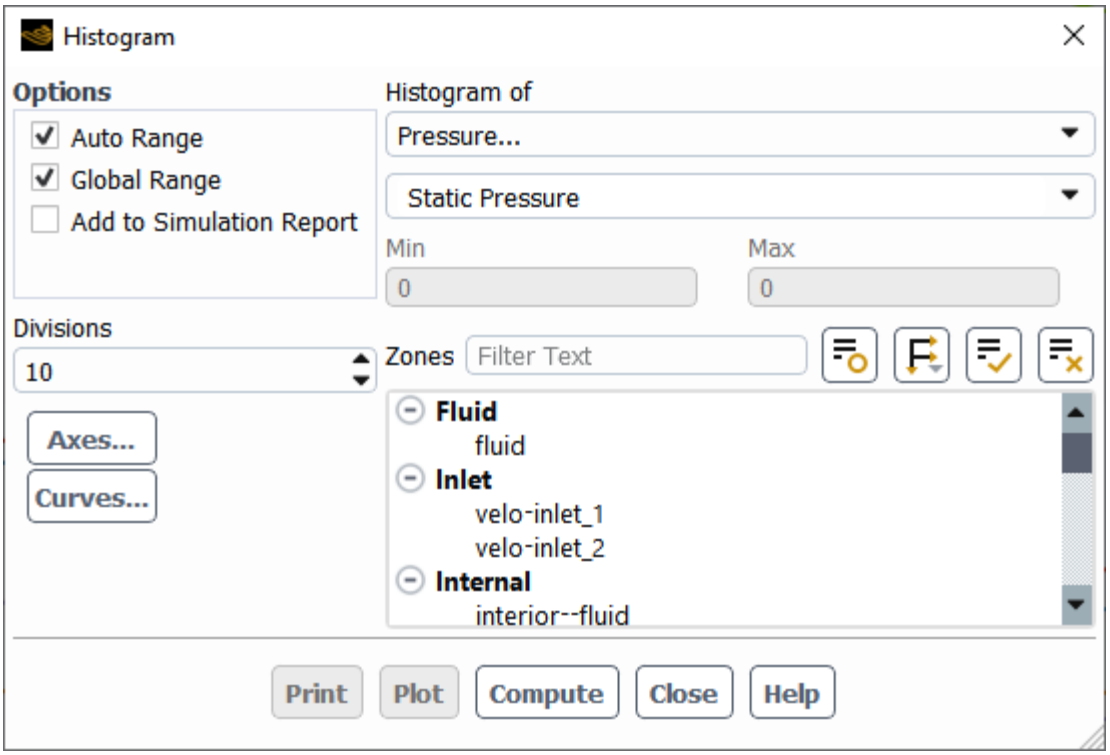
40.11.8.1. Steps for Generating Histogram Plots

You can create a histogram plot of solution data using the Histogram Dialog Box (p. 5295) (Figure 40.121: The Histogram Dialog Box (p. 4073)).

Results → **Plots** → **Histogram** **Edit...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.121: The Histogram Dialog Box



The steps for generating a histogram plot are as follows:

1. Choose the scalar quantity to be plotted in the **Histogram of** drop-down list. Select a category in the upper list and then select the desired quantity in the lower list. (See Field Function Definitions (p. 4167) for an explanation of the variables in the list.)

2. Set the number of data intervals that will be plotted in the histogram in the **Divisions** field. By default there will be 10 intervals ("bars") in the histogram plot to finer intervals, increase the number of **Divisions**. You may want to click **Compute** to update the **Min** and **Max** fields when you are trying to decide how many divisions to plot.

3. Select the face or cell zone under **Zones** for which you want results plotted or printed. If all zones are selected, then the entire domain will be plotted. You can also plot histograms based on the selected **Zone Types**.

4. Set the option described below, if desired, or modify the attributes of the axes or curves as described in Modifying Axis Attributes (p. 4074) and Modifying Curve Attributes (p. 4076).

5. Click **Plot** to generate the histogram plot in the active graphics window.

6. Click **Print** to print out your histogram results on individual zones, or the entire domain. Similarly, you can click **Compute** to calculate your histogram results on individual zones, or the entire domain.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

40.11.8.2. Options for Histogram Plots

In addition to the axis and curve attribute controls mentioned in the procedure above, the only other options for histogram plotting are the ability to specify a subrange of values to be plotted, and the ability to add histograms to your simulation reports.

40.11.8.2.1. Specifying the Range of Values Plotted

By default, the range of values included in the histogram plot is automatically set to the range of values in the entire domain for the selected variable. If you want to focus in on a smaller range of values, you can restrict the range to be displayed.

To manually set the range of values, turn off the **Auto Range** option in the Histogram Dialog Box (p. 5295). The **Min** and **Max** fields will become editable, and you can enter the new range of values to be plotted. To show the default range at any time, click **Compute** and the **Min** and **Max** fields will be updated.

You can also choose to base the minimum and maximum values on the range of values on the selected surfaces, rather than in the entire domain. To do this, turn off the **Global Range** option in the Histogram Dialog Box (p. 5295). The **Min** and **Max** values will be updated when you next click **Compute**.

40.11.8.2.2. Adding Histograms to Your Simulation Reports

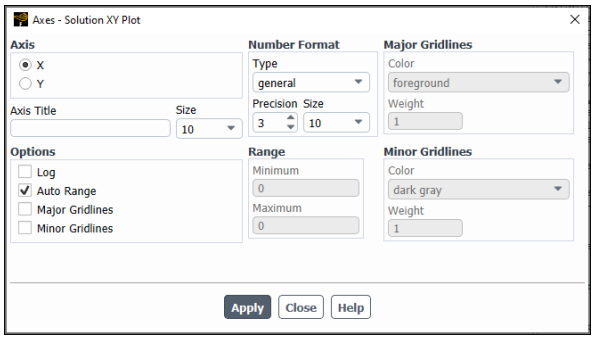
By default, histograms and their data are not automatically included in simulation reports. To include histograms in your simulation reports, select the **Add to Simulation Report** option. Once you have made any additional changes in the dialog box, you can click the **Add to Report** button.

For more information, see Using Simulation Reports (p. 4359).

40.11.9. Modifying Axis Attributes

You can modify the appearance of the XY and parameters that control the labels, scale, range, numbers, and major and minor rules. For each type of plot (solution XY, file XY, profile, residual, histogram, and so on), you can set different axis parameters in the Axes Dialog Box (p. 5313) (Figure 40.122: The Axes Dialog Box (p. 4074)). Note that the title following **Axes** in the dialog box indicates which plot environment you are changing (for example, the **Axes - Solution XY Plot** dialog box controls parameters for solution XY plots).

Figure 40.122: The Axes Dialog Box



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To open the Axes Dialog Box (p. 5313) for a particular plot type, click **Axes...** in the appropriate dialog box (for example, the **Solution XY Plot**, **File XY Plot**, **Plot Profile Data**, **Plot Interpolated Data**, or **Residual Monitors** dialog box).

40.11.9.1. Using the Axes Dialog Box

The Axes Dialog Box (p. 5313) enables you to independently control the characteristics of the ordinate (*y*-axis) and abscissa (*x*-axis) on an XY plot or parameters for one axis or the other, by following the procedure below:

1. Choose the axis for which you want to modify the attributes by selecting **X** or **Y** under **Axis**.
2. Set the desired parameters.
3. Click **Apply** and then choose the other axis and repeat the steps, if desired.

The changes to the axis attributes will appear in the graphics window the next time you generate a plot.

40.11.9.1.1. Changing the Axis Title and Font Size

If you want to modify the title for the axis, you can do so by editing the **Axis Title** text field in the Axes Dialog Box (p. 5313). The title font size is controlled by the **Size** drop-down list located next to the **Axis Title** field.

40.11.9.1.2. Changing the Format of the Data Labels

You can change the format of the labels that define the primary data divisions on the axes using the controls under the **Number Format** heading in the Axes Dialog Box (p. 5313).

- To display the real value with an integral and fractional part (for example, 1.0000), select **float** in the **Type** drop-down list. You can set the number of digits in the fractional part by changing the value of **Precision**.
- To display the real value with a mantissa and exponent (for example, 1.0e-02), select **exponential** in the **Type** drop-down list. You can define the number of digits in the fractional part of the mantissa in the **Precision** field.
- To display the real value with either float or exponential form, depending on the size of the number and the defined **Precision**, choose **general** in the **Type** drop-down list.
- Use the **Size** field to control the size of the font used in displaying the numbers on the axis.

40.11.9.1.3. Choosing Logarithmic or Decimal Scaling

By default, decimal scaling is used for both axes (except for the y-axis in residual plots, which uses a log scale). If you want to change to a logarithmic scale, enable the **Log** option in the Axes Dialog Box (p. 5313). To return to a decimal scale, turn off the **Log** option. Note that when you are using the logarithmic scale, the **Range** values are the exponents; to specify a logarithmic range from 1 to 10000, for example, you will specify a minimum value of 1 and a maximum value of 4.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

40.11.9.1.4. Resetting the Range of the Axis

By default, the extents of the axis will range from the minimum value plotted to the maximum value plotted. If you want to change the range or extents of the axis, you can do so by turning off the **Auto Range** option in the Axes Dialog Box (p. 5313) and setting the new **Minimum** and **Maximum** values for the **Range**. This feature is useful when you are generating a series of plots and you want the extents of one or both of the axes to be the same, even if the range of plotted values differs. For example, if you are generating plots of temperature on several different wall zones, you might want the minimum and maximum temperature on the y-axis to be the same in every plot so that you can easily compare one plot with another. You would determine a temperature range that includes the temperatures on all walls, and use that as the range for the y-axis in each plot.

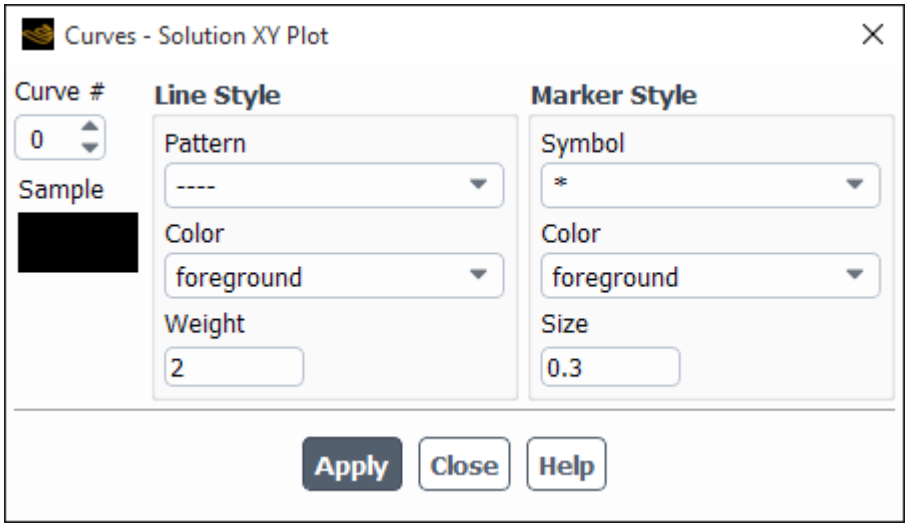
40.11.9.1.5. Controlling the Major and Minor Rules

Ansys Fluent enables you to display major and/or minor rules on the axes. Major and minor rules are the horizontal or vertical lines that mark, respectively, the primary and secondary data divisions and span the whole plot window to produce a "mesh." To add major or minor rules to the plot, enable the **Major Rules** or **Minor Rules** option. You can then specify a color and weight for each type of rule. Under the **Major Rules** or **Minor Rules** heading, select the desired color for the lines in the **Color** drop-down list and specify the line thickness in the **Weight** field. A line of weight 1.0 is normally 1 pixel wide. A weight of 2.0 would make the line twice as thick (that is, 2 pixels wide).

40.11.10. Modifying Curve Attributes

The data curves in XY plots and histograms can be represented by any combination of lines and markers. You can modify the attributes of the curves, including the patterns, weights, and colors of the lines, and the symbols, sizes, and colors of the markers. For each type of plot (solution XY, file XY, profile, residual, and so on), you can set different curve parameters in the Curves Dialog Box (p. 5316) (Figure 40.123: The Curves Dialog Box (p. 4076)). Note that the title following **Curves** in the dialog box indicates which plot environment you are changing (for example, the **Curves - Solution XY Plot** dialog box controls curve parameters for solution XY plots).

Figure 40.123: The Curves Dialog Box



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To open the Curves Dialog Box (p. 5316) for a particular plot type, click **Curves...** in the appropriate dialog box (for example, **Solution XY Plot**, **File XY Plot**, **Plot Profile Data**, **Plot Interpolated Data**, or **Residual Monitors** dialog box).

40.11.10.1. Using the Curves Dialog Box

The Curves Dialog Box (p. 5316) enables you to independently control the characteristics of each data curve in an XY plot or parameters for a curve, you will follow the procedure below:

1. Specify the curve for which you want to modify the attributes by increasing or decreasing the **Curve #** counter. The curves are numbered sequentially, starting from 0. For example, if you were plotting flow-field values on two surfaces, the first surface would be curve 0, and the second, curve 1. If the plot contains only one curve, the **Curve #** is set to 0 and is not editable.

2. Set the desired line and/or marker parameters as described below.

3. Click **Apply** and then choose another **Curve #** and repeat the steps, if desired.

Your changes to the curve attributes will appear in the graphics window the next time you generate a plot.

40.11.10.1.1. Changing the Line Style

You can control the pattern, color, and weight of the line using the controls under the **Line Style** heading:

- To set the line pattern for the curve, choose one of the items in the **Pattern** drop-down list. Except for **center** and **phantom** lines, the list displays examples of the pattern choices. A **center** line alternates a very long dash and a short dash and a **phantom** line alternates a very long dash and a double short dash. Note that selecting the second item in the drop-down list, represented by 4 short dashes, will result in a solid-line curve.

  Important:

  If you do not want the data points to be connected by any type of line (that is, if you plan to use just markers), select the "blank" choice, which is the first item in the **Pattern** list.

- To set the color of the line, pick one of the choices in the **Color** drop-down list.

- To define the line thickness, set the value of **Weight**. A line weight of 1.0 is normally 1 pixel wide. Therefore, a weight of 2.0 would make the line twice as thick (that is, 2 pixels wide).

40.11.10.1.2. Changing the Marker Style

You can control the symbol, color, and size for the data marker using the controls under the **Marker Style** heading:

- To set the symbol used to mark data, choose one of the items in the **Symbol** drop-down list. The list displays examples of the symbol choices. For example, in plotting pressure-coefficient data on the upper and lower surfaces of an airfoil, the symbol /*\ (filled-in upward-pointing

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

triangle) could be used for the marker representing the upper surface data, and the symbol */ (filled-in downward-pointing triangle) could be used for the marker representing the lower surface data.

Important:

If you do not want the data points to be represented by markers (that is, if you plan to use just a line connecting the data points), select the "blank" choice, which is the first item in the **Style** list.

- To set the color of the marker, pick one of the choices in the **Color** drop-down list.
- To define the size of the data marker, set the value of **Size**. A symbol of size 1.0 is 3.0% of the height of the display screen, except for the "." symbol, which is always one pixel.

40.11.10.1.3. Previewing the Curve Style

To see what a particular setting will look like in the plot, you can preview it in the **Sample** window of the Curves Dialog Box (p. 5316). A single marker and/or line will be shown with the specified style attributes.

40.12. Fast Fourier Transform (FFT) Postprocessing

When interpreting time-sequence data from a transient solution, it is often useful to look at the data's spectral (frequency) attributes. For instance, you may want to determine the major vortex-shedding frequency from the time-history of the drag force on a body recorded during an Ansys Fluent simulation. Or, you may want to compute the spectral distribution of static pressure data recorded at a particular location on a body surface. Similarly, you may need to compute the spectral distribution of turbulent kinetic energy using data for fluctuating velocity components. To interpret some of these time dependent data, you need to perform Fourier transform analysis. In essence, the Fourier transform enables you to take any time dependent data and resolve it into an equivalent summation of sine and cosine waves.

Ansys Fluent enables you to analyze your time dependent data using the Fast Fourier Transform (FFT) algorithm. Information on using the FFT algorithm in Ansys Fluent is provided in the following sections:

40.12.1. Limitations of the FFT Algorithm
40.12.2. Windowing
40.12.3. Fast Fourier Transform (FFT)
40.12.4. Using the FFT Utility

40.12.1. Limitations of the FFT Algorithm

The following limitations apply to Ansys Fluent's FFT module:

- The Ansys Fluent FFT module can only read inputs files in the following formats: Ansys Fluent report file, monitor, x-y file, Krichhoff model signal (`*.ark`), and Ffowcs Williams and Hawkings model signal (`*.ard`).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The Ansys Fluent FFT module assumes that the input data have been sampled at equal intervals and are consecutive (in the order of increasing time).
- The lowest frequency that the FFT module can pick up is given by $1/t$, where t is the total sampling time. If the sampled sequence contains frequencies lower than this, these frequencies will be aliased into higher frequencies.
- The highest frequency that the FFT module can pick up is $1/\left(2dt\right)$, where dt is the sampling interval (or time step).

40.12.2. Windowing

The discrete FFT algorithm is based on the assumption that the time-sequence data passed to the FFT corresponds to a single period of a periodically repeating signal. Since, in most situations, the first and the last data points will not coincide, the repeating signal implied in the assumption can often have a large discontinuity. The large discontinuity produces high-frequency components in the resulting Fourier modes, causing an aliasing error. You can condition the input signal before the transform by "windowing" it, in order to avoid this problem.

Suppose that we have N consecutive discrete (time-sequence) data sampled with a constant interval, Δt:

$$\phi_k \equiv \phi\left(t_k\right), \quad t_k \equiv k\Delta t, \qquad k=0,1,2,\ldots,\left(N-1\right) \tag{40.2}$$

Windowing is done by multiplying the original input data (ϕ_j) by a window function, W_j:

$$\tilde{\phi}_j = \phi_j W_j \qquad j=0,1,2,\ldots,\left(N-1\right) \tag{40.3}$$

Ansys Fluent offers four different window functions:

Hamming's window:

$$W_j = \begin{cases} 0.54-0.46\cos\left(\frac{8\pi j}{N}\right) & j \le \frac{N}{8},\ j \ge \frac{7N}{8} \\ 1 & \frac{N}{8} < j < \frac{7N}{8} \end{cases} \tag{40.4}$$

Hanning's window:

$$W_j = \begin{cases} 0.5\left[1-\cos\left(\frac{8\pi j}{N}\right)\right] & j \le \frac{N}{8},\ j \ge \frac{7N}{8} \\ 1 & \frac{N}{8} < j < \frac{7N}{8} \end{cases} \tag{40.5}$$

Bartlett's window:

$$W_j = \begin{cases} \frac{8j}{N} & j \le \frac{N}{8} \\ 8\left(1-\frac{j}{N}\right) & j \ge \frac{7N}{8} \\ 1 & \frac{N}{8} < j < \frac{7N}{8} \end{cases} \tag{40.6}$$

Blackman's window:

$$W_j = \begin{cases} 0.42-0.5\cos\left(\frac{8\pi j}{N}\right)+0.08\cos\left(\frac{16\pi j}{N}\right) & j \le \frac{N}{8},\ j \ge \frac{7N}{8} \\ 1 & \frac{N}{8} < j < \frac{7N}{8} \end{cases} \tag{40.7}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

These window functions preserve a large fraction $(3/4)$ of the original data, affecting only $1/4$ of the data on both ends.

40.12.3. Fast Fourier Transform (FFT)

The Fourier transform utility in Ansys Fluent enables you to compute the Fourier transform of a signal, $\phi(t)$, a real-valued function, from a finite number of its sampled points.

For a periodic set of N sampled points, ϕ_k, the discrete Fourier transform [21] (p. 5714) expresses the signal as a finite trigonometric series:

$$\phi_k = \sum_{n=0}^{N-1} \hat{\phi}_n e^{2\pi i k n/N} \qquad k = 0,1,2,\ldots(N-1) \tag{40.8}$$

where the series coefficients $\hat{\phi}_n$ are computed as

$$\hat{\phi}_n = \frac{1}{N} \sum_{k=0}^{N-1} \phi_k e^{-2\pi i k n/N} \qquad n = 0,1,2,\ldots(N-1) \tag{40.9}$$

Equation 40.8 (p. 4080) and Equation 40.9 (p. 4080) form a Fourier transform pair that enables us to determine one from the other.

Note that when we follow the convention of varying n from 0 to $N-1$ in Equation 40.8 (p. 4080) and Equation 40.9 (p. 4080) instead of from $-N/2$ to $N/2$, the range of index $1 \le n \le N/2-1$ corresponds to positive frequencies, and the range of index $N/2+1 \le n \le N-1$ corresponds to negative frequencies. $n=0$ still corresponds to zero frequency.

For the actual calculation of the transforms, Ansys Fluent adopts the so-called fast Fourier transform (FFT) algorithm, which significantly reduces operation counts in comparison to the direct transform. Furthermore, unlike most FFT algorithms in which the number of data should be a power of 2, the FFT utility in Ansys Fluent employs a prime-factor algorithm [161] (p. 5722). The number of data points permissible in the prime-factor FFT algorithm is any products of mutually prime factors from the set $2,3,4,5,7,8,9,11,13,16$, with a maximum value of $720720 = 5\times7\times9\times11\times13\times16$. Thus, the prime-factor FFT preserves the original data better than the conventional FFT.

Just prior to computing the transform, Ansys Fluent determines the largest permissible number of data points based on the prime factors, discarding the rest of the data. A list of these numbers is provided in Table 40.2: Numbers of Data Points Supported by the Prime-Factor FFT Algorithm (p. 4080).

Table 40.2: Numbers of Data Points Supported by the Prime-Factor FFT Algorithm

2	70	220	572	1386	3120	8580	34320
4	72	234	616	1430	3276	9240	36036
6	78	240	624	1456	3432	9360	40040
8	80	252	630	1540	3640	10010	48048
10	84	260	660	1560	3696	10296	51480
12	88	264	720	1584	3960	10920	55440
14	90	280	728	1638	4004	11088	60060
16	104	286	770	1680	4290	11440	65520

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

18	110	308	780	1716	4368	12012	72072
20	112	312	792	1820	4620	12870	80080
22	120	330	840	1848	4680	13104	90090
24	126	336	858	1872	5040	13860	102960
26	130	360	880	1980	5148	16016	120120
28	132	364	910	2002	5460	16380	144144
30	140	390	924	2184	5544	17160	180180
36	144	396	936	2288	5720	18018	240240
39	154	420	990	2310	6006	18480	360360
40	156	440	1008	2340	6160	20020	720720
42	168	462	1040	2520	6552	20592	
44	176	468	1092	2574	6864	21840	
48	180	504	1144	2640	6930	24024	
52	182	520	1170	2730	7280	25740	
56	198	528	1232	2772	7920	27720	
60	208	546	1260	2860	8008	30030	
66	210	560	1320	3080	8190	32760	

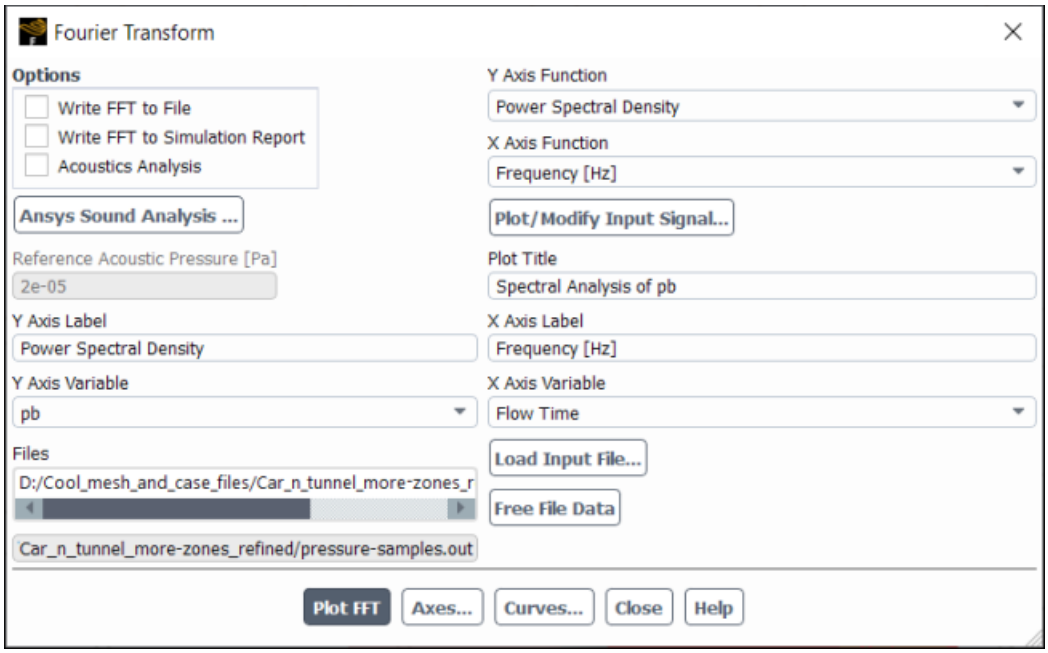
40.12.4. Using the FFT Utility

The Ansys Fluent FFT utility is available through the Fourier Transform Dialog Box (p. 5302) (Figure 40.124: The Fourier Transform Dialog Box (p. 4082)).

Results → **Plots** → **FFT** → **Set Up...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.124: The Fourier Transform Dialog Box



Note:

The threshold behavior of reverting to standard plotting from enhanced interactive plotting (Enhanced Interactive Plots (p. 4058)) does not work for Fast Fourier Transform (FFT) plots.

40.12.4.1. Loading Data for Spectral Analysis

FFT analysis requires an input signal data file consisting of time-sequence data. To load an input signal data file into the Fourier Transform Dialog Box (p. 5302), click **Load Input File...**. This displays The Select File Dialog Box (p. 931) where you can browse through your file directories and locate your data file containing your time-sequence data. To remove a file from the **Files** list, select it and then click **Free File Data**.

If you computed acoustic signals "on the fly", you have the option of processing signal data from a file or processing receiver data stored in memory. To analyze signal data from an existing input file, select **Process File Data** under **Process Options** and proceed as described above. To analyze receiver data stored in memory, select **Process Receiver** under **Process Options** and select the appropriate receiver in the **Receiver** list. If your model results include receiver signals for both the Ffowcs Williams – Hawkings model and the wave equation-based Kirchhoff model, the signals will be taken for the model which is currently active in your Fluent session.

Click **Plot FFT** to display the spectral analysis data and, if you have enabled **Acoustics Analysis**, to calculate the overall sound pressure level in dB based on the **Reference Acoustic Pressure** and display it in the console.

Note:

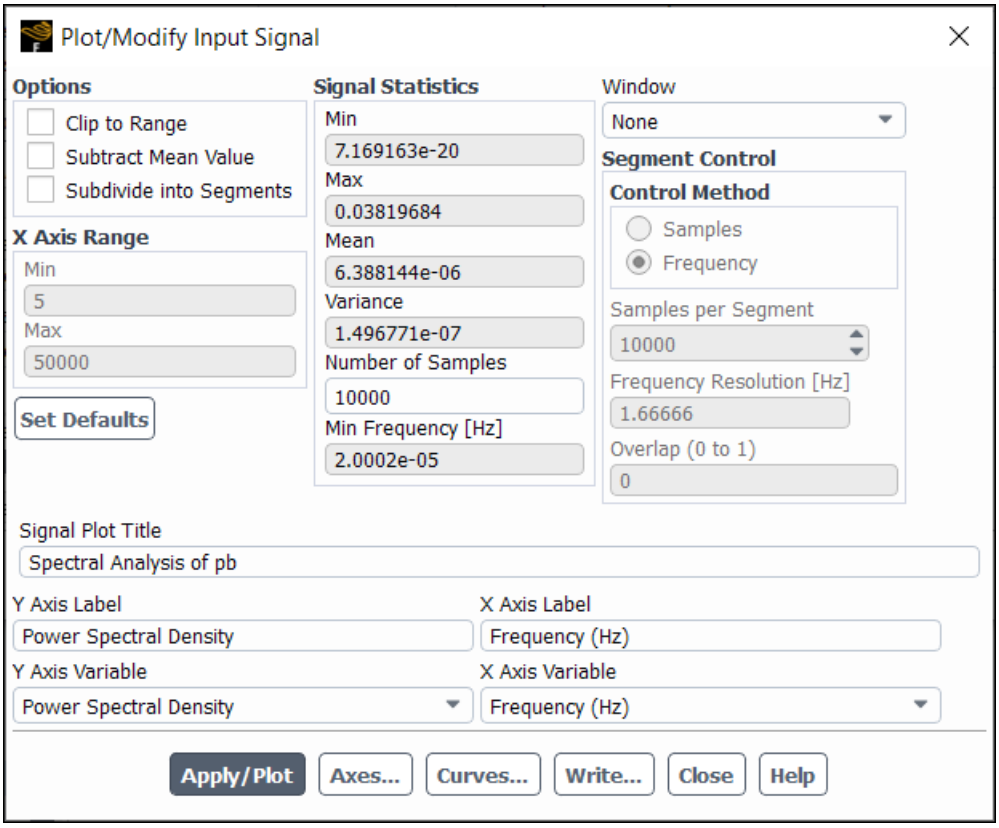
You can specify the y axis and x axis variables if your input report file contains multiple report definitions.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

40.12.4.2. Customizing the Input and Defining the Spectrum Smoothing

With the input signal data file loaded into the Fourier Transform Dialog Box (p. 5302), you may want to view a plot of the input signal and/or customize the data set in preparation for applying the FFT algorithm. To do so, click the **Plot/Modify Input Signal** button to open the Plot/Modify Input Signal Dialog Box (p. 5309) (Figure 40.125: The Plot/Modify Input Signal Dialog Box (p. 4083)).

Figure 40.125: The Plot/Modify Input Signal Dialog Box



The Plot/Modify Input Signal Dialog Box (p. 5309) allows you to analyze a portion of the input signal, view input **Signal Statistics**—**Min**, **Max**, **Mean**, **Variance**, total **Number of Samples**, and **Min Frequency** (that is, the finest possible frequency resolution)—and set title and label information for the input signal plot. Additionally, this dialog box allows you to enable and control the smoothing of the resulting spectrum by subdividing the input signal into multiple segments and averaging the segment-based spectra.

Note:

You can specify the y axis and x axis variables if your input report file contains multiple report definitions.

40.12.4.2.1. Customizing the Input Signal Data Set

By default, the entire data set is analyzed. To analyze a portion of the input signal, enable the **Clip to Range** option and specify the data range by entering **Min** and **Max** values under **X Axis Range**. To have the y-axis quantities reduced by the **Mean** value of the relevant signal property, enable the **Subtract Mean Value** option.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Set Defaults** button will reset the original values for the **Min** and **Max** fields under **X Axis Range** and turn off the **Clip to Range** option.

40.12.4.2.2. Spectrum Smoothing Through Signal Segmentation

For long broadband noise signals, the computed Fourier spectrum may display spurious fluctuations of amplitudes between the neighboring modes. In order to obtain a smooth spectrum, you can split the signal into multiple overlapping segments so that Ansys Fluent can apply the FFT algorithm on each segment and then average the resulting spectra. This procedure allows you to significantly suppress the spurious fluctuations at the cost of coarsening the frequency resolution of the spectrum. Note that the arithmetic averaging of spectra is performed for the squares of Fourier amplitudes, thus conserving the signal energy.

To use the smoothing procedure, enable the **Subdivide into Segments** option and define the segment size and the overlap of the subsequent segments in the **Segment Control** group box. Make a selection from the **Control Method** list to specify how you want to define the segment size: select **Samples** if you want to specify the number of **Samples per Segment**; select **Frequency** if you want to specify the desired **Frequency Resolution** Δf in Hertz units (the segment length in seconds is then equal to $1/\Delta f$). To help determine a suitable segment size, you can use the **Number of Samples** and the **Min Frequency** information in the **Signal Statistics** group box. Note that the final segment length is selected by Ansys Fluent to adhere to the largest supported number of data samples (see Table 40.2: Numbers of Data Points Supported by the Prime-Factor FFT Algorithm (p. 4080)). The actual signal length and the number of segments will be displayed in the console when you plot or write the Fourier spectrum. The number of segments depends on the **Overlap** of the subsequent segments, which is specified regardless of which **Control Method** you select. If you select a **Window** function (see Windowing (p. 4079) for details), it is recommended that you use an **Overlap** value of at least 0.125, in order to cover the signal portion affected by the window function. Generally, an **Overlap** value of 0.5 is recommended. The exact number of samples that are in the overlapping region of the segment can be determined from the segment hop size, which is displayed in the console. The segment hop size is the distance (in terms of samples) between the starting samples of any two adjacent segments (that is, the shift from one segment to the next), and has a minimum value of 1.

The **Set Defaults** button will reset the segment control values and disable the **Subdivide into Segments** option.

40.12.4.2.3. Viewing Data Statistics

To aid in the signal analysis, whether for the entire input signal or for a certain range of data, the **Signal Statistics** group box in the Plot/Modify Input Signal Dialog Box (p. 5309) displays signal information such as the minimum, maximum, and average signal values, as well as the signal variance, total number of samples, and finest possible frequency resolution.

40.12.4.2.4. Customizing Titles and Labels

You can create a new title or edit the original title for the input signal plot by entering a text string in **Signal Plot Title**. Likewise, you can create a new axis label or edit the original axis label by entering a text string into either **Y Axis Label** or **X Axis Label**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

40.12.4.2.5. Applying the Changes in the Input Signal Data

To apply any changes you have made in the Plot/Modify Input Signal Dialog Box (p. 5309) and view a plot of the input signal, click **Apply/Plot**.

40.12.4.3. Customizing the Output

In most practical applications with CFD data, you may want to find out how much power or energy is contained in a certain frequency range, but do not want to distinguish positive and negative frequency. In recognition of this, all the outputs from the FFT module in Ansys Fluent pertain to *one-sided spectra* for the range of positive frequency.

The Fourier Transform Dialog Box (p. 5302) (Figure 40.124: The Fourier Transform Dialog Box (p. 4082)) and Plot/Modify Input Signal Dialog Box (p. 5309) (Figure 40.125: The Plot/Modify Input Signal Dialog Box (p. 4083)) enable you to set several different functions for the x and y axes, apply different FFT windowing techniques, and set various output options.

40.12.4.3.1. Specifying a Function for the Y Axis

You can choose the y-axis function using the **Y Axis Function** drop-down list. Available options for the y-axis functions are as follows. Note that the functions related to acoustics (all of which are measured in dB) are only available when the **Acoustics Analysis** option is enabled.

The definitions are provided for $n \le N/2$, and include contributions from both elements of a complex conjugate pair n and N-n. Note that in Ansys Fluent, the value of N is always an even number.

Power Spectral Density

is the distribution of signal power in the frequency domain. Its value and units depend on the **X Axis Function** choice. For the detailed spectral representation with all resolved harmonics (that is, when **X Axis Function** is either **Frequency**, **Strouhal Number**, or **Fourier Mode**), the **Power Spectral Density** (PSD) has units of the signal magnitude squared over the frequency (for example, Pa^2/Hz) and is defined for the frequency f_n as

$$PSD(f_n) = E(f_n)/\Delta f \qquad n = 1,2,\ldots,N/2 \tag{40.10}$$

where Δf is the frequency step in the discrete spectrum, and the Fourier mode power $E(f_n)$ is computed as

$$E(f_n) = \begin{cases} 0.5\left(2|\hat{\phi}_n|\right)^2 & n = 1,2,\ldots,N/2-1 \\ |\hat{\phi}_n|^2 & n = N/2 \end{cases} \tag{40.11}$$

For the octave analysis (that is, when the **X Axis Function** is either **Octave Band** or **1/3-Octave Band**), the **Power Spectral Density** has units of the signal magnitude squared (for example, Pa^2), and is defined for the frequency band f_{band} as

$$PSD(f_{\text{band}}) = \sum E(f_n) \tag{40.12}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where n includes all of the Fourier modes belonging to the band.

Magnitude

is the amplitude. For the detailed spectral representation with all resolved harmonics (that is, when **X Axis Function** is either **Frequency**, **Strouhal Number**, or **Fourier Mode**), the **Magnitude** (A) is defined for the frequency f_n as

$$(f_n)=\begin{cases} |\hat{\phi}_n| & n=0, N/2 \\ 2|\hat{\phi}_n| & n=1,2,...,N/2-1 \end{cases} \tag{40.13}$$

where $A\left(f_0\right)$ is the mean signal value.

For the octave analysis (that is, when the **X Axis Function** is either **Octave Band** or **1/3-Octave Band**), the **Magnitude** is defined for the frequency band f_{band} as

$$A\left(f_{\text{band}}\right)=\sqrt{2PSD\left(f_{\text{band}}\right)} \tag{40.14}$$

where $PSD\left(f_{\text{band}}\right)$ is calculated according to Equation 40.12 (p. 4085).

Sound Pressure Level (dB)

is the decibel level. For either general or acoustic data, when the sampled data is pressure (for example, static pressure or sound pressure), the **Sound Pressure Level (dB)** (L_{sp}) is calculated in decibel units using

$$L_{\text{sp}}=10\left(\log\frac{PSD}{p_{\text{ref}}^2}\right) \quad (\text{dB}) \tag{40.15}$$

where PSD is the **Power Spectral Density** for either a particular Fourier mode or a particular frequency band (see Equation 40.10 (p. 4085) and Equation 40.12 (p. 4085)). p_{ref} is the reference acoustic pressure, with a default value of 2×10^{-5} Pa; you can revise this value, as described in Enabling the FW-H Acoustics Model (p. 2629).

Sound Amplitude (dB)

is similar to the **Sound Pressure Level (dB)**, and is a logarithmic conversion of the pressure signal **Magnitude** into decibel units. The **Sound Amplitude (dB)** (A_{sp}) is calculated for either a Fourier mode or a frequency band using

$$A_{\text{sp}}=10\left(\log\frac{A}{p_{\text{ref}}}\right) \quad (\text{dB}) \tag{40.16}$$

A-Weighted, Sound Pressure Level (dB A)

is the calculated sound pressure level weighted by the A-scale function to more closely approximate the frequency response of the human ear. A-Weighting is applied for loudness levels below 55 phons (55 dB at 1 kHz) and is the most commonly used weighting function. See Figure 40.126: A-, B-, and C-Weighting Functions (p. 4087) for a graphical representation. This function is only available when the **X Axis Function** is either **Octave Band** or **1/3-Octave Band**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

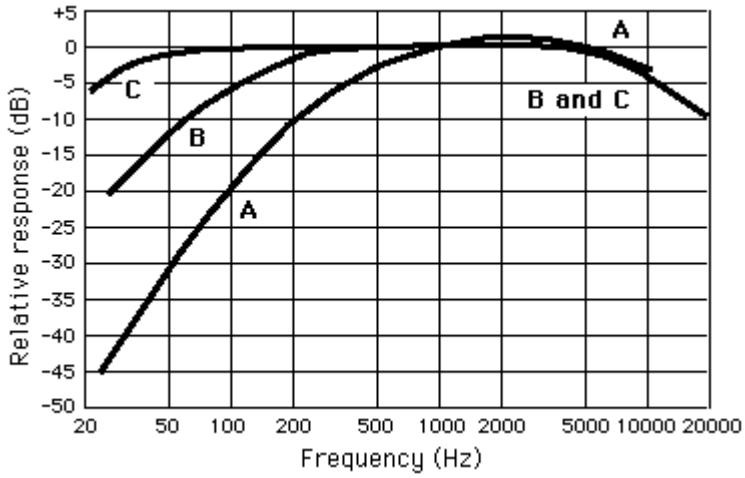
B-Weighted, Sound Pressure Level (dB B)

is the calculated sound pressure level weighted by the B-scale function. B-Weighting is applied to loudness levels between 55 and 85 phons, though it is rarely used. See Figure 40.126: A-, B-, and C-Weighting Functions (p. 4087) for a graphical representation. This function is only available when the **X Axis Function** is either **Octave Band** or **1/3-Octave Band**.

C-Weighted, Sound Pressure Level (dB C)

is the calculated sound pressure level weighted by the C-scale function. C-Weighting is applied for loudness levels above 85 phons and is commonly used for high-intensity sound such as traffic studies. See Figure 40.126: A-, B-, and C-Weighting Functions (p. 4087) for a graphical representation. This function is only available when the **X Axis Function** is either **Octave Band** or **1/3-Octave Band**.

Figure 40.126: A-, B-, and C-Weighting Functions



Further graphical customizations for the y-axis are available by clicking the **Axes...** button. For more information, see Modifying Axis Attributes (p. 4074).

40.12.4.3.2. Specifying a Function for the X Axis

There are three options for the x-axis function you can choose from in order to plot or write the detailed spectrum with all resolved Fourier modes; these three options are related to the discrete frequencies at which the Fourier coefficients are computed. There are also two additional options available when the **Acoustics Analysis** option is enabled, which allow the octave and 1/3 octave band analysis. You can apply specific analytic functions for the x-axis using the **X Axis Function** drop-down list.

Available options for the x-axis functions are as follows. The definitions are provided for $n \leq N/2$, because the corresponding definitions for the y-axis functions include contributions from both elements of a complex conjugate pair n and N-n. Note that in Ansys Fluent, the value of N is always an even number.

Frequency (Hz)

is defined as:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$f_n = \frac{1}{N\Delta t} n \qquad n = 0, 1, 2, \ldots, N/2 \tag{40.17}$$

where N is the number of data points used in the FFT.

Strouhal Number

is the nondimensionalized version of the frequency defined in Equation 40.17 (p. 4088) :

$$St_n \equiv \frac{f_n L_{ref}}{U_{ref}} \tag{40.18}$$

where L_{ref} and U_{ref} are the reference length and velocity scales specified in the Reference Values Task Page (p. 5154).

Fourier Mode

is the index in Equation 40.8 (p. 4080) and/or Equation 40.9 (p. 4080), which represents the n^{th} term in the Fourier transform of the signal.

Octave Band (Hz)

is a range of discrete frequency bands for different octaves within the threshold of hearing. The range of each octave band is double to that of the previous band (see Table 40.3: Octave Band Frequencies and Weightings (p. 4088)).

1/3-Octave Band (Hz)

is a range of discrete frequency bands for different 1/3 octaves within the threshold of hearing.

Table 40.3: Octave Band Frequencies and Weightings

Lower Freq. (Hz)	Center Freq. (Hz)	Upper Freq. (Hz)	dB A	dB B	dB C
11	16	22	-56.7	-28.5	-8.5
22	31.5	45	-39.4	-17.1	-3.0
45	63	90	-26.2	-9.3	-0.8
90	125	180	-16.1	-4.2	-0.2
180	250	355	-8.6	-1.3	0.0
355	500	710	-3.2	-0.3	0.0
710	1000	1400	0.0	0.0	0.0
1400	2000	2800	1.2	-0.1	-0.2
2800	4000	5600	1.0	-0.7	-0.8
5600	8000	11200	-1.1	-2.9	-3.0
11200	16000	22400	-6.6	-8.4	-8.5

Further graphical customizations for the x-axis are available by clicking **Axes...**. For more information, see Modifying Axis Attributes (p. 4074).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

40.12.4.3.3. Specifying Output Options

You can write out the FFT data directly to a file by choosing the **Write FFT to File** option under **Options** in the Fourier Transform Dialog Box (p. 5302). Once the **Write FFT to File** option is selected, click **Write FFT** to display a file selection dialog box where you can choose a file and/or a location to hold the FFT data. If **Acoustics Analysis** is selected, the overall sound pressure level will be calculated in dB (based on the **Reference Acoustic Pressure**) and displayed in the console at this time.

Further customizations for how the FFT data is displayed are available by clicking **Curves...**. For more information, see Modifying Curve Attributes (p. 4076).

40.12.4.3.4. Specifying a Windowing Technique

You can use the various windowing techniques described in Windowing (p. 4079) by selecting any of the **Window** options in the Plot/Modify Input Signal Dialog Box (p. 5309). By default, **None** is selected so that no windowing technique is applied.

40.12.4.3.5. Specifying Labels and Titles

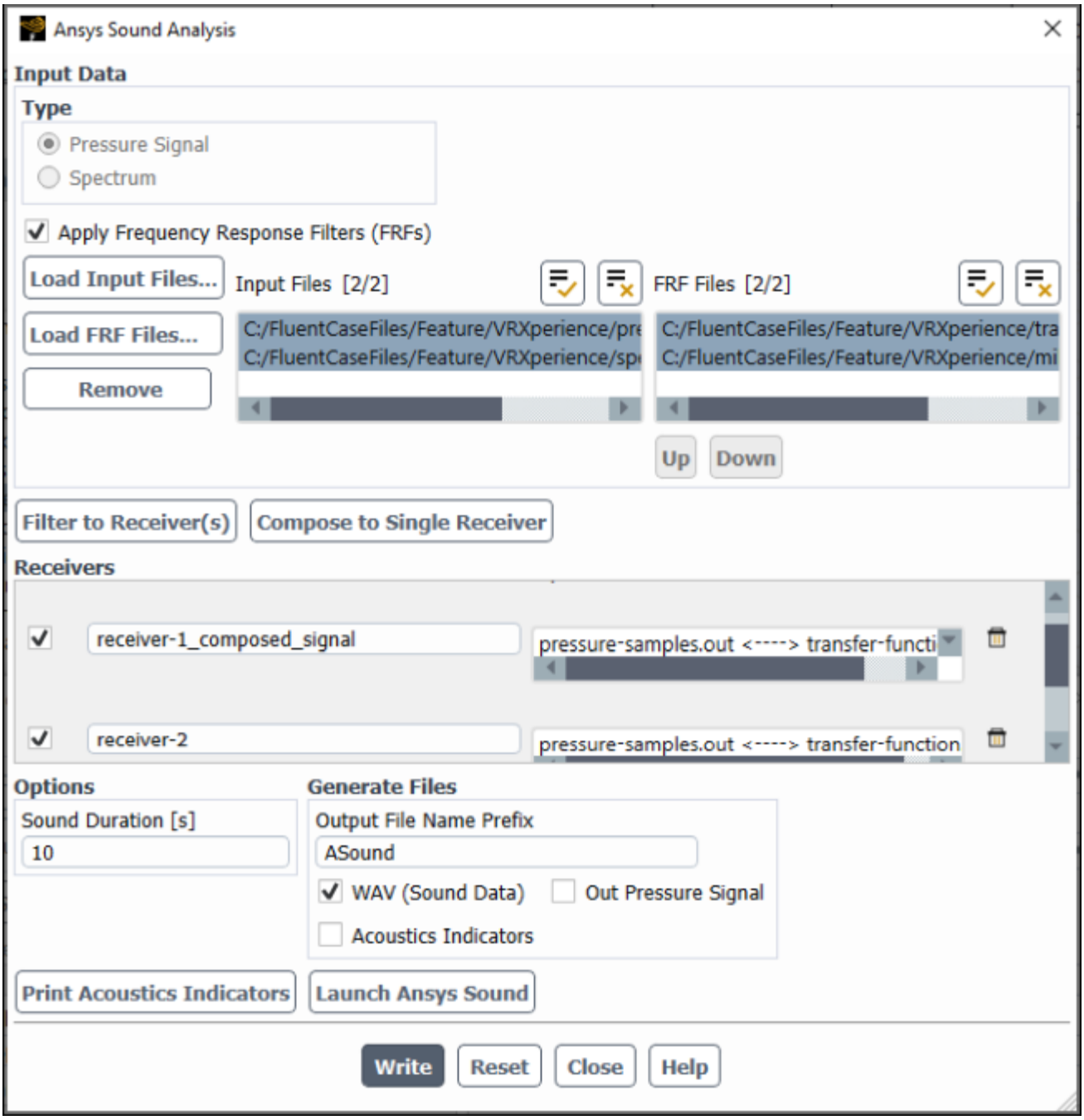
You can assign a title for your FFT plot using the **Plot Title** text field. You can also assign y-axis and x-axis labels for your FFT plot using the **Y Axis Label** and **X Axis Label** text fields, respectively. By default,Ansys Fluent assigns the **Y Axis Label** and the **X Axis Label** to the particular selection of **Y Axis Function** and **X Axis Function**.

40.12.4.4. Performing an Ansys Sound Analysis

You can use the **Ansys Sound Analysis** dialog box to evaluate the acoustic indicators for one or more pressure signal or spectrum files. Using a frequency response filter (FRF) file, you can translate a pressure signal from one location to see the acoustic indicators at another location. You can also combine input files with FRFs to create a merged receiver for output at a single location. Output options include writing a WAV format sound file, output pressure signal, and/or acoustic indicator files. The WAV file can then be read into most suitable media players so you can hear the noise of the simulation results. The WAV file can also be read into Ansys Sound for listening and further analysis or combination with other simulation and measurement data.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 40.127: The Ansys Sound Analysis Dialog Box



To perform the acoustic analysis:

1. Open the **Ansys Sound Analysis** dialog box by clicking **Ansys Sound Analysis...** in Figure 40.124: The Fourier Transform Dialog Box (p. 4082).

2. Select the input file type.

   - **Pressure Signal**—an acoustic pressure file (two column file of time vs. pressure) that typically has an FMD file extension.

     Example pressure signal file:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
"pb"
"Flow Time" "pb"
("Flow Time" "pb")
0.0005 -0.0031175
0.001 0.00365144
0.002 0.00954449
0.003 0.0145043
0.004 0.0184765
0.005 0.0214042
0.006 0.0232245
0.007 0.0238654
0.008 0.0232452
0.009 0.0212717
0.01 0.0178438
```

- **Spectrum**—an FFT output file (two column file of frequency vs. power spectral density).

  Example spectrum file:

```
(title "Spectral Analysis of pb")
(labels "Frequency (Hz)" "Power Spectral Density")

((xy/key/label "pb")
5.0000000e+00 1.6515032e-05
1.0000000e+01 2.0448203e-06
1.5000000e+01 3.7586217e-06
2.0000000e+01 4.5504839e-06
2.5000000e+01 3.7905529e-06
3.0000000e+01 3.5729342e-06
3.5000000e+01 4.6056671e-06
4.0000000e+01 5.1608927e-06
4.5000000e+01 1.9510917e-06
5.0000000e+01 8.7493390e-06
5.5000000e+01 5.8902977e-05
6.0000000e+01 3.5942177e-04
6.5000000e+01 6.3120312e-04
```

3. (Optional) Enable **Apply Frequency Response Filters (FRFs)** to allow the loading of a frequency response filter (FRF) file.

   Example FRF file:

```
AnsysSound_FRF 1
150    8.83959
200    31.1247
250    23.8472
300    5.65476
350    5.98757
400    29.6756
450    34.4814
500    41.3502
550    28.1577
600    19.6379
650    30.9932
700    19.093
750    22.9459
800    16.0768
850    30.0843
900    31.012
950    25.9581
1000    73.0306
```

4. Click **Load Input File...** to select the desired input file(s).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. (If **Apply Frequency Response Filters (FRFs)** is enabled) Click **Load FRF Files...** to select the desired FRF file(s).

6. Select one or more **Input Files** and an equivalent number of **FRF Files** (if applicable).

7. Create one or more receivers:

   - Click **Filter to Receiver(s)** to add one or more receivers that are mapped with one input file and one FRF file per receiver.

   - Click **Compose to Single Receiver** to combine the selected input files and corresponding FRF files into a single receiver.

   *Note that when you have multiple input files selected, the receivers are created assuming that the order of the **FRF Files** list matches the intended correspondence with the **Input Files** list. That is, the first input file will be paired with the first FRF file, the second input file will be paired with the second FRF file, and so on.*

   If you need to reorder the files for proper correspondence, you can click **Up** or **Down** to move the selected transfer function file in the list.

   *The **Name** of the receiver is used when you write the WAV file.*

8. Specify the **Sound Duration** under **Options**.

9. Click **Print Acoustic Indicators**, which computes the acoustic indicators on the active receiver(s) and prints them to the console.

   - `Loudness [ISO 532B,Range -20 / + 140 phones]`—The sensory scale of sound intensity. A sound perceived as twice as loud as another has double the loudness value in sones. Doubling a loudness value in sone (doubling the loudness sensation) corresponds to adding 10 phones to the loudness level. Refer to Loudness in the Ansys Sound: Analysis and Specification User's Guide for additional information.

   - `Sound Level [dB SPL,Range -20 / + 200 dB]`—The equivalent continuous sound level. Its value is the average energy over time of the signal. Refer to Standard Levels in the Ansys Sound: Analysis and Specification User's Guide for additional information.

   - `Sound Level A-weighted [dBA SPL,Range -20 / + 200 dBA]`—This is equivalent to the sound level, but with an A-weighting of the levels. Refer to Standard Levels in the Ansys Sound: Analysis and Specification User's Guide in the for additional information.

   - `Sharpness [DIN 45692, Range 0 / 10 acum]`—An attribute that is related to the timbre of a sound. Sharpness is the subjective attribute describing the perception of the spectral balance of a sound. Sharpness is measured in "acum", which is defined as the sharpness of a narrow band of noise (one Bark) centered at one kHz at 60 dB SPL. Refer to Sharpness in the Ansys Sound: Analysis and Specification User's Guide for additional information.

   - `Tonality DIN 45681, [Range 0 / 100 dB] information is as follows:`

     – `Mean difference DL (dB)`

     – `Tonal Adjustment (dB)`

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - `Number of tones`

    Tonal adjustment Kt DIN 45981 (dB) is based on the DIN 45681:2005-03 standard. The standard describes a method to objectively determine the noise tonality and to determine the tonal adjustment for the evaluation of tonal emissions. Refer to DIN45681 in the Ansys Sound: Analysis and Specification User's Guide for additional information.

- `Articulation Index (ANSI S3.5-1969)`—Ranges from 0 (speech not intelligible) to (speech perfectly clear). Refer to Intelligibility in the Ansys Sound: Analysis and Specification User's Guide for additional information.

Note:

The values of acoustic indicators (all the printed fields except **Loudness**) may vary when repeatedly computed. These differences are due to the Inverse Fast Fourier Transform (IFFT) that is performed, which uses random phases to generate a sound, from which the indicators are computed. The differences are small. *Note that repeated experiments of the same phenomenon are unlikely to have exactly matching measurements, even though the samples sound the same. Refer to the point on the* ***Hybrid/Automatic*** *method in Methods for Sound Creation for additional information.*

---

10. Select the types of files you want to save in the **Generate Files** group box.

    - **WAV (Sound Data)**—sound of the computed analysis that can be played using most media players.
    - **Out Pressure Signal**—output signal that can be read into Ansys Sound.
    - **Acoustic Indicators**—report of the acoustic indicators including sound level, frequency, and so on.

11. Click **Write** to save the specified receiver(s). Note, files are only written for the selected receivers (those with an enabled checkbox).

    Two files are created for each receiver when you click **Write**; a 32-bit file and a 16-bit file. The 16-bit WAV file is well suited for listening in any sound file player or being included in presentations. The 32-bit file contains the real physical values of the samples, but it is not always supported by all audio players (this can lead to distorted sound playback). *Both files are fully supported in Ansys Sound SAS. It will keep the real physical values of each sample (with more precision in the 32-bit file, as the dynamic range is higher) and will not generate any artifacts while listening.* Refer to File Formats for additional information on these files.

12. (Optional) Click **Launch Ansys Sound** to launch Ansys Sound: Analysis and Specification for further solution exploration (Welcome! in the *Ansys Sound: Analysis and Specification User's Guide*).

    Ansys Sound: Analysis and Specification will launch with the selected receiver(s) open with the specified ***Sound Duration****.*

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

40.13. Cumulative Force, Moment, and Coefficients Plots

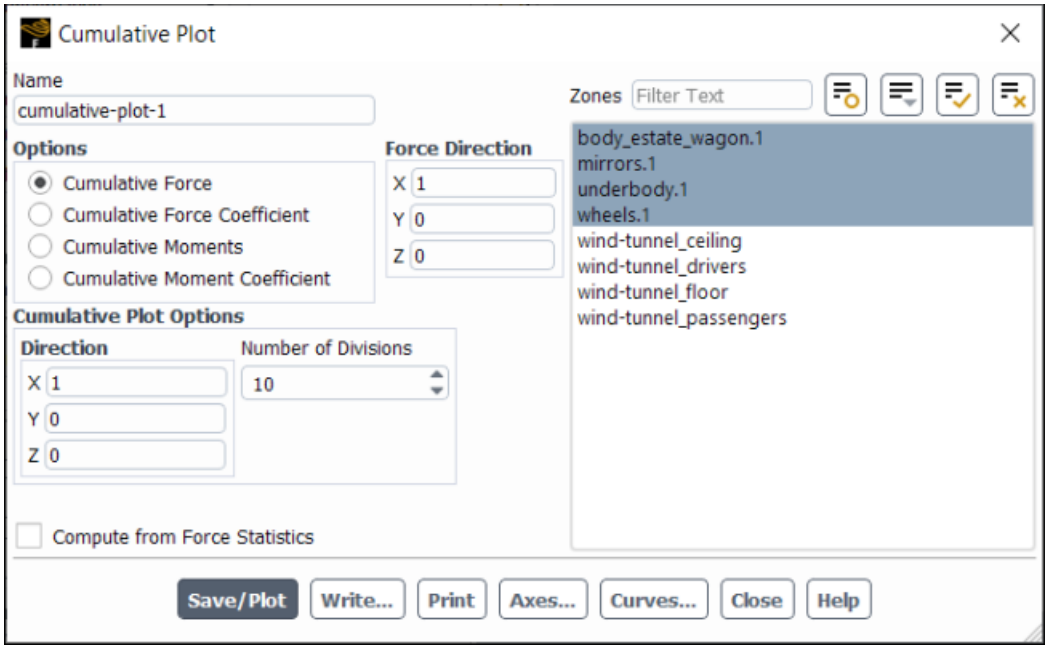
In industries such as aerospace and automotive, it is useful and informative to understand how forces and moments develop along certain directions, such as along a car's length. This provides valuable insights on which region is contributing more forces and moments to the model. Using the **Cumulative Plot** dialog box you can specify the type of cumulative plot (force, moment, force coefficient, or moment coefficient) and select which surfaces to include as well as how many "slices" to make through the model.

40.13.1. Steps for Generating Cumulative Plots

You can create a cumulative plot of force, moment, force coefficients, and moment coefficients using Cumulative Plot Dialog Box (p. 5307) (Figure 40.128: The Cumulative Plot Dialog Box (p. 4094)).

Results → Plots → Cumulative Plot → New...

Figure 40.128: The Cumulative Plot Dialog Box



The steps for generating a cumulative plot are as follows:

1. (Optional) Provide a **Name** for the cumulative plot.
2. Select the type of cumulative plot in the **Options** group box:
   - Cumulative Force
   - Cumulative Force Coefficient
   - Cumulative Moments
   - Cumulative Moment Coefficient

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3. Specify the **Direction** and **Number of Divisions** (in the **Cumulative Plot Options** group box). Along this direction, Fluent creates divisions on the selected surfaces and computes the cumulative force or moments.

   - **Direction**—the planes created according to the **Number of Divisions** will be perpendicular to this direction.

   - **Number of Divisions**—how many slices or cuts perpendicular to the **Direction** will be made through the selected **Wall Zones** for plotting the selected quantity.

4. Specify the **Force Direction** (for **Cumulative Force** and **Cumulative Force Coefficient**) or the **Moment Center** and **Moment Axis** (for **Cumulative Moments** and **Cumulative Moment Coefficient**).

5. (Optional) Enable **Compute from Force Statistics** to use averaged values instead of instantaneous values in the plot. *This option is only available if **Force Statistics** is enabled in the Sampling Options Dialog Box (p. 5232)*.

6. Select the desired **Zones** to include in the cumulative plot.

7. Click **Save/Plot**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 41: Reporting Alphanumeric Data

Ansys Fluent provides tools for computing and reporting integral quantities at surfaces and boundaries. These tools enable you to find the mass flow rate and heat transfer rate through boundaries, the forces and moments on boundaries, and the area, integral, flow rate, average, and mass average (among other quantities) on a surface or in a volume. In addition, you can print histograms of geometric and solution data, set reference values for the calculation of non-dimensional coefficients, and compute projected surface areas. You can also print or save a summary report of the models, boundary conditions, and solver settings in the current case. These features are described in the following sections.

41.1. Reporting Conventions

41.2. Monitoring and Reporting Solution Data

41.3. Creating Output Parameters

41.4. Fluxes Through Boundaries

41.5. Forces on Boundaries

41.6. Projected Surface Area Calculations

41.7. Surface Integration

41.8. Volume Integration

41.9. Efficiency Calculations

41.10. Histogram Reports

41.11. Discrete Phase

41.12. S2S Information

41.13. Reference Values

41.14. Summary Reports of Case Settings

41.15. System Resource Usage

Reporting tools for the discrete phase are described in Postprocessing for the Discrete Phase (p. 2833).

41.1. Reporting Conventions

For 2D problems, Ansys Fluent computes all integral quantities for a unit depth equivalent to 1 meter. This value can be adjusted to match the specific dimension of your application only by manually revising the **Depth** in the Reference Values Task Page (p. 5154) (see Reference Values (p. 4158)).

Important:

The default value of **Depth** will be equivalent to 1 meter, even if the units are changed for **depth** in the Set Units Dialog Box (p. 4693) (for example, if the units for **depth** are changed to cm in the Set Units Dialog Box (p. 4693), the value of **Depth** in the Reference Values Task Page (p. 5154) will be 100 cm).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that inputs for DPM and fluid massflow are always assumed to correspond to a 1m deep version of the model model and are scaled accordingly. For example, if the reference Depth is set to 0.5 m, effectively reducing the model by half, massflows used in volume integral calculations will be half of the original massflows.

For axisymmetric problems, all integral quantities are computed for an angle of 2π radians.

41.2. Monitoring and Reporting Solution Data

During the course of a simulation, you may want to monitor, print, write, and/or plot solution data; this is done using report definitions. A report definition is an object that specifies a certain quantity or set of values to be computed at the end of a solver timestep or iteration. You can then choose to have multiple report definitions printed to the console, written to a single file, or plotted in the same window.

You can use report definitions to monitor convergence, verify mesh independence, and record time-histories. These report definitions can monitor the value of field variables or functions through specified operations, such as surface, volume, force, moment, and flux integrals.

Once you create report definitions, they can be added to a report file. Report files can include multiple report definitions of varied types. Multiple report definitions can be plotted in the same graphics window by adding them to a single report plot. One report definition can be included in multiple report plots and files. For additional information on report files and plots, see Report Files and Report Plots (p. 4123).

Important:

Expression report definitions created prior to release 2019 R3 appear as **Custom Report Definitions** in this release and Fluent may be unable to evaluate them. You can recreate any affected report definitions using the Expression Report Definition (p. 4121) dialog box and they should evaluate correctly.

41.2.1. Creating Report Definitions

You can create report definitions from multiple locations in Fluent, allowing you to quickly define report definitions, regardless of where you are in your workflow. The primary method to create a report definition is through the ribbon. For example:

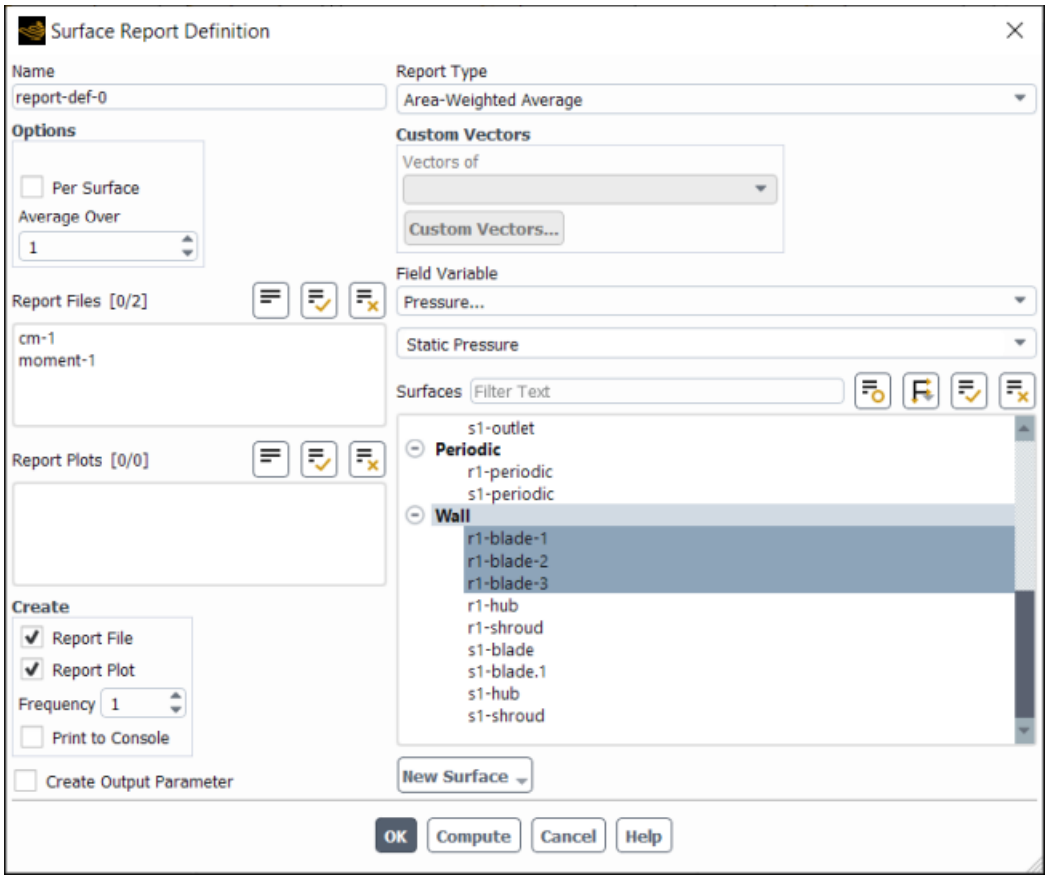
Solution → **Reports** → **Definitions** → **New** → **Surface Report** → **Area-Weighted Average...**

Opens the **Surface Report Definition** dialog box.

Alternative methods for accessing this dialog box are:

- *through the tree, by right-clicking the **Report Definition** branch*
- *through the graphics window via the right-click context menu (when the mesh is displayed and one or more surfaces or volumes are selected)*
- *through the **New** drop-down list in the Figure 41.1: Report Definitions Dialog Box (p. 4102)*

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Many of the setup steps are the same or similar when setting up the various types of report definitions.

To set up the **Surface Report Definition** dialog box:

1. Enter a name for the report definition under **Name**.
2. If you want the value reported on individual surfaces rather than the net results from a group of surfaces, enable the **Per Surface** option.

   Important:

   The **Per Surface** and **Per Zone** options are incompatible with an **Average Over** value greater than 1. *The issue is that the net value does not reflect the specified **Average Over** value. The net value is seen when you click **Compute**, use the report definition in an expression, or as an output parameter.*

   If you intend to use an **Average Over** of 2 or more, then create separate report definitions for each surface or zone.

   ---

3. (optional) To have Fluent calculate a running average for the **Surface Report Definition** you can enter a positive integer greater than 1 (the default) for **Average Over**.

   Specifying a number greater than 1 means that Fluent will print, plot, and write the running average value of the selected variable instead of the current value of the same variable.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The value reported is averaged over the last N *iterations/time steps, where* N *is your specified* ***Average Over*** *value. When the iteration/time step number is lower than* N*, Fluent calculates the average of the available variable values.*

Important:

Increasing the **Average Over** value exposes an additional **Retain Instantaneous Values** option. When enabled, the option ensures that the report file storing the data for this report definition will not only store the averaged values, but the instantaneous values as well. This preserves the flexibility to change the **Average Over** value, because Fluent will have the data require to re-compute a new average.

If you plan on varying the **Average Over** value at any point in the future, it is recommended that you start with an **Average Over** of at least 2, so that **Write Instantaneous Values** is enabled in the containing report file (assuming you keep **Retain Instantaneous Values** enabled when you create the report definition).

---

4. Fluent automatically creates a report file and plot for each report definition (indicated by **Report File** and **Report Plot** being enabled within the **Create** group box).

   If you prefer to write all your report definitions to a single file, you can select an existing file from the **Report Files** list.

   The default behavior of automatically enabling ***Report File*** *and* ***Report Plot*** *for new report definitions is controlled in* ***Preferences*** *by the* ***Automatically Plot and File*** *option (****File****>****Preferences...****>****SimulationReport Definitions****>****Automatically Plot and File****).*

5. If you want the report definition available as an output parameter, enable **Create Output Parameter**.

   Note:

   Using the **Per Surface** or **Per Zone** option with multiple selected surfaces/zones in conjunction with the **Create Output Parameter** option, results in a single output parameter. The output parameter will show a single combined value for all the selected surfaces/zones.

   ---

6. Choose the surface report definition integration method by selecting **Area**, **Integral**, **Standard Deviation**, **Flow Rate**, **Mass Flow Rate**, **Volume Flow Rate**, **Area-Weighted Average**, **Mass-Weighted Average**, **Sum**, **Uniformity Index - Mass Weighted**, **Uniformity Index - Area Weighted**, **Facet Average**, **Facet Minimum**, **Facet Maximum**, **Vertex Average**, **Vertex Minimum**, **Vertex Maximum**, **Custom Vector Based Flux**, **Custom Vector Flux**, or **Custom Vector Weighted Average** from the **Report Type** drop-down list. These methods are described in Surface Integration (p. 4150).

7. If you selected **Custom Vector Based Flux**, **Custom Vector Flux**, or **Custom Vector Weighted Average** from the **Report Type** drop-down list, you must define the vectors in the **Custom Vectors** group box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

8. Unless you are reporting a mass flow rate or volume flow rate, specify the variable or function to be integrated in the **Field Variable** drop-down list. First select the desired category in the upper drop-down list. You can then select one of the related quantities in the lower list. (See Field Function Definitions (p. 4167) for an explanation of the variables in the list.) If you are running a multiphase simulation you may need to select the phase of interest (or **mixture**) from the **Phase** drop-down list (depending on the field variable selected).

   Note:

   - In general, you can select any arbitrary field variable for each report type. However, not all possible combinations are physically meaningful.
   - For multiphase flow, you should use only the **Mass Flow Rate** integral to compute the mass flow rate of the mixture or for an individual phase. Do not use the **Flow Rate** integral of the **Volume Fraction** field variable as this will result in erroneous results.

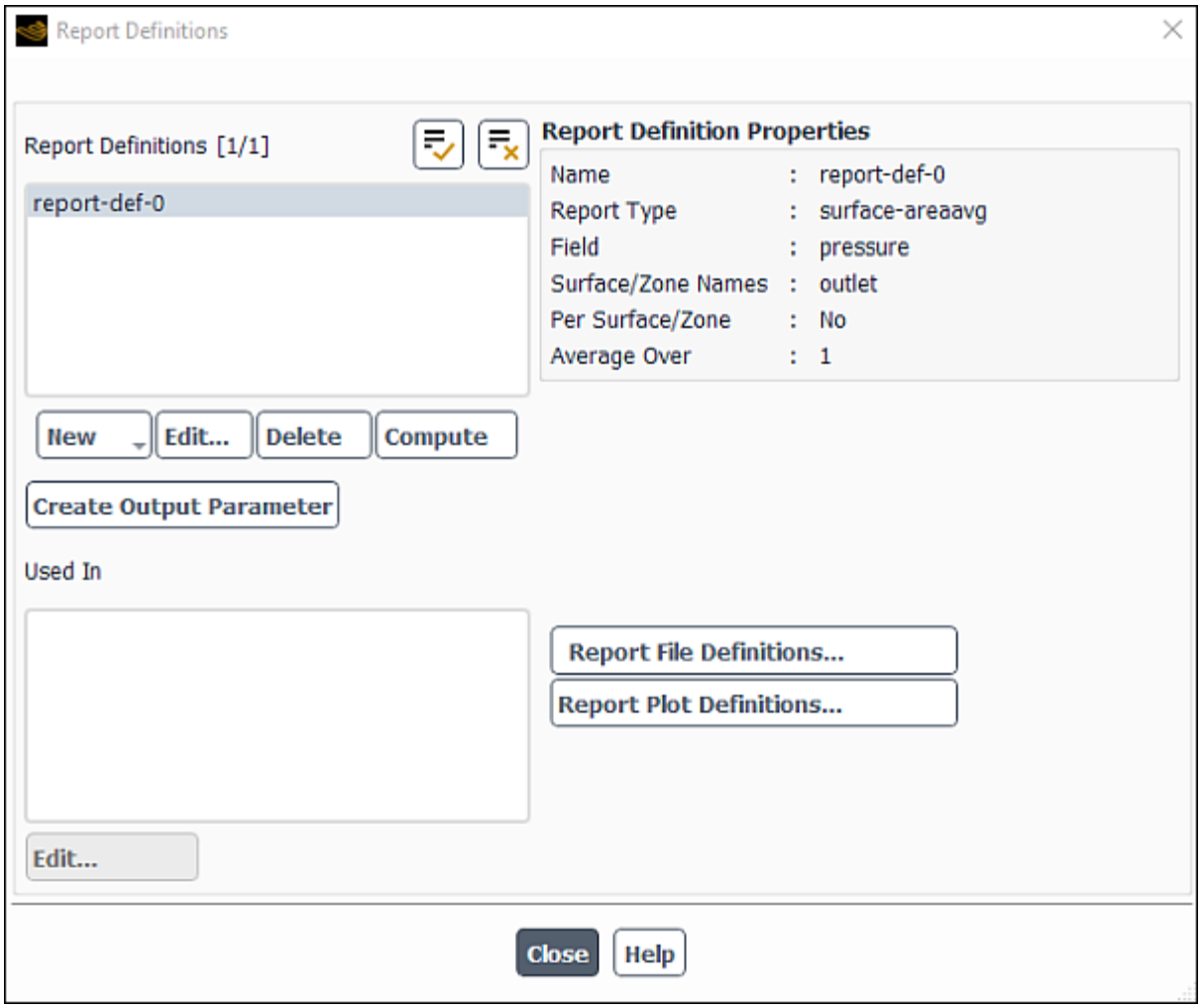
9. Select the **Surfaces** where you want Fluent to evaluate the specified variable.

10. Click **OK** to create the report definition and close the dialog box.

Once you have created a report definition, it appears in the Figure 41.1: Report Definitions Dialog Box (p. 4102).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 41.1: Report Definitions Dialog Box



You can use this dialog box to review and modify the properties of the existing report definitions.

Note:

The report definitions that are automatically created for transient simulations (flow-time, delta-time, iters-per-timestep) are not available for export or copying.

The following sections detail setup instructions that are specific to each type of report definition.

41.2.1.1. Surface Report Definitions
41.2.1.2. Volume Report Definitions
41.2.1.3. Force and Moment Report Definitions
41.2.1.4. Flux Report Definition
41.2.1.5. Mesh Report Definitions
41.2.1.6. Aerodamping (Travelling Wave Method) Report Definition
41.2.1.7. DPM Report Definition
41.2.1.8. User Defined Report Definition

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

41.2.1.9. Expression Report Definition

41.2.1.1. Surface Report Definitions

You can use the Figure 41.2: Surface Report Definition Dialog Box (p. 4104) to create surface report definitions. There are many different types of surface-based report definitions that you can create, including:

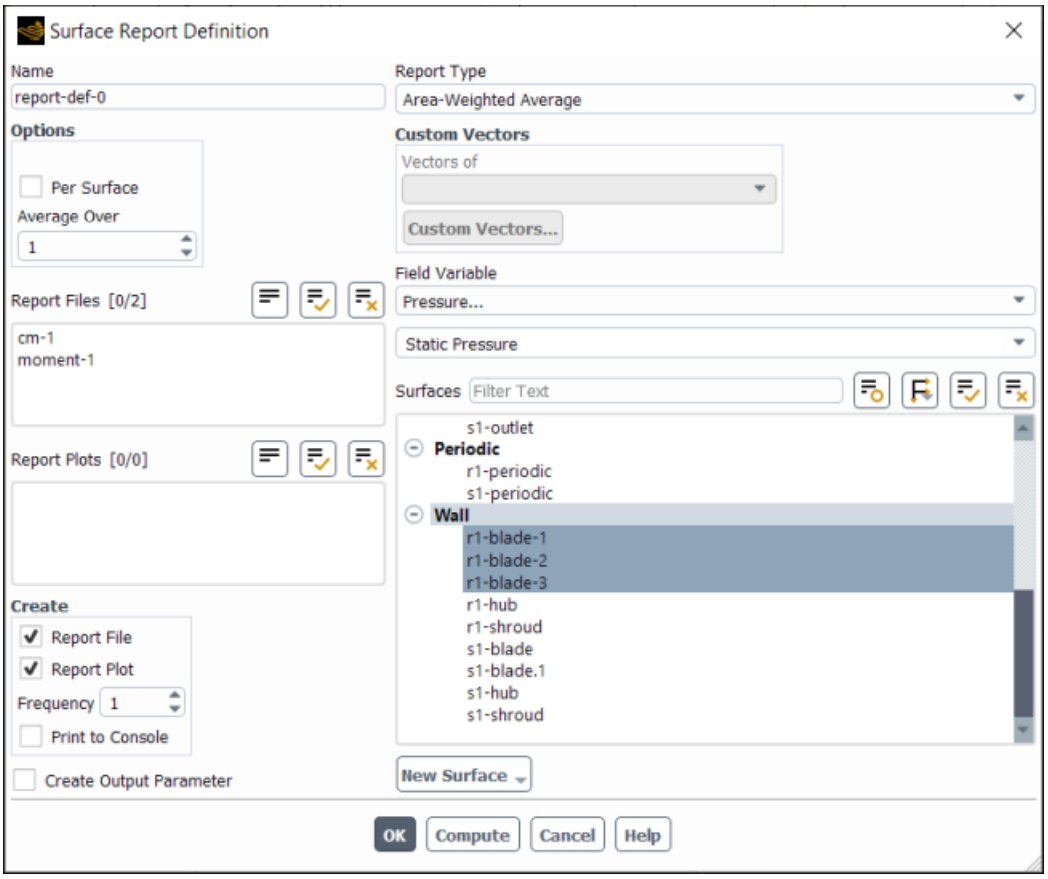
- Integral
- Standard Deviation
- Flow Rate
- Mass Flow Rate (non-field based)
- Volume Flow Rate (non-field based)
- Area-Weighted Average
- Mass-Weighted Average
- Sum
- Uniformity Index — Mass Weighted
- Uniformity Index — Area Weighted
- Facet Average
- Facet Minimum
- Facet Maximum
- Custom Vector Based Flux (custom-vector based)
- Custom Vector Flux (custom-vector based, non-field based)
- Custom Vector Weighted Average (custom-vector based)

To plot or write the surface report definition data you must use a **Report File** or **Report Plot**. If a surface report definition is already included in a report file/plot, then that report file/plot will be highlighted. If you de-select a report file/plot that had included the current report definition and click **OK**, the report definition will be removed from that report file/plot. For additional information on report files and plots, see Report Files and Report Plots (p. 4123).

Solution → **Reports** → **Definitions** → **New** → **Surface Report**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 41.2: Surface Report Definition Dialog Box



The procedure for creating surface report definitions is covered in Creating Report Definitions (p. 4098).

41.2.1.2. Volume Report Definitions

You can use the Figure 41.3: Volume Report Definition Dialog Box (p. 4105) to create volume report definitions. There are many different types of volume reports that you can create, including:

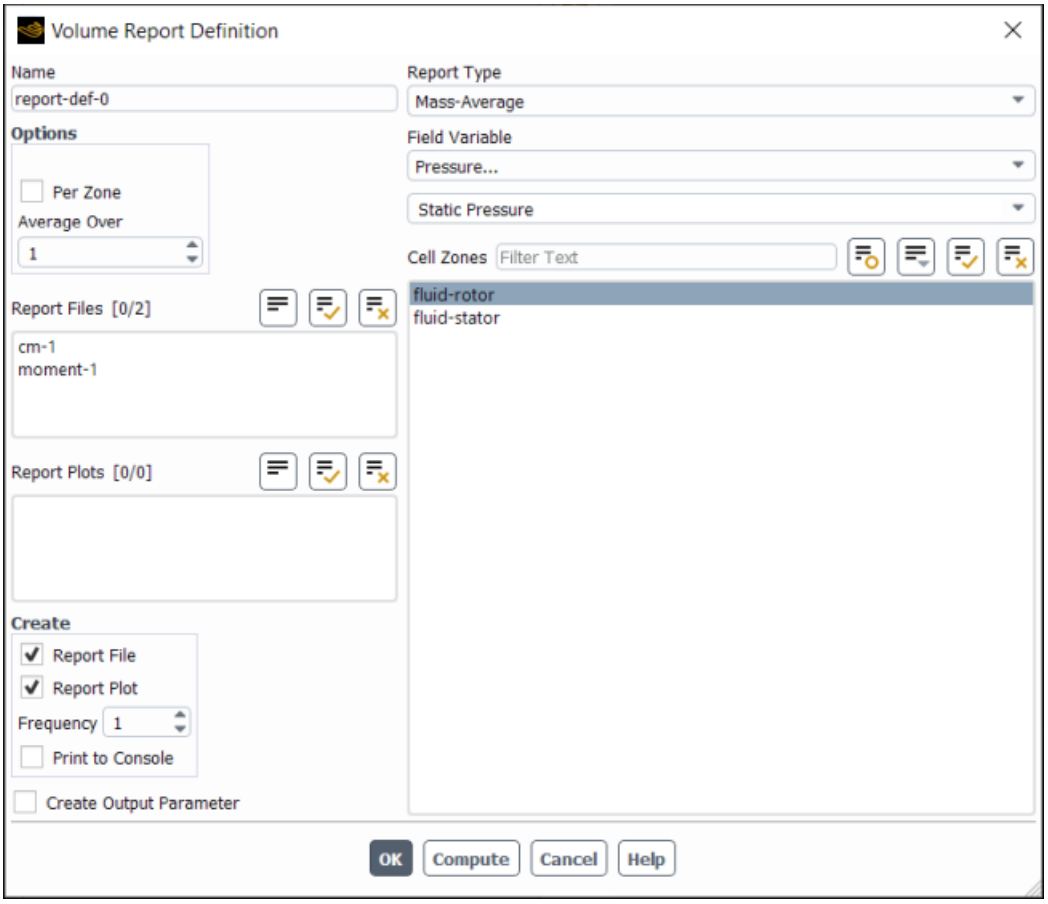
- Volume (non-field based)
- Sum
- Sum*2π (only axi-symmetric)
- Max
- Min
- Volume Integral
- Volume Average
- Mass Integral
- Mass Average

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Mass (non-field based)

Solution → Reports → Definitions → New → Volume Report

Figure 41.3: Volume Report Definition Dialog Box



The common steps for creating a volume report definition are covered in Creating Report Definitions (p. 4098), and the volume report definition-specific steps are as follows:

1. If you want the value reported on individual wall zones rather than the net results from a group of wall zones, enable the **Per Zone** option.

2. Choose the integration method for the volume report definition by selecting **Volume**, **Sum**, **Sum*2Pi** (2D axisymmetric cases only), **Max**, **Min**, **Volume Integral**, **Volume-Average**, **Mass Integral**, **Mass-Average**, or **Mass** in the **Report Type** drop-down list. These methods are described in Volume Integration (p. 4153).

3. Specify the variable or function to be integrated in the **Field Variable** drop-down list. First select the desired category in the upper drop-down list. You can then select one of the related quantities in the lower list. (See Field Function Definitions (p. 4167) for an explanation of the variables in the list.)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. Select the **Cell Zones** where you want Fluent to evaluate the specified variable. *The **Cell Zones** list will show any defined expression volumes, as they are also valid inputs for volume-based computations.*

   Note:

   While cell registers may appear in the cell zone lists, they cannot be used in volume report definitions.

5. Click **OK** to create the report definition and close the dialog box.

41.2.1.3. Force and Moment Report Definitions

Monitoring force coefficients can be useful when you are calculating external aerodynamics, for example, and are especially interested in the lift coefficient. By monitoring these values you may also be able to stop the calculation early and reduce the processing time, as sometimes the force and moment coefficients converge before the residuals have decreased three orders of magnitude. (In such an instance, you should be sure to check the mass flow rate and heat transfer rate as well, to ensure that the mass and energy are being suitably conserved. This is accomplished using either the Figure 41.5: Drag Report Definition Dialog Box (p. 4108), the Figure 41.6: Lift Report Definition Dialog Box (p. 4109), the Figure 41.7: Moment Report Definition Dialog Box (p. 4110), or the Figure 41.4: Force Report Definition Dialog Box (p. 4107)).

To begin setting up force or moment report definitions, first enter appropriate values in the **Reference Values** task page, as described in Reference Values (p. 4158). The relevant values include the following:

- The force coefficients use the reference area, density, and velocity.
- The moment coefficients use the reference area, density, velocity and length.

For information about how these coefficients are calculated, see Computing Forces, Moments, and the Center of Pressure in the Fluent Theory Guide.

Tip:

You can have Fluent report unscaled force values by selecting **Drag Force** or **Lift Force** in the **Report Output Type** group box of the **Drag Report Definition** or **Lift Report Definition** dialog boxes, respectively.

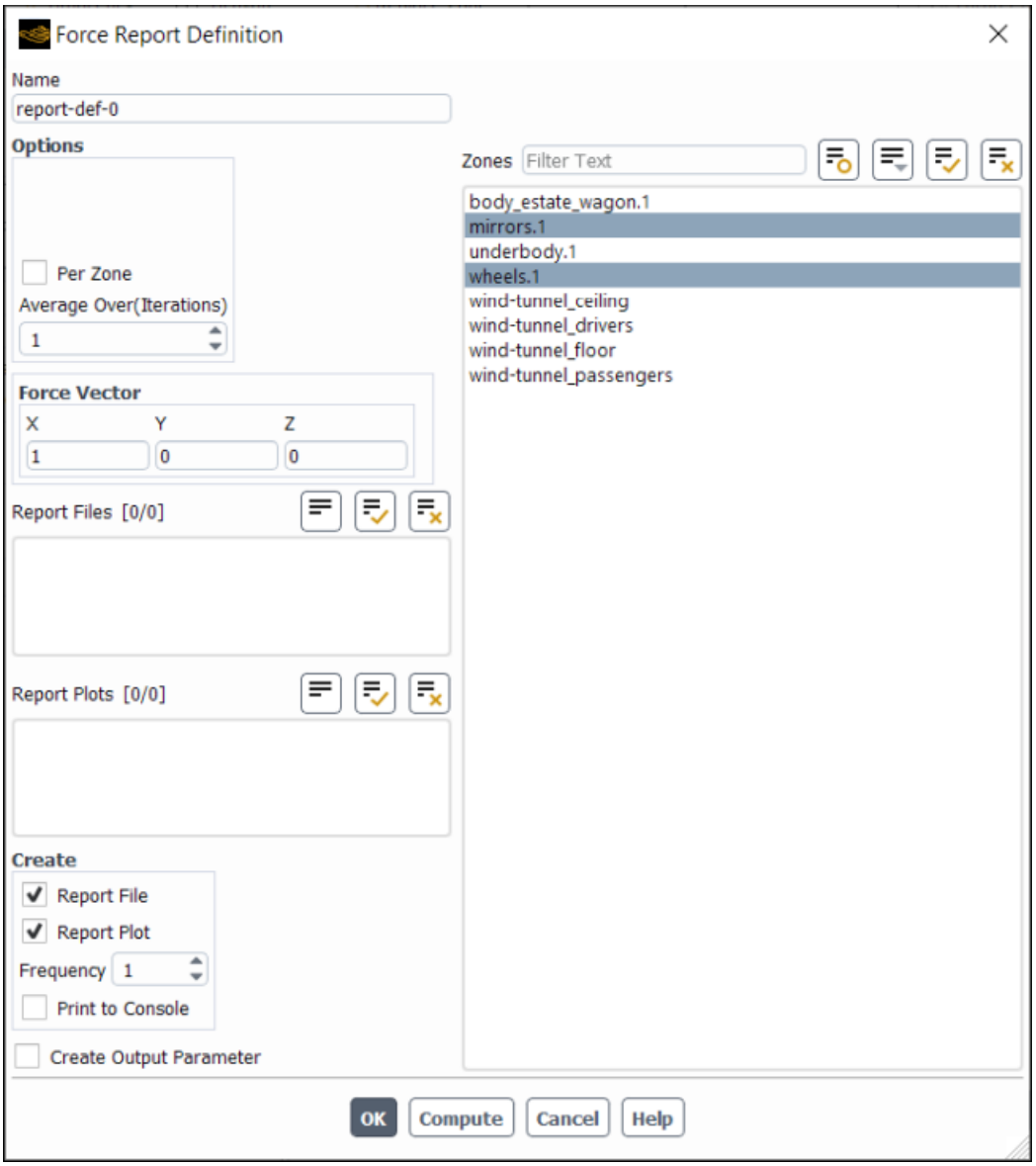
Next, open the appropriate dialog box through the **Solution** tab of the ribbon.

Solution → **Reports** → **Definitions** → **New** → **Force Report**

Select either **Drag...**, **Lift...**, **Moment...**, or **Force...** to open the Figure 41.5: Drag Report Definition Dialog Box (p. 4108), Figure 41.6: Lift Report Definition Dialog Box (p. 4109), Figure 41.7: Moment Report Definition Dialog Box (p. 4110), and Figure 41.4: Force Report Definition Dialog Box (p. 4107). Note that you can only access one of these report definition dialog boxes at a time, though multiple report definitions of the same type can be used during the simulation.

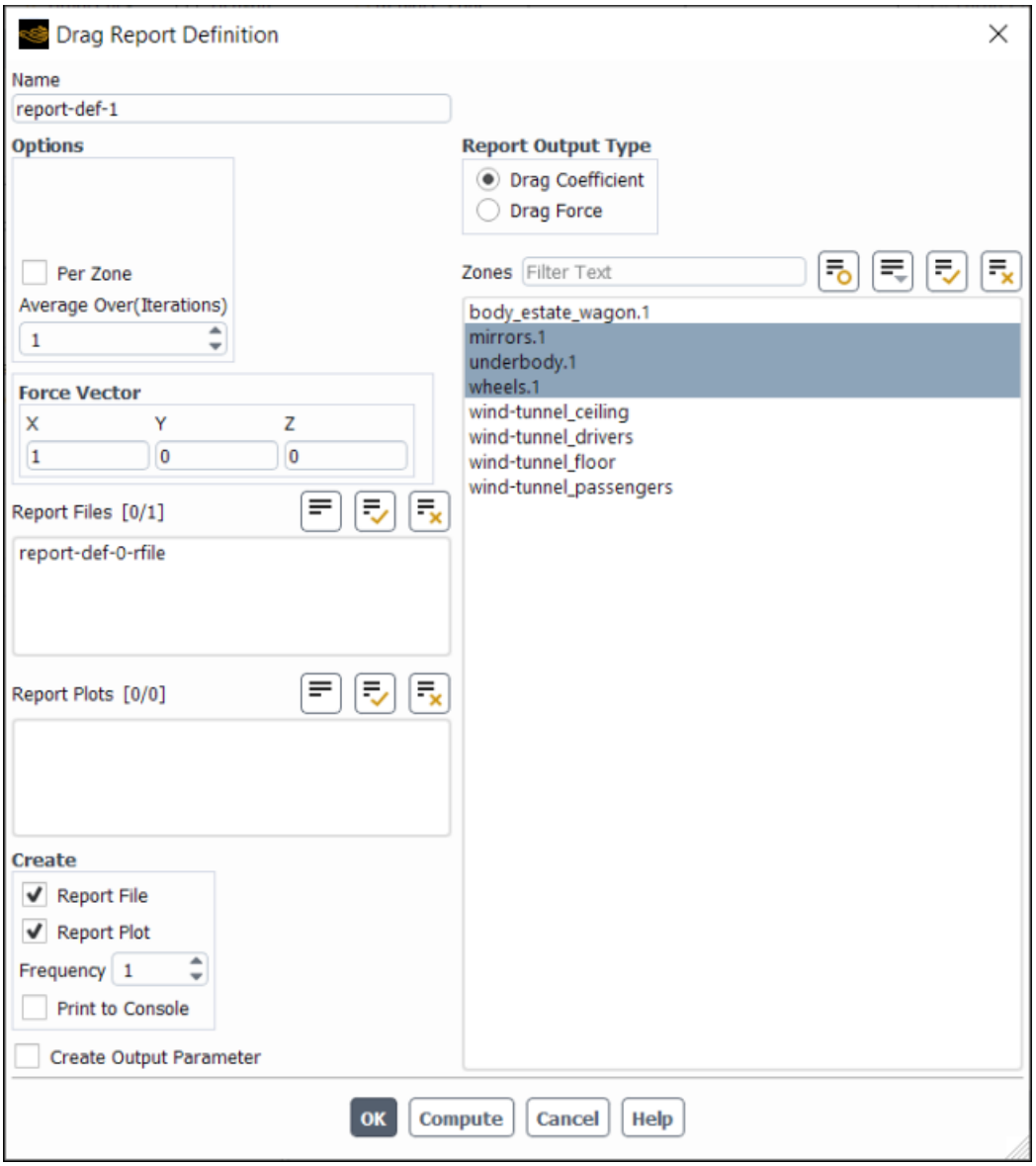
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 41.4: Force Report Definition Dialog Box



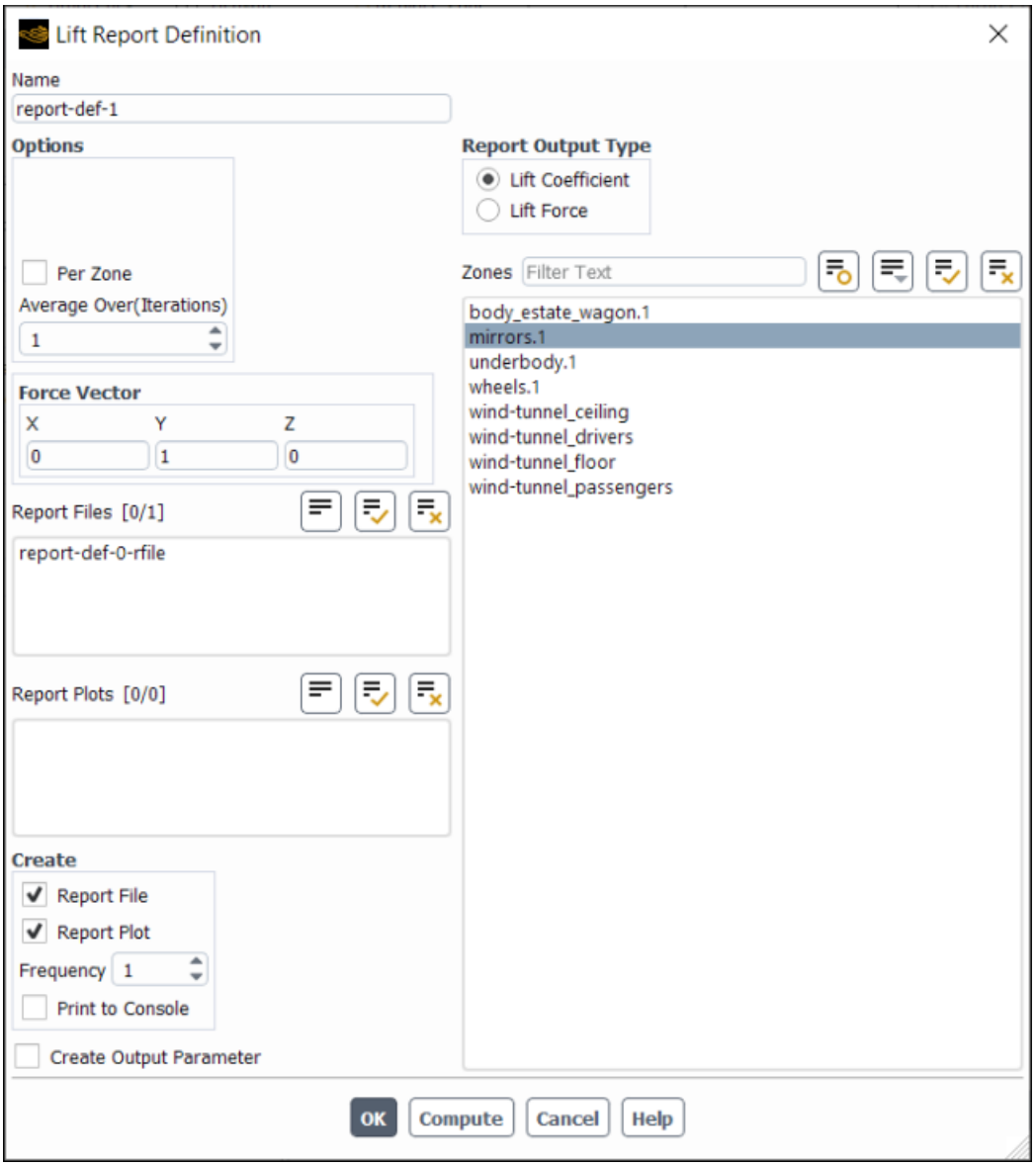
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 41.5: Drag Report Definition Dialog Box



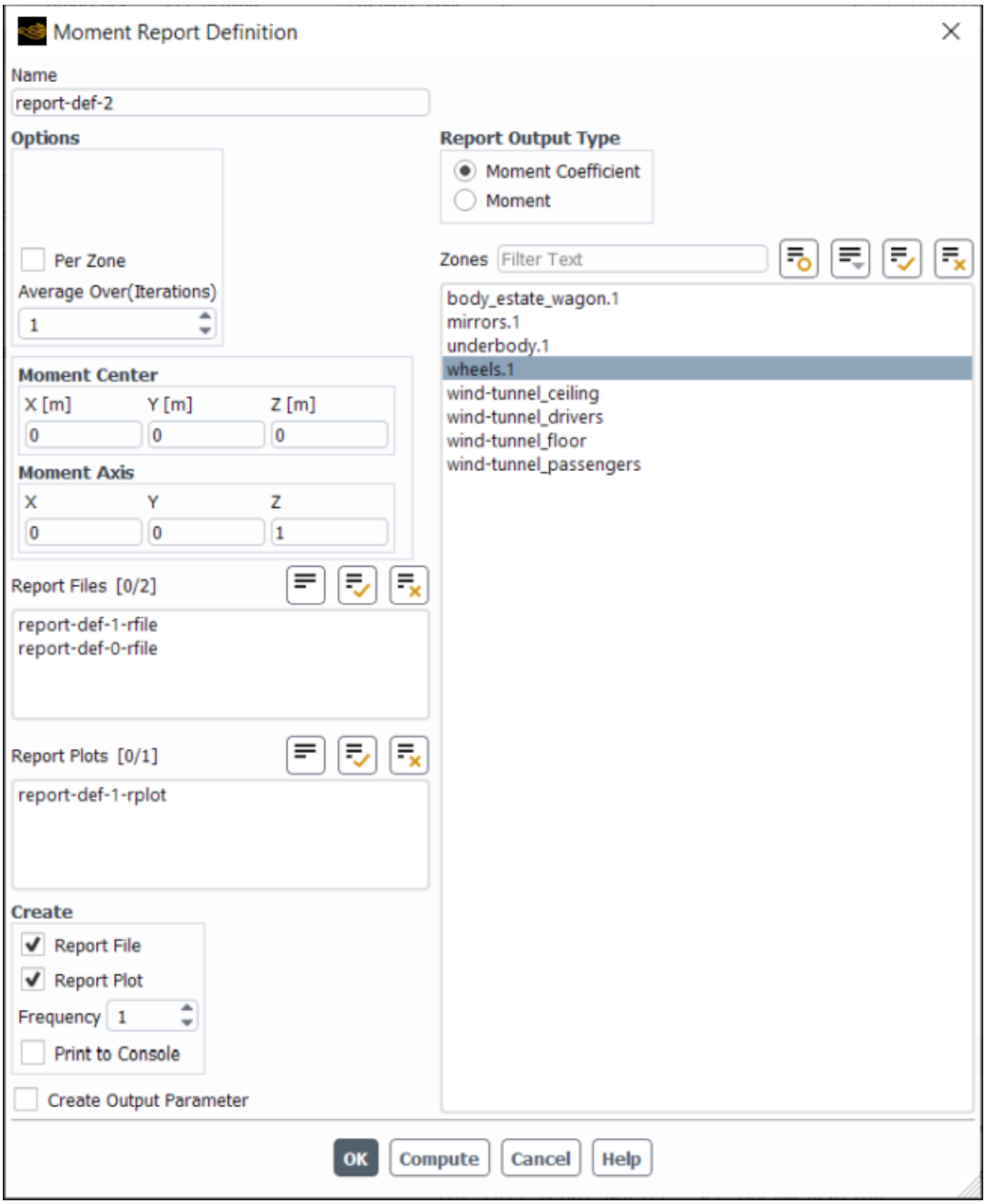
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 41.6: Lift Report Definition Dialog Box



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 41.7: Moment Report Definition Dialog Box



The common steps for creating a force or moment report definition are covered in Creating Report Definitions (p. 4098), and the force/moment report definition-specific steps are as follows:

1. If you want to report the force or moment coefficient data from individual wall zones rather than the net results from a group of wall zones, enable the **Per Zone** option.

2. Depending on the coefficient that will be monitored, perform one of the following steps:

   - In the **Drag Report Definition**, **Lift Report Definition** or **Force Report Definition** dialog box, enter the **X**, **Y**, and **Z** components of the **Force Vector** along which the forces will be computed. By default, the **Force Vector** for the drag coefficient is a unit vector in the x direction, whereas for the lift coefficient it is a unit vector in the y direction.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- In the **Moment Report Definition** dialog box, enter the Cartesian coordinates (**X**, **Y**, and **Z**) of the **Moment Center**, about which moments will be computed. The default **Moment Center** is (**0**, **0**, **0**). You also need to enter the **X**, **Y**, and **Z** components for the **Moment Axis**, along which the moment coefficient will be calculated. By default, the **Moment Axis** is defined as a unit vector in the z direction.

3. Select the appropriate zone(s) for computation of the force/moment/coefficient(s).
4. Click **OK** to save the report definition settings.
5. If you need to revise the settings of a particular report definition after they have been saved, double-click the report definition in the tree.

41.2.1.4. Flux Report Definition

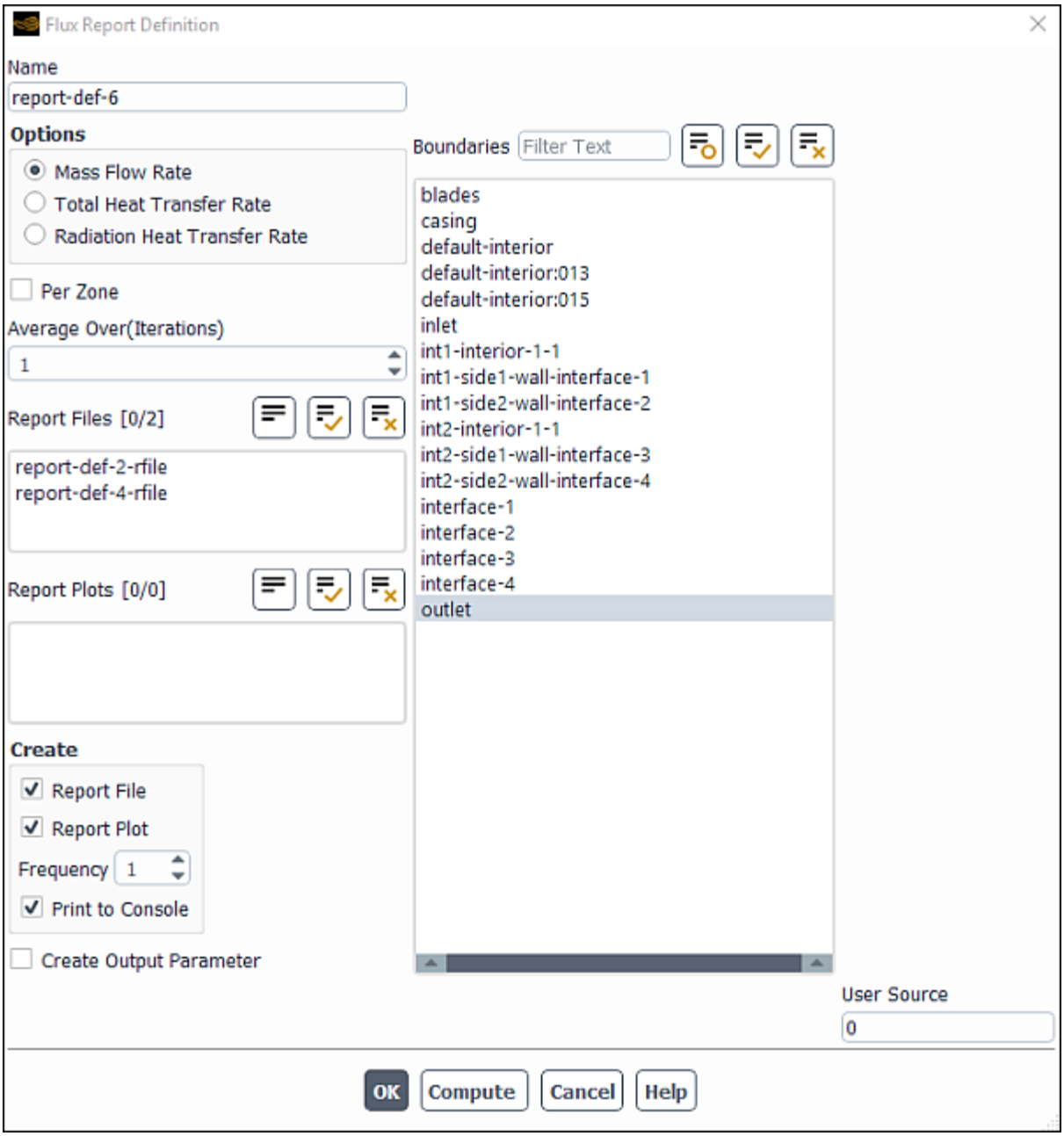
You can use the Figure 41.8: Flux Report Definition Dialog Box (p. 4112) to create flux report definitions. There are many different types of flux-based report definitions that you can create, including:

- Mass Flow Rate
- Total Heat Transfer Rate
- Total Sensible Heat Transfer Rate
- Radiation Heat Transfer Rate
- Pressure Work Rate
- Viscous Work Rate
- Film Mass Flow Rate
- Film Heat Transfer Rate
- DPM Mass Source
- DPM Enthalpy Source
- DPM Sensible Enthalpy Source

Solution → **Reports** → **Definitions** → **New** → **Flux Report**

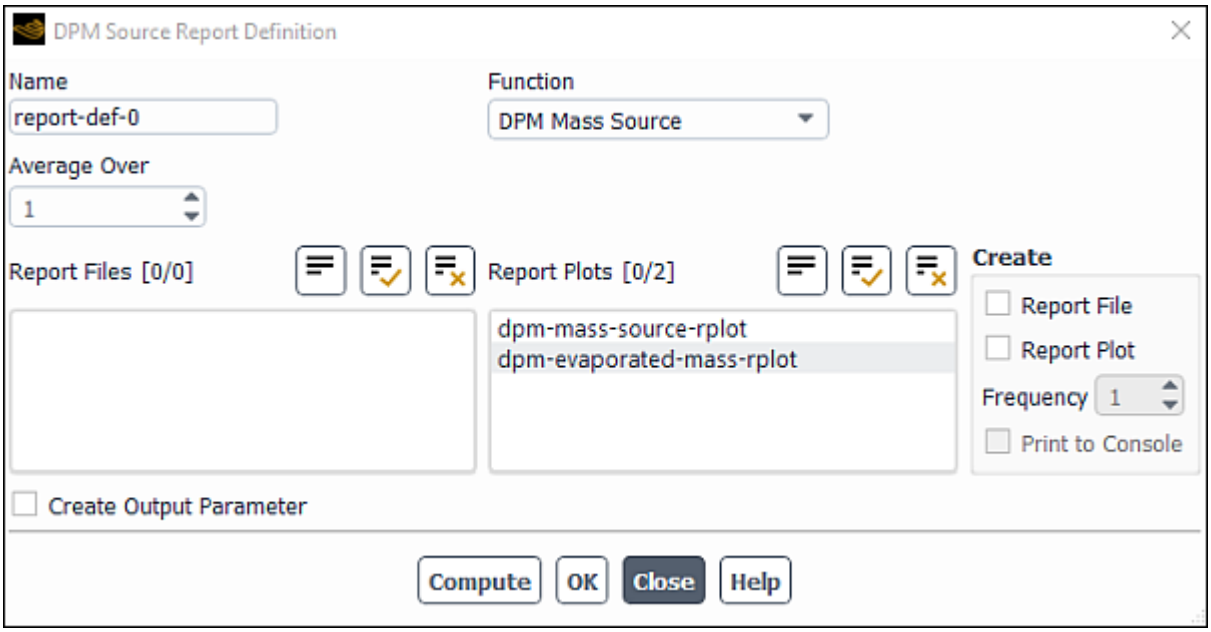
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 41.8: Flux Report Definition Dialog Box



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 41.9: DPM Source Report Definition Dialog Box



The common steps for creating a flux report definition are covered in Creating Report Definitions (p. 4098), and the flux report definition-specific steps are as follows:

1. Select or confirm the type of flux report.

   Note:

   If you are reporting **Mass Flow Rate** on any interior zones, you must enable **Per Zone** to see the mass flow rate value, otherwise it is reported as `0`.

   Fluent reports the mass flow rate at individual interior zones as `0` because there are instances where such non-zero computations would be illogical. For example, if you have a continuous fluid volume containing 10 interior zones, and you report the true mass flow rate through the domain at each interior zone, a summation of the mass flow rate at each interior zone would result in a mass flow rate that is 10 times higher than reality.

2. Select the appropriate boundaries for the flux calculations.

   Note:

   Fluxes can only be computed on boundaries and not on user defined surfaces such as planes, points, iso-surfaces, and so on.

3. Click **OK** to save the report definition settings.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. If you need to revise the settings of a particular report definition after they have been saved, double-click the report definition in the tree.

Note:

If you enable **Create Output Parameter**, the report definition will show the contributions of sources (when applicable) when the output parameter is printed to the **Console**.

For additional information on the available options for DPM source report definitions, see DPM Source Report Definition Dialog Box (p. 5444) in the reference guide.

41.2.1.5. Mesh Report Definitions

You can create mesh report definitions for the following:

- cell count (that is, the number of cells within one or more cell zones)
- face count (that is, the number of faces within one or more boundary zones)

Such reports can be useful for simulations that involve adaption, dynamic mesh remeshing, and so on.

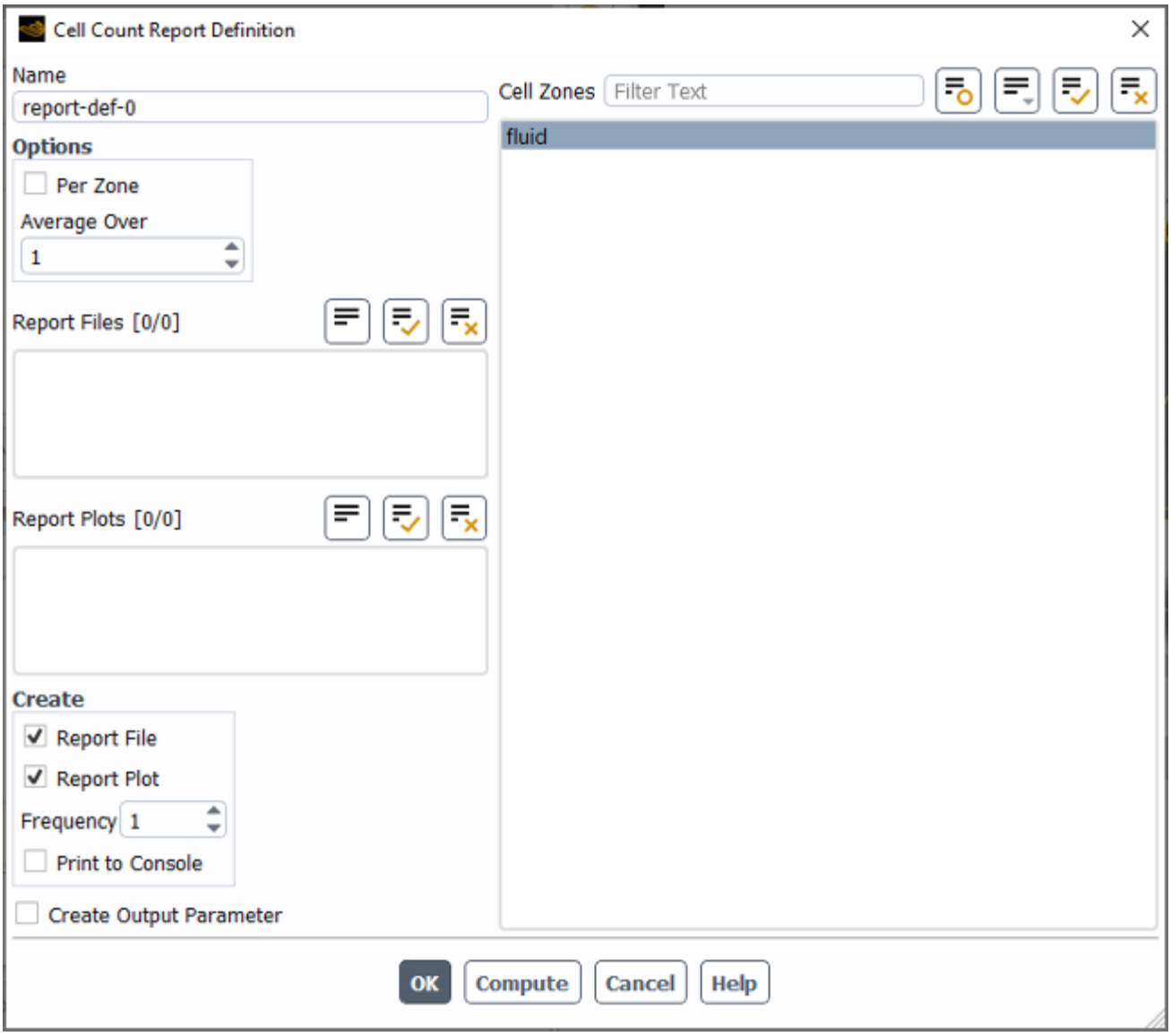
You can open the appropriate dialog box through the **Solution** tab of the ribbon.

Solution → **Reports** → **Definitions** → **New** → **Mesh Report**

Select either **Cell Count...** or **Face Count...** to open the Figure 41.10: Cell Count Report Definition Dialog Box (p. 4115) or Figure 41.11: Face Count Report Definition Dialog Box (p. 4116), respectively. Note that you can only access one of these report definition dialog boxes at a time, though multiple report definitions of both types can be used during the simulation.

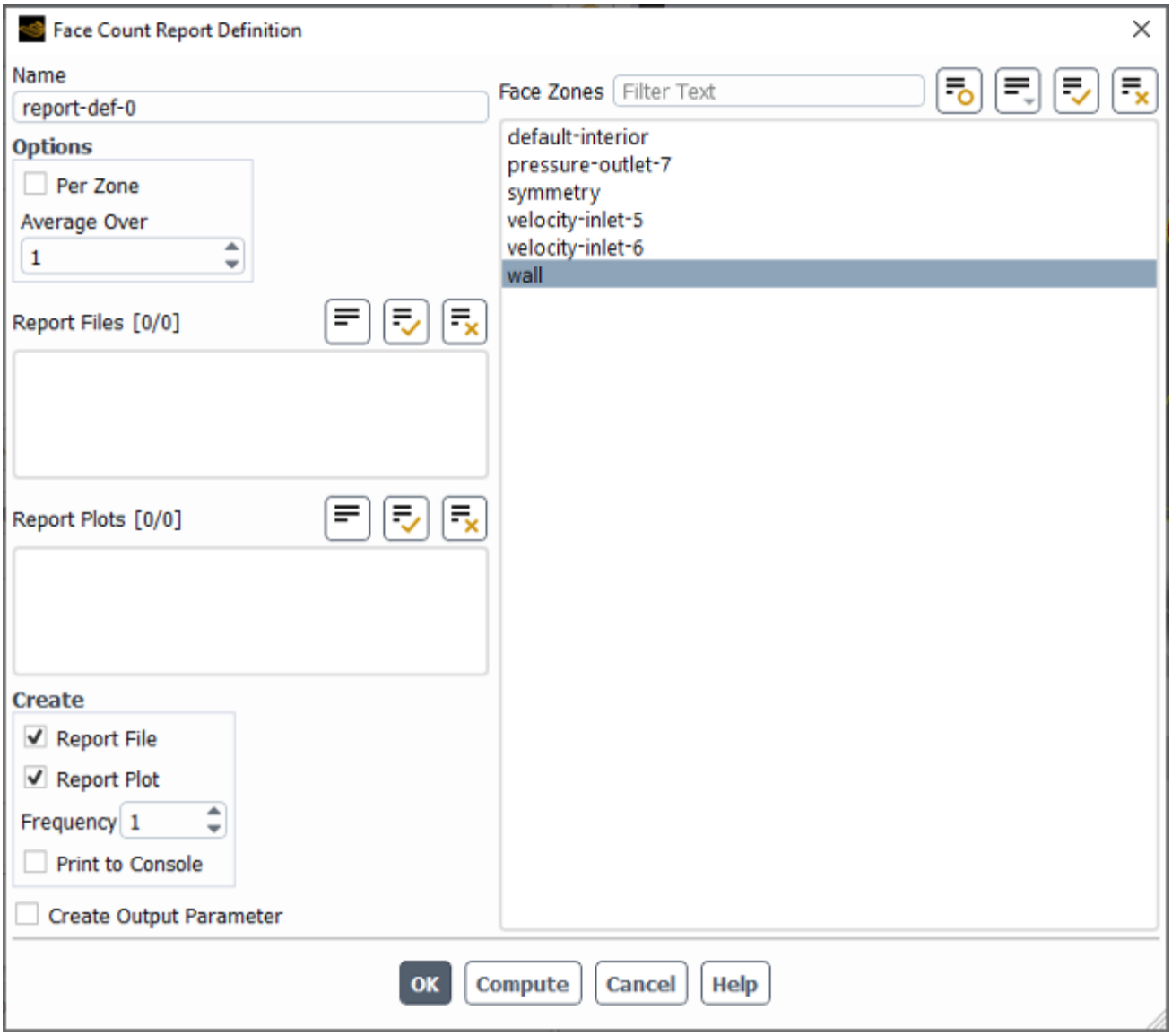
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 41.10: Cell Count Report Definition Dialog Box



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 41.11: Face Count Report Definition Dialog Box



The common steps for creating a mesh report definition are covered in Creating Report Definitions (p. 4098), and the mesh report definition-specific steps are as follows:

1. If you want to report the counts for individual zones rather than a net count from a group of zones, enable the **Per Zone** option.

2. Depending on what is being counted, perform one of the following steps:

   - In the **Cell Count Report Definition** dialog box, select the **Cell Zones** for which you want Fluent to report the number of cells.

   - In the **Face Count Report Definition** dialog box, select the **Face Zones** for which you want Fluent to report the number of faces.

3. Click **OK** to create the report definition and close the dialog box.

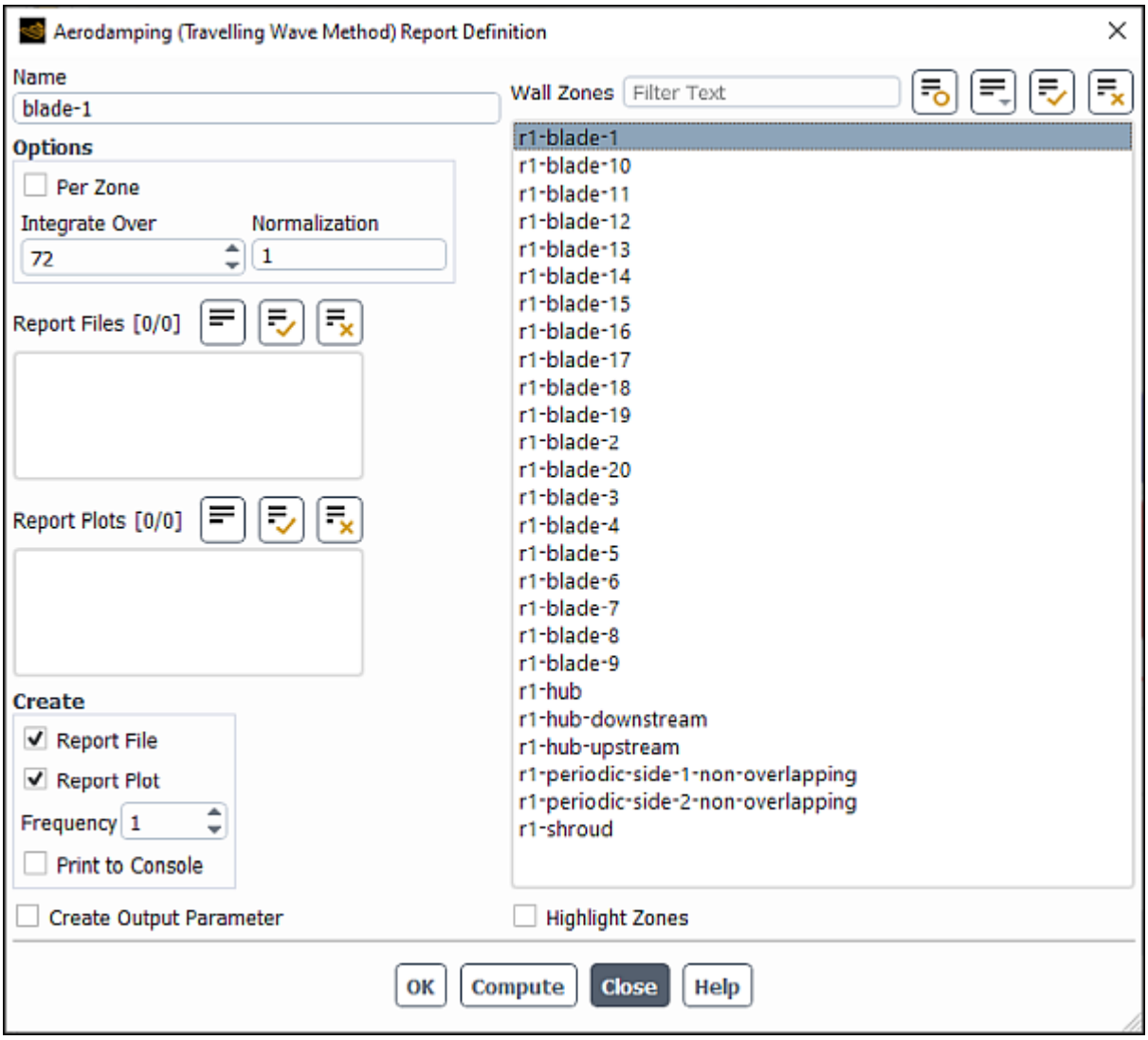
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

41.2.1.6. Aerodamping (Travelling Wave Method) Report Definition

You can use the Figure 41.12: Aerodamping (Travelling Wave Method) Report Definition Dialog Box (p. 4117) to create aerodamping report definitions.

Solution → Reports → Definitions → New → Aeromechanics Report → Aerodamping (Travelling Wave Method)...

Figure 41.12: Aerodamping (Travelling Wave Method) Report Definition Dialog Box



The common steps for creating an Aerodamping (Travelling Wave Method) report definition are covered in Creating Report Definitions (p. 4098), and the Aerodamping (Travelling Wave Method) report definition-specific steps are as follows:

1. If you are modeling multiple (main) blades, select **Per Zone** so that the damping value is calculated for each dynamic mesh zone (which encompasses one main blade and possibly accompanying splitter blades).

2. Set **Integrate Over** to the number of time steps over which to integrate when calculating the damping value; a suggested value is the number of time steps per period.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The number of time steps per period is shown in the **Run Calculation** task page.

Note:

If you change the number of time steps per period in the Run Calculation settings, you probably need to make a corresponding change to the **Integrate Over** setting in the monitor definition.

For additional information on aerodamping, see Aerodynamic Damping (Blade Flutter Analysis) (p. 1972).

41.2.1.7. DPM Report Definition

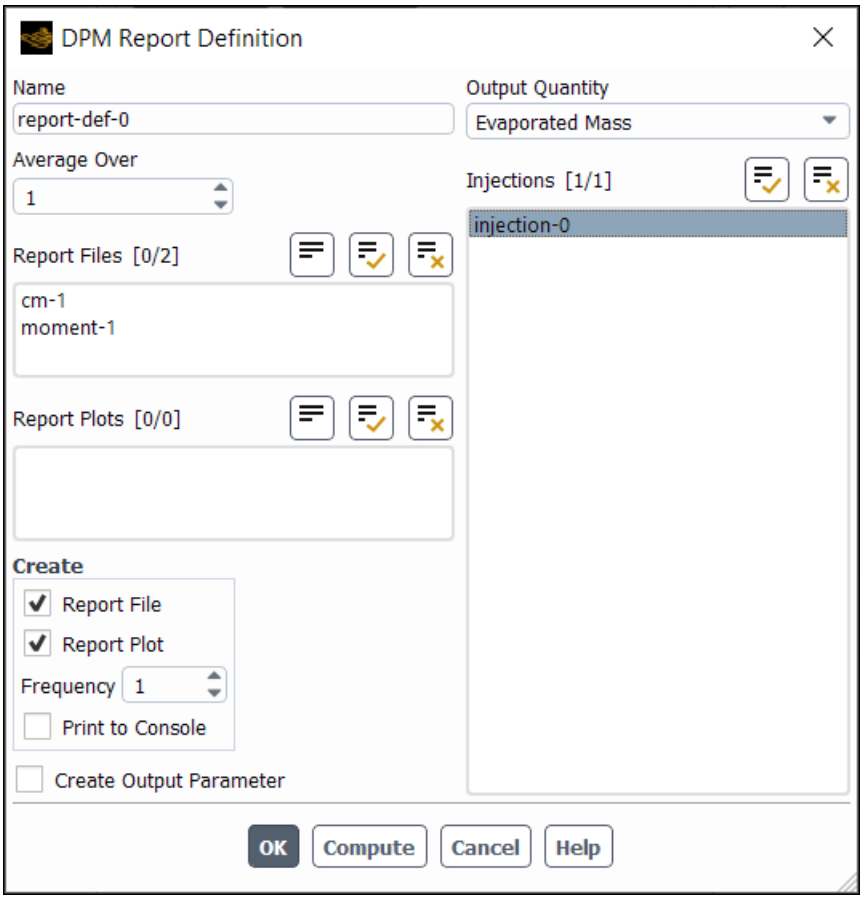
You can use the DPM Report Definition dialog box to create DPM report definitions. The following types of DPM reports are available:

- **Injected Mass** (*unsteady particle tracking only*): Reports the total DPM mass injected into the domain.
- **Mass in Domain** (*unsteady particle tracking only*): Reports the total DPM mass present in the domain.
- **Mass in Fluid** (*unsteady particle tracking only*): Reports all DPM mass currently residing as free-stream particles in the domain. This option is available only with the **wall-film** DPM boundary condition.
- **Mass in Film** (*unsteady particle tracking only*): Reports all DPM mass currently residing in Lagrangian wall-film. This option is available only with the **wall-film** DPM boundary condition.
- **Escaped Mass**: Reports the DPM mass that has left the domain through a certain boundary or boundaries.
- **Evaporated Mass**: Reports the total evaporated DPM mass for the selected injection(s). The quantity accounts for fully evaporated particles as well as all mass that has evaporated from any other particles. For unsteady tracking, this includes particles that are still present in the domain.
- **Penetration Length** (*unsteady particle tracking only*): Reports the total penetration length in meters.

Solution → **Reports** → **Definitions** → **New** → **DPM Report**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 41.13: DPM Report Definition Dialog Box



The common steps for creating a DPM report definition are covered in Creating Report Definitions (p. 4098), and the DPM report definition-specific steps are as follows:

1. Select or confirm the type of DPM report.
2. For the **Escaped Mass** report, if you want to have the escaped mass calculated per injection, enable the text command `report/dpm-zone-summaries-per-injection`.

   The **Injections** selection list will then appear in the **DPM Report Definition** dialog box.
3. From the **Injections** list, select the appropriate injections for the DPM report.

   Note:

   For the **Escaped Mass** report, this step is relevant only if you have enabled the calculation of the escaped mass per injection.
4. For the **Escaped Mass** report, select the boundary(s) where the injected particles exit the domain from the **Boundaries** selection list.
5. For the **Penetration Length** report, you can specify the **Particles Mass Fraction**. You can also enable **User-Specified Origin and Direction** to specify those coordinates.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. For unsteady particle tracking, you can select **Show Mass Flow/Change Rate** (default) for the **Injected Mass**, **Mass in Domain**, **Mass in Fluid**, **Mass in Film**, **Escaped Mass**, and **Evaporated Mass** reports if you want to create a report for the mass flow rate rather than the total mass.

7. Click **OK** to save the report definition settings.

8. If you need to revise the settings of a particular report definition after they have been saved, double-click the report definition in the tree.

Important:

For unsteady particle tracking, if you want to select one or more injection(s) for any **Escaped Mass** report, you must enable the `report/dpm-zone-summaries-per-injection` text command before the first particles are injected; otherwise, some of the reported values may become inconsistent.

For additional information on the available options, see the DPM Report Definition Dialog Box (p. 5441) in the reference guide.

41.2.1.8. User Defined Report Definition

User defined report definitions allow you to create functions that return values that can be plotted, printed, written, or used in other expressions. User defined report definitions can also be used in the **Convergence Conditions** (Convergence Conditions (p. 3696)).

Creating a user defined report definition is a two step process:

1. Create a user defined report definition function.

2. Hook the user defined report definition function to the user defined report definition.

41.2.1.8.1. User Defined Report Definition Function

User defined report definition functions are defined with the `DEFINE_REPORT_DEFINITION_FN` macro, which takes the function name as the argument. The function must return a real value. Once written, it should be compiled and loaded the same way as other user-defined functions, see Interpreting and Compiling UDFs in the *Fluent Customization Manual* for more information.

The function body should be constructed similarly to any C function that returns a real value. Any valid C code can be included in the function body.

`DEFINE_REPORT_DEFINITION_FN` can use all of the APIs available for other user-defined functions, including:

- Last evaluated values of all report definitions.
- Current values of all input parameters that have been set to be used inside user-defined functions.

See `DEFINE_REPORT_DEFINITION_FN` in the *Fluent Customization Manual* for information on writing user defined report definition functions.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

41.2.1.8.2. User Defined Report Definition Function Hooking

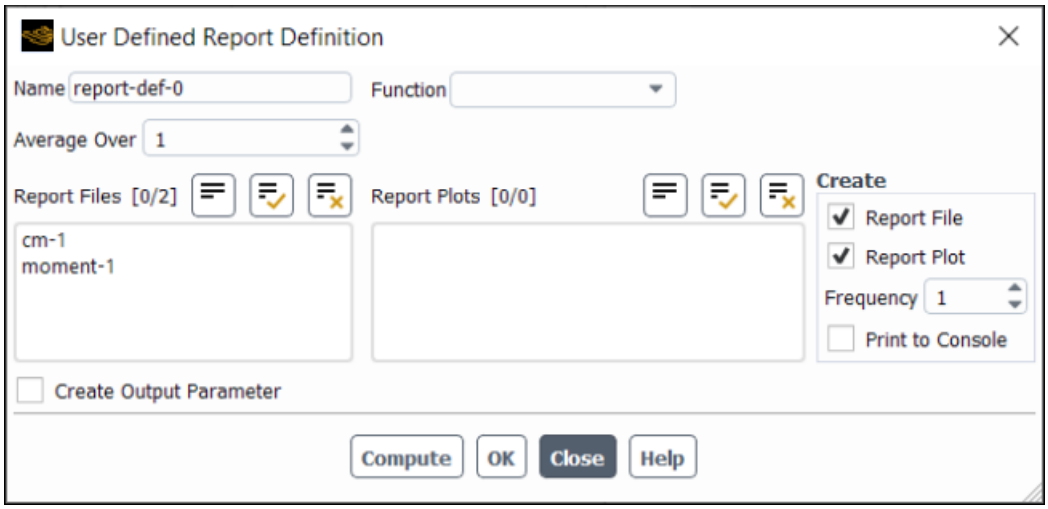
With the user defined report definition function written, you must hook this function to the user defined report definition.

The common steps for creating a user defined report definition are covered in Creating Report Definitions (p. 4098), and the user defined report definition-specific steps are as follows:

1. Open the **User Defined Report definition** dialog box.

   Solution → Report Definitions → New → User Defined

   Figure 41.14: User Defined Report Definition Dialog Box



2. Select your function from the **Function** drop-down list.

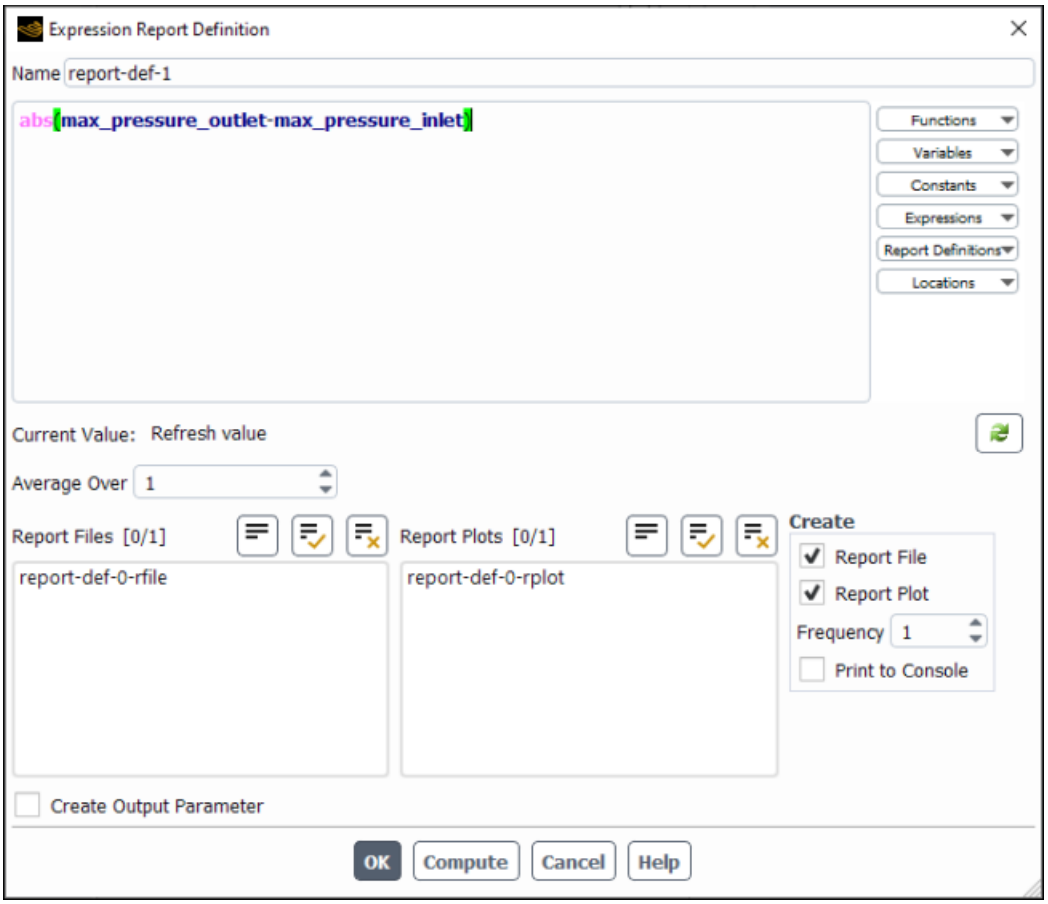
41.2.1.9. Expression Report Definition

You can create expressions from report definitions, allowing you to combine report definitions with mathematical operators to create a new report definition that can be written or plotted. For example, you can create a pressure-drop report for the difference in area-weighted pressures between an inlet and an outlet. You can also use an expression report definition to modify a single report definition with expression functions such as a mathematical function or reduction operation.

Solution → Reports → Definitions → New → Expression

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 41.15: Expression Report Definition Dialog Box



The common steps for creating an expression report definition are covered in Creating Report Definitions (p. 4098), and the expression report definition-specific steps are as follows:

1. Create the **Report Definition**(s) that you will use to create an **Expression**.

2. Use the drop-downs to the right of the **Definition** box to select the report definition(s) and other variables and functions, as required to define your desired expression report definition. *You can also directly enter the expression into the **Definition** box—the drop-downs are available to make the process easier.*

   Note:

   When you use curly brackets "{}" to specify the name of a defined object, such as a report definition or a cell register, automatic suggestions are disabled and you must provide the display name of that object.

3. Click **OK** to create the expression report definition.

Refer to Fluent Expressions Language (p. 1037) for additional information on creating expressions.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

41.2.2. Report Files and Report Plots

Report Files and **Report Plots** are collections of report definitions that are written to a single file (report file) or plotted to a graphics window (report plot) at specified intervals. In steady-state cases, report files are evaluated at the specified iteration, whereas in transient cases you have the option to specify the evaluation frequency based on iterations or timesteps.

*Report definitions in the same **Report Plot** must have the same units.*

Report Files Format:

```
"<report-file-name>"
"Iteration"|"Timestep" "<first-report-def-name> etc.."
("Iteration"|"Timestep" "<report-def-1-name>" "<report-def-2-name>"...)
<Iteration/Timestep> <report-def-1-value> <report-def-2-value>...
```

Report definitions included in a report plot are plotted in the specified window at the specified frequency. It is possible to plot the same report in multiple windows. Both report plots and report files provide the option to print to the console. You can deactivate the writing and plotting of individual report files and plots for a particular calculation; deactivated report files and plots can be re-activated at a later time.

Important:

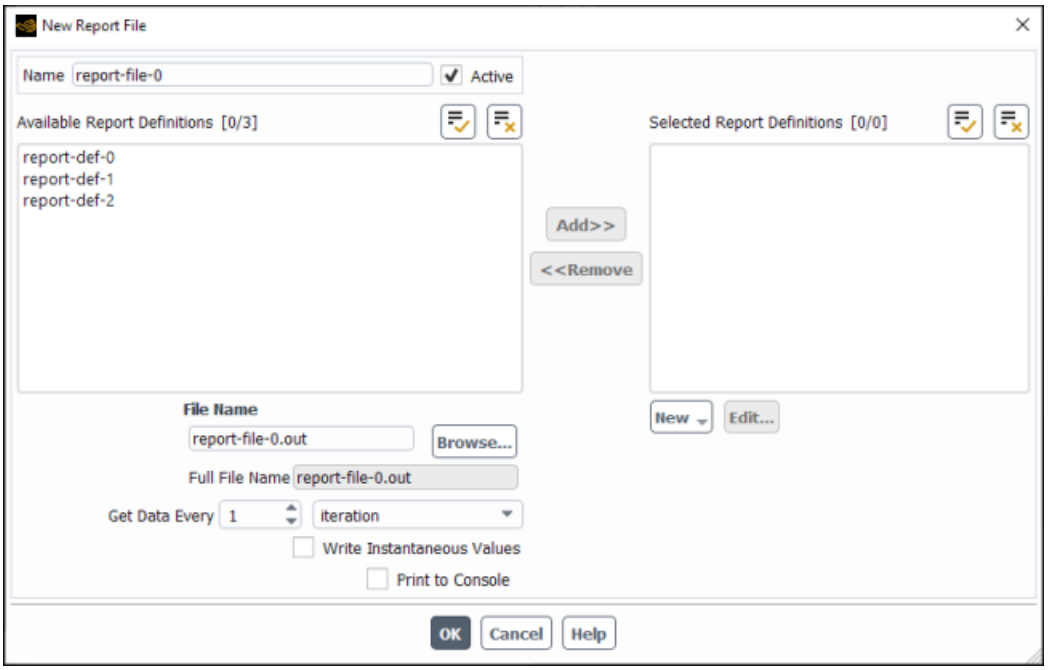
You can control whether **Report Plot** and **Report File** are enabled by default for new report definitions by using the **Automatically Plot and File** option under **Report Definitions** in the **Simulation** branch of **Preferences** (**File**>**Preferences...**).

41.2.2.1. Creating Report Files

In order to create a report file you must have first created a report definition.

Solution → **Monitors** → **Report Files** **New...**

Figure 41.16: New Report File Dialog Box



1. Provide a name for how you want this report file to appear in the Fluent tree.

   *You have the option to deactivate the writing of this report file by de-selecting the **Active** check box to the right of the **Name**.*

2. Select one or more report definitions from **Available Report Definitions** and click **Add>>**, to specify the reports you want written to the same file.

3. Provide an appropriate name in the **File Name** text box. The **Full File Name** that appears below the **File Name** field is created with a run index and iteration/time step index, appended to the base name. *It also provides the full path to the file. You can use the keyboard arrow keys to scroll within this field (<, >).*

4. Specify the frequency at which you want the report definitions written to the file.

   Note:

   In a transient simulation, if you specify an **Average Over** value in a report definition, you must set **Get Data Every** to **time-step** or **flow-time**. If you set **Get Data Every** to **iteration**, Fluent will use the instantaneous value of the report definition.

5. (Optional) Enable the **Write Instantaneous Values** option to retain the flexibility of changing the **Average Over** value of a report definition in a future session (otherwise, Fluent would not have the data necessary to re-compute the average over values). See Moving Average Monitors (p. 4129) for additional information.

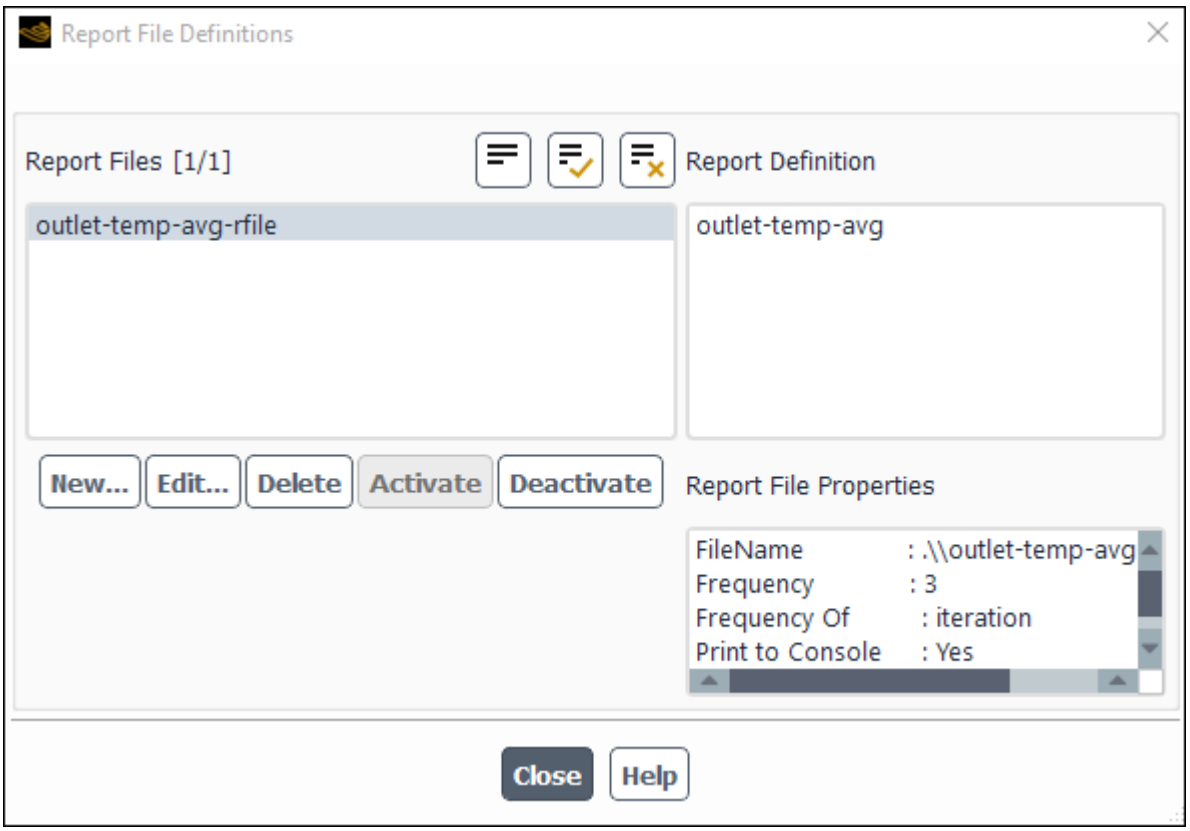
6. Click **OK** to save the new report file.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You have the option to view—and edit —properties of report files that you created previously by selecting **Edit...** when you right-click **Report Files** in the tree, which opens the Figure 41.17: Report File Definitions Dialog Box (p. 4125).

Note that you can also right-click an individual report definition to modify it directly.

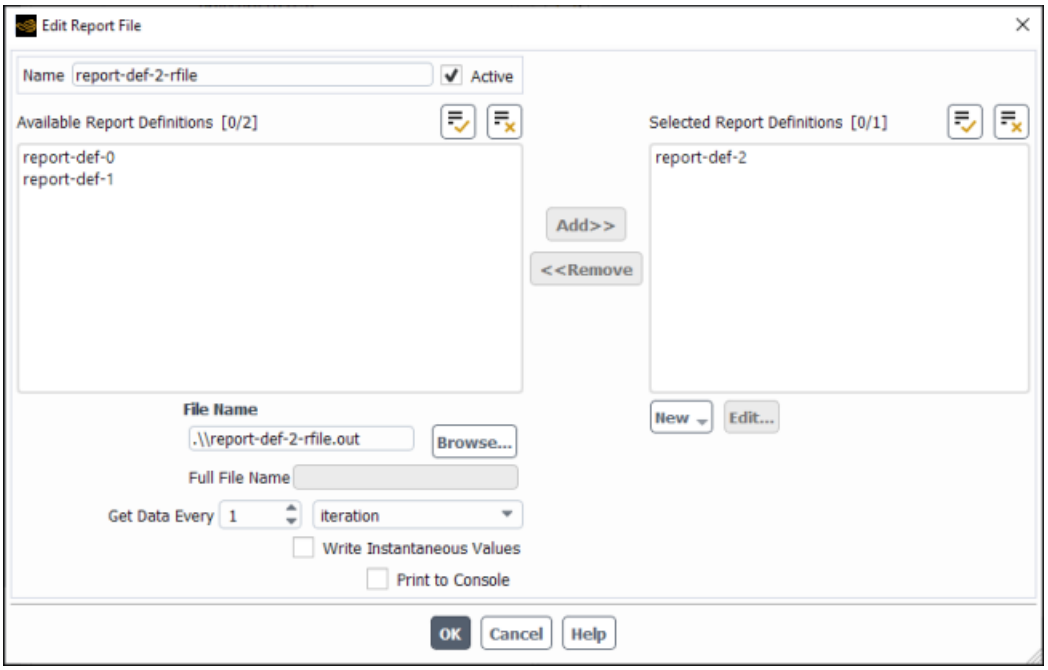
Figure 41.17: Report File Definitions Dialog Box



Selecting a report file to modify opens the Figure 41.18: Edit Report File Dialog Box (p. 4126).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 41.18: Edit Report File Dialog Box

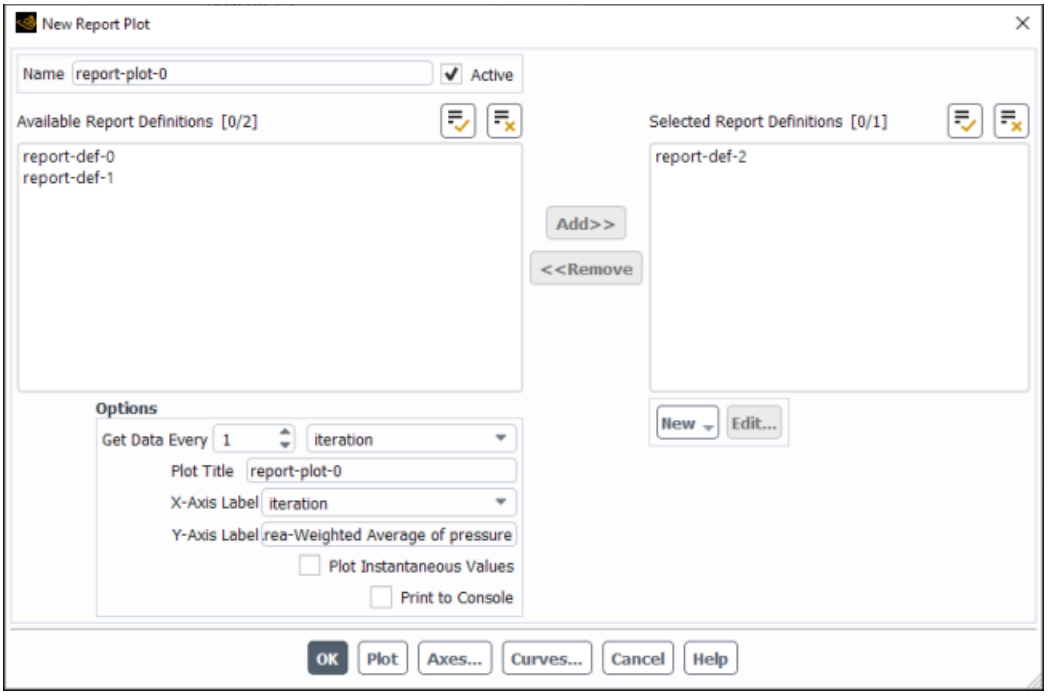


41.2.2.2. Creating Report Plots

In order to create a report plot you must have first created a report definition.

Solution → Monitors → Report Plots → New...

Figure 41.19: New Report Plot Dialog Box



1. Provide a name for how you want this report plot to appear in the Fluent tree.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

*You have the option to deactivate the plotting of this report plot by de-selecting the **Active** check box to the right of the **Name**.*

2. Select one or more report definitions from **Available Report Definitions** and click **Add>>**, to specify the reports you want plotted in the same window.

3. Provide an appropriate names in the **Plot Title** and **Y-Axis Label** group boxes.

4. Specify the frequency at which you want the report definitions plotted.

   Note:

   In a transient simulation, if you specify an **Average Over** value in a report definition, you must set **Get Data Every** to **time-step** or **flow-time**. If you set **Get Data Every** to **iteration**, Fluent will use the instantaneous value of the report definition.

5. (Optional) Enable the **Plot Instantaneous Values** option to include the instantaneous values of the report definition along with the averaged values (averaged based on the **Average Over** value specified in the included report definition).

6. Click **OK** to save the new report plot.

   Note:

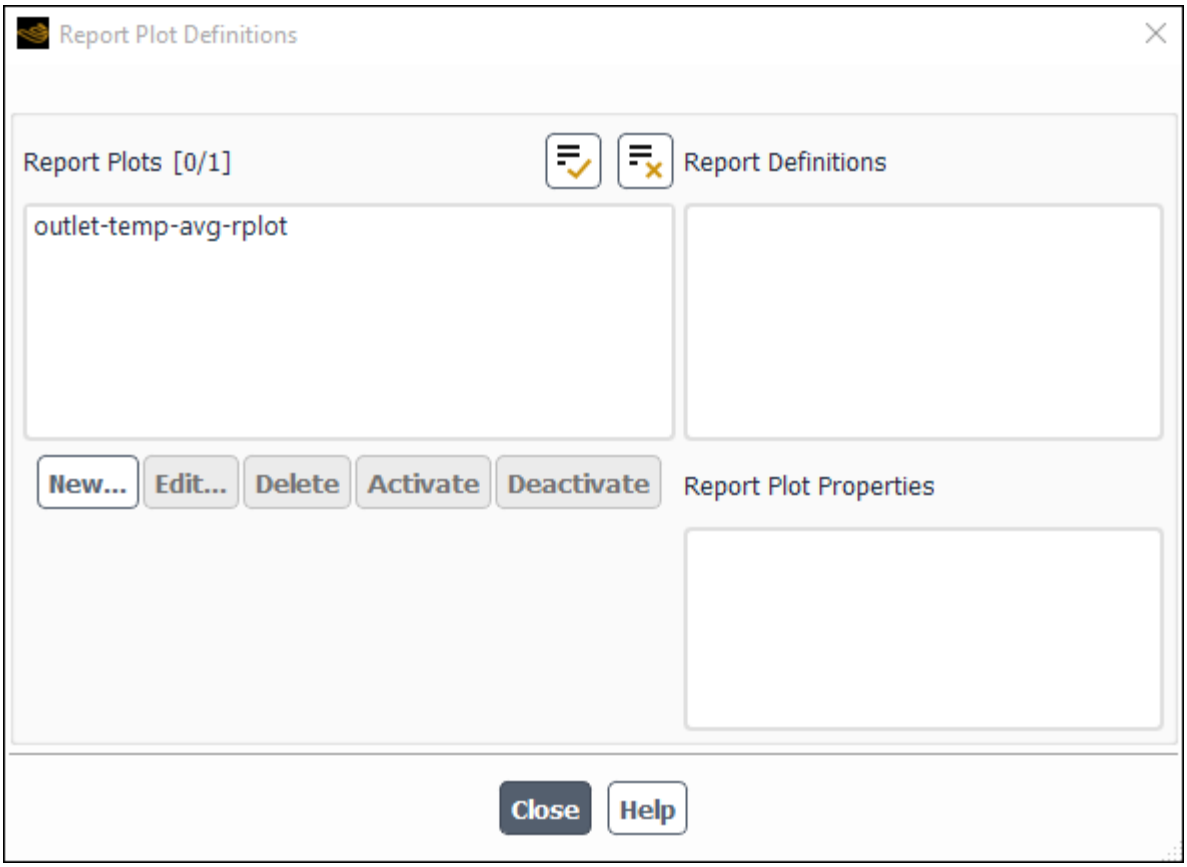
   By Default, Fluent only displays 1000 plot points when a report plot generated in a previous Fluent session is plotted in a new Fluent session. However, you can increase this limit in **Preferences**. Specify the desired maximum number of plot points in the **Report Plot Data History Size** field in the **Simulation** branch of **Preferences** (**File**>**Preferences...**>**Simulation**>**Report Definitions**>**Report Plot Data History Size**).

You have the option to edit report plots that you created previously by selecting **Edit...** when you right-click **Report Plots** in the tree, which opens the Figure 41.20: Report Plot Definitions Dialog Box (p. 4128).

Once you begin iterating, Fluent displays report plots in the graphics window. If you close the graphics window containing a plot, you can redisplay it by right-clicking the report plot in the Outline View tree and selecting **Plot**.

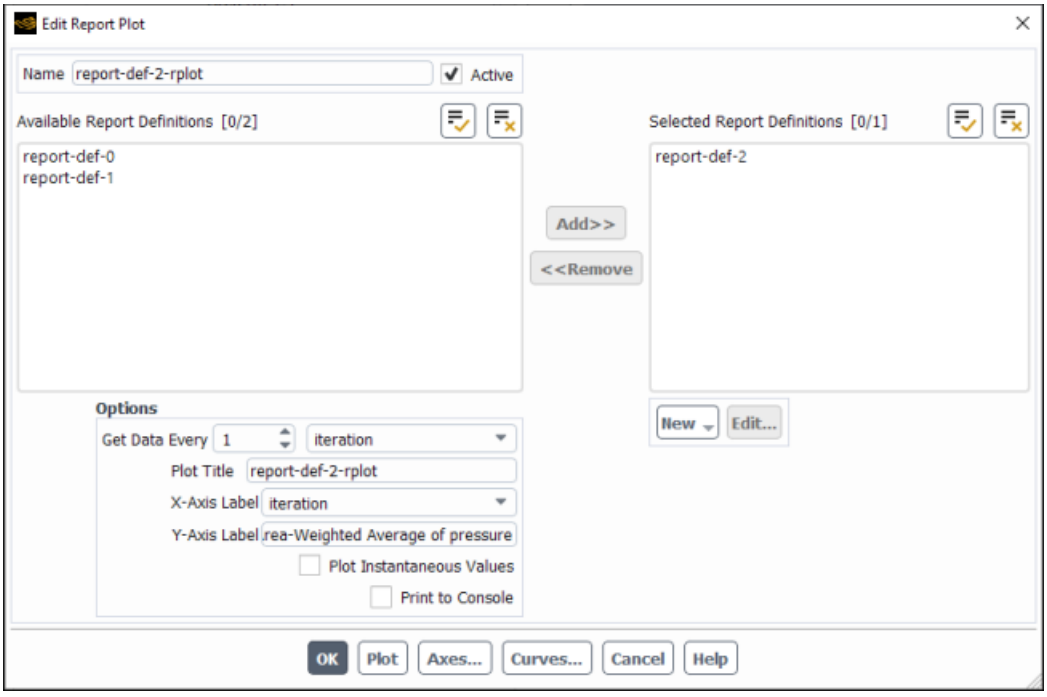
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 41.20: Report Plot Definitions Dialog Box



Selecting a report plot to modify opens the Figure 41.21: Edit Report Plot Dialog Box (p. 4128).

Figure 41.21: Edit Report Plot Dialog Box



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

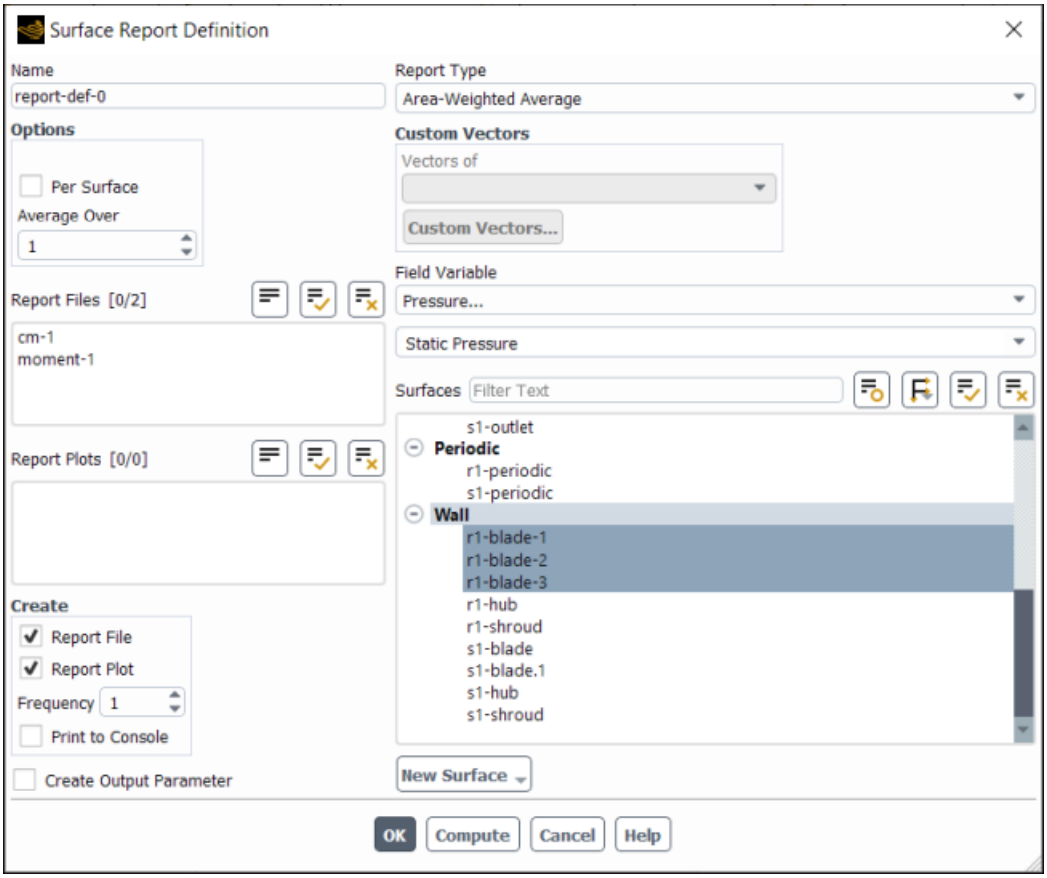
41.2.2.3. Moving Average Monitors

You can vary the number of timesteps or iterations included in the averaging computations for report definitions. Changing the averaging interval can help you smooth fluctuations in an observed quantity or let you see fluctuations that were previously smoothed out.

To control the averaging interval and maintain the ability to do so in a new session:

1. Create a report definition.

   Solution → Report Definitions → New → <Report Category> → <Report Type>



2. Specify interval used for averaging in the **Average Over (Timesteps | Iterations)** field. *The default value of 1 means that you always have the instantaneous value, however, if you plan on modifying the **Average Over** value later, it is recommended that you begin with an **Average Over** of at least 2 when you create the report definition. This ensures that its containing report file will have the **Write Instantaneous Values** option enabled, (assuming you leave **Retain Instantaneous Values** enabled after increasing the **Average Over** value).*

   Important:

   When you increase the reporting interval to a value greater than 1, the **Retain Instantaneous Results** check box appears. You must leave this op-

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

tion checked if you want the flexibility to change the averaging interval in a future session.

3. Confirm **Report Plot** and **Report File** are enabled in the **Create** group box. *Alternatively, you can include the report definition in an existing report file or relevant report plot.*

Important:

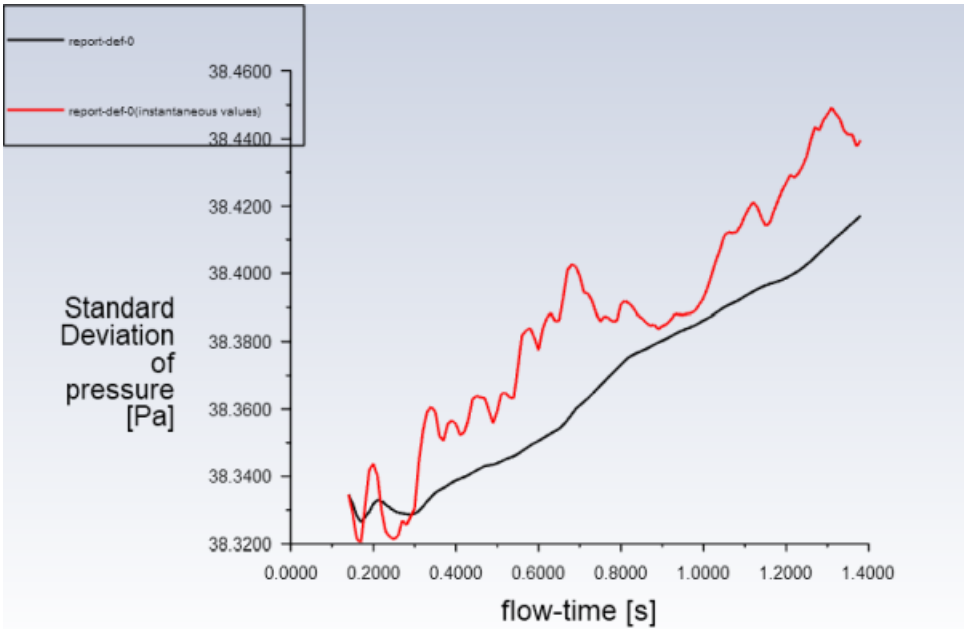
You can control whether **Report Plot** and **Report File** are enabled by default using the **Automatically Plot and File** option under **Report Definitions** in the **Simulation** branch of **Preferences** (**File**>**Preferences...**).

4. Click **OK**.

Once you create your report definitions for monitoring the solution progress, you can change the reporting interval by re-opening the report definition dialog box, modifying the **Average Over (Timesteps | Iterations)** field, and clicking **OK**. The updated averaging interval will be used when you re-plot the report definition using its associated report plot.

In addition to the ability vary the averaging interval, you can also plot the instantaneous values with the averaged values, as shown in Figure 41.22: Plot with Instantaneous and Averaged Values (p. 4130).

Figure 41.22: Plot with Instantaneous and Averaged Values

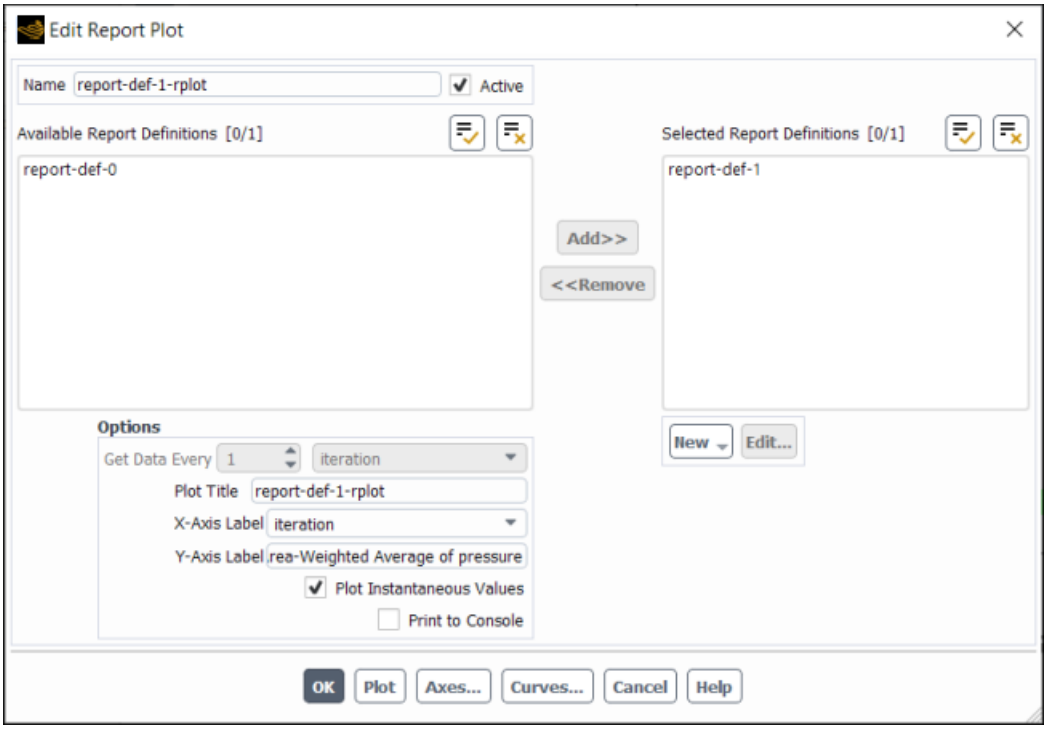


To include the instantaneous values with the average:

1. Open the **Report Plot** dialog box that includes the quantity of interest (*by default, it will have the same name as the corresponding report definition*).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution → **Monitors** → **Report Plot** → **<report-plot-name>** **Edit...**



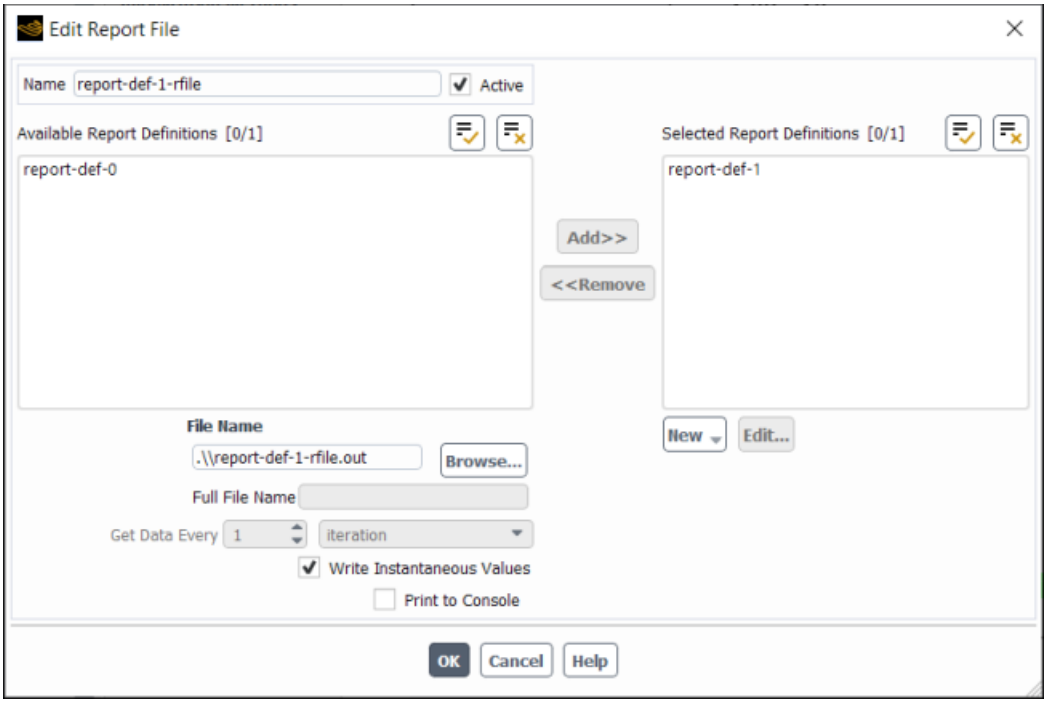
2. Enable **Plot Instantaneous Values**.
3. Click **Plot**.

To ensure that you can include instantaneous values in plots for these data when this case is opened in a new session:

1. Open the **Report File** that includes the quantity of interest.

Solution → **Monitors** → **Report File** → **<report-file-name>** **Edit...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



2. Enable **Write Instantaneous Values**.
3. Click **OK**.

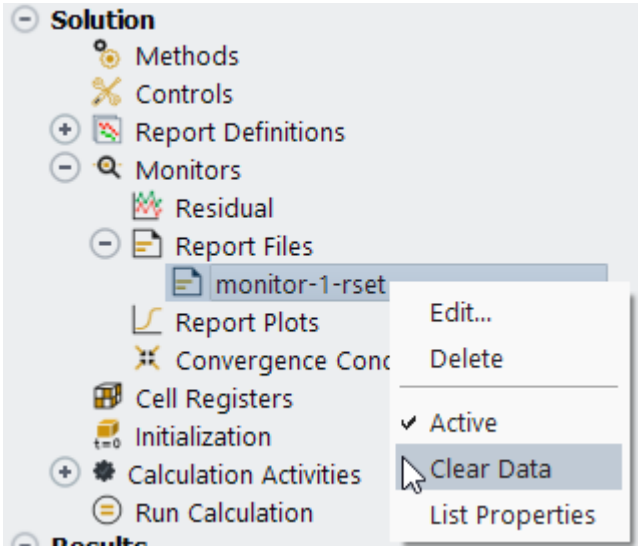
41.2.2.4. Clearing File and Plot Histories

In certain situations you may want to clear the history of report files or plots, without deleting the objects or creating new plots or files for the same quantities.

Solution → Monitors → Report Files → <report-file-name> → Clear Data

Solution → Monitors → Report Plots → <report-plot-name> → Clear Data

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



You can do this by right-clicking the report file or plot that you want to clear and selecting **Clear Data**. For report files, this option deletes the file from the system. A new report file of the same name will be created once you begin iterating. For report plots, this option clears the data from the system memory, so the plot will be empty until you begin iterating again.

41.3. Creating Output Parameters

You can create output parameters, which allow you to compare reporting values for different cases, or include reporting values in the function minimized by the mesh morpher/optimizer. These are single values generated by a variety of reports:

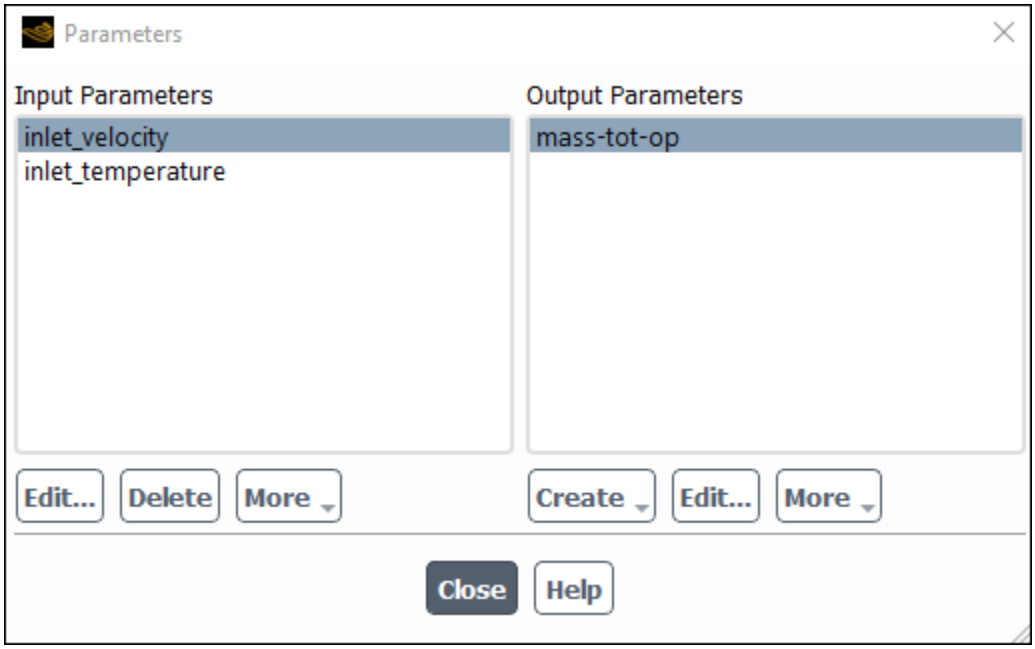
- Fluxes (Fluxes Through Boundaries (p. 4135))
- Forces (Forces on Boundaries (p. 4145))
- Surface integrals (Generating a Surface Integral Report (p. 4151))
- Volume integrals (Generating a Volume Integral Report (p. 4153))
- Surface report definitions (Surface Report Definitions (p. 4103))
- Volume report definitions (Volume Report Definitions (p. 4104))
- Drag report definitions (Force and Moment Report Definitions (p. 4106))
- Lift report definitions (Force and Moment Report Definitions (p. 4106))
- Moment report definitions (Force and Moment Report Definitions (p. 4106))
- Force report definitions (Force and Moment Report Definitions (p. 4106))
- Flux report definitions (Flux Report Definition (p. 4111))
- Cell count report definitions (Mesh Report Definitions (p. 4114))
- Face count report definitions (Mesh Report Definitions (p. 4114))

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Aerodamping (Travelling Wave Method) report definitions (Aerodamping (Travelling Wave Method) Report Definition (p. 4117))
- DPM report definitions (DPM Report Definition (p. 4118))
- User Defined report definitions (User Defined Report Definition (p. 4120))
- Expression report definitions (Expression Report Definition (p. 4121))
- Named expressions (Named Expressions (p. 1054)).

Click **Parameters...** in the **User Defined** ribbon tab (**Field Functions** group box) to open the Parameters Dialog Box (p. 5338), where a list of any previously created input parameters is available.

User Defined → **Field Functions** → **Parameters...**



The list of **Input Parameters** is populated after performing the steps outlined in Defining and Viewing Parameters (p. 1309). The output parameters that you create will be listed under **Output Parameters**.

You can define the output parameters using the reporting dialog boxes, as described in Creating Report Definitions (p. 4098). The reporting dialog boxes are accessible from the **Definitions** drop-down list in the **Solution** ribbon tab (**Reports** group box). Clicking the **Create** drop-down list under **Output Parameters** in the **Parameters** dialog box allows you to select **From Report Definitions...**, which opens the Figure 41.1: Report Definitions Dialog Box (p. 4102). From there you can create any type of report definition.

Note:

Using the **Per Surface** or **Per Zone** option with multiple selected surfaces/zones in conjunction with the **Create Output Parameter** option, results in a single output parameter. The output parameter will show a single combined value for all the selected surfaces/zones.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ensure that the **Average Over** field in any such report definitions is set to 1. An **Average Over** value greater than one, in combination with the **Per Surface | Per Zone** option gives invalid combined net values, because the specified **Average Over** setting is not taken into account in the net value computation.

Once you have saved your output parameters, you can modify their definitions by selecting the parameter in the **Output Parameters** list and clicking **Edit...**. This will open the report dialog box where you can make your changes.

In addition, you can select any of the following commands under the **More** drop-down list:

Delete

removes the selected output parameter from the list of **Output Parameters**.

Rename

allows you to rename the selected output parameter.

Print to Console

reports values to the console window. If you select multiple output parameters, then the output includes values from multiple output parameters.

Print All to Console

outputs the values from all output parameters to the console window.

Write...

allows you to store the output to a file. A dialog box is displayed allowing you to provide a file name.

Write All...

prompts you for a file name and then writes the values for all of the output parameters to a file.

41.4. Fluxes Through Boundaries

This section contains information about generating a flux report. For more background information, see Fluxes Through Boundaries in the Theory Guide.

For additional information, see the following sections:

41.4.1. Generating a Flux Report
41.4.2. Flux Reporting for Reacting Flows
41.4.3. Flux Reporting with Particles
41.4.4. Flux Reporting with Multiphase
41.4.5. Flux Reporting with the DDPM
41.4.6. Flux Reporting with the Potential Solver
41.4.7. Flux Reporting with Other Volumetric Sources

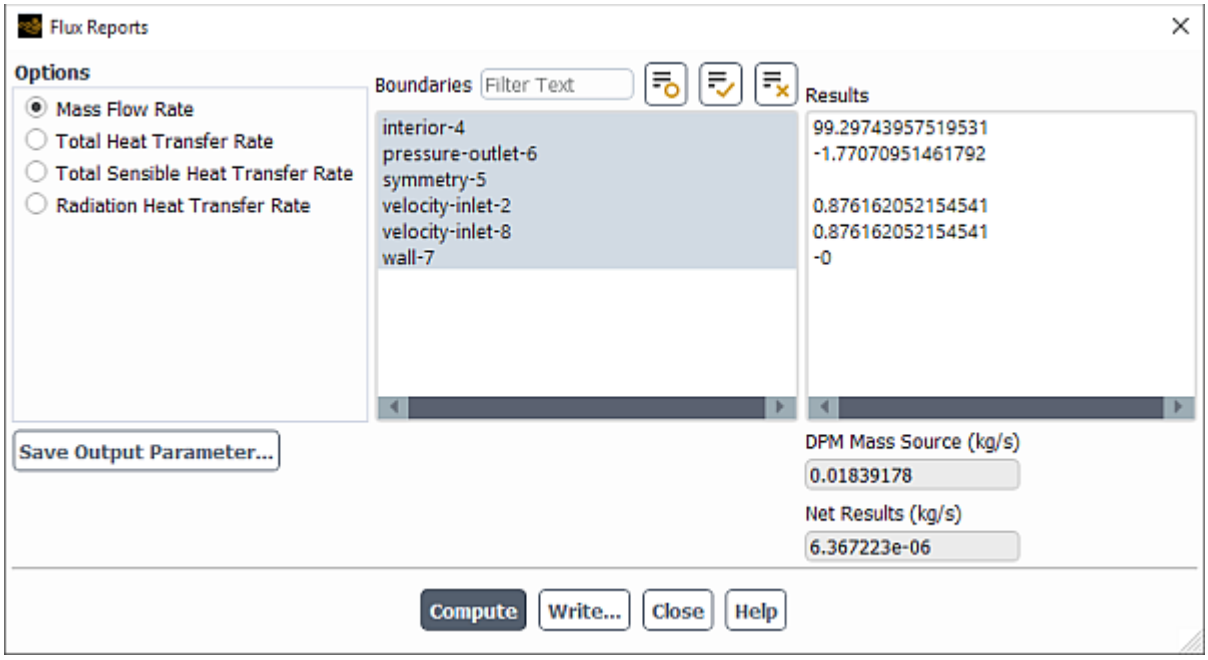
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

41.4.1. Generating a Flux Report

To obtain a report of mass flow rate, total heat transfer rate, or other fluxes on selected boundary zones, use the Flux Reports Dialog Box (p. 5320) (Figure 41.23: The Flux Reports Dialog Box (p. 4136)).

Results → Reports → Fluxes Edit...

Figure 41.23: The Flux Reports Dialog Box



The steps for generating the report are as follows:

1. Specify which flux computation you are interested in by selecting one of the following under **Options**:

   - **Mass Flow Rate**
   - **Total Heat Transfer Rate**

     In cases involving solar radiation, the heat flux associated with the Solar Load model is reported separately under **Total Solar Source**.

   - **Total Sensible Heat Transfer Rate**
   - **Radiation Heat Transfer Rate**

     The net flux from volumetric radiation sources is reported in the **Net Radiation Source** field.

   - **Pressure Work Rate**

     This reports the energy transfer due to pressure when a fluid zone undergoes normal zone motion relative to an adjacent solid zone. This pressure work is on the fluid side only, not on the solid side. This option is only available when the energy equation is enabled, the absolute

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

velocity formulation is selected, and the **Frame Motion** and/or **Mesh Motion** option is enabled in the **Fluid** dialog box.

- **Viscous Work Rate**

  This reports the integral of the product of viscous stress with velocity over the external boundary of the domain with the boundary normal directed inside. Note that after reading case and data files into Ansys Fluent, the viscous work rate is zero. You need to run at least one iteration or time step to obtain the viscous work rate value. This option is available only when the energy equation is enabled and the pressure-based solver is selected.

- **Film Mass Flow Rate**

  This option is available only when the Eulerian Wall Film model is enabled—see Modeling Eulerian Wall Films (p. 3239).

- **Film Heat Transfer Rate**

  This option is available only when the Eulerian Wall Film model is enabled—see Modeling Eulerian Wall Films (p. 3239).

- **Electric Current Rate**

  This reports the total electric current through the selected boundaries. This option is available only if the potential equation is enabled—see Modeling Electric Potential Field and Electrochemistry Models (p. 3265).

2. In the **Boundaries** list, choose the boundary zone(s) on which you want to report fluxes.

   You can use **Filter Text** entry box to filter the **Boundaries** list to show only the boundary zones that match the pattern you enter. For additional information on using the **Filter Text** entry box, see Filter Text Entry Boxes (p. 926).

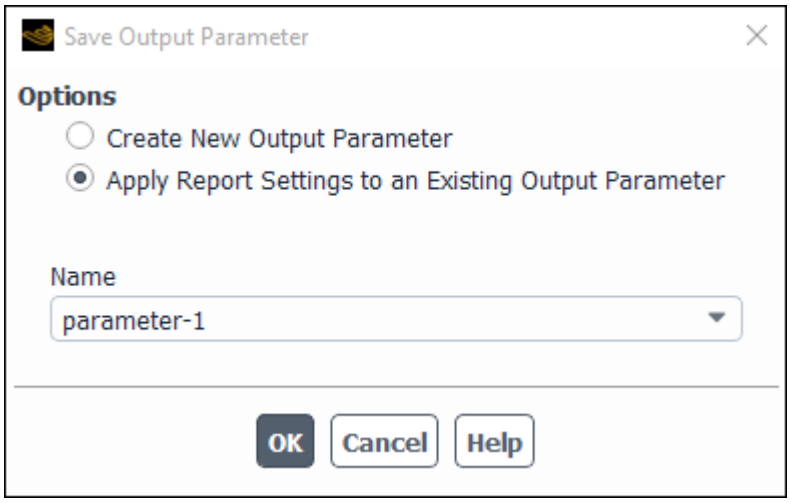
   Note:

   Fluxes can only be computed on boundaries and not on user defined surfaces such as planes, points, iso-surfaces, and so on.

3. To create an output parameter for the reported value, click **Save Output Parameter...**. The Save Output Parameter Dialog Box (p. 5345) (Figure 41.24: The Save Output Parameter Dialog Box (p. 4138)) will open where you will specify the name of the newly created output parameter, or overwrite an existing output parameter of the same type. Ansys Fluent automatically creates generic default names for new input and output parameters (for example, parameter-1, parameter-2, and so on).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 41.24: The Save Output Parameter Dialog Box



After the output parameter is created, it is listed in the Parameters Dialog Box (p. 5338), accessed via the **Parameters...** button in the Reports Task Page (p. 5317). You can create any number of output parameters of this report type.

Note:

If you create an output parameter from a flux report and save the case and data (write), an equivalent report definition will be created when you read in the case file.

4. Click the **Compute** button to display the results of the selected flux computation for each selected boundary zone.

   The **Net Results** field shows the summation of the individual zone flux results. If you specified the mass source terms in the **Fluid** dialog box, the integral mass flow rate value from the user-specified sources is reported in the **User Source** field and is added to the **Net Results**. Similarly, if you specified the energy source terms in the **Fluid** dialog box, the integral total heat transfer rate value from the user-specified sources is reported in the **User Source** field and is added to the net heat balance results reported in **Net Results**.

Important:

- Additional steps must be taken prior to generating a flux report for an interior boundary zone that has the same fluid defined on either side. In such a case, the area vectors of the cell faces associated with the zone may have been automatically defined in an inconsistent manner when the mesh file was read into the solver. Since the flux for each individual cell face is calculated with respect to its area vector, such an inconsistency leads to inaccurate results when the face fluxes are summed to calculate the total flux of the boundary zone.
- To ensure accurate flux results for such an interior zone, you must orient the area vectors by changing the definition of the zone **Type** to **wall**. You should then change the **Type** back to **interior** and proceed to generate the flux report.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Mass Flow Rate** is calculated through the simulation domain or the specified zones. However, if you specify only the interior zone(s), the net mass flow rate may show as 0, because these internal boundaries are skipped when Ansys Fluent computes the net influx and outflux.

Note that the fluxes are reported exactly as computed by the solver. Therefore, they are inherently more accurate than those computed with the **Flow Rate** option in the Surface Integrals Dialog Box (p. 5325) (described in Surface Integration (p. 4150)).

Information on model-specific flux reports can be found in the sections that follow.

41.4.2. Flux Reporting for Reacting Flows

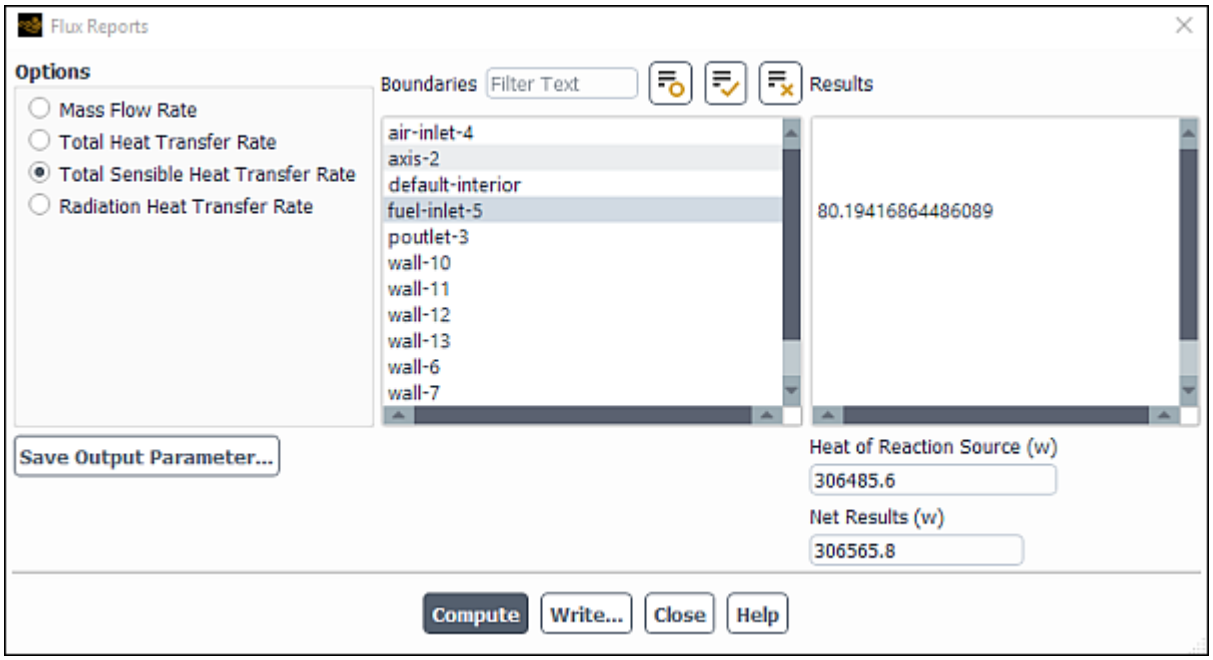
To report heat transfer for reacting flows, one of models in the Species Model Dialog Box (p. 4768) must be enabled for the **Total Sensible Heat Transfer Rate** option to appear in the Flux Reports Dialog Box (p. 5320). For reacting flows, Ansys Fluent produces two kinds of reports that use a different treatment at the flow boundaries:

- **Total Heat Transfer Rate** reports the total enthalpy flux, which consists of the thermal enthalpy, plus the species formation enthalpy when **Volumetric Reactions** are enabled. The heat rate based on this definition is a conserved quantity in reacting flows. See Heat Transfer Theory in the Theory Guide for details.

- **Total Sensible Heat Transfer Rate** reports the total energy flux as defined in Equation 5.5 in the Theory Guide. Note that in reacting flows, this is not a conserved quantity and the addition or removal of heat due to the chemical reactions (Equation 5.11 in the Theory Guide) is reported separately in the **Heat of Reaction Source** field, as shown in Figure 41.25: The Flux Reports Dialog Box (p. 4140). If you have more than one reaction defined in your case, the **Heat of Reaction Source** reported is the sum of the heat for all reactions. For exothermic reactions the **Heat of Reaction Source** is reported as a positive quantity, while for endothermic reactions it will be a negative quantity.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 41.25: The Flux Reports Dialog Box



Important:

Note that both the **Total Heat Transfer Rate** and **Total Sensible Heat Transfer Rate** options report a **Net Result**, which may be used as an indication of the energy balance for the case. In general, and if heat sources other than the heat of reaction and DPM are not included in your problem, the **Net Result** reported in both the **Total Heat Transfer Rate** and **Total Sensible Heat Transfer Rate** options should be a small number for a converged calculation. However, if a reacting case is not well converged for both energy and species transport equations, the **Net Result** reported in the **Total Heat Transfer Rate** and **Total Sensible Heat Transfer Rate** options may differ. In that case, you may consider iterating further to achieve a fully converged solution. In addition, refer to the sections that follow for special considerations when including particles, multiphase models, or other volumetric energy sources.

Important:

Note that for the non-premixed and partially premixed models the **Heat of Reaction Source** is calculated as the difference of the net **Total Heat Transfer Rate** and the net **Total Sensible Heat Transfer Rate**. The **Heat of Reaction** field function is not available for the non-premixed and partially-premixed models.

For the general procedure on generating flux reports, see Generating a Flux Report (p. 4136).

41.4.3. Flux Reporting with Particles

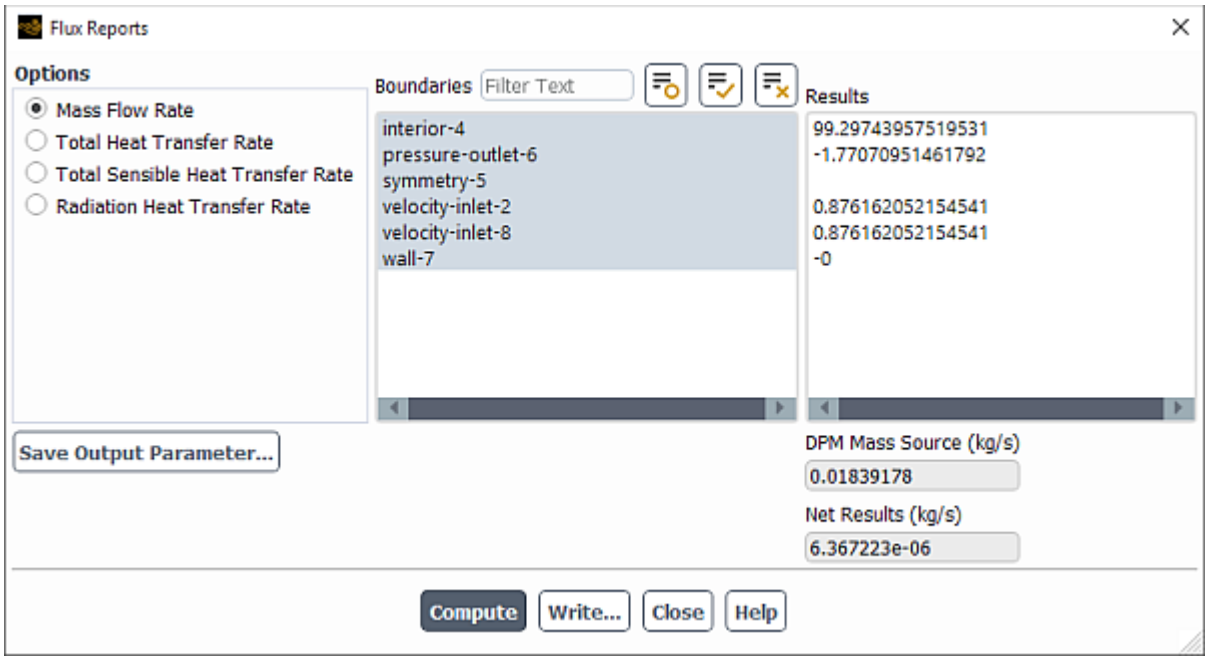
If you are using the discrete phase model (DPM), the contributions from the particle injections are reported separately and are included in the net mass and heat balance results. Consequently, the

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

following quantities are included in the reports (see Figure 41.26: The Flux Reports Dialog Box with DPM (p. 4141)):

- **Mass Flow Rate** flux report: **DPM Mass Source**
- **Total Heat Transfer Rate** flux report: **DPM Enthalpy Source**
- **Total Sensible Heat Transfer Rate** flux report: **DPM Sensible Enthalpy Source**

Figure 41.26: The Flux Reports Dialog Box with DPM



For P1 or discrete ordinates (DO) radiation simulations, if **Particle Radiation Interaction** is selected in the **Physical Models** tab of the **Discrete Phase Model** dialog box, the **Radiation Heat Transfer Rate** reports the difference between the radiation emitted and the radiation absorbed on the surface of the particles in the **DPM Radiation Source** field.

Important:

In the case of reacting flows with the DPM model, the **Heat of Reaction Source** entry reports the heat of all homogeneous reactions in the continuous phase, while the heat released or consumed due to particle reactions (for example, char combustion) is reported in the **DPM Sensible Enthalpy Source** field.

When you are using flux reporting with Lagrangian wall film particles, the total heat transfer rate reported as **Net Results** will equal the surface integral of the **Wall Film Heat Flux** field variable (in the **Wall Film…** category).

For the general procedure on generating flux reports, see Generating a Flux Report (p. 4136).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

41.4.4. Flux Reporting with Multiphase

If you are using any of the multiphase models, the mass or heat rates can be reported separately for each phase and for the mixture phase depending on the model used. If you specified the mass source terms in the cell zone conditions dialog box for a specific phase, then the integral mass flow rate value from the user-specified sources is reported for the relevant phase in the **User Source** field and is added to the net mass balance results reported in **Net Results**. Similarly, if you specified the energy source terms in the cell zone conditions dialog box for a specific phase, the integral total heat transfer rate value from the user-specified sources is reported for the relevant phase in the **User Source** field and is added to the net heat balance results reported in **Net Results**.

Note that if your multiphase model includes mass or heat transfer processes between phases, the mass and heat transferred across the phases will be reported as an imbalance in the report of each phase. In order to check the overall balances for the multiphase cases you should select the mixture phase for your report. In that case, the report will include the sum of the fluxes and sources for all phases included in your model.

If you are solving a multiphase problem that includes chemical reactions, you should be aware of the following conventions when you are requesting a **Total Sensible Heat Transfer Rate** report:

- If you select one of the phases with gas phase chemical reactions, the **Heat of Reaction Source** will only include contributions from reactions in the particular phase.
- When you report the **Total Sensible Heat Transfer Rate** for the mixture phase, the **Heat of Reaction Source** entry will report the sum of the heat of reaction of all gas phase reactions in all phases plus the heat of any heterogeneous reactions that take place.

For the general procedure on generating flux reports, see Generating a Flux Report (p. 4136).

41.4.5. Flux Reporting with the DDPM

If you are using the DDPM, the mass and heat rates across the domain boundaries can be reported for each phase as described in Flux Reporting with Multiphase (p. 4142). In addition, for the discrete (DDPM) phase, the fluxes within the domain are also reported in the following fields of the **Flux Reports** dialog box and the Fluent console:

- **Mass Flow Rate** flux report: **DDPM In-domain Mass Rate**
- **Total Heat Transfer Rate** flux report: **DDPM In-domain Heat Rate**

These quantities report the total mass or heat rate contributions from:

- particles that are released inside the domain (that is, do not enter the domain through the boundaries), and
- particles that disappear inside the domain (that is, do not leave the domain through the boundaries).

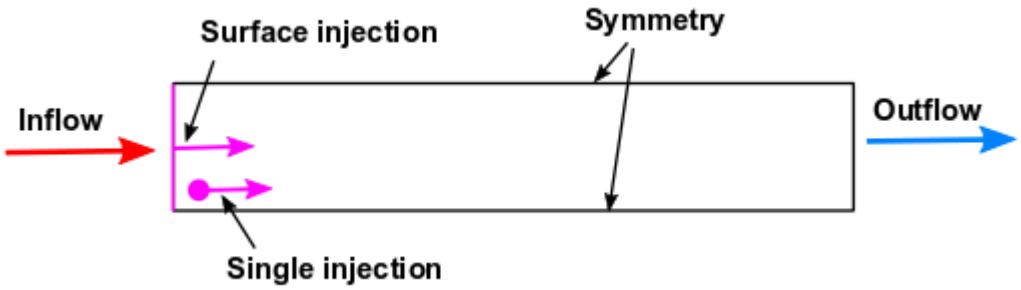
Note that this is valid only for particles that are assigned to a discrete phase. The reported quantities extend the mass and heat rate balances of the standard reports that are available in multiphase simulations only for fluxes through domain boundaries.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The contribution from the DPM injections that have not been assigned to the discrete phase are reported as the DPM mass or enthalpy source, depending on your case. See Flux Reporting with Particles (p. 4140) for details.

Figure 41.27: A Two-Phase Flow with Two Discrete Phase Injections (p. 4143) shows an example of a steady state two-phase flow in a channel with one discrete phase and one fluid phase.

Figure 41.27: A Two-Phase Flow with Two Discrete Phase Injections



Two DPM injections, one surface injection at the inflow boundary and one single injection in the first cell layer adjacent to the inflow boundary, with the same mass flow rate are defined in the domain. Both injections are assigned to the discrete phase. All phases of the two-phase flow are leaving the domain through the outflow boundary.

When generating flux reports for the discrete phase, the mass flow rate for particles entering the domain via the surface injection is reported as a rate through the inlet boundary, while the mass flow rate for particles entering the domain via the single injection is reported as **DDPM In-domain Mass Rate** since the injection occurs within the domain as shown below in the mass flow rate report for the discrete phase. All particles leave the domain through the outlet. Since no mass is exchanged with the primary phase, the net value for the mass flow rate for discrete phase is nearly zero.

The mass flow rate report for the discrete phase is as follows:

```
                  particle-phase
                  Mass Flow Rate               [kg/s]
------------------------------- --------------------
                          inlet              4.5e-05
                         outlet               -9e-05
        DDPM In-domain Mass Rate              4.5e-05
                ---------------- --------------------
                            Net        1.1922239e-14
```

The mass flow rate report for the primary phase is shown below:

```
                     fluid-phase
                  Mass Flow Rate               [kg/s]
------------------------------- --------------------
                          inlet             5.55e-06
                         outlet             -5.1e-06
                DPM Mass Source                    0
                ---------------- --------------------
                            Net              4.5e-07
```

The **DPM Mass Source** is reported as `0` because there are no injections in the domain that have not been assigned to the discrete phase.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The balance for the mixture includes the mass flow rates from all the phases through the selected boundaries, the **DPM Mass source** from standard DPM injections, and the **DDPM In-domain Mass Rate** from the injections assigned to the discrete phase.

The mass flow rate report for the mixture is shown below:

```
                        mixture
                Mass Flow Rate               [kg/s]
------------------------------ --------------------
                         inlet            5.055e-05
                        outlet            -9.51e-05
               DPM Mass Source                    0
      DDPM In-domain Mass Rate              4.5e-05
              --------------- --------------------
                           Net        4.5000001e-07
```

For the general procedure on generating flux reports, see Generating a Flux Report (p. 4136).

Note:

The following limitations currently exist for flux reporting capability with DDPM.

When reading a previously saved data file, the following steps are required to obtain the correct flux reports:

- For steady-state simulations and transient simulations with the injection flow time spread over multiple time steps, you need to perform a single DPM iteration (steady-state cases) or a single time step (transient cases) to update information for the **DDPM In-domain Mass Rate** or **DDPM In-domain Heat Rate**.
- Cases where injections both assigned and not assigned to a discrete phase are present in the domain must be rerun to obtain the correct values for the DPM mass or enthalpy source term.

Since the flux reporting with DDPM is purely a postprocessing functionality for the reporting of in- and outcoming fluxes for the DDPM phases and the mixture, it does not affect the simulation results. Any other postprocessing quantities can be reported as usual, and no additional steps are required.

41.4.6. Flux Reporting with the Potential Solver

If you have solved the potential equation, you can use the **Flux Reports** dialog box to compute the following quantities:

- **Electric Current Rate**

  You can report the total electric current through the selected boundaries.

- **Joule Heating Source**

  When you report **Total Heat Transfer Rate** for cases that involve the potential equation with the **Include Joule Heating in Energy Equation** option, the total Joule heating source in the domain is reported separately in the **Joule Heat Source** field. This value is also counted in the total heat balance calculation reported in the **Net Results** field.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For the general procedure on generating flux reports, see Generating a Flux Report (p. 4136).

41.4.7. Flux Reporting with Other Volumetric Sources

The reported mass and heat balances address the flow that enters or leaves the domain through boundaries and the contributions from DPM sources and user-defined sources; they do not include the contributions from other volumetric sources, such as the heat exchanged in the Heat Exchanger Model. For this reason, a mass or heat imbalance may be reported. In that case, and in a converged calculation, the reported imbalance will be equal to that volumetric source.

For a **Radiation Heat Transfer Rate** flux report, the **Net Radiation Source** field provides the net flux from volumetric radiation sources and is computed as the difference between the emitted and absorbed volumetric radiation sources.

For the general procedure on generating flux reports, see Generating a Flux Report (p. 4136).

41.5. Forces on Boundaries

For wall zones that you select, you can compute and report the forces along a specified vector, the moments about a specified center and along a specified axis, and the coordinates of the center of pressure. This feature is useful for reporting, for instance, aerodynamic quantities such as lift, drag, and moment coefficients, as well as the center of pressure for an airfoil.

For additional information about forces, moments, and the center of pressure, see Computing Forces, Moments, and the Center of Pressure in the Theory Guide.

For additional information, see the following section:

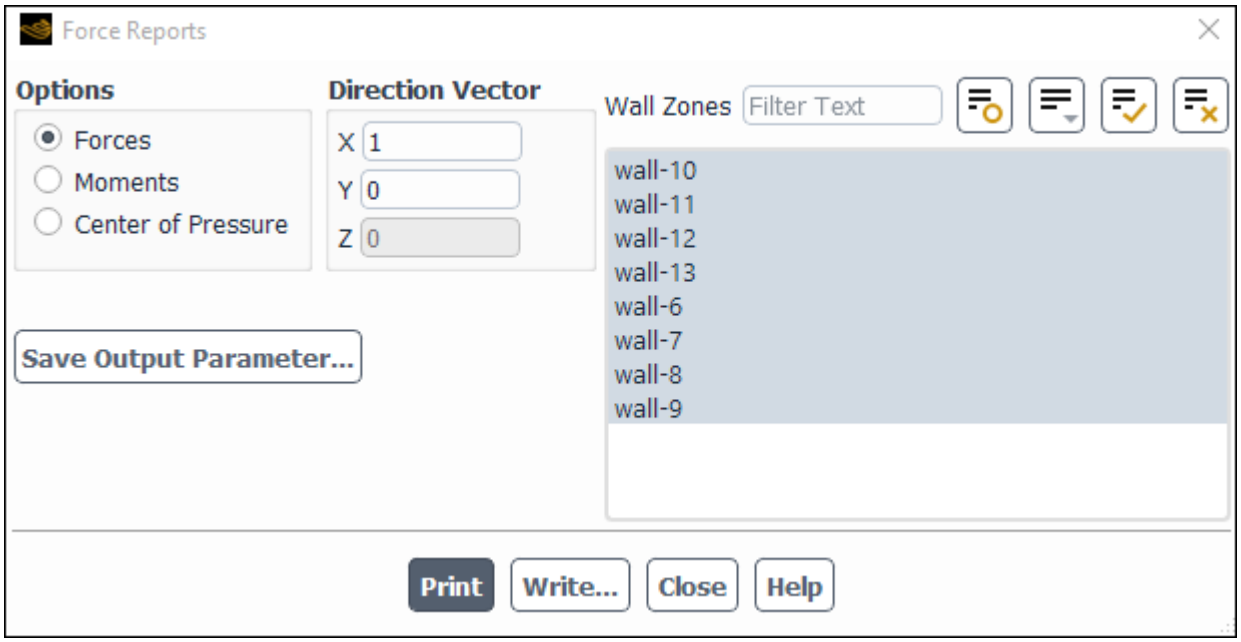
41.5.1. Generating a Force, Moment, or Center of Pressure Report

41.5.1. Generating a Force, Moment, or Center of Pressure Report

To obtain a report (for selected wall zones) of forces along a specified vector, moments about a specified center and along a specified axis, or the center of pressure, use the Force Reports Dialog Box (p. 5322) (Figure 41.28: The Force Reports Dialog Box (p. 4146)).

Results → Reports → Forces Edit...

Figure 41.28: The Force Reports Dialog Box



The steps for generating the report are as follows:

1. Specify the type of report in which you are interested by selecting **Forces**, **Moments**, or **Center of Pressure** from the **Options** list.

2. Define the settings associated with report you are generating:

   a. For a force report, enter the **X**, **Y**, and **Z** components of the **Force Vector** along which the forces will be computed.

   Important:

   If F_x (the x-component of the force) is zero, then either the **Y** or **Z** coordinate can be fixed. If F_y is zero, then either the **X** or **Z** coordinate can be fixed. If F_z is zero, then either the **X** or **Y** coordinate can be fixed.

   b. For a moment report, enter the **X**, **Y**, and **Z** coordinates of the **Moment Center** about which the moments will be computed, as well as the **X**, **Y**, and **Z** components of the **Moment Axis** along which the moments will be computed.

   c. For a center of pressure report, define the line (for 2D geometries) or plane (for 3D geometries) on which you want to calculate the center of pressure. The line or plane must have one of its coordinate values fixed (for example, a line defined as $y=10$). Select the axis (**X**, **Y**, or **Z**) in the **Coordinate** group box, and then enter the fixed **Value**. See the example at the end of this section for further details.

3. Select the **Wall Zones** (which can also include porous zones *when applicable*), where you want forces, moment, or the pressure center reported.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. To create an output parameter for the reported value, click **Save Output Parameter...**. The Save Output Parameter Dialog Box (p. 5345) (Figure 41.24: The Save Output Parameter Dialog Box (p. 4138)) will open where you will specify the name of the newly created output parameter, or overwrite an existing output parameter of the same type. Ansys Fluent automatically creates generic default names for new input and output parameters (for example, parameter-1, parameter-2, and so on).

   After the output parameter is created, it is listed in the Parameters Dialog Box (p. 5338). You can create any number of output parameters of this report type.

   Note:

   If you create an output parameter from a force, moment, or center of pressure report and save the case and data (write), an equivalent report definition will be created when you read in the case file.

5. Click the **Print** button if you want the results displayed in the console window, or click **Write...** to save it to a file.

   If you selected **Forces** under **Options**, the pressure force, viscous force (if appropriate), total forces, pressure coefficient, viscous coefficient, and total coefficients for each selected wall zone will be displayed or saved.

   If you selected **Moments**, the pressure moments, viscous moments (if appropriate), total moments, pressure coefficient, viscous coefficient and total coefficients for the wall zones about the specified center will be displayed or saved. Additionally, the moments and coefficients in the direction of the specified axis will be displayed or saved. The report will include the values for the individual wall zones, as well as the net values for all of the wall zones combined. See Computing Forces, Moments, and the Center of Pressure in the Theory Guide for details about computing forces and moments.

   If you selected **Center of Pressure**, then Ansys Fluent displays or saves the coordinates about which the moment by pressure forces is zero.

   Important:

   You cannot save your output parameter if **Center of Pressure** is selected; **Center of Pressure** is not available as an output parameter since it is a set of coordinates.

Important:

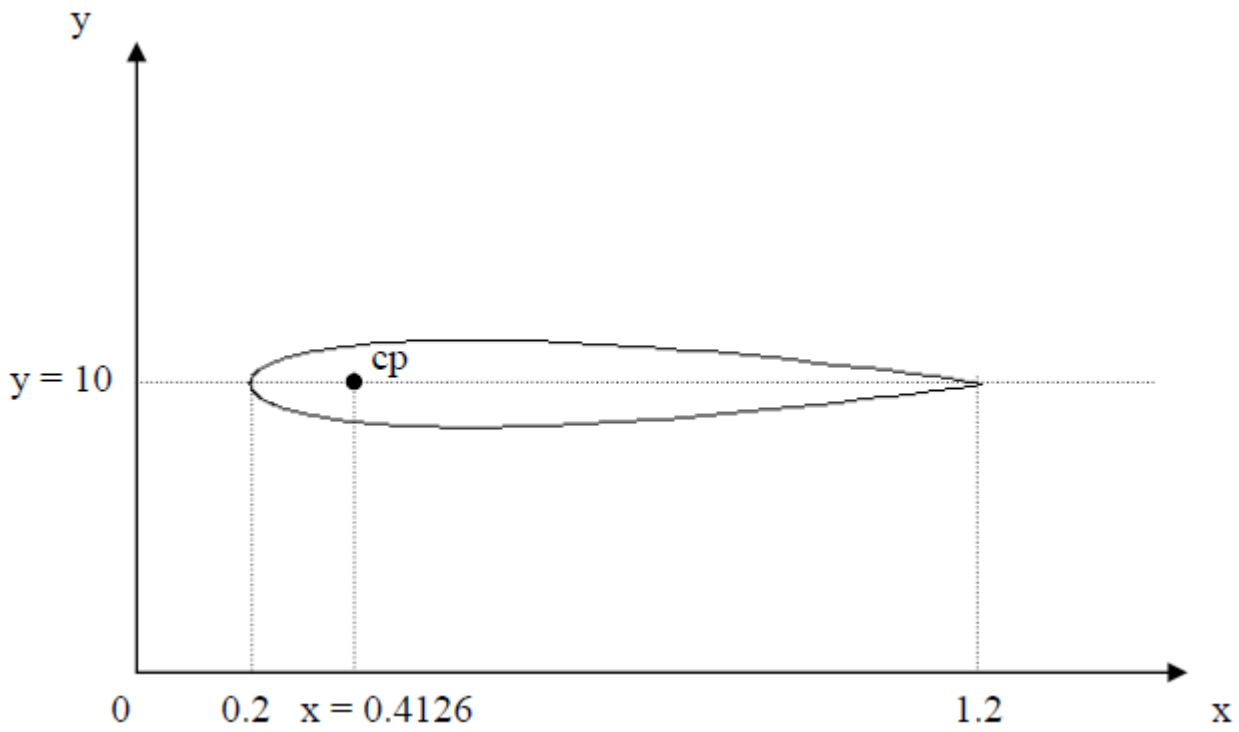
Note that the reported force and moment coefficients are a function of the values entered in the **Reference Values** task page (as described in Computing Forces, Moments, and the Center of Pressure in the Theory Guide). Therefore, appropriate values must be entered in the **Reference Values** task page to get meaningful results.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

41.5.1.1. Example

To demonstrate how you would generate and interpret the center of pressure report, consider an airfoil of chord length 1 m (shown in Figure 41.29: An Airfoil with its Computed Center of Pressure (p. 4148)).

Figure 41.29: An Airfoil with its Computed Center of Pressure

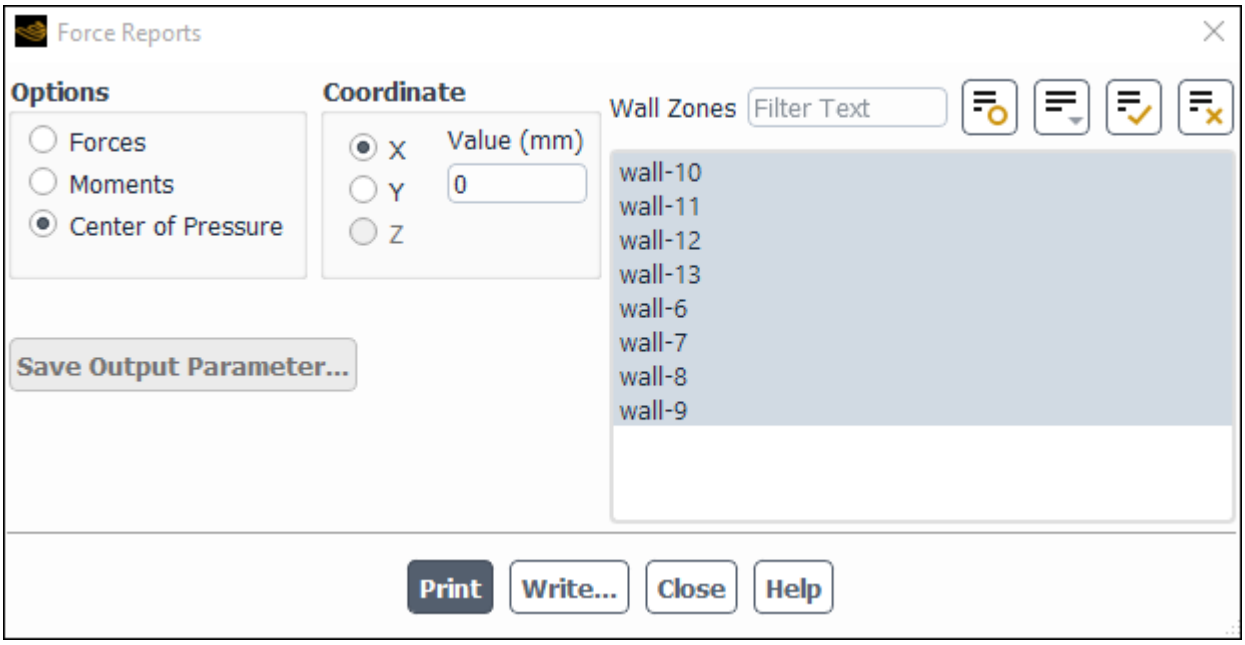


Open the **Force Reports** dialog box and perform the steps that follow.

Results → Reports → Forces Edit...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 41.30: The Force Reports Dialog Box for a Center of Pressure Report



1. Select **Center of Pressure** from the **Options** list.
2. Define the line on which the center of pressure will be calculated. In this case, the **Y** coordinate for the line has a fixed **Value** of `10`.
3. Select the **Wall Zones** that are relevant for the computation.
4. Click **Print** to have the coordinates of the center of pressure displayed in the console window.

The report generated will be in the following form:

```
Pressure Center Coordinates (in m):
X = 0.41267981
Y = 10
```

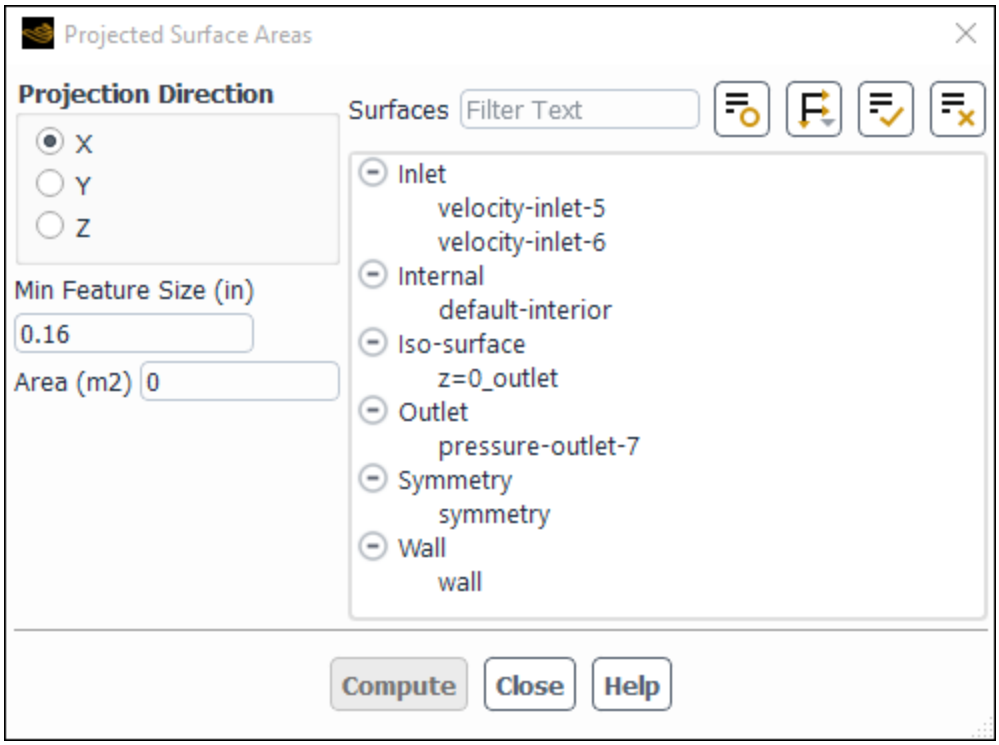
41.6. Projected Surface Area Calculations

You can use the Projected Surface Areas Dialog Box (p. 5324) (Figure 41.31: The Projected Surface Areas Dialog Box (p. 4150)) to compute an estimated area of the projection of selected surfaces along the x, y, or z axis (that is, onto the yz, xz, or xy plane).

Results → Reports → Projected Areas Edit...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 41.31: The Projected Surface Areas Dialog Box



The steps for calculating the projected area are as follows:

1. Select the **Projection Direction** (**X**, **Y**, or **Z**).
2. Choose the surface(s) for which the projected area is to be calculated in the **Surfaces** list.
3. Set the **Min Feature Size** to the length of the smallest feature in the geometry that you want to resolve in the area calculation. (You can just use the default value to start with, if you are not sure of the size of the smallest geometrical feature.)
4. Click **Compute**. The area will be displayed in the **Area** box and in the console window.
5. To improve the accuracy of the area calculation, reduce the **Min Feature Size** by half and recompute the area. Repeat this step until the computed **Area** stops changing (or you reach memory capacity).

This feature is available only for 3D domains.

41.7. Surface Integration

This section describes how to compute surface integrals. For mathematical definitions of the various integral types, refer to Computing Surface Integrals in the Theory Guide.

For additional information, see the following section:

41.7.1. Generating a Surface Integral Report

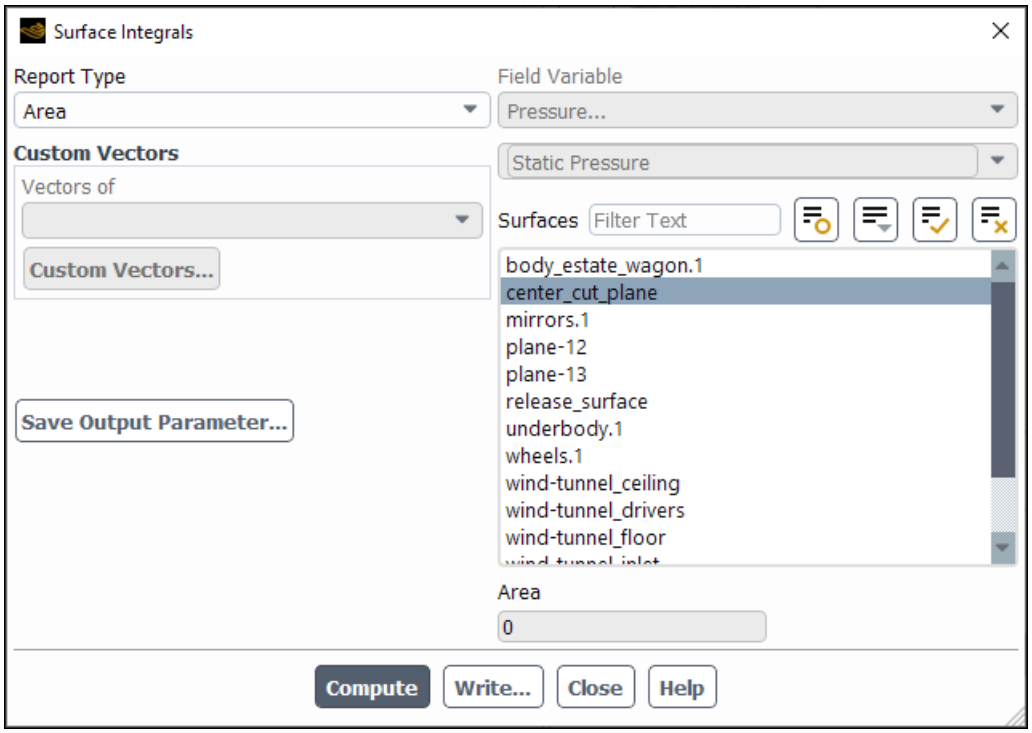
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

41.7.1. Generating a Surface Integral Report

To obtain a report for selected surfaces of the area, custom vector-based flux, custom vector flux, custom vector-weighted average, mass flow rate, or volume flow rate, or the integral, flow rate, standard deviation, sum, facet maximum, facet minimum, uniformity index (weighted by mass or area), vertex maximum, vertex minimum, or mass-, area-, facet-, or vertex-averaged quantity of a specified field variable, use the Surface Integrals Dialog Box (p. 5325) (Figure 41.32: The Surface Integrals Dialog Box (p. 4151)).

Results → Reports → Surface Integrals Edit...

Figure 41.32: The Surface Integrals Dialog Box



The steps for generating the report are as follows:

1. Specify which type of report you are interested in by selecting **Area**, **Area-Weighted Average**, **Custom Vector Based Flux**, **Custom Vector Flux**, **Custom Vector Weighted Average**, **Facet Average**, **Facet Minimum**, **Facet Maximum**, **Flow Rate**, **Integral**, **Mass Flow Rate**, **Mass-Weighted Average**, **Standard Deviation**, **Sum**, **Uniformity Index - Mass Weighted**, **Uniformity Index - Area Weighted**, **Vertex Average**, **Vertex Minimum**, **Vertex Maximum**, or **Volume Flow Rate** in the **Report Type** drop-down list.

2. If you selected **Custom Vector Based Flux**, **Custom Vector Flux**, or **Custom Vector Weighted Average** from the **Report Type** drop-down list, you must define vectors in the **Custom Vectors** group box.

3. If you are performing a multiphase simulation, you may need to select the phase of interest from the **Phase** drop-down list depending on the selected **Report Type**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. If you are generating a report of area, mass flow rate, or volume flow rate, skip to the next step. Otherwise, use the **Field Variable** drop-down lists to select the field variable to be used in the surface integrations. First, select the desired category in the upper drop-down list. You can then select a related quantity from the lower list. (See Field Function Definitions (p. 4167) for an explanation of the variables in the list.)

5. In the **Surfaces** list, choose the surface(s) on which to perform the surface integration.

6. To create an output parameter for the reported value, click **Save Output Parameter...**. The Save Output Parameter Dialog Box (p. 5345) (Figure 41.24: The Save Output Parameter Dialog Box (p. 4138)) will open where you will specify the name of the newly created output parameter, or overwrite an existing output parameter of the same type. Ansys Fluent automatically creates generic default names for new input and output parameters (for example, parameter-1, parameter-2, and so on).

   After the output parameter is created, it is listed in the Parameters Dialog Box (p. 5338). You can create any number of output parameters of this report type.

   Note:

   If you create an output parameter from a surface integral report and save the case and data (write), an equivalent report definition will be created when you read in the case file.

7. Click the **Compute** button. The computed results are printed in the numeric result field and also in the Ansys Fluent console.

   The **Net** *value reported in the Ansys Fluent console corresponds to the specified calculation performed over the list of selected surfaces.*

8. To save the computed results to a file, click the **Write...** button and specify the filename in the resulting **Select File** dialog box.

Note the following:

- Mass averaging "weights" toward regions of higher velocity (that is, regions where more mass crosses the surface).
- Flow rates reported using the Surface Integrals Dialog Box (p. 5325) are not as accurate as those reported with the Flux Reports Dialog Box (p. 5320) (described in Fluxes Through Boundaries (p. 4135)).
- The facet and vertex average options are recommended for zero-area surfaces.
- The uniformity index represents how a specified field variable varies over a surface, where a value of 1 indicates the highest uniformity. The uniformity index can be weighted by area or mass: the area-weighted uniformity index captures the variation of the quantity (for example, the species concentration), whereas the mass-weighted uniformity index captures the variation of the flux (for example, the species flux). See Computing Surface Integrals in the Theory Guide for the equations used to calculate the uniformity index.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

41.8. Volume Integration

This section describes how to compute volume integrals. For mathematical definitions of the various integral types, refer to Computing Volume Integrals in the Theory Guide.

For additional information, see the following section:

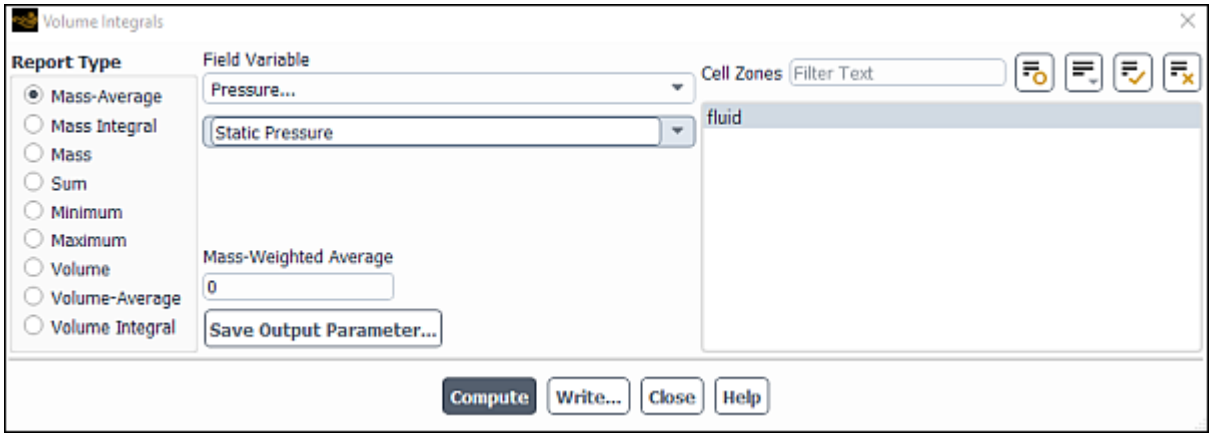
41.8.1. Generating a Volume Integral Report

41.8.1. Generating a Volume Integral Report

To obtain a report (of quantities such as the volume, sum, minimum, maximum, volume integral, volume-weighted average, mass-weighted integral, or mass-weighted average) for selected cell zones for a specified field variable, use the Volume Integrals Dialog Box (p. 5329) (Figure 41.33: The Volume Integrals Dialog Box (p. 4153)).

Results → Reports → Volume Integrals Edit...

Figure 41.33: The Volume Integrals Dialog Box



The steps for generating the report are as follows:

1. Specify which type of report you are interested in by selecting **Volume**, **Sum**, **Sum*2Pi** (only available for 2D Axisymmetric cases), **Maximum**, **Minimum**, **Volume Integral**, **Volume-Average**, **Mass**, **Mass Integral**, or **Mass-Average** under **Report Type**.

2. If you are generating a report of **Volume** or **Mass**, skip to the next step. Otherwise, use the **Field Variable** drop-down lists to select the field variable to be used in the integral, sum, or averaged volume integrations. First, select the desired category in the upper drop-down list. You can then select a related quantity from the lower list. (See Field Function Definitions (p. 4167) for an explanation of the variables in the list.)

3. If you are performing a multiphase simulation, select the phase of interest (or **mixture**) from the **Phase** drop-down list.

4. In the **Cell Zones** list, choose the zones on which to compute the volume, sum, max, min, volume integral, volume-weighted average, mass integral, or mass-averaged quantity. *The **Cell Zones** list will show any defined expression volumes, as they are also valid inputs for volume-based computations.*

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

5. To create an output parameter for the reported value, click **Save Output Parameter...**. The Save Output Parameter Dialog Box (p. 5345) (Figure 41.24: The Save Output Parameter Dialog Box (p. 4138)) will open where you will specify the name of the newly created output parameter, or overwrite an existing output parameter of the same type. Ansys Fluent automatically creates generic default names for new input and output parameters (for example, parameter-1, parameter-2, and so on).

   After the output parameter is created, it is listed in the Parameters Dialog Box (p. 5338). You can create any number of output parameters of this report type.

   Note:

   If you create an output parameter from a volume integral report and save the case and data (write), an equivalent report definition will be created when you read in the case file.

6. Click the **Compute** button. The computed results are printed in the numeric result field and also in the Ansys Fluent console.

   The **Net** *value reported in the Ansys Fluent console corresponds to the specified calculation performed over the list of selected cell zones.*

7. To save the computed results to a file, click the **Write...** button and specify the filename in the resulting **Select File** dialog box.

41.9. Efficiency Calculations

This section describes how to compute efficiencies for a variety of scenarios. For mathematical definitions of the various efficiency types, refer to Efficiency Calculation in the *Fluent Theory Guide*.

For additional information, see the following sections:

41.9.1. Device Efficiency

41.9.2. Calculating Efficiency using Named Expressions

41.9.3. Limitations

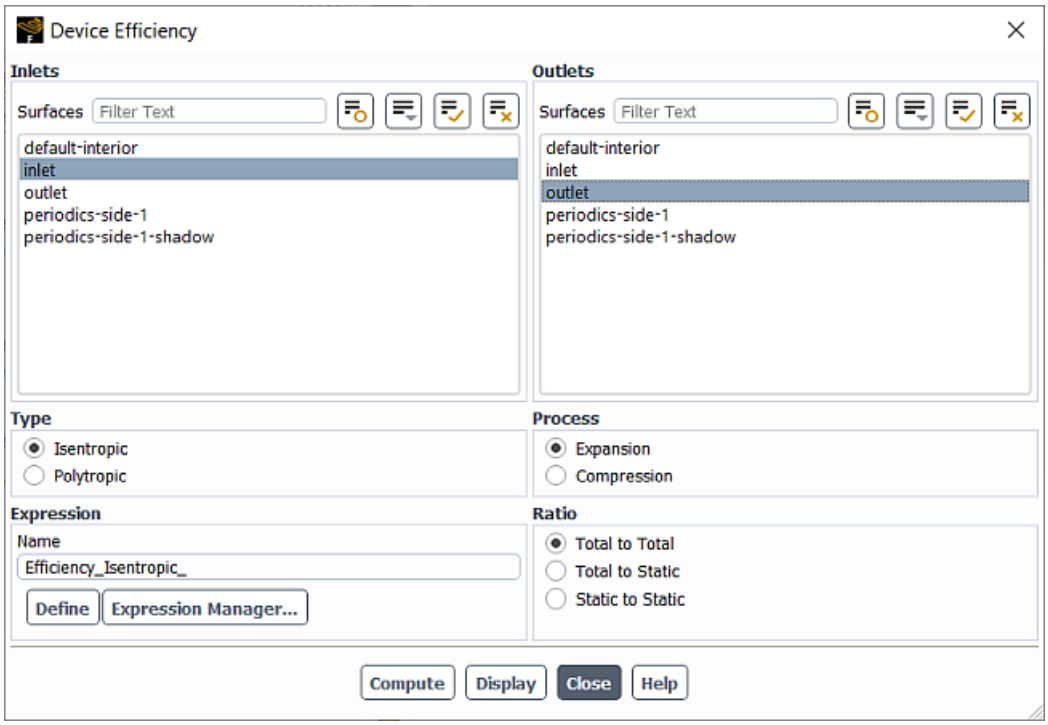
41.9.1. Device Efficiency

You can define and evaluate efficiency for a generic device using the Device Efficiency dialog box (Figure 41.34: The Device Efficiency Dialog Box (p. 4155)) accessed from:

→ **Results** → **Reports** → **Efficiency...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 41.34: The Device Efficiency Dialog Box



1. Locations for the **Inlets** and **Outlets** can be selected using the corresponding surface lists. These lists are filtered to include only the surfaces of supported types.

2. Under **Type**, select whether to compute the **Isentropic** or **Polytropic** efficiency.

3. Under **Process**, select whether to compute the efficiency for the **Expansion** or **Compression** process.

4. Under **Ratio**, you can select which values are used when computing the device efficiency: **Total to Total**, **Total to Static**, or **Static to Static**.

5. You can optionally define an expression based on the dialog box input. Edit or keep the default name (an integer number will be appended to the name automatically) and click **Define**. All available expressions can be viewed by clicking **Expression Manager...**. For more details, see Calculating Efficiency using Named Expressions (p. 4156).

6. If the solution is initialized or computed, you can click **Compute** to get an efficiency value for the selected input.

7. Click **Display** to visualize the selected surfaces.

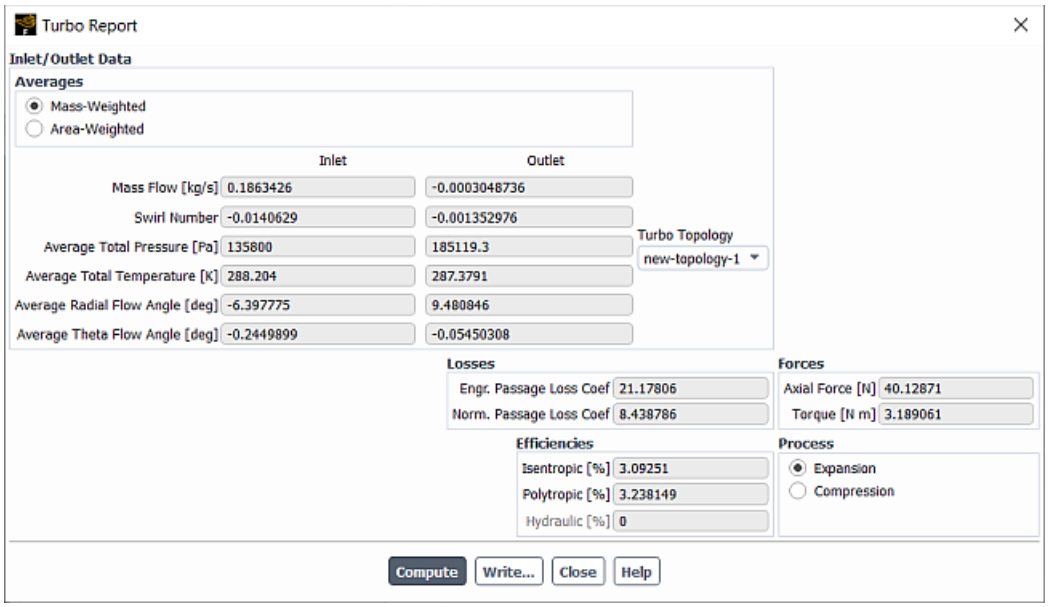
You can also access this tool by using the following console command and following the prompts:

```
/report/efficiency
```

Note that the **Device Efficiency** dialog box is applicable to devices of various types, including but not limited to turbomachines, such as diffusers, nozzles, valves, and other devices with compressible throughflow. However, specifically for turbomachines, an additional option is available to compute efficiency using the **Turbo Report** dialog box as described in Generating Reports of Turbomachinery Data (p. 2041).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

→ **Results** → **Model Specific** → **Turbo Post** → **Report...**



By default, the **Turbo Report** dialog box computes the Total to Total values of isentropic and polytropic efficiencies assuming constant values of the specific heat capacity Cp and heat capacity ratio γ. You can switch to the more general equations which are implemented from the **Device Efficiency** dialog box, by using the following console command:

```
/report/efficiency/use-in-turbo-report?
```

Note that for an ideal gas with constant Cp value, the two methods give nearly identical results. However, for other cases, only the method from the **Device Efficiency** dialog box is able to provide appropriate efficiency values.

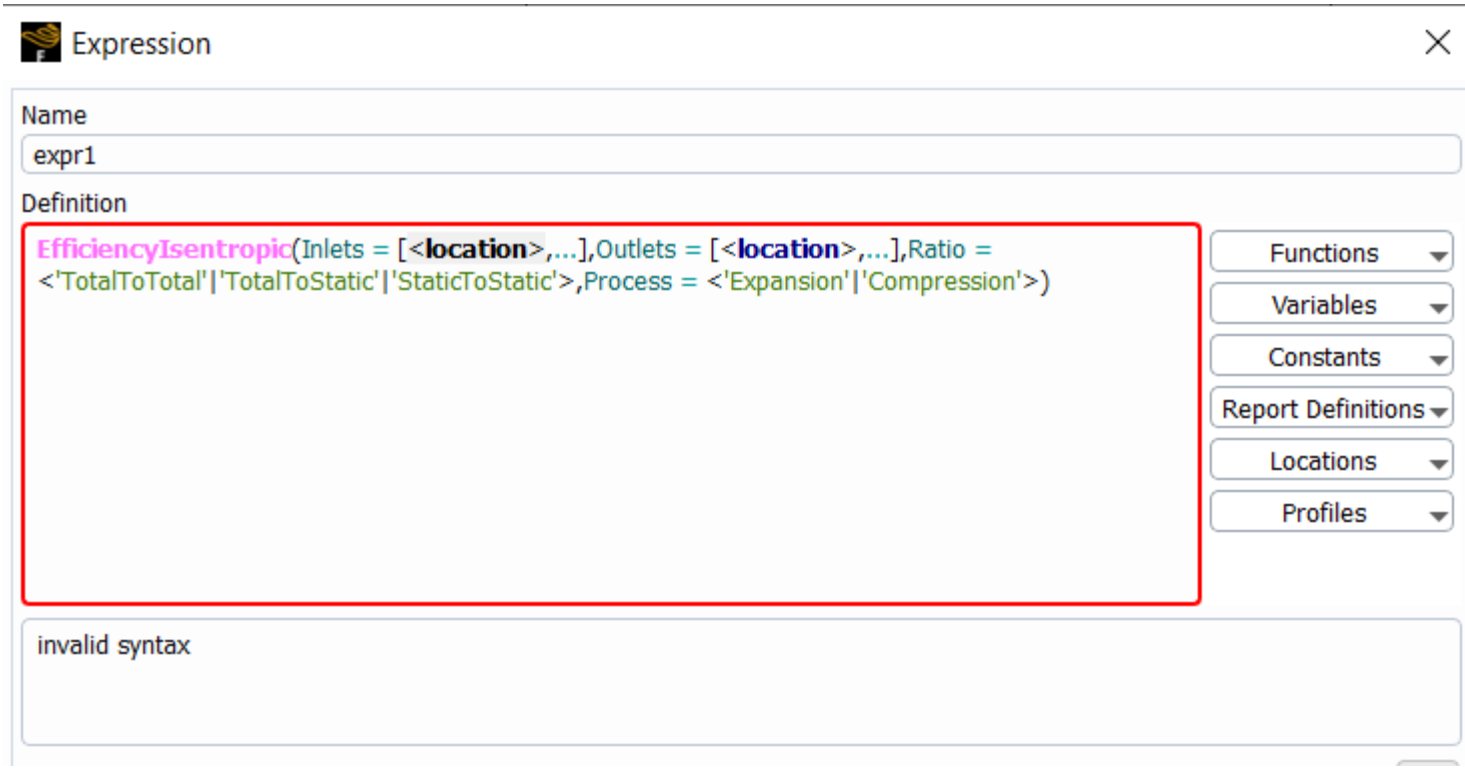
41.9.2. Calculating Efficiency using Named Expressions

You can compute the values of isentropic and polytropic efficiencies for a device with compressible throughflow using **Named Expressions** as shown in Figure 41.35: Expression Dialog Box with Efficiency Calculation (p. 4157).

→ **User Defined** → **User Defined** → **Named Expressions** → **New...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 41.35: Expression Dialog Box with Efficiency Calculation



Functions for both efficiency types are listed under **Functions** → **Reduction** as `EfficiencyIsentropic` and `EfficiencyPolytropic`.

Permeable boundaries of various types (for example, inlets, outlets, interfaces), as well as cut-planes and iso-surfaces, can be chosen as locations. Also, groups of surfaces for postprocessing are supported. For the argument `Inlets`, you can select one or more locations of mixed types. However, multiple locations will still be treated as the single-stream inlet with properties averaged over all selected locations. This averaged inlet can be considered as the starting point for efficiency calculation. Similarly, the end point will be determined by locations selected for the argument `Outlets`. Options for the argument `Process(Expansion and Compression)` specify whether the flow expands or compresses when moving from the inlet to the outlet. Options for the argument `Ratio` (`TotaltoTotal`, `TotaltoStatic` and `StatictoStatic`) determines the values used in computing the efficiency.

41.9.3. Limitations

- Efficiency can be evaluated only for compressible flows of ideal gas, single-component real gas, and condensing water vapor (Wet Steam model).
- Only the following locations are supported: Pressure Inlet, Pressure Outlet, Mass-Flow Inlet, Mass-Flow Outlet, Velocity Inlet, Pressure-far-field, Inlet Vent, Outlet Vent, Intake Fan, Exhaust Fan, Interior, Interface, Cut-plane, Iso-surface, Groups of listed surfaces.
- Multiple inlet or outlet locations are reduced to a single stream with flow properties averaged over all selected.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

41.10. Histogram Reports

In Ansys Fluent, you can print geometric and solution data in the console (text) window in histogram format or plot a histogram in the graphics window. Graphical display of histograms and the procedures for defining a histogram are discussed in Histograms (p. 4072).

The number of cells, the range of the selected variable or function, and the percentage of the total number of cells in the interval will be reported, as in the example below:

```
0 cells below 1.195482 (0 %)
2 cells between 1.195482 and 1.196048 (4.1666667 %)
1 cells between 1.196048 and 1.196614 (2.0833333 %)
0 cells between 1.196614 and 1.19718 (0 %)
0 cells between 1.19718 and 1.197746 (0 %)
2 cells between 1.197746 and 1.198312 (4.1666667 %)
1 cells between 1.198312 and 1.198878 (2.0833333 %)
6 cells between 1.198878 and 1.199444 (12.5 %)
9 cells between 1.199444 and 1.20001 (18.75 %)
25 cells between 1.20001 and 1.200576 (52.083333 %)
2 cells between 1.200576 and 1.201142 (4.1666667
0 cells above 1.201142 (0 %)
```

To generate such a printed histogram, use the Histogram Dialog Box (p. 5295).

Results → Plots → Histogram Edit...

Follow the instructions in Histograms (p. 4072) for generating histogram plots, but click **Print** instead of **Plot** to create the report.

41.11. Discrete Phase

Ansys Fluent allows you to write particle states (position, velocity, diameter, temperature, and mass flow rate) to files at various boundaries and planes (lines in 2D) using the Sample Trajectories Dialog Box (p. 5331) (Figure 24.69: The Sample Trajectories Dialog Box (p. 2874)). Information about discrete phase reporting is discussed in detail in Sampling of Trajectories (p. 2873), Histogram Reporting of Samples (p. 2875), and Summary Reporting of Current Particles (p. 2883).

41.12. S2S Information

Ansys Fluent allows you to view the values of the view factor and radiation emitted from one zone to another. You will use the S2S Information Dialog Box (p. 5603) (Figure 16.47: The S2S Information Dialog Box (p. 2247)) to generate a report of these values. For details on reporting S2S information, refer to Reporting Radiation in the S2S Model (p. 2246).

41.13. Reference Values

You can control the reference values that are used in the computation of derived physical quantities and non-dimensional coefficients. These reference values are used only for postprocessing.

Some examples of the use of reference values include the following:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Force coefficients use the reference area, density, and velocity. In addition, the pressure force calculation uses the reference pressure.
- Moment coefficients use the reference length, area, density and velocity. In addition, the pressure force calculation uses the reference pressure.
- Reynolds number uses the reference length, density, and viscosity.
- Pressure and total pressure coefficients use the reference pressure, density, and velocity.
- Entropy uses the reference density, pressure, and temperature.
- Skin friction coefficient uses the reference density and velocity.
- Heat transfer coefficient uses the reference temperature.
- Turbomachinery efficiency calculations use the ratio of specific heats.

For additional information, see the following sections:

41.13.1. Setting Reference Values

41.13.2. Setting the Reference Zone

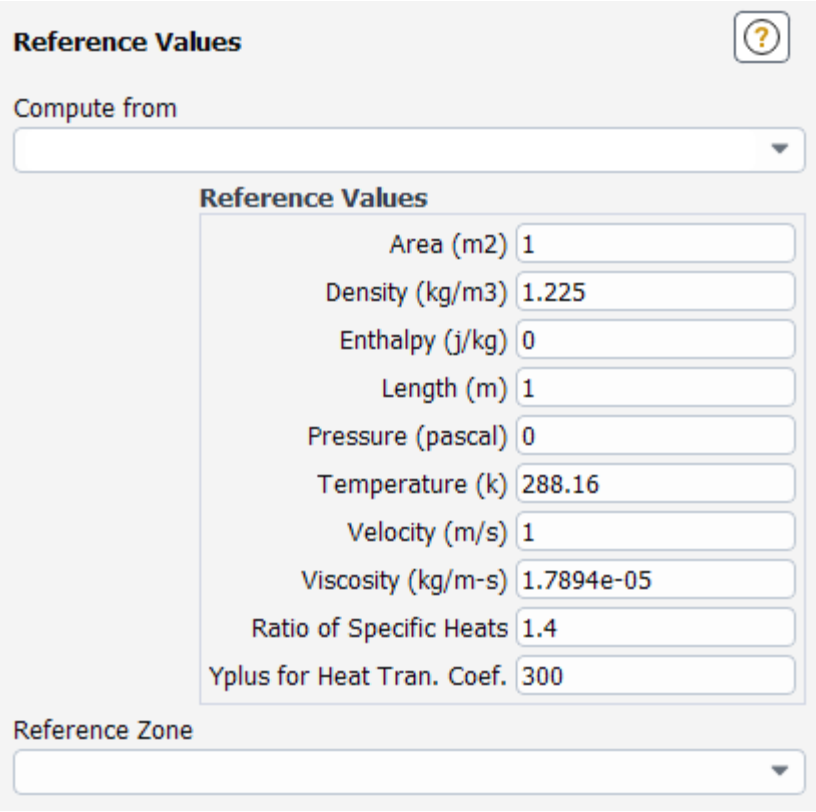
41.13.1. Setting Reference Values

To set the reference quantities used for computing normalized flow-field variables, use the Reference Values Task Page (p. 5154) (Figure 41.36: The Reference Values Task Page (p. 4160)).

Setup → Reference Values

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 41.36: The Reference Values Task Page



You can specify the reference values manually or compute them based on values of physical quantities at a selected boundary zone. The reference values to be set are **Area**, **Density**, **Enthalpy**, **Length**, **Pressure**, **Temperature**, **Velocity**, dynamic **Viscosity**, **Ratio Of Specific Heats**, and **Yplus for Heat Transfer Coefficient**.

For 2D problems, an additional quantity, **Depth**, can also be defined. This quantity will be used for reporting fluxes and forces, as well as relevant variables computed using the Surface Integrals Dialog Box (p. 5325) and the Volume Integrals Dialog Box (p. 5329) (for example, **Area**, **Flow Rate**, **Mass Flow Rate**, **Volume**, and so on). You should verify that the value and units of **Depth** corresponds to the depth dimension of your application prior to reporting any of the variables above.

Important:

The units for **Depth** are set independently from the units for **Length** in the Set Units Dialog Box (p. 4693).

If you want to compute reference values from the conditions set on a particular boundary zone, select the zone in the **Compute From** drop-down list. Note, however, that depending on the boundary condition used, only some of the reference values may be set. For example, the reference length and area will not be set by computing the reference values from a boundary condition; you will need to set these manually.

To set the values manually, simply enter the value for each under the **Reference Values** heading.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

41.13.2. Setting the Reference Zone

If you are solving a flow involving multiple reference frames or sliding meshes, you can plot velocities and other related quantities relative to the motion of a specified "reference zone". Choose the desired zone in the **Reference Zone** drop-down list. Changing the reference zone allows you to plot velocities (and total pressure, temperature, and so on) relative to the motion of different zones. See Modeling Flows with Moving Reference Frames (p. 1769) for details about postprocessing of relative quantities.

41.14. Summary Reports of Case Settings

You may sometimes find it useful to get a report of the current settings in your case. In Ansys Fluent, you can list the settings for physical models, boundary conditions, material properties, and solver controls. This report allows you to get an overview of your current problem definition quickly, instead of having to check the settings in each dialog box.

For additional information, see the following section:

41.14.1. Modified Settings Summary

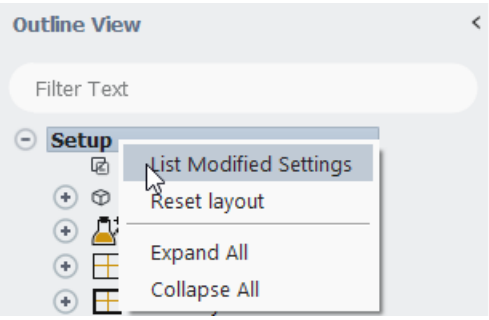
41.14.2. Generating a Summary Report

41.14.1. Modified Settings Summary

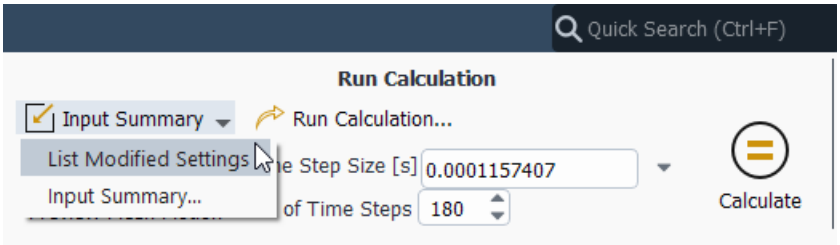
The **Modified Settings Summary** table lets you quickly review the non-default settings in the current case. You can double-click on any listed setting to open the corresponding dialog box or task page, allowing you to modify its value or the value of any related setting.

You can open the **Modified Settings Summary** by:

- right-clicking the **Setup** branch in the tree and selecting **List Modified Settings**. *You can also right-click many of the sub-branches to see a reduced list of modified settings.*



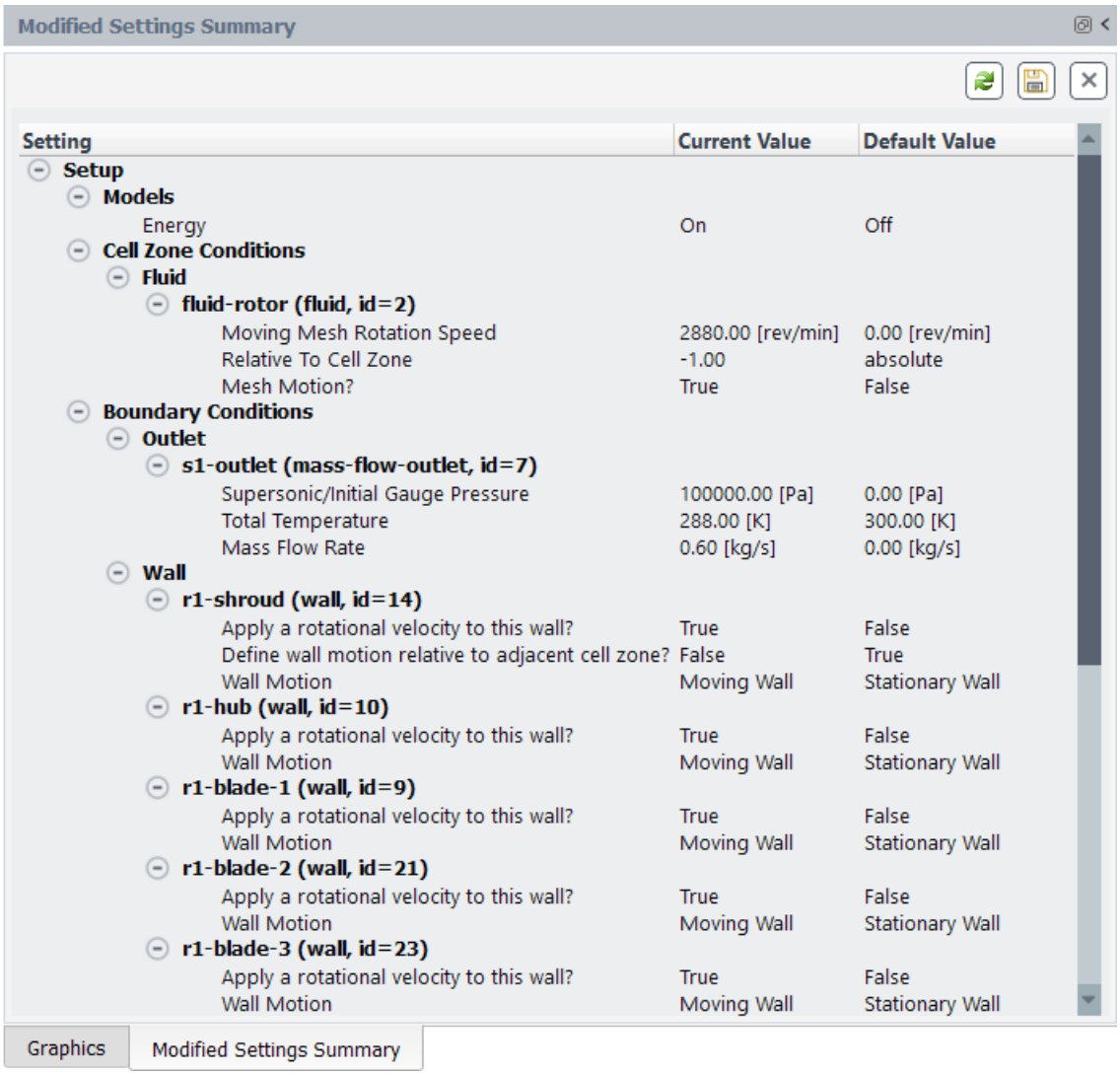
- selecting **List Modified Settings** in the **Input Summary** drop-down list in the **Solution** ribbon tab (**Run Calculation** group box).



The summary table opens tabbed with the graphics window.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 41.37: The Modified Settings Summary Table



Click refresh ( ) after changing settings to update the table.

Click to write the **Modified Settings Summary** table to a file.

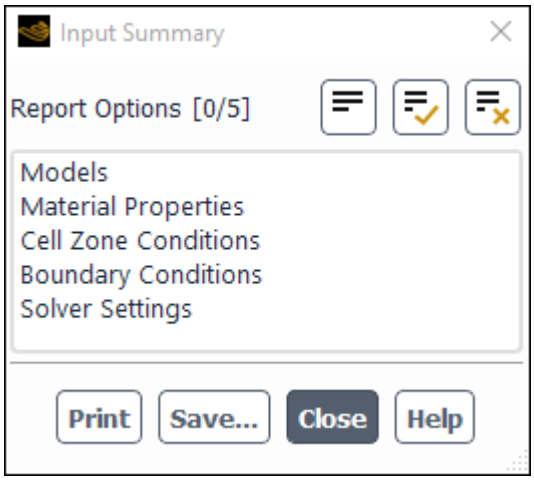
41.14.2. Generating a Summary Report

To generate a summary report you will use the Input Summary Dialog Box (p. 5497) (Figure 41.38: The Input Summary Dialog Box (p. 4163)).

Solution → **Run Calculation** → **Input Summary...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 41.38: The Input Summary Dialog Box



The steps are as follows:

1. Select the information you would like to see in the report (**Models**, **Boundary Conditions**, **Solver Controls**, and/or **Material Properties**) in the **Report Options** list.

2. To print the information to the Ansys Fluent console window, click the **Print** button. To save the information to a text file, click the **Save...** button and specify the filename in the resulting **Select File** dialog box.

41.15. System Resource Usage

You can print several reports of memory and processor usage from the **Parallel** ribbon tab.

41.15.1. Processor Information

41.15.2. Memory Information

41.15.3. Process and Model Timers

41.15.1. Processor Information

You can print out a table summarizing the processor usage on each machine that has compute processes spawned for the current session by clicking **CPU Info** and selecting **CPU Info** in the **Parallel** ribbon tab (**System** group box). For example, if processes have been spawned on three machines, a table similar to the following is displayed:

```
-------------------------------------------------------------------------------
                  | CPU                                  | System Mem (GB)
Hostname          | Sock x Core  Clock (MHz)  Load       | Total        Available
-------------------------------------------------------------------------------
host23            | 8 x 8, HT    2399.85      0.07       | 129.039      89.565
host24            | 8 x 8, HT    2399.95      1.08       | 129.039      32.516
host25            | 8 x 8, HT    2400.4       3.14       | 129.039      62.531
-------------------------------------------------------------------------------
Total             | 192          -            -          | 387.117      184.612
-------------------------------------------------------------------------------
```

Under CPU:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sock x Core

displays the number of processor sockets, number of cores per socket, and whether hyper-threading is used.

Clock (MHz)

is the processor speed.

Load

is the work load on the machine.

Under `System Mem (GB)`:

Total

is the total system memory on the machine.

Available

is the available system memory on the machine.

You can print out information about the GPUs available on the machine by clicking **CPU Info** and selecting **GPU Info** in the **Parallel** ribbon tab (**System** group box). If your machine has one or more suitable GPUs present, you can use these to accelerate parallel processing computations as described in Using General Purpose Graphics Processing Units (GPGPUs) With the Algebraic Multigrid (AMG) Solver (p. 4343).

```
CUDA visible GPUs on host01
  CUDA runtime version 5000
  Driver version 6000
  Number of GPUs 1
    0. Quadro K2100M
       3 SMs
       0.6665 GHz
       2.14748 GBytes
```

41.15.2. Memory Information

You can print out tables summarizing memory usage by node and by host by clicking **CPU Info** and selecting **Memory Usage** in the **Parallel** ribbon tab (**System** group box).

```
-------------------------------------------
       | Virtual Mem Usage (GB)|
ID     | Current      Peak     | Page Faults
-------------------------------------------
host   | 0.0680117    0.0695   | 2.667e+04
n0     | 0.0855195    0.114609 | 7.229e+04
n1     | 0.0869922    0.115367 | 6.988e+04
n2     | 0.0881836    0.117613 | 7.318e+04
n3     | 0.0840781    0.11384  | 6.999e+04
-------------------------------------------
Total  | 0.412785     0.53093  | 3.12e+05
-------------------------------------------

-----------------------------------------------------------------
                    | Virtual Mem Usage (GB)    | System Mem (GB)
Hostname            | Current      Peak         |
-----------------------------------------------------------------
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
host01              | 0.412785     0.53093      | 32.673
------------------------------------------------------------------
Total               | 0.412785     0.53093      |
------------------------------------------------------------------
```

Under `Virtual Mem Usage (GB)`:

Current

is the virtual memory usage at the time the report is generated.

Peak

is the peak virtual memory usage.

(Linux only) Under `Resident Mem Usage (GB)`:

Current

is the resident memory usage at the time the report is generated.

Peak

is the peak resident memory usage.

Page Faults

is the number of page faults that have occurred.

You can use these commands to plan Ansys Fluent jobs and machines accordingly. They may also be useful to diagnose performance problems.

41.15.3. Process and Model Timers

You can print out detailed information about the CPU timings for the current session as well as solver timings by clicking **CPU Info** and selecting **Time Usage** in the **Parallel** ribbon tab (**System** group box). The solver timings presented are described in Checking Parallel Performance (p. 4347).

```
-----------------------------------------
       | CPU Time Usage (Seconds)
ID     | User         Kernel   Elapsed
-----------------------------------------
host   | 1.34161      0.452403 2328.63
n0     | 3.47882      0.702005 2325.88
n1     | 19.9525      1.15441  2325.87
n2     | 19.8433      0.873606 2325.87
n3     | 19.8589      0.764405 2325.86
-----------------------------------------
Total  | 64.4752      3.94683  -
-----------------------------------------

Model Timers (Host)
  Other Models Time:                                  2.858 sec
  Total Time:                                         2.858 sec

Model Timers
  Other Models Time:                                  2.871 sec
  Total Time:                                         2.871 sec

Performance Timer for 36 iterations on 4 compute nodes
```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
  Average wall-clock time per iteration:              0.080 sec
  Global reductions per iteration:                       63 ops
  Global reductions time per iteration:               0.000 sec (0.0%)
  Message count per iteration:                         1955 messages
  Data transfer per iteration:                        2.239 MB
  LE solves per iteration:                               4 solves
  LE wall-clock time per iteration:                   0.045 sec (56.2%)
  LE global solves per iteration:                        4 solves
  LE global wall-clock time per iteration:            0.000 sec (0.4%)
  LE global matrix maximum size:                        20
  AMG cycles per iteration:                           8.833 cycles
  Relaxation sweeps per iteration:                      378 sweeps
  Relaxation exchanges per iteration:                   383 exchanges

  Total wall-clock time:                              2.894 sec
  Total CPU time:                                    11.185 sec
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 42: Field Function Definitions

You must select flow variables for a number of tasks in Ansys Fluent. The values are computed and placed in temporary memory that is allocated for storing the results for each cell. For example, the **Compute** command associated with a dialog box that contains the field variable drop-down list calculates the values of the selected function and places them into temporary storage.

Node, Cell, and Facet Values (p. 4167) and Velocity Reporting Options (p. 4169) provide some general information related to the field variables. In Field Variables Listed by Category (p. 4171), the variables are listed by category in Table 42.2:**Pressure** and **Density** Categories (p. 4175) – Table 42.21:**Acoustics** Category (p. 4203). These tables will also indicate when each variable will be available. Alphabetical Listing of Field Variables and Their Definitions (p. 4205) contains an alphabetical listing of the variables along with their definitions. All variables appear as they would in the variable selection drop-down lists that are contained in many of the Ansys Fluent dialog boxes. Custom Field Functions (p. 4285) explains how you can calculate your own field function.

42.1. Node, Cell, and Facet Values

42.2. Velocity Reporting Options

42.3. Field Variables Listed by Category

42.4. Alphabetical Listing of Field Variables and Their Definitions

42.5. Custom Field Functions

42.1. Node, Cell, and Facet Values

For the following discussion, "surface" refers to a collection of facets, lines or points that are created and manipulated in the **Setting Up Domain** ribbon tab. In most cases, these surfaces are created by computing intersections of constant isovalues with the domain cells or with existing surfaces.

For additional information, see the following sections:

42.1.1. Cell Values

42.1.2. Node Values

42.1.3. Facet Values

42.1.1. Cell Values

Ansys Fluent stores most variables in cells. For postprocessing, the entire region contained within the cell has this value. A surface cell value is the value of the cell that has been intersected by a surface facet or line, or that contains a surface point. Since surface facets and lines are created from the intersection of isovalues and the existing mesh cells, this is a unique definition. Typically, the cell value on a boundary is the value in the cell adjacent to the boundary. For face-only functions like **Wall Shear Stress**, the cell value is the area-weighted average from the face values that define that cell as `c0`. This value is used for the cell values of postprocessing surfaces. But for boundary faces, the cell value actually displays/uses the exact face value.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

42.1.2. Node Values

Node values are explicitly defined or obtained by weighted averaging of the cell data. Various boundary conditions impose values of field variables at the domain boundaries, so mesh node values on these boundary zones are obtained by simple averaging of the adjacent boundary face data. In addition, for several variables (for example, node coordinates or node displacements for the structural model) explicit node values are available at all nodes.

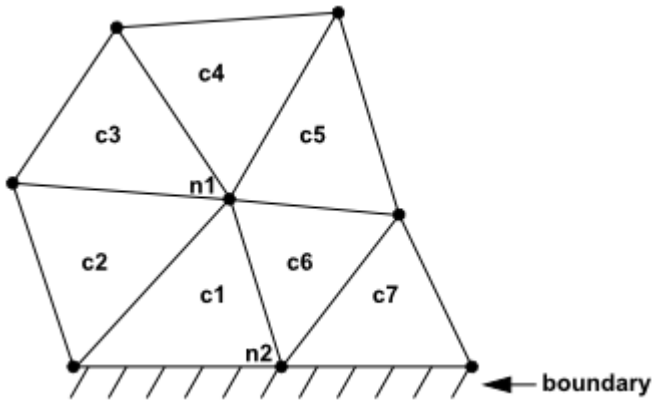
Computation of node values is performed in two steps:

1. Values at all nodes are initialized to the weighted average of the surrounding cell values. The weights are the inverses of the cell volumes that neighbor the nodes.

2. At boundaries, these node values are overwritten with the simple average of the boundary face values. Variables for which explicit node values are available at boundaries are indicated by *bnv* in Table 42.2:**Pressure** and **Density** Categories (p. 4175) – Table 42.21:**Acoustics** Category (p. 4203).

For example, in Figure 42.1: Computing Node Values (p. 4168), the value at node $n1$ will be computed from the weighted average of the values in the surrounding cells ($c1$ — $c6$). The value at node $n2$ will be computed from the simple average of the boundary faces ($bf1$ and $bf2$) if there are explicit boundary values available for the variable in question.

Figure 42.1: Computing Node Values



42.1.2.1. Vertex Values for Points That Are Not Mesh Nodes

The values of the nodes on surfaces are linearly interpolated from the mesh node data. For zone surfaces, the nodes on the surface and the zone correspond; therefore, the values are identical. For surfaces that are not zone surfaces (for example, isosurfaces, plane surfaces, and so on), the node values are interpolated from mesh nodes on the cell faces intersected by the postprocessing surface. For point surfaces and structural point surfaces, the value is interpolated from all the mesh nodes of the cell containing the point.

42.1.3. Facet Values

Facets can be created on preprocessing surfaces and postprocessing surfaces.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

42.1.3.1. Facet Values on Zone Surfaces

The interior facets on a zone surface are associated with two cells (`c0` and `c1`). The values of a specified variable on such facets are computed as the arithmetic average of the two cell values of the selected variable.

The boundary facet values of a primary field variables on a zone surface are taken from the specified boundary conditions. The boundary values of dependent variables are computed from the adjacent cell values (versus using the value of the primary variable specified at the boundary). For exact values of dependent variables at the boundaries, define a custom field function specifying the boundary conditions as the primary variables. For more information on custom field functions, see Custom Field Functions (p. 4285).

42.1.3.2. Facet Values on Postprocessing Surfaces

Each facet on a postprocessing surface is associated with a cell. The values of a specified variable on facets are the same as the cell values of the selected variable in the associated cells (this includes iso surfaces, planes, lines, points, rakes, quadric, and so on).

42.2. Velocity Reporting Options

The following methods are available for reporting velocities:

- Cartesian velocities:

  These velocities are based on the Cartesian coordinate system used by the geometry. To report Cartesian velocities, select **X Velocity**, **Y Velocity**, or **Z Velocity**. This is the most common type of velocity reported.

- Cylindrical velocities:

  These velocities are the axial, radial, and tangential components based on the following coordinate systems:

  – For axisymmetric problems, in which the rotation axis must be the x axis, the x direction is the axial direction and the y direction is the radial direction. (If you model axisymmetric swirl, the swirl direction is the tangential direction.)

  – For 2D problems involving a single cell zone, the z direction is the axial direction, and its origin is specified in the Fluid Dialog Box (p. 4961).

  – For 3D problems involving a single cell zone, the coordinate system is defined by the rotation axis and origin specified in the Fluid Dialog Box (p. 4961).

  – For problems involving multiple zones (for example, multiple reference frames or sliding meshes), the coordinate system is defined by the rotation axis specified in the Fluid Dialog Box (p. 4961) (or Solid Dialog Box (p. 4973)) for the "reference zone". The reference zone is chosen in the Reference Values Task Page (p. 5154), as described in Reference Values (p. 4158). Recall that for 2D problems, you will specify only the axis origin; the z direction is always the axial direction.

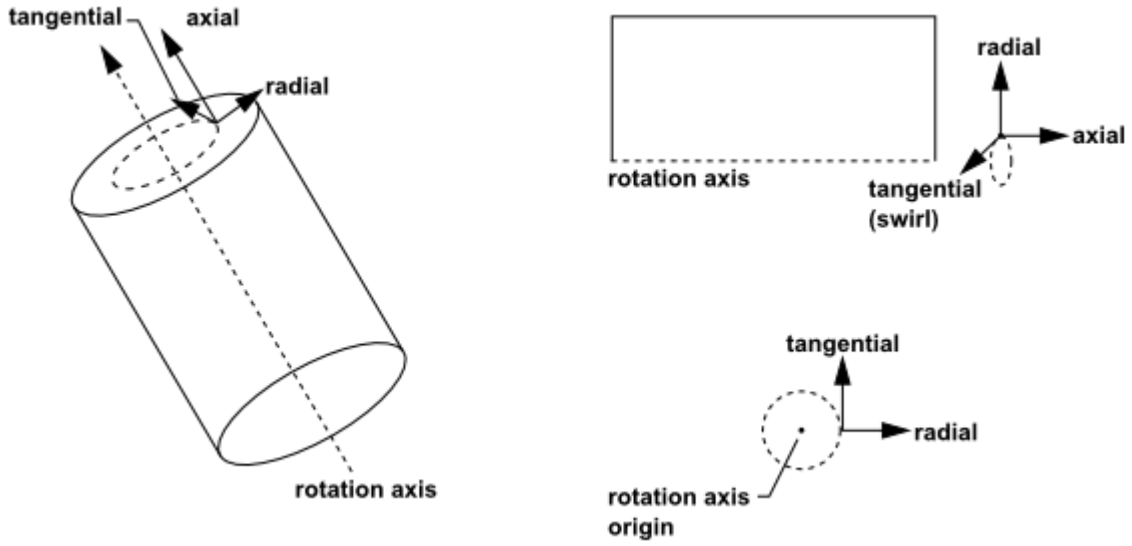
  For all of the above definitions of the cylindrical coordinate system, positive radial velocities point radially out from the rotation axis, positive axial velocities are in the direction of the rotation axis

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

vector, and positive tangential velocities are based on the right-hand rule using the positive rotation axis.

To report cylindrical velocities, select **Axial Velocity**, **Radial Velocity**, and so on. Figure 42.2: Cylindrical Velocity Components in 3D, 2D, and Axisymmetric Domains (p. 4170) illustrates the cylindrical velocities available for different types of domains. For 3D problems, you can report axial, radial, and tangential velocities. For 2D problems, radial and tangential velocities are available. For axisymmetric problems, you can report axial and radial velocities, and, if you are modeling axisymmetric swirl, you can also report the swirl velocity (which is equivalent to the tangential velocity).

Figure 42.2: Cylindrical Velocity Components in 3D, 2D, and Axisymmetric Domains



- Relative velocities: These velocities are based on the coordinate system and motion of a moving reference frame. They are useful when you are modeling your flow using a moving reference frame, a mixing plane, multiple reference frames, or sliding meshes. (For example, see Modeling Flows with Moving Reference Frames (p. 1769) for information about modeling flow in moving zones.) To report relative velocities, select **Relative X Velocity**, **Relative Y Velocity**, **Relative Radial Velocity**, and so on. (Note that you can report relative velocities for both Cartesian and cylindrical components.)

  If you are using a single moving reference frame, the relative velocity values will be reported with respect to the moving frame. If you are using multiple reference frames, mixing planes, or sliding meshes, you will need to specify the frame to which you want the velocities to be relative by choosing the appropriate cell zone as the **Reference Zone** in the Reference Values Task Page (p. 5154) (see Reference Values (p. 4158)). The axis of rotation for each cell zone is defined in the associated Fluid Dialog Box (p. 4961) or Solid Dialog Box (p. 4973). (See Specifying the Rotation Axis (p. 1323) or Defining Zone Motion (p. 1328) for details.)

  Note that if your problem does not involve any moving zones, relative and absolute velocities will be equivalent.

Note that relative velocities can also be used to compute stagnation quantities (total pressure and total temperature), and that the cylindrical coordinate systems described in the second item above are used for defining the **Axial Coordinate** and **Radial Coordinate** as well.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

42.3. Field Variables Listed by Category

The field variables available in each category are summarized in the following tables:

- Table 42.1:**Expressions** Category (p. 4175)
- Table 42.2:**Pressure** and **Density** Categories (p. 4175)
- Table 42.3:**Velocity** Category (p. 4176)
- Table 42.4:**Temperature**, **Radiation**, **Solidification/Melting**, and **Two-Temperature Model** Categories (p. 4178)
- Table 42.5:**Turbulence** Category (p. 4180)
- Table 42.6:**Runtime DFT** Category (p. 4182)
- Table 42.7:**Species**, **Reactions**, **Pdf**, and **Premixed Combustion** Categories (p. 4182)
- Table 42.8:**NOx**, **Soot**, and **Steady|Unsteady Statistics** Categories (p. 4185)
- Table 42.9:**Phases**, **Discrete Phase Model**, **Granular Pressure**, **Granular Temperature**, and **Wall Film** Categories (p. 4187)
- Table 42.10:**Properties** Category (p. 4191)
- Table 42.11:**Eulerian Wall Film** Category (p. 4192)
- Table 42.12:**Sensitivities** Category (p. 4193)
- Table 42.13:**Wall Fluxes**, **User Defined Scalars**, and **User Defined Memory** Categories (p. 4196)
- Table 42.14:**Cell Info** and **Mesh** Categories (p. 4197)
- Table 42.15:**Perforated Walls** Category (p. 4200)
- Table 42.16:**Mesh** Category (Turbomachinery-Specific Variables) (p. 4200)
- Table 42.17:**Residuals** Category (p. 4201)
- Table 42.18:**Derivatives** Category (p. 4201)
- Table 42.19:**Potential** Category (p. 4202)
- Table 42.20:**Lithium** Category (p. 4203)
- Table 42.21:**Acoustics** Category (p. 4203)
- Table 42.22:**Structure** Category (p. 4205)

In these tables, the following restrictions apply to marked variables:

2d	available only for 2D flows
2da	available only for 2D axisymmetric flows (with or without swirl)
2dasw	available only for 2D axisymmetric swirl flows
3d	available only for 3D flows
adj	available only with data from the adjoint solver
adis	available only for adjoint calculations that use the dissipation stabilization scheme
adt	available only when the turbulence adjoint equations are solved
adnn	available only when the neural network model is applied for turbulence model optimization
adjtr	available only after offline training for turbulence model optimization

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

aen	available only when the energy adjoint equation is solved
ark	available only with the Aungier-Redlich-Kwong real gas model
asfft	available only when created using the Acoustic Sources FFT Dialog Box (p. 5224) or related text commands
awe	available only with the acoustics wave equation model
bcde	available only with the bounded central differencing scheme, when boundedness is specified by an expression
bns	available only for broadband noise source models
bnv	node values available at boundaries
cgq	available only when the cell gradient quality criterion is enabled for poor mesh numerics
cmp	available only for compressible flow
con	available only for contact detection simulations using the contact marks method
cpl	available only in the density-based solvers
cv	available only for cell values (**Node Values** option disabled)
cve	available only for cell values and only cell values are exported to Common Fluids Format - Post.
ddpm	available only when the DDPM model is used
des	available when the turbulence model includes the DES turbulence model
dil	not available with full multicomponent diffusion
dm	available only for dynamic mesh calculations
do	available only when the discrete ordinates radiation model is used
dpm	available only for coupled discrete phase calculations
dpmean	available only when Mean Values are enabled in the DPM dialog box
dprms	available only when RMS Values are enabled in the DPM dialog box
dtrm	available only when the discrete transfer radiation model is used
fwh	available only with the Ffowcs Williams and Hawkings acoustics model

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

e	available only for energy calculations
echem	available only with the Electrochemical Reactions model
edc	available only with the EDC model for turbulence-chemistry interaction
emm	available only when the Eulerian multiphase model is used
ewf	available only with the Eulerian Wall Film model
ewt	available only with the enhanced wall treatment
fv	available for face values
fm	available only when the frame motion option is used
gap	available only if a gap region is created as part of the gap model
gran	available only if a granular phase is present
h2o	available only when the mixture contains water
hn	available only for hanging node adaption
id	available only when the ideal gas law is enabled for density
ke	available when one of the k - ε turbulence models is used
kklo	available when the k - kl - ω model is used
kw	available when one of the k - ω turbulence models is used
les	available when the LES turbulence model is used
lwf	available only when the Lagrangian wall film model is used
mc	available only when the Monte Carlo model is used
mcv	available only when check-verbosity is set to 2
melt	available only when the melting and solidification model is used
mix	available only when the multiphase mixture model is used
mm	available only when the mesh motion option is used
mp	available only for multiphase models
netm	available only for porous cell zones with the non-equilibrium thermal model enabled
nox	available only for NOx calculations

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

nv	uses explicit node value function
nvs	uses explicit node value solutions
os	available only when overset meshing is used
p1	available only when the P-1 radiation model is used
pdf	available only for non-premixed combustion calculations
pmx	available only for premixed combustion calculations
pot	available only when the potential equation is solved
ppmx	available only for partially premixed combustion calculations
r	available only when the Rosseland radiation model is used
rad	available only for radiation heat transfer calculations
rc	available only for finite-rate reactions
ret	available only if the `solve/set/advanced/retain-cell-residuals` text command is enabled
rg	available only for the real gas models
rsm	available when the Reynolds stress turbulence model is used
s2s	available only when the surface-to-surface radiation model is used
sa	available when the Spalart-Allmaras turbulence model is used
sas	available when the turbulence model includes Scale-Adaptive Simulation (SAS)
sbes	available when the turbulence model includes the Stress-Blended Eddy Simulation (SBES) turbulence model
scq	available only when the solution and cell quality criterion is enabled for poor mesh numerics
sdes	available when the turbulence model includes the Shielded Detached Eddy Simulation (SDES) turbulence model
seg	available only in the pressure-based solver
sol	available only when the solar model is used
soot	available only for soot calculations
sp	available only for species calculations
spo	available only when a sponge layer is defined

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

sr	available only for surface reactions
sst	available when the Transition SST model is used
stat	available only with data sampling for transient statistics
stcm	available only for stiff chemistry calculations
strc	available only when a structural model is selected
t	available only for turbulent flows
th	available only when thin walls are present
trn	available only for transient flow calculations
turbo	available only when a turbomachinery topology has been defined
udm	available only when a user-defined memory is used
uds	available only when a user-defined scalar is used
v	available only for viscous flows
vof-dpm	available only for VOF-to-DPM model transition calculations

Table 42.1: Expressions Category

Category	Variable
Expressions...	Supported expressions must: • be of type **Double** or **Double Vector**. *If it is of type **Double Vector**, the components and magnitude are available for postprocessing.* • evaluate to a field (and not a single-value) • be fully specified. *For example, if the expression computes mass fraction, the specie must also be included in the definition (`MassFraction(species='co2')`). Similarly, for a multiphase case, the phase must be specified, where required (`Volume-Fraction(phase='water')`).*

Table 42.2: Pressure and Density Categories

Category	Variable
Pressure...	**Static Pressure** (bnv) **Pressure Coefficient**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Dynamic Pressure **Absolute Pressure** (bnv) **Total Pressure** (bnv) **Relative Total Pressure** **Capillary-Pressure** (emm) **Pressure Discontinuity Sensor** (cpl)
Density...	**Density** **Density All** **Schlieren** (cmp) **Rescaled Schlieren** (cmp) **Rescaled Streamwise Schlieren** (cmp) **Logarithmic Streamwise Schlieren** (cmp)

Table 42.3: Velocity Category

Category	Variable
Velocity...	**Velocity Magnitude** (bnv) **X Velocity** (bnv) **Y Velocity** (bnv) **Z Velocity** (3d, bnv) **Swirl Velocity** (2dasw, bnv) **Axial Velocity** (2da or 3d) **Radial Velocity** **Stream Function** (2d) **Tangential Velocity** **Mach Number** (id or rg) **Relative Velocity Magnitude** (bnv) **Relative X Velocity** (bnv) **Relative Y Velocity** (bnv) **Relative Z Velocity** (3d, bnv)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Relative Axial Velocity (2da)
	Relative Radial Velocity (2da)
	Relative Swirl Velocity (2dasw, bnv)
	Relative Tangential Velocity
	Relative Mach Number (id or rg)
	Mesh X-Velocity (nv)
	Mesh Y-Velocity (nv)
	Mesh Z-Velocity (3d, nv)
	Velocity Angle
	Relative Velocity Angle
	Vorticity Magnitude (v)
	Helicity (v, 3d)
	X-Vorticity (v, 3d)
	Y-Vorticity (v, 3d)
	Z-Vorticity (v, 3d)
	Cell Reynolds Number (v, cve)
	Compressible Wall Reynolds Number (e,fv,v,cmp)
	Cell Convective Courant Number (seg, cve)
	Moving Mesh Courant Number (trn, fm or mm or dm)
	Multiphase Minimum Time Scale (trn, fm or mm or dm)
	Cell Acoustic Courant Number (cmp, seg, cve)
	Preconditioning Reference Velocity (cpl)
	Q Criterion Normalized
	Q Criterion Raw

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Lambda 2 Criterion

Table 42.4: Temperature, Radiation, Solidification/Melting, and Two-Temperature Model Categories

Category	Variable
Temperature...	**Static Temperature** (e, bnv, nv) **User Energy Source** (e) **User Volumetric Energy Source** (e) **Total Temperature** (e, nv) **Sensible Enthalpy** (e, nv) **Enthalpy** (e, nv) **Relative Total Enthalpy** (e, nv) **Relative Total Temperature** (e) **Rothalpy** (e, nv) **Fine Scale Temperature** (edc,e) **Wall Temperature** (fv, e, v) **Wall Temperature (Thin)** (th, fv, e, v) **Wall Adjacent Temperature** (fv, e) **Total Enthalpy** (e) **Total Enthalpy Deviation** (e) **Entropy** (e) **Total Energy** (e) **Internal Energy** (e) **Non-Equilibrium Thermal Model Source** (e, netm) **NLBF Averaged Temperature** (emm) **Yplus Based Heat Transfer Reference Temperature** (fv, e, v)
Radiation...	**Absorption Coefficient** (r, p1, do, mc, or dtrm) **Scattering Coefficient** (r, p1, do, or mc) **Refractive Index** (do or mc)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Radiation Temperature (p1,do, or mc) **Incident Radiation** (p1, do, or mc) **Incident Radiation (Band n)** (p1 (non-gray), do (non-gray), or mc (non-gray)) **Volumetric Absorbed Radiation** (p1, do, or mc) **Volumetric Emitted Radiation** (p1, do, or mc) **Wall Irradiation Flux.Normalized Std Deviation** (mc) **Radiation Intensity.Normalized Std Deviation** (mc) **Surface Cluster ID** (fv, s2s)
Solidification/ Melting	**Liquid Fraction** (melt) **Contact Resistivity** (fv, melt) **X Pull Velocity** (melt (if calculated)) **Y Pull Velocity** (melt (if calculated)) **Z Pull Velocity** (melt (if calculated), 3d) **Axial Pull Velocity** (melt (if calculated), 2da) **Radial Pull Velocity** (melt (if calculated), 2da) **Swirl Pull Velocity** (melt (if calculated), 2dasw)
Two-Temperature Model	**Translational-Rotational Temperature** (e, cpl) **Vibrational-Electronic Temperature** (e, cpl) **Internal Energy** (e, cpl) **Translational-Rotational Energy** (e, cpl) **Vibrational-Electronic Energy** (e, cpl) **Translational-Rotational over Vibrational-Electronic Temperature** (e, cpl) **Vibrational-Electronic Conductivity** (e, cpl) **Vibrational-Electronic Specific Heat** (e, cpl) **Mass-Averaged Relaxation Time** (e, cpl)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Frozen Sound Speed (e, cpl) **Dissociation Vibration Source** (e, cpl)

Table 42.5: Turbulence Category

Category	Variable
Turbulence...	**Turbulent Kinetic Energy (k)** (ke, kw, kklo, sst, rsm, les; sbes, sdes, bnv, nv, or emm) **Laminar Kinetic Energy** (kklo) **Total Fluctuation Energy** (kklo) **Turbulent Intensity** (ke, kw, kklo, sst, rsm, les, sbes, sdes) **Intermittency** (sst, kw, sas, des, sbes, sdes) **Intermittency Effective** (sst, sbes (with Transition SST only), sdes (with Transition SST only)) **Momentum Thickness Re** (sst, sbes (with Transition SST only), sdes (with Transition SST only)) **Geometric Roughness Height** (sst, sbes (with Transition SST only), sdes (with Transition SST only)) **UU Reynolds Stress** (rsm; emm) **VV Reynolds Stress** (rsm; emm) **WW Reynolds Stress** (rsm; emm) **UV Reynolds Stress** (rsm; emm) **UW Reynolds Stress** (rsm, 3d; emm) **VW Reynolds Stress** (rsm, 3d; emm) **Resolved UV Reynolds Stress** (sas, des, sbes, sdes, les) **Resolved UW Reynolds Stress** (sas, des, sbes, sdes, les) **Resolved VW Reynolds Stress** (sas, des, sbes, sdes, les)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Turbulent Dissipation Rate (Epsilon) (ke, kw, kklo, sst, sbes, sdes, or rsm; bnv (k-epsilon model only), nv (k-epsilon model only), or emm)
	Specific Dissipation Rate (Omega) (kw, kklo, sst, sbes, sdes, rsm)
	Production of k (ke, kw, kklo, sst, rsm, les; emm)
	Production of laminar k (kklo)
	Modified Turbulent Viscosity (sa, des)
	Turbulent Viscosity (sa, ke, kw, kklo, sst, sbes, sdes, rsm, sas, des, les)
	Turbulent Viscosity (large-scale) (kklo)
	Turbulent Viscosity (small-scale) (kklo)
	Effective Viscosity (sa, ke, kw, kklo, sst, sbes, sdes, rsm, des; emm)
	Turbulent Viscosity Ratio (ke, kw, kklo, sst, sbes, sdes, rsm, sa, des, les; emm)
	LES Subgrid Turbulent Viscosity (les)
	Subgrid Kinetic Energy (les)
	Subgrid Turbulent Viscosity (les)
	Subgrid Turbulent Viscosity Ratio (les)
	Subgrid Effective Viscosity (les)
	Subgrid Filter Length (les)
	Subgrid Test-Filter Length (les)
	Subgrid Dissipation Rate (les)
	Subgrid Dynamic Viscosity Const (les)
	Subgrid Dynamic Prandtl Number (les)
	Subgrid Dynamic Sc of Species (les)
	Subtest Kinetic Energy (les)
	Effective Thermal Conductivity (t, e)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Effective Prandtl Number (t, e) **Wall Ystar** (fv, ke, kw, kklo, sst, sbes, sdes, rsm, les) **Wall Yplus** (fv, t) **Turbulent Reynolds Number (Re_y)** (ke, kw, kklo, sst, sbes, sdes, rsm; ewt) **Relative Length Scale (DES)** (des) **DES TKE Dissipation Multiplier** (des) **Curvature Correction Function fr** (sa, ke, kw, sst, sbes, sdes, sas, des) **Shielding Function for SBES or SDES** (sbes, sdes) **Blending Function for GEKO** (geko) **Built-in Correlation for GEKO Blending Function** (geko)

Table 42.6: Runtime DFT Category

Category	Variable
Runtime DFT...	**<dft-dataset-name>_<frequency>-ba-mag** (stat, trn) **<dft-dataset-name>_<frequency>-ta-mag** (stat, trn) **<dft-dataset-name>_<frequency>-ta-phase** (stat, trn)

Table 42.7: Species, Reactions, Pdf, and Premixed Combustion Categories

Category	Variable
Species...	**Mass fraction of *`species-n`*** (sp, pdf, or ppmx; nv) **Mole fraction of *`species-n`*** (sp, pdf, or ppmx) **Mass Concentration of *`species-n`* (Phase Level)** (sp) **Mass Concentration of *`species-n`* (Mixture Level)** (sp)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Molar Concentration of *species-n* (sp, pdf, or ppmx) **Lam Diff Coef of *species-n*** (sp, dil) **Eff Diff Coef of *species-n*** (t, sp, dil) **Thermal Diff Coef of *species-n*** (sp) **Enthalpy of *species-n*** (sp) ***species-n* Source Term** (rc, cpl) **Surface Deposition Rate of *species-n*** (sr) **Surface Coverage of *species-n*** (sr) **Porous Deposition Rate of *species-n*** (sr) **Relative Humidity** (sp, pdf, or ppmx; h2o) **Time Step Scale** (sp, stcm) **Fine Scale Mass fraction of *species-n*** (edc) **Cell Time Scale** (edc) **EDC Cell Volume Fraction** (edc) **DRG Reduced Number of Species** (sp) **Reactor Net Zone ID** (sp) **Reactor Net Temperature** (sp) **Reactor Net Mass fraction of *species-n*** (sp) **Surface Corrosion Rate of *species-n*** (echem)
Reactions...	**Rate of Reaction-n** (rc) **Kinetic Rate of Reaction-n** (rc) **Turbulent Rate of Reaction-n** (rc, t) **Heat of Reaction** (e, rc) **Net Reaction Rate of *Species-n*** (edc, stcm) **DRG Reduced Number of Reactions** (sp)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Echem Reaction Rate of Reaction-n (echem)
Pdf...	**Mean Mixture Fraction** (pdf or ppmx; nv) **Secondary Mean Mixture Fraction** (pdf or ppmx; nv) **Mixture Fraction Variance** (pdf or ppmx; nv) **Secondary Mixture Fraction Variance** (pdf or ppmx; nv) **Fvar Prod** (pdf or ppmx) **Fvar2 Prod** (pdf or ppmx) **Scalar Dissipation** (pdf or ppmx) **Heat Release Rate** (pdf or ppmx) **PDF Table Adiabatic Enthalpy** (pdf or ppmx) **PDF Table Heat Loss/Gain** (e, pdf or ppmx)
Premixed Combustion...	**Progress Variable** (pmx or ppmx; nv) **Progress Variable Variance** (ppmx; nv) **Scalar Mass Fraction of *`species-n`*** (ppmx) **Forward Reaction Rate of *`PDF scalar-n`*** (ppmx) **Reverse Reaction Rate of *`PDF scalar-n`*** (ppmx) **Damkohler Number** (pmx or ppmx) **Stretch Factor** (pmx or ppmx) **Turbulent Flame Speed** (pmx or ppmx) **Static Temperature** (pmx or ppmx) **Product Formation Rate** (pmx or ppmx) **Laminar Flame Speed** (pmx or ppmx) **Critical Strain Rate** (pmx or ppmx) **Adiabatic Flame Temperature** (pmx or ppmx) **Unburnt Fuel Mass Fraction** (pmx or ppmx)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Strained Laminar Flame Speed (ppmx)

Table 42.8: NOx, Soot, and Steady|Unsteady Statistics Categories

Category	Variable
NOx...	**Mass fraction of Pollutant no** (nox) **Mass fraction of Pollutant hcn** (nox) **Mass fraction of Pollutant nh3** (nox) **Mass fraction of Pollutant n2o** (nox) **Mole fraction of Pollutant no** (nox) **Mole fraction of Pollutant hcn** (nox) **Mole fraction of Pollutant nh3** (nox) **Mole fraction of Pollutant n2o** (nox) **no Density** (nox) **hcn Density** (nox) **nh3 Density** (nox) **n2o Density** (nox) **RMS of Temperature** (nox) **Variance of Species** (nox) **Variance of Species 1** (nox) **Variance of Species 2** (nox) **Rate of NO** (nox) **Rate of Thermal NO** (nox) **Rate of Prompt NO** (nox) **Rate of Fuel NO** (nox) **Rate of N2OPath NO** (nox) **Rate of Reburn NO** (nox) **Rate of SNCR NO** (nox) **Rate of USER NO** (nox)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
Soot...	**Average Number of Particles in Soot Aggregate** (soot) **Mass fraction of soot** (soot) **Mass fraction of Nuclei** (soot) **Mole fraction of soot** (soot) **Normalized Concentration of Nuclei** (soot) **Normalized Soot Aggregation Moments** (soot) **Normalized soot moments** (soot) **Primary Particle Diameter** (soot) **Rate of Coagulation** (soot) **Rate of Nuclei** (soot) **Rate of Nucleation** (soot) **Rate of Oxidation** (soot) **Rate of Soot** (soot) **Rate of Soot Mass Nucleation** (soot) **Rate of Surface Growth** (soot) **Soot Density** (soot) **Soot Mean Diameter** (soot) **Soot Surface Area** (soot) **Soot Volume fraction** (soot)
Steady|Unsteady DPM Statistics...	**Accum DPM Parcels in Cell** (dpm, stat) **Accum DPM Particles in Cell** (dpm, stat) **Mean DPM** *n* (dpm, stat) **RMS DPM** *n* (dpm, stat)
Steady|Unsteady Statistics...	**Mean** *n* (stat) **Mean-** *cff_n* (stat) **RMSE** *n* (stat)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	RMSE- *cff_n* (stat)

Table 42.9: Phases, Discrete Phase Model, Granular Pressure, Granular Temperature, and Wall Film Categories

Category	Variable
Phase Interaction...	**Heat of Heterogeneous Reaction**
	Mass Transfer Rate *n*
	Saturation Vapor Pressure *n*
	Saturation Temperature *n*
	Latent Heat *n*
	Mass Transfer Coefficient (Kl)
	Flow Regime Blending Factors Fcc (mix)
	Flow Regime Blending Factors Fcd (mix)
	Flow Regime Blending Factors Fdc (mix)
	Flow Regime Blending Factors Fdd (mix)
	Slip X-velocity (mix)
	Slip Y-velocity (mix)
	Slip Z-velocity (mix, 3d)
	Relaxation Time (mix)
	Saturation Temperature (emm)
	Liquid Subcooling (emm)
	Wall Superheat (emm)
	Bubble Departure Diameter (emm)
	Bubble Departure Frequency (emm)
	Bubble Waiting Time (emm)
	Bubble Area Fraction (emm)
	Bubble Nucleation Site Density (emm)
Phases...	**Smoothed VOF Gradient-dX** (mp)
	Smoothed VOF Gradient-dY (mp)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Smoothed VOF Gradient-dZ (mp) **Smooth VOF Gradient Magnitude** (mp) **Volume fraction** (mp)
Lump Detection...	**Lump ID** (vof-dpm) **Lump Diameter** (vof-dpm) **Lump Density** (vof-dpm) **Lump Asph. by Rad. Std. Dev.** (vof-dpm) **Lump Asph. by Orthogonality** (vof-dpm) **Lump X, Y, Z Coordinate** (vof-dpm) **Lump Temperature** (vof-dpm, e) **Lump Enthalpy** (vof-dpm, e) **Lump Pressure** (vof-dpm, e)
Discrete Phase Variables...	**DPM Erosion Rate (DNV)** (dpm, cv, fv) **DPM Erosion Rate (Generic)** (dpm, cv, fv) **DPM Erosion Rate (Finnie)** (dpm, cv, fv) **DPM Erosion Rate (McLaury)** (dpm, cv, fv) **DPM Erosion Rate (Oka)** (dpm, cv, fv) **DPM Erosion Rate (Wall Shear)** (dpm, cv, fv, gran) **DPM Accretion** (dpm, cv, fv) **DPM Volume Fraction** (dpm, dpmean) **DPM Parcels in Cell** (dpm, dpmean) **DPM Particles in Cell** (dpm, dpmean) **DPM Number Density** (dpm, dpmean) **DPM Collision Rate** (dpm) **DPM X Velocity** (dpm, dpmean) **DPM Y Velocity** (dpm, dpmean) **DPM Z Velocity** (dpm, dpmean)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	DPM Diameter (dpm, dpmean)
	DPM Density (dpm, dpmean)
	DPM Temperature (dpm, dpmean)
	DPM Enthalpy (dpm, dpmean)
	DPM Specific Heat (dpm, dpmean)
	DPM D20 (dpm, dpmean)
	DPM D30 (dpm, dpmean)
	DPM D32 (dpm, dpmean)
	DPM D43 (dpm, dpmean)
	DPM Granular Temperature (ddpm, gran)
	DPM Conc of <*component*> (dpm, dpmean, sp)
	DPM RMS X Velocity (dpm, dprms)
	DPM RMS Y Velocity (dpm, dprms)
	DPM RMS Z Velocity (dpm, dprms)
	DPM RMS Diameter (dpm, dprms)
	DPM RMS Temperature (dpm, dprms)
	DPM Absorption Coefficient (dpm, rad)
	DPM Emission (dpm, rad)
	DPM Scattering (dpm, rad)
	DPM Concentration (dpm)
	DPM Wall X Force (dpm)
	DPM Wall Y Force (dpm)
	DPM Wall Z Force (dpm)
	DPM Wall Normal Pressure (dpm)
Discrete Phase Sources...	**DPM Mass Source** (dpm)
	DPM X Momentum Source (dpm)
	DPM Y Momentum Source (dpm)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	DPM Z Momentum Source (dpm, 3d) **DPM Swirl Momentum Source** (dpm, 2dasw) **DPM Sensible Enthalpy Source** (dpm, e, rc) **DPM Enthalpy Source** (dpm, e) **DPM Burnout** (dpm, sp, e) **DPM Evaporation/Devolatilization** (dpm, sp, e) **DPM *species-n* Source** (dpm, sp, e) **DPM Mixture Fraction Source** (dpm, pdf or ppmx) **DPM Mixture Fraction Secondary Source** (dpm, pdf or ppmx) **DPM Inert Source** (dpm, pdf or ppmx)
Granular Pressure...	**Granular Pressure** (emm, gran)
Granular Temperature...	**Granular Temperature** (emm, gran)
Wall Film...	**Wall Film Height** (dpm, lwf) **Wall Film Impingement Mass Flux** (dpm, lwf) **Wall Film Face Pressure** (dpm, lwf) **Wall Film Mass** (dpm, lwf) **Wall Film Temperature** (dpm, lwf) **Wall Film Heat Flux** (dpm, lwf) **Wall Film Heat Transfer Coefficient** (dpm, lwf) **Wall Film Surface Temperature** (dpm, lwf) **Wall Film X-Velocity** (dpm, lwf) **Wall Film Y-Velocity** (dpm, lwf) **Wall Film Z-Velocity** (dpm, lwf, 3d) **Wall Film Velocity Magnitude** (dpm, lwf) **Wall Leidenfrost Temperature** (dpm, lwf) **Wall Coverage** (dpm, lwf)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Film Mass Fraction of *material-i* (dpm, lwf)

Table 42.10: Properties Category

Category	Variable
Properties...	**Molecular Viscosity** (v) **Diameter** (mix, emm) **Granular Conductivity** (mix, emm, gran) **Thermal Conductivity** (e, v) **Thermal Conductivity XX** (e, v) **Thermal Conductivity XY** (e, v) **Thermal Conductivity XZ** (e, v) **Thermal Conductivity YX** (e, v) **Thermal Conductivity YY** (e, v) **Thermal Conductivity YZ** (e, v) **Thermal Conductivity ZX** (e, v) **Thermal Conductivity ZY** (e, v) **Thermal Conductivity ZZ** (e, v) **Specific Heat (Cp)** (e) **Specific Heat Ratio (gamma)** (id) **Gas Constant (R)** (id or rg) **Molecular Prandtl Number** (e, v) **Mean Molecular Weight** (seg, pdf) **Sound Speed** (id or rg) **Compressibility Factor** (rg) **Reduced Temperature** (ark) **Reduced Pressure** (ark) **Critical Temperature** (ark, spe) **Critical Pressure** (ark, spe)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Acentric Factor (ark,spe)
	Critical Specific Volume (ark,spe)
	Spinodal Temperature (ark)
	Electric Conductivity (pot)

Table 42.11: Eulerian Wall Film Category

Category	Variable
Eulerian Wall Film...	**Film Thickness** (3d, ewf)
	Film Mass (3d, ewf)
	Film Temperature (3d, ewf)
	Film X-Velocity (3d, ewf)
	Film Y-Velocity (3d, ewf)
	Film Z-Velocity (3d, ewf)
	Film Velocity Magnitude (3d, ewf)
	Film Coverage (3d, ewf)
	Film Effective Pressure (3d, ewf)
	Film Surface X-Velocity (3d, ewf)
	Film Surface Y-Velocity (3d, ewf)
	Film Surface Z-Velocity (3d, ewf)
	Film Surface Velocity Magnitude (3d, ewf)
	Film Surface Temperature (3d, ewf)
	Film Passive Scalar (3d, ewf)
	Film Courant Number (3d, ewf)
	Film Weber Number (3d, ewf)
	Film Stripped Mass (3d, ewf)
	Film Stripped Diam (3d, ewf)
	Film DPM Mass Source (3d, ewf, dpm)
	Film DPM Energy Source (3d, ewf, dpm)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Film DPM X-Momentum Source (3d, ewf, dpm)
	Film DPM Y-Momentum Source (3d, ewf, dpm)
	Film DPM Z-Momentum Source (3d, ewf, dpm)
	Film X-Momentum Source (3d, ewf)
	Film Y-Momentum Source (3d, ewf)
	Film Separated Mass (3d, ewf)
	Film Separated Diam (3d, ewf)
	Film Separation Rate (3d, ewf)
	Film Phase Change Rate (3d, ewf)
	Film Outflow Mass (3d, ewf)
	Film Secondary Phase Mass (3d, ewf)
	Film Secondary Phase Collection Coef (3d, ewf)
	Film VOF Equivalent Thickness (3d, ewf)
	Film Total Equivalent Thickness (3d, ewf)
	Film Volume Fraction (3d, ewf)
	Film Total Equivalent Volume Fraction (3d, ewf)
	Film Height Fraction (3d, ewf)
	Film Total Equivalent Height Fraction (3d, ewf)
	Film VOF Mass Exchange (3d, ewf)
	Film VOF Exchanged Mass (3d, ewf)

Table 42.12: Sensitivities Category

Category	Variable
Sensitivities...	**Magnitude of Sensitivity to Body Forces (Cell Values)** (adj)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Sensitivity to Body Force X-Component (Cell Values) (adj)
	Sensitivity to Body Force Y-Component (Cell Values) (adj)
	Sensitivity to Body Force Z-Component (Cell Values) (adj, 3d)
	Sensitivity to Mass Sources (Cell Values) (adj)
	Sensitivity to Energy Sources (Cell Values) (adj, aen)
	Sensitivity to Turbulent Kinetic Energy Sources (Cell Values) (adj, adt)
	Sensitivity to Specific Dissipation Rate Sources (Cell Values) (adj, adt)
	Artificial Dissipation (adj, adis)
	Sensitivity to Viscosity (adj)
	Shape Sensitivity Magnitude (adj)
	Normal Shape Sensitivity (adj)
	Normal Optimal Displacement (adj)
	Optimal Displacement X-Component (adj)
	Optimal Displacement Y-Component (adj)
	Optimal Displacement Z-Component (adj)
	log10(Shape Sensitivity Magnitude) (adj)
	Shape Sensitivity X Component (adj)
	Shape Sensitivity Y Component (adj)
	Shape Sensitivity Z Component (adj, 3d)
	Sensitivity to Boundary X-Velocity (adj)
	Sensitivity to Boundary Y-Velocity (adj)
	Sensitivity to Boundary Z-Velocity (adj, 3d)
	Sensitivity to Boundary Pressure (adj)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Sensitivity to Boundary Temperature (adj, aen)
	Sensitivity to Boundary Heat Flux (adj, aen)
	Sensitivity to Flow Blockage (adj)
	Sensitivity to GEKO CSEP (adj, adt)
	Sensitivity to GEKO CNW (adj, adt)
	Sensitivity to GEKO CMIX (adj, adt)
	Sensitivity to GEKO Blending Function (adj, adt)
	Sensitivity to CCURV (adj, adt)
	Adjoint GEKO CSEP (adj, adt)
	Adjoint GEKO CNW (adj, adt)
	Adjoint GEKO CMIX (adj, adt)
	Adjoint GEKO BF (adj, adt)
	Adjoint GEKO CSEP (Trained) (adj, adt, adnn, adjtr)
	Adjoint GEKO CNW (Trained) (adj, adt, adnn, adjtr)
	Adjoint GEKO CMIX (Trained) (adj, adt, adnn, adjtr)
	Adjoint GEKO BF (Trained) (adj, adt, adnn, adjtr)
	Non-Equilibrium Parameter (adj, adt, adnn)
	Second Invariant (adj, adt, adnn)
	Third Invariant (adj, adt, adnn)
	Fourth Invariant (adj, adt, adnn)
	Fifth Invariant (adj, adt, adnn)
	Length Ratio (adj, adt, adnn)
	Turbulent Reynolds Number (Scaled) (adj, adt, adnn)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Turbulent Viscosity Ratio (Scaled) (adj, adt, adnn) **Adjoint Local Solution Marker** (adj) **Adjoint X-Velocity Residual** (adj) **Adjoint Y-Velocity Residual** (adj) **Adjoint Z-Velocity Residual** (adj) **Adjoint Continuity Residual** (adj) **Adjoint Local Flow Rate Residual** (adj) **Adjoint Energy Residual** (adj, aen) **Adjoint Turbulent Kinetic Energy Residual** (adj, adt) **Adjoint Specific Dissipation Rate Residual** (adj, adt)

Table 42.13: Wall Fluxes, User Defined Scalars, and User Defined Memory Categories

Category	Variable
Wall Fluxes...	**Wall Shear Stress** (v, emm, fv) **X-Wall Shear Stress** (v, 2d, 3d, emm, fv) **Y-Wall Shear Stress** (v, 2d, 3d, emm, fv) **Z-Wall Shear Stress** (v, 3d, emm, fv) **Axial-Wall Shear Stress** (2da, 2dasw, fv) **Radial-Wall Shear Stress** (2da, 2dasw, fv) **Swirl-Wall Shear Stress** (2dasw, fv) **Skin Friction Coefficient** (v, emm, fv) **Total Surface Heat Flux** (e, v, fv) **Radiation Heat Flux** (fv, rad) **Solar Heat Flux** (sol, fv) **Absorbed Radiation Flux (Band-n)** (do, mc, fv) **Absorbed Visible Solar Flux** (sol, fv)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Absorbed IR Solar Flux (sol, fv) **Reflected Radiation Flux (Band-n)** (do, mc, fv) **Reflected Visible Solar Flux** (sol, fv) **Reflected IR Solar Flux** (sol, fv) **Transmitted Radiation Flux (Band-n)** (do, mc, fv) **Transmitted Visible Solar Flux** (sol, fv) **Transmitted IR Solar Flux** (sol, fv) **Beam Irradiation Flux (Band-n)** (do, mc, fv) **Surface Incident Radiation** (do, dtrm, s2s, mc, fv) **Surface Heat Transfer Coef.** (e, v, fv) **Wall Func. Heat Tran. Coef.** (e, v, fv) **Wall Adjacent Heat Transfer Coef.** (e, fv) **Surface Nusselt Number** (e, v, fv) **Surface Stanton Number** (e, v, fv) **Yplus Based Heat Transfer Coefficient** (e, v, fv) **SMB Single Phase Heat Flux** (mix) **SMB Nucleate Boiling Heat Flux** (mix) **Wall Deposition Thickness** (sr)
User Defined Scalars...	**Scalar-n** (bnv, uds) **Diffusion Coef. of Scalar-n** (bnv, uds)
User Defined Memory...	**User Memory <n>** (bnv, udm) **User Node Memory <n>** (bnv, udm)

Table 42.14: Cell Info and Mesh Categories

Category	Variable
Cell Info...	**Active Cell Partition** (cve) **Stored Cell Partition** (cve)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Cell Id (cve)
	Cell Element Type (cve)
	Cell Zone Type (cve)
	Cell Zone Index (cve)
	Overset Cell Type (cve, os)
	Overset Donor Count (os, cve)
	Overset Receptor Count (os, cve)
	Partition Neighbors (cve)
	Contact Cell Mark (con, cv, cve)
	Gap ID (gap)
	Gap Type (gap)
	Flow-Blocking Gap Interface (gap)
Mesh...	**X-Coordinate** (nv)
	Y-Coordinate (nv)
	Z-Coordinate (3d, nv)
	Axial Coordinate (nv)
	Angular Coordinate (3d, nv)
	Abs. Angular Coordinate (3d, nv)
	Radial Coordinate
	Face Area Magnitude
	X Face Area
	Y Face Area
	Z Face Area (3d)
	Orthogonal Quality
	Aspect Ratio (cve)
	Cell Equiangle Skew (cve)
	Cell Equivolume Skew (cve)
	Cell Volume (cve)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	2D Cell Volume (2da, cve)
	Interface Overlap Fraction
	Cell Wall Distance (cve)
	Face Handedness
	Accumulated Deformation (dpm, dm, fv)
	Gradient Quality Measure (cgq)
	Solution and Cell Quality Measure (scq)
	Mark Poor Elements
	Cell Volume Derivative (cve)
	Cell Volume Error (cve)
	Dynamic Cell Volume
	Dynamic Cell Volume Change (cve)
	Cell Volume Change
	Cell Surface Area (mcv)
	Cell Warpage (mcv)
	Boundary Layer Cells
	Boundary Cell Distance (cve)
	Boundary Normal Distance (cve)
	Boundary Volume Distance (cve)
	Cell Refine Level
	Overset Donor Size Ratio (os)
	X Periodic Displacement
	Y Periodic Displacement
	Z Periodic Displacement
	X Imaginary Periodic Displacement
	Y Imaginary Periodic Displacement
	Z Imaginary Periodic Displacement

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	X Imaginary Periodic Displacement
	Y Imaginary Periodic Displacement
	Z Imaginary Periodic Displacement
	X Relative Mesh Displacement
	Y Relative Mesh Displacement
	Z Relative Mesh Displacement
	X Total Mesh Displacement
	Y Total Mesh Displacement
	Z Total Mesh Displacement
	Sponge Layer Distance (spo)
	Sponge Blending Function (spo)

Table 42.15: Perforated Walls Category

Category	Variable
Perforated Walls...	**Tagged Face** (seg)
	Injection Hole Mass Flow Rate (seg)
	Injection Pressure (seg)
	Conjugate Heat Transfer (seg)

Table 42.16: Mesh Category (Turbomachinery-Specific Variables)

Category	Variable
Mesh...	**Meridional Coordinate** (nv, turbo)
	Abs Meridional Coordinate (nv, turbo)
	Spanwise Coordinate (nv, turbo)
	Abs (H-C) Spanwise Coordinate (nv, turbo)
	Abs (C-H) Spanwise Coordinate (nv, turbo)
	Pitchwise Coordinate (nv, turbo)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Abs Pitchwise Coordinate (nv, turbo)

Table 42.17: Residuals Category

Category	Variable
Residuals...	**Mass Imbalance** (seg)
	Pressure Residual (ret)
	X Velocity Residual (ret)
	Y Velocity Residual (ret)
	Z Velocity Residual (3d, ret)
	Swirl Velocity Residual (2dasw, ret)
	Temperature Residual (e, ret)
	***Species-n* Residual** (sp, ret)
	Time Step (cpl)

Table 42.18: Derivatives Category

Category	Variable
Derivatives...	**BCD Scheme Boundedness** (seg, bcde)
	Strain Rate (v)
	dX-Velocity/dx
	dY-Velocity/dx
	dZ-Velocity/dx (3d)
	dAxial-Velocity/dx (2da)
	dRadial-Velocity/dx (2da)
	dSwirl-Velocity/dx (2dasw)
	d species-n/dx (cpl, sp)
	dX-Velocity/dy
	dY-Velocity/dy
	dZ-Velocity/dy (3d)
	dAxial-Velocity/dy (2da)
	dRadial-Velocity/dy (2da)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	dSwirl-Velocity/dy (2dasw)
	d species-n/dy (cpl, sp)
	dX-Velocity/dz (3d)
	dY-Velocity/dz (3d)
	dZ-Velocity/dz (3d)
	d species-n/dz (cpl, sp, 3d)
	dOmega/dx (2dasw)
	dOmega/dy (2dasw)
	dp-dX (seg)
	dp-dY (seg)
	dp-dZ (seg, 3d)
	dp-dt (trn)
	Pressure Hessian Indicator

Table 42.19: Potential Category

Category	Variable
Potential...	**Electric Potential**
	Electric Current Magnitude
	Joule Heat Source
	X Current
	Y Current
	Z Current
	dPotential/dx
	dPotential/dy
	dPotential/dz
	Electrolyte Potential
	Over Potential
	Transfer Current
	Osmotic Drag

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	Water Phase Change
	Water Content
	Faradaic Current Density (echem)
	Total Surface Corrosion Rate (echem)
	Electrode Surface Potential
	Faradaic Heat Source (echem)
	Total Echem Heat Source (echem)

Table 42.20: Lithium Category

Category	Variable
Lithium...	**Lithium Concentration**
	Magnitude of Lithium Mass Flux Vector
	X Lithium Mass Flux
	Y Lithium Mass Flux
	Z Lithium Mass Flux

Table 42.21: Acoustics Category

Category	Variable
Acoustics...	**Surface dpdt RMS** (fv, fwh)
	Acoustic Power Level (dB) (bns)
	Acoustic Power (bns)
	Jet Acoustic Power Level (dB) (bns, 2da)
	Jet Acoustic Power (bns, 2da)
	Surface Acoustic Power Level (dB) (bns, fv)
	Surface Acoustic Power (bns, fv)
	Lilley's Self-Noise Source (bns)
	Lilley's Shear-Noise Source (bns)
	Lilley's Total Noise Source (bns)
	LEE Self-Noise X-Source (bns)
	LEE Shear-Noise X-Source (bns)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	LEE Total Noise X-Source (bns)
	LEE Self-Noise Y-Source (bns)
	LEE Shear-Noise Y-Source (bns)
	LEE Total Noise Y-Source (bns)
	LEE Self-Noise Z-Source (bns, 3d)
	LEE Shear-Noise Z-Source (bns, 3d)
	LEE Total Noise Z-Source (bns, 3d)
	Pressure Spectrum Im *n* (asfft, 3d)
	Pressure Spectrum Re *n* (asfft, 3d)
	PSD of dp/dt for Octave Band at *x* **Hz** (asfft, 3d)
	PSD of dp/dt for 1/3-Octave Band at *x* **Hz** (asfft, 3d)
	PSD of dp/dt for Const Width Band *n* (asfft, 3d)
	Sound Pressure (awe)
	Sound Potential (awe)
	Sound dP/dt (awe)
	Sound WaveEq Model Source (awe)
	Sound WaveEq Model Source Smoothed (awe)
	Sound WaveEq Model Source Mask (awe)
	Sound Sponge Layer Marker (awe)
	SPL for Octave Band at *x* **Hz (dB)** (asfft, 3d)
	SPL for 1/3-Octave Band at *x* **Hz (dB)** (asfft, 3d)
	SPL for Const Width Band *n* **(dB)** (asfft, 3d)
	static-pressure-tonal-analysis-1-magnitude

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Category	Variable
	static-pressure-tonal-analysis-1-phase

Table 42.22: Structure Category

Category	Variable
Structure...	**X Displacement** (nvs, strc) **Y Displacement** (nvs, strc) **Z Displacement** (nvs, strc, 3d) **Sigma XX** (strc) **Sigma YY** (strc) **Sigma XY** (strc) **Sigma ZZ** (strc, 3d) **Sigma YZ** (strc, 3d) **Sigma XZ** (strc, 3d)

Table 42.23: Aeromechanics Category

Category	Variable
Aeromechanics...	**Wall Work Density** **Wall Power Density**

42.4. Alphabetical Listing of Field Variables and Their Definitions

Below, the variables listed in Table 42.2:**Pressure** and **Density** Categories (p. 4175) – Table 42.21:**Acoustics** Category (p. 4203) are defined. For some variables (such as residuals) a general definition is given under the category name, and variables in the category are not listed individually. When appropriate, the unit quantity is included, as it appears in the **Quantities** list in the Set Units Dialog Box (p. 4693).

<dft-dataset-name>_<frequency>-ba-mag

(in the **Runtime DFT...** category) is the Fourier magnitude of a quantity in a frequency band, evaluated as a square root of the quantity power in a frequency band, see Runtime Discrete Fourier Transformation (p. 3671).

<dft-dataset-name>_<frequency>-ta-mag

(in the **Runtime DFT...** category) is the Fourier magnitude of a quantity for a frequency.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

<dft-dataset-name>_<frequency>-ta-phase

(in the **Runtime DFT...** category) is the Fourier phase of a quantity for a frequency. It is computed in radians in the range [-π, π].

Abs. Angular Coordinate

(in the **Mesh...** category) is the absolute value of the **Angular Coordinate** defined above.

Abs (C-H) Spanwise Coordinate

(in the **Mesh...** category) is the dimensional coordinate in the spanwise direction, from casing to hub. Its unit quantity is **length**.

Abs (H-C) Spanwise Coordinate

(in the **Mesh...** category) is the dimensional coordinate in the spanwise direction, from hub to casing. Its unit quantity is **length**.

Abs Meridional Coordinate

(in the **Mesh...** category) is the dimensional coordinate that follows the flow path from inlet to outlet. Its unit quantity is **length**.

Abs Pitchwise Coordinate

(in the **Mesh...** category) is the dimensional coordinate in the circumferential (pitchwise) direction. Its unit quantity is **angle**.

Absolute Pressure

(in the **Pressure...** category) is equal to the operating pressure plus the gauge pressure. See Operating Pressure (p. 1677) for details. Its unit quantity is **pressure**.

Absorbed Radiation Flux (Band-n)

(in the **Wall Fluxes...** category) is the amount of radiative heat flux absorbed by a semi-transparent wall for a particular band of radiation. Its unit quantity is **heat-flux**.

Absorbed Visible Solar Flux, Absorbed IR Solar Flux

(in the **Wall Fluxes...** category) is the amount of solar heat flux absorbed by a semi-transparent wall or porous jump boundary for a visible or infrared (IR) radiation.

Absorption Coefficient

(in the **Radiation...** category) is the property of a medium that describes the amount of absorption of thermal radiation per unit path length within the medium. It can be interpreted as the inverse of the mean free path that a photon will travel before being absorbed (if the absorption coefficient does not vary along the path). For multi-band cases, Fluent uses the emissivity weighting factor for each band to output a weighted absorption coefficient.

The unit quantity for **Absorption Coefficient** is **length-inverse**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Accumulated Deformation

(in the **Mesh...** category) is the accumulated displacement of the wall surface due to erosion over time. This variable will appear in the **Mesh...** category only for erosion dynamic mesh simulations. See Procedure for the Erosion Coupled with Dynamic Mesh Setup and Solution (p. 2794) for more information.

Accum Parcels in Cell

(in the **Steady | Unsteady DPM Statistics...** category)

- For steady particle tracking, this is the cumulative count of trajectories that has entered the cell since the last reset of the unsteady statistics.
- For unsteady particle tracking, this is the count of particle parcels that has entered the cell since the last reset of the unsteady statistics, where each parcel is weighted by the fraction of the respective fluid flow time step that it actually resides in the cell.

Accum Particles in Cell

(in the **Steady | Unsteady DPM Statistics...** category)

- For steady particle tracking, this is the cumulative sum of trajectories that has entered the cell since the last reset of the unsteady statistics. Each time a trajectory is counted, the time-constant number of particles represented by the trajectory in the cell is added to the sum.
- For unsteady particle tracking, this is the number of particles in all parcels that have entered the cell since the last reset of the unsteady statistics. Each parcel is weighted by the fraction of the respective fluid flow time step that it actually resides in the cell.

Acentric Factor

(in the **Properties...** category) is the mixture acentric factor. This property is available when a composition dependent option is selected for acentric factor in the cases with Aungier-Redlich-Kwong real gas model and species transport.

Acoustic Power

(in the **Acoustics...** category) is the acoustic power per unit volume generated by isotropic turbulence (see Equation 11.16 in the Theory Guide). It is available only when the **Broadband Noise Sources** acoustics model is being used. Its unit quantity is **power** per **volume**.

Acoustic Power Level (dB)

(in the **Acoustics...** category) is the acoustic power per unit volume generated by isotropic turbulence and reported in dB (see Equation 11.19 in the Theory Guide). It is available only when the **Broadband Noise Sources** acoustics model is being used.

Active Cell Partition

(in the **Cell Info...** category) is an integer identifier designating the partition to which a particular cell belongs. In problems in which the mesh is divided into multiple partitions to be solved on multiple processors using the parallel version of Ansys Fluent, the partition ID can be used to determine the extent of the various groups of cells. The active cell partition is used for the current

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

calculation, while the stored cell partition (the last partition performed) is used when you save a case file. See Partitioning the Mesh Manually and Balancing the Load (p. 4319) for more information.

Adiabatic Flame Temperature

(in the **Premixed Combustion...** category) is the adiabatic temperature of burnt products in a laminar premixed flame (T_b in Equation 8.100 in the Theory Guide). Its unit quantity is **temperature**.

Adjoint Continuity Residual

(in the **Sensitivities...** category), intended for users using the adjoint diagnosis, this is the residual of the adjoint continuity equation in each cell.

Adjoint Energy Residual

(in the **Sensitivities...** category), intended for users using the adjoint diagnosis, this field is available for users using the adjoint diagnosis and is the residual of the adjoint energy equation in each cell.

Adjoint GEKO BF

(in the **Sensitivities...** category), This field is available after performing gradient-based turbulence model optimization within the **Gradient-Based Optimizer** and is the optimized blending function (F_{Blend}), which is used to deactivate GEKO parameters inside boundary layers..

Adjoint GEKO BF (Trained)

(in the **Sensitivities...** category), This field is available after performing **Offline training** of the neural network turbulence model within the **Optimizer Design Variables** dialog box, and is the trained blending function parameter (F_{Blend}).

Adjoint GEKO CMIX

(in the **Sensitivities...** category), This field is available after performing gradient-based turbulence model optimization within the **Gradient-Based Optimizer** and is the optimized C_{MIX} parameter, which is used to optimize strength of mixing in free shear flows.

Adjoint GEKO CMIX (Trained)

(in the **Sensitivities...** category), This field is available after performing **Offline training** of the neural network turbulence model within the **Optimizer Design Variables** dialog box, and is the trained C_{MIX} parameter.

Adjoint GEKO CNW

(in the **Sensitivities...** category), This field is available after performing gradient-based turbulence model optimization within the **Gradient-Based Optimizer** and is the optimized C_{NW} parameter, which is used to optimize flow in non-equilibrium near wall regions (such as heat transfer or C_f).

Adjoint GEKO CNW (Trained)

(in the **Sensitivities...** category), This field is available after performing **Offline training** of the neural network turbulence model within the **Optimizer Design Variables** dialog box, and is the trained C_{NW} parameter.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Adjoint GEKO CSEP

(in the **Sensitivities...** category), This field is available after performing gradient-based turbulence model optimization within the **Gradient-Based Optimizer** and is the optimized C_{SEP} parameter, which is used to optimize flow separation from smooth surfaces.

Adjoint GEKO CSEP (Trained)

(in the **Sensitivities...** category), This field is available after performing **Offline training** of the neural network turbulence model within the **Optimizer Design Variables** dialog box, and is the trained C_{SEP} parameter.

Adjoint Local Flow Rate Residual

(in the **Sensitivities...** category), intended for users using the adjoint diagnosis, this is the residual of the adjoint local flow rate equation in each cell.

Adjoint Local Solution Marker

(in the **Sensitivities...** category), intended for expert users, can be plotted to identify those portions of the flow domain where the stabilized adjoint solution advancement scheme is applied. It is preferable to plot this with the **Node Values** disabled in the **Contours** dialog box. In this case, the **Adjoint Local Solution Marker** will take a value between 0 and 1. The `Mode Amplitude Cutoff` defined through the `adjoint/controls/stabilization` text command defines the lower bound for cells where the stabilized scheme is applied.

Adjoint Turbulent Kinetic Energy Residual

(in the **Sensitivities...** category), intended for users using the adjoint diagnosis, this field is available for users using the adjoint diagnosis and is the residual of the adjoint turbulent kinetic energy equation in each cell.

Adjoint Specific Dissipation Rate Residual

(in the **Sensitivities...** category), intended for users using the adjoint diagnosis, This field is available for users using the adjoint diagnosis and is the residual of the adjoint specific dissipation rate in each cell.

Adjoint X-Velocity Residual

(in the **Sensitivities...** category), intended for users using the adjoint diagnosis, this is the x-component of the residual of the adjoint momentum equation in each cell.

Adjoint Y-Velocity Residual

(in the **Sensitivities...** category), intended for users using the adjoint diagnosis, this is the y-component of the residual of the adjoint momentum equation in each cell.

Adjoint Z-Velocity Residual

(in the **Sensitivities...** category), intended for users using the adjoint diagnosis, this is the z-component of the residual of the adjoint momentum equation in each cell.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Angular Coordinate

(in the **Mesh...** category) is the angle between the radial vector and the position vector. The radial vector is obtained by transforming the default radial vector (y-axis) by the same rotation that was applied to the default axial vector (z-axis). This assumes that, after the transformation, the default axial vector (z-axis) becomes the reference axis. The angle is positive in the direction of cross-product between reference axis and radial vector.

Artificial Dissipation

(in the **Sensitivities...** category) is available when the adjoint dissipation stabilization scheme is enabled and shows the location and amount of the nonlinear damping that has been introduced to damp the growth of instabilities that lead to adjoint solution divergence.

Aspect Ratio

(in the **Mesh...** category) is a measure of the stretching of a cell. It is computed as the ratio of the maximum value to the minimum value of any of the following distances: the normal distances between the cell centroid and face centroids (computed as a dot product of the distance vector and the face normal), and the distances between the cell centroid and nodes.

Average Number of Particles in Soot Aggregate

(In the **Soot…** category) is an average number of primary particles in a soot aggregate (n_p in equation Equation 9.176 in the *Fluent Theory Guide*). This quantity is available only with the Method of Moments model.

Axial Coordinate

(in the **Mesh...** category) is the distance from the origin in the axial direction. The axis origin and (in 3D) direction is defined for each cell zone in the Fluid Dialog Box (p. 4961) or Solid Dialog Box (p. 4973). The axial direction for a 2D model is always the z direction, and the axial direction for a 2D axisymmetric model is always the x direction. The unit quantity for **Axial Coordinate** is **length**.

Axial Pull Velocity

(in the **Solidification/Melting...** category) is the axial-direction component of the pull velocity for the solid material in a continuous casting process. Its unit quantity is **velocity**.

Axial Velocity

(in the **Velocity...** category) is the component of velocity in the axial direction. (See Velocity Reporting Options (p. 4169) for details.) For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list. Its unit quantity is **velocity**.

Axial-Wall Shear Stress

(in the **Wall Fluxes...** category) is the axial component of the force per unit area acting tangential to the surface due to friction. Its unit quantity is that of **pressure**.

BCD Scheme Boundedness

(in the **Derivatives...** category) is the non-dimensional boundedness strength of the bounded central differencing scheme, see Bounded Central Differencing Scheme in the *Fluent Theory Guide*.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Beam Irradiation Flux (Band-b)

(in the **Wall Fluxes...** category) is specified as an incident heat flux (W / m^2) for each wavelength band.

Blending Function for GEKO

(in the **Turbulence...** category) is the blending function F_{Blend} for the GEKO model.

Boundary Cell Distance

(in the **Mesh...** category) is an integer that indicates the approximate number of cells from a boundary zone.

Boundary Layer Cells

(in the **Mesh...** category) identifies boundary layer cells, that is, the prismatic cells in layers that are adjacent to a boundary zone (not including periodic zones or shells). Prismatic cells are hexahedral cells or wedge or polyhedral cells that have the same number of nodes on the top and bottom faces relative to the direction that is normal to the boundary. Boundary layer cells have a value of 1, whereas all other cells have a value of 0. Only cell values are useful when displaying this identifier, so you should ensure that the **Node Values** option is disabled in the **Contours** dialog box.

Boundary Normal Distance

(in the **Mesh...** category) is the distance of the cell centroid from the closest boundary zone.

Boundary Volume Distance

(in the **Mesh...** category) is the cell volume distribution based on the **Boundary Volume**, **Growth Factor**, and normal distance from the selected **Boundary Zones** defined in the Boundary (p. 3854) cell register.

Bubble Area Fraction

(in the **Phase Interaction...** category) is the bubble area fraction of the boiling model (A_b in Equation 14.510 and Equation 14.511 in the *Fluent Theory Guide*). This item is available for the Eulerian multiphase boiling model only.

Bubble Departure Diameter

(in the **Phase Interaction...** category) is the bubble departure diameter of the boiling model (D_w in Equation 14.518 through Equation 14.520 in the *Fluent Theory Guide*). This item is available for the Eulerian multiphase boiling model only.

Bubble Departure Frequency

(in the **Phase Interaction...** category) is the bubble departure frequency of the boiling model (f in Equation 14.514 in the *Fluent Theory Guide*). This item is available for the Eulerian multiphase boiling model only.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Bubble Nucleation Site Density

(in the **Phase Interaction...** category) is the bubble nucleation site density of the boiling model (see Equation 14.515 in the *Fluent Theory Guide*). This item is available for the Eulerian multiphase boiling model only.

Bubble Waiting Time

(in the **Phase Interaction...** category) is the bubble waiting time of the boiling model defined as:

$$t_{wt} = C_{wt} / f$$

where C_{wt} is a coefficient to correct the waiting time between departures of consecutive bubbles, and f is the bubble departure frequency.

This item is available for the Eulerian multiphase boiling model only.

Built-in Correlation for GEKO Blending Function

(in the **Turbulence...** category) is the built-in correlation for the **Blending Function for GEKO**. It may be used for defining the effective **Blending Function for GEKO** as an expression referring to the value of the built-in correlation.

Capillary-Pressure

(in the **Pressure...** category) is the capillary pressure acting on a wetting (secondary) phase in a multiphase flow through a porous zone. Note that, although this item is available for both non-wetting and wetting phases, meaningful values are reported only for the wetting phase. See Specifying the Capillary Pressure (p. 1363) for details.

Cell Acoustic Courant Number

(in the **Velocity...** category) is a ratio of the time step size to the acoustic wave propagation time based on the cell size.

Cell Convective Courant Number

(in the **Velocity...** category) is a ratio of the time step size to the convective wave propagation time based on the cell size.

Cell Element Type

(in the **Cell Info...** category) is the integer cell element type identification number. Each cell can have one of the following element types:

```
triangle     1
tetrahedron  2
quadrilateral 3
hexahedron   4
pyramid    5
wedge      6
polyhedra  7
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cell Equiangle Skew

(in the **Mesh...** category) is a nondimensional parameter calculated using the normalized angle deviation method, and is defined as

$$\max\left[\frac{q_{max}-q_e}{180-q_e}, \frac{q_e-q_{min}}{q_e}\right] \tag{42.1}$$

where

q_{max} = largest angle in the face or cell

q_{min} = smallest angle in the face or cell

q_e = angle for an equiangular face or cell (for example, 60 for a triangle and 90 for a square)

A value of 0 indicates a best case equiangular cell, and a value of 1 indicates a completely degenerate cell. Degenerate cells (slivers) are characterized by nodes that are nearly coplanar (collinear in 2D). **Cell Equiangle Skew** applies to all elements.

Cell Equivolume Skew

(in the **Mesh...** category) is a nondimensional parameter calculated using the volume deviation method, and is defined as

$$\frac{(\text{optimal-cell-size})-(\text{cell-size})}{(\text{optimal-cell-size})} \tag{42.2}$$

where optimal-cell-size is the size of an equilateral cell with the same circumradius. A value of 0 indicates a best case equilateral cell and a value of 1 indicates a completely degenerate cell. Degenerate cells (slivers) are characterized by nodes that are nearly coplanar (collinear in 2D). **Cell Equivolume Skew** applies only to triangular and tetrahedral elements.

Cell Id

(in the **Cell Info...** category) is a unique integer identifier associated with each cell.

Cell Info...

includes quantities that identify the cell and its relationship to other cells.

Cell Refine Level

(in the **Mesh...** category) is an integer that indicates the number of times a cell has been subdivided by the adaption process, compared with the original mesh. For example, if one quad cell is split into four quads, the **Cell Refine Level** for each of the four new quads will be 1. If the resulting four quads are split again, the **Cell Refine Level** for each of the resulting 16 quads will be 2. *This value is only available at the cell center and is only exported at the cell center when exported to Common Fluids Format - Post.*

Cell Reynolds Number

(in the **Velocity...** category) is the value of the Reynolds number in a cell. (Reynolds number is a dimensionless parameter that is the ratio of inertia forces to viscous forces.) **Cell Reynolds Number** is defined as

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$Re \equiv \frac{\rho u d}{\mu} \tag{42.3}$$

where ρ is density, u is velocity magnitude, μ is the effective viscosity (laminar plus turbulent), and d is **Cell Volume** $^{1/2}$ for 2D cases and **Cell Volume** $^{1/3}$ in 3D or axisymmetric cases.

Cell Surface Area

(in the **Mesh...** category) is the total surface area of the cell, and is computed by summing the area of the faces that compose the cell. *This value is only available at the cell center and is only exported at the cell center when exported to Common Fluids Format - Post.*

Cell Volume

(in the **Mesh...** category) is the volume of a cell. In 2D the volume is the area of the cell multiplied by the unit depth. For axisymmetric cases, the cell volume is calculated using a reference depth of 1 radian. The unit quantity of **Cell Volume** is **volume**.

Cell Volume Derivative

(in the **Mesh...** category) is the change of a cell volume over time.

Cell Volume Error

(in the **Mesh...** category) is the cell volume over the unsteady cell volume.

2D Cell Volume

(in the **Mesh...** category) is the two-dimensional volume of a cell in an axisymmetric computation. For an axisymmetric computation, the 2D cell volume is scaled by the radius. Its unit quantity is **area**.

Cell Volume Change

(in the **Mesh...** category) is the maximum volume ratio of the current cell and its neighbors.

Cell Wall Distance

(in the **Mesh...** category) is the distribution of the normal distance of each cell centroid from the wall boundaries. Its unit quantity is **length**. By default, it is calculated using an exact geometric method. You have the option to specify that it is instead calculated using a Laplacian approach that estimates the value based on the transport equation; this can be useful if your case requires too much memory with the geometric wall distance method. Note that the transport equation method is not recommended for coarse meshes, since it may produce inaccurate wall distance results and subsequently decrease the accuracy of the solutions. To use the transport equation method, enter the following text command prior to running the calculation: `mesh/wall-distance-method transport-eqn`.

Cell Warpage

(in the **Mesh...** category) is the square root of the ratio of the distance between the cell centroid and cell circumcenter and the circumcenter radius:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\text{warpage}=\sqrt{\frac{\left|\vec{r}_{centroid}-\vec{r}_{circumcenter}\right|}{R_{circumcenter}}} \tag{42.4}$$

This value is only available at the cell center and is only exported at the cell center when exported to Common Fluids Format - Post.

Cell Zone Index

(in the **Cell Info...** category) is the integer cell zone identification number. In problems that have more than one cell zone, the cell zone ID can be used to identify the various groups of cells.

Cell Zone Type

(in the **Cell Info...** category) is the integer cell zone type ID. A fluid cell has a type ID of 1, a solid cell has a type ID of 17, and an exterior cell (parallel solver) has a type ID of 21.

Compressibility Factor

(in the **Properties...** category) is the ratio of the ideal gas density of the fluid divided by the real gas fluid density in the same flow conditions. Compressibility Factor is defined as

$$Z=\frac{P/RT}{\rho} \tag{42.5}$$

where Z is the compressibility factor, P is the absolute pressure, T is the temperature, and $R=R_u/MW$ (the universal gas constant R_u divided by the molecular weight MW). The compressibility factor is available only with the real gas models.

Compressible Wall Reynolds Number

(in the **Velocity...** category) is defined as

$$\text{Re}=\frac{\rho c \Delta h}{\mu} \tag{42.6}$$

where ρ is the density, c is the speed of sound, μ is the laminar viscosity, and Δh is the grid spacing in the normal direction of the wall. The **Compressible Wall Reynolds Number** is defined at the wall and applies only for laminar flows. To accurately predict the heat transfer on the wall, a **Compressible Wall Reynolds Number** less than one is recommended.

Conjugate Heat Transfer

(in the **Perforated Walls...** category) is the heat transfer between solid walls and fluid calculated by Equation 7.133 (p. 1486). Its unit quantity is **power**.

Contact Cell Mark

(in the **Cell Info...** category) is an integer identifier designating whether a cell is marked for flow blocking as part of a contact detection simulation that uses the contact marks method of flow control. Marked cells have a value of 1, whereas unmarked cells have a value of 0. This field variable is only available if you have enabled the following text command: `define/dynamic-mesh/controls/contact-parameters/render-contact-cells?`. Only cell values may be displayed for **Contact Cell Mark**, so you must ensure that the **Node Values** option is disabled in the **Contours** dialog box. See Contact Detection Settings (p. 1890) for more information.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Contact Resistivity

(in the **Solidification/Melting...** category) is the additional resistance at the wall due to contact resistance. It is equal to $R_c\left(1-\beta\right) / h$, where R_c is the contact resistance, β is the liquid fraction, and h is the cell height of the wall-adjacent cell. The unit quantity for **Contact Resistivity** is **thermal-resistivity**.

Critical Pressure

(in the **Properties...** category) is the mixture critical pressure. This property is available when a composition dependent option is selected for critical pressure in the cases with Aungier-Redlich-Kwong real gas model and species transport.

Critical Specific Volume

(in the **Properties...** category) is the mixture critical specific volume. This property is available when a composition dependent option is selected for critical specific volume in the cases with Aungier-Redlich-Kwong real gas model and species transport.

Critical Strain Rate

(in the **Premixed Combustion...** category) is a parameter that takes into account the stretching and extinction of premixed flames (g_{cr} in Equation 8.87 in the Theory Guide). Its unit quantity is **time-inverse**.

Critical Temperature

(in the **Properties...** category) is the mixture critical temperature. This property is available when a composition dependent option is selected for critical temperature in the cases with Aungier-Redlich-Kwong real gas model and species transport

Curvature Correction Function fr

(in the **Turbulence...** category) is the multiplier of the production term when curvature correction is selected for Spalart-Allmaras or two equation models. See Curvature Correction for the Spalart-Allmaras and Two-Equation Models in the *Fluent Theory Guide* for details.

Custom Field Functions...

are scalar field functions defined by you. You can create a custom function using the Custom Field Function Calculator Dialog Box (p. 5428). All defined custom field functions will be listed in the lower drop-down list. See Custom Field Functions (p. 4285) for details.

Damkohler Number

(in the **Premixed Combustion...** category) is a nondimensional parameter that is defined as the ratio of turbulent to chemical time scales.

Density...

includes variables related to density.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Density

(in the **Density...** category) is the mass per unit volume of the fluid. Plots or reports of **Density** include only fluid cell zones. For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list. The unit quantity for **Density** is **density**.

Density All

(in the **Density...** category) is the mass per unit volume of the fluid or solid material. Plots or reports of **Density All** include both fluid and solid cell zones. The unit quantity for **Density All** is **density**.

Derivatives...

are the viscous derivatives. For example, **dX-Velocity/dx** is the first derivative of the x component of velocity with respect to the x -coordinate direction. You can compute first derivatives of velocity, angular velocity, and pressure in the pressure-based solver, and first derivatives of velocity, angular velocity, temperature, and species in the density-based solvers.

DES Length Scale

(in the **Turbulence...** category) is available when you retain the temporary solver memory using the `solve/set/advanced/retain-temporary-solver-mem` text command. It is defined by Equation 4.272 in the Theory Guide for the Spalart-Allmaras based DES model and Equation 4.279 in the Theory Guide for the realizable k-epsilon based DES model. The DES length scale for the BSL / SST k-omega based DES model is defined by $C_{des}\Delta_{\max}=\left(\frac{L_t}{F_{DES}}\right)$, where C_{des} is a calibration constant used in the DES model and has a value of 0.61 and $\Delta_{\max}$ is the maximum local grid spacing $(\Delta x, \Delta y, \Delta z)$.

DES TKE Dissipation Multiplier

(in the **Turbulence...** category) is either the multiplier function F_{DES} for Detached Eddy Simulation (DES) with BSL, SST, or Transition SST (see DES with the BSL or SST k-ω Model and DES with the Transition SST Model in the Theory Guide for details) or the function f_d for DES with Spalart-Allmaras or Realizable k - ε (see Theory Guide DES with the Spalart-Allmaras Model or DES with the Realizable k-ε Model, respectively).

Diameter

(in the **Properties...** category) is the diameter of particles, droplets, or bubbles of the secondary phase selected in the **Phase** drop-down list. Its unit quantity is **length**.

Diffusion Coef. of Scalar-n

(in the **User Defined Scalars...** category) is the diffusion coefficient for the n^{th} user-defined scalar transport equation. See the Fluent Customization Manual for details about defining user-defined scalars.

Discrete Phase Sources...

includes quantities related to the discrete phase model sources. See Modeling Discrete Phase (p. 2671) for details about this model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Discrete Phase Variables...

includes non-source quantities related to the discrete phase model. See Modeling Discrete Phase (p. 2671) for details about this model.

Dissociation Vibration Source

(in the **Two-Temperature Model...** category)is the vibrational-electronic energy change due to chemical reactions (S^{cv} in Equation 5.28 in the *Fluent Theory Guide*).

DPM Absorption Coefficient

(in the **Discrete Phase Variables...** category) is the absorption coefficient for discrete-phase calculations that involve radiation (a in Equation 5.56 in the Theory Guide). Its unit quantity is **length-inverse**.

DPM Accretion

(in the **Discrete Phase Variables...** category) is the accretion rate calculated at a wall boundary:

$$R_{accretion} = \sum_{p=1}^{N} \frac{\dot{m}_p}{A_{face}} \tag{42.7}$$

where $\dot{m}_p$ is the mass flow rate of the particle stream, and A_{face} is the area of the wall face where the particle strikes the boundary. This item will appear only if the optional erosion/accretion model is enabled. See Monitoring Erosion/Accretion of Particles at Walls (p. 2697) for details. The unit quantity for **DPM Accretion** is **mass-flux**.

DPM Burnout

(in the **Discrete Phase Sources...** category) is the exchange of mass from the discrete to the continuous phase for the combustion law (Law 5) and is proportional to the solid phase reaction rate. The burnout exchange has units of **mass-flow** and is reported as the rate occurring in each cell. A unit cell depth is used for 2D cases, and a reference cell depth of 1 radian is used for 2D axisymmetric cases.

DPM Collision Rate

(in the **Discrete Phase Variables...** category) is the time rate of particle-to-particle collisions per unit volume. Its unit quantity is **collision-rate**.

DPM Concentration

(in the **Discrete Phase Variables...** category) is the total concentration of the particles in all phases in a cell. Its unit quantity is **density**. Note that Lagrangian wall film particles are not included when calculating the concentration.

DPM Conc of <*component*>

(in the **Discrete Phase Variables...** category) is the concentration of the chemical <*component*>, as part of DPM particles, in the cell (that is, the mass of that particle component in the cell, divided by the cell volume). Its unit quantity is **density**. When using DDPM, this is calculated separately for each DDPM phase, but always using the entire cell volume. (The name of the component will replace

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

<*component*> in **DPM Conc of <*component*>**, and for DDPM calculations, the field variable name will be appended with the phase name.)

DPM Density

(in the **Discrete Phase Variables...** category) is the mean particle density. This is computed as the mean density of the particles in each cell and is computed per phase. Therefore, when using DDPM this quantity is available for each discrete phase and the mixture phase. Its unit quantity is **density**.

DPM D20

(in the **Discrete Phase Variables...** category) is the surface mean diameter for the discrete phase particles within a cell. This is computed per phase. Therefore, when using DDPM this quantity is available for each discrete phase and the mixture phase. Its unit quantity is **length**. For the definition of the D20 diameter, refer to Summary Reporting of Current Particles (p. 2883).

DPM D30

(in the **Discrete Phase Variables...** category) is the volume mean diameter for the discrete phase particles within a cell. This is computed per phase. Therefore, when using DDPM this quantity is available for each discrete phase and the mixture phase. Its unit quantity is **length**. For the definition of the D30 diameter, refer to Summary Reporting of Current Particles (p. 2883).

DPM D32

(in the **Discrete Phase Variables...** category) is the Sauter mean diameter for the discrete phase particles within a cell. This is computed per phase. Therefore, when using DDPM this quantity is available for each discrete phase and the mixture phase. Its unit quantity is **length**. For the definition of the Sauter diameter, refer to Summary Reporting of Current Particles (p. 2883).

DPM D43

(in the **Discrete Phase Variables...** category) is the mean diameter over volume (also called De Brouckere mean diameter) for the discrete phase particles within a cell. This is computed per phase. Therefore, when using DDPM this quantity is available for each discrete phase and the mixture phase. Its unit quantity is **length**. For the definition of the De Brouckere diameter, refer to Summary Reporting of Current Particles (p. 2883).

DPM Diameter

(in the **Discrete Phase Variables...** category) is the mean particle diameter. This is computed as the mean diameter of the discrete phase particles in each cell and is computed per phase. Therefore, when using DDPM this quantity is available for each discrete phase and the mixture phase. Its unit quantity is **length**.

DPM Emission

(in the **Discrete Phase Variables...** category) is the amount of radiation emitted by a discrete-phase particle per unit volume. Its unit quantity is **heat-generation-rate**.

DPM Enthalpy

(in the **Discrete Phase Variables...** category) is the specific enthalpy. Depending on the particle type, the **DPM Enthalpy** is computed as:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Inert particles: Integral of the material specific heat over a temperature range from 298.15°K to the current particle temperature.
- Droplets: Same as for inert particles, corrected by the latent heat at the reference temperature and the heat of pyrolysis. The latent heat at the reference temperature is automatically calculated from the specified latent heat at the boiling point and the specified specific heats for liquid and vapor.
- Multicomponent: Sum of the mass-fraction-weighted enthalpies of the particle components corrected by the component latent heat at the reference temperature and the heat of pyrolysis.
- Combusting particles: Same as for droplets with a correction for the reaction heat.

Its unit quantity is specified in terms of enthalpy per unit mass.

DPM Enthalpy Source

(in the **Discrete Phase Sources...** category) is the exchange of enthalpy (sensible enthalpy plus heat of formation) from the discrete phase to the continuous phase. The exchange is positive when the particles are a source of heat in the continuous phase. The unit quantity for **DPM Enthalpy Source** is **power** and is reported as the rate of exchange occurring in each cell. A unit cell depth is used for 2D cases, and a reference cell depth of 1 radian is used for 2D axisymmetric cases.

DPM Erosion Rate (DNV)

(in the **Discrete Phase Variables...** category) is the DNV erosion rate calculated using the DNV formulation. (See DNV Erosion Model in the *Fluent Theory Guide* for background information about this model.) This item will appear only if the **DNV** option is enabled in the **Physical Models** tab of the Discrete Phase Model Dialog Box (p. 4845). See Monitoring Erosion/Accretion of Particles at Walls (p. 2697) for details.

DPM Erosion Rate (Generic)

(in the **Discrete Phase Variables...** category) is the erosion rate calculated at a wall boundary face using Equation 12.346 in the *Fluent Theory Guide* based on the values entered in the **Generic Erosion Model Parameters** dialog box (see Setting Particle Erosion and Accretion Parameters (p. 2787)). This item will appear only if the **Erosion/Accretion** option is selected in the **Physical Models** tab of the Discrete Phase Model Dialog Box (p. 4845) and the **Generic Model** option is enabled in the **DPM** tab of the Wall Dialog Box (p. 5085). See Monitoring Erosion/Accretion of Particles at Walls (p. 2697) for details. The unit quantity for **DPM Erosion Rate (Generic)** is **mass-flux**.

DPM Erosion Rate (Finnie)

(in the **Discrete Phase Variables...** category) is the Finnie erosion rate calculated using the Finnie formulation. (See Finnie Erosion Model in the *Fluent Theory Guide* for background information about this model.) This item will appear only if the **Finnie** option is enabled in the **Physical Models** tab of the Discrete Phase Model Dialog Box (p. 4845). See Monitoring Erosion/Accretion of Particles at Walls (p. 2697) for details. The unit quantity for **DPM Erosion Rate (Finnie)** is **mass-flux**.

DPM Erosion Rate (McLaury)

(in the **Discrete Phase Variables...** category) is the McLaury erosion rate calculated using the McLaury formulation. (See McLaury Erosion Model in the *Fluent Theory Guide* for background information about this model.) This item will appear only if the **McLaury** erosion model is enabled in the **Physical Models** tab of the Discrete Phase Model Dialog Box (p. 4845). See Monitoring Erosion/Accretion

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

of Particles at Walls (p. 2697) for details. The unit quantity for **DPM Erosion Rate (McLaury)** is **mass-flux**.

DPM Erosion Rate (Oka)

(in the **Discrete Phase Variables...** category) is the Oka erosion rate calculated using the Oka formulation. (See Oka Erosion Model in the *Fluent Theory Guide* for background information about this model.) This item will appear only if the **Oka** erosion model is enabled in the **Physical Models** tab of the Discrete Phase Model Dialog Box (p. 4845). See Monitoring Erosion/Accretion of Particles at Walls (p. 2697) for details. The unit quantity for **DPM Erosion Rate (Oka)** is **mass-flux**.

DPM Erosion Rate (Wall Shear)

(in the **Discrete Phase Variables...** category) is the abrasive erosion rate calculated using the shear stress formulation. See Modeling Erosion Rates in Dense Flows in the *Fluent Theory Guide* for background information about this model. See Monitoring Erosion/Accretion of Particles at Walls (p. 2697) for details. The unit quantity for **DPM Erosion Rate (Wall Shear)** is **mass-flux**. This quantity is available for dense flows for which DPM erosion is enabled and with the disperse granular secondary phase.

DPM Evaporation/Devolatilization

(in the **Discrete Phase Sources...** category) is the exchange of mass, due to droplet-particle evaporation or combusting-particle devolatilization, from the discrete phase to the evaporating or devolatilizing species. If you are not using the non-premixed combustion model, the mass source for each individual species (**DPM *species-n* Source** , below) is also available; for non-premixed combustion, only this sum is available. The unit quantity for **DPM Evaporation/Devolatilization** is **mass-flow** and is reported as the rate of exchange occurring in each cell. A unit cell depth is used for 2D cases, and a reference cell depth of 1 radian is used for 2D axisymmetric cases.

DPM Granular Temperature

(in the **Discrete Phase Variables...** category) is the mean Granular Temperature for the discrete phase particles within a cell. Its unit quantity is **turbulent kinetic energy**. This quantity is only available when using a granular secondary phase in the Dense Discrete Phase Model.

DPM Mass Source

(in the **Discrete Phase Sources...** category) is the total exchange of mass from the discrete phase to the continuous phase. The mass exchange is positive when the particles are a source of mass in the continuous phase. If you are not using the non-premixed combustion model, **DPM Mass Source** will be equal to the sum of all species mass sources (**DPM *species-n* Source** , below); if you are using the non-premixed combustion model, it will be equal to **DPM Burnout** plus **DPM Evaporation/Devolatilization**. The unit quantity for **DPM Mass Source** is **mass-flow** and is reported as the rate of exchange occurring in each cell. A unit cell depth is used for 2D cases, and a reference cell depth of 1 radian is used for 2D axisymmetric cases.

DPM Mixture Fraction Secondary Source

(in the **Discrete Phase Sources...** category) is the exchange of secondary mixture fraction from the discrete phase to the continuous phase. **DPM Mixture Fraction Secondary Source** is reported as the rate of exchange occurring in each cell. A unit cell depth is used for 2D cases, and a reference cell depth of 1 radian is used for 2D axisymmetric cases.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DPM Mixture Fraction Source

(in the **Discrete Phase Sources...** category) is the exchange of mixture fraction from the discrete phase to the continuous phase. **DPM Mixture Fraction Source** is reported as the rate of exchange occurring in each cell. A unit cell depth is used for 2D cases and a reference cell depth of 1 radian is used for 2D axisymmetric cases.

DPM Inert Source

(in the **Discrete Phase Variables...** category) is the exchange of the inert stream from the discrete phase to the continuous phase. **DPM Inert Source** is reported as the rate of exchange occurring in each cell. A unit cell depth is used for 2D cases, and a reference cell depth of 1 radian is used for 2D axisymmetric cases.

DPM Number Density

(in the **Discrete Phase Variables...** category) is the number of particles per unit cell volume. This is computed per phase. Therefore, when using DDPM this quantity is available for each discrete phase and the mixture phase. Its unit quantity is **volume-inverse**.

DPM Parcels in Cell

(in the **Discrete Phase Variables...** category)

- For steady particle tracking, this is the count of trajectories that entered the cell during the last DPM iteration.
- For unsteady particle tracking, this is the count of particle parcels that entered the cell during the last DPM iteration. Each parcel is weighted by the fraction of the fluid flow time step that it actually resides in the cell.

This is computed per phase. Therefore, when using DDPM this quantity is available for each discrete phase and the mixture phase.

DPM Particles in Cell

(in the **Discrete Phase Variables...** category)

- For steady particle tracking, this is the time-constant number of particles that is represented by all trajectories in the cell.
- For unsteady particle tracking, this is the number of particles in all parcels that entered the cell during the last DPM iteration, where each parcel is weighted by the fraction of the fluid flow time step that it actually resides in the cell.

This is computed per phase. Therefore, when using DDPM this quantity is available for each discrete phase and the mixture phase.

DPM RMS Diameter

(in the **Discrete Phase Variables...** category) is the RMS particle diameter. This is computed as the RMS particle diameter within each cell and is computed per phase. Therefore, when using DDPM, this quantity is available for each discrete phase and the mixture phase. Its unit quantity is length.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DPM RMS Temperature

(in the **Discrete Phase Variables...** category) is the RMS particle temperature. This is computed as the RMS particle temperature within each cell and is computed per phase. Therefore, when using DDPM this quantity is available for each discrete phase and the mixture phase. Its unit quantity is **temperature**.

DPM RMS X, Y, Z Velocity

(in the **Discrete Phase Variables...** category) are the RMS X, Y, and Z Velocity components of the discrete phase. These are computed as the mean discrete phase RMS velocities and are computed per phase. Therefore, when using DDPM these quantities are available for each discrete phase and the mixture phase. Its unit quantity is **velocity**.

DPM Scattering

(in the **Discrete Phase Variables...** category) is the scattering coefficient for discrete-phase calculations that involve radiation (σ_s in Equation 5.56 in the Theory Guide). Its unit quantity is **length-inverse**.

DPM Sensible Enthalpy Source

(in the **Discrete Phase Sources...** category) is the exchange of sensible enthalpy from the discrete phase to the continuous phase. The exchange is positive when the particles are a source of heat in the continuous phase. Its unit quantity is **power** and is reported as the rate of exchange occurring in each cell. A unit cell depth is used for 2D cases, and a reference cell depth of 1 radian is used for 2D axisymmetric cases.

DPM *species-n* Source

(in the **Discrete Phase Sources...** category) is the exchange of mass, due to droplet-particle evaporation or combusting-particle devolatilization, from the discrete phase to the evaporating or devolatilizing species. (The name of the species will replace ***species-n*** in **DPM *species-n* Source** .) These species can be specified in the Set Injection Properties Dialog Box (p. 5608), and their descriptions can be found in Defining Injection Properties (p. 2756). The unit quantity is **mass-flow** and is reported as the rate of exchange occurring in each cell. A unit cell depth is used for 2D cases, and a reference cell depth of 1 radian is used for 2D axisymmetric cases. Note that this variable will not be available if you are using the non-premixed combustion model; use **DPM Evaporation/Devolatilization** instead.

DPM Specific Heat

(in the **Discrete Phase Variables...** category) is the mean particle specific heat. This is computed as the mean specific heat of the particles within each cell and is computed per phase. Therefore, when using DDPM this quantity is available for each discrete phase and the mixture phase. Its unit quantity is **specific-heat**.

DPM Swirl Momentum Source

(in the **Discrete Phase Sources...** category) is the exchange of swirl momentum from the discrete phase to the continuous phase. This value is positive when the particles are a source of momentum in the continuous phase. The unit quantity is **force** and is reported as the rate of exchange occurring

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

in each cell. A unit cell depth is used for 2D cases, and a reference cell depth of 1 radian is used for 2D axisymmetric cases.

DPM Temperature

(in the **Discrete Phase Variables...** category) is the mean particle temperature. This is computed as the mean temperature of the particles within each cell and is computed per phase. Therefore, when using DDPM this quantity is available for each discrete phase and the mixture phase. Its unit quantity is **temperature**.

DPM Volume Fraction

(in the **Discrete Phase Variables...** category) is the volume fraction of the discrete phase. This is computed as the mean discrete phase volume fraction within each cell and is computed per phase. Therefore, when using DDPM this quantity is available for each discrete phase and the mixture phase.

DPM Wall Normal Pressure

(in the **Discrete Phase Variables...** category) is the magnitude of the wall force vector per unit area.

DPM Wall X, Y, Z Force

(in the **Discrete Phase Variables...** category) are the X, Y, and Z components of the DPM particle-wall force. These are computed on all wall boundary conditions. For steady-state simulations, the reported forces are the mean forces exerted by the particles per unit time. For transient simulations, the reported forces are the time-averaged particle-wall forces over the fluid time step.

DPM X, Y, Z Momentum Source

(in the **Discrete Phase Sources...** category) are the exchange of x -, y -, and z -direction momentum from the discrete phase to the continuous phase. These values are positive when the particles are a source of momentum in the continuous phase. The unit quantity is **force** and is reported as the rate of exchange occurring in each cell. A unit cell depth is used for 2D cases, and a reference cell depth of 1 radian is used for 2D axisymmetric cases.

DPM X, Y, Z Velocity

(in the **Discrete Phase Variables...** category) are the X, Y, and Z Velocity components of the discrete phase. These are computed as the mean discrete phase velocities and are computed per phase. Therefore, when using DDPM these quantities are available for each discrete phase and the mixture phase. Its unit quantity is **velocity**.

dPotential/dx, dPotential/dy, dPotential/dz

(in the **Potential...** category) are the x, y, and z components of the potential gradient field ($\frac{\partial \phi}{\partial x}$, $\frac{\partial \phi}{\partial y}$, and $\frac{\partial \phi}{\partial z}$), respectively.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DRG Reduced Number of Reactions

(in the **Reactions...** category) is the number of retained reactions in reduced mechanism. This variable quantifies the size of the reduced mechanism at each cell or particle in the domain.

DRG Reduced Number of Species

(in the **Species...** category) is the number of retained species in reduced mechanism. This variable quantifies the size of the reduced mechanism at each cell or particle in the domain.

Dynamic Cell Volume Change

(in the **Mesh...** category) is the change of a cell volume.

Dynamic Pressure

(in the **Pressure...** category) is defined as $q \equiv \frac{1}{2}\rho v^2$. Its unit quantity is **pressure**.

Echem Reaction Rate of Reaction-n

(in the **Reactions...** category) is the n^{th} electrochemical reaction rate. This quantity is only available if the electrochemical reaction model is enabled.

Eff Diff Coef of *species-n*

(in the **Species...** category) is the sum of the laminar and turbulent diffusion coefficients of a species into the mixture:

$$D_{i,m} + \frac{\mu_t}{\rho Sc_t} \tag{42.8}$$

(The name of the species will replace ***species-n*** in **Eff Diff Coef of *species-n*** .) The unit quantity is **mass-diffusivity**.

Effective Prandtl Number

(in the **Turbulence...** category) is the ratio $\mu_{eff} c_p / k_{eff}$, where μ_{eff} is the effective viscosity, c_p is the specific heat, and k_{eff} is the effective thermal conductivity.

Effective Thermal Conductivity

(in the **Properties...** category) is the sum of the laminar and turbulent thermal conductivities, $k + k_t$, of the fluid. A large thermal conductivity is associated with a good heat conductor and a small thermal conductivity with a poor heat conductor (good insulator). Its unit quantity is **thermal-conductivity**.

Effective Viscosity

(in the **Turbulence...** category) is the sum of the laminar and turbulent viscosities of the fluid. Viscosity, μ, is defined by the ratio of shear stress to the rate of shear. Its unit quantity is **viscosity**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Electric Potential

(in the **Potential...** category) is the electric potential φ used in Equation 7.130 in the *Fluent Theory Guide*.

Electric Conductivity

(in the **Properties...** category) is the electric conductivity σ used in Equation 7.118 in the *Fluent Theory Guide*.

Electric Current Magnitude

(in the **Potential...** category) is the magnitude of the electric current vector in the field, $\left|\sigma \nabla \varphi\right|$.

Electrode Surface Potential

(in the **Potential...** category) is either the electrode potential (φ_{ed} in the Butler-Volmer Equation 7.120 in the *Fluent Theory Guide*) for walls with Faradaic reaction, or the wall potential after considering the contact resistance effect for walls without Faradaic reaction.

Electrolyte Potential

(in the **Potential...** category) is the electrolyte potential ϕ_{mem} used in Equation 20.4 in the *Fluent Theory Guide*.

Enthalpy

(in the **Temperature...** category) is defined differently for compressible and incompressible flows, and depending on the solver and models in use.

For compressible flows,

$$H = \sum_j Y_j H_j \tag{42.9}$$

and for incompressible flows,

$$H = \sum_j Y_j H_j + \frac{p}{\rho} \tag{42.10}$$

where Y_j and H_j are, respectively, the mass fraction and enthalpy of species j. (See **Enthalpy of *species-n*** , below). For the pressure-based solver, the second term on the right-hand side of Equation 42.10 (p. 4226) is included only if the pressure work term is included in the energy equation (see Inclusion of Pressure Work and Kinetic Energy Terms in the Theory Guide). For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list. For all reacting flow models, the **Enthalpy** plots consist of the thermal (or sensible) plus chemical energy. The unit quantity for **Enthalpy** is **specific-energy**.

In the case of the inert model (Using the Non-Premixed Model with the Inert Model in the Theory Guide), the enthalpy in a cell is split into the contributions from the inert and the reacting fractions of the gas phase species in the cell. The cell enthalpy is partitioned as

$$H = \gamma H_{inert} + \left(1 - \gamma\right) H_{pdf} \tag{42.11}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where γ is the fraction of inert species in the cell. The quantity H_{inert} is the enthalpy of the inert species at the cell temperature, similarly H_{pdf} is the enthalpy of the active species at the cell temperature. It is assumed that the cell temperature is common to both inert and active species, so H_{inert}, H_{pdf} and the cell temperature are chosen so that Equation 42.11 (p. 4226) is satisfied.

Enthalpy of *species-n*

(in the **Species...** category) is defined differently depending on the solver and models options in use. The quantity:

$$H_j = \int_{T_{ref,j}}^{T} c_{p,j} dT + h_j^0\left(T_{ref,j}\right) \tag{42.12}$$

where $h_j^0\left(T_{ref,j}\right)$ is the formation enthalpy of species j at the reference temperature $\left(T_{ref,j}\right)$, is reported only for non-adiabatic PDF cases, or if the density-based solver is selected. The quantity:

$$h_j = \int_{T_{ref}}^{T} c_{p,j} dT \tag{42.13}$$

where $T_{ref} = 298.15K$, is reported in all other cases. The unit quantity for **Enthalpy of *species-n*** is **specific-energy**.

Entropy

(in the **Temperature...** category) is a thermodynamic property defined by the equation

$$\Delta S \equiv \int_{rev} \frac{\delta Q}{T} \tag{42.14}$$

$$\Delta S = C_p \ln\left(\frac{T}{T_{ref}}\right) - R \ln\left(\frac{P}{P_{ref}}\right) \tag{42.15}$$

the entropy is computed using the equation

$$\Delta S = C_p \ln\left(\frac{T}{T_{ref}}\right) \tag{42.16}$$

The unit quantity for entropy is **specific-heat**.

Important:

Note that for the real gas models the entropy is computed accordingly by the appropriate equation of state formulation.

Face Area Magnitude

(in the **Mesh...** category) is the magnitude of the face area vector for noninternal faces (that is, faces that only have `c0` and no `c1`). The values are stored on the face itself and used when required. This variable is intended only for zone surfaces and not for other surfaces created for postprocessing.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Face Handedness

(in the **Mesh...** category) is a parameter that is equal to one in cells that are adjacent to left-handed faces, and zero elsewhere. It can be used to locate mesh problems.

Faradaic Current Density

(in the **Potential...** category) is the electric current density produced by the electrochemical reaction at Faradaic interfaces (Equation 7.127 in the *Fluent Theory Guide*). Faradaic current density is positive if electric current flows from the electrode zone to the electrolyte zone, and negative if the electric current flows in the reverse direction.

Faradaic Heat Source

(in the **Potential...** category) is the energy source term due to the electrochemical reaction at Faradaic interfaces (Equation 7.132 in the *Fluent Theory Guide*).

Fifth Invariant

(in the **Sensitivities...** category) This field is available when the **Neural Network Model** has been applied for turbulence model optimization within the **Optimizer Design Variables** dialog box, and is the Fifth Invariant flow feature.

Film Courant Number

(in the **Eulerian Wall Film...** category) is the Courant number of the wall film.

Film Coverage

(in the **Eulerian Wall Film...** category) is the value of film wet area fraction as defined in Equation 17.30 in the *Fluent Theory Guide*. In reporting, the face integral of this variable gives the area covered by the film. This item is available when **Solve Momentum** is enabled in the **Eulerian Wall Film** dialog box.

Film DPM Energy Source

(in the **Eulerian Wall Film...** category) is the additional energy of discrete particles being absorbed into the wall film.

Film DPM Mass Source

(in the **Eulerian Wall Film...** category) is the additional mass of discrete particles being absorbed into the wall film.

Film DPM X-Momentum Source

(in the **Eulerian Wall Film...** category) is the x-component of any additional momentum of discrete particles being absorbed into the wall film.

Film DPM Y-Momentum Source

(in the **Eulerian Wall Film...** category) is the y-component of any additional momentum of discrete particles being absorbed into the wall film.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Film DPM Z-Momentum Source

(in the **Eulerian Wall Film...** category) is the z-component of any additional momentum of discrete particles being absorbed into the wall film.

Film Effective Pressure

(in the **Eulerian Wall Film...** category) is the effective pressure of the wall film.

Film Height Fraction

(in the **Eulerian Wall Film...** category) is the ratio of the film half-thickness to the cell center height next to the film wall.

Film Mass

(in the **Eulerian Wall Film...** category) is the mass of the wall film.

Film Mass Fraction of *material-i*

(in the **Wall Film...** category) is the mass fraction of the discrete phase material i in the wall film. (The name of the material will replace ***material-i*** in **Film Mass Fraction of *material-i*** .)

Film Outflow Mass

(in the **Eulerian Wall Film...** category) is the cumulative mass of film outflow.

Film Passive Scalar

(in the **Eulerian Wall Film...** category) is the passive scalar for the wall film scalar equation.

Film Phase Change Rate

(in the **Eulerian Wall Film...** category) is the phase change rate, with positive value as the condensation rate and negative value as the vaporization rate. This item is available when **Phase Change** is enabled.

Film Secondary Phase Collection Coef

(in the **Eulerian Wall Film...** category) is the coefficient of secondary phase collection. This item is available when **Phase Accretion** is enabled.

Film Secondary Phase Mass

(in the **Eulerian Wall Film...** category) is the mass of secondary phase collection. This item is available when **Phase Accretion** is enabled.

Film Separated Diam

(in the **Eulerian Wall Film...** category) is the diameter of separated wall film droplet. This item is available when **Edge Separation** is enabled.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Film Separation Rate

(in the **Eulerian Wall Film...** category) is the mass of secondary phase collection. This item is available when **Edge Separation** is enabled.

Film Stripped Diam

(in the **Eulerian Wall Film...** category) is the diameter of stripped wall film droplets.

Film Stripped Mass

(in the **Eulerian Wall Film...** category) is the cumulative mass of stripped wall film.

Film Surface Temperature

(in the **Eulerian Wall Film...** category) is the surface temperature of the wall film.

Film Surface Velocity Magnitude

(in the **Eulerian Wall Film...** category) is the magnitude of the surface velocity of the wall film.

Film Surface X-Velocity

(in the **Eulerian Wall Film...** category) is the x-component of the surface velocity of the wall film.

Film Surface Y-Velocity

(in the **Eulerian Wall Film...** category) is the y-component of the surface velocity of the wall film.

Film Surface Z-Velocity

(in the **Eulerian Wall Film...** category) is the z-component of the surface velocity of the wall film.

Film Temperature

(in the **Eulerian Wall Film...** category) is the temperature of the wall film.

Film Thickness

(in the **Eulerian Wall Film...** category) is the thickness of the wall film.

Film Total Equivalent Height Fraction

(in the **Eulerian Wall Film...** category) is the sum of the film height fraction and the height fraction computed from the VOF equivalent film thickness.

Film Total Equivalent Thickness

(in the **Eulerian Wall Film...** category) is the sum of film VOF equivalent thickness and the film thickness.

Film Total Equivalent Volume Fraction

(in the **Eulerian Wall Film...** category) is the sum of the film volume fraction and the liquid phase volume fraction in the VOF solution.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Film Velocity Magnitude

(in the **Eulerian Wall Film...** category) is the magnitude of the velocity of the wall film.

Film VOF Equivalent Thickness

(in the **Eulerian Wall Film...** category) is the equivalent film thickness computed from the liquid phase volume fraction from the VOF solution in the cell adjacent to the film wall.

Film VOF Exchanged Mass

(in the **Eulerian Wall Film...** category) is the total exchanged mass rate between the EWF and VOF multiphase models. A positive value indicates net mass transfer from the VOF to the film solution, and a negative value indicates net mass transfer from the film to the VOF solution.

Film VOF Mass Exchange

(in the **Eulerian Wall Film...** category) is the mass exchange between the EWF and VOF multiphase models during the last flow time step. A positive value indicates mass transfer from the VOF to film solution, and a negative value indicates mass transfer from the film to the VOF solution.

Film Volume Fraction

(in the **Eulerian Wall Film...** category) is the ratio of the film liquid volume to the cell volume next to the film wall.

Film Weber Number

(in the **Eulerian Wall Film...** category) is the Weber number of the wall film.

Film X-Momentum Source

(in the **Eulerian Wall Film...** category) is the x-component of any additional momentum being absorbed into the wall film.

Film Y-Momentum Source

(in the **Eulerian Wall Film...** category) is the y-component of any additional momentum being absorbed into the wall film.

Film X-Velocity

(in the **Eulerian Wall Film...** category) is the x-component of the velocity of the wall film.

Film Y-Velocity

(in the **Eulerian Wall Film...** category) is the y-component of the velocity of the wall film.

Film Z-Velocity

(in the **Eulerian Wall Film...** category) is the z-component of the velocity of the wall film.

Fine Scale Mass Fraction of *`species-n`*

(in the **Species...** category) is the term Y_i^* in Equation 7.42 in the Theory Guide.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fine Scale Temperature

(in the **Temperature...** category) is the temperature of the fine scales, which is calculated from the enthalpy when the reaction proceeds over the time scale (τ^* in Equation 7.45 in the Theory Guide), governed by the Kinetic rates of Equation 7.21 in the Theory Guide. Its unit quantity is **temperature**.

Fine Scale Transfer Rate

(in the **Species...** category) is the transfer rate of the fine scales, which is equal to the inverse of the time scale (τ^* in Equation 7.45 in the Theory Guide). Its unit quantity is **time-inverse**.

1-Fine Scale Volume Fraction

(in the **Species...** category) is a function of the fine scale volume fraction (ξ^* in Equation 7.44 in the Theory Guide). The quantity is subtracted from unity to make it easier to interpret.

Film Separated Mass

(in the **Eulerian Wall Film...** category) is the cumulative mass of separated wall film.

Flow-Blocking Gap Interface

(in the **Cell Info...** category) identifies the interfaces of flow-blocking gap regions. All cells that have at least one face that applies the zero-mass-flux boundary for the gap region have a value of 1, whereas all other cells have a value of 0. For further details, see Controlling Flow in Narrow Gaps for Valves and Pumps (p. 1234).

Flow Regime Blending Factors Fcc, Flow Regime Blending Factors Fcd, Flow Regime Blending Factors Fdc, Flow Regime Blending Factors Fdd

(in the **Phase Interaction...** category) are the blending factors for the continuous-continuous, continuous-dispersed, dispersed-continuous, and dispersed-dispersed phase interactions, respectively, which are used for the flow regime detection. The blending factors must satisfy the following restriction:

$$F_{CC} + F_{CD} + F_{DC} + F_{DD} = 1 \tag{42.17}$$

This item is available only for mixture multiphase cases with the flow regime modeling. For further details, see Flow Regime Modeling in the *Fluent Theory Guide*.

Forward Reaction Rate of *PDF scalar-n*

(in the **Premixed Combustion...** category) is the rate of production of `species-n` due to forward reactions for scalar φ in kg/m ^{3} /s ($\overline{S}_{\varphi,fwd}$ in Equation 8.120 in the *Fluent Theory Guide*). (The name of the PDF scalar will replace *PDF* `scalar-n` in **Forward Reaction Rate of *PDF scalar-n*** .)

Fourth Invariant

This field is available when the **Neural Network Model** has been applied for turbulence model optimization within the **Optimizer Design Variables** dialog box, and is the Fourth Invariant flow feature.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Frozen Sound Speed

(in the **Two-Temperature Model...** category) is defined as $\sqrt{\gamma_{tr} R T_{tr}}$ where γ_{tr} is the translational-rotational heat capacity ratio.

Fvar Prod

(in the **Pdf...** category) is the production term in the mixture fraction variance equation solved in the non-premixed combustion model (that is, the last two terms in Equation 8.5 in the Theory Guide).

Fvar2 Prod

(in the **Pdf...** category) is the production term in the secondary mixture fraction variance equation solved in the non-premixed combustion model. See Equation 8.5 in the Theory Guide.

Gap ID

(in the **Cell Info...** category) is the gap region ID, as identified using the **List** button in the Gap Model Dialog Box (p. 5484). All cells that are not marked for a gap region have a value of 0. For further details, see Controlling Flow in Narrow Gaps for Valves and Pumps (p. 1234).

Gap Type

(in the **Cell Info...** category) is the type of gap region. Cells that are marked as part of a flow-blocking gap region have a value of 1, cells that are marked as part of a flow-modeling gap region have a value of 2, and cells that are not marked for a gap region have a value of 0. For further details, see Controlling Flow in Narrow Gaps for Valves and Pumps (p. 1234).

Gas Constant (R)

(in the **Properties...** category) is the gas constant of the fluid. Its unit quantity is **specific-heat**.

Geometric Roughness Height

(in the **Turbulence...** category) is used in the roughness correlation of the Transition SST model. See Equation 4.189 in the Theory Guide.

Gradient Quality Measure

(in the **Mesh...** category) is a measure used to apply poor mesh numerics to cells when the cell gradient quality criterion is enabled; poor cells have a value closer to 0, whereas good cells have a value closer to 1.

Granular Conductivity

(in the **Properties...** category) is equivalent to the diffusion coefficient in Equation 14.374 in the Theory Guide. For more information, see Granular Temperature in the Theory Guide. Its unit quantity is kg/m-s.

Granular Pressure...

includes quantities for reporting the solids pressure for each granular phase (p_s in Equation 14.345 in the Theory Guide). See Solids Pressure in the Theory Guide for details. Its unit quantity is **pressure**. For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Granular Temperature...

includes quantities for reporting the granular temperature for each granular phase (Θ_s in Equation 14.374 in the Theory Guide). See Granular Temperature in the Theory Guide for details. Its unit quantity is m^2/s^2. For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list.

hcn Density

(in the **NOx...** category) is the mass per unit volume of HCN. The unit quantity is **density**. The **hcn Density** will appear only if you are modeling fuel NOx. See Fuel NOx Formation in the Theory Guide for details.

Heat of Heterogeneous Reaction

(in the **Phase Interaction...** category) is the heat added or removed due to heterogeneous chemical reactions. For exothermic reactions the **Heat of Heterogeneous Reaction** is reported as a positive quantity, while for endothermic reactions it will be a negative quantity. If you have more than one heterogeneous reaction defined in your case, the **Heat of Heterogeneous Reaction** reported is the sum of the heat for all heterogeneous reactions. The unit quantity of **Heat of Heterogeneous Reaction** is Watt.

Heat of Reaction

(in the **Reactions...** category) is the heat added or removed due to chemical reactions. The quantity reported by Fluent is the volumetric heat of reaction, as defined in Equation 5.11 in the Theory Guide multiplied by the cell volume. For exothermic reactions, the heat of reaction is reported as a positive quantity, while for endothermic reactions it is reported as a negative quantity. If you have more than one reaction defined in your case, the **Heat of Reaction** reported is the sum of the heat for all reactions. The unit of measurement for the heat of reaction is Watts. The **Heat of Reaction** is not available for the non-premixed and partially-premixed models.

Heat Release Rate

(in the **Pdf...** category) is the rate of heat generated due to chemical reaction (J/m3/s) as defined in Equation 5.11 in the *Fluent Theory Guide*.

Helicity

(in the **Velocity...** category) is defined by the dot product of vorticity and the velocity vector.

$$H=\left(\nabla\times\vec{V}\right)\cdot\vec{V} \tag{42.18}$$

Vorticity is a measure of the rotation of a fluid element as it moves in the flow field.

Incident Radiation

(in the **Radiation...** category) is the total radiation energy, G, that arrives at a location per unit time and per unit area:

$$G=\int_{\Omega=4\pi} I d\Omega \tag{42.19}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where I is the radiation intensity and Ω is the solid angle. G is the quantity that the P-1 radiation model computes. For the DO radiation model, the incident radiation is computed over a finite number of discrete solid angles, each associated with a vector direction. For the MC model, the radiation intensity is computed by tallying the distance traveled by photons in each cell. The unit quantity for **Incident Radiation** is **heat-flux**.

Incident Radiation (Band n)

(in the **Radiation...** category) is the radiation energy contained in the wavelength band $\Delta\lambda$ for the non-gray P-1, DO, or MC radiation models. Its unit quantity is **heat-flux**.

Injection Hole Mass Flow Rate

(in the **Perforated Walls...** category) is the mass flow rate calculated by Equation 7.131 (p. 1485) (incompressible flow) or Equation 7.132 (p. 1485) (compressible flow). Note that the mass flow rate will be the same for the faces that belong to the same effusion surface (uniform type) or perforated hole (discrete type).

Injection Pressure

(in the **Perforated Walls...** category) is the averaged pressure used in the mass flow rate calculation in Equation 7.131 (p. 1485) (incompressible flow) or Equation 7.132 (p. 1485) (compressible flow). Note that the injection pressure will be the same for the faces that belong to the same effusion surface (uniform type) or perforated hole (discrete type).

Interface Overlap Fraction

(in the **Mesh...** category) is a metric for measuring the quality of the intersection between interface zones for non-conformal interfaces. It is calculated on each element of the interface zone and ranges from 0 to 1, where 0 means no intersection, and 1 means full intersection.

Intermittency

(in the **Turbulence...** category) is a measure of the probability that a given point is located inside a turbulent region. Upstream of transition the intermittency is zero. Once the transition occurs, the intermittency is ramped up to one until the fully turbulent boundary layer regime is achieved.

Intermittency Effective

(in the **Turbulence...** category) is used in the coupling of the Transition model and the SST Transport equations (see Coupling the Transition Model and SST Transport Equations in the Theory Guide for details).

Internal Energy

(in the **Temperature...** or **Two-Temperature Model...** category) is the summation of the kinetic and potential energies of the molecules of the substance per unit volume (and excludes chemical and nuclear energies). **Internal Energy** is defined as $e=c_vT$. Its unit quantity is **specific-energy**.

Jet Acoustic Power

(in the **Acoustics...** category) is the acoustic power for turbulent axisymmetric jets (see Equation 11.20 in the Theory Guide). It is available only when the **Broadband Noise Sources** acoustics model is being used.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Jet Acoustic Power Level (dB)

(in the **Acoustics...** category) is the acoustic power for turbulent axisymmetric jets, reported in dB (see Equation 11.33 in the Theory Guide). It is available only when the **Broadband Noise Sources** acoustics model is being used.

Joule Heat Source

(in the **Potential...** category) is the energy source term due to the Joule heating (Equation 18.2 in the *Fluent Theory Guide*).

Kinetic Rate of Reaction-n

(in the **Reactions...** category) is given by the following expression (see Equation 7.21 in the Theory Guide for definitions of the variables shown here):

$$\hat{R}_r = \Gamma \left(k_{f,r} \prod_{j=1}^{N_r} [C_{j,r}]^{\eta'_{j,r}} - k_{b,r} \prod_{j=1}^{N_r} [C_{j,r}]^{\eta''_{j,r}} \right) \tag{42.20}$$

The reported value is independent of any particular species, and has units of kmol/ m^3 -s.

To find the rate of production/destruction for a given species i due to reaction r, multiply the reported reaction rate for reaction r by the term $M_i\left(\nu''_{i,r} - \nu'_{i,r}\right)$, where M_i is the molecular weight of species i, and $\nu''_{i,r}$ and $\nu'_{i,r}$ are the stoichiometric coefficients of species i in reaction r.

For particle reactions it is the global rate of the particle reaction n expressed in kmol/s/m3. This is computed as $\overline{R}_{j,r} / M_j V$, where $\overline{R}_{j,r}$ is the rate of particle species depletion (or generation) given by Equation 7.102 in the Theory Guide, M_j is the particle species molecular weight, and V is the cell volume.

Lambda 2 Criterion

(in the **Velocity...** category) is a postprocessing quantity used for visual inspection of turbulence structures.

Lam Diff Coef of *species-n*

(in the **Species...** category) is the laminar diffusion coefficient of a species into the mixture, $D_{i,m}$. Its unit quantity is **mass-diffusivity**.

Laminar Flame Speed

(in the **Premixed Combustion...** category) is the propagation speed of laminar premixed flames (U_l in Equation 8.77 in the Theory Guide). Its unit quantity is **velocity**.

Laminar Kinetic Energy (kl)

(in the **Turbulence...category**) is a measure of the "laminar" streamwise fluctuations present in the pre-transitional region of the boundary layer subjected to free-stream turbulence. A transport equation of kl is considered by the k-kl-omega transition model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Latent Heat *n*

(in the **Phase Interaction...** category) is the latent heat for the n^{th} mass transfer mechanism that you defined.

LEE Self-Noise X-Source, LEE Self-Noise Y-Source, LEE Self-Noise Z-Source

(in the **Acoustics...** category) are the self-noise source terms in the linearized Euler equation for the acoustic velocity component (see Equation 11.38 in the Theory Guide). They are available only when the **Broadband Noise Sources** acoustics model is being used.

LEE Shear-Noise X-Source, LEE Shear-Noise Y-Source, LEE Shear-Noise Z-Source

(in the **Acoustics...** category) are the shear-noise source terms in the linearized Euler equation for the acoustic velocity component (see Equation 11.38 in the Theory Guide). They are available only when the **Broadband Noise Sources** acoustics model is being used.

LEE Total Noise X-Source, LEE Total Noise Y-Source, LEE Total Noise Z-Source

(in the **Acoustics...** category) are the total noise source terms in the linearized Euler equation for the acoustic velocity component (see Equation 11.38 in the Theory Guide). The total noise source term is the sum of the self-noise and shear-noise source terms. They are available only when the **Broadband Noise Sources** acoustics model is being used.

Length Ratio

(in the **Sensitivities...** category) This field is available when the **Neural Network Model** has been applied for turbulence model optimization within the **Optimizer Design Variables** dialog box, and is the length ratio flow feature.

LES Subgrid Turbulent Viscosity

(in the **Turbulence...** category) is the eddy viscosity that is determined by the local algebraic sub-grid scale model in an embedded LES zone, which actually affects the momentum transport equations (see Embedded Large Eddy Simulation (ELES) in the Theory Guide). Its unit quantity is **viscosity**.

Lilley's Self-Noise Source

(in the **Acoustics...** category) is the self-noise source term in the linearized Lilley's equation (see Equation 11.42 in the Theory Guide), available only when the **Broadband Noise Sources** acoustics model is being used.

Lilley's Shear-Noise Source

(in the **Acoustics...** category) is the shear-noise source term in the linearized Lilley's equation (see Equation 11.42 in the Theory Guide), available only when the **Broadband Noise Sources** acoustics model is being used.

Lilley's Total Noise Source

(in the **Acoustics...** category) is the total noise source term in the linearized Lilley's equation (see Equation 11.42 in the Theory Guide). The total noise source term is the sum of the self-noise and shear-noise source terms, available only when the **Broadband Noise Sources** acoustics model is being used.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Liquid Fraction

(in the **Solidification/Melting...** category) is the liquid fraction β computed by the solidification/melting model:

$$\beta = \frac{\Delta H}{L} = 0 \quad if \quad T < T_{solidus} \tag{42.21}$$

$$\beta = \frac{\Delta H}{L} = 1 \quad if \quad T > T_{liquidus} \tag{42.22}$$

$$\beta = \frac{\Delta H}{L} = \frac{T - T_{solidus}}{T_{liquidus} - T_{solidus}} \quad if \quad T_{solidus} < T < T_{liquidus} \tag{42.23}$$

Liquid Subcooling

(in the **Phase Interaction...** category) is the liquid subcooling of the boiling model defined as:

$$T_{subcool} = T_{sat} - T_l$$

where T_{sat} and T_l are the saturation and liquid temperatures.

This item is available for the Eulerian multiphase boiling model only.

Lithium Concentration

(in the **Lithium...** category) is the lithium concentration in the electrode (c_s in Equation 18.13 in the *Fluent Theory Guide*) or in the electrolyte (c_e in Equation 18.14 in the *Fluent Theory Guide*). The unit quantity is kmol/m ^{3}.

log10(Shape Sensitivity Magnitude)

(in the **Sensitivities...** category) provides a convenience function that plots $\log_{10}$ of the magnitude, in view of the large range of values possible for the shape sensitivity magnitude. This allows the importance of the surfaces in a domain to be ranked more easily based on how they affect the observation of interest when they are reshaped.

Logarithmic Streamwise Schlieren

(in the **Density...** category) is defined as:

$$Logarithmic\,Streamwise\,Schlieren = \begin{cases} \ln[g+1] & g > 0 \\ -\ln[(-g)+1] & g < 0 \end{cases} \tag{42.24}$$

where $g = \frac{\nabla\rho \cdot u}{\mathrm{MAX}(0.001, |u|)}$, ρ is density, and u is velocity. The term $\nabla\rho \cdot u$ provides directional information on the density gradient. $\nabla\rho \cdot u$ yields a positive value in the shock and a negative value in the expansion fan. Therefore, shocks and expansions can be differentiated using the **Logarithmic Streamwise Schlieren** definition. The Logarithmic function is introduced to display numerical data over a wide range and simultaneously maintain the sign of $\nabla\rho \cdot u$. Plots or reports of **Logarithmic Streamwise Schlieren** include only fluid cell zones.

Lump Asph. by Orthogonality

(in the **Lump Detection...** category) is a measure of the non-sphericity of the liquid lump according to the surface-radius orthogonality method.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Lump Asph. by Rad. Std. Dev.

(in the **Lump Detection...** category) is a measure of the non-sphericity of the liquid lump according to the radius standard deviation method.

Lump Density

(in the **Lump Detection...** category) is a volume-weighted average of the lump density.

Lump Diameter

(in the **Lump Detection...** category) is the diameter of a volume-equivalent sphere.

Lump Enthalpy

(in the **Lump Detection...** category) is the average specific enthalpy of the lump. This quantity is available only when the energy equation is enabled.

Lump ID

(in the **Lump Detection...** category) is a unique integer value for every contiguous portion of liquid. One ID is assigned to the liquid core, and other IDs are assigned to each detached lump.

Lump Pressure

(in the **Lump Detection...** category) is the average pressure of the lump. This quantity is available only when the energy equation is enabled.

Lump Temperature

(in the **Lump Detection...** category) is the average temperature of the lump. This quantity is available only when the energy equation is enabled.

Lump X, Y, Z Coordinate

(in the **Lump Detection...** category) are the X, Y, and Z coordinates of the center of gravity of the liquid lump.

Lump X, Y, Z Velocity

(in the **Lump Detection...** category) are the X, Y, and Z components of the liquid lump velocity vector.

Mach Number

(in the **Velocity...** category) is the ratio of velocity and speed of sound.

Magnitude of Lithium Mass Flux Vector

(in the **Lithium...** category) is the magnitude of the lithium mass flux vector $\vec{N}$ in Equation 18.4 in the *Fluent Theory Guide*. The unit quantity is kmol/m^2 s^{-1}.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Magnitude of Sensitivity to Body Forces (Cell Values)

(in the **Sensitivities...** category) is the magnitude of the adjoint velocity primitive field. This field can be interpreted as the magnitude of the sensitivity of the observable to body force per unit volume. It can be used to identify regions in the domain where small changes to the momentum balance in the flow can have a large or small effect on the observable. This field is often observed to be large, for example, upstream of a body for which drag sensitivity is of interest, with the field diminishing in the upstream direction. This indicates the interference effect for an object positioned at various locations upstream of the object of interest.

Mahalanobis distance

(in the **Sensitivities...** category) This field is available when the **Neural Network Model** is selected for turbulence model optimization within the **Optimizer Design Variables** dialog box. Large values for the **Mahalanobis distance** suggest large deviations in flow features compared to the training data results.

Mark Poor Elements

(in the **Mesh...** category) is a parameter that can be used to mark and/or display invalid and poor elements. The solver variable can take a value of 1 to 6 and can be visualized:

- 1.0 user_defined_poor_elements
- 2.0 poor_quality_cells
- 3.0 modified_centroid_cells
- 4.0 cell adjacent to small_face_area
- 5.0 invalid_cells
- 6.0 solver_identified_poor_cells

By default 3.0, 4.0 and 5.0 are active. 2.0 is recommended.

Mass-Averaged Relaxation Time

(in the **Two-Temperature Model...** category) is the mass-averaged translational-vibrational relaxation time.

Mass Concentration of *species-n* (Phase Level)

(in the **Species...** category) is the mass of a component n in phase p per unit phase volume:

$$\rho_{n,p}=\rho_p y_{n,p}$$

where ρ_p is the density of phase p, and $y_{n,p}$ is the mass fraction of component n in phase p.

Mass Concentration of *species-n* (Mixture Level)

(in the **Species...** category) is the mass of a component n in phase p per unit cell volume:

$$\rho_{n,p} = \rho_p y_{n,p} \alpha_p$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where ρ_p is the density of phase p, $y_{n,p}$ is the mass fraction of component n in phase p, and α_p is the volume fraction of phase p. You can use a volume integral of this quantity to report the mass of a phase component as described in Multiphase Species Transport (p. 3213).

Mass fraction of Pollutant hcn, Mass fraction of Pollutant nh3, Mass fraction of Pollutant no, Mass fraction of Pollutant n2o

(in the **NOx...** category) are the mass of HCN, the mass of NH_3, the mass of NO, and the mass of N_2O per unit mass of the mixture (for example, kg of HCN in 1 kg of the mixture). The **Mass fraction of Pollutant hcn** and the **Mass fraction of Pollutant nh3** will appear only if you are modeling fuel NOx. See Fuel NOx Formation in the Theory Guide for details.

Mass fraction of nuclei

(in the **Soot...** category) is the number of particles per unit mass of the mixture (in units of particles $\times 10^{15}$ /kg) The **Mass fraction of nuclei** will appear only if you use the two-step soot model. See Soot Formation (p. 2602) for details.

Mass fraction of soot

(in the **Soot...** category) is the mass of soot per unit mass of the mixture (for example, kg of soot in 1 kg of the mixture). See Soot Formation (p. 2602) for details.

Mass fraction of *species-n*

(in the **Species...** category) is the mass of a species per unit mass of the mixture (for example, kg of species in 1 kg of the mixture).

Mass Transfer Coefficient (Kl)

(in the **Phase Interaction...** category) is the interphase mass transfer coefficient. Its unit quantity is **velocity**.

Mass Transfer Rate *n*

(in the **Phase Interaction...** category) is the mass transfer rate for the n^{th} mass transfer mechanism that you defined.

Mean *n*

(in the **Steady Statistics...** or **Unsteady Statistics...** category) is the iteration- or time-averaged value of a solution variable *n* (for example, **Static Pressure**). See Performing Steady-State Calculations (p. 3639) and Postprocessing for Time-Dependent Problems (p. 3669) for details.

Mean- *cff_n*

(in the **Steady Statistics...** or **Unsteady Statistics...** category) is the iteration- or time-averaged value of a custom field function *cff_n* (for example, **uns-custom-function-0**). See Performing Steady-State Calculations (p. 3639) and Postprocessing for Time-Dependent Problems (p. 3669) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mean DPM *n*

(in the **Steady Statistics...** or **Unsteady Statistics...** category) is the iteration- or time-averaged value of a discrete phase variable *n* (for example, **Volume Fraction**). See Performing Steady-State Calculations (p. 3639) and Postprocessing for Time-Dependent Problems (p. 3669) for details.

Meridional Coordinate

(in the **Mesh...** category) is the normalized (dimensionless) coordinate that follows the flow path from inlet to outlet. Its value varies from 0 to 1.

Mesh...

includes variables related to the mesh.

Moving Mesh Courant Number

(in the **Velocity...** category) is a non-dimensional value that indicates the number of cells that might be swept in a single time step due to a mesh motion. This field variable is available for transient cases that involve either dynamic mesh or cell zones with frame or mesh motion.

Mesh X-Velocity, Mesh Y-Velocity, Mesh Z-Velocity

(in the **Velocity...** category) are the vector components of the mesh velocity for moving-mesh problems (rotating or multiple reference frames, mixing planes, sliding meshes, or solid motion). Its unit quantity is **velocity**.

Mixture Fraction Variance

(in the **Pdf...** category) is the variance of the mixture fraction solved for in the non-premixed combustion model. This is the second conservation equation (along with the mixture fraction equation) that the non-premixed combustion model solves. (See Definition of the Mixture Fraction in the Theory Guide.)

Modified Turbulent Viscosity

(in the **Turbulence...** category) is the transported quantity $\tilde{\nu}$ that is solved for in the Spalart-Allmaras turbulence model and in DES with the Spalart-Allmaras model (see Equation 4.15 in the Theory Guide). The turbulent viscosity, μ_t, is computed directly from this quantity using the relationship given by Equation 4.16 in the Theory Guide. Its unit quantity is **viscosity**.

Molar Concentration of *`species-n`*

(in the **Species...** category) is the moles per unit volume of a species. Its unit quantity is **concentration**.

Mole fraction of *`species-n`*

(in the **Species...** category) is the number of moles of a species in one mole of the mixture.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mole fraction of Pollutant hcn, Mole fraction of Pollutant nh3, Mole fraction of Pollutant no, Mole fraction of Pollutant n2o

(in the **NOx...** category) are the number of moles of HCN, NH_3, NO, and N_2O in one mole of the mixture. The **Mole fraction of Pollutant hcn** and the **Mole fraction of Pollutant nh3** will appear only if you are modeling fuel NOx. See Fuel NOx Formation in the Theory Guide for details.

Mole fraction of soot

(in the **Soot...** category) is the number of moles of soot in one mole of the mixture.

Molecular Prandtl Number

(in the **Properties...** category) is the ratio $c_p\mu_{lam} / k_{lam}$.

Molecular Viscosity

(in the **Properties...** category) is the laminar viscosity of the fluid. Viscosity, μ, is defined by the ratio of shear stress to the rate of shear. Its unit quantity is **viscosity**. For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list. For a granular phase, the molecular viscosity is multiplied by the granular volume fraction.

Momentum Thickness Re ($Re_{\theta t}$)

(in the **Turbulence...** category) is based on the momentum thickness of the boundary layer. The SST transition model is considering a nonlocal empirical correlation for the value of $Re_{\theta t}$ in the free-stream, based on turbulence intensity, pressure gradient, and so on, and a transport equation to allow the free-stream value to diffuse into the boundary layer.

Multiphase Minimum Time Scale

(in the **Velocity...** category) is the minimum of all the selected and relevant time scales in the whole domain. Note that the final time-step size is estimated only from interfacial cells. This field variable is available for transient VOF cases that involve Multiphase Specific Time Stepping.

nh3 Density, no Density, n2o Density

(in the **NOx...** category) are the mass per unit volume of NH_3, NO, and N_2O. The unit quantity for each is **density**. The **nh3 Density** will appear only if you are modeling fuel NOx. See Fuel NOx Formation in the Theory Guide for details.

NLBF Averaged Temperature

(in the **Temperature...** category) is the average temperature determined by the non-local boundary field (NLBF) model. This item is available when **NLBF Model** is enabled in the in the **Boiling Model** dialog box.

Non-Equilibrium Parameter

(in the **Sensitivities...** category) This field is available when the **Neural Network Model** has been applied for turbulence model optimization within the **Optimizer Design Variables** dialog box, and is the Non-Equilibrium Parameter flow feature.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Non-Equilibrium Thermal Model Source

(in the **Temperature...** category) is the scaled value of thermal conductivity for the fluid zone (γk_f) or for the overlapping solid zone ($(1-\gamma)k_s$), where γ is the porosity, k_f is the fluid phase thermal conductivity (including the turbulent contribution, k_t), and k_s is the solid medium thermal conductivity. These terms can be seen in Equation 7.14 (p. 1338) and Equation 7.15 (p. 1338).

See Non-Equilibrium Thermal Model Equations (p. 1338) and Non-Equilibrium Thermal Model (p. 1355) for details about the non-equilibrium thermal model.

Normal Optimal Displacement

(in the **Sensitivities...** category) shows the normal component of the optimal displacement computed from the adjoint design tool calculation. This field is defined only for portions of walls lying within the control-volume specified for morphing. A positive value of displacement indicates that the surface will be displaced into the flow domain, whereas a negative value of displacement corresponds to wall movement outwards from the flow domain. This field eliminates the component of the optimal displacement vector that lies in the plane of the wall.

Normal Shape Sensitivity

(in the **Sensitivities...** category) shows the normal component of the shape sensitivity. A positive value indicates an orientation directed into the domain, while a negative value indicates that the shape sensitivity is oriented outwards from the domain. This field eliminates the component of the vector shape sensitivity field that lies in the plane of the wall.

Normalized Concentration of Nuclei

(in the **Soot...** category) is the normalized soot particle number density.

Normalized Soot Aggregation Moments

(in the **Soot...** category) is the normalized soot aggregate moments. This quantity is available only with the Method of Moments model.

Normalized soot moments

(in the **Soot...** category) are the normalized soot moments. This quantity is available only with the Method of Moments model.

NOx...

contains quantities related to the NOx model. See NOx Formation (p. 2585) for details about this model.

Optimal Displacement X-Component, Optimal Displacement Y-Component, Optimal Displacement Z-Component

(in the **Sensitivities...** category) are the individual components of the optimal displacement computed from the adjoint design tool calculation. These fields are defined only for portions of walls lying within the control-volume specified for morphing.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Orthogonal Quality

(in the **Mesh...** category) is a measure of the quality of a mesh, and is computed as described in Mesh Quality (p. 1146). The worst cells will have an **Orthogonal Quality** closer to 0, with better cells closer to 1.

Osmotic Drag

(in the **Potential...** category) is the third term in Equation 18.17 in the *Fluent Theory Guide*.

Over Potential

(in the **Potential...** category) is the surface overpotential ζ_{an} and ζ_{cat} in Equation 20.5 and Equation 20.6 in the *Fluent Theory Guide*.

Overset Cell Type

(in the **Cell Info...** category) is an integer value indicating the overset cell type. Each cell can have one of the following values:

Cell Type	Integer Function Value
donor	2
solve	1
receptor	0
orphan	–1
dead	–2

Overset Donor Count

(in the **Cell Info...** category) shows the number of donor cells a receptor cell has for data interpolation. A value of zero indicates that the cell is not a receptor, or that the receptor cell is an orphan cell.

Overset Donor Size Ratio

(in the **Mesh...** category) is the ratio of the donor cell volume (V_d) divided by its smallest receptor cell volume (V_r). This field variable allows you to determine if your donor cells are excessively large and so in need of refinement.

Overset Receptor Count

(in the **Cell Info...** category) shows the number of receptor cells referencing a donor cell. A value of zero indicates that the cell is not a donor cell.

Primary Particle Diameter

(in the **Soot...** category) is a soot primary particle diameter within soot aggregates. This quantity is available only with the Method of Moments model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Partition Boundary Cell Distance

(in the **Mesh...** category) is the smallest number of cells that must be traversed to reach the nearest partition (interface) boundary.

Partition Neighbors

(in the **Cell Info...** category) is the number of adjacent partitions (that is, those that share at least one partition boundary face (interface)). It gives a measure of the number of messages that will have to be generated for parallel processing.

Pdf...

contains quantities related to the non-premixed combustion model, which is described in Modeling Non-Premixed Combustion (p. 2471).

PDF Table Adiabatic Enthalpy

is the adiabatic enthalpy corresponding to the cell value of mixture fraction. For single mixture fraction cases it is given by the following equation:

$$H_{ad}=H_{fuel}f+H_{oxidizer}\left(1-f\right) \tag{42.25}$$

and for cases involving a secondary stream it is given by the following equation:

$$H_{ad}=H_{fuel}f+H_{secondary}f_{sec}+H_{oxidizer}\left(1-f_{sec}-f\right) \tag{42.26}$$

where

f = mixture fraction

f_{sec} = secondary mixture fraction

H_{fuel} = total enthalpy of the fuel stream

$H_{secondary}$ = total enthalpy of the secondary stream

$H_{oxidizer}$ = total enthalpy of the oxidizer stream

For adiabatic cases the **PDF Table Adiabatic Enthalpy** is equal to the value of Enthalpy. The unit of measurement is specific-energy.

PDF Table Heat Loss/Gain

is given by the following equation:

$$h_{loss}=\left(H-H_{min}\right)/\left(H_{ad}-H_{min}\right)-1 \tag{42.27}$$

if the cell enthalpy is less than the adiabatic enthalpy, and by the following equation:

$$h_{gain}=1-\left(H_{max}-H\right)/\left(H_{max}-H_{ad}\right) \tag{42.28}$$

if the cell enthalpy is higher than adiabatic

where

H = total enthalpy

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

H_{ad} = the PDF Table Adiabatic Enthalpy

H_{min} = the minimum Enthalpy defined in the PDF table

H_{max} = the maximum Enthalpy defined in the PDF table

The **PDF Table Heat Loss/Gain** is dimensionless and ranges in value from -1, when H is equal to H_{min}, to +1, when H is equal to H_{max}. If H is equal to the adiabatic enthalpy it will be 0.

Phases...

contains quantities for reporting the volume fraction of each phase. See Modeling Multiphase Flows (p. 2915) for details.

Pitchwise Coordinate

(in the **Mesh...** category) is the normalized (dimensionless) coordinate in the circumferential (pitchwise) direction. Its value varies from 0 to 1.

Porous Deposition Rate of *species-n*

(in the **Species...** category) is the amount of a surface species that is deposited on porous zones. Its unit quantity is **mass-flux**.

Preconditioning Reference Velocity

(in the **Velocity...** category) is the reference velocity used in the coupled solver's preconditioning algorithm. See Preconditioning in the Theory Guide for details.

Premixed Combustion...

contains quantities related to the premixed combustion model, which is described in Modeling Premixed Combustion (p. 2536).

Pressure...

includes quantities related to a normal force per unit area (the impact of the gas molecules on the surfaces of a control volume).

Pressure Coefficient

(in the **Pressure...** category) is a dimensionless parameter defined by the equation

$$C_p = \frac{\left(p - p_{ref}\right)}{q_{ref}} \tag{42.29}$$

and q_{ref} is the reference dynamic pressure defined by $\frac{1}{2}\rho_{ref}v_{ref}{}^2$. The reference pressure, density, and velocity are defined in the Reference Values Task Page (p. 5154).

Pressure Discontinuity Sensor

(in the **Pressure...** category) is a binary identifier based on whether a cell is in proximity to a pressure discontinuity (in which case the value = 1) or not (value = 0). Note that this field variable will display

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

only the strongest part of the shock front and not necessarily the entire shock system; often the shock runs longer than indicated. For details, see Enabling High-Speed Numerics (p. 3608).

Pressure Hessian Indicator

(in the **Derivatives...** category) is an indicator of the solution error, as approximated from the Hessian (the matrix of second derivatives) of the pressure distribution. The error is expected to be higher in cells that have a higher value compared to other cells in the domain, and so this field variable may indicate where refinement is needed to increase accuracy. This indicator is used as part of the predefined criteria for adaption (under **Aerodynamics...** / **Error-based** / **Pressure Hessian Indicator**), as described in Aerodynamics Adaption (p. 3787). For further details on how the indicator is calculated, see the description of the Hessian approach in Approaches For Deriving Field Values (p. 3855).

Pressure Spectrum Im *n*

(in the **Acoustics...** category) is the imaginary part of the complex Fourier amplitudes for Fourier mode *n*, as calculated by the Acoustic Sources FFT Dialog Box (p. 5224). See FFT of Acoustic Sources: Band Analysis and Export of Surface Pressure Spectra (p. 2644) for details.

Pressure Spectrum Re *n*

(in the **Acoustics...** category) is the real part of the complex Fourier amplitudes for Fourier mode *n*, as calculated by the Acoustic Sources FFT Dialog Box (p. 5224). See FFT of Acoustic Sources: Band Analysis and Export of Surface Pressure Spectra (p. 2644) for details.

Product Formation Rate

(in the **Premixed Combustion...** category) is the source term in the progress variable transport equation (S_c in Equation 8.70 in the Theory Guide). Its unit quantity is **time-inverse**.

Production of k

(in the **Turbulence...** category) is the rate of production of turbulence kinetic energy (times density). Its unit quantity is **turb-kinetic-energy-production**. For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list.

Production of laminar k

(in the **Turbulence...** category) is the production of laminar kinetic energy used in the k - kl - ω transition model. See Transport Equations for the k-kl-ω Model in the Theory Guide for more details (Equation 4.150).

Progress Variable

(in the **Premixed Combustion...** category) is a normalized mass fraction of the combustion products (c=1) or unburnt mixture products (c=0), as defined by Equation 8.107 in the Theory Guide.

Progress Variable Variance

(in the **Premixed Combustion...** category) is the variance of the reaction progress variable $\overline{c}$ that is solved in the partially-premixed combustion model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Properties...

includes material property quantities for fluids and solids.

PSD of dp/dt for Octave Band at *x* Hz , PSD of dp/dt for 1/3-Octave Band at *x* Hz , PSD of dp/dt for Const Width Band *n*

(in the **Acoustics...** category) is the power spectral density of the flow pressure time derivative dp/dt. It is an integral value for the standard technical octave band (or the standard technical 1/3-octave band) at the central frequency *x*, or for the user-defined constant width band number *n*. These fields are calculated in the **Acoustic Sources FFT Dialog Box**. See FFT of Acoustic Sources: Band Analysis and Export of Surface Pressure Spectra (p. 2644) for the workflow details, and Specifying a Function for the Y Axis (p. 4085) for the calculation recipe of PSD for a frequency band. Note that the Fourier magnitude of an individual dp/dt harmonic at frequency *f* is computed based on the magnitude of the correspondent harmonic of pressure by multiplying it with $2\pi f$.

Q Criterion Normalized

(in the **Velocity...** category) is a postprocessing quantity used for visual inspection of turbulence structures. It is defined by the equation:

$$Q_{norm}=\frac{0.5(\Omega^2-S^2)}{(v_{ref}^2/l_{ref}^2)} \tag{42.30}$$

where l_{ref} and v_{ref} are reference length and velocity, defined in the Reference Values Task Page (p. 5154).

and the definitions of Ω and S in the following definition of **Q Criterion Raw** in Equation 42.32 (p. 4249).

Note:

In the TUI, **Q Criterion Normalized** is referred to as `q-criterion`.

Q Criterion Raw

(in the **Velocity...** category) is a postprocessing quantity used for visual inspection of turbulence structures. It is defined by the equation:

$$Q=\frac{1}{2}(\Omega^2-S^2) \tag{42.31}$$

where Ω and S are norms of the vorticity and the strain rate tensors, respectively:

$$\Omega=\sqrt{2\Omega_{ij}\Omega_{ij}}\ ,\ S=\sqrt{2S_{ij}S_{ij}}\ ,\ \Omega_{ij}=\frac{1}{2}\left(\frac{\partial U_i}{\partial x_j}-\frac{\partial U_j}{\partial x_i}\right),\ S_{ij}=\frac{1}{2}\left(\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i}\right) \tag{42.32}$$

The definitions of **Vorticity Magnitude** and **Strain Rate**, which are provided separately in this list, are equivalent to the definitions of Ω and S above.

Note:

In the TUI, **Q Criterion Raw** is referred to as `raw-q-criterion`.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Radial Coordinate

(in the **Mesh...** category) is the length of the radius vector in the polar coordinate system. The radius vector is defined by a line segment between the node and the axis of rotation. You can define the rotational axis in the Fluid Dialog Box (p. 4961). (See also Velocity Reporting Options (p. 4169).) The unit quantity for **Radial Coordinate** is **length**.

Radial Pull Velocity

(in the **Solidification/Melting...** category) is the radial-direction component of the pull velocity for the solid material in a continuous casting process. Its unit quantity is **velocity**.

Radial Velocity

(in the **Velocity...** category) is the component of velocity in the radial direction. (See Velocity Reporting Options (p. 4169) for details.) The unit quantity for **Radial Velocity** is **velocity**. For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list.

Radial-Wall Shear Stress

(in the **Wall Fluxes...** category) is the radial component of the force per unit area acting tangential to the surface due to friction. Its unit quantity is that of **pressure**.

Radiation...

includes quantities related to radiation heat transfer. See Modeling Radiation (p. 2192) for details about the radiation models available in Ansys Fluent.

Radiation Heat Flux

(in the **Wall Fluxes...** category) is the rate of radiation heat transfer through the control surface. It is calculated by the solver according to the specified radiation model. Heat flux out of the domain is negative, and heat flux into the domain is positive. The unit quantity for **Radiation Heat Flux** is **heat-flux**.

Radiation Intensity.Normalized Std Deviation

(in the **Radiation...** category) is the statistical standard deviation for the spherical mean radiation intensity.

Radiation Temperature

(in the **Radiation...** category) is the quantity θ_R, defined by

$$\theta_R = \left(\frac{G}{4\sigma}\right)^{1/4} \tag{42.33}$$

where G is the **Incident Radiation**. The unit quantity for **Radiation Temperature** is **temperature**.

Rate of Coagulation

(in the **Soot...** category) is the rate of the coagulation of soot particles.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Rate of NO

(in the **NOx...** category) is the overall rate of formation of NO due to all active NO formation pathways (for example, thermal, prompt, and so on).

Rate of Nucleation

(in the **Soot...** category) is the rate of nucleation of the soot particles.

Rate of Nuclei

(in the **Soot...** category) is the overall rate of formation of nuclei.

Rate of N2OPath NO

(in the **NOx...** category) is the rate of formation of NO due to the N_2O pathway only (only available when N_2O pathway is active).

Rate of Oxidation

(in the **Soot...** category) is the rate of oxidation of the soot particles.

Rate of Prompt NO

(in the **NOx...** category) is the rate of formation of NO due to the prompt pathway only (only available when prompt pathway is active).

Rate of Reburn NO

(in the **NOx...** category) is the rate of formation of NO due to the reburn pathway only (only available when reburn pathway is active).

Rate of SNCR NO

(in the **NOx...** category) is the rate of formation of NO due to the SNCR pathway only (only available when SNCR pathway is active).

Rate of Soot

(in the **Soot...** category) is the overall rate of formation of soot mass.

Rate of Soot Mass Nucleation

(in the **Soot...** category) is the overall rate of soot nucleation.

Rate of Surface Growth

(in the **Soot...** category) is the rate of the soot particle surface growth due to reactions at soot particle surface.

Rate of Thermal NO

(in the **NOx...** category) is the rate of formation of NO due to the thermal pathway only (only available when thermal pathway is active).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Rate of Fuel NO

(in the **NOx...** category) is the rate of formation of NO due to the fuel pathway only (only available when fuel pathway is active).

Rate of USER NO

(in the **NOx...** category) is the rate of formation of NO due to user defined rates only (only available when user-defined function rates are added).

Rate of Reaction-n

(in the **Reactions...** category) is the effective rate of progress of n^{th} reaction. For the finite-rate model, the value is the same as the **Kinetic Rate of Reaction-n**. For the eddy-dissipation model, the value is equivalent to the **Turbulent Rate of Reaction-n**. For the finite-rate/eddy-dissipation model, it is the lesser of the two.

For particle reactions it is the global rate of the particle reaction n expressed in kmol/s/m ^{3}. This is computed as

$$\frac{\overline{R}_{j,r}}{M_j V}$$

where $\overline{R}_{j,r}$ is the rate of particle species depletion (or generation) given by Equation 7.102 in the Theory Guide, M_j is the particle species molecular weight, and V is the cell volume.

Reactions...

includes quantities related to finite-rate reactions. See Modeling Species Transport and Finite-Rate Chemistry (p. 2355) for information about modeling finite-rate reactions.

Reactor Net Mass Fraction of *`Species-n`*

(in the **Species...** category) is the mass of a species per unit mass of the mixture in the reactor network.

Reactor Net Temperature

(in the **Species...** category) is the computed temperature of the reactor network.

Reactor Net Zone ID

(in the **Species...** category) a unique identifier associated with each reactor network.

Reduced Temperature

(in the **Properties...** category) is the ratio T / T_c of the fluid temperature T divided by the critical temperature T_c. The reduced temperature T_r is available only with the Angier-Redlich-Kwong real gas model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reduced Pressure

(in the **Properties...** category) is the ratio P / P_c of the fluid pressure P divided by the critical pressure P_c. The reduced pressure P_r is available only with the Angier-Redlich-Kwong real gas model.

Reflected Radiation Flux (Band-n)

(in the **Wall Fluxes...** category) is the amount of radiative heat flux reflected by a semi-transparent wall for a particular band of radiation. Its unit quantity is **heat-flux**.

Reflected Visible Solar Flux, Reflected IR Solar Flux

(in the **Wall Fluxes...** category) is the amount of solar heat flux reflected by a semi-transparent wall or porous jump boundary for a visible or infrared (IR) radiation.

Refractive Index

(in the **Radiation...** category) is a nondimensional parameter defined as the ratio of the speed of light in a vacuum to that in a material. See Specular Semi-Transparent Walls in the Theory Guide for details.

Relative Axial Velocity

(in the **Velocity...** category) is the axial-direction component of the velocity relative to the reference frame motion. See Velocity Reporting Options (p. 4169) for details. The unit quantity for **Relative Axial Velocity** is **velocity**.

Relative Humidity

(in the **Species...** category) is the ratio of the partial pressure of the water vapor actually present in an air-water mixture to the saturation pressure of water vapor at the mixture temperature. Ansys Fluent computes the saturation pressure, p, from the following equation [129] (p. 5720):

$$\ln\left(\frac{p}{p_c}\right) = \left(\frac{T_c}{T} - 1\right) \times \sum_{i=1}^{8} F_i [\, a(T - T_p)\,]^{i-1} \tag{42.34}$$

where

p_c = 22.089 MPa

T_c = 647.286 K

$F_1 = -7.4192420$

$F_2 = 2.9721000 \times 10^{-1}$

$F_3 = -1.1552860 \times 10^{-1}$

$F_4 = 8.6856350 \times 10^{-3}$

$F_5 = 1.0940980 \times 10^{-3}$

$F_6 = -4.3999300 \times 10^{-3}$

$F_7 = 2.5206580 \times 10^{-3}$

$F_8 = -5.2186840 \times 10^{-4}$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$a = 0.01$

$T_p = 338.15$ K

Relative Mach Number

(in the **Velocity...** category) is the nondimensional ratio of the relative velocity and speed of sound.

Relative Radial Velocity

(in the **Velocity...** category) is the radial-direction component of the velocity relative to the reference frame motion. (See Velocity Reporting Options (p. 4169) for details.) The unit quantity for **Relative Radial Velocity** is **velocity**.

Relative Swirl Velocity

(in the **Velocity...** category) is the tangential-direction component of the velocity relative to the reference frame motion, in an axisymmetric swirling flow. (See Velocity Reporting Options (p. 4169) for details.) The unit quantity for **Relative Swirl Velocity** is **velocity**.

Relative Tangential Velocity

(in the **Velocity...** category) is the tangential-direction component of the velocity relative to the reference frame motion. (See Velocity Reporting Options (p. 4169) for details.) The unit quantity for **Relative Tangential Velocity** is **velocity**.

Relative Total Enthalpy

(in the **Temperature...** category) is defined as $h+\frac{1}{2}w^2$ where h is enthalpy and w is the relative velocity magnitude.

Relative Total Pressure

(in the **Pressure...** category) is the stagnation pressure computed using relative velocities instead of absolute velocities; that is, for incompressible flows the dynamic pressure would be computed using the relative velocities. (See Velocity Reporting Options (p. 4169) for more information about relative velocities.) The unit quantity for **Relative Total Pressure** is **pressure**.

Relative Total Temperature

(in the **Temperature...** category) is the stagnation temperature computed using relative velocities instead of absolute velocities. (See Velocity Reporting Options (p. 4169) for more information about relative velocities.) The unit quantity for **Relative Total Temperature** is **temperature**.

Relative Velocity Angle

(in the **Velocity...** category) is similar to the **Velocity Angle** except that it uses the relative tangential velocity, and is defined as

$$\tan^{-1}\left(-\frac{\text{relative-tangential-velocity}}{\text{axial-velocity}}\right) \tag{42.35}$$

Its unit quantity is **angle**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Relative Velocity Magnitude

(in the **Velocity...** category) is the magnitude of the relative velocity vector instead of the absolute velocity vector. The relative velocity ($\vec{w}$) is the difference between the absolute velocity ($\vec{v}$) and the mesh velocity. For simple rotation, the relative velocity is defined as

$$\vec{w} \equiv \vec{v} - \vec{\Omega} \times \vec{r} \tag{42.36}$$

where $\vec{\Omega}$ is the angular velocity of a moving reference frame about the origin and $\vec{r}$ is the position vector. (See Velocity Reporting Options (p. 4169) for details.) The unit quantity for **Relative Velocity Magnitude** is **velocity**.

Relative X Velocity, Relative Y Velocity, Relative Z Velocity

(in the **Velocity...** category) are the x -, y -, and z -direction components of the velocity relative to the reference frame motion. (See Velocity Reporting Options (p. 4169) for details.) The unit quantity for these variables is **velocity**.

Relaxation Time

(in the **Phase Interaction...** category) is the characteristic time it takes for a particle to reach a steady state. It is calculated as:

$$\tau_p = \frac{\rho_p d_p^2}{18 \mu_q} \tag{42.37}$$

where

τ_p = relaxation time of dispersed phase p

ρ_p = density of dispersed phase p

d_p = diameter of dispersed phase p

μ_q = viscosity of continuous phase q

This item is available only for mixture multiphase cases with the flow regime modeling.

Rescaled Schlieren

(in the **Density...** category) is defined as:

$$\mathrm{Re}scaled\ Schlieren = \frac{|\nabla \rho|}{\rho} \tag{42.38}$$

where ρ is density. Plots or reports of **Rescaled Schlieren** include only fluid cell zones. The unit quantity for **Rescaled Schlieren** is length-inverse.

Rescaled Streamwise Schlieren

(in the **Density...** category) is defined as:

$$\mathrm{Re}scaled\ Streamwise\ Schlieren = \frac{\nabla \rho \cdot u}{\rho \mathrm{MAX}\left(0.001, |u|\right)} \tag{42.39}$$

where ρ is density and u is velocity. The term $\nabla \rho \cdot u$ provides directional information on the density gradient. $\nabla \rho \cdot u$ yields a positive value in the shock and a negative value in the expansion fan.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Therefore, shocks and expansions can be differentiated using the **Rescaled Streamwise Schlieren** definition. Plots or reports of **Rescaled Streamwise Schlieren** include only fluid cell zones. The unit quantity for **Rescaled Streamwise Schlieren** is length-inverse.

Residuals...

contains different quantities for the pressure-based and density-based solvers:

For the density-based solver, by default only the **Time Step** is available. If you enable the retention of cell residuals (as described in Postprocessing Residual Values (p. 3694)), this category also includes the residuals of the continuity, momentum, and energy conservation equations. The displayed residuals are defined as the rate of change of the conserved quantity (as described in Monitoring Residuals (p. 3684)). They are computed as the summation of the Euler, viscous, and dissipation fluxes divided by the local cell volume. Note that the residuals that are made available for postprocessing are the unscaled values.

For the pressure-based solver, only the **Mass Imbalance** in each cell is reported (unless you have requested others, as described in Postprocessing Residual Values (p. 3694)). At convergence, this quantity should be small compared to the average mass flow rate.

Reverse Reaction Rate of *PDF scalar-n*

(in the **Premixed Combustion...** category) is the rate of consumption of *species-n* due to reverse reactions for scalar φ in kg/m ^{3} /s ($\overline{S}_{\varphi,rev}$ in Equation 8.120 in the *Fluent Theory Guide*). (The name of the PDF scalar will replace *PDF scalar-n* in **Reverse Reaction Rate of *PDF scalar-n*** .)

RMS (*species-n*) Mass Fraction

(in the **Species...** category) is the root mean square value of the mass of a species per unit mass of the mixture.

RMSE *n*

(in the **Steady Statistics...** or **Unsteady Statistics...** category) is the root mean squared error value of a solution variable *n* (for example, **Static Pressure**). See Performing Steady-State Calculations (p. 3639) and Postprocessing for Time-Dependent Problems (p. 3669) for details.

RMSE- *cff_n*

(in the **Steady Statistics...** or **Unsteady Statistics...** category) is the root mean squared error value of a custom field function *cff_n* (for example, **uns-custom-function-0**). See Performing Steady-State Calculations (p. 3639) and Postprocessing for Time-Dependent Problems (p. 3669) for details.

RMS DPM *n*

(in the **Steady Statistics...** or **Unsteady Statistics...** category) is the root mean square value of a discrete phase variable *n* (for example, **Volume Fraction**). See Performing Steady-State Calculations (p. 3639) and Postprocessing for Time-Dependent Problems (p. 3669) for details.

RMS of Temperature

(in the **NOx...** category) is the root-mean-square (RMS) of the normalized temperature in the flow field. It is calculated from Equation 9.113 in the Theory Guide.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Rothalpy

(in the **Temperature...** category) is defined as

$$I=h+\frac{w^2}{2}-\frac{u^2}{2} \tag{42.40}$$

where h is the enthalpy, w is the relative velocity magnitude, and u is the magnitude of the rotational velocity $\vec{u}=\vec{\omega}\times\vec{r}$.

Saturation Temperature

(in the **Phase Interaction...** category) is the saturation temperature of the boiling mass transfer mechanism. This item is available for the Eulerian multiphase boiling model only.

Saturation Temperature *n*

(in the **Phase Interaction...** category) is the saturation temperature of the n^{th} mass transfer mechanism that you defined. This item is available only for the evaporation-condensation model. See Including Semi-Mechanistic Boiling (p. 3057) for more information about the semi-mechanistic boiling model.

Saturation Vapor Pressure *n*

(in the **Phase Interaction...** category) is the saturation vapor pressure for the n^{th} cavitation mass-transfer mechanism that you defined.

Scalar-n

(in the **User Defined Scalars...** category) is the value of the n^{th} scalar quantity you have defined as a user-defined scalar. See the Fluent Customization Manual for more information about user-defined scalars.

Scalar Dissipation

(in the **Pdf...** category) is one of two parameters that describes the species mass fraction and temperature for a laminar flamelet in mixture fraction spaces. It is defined as

$$\chi=2D\left|\nabla f\right|^2 \tag{42.41}$$

where f is the mixture fraction and D is a representative diffusion coefficient (see The Flamelet Concept in the Theory Guide for details). Its unit quantity is **time-inverse**.

Scalar Mass Fraction of *`species-n`*

(in the **Premixed Combustion...** category) is the mass fraction of the *`species-n`* in the PDF mixture that is solved as a transported scalar in the partially-premixed FGM model. (The name of the species will replace *`species-n`* in **Scalar Mass Fraction of *`species-n`***).

Scattering Coefficient

(in the **Radiation...** category) is the property of a medium that describes the amount of scattering of thermal radiation per unit path length for propagation in the medium. It can be interpreted as the inverse of the mean free path that a photon will travel before undergoing scattering (if the

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

scattering coefficient does not vary along the path). The unit quantity for **Scattering Coefficient** is **length-inverse**.

Schlieren

(in the **Density...** category) is defined as:

$$Schlieren = |\nabla \rho| \tag{42.42}$$

where ρ is density. Plots or reports of **Schlieren** include only fluid cell zones. The unit quantity for **Schlieren** is density-gradient.

Second Invariant

(in the **Sensitivities...** category) This field is available when the **Neural Network Model** has been applied for turbulence model optimization within the **Optimizer Design Variables** dialog box, and is the Second Invariant flow feature.

Secondary Mean Mixture Fraction

(in the **Pdf...** category) is the mean ratio of the secondary stream mass fraction to the sum of the fuel, secondary stream, and oxidant mass fractions. It is the secondary-stream conserved scalar that is calculated by the non-premixed combustion model. See Definition of the Mixture Fraction in the Theory Guide.

Secondary Mixture Fraction Variance

(in the **Pdf...** category) is the variance of the secondary stream mixture fraction that is solved for in the non-premixed combustion model. See Definition of the Mixture Fraction in the Theory Guide.

Sensible Enthalpy

(in the **Temperature...** category) is available when any of the species models are active and displays only the thermal (sensible) enthalpy.

Sensitivity to Body Force X-Component (Cell Values), Sensitivity to Body Force Y-Component (Cell Values), and Sensitivity to Body Force Z-Component (Cell Values)

(in the **Sensitivities...** category) are the components of the adjoint velocity primitive field. These fields can be interpreted as the magnitude of the sensitivity of the observable to components of a body force per unit volume. Consider a body force distribution, expressed as a force per unit volume. The volume integral of the vector product of that distribution with the components of this field gives a first-order estimate of the net effect of the body force on the observation.

Sensitivity to Boundary Heat Flux

(in the **Sensitivities...** category) is available when the energy adjoint equation is solved and is defined on walls where a heat flux boundary condition is imposed. The field shows the sensitivity of the observation of interest to variations in the boundary heat flux through the wall. Its properties are analogous to those of Sensitivity to Boundary Temperature.

Sensitivity to Boundary Pressure

(in the **Sensitivities...** category) is defined on boundaries where there is a user-specified pressure as part of a boundary condition, such as on a pressure outlet. The field shows the sensitivity of the

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

observation of interest to variations in the boundary pressure across the flow boundary. It is interesting to note that even though the original boundary condition specification may be for a uniform pressure on the domain boundary, the effect of a non-uniform pressure perturbation is available, The effect of any specific boundary pressure change can be estimated as an integral of the product of the change to the pressure with the plotted sensitivity field. Viewing this field will also indicate whether or not the assumption of a uniform pressure is adequate for the simulation.

Sensitivity to Boundary Temperature

(in the **Sensitivities...** category) is available when the energy adjoint equation is solved and is defined on boundaries where a temperature boundary condition is applied. This includes walls, velocity inlets, mass-flow inlets, pressure inlets, and pressure outlets where a backflow temperature may be specified. The field shows the sensitivity of the observation of interest to variations in the boundary temperature across the boundaries. Note that even if the original boundary condition specification is for a uniform temperature on the boundary, the effect of a non-uniform temperature perturbation is available. The effect of any specific boundary temperature change can be estimated as an integral of the product of the change to the temperature with the plotted sensitivity field. This field can be used to indicate whether or not the assumption of a uniform temperature is adequate for the simulation.

Sensitivity to Boundary X-Velocity, Sensitivity to Boundary Y-Velocity, and Sensitivity to Boundary Z-Velocity

(in the **Sensitivities...** category) are defined on those boundaries where a user-specification of a boundary velocity is made for the original flow calculation. This includes no-slip walls. The field shows how sensitive the observable of interest is to changes in the boundary velocity at any point. It is interesting to note that even though the original boundary condition specification may be for a uniform velocity on the domain boundary, the effect of a non-uniform velocity perturbation is available, The effect of any specific boundary velocity change can be estimated as an integral of the vector product of the change to the velocity with the plotted sensitivity field. A plot of this quantity on a velocity inlet, for example, can be very useful for assessing whether or not the inlet is positioned too close to key parts of the system. That is, it addresses the question of whether or not the flow domain is too small to achieve a successful computation of the performance measure of interest. Viewing this field will also indicate whether or not the assumption of a uniform inflow is adequate.

Sensitivity to CCURV

(in the **Sensitivities...** category) This field is available when the turbulence adjoint equations are solved for a case in which the Curvature Correction option is enabled in the **Viscous Model** dialog box. It is the sensitivity of the observable with respect to the curvature correction parameter, **CCURV**, which is used to optimize the strength of the curvature correction if needed for a specific flow.

Sensitivity to Energy Sources (Cell Values)

(in the **Sensitivities...** category) is available when the energy adjoint equation is solved and is the primitive adjoint temperature field. It can be interpreted as the sensitivity of the observable with respect to thermal energy sources or sinks per unit volume in the domain.

Sensitivity to Flow Blockage

(in the **Sensitivities...** category) is provided as a convenient tool for identifying portions of the flow domain where the introduction of blockages or obstructions in the flow can affect the observation

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

of interest. Consider a blockage in the flow that generates a reaction force on the flow that is proportional to the local flow speed, and acting in the opposite direction to the local flow: $F=-\alpha(r)v$ where α is a local coefficient for the reaction force. The local contribution of this force on the observation of interest is determined by the vector product of this force with the adjoint velocity field. The flow blockage field that is plotted is $-v\cdot\tilde{v}$, namely the negative of the vector product of the flow velocity and the adjoint velocity (Cell Value).

Sensitivity to GEKO Blending Function

(in the **Sensitivities...** category) is available when the turbulence adjoint equations are solved and is the sensitivity of the observable with respect to the blending function, F_{Blend}, which is used to deactivate GEKO parameters inside boundary layers.

Sensitivity to GEKO CMIX

(in the **Sensitivities...** category) is available when the turbulence adjoint equations are solved and is the sensitivity of the observable with respect to the C_{MIX} parameter, which is used to optimize strength of mixing in free shear flows.

Sensitivity to GEKO CNW

(in the **Sensitivities...** category) is available when the turbulence adjoint equations are solved and is the sensitivity of the observable with respect to the C_{NW} parameter, which is used to optimize flow in non-equilibrium near wall regions (such as heat transfer or C_f).

Sensitivity to GEKO CSEP

(in the **Sensitivities...** category) is available when the turbulence adjoint equations are solved and is the sensitivity of the observable with respect to the C_{SEP} parameter, which is used to optimize flow separation from smooth surfaces.

Sensitivity to Mass Sources (Cell Values)

(in the **Sensitivities...** category) is the primitive adjoint pressure field. This field can be interpreted as the sensitivity of the observable with respect to mass sources or sinks in the domain. Consider a mass source / sink distribution, expressed as mass flow rate per unit volume. The volume integral of that distribution, weighted by the local value of this field, gives the effect of the sources / sinks on the observation. When plotted on a boundary, this field indicates the effect of the addition or removal of fluid from the domain upon the quantity of interest. It is important to note that in this scenario the effect of the momentum of the fluid that is added or removed is not taken into account. The boundary velocity sensitivity should be plotted if that effect is also of interest.

Sensitivity to Specific Dissipation Rate Sources (Cell Values)

(in the **Sensitivities...** category) is available when the turbulence adjoint equations are solved and is the sensitivity of the observable with respect to the specific dissipation rate (ω) sources per unit volume in the domain.

Sensitivity to Turbulent Kinetic Energy Sources (Cell Values)

(in the **Sensitivities...** category) is available when the turbulence adjoint equations are solved and is the sensitivity of the observable with respect to the turbulent kinetic energy (k) sources per unit volume in the domain.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sensitivity to Viscosity

(in the **Sensitivities...** category) shows the sensitivity of the quantity of interest to variations in the turbulent effective viscosity for a turbulent problem, or the laminar viscosity in a laminar case. The sensitivity is normalized by the cell volume to account for cell size variations in the mesh.

Shape Sensitivity Magnitude

(in the **Sensitivities...** category) is the magnitude of the sensitivity of the observable with respect to a deformation applied to the mesh (both boundary and interior mesh). When plotted on the surface of a body the locations where this quantity is large indicates where small changes to the surface shape can have a large effect on the observable of interest. If the shape sensitivity magnitude is small then the effect of shape changes in this region can be expected to have a small effect on the observable of interest. When viewing this field, it is often observed that the magnitude varies by many orders of magnitude. Contour plots will clearly draw attention to regions with the highest sensitivity (often sharp edges and corners). However, it should be remembered that a relatively small surface movement that is distributed over a large area can have a cumulative effect that is large.

Shape Sensitivity X Component, Shape Sensitivity Y Component, and Shape Sensitivity Z Component

(in the **Sensitivities...** category) are the individual components of the sensitivity of the observable of interest with respect to the mesh node locations. It is plotted as cell data and is computed as the average of the nodal sensitivities for a given cell, divided by the cell volume. Note that for this discrete adjoint solver the sensitivity of the result with respect to node locations both on and off boundaries is computed. The normalization by cell volume indicates that the fields that are plotted are the weighting factors for a continuous spatial deformation field. (Note that the nodal sensitivity data itself is used when mesh morphing is performed, and predictions about the effect of shape changes are made.)

Shielding Function for SBES or SDES

(in the **Turbulence...** category) is the shielding function f_{SDES} used as part of the Stress-Blended Eddy Simulation turbulence model and the Shielded Detached Eddy Simulation turbulence model. For details, see Stress-Blended Eddy Simulation (SBES) and Shielded Detached Eddy Simulation (SDES) in the *Fluent Theory Guide*, as well as Including the SDES or SBES Model with RANS Models (p. 2140).

Sigma XX, Sigma YY, Sigma XY, Sigma ZZ, Sigma YZ, Sigma XZ

(in the **Structure...** category) are the components of the stress tensor. These variables are intended only for solid zones and/or their adjacent walls in intrinsic fluid-structure interaction (FSI) simulations.

Skin Friction Coefficient

(in the **Wall Fluxes...** category) is a nondimensional parameter defined as the ratio of the wall shear stress and the reference dynamic pressure

$$C_f \equiv \frac{\tau_w}{\frac{1}{2}\rho_{ref} v_{ref}^2} \tag{42.43}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where τ_w is the wall shear stress, and ρ_{ref} and v_{ref} are the reference density and velocity defined in the Reference Values Task Page (p. 5154). For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list.

Slip X-velocity, Slip Y-velocity, Slip Z-velocity

(in the **Phase Interaction...** category) are the x, y, and z components of the slip velocity vector, which is the difference between velocity vectors of phases i and j:

$$\vec{V}_{ij} = \vec{V}_j - \vec{V}_i \tag{42.44}$$

Each component of the slip velocity can be postprocessed separately for each phase pair. This item is available only for mixture multiphase cases with the flow regime modeling. For further details, see Flow Regime Modeling in the *Fluent Theory Guide*.

SMB Nucleate Boiling Heat Flux

(in the **Wall Fluxes...** category) is the nucleate boiling phase heat flux in the semi-mechanistic boiling model. This item is available only when **Semi-Mechanistic** is selected as the **Boiling Model** in the **Evaporation-Condensation Model** dialog box.

SMB Single Phase Heat Flux

(in the **Wall Fluxes...** category) is the single phase heat flux in the semi-mechanistic boiling model. This item is available only when **Semi-Mechanistic** is selected as the **Boiling Model** in the **Evaporation-Condensation Model** dialog box.

Smoothed VOF Gradient-dX, Smoothed VOF Gradient-dY, Smoothed VOF Gradient-dY

(in the **Phases...** category) are the components of the volume fraction gradient based on the smoothed volume fraction field. These variables are only available when you enable **Surface Tension Force Modeling** in the **Multiphase Model** dialog box (**Phase Interaction** > **Forces** tab).

Smooth VOF Gradient Magnitude

(in the **Phases...** category) is the magnitude of the smoothed volume fraction gradient. This variable is only available when you enable **Surface Tension Force Modeling** in the **Multiphase Model** dialog box (**Phase Interaction** > **Forces** tab).

Solar Heat Flux

(in the **Wall Fluxes...** category) is the rate of solar heat transfer through the control surface. Heat flux out of the domain is negative and heat flux into the domain is positive.

Solidification/Melting...

contains quantities related to solidification and melting.

Solution and Cell Quality Measure

(in the **Mesh...** category) is a measure that indicates the cells that have poor mesh numerics applied to them when the solution and cell quality criterion is enabled; after an iteration is calculated, cells with a criterion value below the threshold (that is, poor cells) and their neighboring cells are assigned a value of 1, whereas all other cells (that is, good cells) are assigned a value of 0.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Soot...

contains quantities related to the **Soot** model, which is described in Soot Formation (p. 2602).

Soot Density

(in the **Soot...** category) is the mass per unit volume of soot. The unit quantity is **density**. See Fuel NOx Formation in the Theory Guide for details.

Soot Mean Diameter

(in the **Soot...** category) is the average diameter of the soot particles. This quantity is available only with the Method of Moments model.

Soot Surface Area

(in the **Soot...** category) is the total surface area of the soot particles. This quantity is available only with the Method of Moments model.

Soot Volume fraction

(in the **Soot...** category) is the volume fraction of soot.

Sound Speed

(in the **Properties...** category) is the acoustic speed. It is computed from $\sqrt{\frac{\gamma p}{\rho}}$. Its unit quantity is **velocity**.

Important:

Note that for the real gas models the sound speed is computed accordingly by the appropriate equation of state formulation.

Sound dP/dt

(in the **Acoustics...** category) is the time derivative of the Sound Pressure. It is available only when the acoustics wave equation model is used. Its unit quantity is pressure divided by time.

Sound Pressure

(in the **Acoustics...** category) is the acoustic pressure, resulting from the acoustics wave equation. It is available only when the acoustics wave equation model is used. Its unit quantity is pressure.

Sound Potential

(in the **Acoustics...** category) is the acoustic potential, which is a potential of the acoustics particle velocity, resulting from the acoustics wave equation. It is available only when the acoustics wave equation model is used. Its unit quantity is velocity times length.

Sound Sponge Layer Marker

(in the **Acoustics...** category) is the marker showing a sponge region of the acoustics wave equation. It is available only when the acoustics wave equation model is used.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sound WaveEq Model Source

(in the **Acoustics...** category) is the original model source term of the acoustics wave equation, before the application of the time filter. It is available only when the acoustics wave equation model is used. Its unit quantity is square of velocity divided by time.

Sound WaveEq Model Source Mask

(in the **Acoustics...** category) is the marker showing a masking region of the model source term of the acoustics wave equation. It is available only when the acoustics wave equation model is used.

Sound WaveEq Model Source Smoothed

(in the **Acoustics...** category) is the model source term of the acoustics wave equation, smoothed by the time filter. It is available only when the acoustics wave equation model is used. Its unit quantity is square of velocity divided by time.

Spanwise Coordinate

(in the **Mesh...** category) is the normalized (dimensionless) coordinate in the spanwise direction, from hub to casing. Its value varies from 0 to 1.

***species-n* Source Term**

(in the **Species...** category) is the source term in each of the species transport equations due to reactions. The unit quantity is always kg/m ^{3} -s.

Species...

includes quantities related to species transport and reactions.

Specific Dissipation Rate (Omega)

(in the **Turbulence...** category) is the rate of dissipation of turbulence kinetic energy in unit volume and time. Its unit quantity is **time-inverse**.

Specific Heat (Cp)

(in the **Properties...** category) is the thermodynamic property of specific heat at constant pressure. It is defined as the rate of change of enthalpy with temperature while pressure is held constant. Its unit quantity is **specific-heat**.

Specific Heat Ratio (gamma)

(in the **Properties...** category) is the ratio of specific heat at constant pressure to the specific heat at constant volume.

Spinodal Temperature

(in the **Properties...** category) is the temperature of the gas phase at which the derivative of pressure with respect to molar volume becomes positive. The spinodal temperature defines the point beyond which the equation of state is no longer valid. If the temperature of your case approaches the spinodal temperature in some regions, this indicates that the flow conditions in these regions

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

probably fall inside the saturation dome. The **Spinodal Temperature** is available only with the Cubic Equation of State Real Gas models.

SPL for Octave Band at *x* Hz (dB)

(in the **Acoustics...** category) is the surface pressure level (in decibels) of the standard technical octave band at the octave central frequency *x*, as calculated by the Acoustic Sources FFT Dialog Box (p. 5224). See FFT of Acoustic Sources: Band Analysis and Export of Surface Pressure Spectra (p. 2644) for details. Note that the transformation to the decibel units is done by default using the standard acoustic reference pressure value of 2 x 10^{-5} Pa; you can change this value using the Acoustics Model Dialog Box (p. 4860).

SPL for 1/3-Octave Band at *x* Hz (dB)

(in the **Acoustics...** category) is the surface pressure level (in decibels) of the standard technical 1/3-octave band at the 1/3-octave central frequency *x*, as calculated by the Acoustic Sources FFT Dialog Box (p. 5224). See FFT of Acoustic Sources: Band Analysis and Export of Surface Pressure Spectra (p. 2644) for details. Note that the transformation to the decibel units is done by default using the standard acoustic reference pressure value of 2 x 10^{-5} Pa; you can change this value using the Acoustics Model Dialog Box (p. 4860).

SPL for Const Width Band *n* (dB)

(in the **Acoustics...** category) is the surface pressure level (in decibels) of the user-defined constant band *n*, as calculated by the Acoustic Sources FFT Dialog Box (p. 5224). See FFT of Acoustic Sources: Band Analysis and Export of Surface Pressure Spectra (p. 2644) for details. Note that the transformation to the decibel units is done by default using the standard acoustic reference pressure value of 2 x 10^{-5} Pa; you can change this value using the Acoustics Model Dialog Box (p. 4860).

Sponge Blending Function

(in the **Mesh...** category) shows how a blending function changes across a sponge layer, where a value of 0 means the solver density is used and a value of 1 means the far-field value is used. For details on sponge layers, see Sponge Layers (p. 2667).

Sponge Layer Distance

(in the **Mesh...** category) is the distance from each cell inside a sponge layer to the interior boundary of a sponge layer. It changes linearly from 0 at the interior boundary to the thickness value of the sponge layer at the boundary zone. For details on sponge layers, see Sponge Layers (p. 2667).

Static Pressure

(in the **Pressure...** category) is the static pressure of the fluid. It is a gauge pressure expressed relative to the prescribed operating pressure. The absolute pressure is the sum of the **Static Pressure** and the operating pressure. Its unit quantity is **pressure**.

Static Temperature

(in the **Temperature...** and **Premixed Combustion...** categories) is the temperature in the fluid / solid. Its unit quantity is **temperature**. For a wall or shadow wall surface, the **Static Temperature** is the temperature of the adjacent fluid / solid cells; for a shell surface (see Postprocessing

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Shells (p. 2187)), it is the temperature of the shell layer cells on the c0 side (that is, the side closer to the associated wall surface).

Note that **Static Temperature** will appear in the **Premixed Combustion...** category only for adiabatic premixed combustion calculations. See Postprocessing for Premixed Combustion Calculations (p. 2541).

Steady Statistics...

includes mean and root mean square error (RMSE) values of solution variables and custom field functions derived from steady state flow calculations (with **Data Sampling for Steady Statistics** enabled).

Stored Cell Partition

(in the **Cell Info...** category) is an integer identifier designating the partition to which a particular cell belongs. In problems in which the mesh is divided into multiple partitions to be solved on multiple processors using the parallel version of Ansys Fluent, the partition ID can be used to determine the extent of the various groups of cells. The active cell partition is used for the current calculation, while the stored cell partition (the last partition performed) is used when you save a case file. See Partitioning the Mesh Manually and Balancing the Load (p. 4319) for more information.

Strain Rate

(in the **Derivatives...** category) relates shear stress to the viscosity. Also called the shear rate ($\dot{\gamma}$ in Equation 8.36 (p. 1640)), the strain rate is related to the second invariant of the rate-of-deformation tensor $\overline{\overline{D}}$. Its unit quantity is **time-inverse**. In 3D Cartesian coordinates, the strain rate, S, is defined as

$$S^2 = \left[\frac{\partial u}{\partial x}\left(\frac{\partial u}{\partial x}+\frac{\partial u}{\partial x}\right)+\frac{\partial u}{\partial y}\left(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x}\right)+\frac{\partial u}{\partial z}\left(\frac{\partial u}{\partial z}+\frac{\partial w}{\partial x}\right)\right] + \left[\frac{\partial v}{\partial x}\left(\frac{\partial v}{\partial x}+\frac{\partial u}{\partial y}\right)+\frac{\partial v}{\partial y}\left(\frac{\partial v}{\partial y}+\frac{\partial v}{\partial y}\right)+\frac{\partial v}{\partial z}\left(\frac{\partial v}{\partial z}+\frac{\partial w}{\partial y}\right)\right] + \left[\frac{\partial w}{\partial x}\left(\frac{\partial w}{\partial x}+\frac{\partial u}{\partial z}\right)+\frac{\partial w}{\partial y}\left(\frac{\partial w}{\partial y}+\frac{\partial v}{\partial z}\right)+\frac{\partial w}{\partial z}\left(\frac{\partial w}{\partial z}+\frac{\partial w}{\partial z}\right)\right] \tag{42.45}$$

For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list.

Strained Laminar Flame Speed

(in the **Premixed Combustion...** category) is the speed of the strained laminar flame inside the domain. The strained laminar flame speed is stored as a function of mixture fraction and strain rate. This item is available only for the partially-premixed FGM model. See Modeling Strained Laminar Flame Speed (p. 2567) for more information.

Stream Function

(in the **Velocity...** category) is formulated as a relation between the streamlines and the statement of conservation of mass. A streamline is a line that is tangent to the velocity vector of the flowing fluid. For a 2D planar flow, the stream function, ψ, is defined such that

$$\rho u \equiv \frac{\partial \psi}{\partial y} \rho v \equiv -\frac{\partial \psi}{\partial x} \tag{42.46}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

constant values of stream function defining two streamlines is the mass rate of flow between the streamlines.

The accuracy of the stream function calculation is determined by the text command `/display/set/n-stream-func`.

Stretch Factor

(in the **Premixed Combustion...** category) is a nondimensional parameter that is defined as the probability of unquenched flamelets (G in Equation 8.83 in the Theory Guide).

Subcritical Condition

(in the **Properties...** category) has a value of 1 if the flow condition is subcritical and 0 if the flow condition is supercritical and is available with the Cubic Equation of State Real Gas models.

Subgrid Dissipation Rate

(in the **Turbulence...** category) is the turbulence dissipation rate of the unresolved eddies, eps_{sgs}, only active for the LES and Kinetic Energy Subgrid-Scale Model. It is defined as

$$eps_{sgs} = C_{eps} \cdot \frac{k_{sgs}^{3/2}}{L_s} \tag{42.47}$$

Its unit quantity is **turbulent-energy-diss-rate**.

Subgrid Dynamic Prandtl Number

(in the **Turbulence...** category) is used in the calculation of the subgrid-scale turbulent flux of a scalar φ. See Equation 4.304 in Subgrid-Scale Models in the Theory Guide.

Subgrid Dynamic Sc of Species

(in the **Turbulence...** category) is used in the calculation of the subgrid-scale turbulent flux for Species (see Subgrid Dynamic Prandtl Number).

Subgrid Dynamic Viscosity Const

(in the **Turbulence...** category) is the Smagorinsky model constant as determined by the dynamic procedure described in Dynamic Smagorinsky-Lilly Model in the Theory Guide). Additional information with respect to the Embedded LES (E-LES) model can be found in Postprocessing for Turbulent Flows (p. 2151).

Subgrid Filter Length

(in the **Turbulence...** category) is a mixing length for subgrid scales of the LES turbulence model (defined as L_S in Equation 4.307 in the Theory Guide).

Subgrid Kinetic Energy

(in the **Turbulence...** category) is the turbulence kinetic energy per unit mass of the unresolved eddies, k_{sgs}, calculated using a transport equation, only active for the LES and Kinetic Energy Subgrid-Scale Model. It is defined as

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$k_{sgs}=\frac{1}{2}\left(\overline{u_k^2}-\overline{u}_k^2\right) \tag{42.48}$$

Additional information with respect to the Embedded LES (E-LES) model can be found in Postprocessing for Turbulent Flows (p. 2151). Its unit quantity is **turbulent-kinetic-energy**.

Subgrid Test–Filter Length

(in the **Turbulence...** category) is the test filter width $\hat{\Delta}$ described in Dynamic Smagorinsky-Lilly Model in the Theory Guide). Additional information with respect to the Embedded LES (E-LES) model can be found in Postprocessing for Turbulent Flows (p. 2151).

Subgrid Turbulent Viscosity

(in the **Turbulence...** category) is the turbulent (dynamic) viscosity of the fluid calculated using the LES turbulence model. It expresses the proportionality between the anisotropic part of the subgrid-scale stress tensor and the rate-of-strain tensor. (See Equation 4.298 in the Theory Guide.) Its unit quantity is **viscosity**.

Subgrid Turbulent Viscosity Ratio

(in the **Turbulence...** category) is the ratio of the subgrid turbulent viscosity of the fluid to the laminar viscosity, calculated using the LES turbulence model.

Subtest Kinetic Energy

(in the **Turbulence...** category) is the turbulence kinetic energy of filtered eddies, kf, only active for the LES and Kinetic Energy Subgrid-Scale Model. It is defined as

$$k_f=L_1+L_2+L_3 \tag{42.49}$$

with L_i being the normal components of the Leonard stress.

Its unit quantity is **turbulent-kinetic-energy**

Surface Acoustic Power

(in the **Acoustics...** category) is the **Acoustic Power** per unit area generated by boundary layer turbulence (see Equation 11.37 in the Theory Guide). It is available only when the **Broadband Noise Sources** acoustics model is being used. Its unit quantity is **power** per **area**.

Surface Acoustic Power Level (dB)

(in the **Acoustics...** category) is the **Acoustic Power** per unit area generated by boundary layer turbulence, and represented in dB (see Equation 11.37 in the Theory Guide). It is available only when the **Broadband Noise Sources** acoustics model is being used.

Surface Cluster ID

(in the **Radiation...** category) is used to view the distribution of surface clusters in the domain. Each cluster has a unique integer number (ID) associated with it.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Surface Corrosion Rate of *species-n*

(in the **Species...** category) is the rate of n^{th} solid species consumption/deposition. A positive value indicates corrosion of the solid species on the interface, and a negative value indicates deposition of the solid species.

Surface Coverage of *species-n*

(in the **Species...** category) is the amount of a surface species that is deposited on the substrate at a specific point in time.

Surface Deposition Rate of *species-n*

(in the **Species...** category) is the amount of a surface species that is deposited on the substrate. Its unit quantity is **mass-flux**.

Surface dpdt RMS, definition

(in the **Acoustics...** category) is the RMS value of the time-derivative of static pressure ($\partial p / \partial t$). It is available when the **Ffowcs-Williams & Hawkings** acoustics model is being used.

Surface Heat Transfer Coef.

(in the **Wall Fluxes...** category), as defined in Ansys Fluent, is given by the equation

$$h_{eff} = \frac{q}{T_{wall} - T_{ref}} \tag{42.50}$$

T_{ref} is the reference temperature defined in the Reference Values Task Page (p. 5154). Note that T_{ref} is a constant value that should be representative of the problem. Its unit quantity is the **heat-transfer- coefficient**.

Surface Incident Radiation

(in the **Wall Fluxes...** category) is the net incoming radiation heat flux on a surface. Its unit quantity is **heat-flux**.

Surface Nusselt Number

(in the **Wall Fluxes...** category) is a local nondimensional coefficient of heat transfer defined by the equation

$$Nu = \frac{h_{eff} L_{ref}}{k} \tag{42.51}$$

Surface Stanton Number

(in the **Wall Fluxes...** category) is a nondimensional coefficient of heat transfer defined by the equation

$$St = \frac{h_{eff}}{\rho_{ref} v_{ref} c_p} \tag{42.52}$$

v_{ref} are reference values of density and velocity defined in the Reference Values Task Page (p. 5154), and c_p is the specific heat at constant pressure.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Swirl Pull Velocity

(in the **Solidification/Melting...** category) is the tangential-direction component of the pull velocity for the solid material in a continuous casting process. Its unit quantity is **velocity**.

Swirl Velocity

(in the **Velocity...** category) is the tangential-direction component of the velocity in an axisymmetric swirling flow. See Velocity Reporting Options (p. 4169) for details. The unit quantity for **Swirl Velocity** is **velocity**. For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list.

Swirl-Wall Shear Stress

(in the **Wall Fluxes...** category) is the swirl component of the force per unit area acting tangential to the surface due to friction. Its unit quantity is that of **pressure**.

Tagged Face

(in the **Perforated Walls...** category) shows the tagged faces where the injection or extraction conditions are applied. A value of 1 indicates an injection face. A value of -1 indicates an extraction face.

Tangential Velocity

(in the **Velocity...** category) is the velocity component in the tangential direction. (See Velocity Reporting Options (p. 4169) for details.) The unit quantity for **Tangential Velocity** is **velocity**.

Temperature...

indicates the quantities associated with the thermodynamic temperature of a material.

Thermal Conductivity

(in the **Properties...** category) is a parameter (k) that defines the conduction rate through a material via Fourier's law ($q=-k\nabla T$). A large thermal conductivity is associated with a good heat conductor and a small thermal conductivity with a poor heat conductor (good insulator). Its unit quantity is **thermal-conductivity**. Note that when postprocessing solid material properties, the Thermal Conductivity will be displayed as zero if an anisotropic thermal conductivity model is being used for a solid material.

Thermal Conductivity XX

(in the **Properties...** category) is a parameter (k_{11}) that defines the conduction rate in the $i=1$, $j=1$ components (XX in the global reference frame) of the thermal conductivity of a material via Fourier's law ($q_i=-k_{ij}\frac{\partial T}{\partial x_j}$). The components of thermal conductivity are only available when **anisotropic** or **orthotropic** is selected as the method for specifying **Thermal Conductivity** for one or more materials in the **Create/Edit Materials** dialog box.

Thermal Conductivity XY

(in the **Properties...** category) is a parameter (k_{12}) that defines the conduction rate in the $i=1$, $j=2$ components (XY in the global reference frame) of the thermal conductivity of a material via Fourier's

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

law ($q_i = -k_{ij}\frac{\partial T}{\partial x_j}$). The components of thermal conductivity are only available when **anisotropic** or **orthotropic** is selected as the method for specifying **Thermal Conductivity** for one or more materials in the **Create/Edit Materials** dialog box.

Thermal Conductivity XZ

(in the **Properties...** category) is a parameter (k_{13}) that defines the conduction rate in the i=1, j=3 components (XZ in the global reference frame) of the thermal conductivity of a material via Fourier's law ($q_i = -k_{ij}\frac{\partial T}{\partial x_j}$). The components of thermal conductivity are only available when **anisotropic** or **orthotropic** is selected as the method for specifying **Thermal Conductivity** for one or more materials in the **Create/Edit Materials** dialog box.

Thermal Conductivity YX

(in the **Properties...** category) is a parameter (k_{21}) that defines the conduction rate in the i=2, j=1 components (YX in the global reference frame) of the thermal conductivity of a material via Fourier's law ($q_i = -k_{ij}\frac{\partial T}{\partial x_j}$). The components of thermal conductivity are only available when **anisotropic** is selected as the method for specifying **Thermal Conductivity** for one or more materials in the **Create/Edit Materials** dialog box.

Thermal Conductivity YY

(in the **Properties...** category) is a parameter (k_{22}) that defines the conduction rate in the i=2, j=2 components (YY in the global reference frame) of the thermal conductivity of a material via Fourier's law ($q_i = -k_{ij}\frac{\partial T}{\partial x_j}$). The components of thermal conductivity are only available when **anisotropic** or **orthotropic** is selected as the method for specifying **Thermal Conductivity** for one or more materials in the **Create/Edit Materials** dialog box.

Thermal Conductivity YZ

(in the **Properties...** category) is a parameter (k_{23}) that defines the conduction rate in the i=2, j=3 components (YZ in the global reference frame) of the thermal conductivity of a material via Fourier's law ($q_i = -k_{ij}\frac{\partial T}{\partial x_j}$). The components of thermal conductivity are only available when **anisotropic** or **orthotropic** is selected as the method for specifying **Thermal Conductivity** for one or more materials in the **Create/Edit Materials** dialog box.

Thermal Conductivity ZX

(in the **Properties...** category) is a parameter (k_{31}) that defines the conduction rate in the i=3, j=1 components (ZX in the global reference frame) of the thermal conductivity of a material via Fourier's law ($q_i = -k_{ij}\frac{\partial T}{\partial x_j}$). The components of thermal conductivity are only available when **anisotropic** is selected as the method for specifying **Thermal Conductivity** for one or more materials in the **Create/Edit Materials** dialog box.

Thermal Conductivity ZY

(in the **Properties...** category) is a parameter (k_{32}) that defines the conduction rate in the i=3, j=2 components (ZY in the global reference frame) of the thermal conductivity of a material via Fourier's

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

law ($q_i = -k_{ij}\frac{\partial T}{\partial x_j}$). The components of thermal conductivity are only available when **anisotropic** is selected as the method for specifying **Thermal Conductivity** for one or more materials in the **Create/Edit Materials** dialog box.

Thermal Conductivity ZZ

(in the **Properties...** category) is a parameter (k_{33}) that defines the conduction rate in the i=3, j=3 components (ZZ in the global reference frame) of the thermal conductivity of a material via Fourier's law ($q_i = -k_{ij}\frac{\partial T}{\partial x_j}$). The components of thermal conductivity are only available when **anisotropic** or **orthotropic** is selected as the method for specifying **Thermal Conductivity** for one or more materials in the **Create/Edit Materials** dialog box.

Thermal Diff Coef of *species-n*

(in the **Species...** category) is the thermal diffusion coefficient for the n^{th} species ($D_{T,i}$ in Equation 7.2, Equation 7.4, and Equation 7.8 in the *Fluent Theory Guide*). Its unit quantity is **viscosity**.

Third Invariant

(in the **Sensitivities...** category) This field is available when the **Neural Network Model** has been applied for turbulence model optimization within the **Optimizer Design Variables** dialog box, and is the Third Invariant flow feature.

Time Step

(in the **Residuals...** category) is the local time step size of the cell, Δt, at the current iteration level. Its unit quantity is **time**.

Time Step Scale

(in the **Species...** category) is the factor by which the time step is reduced for the stiff chemistry solver (available in the density-based solver only). The time step is scaled down based on an eigenvalue and positivity analysis.

Total Echem Heat Source

(in the **Potential...** category) is the summation of the Joule and Faradaic heat sources. The total heat source due to electrochemical chemical reactions is added to the energy equation.

Total Energy

(in the **Temperature...** category) is the total energy per unit mass. Its unit quantity is **specific-energy**. For all species models, plots of **Total Energy** include the sensible, chemical and kinetic energies. For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list.

Total Enthalpy

(in the **Temperature...** category) is defined as $H+\frac{1}{2}v^2$ where H is the **Enthalpy**, as defined in Equation 5.8 in the Theory Guide, and v is the velocity magnitude. Its unit quantity is **specific-energy**. For all species models, plots of **Total Enthalpy** consist of the sensible, chemical and kinetic energies. For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Total Enthalpy Deviation

(in the **Temperature...** category) is the difference between **Total Enthalpy** and the reference enthalpy, $H+\frac{1}{2}v^2-H_{ref}$, where H_{ref} is the reference enthalpy defined in the Reference Values Task Page (p. 5154). However, for non-premixed and partially premixed models, **Total Enthalpy Deviation** is the difference between **Total Enthalpy** and total adiabatic enthalpy (total enthalpy where no heat loss or gain occurs). The unit quantity for **Total Enthalpy Deviation** is **specific-energy**. For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list.

Total Pressure

(in the **Pressure...** category) is the pressure at the thermodynamic state that would exist if the fluid were brought to zero velocity and zero potential. For compressible flows, the total pressure is computed using isentropic relationships. For constant c_p, this reduces to:

$$p_0=p\left[1+\frac{\gamma-1}{2}M^2\right]^{\gamma/(\gamma-1)} \tag{42.53}$$

For incompressible flows (constant density fluid), we use Bernoulli's equation, $p_0=p+p_{dyn}$, where p_{dyn} is the local dynamic pressure. Its unit quantity is **pressure**.

Important:

Note that in the postprocessing, the total pressure is presented as gauge pressure, for compressible and incompressible flows. If the total absolute pressure is needed, then add the value of the reference pressure to the total gauge pressure.

Total Surface Corrosion Rate

(in the **Potential...** category) is the total rate of solid species consumption/deposition. A positive value indicates corrosion of the solid species on the interface, and a negative value indicates deposition of the solid species. It's computed as the sum of the surface corrosion rates (**Surface Corrosion Rates of *species-n*** reported under the **Species** category) of all species.

Total Surface Heat Flux

(in the **Wall Fluxes...** category) is the rate of total heat transfer through the control surface. It is calculated by the solver according to the boundary conditions being applied at that surface. By definition, heat flux out of the domain is negative, and heat flux into the domain is positive. The unit quantity for **Total Surface Heat Flux** is **heat-flux**.

Total Temperature

(in the **Temperature...** category) is the temperature at the thermodynamic state that would exist if the fluid were brought to zero velocity. For compressible flows, the total temperature is computed from the total enthalpy using the current c_p method (specified in the Create/Edit Materials Dialog Box (p. 4885)). For the Two-Temperature model, local thermodynamic equilibrium is assumed when computing the total temperature from the total enthalpy. For incompressible flows, the total temperature is equal to the static temperature, unless kinetic energy is explicitly added. For solid zones, the total temperature is equal to the static temperature. The unit quantity for **Total Temperature** is **temperature**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Transfer Current

(in the **Potential...** category) is the volumetric transfer current R_{an} and R_{cat} in Equation 20.5 and Equation 20.6 in the *Fluent Theory Guide*.

Translational-Rotational Energy

(in the **Two-Temperature Model...** category) is the sum of the translational and rotational energy ($e_{t,s}$ and $e_{r,s}$ in Equation 5.38 in the *Fluent Theory Guide*).

Translational-Rotational over Vibrational-Electronic Temperature:

(in the **Two-Temperature Model...** category) is the ratio of translational-rotational temperature over vibrational-electronic temperature.

Translational-Rotational Temperature

(in the **Two-Temperature Model...** category) is T_{tr} in Equation 5.23 in the *Fluent Theory Guide*. It is equivalent to the **Static Temperature** field under the **Temperature...** category.

Transmitted Radiation Flux (Band-n)

(in the **Wall Fluxes...** category) is the amount of radiative heat flux transmitted by a semi-transparent wall for a particular band of radiation. Its unit quantity is **heat-flux**.

Transmitted Visible Solar Flux, Transmitted IR Solar Flux

(in the **Wall Fluxes...** category) is the amount of solar heat flux transmitted by a semi-transparent wall or porous jump boundary for a visible or infrared radiation.

Turbulence...

includes quantities related to turbulence. See Modeling Turbulence (p. 2061) for information about the turbulence models available in Ansys Fluent.

Turbulence Intensity

(in the **Turbulence...** category) is the ratio of the magnitude of the RMS turbulent fluctuations to the reference velocity:

$$I=\frac{\sqrt{\frac{2}{3}k}}{v_{ref}} \tag{42.54}$$

where k is the turbulence kinetic energy and v_{ref} is the reference velocity specified in the Reference Values Task Page (p. 5154). The reference value specified should be the mean velocity magnitude for the flow. Note that turbulence intensity can be defined in different ways, so you may want to use a custom field function for its definition. See Custom Field Functions (p. 4285) for more information.

Turbulent Dissipation Rate (Epsilon)

(in the **Turbulence...** category) is the turbulent dissipation rate. Its unit quantity is **turbulent-energy-diss-rate**. This quantity is available for the k-epsilon and k-omega based turbulence models, where the epsilon/omega relationship is defined as

$$\varepsilon=0.09k\omega \tag{42.55}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list.

Turbulent Flame Speed

(in the **Premixed Combustion...** category) is the turbulent flame speed computed by Ansys Fluent using Equation 8.77 in the Theory Guide. Its unit quantity is **velocity**.

Turbulent Kinetic Energy (k)

(in the **Turbulence...** category) is the turbulence kinetic energy per unit mass defined as

$$k=\frac{1}{2}\overline{u_i'u_i'} \tag{42.56}$$

Turbulent Rate of Reaction-n

(in the **Reactions...** category) is the rate of progress of the n^{th} reaction computed by Equation 7.39 or Equation 7.40 (in the Theory Guide). For the "eddy-dissipation" model, the value is the same as the **Rate of Reaction-n**. For the "finite-rate" model, the value is zero.

Turbulent Reynolds Number (Re_y)

(in the **Turbulence...** category) is a nondimensional quantity defined as

$$\frac{\rho d\sqrt{k}}{\mu_{lam}} \tag{42.57}$$

where k is turbulence kinetic energy, d is the distance to the nearest wall, and μ_{lam} is the laminar viscosity.

Turbulent Reynolds Number (Re_t)

(in the **Turbulence...** category) is a nondimensional quantity defined as

$$\mathrm{Re}_t=\left(\frac{\rho}{\mu}\right)\times\text{turb_velocity_scale}\times\text{turb_length_scale} \tag{42.58}$$

Turbulent Reynolds Number (Scaled)

(in the **Sensitivities...** category) This field is available when the **Neural Network Model** has been applied for turbulence model optimization within the **Optimizer Design Variables** dialog box, and is the scaled turbulent reynolds number flow feature.

Turbulent Viscosity

(in the **Turbulence...** category) is the turbulent viscosity of the fluid computed using the turbulence model. Its unit quantity is **viscosity**. For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list. Additional information with respect to the Embedded LES (E-LES) model can be found in Postprocessing for Turbulent Flows (p. 2151).

Turbulent Viscosity (large-scale)

(in the **Turbulence...** category) is used in the k - kl - ω transition model.See Transport Equations for the k-kl-ω Model in the Theory Guide for more details (Equation 4.151).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Turbulent Viscosity (small-scale)

(in the **Turbulence...** category) is used in the k - kl - ω transition model. See Transport Equations for the k-kl-ω Model in the Theory Guide for more details (Equation 4.145).

Turbulent Viscosity Ratio

(in the **Turbulence...** category) is the ratio of turbulent viscosity to the laminar viscosity. Additional information with respect to the Embedded LES (E-LES) model can be found in Postprocessing for Turbulent Flows (p. 2151).

Turbulent Viscosity Ratio (Scaled)

(in the **Sensitivities...** category) This field is available when the **Neural Network Model** has been applied for turbulence model optimization within the **Optimizer Design Variables** dialog box, and is the scaled turbulent viscosity ratio flow feature.

User Memory <n>

(in the **User Defined Memory...** category) is the value of the quantity in the n^{th} user-defined memory location.

User Node Memory <n>

(in the **User Defined Memory...** category) is the value of the quantity in the n^{th} user-defined node memory location.

Unburnt Fuel Mass Fraction

(in the **Premixed Combustion...** category) is the mass fraction of unburnt fuel. This function is available only for non-adiabatic models.

Unsteady Statistics...

includes mean and root mean square error (RMSE) values of solution variables and custom field functions derived from transient flow calculations.

User Defined Memory...

includes quantities that have been allocated to a user-defined memory location. See the separate Fluent Customization Manual for details about user-defined memory.

User-Defined Scalars...

includes quantities related to user-defined scalars. See the separate Fluent Customization Manual for information about using user-defined scalars.

User Energy Source

(in the **Temperature...** category) is the user-defined energy source term in watts.

User Volumetric Energy Source

(in the **Temperature...** category) is the user-defined energy source term in watts/volume.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UU Reynolds Stress

(in the **Turbulence...** category) is the $\overline{u'^2}$ stress.

UV Reynolds Stress

(in the **Turbulence...** category) is the $\overline{u'v'}$ stress.

Resolved UV Reynolds Stress

(in the **Turbulence...** category) is defined as:

$$\overline{u'v'}(t=T)=\frac{\sum_{t=0}^{T} uv\cdot dt}{\sum_{t=0}^{T} dt}-\frac{\left(\sum_{t=0}^{T} u\cdot dt\right)\left(\sum_{t=0}^{T} v\cdot dt\right)}{\left(\sum_{t=0}^{T} dt\right)^2} \tag{42.59}$$

at time $t=T$. This variable is only available if you enable **Data Sampling for Time Statistics** in the Run Calculation Task Page (p. 5207). Time $t=0$ represents the time **Data Sampling for Time Statistics** is first enabled, which is not necessarily the first timestep.

UW Reynolds Stress

(in the **Turbulence...** category) is the $\overline{u'w'}$ stress.

Resolved UW Reynolds Stress

(in the **Turbulence...** category) is defined as:

$$\overline{u'w'}(t=T)=\frac{\sum_{t=0}^{T} uw\cdot dt}{\sum_{t=0}^{T} dt}-\frac{\left(\sum_{t=0}^{T} u\cdot dt\right)\left(\sum_{t=0}^{T} w\cdot dt\right)}{\left(\sum_{t=0}^{T} dt\right)^2} \tag{42.60}$$

at time $t=T$. This variable is only available if you enable **Data Sampling for Time Statistics** in the Run Calculation Task Page (p. 5207). Time $t=0$ represents the time **Data Sampling for Time Statistics** is first enabled, which is not necessarily the first timestep.

Variance of Species

(in the **NOx...** category) is the variance of the mass fraction of a selected species in the flow field. It is calculated from Equation 9.113 or Equation 9.114 in the *Fluent Theory Guide*.

Variance of Species 1, Variance of Species 2

(in the **NOx...** category) are the variances of the mass fractions of the selected species in the flow field. They are each calculated from Equation 9.113 or Equation 9.114 in the *Fluent Theory Guide*.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Velocity...

includes the quantities associated with the rate of change in position with time. The instantaneous velocity of a particle is defined as the first derivative of the position vector with respect to time, $d\vec{r} / dt$, termed the velocity vector, $\vec{v}$.

Velocity Angle

(in the **Velocity...** category) is defined as follows:

For a 2D model,

$$\tan^{-1}\left(\frac{\text{y-velocity-component}}{\text{x-velocity-component}}\right) \tag{42.61}$$

For a 2D or axisymmetric model,

$$\tan^{-1}\left(\frac{\text{radial-velocity-component}}{\text{axial-velocity-component}}\right) \tag{42.62}$$

For a 3D model,

$$\tan^{-1}\left(\frac{\text{tangential-velocity-component}}{\text{axial-velocity-component}}\right) \tag{42.63}$$

Its unit quantity is **angle**.

Velocity Magnitude

(in the **Velocity...** category) is the speed of the fluid. Its unit quantity is **velocity**. For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list.

Vibrational-Electronic Conductivity

(in the **Two-Temperature Model...** category) is the vibrational-electronic conductivity of the mixture.

Vibrational-Electronic Energy

(in the **Two-Temperature Model...** category) is the sum of the vibrational and electronic energy ($e_{v,s}$ and $e_{e,s}$ in Equation 5.38 in the *Fluent Theory Guide*).

Vibrational-Electronic Specific Heat

(in the **Two-Temperature Model...** category) is the vibrational-electronic specific heat of the mixture.

Vibrational-Electronic Temperature

(in the **Two-Temperature Model...** category) is T_{ve} in Equation 5.23 in the *Fluent Theory Guide*.

Volume fraction

(in the **Phases...** category) is the volume fraction of the selected phase in the **Phase** drop-down list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Volumetric Absorbed Radiation

(in the **Radiation...** category) is the spherical mean absorbed radiation and, for a single spectral band, can be defined as:

$$4\pi I\alpha \tag{42.64}$$

where I is the radiation intensity and α is the absorption coefficient.

Volumetric Emitted Radiation

(in the **Radiation...** category) is the spherical mean emitted radiation and, for a single spectral band, can be defined as.

$$4\varepsilon\sigma T^4\alpha \tag{42.65}$$

where ε is the blackbody fraction, σ is the Stefan-Boltzmann constant, T is the temperature, and α is the absorption coefficient.

Vorticity Magnitude

(in the **Velocity...** category) is the magnitude of the vorticity vector. Vorticity is a measure of the rotation of a fluid element as it moves in the flow field, and is defined as the curl of the velocity vector:

$$\xi = \nabla \times \vec{V} \tag{42.66}$$

VV Reynolds Stress

(in the **Turbulence...** category) is the $\overline{v'^2}$ stress.

VW Reynolds Stress

(in the **Turbulence...** category) is the $\overline{v'w'}$ stress.

Resolved VW Reynolds Stress

(in the **Turbulence...** category) is defined as:

$$\overline{v'w'}(t=T) = \frac{\sum_{t=0}^{T} vw \cdot dt}{\sum_{t=0}^{T} dt} - \frac{\left(\sum_{t=0}^{T} v \cdot dt\right)\left(\sum_{t=0}^{T} w \cdot dt\right)}{\left(\sum_{t=0}^{T} dt\right)^2} \tag{42.67}$$

at time $t=T$. This variable is only available if you enable **Data Sampling for Time Statistics** in the Run Calculation Task Page (p. 5207). Time $t=0$ represents the time **Data Sampling for Time Statistics** is first enabled, which is not necessarily the first timestep.

Wall Adjacent Heat Transfer Coef.

(in the **Wall Fluxes...** category) is defined as:

$$h = \frac{q}{(T_w - T_{ac})} \tag{42.68}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where q is the surface heat flux, T_w is the wall face temperature, and T_{ac} is the wall adjacent cell temperature.

Wall Adjacent Temperature

(in the **Temperature...** category) is the temperature of the fluid cell adjacent to the wall.

Wall Coverage

(in the **Wall Film...** category) is the fraction of the wall surface covered by the wall film.

Wall Deposition Thickness

(in the **Wall Fluxes...** category) is the thickness of the solid species deposition on the wall. This item is available only for transient species transport cases that involve wall surface reactions and is calculated as:

$$h = h_0 + \int \frac{\dot{m}}{\rho} \Delta t$$

where

h_0 = value specified for **Initial Deposition Thickness** in the **Wall** dialog box (**Species** tab).

$\dot{m}$ = deposition rate of the surface species on the wall

ρ = value specified for **Solid Species Density** in the **Wall** dialog box (**Species** tab).

Δt = time step

Wall Film Face Pressure

(in the **Wall Film...** category) is the face pressure of the wall film.

Wall Film Height

(in the **Wall Film...** category) is the height of the wall film.

Wall Film Heat Flux

(in the **Wall Film...** category) is the heat flux of the wall film to the wall.

Wall Film Heat Transfer Coefficient

(in the **Wall Film...** category) is the film to wall heat transfer coefficient of the wall film.

Wall Film Impingement Mass Flux

(in the **Wall Film...** category) is the mass flux impinging on the wall film.

Wall Film Mass

(in the **Wall Film...** category) is the mass of the wall film (in units of mass).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Wall Film Surface Temperature

(in the **Wall Film…** category) is the surface temperature of the wall film.

Wall Film Temperature

(in the **Wall Film…** category) is the temperature of the wall film.

Wall Film Velocity Magnitude

(in the **Wall Film…** category) is the velocity magnitude of the wall film.

Wall Film X-Velocity

(in the **Wall Film…** category) is the x-component of the velocity of the wall film.

Wall Film Y-Velocity

(in the **Wall Film…** category) is the y-component of the velocity of the wall film.

Wall Film Z-Velocity

(in the **Wall Film…** category) is the z-component of the velocity of the wall film.

Wall Fluxes...

includes quantities related to forces and heat transfer at wall surfaces.

Wall Func. Heat Tran. Coef.

is generally defined by the equation

$$h_{eff} = \frac{\rho c_p u^*}{T_c^*} - \frac{D/\left(T_w - T_p\right)}{T_c^*} \tag{42.69}$$

where c_p is the specific heat, u^* is the near-wall turbulence velocity scale, D is the viscous heating (included only for variable density flows, Equation 4.345 in the *Fluent Theory Guide*), T_c^* is the dimensionless law-of-the-wall temperature defined for the standard wall functions in Equation 4.344 in the *Fluent Theory Guide*. For the enhanced wall treatment, the combined forms for D and T_c^* (Equation 4.344 and Equation 4.345 in the *Fluent Theory Guide*) are replaced by the corresponding functions obtained using the blending of Kader, explained in Enhanced Wall Treatment for Momentum and Energy Equations in the *Fluent Theory Guide*.

Wall Irradiation Flux.Normalized Std Deviation

(in the **Radiation...** category) is the statistical standard deviation for the hemispherical mean irradiation flux.

Wall Leidenfrost Temperature

(in the **Wall Film…** category) is the wall temperature at which the insulating vapor layer fully covers the hot wall preventing the liquid from boiling rapidly.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Wall Power Density

(in the **Aeromechanics...** category) is the power performed on the wall by the fluid per unit of area. It is calculated as:

$$wall\ power\ density = pressure \times \left(\frac{\Delta\vec{X}}{\Delta t} \cdot \vec{A}\right)[W/m^2] \tag{42.70}$$

where $\frac{\Delta\vec{X}}{\Delta t}$ is the mesh velocity and $\vec{A}$ is the area unit vector.

Wall Shear Stress

(in the **Wall Fluxes...** category) is the magnitude of the wall shear stress vector, that is, the force per unit area acting tangential to the surface due to friction. Its unit quantity is that of **pressure**. For cases that are not axisymmetric, you can view the components of the vector through the **X-Wall Shear Stress**, **Y-Wall Shear Stress** and (for 3D cases) **Z-Wall Shear Stress** field variables; for 2D axisymmetric cases, you can view the components of the vector through the **Axial-Wall Shear Stress**, **Radial-Wall Shear Stress**, and (for swirling flows) **Swirl-Wall Shear Stress** field variables. When **Wall Shear Stress** is used to visualize a multiphase flow simulation result, its value corresponds to the selected phase in the **Phase** drop-down list. For more information, see Shear-Stress Calculation Procedure at Wall Boundaries (p. 1480).

Wall Superheat

(in the **Phase Interaction...** category) is the wall superheat of the boiling model defined as:

$$T_{sup} = T_w - T_{sat}$$

where T_w and T_{sat} are the wall and saturation and liquid temperatures.

This item is available for the Eulerian multiphase boiling model only.

Wall Temperature

(in the **Temperature...** category) is the temperature on a surface (including wall surfaces, shadow wall surfaces, and shell surfaces). For an illustration of such surfaces for thin walls and shells, see Figure 7.56: A Thin Wall (p. 1471) and Figure 16.8: A Boundary Wall with Shell Conduction (p. 2183), respectively.

Wall Temperature (Thin)

(in the **Temperature...** category) is for thin walls only, and reports the temperature on the surface that is separated from the fluid / solid cells by the wall thickness, as shown in Figure 7.56: A Thin Wall (p. 1471). Note that the wall thermal boundary conditions are applied on this surface.

Wall Work Density

(in the **Aeromechanics...** category) is the work performed on the wall by the fluid per unit of area. It is calculated as:

$$wall\ work\ density = pressure \times \left(\Delta\vec{X} \cdot \vec{A}\right)[J/m^2] \tag{42.71}$$

where $\Delta\vec{X} = \vec{X} - \vec{X}_{N-1}$ is the relative mesh displacement and $\vec{A}$ is the area unit vector.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Wall Yplus

(in the **Turbulence...** category) is a nondimensional parameter defined by the equation

$$y^{+}=\frac{\rho u_{\tau} y_{P}}{\mu} \tag{42.72}$$

where $u_{\tau}=\sqrt{\tau_{w}/\rho_{w}}$ is the friction velocity, y_{P} is the distance from point P to the wall and is adopted for certain two-equation turbulence models in combination with the Standard Roughness Model, ρ is the fluid density, and μ is the fluid viscosity at point P. See Near-Wall Treatments for Wall-Bounded Turbulent Flows in the *Fluent Theory Guide* and Wall Roughness Effects in Turbulent Wall-Bounded Flows in the *Fluent Theory Guide* for details. For multiphase models, this value corresponds to the selected phase in the **Phase** drop-down list.

Wall Ystar

(in the **Turbulence...** category) is a nondimensional parameter defined by the equation

$$y^{*}=\frac{\rho C_{\mu}^{1/4} k_{P}^{1/2} y_{P}}{\mu} \tag{42.73}$$

where k_{P} is the turbulence kinetic energy at point P, y_{P} is the distance from point P to the wall and is adopted for certain two-equation turbulence models in combination with the Standard Roughness Model, ρ is the fluid density, and μ is the fluid viscosity at point P. See Near-Wall Treatments for Wall-Bounded Turbulent Flows in the *Fluent Theory Guide* and Wall Roughness Effects in Turbulent Wall-Bounded Flows in the *Fluent Theory Guide* for details.

Water Content

(in the **Potential...** category) is the water content λ used in Equation 18.21 in the *Fluent Theory Guide*. This item is available only for the Electrolyses and H2 Pump model when the **Water Content** option is enabled in the **Potential/Electrochemistry** dialog box (**Model** tab).

Water Phase Change

(in the **Potential...** category) is the is the rate of the mass change between water vapor and liquid water, S_{gl} in Equation 18.17 in the *Fluent Theory Guide*. This item is available only for the Electrolyses and H2 Pump model when the **Phase Change** option is enabled in the **Potential/Electrochemistry** dialog box (**Model** tab).

WW Reynolds Stress

(in the **Turbulence...** category) is the $\overline{w'^{2}}$ stress.

X-Coordinate, Y-Coordinate, Z-Coordinate

(in the **Mesh...** category) are the Cartesian coordinates in the x -axis, y -axis, and z -axis directions respectively. The unit quantity for these variables is **length**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

X Current, Y Current, Z Current

(in the **Potential...** category) are the x, y, and z components of electric current ($\sigma\frac{\partial\phi}{\partial x}$, $\sigma\frac{\partial\phi}{\partial y}$, and $\sigma\frac{\partial\phi}{\partial z}$), respectively.

X Displacement, Y Displacement, Z Displacement

(in the **Structure...** category) are the components of the displacement. These variables are intended only for solid zones and/or their adjacent walls in intrinsic fluid-structure interaction (FSI) simulations.

X Face Area, Y Face Area, Z Face Area

(in the **Mesh...** category) are the components of the face area vector for noninternal faces (that is, faces that only have `c0` and no `c1`). The values are stored on the face itself and used when required. These variables are intended only for zone surfaces and not for other surfaces created for postprocessing.

X Imaginary Periodic Displacement, Y Imaginary Periodic Displacement, Z Periodic Displacement

(in the **Mesh...** category) are the interpolated imaginary mode shape profile components.

X Lithium Mass Flux, Y Lithium Mass Flux, Z Lithium Mass Flux

(in the **Lithium...** category) are the x, y, and z components of the mass flux vector of the lithium-ion species ($\vec{N}$ in Equation 18.4 in the *Fluent Theory Guide*). The unit quantity is kmol/m^2 s^{-1}.

X Periodic Displacement, Y Periodic Displacement, Z Periodic Displacement

(in the **Mesh...** category) are the interpolated real mode shape profile components.

X Pull Velocity, Y Pull Velocity, Z Pull Velocity

(in the **Solidification/Melting...** category) are the x, y, and z components of the pull velocity for the solid material in a continuous casting process. The unit quantity for each is **velocity**.

X Relative Mesh Displacement, Y Relative Mesh Displacement, Z Relative Mesh Displacement

(in the **Mesh...** category) are components for the difference between the current time step and the previous time step mesh node position.

X Total Mesh Displacement, Y Total Mesh Displacement, Z Total Mesh Displacement

(in the **Mesh...** category) are components for the difference between the current time step and the reference (non-deformed) mesh node position.

X Velocity, Y Velocity, Z Velocity

(in the **Velocity...** category) are the components of the velocity vector in the x -axis, y -axis, and z -axis directions, respectively. The unit quantity for these variables is **velocity**. For multiphase models, these values correspond to the selected phase in the **Phase** drop-down list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

X-Vorticity, Y-Vorticity, Z-Vorticity

(in the **Velocity...** category) are the x, y, and z components of the vorticity vector.

X-Wall Shear Stress, Y-Wall Shear Stress, Z-Wall Shear Stress

(in the **Wall Fluxes...** category) are the x, y, and z components of the force per unit area acting tangentially to the surface due to friction. The unit quantity for these variables is that of **pressure**. For multiphase models, these values correspond to the selected phase in the **Phase** drop-down list.

Yplus Based Heat Transfer Coefficient

(in the **Wall Fluxes...** category) is based on the temperature of the fluid, which is estimated at a user-specified y^+ value, from the Reference Values Task Page (p. 5154):

$$h_{y^+} = \frac{q}{T_{wall} - T_{y^+ \text{for HTC}}} \tag{42.74}$$

where q is the surface heat flux, T_{wall} is the wall face temperature, and $T_{y^+ \text{for HTC}}$ is the **Yplus Based Heat Transfer Reference Temperature**.

Like the pair of field variables **Wall Adjacent Heat Transfer Coefficient** and **Wall Adjacent Temperature**, the**Yplus Based Heat Transfer Coefficient** h_{y^+} and its reference temperature$T_{y^+ \text{for HTC}}$ are both local, hence more suitable for complex flows than **Surface Heat Transfer Coefficient** with its single global T_{ref}, Equation 42.50 (p. 4269). Different from the **Wall Adjacent Heat Transfer Coefficient**, h_{y^+} is approximately mesh-insensitive due to the reconstruction of the reference temperature $T_{y^+ \text{for HTC}}$ at the user-specified mesh-independent y^+ value

Yplus Based Heat Transfer Reference Temperature

(in the **Temperature...** category) is the temperature of fluid at the y^+ value, which is specified by a user as **Yplus for Heat Tran. Coef.** in the Reference Values Task Page (p. 5154). $T_{y^+ \text{for HTC}}$ is calculated from the logarithmic temperature profile $T^+(y^+)$ at a distance from the wall corresponding to the specified y^+ value:

$$T_{y^+ \text{for HTC}} = T_{wall} - \frac{q}{\rho C_p u^*} T^+(y^+ \text{for HTC}) \tag{42.75}$$

were q is the surface heat flux, u^* is the wall velocity scale of turbulence (see Equation 4.343 in the Theory Guide), fluid density ρ and specific heat capacity C_p are taken from the wall-adjacent cell. To approximate the local external fluid temperature by $T_{y^+ \text{for HTC}}$, you should specify the y^+ value possibly close to its maximum in the logarithmic law range for a simulated flow.

42.5. Custom Field Functions

In addition to the basic field variables provided by Ansys Fluent (and described in Alphabetical Listing of Field Variables and Their Definitions (p. 4205)), you can also define your own field functions to be used in conjunction with any of the commands that use these variables (contour and vector display, XY plots, and so on). This capability is available with the Custom Field Function Calculator Dialog Box (p. 5428). You can use the default field variables, previously defined calculator functions, and calculator operators to create new functions. (Several sample functions are described in Sample Custom Field Functions (p. 4290).)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Any field functions that you define will be saved in the case file the next time that you save it. You can also save your custom field functions to a separate file (as described in Manipulating, Saving, and Loading Custom Field Functions (p. 4289)), so that they can be used with a different case file.

Important:

- Note that all custom field functions are evaluated and stored in SI units.

  Any solver-defined flow variables that you use in your field-function definition will be automatically converted if they are not already in SI units, but you must be careful to enter constants in the appropriate units. Note also that explicit node values are not available for custom field functions; all node values for these functions will be computed by averaging the values in the surrounding cells, as described in Node Values (p. 4168).

- Use expressions (Fluent Expressions Language (p. 1037)) to evaluate any boundary or face value quantities (indicated with an *fv* in Field Variables Listed by Category (p. 4171)). Custom field functions do not compute face values directly, rather they compute cell values as described in Cell Values (p. 4167), then use the cell values to interpolate back to the surfaces. This could lead to an incorrect value, where, for example, heat flux shows a non-zero value on a symmetry boundary.

- Custom field functions interpolate cell values to nodes prior to performing the computations of the provided definition. This approach differs from Expressions (Fluent Expressions Language (p. 1037)), which compute according to the provided expression definition, then interpolate the cell values to the nodes. The observable difference in these two approaches appears in cases where there are steep gradients in cell values.

For additional information, see the following sections:

42.5.1. Creating a Custom Field Function

42.5.2. Manipulating, Saving, and Loading Custom Field Functions

42.5.3. Sample Custom Field Functions

42.5.1. Creating a Custom Field Function

To create your own field function, you will use the Custom Field Function Calculator Dialog Box (p. 5428) (Figure 42.3: The Custom Field Function Calculator Dialog Box (p. 4287)). This dialog box allows you to define field functions based on existing functions, using simple calculator operators. Any functions that you define will be added to the list of default flow variables and other field functions provided by the solver.

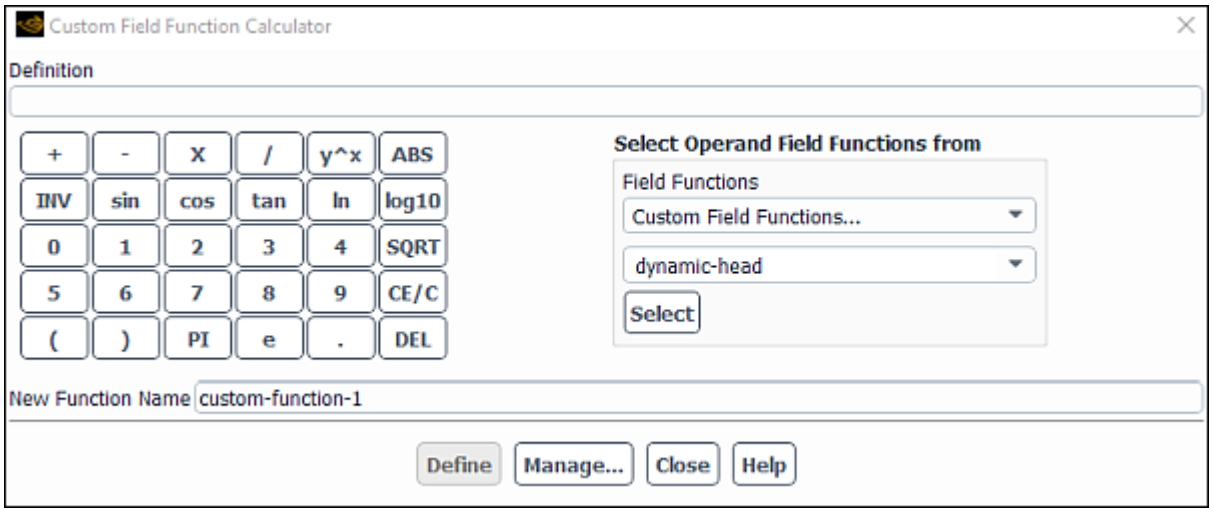
Parameters & Customization → **Custom Field Functions** **New...**

Important:

Recall that you must enter all constants in the function definition in SI units.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 42.3: The Custom Field Function Calculator Dialog Box



The steps for creating a custom field function are as follows:

1. Use the calculator buttons and the **Field Functions** list and **Select** button to specify the function definition, as described below. (As you select each item from the **Field Functions** list or click a button in the calculator keypad, its symbol will appear in the **Definition** text entry box. You *cannot* edit the contents of this box directly; if you want to delete part of a function, use the **DEL** button on the keypad.)

   Important:

   The range of integers and real numbers that can be stored is as follows:

   $-2147483648 > integers < 2147483647$

   $-1.79769e+308 > real < 1.79769e+308$

   Note that using a number less than 1e-39 may produce inaccurate results, while values less than 1e-45 will produce a result of zero.

2. Specify the name of the function in the **New Function Name** field.

   Important:

   Be sure that you do not specify a name that is already used for a standard field function (for example, `velocity-magnitude`); you can see a complete list of the predefined field functions in Ansys Fluent by selecting the `display/contours` text command and viewing the available choices for `contours of`.

3. Click the **Define** button.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When you click **Define**, the solver will create the function and add it to the list of **Custom Field Functions** within the drop-down list of available field functions. The **Define** push button is grayed out after you create a new function or if the **Definition** text entry box is empty.

Should you decide to rename or delete the function after you have completed the definition, you can do so in the Field Function Definitions Dialog Box (p. 5476), which you can open by clicking on the **Manage...** push button. See Manipulating, Saving, and Loading Custom Field Functions (p. 4289) for details.

42.5.1.1. Using the Calculator Buttons

Your function definition can include many basic calculator operations (for example, addition, subtraction, multiplication, square root). When you select a calculator button (by clicking on it), the appropriate symbol will appear in the **Definition** text entry box. The meaning of the buttons is straightforward; they are similar to the buttons you would find on any standard calculator. You should, however, note the following:

- The **CE/C** button will clear the entire **Definition** and the **New Function Name**, if you have entered one. The **DEL** button will delete only the last entry in the **Definition** text entry box. You can use **DEL** to delete characters one at a time, starting with the last one entered.
- To obtain the inverse trigonometric functions arcsin, arccos, and arctan, click the **INV** button before selecting **sin**, **cos**, or **tan**.
- The **ABS** button yields the absolute value of the number that follows it. Likewise, the **ln** button yields the natural logarithm of the number that follows it, and the **log10** button yields the base 10 logarithm function of the number that follows it.

  Important:

  log10 and **ln** will be calculated for values greater than 0. For values less than or equal to 0, the resultant value will be zero.

- The **PI** button represents π and the **e** button represents the base of the natural logarithm system (which is approximately equal to 2.71828).

42.5.1.2. Using the Field Functions List

Your function definition can include any of the field functions defined by the solver (and listed in Alphabetical Listing of Field Variables and Their Definitions (p. 4205)) or by you. You can also use any report definitions that you have created (see Monitoring and Reporting Solution Data (p. 4098) for additional information). To include one of these variables/functions in your function definition, select it in the **Field Functions** drop-down list and then click the **Select** button below the list. The symbol

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

for the selected item will appear in the **Definition** text entry box (for example, **p** will appear if you select **Static Pressure**).

Note:

If a custom field function definition uses a report definition, then it cannot be exported to a file.

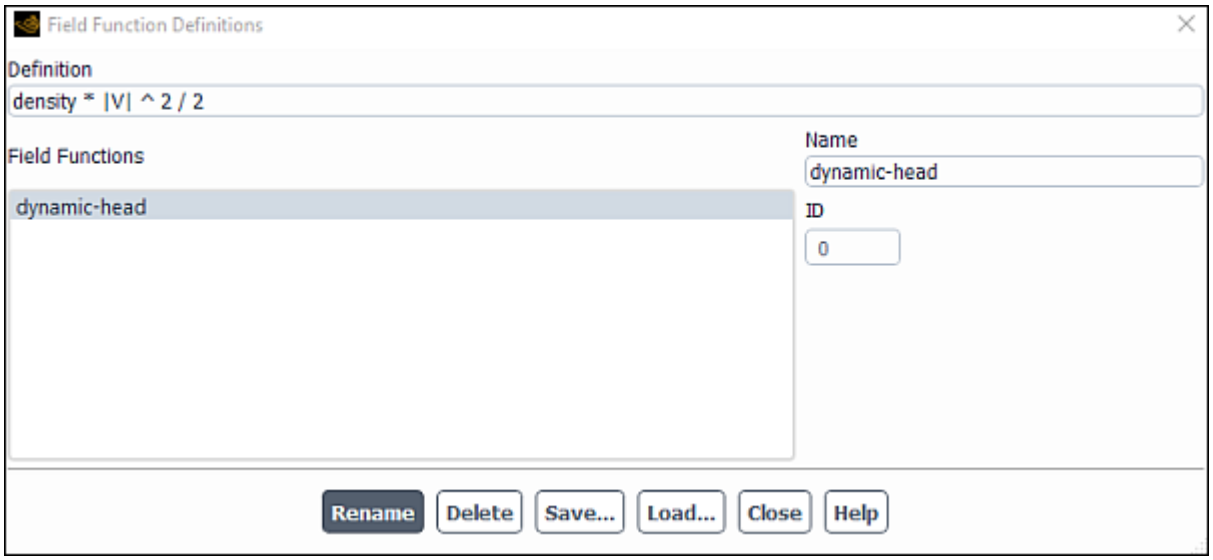
42.5.2. Manipulating, Saving, and Loading Custom Field Functions

Once you have defined your field functions, you can manipulate them using the Field Function Definitions Dialog Box (p. 5476) (Figure 42.4: The Field Function Definitions Dialog Box (p. 4289)). You can display a function definition to be sure that it is correct, delete the function if you decide that it is incorrect and must be redefined, or give the function a new name. You can also save custom field functions to a file or read them from a file. The custom field function file allows you to transfer your custom functions between case files.

To open the Field Function Definitions Dialog Box (p. 5476), right-click **Custom Field Functions** (under the **Parameters & Customization** branch of the tree) and select **Manage...**. You can also click the **Manage...** button in the Custom Field Function Calculator Dialog Box (p. 5428).

To open the Field Function Definitions Dialog Box (p. 5476), right-click **Custom Field Functions** (under the **Parameters & Customization** branch of the tree) and select **Manage...**. You can also click the **Manage...** button in the Custom Field Function Calculator Dialog Box (p. 5428).

Figure 42.4: The Field Function Definitions Dialog Box



The following actions can be performed in the Field Function Definitions Dialog Box (p. 5476):

- To check the definition of a function, select it in the **Field Functions** list. Its definition will be displayed in the **Definition** field. This display is for informational purposes only; you cannot edit it. If you want to change a function definition, you must delete the function and define it again in the Custom Field Function Calculator Dialog Box (p. 5428).
- To delete a function, select it in the **Field Functions** list and click the **Delete** button.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- To rename a function, select it in the **Field Functions** list, enter a new name in the **Name** field, and click the **Rename** button.

  Important:

  Be sure that you do not specify a name that is already used for a standard field function (for example, `velocity-magnitude`); you can see a complete list of the predefined field functions in Ansys Fluent by selecting the `display/contours` text command and viewing the available choices for `contours of`.

- To save all of the functions in the **Field Functions** list to a file, click the **Save...** button and specify the file name in The Select File Dialog Box (p. 931).

- To read custom field functions from a file that you saved as described above, click the **Load...** button and specify the file name in the resulting **Select File** dialog box. (Custom field function files are valid Scheme functions, and can also be loaded with the **File/Read/Scheme...** ribbon tab item, as described in Reading Scheme Source Files (p. 972).)

42.5.3. Sample Custom Field Functions

When you are checking the results of your simulation, you may find it useful to define some of the following field functions:

- To define a function that determines the ratio of static pressure to inlet total pressure, use the relationship

$$R=\frac{p+p_{op}}{p_{to}+p_{op}} \tag{42.76}$$

  where p is the static pressure calculated by the solver, p_{to} is the inlet total pressure, and p_{op} is the operating pressure for the problem. Use the solver-defined function **Static Pressure** for p, and the numerical value that you specified for **Gauge Total Pressure** in the Pressure Inlet Dialog Box (p. 5051) for p_{to}. Specify the value of the operating pressure to be the value that you set in the Operating Conditions Dialog Box (p. 4978). As discussed in Operating Pressure (p. 1677), all pressures in Ansys Fluent are gauge pressures relative to the operating pressure. If the operating pressure is zero, as is generally the case for compressible flow calculations, the expression for the pressure ratio reduces to

$$PR=\frac{p}{p_{to}} \tag{42.77}$$

- To define a function that determines the critical velocity ratio v/a_*, a parameter that is sometimes used in turbomachinery calculations, use the relationship

$$\frac{v}{a_*}=\left[\left(\frac{\gamma+1}{\gamma-1}\right)\left(1-PR^{(\gamma-1)/\gamma}\right)\right]^{1/2} \tag{42.78}$$

  In this relationship, a_* is the critical velocity (that is, the velocity that would occur for the same stagnation conditions if $M=1$), γ is the ratio of specific heats, and PR is the pressure ratio defined in Equation 42.77 (p. 4290) for which you created your own function. For γ, ratio of specific heats, select **Specific Heat Ratio (gamma)** in the **Properties...** category. To include PR, select **Custom**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Field Functions... in the first drop-down list under **Field Functions**, and then select from the second list the function name that you assigned PR.

- Suppose you have swirling flow in a pipe, aligned with the z axis, and you want to calculate the flow rate of angular momentum through a cross-sectional plane:

$$\int \rho r v_\theta \vec{v} \cdot d\vec{A} \tag{42.79}$$

You can create a function for the product rv_θ, where r is the **Radial Coordinate** and v_θ is the **Tangential Velocity**. Then use the Surface Integrals Dialog Box (p. 5325) to compute the flow rate of this quantity.

Important:

The custom field function containing model dependent functions (like temperature when the energy equation is enabled) will be computed only when those models are still active.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 43: Parallel Processing

The following sections describe the parallel-processing features of Ansys Fluent.

43.1. Introduction to Parallel Processing

43.2. Starting Parallel Ansys Fluent Using Fluent Launcher

43.3. Starting Parallel Ansys Fluent on a Windows System

43.4. Starting Parallel Ansys Fluent on a Linux System

43.5. Mesh Partitioning and Load Balancing

43.6. Using General Purpose Graphics Processing Units (GPGPUs) With the Algebraic Multigrid (AMG) Solver

43.7. Controlling the Threads

43.8. Checking Network Connectivity

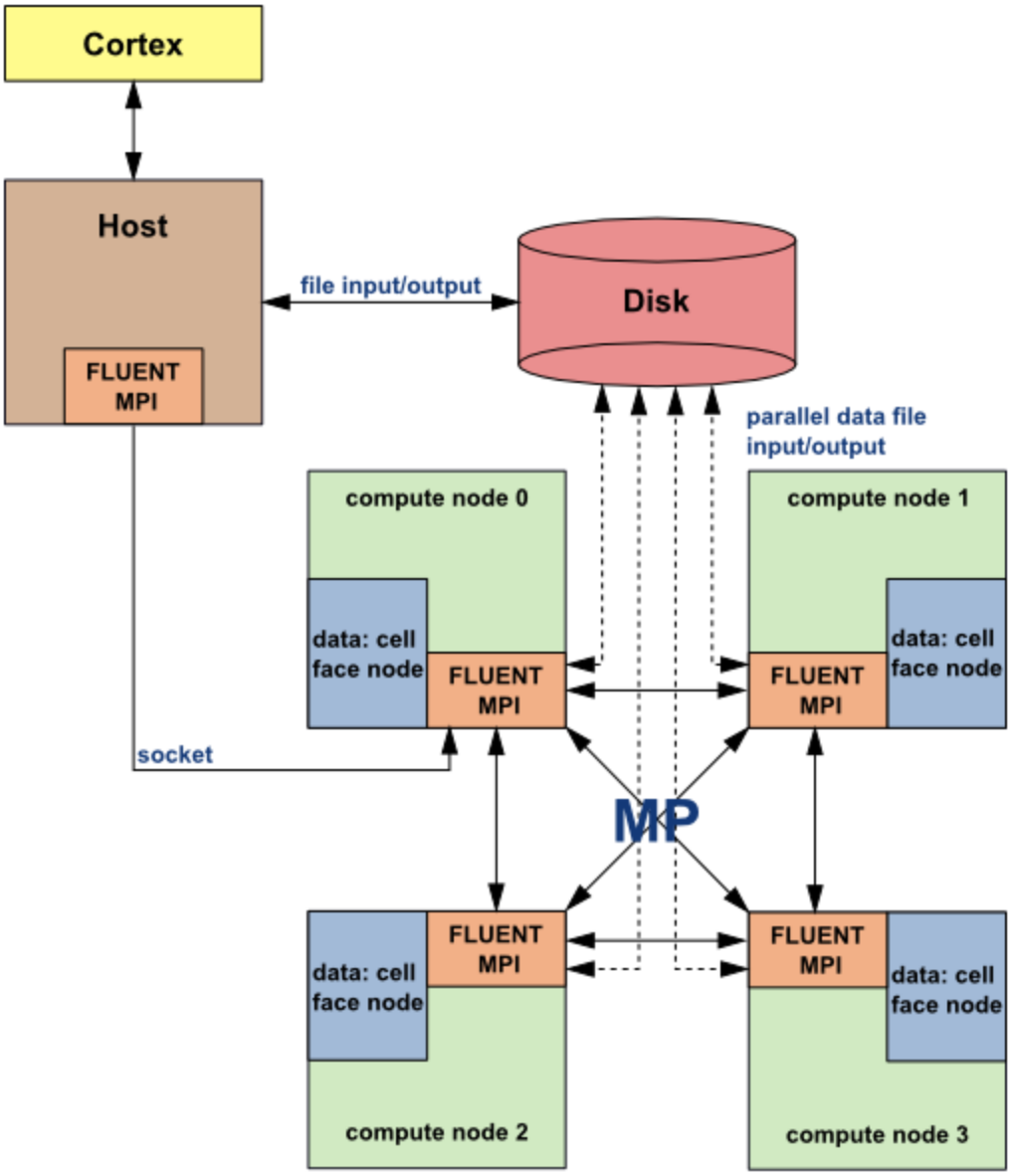
43.9. Checking and Improving Parallel Performance

43.1. Introduction to Parallel Processing

Processing in Ansys Fluent involves an interaction between Ansys Fluent, a host process, and one or more compute-node processes. Ansys Fluent interacts with the host process and the compute node(s) using a utility called `cortex` that manages Ansys Fluent's user interface and basic graphical functions.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 43.1: Ansys Fluent Architecture



For serial processing, the Ansys Fluent solver uses only a single compute node. For parallel processing, a solution is computed using multiple compute nodes that may be executing on the same computer, or on different computers in a network (Figure 43.1: Ansys Fluent Architecture (p. 4294)).

Parallel Ansys Fluent splits up the mesh and data into multiple partitions, then assigns each mesh partition to a different compute node. The number of partitions is equal to or less than the number of processors (or cores) available on your compute cluster. The compute-node processes can be executed on a massively parallel computer, a multiple-CPU workstation, or a network cluster of computers.

Generally, as the number of compute nodes increases, turnaround time for solutions will decrease. This is referred to as solver "scalability." However, beyond a certain point, the ratio of network communication

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

to computation increases, leading to reduced parallel efficiency, so optimal system sizing is important for simulations.

Ansys Fluent uses a host process that does not store any mesh or solution data. Instead, the host process only interprets commands from Ansys Fluent's graphics-related interface, `cortex`.

The host distributes those commands to the other compute nodes via a socket interconnect to a single designated compute node called `compute-node-0`. This specialized compute node distributes the host commands to any additional compute nodes. Each compute node simultaneously executes the same program on its own data set. Communication from the compute nodes to the host is possible only through `compute-node-0` and only when all compute nodes have synchronized with each other.

Each compute node is virtually connected to every other compute node, and relies on inter-process communication to perform such functions as sending and receiving arrays, synchronizing, and performing global operations (such as summations over all cells). Inter-process communication is managed by a message-passing library. For example, the message-passing library could be a vendor implementation of the Message Passing Interface (MPI) standard, as depicted in Figure 43.1: Ansys Fluent Architecture (p. 4294).

All of the parallel Ansys Fluent processes (as well as the serial process) are identified by a unique integer ID. The host collects messages from `compute-node-0` and performs operations (such as printing, displaying messages, and writing to a file) on all of the data.

For additional information, see the following section:

43.1.1. Recommended Usage of Parallel Ansys Fluent

43.1.1. Recommended Usage of Parallel Ansys Fluent

The recommended procedure for using parallel Ansys Fluent is as follows:

1. Select your case file by clicking **Case** under **Get Started With...** in the Fluent Launcher.
2. Specify the number of parallel processors and start Ansys Fluent. For details, see Starting Parallel Ansys Fluent on a Windows System (p. 4306) and Starting Parallel Ansys Fluent on a Linux System (p. 4311).
3. Review the partitions and perform partitioning again, if necessary. See Checking the Partitions (p. 4338) for details on checking your partitions. Note that there are other approaches for partitioning, including manual partitioning in either the serial or the parallel solver. For details, see Mesh Partitioning and Load Balancing (p. 4315).
4. Calculate a solution. See Checking and Improving Parallel Performance (p. 4346) for information on checking and improving the parallel performance.

Note:

Due to limitations imposed by several MPI implementations, Ansys Fluent performance on heterogeneous clusters involving either operating system or processor family differences may not be optimal, and in certain cases cause failures. You are urged to use caution in such parallel operating environments.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

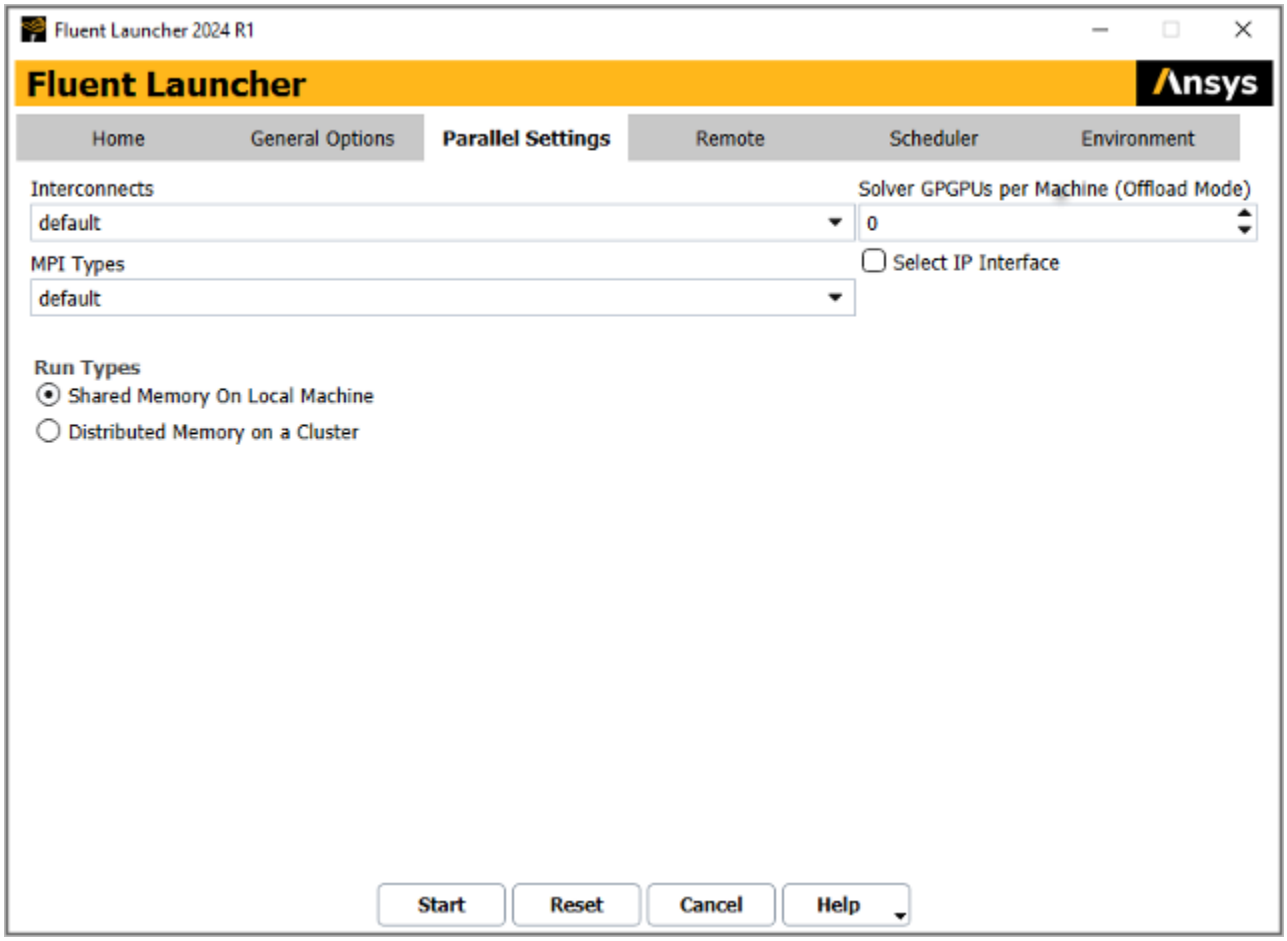
43.2. Starting Parallel Ansys Fluent Using Fluent Launcher

Whether you start Ansys Fluent either from the Linux or Windows command line with no arguments, from the Windows Programs menu, or from the Windows desktop, Fluent Launcher will appear (for details, see Starting Ansys Fluent Using Fluent Launcher (p. 180) in the Getting Started (p. 1) part of this manual), where you can select the mesh or case file that you would like to start with, as well as other options (for example, whether you want a single-precision or double-precision calculation).

Parallel calculation options can be set up by specifying a number of **Solver Processes** greater than 1 under **Parallel (Local Machine)** in the Fluent Launcher.

The **Parallel Settings** tab allows you to specify settings for running Ansys Fluent in parallel.

Figure 43.2: The Parallel Settings Tab of Fluent Launcher



- If your machines are equipped with appropriate General Purpose Graphical Processing Units (GPGPUs) you can indicate that these should be used in offload mode for Algebraic Multigrid (AMG) solver acceleration and/or Discrete Ordinates (DO) solver acceleration by setting the **Solver GPGPUs per Machine (Offload Mode)**. Note that the number of solver processes per machine must be the same for all machines and that the number of processes per machine must be evenly divisible into the value you specify for **Solver GPGPUs per Machine (Offload Mode)**. That is, for *nprocs* solver processes running on *M* machines using *ngpgpus* GPGPUS per machine:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$nprocs \bmod M = 0$$
$$\left(\frac{nprocs}{M}\right) \bmod ngpgpus = 0$$

Table 43.1: Examples for GPGPUs per Machine (p. 4297) presents several examples illustrating the relationship between number of machines, number of solver processes, and GPGPUs per machine.

Table 43.1: Examples for GPGPUs per Machine

	Example 1	Example 2	Example 3
Number of Machines (`M`)	1	4	4
Number of Solver Processes (`nprocs`)	4	12	22
Valid values for GPGPUs per Machine (`ngpgpus`)	1, 2, 4	1, 3	`ngpgpus` will be ignored and GPGPU acceleration will be disabled (`M` does not evenly divide `nprocs`)

See Using General Purpose Graphics Processing Units (GPGPUs) With the Algebraic Multigrid (AMG) Solver (p. 4343) and/or Accelerating Discrete Ordinates (DO) Radiation Calculations (p. 4352) for more information about using GPGPU acceleration.

- Specify the interconnect in the **Interconnects** drop-down list. The default setting is recommended. For a symmetric multi-processor (SMP) system, the default setting uses shared memory for communication. On Windows, the best available interconnect is automatically used.

  (Linux only) If you prefer to select a specific interconnect, you can choose **ethernet**, **ethernet.efa**, or **infiniband**. For more information about these interconnects, see Table 43.5: Supported Interconnects for Linux Platforms (Per Platform) (p. 4313), Table 43.6: Available MPIs for Linux Platforms (p. 4313), and Table 43.7: Supported MPIs for Linux Architectures (Per Interconnect) (p. 4314).

- Specify the type of message passing interface (MPI) you require for the parallel computations in the **MPI Types** field. The list of MPI types varies depending on the selected release and the selected architecture. There are several options, based on the operating system of the parallel cluster. For more information about the available MPI types, see Table 43.2: Supported Interconnects for the Windows Platform (p. 4308) - Table 43.3: Available MPIs for Windows Platforms (p. 4308).

  Important:

  It is your responsibility to make sure the interconnects and the MPI types are compatible. If incompatible inputs are used, Fluent Launcher resorts to using the default values.

- Specify the type of parallel calculation under **Run Types**:

  – Select **Shared Memory on Local Machine** if the parallel calculations are performed by sharing memory allocations on your local machine.

  – Select **Distributed Memory on a Cluster** if the parallel calculations will be distributed among several machines.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can select **Machine Names** and enter the machine names directly into the text field as a list. Machine names can be separated either by a comma or a space. This is not recommended for a long list of machine names.

Alternatively, you can select **File Containing Machine Names** to specify a hosts file (a file that contains the machine names), or you can use the button to browse for a hosts file. If you do not browse for the hosts file and it is not located in the **Working Directory** you specified in the **Home** tab, you must enter the full pathname to the file. On Linux, you may also need to enter the full pathname in some scheduler and custom-scripted scenarios; you will know this is the case if Fluent prints the following error message:

```
Error: file "hosts.txt" does not exist!
```

To edit an existing hosts file, click the button.

By default, Fluent allocates ranks to machines in contiguous blocks, where the block sizes are as equal as possible. You can control the per machine block allocation size using the `machine` *X* `:` *Y* convention in the hosts specification, where *Y* is the process block count for `machine` *X*. The process assignment will cycle through the machine list until all processes are allocated in specified blocks. A fully round-robin assignment of processes can be achieved by setting the machine block allocation sizes to 1 (for example, `machine1:1`, `machine2:1`, and so on).

- Enable the **Select IP Interface** option and make a selection from the drop-down list that appears if you would like to specify the IP interface to be used by the host process. This is equivalent to the `-host_ip=host:ip` command line option. An example of when you might use this option is when you are using distributed memory on multiple machines and your security software is dropping the active socket connections used by Ansys Fluent (resulting in the following message in the console: `The fl process could not be started.`); while it is preferable to avoid this by creating an exception for Ansys Fluent in your security software, you could instead select a suitable IP interface.

- For certain platforms, enable the **Use Job Scheduler** option in the **Scheduler** tab of Fluent Launcher if the parallel calculations are to be performed using a designated Job Scheduler (for details, see Setting Parallel Scheduler Options in Fluent Launcher (p. 4298)).

For additional information, see the following sections:

43.2.1. Setting Parallel Scheduler Options in Fluent Launcher

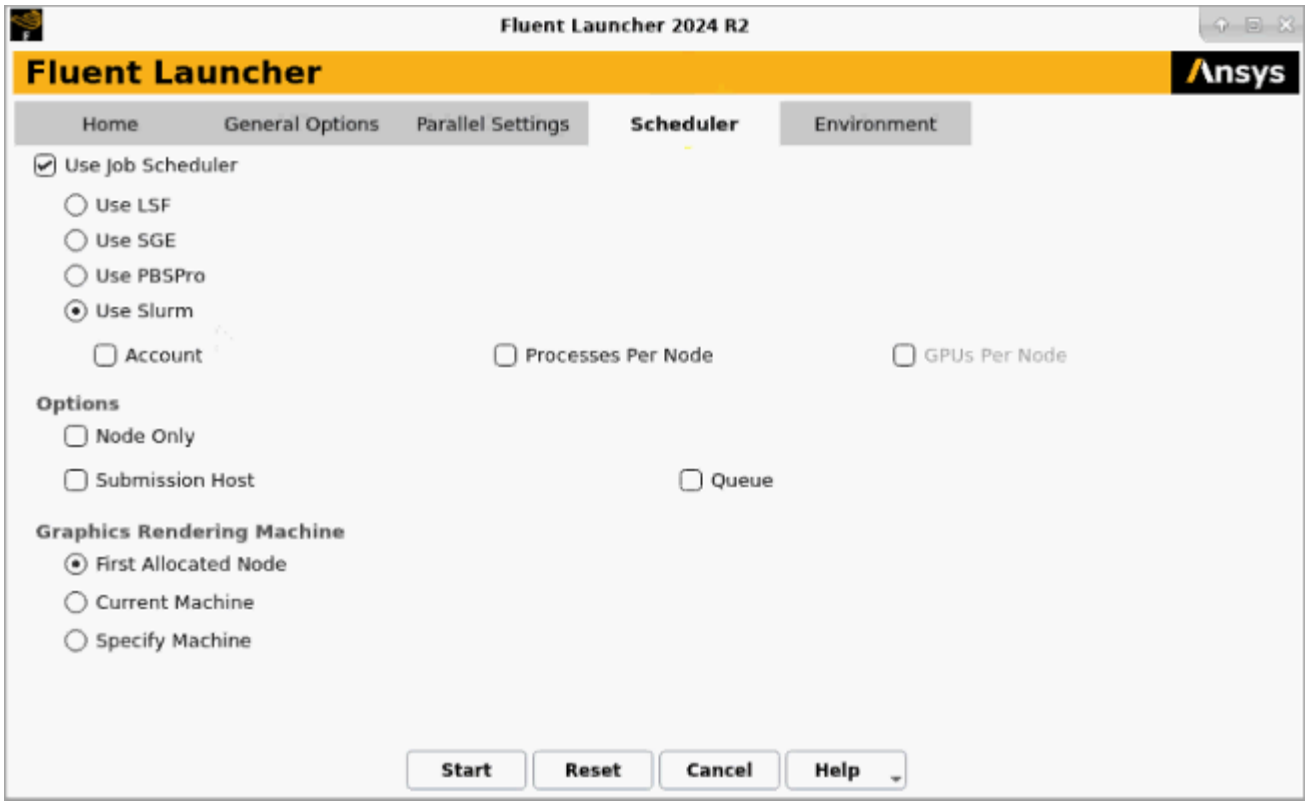
43.2.2. Setting Additional Options When Running on Remote Linux Machines

43.2.1. Setting Parallel Scheduler Options in Fluent Launcher

The **Scheduler** tab allows you to specify settings for running Ansys Fluent with various job schedulers (for example, the Microsoft Job Scheduler for Windows, or LSF, SGE, PBS Professional, and Slurm on Linux).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 43.3: The Scheduler Tab of Fluent Launcher (Linux Version)



For Windows 64-bit, with MSMPI, you can specify that you want to use the Job Scheduler by enabling **Use Job Scheduler** in the **Scheduler** tab. Once enabled, you can then enter a machine name in the **Compute Cluster Head Node Name** text field. If you are running Ansys Fluent on the head node, then you can keep the field empty. This option translates into the proper parallel command line syntax for using the Microsoft Job Scheduler (for details, see Starting Parallel Ansys Fluent with the Microsoft Job Scheduler (p. 4308)).

If you want Ansys Fluent to start after the necessary resources have been allocated by the Scheduler, then select the **Start When Resources are Available** check box.

For Linux (or when the **Use Remote Linux Nodes** option is enabled in the **Remote** tab on Windows), enable **Use Job Scheduler** in the **Scheduler** tab to use one of four available job schedulers.

- Select **Use LSF** to use the LSF load management system with or without checkpointing. If you enable **Use Checkpointing**, then you can specify a checkpointing directory by enabling **Checkpointing Directory** and selecting it from the list below. By default, the current working directory is used. In addition, you can enable **Automatic Checkpoint with Setting of Period** and specify a numerical value for the frequency of automatic checkpointing in the field below.

  For more information, see Setting Job Scheduler Options When Running on Remote Linux Machines (p. 4303) or Part 1: Running Fluent Under LSF.

- Select **Use SGE** to use the SGE load management system. You can choose to make a selection for the **SGE qmaster**, as well as the **SGE Queue**, or the **SGE pe**. Alternatively, with Linux you can enable the **Use SGE settings** check box and specify the location and name of the SGE configuration file.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For more information, see Setting Job Scheduler Options When Running on Remote Linux Machines (p. 4303) or Part 3: Running Fluent Under SGE.

- Select **Use PBSPro** to use the PBS Professional load management system.

  For more information, see Setting Job Scheduler Options When Running on Remote Linux Machines (p. 4303) or Part 2: Running Fluent Under PBS Professional.

- Select **Use Slurm** to use the Slurm load management system. You can enable the **Account** option and make a selection from the list below. If you want to set the number of node processes per cluster node (rather than leaving it to the cluster configuration), you can enable the **Processes Per Node** option and enter a value in the field below. If you enabled the **Native GPU Solver** option in the **Home** tab, you can enable the **GPUs Per Node** option and enter a value in the field below.

  For more information, see Setting Job Scheduler Options When Running on Remote Linux Machines (p. 4303) or Part 4: Running Fluent Under Slurm.

You have the following options under **Options**:

- Enable the **Node Only** option to specify that Cortex and host processes are launched before the job submission and that only the parallel node processes are submitted to the scheduler.

- For SGE and PBS Professional, you can enable the **Tight Coupling** option to enable a job-scheduler-supported native remote node access mechanism in Linux. For details about the MPI / job scheduler combinations that are supported for this tight coupling, see Running Fluent Using a Load Manager.

- For LSF, PBS Professional, and Slurm, you can the enable the **Submission Host** option and make a selection from the drop-down list to specify the submission host name for submitting the job. This is needed if the machine you are using to run the launcher cannot submit jobs to the job scheduler.

- For LSF, PBS Professional, and Slurm, you can enable the **Queue** option and make a selection from the drop-down list to specify the queue name or (for Slurm) partition name.

If you experience poor graphics performance when using a job scheduler in Linux, you may be able to improve performance by changing the machine on which Cortex (the process that manages the graphical user interface and graphics) is running. The **Graphics Rendering Machine** list provides the following options:

- Select **First Allocated Node** if you want Cortex to run on the same machine as that used for compute node 0. Note that this is not available if you have enabled the **Node Only** option.

- Select **Current Machine** if you want Cortex to run on the same machine used to start Fluent Launcher.

- Select **Specify Machine** if you want Cortex to run on a specified machine, which you select from the drop-down list below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that if you enable the **Tight Coupling** option, it is not used if the Cortex process is launched after the job submission (which is the default when not using the **Node Only** option) and is run outside of the scheduler environment by using **Current Machine** or **Specify Machine**.

Important:

(Exceed onDemand Only) If you select the **Current Machine** or **Specify Machine**, you must also set the `CORTEX_PRE=ssrun` environment variable to specify the server side rendering to ensure accelerated graphics performance.

For example: `CORTEX_PRE=/opt/Exceed_connection_server_13.8_64/bin/ssrun`.

Note that the path to `ssrun` may be different for your specific environment.

Note:

Some Linux scheduler options available from the command line are not available in Fluent Launcher, such as specifying the scheduler job submission machine name or setting the scheduler standard error file or standard output file. For details, see Scheduler Options (p. 200).

For Windows, you also have the ability to run in batch mode (using the **Run in Batch Mode** check box) when you provide a journal file that exits Ansys Fluent at the end of the run.

For machines running Microsoft HPC Pack 2008 or newer, you also have the following options to choose from:

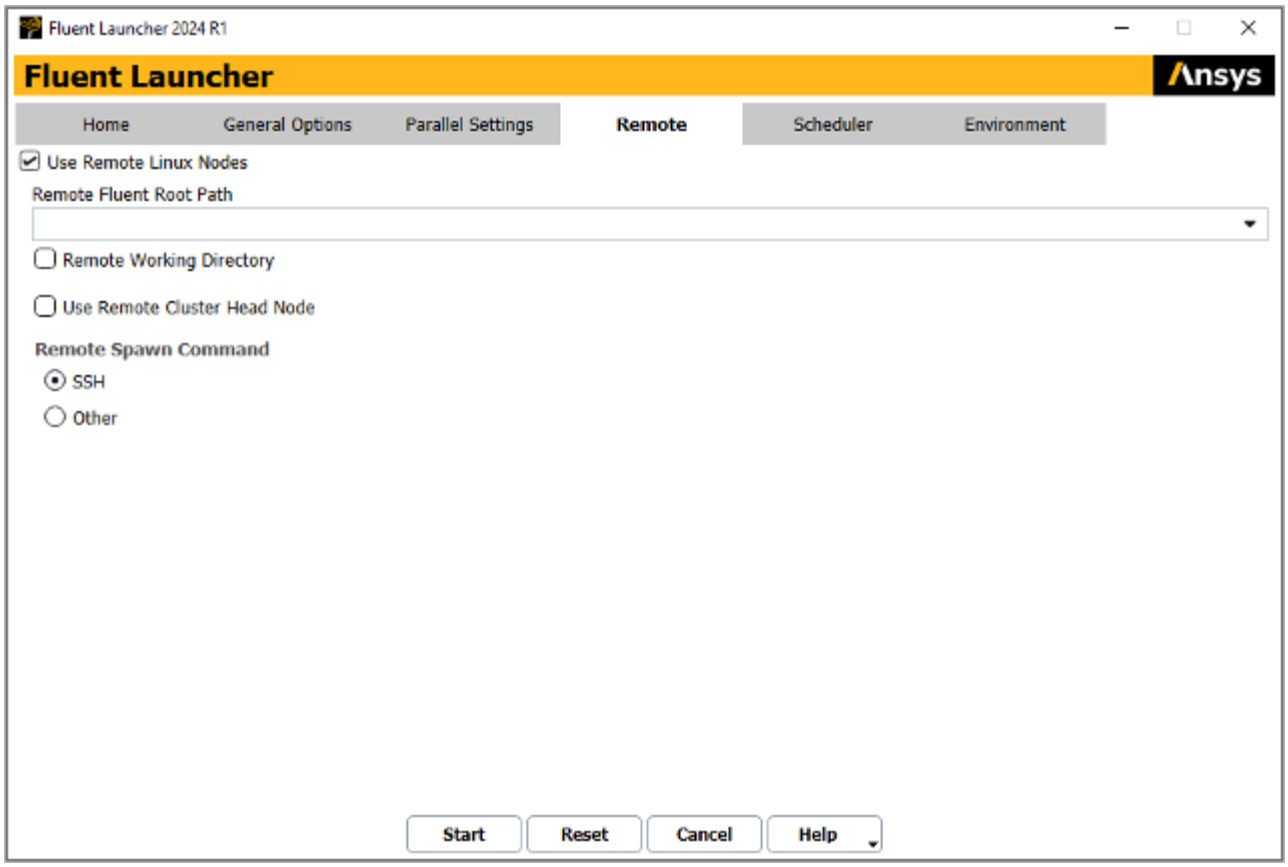
- **Job Template** allows you to create a custom submission policy to define the job parameters for an application. The cluster administrator can use job templates to manage job submission and optimize cluster usage.
- **Node Group** allows you to specify a collection of nodes. Cluster administrators can create groups and assign nodes to one or more groups.
- **Processor Unit** allows you to choose the following:
  - **Core** refers to a single computing unit in a machine. For example, a quad-core processor has 4 cores.
  - **Socket** refers to a set of tightly integrated cores as on a single chip. Machines often have 2 or more sockets, each socket with multiple cores. A dual CPU, hexcore processor, for example, having a total of 12 cores.
  - **Node** refers to a named host, that is, a single machine used as part of a cluster. Typical clusters range from a few to tens, hundreds, or sometimes thousands of machines.

43.2.2. Setting Additional Options When Running on Remote Linux Machines

The **Remote** tab allows you to specify settings for running Ansys Fluent parallel simulations on Linux machines (either in serial or on parallel Linux clusters) via the Windows interface.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 43.4: The Remote Tab of Fluent Launcher



To access remote 64-bit Linux clusters for your parallel calculation, enable the **Use Remote Linux Nodes** option. You can then specify the remote Ansys Fluent Linux installation root path in the **Remote Fluent Root Path** field (for example, *path*`/ansys_inc/v242/fluent`, where *path* is the Linux machine directory in which you installed Ansys Fluent). The **Remote Working Directory** option and field allows you to specify a working directory for the remote Linux nodes, other than the default `temp` directory.

Select one of the following **Remote Spawn Command**s to connect to the remote node:

- **SSH** (the default) will use SSH to spawn nodes from the local Windows machine to the Linux head node as well as from the Linux head node to the compute nodes. To use SSH with Ansys Fluent, you must set up password-less SSH access.
- **Other** allows you to provide other compatible remote shell commands.

Enable the **Use Remote Cluster Head Node** field and specify the remote node to which Ansys Fluent will connect for spawning (for example, via `ssh`). If this is not provided, then Ansys Fluent will try to use the first machine in the machine list or file (defined using the **Distributed Memory on a Cluster** run type in the **Parallel Settings** tab). If SGE, PBS Professional, or Slurm is chosen as the job scheduler, then the same purpose will be served by **Submission Host** or (for SGE) **SGE qmaster**.

In addition to using the settings in the **Remote** tab in Fluent Launcher, the following command line options are also available when starting Ansys Fluent from the command line:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

-nodepath=path

is the path on the remote machine where Ansys Fluent is installed.

-node0=machine name

is the machine from which to launch other nodes.

-nodehomedir=directory

is the directory that becomes the current working directory for all the nodes. Additionally, this will be used as a scratch area for temporary files that are created on the nodes.

-rsh=secure shell command

is the command that will be used to launch executables remotely. This option defaults to `ssh.exe` but can point to any equivalent program. The form of this command should be that it should not wait for additional inputs such as passwords. For example, if you install SSH, and try to launch in mixed mode using `ssh`, the launch may fail unless you have set up a login for SSH without a password.

As there are known issues with launching Ansys Fluent in mixed Windows/Linux mode from cygwin, it is recommended that you use the command prompt (`cmd.exe`).

When working with mixed Linux and Windows runs that employ user-defined functions (UDFs), note the following:

- The file that you have opened for reading / writing on the host machine will not be available on remote nodes and vice-versa. You may therefore have to transfer data present on the nodes to the host and write it from host, (or distribute the data from the host to the nodes after reading the data from the host).
- The loading of multiple UDF libraries into the same session is not supported. As a workaround, you can compile multiple UDF files into same UDF library.

43.2.2.1. Setting Job Scheduler Options When Running on Remote Linux Machines

By selecting the **Use Remote Linux Nodes** option and the **Use Job Scheduler** option in Fluent Launcher, you can set job scheduler options for the remote Linux machines you are accessing for your CFD analysis.

When these options are enabled in Fluent Launcher, you can use the **Scheduler** tab to set parameters for either LSF, SGE, PBS Professional, or Slurm job schedulers. You can learn more about each of the schedulers by referring to the Load Management Documentation.

The following list describes the various controls that are available in the **Scheduler** tab:

Use LSF

allows you to use the LSF job scheduler.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Use Checkpointing

allows you to use checkpointing with LSF. By default, the checkpointing directory will be the current working directory. You can set the following:

Checkpointing Directory

allows you to specify a checkpointing directory that is different from the current working directory.

Automatic Checkpoint with Setting of Period

allows you to specify that the checkpointing is done automatically at a set time interval. Enter the period (in minutes) in the field, otherwise checkpointing will not occur unless you call the `bchkpnt` command.

Use SGE

allows you to use the SGE job scheduler.

SGE qmaster

is the machine in the SGE job submission host list. SGE will allow the **SGE qmaster** node to summon jobs. By default, **localhost** is specified for **SGE qmaster**. Note that the button allows you to check the job status.

SGE Queue

is the queue where you want to submit your Ansys Fluent jobs. Note that you can use the button to contact the **SGE qmaster** for a list of queues. Leave this field blank if you want to use the default queue.

SGE pe

is the parallel environment where you want to submit your Ansys Fluent jobs. The parallel environment must be defined by an administrator. For more information about creating a parallel environment, refer to the SGE documentation. Leave this field blank if you want to use the default parallel environment.

Use PBSPro

allows you to use the PBS Professional job scheduler.

Use Slurm

allows you to use the Slurm job scheduler.

Account

specifies the Slurm account.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Processes Per Node

specifies the number of node processes per cluster node (rather than leaving it to the cluster configuration).

GPUs Per Node

specifies the number of GPUs per cluster node. This is only available for editing if you have enabled the **Native GPU Solver** option in the **Home** tab.

You have the following options under **Options**:

Submission Host

specifies the submission host name for submitting the job. This is needed if the machine you are using to run the launcher cannot submit jobs to the job scheduler. This is only available for LSF, PBS Professional, and Slurm.

Queue

specifies the queue name or (for Slurm) partition name. This is only available for LSF, PBS Professional, and Slurm.

Important:

While running on remote Linux machines using any one of the Job Scheduler options, if the submitted job is in the job queue because of unavailable requested resources, then the Ansys Fluent graphical user interface will remain open until resources are available and the job starts running.

Note:

If you experience poor graphics performance when using a job scheduler in Linux, you may be able to improve performance by specifying that Cortex run on a specified machine when you start Fluent. This can be done by using the `-scheduler=<scheduler>` command line option with `-gui_machine=<hostname>` or `-gui_machine` (see Scheduler Options (p. 200) and Graphics and Files Options (p. 196)), or by selecting **Specify Machine** or **Current Machine** from the **Graphics Rendering Machine** list in the **Scheduler** tab of Fluent Launcher. When running under Slurm, such specification is also needed to allow dynamic spawning (which is described in Dynamically Spawning Processes Between Fluent Meshing and Fluent Solution Modes (p. 220)), as well as the combination of Slurm + Open MPI + distributed memory on a cluster.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

43.3. Starting Parallel Ansys Fluent on a Windows System

You can run Ansys Fluent on a Windows system using either the graphical user interface (for details, see Starting Parallel Ansys Fluent Using Fluent Launcher (p. 4296)) or command line options (for details, see Starting Parallel Ansys Fluent on a Windows System Using Command Line Options (p. 4306)).

Important:

See the separate installation instructions for more information about installing parallel Ansys Fluent for Windows. The startup instructions below assume that you have properly set up the necessary software, based on the appropriate installation instructions.

For additional information, see the following section:

43.3.1. Starting Parallel Ansys Fluent on a Windows System Using Command Line Options

43.3.1. Starting Parallel Ansys Fluent on a Windows System Using Command Line Options

To start the parallel version of Ansys Fluent using command line options, you can use the following syntax in a Command Prompt window:

`fluent` *version* `-t` *nprocs* `[-gpgpu=` *ngpgpus* `]` `[-p` *interconnect* `]` `[-mpi=` *mpi_type* `]` `[-cnf=` *hosts* `]`

where

- *version* must be replaced by the version of Ansys Fluent you want to run (`2d`, `3d`, `2ddp`, or `3ddp`).
- `-t` *nprocs* specifies the number of processes to use. When the `-cnf` option is present, the *hosts* argument is used to determine which machines to use for the parallel job. For example, if there are 8 machines listed in the hosts file and you want to run a job with 4 processes, set *nprocs* to 4 (that is, `-t4`) and Ansys Fluent will use the first 4 machines listed in the hosts file. Note that this does not apply to the Compute Cluster Server (CCS). If the `-gpgpu` option is used, *nprocs* must be chosen such that the number of solver processes per machine is equal on all machines.
- `-p` *interconnect* (optional) specifies the type of interconnect. The `ethernet` interconnect is used by default if the option is not explicitly specified. See Table 43.2: Supported Interconnects for the Windows Platform (p. 4308), Table 43.3: Available MPIs for Windows Platforms (p. 4308), and Table 43.4: Supported MPIs for Windows Architectures (Per Interconnect) (p. 4308) for more information.
- `-mpi=` *mpi_type* (optional) specifies the MPI implementation. If the option is not specified, the default MPI for the given interconnect (Intel MPI or, with more than 64 logical cores, Microsoft MPI) will be used. The use of the default MPI is recommended. The available MPIs for Windows are shown in Table 43.3: Available MPIs for Windows Platforms (p. 4308).
- `-cnf=` *hosts* (optional) specifies the hosts file, which contains a list of the machines on which you want to run the parallel job; if this option is not used, then the session will run on the local machine. If the hosts file is not located in the folder where you are typing the startup command, you must supply the full pathname to the file.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can use a plain text editor such as Notepad to create the hosts file. The only restriction on the filename is that there should be no spaces in it. For example, `hosts.txt` is an acceptable hosts file name, but `my hosts.txt` is not.

Your hosts file (for example, `hosts.txt`) might contain the following entries:

```
machine1
machine2
```

Important:

The last entry must be followed by a blank line.

If a machine in the network is a multiprocessor, you can list it more than once. For example, if `machine1` has 2 CPUs, then, to take advantage of both CPUs (and similarly for multicore machines), the `hosts.txt` file should list `machine1` twice:

```
machine1
machine1
machine2
```

By default, Fluent allocates ranks to machines in contiguous blocks, where the block sizes are as equal as possible. You can control the per machine block allocation size using the `machine` *X* `:` *Y* convention in the hosts specification, where *Y* is the process block count for `machine` *X*. The process assignment will cycle through the machine list until all processes are allocated in specified blocks. A fully round-robin assignment of processes can be achieved by setting the machine block allocation sizes to 1 (for example, `machine1:1`, `machine2:1`, and so on).

- `-gpgpu=` *ngpgpus* specifies the number of GPGPUs per machine to use in offload mode to accelerate the Algebraic Multigrid (AMG) solver and/or the Discrete Ordinates (DO) radiation model calculations. Note that when this option is used, the number of solver process per machine must be equal on all machines and *ngpgpus* must be chosen such that the number of solver processes per machine is an integer multiple of *ngpgpus*. That is, for *nprocs* solver processes running on *M* machines using *ngpgpus* GPGPUS per machine:

$$nprocs \bmod M = 0$$
$$\left(\frac{nprocs}{M}\right) \bmod ngpgpus = 0$$

  See Using General Purpose Graphics Processing Units (GPGPUs) With the Algebraic Multigrid (AMG) Solver (p. 4343) and/or Accelerating Discrete Ordinates (DO) Radiation Calculations (p. 4352) for more information about using GPGPU acceleration.

For example, the full command line to start a 3D parallel job on the first 4 machines listed in a hosts file called `hosts.txt` is as follows:

```
fluent 3d -t4 -cnf=hosts.txt
```

As another example, the full command line to start a 3D symmetrical multiprocessing (SMP) parallel job on 4 machines is as follows:

```
fluent 3d -t4
```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

In either case, the default communication library (Intel MPI or, with more than 64 logical cores, Microsoft MPI), and the default interconnect (automatically selected by the MPI used, or `ethernet`) will be used since these options are not specified.

The first time that you try to run Ansys Fluent in parallel, you will be prompted for information about the current Windows account.

The supported interconnects for dedicated parallel `win64` Windows machines, the associated MPIs for them, and the corresponding syntax are listed in Table 43.2: Supported Interconnects for the Windows Platform (p. 4308) - Table 43.4: Supported MPIs for Windows Architectures (Per Interconnect) (p. 4308).

Table 43.2: Supported Interconnects for the Windows Platform

Platform	Processor	Architecture	Interconnects
Windows	64-bit	`win64`	`ethernet` (default), `infiniband`

Table 43.3: Available MPIs for Windows Platforms

MPI	Syntax (flag)	Communication Library	Notes
`msmpi`	`-mpi=msmpi`	Microsoft MPI	(1), (2)
`intel`	`-mpi=intel`	Intel MPI	(1), (3), (4)

(1) Used with Shared Memory Machine (SHM) where the memory is shared between the processors on a single machine.

(2) Used with a job scheduler to run on a Windows high performance computing server (HPC).

(3) Used with Distributed Memory Machine (DMM) where each processor has its own memory associated with it.

(4) By default, the Intel MPI may fail when mixing hardware for compute nodes. As a workaround, you can use the following environment setting:

```
I_MPI_PLATFORM none
```

Table 43.4: Supported MPIs for Windows Architectures (Per Interconnect)

Architecture	Ethernet	InfiniBand
`win64`	`intel` (default with less than 64 logical cores), `msmpi` (default with more than 64 logical cores)	`msmpi`

43.3.1.1. Starting Parallel Ansys Fluent with the Microsoft Job Scheduler

The Microsoft Job Scheduler allows you to manage multiple jobs and tasks, allocate machine resources, send tasks to compute nodes, and monitor jobs, tasks, and compute nodes.

The Windows Server operating systems include a compute cluster server (CCS) and a high performance computing server (HPC) that combines the Microsoft MPI type (`msmpi`) with the Microsoft Job Scheduler. Ansys Fluent provides a means of using the Microsoft Job Scheduler using the following flag in the parallel command:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

`-ccp` *head-node-name*

where `-ccp` indicates the use of the compute cluster server package, and *head-node-name* indicates the name of the head node of the machine cluster.

For example, if you want to use the Microsoft Job Scheduler to run a 3D model on 2 nodes, the corresponding command syntax would be:

`fluent 3d -t2 -ccp` *head-node-name*

Important:

The Intel MPI (`intel`) is not supported with the Microsoft Job Scheduler.

Note:

When using Microsoft Job Scheduler, the best interconnect is automatically selected by MS-MPI.

Though the usage described previously is recommended as an initial starting point for running Ansys Fluent with the Microsoft Job Scheduler, there are further options provided to meet your specific needs. Ansys Fluent allows you to do any of the following with the Microsoft Job Scheduler:

- Request resources from the Microsoft Job Scheduler first, before you launch Ansys Fluent.

  This is done by first submitting a job that will run until canceled, as shown in the following example:

  `job new/scheduler:` *head-node-name* `/numprocessors:2 /rununtilcanceled:true`

  This example requests a 2-node resource on a cluster named *head-node-name*. You will see that a job is created with the job ID *job-id*:

  `job submit/scheduler:` *head-node-name* `/id:` *job-id*

  Then check if the resources have been allocated:

  `job view` *job-id* `/scheduler:` *head-node-name*

  If the resources are ready, you can start Ansys Fluent using the job ID:

  `fluent 3d -t2 -ccp` *head-node-name* `-jobid=` *job-id*

  This job will be reusable until you decide to cancel it, at which point you must enter the following:

  `job cancel` *job-id* `/scheduler:` *head-node-name*

- Have Ansys Fluent submit a CCS job, but delay the launching of Ansys Fluent until the actual resources are allocated.

  This is done by specifying the job ID as `-1`, as shown in the following example:

  `fluent 3d -t2 -ccp` *head-node-name* `-jobid=-1`

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you want to stop the job application, click the **Cancel** button. Ansys Fluent will prompt you for confirmation, and then clean up the pending job and exit.

- Run your job using XML template files.

  This is done by first creating an XML template file, such as shown in the following example:

```
<?xml version="1.0" encoding="utf-8"?>
<Job
xmlns:m:xsi="http://www.w3.org/2001/XMLSchema-instance"
xmlns:m:xsd="http://www.w3.org/2001/XMLSchema"
SoftwareLicense=""
MaximumNumberOfProcessors="4"
MinimumNumberOfProcessors="4"
Runtime="Infinite"
IsExclusive="true"
Priority="Normal"
Name="Name_of_job"
Project="Fluent runs"
RunUntilCanceled="false">
<Tasks xmlns:m="http://www.microsoft.com/ComputeCluster/">
 <Task
   MaximumNumberOfProcessors="4"
   MinimumNumberOfProcessors="4"
   Depend=""
   WorkDirectory="\\file-server\home\user"
   Stdout="fluent-case.%CCP_JOBID%.out"
   Stderr="fluent-case.%CCP_JOBID%.err"
   Name="My Task"
   CommandLine="\\head-node\fluent-sharename\win64\fluent.exe 3d -i bsi.jou -t4"
   IsExclusive="true"
   IsRerunnable="false"
   Runtime="Infinite">
 </Task>
</Tasks>
</Job>
```

  where `fluent-sharename` is the name of the shared directory pointing to where Ansys Fluent is installed (for example, `C:\Program Files\ANSYS Inc\v242\fluent`).

  Important:

  Note that you must create a journal file that exits Ansys Fluent at the end of the run, and refer to it using the `-i` flag in your XML template file (`bs1.jou` in the previous example).

  After you have saved the file and given it a name (for example, `job1.xml`), you can submit the job as shown:

```
job submit /jobfile:job1.xml
```

- Run the job in batch mode without displaying the Ansys Fluent graphical user interface.

  The following is an example of such a batch mode job:

```
job submit /scheduler: head-node-name

/numprocessors:2 /workdir:\\file-server\home\user\

\\head-node\fluent-sharename\ntbin\win64\fluent.exe 3d -t2 -i bs1.jou
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where `fluent-sharename` is the name of the shared directory pointing to where Ansys Fluent is installed (for example, `C:\Program Files\ANSYS Inc\v242\fluent`).

Important:

- Note that you must create a journal file that exits Ansys Fluent at the end of the run, and refer to it using the `-i` flag in your batch mode job submission (`bs1.jou` in the previous example).
- You can start Ansys Fluent jobs from any machine on which is installed either the full CCP or the CCP client tools, but note that all the machines must have the same version installed.

43.4. Starting Parallel Ansys Fluent on a Linux System

You can run Ansys Fluent on a Linux system using either the graphical user interface (for details, see Starting Parallel Ansys Fluent Using Fluent Launcher (p. 4296)) or command line options (for details, see Starting Parallel Ansys Fluent on a Linux System Using Command Line Options (p. 4311) and Setting Up Your Secure Shell Clients (p. 4314)).

Important:

On some clusters without accelerated graphics, Fluent may not accept keyboard inputs. If you encounter this behavior, set the `QT_XKB_CONFIG_ROOT` environment variable equal to `/usr/share/X11/xkb`.

For additional information, see the following sections:

43.4.1. Starting Parallel Ansys Fluent on a Linux System Using Command Line Options

43.4.2. Setting Up Your Secure Shell Clients

43.4.1. Starting Parallel Ansys Fluent on a Linux System Using Command Line Options

To start the parallel version of Ansys Fluent using command line options, you can use the following syntax in a command prompt window:

`fluent` *version* `-t` *nprocs* `[-gpgpu=` *ngpgpus* `] [-p` *interconnect* `] [-mpi=` *mpi_type* `] [-cnf=` *hosts* `]`

where

- *version* must be replaced by the version of Ansys Fluent you want to run (`2d`, `3d`, `2ddp`, or `3ddp`).
- `-t` *nprocs* specifies the number of processes to use. When the `-cnf` option is present, the *hosts* argument is used to determine which machines to use for the parallel job. For example, if there are 10 machines listed in the hosts file and you want to run a job with 5 processes, set *nprocs* to 5 (that is, `-t5`) and Ansys Fluent will use the first 5 machines listed in the hosts file. Note that if

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the `-gpgpu` option is used, *nprocs* must be chosen such that the number of solver processes per machine is equal on all machines.

- `-p` *interconnect* (optional) specifies the type of interconnect. The `auto-select` interconnect is used by default so that the best available interconnect is used if the option is not explicitly specified. See Table 43.5: Supported Interconnects for Linux Platforms (Per Platform) (p. 4313), Table 43.6: Available MPIs for Linux Platforms (p. 4313), and Table 43.7: Supported MPIs for Linux Architectures (Per Interconnect) (p. 4314) for more information.

- `-mpi=` *mpi_type* (optional) specifies the type of MPI. If the option is not specified, the default MPI for the given interconnect will be used (the use of the default MPI is recommended). The available MPIs for Linux are shown in Table 43.6: Available MPIs for Linux Platforms (p. 4313).

- `-cnf=` *hosts* (optional) specifies the hosts file, which contains a list of the machines on which you want to run the parallel job; if this option is not used, then the session will run on the local machine. If the hosts file is not located in the directory where you are typing the startup command, you must supply the full pathname to the file. You may also need to supply the full pathname in some scheduler and custom-scripted scenarios; you will know this is the case if Fluent prints the following error message:

  ```
  Error: file "hosts.txt" does not exist!
  ```

  You can use a plain text editor to create the hosts file. The only restriction on the filename is that there should be no spaces in it. For example, `hosts.txt` is an acceptable hosts file name, but `my hosts.txt` is not.

  Your hosts file (for example, `hosts.txt`) might contain the following entries:

  ```
  machine1
  machine2
  ```

  If a machine in the network is a multiprocessor, you can list it more than once. For example, if `machine1` has 2 CPUs, then, to take advantage of both CPUs, the `hosts.txt` file should list `machine1` twice:

  ```
  machine1
  machine1
  machine2
  ```

  As an alternative to a hosts file, you can simply type the names of the machines in a list separated by commas, as shown in the following example: `-cnf=machine1,machine1,machine2`.

  By default, Fluent allocates ranks to machines in contiguous blocks, where the block sizes are as equal as possible. You can control the per machine block allocation size using the `machine` *X* `:` *Y* convention in the hosts specification, where *Y* is the process block count for `machine` *X*. The process assignment will cycle through the machine list until all processes are allocated in specified blocks. A fully round-robin assignment of processes can be achieved by setting the machine block allocation sizes to 1 (for example, `machine1:1`, `machine2:1`, and so on).

- `-gpgpu=` *ngpgpus* specifies the number of GPGPUs per machine to use in offload mode to accelerate the Algebraic Multigrid (AMG) solver and/or the Discrete Ordinates (DO) radiation model calculations. Note that when this option is used, the number of solver process per machine must be equal on all machines and *ngpgpus* must be chosen such that the number of solver processes

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

per machine is an integer multiple of *ngpgpus*. That is, for *nprocs* solver processes running on *M* machines using *ngpgpus* GPGPUS per machine:

$$nprocs \bmod M = 0$$
$$\left(\frac{nprocs}{M}\right) \bmod ngpgpus = 0$$

See Using General Purpose Graphics Processing Units (GPGPUs) With the Algebraic Multigrid (AMG) Solver (p. 4343) and/or Accelerating Discrete Ordinates (DO) Radiation Calculations (p. 4352) for more information about using GPGPU acceleration.

For example, to use the InfiniBand interconnect, and to start the 3D solver with 4 compute nodes on the machines defined in the text file called `fluent.hosts`, you can enter the following in the command prompt:

```
fluent 3d -t4 -pinfiniband -cnf=fluent.hosts
```

Note that if the optional `-cnf=` *hosts* is specified, a compute node will be spawned on each machine listed in the file *hosts*.

During startup, Ansys Fluent will create very small text files in the `/tmp` area. If you would like to specify a different location for these files, set the following environment variable: `export FL_TMP-DIR=` *directory*.

The supported interconnects for parallel Linux machines are listed below (Table 43.5: Supported Interconnects for Linux Platforms (Per Platform) (p. 4313), Table 43.6: Available MPIs for Linux Platforms (p. 4313), and Table 43.7: Supported MPIs for Linux Architectures (Per Interconnect) (p. 4314)), along with their associated communication libraries, the corresponding syntax,

Table 43.5: Supported Interconnects for Linux Platforms (Per Platform)

Platform	Processor	Architecture	Interconnects/Systems*
Linux	64-bit	`lnamd64`	`ethernet`, `ethernet.efa`**, `infiniband`

(*) Node processes on the same machine communicate by shared memory. Ansys Fluent lets the MPI autoselect the best interconnect available on the system. Users can specify an interconnect to override that selection. Ethernet is the fallback choice.

(**) `ethernet.efa` is only a valid setup in Amazon Web Service (AWS) instances with the EFA interface.

Table 43.6: Available MPIs for Linux Platforms

MPI	Syntax (flag)	Communication Library	Notes
`intel`	`-mpi=intel`	Intel MPI	General purpose for SMPs and clusters (1)
`openmpi`	`-mpi=openmpi`	Open MPI	Open source MPI-2 implementation. For both SMPs and clusters. (2)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(1) By default, the Intel MPI may fail when mixing hardware for compute nodes. As a workaround, you can use the following environment setting:

```
I_MPI_PLATFORM none
```

(2) This version of Open MPI is dependent on the following versions (or higher) of software:

- OFED version 4.4
- The versions of LSF, UGE, PBS, and/or Slurm listed with the other supported job schedulers and queuing systems posted on the Platform Support section of the Ansys Website.

Table 43.7: Supported MPIs for Linux Architectures (Per Interconnect)

Architecture	Ethernet	Ethernet.EFA	InfiniBand	Omni-Path*
lnamd64	intel (default) and openmpi	intel (default) and openmpi	intel (default) and openmpi	intel (default)

(*) For Intel Omni-Path architecture, Omni-Path software 10.2 or higher is recommended.

(Linux Only) To enable a job-scheduler-supported native remote node access mechanism, you can use the -scheduler_tight_coupling command line option. For details about the MPI / job scheduler combinations that are supported for this tight coupling, see Running Fluent Using a Load Manager.

43.4.2. Setting Up Your Secure Shell Clients

For cluster computing on Linux systems, most parallel versions of Ansys Fluent will require the user account set up such that you can connect to all nodes on the cluster (using the secure shell (ssh) client) without having to enter a password each time for each machine.

Provided that the server daemon (sshd) is running, this section briefly describes how you can configure your system in order to use Ansys Fluent for parallel computing.

43.4.2.1. Configuring the ssh Client

The secure shell client (ssh) is used widely. Depending on the specific protocol and the version deployed, configuration involves a few steps. SSH1 and SSH2 are two current protocols. OpenSSH is an open implementation of the SSH2 protocol and is backwards compatible with the SSH1 protocol. To add a client machine, with respect to user configuration, the following steps are involved:

1. Generate a public-private key pair using ssh-keygen (or using a graphical user interface client). For example:

   ```
   % ssh-keygen -t dsa
   ```

   where it creates a Digital Signature Authority (DSA) type key pair.

2. Place your public key on the remote host.

   - For SSH1, insert the contents of the client (~/.ssh/identity.pub) into the server (~/.ssh/authorized_keys).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For SSH2, insert the contents of the client (`~/.ssh/id_dsa.pub`) into the server (`~/.ssh/authorized_keys2`).

The client machine is now added to the access list and you are no longer required to type in a password each time. For additional information, consult your system administrator or refer to your system documentation.

43.5. Mesh Partitioning and Load Balancing

Information about mesh partitioning and load balancing is provided in the following sections:

43.5.1. Overview of Mesh Partitioning
43.5.2. Partitioning the Mesh Automatically
43.5.3. Partitioning the Mesh Manually and Balancing the Load
43.5.4. Using the Partitioning and Load Balancing Dialog Box
43.5.5. Mesh Partitioning Methods
43.5.6. Checking the Partitions
43.5.7. Load Distribution
43.5.8. Troubleshooting

43.5.1. Overview of Mesh Partitioning

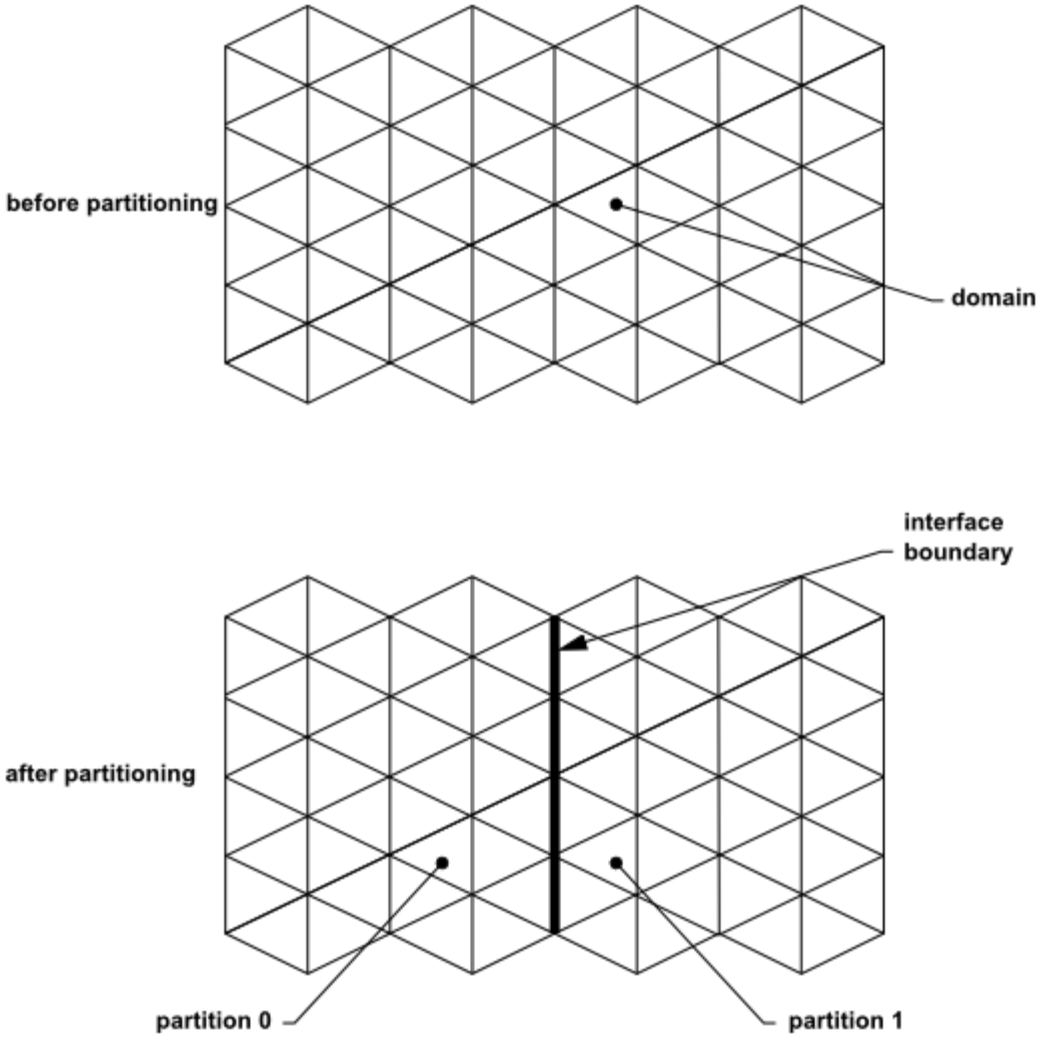
When you use the parallel solver in Ansys Fluent, you must partition or subdivide the mesh into groups of cells that can be solved on separate processors (see Figure 43.5: Partitioning the Mesh (p. 4316)). You can either use the automatic partitioning algorithms when reading an unpartitioned mesh into the parallel solver (recommended approach, described in Partitioning the Mesh Automatically (p. 4316)), or perform the partitioning yourself in the serial solver or after reading a mesh into the parallel solver (as described in Partitioning the Mesh Manually and Balancing the Load (p. 4319)). In either case, the available partitioning methods are those described in Mesh Partitioning Methods (p. 4331). You can partition the mesh before or after you set up the problem (by defining models, boundary conditions, and so on).

Note that the relative distribution of cells among compute nodes will be maintained during mesh adaption, so manual repartitioning after adaption is not required. For details, see Load Distribution (p. 4342).

If you use the serial solver to set up the problem before partitioning, the machine on which you perform this task must have enough memory to read in the mesh. If your mesh is too large to be read into the serial solver, you can read the unpartitioned mesh directly into the parallel solver (using the memory available in all the defined hosts) and have it automatically partitioned. In this case you will set up the problem after an initial partition has been made. You will then be able to manually repartition the case if necessary. See Partitioning the Mesh Automatically (p. 4316) and Partitioning the Mesh Manually and Balancing the Load (p. 4319) for additional details and limitations, and Checking the Partitions (p. 4338) for details about checking the partitions.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 43.5: Partitioning the Mesh



Note:

When you create a Fluent mesh in parallel and read it into the Fluent solver, any partitions set up in meshing mode are preserved when the number of nodes is the same between meshing and solution modes. Performance is impacted, however, when the mesh is not balanced in Fluent meshing (as in the case of tetrahedral and prism cells).

For improved computational efficiency, automatic repartitioning occurs for any imbalanced Fluent (tetrahedral and prism) mesh when it is read into the Fluent solver. This happens regardless of any differences in the parallel node distribution of the mesh. In other words, upon being read into the Fluent parallel solver, an imbalanced parallel tetrahedral or prism mesh created in the Fluent mesher will be automatically repartitioned.

43.5.2. Partitioning the Mesh Automatically

For automatic mesh partitioning, you can select the partition method and other options for creating the mesh partitions before reading a case file into the parallel version of the solver. For some of the methods, you can perform pretesting to ensure that the best possible partition is performed. See

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mesh Partitioning Methods (p. 4331) for information about the partitioning methods available in Ansys Fluent.

Note:

Architecturally aware partitioning (see Partitioning (p. 4319)) is performed automatically when the case file is read. If the maximum inter-machine communication is reduced by more than 5%, the new partition mapping will be applied, and a message is displayed in the console, for example:

```
inter-node communication reduction by architecture-aware remap-
ping: 47%
```

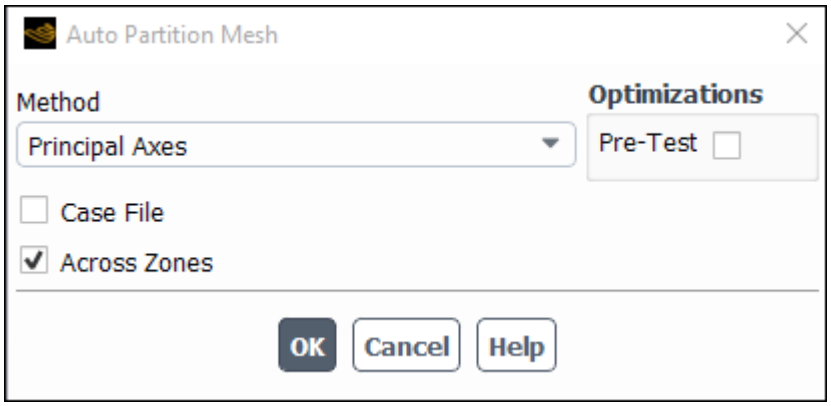
While the message indicates actual point-to-point network traffic reduction, solver computational performance improvement may be somewhat less, and depends on the case and the system network configuration.

The procedure for partitioning automatically in the parallel solver is as follows:

1. (optional) Set the partitioning parameters in the **Auto Partition Mesh** dialog box (Figure 43.6: The Auto Partition Mesh Dialog Box (p. 4317)).

   Parallel → General → Auto Partition...

   Figure 43.6: The Auto Partition Mesh Dialog Box

   If you are reading in a mesh file or a case file for which no partition information is available, and you keep the **Case File** option turned on, Ansys Fluent will partition the mesh using the method displayed in the **Method** drop-down list. By default, the **Metis** method is used; as part of this method, the Laplace smoothing method is enabled by default so that adjacent highly-stretched cells are grouped together, which can significantly improve convergence and scalability. Such Laplace smoothing is controlled through preferences, as described in Preferences for Advanced Auto-Partitioning Methods (p. 4318).

   If you want to specify the partitioning method and associated options yourself, the procedure is as follows:

   a. Turn off the **Case File** option. The other options in the dialog box will become available.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. Select the partition method in the **Method** drop-down list. The choices are the techniques described in Partition Methods (p. 4332).

c. You can choose to independently apply partitioning to each cell zone, or you can allow partitions to cross zone boundaries using the **Across Zones** check button (which is enabled by default). It is recommended that you leave this option enabled, as otherwise the resulting partitions may be too granularized, which can compromise performance. Note that disabling this option has no effect when you have selected **Metis** for the **Method**.

d. If you have chosen the **Principal Axes** or **Cartesian Axes** method, you can improve the partitioning by enabling the automatic testing of the different bisection directions before the actual partitioning occurs. To use pretesting, turn on the **Pre-Test** option. Pretesting is described in Pretesting (p. 4337).

e. Click **OK**.

If you have a case file where you have already partitioned the mesh, and the number of partitions divides evenly into the number of compute nodes, you can keep the default selection of **Case File** in the **Auto Partition Mesh** dialog box. This instructs Ansys Fluent to use the partitions in the case file.

2. Read the case file.

File → **Read** → **Case...**

43.5.2.1. Preferences for Advanced Auto-Partitioning Methods

If you want auto-partitioning to prioritize convergence and scalability during solution, you can select a Laplace smoothing option from the **Advanced Partitioning Method For Better Convergence** drop-down list in the **General** branch of the **Preferences** dialog box (accessed through **File**/**Preferences...**). Through these options, the Laplace smoothing method is enabled so that adjacent highly-stretched cells are grouped together in order to prevent partition boundaries from passing through areas of high-aspect-ratio cells. There are two options:

- **Laplace Auto-Partition**

  For a case written using release 2023 R2 or later, you can select this option with the **Metis** auto-partition method, and Laplace smoothing will be directly based on the cell group information written into the case file. This option is selected by default with the **Metis** method.

- **Laplace Auto-Repartition**

  For a case written using a release previous to 2023 R2, you can select this option, so that the cell group information is created during the repartition and migration steps that take place after the auto-partitioning when reading a case file. This option can be used with any auto-partition method.

43.5.2.2. Reporting During Auto Partitioning

As the mesh is automatically partitioned, some information about the partitioning process will be displayed in the console. If you want additional information, you can display a report from the **Partitioning and Load Balancing** dialog box after the partitioning is completed.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parallel → General → Partition/Load Balance...

When you click the **Print Active Partitions** or **Print Stored Partitions** button in the **Partitioning and Load Balancing** dialog box, Ansys Fluent will display the partition ID, number of cells, faces, and interfaces, and the ratio of interfaces to faces for each active or stored partition in the console. In addition, it will display the minimum and maximum cell, face, interface, and face-ratio variations. For details, see Interpreting Partition Statistics (p. 4339). You can examine the partitions graphically by following the directions in Checking the Partitions (p. 4338).

43.5.3. Partitioning the Mesh Manually and Balancing the Load

Automatic partitioning in the parallel solver (described in Partitioning the Mesh Automatically (p. 4316)) is the recommended approach to mesh partitioning, but it is also possible to partition the mesh manually in either the serial solver or the parallel solver. After automatic or manual partitioning, you will be able to inspect the partitions created (for details, see Checking the Partitions (p. 4338)) and optionally repartition the mesh, if necessary. Again, you can do so within the serial or the parallel solver, using the **Partitioning and Load Balancing** dialog box. A partitioned mesh may also be used in the serial solver without any loss in performance.

43.5.3.1. Guidelines for Partitioning the Mesh

The following steps are recommended for partitioning a mesh manually:

1. Partition the mesh using the default method (**Metis**). **Metis** will generally produce the best quality partitions for most problems and no further user intervention should be necessary. Note that the **Cartesian Axes** method can be a reasonable alternative with less memory overhead.

2. Examine the partition statistics, which are described in Interpreting Partition Statistics (p. 4339). Your aim is to minimize the `Maximum` value of the `Partition boundary face count ratio`, while maintaining a balanced load. For a given mesh, the `Mean cell count deviation` is a measure of the maximum load imbalance for a method. If the statistics are not satisfactory for a problem, you can try one of the other partitioning methods.

Instructions for manual partitioning are provided below.

43.5.4. Using the Partitioning and Load Balancing Dialog Box

43.5.4.1. Partitioning

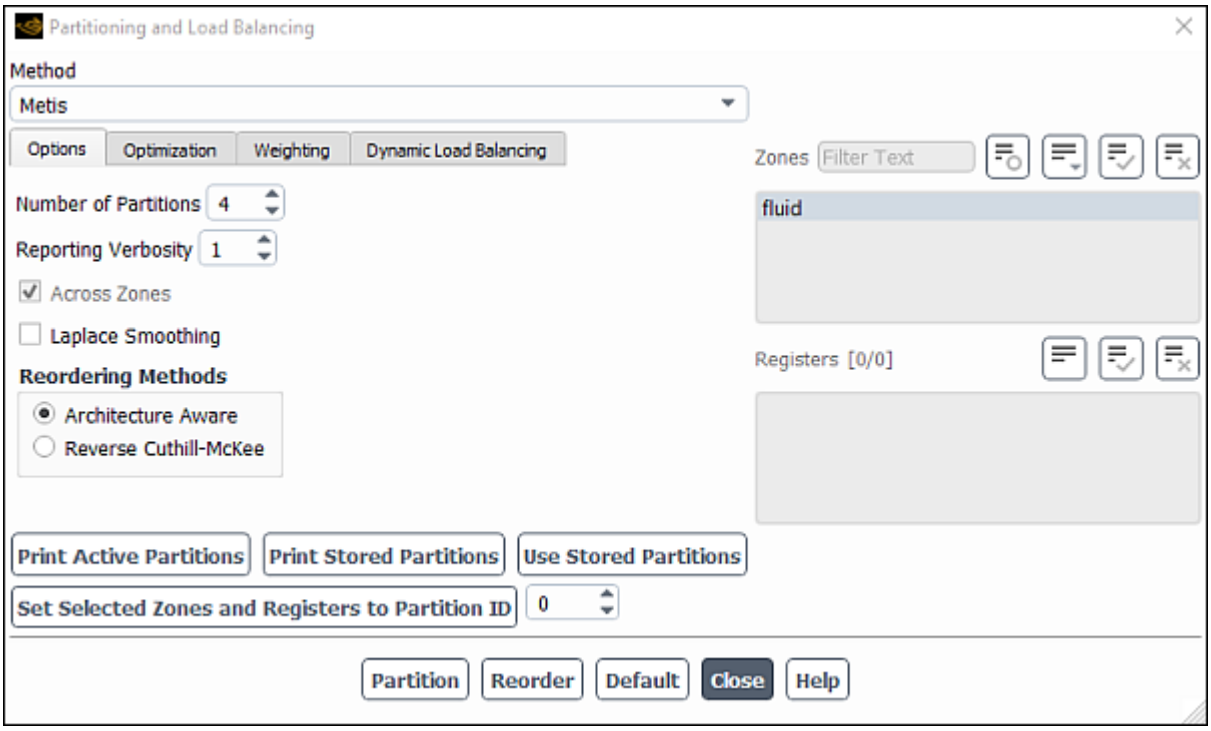
In order to partition the mesh, you must select the partition method for creating the mesh partitions, set the number of partitions, select the zones and/or registers, and choose the optimizations to be used. For some methods, you can also perform pretesting to ensure that the best possible partition is performed. Once you have set all the parameters in the **Partitioning and Load Balancing** dialog box to your satisfaction, click the **Partition** button to subdivide the mesh into the selected number of partitions using the prescribed method and optimization(s). For recommended partitioning strategies see Guidelines for Partitioning the Mesh (p. 4319).

You can set the relevant inputs in the **Partitioning and Load Balancing** dialog box (Figure 43.7: The Partitioning and Load Balancing Dialog Box (p. 4320)) in the following manner:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parallel → General → Partition/Load Balance...

Figure 43.7: The Partitioning and Load Balancing Dialog Box



1. Select the **Method** from the drop-down list. The choices are described in Partition Methods (p. 4332).

2. In the **Options** tab

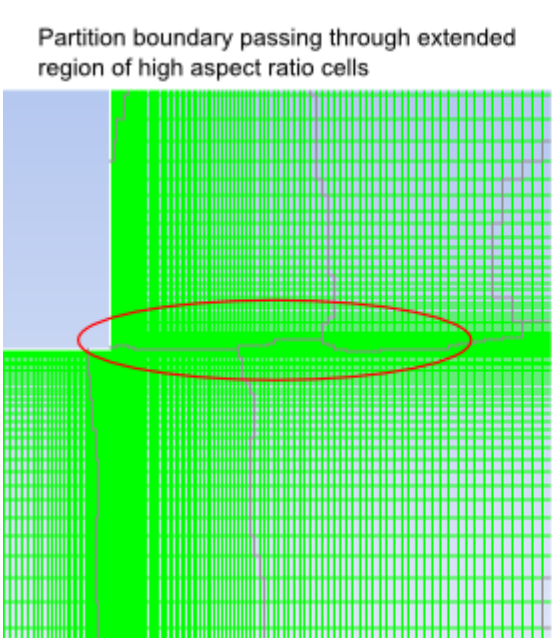
   a. Set the desired number of mesh partitions in the **Number of Partitions** field. You can use the counter arrows to increase or decrease the value, instead of typing in the box. The number of mesh partitions must be an integer number that is divisible by the number of processors available for parallel computing.

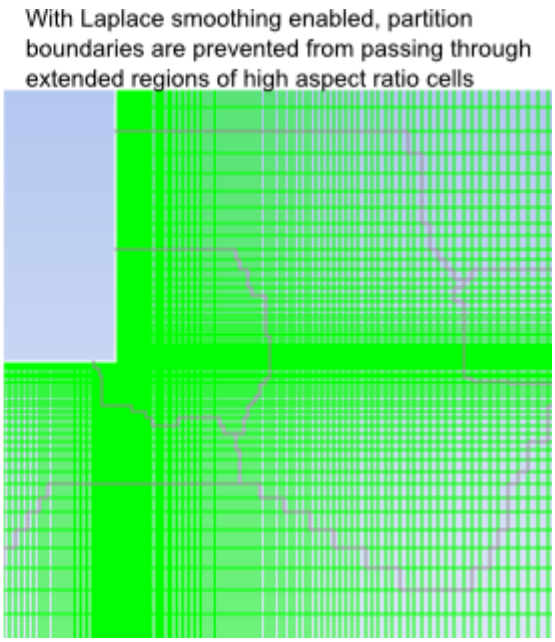
   b. Set the **Reporting Verbosity**. This allows you to control what is displayed in the console. For details, see Reporting During Partitioning (p. 4328).

   c. You can choose to independently apply partitioning to each cell zone, or you can allow partitions to cross zone boundaries using the **Across Zones** check button (which is enabled by default). It is recommended that you leave this option enabled, as otherwise the resulting partitions may be too granularized, which can compromise performance. Note that disabling this option has no effect when you have selected **Metis** for the **Method**.

   d. If you are using the **Metis** method, you have the option of enabling **Laplace Smoothing**. This option can be used to prevent partition boundaries from passing through areas of high aspect ratio cells. This can improve the convergence and scalability of the flow and conjugate heat transfer computations for cases with regions of highly stretched cells. After enabling **Laplace Smoothing**, you can specify the **Cutoff Aspect Ratio**. The **Cutoff Aspect Ratio** corresponds roughly to the maximum aspect ratio allowable along a partition boundary.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.





e. Select the **Reordering Method** for partitions to optimize parallel performance:

- **Architecture Aware**: This is the default option and it accounts for the system architecture and network topology in remapping the partitions to the processors.
- **Reverse Cuthill-McKee**: This option minimizes the bandwidth of the compute-node connectivity matrix (the maximum distance between two connected processes) without incorporating the system architecture.

The reordering methods are parallel performance tuning options. After the case is initially partitioned for parallel processing, the partition reordering step will remap the partitions in a more optimal way to improve parallel performance.

Important:

The Architecture-aware reordering method is not applicable when only a single machine is used for the simulation.

After initially loading the case into a parallel session, you can click the **Reorder** button to reorder the partitions. The necessary algorithms are executed and Ansys Fluent will report if it can find a more optimal mapping for the partitions, as well as the potential improvement in inter-machine communications. If the reported improvement is significant (say, more than 5%), then you can click the **Use Stored Partitions** button to use the new partition mapping. This will generally entail large data transfers amongst all the processes, and another reliable method to use the new partitions would be to write out a case file and load it back in to a new parallel session. The process is similar to re-partitioning with a new partitioning method, for example. Note that sometimes, depending on the cluster configuration and initial case partitioning, and if the partitions have already been reordered, no improvement is possible, and this will be reported in the console after clicking the **Reorder** button. You can simply continue in this case, and there will be no effect on the simulation. Also, note that partition

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

reordering is specific to the current parallel configuration and should be repeated if the number of machines used changes during subsequent computations.

3. In the **Optimization** tab

   a. You can enable and control the desired optimization methods (described in Optimizations (p. 4336)). You can enable the **Merge** and **Smooth** schemes by enabling the check button next to each one. For each scheme, you can also set the number of **Iterations**. Each optimization scheme will be applied until appropriate criteria are met, or the maximum number of iterations has been executed. If the **Iterations** counter is set to 0, the optimization scheme will be applied until completion, with no limit on maximum number of iterations.

   b. Choosing the **Principal Axes** or **Cartesian Axes** method, you can improve the partitioning by enabling the automatic testing of the different bisection directions before the actual partitioning occurs. To use pretesting, enable the **Pre-Test** option. Pretesting is described in Pretesting (p. 4337).

4. In the **Zones** and/or **Registers** lists, select the zone(s) and/or register(s) for which you want to partition. For most cases, you will select all **Zones** (the default) to partition the entire domain. See below for details.

5. You can assign selected **Zones** and/or **Registers** to a specific partition ID by entering a value for the **Set Selected Zones and Registers to Partition ID**. For example, if the **Number** of partitions for your mesh is 2, then you can only use IDs of 0 or 1. If you have three partitions, then you can enter IDs of 0, 1, or 2. This can be useful in situations where the gradient at a region is known to be high. In such cases, you can mark the region or zone and set the marked cells to one of the partition IDs, thereby preventing the partition from going through that region. This in turn will facilitate convergence. This is also useful in cases where mesh manipulation tools are not available in parallel. In this case, you can assign the related cells to a particular ID so that the mesh manipulation tools are now functional.

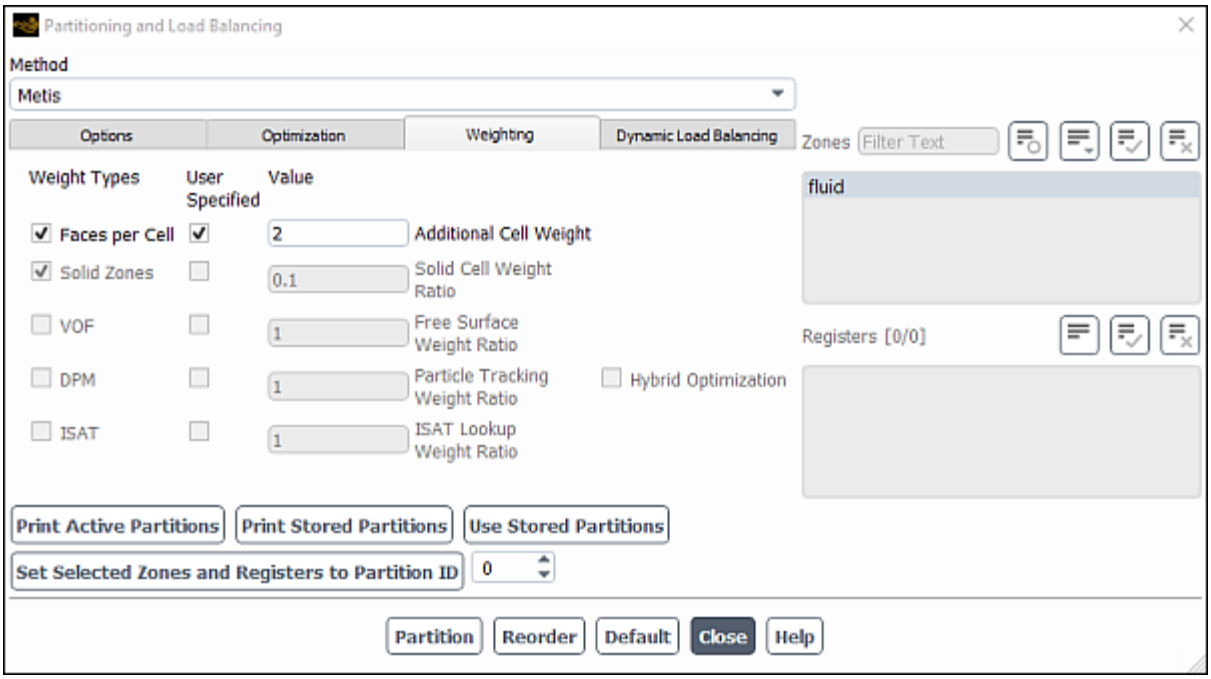
   If you are running the parallel solver, and you have marked your region and assigned an ID to the selected **Zones** and/or **Registers**, click the **Use Stored Partitions** button to make the new partitions valid.

   Refer to the example described later in this section for a demonstration of how selected registers are assigned to a partition (Example of Setting Selected Cell Registers to Specified Partition IDs (p. 4325)).

6. In the **Weighting** tab (Figure 43.8: The Weighting Tab in the Partitioning and Load Balancing Dialog Box (p. 4323)), you can set the appropriate weights prior to partitioning the mesh, to improve load balancing and overall performance. You can control weights for cells, solid cell zones, VOF, DPM, and ISAT table lookup. You can rely on Ansys Fluent timers to set the weight scaling and optionally modify it (by enabling the **User Specified** option); alternatively, you can use model-weighted partitioning, so that Fluent automatically calculates the weighting based on the cell count and the models and attributes used as weights.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 43.8: The Weighting Tab in the Partitioning and Load Balancing Dialog Box



a. Enable **Faces per Cell** so that the partitioning assigns a weight to each cell based on its number of faces. This type of weighting is advantageous when the case has mixed or polyhedral cell zones. If you enable the **User Specified** check box, the weight assigned to each cell will be the number of faces plus the **Additional Cell Weight** you enter in the number-entry box under **Value**. By default the **Faces per Cell** weighting is enabled with the **Additional Cell Weight** set to **2**.

b. Enable **Solid Zones** weighting so that the partitioning takes solid cells into consideration. If you enable the **User Specified** check box, you can specify a **Value** for the **Solid Cell Weight Ratio**. This value is relative to the fluid cell weighting; typically, it should be less than 1, since the calculation is usually quicker and less computationally expensive for the solid zone compared to the fluid zone. When using model-weighted partitioning, the default value of `0.1` is appropriate, otherwise a larger value may be more suitable. For cases that have solid zones, the **Solid Zones** weighting is enabled by default.

> **Note:**
>
> The solid cell count must be within 30–70% of the total cell count, otherwise the model-weighted partitioning will not consider solid cells.

c. Enable **VOF** weighting to allow the partitioning to consider the imbalance caused by the free surface reconstruction with the geo-reconstruct scheme. Therefore, it is only available when using the VOF model with geometric reconstruction. You may use the user-specified value before timers are collected, or if you want to specify a value other than timing statistics. The specified value is the VOF proportion of the total computational effort.

d. Enable **DPM** weighting to set the weight of DPM particles relative to the continuous phase. DPM weights are valid when you have particle tracking in your simulation, where the user-

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

specified value is the DPM proportion of the total computational effort relative to the continuous phase. Note that this is available only when you have injections defined. For details, see Modeling Discrete Phase (p. 2671).

The DPM weight takes into account the distribution of the tracking effort over the partitions and it is available after at least one calculation step with particle tracking. Displaying Particle Tracks does not change the weights. The computational effort is determined by the number of DPM steps performed in each cell. This weight becomes more important when the time for the particle tracking of particles exceeds the time for solving the flow. Enabling this option in the **Weighting** tab enables the counting of the particle steps in the cells. These values are available for contour and vector plots when using the **Discrete Phase Model** and **DPM Steps per Cell** variable. After repartitioning, the DPM weights are reset before the next particle tracking. It is generally preferable to partition along the dominant path of the particles in order to minimize particles crossing partition boundaries and thereby reducing associated communication costs. However, partitioning should also consider load balance for the other models, especially the continuous phase, and model weighting provides a means to effectively load balance the overall simulation.

Select the **Hybrid Optimization** option to enable the hybrid optimization partition weighting method for DPM. This method balances the load across machines, and, within each machine, the hybrid parallel DPM method is used to make sure the load is balanced by multi-threading. First, the domain is split based on the model weights of each cell and then partitioned across a number of machines. Finally, each machine is partitioned according to the number of cores. This allows you to have a balanced number of cells in each partition, at the same time having a balanced number of particles on each machine, which will be further balanced by the hybrid DPM method. This optimization option is also applicable to the discrete element method (DEM) collision model.

e. Enable **ISAT** weighting to balance the load during the ISAT table lookup for the stiff-chemistry Laminar, EDC or PDF Transport models. The ISAT algorithm builds an unstructured table in N species dimensions for storage and retrieval of the chemistry mappings. Since chemistry is usually computationally expensive, this storage/retrieval can be very time-consuming (for information about ISAT, refer to In-Situ Adaptive Tabulation (ISAT) in the Theory Guide). Each parallel node builds its own table, and there is no message passing to tables on other nodes. As some nodes may have more chemical reactions than others (for example one parallel node may contain just air at a constant temperature, in which case the ISAT table will contain only one entry and calculation will be rapid), there may be a load imbalance. The dynamic load balancing algorithm will migrate cells from high computational load nodes to low computational load nodes.

   If you decide to specify a value, this user-specified value is the ISAT proportion of the total computational effort.

f. For the **Metis** partition method, you have the option of using model-weighted partitioning. The objective of model-weighted partitioning is to balance the overall number of cells, as well as the time needed for the selected models (that is, the enabled **Weight Types**, described previously). This is specifically useful for cases that could potentially lead to a load imbalance—for example: when using the discrete phase model, as the distribution of particles could be different across partitions and thus cause an imbalance; or for cases with a large proportion of solid zones, as solid zones require less processing time / expense than fluid zones.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Each model is considered as a constraint, and a base constraint is automatically introduced for the overall number of cells. When partitioning, ANSYS Fluent will automatically calculate the weights for these constraints, and balance each of them.

To use model-weighted partitioning, ensure that the **Metis** partition method is selected, and enable the appropriate models under **Weight Types** in the **Weighting** tab; note that for **VOF**, **DPM**, and **ISAT**, the associated **User Specified** and **Value** settings are not relevant). Then make sure that the following text command is enabled (which it is by default) prior to partitioning:

`parallel` → `partition` → `set` → `model-weighted-partition`

Note that you can get additional information specific to the constraints by setting the **Reporting Verbosity** to 2 in the **Options** tab.

7. When using the dynamic mesh model in your parallel simulations, the **Partition** dialog box includes an **Auto Repartition** option and a **Repartition Interval** setting. These parallel partitioning options are provided because Ansys Fluent migrates cells when remeshing and smoothing are performed. Therefore, the partition interface becomes very wrinkled and the load balance may deteriorate. By default, the **Auto Repartition** option is selected, where a percentage of interface faces and loads are automatically traced. When this option is selected, Ansys Fluent automatically determines the most appropriate repartition interval based on various simulation parameters. Sometimes, using the **Auto Repartition** option provides insufficient results, therefore, the **Repartition Interval** setting can be used. The **Repartition Interval** setting lets you to specify the interval (in time steps or iterations respectively) when a repartition is enforced. When repartitioning is not desired, you can set the **Repartition Interval** to zero.

Important:

Note that when dynamic meshes and remeshing is utilized, updated meshes may be slightly different in parallel Ansys Fluent (when compared to serial Ansys Fluent or when compared to a parallel solution created with a different number of compute nodes), resulting in very small differences in the solutions.

---

8. Click the **Partition** button to partition the mesh.

9. Click the **Use Stored Partitions** button if you decide that the new partitions are better than the previous ones (if the mesh was already partitioned). This makes the newly stored cell partitions the active cell partitions. The active cell partition is used for the current calculation, while the stored cell partition (the last partition performed) is used when you save a case file.

43.5.4.1.1. Example of Setting Selected Cell Registers to Specified Partition IDs

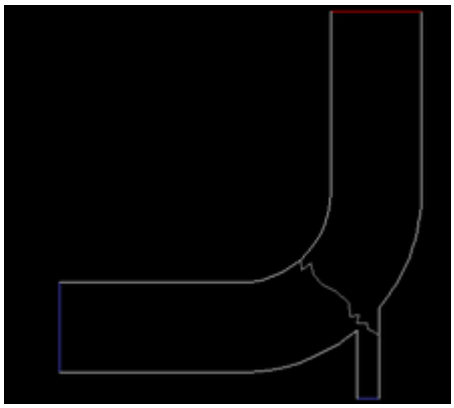
1. Start Ansys Fluent in parallel. The case in this example was partitioned across two nodes.

2. Read in your case.

3. Display the mesh with the **Partitions** option enabled in the **Mesh Display** dialog box (Figure 43.9: The Partitioned Mesh (p. 4326)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 43.9: The Partitioned Mesh



4. Mark your cells using a region cell register (for details, see Region (p. 3852)). This creates a cell register.

5. Open the **Partitioning and Load Balancing** dialog box.

6. Set the **Set Selected Zones and Registers to Partition ID** to `0` and click the corresponding button. This displays the following output in the Ansys Fluent console:

```
>> 2 Active Partitions:
  ------------------------------------------------------------------------
  Collective Partition Statistics:      Minimum      Maximum   Total
  ------------------------------------------------------------------------
  Cell count                            459          459       918
  Mean cell count deviation             0.0%         0.0%
  Partition boundary cell count         11           11        22
  Partition boundary cell count ratio   2.4%         2.4%      2.4%

  Face count                            764         1714       2461
  Mean face count deviation             -38.3%       38.3%
  Partition boundary face count         13          13        17
  Partition boundary face count ratio   0.8%        1.7%      0.7%

  Partition neighbor count              1           1
  ------------------------------------------------------------------------
  Partition Method                      Metis
  Stored Partition Count                2
Done.
```

7. Click the **Use Stored Partitions** button to make the new partitions valid. This migrates the partitions to the compute-nodes. The following output is then displayed in the Ansys Fluent console:

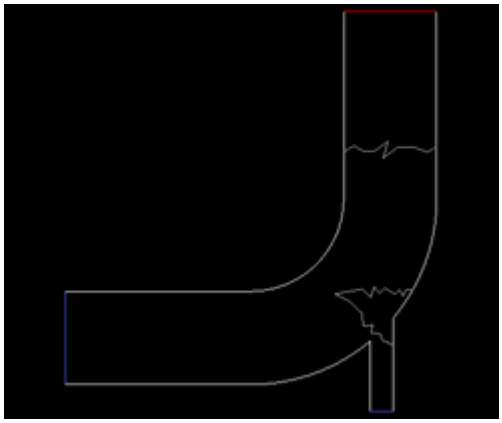
```
Migrating partitions to compute-nodes.
>> 2 Active Partitions:
     P   Cells  I-Cells  Cell Ratio   Faces  I-Faces  Face Ratio  Neighbors
     0     672      24        0.036    2085       29      0.014          1
     1     246      24        0.098     425       29      0.068          1
  ------------------------------------------------------------------------
  Collective Partition Statistics:     Minimum       Maximum      Total
  ------------------------------------------------------------------------
  Cell count                           246           672          918
  Mean cell count deviation            -46.4%        46.4%
  Partition boundary cell count        24            24           48
  Partition boundary cell count ratio  3.6%          9.8%         5.2%
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
  Face count                          425           2085          2461
  Mean face count deviation           -66.1%        66.1%
  Partition boundary face count       29            29            49
  Partition boundary face count ratio 1.4%          6.8%          2.0%

  Partition neighbor count            1             1
  ------------------------------------------------------------------------
  Partition Method                    Metis
  Stored Partition Count              2
Done.
```

8. Display the mesh (Figure 43.10: The Partitioned ID Set to Zero (p. 4327)).

Figure 43.10: The Partitioned ID Set to Zero



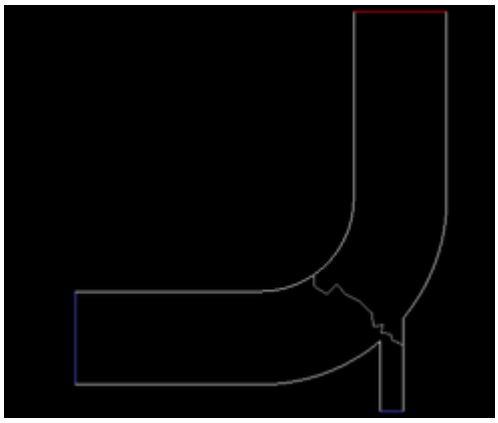
9. This time, set the **Set Selected Zones and Registers to Partition ID** to 1 and click the corresponding button. This displays a report in the Ansys Fluent console.

10. Click the **Use Stored Partitions** button to make the new partitions valid and to migrate the partitions to the compute-nodes.

11. Display the mesh (Figure 43.11: The Partitioned ID Set to 1 (p. 4328)). Notice now that the partition appears in a different location as specified by your partition ID.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 43.11: The Partitioned ID Set to 1



Important:

Although this example demonstrates setting selected registers to specific partition IDs in parallel, it can be similarly applied in serial.

43.5.4.1.2. Partitioning Within Zones or Registers

The ability to restrict partitioning to cell zones or registers gives you the flexibility to apply different partitioning strategies to subregions of a domain. For example, if your geometry consists of a cylindrical plenum connected to a rectangular duct, you may want to partition the plenum using the **Cylindrical Axes** method, and the duct using the **Cartesian Axes** method.

If the plenum and the duct are contained in two different cell zones, you can select one at a time and perform the desired partitioning, as described in Using the Partitioning and Load Balancing Dialog Box (p. 4319). If they are not in two different cell zones, you can create a cell register (basically a list of cells) for each region using the functions that are used to mark cells for adaption. These functions allow you to mark cells based on physical location, cell volume, gradient or isovalue of a particular variable, and other parameters. See Adapting the Mesh (p. 3773) for information about marking cells for adaption. Using Cell Registers (p. 3851) provides information about creating cell registers. Once you have created a cell register, you can partition within it as described in Example of Setting Selected Cell Registers to Specified Partition IDs (p. 4325).

Important:

Note that partitioning within zones or registers is not available when **Metis** is selected as the partition **Method**.

For dynamic mesh applications, Ansys Fluent stores the partition method used to partition the respective zone. Therefore, if repartitioning is done, Ansys Fluent uses the same method that was used to partition the mesh.

43.5.4.1.3. Reporting During Partitioning

As the mesh is partitioned, information about the partitioning process will be displayed in the console. By default, the number of partitions created, the time required for the partitioning, and

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the minimum and maximum cell, face, interface, and face-ratio variations will be displayed (for details, see Interpreting Partition Statistics (p. 4339)). If you increase the **Reporting Verbosity** to 2 from the default value of 1, the partition method used, the partition ID, number of cells, faces, and interfaces, and the ratio of interfaces to faces for each partition will also be displayed in the console. If you decrease the **Reporting Verbosity** to 0, only the number of partitions created and the time required for the partitioning will be reported.

You can request a portion of this report to be displayed again after the partitioning is completed. When you click the **Print Active Partitions** or **Print Stored Partitions** button in the serial or parallel solver, Ansys Fluent will display the partition ID, number of cells, faces, and interfaces, and the ratio of interfaces to faces for each active or stored partition in the console. In addition, it will display the minimum and maximum cell, face, interface, and face-ratio variations. For details, see Interpreting Partition Statistics (p. 4339).

Important:

Recall that to make the stored cell partitions the active cell partitions you must click the **Use Stored Partitions** button. The active cell partition is used for the current calculation, while the stored cell partition (the last partition performed) is used when you save a case file.

43.5.4.1.4. Resetting the Partition Parameters

If you change your mind about your partition parameter settings, you can easily return to the default settings assigned by Ansys Fluent by clicking on the **Default** button. When you click the **Default** button, it will become the **Reset** button. The **Reset** button allows you to return to the most recently saved settings (that is, the values that were set before you clicked on **Default**). After execution, the **Reset** button will become the **Default** button again.

43.5.4.2. Load Balancing

A dynamic load balancing capability is available in Ansys Fluent. The principal reason for using parallel processing is to reduce the turnaround time of your simulation, which may be achieved by the following means:

- Faster machines, for example, faster CPU, memory, cache, and communication bandwidth between the CPU and memory
- Faster interconnects, for example, smaller latency and larger bandwidth
- Better Load balancing, for example, load is evenly distributed and CPUs are not idled during calculation

The first two evolve at the pace of computer technology, which is beyond the scope of this document. The third item is regarding optimization of available computation power. Here we are mainly talking about load balancing on dedicated homogeneous resources, which is often the case nowadays. If you are not using a dedicated homogeneous resource, you may need to account for differences in CPU speeds during partitioning by specifying a load distribution (for details, see Load Distribution (p. 4342)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

On a dedicated homogeneous system, the key for load balancing is how to evaluate the computational requirement of each cell. By default, Ansys Fluent assumes that each cell requires the same computational work, but this is often not the case. For example

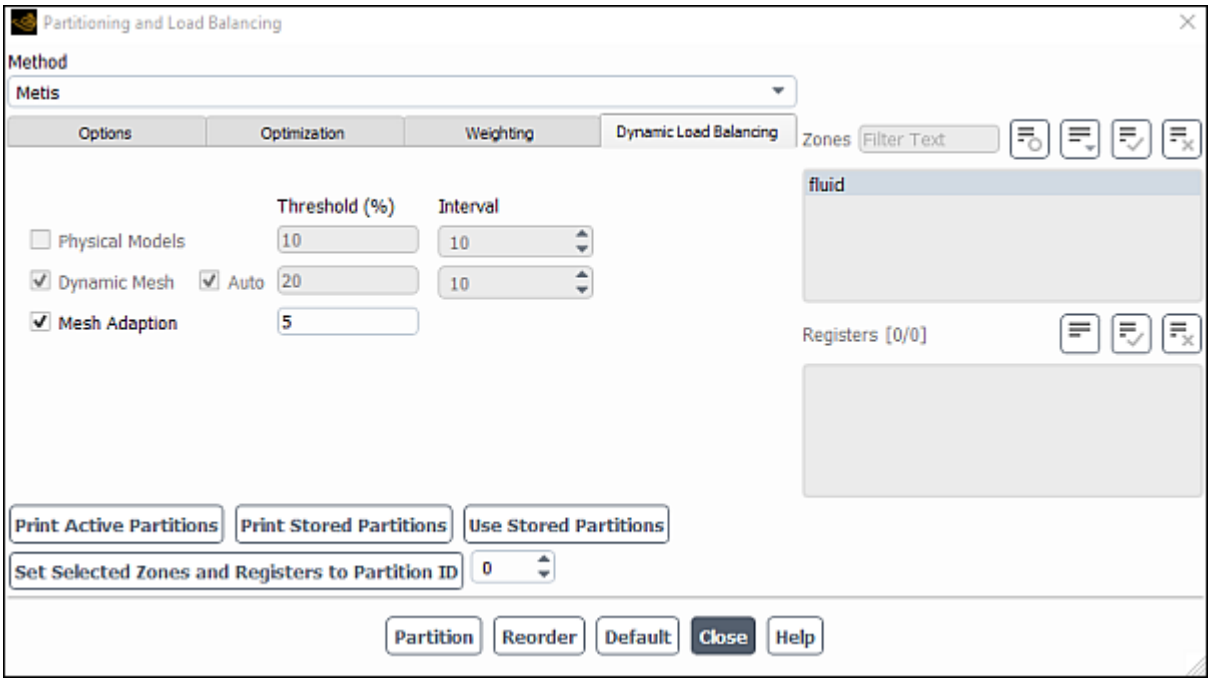
- A hexahedral cell demands more CPU and memory than a tetrahedral cell.
- A cell with particle tracking will use more time than a cell without particle tracking.
- ISAT species model cells may have magnitude differences in time usage.

To balance these differences, ideally, the time used in each cell could be recorded and load balance achieved based on these detailed timing statistics. However, this can be expensive and such low level timings can be unreliable in any case. Instead, we identify features causing computational imbalance and record time usage for these models in aggregate. For a more detailed description of this, refer to Partitioning (p. 4319) in the discussion of the **Weighting** tab. In addition, the imbalance may happen dynamically during run time, for example

- The mesh may be changed by adaption or mesh movement.
- In unsteady cases, particle tracking may move from one region to another region.

Dynamic load balancing has been implemented for better scalability of cases with imbalanced physical or geometrical models, thereby reducing the simulation time. The implementation considers weights from these models scaled by CPU time usage. Load balancing for DPM, VOF, cell type (number of faces per cell), and solid zones can be performed. In addition, cell weight based load balancing and machine load distribution can also be specified (for details, see Load Distribution (p. 4342)). Ansys Fluent takes the weights from physical models and considers them for partitioning. The weights are assembled based on the time used by each physical model. For dynamic load balancing, the load is checked and balanced based on your specified imbalance threshold. To apply dynamic load balancing on the various models, click the **Dynamic Load Balancing** tab and select the required balancing as follows:

Figure 43.12: The Dynamic Load Balancing Tab



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Enable **Physical Models** load balancing during iterations so that the load will be evaluated for time usage and weight distribution, based on the **Interval** that you provide. If the imbalance exceeds the specified **Threshold**, then repartitioning will be performed by considering the selected weights. **Physical Models** load balancing will only be available when you have the specific physical models enabled in the case. You will be prompted to enable the weights for those models. When weights for the physical models are all disabled, you will be prompted to disable **Physical Models** load balancing.

   Note:

   Applying load balancing too frequently may cause performance degradation due to the additional cost of migrating cells for the new partition layout.

2. Enable **Dynamic Mesh** if there is any dynamic mesh movement. Load balancing, based on the number of cells, will be checked and balanced if the imbalance threshold is exceeded. These parallel partitioning options are provided because with mesh motion, when remeshing and smoothing are performed, the partition interface can become very wrinkled and load balance may deteriorate. By default, the **Auto** option is selected, where a percentage of interface faces and loads are automatically traced. When this option is selected, Ansys Fluent automatically determines the most appropriate repartitioning interval based on various simulation parameters. However, sometimes, the frequency of load balancing from the **Auto** option may be inadequate, and then the **Interval** setting can be explicitly set. The **Interval** setting lets you specify the interval (in time steps or iterations, respectively) when load balancing is enforced. When load balancing is not desired, you may disable **Dynamic Mesh** load balancing. Dynamic Mesh load balancing is only available when you have dynamic models enabled in your case.

   Important:

   Note that when dynamic meshes and remeshing are utilized, updated meshes may be slightly different in parallel Ansys Fluent (when compared to serial Ansys Fluent or when compared to a parallel solution created with a different number of compute nodes), resulting in very small differences in the solutions.

3. Enable **Mesh Adaption**. Any time mesh adaption occurs, load balancing, based on the number of cells, will be checked and balanced if the imbalance threshold is exceeded. If problems arise in your computations due to adaption, you can disable the load balancing for **Mesh Adaption**.

43.5.5. Mesh Partitioning Methods

Partitioning the mesh for parallel processing has three major goals:

- Create partitions with equal numbers of cells.
- Minimize the number of partition interfaces — that is, decrease partition boundary surface area.
- Minimize the number of partition neighbors.

Balancing the partitions (equalizing the number of cells) ensures that each processor has an equal load and that the partitions will be ready to communicate at about the same time. Since communic-

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ation between partitions can be a relatively time-consuming process, minimizing the number of interfaces can reduce the time associated with this data interchange. Minimizing the number of partition neighbors reduces the chances for network and routing contentions. In addition, minimizing partition neighbors is important on machines where the cost of initiating message passing is expensive compared to the cost of sending longer messages. This is especially true for workstations connected in a network.

The partitioning schemes in Ansys Fluent use bisection or METIS algorithms to create the partitions, but unlike other schemes that require the number of partitions to be a factor of two, these schemes have no limitations on the number of partitions. You will create as many partitions as there are computing units (cores based on processors and machines) available for your simulation.

43.5.5.1. Partition Methods

The mesh is partitioned using a bisection or METIS algorithm. The selected algorithm is applied to the parent domain, and then recursively applied to the subdomains. For example, to divide the mesh into four partitions with a bisection method, Fluent will bisect the entire (parent) domain into two child domains, and then repeat the bisection for each of the child domains, yielding four partitions in total. To divide the mesh into three partitions with a bisection method, Fluent will "bisect" the parent domain to create two partitions—one approximately twice as large as the other—and then bisect the larger child domain again to create three partitions in total. METIS uses graph partitioning techniques that generally provide more optimal partitions than the geometric methods.

The mesh can be partitioned using one of the algorithms listed below. The most efficient choice is problem-dependent, so you can try different methods until you find the one that is best for your problem. See Guidelines for Partitioning the Mesh (p. 4319) for recommended partitioning strategies.

Cartesian Axes

bisects the domain based on the Cartesian coordinates of the cells (see Figure 43.13: Partitions Created with the Cartesian Axes Method (p. 4334)). It bisects the parent domain and all subsequent child subdomains perpendicular to the coordinate direction with the longest extent of the active domain. It is often referred to as coordinate bisection.

Cartesian Strip

uses coordinate bisection but restricts all bisections to the Cartesian direction of longest extent of the parent domain (see Figure 43.14: Partitions Created with the Cartesian Strip or Cartesian X-Coordinate Method (p. 4335)). You can often minimize the number of partition neighbors using this approach.

Cartesian X-, Y-, Z-Coordinate

bisects the domain based on the selected Cartesian coordinate. It bisects the parent domain and all subsequent child subdomains perpendicular to the specified coordinate direction. (See Figure 43.14: Partitions Created with the Cartesian Strip or Cartesian X-Coordinate Method (p. 4335).)

Cartesian R Axes

bisects the domain based on the shortest radial distance from the cell centers to that Cartesian axis (x, y, or z) whichever produces the smallest interface size. This method is available only in 3D.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cartesian RX-, RY-, RZ-Coordinate

bisects the domain based on the shortest radial distance from the cell centers to the selected Cartesian axis (x, y, or z). These methods are available only in 3D.

Cylindrical Axes

bisects the domain based on the cylindrical coordinates of the cells. This method is available only in 3D.

Cylindrical R-, Theta-, Z-Coordinate

bisects the domain based on the selected cylindrical coordinate. These methods are available only in 3D.

Metis

uses the METIS software package for partitioning irregular graphs, developed by Karypis and Kumar at the University of Minnesota and the Army HPC Research Center. It uses a multilevel approach in which the vertices and edges on the fine graph are coalesced to form a coarse graph. The coarse graph is partitioned, and then uncoarsened back to the original graph. During coarsening and uncoarsening, algorithms are applied to permit high-quality partitions. METIS routines can handle partitioning with model-weighted multiple constraints: when automatically partitioning, solid cell zones are weighted with a default value of 0.1 relative to the fluid cell weighting; when manually partitioning, you can control weighting for cells, solid cell zones, VOF, DPM, and ISAT table lookup (as described in Partitioning (p. 4319)). Detailed information about METIS can be found in [75] (p. 5717) and [76] (p. 5717).

Important:

If you create non-conformal interfaces, and generate virtual polygonal faces, your METIS partition can cross non-conformal interfaces by using the connectivity of the virtual polygonal faces. This improves load balancing for the parallel solver and minimizes communication by decreasing the number of partition interface cells.

Polar Axes

bisects the domain based on the polar coordinates of the cells (see Figure 43.17: Partitions Created with the Polar Axes or Polar Theta-Coordinate Method (p. 4336)). This method is available only in 2D.

Polar R-Coordinate, Polar Theta-Coordinate

bisects the domain based on the selected polar coordinate (see Figure 43.17: Partitions Created with the Polar Axes or Polar Theta-Coordinate Method (p. 4336)). These methods are available only in 2D.

Principal Axes

bisects the domain based on a coordinate frame aligned with the principal axes of the domain (see Figure 43.15: Partitions Created with the Principal Axes Method (p. 4335)). This reduces to Cartesian bisection when the principal axes are aligned with the Cartesian axes. The algorithm is also referred to as moment, inertial, or moment-of-inertia partitioning.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This is the default bisection method in Ansys Fluent.

Principal Strip

uses moment bisection but restricts all bisections to the principal axis of longest extent of the parent domain (see Figure 43.16: Partitions Created with the Principal Strip or Principal X-Coordinate Method (p. 4336)). You can often minimize the number of partition neighbors using this approach.

Principal X-, Y-, Z-Coordinate

bisects the domain based on the selected principal coordinate (see Figure 43.16: Partitions Created with the Principal Strip or Principal X-Coordinate Method (p. 4336)).

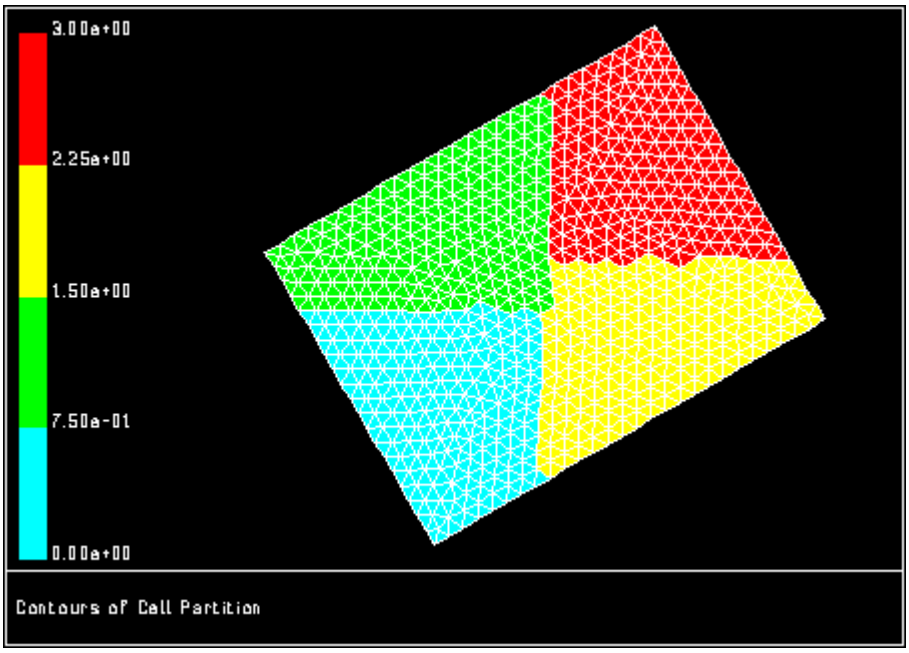
Spherical Axes

bisects the domain based on the spherical coordinates of the cells. This method is available only in 3D.

Spherical Rho-, Theta-, Phi-Coordinate

bisects the domain based on the selected spherical coordinate. These methods are available only in 3D.

Figure 43.13: Partitions Created with the Cartesian Axes Method



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 43.14: Partitions Created with the Cartesian Strip or Cartesian X-Coordinate Method

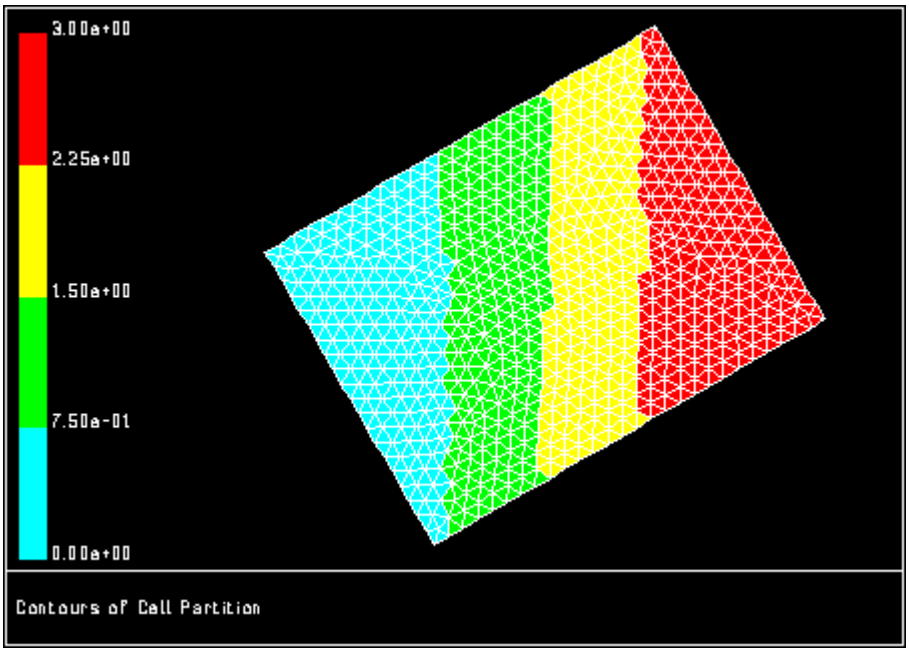
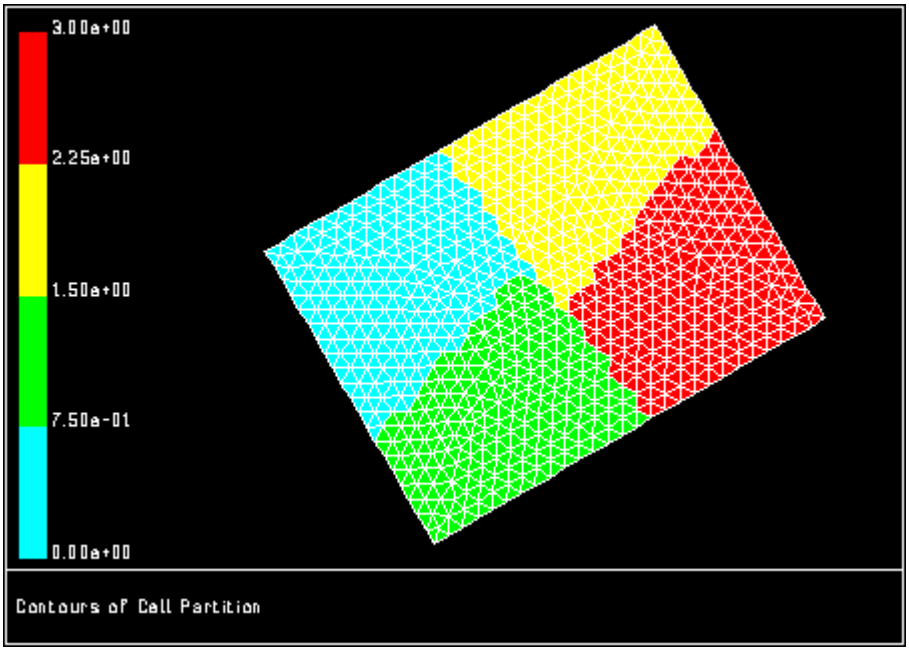


Figure 43.15: Partitions Created with the Principal Axes Method



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 43.16: Partitions Created with the Principal Strip or Principal X-Coordinate Method

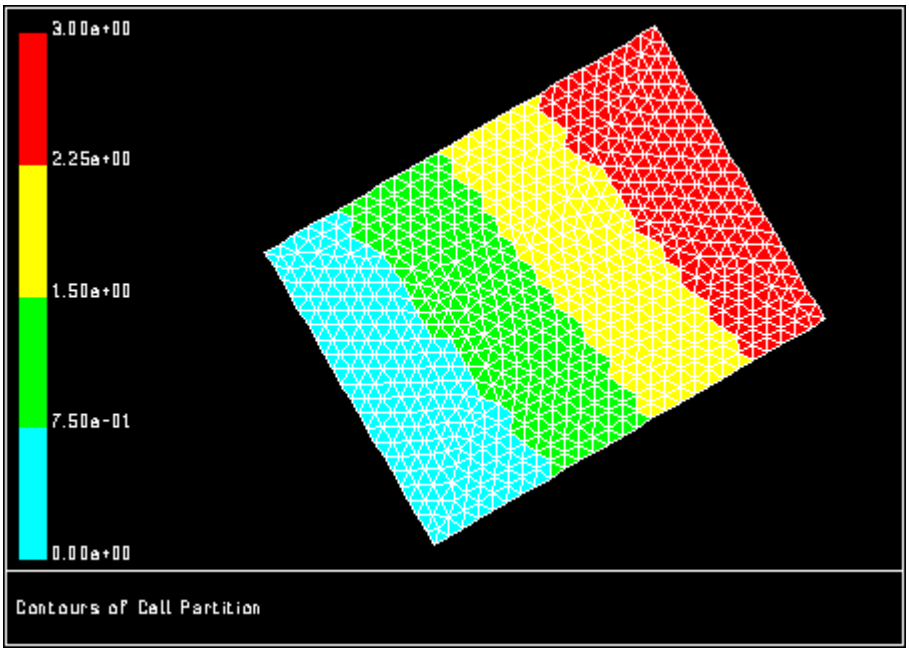
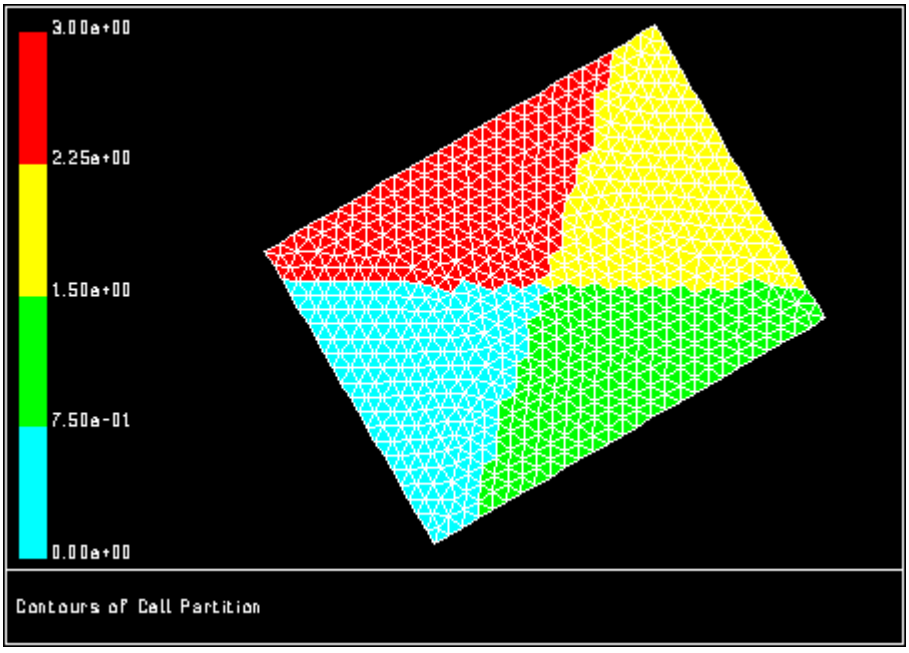


Figure 43.17: Partitions Created with the Polar Axes or Polar Theta-Coordinate Method



43.5.5.2. Optimizations

Additional optimizations can be applied to improve the quality of the mesh partitions. The heuristic of bisecting perpendicular to the direction of longest domain extent is not always the best choice for creating the smallest interface boundary. A pre-testing operation (for details, see Pretesting (p. 4337)) can be applied to automatically choose the best direction before partitioning. In addition, the following iterative optimization schemes exist:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Smooth

attempts to minimize the number of partition interfaces by swapping cells between partitions. The scheme traverses the partition boundary and gives cells to the neighboring partition if the interface boundary surface area is decreased. (See Figure 43.18: The Smooth Optimization Scheme (p. 4337).)

Merge

attempts to eliminate orphan clusters from each partition. An orphan cluster is a group of cells with the common feature that each cell within the group has at least one face that coincides with an interface boundary. (See Figure 43.19: The Merge Optimization Scheme (p. 4337).) Orphan clusters can degrade multigrid performance and lead to large communication costs.

Figure 43.18: The Smooth Optimization Scheme

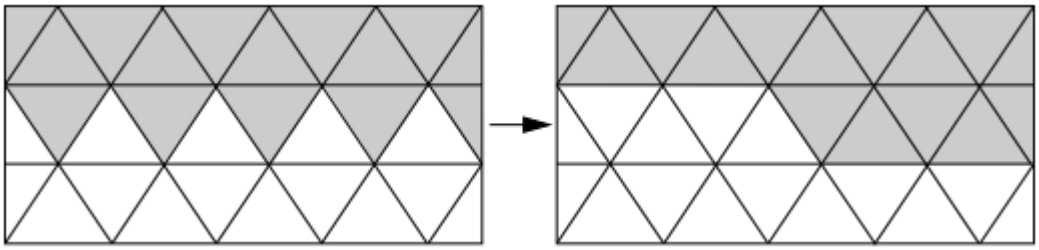
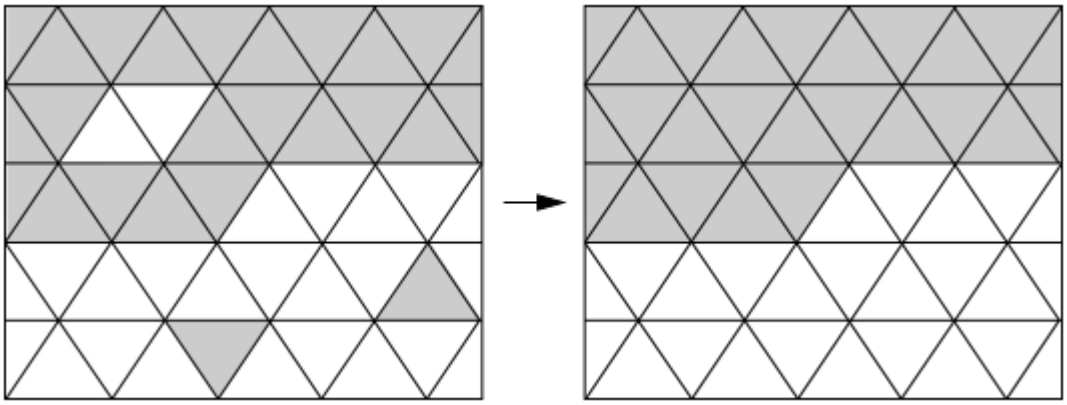


Figure 43.19: The Merge Optimization Scheme



In general, the Smooth and Merge schemes are relatively inexpensive optimization tools.

43.5.5.3. Pretesting

If you choose the **Principal Axes** or **Cartesian Axes** method, you can improve the bisection by testing different directions before performing the actual bisection. If you choose not to use pretesting (the default), Ansys Fluent will perform the bisection perpendicular to the direction of longest domain extent.

If pretesting is enabled, it will occur automatically when you click the **Partition** button in the Partitioning and Load Balancing Dialog Box (p. 5567), or when you read in the mesh if you are using automatic partitioning. The bisection algorithm will test all coordinate directions and choose the one which yields the fewest partition interfaces for the final bisection.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that using pretesting will increase the time required for partitioning. For 2D problems partitioning will take 3 times longer than without pretesting, and for 3D problems it will take 4 times longer.

43.5.5.4. Using the Partition Filter

As noted above, you can use the METIS partitioning method through a filter in addition to within the **Auto Partition Mesh** and **Partitioning and Load Balancing** dialog boxes. To perform METIS partitioning on an unpartitioned mesh, use the **File/Import/Partition/Metis...** ribbon tab item.

File → **Import** → **Partition** → **Metis...**

Ansys Fluent will use the METIS partitioner to partition the mesh, and then read the partitioned mesh. The number of partitions will be equal to the number of processes. You can then proceed with the model definition and solution.

Important:

Direct import to the parallel solver through the partition filter requires that the host machine has enough memory to run the filter for the specified mesh. If not, you must run the filter on a machine that does have enough memory. You can either start the parallel solver on the machine with enough memory and repeat the process described above, or run the filter manually on the new machine and then read the partitioned mesh into the parallel solver on the host machine.

To manually partition a mesh using the partition filter, enter the following command:

`utility partition` *input_filename partition_count output_filename*

where *input_filename* is the filename for the mesh to be partitioned, *partition_count* is the number of partitions desired, and *output_filename* is the filename for the partitioned mesh. You can then read the partitioned mesh into Fluent (using the standard **File/Read/Case...** ribbon tab item) and proceed with the model definition and solution.

When the **File/Import/Partition/Metis...** ribbon tab item is used to import an unpartitioned mesh into the parallel solver, the METIS partitioner partitions the entire mesh. You may also partition each cell zone individually, using the **File/Import/Partition/Metis Zone...** ribbon tab item.

File → **Import** → **Partition** → **Metis Zone...**

This method can be useful for balancing the work load for cases that have few cell zones.

43.5.6. Checking the Partitions

After partitioning a mesh, you should check the partition information and examine the partitions graphically.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

43.5.6.1. Interpreting Partition Statistics

You can request a report to be displayed after partitioning (either automatic or manual) is completed. Click the **Print Active Partitions** or **Print Stored Partitions** button in the **Partitioning and Load Balancing** dialog box.

Ansys Fluent distinguishes between two cell partition schemes: the active cell partitions and the stored cell partitions. Initially, both are set to the cell partitions that were established upon reading the case file. If you re-partition the mesh using the **Partitioning and Load Balancing** dialog box, the new partitions will be referred to as the stored cell partitions. To make them the active cell partitions, you must click the **Use Stored Partitions** button in the **Partitioning and Load Balancing** dialog box in the parallel version of Ansys Fluent. The active cell partitions are used for the current calculation, while the stored cell partitions (determined from the last partitioning performed) are used when you save a case file. This distinction is made mainly to allow you to partition a case on one machine or network of machines and solve it on a different one. Thanks to the two separate partitioning schemes, you could use the parallel solver with a certain number of compute nodes to subdivide a mesh into an arbitrary different number of partitions, suitable for a different parallel machine, save the case file, and then load it into the designated machine.

The output generated when you print the partitions consists of tabulated information about the active or stored partitioning scheme. A typical output for a mesh with 4 partitions is as follows:

```
>> 4 Active Partitions:
    P   Cells I-Cells Cell Ratio   Faces I-Faces Face Ratio Neighbors Load
    0    3520     142      0.040   11399     195      0.017         1    1
    1    3298     115      0.035   10678     151      0.014         1    1
    2    3451     305      0.088   11404     372      0.033         2    1
    3    3583     332      0.093   11586     416      0.036         2    1

   ------------------------------------------------------------------------
   Collective Partition Statistics:        Minimum   Maximum   Total
   ------------------------------------------------------------------------
   Cell count                              3298      3583      13852
   Mean cell count deviation               -4.8%     3.5%
   Partition boundary cell count           115       332       894
   Partition boundary cell count ratio     3.5%      9.3%      6.5%

   Face count                              10678     11586     44500
   Mean face count deviation               -5.2%     2.8%
   Partition boundary face count           151       416       567
   Partition boundary face count ratio     1.4%      3.6%      1.3%

   Partition neighbor count                1         2
   ------------------------------------------------------------------------
   Partition Method                        Metis
   Stored Partition Count                  4
```

The first table in the output displays per-partition statistics of interest:

P

the partition ID

Cells

the number of cells in the partition

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

I-Cells

the number of interface cells in the partition (that is, cells that lie on the partition interfaces)

Cell Ratio

the ratio of interface cells to total cells for the partition

Faces

the number of faces in the partition

I-Faces

the number of interface faces in the partition (that is, faces that lie on partition interfaces)

Face Ratio

the ratio of interface faces to total faces for the partition

Neighbors

the number of neighbor partitions

Load

the desired relative load on this node in proportion to the other nodes. See Load Distribution (p. 4342) for details.

Note that partition IDs correspond directly to compute node IDs when a case file is read into the parallel solver. When the number of partitions in a case file is larger than the number of compute nodes, but is evenly divisible by the number of compute nodes, then the distribution is such that partitions with IDs 0 to $(M-1)$ are mapped onto compute node 0, partitions with IDs M to $(2M-1)$ onto compute node 1, and so on, where M is equal to the ratio of the number of partitions to the number of compute nodes.

The second table in the output displays `Minimum`, `Maximum`, and (where applicable) `Total` values for various partition statistics:

Cell Count

the number of cells in the partitions (corresponding to `Cells` in the per-partition table)

Mean cell count deviation

the deviation of an individual partition cell count from the mean partition cell count

Partition boundary cell count

the number of cells that lie on partition interfaces (corresponding to `I-Cells` in the per-partition table)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Partition boundary cell count ratio

the ratio of the number of cells that lie on partition interfaces to the total number of cells in the partition (corresponding to `Cell Ratio` in the per-partition table)

Face Count

the number of faces in the partitions (corresponding to `Faces` in the per-partition table)

Mean face count deviation

the deviation of an individual partition face count from the mean partition face count

Partition boundary face count

the number of faces that lie on partition interfaces (corresponding to `I-Faces` in the per-partition table)

Partition boundary face count ratio

the ratio of the number of faces that lie on partition interfaces to the total number of faces in the partition (corresponding to `Face Ratio` in the per-partition table)

Partition neighbor count

the number of neighbors for a given partition (corresponding to `Neighbors` in the per-partition table)

Finally, the `Partition Method` and `Stored Partition Count` are displayed.

Your aim is to minimize the `Maximum` value of the `Partition boundary face count ratio`, while maintaining a balanced load. For a given mesh, the `Mean cell count deviation` is a measure of the maximum load imbalance for a method. If the statistics are not satisfactory for a problem, you can try one of the other partitioning methods.

If there is an overset mesh for which the solution has been initialized, an additional partition table with solve and dead cells is included in the partitioning report (see Overset Cell Marks (p. 1233) for the definitions of such cells):

```
>> Overset partition statistics:
    P   Cells Solve-cells Dead-cells  Ext donors
    0      66          53          4          12
    1      66          55          4          10

   ------------------------------------------------------------------------
   Overset Partition Statistics:           Minimum   Maximum   Total
   ------------------------------------------------------------------------
   Cell count                              66        66        132
   Mean cell count deviation               0.0%      0.0%

   Solve cell count                        53        55        108
   Mean solve cell count deviation         -1.9%     1.9%

   Dead cell count                         4         4         8
   Mean dead cell count deviation          0.0%      0.0%

   Ext donors                              10        12        22
   ------------------------------------------------------------------------
```

```
   Partition Method                        Metis
   Stored Partition Count                  2
```

43.5.6.2. Examining Partitions Graphically

To further aid interpretation of the partition information, you can draw contours of the mesh partitions (see the figures in Partition Methods (p. 4332)).

Results → **Graphics** → **Contours** **Edit...**

To display the active cell partition or the stored cell partition (which were described above), select **Active Cell Partition** or **Stored Cell Partition** in the **Cell Info...** category of the **Contours Of** drop-down list, and turn off the display of **Node Values** (for details, see Displaying Contours and Profiles (p. 3896) for information about displaying contours).

Important:

If you have not already done so in the setup of your problem, you must perform a solution initialization in order to use the **Contours** dialog box.

43.5.7. Load Distribution

If the speeds of the processors that will be used for a parallel calculation differ significantly, you can specify a load distribution for partitioning, using the `load-distribution` text command.

`parallel` → `partition` → `set` → `load-distribution`

For example, if you will be solving on three compute nodes, and one machine is twice as fast as the other two, then you may want to assign twice as many cells to the first machine as to the others (that is, a load vector of (2 1 1)). During subsequent mesh partitioning, partition 0 will end up with twice as many cells as partitions 1 and 2.

For this example, you need to start up Ansys Fluent such that compute node 0 is the fast machine, since partition 0, with twice as many cells as the others, will be mapped onto compute node 0. Alternatively, in this situation, you could enable the load balancing feature (described in Load Balancing (p. 4329)) to have Ansys Fluent automatically attempt to discern any difference in load among the compute nodes.

43.5.8. Troubleshooting

When running a calculation using parallel Ansys Fluent, you may encounter a warning message in the console that reports problems related to the partitioning. The following is an example of such a warning:

```
#AMG# Warning: The global matrix size (1273286) is too large, and may
adversely affect the parallel performance. See the Ansys Fluent User's
Guide for information on troubleshooting partitioning issues.
```

The following are possible reasons for partitioning problems, along with recommendations for reducing them:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The presence of solid zones may cause a partition to have a very small amount of fluid cells, or none at all. To avoid this, it is recommended that you use the **Metis** method with automatic or manual partitioning; for the latter, you must ensure that **Solid Zones** weighting and model-weighted partitioning are enabled, with a suitable **Value** for the solid-to-fluid cell weighting (for example, `0.1`).

- A partition may have a small number of cells if you have set up a load distribution for partitioning. Such settings should be disabled, by using the `load-distribution` text command (described in Load Distribution (p. 4342)) and entering a value of `1` for each of the previously defined partitions.

- Some model settings (for example, shell conduction) can encapsulate some cells, which may cause difficulties with the coarsening process. To remedy this situation, you can either try a different partitioning method, or you can enable the global coarsening checking criteria with the following rpvar setting:

  ```
  (rpsetvar 'amg/parallel/global-check-coarsening? #t)
  ```

- Coupled walls are encapsulated as part of the shell conduction model and S2S model. If you have partitioning problems, you can try reverting to the encapsulation routine used prior to version 16.0 by disabling the enhanced encapsulation:

  `define` → `models` → `shell-conduction` → `enhanced-encapsulation?`

43.6. Using General Purpose Graphics Processing Units (GPGPUs) With the Algebraic Multigrid (AMG) Solver

You can accelerate the Algebraic Multigrid (AMG) solver inside Fluent by using General Purpose Graphics Processing Units (GPGPUs) in offload mode, provided that suitable hardware is available on your compute machines. When set up, you can use GPGPU acceleration for AMG computations in a Fluent session on linear systems with up to 5 coupled equations. Using GPGPUs requires HPC licenses. Licensing details can be found in HPC Licensing in the *Ansys, Inc. Licensing Guide*.

43.6.1. Requirements

GPUs that are supported in the current release are posted on the Platform Support section of the Ansys Website.

When starting the parallel Fluent session the following conditions must be met:

- The number of solver processes per machine must be equal on all machines.

- The number solver processes per machine must be evenly divisible by the specified number of GPGPUs per machine.

43.6.2. Limitations

GPGPU acceleration for the AMG solver is subject to the following limitations:

- When using Fluent in Workbench, the number of GPGPUs cannot be set through:

  – Properties of Setup Cells

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - Properties of Solution Cells
  - Properties of Solution Cells through RSM update
  - Properties of Parameter Sets through RSM update
- GPGPU acceleration of the AMG solver will not be used in the following cases:
  - The population balance model is active.
  - The Eulerian multiphase model is active.
  - The system has more than 5 coupled equations.

43.6.3. Using and Managing GPGPUs

In order to use GPGPUs in offload mode to accelerate the AMG solver, you must specify how many GPGPUs are to be used per machine by using the **Solver GPGPUs per Machine (Offload Mode)** setting in the Fluent Launcher or by using the `-gpgpu=` *ngpgpus* command line option. For details, refer to the following sections:

Starting Parallel Ansys Fluent Using Fluent Launcher (p. 4296)
Starting Parallel Ansys Fluent on a Windows System Using Command Line Options (p. 4306)
Starting Parallel Ansys Fluent on a Linux System (p. 4311)

Once the Fluent session is running, you can view and/or select the available GPGPUs on the system using the following TUI commands:

`parallel/gpgpu/show`

Displays the available GPGPUs on the system.

`parallel/gpgpu/select`

Selects the GPGPUs to use in offload mode. Note that you can only select up to the number of GPGPUs that you specified on the command line or in Fluent Launcher when starting the session.

By default, GPGPU acceleration is applied automatically to coupled systems and not to scalar systems because scalar systems typically are not as computationally expensive. However, if desired you can enable/disable GPGPU acceleration of the AMG solver for coupled and scalar systems by using the following text command menu:

```
solve/set/amg-options/amg-gpgpu-options/
```

The menu contains commands for each supported equation type, allowing you to enable/disable GPGPU acceleration, choose between AMG and FGMRES solvers, and specify various solver options.

Note:

When using this text command menu, you must ensure that GPGPU acceleration is enabled for only coupled equations or only scalar equations, rather than a mix of both.

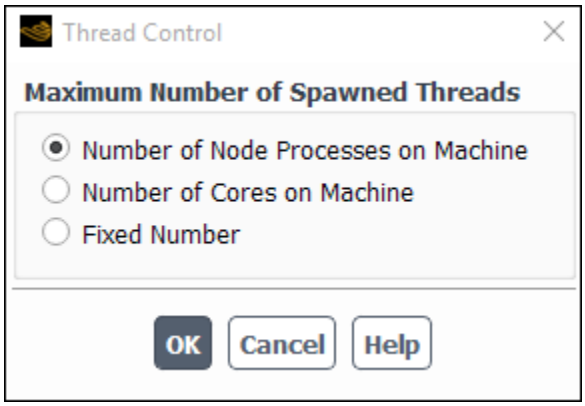
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

43.7. Controlling the Threads

You can control the maximum number of threads on each machine by using the **Thread Control** dialog box (Figure 43.21: The Parallel Connectivity Dialog Box (p. 4346)).

Parallel → General → Thread Control...

Figure 43.20: The Thread Control Dialog Box



You have the following options when using the **Thread Control** dialog box:

- **Number of Node Processes on Machine**

  This is the default option. When this option is chosen, the maximum number of threads on each machine is equal to the number of Ansys Fluent node processes on each machine.

- **Number of Cores on Machine**

  When this option is chosen, the maximum number of threads on each machine is equivalent to the number of cores on the machine. Ansys Fluent obtains the number of cores from the operating system. This may be applicable when the multi-threaded part of the calculation is dominating the computation time, and you want to take full advantage of computational resources.

  If, for example, you have a very small case with regard to the number of cells, but a large number of particles to be tracked, you may want to spawn one Ansys Fluent node process on each machine, but use the maximum number of cores in order to get a good overall performance.

- **Fixed Number**

  When this option is chosen, you may specify the maximum number of threads that can be spawned on each machine in the number-entry box below **Fixed Number**. This may only be applicable when you want to have fine control of the number of threads on each machine; it is not recommended in general.

43.8. Checking Network Connectivity

For any compute node, you can print network connectivity information that includes the hostname, architecture, process ID, and ID of the selected compute node and all machines connected to it. The ID of the selected compute node is marked with an asterisk.

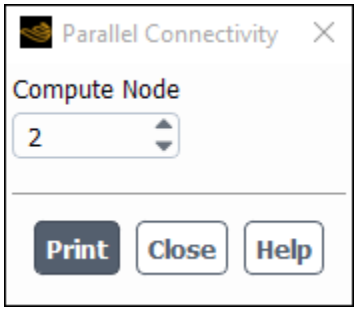
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The ID for the Ansys Fluent host process is always `host`. The compute nodes are numbered sequentially starting from `node-0`. All compute nodes are completely connected. In addition, compute node 0 is connected to the host process.

To obtain connectivity information for a compute node, you can use the Parallel Connectivity Dialog Box (p. 5557) (Figure 43.21: The Parallel Connectivity Dialog Box (p. 4346)).

Parallel → Network → Connectivity...

Figure 43.21: The Parallel Connectivity Dialog Box



Indicate the compute node ID for which connectivity information is desired in the **Compute Node** field, and then click the **Print** button. Sample output for compute node 0 is shown below:

```
-------------------------------------------------------------------------------------------
ID      Hostname              Core        O.S.        PID     Vendor
-------------------------------------------------------------------------------------------
n5      host25                2/64        Linux-64    18909   AMD Opteron(tm) 6278
n2      host25                1/64        Linux-64    18908   AMD Opteron(tm) 6278
n4      host24                2/64        Linux-64    32939   AMD Opteron(tm) 6278
n1      host24                1/64        Linux-64    32938   AMD Opteron(tm) 6278
host    host23                -           Linux-64    38427   AMD Opteron(tm) 6278
n3      host23                2/64        Linux-64    38755   AMD Opteron(tm) 6278
n0*     host23                1/64        Linux-64    38754   AMD Opteron(tm) 6278
```

43.9. Checking and Improving Parallel Performance

Fluent offers several tools to help you optimize the performance of your parallel computations. You can check the utilization of your hardware using the parallel check feature. To determine how well the parallel solver is working, you can measure computation and communication times, and the overall parallel efficiency, using the performance meter. You can also control the amount of communication between compute nodes in order to optimize the parallel solver, and take advantage of the automatic load balancing feature of Ansys Fluent.

Information about checking and improving parallel performance is provided in the following sections:

43.9.1. Parallel Check

43.9.2. Checking Parallel Performance

43.9.3. Optimizing the Parallel Solver

43.9.4. Clearing the Linux File Cache Buffers

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

43.9.1. Parallel Check

You can use the **Check** command in the **Parallel** ribbon tab to check various factors that affect parallel performance. Checks are performed to identify the following issues:

- CPU cores are overloaded
- CPU clock is throttled
- System memory usage is too high
- A faster interconnect is available
- Partitions are imbalanced (if a valid mesh is loaded)

43.9.2. Checking Parallel Performance

The performance meter allows you to report the wall clock time elapsed during a computation, as well as message-passing statistics. Since the performance meter is always enabled, you can access the statistics by displaying them after the computation is completed. To view the current statistics, click **Usage** in the **Parallel** ribbon tab (**Timer** group box).

Parallel → Timer → Usage

Performance statistics will be displayed in the console.

To clear the performance meter so that you can eliminate past statistics from the future report, click **Reset** in the **Parallel** ribbon tab (**Timer** group box).

Parallel → Timer → Reset

The following example demonstrates how the current parallel statistics are displayed in the console:

```
Performance Timer for 1 iterations on 4 compute nodes
Average wall-clock time per iteration:           4.901 sec
Global reductions per iteration:                   408 ops
Global reductions time per iteration:            0.000 sec (0.0%)
Message count per iteration:                       801 messages
Data transfer per iteration:                     9.585 MB
LE solves per iteration:                           12 solves
LE wall-clock time per iteration:                2.445 sec (49.9%)
LE global solves per iteration:                    27 solves
LE global wall-clock time per iteration:         0.246 sec (5.0%)
AMG cycles per iteration:                          64 cycles
Relaxation sweeps per iteration:                 4160 sweeps
Relaxation exchanges per iteration:                920 exchanges

Total wall-clock time:                           4.901 sec
```

A description of the parallel statistics is as follows:

- `Average wall-clock time per iteration` describes the average real (wall clock) time per iteration.
- `Global reductions per iteration` describes the number of global reduction operations (such as variable summations over all processes). This requires communication among all processes.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

A global reduction is a collective operation over all processes for the given job that reduces a vector quantity (the length given by the number of processes or nodes) to a scalar quantity (for example, taking the sum or maximum of a particular quantity). The number of global reductions cannot be calculated from any other readily known quantities. The number is generally dependent on the algorithm being used and the problem being solved.

- `Global reductions time per iteration` describes the time per iteration for the global reduction operations.

- `Message count per iteration` describes the number of messages sent between all processes per iteration. This is important with regard to communication latency, especially on high-latency interconnects.

  A message is defined as a single point-to-point, send-and-receive operation between any two processes. This excludes global, collective operations such as global reductions. In terms of domain decomposition, a message is passed from the process governing one subdomain to a process governing another (usually adjacent) subdomain.

  The message count per iteration is usually dependent on the algorithm being used and the problem being solved. The message count and the number of messages that are reported are totals for all processors.

  The message count provides some insight into the impact of communication latency on parallel performance. A higher message count indicates that the parallel performance may be more adversely affected if a high-latency interconnect is being used. Ethernet has a higher latency than InfiniBand. Therefore, a high message count will more adversely affect performance with Ethernet than with InfiniBand.

  To check the latency of the overall cluster interconnect, refer to Checking Latency and Bandwidth (p. 4349).

- `Data transfer per iteration` describes the amount of data communicated between processors per iteration. This is important with respect to interconnect bandwidth.

  Data transfer per iteration is usually dependent on the algorithm being used and the problem being solved. This number generally increases with increases in problem size, number of partitions, and physics complexity.

  The data transfer per iteration may provide some insight into the impact of communication bandwidth (speed) on parallel performance. The precise impact is often difficult to quantify because it is dependent on many things including: ratio of data transfer to calculations, and ratio of communication bandwidth to CPU speed. The unit of data transfer is a byte.

  To check the bandwidth of the overall cluster interconnect, refer to Checking Latency and Bandwidth (p. 4349).

- `LE solves per iteration` describes the number of linear systems being solved per iteration. This number is dependent on the physics (non-reacting versus reacting flow) and the algorithms (pressure-based versus density-based solver), but is independent of mesh size. For the pressure-based solver, this is usually the number of transport equations being solved (mass, momentum, energy, and so on).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- `LE wall-clock time per iteration` describes the time (wall-clock) spent doing linear equation solvers (that is, multigrid).
- `LE global solves per iteration` describes the number of solutions on the coarsest level of the AMG solver where the entire linear system has been pushed to a single processor (`n0`). The system is pushed to a single processor to reduce the computation time during the solution on that level. Scaling generally is not adversely affected because the number of unknowns is small on the coarser levels.
- `LE global wall-clock time per iteration` describes the time (wall-clock) per iteration for the linear equation global solutions.
- `AMG cycles per iteration` describes the average number of multigrid cycles (V, W, flexible, and so on) per iteration.
- `Relaxation sweeps per iteration` describes the number of relaxation sweeps (or iterative solutions) on all levels for all equations per iteration. A relaxation sweep is usually one iteration of Gauss-Siedel or ILU.
- `Relaxation exchanges per iteration` describes the number of solution communications between processors during the relaxation process in AMG. This number may be less than the number of sweeps because of shifting the linear system on coarser levels to a single node/process.
- `Time-step updates per iteration` describes the number of sub-iterations on the time step per iteration.
- `Time-step wall-clock time per iteration` describes the time per sub-iteration.
- `Total wall-clock time` describes the total wall-clock time.

The most relevant quantity is the `Total wall clock time`. This quantity can be used to gauge the parallel performance (speedup and efficiency) by comparing this quantity to that from the serial analysis.

43.9.2.1. Checking Latency and Bandwidth

You can check the latency and bandwidth of the overall cluster interconnect, to help identify any issues affecting Ansys Fluent scalability, by clicking **Latency** and **Bandwidth** in the **Parallel** ribbon tab (**Network** group box).

Parallel → Network → Latency

Depending on the number of machines and processors being used, a table containing information about the communication speed for each node will be displayed in the console. The table will also summarize the minimum and maximum latency between two nodes.

Consider the following example when checking for latency:

```
Latency (usec) with 1000 samples [1.83128 sec]
 ----------------------------------------
 ID       n0     n1     n2    n3    n4    n5
 ----------------------------------------
 n0            48.0   48.2  48.2  48.3   *50
 n1    48.0           48.2  48.3  48.3   *48
 n2    48.2    48.2         48.8  49.1   *53
```

```
n3    48.2    48.3    *49        48.6  48.5
n4    48.3    48.3   49.1  48.6         *50
n5    49.7    48.5    *53  48.5  49.7

----------------------------------------
Min: 47.9956 [n0<-->n1]
Max: 52.6836 [n5<-->n2]
----------------------------------------
```

Important:

In the above table, (*) is the maximum value in that row. The smaller the latency, the better.

Six processors (n0 to n5) are spawned. The latency between n0 and n1 is 48.0 μs. Similarly, the latency between n1 and n2 is 48.2 μs. The minimum latency occurs between n0 and n1 and the maximum latency occurs between n2 and n5, as noted in the table. Checking the latency is particularly useful when you are not seeing expected speedup on a cluster.

Parallel → Network → Bandwidth

In addition to checking for latency, you can check your bandwidth. A table containing information about the amount of data communicated within one second between two nodes is displayed in the console. The table will also summarize the minimum and maximum bandwidth between two nodes.

Consider the following example when checking for bandwidth:

```
Bandwidth (MB/s) with 5 messages of size 4MB [4.36388 sec]
-----------------------------------------
ID      n0     n1     n2     n3     n4     n5
-----------------------------------------
n0          111.8    *55  111.8   97.5  101.3
n1   111.8          69.2   98.7  111.7    *51
n2    54.7   69.2          72.9  104.8    *45
n3   111.8   98.7   72.9          64.0    *45
n4    97.6  111.7  104.8    *64          76.9
n5   101.2   50.9   45.5    *45    76.9

-----------------------------------------
Min: 45.1039 [n5<-->n3]
Max: 111.847 [n0<-->n3]
-----------------------------------------
```

Important:

In the above table, (*) is the minimum value in that row. The larger the bandwidth, the better.

The bandwidth between n0 and n1 is 111.8 MB/s. Similarly, the bandwidth between n1 and n2 is 69.2 MB/s. The minimum amount of bandwidth occurs between n3 and n5 and the maximum occurs between n0 and n3, as noted in the table. Checking the bandwidth is particularly useful when you cannot see good scalability with relatively large cases.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

43.9.3. Optimizing the Parallel Solver

43.9.3.1. Increasing the Report Interval

In Ansys Fluent, you can reduce communication and improve parallel performance by increasing the report interval for residual printing/plotting or other solution monitoring reports. You can modify the value for **Reporting Interval** in the Run Calculation Task Page (p. 5207).

Solution → Run Calculation → Calculate...

Important:

Note that you will be unable to interrupt iterations until the end of each report interval.

43.9.3.2. Accelerating View Factor Calculations Using General Purpose Graphics Processing Units (GPGPUs)

When using the **Surface to Surface (S2S)** radiation model, you can accelerate the calculation of the view factors with the following methods:

- the **Hemicube** method with the **Cluster to Cluster** basis
- the **Ray Tracing** method

For the accelerated hemicube method, a combination of MPI/OpenMP/OpenCL models is used to speed up the view factor computations. Irrespective of the number of MPI processes launched, only one MPI process/machine is used for computing view factors. On each machine, one MPI process spawns several OpenMP threads that actually compute the view factors. The number of accelerated hemicube processes will be same as the number of Ansys Fluent processes. If OpenCL-capable GPUs are available, then a portion of the view factor computations are done on GPUs in offload mode using OpenCL, to further speed up the computation.

To use the accelerated hemicube method, set up the **View Factors and Clustering** dialog box and then enter the following text command to compute, write, and read a file with the surface cluster information and view factors: `/define/models/radiation/s2s-parameters/compute-clusters-and-vf-accelerated`. You will be prompted for the name of the file, the number of GPUs to be used, and the ratio of the work load on 1 GPU vs 1 CPU OpenMP thread (this is based on the time consumed by the GPU and the CPU). At the end of the view factor computations, a recommendation is printed for the GPU/CPU work load ratio to use in future computations.

Note that for the accelerated hemicube method, the OpenCL library should be accessible through the appropriate environment variable (`LD_LIBRARY_PATH` on lnamd64 or `%path%` on win64) in order to use the GPU for view factor computations. By default on lnamd64, `/usr/lib64` is searched, but if the library is installed in another location, then that location should be specified in the `LD_LIBRARY_PATH` variable.

When using the accelerated ray tracing method with a GPU in offload mode, you must have one NVIDIA GPU (and only one) available on node-0. The NVIDIA Optix library is used for tracing the rays. Currently the Ampere GPUs are not supported with this method.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

To use the accelerated ray tracing method, set up the **View Factors and Clustering** dialog box and then enter the following text command to compute, write, and read a file with the surface cluster information and view factors: `/define/models/radiation/s2s-parameters/compute-clusters-and-vf-accelerated`. You will be prompted for the name of the file.

Note:

CFF (.h5) format is not supported with the accelerated view factor calculations.

43.9.3.3. Accelerating Discrete Ordinates (DO) Radiation Calculations

The accelerated discrete ordinates (DO) radiation solver is computationally faster than the standard DO solver. Note that even though the accelerated DO solver may take more iterations to converge, the overall simulation time is shorter. If NVIDIA GPGPUs are set up in offload mode for the Fluent session, the accelerated DO solver will accelerate the DO computations by using the GPGPUs; in the absence of GPGPUs, this solver can still be used with the CPU cores to accelerate the DO computations.

After you have selected the DO model in the **Radiation Model** dialog box, you can enable the accelerated DO solver by using the following text command:

`define` → `models` → `radiation` → `do-acceleration?`

Note that the accelerated DO solver uses the first-order upwind scheme (and ignores whatever selection you have made for the **Discrete Ordinates** spatial discretization scheme in the **Solution Methods** task page), along with an explicit relaxation of 1.0.

If you plan to use GPGPUs with the accelerated DO solver, it is recommended that you run NVIDIA's multi-process server (MPS) before launching Ansys Fluent using the following command: `nvidia-cuda-mps-control -d`. It is known to improve the robustness and performance of the GPGPU computations with the multiple Fluent processes. Then you must specify how many GPGPUs are to be used per machine when you launch Fluent: you can use the **Solver GPGPUs per Machine (Offload Mode)** setting in Fluent Launcher or use the `-gpgpu=` *ngpgpus* command line option. For details, refer to the following sections:

Starting Parallel Ansys Fluent Using Fluent Launcher (p. 4296)

Starting Parallel Ansys Fluent on a Windows System Using Command Line Options (p. 4306)

Starting Parallel Ansys Fluent on a Linux System (p. 4311)

Note that using GPGPUs requires HPC licenses. Licensing details can be found in HPC Licensing in the *Ansys, Inc. Licensing Guide*.

Once the Fluent session is running, you can view and/or select the available GPGPUs on the system using the following TUI commands:

`parallel/gpgpu/show`

Displays the available GPGPUs on the system.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

parallel/gpgpu/select

Selects the GPGPUs to use in offload mode. Note that you can only select up to the number of GPGPUs that you specified on the command line or in Fluent Launcher when starting the session.

Currently the Ampere GPUs are not supported with the accelerated radiation solver on Windows platforms.

43.9.3.3.1. Accelerated DO Model Limitations

The accelerated DO solver is incompatible with some models and settings. When necessary, Fluent will automatically revert to the standard DO solver when the calculation is started and print a message about the conflict. The known incompatibilities are as follows:

- Shells
- Axisymmetric cases
- Eulerian multiphase
- UDF-specified DOM sources
- UDF-specified emissivity weighting factor
- Porosity scaling of radiation
- Pixelated interior
- Non-isotropic scattering phase function
- Non-participating medium
- Non-conformal interfaces
- Supports only upwind scheme and explicit relaxation of 1.0.

43.9.4. Clearing the Linux File Cache Buffers

Processing performance can significantly decrease when the file cache buffers of a Linux machine are full. While this is true in serial, it is more often a concern when solving large cases in parallel, particularly when using AMD processors. If you see a performance decrease even though the case / machine setup has not changed, that is an indication that the file cache buffers may be to blame.

The filling of the file cache buffers can happen over a period of time as a result of input-output activity. Even after the Ansys Fluent session is exited, by default the operating system does not free up the file cache buffers immediately (unless the operating system is unable to satisfy a malloc sub-routine request with available free memory). During memory allocation for a parallel case, this can result in the allocation of memory from a different NUMA domain, and consequently can have a significant impact on performance.

To resolve this issue on Linux machines, you must first ensure that all of the relevant machines are idle (so that you do not adversely affect any jobs that are running). Then you can clear the file cache buffers by performing one of the following actions:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Include the `-cflush` option when launching Ansys Fluent from the command line.

  This option ensures that the file cache buffers are flushed in a separate operation. While this process may take a few minutes to complete (depending on the total memory of the system), it does not require you to have root privileges.

or

- Enter the `(drop-cache)` Scheme command (either in the Ansys Fluent console or through your journal file) after launching Fluent but before you read the case file.

  This command will instantaneously clear the pagecaches, dentries, and inodes.

  Important:

  Note that in order to use the `(drop-cache)` command, you must have sudo administrative privileges for the `/sbin/sysctl vm.drop_caches=3` command.

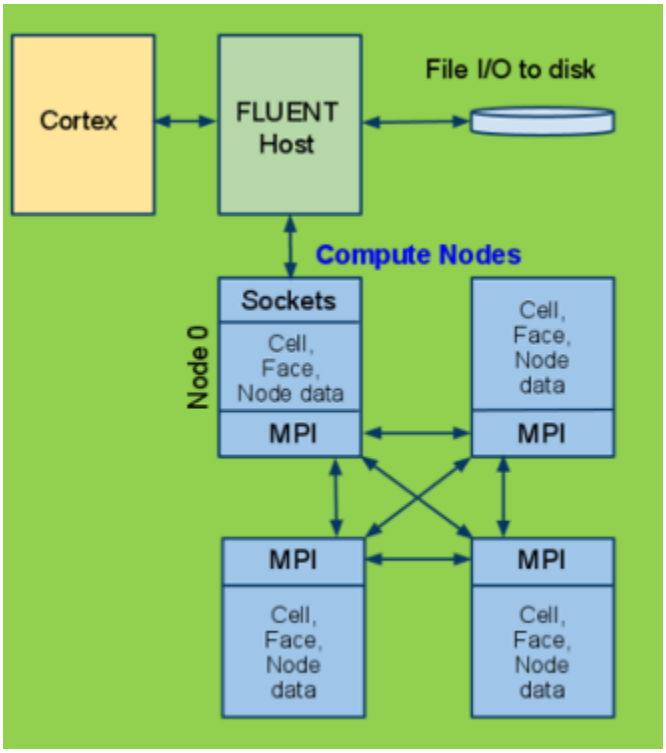
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 44: Running Ansys Fluent on Arm Compute Nodes

This chapter contains information about the support for running Fluent on the 64-bit extension of the Arm architecture (aarch64), along with the Armv8.2-A extensions.

The version of Fluent for Arm runs entirely on the Arm systems. Only batch mode is supported, and no user interface (UI) or graphics are provided. This workflow is shown in Figure 44.1: The Workflow for Fluent on Arm Compute Nodes (p. 4355).

Figure 44.1: The Workflow for Fluent on Arm Compute Nodes



Note:

Fluent for Arm is not supported on Windows.

For details, see the following sections.

44.1. Setting up Fluent for Arm

44.2. Message Passing Interface (MPI) Supported with Fluent for Arm

44.3. Starting Fluent for Arm

44.4. Running Fluent for Arm with a Job Scheduler

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

44.5. Fluent for Arm Known Limitations

44.1. Setting up Fluent for Arm

To set up Fluent for Arm, perform the following steps:

1. Request the Fluent for Arm package from Ansys support and download it. This package provides an aarch64 installation for Arm.
2. Install the Fluent for Arm package:
   a. Extract the package contents using the following command:
   ```
   tar xf Fluent_install-lnarm64-24.2.0-<BID>.tar
   ```
   where <BID> is the build ID.
   b. Run the installer using the following command, and follow the prompts:
   ```
   sh Fluent_install-lnarm64-24.2.0.sh
   ```

   Note:

   It is recommended that you use one of the following on Arm:

   - Rocky Linux 8.9*
   - Red Hat Enterprise Linux (RHEL) 8.8* / 9.3*

   (*) Prerequisites: libICE, libnsl2, libSM, pcre2-utf16.

3. Set up the environment.

44.2. Message Passing Interface (MPI) Supported with Fluent for Arm

Fluent for Arm provides Open MPI 4.0.5 within the package. Open MPI is the default MPI and the only MPI available in the package.

44.3. Starting Fluent for Arm

Fluent can be run from the command line only. For example, you could start Fluent on an Arm machine by entering:

```
fluent 3ddp -i fluid_flow.jou
```

Note that this will default to the `-g` option and so will not provide graphics nor the graphical user interface.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

44.4. Running Fluent for Arm with a Job Scheduler

Fluent for Arm can be run with the Slurm job scheduler instead of manually providing a hosts file. With Slurm, you can use the standard options described in Running Fluent Under Slurm in the *Running Fluent Using a Load Manager* manual. For example:

```
fluent 3ddp -i fluid_flow.jou -scheduler=slurm
```

44.5. Fluent for Arm Known Limitations

The following features are not supported when running Fluent for Arm:

- Graphical user interface (GUI)
- Graphics and batch image rendering
- Fluent Launcher
- Meshing mode
- The following Fluent meshing-to-solver workflows
  - CAD import
- Functional Mock-up Unit (FMU)
- Monte Carlo (MC) radiation model
- Dynamic mesh with radial basis function smoothing
- The conversion of `.gtm` to `.def` as part of a turbomachinery guided workflow
- Interpreted user-defined functions (UDFs)
- View factor files in the CFF S2S format
- Fluent native GPU solver
- Parametric studies
- The `fe2ram` filter
- Modifying the geometry based on the adjoint solution, using the **Design Tool** with **Enhanced** selected for the **Constraint Method**
- Geometry parameterization (accessed by clicking **Parameterize and Explore...** in the **Design** ribbon tab)
- Accelerating calculations using GPGPUs in offload mode
- Intel MKL
- Sundials with LAPACK

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- All linx64 beta features
- Anisotropic mesh adaption (beta feature)
- The following Integrations
  - CGNS export (VKI)
  - CHEMKIN CFD Solver
  - EnSight
  - FENSAP-ICE
  - System Coupling
  - Icepak
- The following Add-ons
  - Reduced Order Model (ROM)
  - PCB Model
- The following Workspaces
  - Remote Visualization Client
  - Fluent Icing
  - Fluent Aero
  - Fluent Materials Processing
  - Fluent Post-Analysis

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 45: Using Simulation Reports

You can create HTML or PDF reports for your Ansys Fluent simulations.

45.1. Overview of Simulation Reports

45.2. Preparing Simulation Reports

45.3. Generating Simulation Reports

45.4. Viewing Simulation Reports

45.5. Saving Simulation Reports

45.6. Additional Simulation Report Options

45.7. Customizing Simulation Reports

45.8. Generating Reports Using the Text User Interface (TUI)

45.1. Overview of Simulation Reports

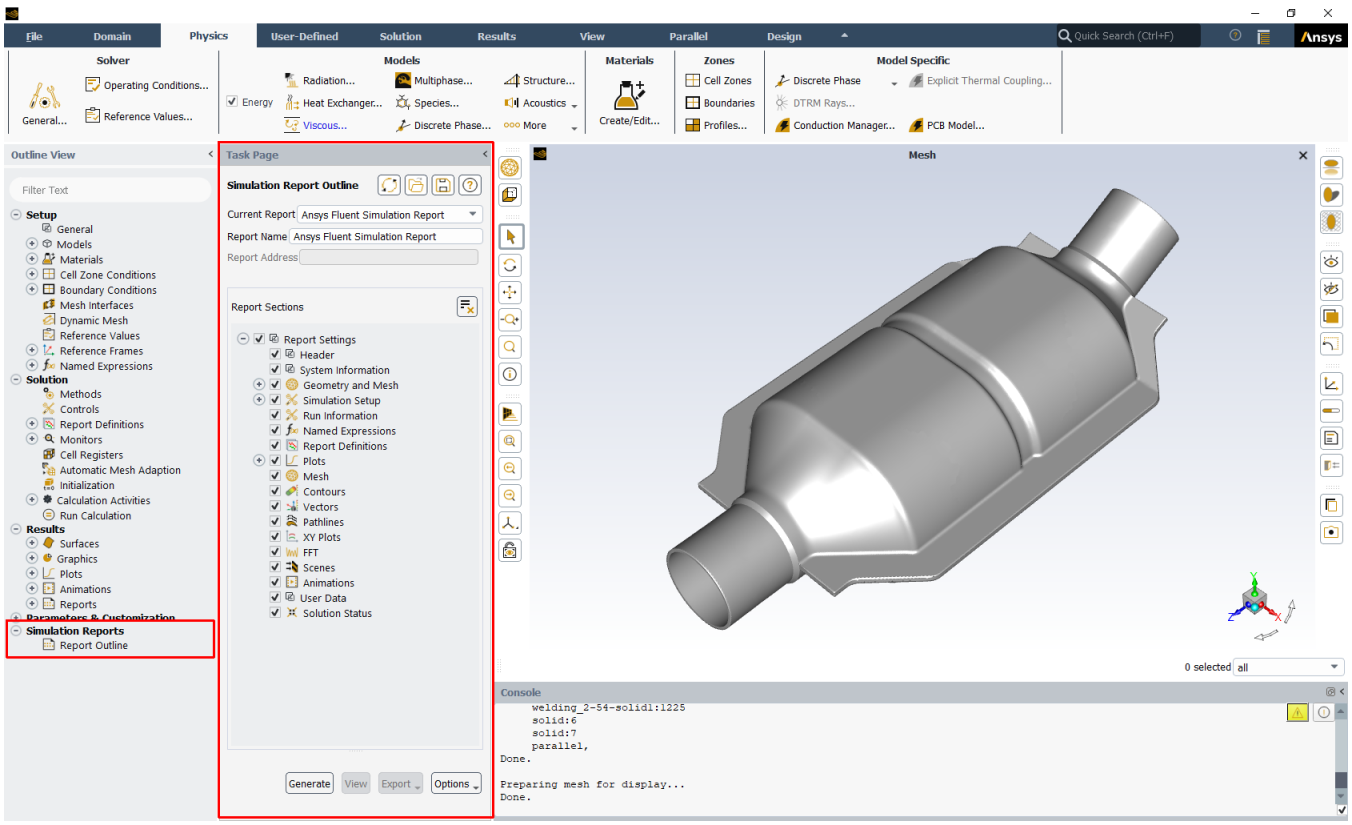
Ansys Fluent simulation reports make it possible for you to summarize and share the setup and results of your simulations. Key aspects of your simulation are compiled from your project settings and can be exported in either HTML or PDF format, or as a PowerPoint presentation. Reports can be customized to suit your specific needs.

When you have a complete Ansys Fluent simulation (with saved case and data files), you can open the **Report Outline** using the **Simulation Reports** task page from the Outline View.

Simulation Reports → **Report Outline**

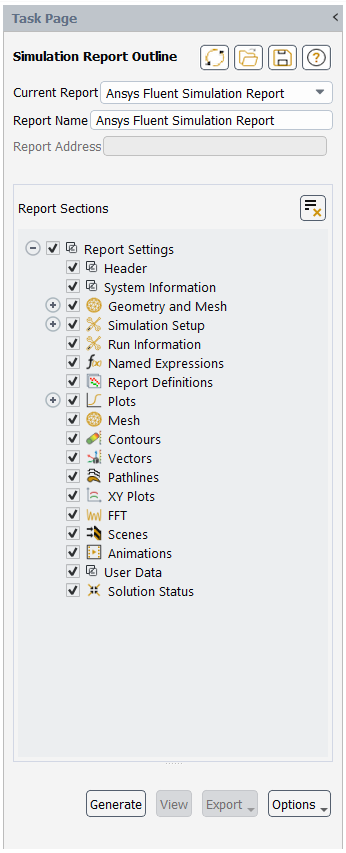
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 45.1: Accessing the Simulation Report Outline Task Page



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 45.2: The Simulation Report Outline Task Page



The task page contains basic information for the simulation report.

- The **Current Report** field allows you to select from a list of pre-existing reports.
- The **Report Name** allows you to provide a name for the report. The field remains available so that you can rename your report even after it has been generated.
- The **Report Address** allows you to use a sharable URL for a generated report.
- The **Report Sections** includes options for filling in descriptions for the various portions of the report.

Note:

You can change individual report settings in the **Simulation Reports** task page and continue to generate new reports. For example, you can add text descriptions for some sections, select or deselect sections in the tree, or change the view or plot settings for contours or report plots. This can all be done from the **Simulation Reports** task page as you click different tree nodes, and you can continue to generate new reports with different selections or descriptions as needed.

Once the report sections are filled out accordingly (see Preparing Simulation Reports (p. 4362)), you can generate a new report using the current settings (see Generating Simulation Reports (p. 4366)), or view other reports, if available (see Viewing Simulation Reports (p. 4367)). Customized settings can also be saved your later use (Customizing Your Report Settings Using Templates (p. 4385)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

45.1.1. Limitations for Simulations Reports

The following limitation exists when using simulation reports in Fluent:

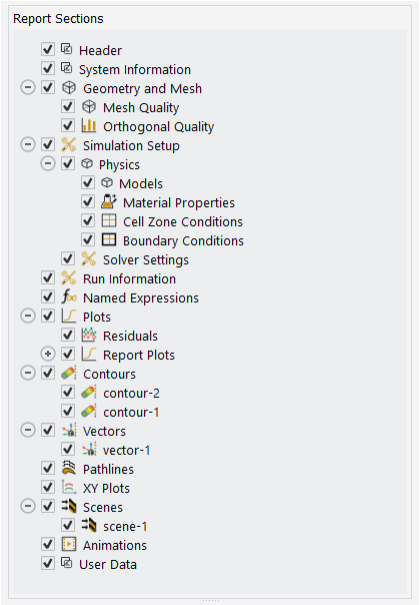
- Since there can be multiple Fluent sessions using the same working directory at the same time to both access the active simulation report server for that working directory, if there are two or more Fluent sessions connected to a report server, the server will shut down if the first Fluent session that started the server is ever closed. The second Fluent session may not be impacted since the server may automatically restart the next time it is needed, however, if the server is being used by the second Fluent session at the time that the first Fluent session is closed, there could be errors and/or report data corruption for the second Fluent session.
- On Linux, animations saved as part of your HTML simulation report may not be directly played within your browser if your browser does not directly support H264 MP4 video. In such cases, your browser will require an appropriate plug-in to support H264 MP4 videos.
- On Linux, you must generate simulation reports *only* while using graphics acceleration (for example, using OpenGL2 driver with accelerated graphics).
- Animations, and AVZ files, created for your simulation report cannot be visualized within the Fluent interface.
- On Linux, generating simulation reports may cause Fluent to fail due to the existence of intermediate files that may remain from a previous release. The workaround is to delete the older local files (`/.local/share/fluent/QtWebEngine`) from your Linux home.
- The **Application Font Size** setting specified in Preferences does not apply to simulation reports.
- Simulation reports do not show species mass fractions for boundary conditions.
- When you export your simulation report as a PowerPoint presentation, named expression tables, that can be long and extensive, are not automatically split across multiple slides.

45.2. Preparing Simulation Reports

By default, the simulation report contains basic information about your simulation, with the ability to provide some additional information. Select a node under **Report Sections** to view that section's information.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 45.3: The Report Sections Outline View



Header information includes fields for the **Analyst** name, and the **Date** (by default). Additionally, you can specify your **Company** name and your **Logo** image file. Further more, there are additional options for **URL**, **Phone** and **Fax** numbers, **Address** information, and a **Description** for the simulation.

You can provide custom **Description**s for the remaining sections:

- **System Information**
- **Geometry and Mesh**, including the **Mesh Quality** and **Orthogonal Quality** sections
- **Simulation Setup**, including the **Solver Settings** and **Physics** sections (which itself contains sections for **Models**, **Material Properties**, **Cell Zone Conditions**, and **Boundary Conditions**)
- **Run Information**
- **Solution Status**
- **Named Expressions**
- **Report Definitions**
- **Plots**, including **Residuals** and **Report Plots**, if available. Additionally, you can specify **Line Type**(s) and **Line Marker**(s) for your plots.
- Any available graphics objects that you want to include in your report, such as **Mesh**, **Contours**, **Vectors**, **Pathlines**, **LICs**,**XY Plots**, **FFT**, **Scenes**, **Animations**, **Histograms**, or your own custom view(s). Additionally, you can display the object using a particular orientation using the **View** field (front, back, left, bottom, isometric, etc.). Use the **Preview View** button to see what the image will look like once it is in the report. The isometric view is the default orientation.
- **User Data**, if available.
- **Solution Status**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Each of these components can be disabled so that they are not included in the final report.

Use the icon to deselect the actively selected node and clear any displayed settings.

45.2.1. Setting General Report Properties

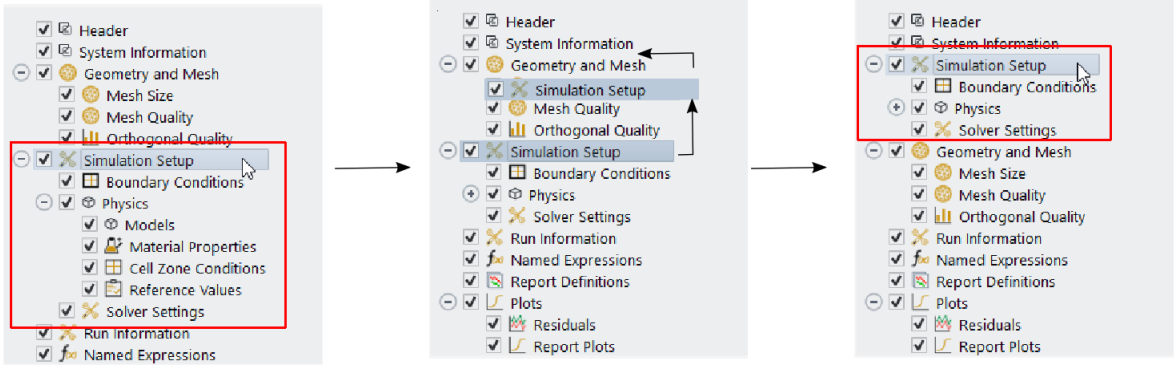
Under **Report Settings**, there are several aspects of your simulation report that you can control from a global perspective.

- **Visibility** - use the check boxes to include/exclude all or parts of the report.
- **Layout Settings** - choose whether the layout of your results plots (such as contours, vectors, etc.) will be included in your report as a series of tabs, or in tabular columns. These settings will be applied to all the graphics results images in your report. See Changing the Layout of Your Results (p. 4384) for details.
- **Image Settings** - choose the default view orientation and resolution for your included graphics results (such as saved views/states, or common orientations such as top, bottom, left, right, and so on). See Customizing Your Report Image Settings (p. 4384) for details.
- **Theme Settings** - choose the **Use Dark Theme** option to present your simulation report (and included plots) with a dark background and white text, with any graphics result images remaining as they were originally captured. The **Use Dark Theme** option will be enabled by default if you have chosen to run Ansys Fluent using the **Dark** theme preference (see Customizing the Graphical User Interface (p. 935) and Setting User Preferences/Options (p. 937)).

45.2.2. Organizing Your Simulation Report

To obtain a more customized report, you can reorganize elements of your report prior to generating it. To do so, drag and drop elements of the report section tree onto other portions of the tree to rearrange the order and contents of various sections of your simulation report.

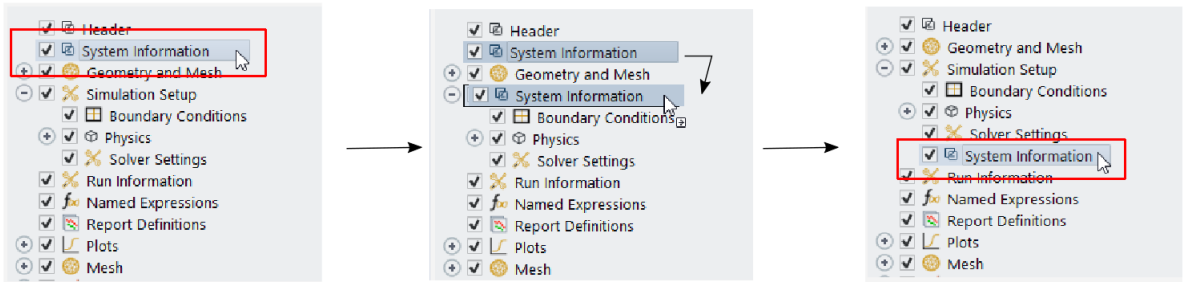
Figure 45.4: Drag-and-Drop Rearrangement of Nodes



Typically, you can drag and drop one or more selected item(s) onto a node and it/they will be added to the bottom of the sub-nodes.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 45.5: Drag-and-Drop Onto a Node

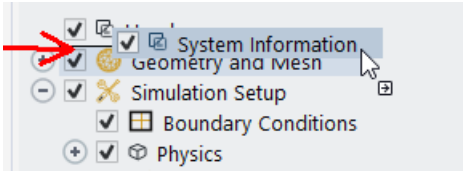


As you select items in the tree and drag them elsewhere in the tree, the cursor changes accordingly, indicating whether you can ( ) or cannot ( ) drop the selected item at that location.

Figure 45.6: Drag and Drop Indicators

When trying to position a selection between two nodes, a line appears between the two nodes, indicating that placement between those nodes is acceptable.

Figure 45.7: Positioning Between Two Nodes



The following caveats exist when organizing your report sections.

- The **Header** section must always be at the beginning of the report, and therefore at the top node of the tree.
- Elements of specialized container nodes - ones that have their sub-nodes arranged as tabs in their respective containers (such as **Contours**, **Vectors**, etc.) - cannot have their contents changed. Nothing can be added to these containers, and nothing from these containers can be moved elsewhere.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

45.3. Generating Simulation Reports

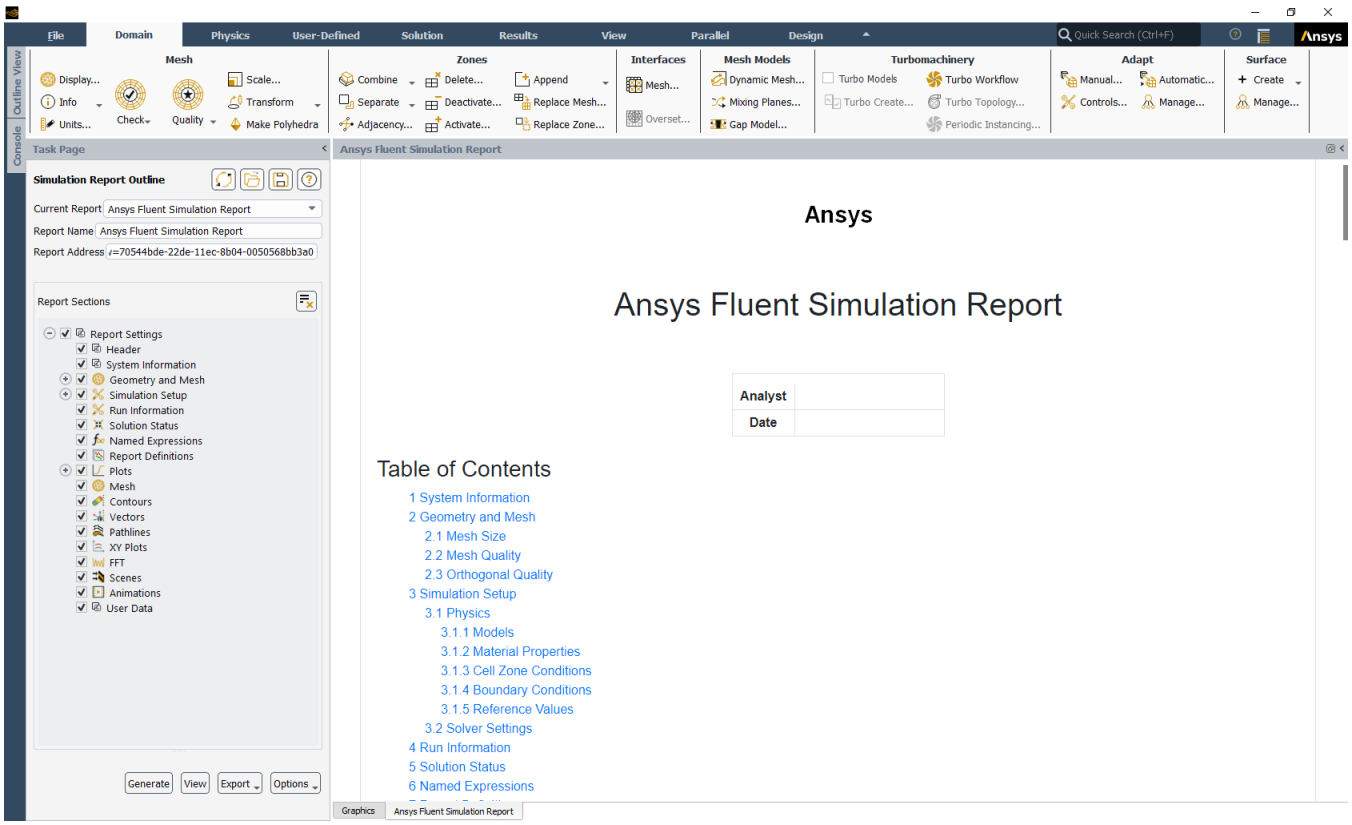
When you are satisfied with your simulation setup and results, and have configured your report to your satisfaction, generate a report using the **Generate** button at the bottom of the **Simulation Reports** task page.

Note:

When you first generate a report, Ansys Fluent may initially take some extra time to create and load the report, since it starts the Fluent report server, which takes a few moments to initialize.

While the report is being generated, a progress bar displays the status of the report generation. You can interrupt the report generation by clicking the **Stop** button to the right of the progress bar.

Figure 45.8: A Generated Report



See Viewing Simulation Reports (p. 4367) for information about the generated report.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

45.4. Viewing Simulation Reports

Once the report is generated, it appears in its own tab alongside the graphics window. Alternatively, you can use the **View** button in the **Simulation Report** task page for a selected report.

Note:

Report settings for all generated reports will be saved with the case file. As long as the `FluentReportServer` folder is not moved or deleted from the working directory, you can read in a case file where you have previously generated a report and immediately view the report using the **View** button without needing to generate it again.

Use the hyperlinked table of contents to jump to different portions of the report.

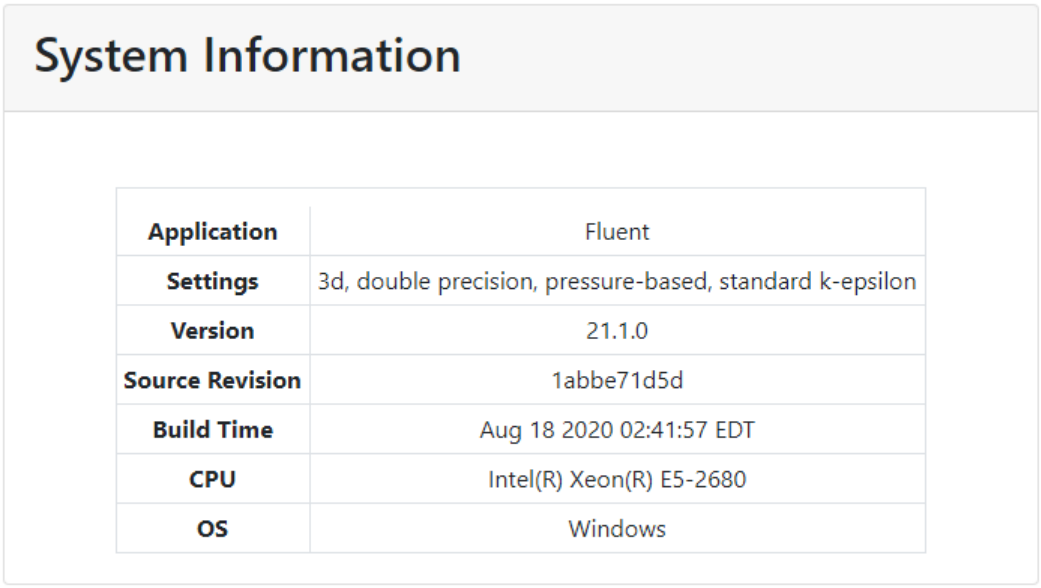
45.4.1. Viewing System Information
45.4.2. Viewing Geometry and Mesh Information
45.4.3. Viewing Simulation Setup Information
45.4.4. Viewing Run Information
45.4.5. Viewing Solution Status
45.4.6. Viewing Named Expression Information
45.4.7. Viewing Report Definition Information
45.4.8. Viewing Plot Information
45.4.9. Viewing Contours, Vectors, Pathlines, LICs, XY Plots, Scenes, Histograms, and Animations

45.4.1. Viewing System Information

System Information information automatically includes details for the **Application** (the product name, such as Fluent), **Settings** (such as 3D, double-precision, etc.), **Version** (such as 20.2, etc.), **Source Revision**, **Build Time**, **CPU**, and **OS**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 45.9: An Example of the System Information Report Section



System Information

Application	Fluent
Settings	3d, double precision, pressure-based, standard k-epsilon
Version	21.1.0
Source Revision	1abbe71d5d
Build Time	Aug 18 2020 02:41:57 EDT
CPU	Intel(R) Xeon(R) E5-2680
OS	Windows

45.4.2. Viewing Geometry and Mesh Information

Geometry and Mesh information includes a default **Mesh Quality** table of the simulation's meshing statistics including: region names, their volume fill type, the minimum orthogonal quality, and the maximum aspect ratio.

Figure 45.10: An Example of the Geometry and Mesh Report Section (Mesh Quality)

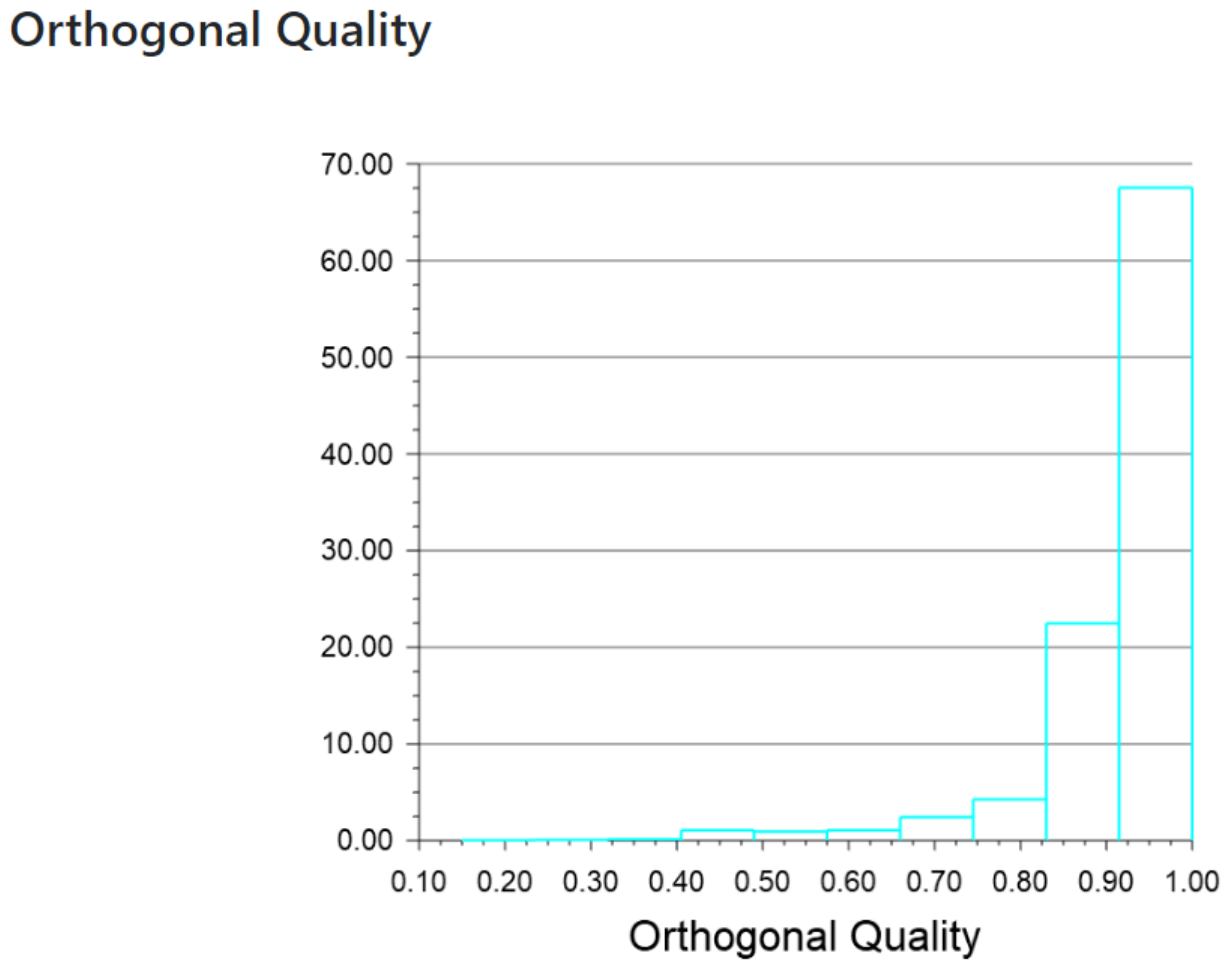
Mesh Quality

Name	Type	Min Orthogonal Quality	Max Aspect Ratio
fluid:0	Poly Cell	0.15053822	59.103277
fluid:1	Poly Cell	0.20037727	28.999819
fluid:5	Poly Cell	0.1636869	35.291213
honeycomb-solid1:1201	Poly Cell	0.40190728	15.246641
honeycomb_af0-solid1:1177	Poly Cell	0.39459723	14.429885
pipe_connetions_cc-71-solid1:1207	Poly Cell	0.49068145	8.1465026
pipe_connetions_cc-solid1:1213	Poly Cell	0.48511881	8.6792763
quadribbed_body_bottom_af0-solid:1162	Poly Cell	0.35431728	11.465975
quadribbed_body_top_af1-solid:1165	Poly Cell	0.35293116	12.659486
recessed_cushioningmat-solid1	Poly Cell	0.55997771	5.3719419
recessed_cushioningmat_af0-solid1	Poly Cell	0.63515375	5.7653949
sensing_element-65-solid:1198	Poly Cell	0.20195385	10.257318
sensor_innertube-67-solid:1204	Poly Cell	0.15000073	56.140799

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You are also presented with a default **Orthogonal Quality** chart.

Figure 45.11: An Example of the Geometry and Mesh Report Section (Orthogonal Quality)



45.4.3. Viewing Simulation Setup Information

Simulation Setup information includes the **Solver Settings** and **Physics** sections (which itself contains sections for **Models**, **Material Properties**, **Cell Zone Conditions**, and **Boundary Conditions**).

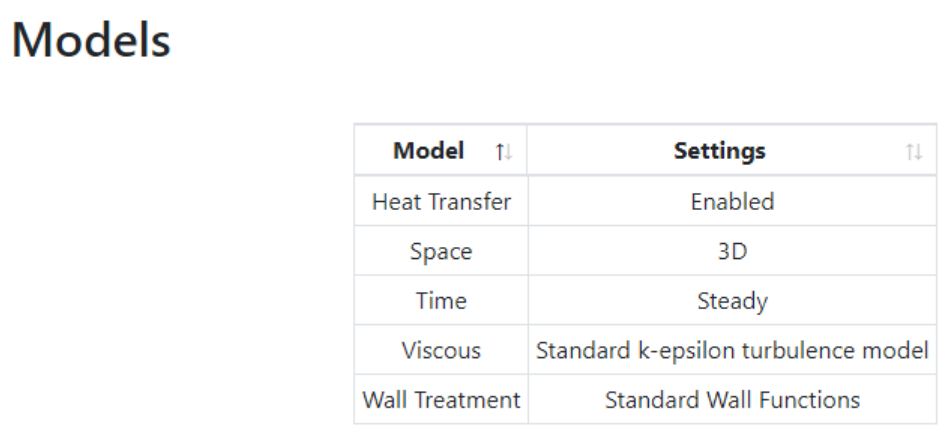
Note:

Under **Simulation Setup**, the **Show Only Modified Settings** check box is enabled by default so that the report only displays settings that have been changed from the default values.

The **Models** section, under **Physics**, includes the numerical models that have been enabled in the simulation, such as the turbulence, heat transfer, etc.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 45.12: An Example of the Physics / Models Settings

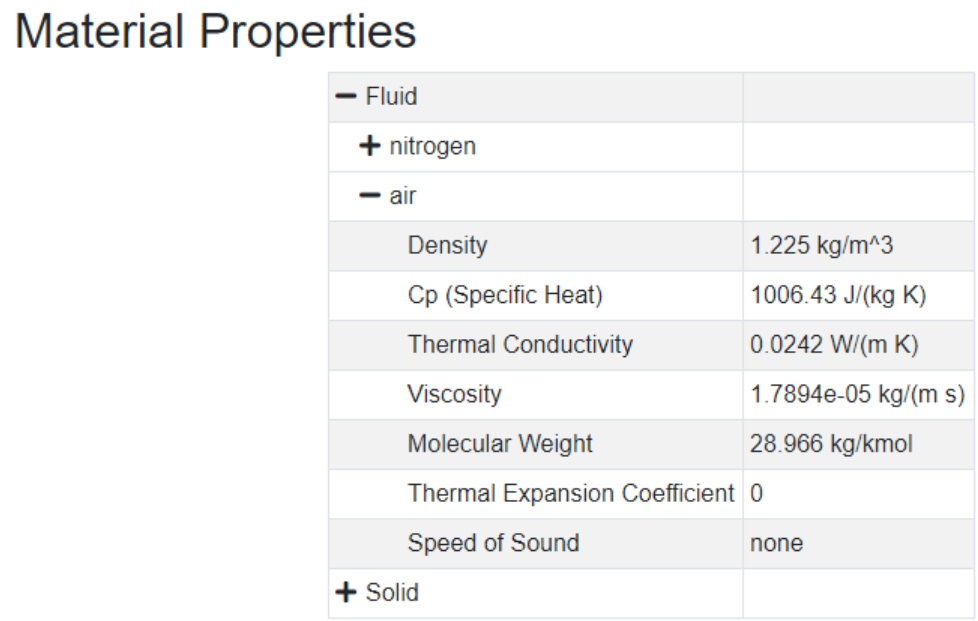


Models

Model	Settings
Heat Transfer	Enabled
Space	3D
Time	Steady
Viscous	Standard k-epsilon turbulence model
Wall Treatment	Standard Wall Functions

The **Material Properties** section, under **Physics**, includes the properties of the materials used in the simulation, such as the density, molecular weight, etc.

Figure 45.13: An Example of the Physics / Material Properties Settings



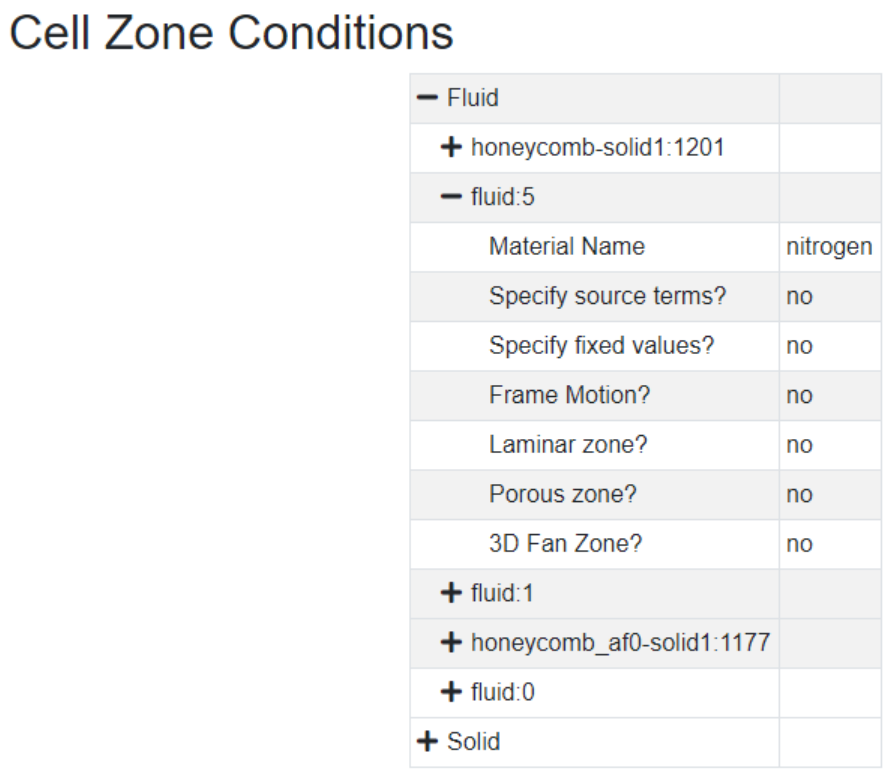
Material Properties

− Fluid	
+ nitrogen	
− air	
Density	1.225 kg/m^3
Cp (Specific Heat)	1006.43 J/(kg K)
Thermal Conductivity	0.0242 W/(m K)
Viscosity	1.7894e-05 kg/(m s)
Molecular Weight	28.966 kg/kmol
Thermal Expansion Coefficient	0
Speed of Sound	none
+ Solid	

The **Cell Zone Conditions** section, under **Physics**, includes the specific setting for all cell zones (fluid and solid) used in the simulation, such as the material assignment, whether its a porous zone or not, whether there is motion or not, etc.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 45.14: An Example of the Physics / Cell Zone Conditions Section



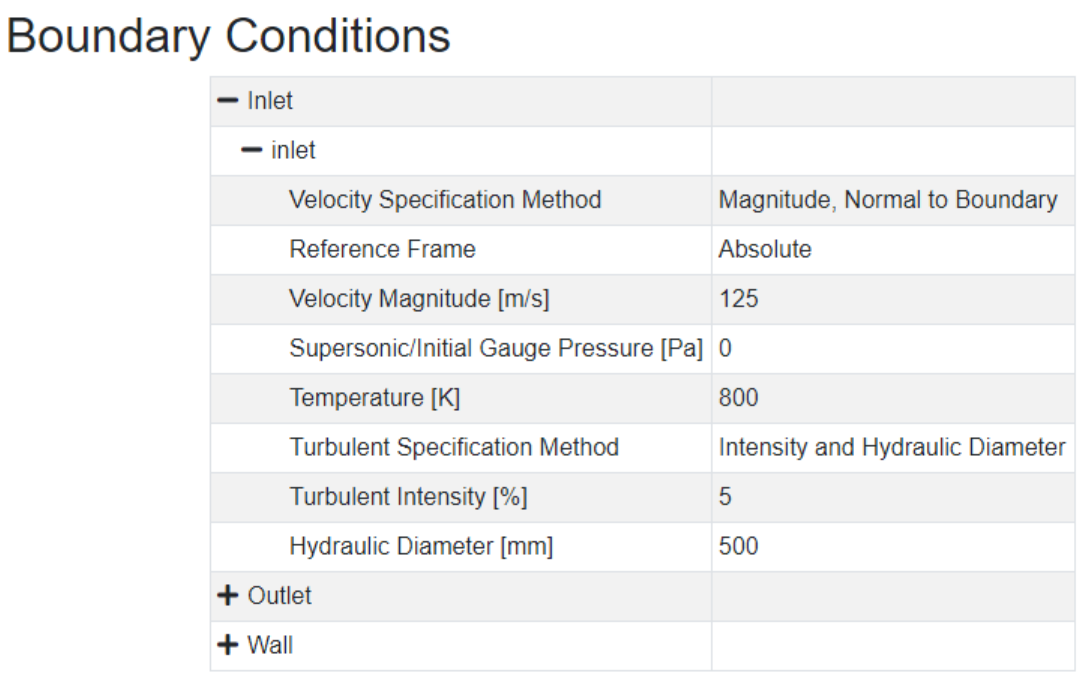
Cell Zone Conditions

− Fluid	
+ honeycomb-solid1:1201	
− fluid:5	
Material Name	nitrogen
Specify source terms?	no
Specify fixed values?	no
Frame Motion?	no
Laminar zone?	no
Porous zone?	no
3D Fan Zone?	no
+ fluid:1	
+ honeycomb_af0-solid1:1177	
+ fluid:0	
+ Solid	

The **Boundary Conditions** section, under **Physics**, includes the specific setting for all boundaries (inlets, outlets, walls, etc.) used in the simulation, such as the velocity specification, temperature, etc.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 45.15: An Example of the Physics / Boundary Conditions Section



Boundary Conditions

− Inlet	
− inlet	
Velocity Specification Method	Magnitude, Normal to Boundary
Reference Frame	Absolute
Velocity Magnitude [m/s]	125
Supersonic/Initial Gauge Pressure [Pa]	0
Temperature [K]	800
Turbulent Specification Method	Intensity and Hydraulic Diameter
Turbulent Intensity [%]	5
Hydraulic Diameter [mm]	500
+ Outlet	
+ Wall	

Note:

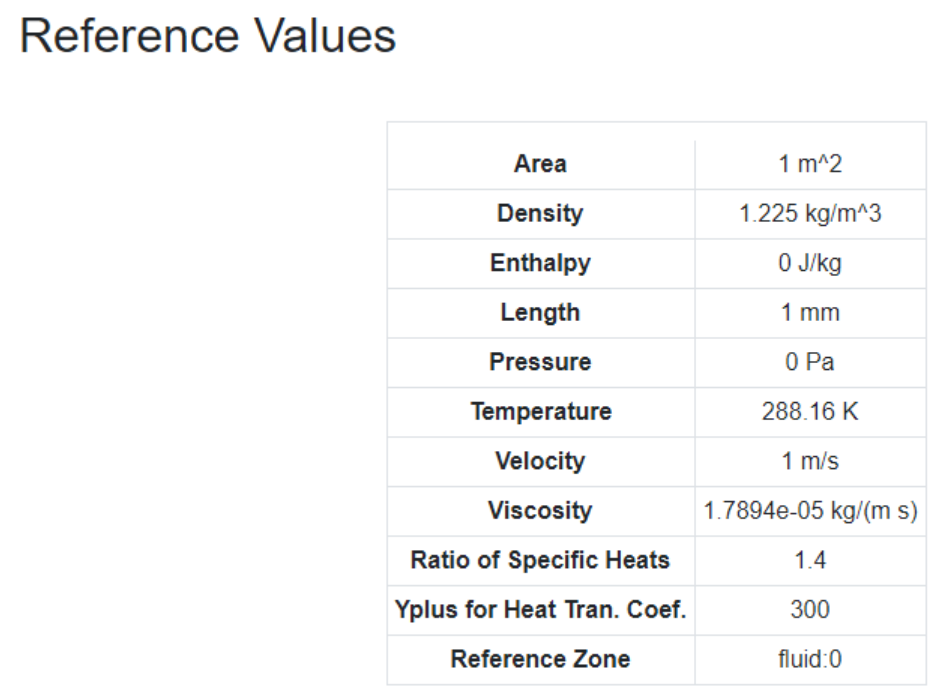
If you use any named expressions as part of your boundary condition definitions, the named expressions are displayed in the Boundary Condition table as hyperlinks that take you to the Named Expressions section of the simulation report.

− inlet	
Velocity Specification Method	Magnitude, Normal to Boundary
Reference Frame	Absolute
Velocity Magnitude	velocity_expression
Supersonic/Initial Gauge Pressure [Pa]	0
Temperature [K]	800
Turbulent Specification Method	Intensity and Hydraulic Diameter
Turbulent Intensity [%]	10
Hydraulic Diameter [mm]	500

The **Reference Values** section, under **Physics**, includes the reference values for several key physics-related variables in your simulation, such as density, pressure, temperature, etc.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 45.16: An Example of the Physics / Reference Values Section

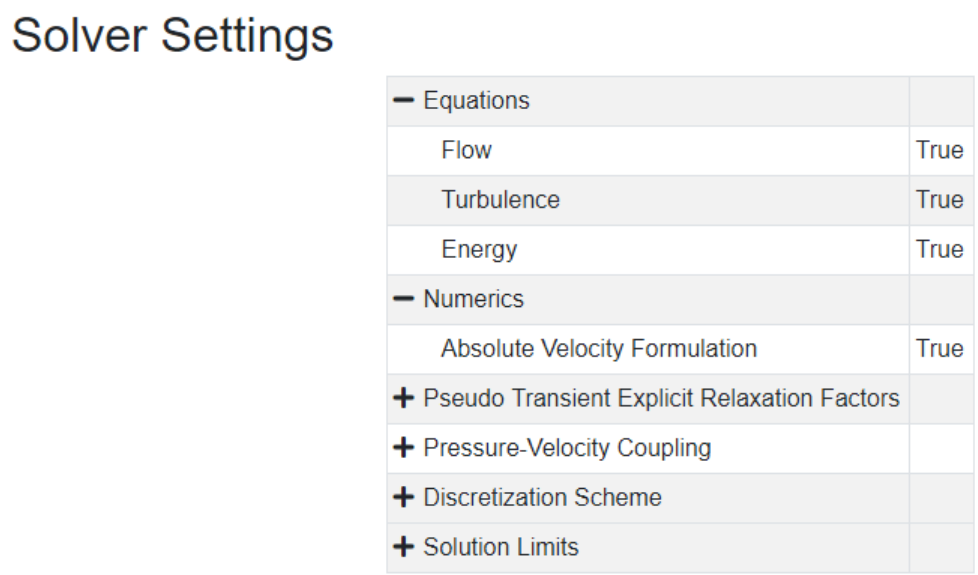


Reference Values

Area	1 m^2
Density	1.225 kg/m^3
Enthalpy	0 J/kg
Length	1 mm
Pressure	0 Pa
Temperature	288.16 K
Velocity	1 m/s
Viscosity	1.7894e-05 kg/(m s)
Ratio of Specific Heats	1.4
Yplus for Heat Tran. Coef.	300
Reference Zone	fluid:0

The **Solver Settings** section includes the numerical parameters of the simulation, such as the equations solved, discretization scheme, etc.

Figure 45.17: An Example of the Solver Settings Section



Solver Settings

− Equations	
Flow	True
Turbulence	True
Energy	True
− Numerics	
Absolute Velocity Formulation	True
+ Pseudo Transient Explicit Relaxation Factors	
+ Pressure-Velocity Coupling	
+ Discretization Scheme	
+ Solution Limits	

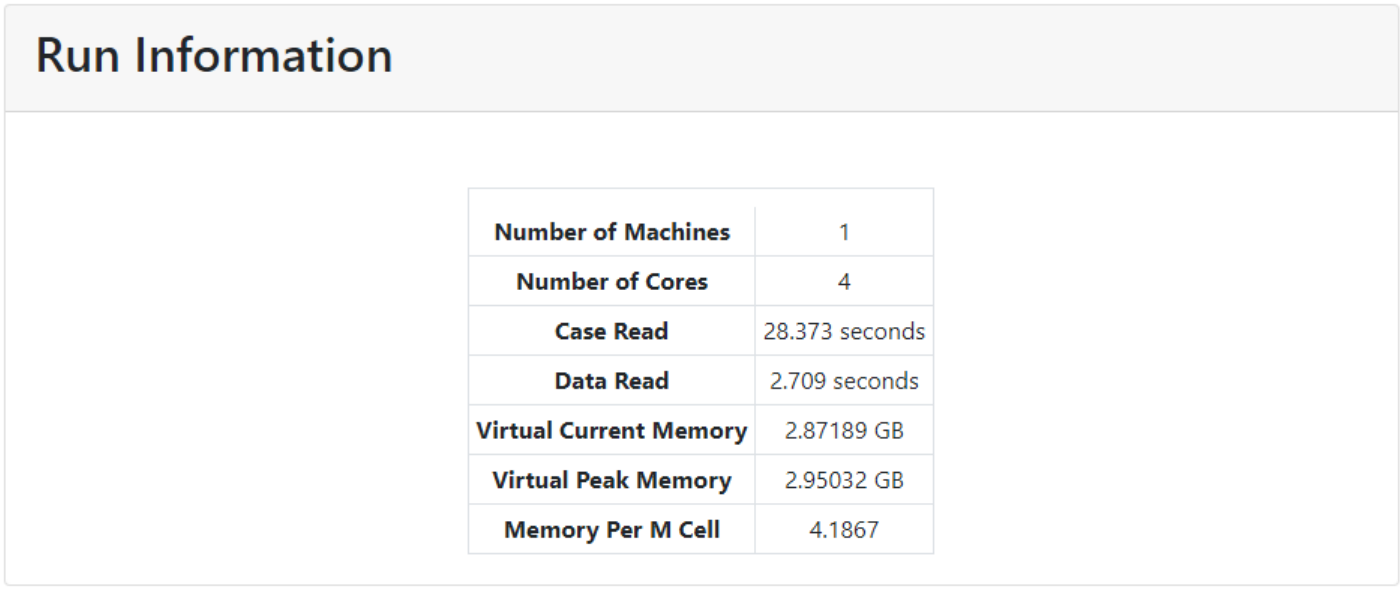
45.4.4. Viewing Run Information

The **Run Information** section contains information such as the number of machines and cores used, memory usage, etc.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You also have the option of adding licensing information to your report by selecting the **Include License Usage** option.

Figure 45.18: An Example of the Run Information Section



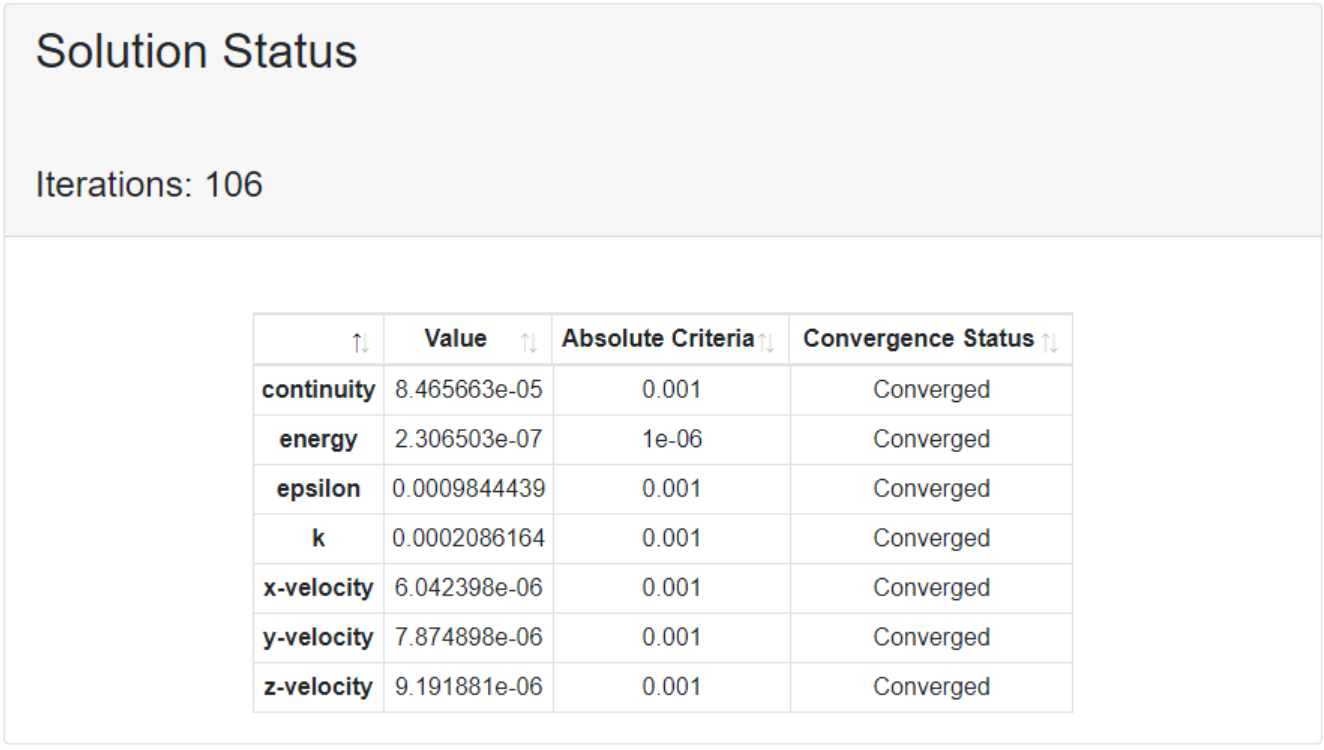
Run Information

Number of Machines	1
Number of Cores	4
Case Read	28.373 seconds
Data Read	2.709 seconds
Virtual Current Memory	2.87189 GB
Virtual Peak Memory	2.95032 GB
Memory Per M Cell	4.1867

45.4.5. Viewing Solution Status

The **Solution Status** section contains information related to the simulation residuals and the convergence status.

Figure 45.19: An Example of the Solution Status Section



Solution Status

Iterations: 106

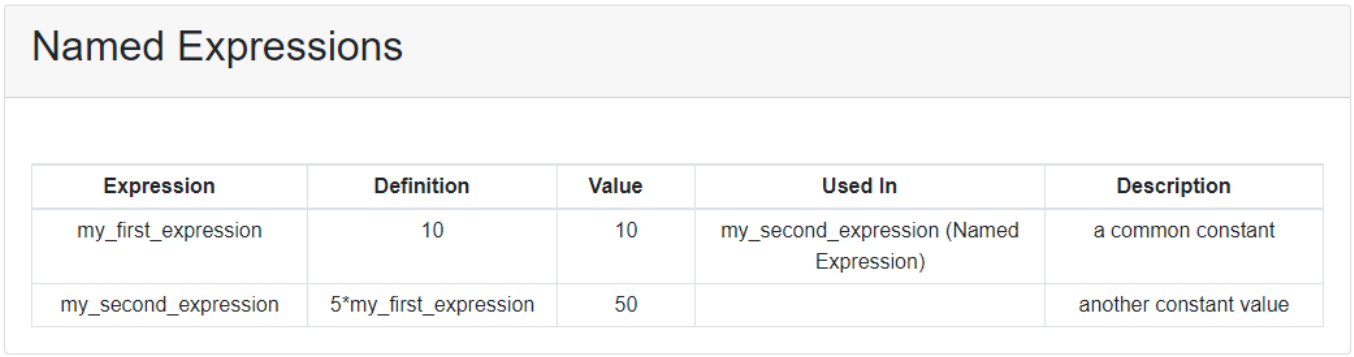
	Value	Absolute Criteria	Convergence Status
continuity	8.465663e-05	0.001	Converged
energy	2.306503e-07	1e-06	Converged
epsilon	0.0009844439	0.001	Converged
k	0.0002086164	0.001	Converged
x-velocity	6.042398e-06	0.001	Converged
y-velocity	7.874898e-06	0.001	Converged
z-velocity	9.191881e-06	0.001	Converged

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

45.4.6. Viewing Named Expression Information

The **Named Expressions** section includes a table of any named expressions from your simulation, along with their definitions, descriptions and where they are used

Figure 45.20: An Example of the Named Expressions Section



Named Expressions

Expression	Definition	Value	Used In	Description
my_first_expression	10	10	my_second_expression (Named Expression)	a common constant
my_second_expression	5*my_first_expression	50		another constant value

Note:

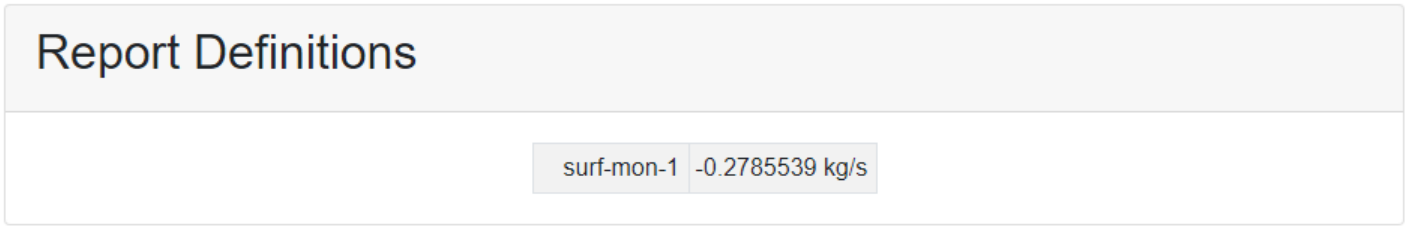
Any named expressions that are utilized as boundary conditions are displayed in the Named Expressions table as hyperlinks that take you to the Boundary Conditions section of the simulation report.

Expression	Definition	Value	Used In	Description
my_first_expression	10	10	my_second_expression (Named Expression)	a common constant
my_second_expression	5*my_first_expression	50		another constant value
velocity_expression	150 [m^s-1]	150 [m s^-1]	inlet (Velocity Magnitude)	a sample expression

45.4.7. Viewing Report Definition Information

The **Report Definitions** section includes a table of any report definitions from your simulation.

Figure 45.21: An Example of the Report Definitions Section



Report Definitions

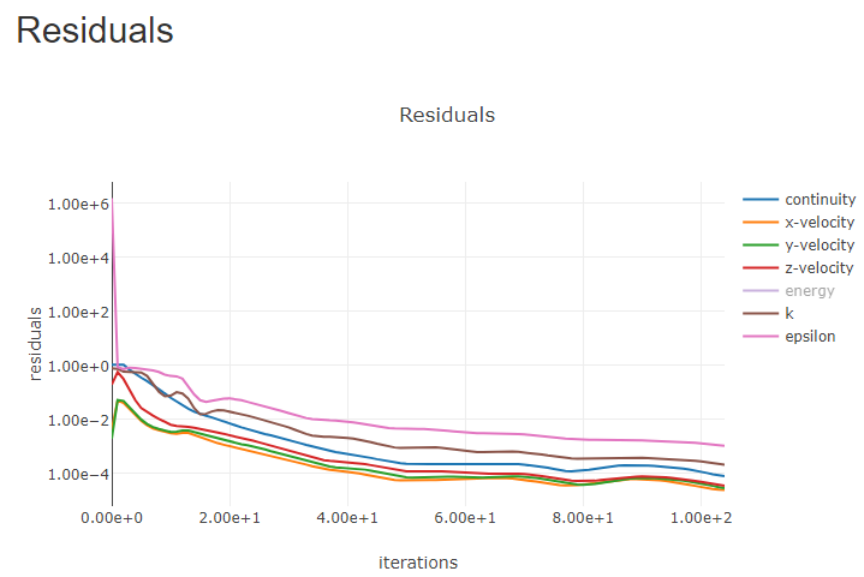
surf-mon-1	-0.2785539 kg/s

45.4.8. Viewing Plot Information

The **Plots** section includes **Residuals** plots and **Report Plots**, if available in your simulation.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 45.22: An Example of the Plots Section (Residuals)



Note:

You can adjust the visible range of the axes by clicking and dragging the mouse along the plot (vertically or horizontally).

Note:

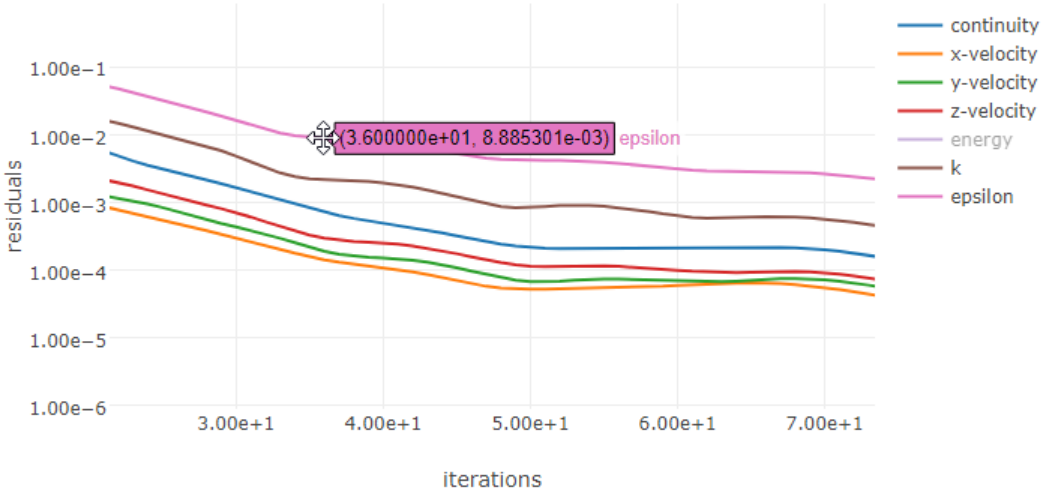
You can also zoom into the plot by clicking and dragging the mouse along the plot (diagonally).

There are several built-in features for viewing plot information, using the toolbar in the upper right-hand corner.

- **Download Plot as a PNG**: allows you to save a copy of a plot as an image file. Specify the name and location of your file, or use the default name and location (the default location is your working folder).
- **Zoom**: allows you to adjust the zoom level.
- **Pan**: allows you to move vertically or horizontally across the plot.
- **Zoom in**: allows you to zoom into the view.
- **Zoom out**: allows you to zoom out of the view.
- **Autoscale**: allows you to return to the view.
- **Reset axes**: allows you to return to the original axis orientation.
- **Show closest data on hover**: allows you to see specific data as you hover over the plot.

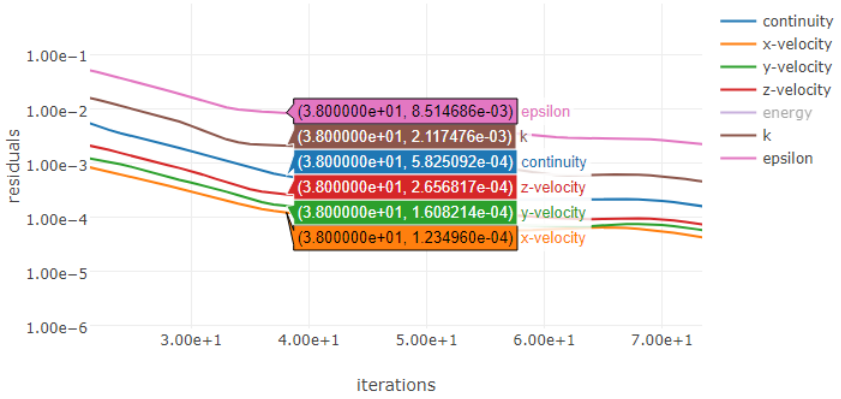
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 45.23: Hovering and Showing Data in a Plot



- **Compare data on hover**: allows you to see and compare specific data as you hover over the plot.

Figure 45.24: Hovering and Comparing Data in a Plot



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

45.4.9. Viewing Contours, Vectors, Pathlines, LICs, XY Plots, Scenes, Histograms, and Animations

Graphical displays of **Contours**, **Vectors**, **Pathlines**, **LICs**, **XY Plots**, **Scenes**, **Histograms**, and **Animations** are able to be included in your report if they are available in your simulation.

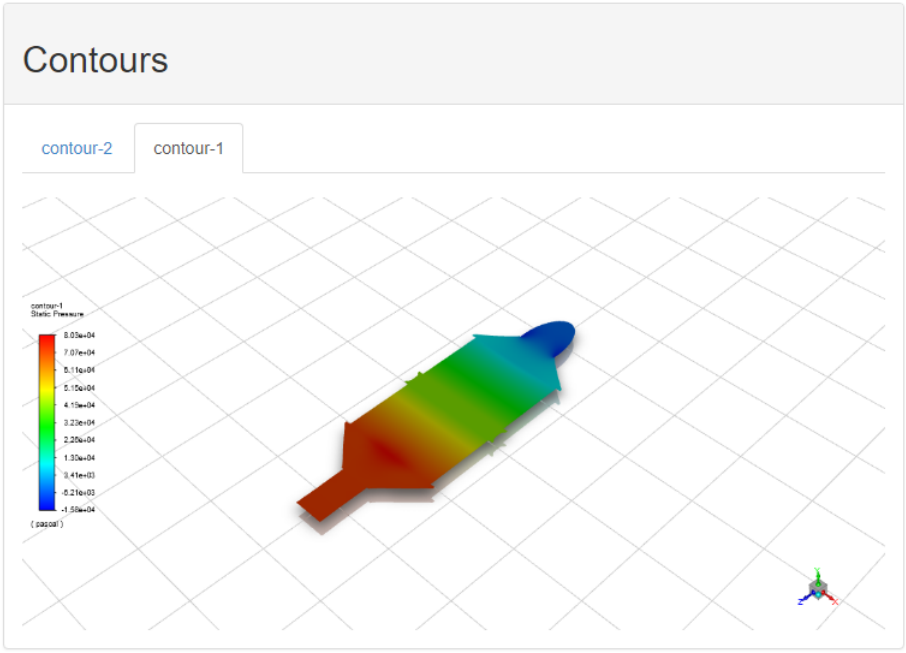
Note:

The layout of your results can also be arranged in a tabular format. See Changing the Layout of Your Results (p. 4384) for details.

Note:

AVZ animations are represented in your simulation report using static proxy images. You can play an embedded AVZ animation directly within your simulation report. Hovering over the static proxy image displays a 'play' button that you can then select to play the embedded animation.

Figure 45.25: Example of Contours



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 45.26: Example of Vectors

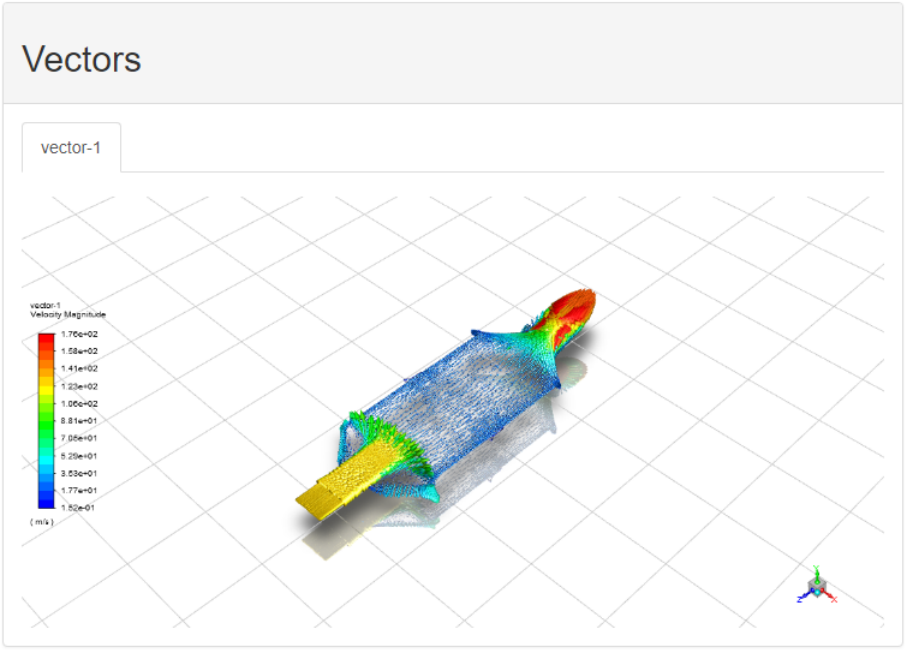
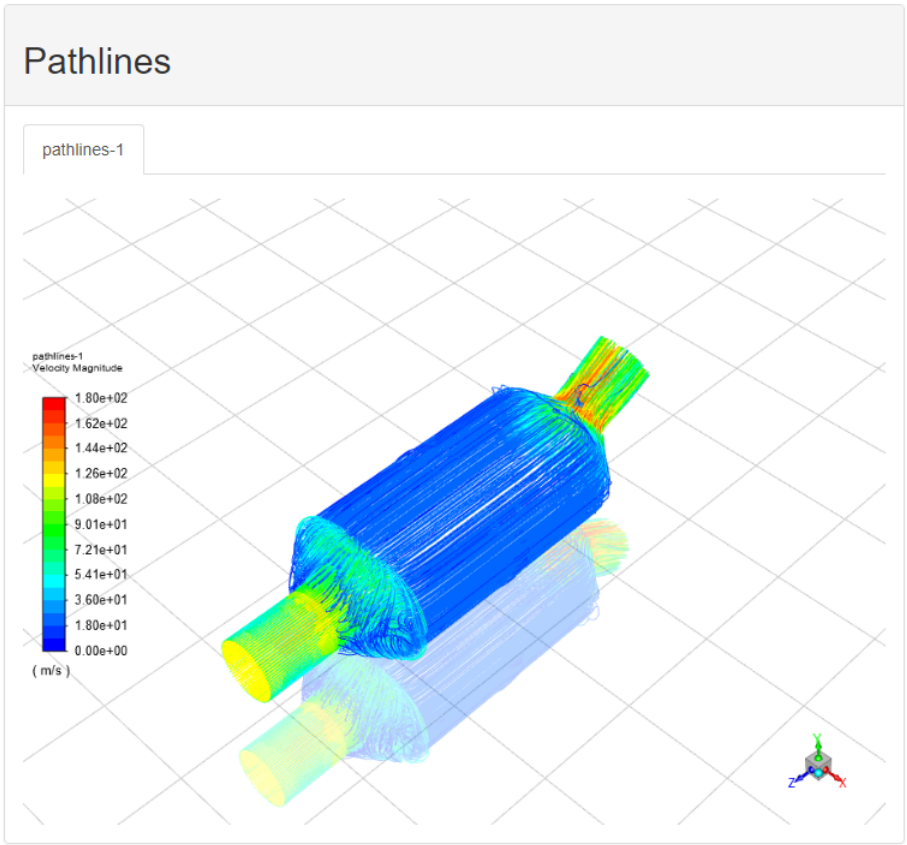
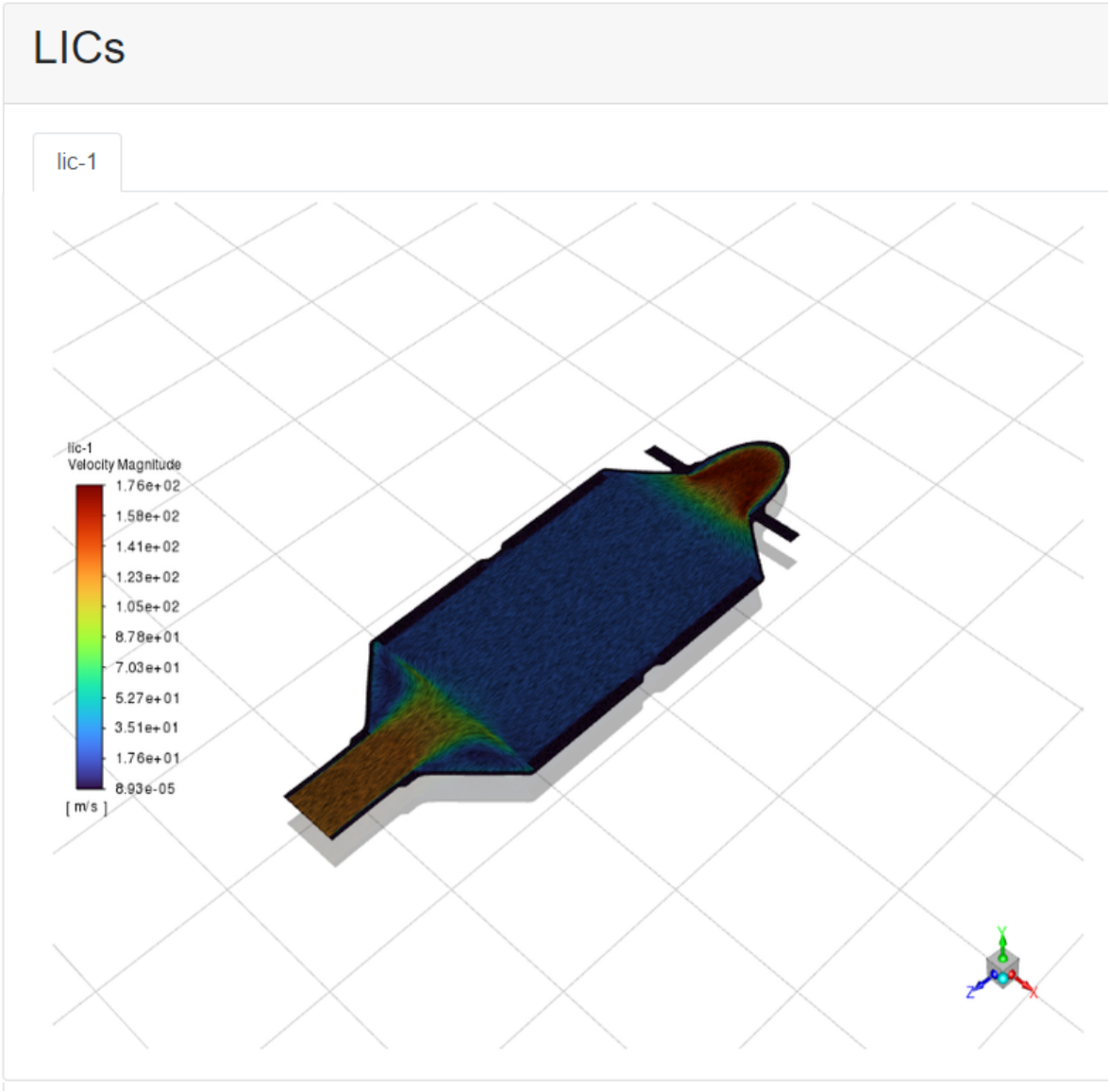


Figure 45.27: Example of Pathlines



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 45.28: Example of LICs



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 45.29: Example of XY Plots

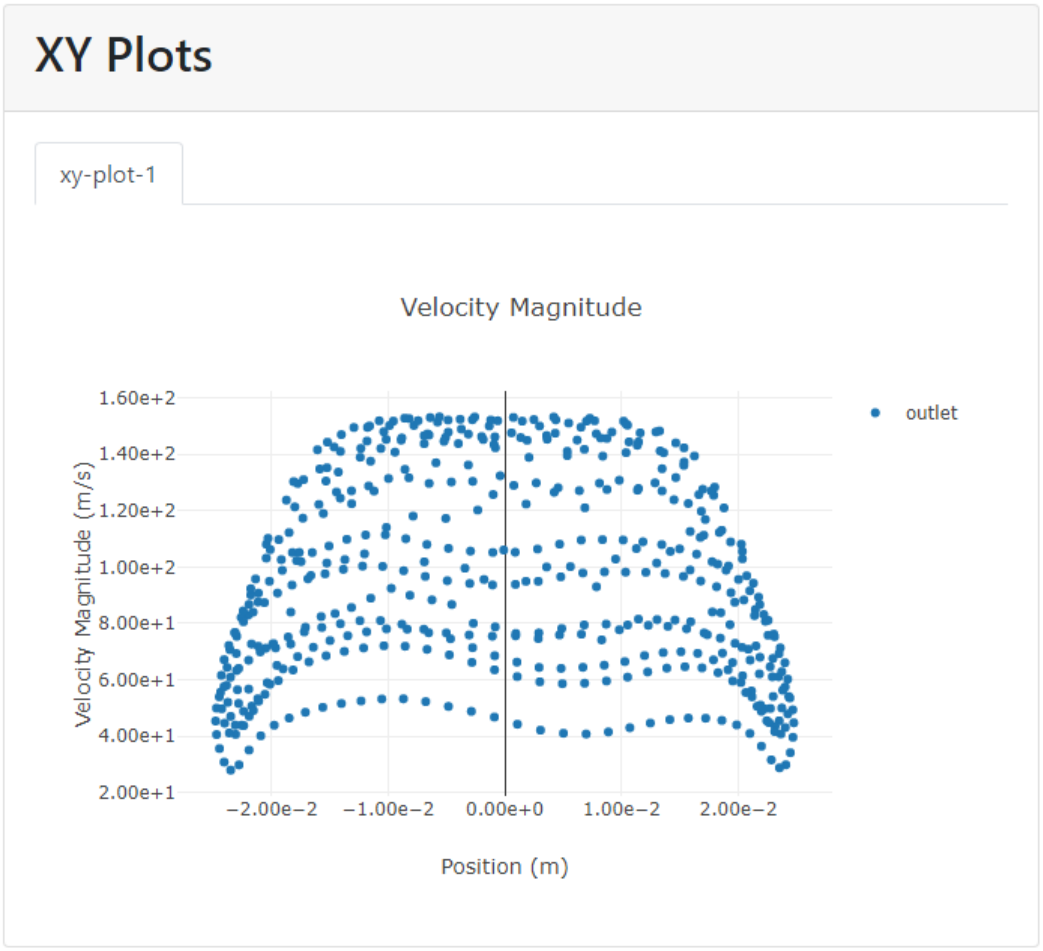
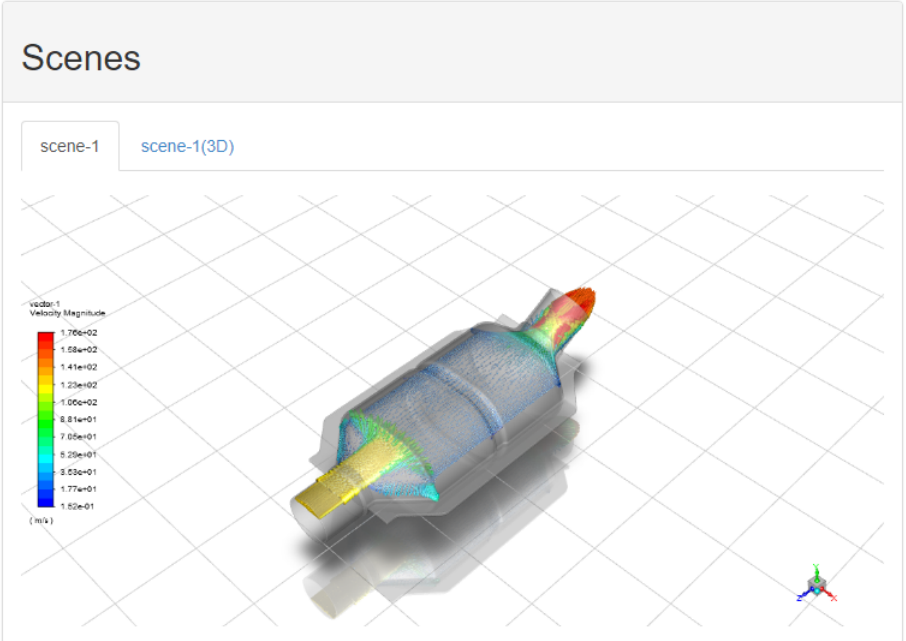
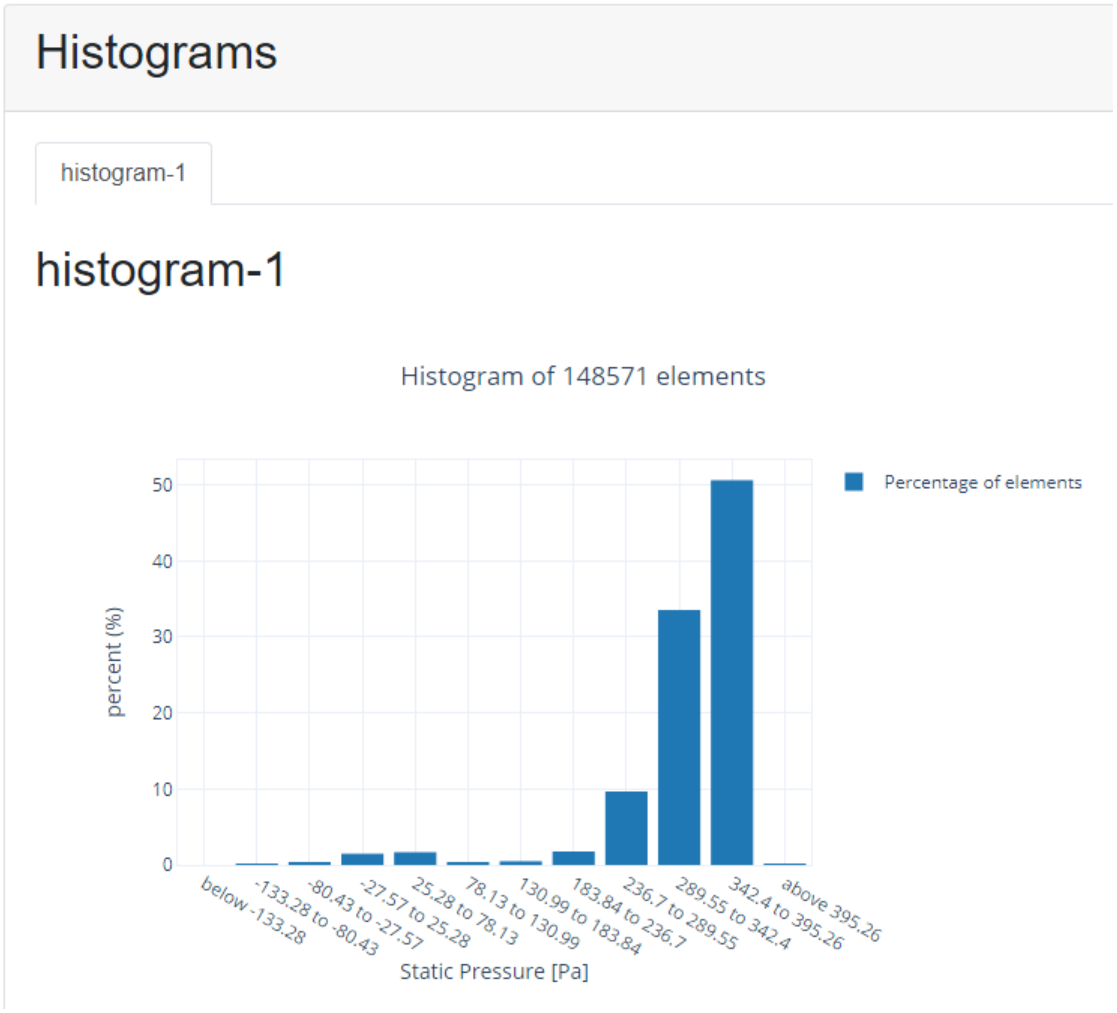


Figure 45.30: Example of a Scene



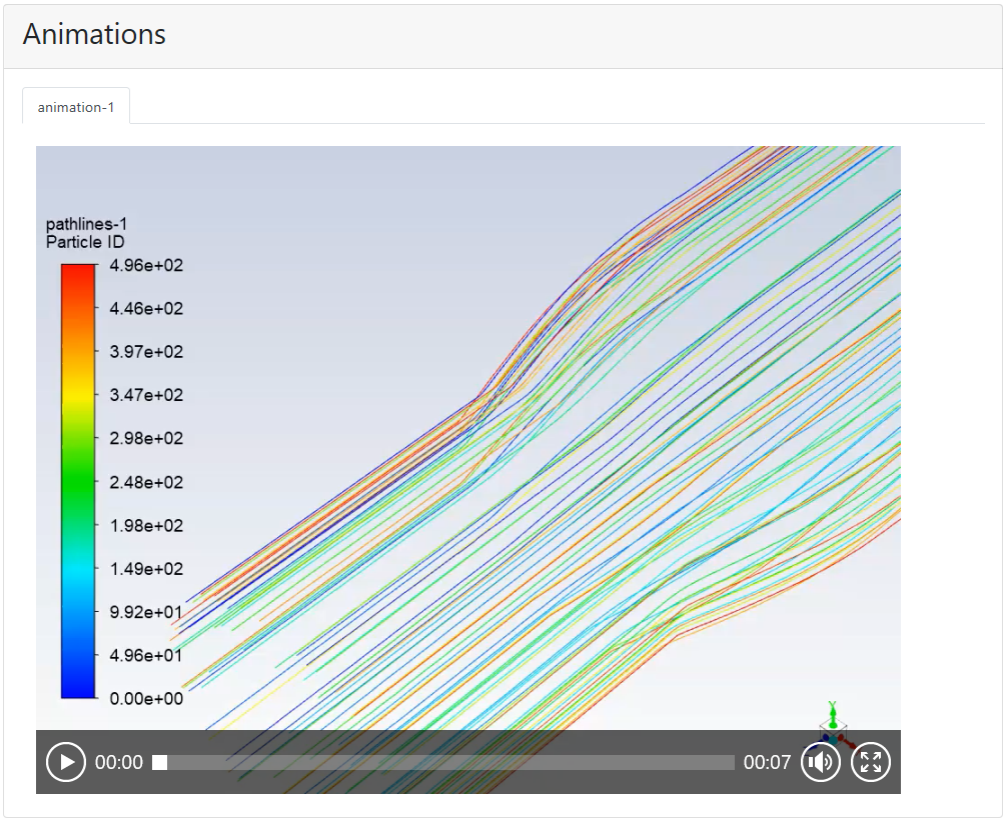
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 45.31: Example of a Histogram



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 45.32: Example of an Animation



45.5. Saving Simulation Reports

Once generated, you can save your reports as PDF or standalone HTML using the **Export** button. You can also export a PowerPoint presentation where, for your convenience, you are supplied with a default Ansys PowerPoint template. By default, the output is saved to your working directory, though you can specify any desired location. As such, you can run variations of the same simulation and generate the corresponding reports separately for later viewing and analysis.

Note:

Exporting a simulation report as a PDF file under Linux must be performed in an environment that supports X11 applications. This includes running an X11 server and having a properly configured X11 display environment with proper authentication, etc.

45.6. Additional Simulation Report Options

Use the **Options** button to access additional report options such as:

- **Delete Report**: removes the generated report and any associated images and files.
- **Duplicate Report Settings**: creates a copy of the report (and any custom settings it may have) and loads it into the **Simulation Report Outline**. The **Report Name** of the copy is the name of

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the original report appended with a "1" at the end. You can revisit previous reports (and their copies) by using the **Current Report** drop-down menu.

- **Reset to Default Settings**: resets your report settings to their default values. You can also use the corresponding icon ( ) at the top of the **Simulation Report Outline** task page.

45.7. Customizing Simulation Reports

45.7.1. Customizing Your Report Image Settings

Under **Image Settings**, you can choose the default view orientation and resolution for your included graphics results (such as saved views/states, or common orientations such as top, bottom, left, right, and so on).

The default is **Object State View** (the most recent saved state of the corresponding view). These settings will be applied to all the graphics results images in your report. Use the **Preview View** button to display a preview of an image with your image settings in the graphics window.

Customize the image resolution using the **X Resolution** and the **Y Resolution** fields. These let you control the resolution at which the graphics are captured for use in the simulation report. By default, the values are set to those found in the **Save Picture** dialog (Using the Save Picture Dialog Box (p. 1020)). Use the **Capture Current Resolution** button to obtain the current resolution settings for the current graphics window.

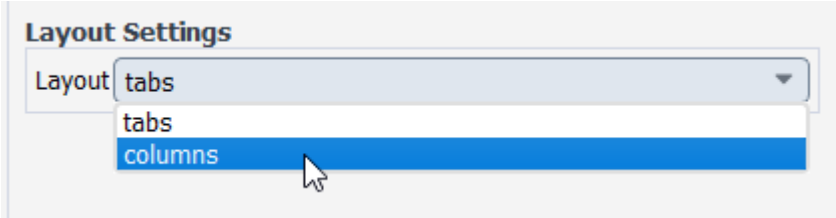
If you want to save your results as separate image files (hard copies), enabling the **Save Hardcopy** check box will save a separate image file to your working folder.

45.7.2. Changing the Layout of Your Results

For the various types of results plots (contours, vectors, etc.) you can change various aspects of the general layout under **Layout Settings**.

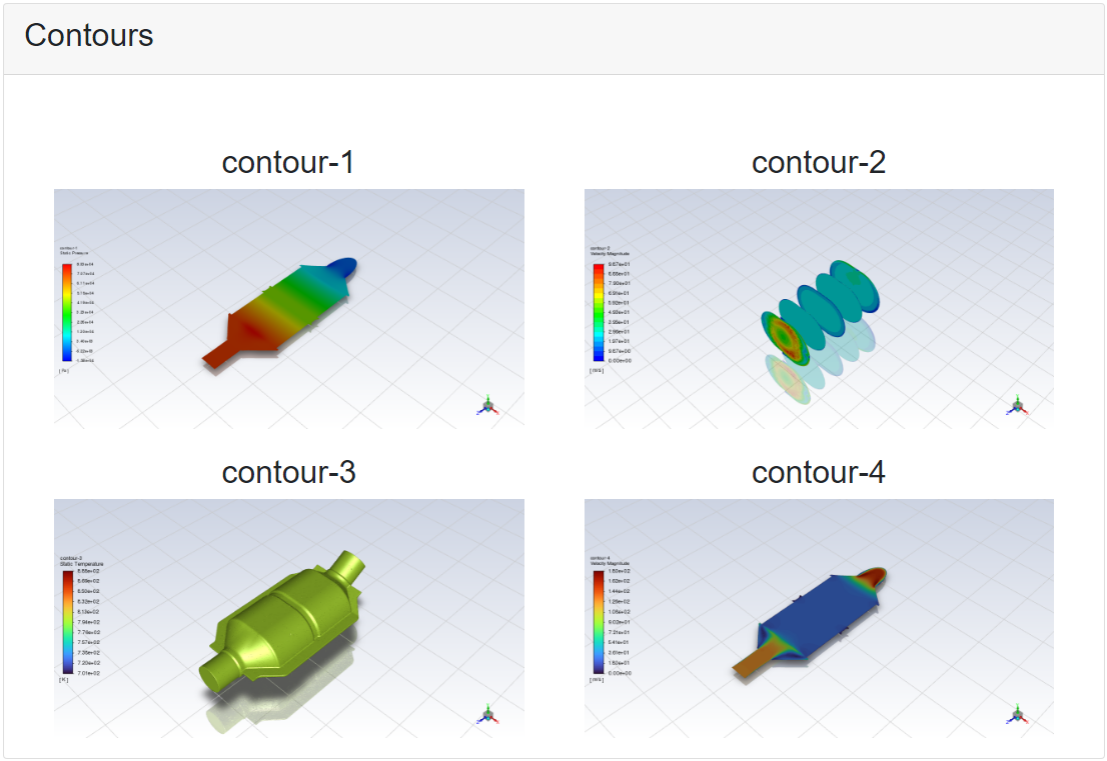
Use the **Layout** field to control the arrangement of multiple plots. By default, when **tabs** is selected, multiple plots of the same type are presented in separate tabs. You can also choose **columns** so that multiple plots are arranged in a tabular format and can be viewed at the same time. When you choose **columns**, the **Number of Columns** field becomes available and is set to 1 by default. but can be set to an appropriate value.

Figure 45.33: Layout Settings for Result Plots



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 45.34: Example of Contours (Columns)



The layout setting can be applied per category (just for a series of contour plots, or a series of vector plots, etc), or as a general setting to apply to all your result plots (see Setting General Report Properties (p. 4364) for more information).

45.7.3. Customizing Your Report Settings Using Templates

You can save your custom simulation report configuration settings (availability of particular nodes, headings, graphics, notes, etc.) as a separate template file so that you can easily recreate your specialized simulation reports for later use. You can access report templates using the icons in the upper portion of the task page.

- : opens a file browser where you can locate and select a Fluent simulation report template (`*.fsrt`) file that contains customized settings.
- : opens a file browser where you can specify the location and the name of a Fluent simulation report template (`*.fsrt`) file where you can save your report settings for later use.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Customizations to your reports can be the inclusion or exclusion of any sections of the report, the order of the sections, notes, and the inclusion or exclusion of various other details of the report.

Note:

While recording a journal of your session, whenever your simulation involves setting up and generating a simulation report, Fluent uses locally stored report templates during the journaling process to keep track of your report settings.

45.7.4. Adding Additional Graphics to Your Report

If you would like to add one or more specific graphics that are not part of the default definitions of your report, you can add it as part of the **User Data** node of the report.

1. Add external graphic images to your report:
   a. Specify the appropriate **Image Path** for your external graphics image file, or browse for the appropriate file and its directory.
   b. Specify the appropriate **Image Name** for your external graphics image file.
   c. Select **Add Image** to add the image to your report.
2. Add graphics from your simulation directly to your report:
   a. In your simulation, proceed to set up a specific view in the graphics window, whether its a specifically orientated mesh display, plane cut, etc.
   b. Right-click in the graphics window and select **Add to Report** from the context menu. This will capture the graphics window contents.
3. Note the new image file listed under **User Data** in the **Simulation Report** task page.
4. Click **Generate** to regenerate the report with the new image under the **User Data** section.

45.7.5. Hiding and Showing Report Sections

By default, all report sections are included in a simulation report, as indicated by the check box in the **Report Section** outline view. To obtain a more simplified and customized report, deselect any of the selected nodes and click **Generate**. The resulting report will include only those selected sections.

45.8. Generating Reports Using the Text User Interface (TUI)

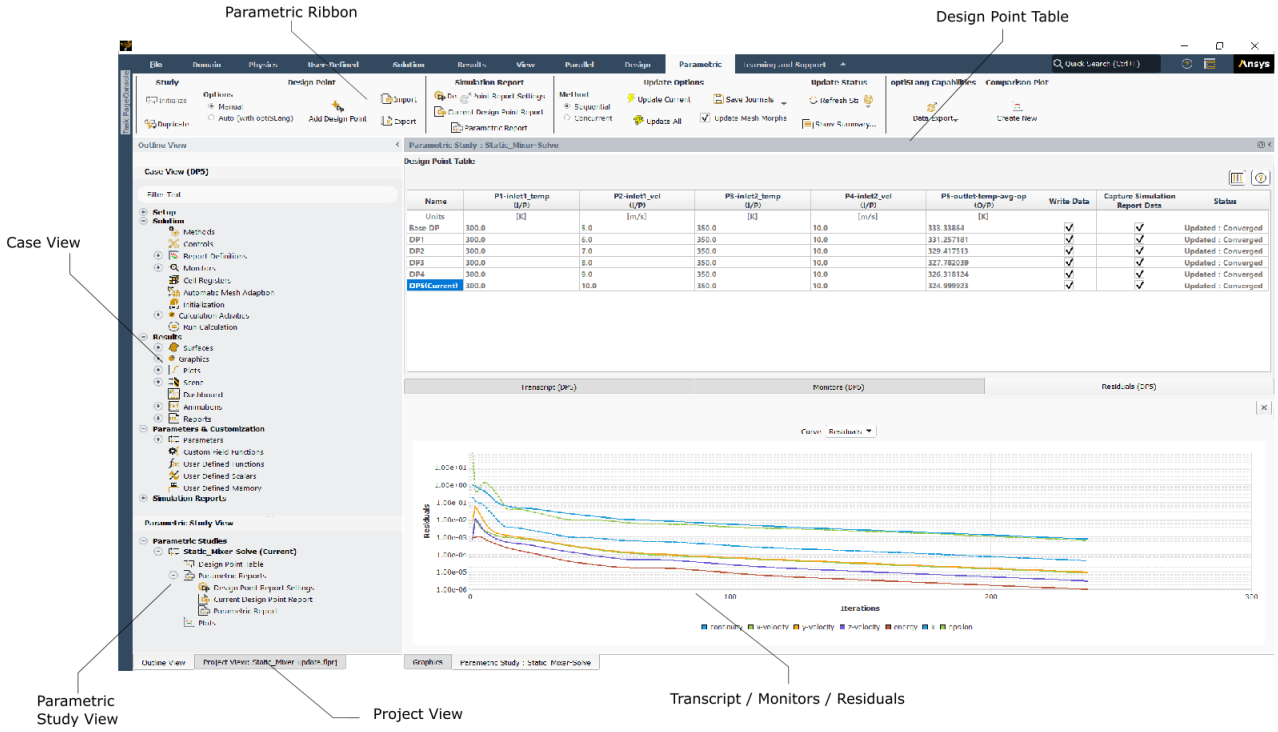
There are several commands available for working with your simulation reports via the Fluent text command user interface (TUI), under the `/report/simulation-reports/` menu. See `report/` for more details about the available commands.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 46: Performing Parametric Studies

If you have an Ansys Fluent case file with input and/or output parameters, you can perform a parametric case study directly within the Ansys Fluent interface. Use the **Parametric** tab of the ribbon to access your parameters and control the parametric study (see Using the Parametric Ribbon (p. 4390) for details).

Figure 46.1: Various Views and Components of a Parametric Study



Once you initialize the parametric study in the ribbon, you can then:

- Add design points to your initial case, and access current design points defined for the parametric study. See Working With the Design Point Table (p. 4393) for details.
- Monitor job submission settings where you can control how your parametric run(s) are processed (either remotely, or locally in various configurations).
- Monitor the status of your various parametric solution(s) by viewing solution transcripts, residual plots, and/or monitor plots. See Monitoring and Viewing Design Point Update Status (p. 4417) for details.
- Access simulation reports for individual design points and full parametric runs. See Creating Simulation Reports for Design Points and Parametric Studies (p. 4424) for details.
- When using optiSLang to define and manage your desgin points, you can directly export your design point data to optiSLang and/or access optiSLang directly. See Exporting Parametric Designs to optiSLang (p. 4420) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Use the modified Outline View, where you can see the currently loaded case/data settings, as well as specific input and output parameter settings and the design point table. See Using the Outline View for Parametric Studies (p. 4391) for details.
- Access the **Project View** (using the upper right hand Layout menu) where you can view the various files in your parametric study.

Note:

When you first initialize a parametric study, Ansys Fluent prompts you to create a specialized *project* file to manage various input/output files when performing parametric simulation(s). See Organizing Parametric Studies Using Projects (p. 4438) for more information.

46.1. Prerequisites
46.2. Limitations
46.3. Getting Started With Your Parametric Study
46.4. Using the Parametric Ribbon
46.5. Using the Outline View for Parametric Studies
46.6. Working With the Design Point Table
46.7. Monitoring and Viewing Design Point Update Status
46.8. Creating Simulation Reports for Design Points and Parametric Studies
46.9. Setting Preferences for Parametric Studies
46.10. Viewing the Current Case Parameters
46.11. Managing Files for Your Parametric Studies

Note:

Parametric studies also include a text user interface, described in Parametric Study Text Commands . In addition, the text user interface is read-only when using the CFD-Pro license level.

46.1. Prerequisites

Using parametric simulations has the following prerequisites:

- Typically, you should have a `.cas` file with input and output parameters defined. Alternatively, you can read a case file, initialize a parametric study, and then add any input and/or output parameters as needed.
- All post-processing images, plots and other quantities must already be included in the design points and their reports already set up prior to updating the "Base DP" design point.
- If you want to use automatic designs of experiments (DOEs) creation, an optiSLang installation is required, though you do not necessarily need a license.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

46.2. Limitations

Using parametric simulations has the following limitations:

- When updating design points in a parametric study sequentially and capturing simulation report data, a simulation report-related state file will be written to the current simulation directory of the project folder, rather than in the simulation report folder. The file does not get registered with the project, and can be ignored or deleted without any issues.
- When saving parametric projects on Windows, the **Save As Copy** command can be problematic if there are too many characters in the project directory (> 170 characters).
- Parameters are not yet supported in Fluent in meshing mode
- Graphical interactions within the Parametric Study window and the Project View are not captured while recording journals.
- When updating local parametric runs concurrently, you cannot update design points when the base session is launched in distributed mode.
- Design points created in optiSLang cannot be updated in batch mode on Linux where the graphical user interface is not available.
- When you export your simulation report as a PowerPoint presentation, named expression tables, that can be long and extensive, are not automatically split across multiple slides.
- If multiple report definition monitors are combined into a single report file, not all of the monitors may appear in the monitor plot accessed from the Parametric Design Table context menu. This can be avoided by placing the data for each monitor into its own report file.
- If parametric projects are given long names and/or placed deep in a directory structure, problems may occur due to limitations in the length of the file path (for example, the Windows filepath limit is ~256 characters).
- If you choose to stop the processing of a batch of design point updates through optiSLang optimization, the individual design point status may be designated as failed in Fluent, even for those design points that were previously completed. This is because the optiSLang server no longer retains the outputs for any design point updates that have already been completed.

46.3. Getting Started With Your Parametric Study

If you have an existing case file, you can perform a parametric analysis of the case file using the following steps:

1. Open the case file of interest in Fluent, and define any input and/or output parameters of interest if you have not already done so.
2. In the **Parametric** Ribbon, under **Study**, select **Initialize** to prepare the necessary files and to activate the rest of the ribbon functions.
3. Set up and run your parametric study.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

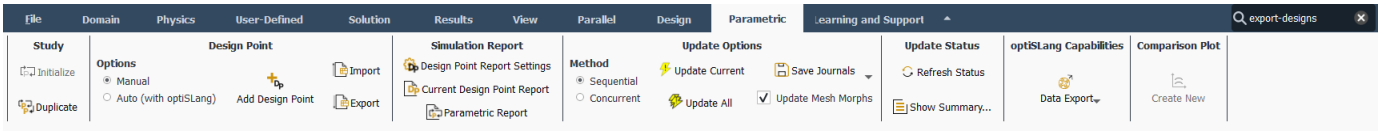
You can use the **Parametric** Ribbon as your workflow where you can add design points, set up report settings, update design points and review their status (see Using the Parametric Ribbon (p. 4390) for more information), or you can use the Outline View to set up and perform your analysis (see Using the Outline View for Parametric Studies (p. 4391) for more information).

Run your parametric study by updating your design points, generating individual solutions and results for each design point (see Working With the Design Point Table (p. 4393) for more information).

4. Analyze the results of your study, by analyzing individual results in Fluent, or by generating and studying parametric simulation reports and comparing data between design points (see Monitoring and Viewing Design Point Update Status (p. 4417) and Creating Simulation Reports for Design Points and Parametric Studies (p. 4424) for more information).

46.4. Using the Parametric Ribbon

The **Parametric** tab of the ribbon lets you initialize your parametric study in Fluent, and provides key functionality when running parametric flow simulations.



- **Study**: allows you to **Initialize** or to **Duplicate** the parametric study (if the need arises) when performing your parametric analysis.

- **Design Point**: provides easy access to creating a new design point for your parametric simulation, either manually or automatically. The new design point is added to the **Design Point Table** where you can easily provide new settings for a parametric study. See Adding Design Points (p. 4401) for more information.

  Additionally, you can use comma-separated value (CSV) data for your parametric study. For instance, use the **Export...** option to export the contents of your design point table to an external `.csv` file. Likewise, use the **Import...** option to import an existing CSV file directly into the parametric table. See Importing and Exporting Design Point Tables (p. 4410) for more information.

- **Simulation Report**: provides options for the simulation reports that you can generate for the base case, the current case, and for the entire parametric simulation study that includes all design points. See Creating Simulation Reports for Design Points and Parametric Studies (p. 4424) for details.

- **Update Options**: choose whether to run your design point simulations sequentially (in the same Fluent session) or concurrently (in a new Fluent session). You also have easy access to update the current design point, or update all design points. See Updating Design Points (p. 4395) for more information.

  Use the **Save Journals** option to customize how any journals are created for any design point that needs to be updated. You can then submit these journals to a job scheduler or run the journals in

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

different Fluent session(s). See Saving Journals When Updating Design Points (p. 4411) for more information.

Note:

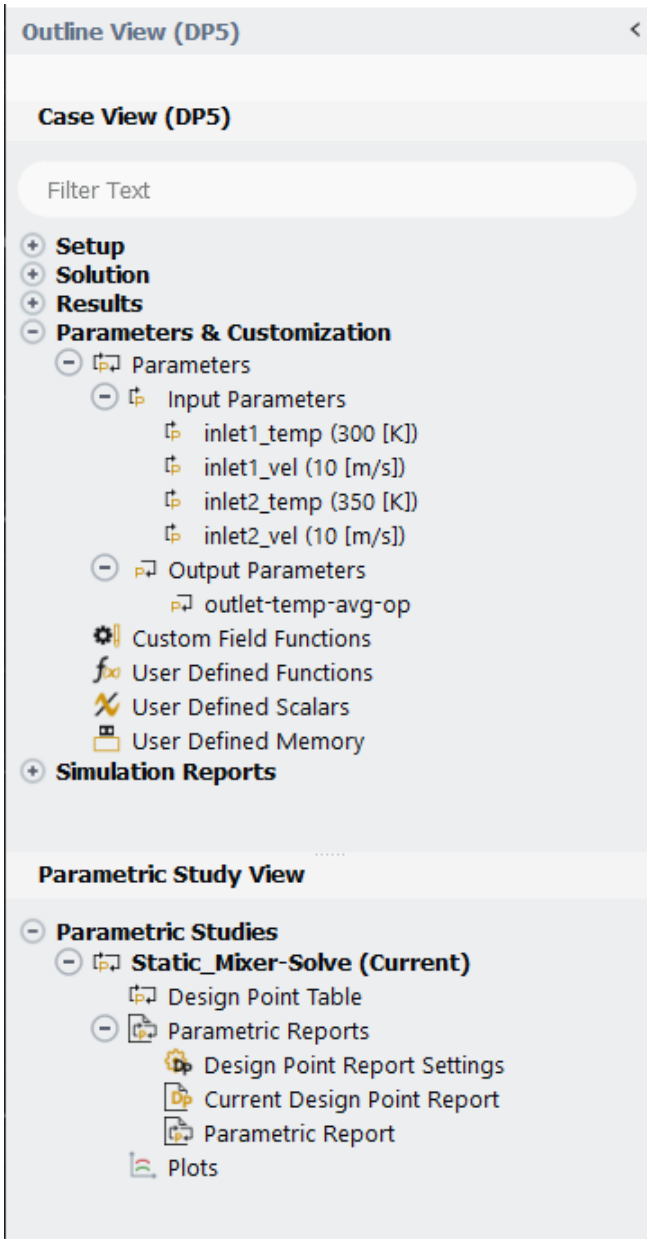
Performing concurrent updates and saving journal files are not available when using the CFD-Pro license level.

- **Update Status**: provides a summary of the status of your simulations, as well as the ability to refresh the overall status of the design points getting updated concurrently. See Monitoring and Viewing Design Point Update Status (p. 4417) for more information.
- **optiSLang Capabilities**: provides access to exporting your optiSLang data. See Exporting Parametric Designs to optiSLang (p. 4420) for more information.
- **Comparison Plot**: provides easy access to creating a new plots where results between design points can be compared. See Comparing Parametric Plots (p. 4421) for more information.

46.5. Using the Outline View for Parametric Studies

For parametric studies, the **Outline View** is split into two views:

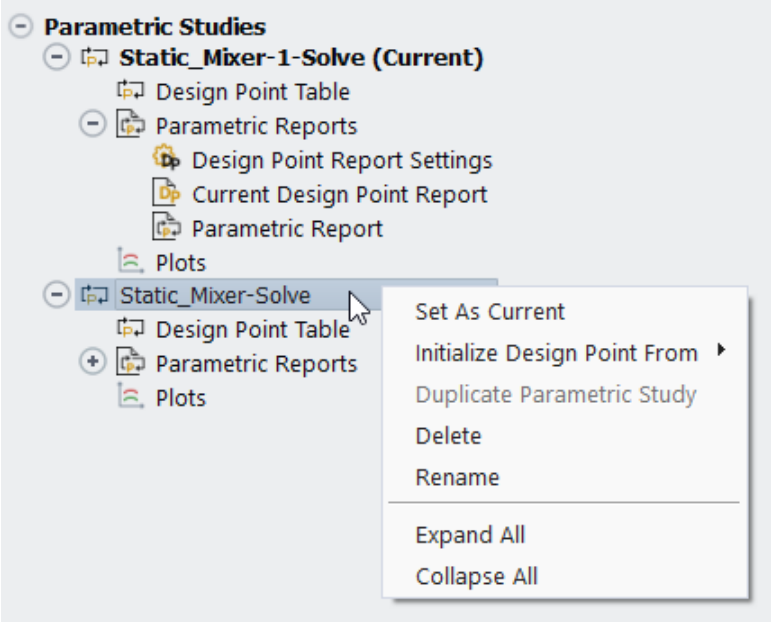
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



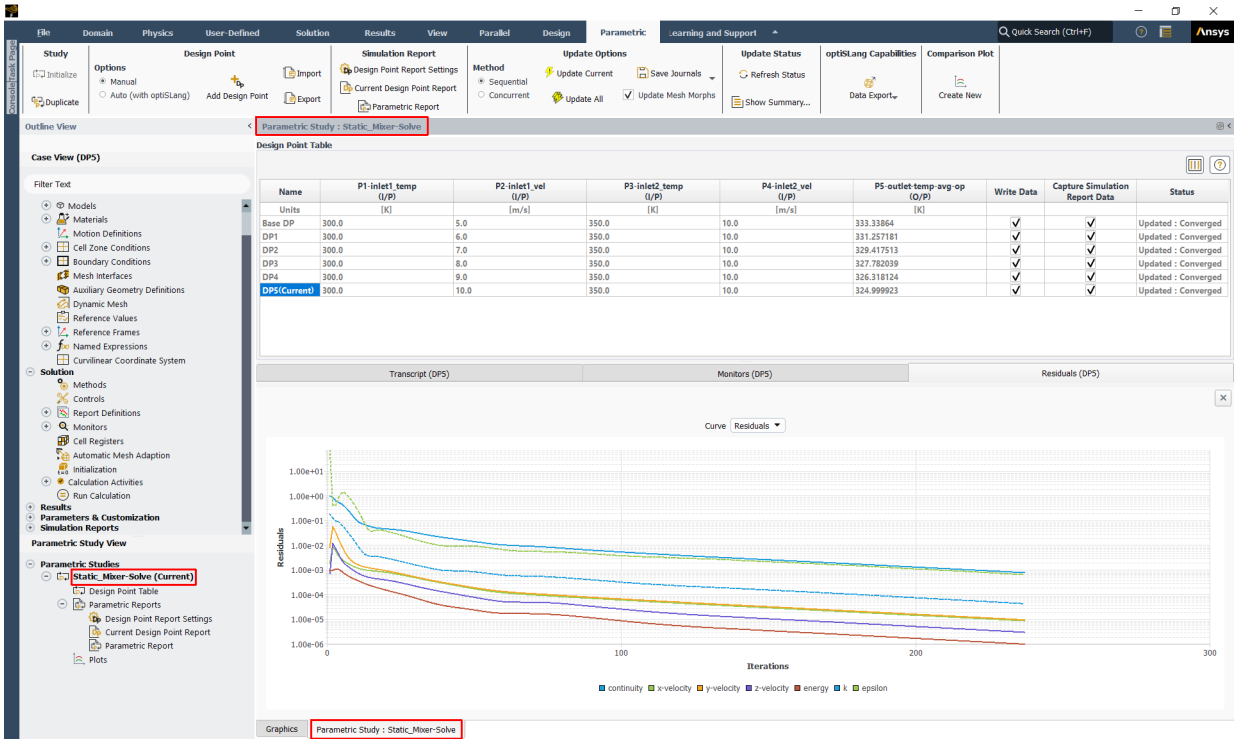
- Use the **Case View** to access and review the case and data settings for the currently loaded design point. Note that if the base design point is the current design point, and is in a cleared state, you can make changes to the base design point here and then (re-)run the study.
- Use the **Parametric Study View** to access and review various aspects of your design point study/studies, such as the corresponding input/output parameters, parametric design point table, parametric reports, and plots.

  Use the context menu to change the current study, initialize design point data, duplicate, delete, or rename a study.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The currently loaded parametric study is highlighted in bold, and its associated information is displayed in the correspondingly named Parametric Study pane (this will contain the design point table, input/output parameters table, etc, for the current parametric study).



46.6. Working With the Design Point Table

By default, Ansys Fluent displays the **Design Point Table** in the **Parametric Study View**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Design Point Table

Name	P1-inlet1_temp (I/P)	P2-inlet1_vel (I/P)	P3-inlet2_temp (I/P)	P4-inlet2_vel (I/P)	P5-outlet-temp-avg-op (O/P)	Write Data	Capture Simulation Report Data	Status
Units	[K]	[m/s]	[K]	[m/s]	[K]			
Base DP(Current)	300.0	5.0	350.0	10.0	333.3386	☑	☑	Updated : Converged
DP1	300.0	6.0	350.0	10.0	331.2571	☑	☑	Updated : Converged
DP2	300.0	7.0	350.0	10.0	329.4175	☑	☑	Updated : Converged
DP3	300.0	8.0	350.0	10.0	327.782	☑	☑	Updated : Converged
DP4	300.0	9.0	350.0	10.0	326.3181	☑	☑	Updated : Converged
DP5	300.0	10.0	350.0	10.0	324.9999	☑	☑	Updated : Converged

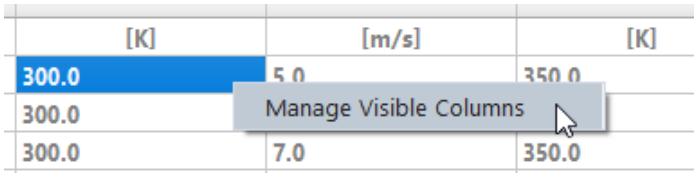
The table provides:

- Values for the design point input/output parameters. Table cells for input and output parameters expect numerical values, and automatically highlight any improper string entries.
- Options for whether or not to **Write Data** during the design point update. By default, this option is not enabled for newly added design points, and must be enabled manually for data to be written during the solution. Alternatively, you can set the **Write Data by Default** preference accordingly (see Setting Preferences for Parametric Studies (p. 4436)).
- Options for whether or not to **Capture Simulation Report Data** for use in the design point's simulation report.
- Status of the design point's run progress.
- Customization options such as hiding (and re-exposing) table columns (see Customizing the Design Point Table (p. 4394)).

46.6.1. Customizing the Design Point Table
46.6.2. Updating Design Points
46.6.3. Adding Design Points
46.6.4. Operating on Design Points
46.6.5. Importing and Exporting Design Point Tables
46.6.6. Saving Journals When Updating Design Points
46.6.7. Accounting for Mesh Morphing During Parametric Updates
46.6.8. Parametric Design Point Process Details and Case Change Considerations

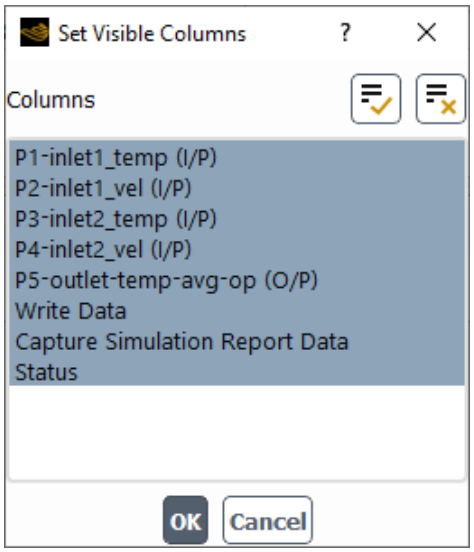
46.6.1. Customizing the Design Point Table

You can customize the design point table by showing and hiding specific columns. You can use either the **Select Visible Columns** icon ( ) in the design point table, or use the **Manage Visible Columns** option from the context menu on a particular cell.



This displays the **Set Visible Columns** dialog box:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The **Set Visible Columns** dialog box allows you select or deselect all or some of the columns in the design point table that you want to hide or expose in the table. All currently exposed column names are selected in the **Columns** list. Click a particular column name to deselect it from the list of visible columns. Use the icon to select all columns. Likewise, use the icon to deselect all columns. Once your selection(s) have been made, click **OK** to save your settings, dismiss the dialog box, and return to the design point table. The table now displays only the columns designated to be visible.

46.6.2. Updating Design Points

You have the ability to update the current design point in your study, or to update all design points in your study. In either case, Ansys Fluent will perform the calculations and update the results based on the parametric value(s).

Using the **Update Options** in the Ribbon, you can update your design point solutions either in the same session of Ansys Fluent (**Sequential**), or by invoking a new session (**Concurrent**).

Note:

Performing concurrent updates is not available when using the CFD-Pro license level. If a parametric project is opened using the CFD-Pro license level that was previously configured using both concurrent updates and/or remote updates, Fluent will display a corresponding warning and the study will proceed to open using sequential updates instead.

46.6.2.1. Updating Design Points in the Same Session Versus a New Session

46.6.2.2. Updating the Current Design Point

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

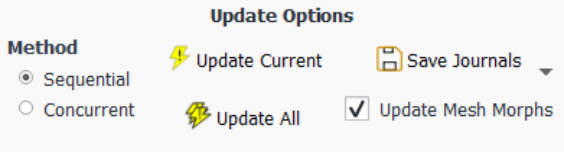
46.6.2.3. Updating All Design Points

Note:

When remotely updating a design point, there is an intentional delay (180 seconds) between the time when the design point calculations are complete and the time when the results are available and the status is updated.

46.6.2.1. Updating Design Points in the Same Session Versus a New Session

To better control how the design points are updated, use the **Sequential** option or the **Concurrent** option under **Update Options** in the **Parametric** tab of the Ribbon.

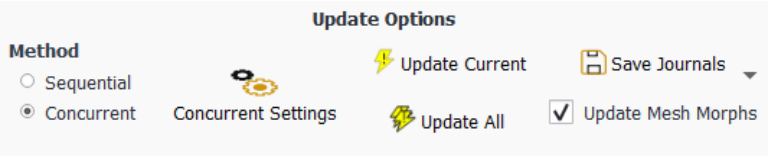


Running Sequential Sessions

By default, Ansys Fluent uses the **Sequential** option to update the design points in the currently open Fluent session. Design points that are marked as **Needs Update**, **Interrupted**. and **Queued in Fluent** in the Design Point Table are sequentially updated one after another. The current session remains busy while the design point update is in progress.

Interrupting a design point update while using the **Sequential** option is the same as interrupting a regular Fluent session using the **Stop** button. If you had multiple design points and had selected **Update All**, you will be prompted to determine if you want to interrupt only the current running design point, or if you wanted to stop all the design points that otherwise would have been updated as a part of **Update All**.

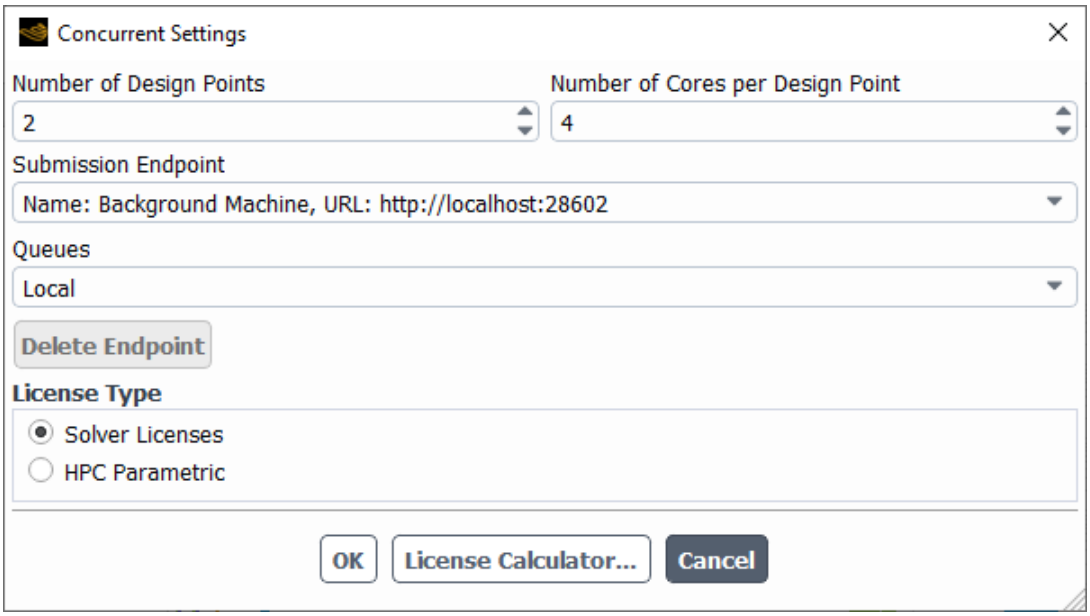
Running Concurrent Sessions



Use the **Concurrent** option to start a new Fluent session in batch mode where journaling is utilized to update the design points in that Fluent session. The new session uses the same mode and precision as the current session.

Adjust the settings for concurrent runs by clicking **Concurrent Settings** in the ribbon to display the **Concurrent Settings** dialog where you can change the number of design points and/or change the number of cores for each, as well as specify licensing options.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



A single session is created to update a single design point, so there is a one-to-one relationship between the design points and the sessions that are launched. Fluent uses parallel processing so that multiple design points can be updated simultaneously. You can provide the **Number of Design Points**, thus providing the number of parallel design points to update. In addition, you can provide the **Number of Cores per Design Point**, thus specifying the number of cores to allocate for each design point run. The current Fluent session remains available and you can continue using the current session for other work. Note that the new session that gets launched uses the same number of nodes as the base session (that is, the current Fluent session).

To run concurrent design point updates using distributed memory, use the **Submission Endpoint** drop-down menu to choose or register a job submission endpoint location URL.

Note:

Keep in mind that Fluent allows you to:

- Run concurrent sessions on the local machine (shared memory), and
- Perform concurrent updates on a distributed memory cluster. As a requirement, a Distributed Compute Gateway endpoint needs to be deployed and configured with suitable HPC Configurations. For more information, see Distributed Compute Gateway User's Guide.

To create a new job submission endpoint and its location:

- Select the **Register a New Endpoint** option in the **Submission Endpoint** drop-down list.
- Enter a **Name** for the new endpoint.
- Enter a valid **URL**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Click the **Register Endpoint** button to officially register the location and to add its name and URL to the **Submission Endpoint** drop-down list.

  Note:

  To remove a registered submission endpoint, select the endpoint in the **Submission Endpoint** drop-down list and click **Delete Endpoint**. Note, however, that you cannot delete the `localhost` endpoint.

The first time you select and access a registered endpoint URL, you will be prompted for a valid user name and password, and once connected, the available **Queues** are populated in the **Concurrent Settings** dialog box where you can pick the queue of your choice.

In addition, you can specify the **License Type** for your concurrent runs. You can choose from **Solver Licenses** such that the concurrent runs would employ standard Fluent licensing.

Alternatively, you can select **HPC Parametric** to use either standard HPC parametric licensing or optiSLang licensing, depending on the value of the optiSLang **License Type**. For instance:

- Set the license type to **None** to use standard HPC parametric licensing.
- Set the license type to **Premium** or **Enterprise** to use Ansys optiSLang licensing. You can then select the number of optiSLang licenses that you want to use.

  Note:

  Starting at the 2024 R1 release, the certificates required by HPC services for performing concurrent design point updates inside a Fluent parametric study are no longer packaged in the`\v241\RSM\bin\HPCServices\certificates` folder. Because of this packaging change, even performing concurrent design point updates on a local machine requires that you now must first generate the necessary certificates. See Obtaining and Installing an SSL Certificate for more information.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Click the **Licensing Calculator** to open the **Ansys Parametric Licensing Calculator** web site in your Guide, where we discuss the Concurrent Settings dialog:browser where you can determine your specific licensing requirements for your parametric run.

Note:

The parametric study checks out the specified number of optiSLang licenses even if fewer licenses could have been used to meet the parametric study licensing needs.

Note:

The **Concurrent** option applies to running concurrent Fluent sessions on either distributed memory machines or the same shared memory machines.

Note:

The base case is the case that is used for all the Design Points updates. When you perform any changes to the case (which is not already present in a base case) then you can create a new parametric study, reset a current study, or discard the changes you have introduced.

To start updating your concurrent design point session(s), click the **Update Current** or the **Update All** buttons in the Ribbon. The **Status** of your sessions will change to **Queued** in the design point table. When updating design points using distributed memory, the number of design points assigned to be **Queued** is based on the number of concurrent design points that you have set. The remaining design points are assigned to be **Queued in Fluent** and are submitted to the cluster for updating once all **Queued** design points are completed. Once queued, you can check the status of the runs by clicking the **Refresh Status** button in the Ribbon, and the status in the design point table will change accordingly.

While your concurrent session's design point(s) are being updated (that is, when the status is either **Queued**, **Queued in Fluent** or **Updating...**), you can use the following context menu options:

- **Stop Selected Design Point Update**: interrupts only the selected design point.
- **Stop All Updating in Concurrent Session**: interrupt all design points.
- **Show Status in Web Browser**: directly launches a web browser to view the run status of the selected design point in an HPC Web portal.

Note that you can set the **Enable Auto Refresh of the Status of the Design Point** preference accordingly (see Setting Preferences for Parametric Studies (p. 4436)). This preference must be enabled when updating design points concurrently using distributed memory. Fluent will monitor and retrieve the latest status of submitted design point status from the cluster and, based on that information, determines how and when to submit the next design points. In addition, when the concurrently submitted design point is in the **Updating...** status, the **Show** option is available in the context menu where you can access and display transcript, residuals and monitors for the selected design point.

In addition, note that you can submit the concurrent design points, close the current Fluent session, later launching a Fluent session, open that project and you will see the updated status of the sub-

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

mitted design points using the **Refresh Status** button in Ribbon (see Monitoring and Viewing Design Point Update Status (p. 4417) for additional information). After the concurrent design points are submitted, the current session is no longer connected with those design points and therefore you can close the current session, if desired. When submitting design points using distributed memory, the base Fluent session needs to remain open until all the design points are updated - unlike on shared memory machines where you are able to close the base Fluent session after clicking **Update All**.

Note:

Currently, by default, the concurrent sessions will consume standard Fluent solver licenses.

46.6.2.1.1. Running Locally Concurrent Design Point Updates with GPUs

While the GPU solver (see Using the Fluent Native GPU Solver (p. 3755)) supports updating parametric studies sequentially for variations in supported parameters, you can also update design points concurrently for locally defined sessions.

When the **Native GPU Solver** option is enabled in the Fluent Launcher for the base Fluent session, the **Concurrent Settings** dialog will show the **Use Native GPU Solver** check box, allowing you to make use of GPU resources for concurrent design point updates defined locally (that is, running on a single shared-memory machine equipped with one or more GPGPU accelerators where the parametric study was initialized).

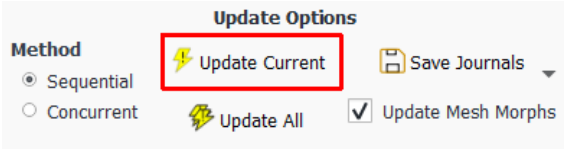
In the case of multiple GPU cards, instead of sharing the GPU cards across the design points, Fluent runs one design point on a single GPU. So, for local concurrent runs, Fluent will continue iterating over the available GPUs and spawn the design points one by one on each of the GPUs.

For example, if you selected 4 GPUs in the Fluent launcher and 4 concurrent design points, Fluent starts the first design point on the first GPU, the second design point on the second GPU, and so forth.

If you set the number of concurrent design points to be more than the number of the available GPU cards, then the design points will share some of the GPU cards.

46.6.2.2. Updating the Current Design Point

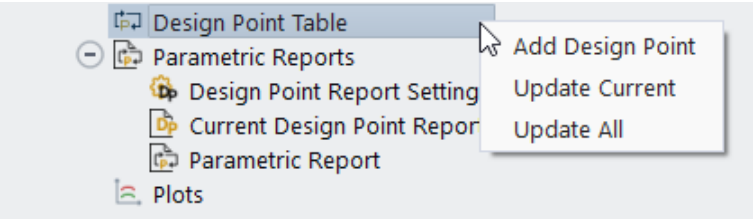
To update the current design point, use the **Update Current** option under **Update Options** in the **Parametric** tab of the Ribbon.



You can also use the context menu in the **Parametric Study View** to update the current design point. Right-click the **Parametric Studies/Design Point Table** tree item and select **Update Current** from the menu that opens.

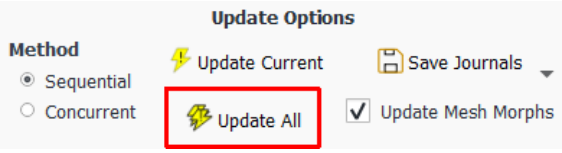
Parametric Study View → **Parametric Studies** → **Design Point Table** → **Update Current**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



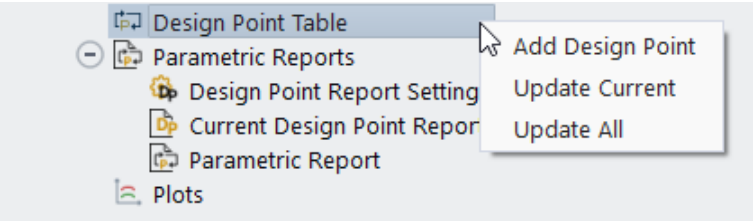
46.6.2.3. Updating All Design Points

To update all design points in your parametric study, use the **Update All** option under **Update Options** in the **Parametric** tab of the Ribbon.



You can also use the context menu in the **Parametric Study View** to update all the design points. Right-click the **Parametric Studies/Design Point Table** tree item and select **Update All** from the menu that opens.

Parametric Study View → **Parametric Studies** → **Design Point Table** → **Update All**



46.6.3. Adding Design Points

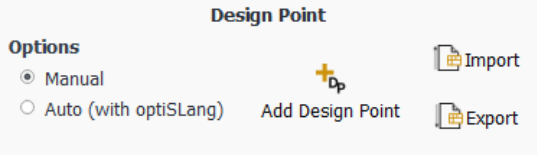
Design points can be manually added to your parametric study or automatically added to your study using Ansys optiSLang.

46.6.3.1. Manually Adding Design Points

46.6.3.2. Automatically Adding Design Points

46.6.3.1. Manually Adding Design Points

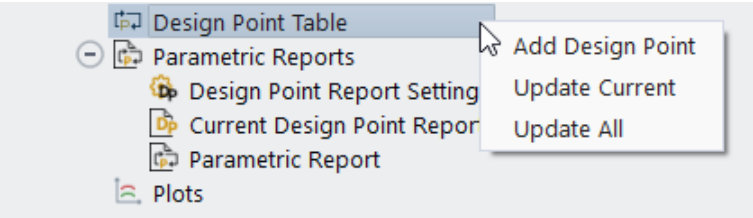
You can manually add design points to your parametric study by changing the values for various input parameters. To manually add a new design point to the **Design Point Table**, enable the **Manual** option and select the **Add Design Point** option under **Design Point** in the **Parametric** tab of the Ribbon.



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also use the context menu in the **Parametric Study View** to add a design point. Right-click the **Parametric Studies/Design Point Table** tree item and select **Add Design Point** from the menu that opens.

Parametric Study View → **Parametric Studies** → **Design Point Table** → **Add Design Point**



46.6.3.2. Automatically Adding Design Points

If Ansys optiSLang is installed, design points can be automatically added to your parametric study using optiSLang's Design of Experiments methods. To automatically add a new design point to the **Design Point Table**, enable the **Auto (with optiSLang)** option and select the **Add Design Points...** option under **Design Point** in the **Parametric** tab of the Ribbon.

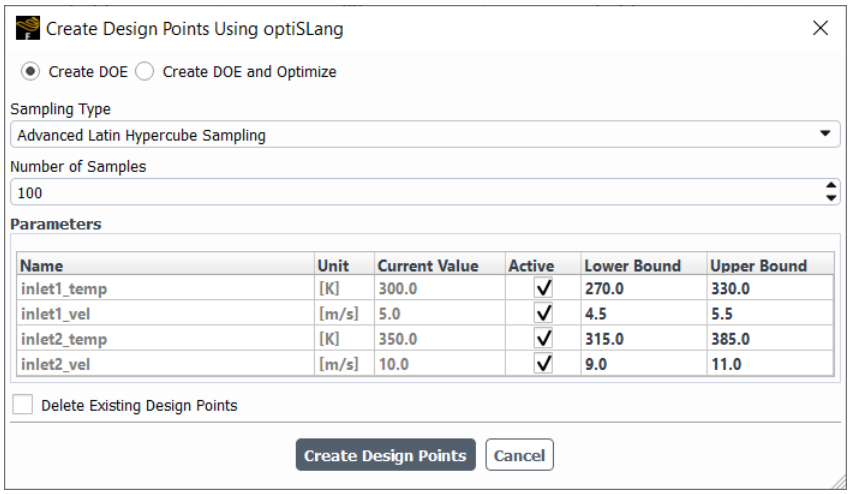
This displays the **Create Design Points Using optiSLang** dialog box where you can add design points to your study and set various optiSLang-specific properties to optimize your analysis. From here, you can choose to create your design of experiments with or without full optimization.

46.6.3.2.1. Creating Designs of Experiments for optiSLang

46.6.3.2.2. Creating and Optimizing Designs of Experiments for optiSLang

46.6.3.2.1. Creating Designs of Experiments for optiSLang

In the **Create Design Points Using optiSLang** dialog, you can choose to create your design of experiments using the **Create DOE** option:



From here, you have the following options:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sampling Type

Allows you to choose from a variety of optiSLang sampling algorithms, where available choices depend on your optiSLang licensing level. For instance, by default, the following types are available:

- **Koshal Linear**
- **Koshal quadratic**
- **Full Factorial**
- **Full combinatorial**
- **Central composite**
- **Star Points**

When using the optiSLang Pro license, however, the following types are also available:

- **D-optimal linear**
- **D-optimal quadratic**
- **D-optimal customizable**
- **Plain Monte Carlo**
- **Latin Hypercube Sampling**
- **Advanced Latin Hypercube Sampling**
- **Space filling Latin Hypercube Sampling**
- **Sobol Sequence**

Number of Samples/Number of Levels

Allows you to choose the number of samples for your assortment of design points. This field is dependant on the selected **Sampling Type** and may be read-only and restricted to certain values for some types.

Number of Designs

For some **Sampling Type**s, the number of designs cannot be known ahead of time. In such cases, Fluent displays the non-editable **Number of Designs** field and the **Calculate** button. Use the **Calculate** button to have optiSLang determine the **Number of Designs** based on the number of levels passed and the number of input and output parameters.

Table

Displays the currently available input variables. Table properties include the input variable **Name**, **Unit**, **Current Value**, whether it is **Active** or not, **Lower Bound** (minimum value), and

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Upper Bound (maximum value). Only the **Lower Bound** and the **Upper Bound** values are editable and, by default are equivalent to +/- 10% of the **Current Value**.

Note:

If you deselect the **Active** field for an input parameter, its **Lower Bound** and **Upper Bound** are set to its **Current Value** which will then be used as a default value for that parameter.

Delete Existing Design Points

Allows you to remove all existing design points once the **Create Design Points** button is selected and the dialog dismissed.

Once your choices and selections have been made, click **Create Design Points** to add your design points to the parametric analysis.

Note:

When processing design points, Fluent needs to know the location of the Ansys optiSLang installation. Depending on whether you are working on the Windows or Linux platforms, this location may be defined differently in different circumstances. For instance, in most cases, optiSLang can be detected if it is installed on the same machine or in the same directory as defined by the `AWP_ROOT` environment variable. In other cases, optiSLang can be detected if it is installed on some other location, another machine, or elsewhere in a network as defined by the `OPTISLANG_EXE` and the `OPTISLANG_HOME` environment variables.

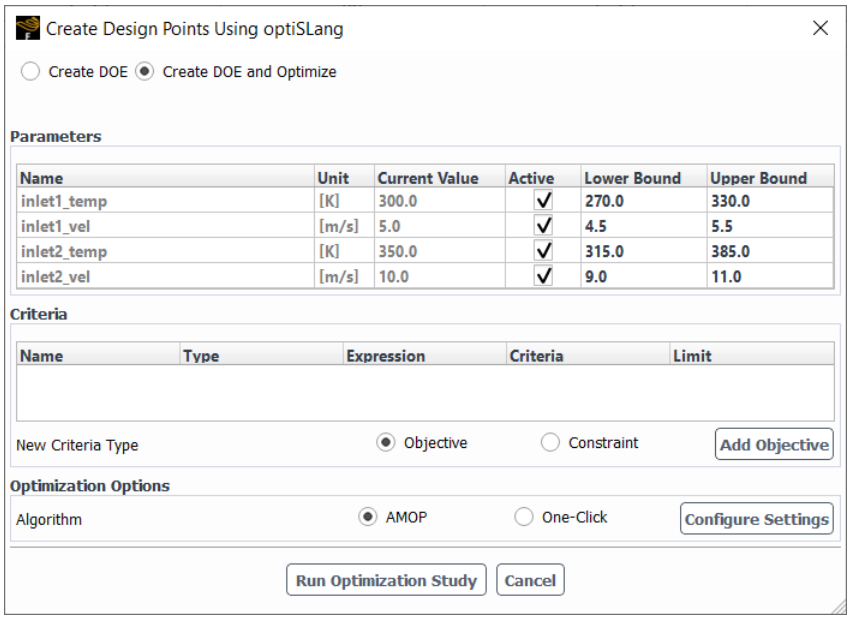
On both Windows and Linux platforms, Fluent searches for the location of optiSLang by first checking the `AWP_ROOT` environment variable, followed by `OPTISLANG_EXE` and then finally `OPTISLANG_HOME`.

Refer to Parametric Study Text Commands for more information regarding using the equivalent text command interface (TUI) commands under the `parametric-study/design-points/auto-create/` menu.

46.6.3.2.2. Creating and Optimizing Designs of Experiments for optiSLang

In the **Create Design Points Using optiSLang** dialog, you can choose to create and optimize your design of experiments using the **Create DOE and Optimize** option (when using an optiSLang Pro license):

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



From here, you have the following options:

Parameters Table

Displays the currently available input variables. Table properties include the input variable **Name**, **Unit**, **Current Value**, whether it is **Active** or not, **Lower Bound** (minimum value), and **Upper Bound** (maximum value). Only the **Lower Bound** and the **Upper Bound** values are editable and, by default are equivalent to +/- 10% of the **Current Value**.

Note:

If you deselect the **Active** field for an input parameter, its **Lower Bound** and **Upper Bound** are set to its **Current Value** which will then be used as a default value for that parameter.

Criteria

You can choose to create a table of objective-based and/or constraint-based criteria for your optimization study.

- For objective-based criteria, select **Objective** and click **Add Objective** to add a row to the **Criteria** table.
- For constraint-based criteria, select **Constraint** and click **Add Constraint** to add a row to the **Criteria** table.

Once added, you can adjust the **Name**, view its **Type**, assign an available**Expression**, specify a **Criteria** (as either **<=** or **>=** for constraint-based criteria or as either **MIN** or **MAX** for objective-based criteria) and set a value for the **Limit** accordingly.

To remove an entry in the table, right-click the row's **Type** cell and select **Delete** from the context menu.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

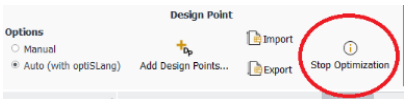
Optimization Options

You can choose from either the **AMOP** optimization algorithm and its default settings or the **One-Click** optimization option and its default settings. In each case, you can keep the default settings, or you can select **Configure Settings** to access additional optimization settings for each algorithm. See Configuring optiSLang AMOP Settings (p. 4407) or Configuring optiSLang One-Click Settings (p. 4409) for more details.

When your settings are complete, you can click the **Run Optimization Study** to perform the optimization operations according to your settings.

As the optimization study is being run, Fluent and optiSLang communicate with each other to create and update multiple design points. optiSLang sends new design of experiment information to Fluent while Fluent creates and updates the design points accordingly.

For sequential runs, you can interrupt the run by using the **Stop** button at the bottom right of the interface. For concurrent runs, you can select the **Stop Optimization** button in the Parametric ribbon.



Once complete, a table summarizing the results of the optimization is available in Fluent.

Figure 46.2: An Example of the Optimization Results Table (AMOP) (p. 4406) describes an example of a results table for an AMOP analysis that contain columns for objectives and constraints and design points that do not meet the constraints are highlighted in red.

Figure 46.2: An Example of the Optimization Results Table (AMOP)

Optimization Results : AMOP

Name	Feasible	Status	inlet2_temp	inlet2_vel	outlet_temp-op	mfr_out-op	obj	constr
DP1	True	Succeeded	202.5	26.25	49.488654	-0.28183	-0.28183	49.488654 <= 50
DP2	False	Succeeded	232.5	25.75	52.907536	-0.281549	-0.281549	52.907535999999999 <= 50
DP3	False	Succeeded	237.5	22.75	50.800162	-0.276878	-0.276878	50.800162 <= 50
DP4	False	Succeeded	247.5	24.75	53.90324	-0.279286	-0.279286	53.903239999999998 <= 50
DP5	True	Succeeded	207.5	23.25	47.779889	-0.278027	-0.278027	47.779889000000003 <= 50
DP6	False	Succeeded	242.5	26.75	55.115474	-0.28289	-0.28289	55.115474000000001 <= 50
DP7	False	Succeeded	227.5	23.75	50.472884	-0.278481	-0.278481	50.472884000000002 <= 50
DP8	True	Succeeded	217.5	24.25	49.742036	-0.278943	-0.278943	49.742035999999998 <= 50
DP9	False	Succeeded	222.5	27.25	52.906933	-0.282442	-0.282442	52.906932999999998 <= 50
DP10	True	Succeeded	212.5	25.25	49.991301	-0.280658	-0.280658	49.991301000000002 <= 50

Likewise, Figure 46.3: An Example of the Optimization Results Table (OCO) (p. 4406) describes another example of a results table for an OCO analysis where the best design is highlighted in green.

Figure 46.3: An Example of the Optimization Results Table (OCO)

Optimization Results : OCO

Name	Feasible	Status	inlet2_temp	inlet2_vel	outlet_temp-op	obj
DP11	True	Succeeded	212.5	25.25	49.991301	49.991301
DP12	True	Succeeded	225.0	27.0	53.059449	53.059449
DP13	True	Succeeded	245.0	24.0	52.821373	52.821373
DP14	True	Succeeded	215.0	23.0	48.458162	48.458162
DP15	True	Succeeded	235.0	26.0	53.449766	53.449766
DP16	True	Succeeded	205.0	25.0	48.843908	48.843908
DP17	True	Succeeded	215.0	25.363817	50.395142	50.395142
DP18	True	Succeeded	212.455346	23.0	48.16592	48.16592
DP19	True	Succeeded	200.0	22.5	46.367454	46.367454
DP20	True	Succeeded	212.410691	22.779127	47.959444	47.959444

Parametric designs can be analyzed further using the **Data Export** options (**Open OMDB in Postprocessor** or **Export Optimization Results into CSV**) that are available in the **optiSLang**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Capabilities category of the Parametric Ribbon. See Exporting Parametric Designs to optiSLang (p. 4420) for more information.

Upon re-opening a parametric project that already has an optimization study defined, when you subsequently revisit the **Create Design Points Using optiSLang** dialog and select the **Create DOE and Optimize** option, Fluent retrieves the optimization settings from optiSLang and you are provided with an option to view the optimization results table again.

Note:

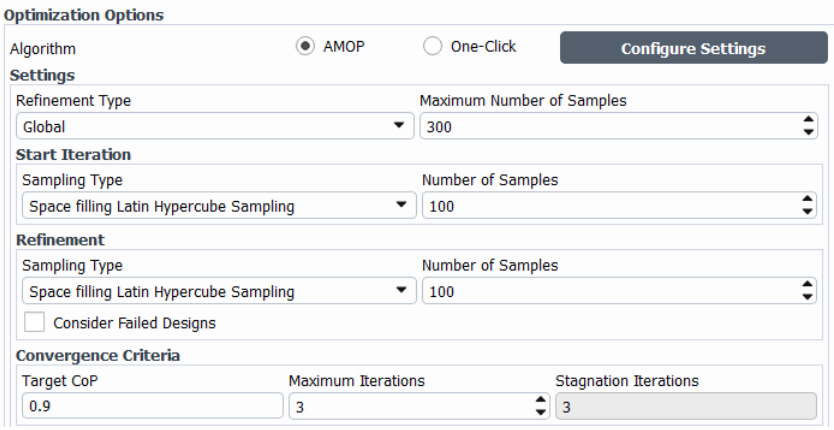
Using the **Create DOE and Optimize** option requires an optiSLang Pro license.

Note:

Since the optiSLang server does not retain the outputs from any already completed designs from a particular batch, if you choose to stop processing a batch of design point updates, the individual design point status will be designated as failed in Fluent, even for those design points that were previously successful.

Refer to Parametric Study Text Commands for more information regarding using the equivalent text command interface (TUI) commands under the `parametric-study/design-points/auto-create/optimize/` menu.

46.6.3.2.2.1. Configuring optiSLang AMOP Settings



The following options are available for AMOP configuration in optiSLang. For additional details, see Adaptive Metamodel of Optimal Prognosis (AMOP) Node in the optiSLang documentation.

- **Settings**: general algorithm settings.
  - **Refinement Type**: choose from either **Global** or **Local** refinement. **Global** refinement adds new data points using a global sampling. **Local** refinement adds new data points in regions where the best improvement of the criteria is expected.
  - **Maximum Number of Samples**: Sets the total number of samples.
- **Start Iteration**: settings relevant to the initial iteration.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - **Sampling Type**: Specifies the sampling method or deterministic design of experiment scheme for the initial iteration. All DOE schemes are available.

  - **Number of Samples**: Specifies the sample number.

- **Refinement**: settings relevant to refinement.

  - **Sampling Type**: (Global refinement type only) Specifies the sampling method for the refinement iterations. Only random sampling schemes are available.

  - **Number of Samples**: (Global refinement type only) Specifies the sample number for each iteration of the refinement sampling method.

  - **Importance of Sample Density**: (Local refinement type only) Only the sample density is considered in the generation of refinement points.

  - **Importance of Local CoP**: (Local refinement type only) The refinement points are created in regions with large local approximation errors.

  - **Importance of Optimization Criteria**: (Local refinement type only) The objective functions and constraints are considered for new refinement points. In case of single and multiple objectives, the region with expected maximum improvement of the objective functions are refined. If only constraints are defined, the refinement points are created in regions, where the constraints are expected to be fulfilled.

  - **Number of Samples per Iteration**: (Local refinement type only) Number of designs in the local refinement iterations.

  - **Consider Failed Designs**: Positions of previous failed designs are considered in the global and local refinement procedures. If unchecked, their positions are ignored in the density estimate.

- **Convergence Criteria**: settings relevant to convergence.

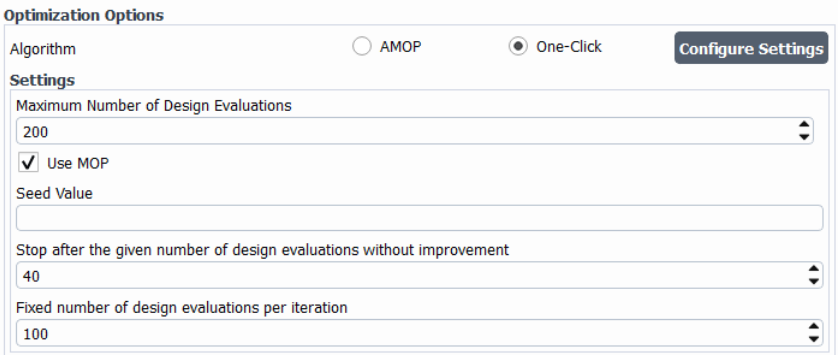
  - **Target CoP**: Sets the minimum Coefficient of Prognosis (CoP), required for all selected responses. If this value is reached, the AMOP algorithm has converged. In the global refinement, the global CoP values of all responses are considered. Tn the local approach the minimum local CoPs of all samples are used as error measure for the local CoP refinement.

  - **Maximum Iterations**: The algorithm stops at the maximum number of iterations, if the convergence criterion is not reached. This number considers the initial and the refinement iterations.

  - **Stagnation Iterations**: Considered in the local criteria-based refinement for single-objective optimization. The refinement stops if no improvement of the best design is obtained within the specified number of iterations.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

46.6.3.2.2.2. Configuring optiSLang One-Click Settings



The following options are available for One-Click (OCO) configuration in optiSLang. For additional details, see One-Click Optimization (OCO) in the optiSLang documentation.

- **Settings**: settings relevant to the OCO algorithm.
  - **Maximum Number of Design Evaluations**: Maximum number of design evaluations to compute before stopping.
  - **Use MOP**: Accelerate convergence and save the number of evaluations using the MOP. Deactivate when dealing with computational inexpensive evaluations.
  - **Seed Value**: Specify a seed value to be used by any algorithms containing stochastic components, instead of a randomly generated value.
  - **Stop after the given number of design evaluations without improvement**: Stops the optimization when no improvement of the best individual is seen after this number of design evaluations and the minimum number of design evaluations has been generated.
  - **Fixed number of design evaluations per iteration**: Defines the number of design evaluations generated per iteration. When set to no value, the number of design evaluations can vary from one iteration to another one. The maximum allowable value is 999.

46.6.4. Operating on Design Points

During the course of setting up and running your design point study, you can access various operations for any given design point(s) through the context menu:

- **Set As Current** lets you load the selected design point into the current session. The **Set As Current (No Data)** option is to inform you that this design point cannot be loaded as the current design point since there is no data yet available.
- **Update Selected Design Point** lets you update the currently selected design point(s).
- **Delete Design Point** lets you remove the selected design point(s) from the table.
- **Duplicate Design Point** lets you add a new design point by copying the values of the input parameters of the selected design point. This can only be performed on a single design point at a time.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Clear Data and Edit Values** lets you remove the generated data for the selected design point(s).
- **Show** lets you see the **Transcript**, **Monitor** plots, and **Residual** plots for the selected design point.
- **Change Status To:** lets you toggle the design point's status between **"Needs Update"** and **"Do Not Update"**.

46.6.5. Importing and Exporting Design Point Tables

Using the Design Point category of the Parametric Ribbon, use the **Import** and **Export** buttons to import and export comma-separated value (CSV) data for your parametric study.

- Use the **Import** option to import an existing CSV file directly into the parametric table.

Note:

When importing design point data, keep the following in mind:

– If design point data matches your existing design point table, you are prompted to either overwrite the existing matching design point data, or to append the data to the existing design point table.

– Any data related to the **Name** column are ignored. Design point names are managed by Ansys Fluent.

– Any data related to the **Units** column are ignored. Only design point values are imported, regardless of any specified units, however, a warning will be displayed when the units in the imported CSV file do not match with the units of the currently loaded case.

– Regardless of the **Status** of the design points in the imported CSV file, upon import, the status of all design points is set to **Needs Update**.

– The column headings in the imported CSV file must exactly match the names of the input parameters, otherwise they are not considered and the remaining columns (excluding **Write Data**, **Capture Simulation Report Data**) are ignored.

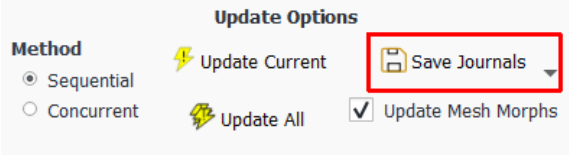
– If there are no corresponding columns for **Write Data** and **Capture Simulation Report Data** in the imported CSV file, then their values are initialized to the default values for the imported design points, where, by default, **Write Data** is not enabled and **Capture Report Data** is enabled. Alternatively, you can set the **Write Data by Default** preference accordingly (see Setting Preferences for Parametric Studies (p. 4436)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Use the **Export** option to export the contents of your design point table to an external `.csv` file. Exporting design point data will write all data from all columns to the specified file.

46.6.6. Saving Journals When Updating Design Points

Using the Design Point category of the Parametric Ribbon, use the **Save Journals** drop-down to customize how any journals are created for any design point that needs to be updated. You can then submit these journals to a job scheduler or run the journals in different Fluent session(s).



- Use the **Separate Journal...** option to create a separate journal file for each design point that needs to be updated.
- Use the **Single Journal...** option to create a single journal file for all design points that need to be updated.

You can also choose the same options from the context menu for selected design point(s) in the design point table (when their status is **Needs Update**).

Note:

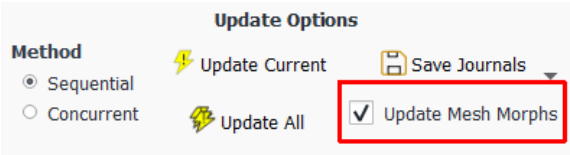
Since Fluent saves data into the project directory when updating the design point(s), ensure that the project directory is accessible from the appropriate machine when submitting journal files to a job scheduler or running the journals in different Fluent session(s).

Note:

Saving journal files is not available when using the CFD-Pro license level.

46.6.7. Accounting for Mesh Morphing During Parametric Updates

If your parametric analysis contains any mesh morphing capabilities, you can incorporate any mesh morphing operations into the parametric updates by selecting the **Update Mesh Morphs** check box in the **Update Options** category of the Parametric Ribbon.



When enabled, this option executes any configured morphing definitions that are specified in the **Parameterize and Explore** in the Design Ribbon. If there are any defined input parameters attached

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

to mesh morphing definitions, then Fluent executes the mesh morphing operation(s) after updating the parameter values and before solution initialization.

Note:

Mesh morphing should only be performed either by enabling the **Update Mesh Morphs** checkbox in the Parametric Ribbon, or by using the **Automatically Initialize and Modify Case** option in the **Calculation Activities** task page.

46.6.8. Parametric Design Point Process Details and Case Change Considerations

Parametric runs can be better understood by looking at what takes place at the beginning and the end of updating the initial design point and all subsequent design points.

Initially, as the first design point ("Base DP") starts updating, Ansys Fluent:

- Creates a copy of the current case, and assigns it to be the *"base case"*. In addition, the design point's simulation report template is retained, in case the design point must be reset later. Any changes, therefore, that are performed before the initial design point get updated (such as adding new contours, adding new parameters, making changes to the settings, and so on), and will be captured for all subsequent design points.
- Opens the transcript.
- Sets the current working directory to the initial design point.

Note:

You must initialize and define all post processing quantities that you wish to capture for the complete study. In addition, you must update the base case's design point report settings to include all desired design point quantities that are to be included for all the design points.

As the initial design point ("Base DP") completes its update, Ansys Fluent:

- Writes the case and data for the design point. If **Use Base Data** (in the context menu of the **Parametric Study View**) is enabled, then Fluent also assigns the design point's data to the base case.
- Captures the simulation report data for the design point (through the **Capture Simulation Report Data** column of the design point table).
- Closes the transcript.

As for any other design points that are part of your study, when they are first being updated, Ansys Fluent:

- Checks if you have made any changes to the currently loaded case after the solution is complete. If there are changes, Fluent prompts you to determine if you would like to create a new study,

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

discard the changes, or reset the study and its design points and use these changes. This is done to make sure that all design points are consistent.

- Loads the base case, applies the new parameters, initializes or reads the base case ("Base DP") data if requested, and continues the calculations.
- Depending on how you are initializing and updating your design point data, additional actions may be involved. See Initializing Design Point Data (p. 4413) for more information.

Likewise, as your study's other design points are completing their updates, Ansys Fluent:

- Captures the simulation report data and writes the data file, if requested (through the **Capture Simulation Report Data** and the **Write Data** columns of the design point table).
- Updates the design point status.
- Closes the transcript.

Note:

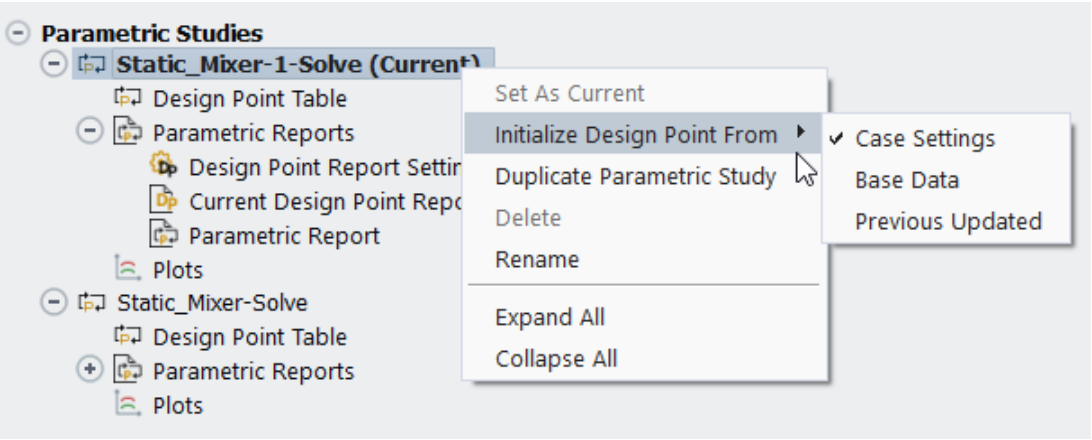
You should use relative path for all input and export files so that they will be written inside the current design point folder.

Note:

At times, a data file can be used for initialization (for example, when using the **Use Solution Data From File** option in the **Automatic Solution Initialization and Case Modification** dialog). In such cases, the parametric study resets the time step, iterations, and residuals, but the study itself uses the actual solution data.

46.6.8.1. Initializing Design Point Data

You have the choice of initializing your design point data using data from either the current case settings, the base design point's data, or from previously updated design points. You can access this choice by using the context menu in the **Parametric Study View** portion of the Outline View.



The options to initialize the design points are as follows:

- **Case Settings**: assigns the design point's data to the case settings.
- **Base Data**: assigns the design point's data to the base case.
- **Previous Updated**: assigns the design point's data to the data from the previously updated design point.

  When you enable **Previous Updated**, you are choosing to initialize your design point data using the previously updated design points, the behavior is dependant on how you update your design points (see Updating Design Points (p. 4395)).

  – **Sequential Update**

    When you update the design points sequentially, Fluent checks for the previous design point's data that is available in order to initialize the current design point. If there is no previous design point data, then the case settings are utilized instead. So, when sequentially updating the design points (for example DP1, DP2, DP3, etc.), the data of the first design point (DP1) is used to initialize the second design point (DP2). Likewise, the data of the second design point (DP2) is used to initialize the data for the third design point (DP3), and so forth.

  – **Concurrent Update (Local)**

    When you update the design points concurrently in a local environment, Fluent retrieves the most recently updated design point information to be used for all submitted design points.

  – **Concurrent Update (Remote)**

    When you update the design points concurrently in a distributed cluster environment, since design points are submitted in batches (based on the number of concurrent design points that you have set), Fluent retrieves the data such that every batch of design points will use the data of any of the design points from the previously updated batch. In other words, Fluent retrieves the data file directly on the endpoint, and uses it to initialize the case of the current design point. If the data file cannot be found, then the case settings are used for initialization instead. In addition, when this option is enabled, Fluent does not explicitly transfer any data file from the base session to the endpoint.

  All of these options are only available before or after updating the base design point.

  Note:

  When the **Previous Updated** option is enabled, Fluent will always write the data for the design points, regardless of whether the **Write Data** option is enabled or not in the design point table.

46.6.8.2. Reusing Your Case File

In some cases, you can re-use the case file prior to updating your design points. Specifically, for sequential runs of studies that contain only operating condition parameter changes and when there

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

are no mesh changes (such as dynamic mesh, mesh morphing, and/or mesh adaption), use the **Read Case Before Each Design Point Update** context menu option in the Parametric Study View to control how you manage design point updates using the case file.

- Setting this option to **Yes** (the default), Fluent reads the case and does not use the already loaded case file.
- Setting this option to **No** allows Fluent to not read any additional case file and to re-use the already loaded case and simply re-initialize the study or continue your work as needed, depending on the design point initialization method (see Initializing Design Point Data (p. 4413)). The following scenarios are supported, and the behavior can vary based on the design point initialization method. The following list describes the scenarios and the behavior, such that:
  - If you initialize the workflow from the case file (**Case Settings**), after updating parameters, Fluent initializes and resets.

Read Case Before Each Design Point Update = Yes	**Read Case Before Each Design Point Update = No**
Set the design point to be updated as the current design point.	Set the design point to be updated as the current design point.
Read the base case.	Apply the input parameter values.
Apply the input parameter values.	Initialize a case (even if data is loaded in the session).
Initialize a case.	Solve.
Solve.	

  - If you initialize the workflow from the base design point (**Base Data**), Fluent will read the base design point and continue.

Read Case Before Each Design Point Update = Yes	**Read Case Before Each Design Point Update = No**
Set the design point to be updated as the current design point.	Set the design point to be updated as the current design point.
Read the base case.	Read the base design point data.
Read the base design point data.	Apply the input parameter values.
Apply the input parameter values.	Solve.
Solve.	

  - If you initialize the workflow from a previously updated design point (**Previous Updated**), Fluent will continue using the data from the previously updated design point.

Read Case Before Each Design Point Update = Yes	**Read Case Before Each Design Point Update = No**
Set the design point to be updated as the current design point.	Set the design point to be updated as the current design point.

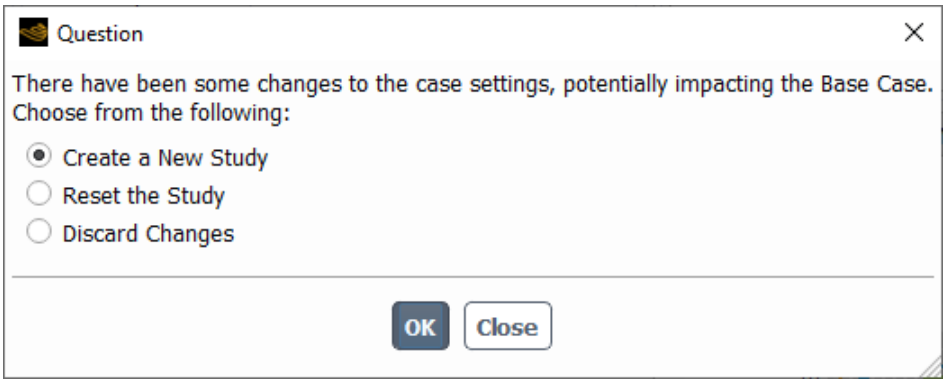
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Read Case Before Each Design Point Update = Yes	Read Case Before Each Design Point Update = No
Read the previous case. For concurrent updates only since, for sequential updates, the previous case is already loaded in the session.	Apply the input parameter values.
Read the previous data. For concurrent updates only since, for sequential updates, the previous data is already loaded in the session.	Solve.
Apply the input parameter values.	
Solve.	

46.6.8.3. Managing Case File Changes

For an already up-to-date base design point, Fluent provides some flexibility for managing your work when making any changes to your study's additional case file(s).

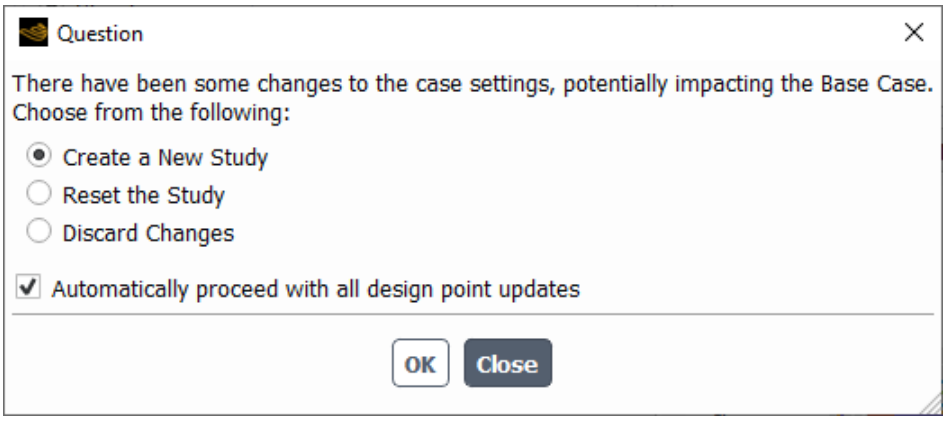
When adding or deleting any input or output parameters, or when you change some case settings and try to make another design point as current, Fluent prompts you to determine if you would like to create a new study, discard the changes, or reset the study and its design points and use these changes.



46.6.8.4. Managing Case Changes During Design Point Updates

When updating your design points, if you have made any explicit changes to an otherwise up-to-date base case (other than changing input variable settings using the design point table), Fluent prompts you to determine if you would like to create a new study, discard the changes, or reset the study and its design points and use these changes.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



If you choose to **Create a New Study** or to **Reset the Study**, you also have the option to **Automatically proceed with design point updates**, using the corresponding check box (since you were already in the process of updating the design points).

Note:

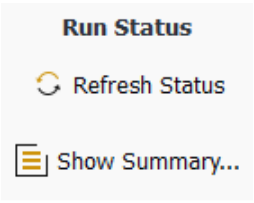
This dialog box also appears in two scenarios after the base design point is updated:

- when adding or deleting any input or output parameters, and
- when you have changed one or more case settings and proceed to make any other design point the current design point.

In either of the scenarios, the dialog does not include the option to **Automatically proceed with design point updates**.

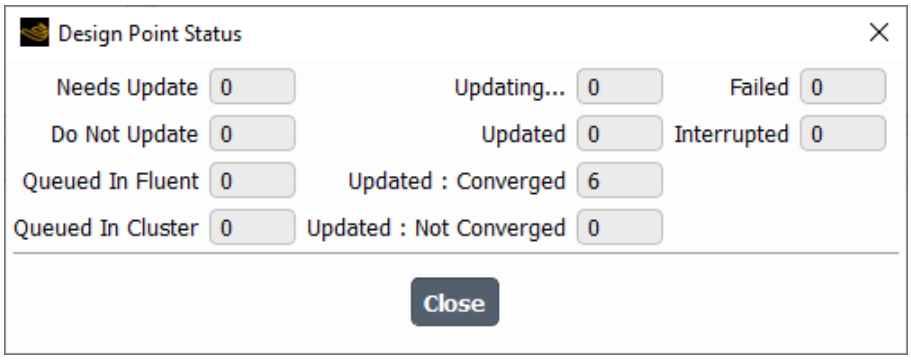
46.7. Monitoring and Viewing Design Point Update Status

As your design points are updated, you can monitor and view their status in the Design Point Table (under **Status**), as well as in the **Update Status** category of the Parametric Ribbon.



Use the **Show Summary...** button to display the **Count of Design Points Based on Status** dialog box that provides a summary of the various state(s) of your design point(s).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



From here, you can monitor how many simulation runs:

- **Needs Update**: indicates how many design point simulation runs still need to be updated.
- **Do Not Update**: indicates how many design point simulations have been designed not to be updated.
- **Queued in Fluent**: indicates how many design point simulations are waiting to be executed in Fluent.
- **Queued in Cluster**: indicates how many design point simulations are waiting to be executed in the cluster.
- **Updating...**: indicates how many design point simulations are currently being updated.
- **Updated**: indicates how many design point simulations have been updated.
- **Updated: Converged**: indicates how many design point simulations have been updated and have a converged solution.
- **Updated: Not Converged**: indicates how many design point simulations have been updated and do not have a converged solution.
- **Failed**: indicates how many design point simulations have failed to be updated.
- **Interrupted**: indicates how many design point simulations have been interrupted.

Note:

Whenever any design point is in the **Updating...** or **Queued in Fluent** or **Queued in Cluster** state:

- Under **Study** in the Ribbon, the **Duplicate** button is disabled.
- Under **Update Options** in the Ribbon, the **Sequential** and **Concurrent** radio buttons are disabled.
- In **Parametric Study Tree View**, the **Set As Current**, and the **Duplicate** context menu options are disabled.
- Corresponding text commands for these functions are also disabled.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

As the design point simulation runs progress, click the **Refresh Status** button to obtain the latest status of the design points that are being updated in the session, if any. Preferences are available for automatically refreshing your design points, see Setting Preferences for Parametric Studies (p. 4436) for details.

Alternatively, you can right-click the design point name in the **Design Point Table**, and select the **Show:** option. Here, you can show the run's transcript details, its monitors, residuals, or all of them in a series of tabs.

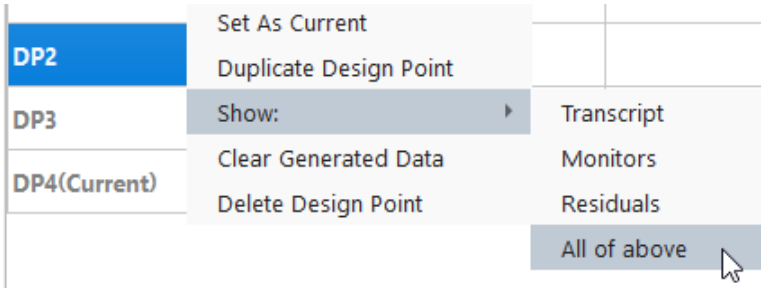


Figure 46.4: Transcript Tab

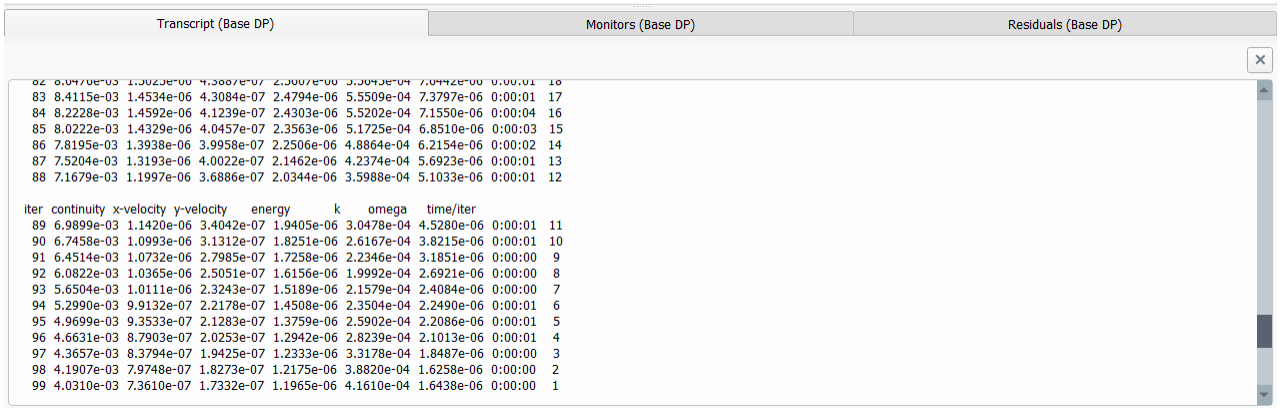
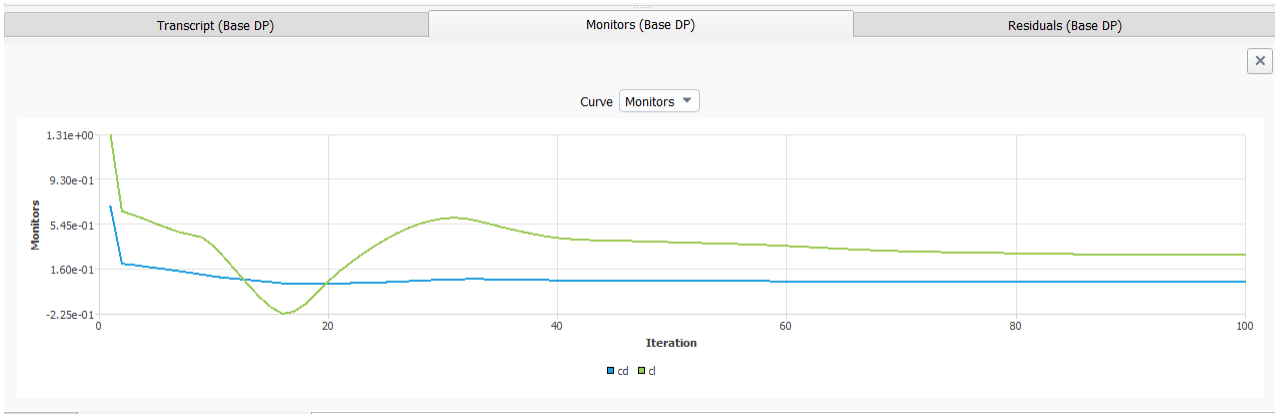
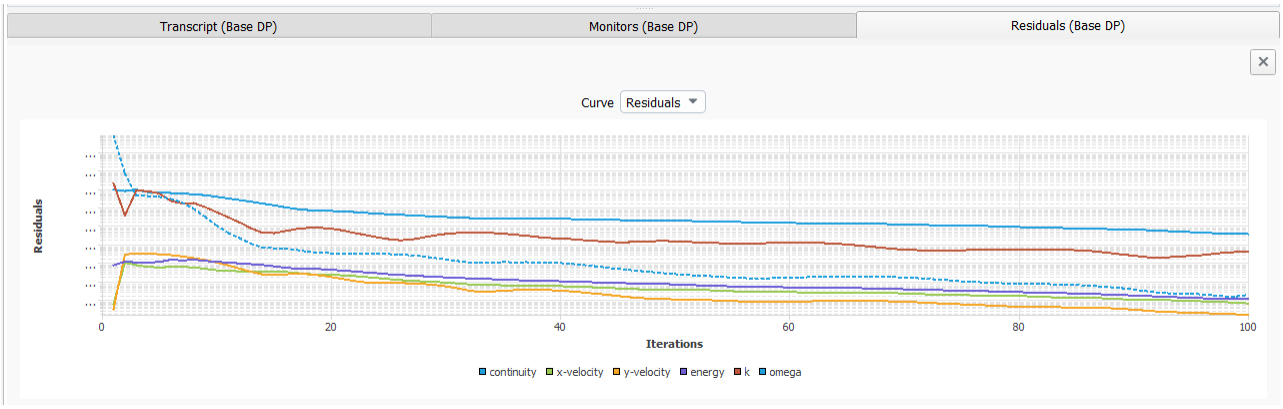


Figure 46.5: Monitors Tab



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 46.6: Residuals Tab



Note:

To ensure residual plots always represent the true progress of the solution through the design points, it is recommended that you set the **Iterations to Plot** and **Iterations to Store** settings in the **Residual Monitors** dialog box to a value that represents the maximum number of iterations that are possible given the current solver settings (maximum number of iterations per timestep multiplied by the number of timesteps).

Graphical plots such as monitors and residuals can be altered according to your needs. See Customizing Graphical Plots (p. 4423) for more information.

46.7.1. Exporting Parametric Designs to optiSLang

46.7.2. Comparing Parametric Plots

46.7.3. Customizing Graphical Plots

46.7.1. Exporting Parametric Designs to optiSLang

When you have more than one updated design point, you can export your Fluent parametric design point data and use it in optiSLang by using the **Data Export** drop-down menu in the **optiSLang Capabilities** category of the Parametric Ribbon.

- Use the **Export Designs** option to generate an optiSLang-compatible output file to a location of your choice, containing the details of the updated design points.
- When automatically adding design points using optiSLang (Automatically Adding Design Points (p. 4402)), you can open optiSLang directly using the **Go to optiSLang** option.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Go to optiSLang** button is available when designs of experiments (DOEs) are created using optiSLang and the optiSLang project exists in the simulation directory. Clicking on this button creates a copy of the optiSLang project from the simulation directory (`*.opf`), places it inside your working directory (as designated in the Fluent Launcher, and opens the copied project in optiSLang.

Note:

Since optiSLang and Fluent are not connected in any way, any changes made in the open optiSLang project are not available in Fluent.

- Once you have updated your design points, use the **Open Designs in Postprocessor** option to open the optiSLang postprocessor using a generated optiSLang-compatible output file (located, by default, under the current simulation directory) containing the details of the updated design points.
- For optimization studies (Creating and Optimizing Designs of Experiments for optiSLang (p. 4404)), once you have updated your design points, use the **Open OMDB in Postprocessor** option to load the optiSLang Monitoring Database into the optiSLang postprocessor. The database contains the details of the optimizations such as all of the designs with their input parameter values, their responses, and so on. Opening the database in the optiSLang postprocessor allows you access to various different types of charts.
- For optimization studies (Creating and Optimizing Designs of Experiments for optiSLang (p. 4404)), once you have updated your design points, use the **Export Optimization Results into CSV** option to convert your optimization results into a comma separated value (CSV) file that can be examined as a spreadsheet.

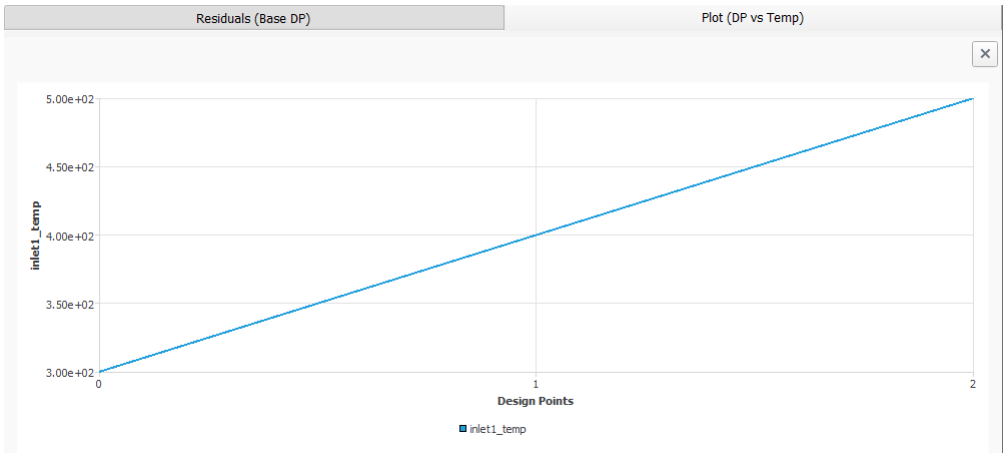
The equivalent text command interface (TUI) commands (`export-designs`, `go-to-optislang`, and `open-designs-in-optislang-postprocessor`) exist under the `parametric-study/design-points/auto-create/` menu.

46.7.2. Comparing Parametric Plots

Use the **Parametric Plots** dialog to create and compare plots between a parametric study's design points and their input and output parameters. Double-click the **Plots** node in the Parametric Study View to open the **Parametric Plots** dialog.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

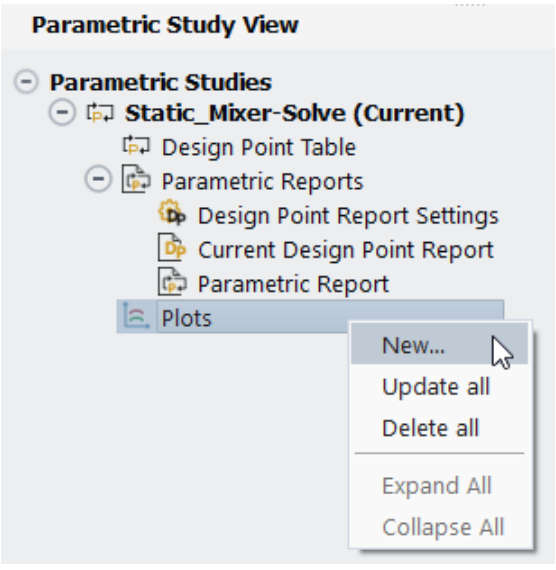
Figure 46.7: Example of a Comparative Plot: Inlet Temperature vs Design Point



You can also create new comparison plots for your design point runs using the **Create New** option under **Comparison Plot** in the **Parametric** tab of the Ribbon.

Alternatively, you can select the **New...** option from the **Plots** context menu to open the task page.

Parametric Study View → **Parametric Studies** → **Plots** → **New...**



With the **Parametric Plots** dialog, you can create and display plots that contain your design point input parameters and output parameters and easily visualize them in the Parametric Study view.

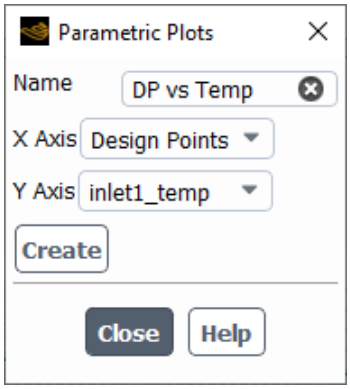
1. Specify a **Name** for the parametric plot.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Use the **X Axis** and the **Y Axis** drop down lists to specify the plot's x-axis and y-axis variables. These lists contains your study's input and output parameters.

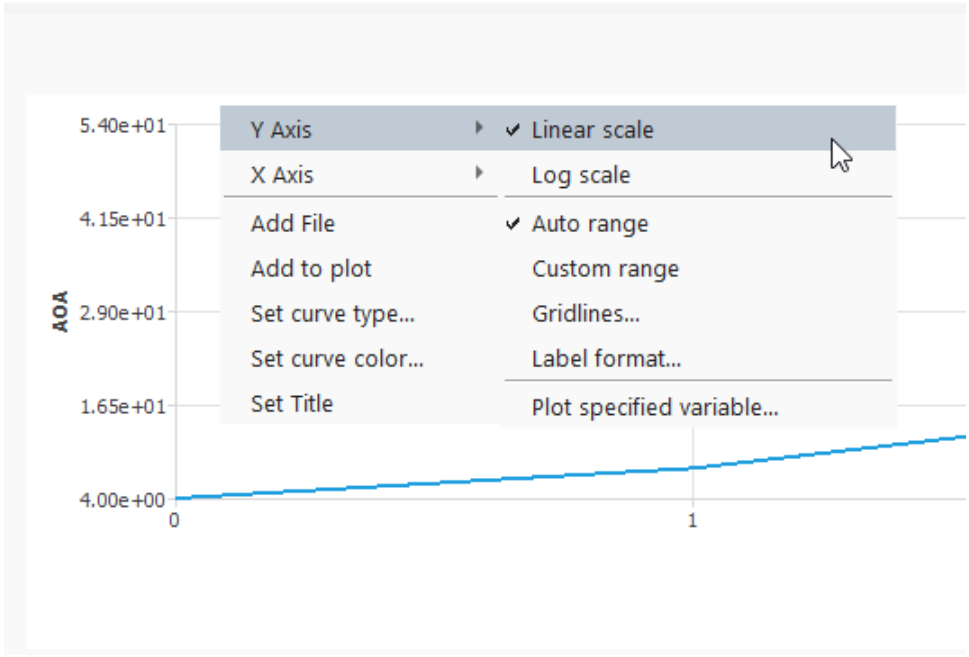
3. Click **Create** to create the plot, add it to the plots tree, and display it in the assigned window.

Figure 46.8: Parametric Plots Dialog



46.7.3. Customizing Graphical Plots

Your design point monitor and residual plots can be customized by using the plot's context menu to expose various customization menus.



With this context menu, you can customize the following plot settings:

- Set X and Y axes
  - Change the scale to linear or logarithmic
  - Change the range of the plot
  - Change the number of grid lines

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - Change the label formatting
  - Add a specific variable to the plot
- Add a file to the data set
- Add a curve to the plot
- Change the type of curve
- Change the color of the curve
- Change the title of the plot

46.8. Creating Simulation Reports for Design Points and Parametric Studies

Just as you can create and customize simulation reports in Ansys Fluent (Using Simulation Reports (p. 4359)), you can also generate separate design point reports, as well as reports for the parametric study itself.

- To define specific settings that will be applied to your individual design point-specific report(s), in the **Simulation Report Outline** task page, select **Design Point Settings** for the **Current Report**. See Creating Design Point Reports for Your Simulation (p. 4424) for more information.
- To define specific settings that will be applied to a parametric report, in the **Simulation Report Outline** task page, select **Parametric Report** for the **Current Report**. See Creating Parametric Reports for Your Simulation (p. 4428) for more information.

Parametric and design point reports can also be accessed by using the **Simulation Report** category of the Parametric Ribbon, or by the **Parametric Reports** node of the **Parametric Study View** in the Outline View.

46.8.1. Creating Design Point Reports for Your Simulation

You can create customized simulation reports for your design points in your parametric study. For instance, you can create one type of simulation report for your initial ("base") design point, and other, completely different, custom report settings for each individual design point.

Note:

These settings are the same as a standard Ansys Fluent Simulation Report, and are described in Using Simulation Reports (p. 4359).

46.8.1.1. Accessing Design Point Report Settings

46.8.1.2. Overview of Design Point Report Settings

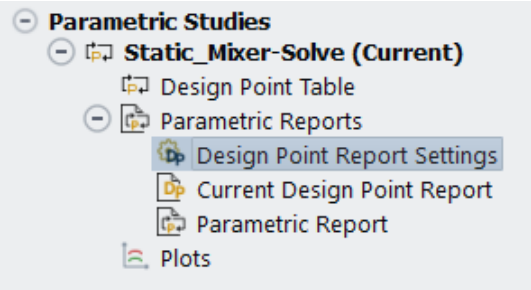
46.8.1.3. Generating, Viewing, and Saving Your Design Point Reports

46.8.1.1. Accessing Design Point Report Settings

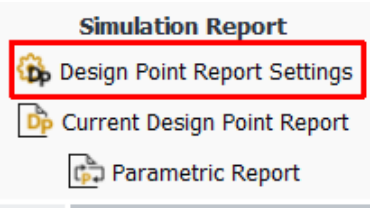
To gain access to the settings that can be applied to your design point report:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Double-click the **Design Point Report Settings** node in the **Parametric Study View** for your particular study.

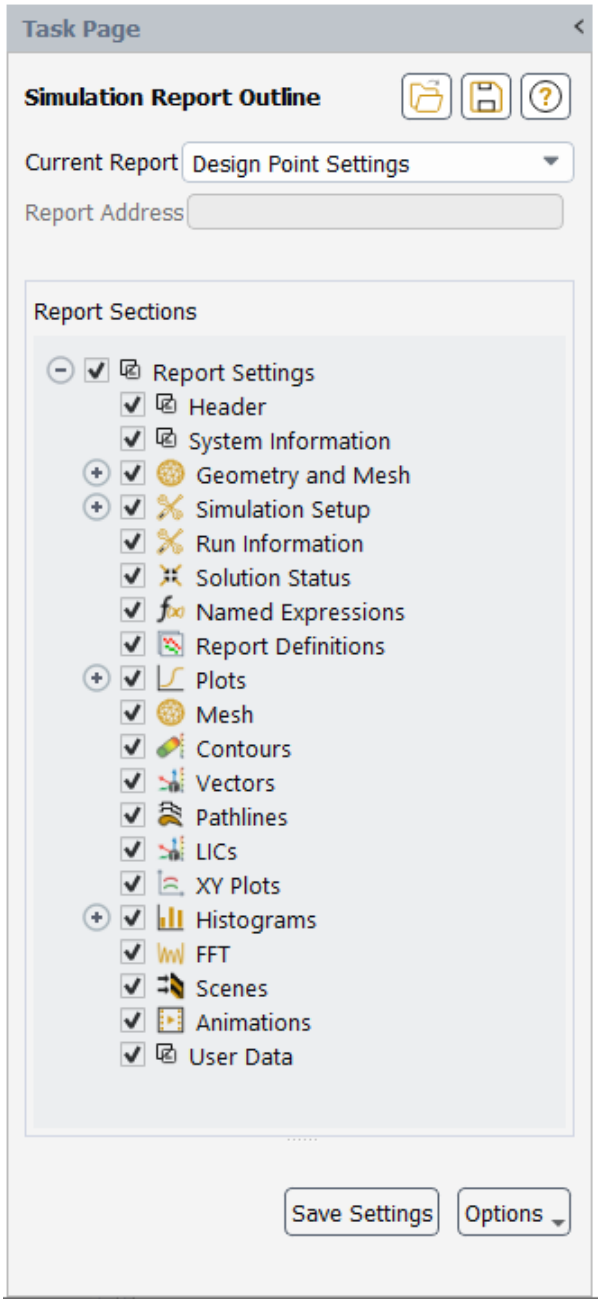


- Alternatively, click **Base Design Point Report Settings** in the **Simulation Report** category of the Parametric Ribbon



Either of these actions displays the **Simulation Report Outline** task page (with the **Current Report** set to **Design Point Settings**), where general settings can be made for the design point reports .

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



These setting categories are described in detail in Overview of Design Point Report Settings (p. 4427)

When you are satisfied with your settings, click **Save Settings** to have these settings available for all of your design point reports. Use the **Options** button to restore default settings.

Note:

For any report settings that you want to apply to a design point report, make sure that you save your design point report settings *before* updating the relevant design points.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

46.8.1.2. Overview of Design Point Report Settings

You can control the *visibility* of design point report categories using the check boxes to include/exclude all or parts of the report. You can also control the *arrangement* of design point report categories by reorganizing elements of your report prior to generating it. To do so, drag and drop elements of the report section tree onto other portions of the tree to rearrange the order and contents of various sections of your simulation report. For more information, see Organizing Your Simulation Report (p. 4364).

Design point reports contain the following sections (with their own **Descriptions** field that you can provide):

- Under **Report Settings**, there are several aspects of your design point report that you can control from a global perspective.

  Layout Settings - choose whether the layout of your results plots (such as contours, vectors, etc.) will be included in your report as a series of tabs, or in tabular columns. These settings can be applied to all the graphics results images in your report. These settings can be applied to the report as a whole, or to individual graphical object categories. See Changing the Layout of Your Results (p. 4384) for details.

  Image Settings - choose the default view orientation and resolution for your included graphics results (such as saved views/states, or common orientations such as top, bottom, left, right, and so on). See Customizing Your Report Image Settings (p. 4384) for details.

  – **Header** information includes fields for the **Analyst** name, and the **Date** (by default). Additionally, you can specify your **Company** name and your **Logo** image file. Further more, there are additional options for **URL**, **Phone** and **Fax** numbers, **Address** information, and a **Description** for the simulation.

  – **System Information**

  – **Geometry and Mesh**, including sections for **Mesh Size**, **Mesh Quality**, and **Orthogonal Quality**.

  – **Simulation Setup** (where you can opt to **Show Only Modified Settings**), including sections for **Physics** (that includes **Models**, **Material Properties**, **Cell Zone Conditions**, **Boundary Conditions**, and **Reference Values**) and **Solver Settings**.

  – **Run Information** includes an option to **Include License Usage**.

  – **Solution Status**

  – **Named Expressions** provides a means to customize the named expressions within your report. You can choose to include following details of your expressions: **Definition**, **Value**, **Unit**, **Used In**, and **Description**. All options are selected by default. Deselect those options that you do not want to include in your report.

  – **Report Definitions**

  – **System Information**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

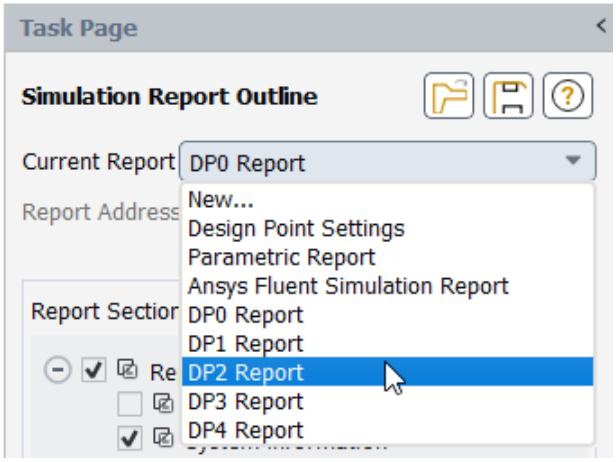
– Any available graphics objects that you want to include in your report, such as **Plots**, **Mesh**, **Contours**, **Vectors**, **Pathlines**, **LICs**, **XY Plots**, **Histograms**, **FFT**, **Scenes**, **Animations**, or your own custom view(s).

  For these objects, you can also provide **Layout Settings**, if desired, where you can choose whether the layout of your results plots (such as contours, vectors, etc.) will be included in your report as a series of tabs, or in tabular columns. These settings can be applied to all the graphics results images in your report. These settings can be applied to the report as a whole, or to individual graphical object categories. See Changing the Layout of Your Results (p. 4384) for details.

  Additionally, you can display the object using a particular orientation using the **View** field (front, back, left, bottom, isometric, etc.). Use the **Preview View** button to see what the image will look like once it is in the report. The isometric view is the default orientation.

46.8.1.3. Generating, Viewing, and Saving Your Design Point Reports

When saved, general design point report settings (described in Overview of Design Point Report Settings (p. 4427)) are applied to all design points in the parametric study. When you update the design points in your parametric study, Fluent generates the individual reports for each design point. You can view and edit selected design point reports (and their settings) if you require additional customization. For a given parametric study's list of design points, you can access individual settings for each design point report in the **Current Report** drop-down list.



When you have configured your design point report settings to your satisfaction, generate a design point report using the **Generate** button at the bottom of the task page. See Generating Simulation Reports (p. 4366) for more information.

Once the design point report is generated, it appears in its own tab alongside the graphics window. Alternatively, you can use the **View** button at the bottom of the task page. See Viewing Simulation Reports (p. 4367) for more information.

Once generated, you can save your reports as PDF or standalone HTML using the **Export** button at the bottom of the task page. See Saving Simulation Reports (p. 4383) for more information.

46.8.2. Creating Parametric Reports for Your Simulation

You can create customized simulation reports for your parametric study.

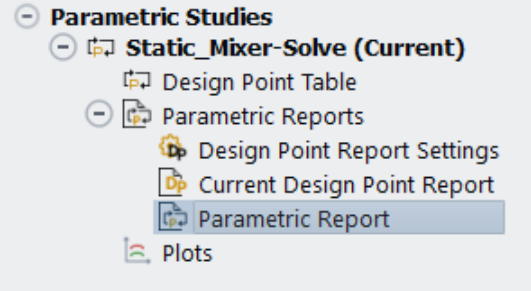
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

46.8.2.1. Accessing Parametric Report Settings
46.8.2.2. Overview of Parametric Report Settings
46.8.2.3. Comparing Parametric Results
46.8.2.4. Generating, Viewing, and Saving Your Parametric Reports

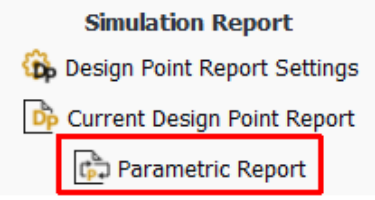
46.8.2.1. Accessing Parametric Report Settings

To gain access to the settings that can be applied to your parametric report:

- Double-click the **Parametric Report** node in the **Parametric Study View** for your particular study.

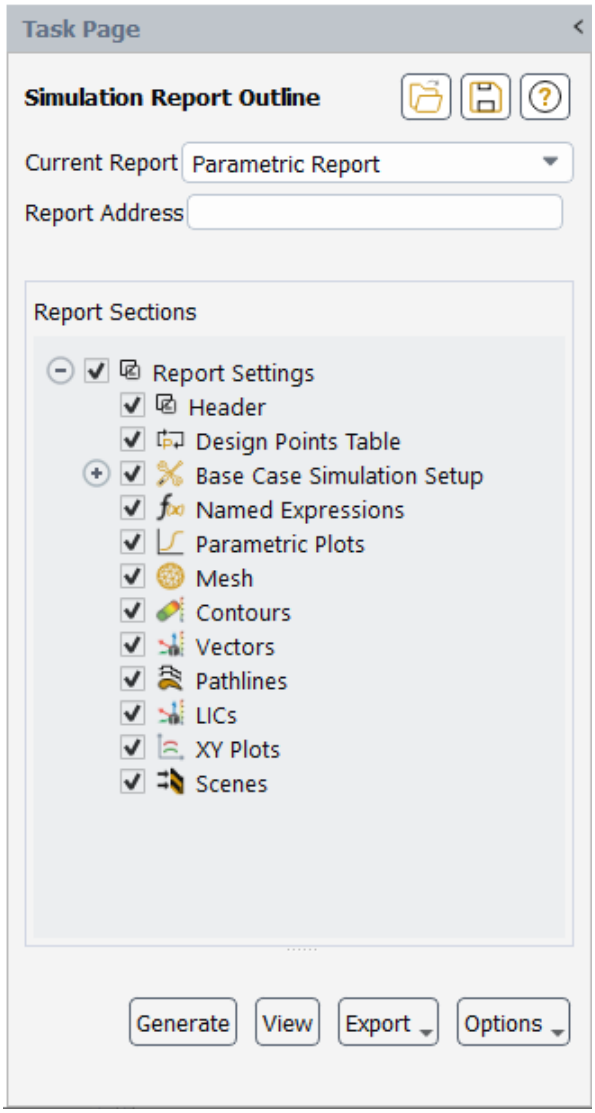


- Alternatively, click **Parametric Report** in the **Simulation Report** category of the Parametric Ribbon



Either of these actions displays the **Simulation Report Outline** task page (with the **Current Report** set to **Parametric Report**), where general settings can be made for the parametric report .

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



These setting categories are described in detail in Overview of Parametric Report Settings (p. 4430)

46.8.2.2. Overview of Parametric Report Settings

You can control the *visibility* of report categories using the check boxes to include/exclude all or parts of the report. You can also control the *arrangement* of report categories by reorganizing elements of your report prior to generating it. To do so, drag and drop elements of the report section tree onto other portions of the tree to rearrange the order and contents of various sections of your simulation report. For more information, see Organizing Your Simulation Report (p. 4364).

Parametric reports contain the following sections (with their own **Descriptions** field that you can provide).

- Under **Report Settings**, there are several other aspects of your parametric report that you can control from a global perspective.

  – **Layout Settings** - choose whether the layout of your results plots (such as contours, vectors, etc.) will be included in your report as a series of tabs, or in tabular columns. These settings can be applied to all the graphics results images in your report. These settings can be applied

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

to the report as a whole, or to individual graphical object categories. See Changing the Layout of Your Results (p. 4384) for details.

– **Image Slider View Settings** - choose whether your results plots (such as contours, vectors, etc.) will include a **standard** slider, a **comparison** slider, or **both**. These settings can be applied to the report as a whole, or to individual graphical object categories. See Comparing Parametric Results (p. 4433) for details.

– **Image Settings** - choose the default view orientation and resolution for your included graphics results (such as saved views/states, or common orientations such as top, bottom, left, right, and so on). See Customizing Your Report Image Settings (p. 4384) for details.

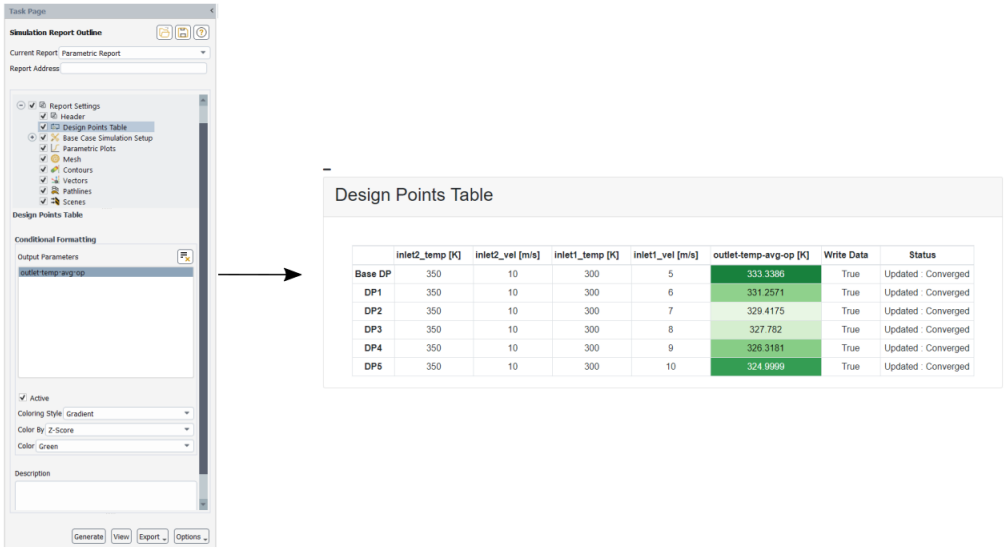
– **Theme Settings** - choose the **Use Dark Theme** option to present your simulation report (and included plots) with a dark background and white text, with any graphics result images remaining as they were originally captured. The **Use Dark Theme** option will be enabled by default if you have chosen to run Ansys Fluent using the **Dark** theme preference (see Customizing the Graphical User Interface (p. 935) and Setting User Preferences/Options (p. 937)).

Other categories include:

– **Header** information includes fields for the **Analyst** name, and the **Date** (by default). Additionally, you can specify your **Company** name and your **Logo** image file. Further more, there are additional options for **URL**, **Phone** and **Fax** numbers, **Address** information, and a **Description** for the simulation.

– **Design Point Table**. Here you can choose **Conditional Formatting** options for your output parameters, customizing how they appear in the report's design point table.

Figure 46.9: An Example of Conditional Formatting for the Design Point Table



Design Point Table Settings

Parametric Report

The following options are available:

→ Use the **Active** check box to include or exclude the conditional coloring and styling settings in the report. When enabled, the conditional settings are applied to the design point table column, otherwise, when disabled, no conditional settings are applied to the table column.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

→ Choose a **Coloring Style** (**Solid** or **Gradient**).

- When the **Coloring Style** is set to **Solid**, you can then choose to **Color By** either **Z-Score** or **Value Range**.

  When you specify **Value Range**, you can choose to apply colors to the table based on values that are **Above a Minimum Value**, values that are **Below a Maximum Value**, or to select **Between a Minimum and a Maximum Value** where you can specify a **Minimum Value** and **Maximum Value**.

- When the **Coloring Style** is set to **Gradient**, you can choose to **Color By** either **Z-Score** or **Magnitude**

→ Choose the **Color** as either **Red**, **Green**, or **Blue**.

– **Base Case Simulation Setup**, including the **Solver Settings** and **Physics** sections (which itself contains sections for **Models**, **Material Properties**, **Cell Zone Conditions**, and **Boundary Conditions**) and **Reference Values**

– **Parametric Plots**. Here, you can also define **Layout Settings** where you can determine whether the layout of your results plots (such as contours, vectors, etc.) will be included in your report as a series of tabs, or in tabular columns. These settings can be applied to all the graphics results images in your report. These settings can be applied to the report as a whole, or to individual graphical object categories. See Changing the Layout of Your Results (p. 4384) for details.

– Any available graphics objects that you want to include in your report, such as **Named Expressions**, **Mesh**, **Contours**, **Vectors**, **Pathlines**, **LICs**, **XY Plots**, **Scenes**, or your own custom view(s). Additionally, you can display the object using a particular orientation using the **View** field (front, back, left, bottom, isometric, etc.). Use the **Preview View** button to see what the image will look like once it is in the report. The isometric view is the default orientation.

Note:

For **Named Expressions**, you can use the **Expression Details** list to customize the named expressions within your report. You can choose to include the following details of your expressions: **Definition**, **Unit**, **Used In**, and **Description**. All options are selected by default. Deselect those options that you do not want to include in your report.

Note:

The parametric report's table of named expressions does not include any named expressions that are used as input parameters. This is because the definition for these expressions originates from the input parameter value for the design point that is designated to be the current design point when you generate the parametric report. So, the definition appears static in the parametric report, when in reality it is not static but instead is based on the input parameter value for each design point. The input parameter values for each design point are still found in the design point table section of the report where the context is more sensible.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

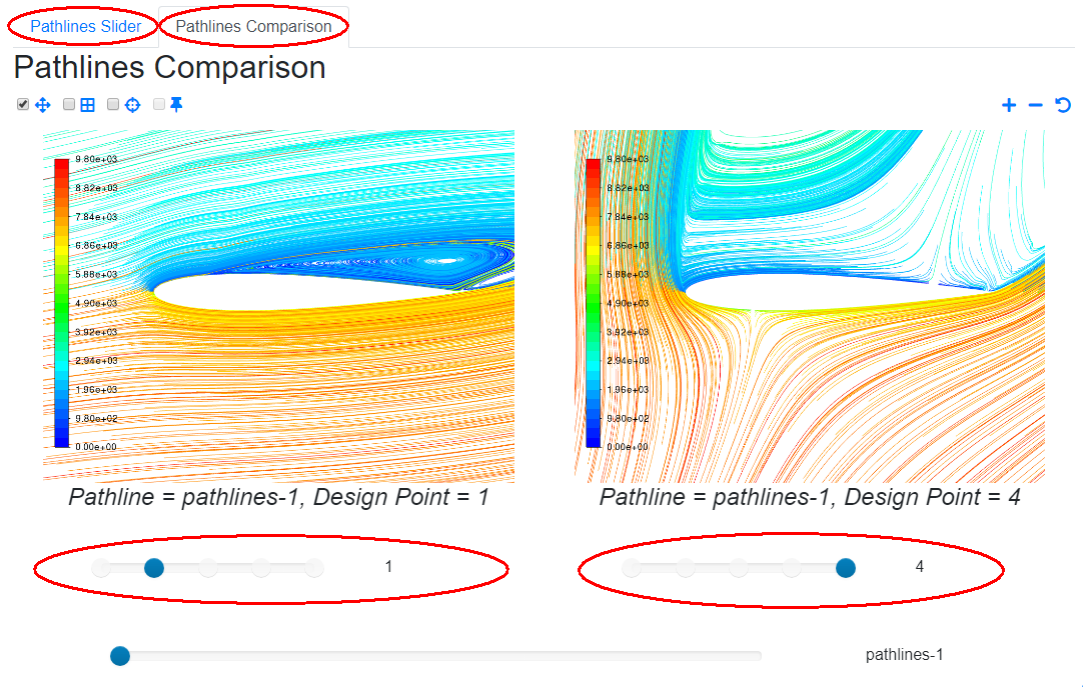
The individual design point report's table of named expressions, however, does include such input parameter named expressions. This is because the inclusion of the static expression definition makes more sense in the context of the single design point and case file.

46.8.2.3. Comparing Parametric Results

As part of the **Report Settings**, under **Image Slider View Settings**, you can determine whether your results plots (such as contours, vectors, etc.) will include a **standard** slider, a **comparison** slider, or **both**.

Choosing the **standard** option provides a single tab where you can use a standard slider to view the results for one design point at a time. Choosing the **comparison** option provides a single tab where two results can be visualized side-by-side with their own sliders. Choosing the **both** option provides both a standard slider tab, and the comparison tab.

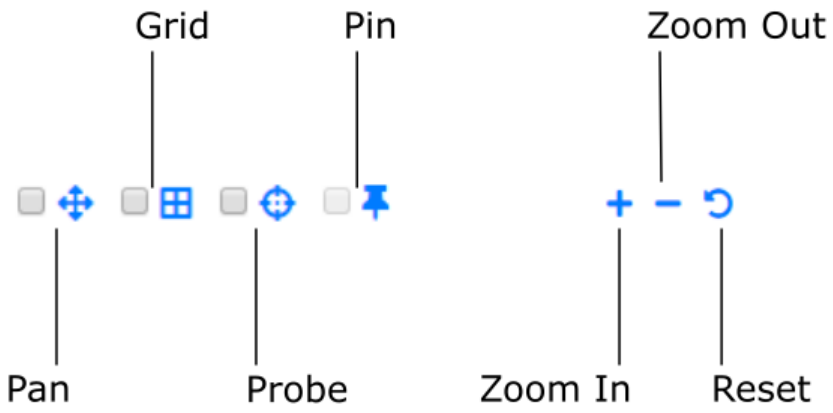
Figure 46.10: An Example of Using Both Slider Views for Pathline Comparison



The comparison view contains useful tools to interactively analyze the details of your plots within your report.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 46.11: Results Comparison Tools



To enable the **Pan**, **Probe**, **Grid**, and **Pin** features, select the corresponding check box.

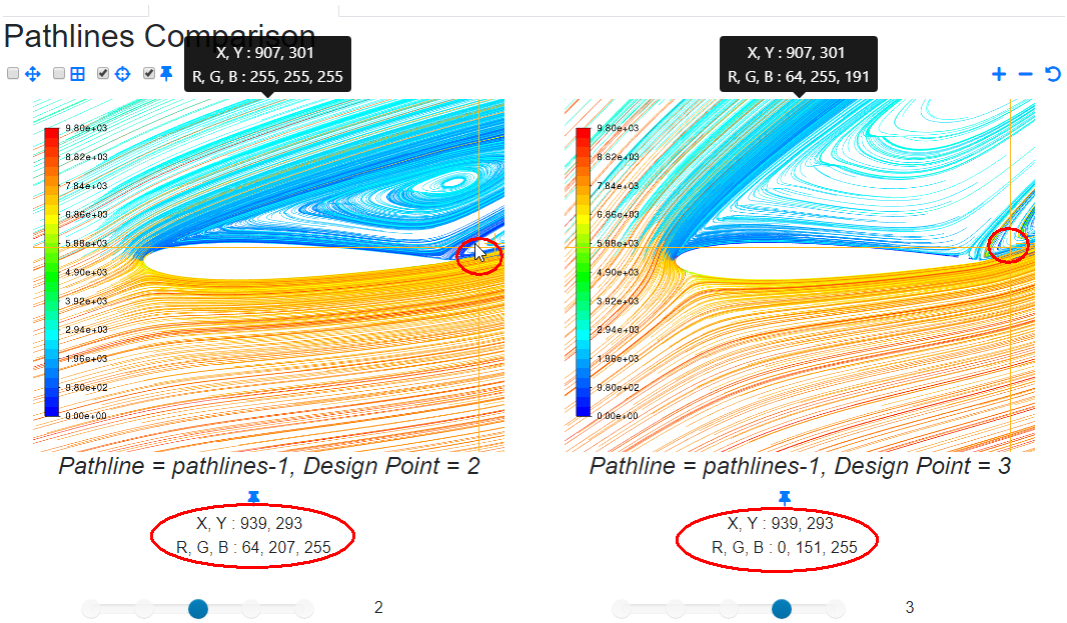
Note:

You cannot use the **Probe** in conjunction with either the **Pan** or the **Grid** feature at the same time.

- **Pan** lets you move the move around the displayed graphics, especially useful when you use the **Zoom In** and **Zoom Out** features.
- **Grid** lets you add grid lines to the graphics display.
- **Probe** lets you explore the displayed results and obtain specific values within the display (X, Y coordinates, as well as RGB color values).
- **Pin**, when used in conjunction with the **Probe**, lets you explore the displayed results and obtain specific values within the display (X, Y coordinates, as well as RGB color values), and upon double-clicking, leaves a 'pinned' marker on the display and reports the values for that pinned probe at the bottom of the comparison chart.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 46.12: Example of Using a Pinned Probe to Compare Pathline Results

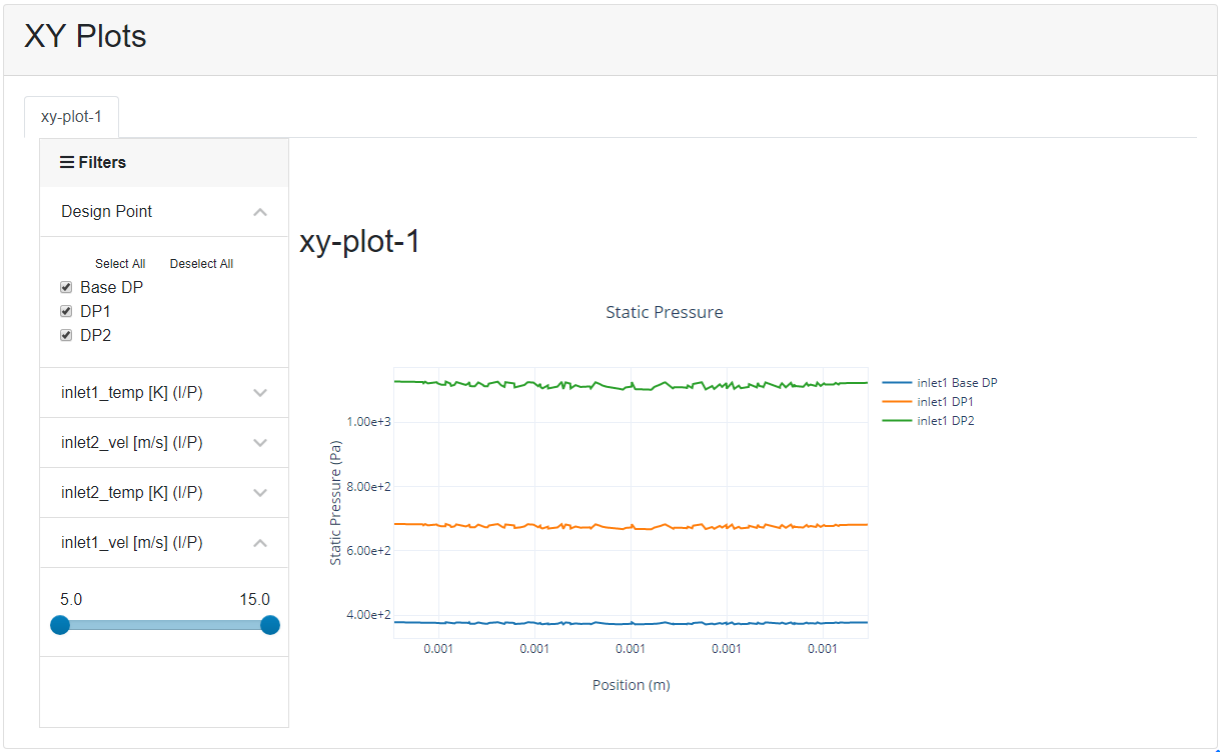


When used in conjunction with the corresponding sliders, you can investigate your results at the same point for various design points.

In addition to results plots, you can also compare XY plots between design points in the parametric report. Filters can be applied within the plot that can show and/or hide relevant design point data. You can also filter the results by input parameter value. In addition, any active filters applied to the plot comparison view are listed at the top of the plot area, along with a button to remove any filters (and reset the filter to its default state), allowing you a convenient way to reset one or more filters quickly and revert back to the original comparison plot.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 46.13: Example of Comparing XY Plots Between Design Points



Note:

Parametric reports exported to HTML will show the dynamic results of the XY plot comparison however, those exported to PowerPoint and/or PDF will only present a static view of the XY plot based on settings used at the time the report was generated.

46.8.2.4. Generating, Viewing, and Saving Your Parametric Reports

When you are satisfied with your simulation setup and results, and have configured your report to your satisfaction, generate a parametric report using the **Generate** button at the bottom of the task page. See Generating Simulation Reports (p. 4366) for more information.

Once the parametric report is generated, it appears in its own tab alongside the graphics window. Alternatively, you can use the **View** button at the bottom of the task page. See Viewing Simulation Reports (p. 4367) for more information.

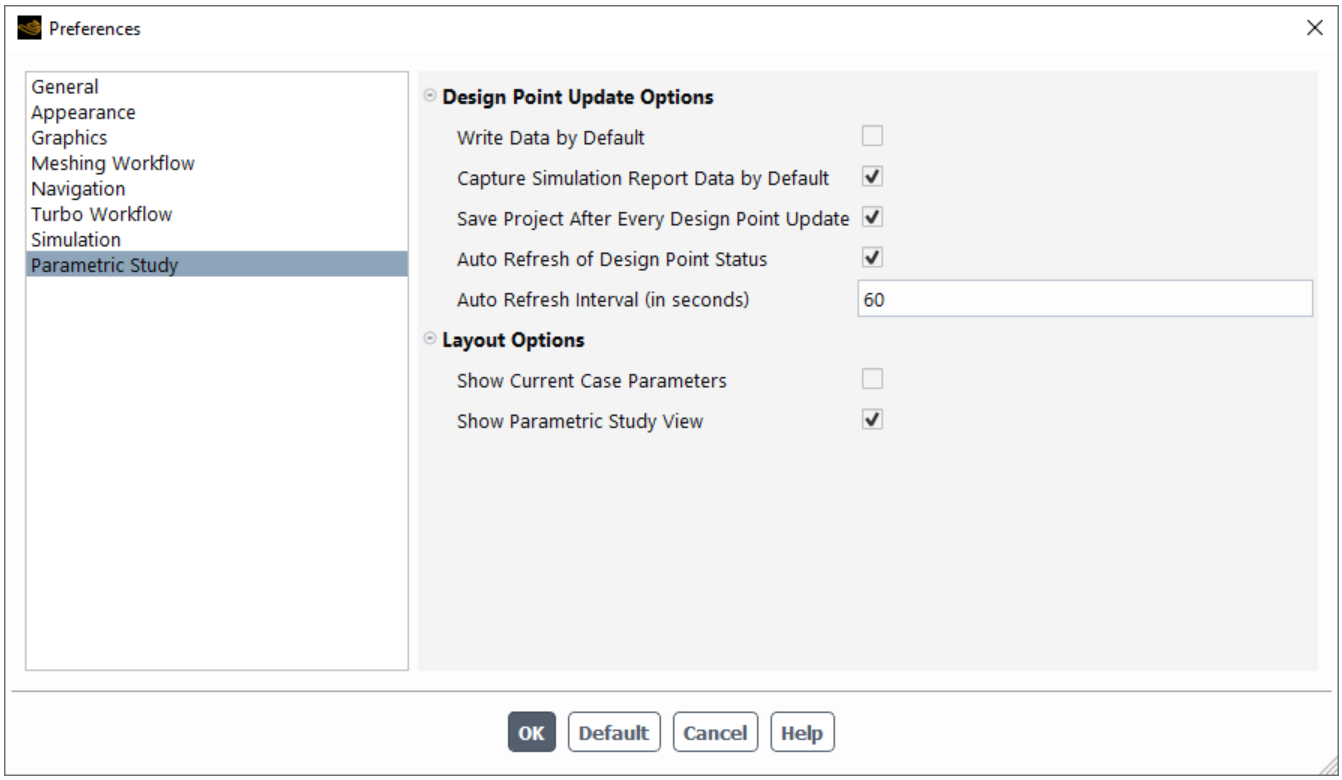
Once generated, you can save your reports as PDF or standalone HTML using the **Export** button at the bottom of the task page. See Saving Simulation Reports (p. 4383) for more information.

46.9. Setting Preferences for Parametric Studies

Certain preferences are available when working with parametric studies. They can be accessed using the Fluent **Preferences** dialog (under the **File** menu), in the **Parametric Study** category.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 46.14: Preferences for Parametric Studies



The preferences are arranged according to:

- **Design Point Update Options**
  - **Write Data by Default** - allows you to have the **Write Data** column in the design point table enabled by default so that all design points will save the appropriate data upon update.
  - **Capture Simulation Report Data by Default** - allows you to have the **Capture Simulation Report Data** column in the design point table enabled by default so that all design points will save the appropriate simulation report data upon update.
  - **Save Project After Every Design Point Update** - allows you to automatically save the design point project whenever a design point is updated.
  - **Auto Refresh of Design Point Status** - allows you to automatically refresh the status of your design points given a specified time interval.
  - **Auto Refresh Interval** - when automatically refreshing design points, this field allows you to perform an automatic refresh of the design points after a specified time interval. While the default value of time interval is 10 seconds (with a minimum value of 5 seconds), you can change this time interval based on the particular case, the number of iterations set, and the number of cores allocated.
- **Layout Options**
  - **Show Current Case Parameters** - allows you to expose the **Current Case Parameters** table.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– **Show Parametric Study View** - allows you to expose the **Parametric Study View**.

46.10. Viewing the Current Case Parameters

To display a table of the parameters for the current case, use the **Show Current Case Parameters** option under **Layout Options** in the Fluent **Preferences** dialog (under the **File** menu), in the **Parametric Study** category (see Setting Preferences for Parametric Studies (p. 4436) for details.

Current Case Parameters

ID	Parameter Name	Value	Unit
Input Parameters			
P1	inlet1_temp	300	[K]
P2	inlet1_vel	10	[m/s]
P3	inlet2_temp	350	[K]
P4	inlet2_vel	10	[m/s]
Output Parameters			
P5	outlet-temp-avg-op	324.9999	[K]

This table contains your current case's input and output parameters along with their values and units.

46.11. Managing Files for Your Parametric Studies

A parametric study can involve multiple, varying individual simulations, each having their own case and data files, in addition to intermediate and accessory files (for example, mesh files, report definition files, transcript files, journal files, plot files, and so on). Ansys Fluent helps you manage your files using "projects" and knowing when and where to set the current working directory.

46.11.1. Organizing Parametric Studies Using Projects

46.11.2. Managing Design Point Files

46.11.3. Using Lightweight Parametric Projects

46.11.1. Organizing Parametric Studies Using Projects

Since a parametric study can contain multiple, individual simulations, each having their own associated case and data files. Ansys Fluent organizes multiple, related case and data file sets using a *project* file (using a `*.flprj` file extension), and is especially useful in the context of managing multiple parametric studies.

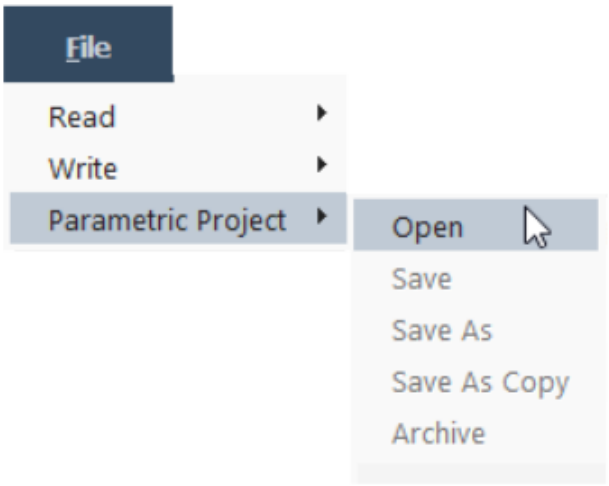
46.11.1.1. Reading and Writing Project Files

46.11.1.2. Using the Parametric Project View

46.11.1.1. Reading and Writing Project Files

Use the **Parametric Project** option under the **File** menu to read and write project files.

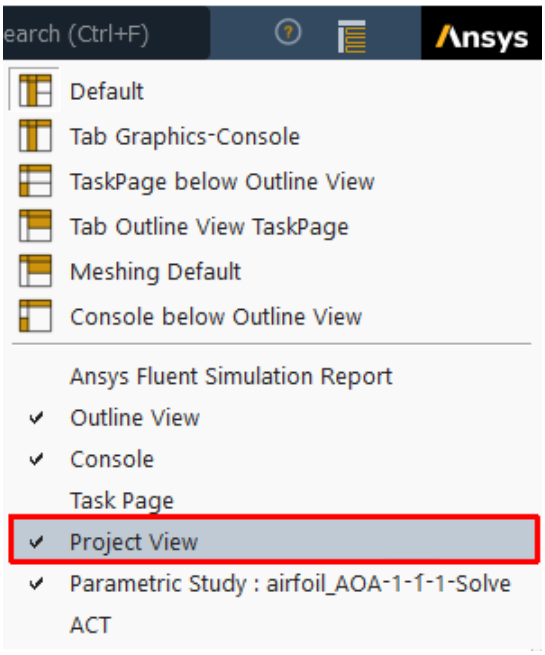
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Using this menu, you can open and save project files as required. In addition, you can also archive a project file for later analysis or sharing purposes.

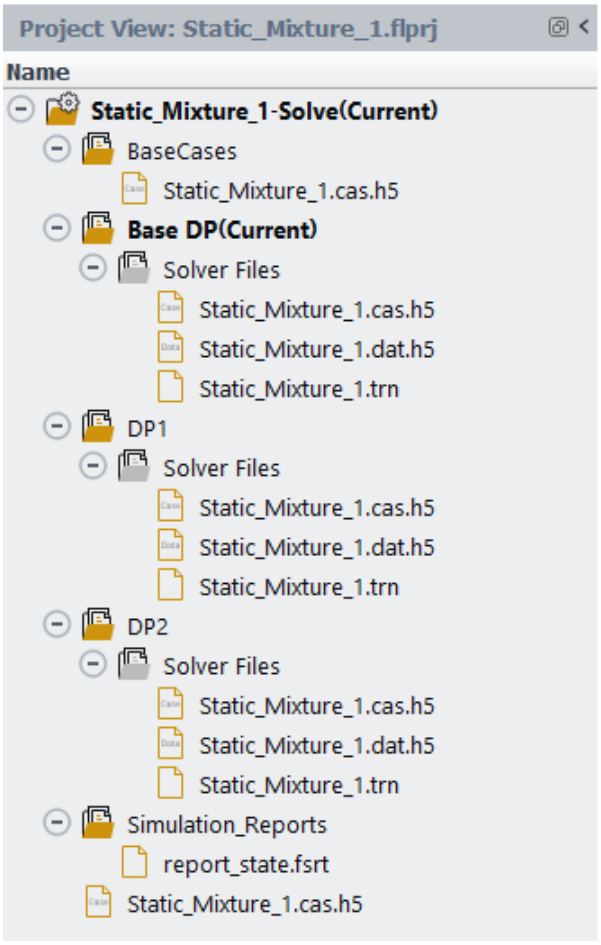
46.11.1.2. Using the Parametric Project View

The **Project View** is available so you can see the various input and output files that make up the Ansys Fluent parametric simulation project. This view gives you an overview of the various files associated with your design point analysis. You can access the view by selecting **Project View** from the list of available views in the Ansys Fluent interface.



Once the menu is selected, the **Project View** tab is visible in the interface.

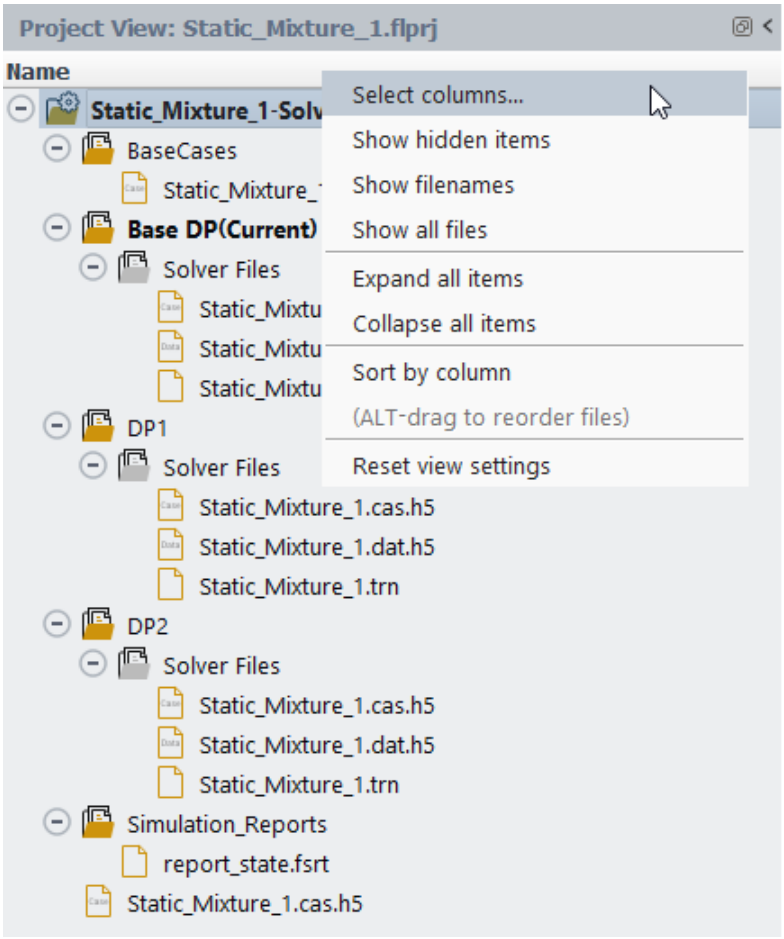
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Output files are available if **Write Data** has been enabled for a design point in the **Design Point Table**. You can also see case and data files for the various design point simulations in your project, as well as simulation reports. Other output files, such as `.out` files that are generated during an update are also visible here irrespective of the **Write Data** option's value.

Customize your Project View by right-clicking next to the **Name** to open the context menu options.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

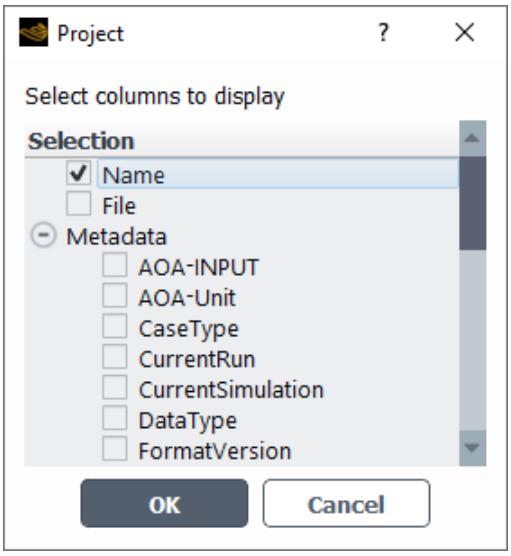


Note:

If the **Automatic Transcript** option is enabled in **Preferences**, then Fluent will also produce a transcript file for the design point.

From this context menu, you can expose and rearrange features of the Project View to your liking. Use the **Select Columns...** option to open the **Project** dialog where you can select from a number of options to expose additional columns in your Project View.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



46.11.2. Managing Design Point Files

When setting a design point to be the "current" design point, or when updating a design point (even when you enable or disable a design point's **Write Data** option), Fluent changes the working directory to be the design point's directory so that all relevant files are subsequently saved to that location.

In addition, when updating a design point, Fluent also registers any generated files with the associated project. These files are visible under the corresponding design point in the **Project View** (see Using the Parametric Project View (p. 4439)).

Note:

After you initialize a parametric study, for *any new animation definitions* that you subsequently create using the **Animation Definition** dialog box, Fluent sets the **Storage Directory** field to '.' and disables it, thereby storing these newly created animation-related files into the study's current design point folder or directory. Once the new animations are defined, when you later update the design points, all animation-related files will now be stored inside the corresponding design point's directory under the parametric project. This does not apply to any pre-existing animation definitions defined prior to initializing the parametric study.

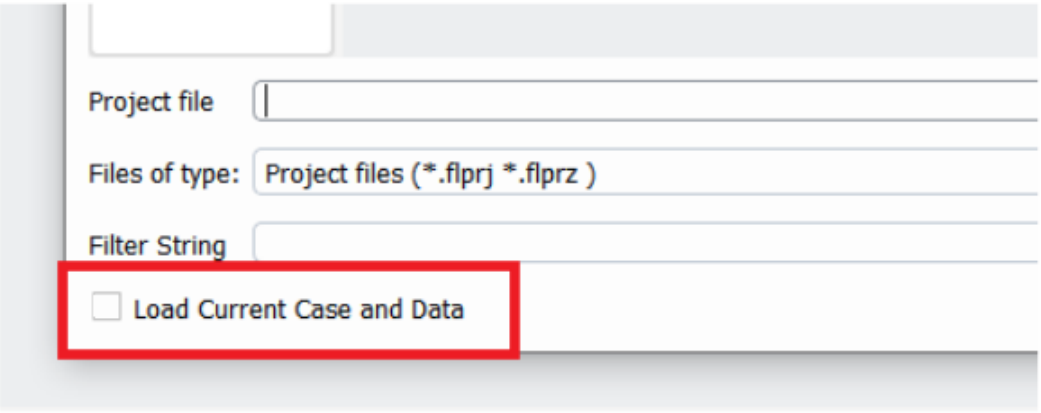
Note:

When updating a parametric study sequentially, this extra file will get created in the current simulation directory in the project. The file does not get registered with the project, and it is harmless and can be ignored or deleted without any issues.

46.11.3. Using Lightweight Parametric Projects

You can choose to open a parametric project without loading any case and/or data files into the session (also known as a "lightweight" project), by *deselecting* the **Load Current Case and Data** option in the project's **Select File** dialog.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Since the case file is not loaded into the session, Fluent can retrieve parametric information from the lightweight parametric project file. As such, you are able to update the design points in either **Sequential** and **Concurrent** mode (see Updating Design Points (p. 4395)). In addition, you can also add design points to the lightweight parametric project using Ansys optiSLang (see Automatically Adding Design Points (p. 4402)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 47: Remotely Accessing Your Simulations Using Ansys Fluent's Web Interface

You can use the Ansys Fluent web interface to remotely interact with and monitor the progress of sometimes lengthy CFD simulations. This chapter provides an overview of how to set up and use the Ansys Fluent web interface.

47.1. Overview

47.2. Getting Started With Ansys Fluent's Web Interface

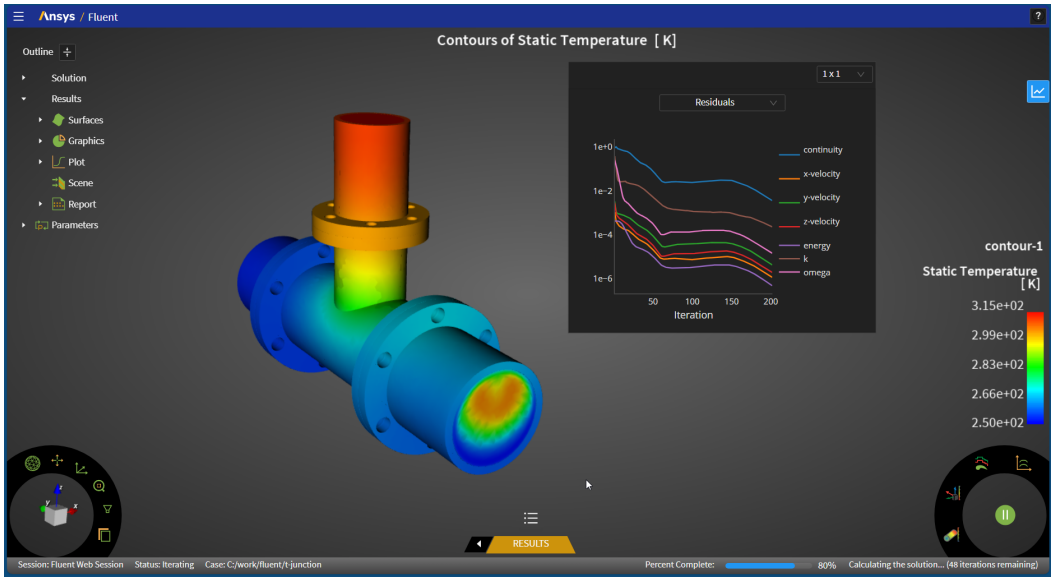
47.3. Using Ansys Fluent's Web Interface

47.4. Setting Up Your Environment (System Administrators)

47.1. Overview

When some CFD simulations can typically be very time-intensive, sometimes running for several days or weeks to complete, you can use Ansys Fluent's web interface to periodically review the progress of these lengthy simulations.

Figure 47.1: Ansys Fluent's Web Interface



Ansys Fluent's web interface is based on starting a web server in your Fluent session for:

- Starting/stopping transcript, monitor, and progress streaming

Ansys Fluent's web interface allows you to:

- Stream transcript, monitor, and progress

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Access field data for postprocessing

From within Ansys Fluent's web interface, you can

- Access the list of your running simulation sessions
- Connect to one or more existing sessions and disconnect from connected sessions
- Create/modify/delete surfaces
- Create/modify/delete graphics objects
- Display graphics objects
- Pause/Resume the simulation
- Interrupt/Continue the simulation
- Modify a limited collection of settings
- Read and/or write case files of your simulation sessions

Important:

Note that any changes made in the Fluent web interface are not necessarily immediately reflected in the dialog boxes, panels, and task pages of the corresponding open Fluent desktop session. Likewise, any changes made in the open Fluent desktop session are not necessarily immediately reflected in the dialog boxes and panels of the corresponding Fluent web interface session.

To reflect any changes that may have been made using the web interface in any open dialog boxes and panels and task pages in the open Fluent desktop session, you should close and re-open any panels, dialog boxes, and/or task pages in the open Fluent desktop session. Likewise, to reflect any changes that may have been made using the open Fluent desktop session in any open dialog boxes and panels in the open Fluent web session, you should close and re-open any dialog boxes and panels in the open Fluent web session.

Note:

Ansys Fluent's web interface only supports features that are available under **Pro** licensing (see Program Capabilities (p. 147)), such as access to a reduced set of Fluent solver capabilities allowing the solution of incompressible and compressible steady-state and transient, single-phase, turbulent, non-reacting flows, and heat transfer.

Note that only supported options and functionalities are displayed in the interface.

Note:

While calculations are progressing, you are intentionally not allowed to make changes to your simulation settings and/or perform certain actions, such as:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Reading case/data files.
- Starting another calculation.
- Editing solution and/or initialization settings, such as the number of iterations, timesteps, etc.

47.1.1. Requirements

Ansys Fluent's web interface has the following requirements:

- Supported web browsers: recent versions of the Chrome, Firefox, and Edge.
- Supported operating systems: Windows 10, 11, Linux.

47.1.2. Limitations

You will note the following limitations when using Ansys Fluent's web interface:

- When writing report definitions, the web session may not initially provide a warning that you are attempting to overwrite an existing file. When doing do, press **Enter** in the **File Name** box for a second time, then the session will allow you to append or overwrite accordingly.
- While calculations are progressing, you are intentionally not allowed to make changes to your simulation settings and/or perform certain actions, such as editing model settings.
- Mouse interactions may at times be limited while the web interface is busy processing a request.
- Performance issues may be experienced when working with large mesh sizes, based on memory allocation and depending on the web browser.

  To increase the performance for especially graphically intensive simulations, consider increasing the custom graphics performance of the web browser (for example, Chrome, Firefox, Edge, etc.).
- Pathlines are not available when running without graphics (using `-g` mode) since pathlines cannot be rendered without graphics.
- While using the Firefox web browser, clicking and dragging slider controls to change values causes the entire panel to be selected and dragged.
- Since the REST and WebSocket responses use gzip data compression, when both server and client are on the same machine, the overhead of compression and decompression increases the total rendering time. To overcome this, the environment variable `FLUENT_WEBSERVER_DISABLE_GRAPHICS_COMPRESSION` (when set to 1) can be used to disable the compression for graphics data transfer, improving the total rendering time for local environments.

47.2. Getting Started With Ansys Fluent's Web Interface

There are several ways to start your web session and connect to existing sessions:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Start a Fluent session and open a case and data file.

   Note:

   You can still access your Fluent session using the web interface even when you start Fluent in batch mode without a graphics window and user interface, that is, with the `-g` or the `-gu` command line options.

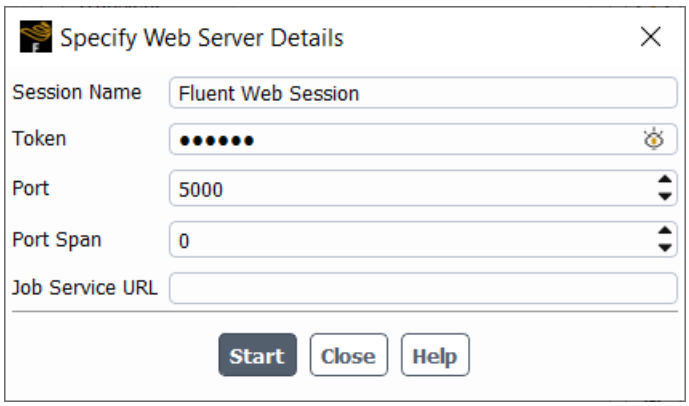
2. Start the web server. You can either:

   - Use the **File** menu.

     File → Applications → Web Server → Start...

     This opens the **Specify Web Server Details** dialog box.

     Figure 47.2: The Specify Web Server Details Dialog Box

     See Specifying Web Server Details (p. 4449) for more information about the available options.

     Once the web server is started, you can print out details of your settings to the console window using the **File > Applications > Web Server > Print Web Server Info** option.

     You can also stop the web server using the **File > Applications > Web Server > Stop** option.

   - Use the Fluent Launcher.

     In the **General** tab (see Setting General Options in Fluent Launcher (p. 184)), enable **Start Web Server** to access settings for the Fluent web service so that you can use the Fluent Web Server and remotely connect to a Fluent session.

     See Specifying Web Server Details (p. 4449) for more information about the available options.

   - Use the Fluent command line (see Command Line Startup Options (p. 191)), with the following available options:

     – `-ws`: starts the web service with default settings

     – `-ws=<x>`: starts the web service with a specified session name

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - `-ws-port=<x>`: specifies the port number for starting the web server
  - `-ws-portspan=<x>`: specifies the port span for starting the web server
  - `-ws-js-url=<x>`: specifies the job service URL for registering the web server

- Enter the following in the console window:

```
/server> start-web-server
Enter a name for the web server ["Fluent web session"] Fluent web session
Enter listening port [5000] 5000
Enter number of ports to try starting from given 'port' [0] 0
Enter the job service URL ["http://my-sample-server.com:4000"] "http://my-sample-server.com:4000"
```

47.2.1. Specifying Web Server Details
47.2.2. Connecting to Your Simulation Web Sessions

47.2.1. Specifying Web Server Details

The web server details are comprised of the following:

Session Name

The name of the web session.

Token

A unique string that you can use to access your web session.

Port

The listening port used by the web service.

Port Span

The number of ports to attempt to connect to if the given port is unavailable.

For example, given a port designation of 5000, if the port span is set to 0 (a special case), then any port can be used if port 5000 is unavailable. However, if the port span is set to 1, then only port 5000 is attempted, generating a message if the port is unavailable. Likewise, if the port span is set to 2, then only ports 5000 and 5001 are attempted, generating a message if the ports are unavailable, and so on.

Job Service URL

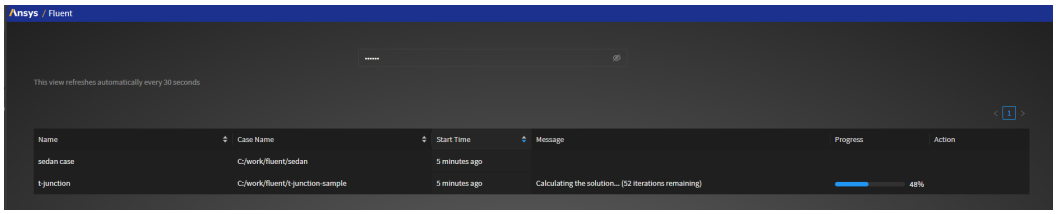
The web address that will serve the job. See Setting Up Your Environment (System Administrators) (p. 4488) for more information.

47.2.2. Connecting to Your Simulation Web Sessions

When you first gain access to Ansys Fluent's web interface, you are initially presented with the **Home** page, where you can enter your token string and see a list of available Fluent sessions that also use the token, providing you with access to the simulation(s).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 47.3: The Session Page for Ansys Fluent's Web Interface



Important:

For security reasons, you should not share your token or network path with anyone.

Once you identify a web session, you can click the name and Ansys Fluent's web interface will open the specified web session.

47.3. Using Ansys Fluent's Web Interface

After you have chosen a session in the **Home** page, Ansys Fluent's web interface opens your simulation session.

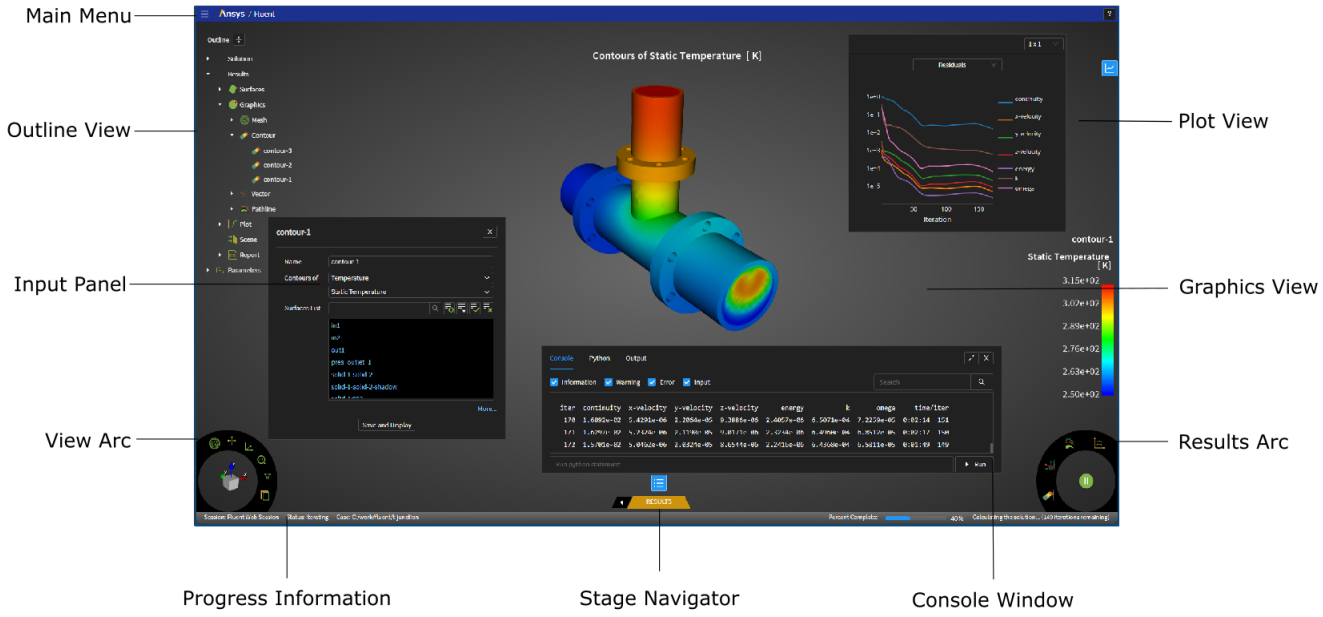
47.3.1. Overview of the Graphical Interface
47.3.2. Interacting with Your Simulation Setup
47.3.3. Interacting with Your Solution Settings
47.3.4. Interacting with the Results of Your Simulation
47.3.5. Working With Parameters

47.3.1. Overview of the Graphical Interface

The graphical interface for Ansys Fluent's web interface is similar to the standard Fluent graphical interface, but different in some significant ways.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

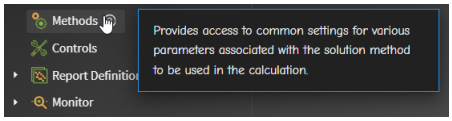
Figure 47.4: Components of Ansys Fluent's Web-Based User Interface



Note:

When available, field-level user assistance provides guidance and information for a selected field.

Figure 47.5: An Example of Field-Level User Assistance



- The **Main** menu is where you can return to the Home page where you can sign into other remote sessions, as well as read in and write out your case and data files, and access key user preferences. See The Main Menu (p. 4453) for more information.
- The **Outline View** is a simplified version of the Fluent's standard outline view. See The Outline View (p. 4454) for more information.
- The **View Arc** is a toolbar containing various view-related tools such as mesh display, fit-to-window, view orientation tools, and so on. See The View Arc (p. 4458) for more information.
- You can view pertinent details of your simulation status along the bottom of the window. See The Status Bar (p. 4459) for more information.
- The **Stage Navigator** is a tool that lets you navigate your simulation components (for example, **Solution** and **Results**). You can move through each component of the display to walk yourself though the stages of your simulation. Depending on the simulation mode, the outline view will change accordingly, presenting you with mode-specific tools and information. See The Stage Navigator (p. 4460) for more information.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The **Console** is available to see more information about the solution progress, the solution transcript, and any messages that the solution may generate.

  Note:

  Depending on the license level used in the Fluent session you are connected to (Enterprise or Premium), you can also enter and view Python commands in this window to augment your current simulation. This feature is not available if you are using the Pro level license.

  See The Console Panel (p. 4460) for more information.

- The **Results Arc** is a toolbar with results-related tools such as contour, vector, pathline display tools. In addition, you can also perform solution-related tasks such as solution initialization and running calculations, and view your session's results-based objects (contours, vectors, etc.). See The Results Arc (p. 4463) for more information.

- The **Graphics View** is where you can visualize your computational mesh, make graphics-related settings. See The Graphics Window (p. 4456) for more information.

- The **Plots View** is where you can visualize any 2D plotting formation such as residual plots, report definition plots, and any XY plots you may create. See The Plots View (p. 4465) for more information.

- Various input panels and dialog boxes are available for setting components of your simulation. See Property Panels (p. 4466) for more information.

- A **Help** menu is available ( ) that presents an overview of the user interface as well as a link to this document. In additon, an **About...** option displays information related to the current version of the web interface.

47.3.1.1. The Main Menu
47.3.1.2. The Outline View
47.3.1.3. The Graphics Window
47.3.1.4. The View Arc
47.3.1.5. The Status Bar
47.3.1.6. The Stage Navigator
47.3.1.7. The Console Panel
47.3.1.8. The Results Arc
47.3.1.9. The Plots View
47.3.1.10. Property Panels
47.3.1.11. Working with Units
47.3.1.12. Working with Expressions
47.3.1.13. Editing Multiple Objects at Once

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

47.3.1.1. The Main Menu

The main menu is equivalent to Fluent's standard **File** menu with many of the same options for reading, writing, importing, and exporting your simulation's various types of files (for example, mesh, case, data, etc.).

- **Read** allows you to open common Fluent files such the mesh, case, case settings, data, case and data, field functions, injections, and profiles.

  Note:

  When reading files, you can type in the file path and name in the corresponding field, or you can use the **Browse...** button to open a **Select File(s)** dialog box where you can browse the file system for the location and name of the file(s) you require. Note that the dialog box displays files on the "remote" file system (where Fluent is running) not the "local" machine (where the web browser is running).

- **Write** allows you to save common Fluent files such as the case, data, and case and data files. You can also replace the mesh, as well as start and stop your transcript files.

  Note:

  When writing files, you can type in the file path and name in the corresponding field, or you can use the **Browse...** button to open a **Select File(s)** dialog box where you can browse the file system for the location and name of the file(s) you require. Note that the dialog box displays files on the "remote" file system (where Fluent is running) not the "local" machine (where the web browser is running).

- **Import** allows you to import various files into your session, including Chemkin and Functional Mock-up Unit (FMU) files.
- **Show Configuration** displays the current session's configuration settings in the console window.
- **Preferences...** displays the Fluent **Preferences** dialog box where user preferences are located when using the web interface.
  - The **Theme** option allows you to choose between the default **dark** color theme or a **light** color theme for the web interface.
  - The **Show mesh on connection** option allows you to choose whether or not to display the session's mesh in the graphics window immediately upon connecting to the session.
  - The **Show all available settings (beta)** option allows you to choose whether or not to make available more of the solver's more advanced settings and capabilities that

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

have not been fully tested. This option is available when you run your Fluent session using the Enterprise license.

Note:

Ansys Fluent's web interface only supports features that are available under **Pro** licensing (see Program Capabilities (p. 147)), such as access to a reduced set of Fluent solver capabilities allowing the solution of incompressible and compressible steady-state and transient, single-phase, turbulent, non-reacting flows, and heat transfer.

Additional features that are exposed using this preference have not been fully tested.

  - The **Show Ansys logo** option allows you to choose whether or not to display the Ansys logo in the graphics window.

- **Disconnect** allows you to leave the current Fluent session from within the web interface.
- **Shutdown Fluent** allows you to close the current Fluent session from within the web interface.

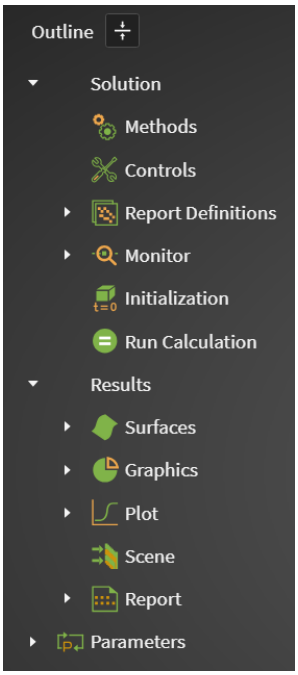
For more detailed information about these options, see File Ribbon Tab (p. 5347).

47.3.1.2. The Outline View

Ansys Fluent's web interface Outline View is similar to Fluent's standard outline view, allowing you to access and interact with various portions of your simulation project. Ansys Fluent's web interface displays the corresponding node according to the presence of the case or data file in Fluent. If data is present, then the Results branch is opened by default. so you can analyze your results. If only a case file is present, then the Solution branch is opened by default. If there is no case or data, then the Setup branch is available. Branches can be expanded and collapsed to suit your requirements. In addition, as you change stages using the Stage Navigator (The Stage Navigator (p. 4460)), the Outline View changes accordingly.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 47.6: Outline View



Generally, you can add, delete, and rename objects by using the context menu on selected items in the tree. Depending on the context of where you are in the Outline View, certain objects have a context menu that has one or more of the following options available:

- Rename
- New
- Copy to Clipboard (copying objects (such as report definitions, graphics objects such as contours, etc.) in the tree from within the same session or even between different sessions. Use this option to copy the selected object to your system clipboard, where you can then paste it into another selected object of the same type in the tree.
- Paste from Clipboard (pasting objects (such as report definitions, graphics objects such as contours, etc.) in the tree from within the same session or even between different sessions. Use this option to paste copied objects from your system clipboard into another selected object of the same type in the tree.
- Display (for showing selected objects in the graphics window)
- Duplicate
- Delete
- Type (for reassigning type for cell zones or boundary conditions)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- List Details

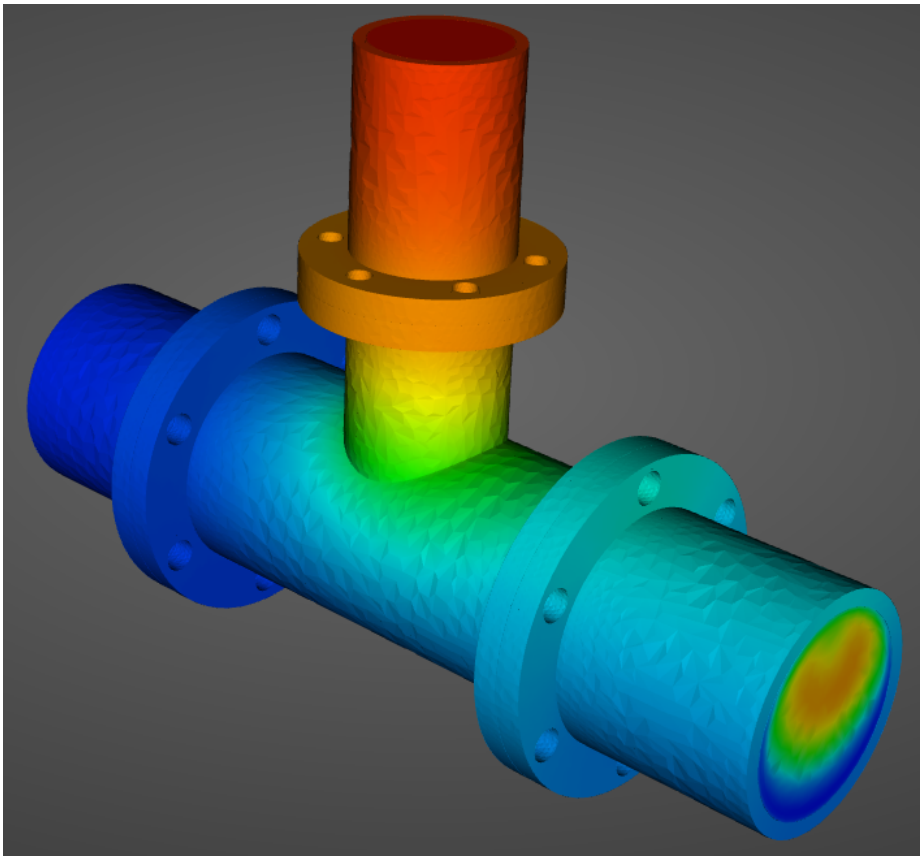
Note:

- To select multiple items in the Outline View at the same time, hold down the **Ctrl** key while selecting non-adjacent items or hold down the **Shift** key while selecting adjacent items. Context menu options are available when applicable.
- At the top level of the Outline View, cell zone and boundary condition can be displayed in various ways through the **Group By** context menu option where you have the choice of displaying a **List View**, by **Name**, by **Adjacency** (for boundaries only), or by **Zone Type** (the default).
- The Outline View can be relocated by hovering the mouse over the top of the tree until the cursor changes from the 'pointer' to the 'pan' symbol.

47.3.1.3. The Graphics Window

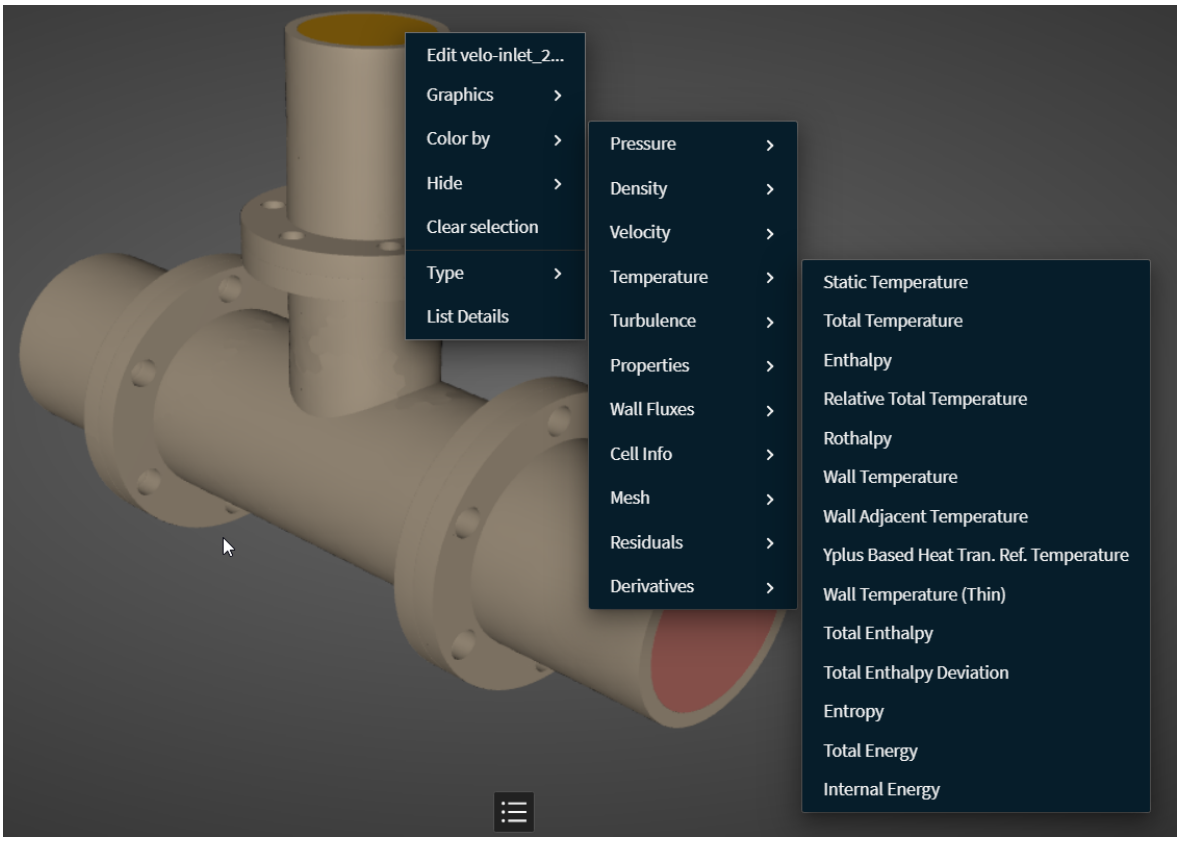
The graphics window displays the graphical output of your simulation (mesh, contours, pathlines, etc.). Along with being able to select one or more objects, rotate, zoom, pan, etc. within the window, you can also interact with the contents by right-clicking to display a context menu that corresponds with your current stage.

Figure 47.7: A Contour Object Displayed in the Graphics Window



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 47.8: A Context Menu for a Selected Boundary

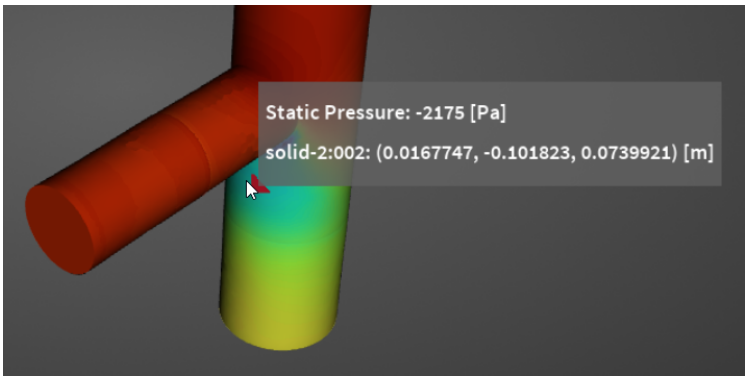


Supported operations within the graphics window depend on the selected object(s) and include:

- Single selection
- Box selection
- Interactively edit/show/hide selected items (via context menu, e.g., changing the zone type, etc.)
- Interactively create additional graphics objects (via context menu)
- Interactively move and resize the color map for results-based plots such as contours, vectors, etc. You can restore the color map to its original location by using the **Shift**+**r** key combination.
- Interactively use the mouse to probe your results to visualize specific values in specific areas. This is available using the corresponding mode in the View Arc (see The View Arc (p. 4458)) where you click a portion of the graphics object (for example, contour, vector, etc.) to display a listing of results-based details for the specified portion of the plot, such as its position, the name of the surface, and the current value of the field variable at that location. You can add multiple probes to the display by holding down the **Ctrl** key as you pick probe locations that persist as you manipulate the plots in the graphics window. Press **ESC** to dismiss the probe display(s). Probe information is preserved when you save the graphics view to a file.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 47.9: Accessing the Mouse Probe in the Graphics Window



For more general information about graphics windows in Fluent, see Displaying Graphics (p. 3869).

47.3.1.4. The View Arc

Ansys Fluent's web interface includes a View Arc, a circular toolbar located within the window, containing various view-related tools such as mesh display, fit-to-window, view orientation tools, and so on.

Figure 47.10: View Arc

Options include:

- **Mesh display options**: tools that allow you to create a new mesh object or display an existing mesh object.
- **Left mouse button options**: tools that allow you to choose between various left mouse behaviors such as selection, rotation, panning, zooming in or out, probing, etc. Note that the mouse icon changes according to your selection.

  Note that by default, the rotation mode allows rotation about the center of the screen. You can rotate around a specific point using the **Ctrl** key while in rotation mode.
- **View options**: tools that allow you to quickly orient the display to your liking (for example, isometric view, +/-X, +/-Y, and +/-Z directions).
- **Fit to window**: tool that allows you to quickly fit the display to the window.
- **Filter Selections**: tool that allows you to quickly filter your selections in the graphics window based on some or all of the available types.

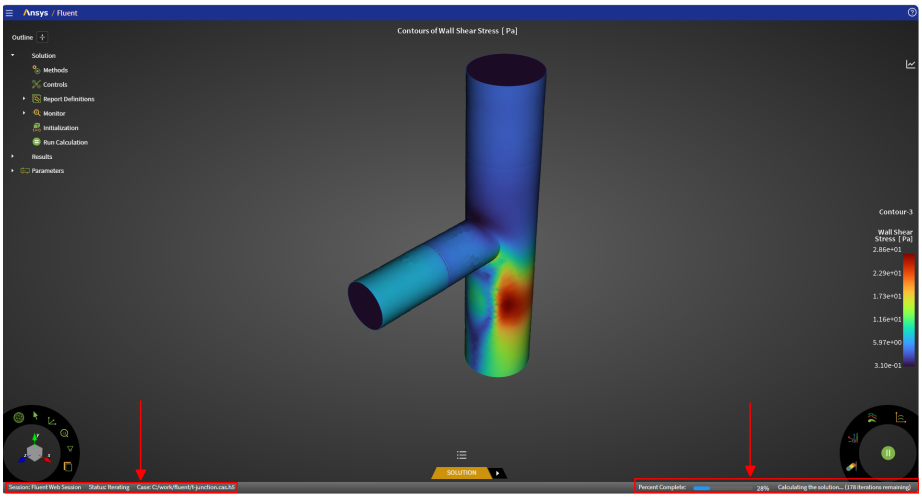
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Save to clipboard or file**: tools that allow you to quickly save the display to the clipboard or to a file.

47.3.1.5. The Status Bar

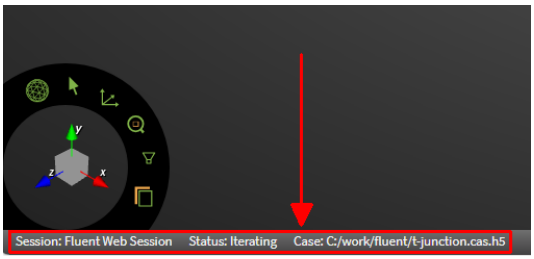
General information about your simulation can be found in the status bar at the bottom of the window.

Figure 47.11: Displaying Session and Solution Information in the Status Bar



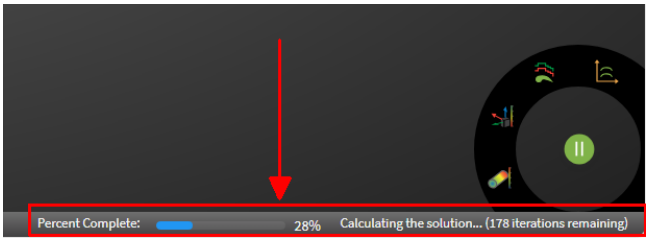
Details about your simulation (session name, status, case name) can be found in the left side of the status bar.

Figure 47.12: Displaying Session Information in the Status Bar



While a calculation is underway, you can view its progress in the status bar at the bottom of the browser window.

Figure 47.13: Displaying Solution Information in the Status Bar

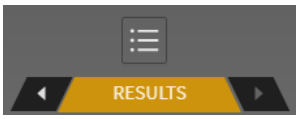


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

47.3.1.6. The Stage Navigator

The Stage Navigator is a means of traversing (using the arrows) through the various stages of your simulation. As you do, the Outline View (The Outline View (p. 4454)) changes to expose the relevant nodes that correlate to the various stages in the simulation.

Figure 47.14: Stage Navigator



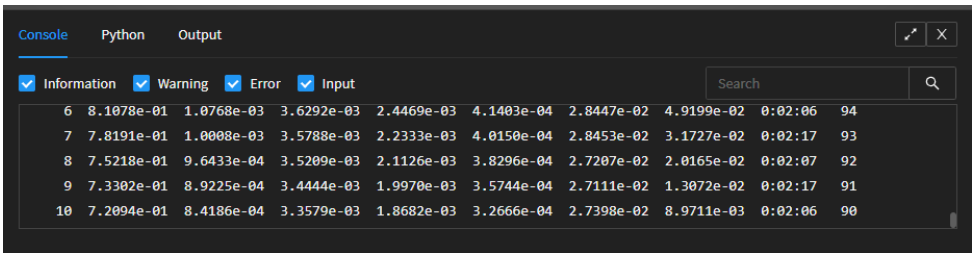
47.3.1.7. The Console Panel

The Console panel is your primary access to any informative messages, Python commands, and output from a simulation. To open the Console panel, click the corresponding icon ( ).

Console Messaging

The panel has a Console tab that will contain transcript data, informative messages about relevant simulation progress, warnings, errors, etc.

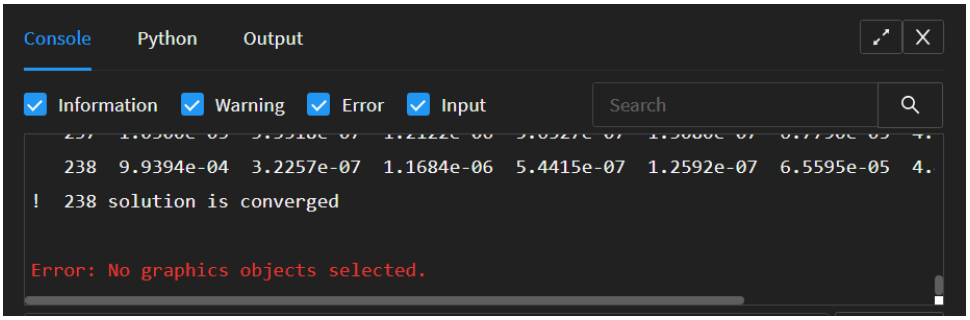
Figure 47.15: The Console Window



You are alerted of any errors via changes to the Console icon, for example:

Figure 47.16: The Console Icon With Two Alerts

Figure 47.17: The Console Window With Alerts

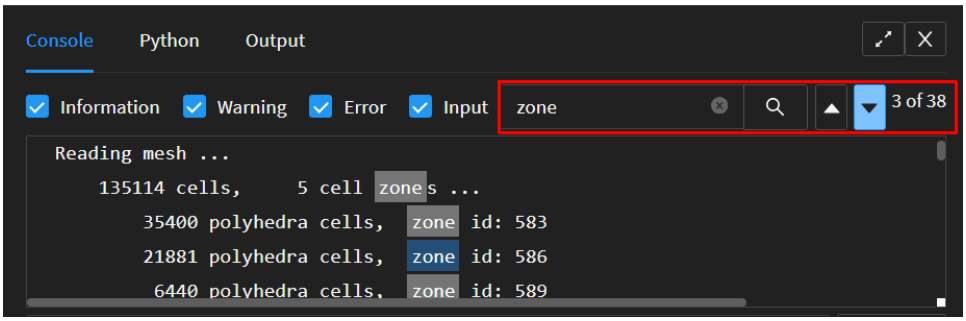


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can also filter messages by types – information, warning, error, and output.

You can also search the Console output using the **Search** field.

Figure 47.18: Searching the Contents of the Console Window



Access to Python

The Python tab shows Python commands while you interact with the session.

Figure 47.19: The Python Tab of the Console Window

Viewing Simulation Output

The Output tab contains basic output messages from the solver.

Figure 47.20: The Output Tab of the Console Window

Note:

The window can be relocated by hovering the mouse over the top of the tree until the cursor changes from the 'pointer' to the 'pan' symbol. You can also make the window

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

full size or close it altogether using the icons in the upper right corner. You can also close the window by clicking the corresponding icon above the Stage Navigator.

Note:

You can capture all text within the window by using **Ctrl-A**.

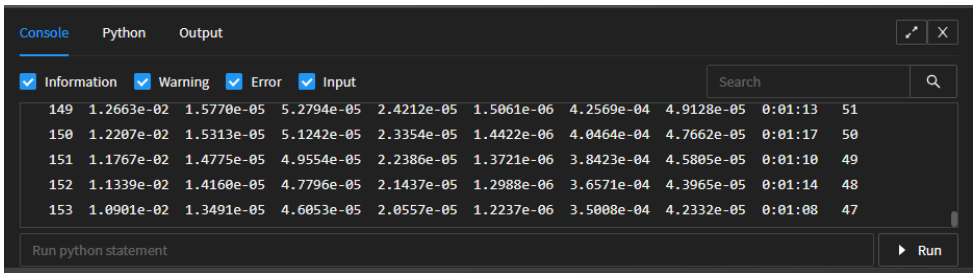
47.3.1.7.1. Using the Console to Interact with Your Session Using Python

Depending on the license level used in the Fluent session you are connected to (Enterprise or Premium), you can view and enter Python commands in this window to augment your current simulation.

Note:

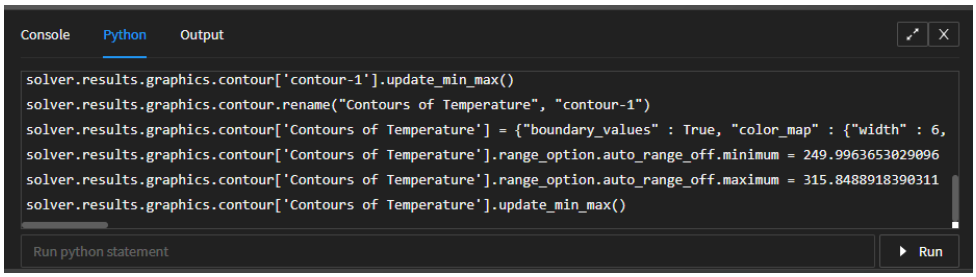
This feature is not available if you are using a Pro level license.

Figure 47.21: The Console Window with Python Command Access



In addition, the Console window also has a Python tab that allows you to view and enter Python commands and directly interact with the Fluent session using the collection of Pythonic interfaces for Fluent.

Figure 47.22: The Python Tab of the Console Window



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The Console window, the Python window, and the Output window all have a command entry field where you can type a Python command. Once you type in your command, click the **Run** button to run the command, or click the **Clear** button to remove the command from the field.

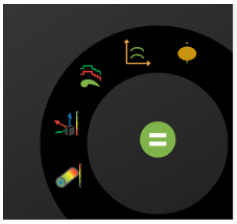
Note:

For both the Console window and the Python window, you can capture all text within the window by using **Ctrl-A**.

47.3.1.8. The Results Arc

Ansys Fluent's web interface includes a Results Arc, a circular toolbar located within the window, containing various results-related tools such as displaying contours, vectors, and so on.

Figure 47.23: Results Arc

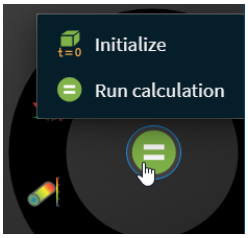


Options include:

- **Contour display options**: tools that allow you to create a new contour object or display an existing contour object.
- **Vector display options**: tools that allow you to create a new vector object or display an existing vector object.
- **Pathline display options**: tools that allow you to create a new pathline object or display an existing pathline object.
- **XY plot display options**: tools that allow you to create a new XY plot object or display an existing XY plot object.
- **Mirror plane options**: tools that allow you to create a new mirror plane or show/hide an existing mirror plane(s). See Creating and Displaying Mirror Plane Objects (p. 4480) for details.

You can also use the **Calculate** button to quickly initialize or to start a calculation.

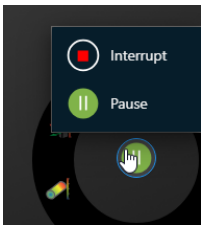
Figure 47.24: Initializing or Running a Calculation



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

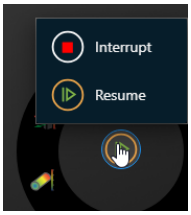
Once the calculations are in progress, you can choose to either **Interrupt** or **Pause** the solution progression.

Figure 47.25: Interrupting or Pausing a Calculation



Once the calculations are paused, you can choose to either **Interrupt** or **Resume** the solution progression.

Figure 47.26: Interrupting or Resuming a Calculation



Note:

There is a subtle difference between pausing and interrupting your calculations, depending on whether you are running a transient or a steady-state simulation, or if you are running a journal file or not.

When you choose to **Pause** a solution, for steady-state simulations, the solver progresses until it reaches the earliest safe stopping point after the current iteration, whereas for transient simulations, the solver progresses until it reaches the earliest safe stopping point after the current time step.

When you choose to **Interrupt** a solution and you are running a journal file, the act of interrupting the solution leads the solver to stop and exit from the journal file.

Note:

While running standard Fluent in batch mode, by default, Fluent does not exit when the iterations are complete, unless an explicit end-of-file mechanism (that is, when standard input (`stdin`) is not present) is manually in place via a script, etc.

During a batch calculation, if an error occurs, or if you choose to interrupt the batch calculation, then the journal is stopped. If the web server is running, Fluent will remain active and available, allowing you to interact with it using the web interface just as you would in an interactive session.

- If there is a specified value for the **Idle Timeout** preference in Fluent, then Fluent will shut down if there is no activity in the web interface after the specified idle time out

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

period. Just as in Fluent, a countdown message is displayed in the web interface before shutting down the session due to inactivity. For instance:

```
This process has been idle for 33 secs, It will start exiting
after 27 secs. The case and data will be saved before
exiting. Press Close to continue to work.
```

- If there is no specified value for the **Idle Timeout** preference in Fluent, one of two things can happen, regardless of if there is an explicit `exit` command in the journal file:
  - If an explicit end-of-file mechanism is in place for the connected batch session, then a timeout period of 30 minutes is automatically applied and a corresponding message will immediately appear in the console:

    ```
    Setting Idle time of 30 min after which session will exit.
    ```

    This timeout period is only started when the journal is stopped and when the batch session is transformed into an interactive session.
  - If an explicit end-of-file mechanism is *not* in place for the connected batch session, then no message appears, meaning that Fluent is still running and will need to be manually stopped.

Note:

While calculations are progressing, you are intentionally not allowed to make changes to your simulation settings and/or perform certain actions, such as:

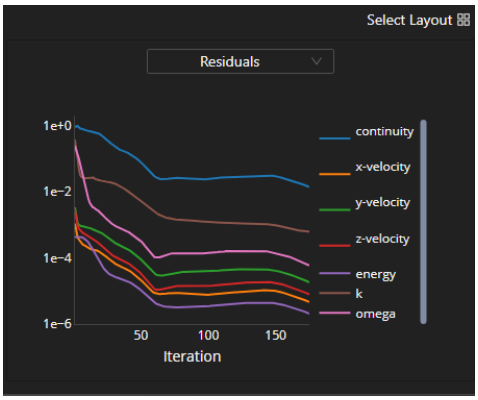
- Reading case/data files.
- Starting another calculation.
- Editing model settings.
- Editing solution and/or initialization settings, such as the number of iterations, timesteps, etc.

47.3.1.9. The Plots View

Ansys Fluent's web interface includes a Plots View, where you can visualize your calculation's residual plots, as well as any report definition plots for your simulation. Click the title to open the corresponding plot.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 47.27: Plots View



Note:

The window can be relocated by hovering the mouse over the top of the window until the cursor changes from the 'pointer' to the 'pan' symbol. You can minimize the window using the icons in the upper right corner.

When the window is not in its docked position, you can also resize the window by hovering the mouse over the bottom edge or the left edge of the window until the cursor changes to the 'edge' symbol.

Hovering the mouse over the plots reveals specific data values and exposes some assorted controls for plotting:

- **Download plot as png**: saves the plot as a PNG file, prompting you to browse and specify the name and location of the file.
- **Zoom**: use the mouse to draw a box that the view will zoom into.
- **Pan**: use the mouse to pan across the XY plot.
- **Zoom in**: zoom into the current view.
- **Zoom out**: zoom out of the current view.
- **Auto-scale**: resets the view to the original size and scale.
- **Reset Axes**: resets the X and Y axes.

For more information about using plots, see Working With Plots and Your Simulation Results (p. 4482).

47.3.1.10. Property Panels

You can access and edit the attributes of various objects in your simulation through the various property panels that are available. The properties panels are similar to Fluent's task pages and dialog boxes, in that they allow you to define settings, enter, and postprocess results. Note that your settings are saved as you define them in a properties panel. There are also some dialog boxes (which you may open from the Outline View or settings in the properties panel) which may have an explicit mechanism to save your work.

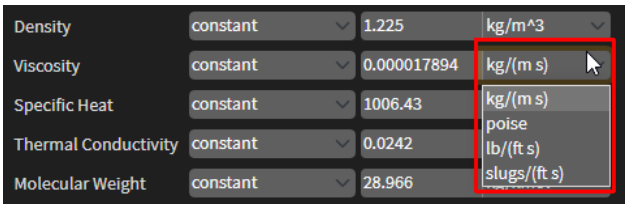
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The content of the various property panels is similar to those found in Fluent, however, there are some differences. For instance, the visual layout of the groups, labels, and controls is more streamlined, as are the handling of units and expressions.

47.3.1.11. Working with Units

Unit management is more flexible from Fluent's handling of units. Using the corresponding drop-down menu, you can change the units accordingly on a case-by-case basis, if desired.

Figure 47.28: Changing Units for a Given Property



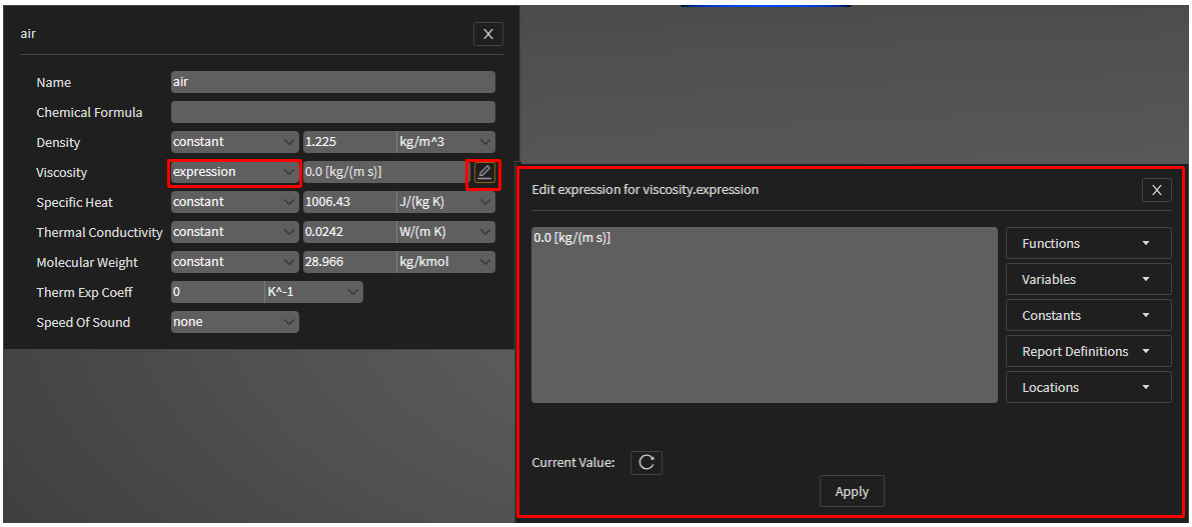
Units can be managed from the General properties in the Outline View.

47.3.1.12. Working with Expressions

Handling expressions is similar to Fluent, however, there are two ways to work with expressions in Ansys Fluent's web interface.

- Choosing an expression for certain material properties, when available. In these cases, you can choose to represent your material property using an expression by choosing **expression** from the corresponding drop-down menu. The expression is listed, and you can then edit the expression using the corresponding icon ( ) to open the expression editor.

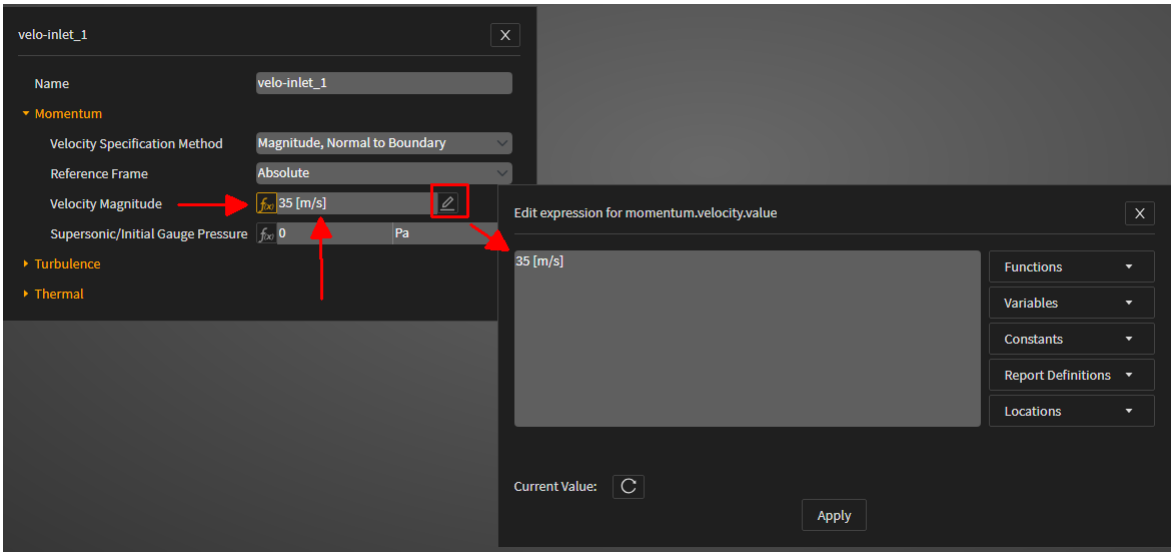
  Figure 47.29: Using Expressions for Material Properties



- Using an expression for boundary conditions, when available. In these cases, an expression icon ( ) can be selected ( ) to enable the use of an expression for a property. When

selected, the expression is presented and can be changed by clicking the corresponding icon ( ) to open the expression editor.

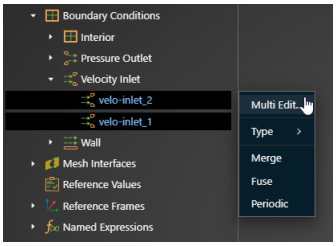
Figure 47.30: Using Expressions for Boundary Conditions



47.3.1.13. Editing Multiple Objects at Once

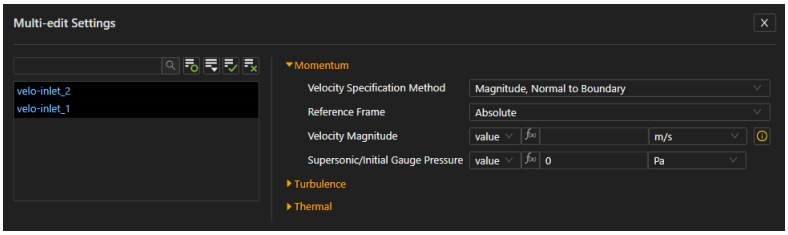
Especially for boundary conditions of the same type, you are able to edit more than one object at a time and apply one or more of an object's properties to another. For instance, you can select two velocity-inlet conditions, right-click and select **Multi-Edit** from the context menu.

Figure 47.31: Editing Multiple Boundary Conditions from the Context Menu



This displays the **Multi-Edit Settings** panel where you can see the selected objects, and their common properties.

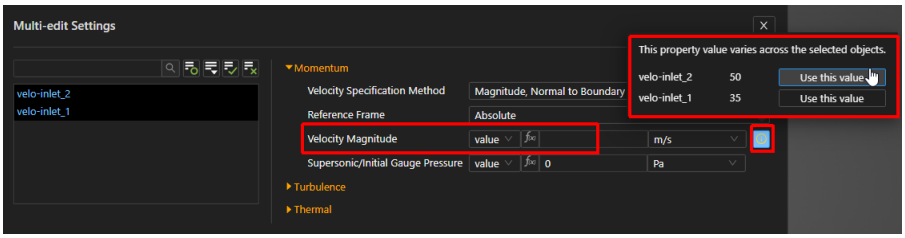
Figure 47.32: An Example of the Multi-Edit Settings Panel



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For common, active properties, an informational icon ( ) next to an empty property indicates a property where you can choose a value from one of the selected objects.

Figure 47.33: An Example of Using the Multi-Edit Settings Panel



For more information about multiple selection using Fluent, see Editing Multiple Boundary Conditions at Once (p. 1521).

47.3.2. Interacting with Your Simulation Setup

One aspect of Ansys Fluent's web interface is the ability to interact with and change your Fluent simulation set up within the web interface. The Setup branch of the Outline view allows you to access and modify simulation settings (models, materials, cell zone and boundary conditions, etc.).

Note:

While calculations are progressing, you are intentionally not allowed to make changes to your simulation settings and/or perform certain actions, such as:

- Reading case/data files.
- Starting another calculation.
- Editing model settings.
- Editing solution and/or initialization settings, such as the number of iterations, timesteps, etc.

The following sections describe operations available for setting up your simulation in Ansys Fluent's web interface.

47.3.2.1. Accessing General Settings
47.3.2.2. Accessing Model Settings
47.3.2.3. Accessing Material Settings
47.3.2.4. Accessing Cell Zone Settings
47.3.2.5. Accessing Boundary Condition Settings
47.3.2.6. Accessing Reference Values Settings
47.3.2.7. Accessing Reference Frame Settings
47.3.2.8. Accessing Named Expression Settings

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

47.3.2.1. Accessing General Settings

You are able to access your simulation's general properties using Ansys Fluent's web interface.

Setup → General

From here, you can view or make changes to settings such as the solver type or operating conditions such as the gravitational acceleration, and unit settings. The available general settings are similar to those found in the standard Fluent solver. For more detailed information about them, see General Task Page (p. 4684).

47.3.2.2. Accessing Model Settings

You are able to access your simulation's model properties using Ansys Fluent's web interface.

Setup → Models

From here, you can view or make changes to the models available to your session such as the consideration of energy, turbulence (viscous), species, and discrete phase. The available model settings are similar to those found in the standard Fluent solver. For more detailed information about them, see the Fluent documentation (e.g., Models Task Page (p. 4696)).

- **Energy**: Energy Dialog Box (p. 4714)
- **Viscous**: Viscous Model Dialog Box (p. 4715)
- **Species**: Species Model Dialog Box (p. 4768)
- **Discrete Phase**: Discrete Phase Model Dialog Box (p. 4845)

47.3.2.3. Accessing Material Settings

You are able to access your simulation's material properties using Ansys Fluent's web interface.

Setup → Materials

From here, you can create new or view and make changes to the fluid and solid (and mixture, if applicable) materials available in your session, such as the material's density, specific heat, thermal conductivity, molecular weight, and so on. You can even access other material databases as well. The available material settings are similar to those found in the standard Fluent solver. For more detailed information about creating new and editing existing materials, see Create/Edit Materials Dialog Box (p. 4885).

47.3.2.4. Accessing Cell Zone Settings

You are able to access your simulation's cell zone conditions using Ansys Fluent's web interface.

Setup → Cell Zone Conditions

From here, you can create new or view and make changes to the fluid and solid cell zones conditions that are available in your session, such as a cell zone's assignment, reference frame, porosity, motion,

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

source terms, and so on. The available settings are similar to those found in the standard Fluent solver. For additional information about cell zone properties in the standard Fluent solver, see Cell Zone Conditions Task Page (p. 4958).

- For more information about **Fluid** cell zone conditions, see Fluid Dialog Box (p. 4961).
- For more information about **Solid** cell zone conditions, see Solid Dialog Box (p. 4973).

Note:

Cell zones can be displayed in the Outline View in various ways through the **Group By** context menu option where you have the choice of displaying a **List View**, by **Name**, or by **Zone Type** (the default).

47.3.2.5. Accessing Boundary Condition Settings

You are able to access your simulation's boundary conditions using Ansys Fluent's web interface.

Setup → Boundary Conditions

From here, you can create new or view and make changes to existing boundary conditions that are available in your session, such as a inlets, outlet, walls, and so on. The available settings for each boundary type are similar to those found in the standard Fluent solver. For more detailed information about them, see Boundary Conditions Task Page (p. 4991).

Note:

Boundary condition can be displayed in the Outline View in various ways through the **Group By** context menu option where you have the choice of displaying a **List View**, by **Name**, by **Adjacency**, or by **Zone Type** (the default).

47.3.2.6. Accessing Reference Values Settings

You are able to access your simulation's reference values using Ansys Fluent's web interface.

Setup → Reference Values

From here, you can create new reference values or view and make changes to existing reference values that are available in your session, such as area, density, enthalpy, length, pressure, and so on. The available settings are similar to those found in the standard Fluent solver. For more detailed information about them, see Reference Values Task Page (p. 5154).

47.3.2.7. Accessing Reference Frame Settings

You are able to access your simulation's reference frames using Ansys Fluent's web interface.

Setup → Reference Frames

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

From here, you can create new reference frames or view and make changes to existing reference frames that are available in your session, such as its motion or the initial state. The available settings are similar to those found in the standard Fluent solver. For more detailed information about them, see Reference Frame Dialog Box (p. 5592).

47.3.2.8. Accessing Named Expression Settings

You are able to access your simulation's named expressions using Ansys Fluent's web interface.

Setup → Named Expressions

From here, you can create new named expressions, create and view plots of your expressions, or view and make changes to existing named expressions that are available in your session. The available settings are similar to those found in the standard Fluent solver. For more detailed information about them, see Expression Manager Dialog Box (p. 5468).

47.3.3. Interacting with Your Solution Settings

Another aspect of Ansys Fluent's web interface is the ability to interact with and change your Fluent simulation settings within the web interface.

Note:

While calculations are progressing, you are intentionally not allowed to make changes to your simulation settings and/or perform certain actions, such as:

- Reading case/data files.
- Starting another calculation.
- Editing model settings.
- Editing solution and/or initialization settings, such as the number of iterations, timesteps, etc.

The following sections describe operations available for solution settings in Ansys Fluent's web interface.

47.3.3.1. Setting Solution Methods
47.3.3.2. Setting Solution Controls
47.3.3.3. Setting Report Definitions
47.3.3.4. Setting Solution Monitors
47.3.3.5. Initializing the Solution
47.3.3.6. Running the Calculations
47.3.3.7. Interrupting and Pausing Calculations

47.3.3.1. Setting Solution Methods

Various means of setting your solution methods are available

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution → Methods

From here, you can view and make changes to solution methods, such as the discretization scheme for pressure, momentum, and so on. The available settings are similar to those found in the standard Fluent solver. For more detailed information about them, see Solution Methods Task Page (p. 5157).

47.3.3.2. Setting Solution Controls

Various means of setting your solution controls are available

Solution → Methods

From here, you can view and make changes to solution controls, such as relaxation factors, equations, limits, and so on. The available settings are similar to those found in the standard Fluent solver. For more detailed information about them, see Solution Controls Task Page (p. 5163).

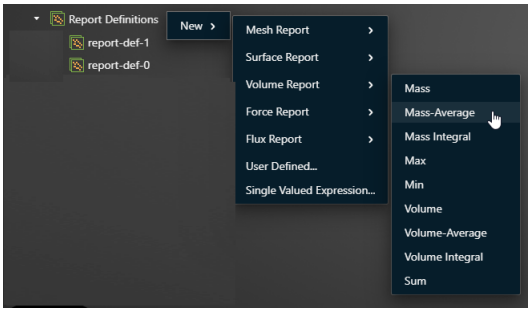
47.3.3.3. Setting Report Definitions

Various means of setting your solution's report definitions are available

Solution → Report Definitions

From here, you can create new or view and make changes to existing report definitions in your session, such as report definitions for mesh, surface, volume, force, lift, drag, moment, flux, user-defined, and single-valued expressions.

Figure 47.34: Using The Context Menu to Create Report Definitions



The available settings are similar to those found in the standard Fluent solver. For more detailed information about them, see Report Definitions Dialog Box (p. 5594).

47.3.3.4. Setting Solution Monitors

Various means of setting your solution's monitors are available

Solution → Monitors

From here, you can create new or view and make changes to existing solution monitors in your session, such as residual, report file, report plot, and convergence condition monitors. The available settings are similar to those found in the standard Fluent solver. For more detailed information about them, see the Fluent documentation.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Residuals** (see Residual Monitors Dialog Box (p. 5599))
- **Report Files** (see Report File Definitions Dialog Box (p. 5596))
- **Report Plots** (see Report Plot Definitions Dialog Box (p. 5597))
- **Convergence Conditions** (see Convergence Conditions Dialog Box (p. 5416))

47.3.3.5. Initializing the Solution

Various means of initializing your solution are available

Solution → Initialization

From here, you can view and make changes to solution controls such as the initialization type, the reference frame, various default solution settings, and so on. The available settings are similar to those found in the standard Fluent solver. For more detailed information about them, see Solution Initialization Task Page (p. 5179).

Note:

You can also initialize and start the calculations using the Results Arc ( ).

47.3.3.6. Running the Calculations

Various means of running the calculations methods are available

Solution → Run Calculation

From here, you can view and make changes to solution controls important when running the calculations, such as time step settings, the number of iterations, reporting interval, and so on. The available settings are similar to those found in the standard Fluent solver. For more detailed information about them, see Run Calculation Task Page (p. 5207).

Note:

You can also initialize and start the calculations using the Results Arc ( ).

47.3.3.7. Interrupting and Pausing Calculations

Once a calculation is started, you can either interrupt or pause the Fluent solver. When you interrupt a calculation, Fluent formally exits the iteration loop, however, when you pause a calculation, the Fluent solver is paused without exiting the iteration loop.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When using a journal file, interrupting a calculation will completely exit the journal file while pausing a calculation lets you continue to use your journal file.

Note:

To improve performance, you should pause Fluent when you are post-processing results during a calculation.

47.3.4. Interacting with the Results of Your Simulation

A key aspect of Ansys Fluent's web interface is the ability to analyze the results of your calculations as the Fluent client is solving a fluid flow simulation. In addition, using Ansys Fluent's web interface, you can create and edit your results as the solver is running.

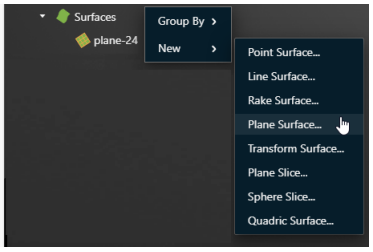
The following sections describe operations available for graphics objects in Ansys Fluent's web interface.

47.3.4.1. Working With Surfaces and Your Simulation Results
47.3.4.2. Working With Graphics Objects and Your Simulation Results
47.3.4.3. Working With Plots and Your Simulation Results
47.3.4.4. Creating and Displaying Scenes and Your Simulation Results
47.3.4.5. Creating and Displaying Reports for Your Simulation Results

47.3.4.1. Working With Surfaces and Your Simulation Results

You can create different types of surfaces to visualize your results, including points, lines, rakes, planes, iso-surfaces, iso-clips, transform surfaces, plane slices, sphere slices, and quadric surfaces. Values are interpolated to the smoothed out boundary lines.

Figure 47.35: Using The Context Menu to Create Surfaces



For more information on working with surfaces in Fluent, see Creating Surfaces and Cell Registers for Displaying and Reporting Data (p. 3813).

47.3.4.1.1. Creating and Displaying Point Surfaces
47.3.4.1.2. Creating and Displaying Line Surfaces
47.3.4.1.3. Creating and Displaying Rake Surfaces
47.3.4.1.4. Creating and Displaying Plane Surfaces
47.3.4.1.5. Creating and Displaying Iso Surfaces
47.3.4.1.6. Creating and Displaying Iso-Clip Surfaces
47.3.4.1.7. Creating and Displaying Transform Surfaces

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

47.3.4.1.8. Creating and Displaying Plane Slice Surfaces
47.3.4.1.9. Creating and Displaying Sphere Slice Surfaces
47.3.4.1.10. Creating and Displaying Quadric Surfaces

47.3.4.1.1. Creating and Displaying Point Surfaces

You may be interested in displaying results at a single point in the domain. For example, you may want to monitor the value of some variable or function at a particular location. To do this, you must first create a "point" surface, which consists of a single point. When you display node-value data on a point surface, the value displayed is a linear average of the neighboring node values. If you display cell-value data, the value at the cell in which the point lies is displayed.

Results → Surfaces → Point New

Specify the location of the point. Enter the coordinates (**x, y, z**) under **Coordinates**.

47.3.4.1.2. Creating and Displaying Line Surfaces

A line is simply a line that extends up to and includes the specified endpoints; data points will be located where the line intersects the faces of the cell, and consequently may not be equally spaced.

Results → Surfaces → Line New

1. Specify the location of the **P0** (start point).
2. Specify the location of the **P1** (end point).

47.3.4.1.3. Creating and Displaying Rake Surfaces

A rake consists of a specified number of points equally spaced between two specified endpoints.

Results → Surfaces → Rake New

1. Specify the location of the **P0** (start point).
2. Specify the location of the **P1** (end point).
3. Set the **Number of points** to include in the rake surface.

47.3.4.1.4. Creating and Displaying Plane Surfaces

To display flow-field data on a specific plane in the domain, you will use a plane surface. You can create surfaces that cut through the solution domain along arbitrary planes (3D cases only).

There are three types of plane surfaces that you can create:

- Coordinate system-based—the plane is created in the YX, ZX, or XY directions, bounded by the extents of the domain.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Point and normal—the plane orientation is determined by selecting a point and specifying a direction normal to that plane. The extents of the plane are the edges of the domain.
- Three points—the plane orientation and extents are bounded by three points that you can specify.

You can create a plane surface by specifying the **Method** as **yz**, **xz**, **xy** then specify the corresponding **X**, **Y**, or **Z** value.

47.3.4.1.5. Creating and Displaying Iso Surfaces

If you want to If you want to display results on cells that have a constant value for a specified variable, you will need to create an iso-surface of that variable. Generating an iso-surface based on *x*, *y*, or *z* coordinate, for example, will give you an *x*, *y*, or *z* cross-section of your domain; generating an iso-surface based on pressure will enable you to display data for another variable on a surface of constant pressure. You can create an iso-surface from an existing surface or from the entire domain. Furthermore, you can restrict any iso-surface to a specified cell zone.

Results → Surfaces → Iso-Surface → New

To create an iso-surface:

1. Click the **Field** drop-down menu to specify the field you want to view.
2. (Optional) Specify a **Surface** to only display iso-values on specific surfaces.
3. (Optional) Specify a **Zone** to only display iso-values on specific zones.
4. Set the range by specifying the **Minimum** and **Maximum**.
5. Enter the iso-values into the **Iso-Value** field. You can enter multiple iso-values into the field separated by spaces.

47.3.4.1.6. Creating and Displaying Iso-Clip Surfaces

If you have created a surface, but you do not want to use the whole surface to display data, you can clip the surface between two isovalues to create a new surface that spans a specified subrange of a specified scalar quantity. The clipped surface consists of those points on the selected surface where the scalar field values are within the specified range.

Results → Surfaces → Iso Clip → New

Specify the variables for **Clip to Values of** as well as one or more **Clip Surfaces** and click **Display**.

47.3.4.1.7. Creating and Displaying Transform Surfaces

You can create a new data surface from an existing surface by rotating and/or translating the original surface. For example, you can rotate the surface of a complicated turbomachinery blade to plot data in the region between blades. You can also create a new surface at a constant normal distance from the original surface.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Results → Surfaces → Transform Surface **New**

Specify one or more **Surface**s, as well as any rotational and/or translational properties as needed, then click **Display**.

47.3.4.1.8. Creating and Displaying Plane Slice Surfaces

If you want to display data on a planar surface, you can specify the surface by entering the coefficients of the function that defines it.

Results → Surfaces → Plane Slice **New**

The surface will consist of all points on the domain that satisfy the equation $ix*x+iy*y+iz*z=\text{distance}$. You will specify **ix** (the coefficient of x), **iy** (the coefficient of y), **iz** (the coefficient of z), and **Distance** (the distance of the line or plane from the origin), then click **Display**.

47.3.4.1.9. Creating and Displaying Sphere Slice Surfaces

If you want to display data on a spherical surface, you can specify the surface by entering the coefficients of the function that defines it.

Results → Surfaces → Sphere Slice **New**

The surface will consist of all points on the domain that satisfy the equation $(x-x0)^2+(y-y0)^2+(z-z0)^2=r^2$. You will specify **x0**, **y0**, **z0** (the x, y, and z coordinates of the sphere or circle's center) and **r** (the radius)

Specify one or more **Center** properties as needed, as well as a **Radius**, then click **Display**.

47.3.4.1.10. Creating and Displaying Quadric Surfaces

If you want to display data on a general quadric surface, you can specify the surface by entering the coefficients of the quadric function that defines it. This feature provides you with an explicit method for defining surfaces.

Results → Surfaces → Quadric Surface **New**

The surface will consist of all points in the domain that satisfy the general quadric function $Q=\text{value}$. You will specify the coefficients of the quadric function Q (the coefficients of the terms x^2, y^2, z^2, xy, yz, zx, x, y, z and the constant term).

Specify one or more **Attributes** as needed, as well as a **Value**, then click **Display**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

47.3.4.2. Working With Graphics Objects and Your Simulation Results

Ansys Fluent's web interface can create new, and access, display, and edit existing graphics objects that are defined on the client.

Note:

By default, Ansys Fluent's web interface displays graphics object results using the **field-velocity** color map.

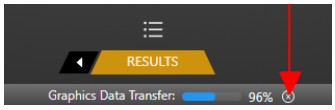
Note:

The name of the graphics object is used as the title of the display.

Note:

You can cancel the rendering of complex graphical objects that may take an excessive amount of time by clicking the **Stop** icon ( ) near the progress bar while the rendering is taking place.

Figure 47.36: Pausing Graphics Rendering



The following sections describe operations available for graphics objects in Ansys Fluent's web interface.

47.3.4.2.1. Creating and Displaying Mesh Objects
47.3.4.2.2. Creating and Displaying Contours Objects
47.3.4.2.3. Creating and Displaying Vector Objects
47.3.4.2.4. Creating and Displaying Pathline Objects
47.3.4.2.5. Creating and Displaying Mirror Plane Objects

47.3.4.2.1. Creating and Displaying Mesh Objects

You can draw the mesh for all or part of your domain.

Results → Graphics → Mesh → New

For the new mesh object, make the necessary changes and Click **Display** to see the new mesh object in the graphics view.

For more information on working with meshes in Fluent, see Displaying the Mesh (p. 3872).

47.3.4.2.2. Creating and Displaying Contours Objects

Ansys Fluent's web interface enables you to plot contour lines or profiles. Contour lines are lines of constant magnitude for a selected variable (isotherms, isobars, and so on).

Results → Graphics → Contour → New

For a new contour object, make the necessary changes and click **Display** to see the new contour object in the graphics view.

For more information on working with contours in Fluent, see Displaying Contours and Profiles (p. 3896).

47.3.4.2.3. Creating and Displaying Vector Objects

You can draw vectors in the entire domain, or on selected surfaces. By default, one vector is drawn at the center of each cell (or at the center of each facet of a data surface), with the length and color of the arrows representing the velocity magnitude. The spacing, size, and coloring of the arrows can be modified, along with several other vector plot settings. Velocity vectors are the default, but you can also plot vector quantities other than velocity. Note that cell-center values are always used for vector plots, and you cannot plot node-averaged values.

Results → Graphics → Vector → New

For the new vector object, make the necessary changes and click **Display** to see the new vector object in the graphics view.

For more information on working with vectors in Fluent, see Displaying Vectors (p. 3912).

47.3.4.2.4. Creating and Displaying Pathline Objects

Pathlines are used to visualize the flow of massless particles in the problem domain. The particles are released from one or more surfaces present in your domain.

Results → Graphics → Pathline → New

For the new pathline object, make the necessary changes and click **Display** to see the new pathline object in the graphics view.

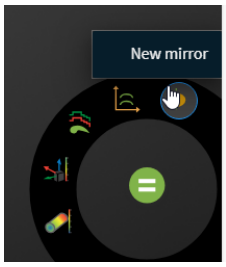
For more information on working with pathlines in Fluent, see Displaying Pathlines (p. 3922).

47.3.4.2.5. Creating and Displaying Mirror Plane Objects

You can create special view-related objects for mirroring your geometry about a symmetrical plane using the Results Arc.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 47.37: Creating Mirror Planes Using the Results Arc



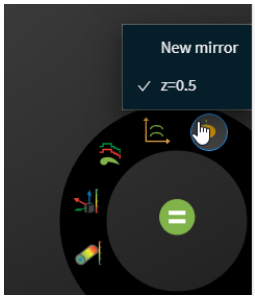
You can also create new mirror planes through the Outline View.

Results → Graphics → Views → Mirror Planes → New

In either case, this displays the **New Mirror Plane** dialog box (see Creating New Mirror Planes (p. 4481)).

Once created, you can access your mirror planes through the arc (a check mark next to the name of the mirror plane indicates that it is visible):

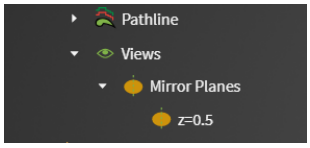
Figure 47.38: Accessing Mirror Planes Using the Results Arc



You can also access existing mirror planes through the Outline View:

Results → Graphics → Views → Mirror Planes

Figure 47.39: Accessing Mirror Planes Using the Outline View



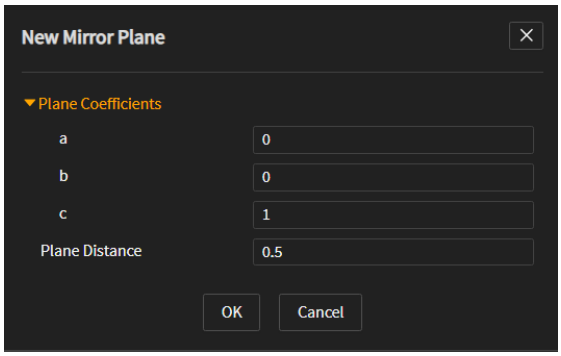
For more information on working with mirror planes in Fluent, see Mirroring and Periodic Repeats (p. 4036).

47.3.4.2.5.1. Creating New Mirror Planes

Using the **New Mirror Plane** dialog box, you can specify the **Plane Coefficients** and the **Plane Distance**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 47.40: The New Mirror Plane Dialog Box



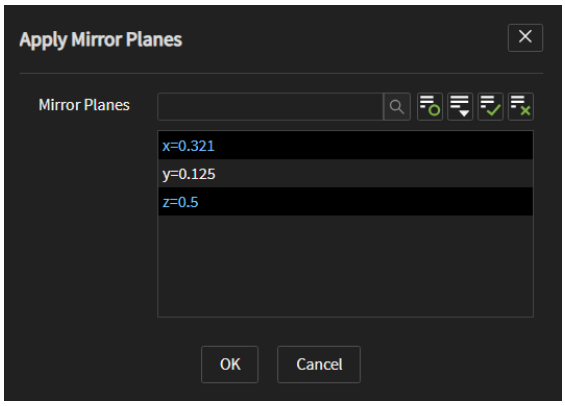
47.3.4.2.5.2. Applying Your Mirror Planes

When you have multiple mirror planes available, but only need to use a select few, you can open the **Apply Mirror Planes** dialog box to pick and choose which planes to use.

Results → Graphics → Views Apply Mirror Planes

This displays the **Apply Mirror Planes** dialog box:

Figure 47.41: The New Mirror Plane Dialog Box



Use the **Apply Mirror Planes** dialog box to pick which mirror planes to employ.

47.3.4.3. Working With Plots and Your Simulation Results

In addition to the many graphics tools already discussed, Ansys Fluent's web interface also provides tools that enable you to generate XY plots of solution, file, profile, and residual data.

Note:

Plots can be relocated by hovering the mouse over the top of the plot window until the cursor changes from the 'pointer' to the 'pan' symbol.

When the window is not in its docked position, you can also resize the window by hovering the mouse over the bottom edge or the left edge of the window until the cursor changes to the 'edge' symbol.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

47.3.4.3.1. Creating and Displaying XY Plots
47.3.4.3.2. Generating Histogram Data
47.3.4.3.3. Generating Cumulative Data
47.3.4.3.4. Viewing Residual Plots
47.3.4.3.5. Viewing Report Definition Plots
47.3.4.3.6. Displaying Multiple Plots
47.3.4.3.7. Analyzing Data in Plots

47.3.4.3.1. Creating and Displaying XY Plots

An XY (abscissa/ordinate) plot is a line and/or symbol chart of data. Virtually any defined variable or function is accessible for this type of plot.

Results → Plot → XY Plot New

For the new XY plot object, make the necessary changes and click **Display** to see the new XY plot object in a separate window.

You can drill down into the specifics of the plot data interactively. See Analyzing Data in Plots (p. 4485) for details.

For more information on working with XY plots in Fluent, see XY Plots (p. 4057).

47.3.4.3.2. Generating Histogram Data

A histogram plot is a bar chart of data. It is a representation of a frequency of distribution by means of rectangles of widths representing class intervals and with areas proportional to the corresponding frequencies. The web interface allows you to generate histogram data.

Results → Plot → Histogram

To generate histogram data, make the necessary changes (to such properties as **Auto Range**, **Number of Divisions**, and whether or not to include **All Zones**). Click **Print** to see the new histogram plot details in a console panel, or click **Plot** to see a graphical representation of the data.

For more information, see Histograms (p. 4058).

47.3.4.3.3. Generating Cumulative Data

A cumulative plot allows you to analyze the development of force, moments, or coefficients for your model in the any desired location or direction.

Results → Plot → Cumulative

To generate cumulative data, make the necessary changes (such as setting the cumulative force/moment or their coefficients under **Option**, one or more **Zones**, **Split Direction**, **Number of Divisions**, and the **Force Direction**) and click **Plot** to see the new cumulative plot details.

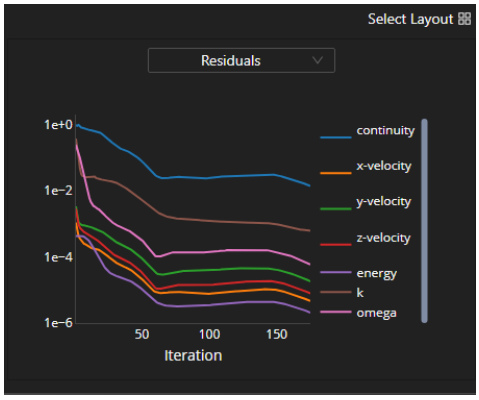
For more information, see Cumulative Force, Moment, and Coefficients Plots (p. 4094).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

47.3.4.3.4. Viewing Residual Plots

Residual history is displayed using an XY plot. The abscissa of the plot corresponds to the number of iterations and the ordinate corresponds to the log-scaled residual values.

Figure 47.42: Residual Plot

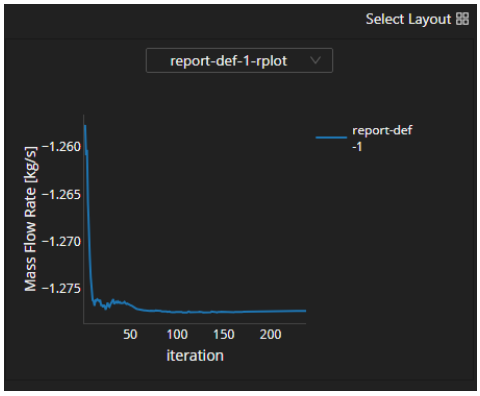


You can drill down into the specifics of the plot data interactively. See Analyzing Data in Plots (p. 4485) for details.

47.3.4.3.5. Viewing Report Definition Plots

For simulations that have report definitions, you can see the corresponding plot in the Plot View.

Figure 47.43: A Report Definition Plot



Note:

Click the title of the plot to display other report definition plots that are also be available

---

You can drill down into the specifics of the plot data interactively. See Analyzing Data in Plots (p. 4485) for details.

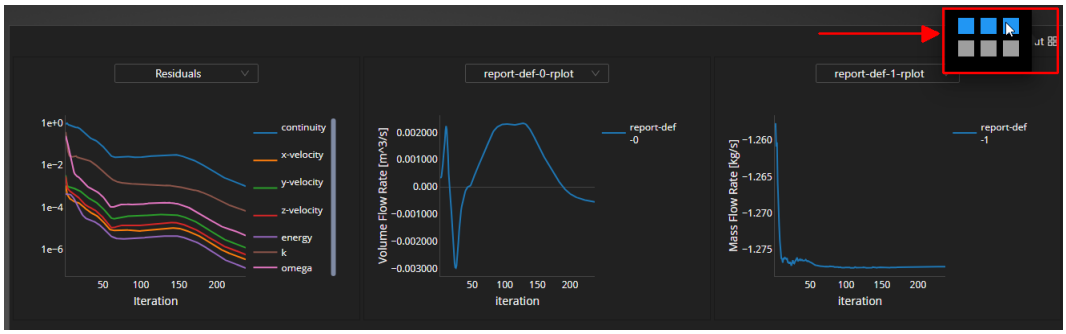
For more information on working with reports in Fluent, see Report Files and Report Plots (p. 4123).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

47.3.4.3.6. Displaying Multiple Plots

If your simulation has several plots defined, you can use the Plots View to view them individually (using the title's drop-down menu) or as a collective set of tiled plots. Use the drop-down menu in the upper right hand corner to see possible layout options.

Figure 47.44: Displaying Multiple Plots (1x3)

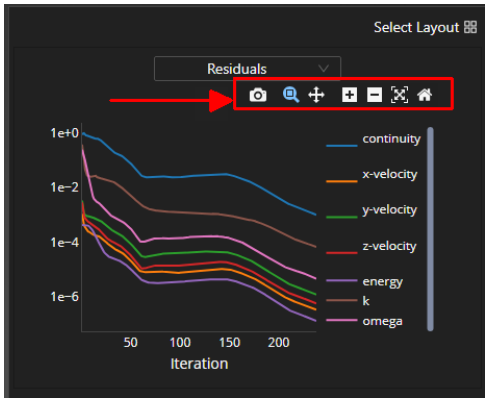


The order of the plot display can be customized to your liking by using the drop-down menu to pick and choose the plots in the corresponding window.

47.3.4.3.7. Analyzing Data in Plots

Hovering the mouse over the plots reveals specific data values and exposes some assorted controls for plotting:

Figure 47.45: Analyzing Data for a Plot



- **Download plot as png**: saves the plot as a PNG file.
- **Zoom**: use the mouse to draw a box that the view will zoom into.
- **Pan**: use the mouse to pan across the XY plot.
- **Zoom in**: zoom into the current view.
- **Zoom out**: zoom out of the current view.
- **Auto-scale**: resets the view to the original size and scale.
- **Reset Axes**: resets the X and Y axes.

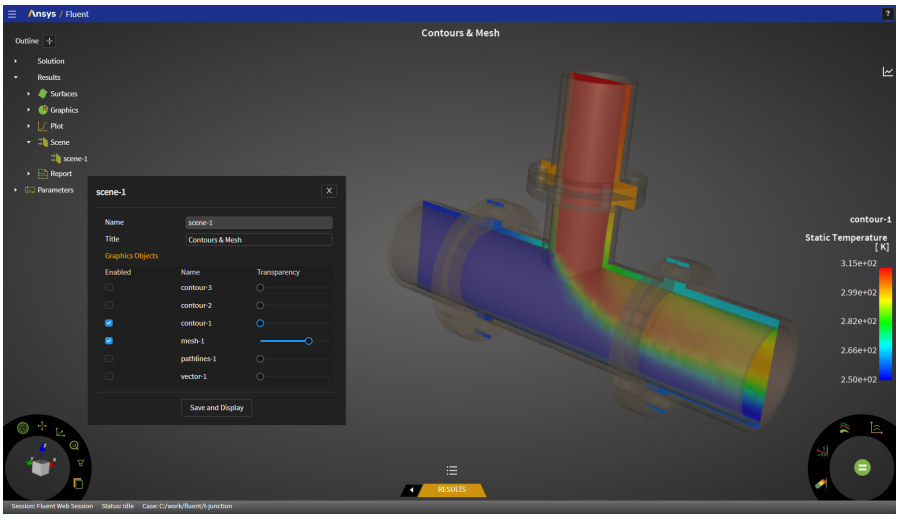
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

47.3.4.4. Creating and Displaying Scenes and Your Simulation Results

Results → Scene New

For the new scene object, open the property editor to enable/disable the display of multiple graphics plots within a single window. In the editor, you can set up different configurations of graphics results and plots from your CFD simulation. Scenes can be comprised of mesh displays, contour, vector, and pathline plots, and so forth, and the transparencies for each can be customized.

Figure 47.46: Building a Scene



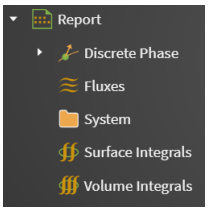
For more detailed information about scenes, see Displaying a Scene (p. 3934).

47.3.4.5. Creating and Displaying Reports for Your Simulation Results

You can set up reports for your CFD simulation using the Outline View:

Results → Reports

Figure 47.47: Reports



Reports can be compiled for fluxes, forces, projected areas, surface and volume integrals, among others.

Discrete Phase

allows you to report on one of the following report types:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Histogram

allows you to set the following:

Histogram Options

displays options for setting up your histogram, such as whether or not to include such properties as **Auto Range**, **Correlation**, **Cumulative Curve**, **Diameter Statistics**, **Histogram Mode**, **Percentage**, **Logarithmic**, and **Weighting**.

Histogram Parameters

displays options for setting up the number of **Divisions** for your histogram

For more information, see Trajectory Sample Histograms Dialog Box (p. 5333).

Sample Trajectories

selecting this item opens the property editor for setting up discrete phase sample trajectories output files. For more information, see Sample Trajectories Dialog Box (p. 5331).

Fluxes

selecting this item opens the property editor for fluxes where you can set up your reports for quantities such as **Mass Flow**, **Heat Transfer**, **Radiation Heat Transfer**, and **Viscous Work**. For more information, see Flux Reports Dialog Box (p. 5320).

System

selecting this item opens the property editor where you can print out reports for process, system, GPGPU, and time statistics for your simulation.

Surface Integrals

selecting this item opens the property editor for surface integrals where you can set up your reports for quantities such as **Axes**, **Area Weighted Average**, **Vector Based Flux**, **Vector Flux**, **Vector Weighted Average**, **Facet Average**, **Facet Minimum**, **Facet Maximum**, **Flow Rate**, **Integral**, **Mass Flow Rate**, **Mass Weighted Average**, **Standard Deviation**, **Sum**, **Uniformity Index Area Weighted Average**, **Uniformity Index Mass Weighted Average**, **Vertex Average**, **Vertex Minimum**, **Vertex Maximum**, and **Volume Flow Rate**. For details, see Surface Integrals Dialog Box (p. 5325).

Volume Integrals

selecting this item opens the property editor for volume integrals where you can set up your reports for quantities such as **Mass Average**, **Mass Integral**, **Mass**, **Sum**, **Minimum**, **Maximum**, **Volume**, **Volume Average**, and **Volume Integral**. For details, see Volume Integrals Dialog Box (p. 5329).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For more detailed information about reports, see Reporting Alphanumeric Data (p. 4097).

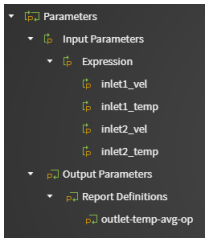
Note:

In the web interface, Fluent computes the report definitions when you click **OK** in the corresponding panel. You can generate a written report by enabling the **Write to File** option and clicking **OK**; Fluent will then prompt you for a file name and location in order to write the report.

47.3.5. Working With Parameters

The Parameters section of the Outline view is similar to that in Fluent, providing easy access to any of your simulation's defined input and output parameters.

Figure 47.48: Example of Input and Output Parameters



Once defined, you can quickly access input and output parameters here to view and/or change their values accordingly.

47.4. Setting Up Your Environment (System Administrators)

This section is designed to help system administrators deploy the Fluent Job Service in on-premise environments.

About the Fluent Job Service

The Fluent job service is a RESTful service developed as a companion to the Fluent web server feature. It makes it easier to access Ansys Fluent's web interface for remote HPC jobs. When a Fluent job starts on an HPC cluster, it registers connection information (such as the server URL and a token) with the job service. The customer can access the server URL from the job service web page by providing the (secret) job token (s)he specifies when submitting the job. It also allows the user to collaborate with other co-workers by sharing the token.

The job service is implemented as a HTTP service using `Node.js/Express.js` and is built as Winx64 and Linux executables. A single instance of the job service should be enough for moderate workloads (a few hundred concurrent users) but if needed, multiple instances can be run in parallel.

Important:

For security reasons, you should not share your token or network path with anyone.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Licensing

The job service is available for free.

Packaging

The job service executable is located in the Fluent installation directory at:

- **Windows 64:**

```
v242\fluent\jobservice\winx64\jobservice.exe
```

- **Linux 64:**

```
v242/fluent/jobservice/linx64/jobservice
```

Hardware Requirements

The service can be run on a single CPU, 4GB RAM VM (Winx64 or Linux).

Deployment

The following steps describe the actions needed to deploy the service:

1. Identify the machine to run the service.
2. Put the executable somewhere on the machine (there is no installation step).
3. Copy `config.js` to `v242/fluent/fluent.24.2.0/web/ui.`
4. Run the executable. If you are running the job service in any directory other than its parent directory, then you need to provide the absolute path to `v242/fluent/fluent.24.2.0/web/ui` as the command line option `-u` (see below).

   The following usage instructions apply to the job service:

```
Usage: jobservice.exe [options]
Options:
  -s, --storage       Storage type     [choices: "file", "db"] [default: "file"]
  -p, --port          Port number                                [default: 4000]
  -d, --dbpath        Path to database json file
  -c, --protocol      Protocol      [choices: "http", "https"] [default: "http"]
  -u, --uipath        Path to ui directory containing index.html and other UI
                      files               [default: "../../fluent24.2.0/web/ui"]
      --helppath      Path to help directory
                                          [default: "../../../commonfiles/help"]
  -l, --certlocation  Path to directory containing ssl certificate (named
                      jobservice.crt) and key (named jobservice.key).
                       [default: "Script directory [C:\snapshot\s\job-service]"]
  -h, --help          Show help                                        [boolean]
```

5. On Linux, it is possible to run the service using `system` to ensure that the service starts automatically and keeps running in event of a problem.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

47.4.1. Supporting HTTPS

For HTTPS support, the Fluent web server requires the certificate files to be provided in one of the following two ways.

1. A custom certificate directory.

   The environment variable `FLUENT_WEBSERVER_CERTIFICATE_ROOT` should be set to the directory path that contains the certificate files.

   At this location, Fluent will look for the following files.

   a. `webserver.crt` (the SSL certificate file)

   b. `webserver.key` (the corresponding private key file)

   c. `dh.pem` (the DH parameter file)

   Note:

   The web server will not start if this environment variable is set, but the above files are absent.

2. A default directory.

   If the environment variable `FLUENT_WEBSERVER_CERTIFICATE_ROOT` is not set, the web server checks the `/web/certificate/` directory in the Fluent installation directory defined by the `FLUENT_PROD_DIR` environment variable.

   For example, `C:\Program Files\ANSYS Inc\v242\fluent\fluent24.2\web\certificate`

   If the certificate files are not present, then the web server is started in normal HTTP mode.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 48: Design Analysis and Optimization

In this chapter, the setup and use of the adjoint solver and the mesh morpher/optimizer in Ansys Fluent are described. Both tools offer shape optimization capabilities, but the scope is complementary. The fundamental difference in the technology is that the adjoint solver uses gradient-based optimization by calculating derivatives, whereas the mesh morpher/optimizer uses a strategy that is not based on gradients to decide design updates.

The tools have a common approach to change a design by morphing the computational mesh. This has the benefit that you need not return to the original geometry in order to update the design for the purposes of the optimization.

The two key considerations when deciding whether to use the adjoint solver or the mesh morpher/optimizer are as follows:

- Is the flow physics / observable of interest supported by the adjoint solver? (For details about adjoint capabilities, see the following sections: Basic Assumptions and Consistency Checks (p. 4510) and General Observables (p. 4493)) If not, then the mesh morpher/optimizer is the better option, since there are very few limitations in the supported physics and choice of quantity of interest with this approach. It is important to verify that the quantity of interest can be specified as an adjoint solver observable.
- Is the dimension of the design space high? Detailed shape optimization of 3D geometries can involve design spaces with large dimensions (thousands of degrees of freedom or more). The adjoint solver is very effective in this scenario, whereas the use of the mesh morpher/optimizer becomes computationally prohibitive when the number of degrees of freedom exceeds more than, say, 10–20.

48.1. The Adjoint Solver
48.2. Using the Adjoint Solver
48.3. Geometry Parameterization and Exploration
48.4. The Mesh Morpher/Optimizer
48.5. Using the Mesh Morpher/Optimizer

48.1. The Adjoint Solver

An adjoint solver is a specialized tool that extends the scope of the analysis provided by a conventional flow solver by providing detailed sensitivity data for the performance of a fluid system.

In order to perform a simulation using the Ansys Fluent standard flow solvers, a user supplies system geometry in the form of a computational mesh, specifies material properties and physics models, and configures boundary conditions of various types. The conventional flow solver, once converged, provides a detailed data set that describes the flow state governed by the flow physics that are being modeled. Various postprocessing steps can be taken to assess the performance of the system.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If a change is made to any of the data that defines the problem, then the results of the calculation can change. The degree to which the solution changes depends on how sensitive the flow is to the particular parameter that is being adjusted. Indeed, the derivative of the solution data with respect to that parameter quantifies this sensitivity to first order. Determining these derivatives is the domain of *sensitivity analysis*.

There is a large collection of derivative data that can be computed for a fluid system, given the extensive set of input data that is required, and the extensive flow data that is produced. The matrix of derivatives of output data with respect to input data can be vast. Depending upon the goal of the analysis only a portion of this derivative data may be needed for engineering analysis and decision-making.

The adjoint solver accomplishes the remarkable feat of calculating the derivative of a single engineering observation with respect to a very large number of input parameters simultaneously via a *single computation*. The engineering observation could be a measure of the system performance, such as the lift or drag on an airfoil, or the total pressure drop through a system. Most importantly, the derivatives with respect to the geometric shape of the system are found.

Understanding such sensitivities in a fluid system can provide extremely valuable engineering insight. A system that is highly sensitive may exhibit strong variability in performance due to small variations in manufacturing or variations in the environment in which it is operating. Alternatively, high-sensitivity may be leveraged for fluid control, with a small actuator being able to induce strong variations in behavior. Yet another perspective is that sensitivity of a performance measure implies that the device in question is not fully optimized and there is still room for improvement—assuming that constraints do not preclude further gains.

The sensitivities of a fluid system provided by an adjoint solver satisfy a central need in gradient-based shape and turbulence model optimization. This makes an adjoint solver a unique and powerful engineering tool for design optimization.

Adjoint data can also play a role in improving solver numerics. Regions of high sensitivity are indicative of areas in the flow where discretization errors can potentially have a strong effect. This information can be used to guide how best to refine a mesh to improve flow solution accuracy.

The process of computing an adjoint solution resembles that for a standard flow calculation in many respects. The adjoint solver solution advancement method is specified, residual monitors configured, and the solver is initialized and run through a sequence of iterations to convergence. One notable difference is that a scalar-valued observation is selected as being of interest prior to starting the adjoint calculation.

Once the adjoint solution is converged the derivative of the observable with respect to turbulence model parameters, or with respect to the position of each and every point on the surface of the geometry is made available, and the sensitivity of the observation to specific settings can be found. This remarkable feature of adjoint solutions has been known for hundreds of years, but only in the last few decades has the significance for computational physics analysis been recognized widely.

The power of this methodology is highlighted when an alternative for assembling the same information is considered. Imagine a sequence of flow calculations in which each point on an airfoil surface is moved in turn a set small distance in the surface-normal direction, and the flow and drag recomputed. If there are N points on the surface, then N flow calculations are required to build the data set. Considering that the same data is provided by a *single* adjoint computation, the adjoint approach has an enormous advantage. Remember that for even modest 3D flow computations there may be many thousands of coordinates or more on a surface.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Once the adjoint is computed it can be used to guide intelligent design modifications to a system. After all, the adjoint sensitivity data provides a map across the entire surface of the geometry of the effect of moving the surface. Design modifications can be most effective if made in regions of high sensitivity since small changes will have a large effect upon the engineering quantity of interest. This principle of making changes to a system in proportion to the local sensitivity is the foundation for the simple gradient algorithm for design optimization.

Once a candidate change in shape or other boundary condition has been selected, the effect of that change can be estimated using the computed derivative data. This amounts to a first order extrapolation using a Taylor series expansion around the baseline flow state. Clearly if a modification is chosen that is large enough that nonlinear effects become important then the accuracy of the predicted change cannot be guaranteed.

48.1.1. General Observables

Several observables are available and serve as the foundation for specifying the quantity that is of interest for the computation. Basic quantities such as forces and moments can be defined and each is given a name. The following types of observables are available:

- expression: Fluent expressions (see Defining Observables Using Expressions (p. 4520)).
- Force: the aerodynamic force in a specified direction on one or more walls.
- Moment of force: the aerodynamic moment about a specified moment center and moment axis. The integration is made over a specified collection of wall zones. Note that in 2D, the moment axis is normal to the plane of the flow.
- Swirl: the moment of the mass flow (with velocity $\underline{u}$) relative to an axis defined by a point, $\underline{r_c}$, and a direction $\underline{d}$. The two options are:
  - Swirl integral

$$\int_V \rho\left(\underline{r}\times\underline{u}\right)\cdot\underline{d}\,dV \tag{48.1}$$

    where V denotes the volume over which the integration is made, and $\underline{r}$ denotes the relative position to the point $\underline{r_c}$.

  - Normalized swirl integral

$$\frac{\int_V \rho\left(\underline{r}\times\underline{u}\right)\cdot\underline{d}\,dV}{\int_V \rho\left(\underline{r}\times\left(\underline{d}\times\underline{r}\right)\right)\cdot\underline{d}\,dV} \tag{48.2}$$

    Note:

    These integrals can be used to construct quantities such as tumble ratio that are of significance for internal combustion engine analysis.

- pressure drop between an inlet (or group of inlets) and an outlet (or group of outlets)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The total pressure is computed by area-weighted averaging of the total pressure values on selected surfaces.

- fixed value: a simple fixed value can be specified and used in the assembly of the observables. This is convenient when some scaling or normalization of the observable is desired. It must be noted that this value is treated strictly as a constant for the purposes of any derivative calculations.
- surface integral: a variety of surface integrals can be constructed for a specified field variable on a set of user-selected surfaces (including zone surfaces, isosurfaces, and clipped surfaces).
  - Facet sum: a simple sum of the field value on each facet of the computational mesh.

$$\sum_f \varphi_f \tag{48.3}$$

  - Facet average: the facet sum is divided by the total number of facets, N_f.

$$\overline{\varphi}=\frac{\sum_f \varphi_f}{N_f} \tag{48.4}$$

  - Facet variance: the sum of the squares of the deviations from the facet average divided by the total number of facets.

$$\frac{\sum_f (\varphi_f-\overline{\varphi})^2}{N_f} \tag{48.5}$$

  - Integral: the sum of the field value on each face multiplied by the face area.

$$\sum_f \left|\underline{A}_f\right|\varphi_f \tag{48.6}$$

  - Area-weighted average: the integral divided by the total face area.

$$\frac{\sum_f \left|\underline{A}_f\right|\varphi_f}{\sum_f \left|\underline{A}_f\right|} \tag{48.7}$$

  - Area-weighted variance: the sum of the squares of the deviations from the area-weighted average, divided by the total face area.

$$\frac{\sum_f \left|\underline{A}_f\right|(\varphi_f-\overline{\varphi})^2}{\sum_f \left|\underline{A}_f\right|} \tag{48.8}$$

  - Mass-weighted integral: the sum of the field value on each face, weighted by the magnitude of the local mass flow through the face.

$$\sum_f \left|\rho \underline{u}_f \cdot \underline{A}_f\right|\varphi_f \tag{48.9}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– Mass-weighted average integral: the mass-weighted integral divided by the sum of the magnitudes of the local mass flow rates through the faces on which the integral is defined.

$$\frac{\sum_f |\rho \underline{u}_f \cdot \underline{A}_f| \varphi_f}{\sum_f |\rho \underline{u}_f \cdot \underline{A}_f|} \tag{48.10}$$

– Mass-weighted variance: the mass-weighted integral of the square of the deviation from the mass-weighted average, divided by the sum of the magnitudes of the local mass flow rates through the faces on which the integral is defined.

$$\frac{\sum_f |\rho \underline{u}_f \cdot \underline{A}_f| \left(\varphi_f - \overline{\varphi}\right)^2}{\sum_f |\rho \underline{u}_f \cdot \underline{A}_f|} \tag{48.11}$$

– Flow-rate weighted: the rate at which the field is convected through the defined surfaces.

$$\sum_f \left(\rho \underline{u}_f \cdot \underline{A}_f\right) \varphi_f \tag{48.12}$$

– The above surface integrals can be computed using the following field variables:

→ Pressure

→ Total pressure

→ Mass flow per unit area

→ Temperature (when adjoint energy is enabled)

→ Total temperature (when adjoint energy is enabled)

→ Heat flux (when adjoint energy is enabled)

→ turbulent kinetic energy k (when adjoint turbulence is enabled)

→ specific dissipation rate ω (when adjoint turbulence is enabled)

→ user-defined expressions (see Defining Observables Using Expressions (p. 4520))

• volume integral: you can compute a variety of volume integrals of a chosen field variable.

– Volume: computes the total volume of the selected zones as a summation of the individual cell volumes.

$$\int dV = \sum_{i=1}^{n} |V_i| \tag{48.13}$$

– Sum: computes the sum of a chosen field variable over the selected zones.

$$\sum_{i=1}^{n} |\varphi_i| \tag{48.14}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– Volume Integral: is calculated by summing the product of cell volume and the selected field variable over the selected zones.

$$\int \varphi dV = \sum_{i=1}^{n} \varphi_i |V_i| \tag{48.15}$$

– Volume-Weighted Average: is computed by dividing the volume integral of a selected field variable (Equation 48.15 (p. 4496)) by the total volume of the selected zones.

$$\int \varphi dV = \frac{1}{V} \sum_{i=1}^{n} \varphi_i |V_i| \tag{48.16}$$

– Volume Variance: is the volume-weighted integral of the square of the deviation from the volume-weighted average (Equation 48.16 (p. 4496)), divided by the total volume of the selected zone.

$$\frac{\sum_{i=1}^{n} |V_i| \left(\varphi_i - \overline{\varphi}\right)^2}{\sum_{i=1}^{n} |V_i|} \tag{48.17}$$

– Mass Integral: is computed by summing the product of density, cell volume, and the chosen field variable.

$$\int \rho \varphi dV = \sum_{i=1}^{n} \rho_i \varphi_i |V_i| \tag{48.18}$$

– Mass: is computed by summing the product of cell density and cell volume.

$$\int \rho dV = \sum_{i=1}^{n} \rho_i |V_i| \tag{48.19}$$

– Mass-Weighted Average: is computed by dividing the Mass Integral of the chosen field variable (Equation 48.18 (p. 4496)) by the total Mass (Equation 48.19 (p. 4496)).

$$\frac{\int \rho \varphi dV = \sum_{i=1}^{n} \rho_i \varphi_i |V_i|}{\int \rho dV = \sum_{i=1}^{n} \rho_i |V_i|} \tag{48.20}$$

– Mass Variance: is computed as the mass-weighted summation of the square of the deviation of the chosen field variable from its mass-weighted average (Equation 48.20 (p. 4496)) divided by the total mass (Equation 48.19 (p. 4496)).

$$\frac{\sum_{i=1}^{n} \rho_i |V_i| \left(\varphi_i - \overline{\varphi}\right)^2}{\sum_{i=1}^{n} \rho_i |V_i|} \tag{48.21}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– Target-volume-integral: is the volume integral of the square of the deviation of the current field variable (ϕ for a scalar quantity or $\omega * d$ for a vector quantity) and the target field variable UDM_i.

Note:

Non-constant target field variable data must be written into user-defined memory as outlined in Defining Observables for Turbulence Model Optimization (p. 4525) . For a constant target quantity, it is not necessary to write to user-defined memory.

For targeting a scalar quantity the equation is as follows:

$$J = \int (\phi - UDM_i)^2 d\Omega \tag{48.22}$$

For targeting a vector quantity the equation is as follows:

$$J = \int (\omega * d - UDM_i)^2 d\Omega \tag{48.23}$$

The volume integrals listed above can be computed for the following field variables:

– Pressure

– Total Pressure

– Temperature (when adjoint energy is enabled)

– Total temperature (when adjoint energy is enabled)

– Velocity

– Velocity Magnitude

– Vorticity

– Vorticity Magnitude

– Turbulence Production: $\frac{\partial u_i}{\partial x_j}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)$

– turbulent kinetic energy k (when adjoint turbulence is enabled)

– specific dissipation rate ω (when adjoint turbulence is enabled)

– user-defined expressions (see Defining Observables Using Expressions (p. 4520))

48.1.2. General Operations

Several operations are available to combine observables in various ways, if needed, to make a wide variety of compound observables. The following operations are available:

- ratio of two quantities
- product of two quantities

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- linear combination of N quantities $q_0 \ldots q_{N-1}$, with constant coefficients $a_0 \ldots a_{N-1}$ raised to various powers $v_0 \ldots v_{N-1}$, plus a constant offset c. The following combination types are available:

  – sum:

$$c + \sum_{i=0}^{N-1} a_i q_i^{v_i} \tag{48.24}$$

  – product:

$$c + \prod_{i=0}^{N-1} a_i q_i^{v_i} \tag{48.25}$$

  – average:

$$c + \left(aq^v\right)_{\text{Avg}} \tag{48.26}$$

  where

$$\left(aq^v\right)_{\text{Avg}} = \frac{\sum_{i=0}^{N-1} a_i q_i^{v_i}}{N} \tag{48.27}$$

  – mean deviation:

$$c + \frac{\sum_{i=0}^{N-1} \left| a_i q_i^{v_i} - \left(aq^v\right)_{\text{Avg}} \right|}{N} \tag{48.28}$$

  – mean variance:

$$c + \frac{\sum_{i=0}^{N-1} \left(a_i q_i^{v_i} - \left(aq^v\right)_{\text{Avg}} \right)^2}{N} \tag{48.29}$$

  – minimum:

$$c + \min\left(a_i q_i^{v_i} \right) \tag{48.30}$$

  – maximum:

$$c + \max\left(a_i q_i^{v_i} \right) \tag{48.31}$$

- arithmetic average of N quantities $q_0 \ldots q_{N-1}$

$$\overline{q} = \frac{\sum_{i=0}^{N-1} q_i}{N} \tag{48.32}$$

- mean variance of N quantities

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\frac{\sum_{i=0}^{N-1}(q_i-\bar{q})^2}{N} \tag{48.33}$$

- unary operation (sin, cosine, and so on) on observable value q :
  - $1/q$
  - $|q|$
  - $\sqrt{q}$
  - $\sin q$
  - $\cos q$
  - $\tan q$
  - $\sin^{-1} q$
  - $\cos^{-1} q$
  - $\tan^{-1} q$
  - $\ln q$
  - $\log 10\ q$

48.1.3. Discrete Versus Continuous Adjoint Solver

When an adjoint solver is developed there is a key design decision that is made regarding the implementation, namely whether to use a *continuous* or a *discrete* approach. Both approaches are intended to compute the same sensitivity data—however, the approaches are remarkably different.

While a user of the adjoint solver may experience no particular difference in workflow for the two separate solver types, it is important to be aware that there are two primary classes of adjoint solver and the approach chosen can have implications for the accuracy of the results.

A *continuous adjoint solver* relies heavily on mathematical properties of the partial-differential equations that define the physics of the problem. In this case those equations are the Navier-Stokes equations. With this approach an adjoint partial differential equation set is formulated explicitly and is accompanied by adjoint boundary conditions that are also derived mathematically. Only after this derivation is complete can the adjoint partial differential equations be discretized and solved, often with extensive re-use of existing solver machinery. This class of solver was implemented by Ansys Fluent as a research effort.

Such a solver has the benefit that it is decoupled largely from the original flow solver. They share only the fact that they are based on the Navier-Stokes equations. The process of discretizing and solving the partial differential equations in each case could in principle be very different indeed. While this flexibility may be appealing it can also be the downfall of the approach. Inconsistencies in modeling, discretization and solution approaches can pollute the sensitivity information significantly, es-

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

pecially for problems with wall functions and complex engineering configurations such as those of interest to Ansys Fluent users.

The continuous adjoint approach can be effective for some classes of problems. However, until there is a significant advance in handling some key challenges, Ansys, Inc. has concluded that it is an unsuitable approach for meeting the needs of our broad client base for the classes of problems of interest to them.

A *discrete adjoint solver* is based not on the form of the partial differential equations governing the flow, but the particular discretized form of the equations used in the flow solver itself. The sensitivity of the discretized equations forms the basis for the sensitivity calculation. In this approach the adjoint solver is much more tightly tied to the specific implementation of the original flow solver. This has been observed to yield sensitivity data that provides valuable engineering guidance for the classes of problem of interest to Ansys Fluent users, including problems with wall functions.

For the above reasons, the discrete adjoint approach has been adopted for the Ansys Fluent adjoint solver.

48.1.4. Discrete Adjoint Solver Overview

As discussed in the previous section, a discrete adjoint approach to solving the adjoint problem has been adopted, since it is believed to provide the most useful engineering sensitivity data for the classes of problems of interest to Ansys clients.

An adjoint method can be used to compute the derivative of an observation of interest for the fluid system with respect to all the user-specified parameters, *with any changes that arise in the flow variables themselves eliminated*. There are three key ingredients to consider when developing the method:

1. All of the user-specified inputs:

   - All values set by a user in the boundary condition panels for each boundary in the problem.
   - The computational mesh. More specifically the locations of the nodes of the mesh and how they define the edges, faces, and ultimately the cells used in the finite-volume computation. This includes both interior as well as boundary nodes.
   - Material properties.
   - Model parameters such as model coefficients for turbulence models.

   Note that the settings that define the problem are being distinguished from settings that define how the solution advancement is to be performed to converge the problem. Only the former are of interest here. For the sake of clarity, let us denote the vector of all of the values in the list by $\underline{c}$. These are considered to be the control variables for the problem, that is, the variables that a user can set explicitly that affect the solution.

   It is worth noting that the topological definition of the mesh is fixed—it is not considered to be a control variable here. The effect of collapsing cells or remeshing on the flow solution is not addressable without further development in view of the discrete changes that are implied. This topic is beyond the scope of the current document.

2. The governing equations for the fluid system:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The main effort in a flow computation is in the determination of the flow state, namely the velocity, pressure, density and possibly other fluid-related variables. For a cell-centered finite-volume scheme, the flow state is defined at the cell centroids by a vector of real values. In the simplest case these values are the pressure and flow velocity components. Let the vector of the variables in the v^{th} cell be denoted here by $\underline{q}^v$.

At convergence the flow variables satisfy

$$\mathcal{R}_i^\mu\left(\underline{q}^0,\underline{q}^1,\ldots,\underline{q}^{M-1};\underline{c}\right)=0,\ \mu=0,\ldots,M-1,\ i=0,\ldots,L-1 \tag{48.34}$$

where M is the number of cells in the problem, and there are L conditions on each cell. This expression is a compact way of denoting conservation of mass and momentum and other constraints.

3. The engineering observation of interest:

   Let

$$\mathcal{J}\left(\underline{q}^0,\underline{q}^1,\ldots,\underline{q}^{M-1};\underline{c}\right) \tag{48.35}$$

   denote a scalar of interest that depends both on the flow state and perhaps directly on the control variables. It is assumed that the observable is differentiable with respect to both the flow and the controls. The inclusion of the control variables here is essential since in many cases the mesh geometry is included directly in the evaluation of the observable. For example, the evaluation of the force on a boundary involves the wall normal and face areas, which change when mesh nodes are moved.

The goal is to determine the sensitivity of the observation with respect to the user-specified control variables. What makes defining this relationship more challenging is the fact that changing the user inputs changes the flow, which indirectly changes the engineering observation. The adjoint method has a specific role in managing this chain of influences by providing a mechanism for eliminating the specific changes that happen in the flow whenever the inputs change.

If a variation δc_j is introduced into the control variables then a linearization of the governing equations (Equation 48.34 (p. 4501)) shows that the variations in the flow state δq_j^v must satisfy

$$\frac{\partial R_i^\mu}{\partial q_j^v}\delta q_j^v=-\frac{\partial R_i^\mu}{\partial c_j}\Big|_q\delta c_j,\ \mu=0,\ldots,M-1,\quad i=0,\ldots,L-1. \tag{48.36}$$

where there is an implied summation over j and v, and $\Big|_q$ denotes that the flow solution is held constant while the derivative is taken.

Meanwhile, if both the control variables and the flow state change, then the observation will change:

$$\delta\mathcal{J}=\frac{\partial\mathcal{J}}{\partial q_j^v}\delta q_j^v+\frac{\partial\mathcal{J}}{\partial c_j}\Big|_q\delta c_j. \tag{48.37}$$

The particular way in which the flow responds to the changes in the control variables can be computed using (Equation 48.36 (p. 4501)) only after specific changes, δc_j, have been chosen. It is prohibitive to consider solving (Equation 48.36 (p. 4501)) for more than a handful of prescribed changes δc_j because of the excessive computing time that would be needed. However, when redesigning the shape of parts of a system there may be pressure to explore a large number of candidate modifications. This conflict is reconciled by eliminating the variations of the flow solution from the expression (Equa-

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

tion 48.37 (p. 4501)) and producing an explicit relationship between changes in the control variables and the observation of interest.

This is accomplished by taking a weighted linear combination of the linearized governing equations (Equation 48.36 (p. 4501)) in a very particular way. A set of adjoint variables $\tilde{q}^{\mu}$ is introduced with a one-to-one correspondence with the governing equations (Equation 48.34 (p. 4501)). This results in a relationship

$$\left[\tilde{q}_i^{\mu}\frac{\partial R_i^{\mu}}{\partial q_j^{\nu}}\right]\delta q_j^{\nu} = -\tilde{q}_i^{\mu}\frac{\partial R_i^{\mu}}{\partial c_j}\bigg|_q \delta c_j. \tag{48.38}$$

The term in square brackets on the left is now matched to the coefficient for the variation in the flow in (Equation 48.37 (p. 4501)) in order to define values for the adjoint variables:

$$\frac{\partial R_i^{\mu}}{\partial q_j^{\nu}}\tilde{q}_i^{\mu} = \frac{\partial \mathcal{J}}{\partial q_j^{\nu}}. \tag{48.39}$$

These are the discrete adjoint equations, and the solution of this system that is the primary goal for the adjoint solver. It is important to recognize that these equations have not been derived. They are defined in this way with a specific goal in mind—the elimination from (Equation 48.37 (p. 4501)) of the perturbations to the flow field as shown below:

$$\begin{aligned}\delta\mathcal{J} &= \frac{\partial \mathcal{J}}{\partial q_j^{\nu}}\delta q_j^{\nu} + \frac{\partial \mathcal{J}}{\partial c_j}\bigg|_q \delta c_j \\ &= \tilde{q}_i^{\mu}\frac{\partial R_i^{\mu}}{\partial q_j^{\nu}}\delta q_j^{\nu} + \frac{\partial \mathcal{J}}{\partial c_j}\bigg|_q \delta c_j \\ &= \left\{\frac{\partial \mathcal{J}}{\partial c_j}\bigg|_q - \tilde{q}_i^{\mu}\frac{\partial R_i^{\mu}}{\partial c_j}\bigg|_q\right\}\delta c_j\end{aligned} \tag{48.40}$$

Note the use of Equation 48.38 (p. 4502) and Equation 48.39 (p. 4502) in the derivation of Equation 48.40 (p. 4502) . The flow perturbation has now been eliminated from the expression, yielding a direct relation between the control variables and the observable of interest.

There are several important observations to be made about (Equation 48.39 (p. 4502)):

- The dimension of the problem to be solved is the same as the original flow problem, although the adjoint problem is linear.
- While the adjoint solution may be considered strictly as a vector of numeric values, experience with the continuous adjoint provides guidance on how the adjoint solution can be interpreted. The vector of weights associated with the components of the residual of the momentum equation in each cell is termed the *adjoint velocity*. The adjoint value associated with the residual of the continuity equation is termed the *adjoint pressure*.
- The right hand side is defined purely on the basis of the observable that is of interest.
- The matrix on the left hand side is the *transpose* of the Jacobian of the governing system of equations (Equation 48.34 (p. 4501)). This seemingly innocent transposition has a very dramatic impact on how the adjoint system is solved. This will be discussed below.
- The adjoint equations are defined by the current state of the flow, and the specific physics that is employed in the modeling. Each adjoint solution is specific to the flow state.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

At first glance it appears that solving the adjoint problem may be straightforward. After all it simply involves setting-up and solving a linear (albeit large) system of equations. In practice both steps can represent a significant challenge, especially when the problem is large.

The evaluation of the residuals of the flow equations is an integral part of the pre-existing Ansys Fluent flow solvers. However, it is necessary to compute the Jacobian of the system $\partial R_i^\mu / \partial q_j^\nu$ and then transpose it, or at the very least to be able to make a matrix-free transpose matrix-vector product with the adjoint solution. There are several technical approaches to accomplishing this task whose description goes beyond the scope of this document. Suffice to say that it is not trivial to encode this functionality, but that it has been done successfully here.

For the present implementation, a pre-conditioned iterative scheme, based on pseudo-time marching, is adopted to solve the adjoint system. The equations that define the advancement process can be written as

$$\left\{\frac{P_{i,j}^{\mu,\nu}}{CFL\Delta t}+L_{ij}^{\mu,\nu}\right\}\Delta\tilde{q}_i^\nu=\frac{\partial \mathcal{J}}{\partial q_j^\mu}-\frac{\partial R_i^\nu}{\partial q_j^\mu}\tilde{q}_i^\nu \tag{48.41}$$

where Δt is a local time step size based on the local flow conditions, CFL is a user-specified CFL number. The matrix $L_{ij}^{\mu,\nu}$ is a simplified form of the system Jacobian that is amenable to solution using the AMG linear solver that is the workhorse of the conventional flow solver. The preconditioning matrix $P_{i,j}^{\mu,\nu}$ is a diagonal matrix. Artificial compressibility is introduced on the adjoint continuity equation to aid in the relaxation of the adjoint pressure field.

The correction $\Delta\tilde{q}_i^\mu$ is under-relaxed and added to the adjoint solution $\tilde{q}_i^\mu$ and the process repeated until the right hand side is adequately small.

48.1.5. Adjoint Solver Stabilization

When applied to problems with large cell counts and complex geometry, adjoint solvers sometimes experience stability issues. These instabilities can be associated with small scale unsteadiness in the flow field and/or strong shear, and tend to be restricted to small and isolated regions of the flow domain. Despite the spatial localization of these instabilities, the linearity of the adjoint problem provides no intrinsic limit on their growth during solution advancement. Their presence, if not handled, can disrupt the entire adjoint calculation despite the problem occurring in sometimes just a few cells. A stabilized solution scheme will be required to obtain adjoint solutions for problems at high Reynolds number in which there is strong shear and/or complex geometry.

Two stabilization schemes are available in Fluent in order to overcome these stability difficulties when larger cases are being solved. You can choose to apply one of them at all times, or use a blended strategy where both schemes are used at different points in the solution. The stabilization schemes are designed to intervene only when the standard advancement scheme is experiencing instability.

Dissipation scheme

The dissipation scheme provides stabilization for the solution advancement of the adjoint solution by introducing nonlinear damping strategically into the calculation domain. The strategy is intended to provide minimal intervention in order to damp the growth of instabilities that lead to adjoint solution divergence. A marker is tracked, based on the state of the adjoint solution, and damping is applied directly to the adjoint solution in regions where the marker becomes large.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Unlike the residual minimization scheme, the dissipation scheme can affect the adjoint solution slightly. In general, the spatial order of the damping is chosen to be one order larger than the adjoint calculation order. This means that the formal order of accuracy of the adjoint solver is unaffected by the addition of the dissipation scheme. In practice, regions of intense dissipation can appear for some cases which may result in isolated spots and/or streaks in the solution on surfaces. However, the scale of these solution features is such that they are often easily smoothed during postprocessing to generate design changes.

Residual minimization scheme

The residual minimization scheme uses a GMRES-like scheme to approximate the solution by the vectors in a Krylov subspace with minimal residual. The overall adjoint residual is guaranteed to decrease. This is a unique benefit compared to the other available schemes. Note that this scheme is expensive with regard to memory. When it is enabled, each main iteration consists of multiple sub-iterations, where one sub-iteration is equivalent to one normal adjoint iteration.

Some additional computational overhead is associated with the stabilization schemes. In general, the dissipation scheme is the least resource-intensive.

For information about using the stabilization schemes, refer to Stabilization Strategies, Schemes, and Settings (p. 4533).

48.1.6. Solution-Based Adaption

An adjoint solution provides guidance on where best to adapt a computational mesh in order to resolve quantities of engineering interest.

Once the governing equations for the system (Equation 48.34 (p. 4501)) have been converged there remains a discretization error, ε_i^μ, such that

$$\mathcal{R}_i^\mu\left(\underline{q}^0,\underline{q}^1,\ldots,\underline{q}^{M-1};\underline{c}\right)=\varepsilon_i^\mu,\ \mu=0,\ldots,M-1,\quad i=0,\ldots,L-1. \tag{48.42}$$

While specific estimates for this discretization error may be tricky to define, it is often estimated to be $O\left(h^p\right)$, where h is the local grid size, and p is the order of the discretization scheme. That is, $p=1$ for a first-order scheme and $p=2$ for a second order scheme. Alternatively, ε_i^μ can be considered to be the residual associated with a solution that is not converged fully.

The correction to the flow field, δq_j^ν, that compensates for this inhomogeneity is given by

$$\frac{\partial R_i^\mu}{\partial q_j^\nu}\delta q_j^\nu=-\varepsilon_i^\mu \tag{48.43}$$

from which it follows quickly that

$$\delta\mathcal{J}=\frac{\partial\mathcal{J}}{\partial q_j^\nu}\delta q_j^\nu=-\varepsilon_i^\mu\tilde{q}_i^\mu. \tag{48.44}$$

This simple expression provides an estimate of the effect of the presence of ε_i^μ on the observation, $\mathcal{J}$.

The presence of discretization errors, or lack of convergence, on the engineering quantity of interest is assessed by weighting the inhomogeneous term by the local adjoint solution. It is clear that even

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

in regions of the domain where the residuals or discretization errors are small, an accompanying adjoint velocity or pressure that is large in magnitude implies that there may be a significant source of error in the observable. A finer mesh in regions where the adjoint is large will reduce the influence of discretization errors that may adversely affect the engineering result of interest. In practice, adapting cells which have large magnitude adjoint velocity and/or adjoint pressure will achieve this goal.

48.1.7. Using The Data To Improve A Design

Adjoint sensitivity data can be used to guide how to modify a system in order to improve the performance. The observable of interest can be made larger or smaller, depending upon the engineering goal.

A common strategy for deciding how to modify the system is based on the gradient algorithm. The underlying principle is quite simply that modifying a system in a manner to which it is most sensitive maximizes the effect of the change. The change to a control variable is made in proportion to the sensitivity of the value of interest with respect to that control variable.

Denote the sensitivity of the cost with respect to shape by

$$\delta \mathcal{J} = \frac{\partial \mathcal{J}}{\partial x_j^n} \delta x_j^n \tag{48.45}$$

where x_j^n is the j^{th} coordinate of the n^{th} node in the mesh. Here x_j^n is a notation for the subset of the control variables c_j for the system that correspond to mesh node positions. Then an adjustment

$$\delta x_j^n = \lambda \frac{\partial \mathcal{J}}{\partial x_j^n} \tag{48.46}$$

will provide the maximum adjustment to $\mathcal{J}$ for given L^2 norm of δx_j^n, where λ is an arbitrary scaling factor. Note that λ can be picked to be positive or negative depending upon whether $\mathcal{J}$ is to be increased or decreased respectively. This is essentially a statement of the method of steepest descent.

Furthermore, the change is estimated to first order to be

$$\delta \mathcal{J} = \lambda \frac{\partial \mathcal{J}}{\partial x_j^n} \frac{\partial \mathcal{J}}{\partial x_j^n}. \tag{48.47}$$

For a sufficiently small adjustment, the change to the observation will strictly have the same sign as the scaling factor λ, provided the gradient is not identically zero.

In regions where the sensitivity is high, small adjustments to the shape will have a large effect on the observable. This corresponds directly with the idea of engineering robustness. If a configuration shows high sensitivities, then the performance will likely be subject to large performance variations if there are manufacturing inconsistencies. For a robust design, the goal is to have the sensitivity be tolerably small.

In practical cases the field $\partial \mathcal{J} / \partial x_j^n$ can be noisy, especially for large, turbulent flow problems. If the noisy field is used directly to modify a boundary shape using (Equation 48.46 (p. 4505)), then the modified surface can have many inflections. This is not helpful for engineering design work. In the next section, the use of mesh morphing technology not only to smooth the sensitivity field, but also to provide smooth boundary and interior mesh deformation, is described.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

48.1.7.1. Smoothing and Mesh Morphing

As has been noted, for typical engineering problems, the shape sensitivity field can have smoothness properties that are not adequate to define a shape modification. Mesh morphing technology is used here for both two- and three-dimensional systems in a two-fold role. The first role is as a smoother for the surface sensitivity field. The second role is to provide smooth distortions not only of the boundary mesh, but also the interior mesh. This approach is very appealing since it functions for arbitrary mesh cell types.

A Cartesian or cylindrical region is defined such that it encompasses all or a part of the problem domain. Only the mesh nodes that fall within this region may move as a result of the morphing operation.

In general, an optimization problem may include both free-form minimization or maximization of the observable(s) and constraints that are localized in space. This requires approaches that can provide a deformation field that is well-behaved and consistent with manufacturability requirements, while still allowing locally sharper deformations where required to satisfy the imposed constraints. For the adjoint solver, Fluent provides three different morphing techniques: one based on polynomials, one that uses direct interpolation, and another that uses the radial basis function. Note that these methods are based on different design spaces, and so will produce different morphing results. In addition to the three morphing techniques, for the adjoint solver, Fluent offers two different constraint methods for handling design conditions: the standard constraint method and the enhanced constraint method.

For the standard constraint method, the free-form motion and design constraints are based only on the selected morphing method. When using the standard constraint method, the radial basis function morphing method will provide a balance between producing a good quality mesh, efficiently handling the design conditions, and requiring fewer user settings, therefore it is the recommended method. However, when there are no design conditions, including all design conditions and the fixed zone condition, the polynomials-based approach is recommended due to its good mesh quality and computational speed. Additionally, when there are excessive design conditions, excluding bounded-by-plane conditions and bounded-by-surfaces conditions, the direct interpolation method is recommended due to it's efficiency.

The enhanced constraint method is available for the polynomials and radial basis function morphing methods. With the enhanced constraint method, the specified morphing method will drive the free-form deformation based on the design objective and the region condition. The design condition is then fulfilled by secondary morphing techniques (outlined in Radial Basis Function (p. 4508)). This provides better balance for morphing in terms of efficiency and accuracy to handle design conditions, as well as fewer user settings and better mesh quality when compared to the standard constraint method. However, the enhanced constraint method has the following limitations:

- More memory is required on node process `0`.
- The target change for multiple objectives may not be strictly respected.
- Region conditions such as symmetric, invariance, and periodic morphing may not be strictly respected (see Defining Region Conditions (p. 4558)).
- Free parameters are not supported for the translation, scaling, rotation, rigid body, and prescribed profile design conditions when the prescribed option is disabled (see Defining Conditions for the Deformation (p. 4560)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For most applications, the enhanced constraint method is the recommended method to handle design conditions. Additionally, with the enhanced constraint method the polynomials morphing method will be faster than the radial basis function morphing method.

For more details about each morphing method, see the following sections.

48.1.7.1.1. Polynomials-Based Approach

Note:

For simplicity, the smoothing and morphing equations presented here are for the case of a 2D domain. The same approach is extended and used for 3D problems.

For the polynomials-based morphing approach, a local (u, v) coordinate system is defined, where $u \in [0,1]$ and $v \in [0,1]$. A regular array of $N_u \times N_v$ control points is then distributed in the control volume. The motion of the mesh at any point in the domain is determined by the movement of the control points. Bernstein polynomials and B-splines are used to map the control point motions to the computational mesh nodes.

The i^{th} Bernstein polynomial of degree ℓ is:

$$B_{i,\ell}(u) = \binom{\ell}{i} u^i (1-u)^{n-i} \tag{48.48}$$

The i^{th} B-spline of degree p is the non-periodic B-spline that uses a knot-vector, $\{t_i\}$, where:

$$t_i = \begin{cases} 0 & i \le p \\ (i-p)/(\ell-2p) & p < i < \ell - p \\ 1 & i \ge \ell - p \end{cases} \tag{48.49}$$

The value of the B-spline is defined recursively as:

$$\begin{aligned} S_{i,0}(t) &= \begin{cases} 1 & t_i < t < t_{i+1}; t_i < t_{i+1} \\ 0 & \text{otherwise} \end{cases} \\ S_{i,j}(t) &= \frac{t - t_i}{t_{i+j} - t_i} S_{i,j-1}(t) + \frac{t_{i+j+1} - t}{t_{i+j+1} - t_{i+1}} S_{i+1,j-1}(t) \end{aligned} \tag{48.50}$$

Depending on the degree of the B-spline, the function is zero on some portion of the unit interval. This is in contrast to the Bernstein polynomials that are strictly non-zero everywhere on the unit interval.

Let x_i^v denote the i^{th} coordinate of the v^{th} mesh node, and let Δx_i^v denote the change in the position of the node due to the morphing process. Let $\Delta \eta_{jk}^i$ denote the displacement of the $(j,k)^{\text{th}}$ control point associated with the Bernstein polynomials in the i^{th} coordinate direction, and let $\Delta \zeta_{jk}^i$ denote the displacement of the $(j,k)^{\text{th}}$ control point associated with the B-splines. If (u^v, v^v) denotes the position of the v^{th} mesh node, then the displacement of the node is defined by the superposition:

$$\Delta x_i^v = \sum_{j=0}^{N_u} \sum_{k=0}^{N_v} \left(\Delta \eta_{jk}^i B_{j,\ell}(u^v) B_{k,m}(v^v) + \Delta \zeta_{jk}^i S_{j,\ell}(u^v) S_{k,m}(v^v) \right) \tag{48.51}$$

The control-point movement for the Bernstein polynomials controls the large-scale smooth deformation, while the control-point movement for the B-splines controls fine-scale motions.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

48.1.7.1.2. Direct Interpolation Method

In the direct interpolation method, the deformation of the interior mesh can be viewed as a projection of the deformation from the boundary into the interior. The displacement of the interior mesh is calculated as a weighted average of all boundary node displacement:

$$\Delta x_v^i = \frac{\sum_{j=0}^{N_b} w_j(r_{ij}) \Delta x_b^j}{\sum_{j=0}^{N_b} w_j(r_{ij})} \tag{48.52}$$

where Δx_v^i and Δx_b^j are the interior and boundary node displacements respectively, w_j is the weighting function, and r_{ij} is the distance between the i^{th} and j^{th} nodes.

Note that while the direct interpolation method has significant advantages, it also has the following limitations:

- The interior mesh quality may not be as good as that produced by the polynomials-based approach and radial basis function (though you may be able to mitigate this by using the **Improve Mesh** dialog box). Boundary zones are also more at risk: they may cross / penetrate themselves or other boundary zones, which may or may not generate a warning about cells with negative volumes; as a result, you must visually review the mesh node changes, and reduce the target changes as needed.
- Non-conformal interfaces should not undergo free deformation, as the morphing may move the two sides apart.
- Some region conditions are not supported, such as invariance and some region boundary continuity options.

48.1.7.1.3. Radial Basis Function

For the radial basis function, the deformation of the mesh is interpolated from the control points.

Radial Basis Function Interpolation:

$$\Delta x_v^i = \sum_{j=1}^{N_c} a_j^i \phi\left(|x_v - x_{cj}|\right) + a_0^i \tag{48.53}$$

Where x_v is the interpolant location, x_c is the set of N_c control point locations, *i* is the *x, y, z* component, and ϕ is the kernel function. The kernel function is responsible for smoothness of deformation and can be selected under the **Numerics** tab of the design tool.

Note:

The linear solver is used to solve the radial basis function equation, therefore the following nonlinear region conditions are not well supported:

- When **Symmetric** is enabled and/or motion is disabled in theta and radial directions (under **Region Conditions** tab of the **Design Tool**), the target change may not be the same as requested and the design conditions may not be well satisfied.
- Invariant region conditions are not supported.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

48.2. Using the Adjoint Solver

This section describes the process for loading the adjoint solver module, as well as setting up, running, and postprocessing the adjoint solutions. This section also demonstrates how the adjoint solution can guide the modification of the boundary shapes or turbulence model parameters.

The typical use of the adjoint solver involves the following steps (though not all of these may be necessary for every workflow):

1. Load or compute a conventional flow solution.
2. Specify the observable(s) of interest.
3. Set the adjoint solver controls.
4. Set the adjoint solver monitors and convergence criteria.
5. Initialize the adjoint solution and iterate to convergence.
6. For shape optimization:
    a. Postprocess the adjoint solution to extract the sensitivity of the observable with respect to the boundary condition settings and the shape of the geometry.
    b. Set up the **Design Tool** and calculate the optimal design change.
    c. Modify the boundary shapes based on shape-sensitivity data and recompute the flow solution. The former can be done through the **Design Tool**, whereas multiple iterations of both the former and the latter can be performed by the **Gradient-Based Optimizer**.
7. For turbulence model optimization:
    a. Postprocess the adjoint solution to extract the sensitivity of the observable with respect to the turbulence model settings and the shape of the geometry.
    b. Use the **Gradient-Based Optimizer** and **Turbulence Model Design Tool** to modify the turbulence model parameters based on flow-sensitivity data and construct an augmented turbulence model.

This section provides information about using the Ansys Fluent adjoint solver in the following sections:

48.2.1. Model Considerations for Using Adjoint Solver
48.2.2. Defining Observables
48.2.3. Solving the Adjoint
48.2.4. Postprocessing of Adjoint Solutions
48.2.5. Modifying the Geometry Using the Design Tool
48.2.6. Using the Gradient-Based Optimizer

In addition, see the Fluent Tutorials for more information about using the adjoint solver. (To access tutorials and their input files, go to the tutorials area of the customer site.)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

48.2.1. Model Considerations for Using Adjoint Solver

The current adjoint solver implementation provides basic adjoint solutions that accompany a conventionally-computed flow solution provided certain criteria are met.

48.2.1.1. Basic Assumptions and Consistency Checks

48.2.1.2. User-Defined Sources

48.2.1.1. Basic Assumptions and Consistency Checks

The adjoint solver is implemented on the following basis:

- The flow state is for a steady single-phase flow, or ideal gas, that is either laminar or turbulent.
- For turbulent flows, the k-ε and k-ω models are supported. When the adjoint turbulence equation is not enabled, a *frozen turbulence* assumption is made, in which the effect of changes to the state of the turbulence is not taken into account when computing sensitivities. When the adjoint turbulence equation is enabled with the GEKO turbulence model, the effect of changes to the state of the turbulence will be taken into account when computing sensitivities.
- The adjoint solver supports both user-defined scalar (UDS) and species transport equations, up to 50 adjoint UDS/adjoint species equations can be specified. When the adjoint UDS/adjoint species equations are not enabled, a *frozen UDS/frozen species* assumption is made, where the effect of changes to the state of the UDS/species are not taken into account when computing sensitivities. When the adjoint UDS/adjoint species equations are enabled, the effect of changes to the state of the UDS/species will be taken into account when computing sensitivities.

  Adjoint species is supported for the following species models and species settings:

  – **Species Transport** model

  – **Chemistry Solver**

    → **None - Direct Source**

    → **Stiff Chemistry Solver**

  – **Volumetric** reactions

  – **Diffusion Energy Source** species transport options

  – All **Turbulence-Chemistry Interaction** options.

  For details on these settings see Species Model Dialog Box (p. 4768).
- The adjoint solver does not support 2d axisymmetric cases.
- For moving (that is, non-inertial) reference frame problems, the single reference frame (SRF) and multiple reference frame (MRF) approaches are supported (as long as the absolute velocity formulation is used), while the mixing plane model (MPM) is not. Note that for the MRF approach, the adjoint solver does not support having a zone with a frame of motion that is relative to an already moving reference frame.
- Solid zones are supported, as long as they do not use a moving reference frame approach.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- By default, the adjoint solver uses spatial discretization methods that are chosen for stability. If desired, you can select more accurate methods, as described in Using the Adjoint Solution Methods Dialog Box (p. 4526).
- The adjoint solver supports constant values for all material properties; non-constant functions are only supported for the following. If you are using a non-constant function not included in this list, the sensitivity accuracy may be reduced; even so, for most cases the sensitivity results will still provide useful information.
  - **Density** :
    → **ideal-gas**
    → **incompressible-ideal-gas**
    → **piecewise-linear**
    → **piecewise-polynomial**
    → **polynomial**
    → **expression** (as a function of temperature only)
  - **Cp (Specific Heat)** :
    → **piecewise-linear**
    → **piecewise-polynomial**
    → **polynomial**
    → **nasa-9-piecewise-polynomial**
    → **kinetic-theory**
  - **Thermal Conductivity** :
    → **piecewise-linear**
    → **piecewise-polynomial**
    → **polynomial**
    → **expression** (as a function of temperature only)
    → **kinetic-theory**
  - **Viscosity**
    → **piecewise-linear**
    → **piecewise-polynomial**
    → **polynomial**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

→ **expression**

→ **power-law**

→ **sutherland**

→ **kinetic-theory**

– **Mass Diffusivity**

→ **constant-dilute-appx**

→ **dilute-approx**

→ **unity-lewis-number**

– **UDS Diffusivity**

→ **polynomial**

→ **expression** (as a function of temperature or pressure)

- The boundary conditions are only of the following types:
  – Wall boundaries are supported with the following limitations:

    → Thin walls and shells are not supported.

    → Only **Heat Flux**, **Temperature**, and **via System Coupling** are supported for **Thermal** walls.

    → Only **No Slip** is supported for the **Shear Condition**.

  – Velocity inlet
  – Mass-flow inlet
  – Mass-flow outlet
  – Pressure inlet
  – Pressure outlet

    Note that the **Prevent Reverse Flow** option is not considered in the adjoint calculation.

  – Pressure far-field
  – Symmetry
  – Rotational and translation periodic
- Most of the boundary conditions in the above list can be defined using profiles, however, ensure that there are no warnings as described below upon initializing the adjoint solver.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Constant and user-defined sources defined by either an expression or UDF are supported for the momentum equation and for scalar transport equations such as the energy equation, turbulence equations, and UDS/species transport equations. See User-Defined Sources (p. 4513) for details.

It is important to note that these requirements are not strict conditions for the conventional flow solver, but rather modeling limitations for the adjoint solver.

When the adjoint solver is initialized or an observable is evaluated, and before iterations are performed, a series of checks is performed to determine the suitability of the existing flow solution for analysis with the adjoint solver. Two types of message may appear:

- *Warning message:* There are certain physics models and boundary condition types that are not explicitly modeled in the adjoint solver, but their absence does not disallow you from proceeding with the calculation. In this case, a warning message will be printed that explains the nature of the inconsistency.

  ```
  Checking adjoint setup...
   -- Warning: Model is active but not included in adjoint calculation: P1 radiation model Done
  ```

  The adjoint solver will still run in this case but the quality of the adjoint solution data can be expected to be poorer as a result of the inconsistency. This is because in such cases the unsupported settings will revert to corresponding supported settings for the adjoint solver. Though the calculation will proceed, it is important to note that the results produced should be considered on this basis. When reverting back to the fluid calculation, the original settings will be preserved and the modified settings are not migrated onto the original case. The warning indicates that the setting change will be made automatically and specifically for the adjoint solution.

- *Error message:* There are some model and boundary conditions that are incompatible with the computation of an adjoint solution with the current adjoint solver implementation. In this case, an explanatory error message will be printed in the console window.

  ```
  Checking adjoint setup...
   ** Unable to proceed: Model is active but not compatible with adjoint calculation:
    Transition SST turbulence model
  Done
  ```

  The adjoint solver will not run in this case and changes must be made manually to the settings identified in the console before the solver can be run.

48.2.1.2. User-Defined Sources

User-defined sources can be created for the momentum equation or for scalar transport equations such as the energy equation, turbulence equations, species transport equations, and user-defined scalar (UDS) transport equations. User-defined sources that are defined in compiled UDFs can be accounted for in the application of the adjoint solver, provided that the source derivatives with respect to flow variables and cell centroid coordinates are provided. This is achieved by creating the source term UDF definition in the usual fashion, with the exception that a special UDF macro is used, `DEFINE_SOURCE_AE` (The suffix _AE denotes Adjoint Enabled).

`DEFINE_SOURCE_AE` (`name` , `c`, `t` , `dS`, `eqn`)

Argument Type	Description
`symbol name`	UDF name

Argument Type	Description
`cell_t c`	Index that identifies cell on which the source term is to be applied
`Thread *t`	Pointer to cell thread
`real dS[]`	Array that contains the derivative of the source term with respect to the dependent variable of the transport equation, as well as the flow variables and cell centroid position
`int eqn`	Equation number

Function returns

`real`

Consider a source that may depend on velocity, pressure, cell-centroid coordinates, temperature, tke coefficient, omega equation, uds equations, and species equations:

$$S\left(u,v,w,p,x_c,y_c,z_c,t,\text{tke},\omega,\text{uds-n},\text{species-n}\right)$$

Where,

- u, v, and w are the Cartesian velocity components in the cell.
- p is the pressure in the cell.
- $\left(x_c,y_c,z_c\right)$ are the cell centroid coordinates as defined by the `C_CENTROID` macro.
- t is the temperature in the cell.
- tke is the value of the turbulence kinetic energy in the cell.
- ω is the specific dissipation rate in the cell.
- uds-n is the (n)th user defined scalar in the cell.
- species-n is the (n)th species mass fraction in the cell.

If the source strength depends on one or more of these quantities, then the derivatives of the source strength with respect to those variables must be computed and filled in as entries to the `dS[]` array. In particular,

$$\text{dS[D_U]}=\frac{\partial S}{\partial u}$$

$$\text{dS[D_V]}=\frac{\partial S}{\partial v}$$

$$\text{dS[D_W]}=\frac{\partial S}{\partial w}$$

$$\text{dS[D_P]}=\frac{\partial S}{\partial p}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$$\mathrm{dS[D_X_CENTROID]}=\frac{\partial S}{\partial x_c}$$

$$\mathrm{dS[D_Y_CENTROID]}=\frac{\partial S}{\partial y_c}$$

$$\mathrm{dS[D_Z_CENTROID]}=\frac{\partial S}{\partial z_c}$$

$$\mathrm{dS[EQ_ENERGY]}=\frac{\partial S}{\partial t}$$

$$\mathrm{dS[EQ_TKE]}=\frac{\partial S}{\partial \mathrm{TKE}}$$

$$\mathrm{dS[EQ_SDR]}=\frac{\partial S}{\partial \omega}$$

$$\mathrm{dS[EQ_UDS\text{-}N]}=\frac{\partial S}{\partial \mathrm{uds\text{-}n}}$$

$$\mathrm{dS[EQ_SPECIES\text{-}N]}=\frac{\partial S}{\partial \mathrm{species\text{-}n}}$$

must be defined in the UDF if there is a dependency of the source upon them. The specification of this derivative information is essential for sensitivities, including shape sensitivity, to be computed correctly by the adjoint solver.

48.2.2. Defining Observables

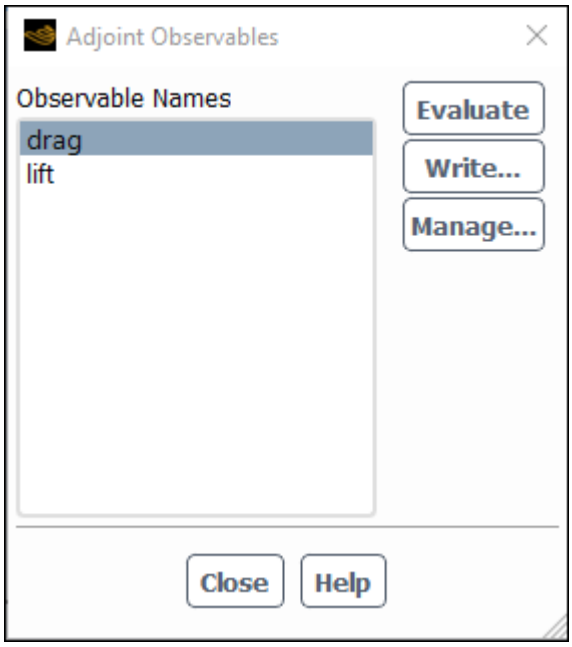
Defining an observable is a key initial step that must be performed for any adjoint calculation. The observable is the quantity for which sensitivities are sought. Observables can be defined based on flow variables, or as operations on other observables. For a list of the types of observables you can define refer to General Observables (p. 4493) and General Operations (p. 4497). You can define multiple observables, but only one observable at a time can be selected for the adjoint sensitivity calculation.

Observables are created and selected in the **Adjoint Observables** dialog box, which is accessed by clicking **Observable...** in the **Design** ribbon tab (**Gradient-Based** group box). (Figure 48.1: Adjoint Observables Dialog Box (p. 4516)).

Design → **Gradient-Based** → **Observable...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 48.1: Adjoint Observables Dialog Box



For more information, see the following sections:

48.2.2.1. Creating New Observables

48.2.2.2. Editing Observable Definitions

48.2.2.3. Defining Observables Using Expressions

48.2.2.4. Selecting an Observable for Sensitivity Calculation

48.2.2.5. Defining Observables for Turbulence Model Optimization

48.2.2.1. Creating New Observables

You can create a new observable by defining a single valued expession or by clicking **Manage...** in the **Adjoint Observables** dialog box and then **Create...** in the **Manage Adjoint Observables** dialog box.

From the **Create...** drop-down list you can choose from several observable types and operation types. Various types of observable quantities can be defined and the available types of observables are described in General Observables (p. 4493).

From the **Observable types** drop-down you can pick an observable from the list. Similarly, you can select an operation from the **Operation types** drop-down list. You can also select an observable to combine existing observables in various ways or apply unary operations to create a wide variety of compound observables. The operations that you can apply to existing observables are described in General Operations (p. 4497).

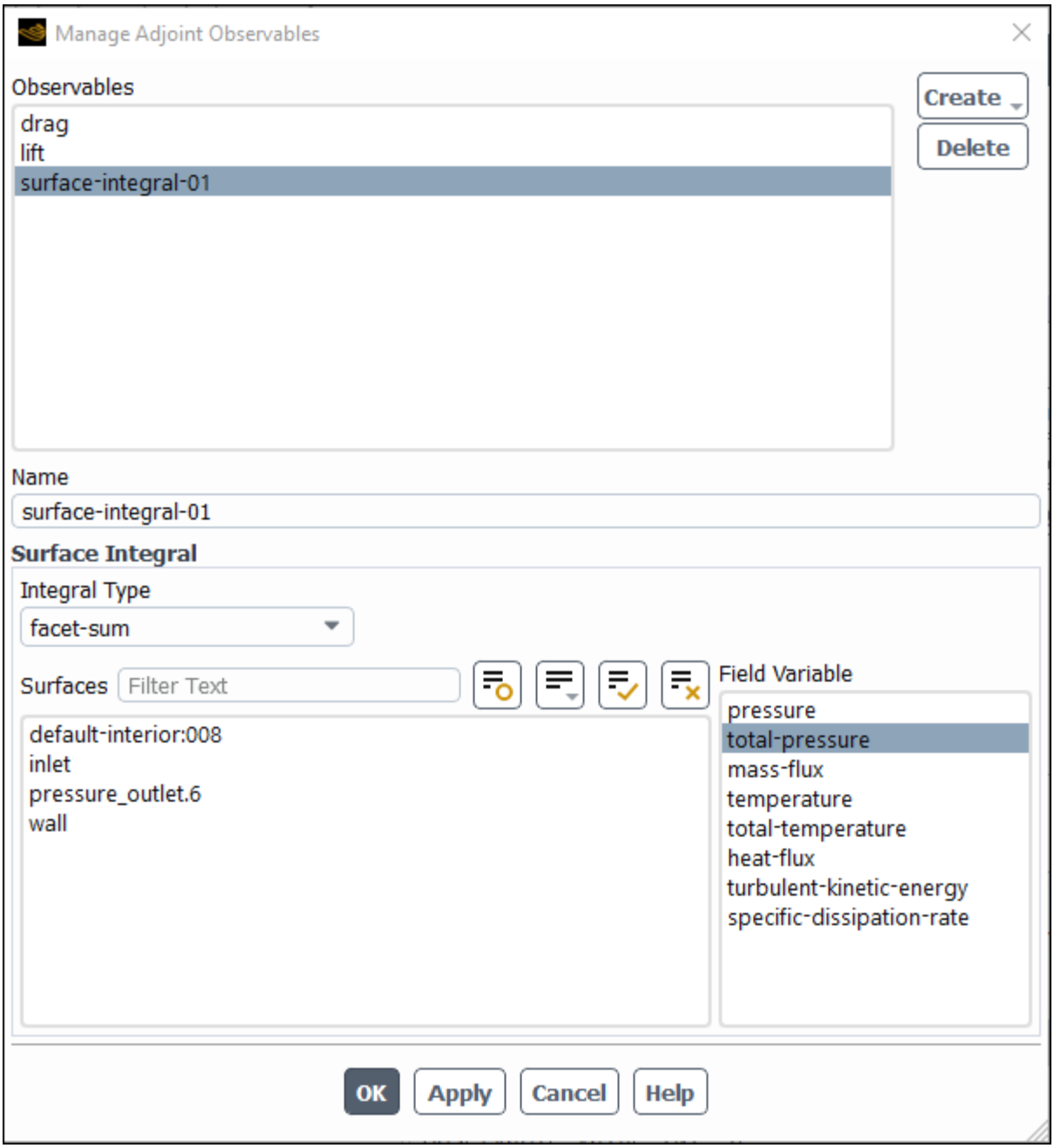
Once selected, you must edit the definition of the new observable within the **Manage Adjoint Observables** dialog box(Editing Observable Definitions (p. 4517)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

48.2.2.2. Editing Observable Definitions

Once an observable is created, it appears in the **Observables** list. When you select an observable in the **Observables** list, the **Manage Adjoint Observables** dialog box changes to expose various properties that can be assigned for the selected observable.

Figure 48.2: Manage Adjoint Observables Dialog Box



Once the observable properties are defined (Inputs for Observable Types (p. 4518)), click **Apply** to apply the settings and proceed to define other observables, or click **OK** in the **Manage Adjoint**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Observables dialog box to apply the settings and close the dialog box. Available observables are described in General Observables (p. 4493).

Note:

It is permissible to leave observable definitions incomplete. The **Manage Adjoint Observables** dialog box can be considered as a workspace for the definition and manipulation of observables. Only those operations with complete definitions will appear in the **Adjoint Observables** dialog box. Operations with undefined fields, or operations that depend on themselves, will be excluded from the list of usable observables.

Inputs for Observable Types

The inputs for each type of observable are summarized below:

- For a **force** observable:

  1. Select the walls that are to contribute to the force of interest in the **Wall Zones** list.
  2. Define the direction in which the force is to be computed by entering the components of this direction in the **X Component**, **Y Component**, and **Z Component** (for 3D) fields.

- For a **moment of force** observable:

  1. Enter the **X**, **Y** , and **Z** (for 3D) components of the **Moment Center**.
  2. Enter the **X**, **Y**, and **Z** (for 3D) components of the **Moment Axis**.
  3. Select a wall in the **Wall Zones** list.

- For a **swirl** observable:

  1. Enter the **X**, **Y**, and **Z** (for 3D) components of the **Swirl Center**.
  2. Enter the **X**, **Y**, and **Z** (for 3D) components of the **Swirl Axis**.
  3. Select a fluid in the **Fluid Zones** list.

- For a **pressure-drop** observable:

  1. Select the inlets and outlets between which the total pressure drop is to be computed using the **Inlets** and **Outlets** lists.

- For a **fixed value** observable, perform the following:

  1. Enter the **Value** of the **Constant**.

- For a **surface-integral** observable:

  1. Select the type of integral from the **Integral Type** drop-down list. Available integral types are described in General Observables (p. 4493).
  2. Select a **Surface** and the corresponding **Field Variable**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For a **volume-integral** observable:

  1. Select the type of integral from the **Volume Integral Type** drop-down list. Available integral types are described in General Observables (p. 4493).

  2. Select the **Field Variable** of interest. If the chosen field variable is a vector quantity, you will need to specify a **Direction** along which the field variable will be evaluated.

  3. If using the **Target-volume-integral** as the **Volume Integral Type**, specify the **user-defined memory** location of your target field variable data from the **Target Field (User Define Memory)** drop-down list.

  4. Under **Integration Domain**, select whether to integrate over selected **Zones**, a **Box Region**, or a **Cell Register**.

  5. Select the **Zones** for integration, enter the coordinates of the rectangular (in 2D) or hexahedral (in 3D) integration region under **Box Settings**, or make a selection from the **Cell Register** drop-down list.

- For a **ratio** observable:

  1. Select an observable from the **Numerator** drop-down list to represent the numerator of the ratio.

  2. Select an observable from the **Denominator** drop-down list to represent the denominator of the ratio.

- For a **product** observable:

  1. Select two observables from the corresponding drop-down list that will be used to compute their product.

- For a **linear combination** observable:

  1. Under **Linear Combination of powers** , enter a **Constant** value.

  2. Enter the number of **Components** .

  3. Make a selection from the **Combination Type** drop-down list. For details, see General Operations (p. 4497).

  4. For each component, enter a corresponding **Coefficient**, an **Observable**, and a **Power**.

- For an **arithmetic average** observable:

  1. Enter the number of **Components** .

  2. Select observables from the corresponding **Observable** lists.

- For a **mean-variance** observable:

  1. Enter the number of **Components** .

  2. Select observables from the corresponding **Observable** lists.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- For a **unary operation** observable:

  1. Select an operation from the drop-down list (see General Observables (p. 4493) for a list of available operators).

  2. Select an observable from the corresponding drop-down list upon which the unary operation is to be applied.

48.2.2.3. Defining Observables Using Expressions

Expressions can be used as adjoint observables or as field variables for **Surface Integral** and **Volume Integral** observables. Expressions used as observables can be selected from the list of **Observable Names** within the **Adjoint Observables** dialog box. For **Surface Integral** and **Volume Integral** observables, expressions can be selected from the **Field Variable** drop-down list within the **Manage Adjoint Observables** dialog box.

For both expression observables and surface/volume integral observables defined with an expression, the expression can be a combination of single-valued and field expressions within a binary operator, for example:

```
VolumeAve((P - VolumeAve(P, ['fluid:011']))^2, ['fluid:011'])
```

where P is a field expression and `VolumeAve(P, ['fluid:011'])` is a single-valued expression.

Apart from combined single-valued and field expressions, only single-valued scalar expressions can be used as observables and only field expressions can be used to define surface/volume integral observables.

Single-valued or field expressions can not be combined with vector expressions. For example, the following expression is not supported:

- `Average(Velocity.x,['wall'],Weight=None)*Force(['wall'])`

Where `Average(Velocity.x,['wall'],Weight =None)`is a scalar single-valued expression and `Force(['wall'])` is a vector single-valued expression.

However, the above expression is supported by using the component of the vector `Force(['wall'])` and written as:

- `Average(Velocity.x,['wall'],Weight =None)*Force(['wall']).x`

Observables can not be created for expressions containing **Flux** functions, **Report Definitions**, and cannot contain the following **Mathematical** functions:

- **gradient**
- **normalize**

Note that when defining an expression for use in an adjoint observable, it is recommended to not use discontinuous functions (e. g. **floor**, **min**, or **max**) due to a possibly invalid differentiation. However you can use a similar continuous function to mimic the behavior, such as using the softmax function to minic the max function.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If an adjoint observable is not fully supported, a warning message similar to the following will be printed in the console when the observable is selected, or before running the adjoint solver:

```
Warning: the adjoint observable does not support the expression: DensityAll.


Warning: the adjoint observable does not support the expression: mul(P, expr3).
         The child expressions are not the same type: field/single-valued scalar/
         single-valued vector.
```

Expressions containing the following **Variables** can be used as observables:

- **Pressure...**
  - **Static Pressure**
  - **Pressure Coefficient**
  - **Dynamic Pressure**
  - **Absolute Pressure**
  - **Total Pressure**
- **Sensitivities...**
  - **Adjoint GEKO CSEP**
  - **Adjoint GEKO CNW**
  - **Adjoint GEKO CMIX**
  - **Adjoint GEKO BF**
- **Mesh...**
  - **Face Area Magnitude**
  - **X Face Area**
  - **Y Face Area**
  - **Z Face Area**
  - **Cell Volume**
  - **X-Coordinate**
  - **Y-Coordinate**
  - **Z-Coordinate**
  - **Position.mag**
- **Turbulence...**
  - **Turbulent Kinetic Energy (k)**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - **Turbulent Viscosity**
  - **Effective Viscosity**
  - **Turbulent Dissipation Rate (Epsilon)**
  - **Specific Dissipation Rate (Omega)**
  - **Turbulent Intensity**
  - **Turbulent Viscosity Ratio**
  - **Production of k**
  - **Turbulent Reynolds Number (Re_y)**
  - **Blending Function for GEKO**
  - **Curvature Correction Function fr**
- **Density...**
  - **Density**
- **Properties...**
  - **Sound Speed**
  - **Thermal Conductivity**
  - **Specific Heat (Cp)**
- **Species...**
  - **Eff Diff Coef of *species-n***
  - **Lam Diff Coef of *species-n***
  - **Enthalpy of *species-n***
  - **Mass fraction of *species-n***
  - **Molar Concentration of *species-n***
  - **Mole fraction of *species-n***
- **Velocity...**
  - **Velocity Magnitude**
  - **X Velocity**
  - **Y Velocity**
  - **Z Velocity**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - **Axial Velocity**
  - **Radial Velocity**
  - **Tangential Velocity**
  - **Velocity Angle**
  - **Vorticity Magnitude**
  - **X-Vorticity**
  - **Y-Vorticity**
  - **Z-Vorticity**
  - **Mach Number**
  - **Cell Reynolds Number**
  - **Q Criterion Normalized**
  - **Q Criterion Raw**
  - **Lambda 2 Criterion**
- **Derivatives...**
  - **Strain Rate**
  - **dX-Velocity/dx**
  - **dY-Velocity/dx**
  - **dZ-Velocity/dx**
  - **dVelocitydx.mag**
  - **dX-Velocity/dy**
  - **dY-Velocity/dy**
  - **dZ-Velocity/dy**
  - **dVelocitydy.mag**
  - **dX-Velocity/dz**
  - **dY-Velocity/dz**
  - **dZ-Velocity/dz**
  - **dVelocitydz.mag**
  - **dp-dX**

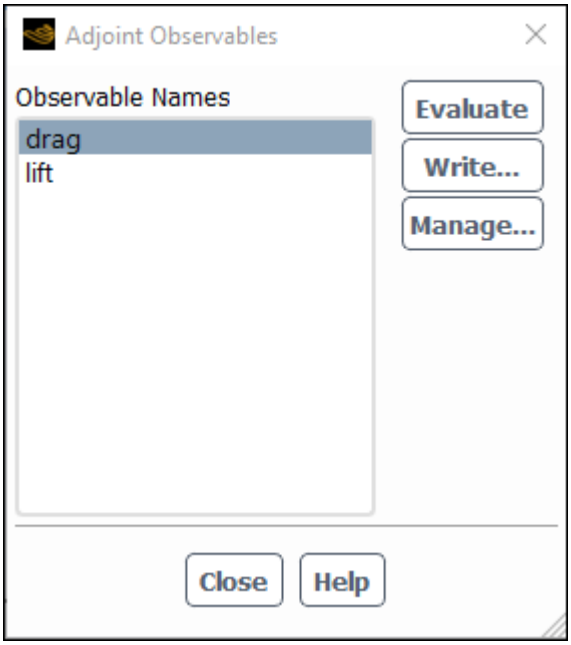
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - **dp-dY**
  - **dp-dZ**
  - **dpd.mag**
- **Temperature...**
  - **Static Temperature**
  - **Total Temperature**
  - **Entropy**

For more information on Fluent expressions see Expression Operations and Functions (p. 1038).

48.2.2.4. Selecting an Observable for Sensitivity Calculation

Figure 48.3: Adjoint Observables Dialog Box



You can select which observable to use for the sensitivity calculation in the list of **Observable Names**.

Note:

- Only one observable can be used in any one adjoint calculation. In the **Adjoint Observables** dialog box, the observable that is currently highlighted is the observable that will be used in the sensitivity calculation.
- Any observables that have been created but which are missing required inputs will not appear in the list of **Observable Names**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Clicking the **Evaluate** button computes the current value of the selected observable quantity and prints the result in the console window. Clicking the **Write...** button provides the option to write the result to a named file.

The **Sensitivity Orientation** determines the sign of the postprocessed sensitivities. For instance, if you select **Maximize** then changes in the direction of positive sensitivities will increase the value of the observable. A standard rule of thumb can be applied: if you want to improve the solution, then follow the direction of the sensitivity vectors. Another standard rule of thumb can also be applied for postprocessed scalars: if you want to improve the solution, then increase values where the sensitivity is positive and/or decrease values where the sensitivity is negative.

48.2.2.5. Defining Observables for Turbulence Model Optimization

For turbulence model optimization (see Using the Turbulence Model Design Tool (p. 4599)), optimization observables should be defined as the difference between the results computed from your RANS simulation and experimental data or data from a high-fidelity simulation. Optimization observables can be specified to target a constant quantity (such as lift or drag) or target field data (such as a velocity or pressure field). Note that targeted field data must either be from experimental data or exported into user-defined memory (UDM) from a high-resolution RANS simulation or time-averaged flow solution of an SBES simulation.

Target field data can be written to user-defined memory using the following text command:

```
/adjoint/utilities/interpolate/write-data
```

To import target field data into user-defined memory for the case being optimized enter the following text command:

```
/adjoint/utilities/interpolate/read-data
```

For more information on creating and editing the following observables see Creating New Observables (p. 4516) and Editing Observable Definitions (p. 4517).

For targeting field data, create a **Volume Integral** observable and configure the following settings:

1. From the **Volume Integral Type** drop-down list, select **Target-volume-integral** which is the norm of the difference between the field computed from your RANS simulation and the target field data.

2. Specify the **Volume Integral** settings for your optimization problem, including selecting the appropriate **User-Defined Memory** location of your imported target data.

For targeting a constant quantity (e.g **force**), the observable should be defined as the absolute value of the sum of the quantity computed by your RANS simulation and the target quantity.

The general procedure for defining an observable to target a constant quantity is as follows:

1. Create a new **Observable** for the quantity of interest (e.g **force**).

2. Create a **linear combination** operation to define the difference between the computed quantity and the target quantity.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a. Within the **Linear Combination of powers** group box, select **Sum** from the **Combination Type** drop-down list and retain the value of `1` for the **Coefficient** , **Components**, and **Power** settings.

b. Select the observable you created for your quantity of interest from the **Observable** drop-down list.

c. Enter the negation of the value of your target quantity under **Constant**. For example if your target quantity is `5000` for lift, enter `-5000`.

3. Create a **unary operation** for the absolute value of the **linear combination** operation defined in the previous step.

48.2.3. Solving the Adjoint

Once an observable is defined as described in Defining Observables (p. 4515), the solution of the adjoint for that observable can be computed based on the current flow solution. The steps for solving the adjoint are similar to those for solving the flow and are detailed in the following sections:

48.2.3.1. Using the Adjoint Solution Methods Dialog Box

48.2.3.2. Using the Adjoint Solution Controls Dialog Box

48.2.3.3. Working with Adjoint Residual Monitors

48.2.3.4. Printing and Postprocessing the Adjoint Equation Residuals

48.2.3.5. Running the Adjoint Calculation

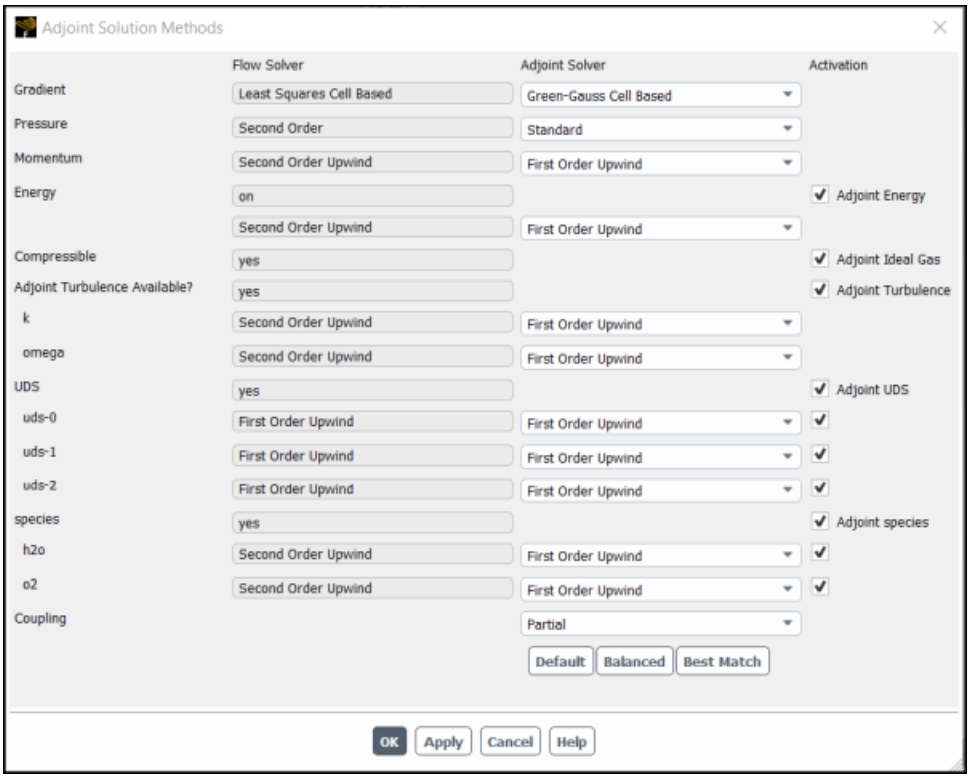
48.2.3.1. Using the Adjoint Solution Methods Dialog Box

You can specify the methods used for computing the adjoint solutions in the **Adjoint Solution Methods** dialog box. To open the **Adjoint Solution Methods** dialog box, click **Methods...** in the **Design** ribbon tab (**Gradient-Based** group box).

Design → **Gradient-Based** → **Methods...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 48.4: Adjoint Solution Methods Dialog Box



The **Adjoint Solution Methods** dialog box shows a side-by-side comparison of the schemes used for the flow solver and for the adjoint solver. Using the same scheme for the adjoint solution and the flow solution yields the most accurate discrete derivative calculation when the adjoint solution is converged. However, not all schemes used for the flow solver are supported for the adjoint solver. In these cases an alternate adjoint scheme must be used. This does not typically lead to severe deterioration of the adjoint results quality. Even if the same scheme is available for the adjoint solver this is not always practical because stability may be reduced with some schemes. Therefore, you can use the drop-down lists under **Adjoint Solver** to select alternate schemes as needed.

In some cases it may not be desirable to solve the energy adjoint even if energy is solved in the flow solver. For example, consider an incompressible flow where the specified observable does not involve thermal quantities. In this case the adjoint for the energy equation is identically zero, but its inclusion would add an unnecessary numerical burden. You can disable the **Adjoint Energy** option to avoid solving the energy adjoint.

If the primary flow is a compressible ideal gas, you can specify that the physics of the compressible flow are modeled in the adjoint solver by enabling the **Adjoint Ideal Gas** option. Enabling this option automatically enables the **Adjoint Energy** option as well. If the flow is not a compressible ideal gas, it will be treated as being incompressible for the purposes of the adjoint calculation.

When the flow solver uses the generalized k - ω (GEKO) turbulence model (for details, see Setting up the Generalized k-ω (GEKO) Model (p. 2090)), you can enable the **Adjoint Turbulence** option in order to take the effect of changes to the state of the turbulence into account when computing sensitivities for the adjoint calculations, rather than using a frozen turbulence assumption. This has the following advantages:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The shape sensitivity will be more accurate. This is helpful when high accurate shape sensitivity is needed, such as:
  - when the design is close to optimal design
  - when the turbulence state can change significantly with respect to the design changes
  - when performing optimization with multiple objectives
- Sensitivities for GEKO parameters and the curvature correction parameter (**CCURV**) can be can be calculated at each cell, which can help you to tune the GEKO model.

Including turbulence in the adjoint calculations can make it harder to reach convergence, and will increase the computational and memory cost. For this reason, it is suggested that you only enable turbulence adjoint equations when it is necessary; in most cases, the frozen turbulence assumption should be fine.

Note:

You must make sure that the **Intermittency Transition Model** option was disabled for the flow solution, as it is not supported with adjoint turbulence. However, the **Algebraic Transition Model** is supported with adjoint turbulence.

When one or multiple user-defined scalar (UDS) transport equations are defined, you can specify the adjoint solver to compute the adjoint for the UDS equations by enabling the **Adjoint UDS** option. Additionally, when species transport is defined, the adjoint for species transport equations can be computed by enabling the **Adjoint species** option. After enabling **Adjoint UDS** or **Adjoint species**, you can enable the desired UDS or species transport equations to be computed by the adjoint solver. Up to 50 adjoint UDS/adjoint species equations can be enabled.

If the UDS or species transport equations are not necessary for optimization, it is recommended to disable **Adjoint UDS** and **Adjoint species** to save time and memory and improve convergence. For example, disabling **Adjoint UDS** and **Adjoint species** is recommended for the following scenarios:

- When a specific observable is not significantly influenced by UDS or species transport equations.
- When the state of UDS or species transport equations do not change significantly with respect to the design changes.
- During the early stages of optimization, where achieving sensitivity accuracy is less important.

The adjoint solver utilizes the algebraic multigrid (AMG) algorithm to solve the correction equations (adjoint equations) and each AMG block consists of an N x N matrix, where N is the number of adjoint equations (for more information on the algebraic multigrid in Ansys Fluent see Algebraic Multigrid (AMG) in the Fluent theory guide). The adjoint solver can be specified to be fully coupled or partially coupled by selecting **Full** or **Partial** respectively in the **Coupled** drop-down list. When **Full** is selected, all adjoint equations are coupled within a single AMG block. The adjoint solver is set to partially coupled by default, and with **Partial** selected the adjoint continuity equation and adjoint momentum equations are coupled within one AMG block while all other adjoint equations are solved individually in segregated AMG blocks. For example, the adjoint solver must solve the following seven adjoint equations for a 3D turbulence case:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- adjoint x-momentum
- adjoint y-momentum
- adjoint z-momentum
- adjoint continuity
- adjoint flow rate
- adjoint k
- adjoint omega

When the adjoint solver is fully coupled, all of the adjoint equations will be solved within a single `7 x 7 = 49` AMG block. For the partially coupled adjoint solver, the AMG block for the adjoint continuity and adjoint momentum equations will be `4 x 4 = 16`, while all other adjoint equations are solved seperately in individual `1 x 1 = 1` AMG blocks.

The significant reduction in the size of the AMG blocks for the partially coupled solver both reduces memory requirements and improves speed per iteration significantly as well, especially when **Adjoint Turbulence** and/or **Adjoint Energy** are enabled. Note that while the partially coupled adjoint solver significantly improves performance, the outer convergence may be slightly reduced.

Clicking the **Default** button will disable the **Adjoint Turbulence** equation and set the spatial discretization methods as follows:

- **Green-Gauss Cell Based** for **Gradient**.
- **Standard** for **Pressure** .
- **First Order Upwind** for **Momentum** and **Energy** (chosen due to their stability).
- **Partial** for **Coupling** .

Clicking the **Best Match** button will attempt to match the adjoint schemes to the schemes used for the flow solver which should in general provide more accurate calculations at convergence. Where the schemes cannot be matched, the **Default** settings will be used.

Clicking the **Balanced** button will provide a balance between stability and accuracy settings. Due to momentum discretization and adjoint turbulence having the largest affect on convergence, the **Balanced** setting will disable **Adjoint Turbulence**, set **Momentum** to **First Order Upwind**, and use **Best Match** settings for all other methods.

By default the **Rhie-Chow: momentum based** flux type is used for the adjoint solver. However, the adjoint solver can be specified to use the same flux type as that of the flow solver using the following text command:

```
adjoint/expert/match-fluent-flux-type?
```

By default, the adjoint solver will incorporate 1st order poor mesh numerics used by the conventional flow solver as outlined in Robustness with Meshes of Poor Quality (p. 3741). Poor mesh numerics can be disabled using the following command:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
/adjoint/expert/undo-2024r1-default-changes?
```

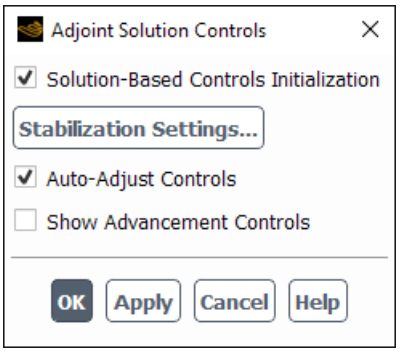
48.2.3.2. Using the Adjoint Solution Controls Dialog Box

The advancement of the adjoint solution to reach a converged solution is an essential part of the analysis. The solution algorithm for the adjoint solver is similar to the coupled pressure-based solver that is available for conventional flow computations in Ansys Fluent.

By default, the adjoint solver advancement settings will be automatically adjusted during calculation to encourage convergence, with initial values determined by the state of the flow solution. If needed, you can adjust the advancement controls in the **Adjoint Solution Controls** dialog box.

To open the **Adjoint Solution Controls** dialog box, click **Solver Controls...** in the **Design** ribbon tab (**Gradient-Based** group box).

Design → **Gradient-Based** → **Solver Controls...**



Controls

Solution-Based Controls Initialization

When enabled, the advancement controls are selected automatically when the adjoint solution is initialized based on the state of the flow.

Stabilization Settings...

This button opens the **Stabilized Strategy and Scheme Settings** dialog box, which you can use to define the stabilization strategy, scheme(s), and settings for the adjoint solution. For details, see Stabilization Strategies, Schemes, and Settings (p. 4533).

Auto-Adjust Controls

When enabled, the convergence of the AMG solver and the adjoint residuals are monitored during solution advancement. Based on the trends observed, the Courant number is automatically adjusted to encourage reliable solution advancement and convergence. This is especially useful when a new type of problem is being solved for which appropriate settings are not initially clear. You can enable or disable **Auto-Adjust Controls** at any time during the solution.

The Courant number starts at the value you defined in the **Advancement Controls** group box or the value determined by the **Solution-Based Controls Initialization** option. When the residual minimization scheme is used, over the first two iterations the Courant number will be increased as high as possible (while still allowing the AMG solver to converge), and will then remain un-

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

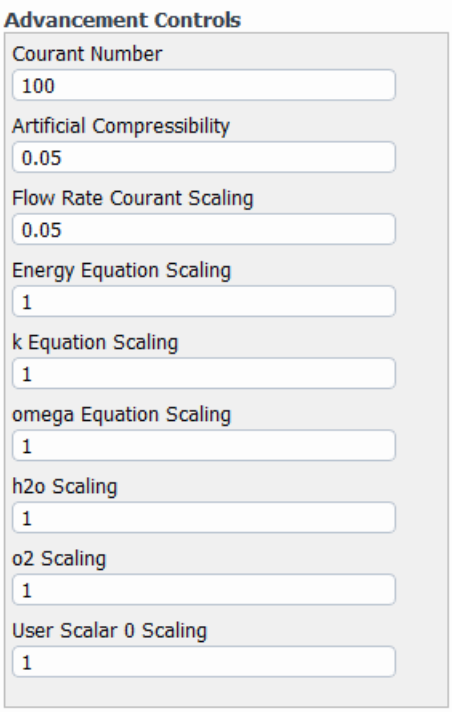
changed; for all other setups, the Courant number will be adjusted to ensure the AMG solver converges, with the goal of reducing the AMG residuals at the rate of 1 order of magnitude over 6-8 iterations. Messages will appear as the calculation progresses indicating the adjustments that are being made.

Show Advancement Controls

This option toggles the visibility of the various advancement controls for direct user specification.

Advancement Controls

contains the advancement controls.



Apply Preconditioning

Enables solution preconditioning. This is needed for most cases involving turbulent flow. If preconditioning is enabled, additional controls are available.

Courant Number

A higher number corresponds to a more aggressive advancement of the computation at the risk of instability. Disabling preconditioning corresponds to an infinite Courant number.

Artificial Compressibility

A nonzero value introduces artificial compressibility into the computation of the adjoint continuity equation. A value of `1.0` or less is reasonable.

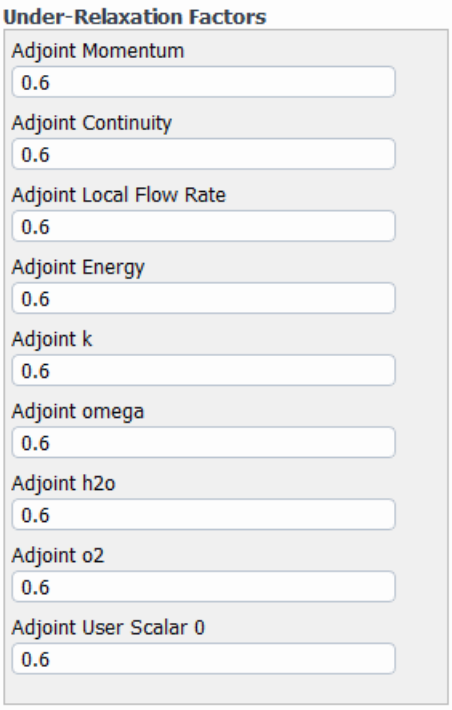
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Flow Rate Courant Scaling , Energy Equation Scaling , k Equation Scaling , omega Equation Scaling, *species-n* Scaling, User Scalar n Scaling

These values should be larger than zero (default value is `0.05` for flow rate and `1` for all other equations). A smaller value implies a less aggressive algorithm that encourages stability of the AMG linear solver.

Under-Relaxation Factors

contains the under-relaxation factors.

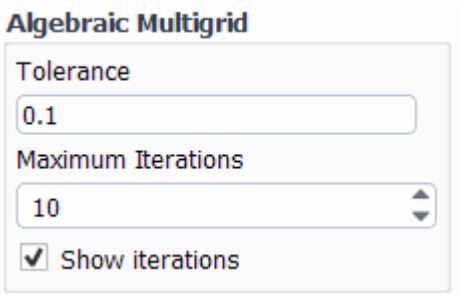


Adjoint Momentum, Adjoint Continuity, Adjoint Local Flow Rate, Adjoint Energy , Adjoint k, Adjoint omega, Adjoint *species-n*, Adjoint User Scalar n

Each of these can be set to a value between `0.0` to `1.0`. A higher value leads to a more aggressive algorithm that is less likely to be stable. A value of `1.0` for each can be used for some simple cases without difficulty.

Algebraic Multigrid

contains the algebraic multigrid controls.



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tolerance

The tolerance used for judging convergence.

Maximum Iterations

The maximum number of inner iterations of the AMG solver.

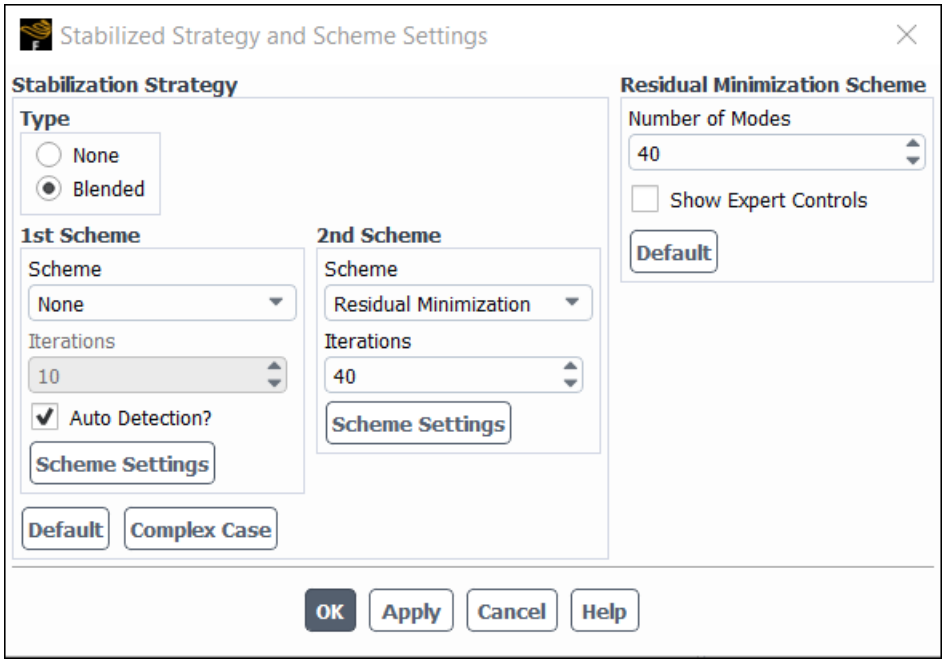
Show Iterations

If enabled, a more verbose iteration history is printed in the text console during iterations. The details of the inner iteration can be useful when deciding on appropriate **Courant Number**, **Artificial Compressibility**, and **Flow Rate Courant Scaling**. If many inner iterations are needed, or if the inner iterations diverge, this signals that a reduction in **Courant Number** may be needed. Alternatively, an increase in **Artificial Compressibility** or a reduction in the **Flow Rate Courant Scaling** may be sufficient for the AMG iterations to converge.

48.2.3.2.1. Stabilization Strategies, Schemes, and Settings

For an overview of when stabilization is needed and descriptions of the available schemes, see Adjoint Solver Stabilization (p. 4503). You can define a stabilization strategy in the **Stabilized Strategy and Scheme Settings** dialog box, which is opened by clicking the **Stabilization Settings...** button in the **Adjoint Solution Controls** dialog box.

Figure 48.5: The Stabilized Strategy and Scheme Settings Dialog Box



Selecting **None** from the **Type** list in the **Stabilization Strategy** group box on the left allows you to select a single stabilization method to be used throughout the calculation, or none at all. You specify your preference using the **Scheme** drop-down list in the **Current Scheme** group box, then click the **Scheme Settings** button to define the scheme settings on the right.

Alternatively, you can select **Blended** from the **Type** list in the **Stabilization Strategy** group box on the left, so that either no scheme (**None**) or the **Dissipation** scheme is applied at the beginning

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

of the calculation, depending on your selection from the **Scheme** drop-down list in the **1st Scheme** group box; then a second scheme (the **Residual Minimization** scheme) is used for the remainder of the calculation. If you enable the **Auto Detection?** option, the first scheme will be used until divergence or slow convergence in the adjoint residual is detected; otherwise, you specify an explicit number of **Iterations** to be run before switching to the second scheme. The latter can be useful for cases that take a long time to diverge or show signs of slow convergence with the first scheme, as you can force an early switch to the residual minimization scheme (which generally converges quickly). You can also specify the maximum number of **Iterations** to allow for the **2nd Scheme**, as the residual minimization scheme can be expensive on a per iteration basis. Note that settings used for the dissipation and residual minimization schemes are defined within the **Dissipation Scheme** and **Residual Minimization Scheme** group boxes on the right, which will appear after clicking the **Scheme Settings** button in the **1st Scheme** and **2nd Scheme** group boxes, respectively. The controls for each scheme are described in the sections that follow.

Note:

After your first calculation is complete, the second stabilization scheme will be used at the start of all subsequent calculations, unless you click the **Initialize Strategy** button in the **Run Adjoint Calculation** dialog box prior to calculating. Initializing the strategy is recommended when there is an operation that will cause a residual jump (such as morphing the mesh), as the first scheme can resolve some of the modes more efficiently.

When using the **Blended** strategy, note that you can use the **Default** button to revert any changes you have made to the settings. You can also use the **Complex Case** button to use settings that are more appropriate for complex cases that would benefit from the initial use of the dissipation scheme to damp local diverging modes, before switching to the residual minimization scheme; note that for some cases, you may need to increase the **Iterations** for the dissipation scheme or increase the damping factor in the **Dissipation Scheme** group box (for example, to `100`) to allow the scheme to fully damp the modes.

The stabilization settings for each scheme are described in the following sections:

48.2.3.2.1.1. Dissipation Scheme

48.2.3.2.1.2. Residual Minimization Scheme

48.2.3.2.1.1. Dissipation Scheme

The dissipation scheme operates by introducing nonlinear damping strategically into the calculation domain. The strategy is intended to provide minimal intervention in order to damp the growth of instabilities that lead to adjoint solution divergence. A marker is tracked, based on the state of the adjoint solution, and the damping is applied directly to the adjoint solution in regions where the marker becomes large.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 48.6: The Dissipation Scheme Settings

When **Dissipation** is selected from the **Scheme** drop-down list within the **Current Scheme** or **1st Scheme** group box (depending on the specified **Stabilization Strategy**), you can then click the **Scheme Settings** button to open the **Dissipation Scheme** group box on the right and define the following settings:

Damping Factor

The overall level of dissipation is proportional to this value although the damping level is determined ultimately through a nonlinear process.

Damping Relaxation

The damping relaxation can be used to control the rate at which the dissipation is updated as the adjoint solution progresses. As you decrease the value from 1, the rate at which the dissipation is updated is decreased.

Damping Order

The spatial order of the dissipation. A higher order leads to more intense and localized damping. This can typically be set to be one order larger than the adjoint calculation spatial order.

Suppression

As part of the dissipation scheme, the growth of instabilities is stopped by the effect of the dissipation; enabling the **Suppression** option ensures that these undesirable patterns will then also decay as the calculation progresses.

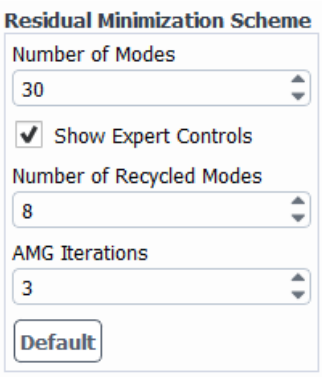
Since the order of discretization of the dissipation scheme is usually set higher than the adjoint, the formal order of accuracy of the adjoint solution is unaffected by the addition of dissipation. However, regions of intense dissipation can appear for some cases when using this scheme. This can sometimes appear as isolated spots and streaks in the adjoint solution on surfaces. The scale of these features is such that they are often easily smoothed during postprocessing to generated design changes.

48.2.3.2.1.2. Residual Minimization Scheme

The residual minimization scheme operates by building a Krylov subspace and using it to build a solution with minimum residual.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 48.7: The Residual Minimization Scheme Settings



When **Residual Minimization** is selected from the **Scheme** drop-down list within the **Current Scheme** or **2nd Scheme** group box (depending on the specified **Stabilization Strategy**), you can then click the **Scheme Settings** button to open the **Residual Minimization Scheme** group box on the right and define the following settings:

Number of Modes

This specifies the number of modes in the Krylov subspace used to approximate the solution. Increasing the number of modes will increase the convergence and reduce the final residual. The convergence rate may increase sharply when the number of modes is larger than a certain threshold. Note that the memory needed for this scheme is proportional to (number of adjoint equations) * (number of modes + number of recycled modes). You should ensure that you have enough memory for the value you enter here; an estimation of the additional memory needed at the peak is printed in the console when you click **Apply** .

Show Expert Controls

Enabling this option reveals the expert controls described below, so that you can adjust them from the default values.

Number of Recycled Modes

This specifies the number of modes that are selected, stored, and used for the next iteration to speed up the convergence. The default value is a quarter of the number of modes.

For the initial step of the residual minimization scheme, the number of sub-iterations is equal to the number of modes; for later steps, it is the difference of the number of modes and the number of recycled modes. Therefore, increasing the number of recycled modes will reduce the number of sub-iterations for each main iteration; while this relation is straightforward, the effect on the overall convergence ratio is not, as it is case dependent.

AMG Iterations

This specifies the fixed number of AMG iterations in each sub-iteration. Increasing the number may improve the overall convergence, but it also increases the time for each sub-iteration.The residual minimization scheme is unique in that it does not require the AMG solver to reduce the inner residual by at least one order; it only requires that the AMG solver not be diverging. In fact, a more aggressive setting of preconditioning may slow

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

down the inner AMG convergence, but improve the overall outer loop convergence. Such an improvement can be significant.

Note:

The following tips can be helpful when using the residual minimization scheme:

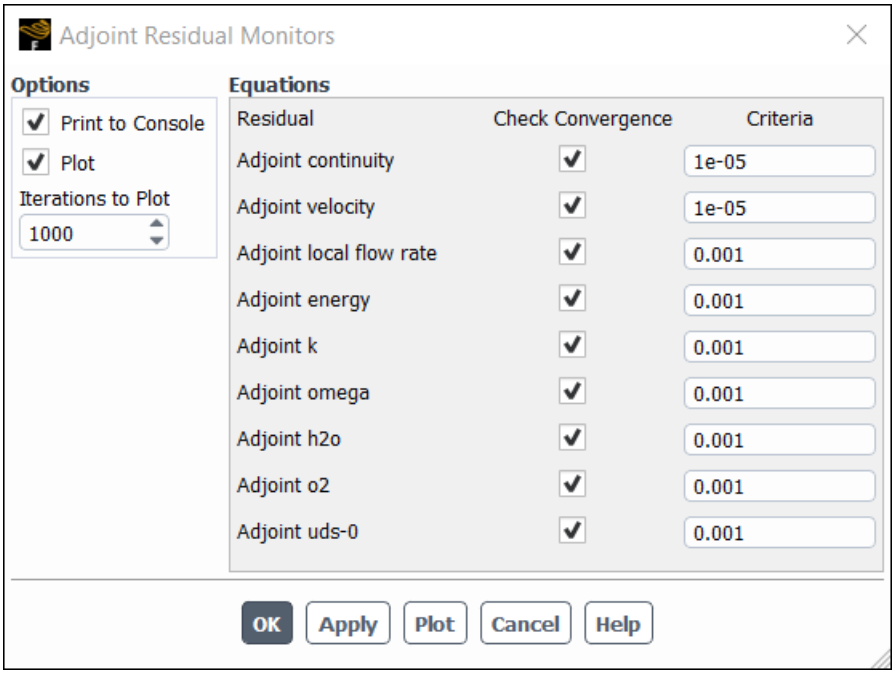
- It is highly recommended that you first run the adjoint solver without a stabilization scheme until the adjoint residual starts to diverge, and then enable the residual minimization scheme. The first run damps out many stable modes in the adjoint residual and reduces the number of modes required in the residual minimization scheme. Note that this can easily be done by selecting the **Blended** strategy in the **Stabilization Strategy** group box.
- The overall adjoint residual will be printed in the console. The residual should either decrease monotonically or stall within each main iteration or between main iterations. If you do not see such behavior, click the **Initialize Stabilization** button in the **Run Adjoint Calculation** dialog box to initialize the scheme.

48.2.3.3. Working with Adjoint Residual Monitors

The progress of the iterations of the adjoint solver and the monitoring of the convergence is controlled in the **Adjoint Residual Monitors** dialog box. This dialog box is accessed by clicking **Monitors...** in the **Design** ribbon tab (**Gradient-Based** group box).

Design → Gradient-Based → Monitors...

Figure 48.8: Adjoint Residual Monitors Dialog Box



The steps to define the monitor behavior are as follows:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Decide whether or not the residuals are to be printed in the console by setting the **Print to Console** check box as desired.

2. Decide whether or not the residuals are to be plotted in the main window by setting the **Plot** check box as desired.

3. Set how many iterations are to be shown in the residual curves that are plotted in the **Iterations to Plot** field.

4. Enable and/or disable those values that are to be used as criteria for convergence, and set the **Criteria** in each case. The options are to test the residuals for the following equations: **Adjoint continuity**, **Adjoint velocity**, **Adjoint local flow rate** , **Adjoint energy**, **Adjoint k**, **Adjoint omega**, **Adjoint *species-n***, and/or **Adjoint uds-n**.

Click the **OK** or **Apply** buttons to confirm that the settings are acceptable. Clicking **Plot** will cause existing adjoint residuals to be plotted in the main graphics window.

48.2.3.4. Printing and Postprocessing the Adjoint Equation Residuals

If the adjoint calculation does not converge properly, you can print and/or postprocess the adjoint equation residuals, as this can provide useful information for the diagnosis of convergence issues. Convergence issues can be caused by:

- Flow instability such as high shear flow, flow separation, or reverse flow
- Poor mesh quality
- A specific boundary condition
- Unsupported models in Fluent, as mentioned in the warning message

To help understand the cause of your particular convergence issue, you can enable the following text commands prior to running the adjoint calculation:

- `adjoint` → `expert` → `diagnosis` → `print-residuals-by-zone`

  This text command specifies that the residuals are printed in the console during the calculation in a format that allows you to see if the high residual is located at a specific boundary zone.

- `adjoint` → `expert` → `diagnosis` → `retain-cell-residuals`

  This text command saves the residuals as field variables that can be postprocessed, so that you can visualize the high residual region to check if any of the previously mentioned reasons are the cause of convergence issues. Occasionally, the divergence only occurs locally, and most of the residual is located at a small region. In this case, even though the adjoint residual value is not low, the adjoint solution can still be used to improve your design if you can avoid morphing that region.

- `adjoint` → `expert` → `diagnosis` → `print-residuals-by-distribution`

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This text command specifies that the distribution of the residuals is printed in the console during the calculation, so that you can see if a high percentage of the residuals is located in a small number or percentage of cells.

Note:

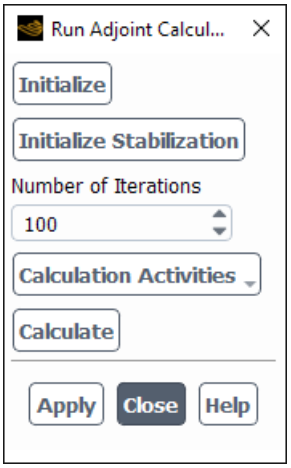
After setting up the text user command, you need to run at least 1 adjoint iteration to print / save the residual or receive a message.

48.2.3.5. Running the Adjoint Calculation

Initialization and execution of the adjoint solver is accomplished in the **Run Adjoint Calculation** dialog box. This dialog box is accessed by clicking **Calculate...** in the **Design** ribbon tab (**Gradient-Based** group box).

Design → Gradient-Based → Calculate...

Figure 48.9: Run Adjoint Calculation Dialog Box



The function of this dialog box is as follows:

Initialize

sets the value of the adjoint velocity and pressure to zero everywhere in the problem domain.

Initialize Strategy

initializes the stabilization strategy, such that the first stabilization scheme is used at the start of the next calculation rather than continuing to use the second stabilization scheme. This button is only available when the blended stabilization strategy is used. For details, see Stabilization Strategies, Schemes, and Settings (p. 4533).

Initialize Stabilization

clears all of the unstable modes that have been identified in calculations using the stabilization scheme, but leaves the adjoint solution unchanged. This button is only available when the residual minimization scheme is enabled.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Number of Iterations

sets the number of iterations for the calculation. Note that if you are using a blended stabilization strategy, the calculation may end sooner if the total of the iterations you specified in the **Stabilized Strategy and Scheme Settings** dialog box (described in Stabilization Strategies, Schemes, and Settings (p. 4533)) is less than the value you enter here.

Calculation Activities

allows you to access dialog boxes so you can set up various tasks that can be performed during the adjoint calculation. This drop-down list provides the following selections:

Autosave...

opens the **Adjoint Autosave** dialog box. For details, see Automatic Saving of Case and Data Files During an Adjoint Calculation (p. 4540).

Execute Commands...

opens the Execute Commands Dialog Box (p. 5201) , which you can use to set up commands to be executed during the adjoint calculation (such as postprocessing commands that export fields). For details, see Executing Commands During the Calculation (p. 3700). Note that once you have opened this dialog box from **Run Adjoint Calculation** dialog box, the following selections become available in the **When** drop-down list:

Adjoint Iteration

This specifies that execution of the command is based on the adjoint iteration. It is executed immediately before each adjoint advancement step.

Design Iteration

This specifies that execution of the command is based on the design iteration. It is executed after the design calculation and before any animation recording, both of which are before the morphing of the mesh.

Note that unexpected behavior may result if the executed command changes settings for the adjoint calculation.

Calculate

advances the adjoint solver by the **Number of Iterations** specified in the adjacent field.

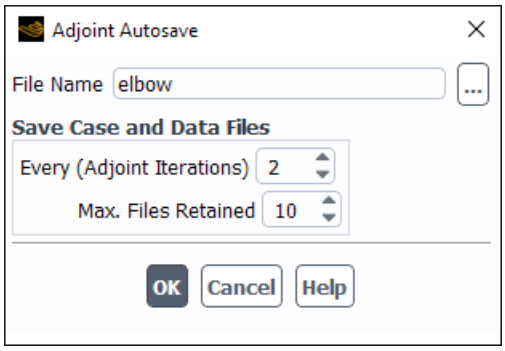
Depending on the monitor settings, the residuals may be printed in the console and/or plotted in the main graphics window.

48.2.3.5.1. Automatic Saving of Case and Data Files During an Adjoint Calculation

The **Adjoint Autosave** dialog box allows you to specify automatic saving of case and data files at specified intervals during an adjoint calculation. This dialog box is accessed by selecting **Autosave...** from the **Calculation Activities** drop-down list in the **Run Adjoint Calculation** dialog box (described in Running the Adjoint Calculation (p. 4539)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 48.10: Adjoint Autosave Dialog Box



The function of this dialog box is as follows:

File Name

specifies the root name for the files that are saved. The adjoint iteration and an appropriate suffix will be added to the specified root name; by default, the suffix will be `.cas.h5` or `.dat.h5`.

Save Case and Data Files

provides the following controls:

Every (Adjoint Iterations)

specifies the frequency (in adjoint iterations) with which case and data files are saved. The default value is set to `0` , indicating that no automatic saving is performed.

Max. Files Retained

specifies the maximum number of case and data files that can be retained. If you have constraints on the disk space, you can restrict the number of files to be saved using this field. After saving the specified number of files, Ansys Fluent will overwrite the earliest existing files. The default value for this field is `5` . Entering a value of `0` ensures that all of the files are retained.

48.2.4. Postprocessing of Adjoint Solutions

Adjoint solution data can be postprocessed so that it provides both qualitative and quantitative views of the effect of many types of change that may be imposed on a system. While shape changes are often of particular interest there is a very rich data set, one as large as the original flow field, available for exploration. Postprocessing tools are provided with the intention of permitting the adjoint data to be mined in such a way that it provides useful supporting information for an engineer who is making design decisions for a system, or has questions about the reliability of the original flow calculation.

Since the Ansys Fluent adjoint solver is a discrete adjoint solver, the primitive adjoint solution data provides sensitivity to changes cell-wise for the computational mesh. Normalization of these results by the cell volume provides a *mesh-independent* view into the data set.

Information regarding postprocessing the adjoint solutions can be found in the following sections:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

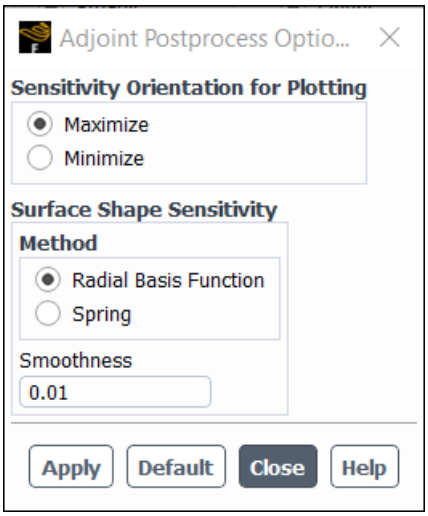
48.2.4.1. Using the Adjoint Postprocess Options Dialog Box
48.2.4.2. Field Data
48.2.4.3. Scalar Data

48.2.4.1. Using the Adjoint Postprocess Options Dialog Box

Settings for postprocessing adjoint sensitivity data is accomplished in the **Adjoint Postprocess Options** dialog box. This dialog box is accessed by clicking **Postprocess Options...** in the **Design** ribbon tab (**Gradient-Based** group box).

Design → **Gradient-Based** → **Postprocess Options...**

Figure 48.11: Adjoint Postprocess Options Dialog Box



The function of this dialog box is as follows:

Sensitivity Orientation for Plotting

provides the following controls:

Maximize

specifies changes in the direction of positive sensitivities to increase the value of the observable.

Minimize

specifies changes in the direction of negative sensitivities to decrease the value of the observable.

When defining **Sensitivity Orientation for Plotting**, a standard rule of thumb can be applied: if you want to improve the solution using the **Maximize** or **Minimize** settings, then follow the direction of the sensitivity vectors. Another standard rule of thumb can also be applied for postprocessed scalars: if you want to improve the solution, then increase values where the sensitivity is positive and/or decrease values where the sensitivity is negative.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Method

By default the **Radial Basis Function** is used to calculate and smooth the surface shape sensitivity. However, you can enable the **Spring** option to reduce memory and computational cost. The spring method assumes that the interior mesh motion is governed by the surface mesh motion, using the spring analogy equation. The adjoint of the spring equation is then solved to back propagate the interior shape sensitivity onto the surface, resulting in a physical shape sensitivty that is not smoothed. To smooth the shape sensitivty, Laplace surface smoothing is used based on the defined smoothness settings to achieve a smoothed surface shape sensitivity.

Smoothness

Specifies the smoothness of the **Sensitivity to Surface Shape** field, which is smoothed using either the radial basis function (when **Radial Basis Function** is enabled), or by solving an adjoint spring equation (when **Spring** is enabled). Decreasing this value will accentuate the effect of local geometric features on the **Sensitivity to Surface Shape** field.

48.2.4.2. Field Data

Adjoint solution data can be postprocessed using standard Ansys Fluent postprocessing tools including contours, vectors, xy-plots, histograms, and surface and volume integrals.

Under **Sensitivities...** (for example, in the **Contours** dialog box), you can find the following fields:

Magnitude of Sensitivity to Body Forces (Cell Values)

This field is the magnitude of the adjoint velocity primitive field. This field can be interpreted as the magnitude of the sensitivity of the observable to body force per unit volume. It can be used to identify regions in the domain where small changes to the momentum balance in the flow can have a large or small effect on the observable. This field is often observed to be large, for example, upstream of a body for which drag sensitivity is of interest, with the field diminishing in the upstream direction. This indicates the interference effect for an object positioned at various locations upstream of the object of interest.

Sensitivity to Body Force X-Component (Cell Values), Sensitivity to Body Force Y-Component (Cell Values), and Sensitivity to Body Force Z-Component (Cell Values)

These are the components of the adjoint velocity primitive field. These fields can be interpreted as the magnitude of the sensitivity of the observable to components of a body force per unit volume. Consider a body force distribution, expressed as a force per unit volume. The volume integral of the vector product of that distribution with the components of this field gives a first-order estimate of the net effect of the body force on the observation.

Sensitivity to Mass Sources (Cell Values)

This field is the primitive adjoint pressure field. This field can be interpreted as the sensitivity of the observable with respect to mass sources or sinks in the domain. Consider a mass source / sink distribution, expressed as mass flow rate per unit volume. The volume integral of that distribution, weighted by the local value of this field, gives the effect of the sources / sinks on the observation. When plotted on a boundary, this field indicates the effect of the addition or removal of fluid from the domain upon the quantity of interest. It is important to note that in this scenario the effect of the momentum of the fluid that

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

is added or removed is not taken into account. The boundary velocity sensitivity should be plotted if that effect is also of interest.

Sensitivity to Energy Sources (Cell Values)

This field is available when the energy adjoint equation is solved and is the primitive adjoint temperature field. It can be interpreted as the sensitivity of the observable with respect to the thermal energy sources or sinks per unit volume in the domain.

Sensitivity to Turbulent Kinetic Energy Sources (Cell Values)

This field is available when the turbulence adjoint equations are solved and is the sensitivity of the observable with respect to the turbulent kinetic energy (k) sources per unit volume in the domain.

Sensitivity to Specific Dissipation Rate Sources (Cell Values)

This field is available when the turbulence adjoint equations are solved and is the sensitivity of the observable with respect to the specific dissipation rate (ω) sources per unit volume in the domain.

Sensitivity to Scalar-n Sources (Cell Values)

This field is available when the specified UDS adjoint equations are solved and is the sensitivity of the observable with respect to the user-defined scalar sources per unit volume in the domain.

Sensitivity to *species-n* Sources (Cell Values)

This field is available when the specified species adjoint equations are solved and is the sensitivity of the observable with respect to the species sources per unit volume in the domain.

Artificial Dissipation

This field is available when the adjoint dissipation stabilization scheme is enabled and shows the location and amount of the nonlinear damping that has been introduced to damp the growth of instabilities that lead to adjoint solution divergence.

Sensitivity to Viscosity

This field shows the sensitivity of the quantity of interest to variations in the turbulent effective viscosity for a turbulent problem, or the laminar viscosity in a laminar case. The sensitivity is normalized by the cell volume to account for cell size variations in the mesh.

Shape Sensitivity Magnitude

This field is the magnitude of the sensitivity of the observable with respect to a deformation applied to the mesh (both boundary and interior mesh). When plotted on the surface of a body the locations where this quantity is large indicates where small changes to the surface shape can have a large effect on the observable of interest. If the shape sensitivity magnitude is small then the effect of shape changes in this region can be expected to have a small effect on the observable of interest. When viewing this field, it is often observed that the magnitude varies by many orders of magnitude. Contour plots will clearly draw

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

attention to regions with the highest sensitivity (often sharp edges and corners). However, it should be remembered that a relatively small surface movement that is distributed over a large area can have a cumulative effect that is large.

Normal Shape Sensitivity

This field shows the normal component of the shape sensitivity. A positive value indicates an orientation directed into the domain, while a negative value indicates that the shape sensitivity is oriented outwards from the domain. This field eliminates the component of the vector shape sensitivity field that lies in the plane of the wall.

Normal Surface Shape Sensitivity

This field shows the normal component of the surface shape sensitivity, which is an accumulation of surrounding nodal sensitivities to the surface. A positive value indicates an orientation directed into the domain, while a negative value indicates that the surface shape sensitivity is oriented outwards from the domain. The accumulation of surrounding nodal sensitivities produces a more accurate representation of the physical surface sensitivity when compared to the **Normal Shape Sensitivity** field which uses raw nodal sensitivity data. The normal surface shape sensitivity field is smoothed using the radial basis function and smoothness settings can be adjusted within the **Adjoint Postprocess Options** dialog box.

Normal Optimal Displacement

This field shows the normal component of the optimal displacement computed from the adjoint design tool calculation. This field is defined only for portions of walls lying within the control-volume specified for morphing. A positive value of displacement indicates that the surface will be displaced into the flow domain, whereas a negative value of displacement corresponds to wall movement outwards from the flow domain. This field eliminates the component of the optimal displacement vector that lies in the plane of the wall.

Optimal Displacement X-Component , Optimal Displacement Y-Component , Optimal Displacement Z-Component (in 3D)

These fields are the individual components of the optimal displacement computed from the adjoint design tool calculation. These fields are defined only for portions of walls lying within the control-volume specified for morphing.

log10(Shape Sensitivity Magnitude)

In view of the large range of values possible for the shape sensitivity magnitude, a convenience function that plots $\log_{10}$ of the magnitude is provided. This allows the importance of the surfaces in a domain to be ranked more easily based on how they affect the observation of interest when they are reshaped.

Shape Sensitivity X-Component, Shape Sensitivity Y-Component, and Shape Sensitivity Z-Component (in 3D)

These fields are the individual components of the sensitivity of the observable of interest with respect to the mesh node locations. It is plotted as cell data and is computed as the average of the nodal sensitivities for a given cell, divided by the cell volume. Note that for this discrete adjoint solver the sensitivity of the result with respect to node locations both

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

on and off boundaries is computed. The normalization by cell volume indicates that the fields that are plotted are the weighting factors for a continuous spatial deformation field. (Note that the nodal sensitivity data itself is used when mesh morphing is performed, and predictions about the effect of shape changes are made.)

Sensitivity to Boundary X-Velocity , Sensitivity to Boundary Y-Velocity, and Sensitivity to Boundary Z-Velocity (in 3D)

These fields are defined on those boundaries where a user-specification of a boundary velocity is made for the original flow calculation. This includes no-slip walls. The field shows how sensitive the observable of interest is to changes in the boundary velocity at any point. It is interesting to note that even though the original boundary condition specification may be for a uniform velocity on the domain boundary, the effect of a non-uniform velocity perturbation is available. The effect of any specific boundary velocity change can be estimated as an integral of the vector product of the change to the velocity with the plotted sensitivity field. A plot of this quantity on a velocity inlet, for example, can be very useful for assessing whether or not the inlet is positioned too close to key parts of the system. That is, it addresses the question of whether or not the flow domain is too small to achieve a successful computation of the performance measure of interest. Viewing this field will also indicate whether or not the assumption of a uniform inflow is adequate.

Sensitivity to Boundary Pressure

This field is defined on boundaries where there is a user-specified pressure as part of a boundary condition, such as on a pressure outlet. The field shows the sensitivity of the observation of interest to variations in the boundary pressure across the flow boundary. It is interesting to note that even though the original boundary condition specification may be for a uniform pressure on the domain boundary, the effect of a non-uniform pressure perturbation is available, The effect of any specific boundary pressure change can be estimated as an integral of the product of the change to the pressure with the plotted sensitivity field. Viewing this field will also indicate whether or not the assumption of a uniform pressure is adequate for the simulation.

Sensitivity to Boundary Temperature

This field is available when the energy adjoint equation is solved and is defined on boundaries where a temperature boundary condition is applied. This includes walls, velocity inlets, mass-flow inlets, pressure inlets, pressure far-field boundaries, and pressure outlets where a backflow temperature may be specified. The field shows the sensitivity of the observation of interest to variations in the boundary temperature across the boundaries. Note that even if the original boundary condition specification is for a uniform temperature on the boundary, the effect of a non-uniform temperature perturbation is available. The effect of any specific boundary temperature change can be estimated as an integral of the product of the change to the temperature with the plotted sensitivity field. This field can be used to indicate whether or not the assumption of a uniform temperature is adequate for the simulation.

Sensitivity to Boundary Heat Flux

This field is available when the adjoint energy equation is solved and is defined on walls where a heat flux boundary condition is imposed. The field shows the sensitivity of the observation of interest to variations in the boundary heat flux through the wall. Its properties are analogous to those of Sensitivity to Boundary Temperature.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sensitivity to Scalar-n Boundary Value

This field is available when the specified UDS adjoint equations are solved and is defined on boundaries where a user-defined scalar boundary condition with specified value is imposed. The field shows the sensitivity of the observation of interest to variations in the user-defined scalar boundary value across the flow boundary.

Sensitivity to Scalar-n Boundary Flux

This field is available when the specified UDS adjoint equations are solved and is defined on boundaries where a user-defined scalar boundary condition with specified flux is imposed. The field shows the sensitivity of the observation of interest to variations in the user-defined scalar boundary flux across the flow boundary.

Sensitivity to *species-n* Boundary Value

This field is available when the species adjoint equations are solved and is defined on boundaries where a species boundary condition with a specified mass fraction is imposed. The field shows the sensitivity of the observation of interest to variations in the species mass fraction across the flow boundary.

Sensitivity to *species-n* Boundary Flux

This field is available when the species adjoint equations are solved and is defined on boundaries where a species boundary condition with specified flux (mass) is imposed. The field shows the sensitivity of the observation of interest to variations in the species boundary flux (mass) across the flow boundary.

Sensitivity to Flow Blockage

This field is provided as a convenient tool for identifying portions of the flow domain where the introduction of blockages or obstructions in the flow can affect the observation of interest. Consider a blockage in the flow that generates a reaction force on the flow that is proportional to the local flow speed, and acting in the opposite direction to the local flow: $F=-\alpha(r)v$ where α is a local coefficient for the reaction force. The local contribution of this force on the observation of interest is determined by the vector product of this force with the adjoint velocity field. The flow blockage field that is plotted is $-v\cdot\tilde{v}$, namely the negative of the vector product of the flow velocity and the adjoint velocity (Cell Value).

Sensitivity to GEKO CSEP

This field is available when the turbulence adjoint equations are solved and is the sensitivity of the observable with respect to the C_{SEP} parameter, which is used to optimize flow separation from smooth surfaces.

Sensitivity to GEKO CNW

This field is available when the turbulence adjoint equations are solved and is the sensitivity of the observable with respect to the C_{NW} parameter, which is used to optimize flow in non-equilibrium near wall regions (such as heat transfer or C_f).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sensitivity to GEKO CMIX

This field is available when the turbulence adjoint equations are solved and is the sensitivity of the observable with respect to the C_{MIX} parameter, which is used to optimize strength of mixing in free shear flows.

Sensitivity to GEKO Blending Function

This field is available when the turbulence adjoint equations are solved and is the sensitivity of the observable with respect to the blending function, F_{Blend}, which is used to deactivate GEKO parameters inside boundary layers.

Sensitivity to CCURV

This field is available when the turbulence adjoint equations are solved for a case in which the Curvature Correction option is enabled in the **Viscous Model** dialog box. It is the sensitivity of the observable with respect to the curvature correction parameter, **CCURV** , which is used to optimize the strength of the curvature correction if needed for a specific flow.

Adjoint GEKO CSEP

This field is available after performing gradient-based turbulence model optimization within the **Gradient-Based Optimizer** and is the optimized C_{SEP} parameter, which is used to optimize flow separation from smooth surfaces.

Adjoint GEKO CNW

This field is available after performing gradient-based turbulence model optimization within the **Gradient-Based Optimizer** and is the optimized C_{NW} parameter, which is used to optimize flow in non-equilibrium near wall regions (such as heat transfer or C_f).

Adjoint GEKO CMIX

This field is available after performing gradient-based turbulence model optimization within the **Gradient-Based Optimizer** and is the optimized C_{MIX} parameter, which is used to optimize strength of mixing in free shear flows.

Adjoint GEKO BF

This field is available after performing gradient-based turbulence model optimization within the **Gradient-Based Optimizer** and is the optimized blending function (F_{Blend}), which is used to deactivate GEKO parameters inside boundary layers..

Adjoint GEKO CSEP (Trained)

This field is available after performing **Offline training** of the neural network turbulence model within the **Optimizer Design Variables** dialog box, and is the trained C_{SEP} parameter.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Adjoint GEKO CNW (Trained)

This field is available after performing **Offline training** of the neural network turbulence model within the **Optimizer Design Variables** dialog box, and is the trained C_{NW} parameter.

Adjoint GEKO CMIX (Trained)

This field is available after performing **Offline training** of the neural network turbulence model within the **Optimizer Design Variables** dialog box, and is the trained C_{MIX} parameter.

Adjoint GEKO BF (Trained)

This field is available after performing **Offline training** of the neural network turbulence model within the **Optimizer Design Variables** dialog box, and is the trained blending function parameter (F_{Blend}).

Non-Equilibrium Parameter

This field is available when the **Neural Network Model** has been applied for turbulence model optimization within the **Optimizer Design Variables** dialog box, and is the Non-Equilibrium Parameter flow feature.

Second Invariant

This field is available when the **Neural Network Model** has been applied for turbulence model optimization within the **Optimizer Design Variables** dialog box, and is the Second Invariant flow feature.

Third Invariant

This field is available when the **Neural Network Model** has been applied for turbulence model optimization within the **Optimizer Design Variables** dialog box, and is the Third Invariant flow feature.

Fourth Invariant

This field is available when the **Neural Network Model** has been applied for turbulence model optimization within the **Optimizer Design Variables** dialog box, and is the Fourth Invariant flow feature.

Fifth Invariant

This field is available when the **Neural Network Model** has been applied for turbulence model optimization within the **Optimizer Design Variables** dialog box, and is the Fifth Invariant flow feature.

Length Ratio

This field is available when the **Neural Network Model** has been applied for turbulence model optimization within the **Optimizer Design Variables** dialog box, and is the length ratio flow feature.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Turbulent Reynolds Number (Scaled)

This field is available when the **Neural Network Model** has been applied for turbulence model optimization within the **Optimizer Design Variables** dialog box, and is the scaled turbulent reynolds number flow feature.

Turbulent Viscosity Ratio (Scaled)

This field is available when the **Neural Network Model** has been applied for turbulence model optimization within the **Optimizer Design Variables** dialog box, and is the scaled turbulent viscosity ratio flow feature.

Mahalanobis distance

This field is available when the **Neural Network Model** is selected for turbulence model optimization within the **Optimizer Design Variables** dialog box. Large values for the **Mahalanobis distance** suggest large deviations in flow features compared to the training data results.

Adjoint Local Solution Marker

This field intended for expert users, can be plotted to identify those portions of the flow domain where the stabilized adjoint solution advancement scheme is applied. It is preferable to plot this with the **Node Values** disabled in the **Contours** dialog box. In this case, the **Adjoint Local Solution Marker** will take a value between 0 and 1. The `Mode Amplitude Cutoff` defined through the `adjoint/controls/stabilization` text command defines the lower bound for cells where the stabilized scheme is applied.

Adjoint X-Velocity Residual

This field is available for users using the adjoint diagnosis and is the x-component of the residual of the adjoint momentum equation in each cell.

Adjoint Y-Velocity Residual

This field is available for users using the adjoint diagnosis and is the y-component of the residual of the adjoint momentum equation in each cell.

Adjoint Z-Velocity Residual

This field is available for users using the adjoint diagnosis and is the z-component of the residual of the adjoint momentum equation in each cell.

Adjoint Continuity Residual

This field is available for users using the adjoint diagnosis and is the residual of the adjoint continuity equation in each cell.

Adjoint Local Flow Rate Residual

This field is available for users using the adjoint diagnosis and is the residual of the adjoint local flow rate equation in each cell.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Adjoint Energy Residual

This field is available for users using the adjoint diagnosis and is the residual of the adjoint energy equation in each cell.

Adjoint Turbulent Kinetic Energy Residual

This field is available for users using the adjoint diagnosis and is the residual of the adjoint turbulent kinetic energy equation in each cell.

Adjoint Specific Dissipation Rate Residual

This field is available for users using the adjoint diagnosis and is the residual of the adjoint specific dissipation rate in each cell.

The surface shape-sensitivity fields contain the derivatives of the chosen observable with respect to the position of the walls of the problem.

In the **Vectors** dialog box, under **Vectors of**, you can find the following custom vector fields:

Sensitivity to Body Forces (Cell Values)

This vector field shows how sensitive the observable of interest is to the presence of body forces within the flow. A body force that locally has a component in the direction of the body-force sensitivity vector will lead to an increase in the observation of interest. The larger the sensitivity vector, the larger the effect that a body force can have on the observation of interest. The first-order effect of a body force distribution per unit volume can be computed as a volume integral of the vector product of the body force distribution with this field.

Sensitivity to Shape

This vector field shows the postprocessed adjoint solution, independent of any morphing settings, that defines the sensitivity of the observable of interest to movement of the walls. Deformation of a wall where this vector is large can be expected to have a significant effect on the observable. In contrast, deformations of the wall in locations where this vector is small in magnitude, to first order, will have no effect on the observable.

Sensitivity to Surface Shape

This vector field shows the postprocessed adjoint solution, independent of any morphing settings, and accumulates the sensitivites of surrounding nodes to the surface. The accumulation of surrounding nodal sensitivities produces a more accurate representation of the physical surface sensitivity when compared to the **Sesnitivity to Shape** vector field which uses raw nodal sensitivity data. Surface shape sensitivity is smoothed using the radial basis function and smoothness can be adjusted within the **Adjoint Postprocess Options** dialog box.

Optimal Displacement

The optimal displacement field is defined once a design change has been computed using the Design Tool. This vector field shows the optimal displacement based on the design goals that have been specified and any prescribed boundary motions. Note that when plotting this vector field, if the **Auto Scale** option is deselected, and the vector **Scale** option is set to 1, then the vectors as shown display the absolute displacement that will occur.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sensitivity to Boundary Velocity

This sensitivity field is defined on boundaries where velocities are specified as an input for the boundary condition. This includes no-slip, and slip walls. When plotted, this field illustrates where a change to the boundary velocity can affect the observation of interest. When plotted on a wall, it shows how the imposition of a nonzero velocity condition on the wall will affect the observation. This can include movement in the plane of the wall itself. When plotted, for example on a velocity inlet, this field illustrates how local adjustments to the velocity of the incoming flow affect the observation of interest. Integrating a potential change to the velocity field, weighted by this sensitivity field over the boundary provides a first order estimate of the effect of the change on the observation of interest.

Sensitivity to Boundary Mass Flux

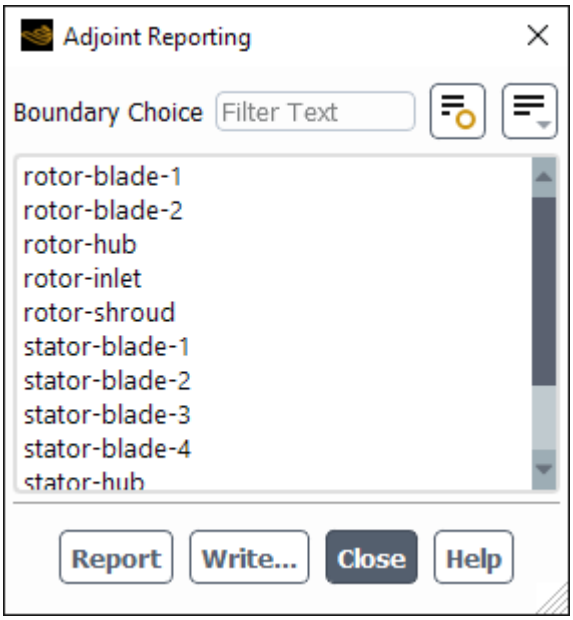
This sensitivity field that can be used to visualize the effect of adding or removing fluid via a wall-normal jet. This can be used, for example, to identify regions on walls where the introduction of suction could have a significant effect on the observable that has been specified. A plot of the **Sensitivity to Boundary Velocity** in general provides a richer view, but sometimes at the expense of the clarity of the visualization.

48.2.4.3. Scalar Data

The computed adjoint solution can provide sensitivity information for individual boundary condition settings through the **Adjoint Reporting** dialog box. This dialog box is accessed by clicking **Reporting...** in the **Design** ribbon tab (**Gradient-Based** group box).

Design → Gradient-Based → Reporting...

Figure 48.12: Adjoint Reporting Dialog Box



Boundary condition sensitivity data is retrieved as follows:

- Select a single boundary of the problem in the **Boundary Choice** selection box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Click **Report** to see the boundary condition settings and the sensitivity of the observable with respect to those settings. An example of a report is shown below:

```
Velocity inlet id 5
Velocity magnitude = 4.000000000e+01 (m/s),
Sensitivity = 5.429127866e+01 ((n)/(m/s))
```

  In this case, the report shows that for every change of the boundary velocity by 1 m/s, a change in the predicted observable (in this case a force) of 54.2 N is expected. This prediction is based on a linear extrapolation.

- Click **Write...** to write the report for the selected boundary to a file.

48.2.5. Modifying the Geometry Using the Design Tool

Tools to modify the boundary and interior mesh of the problem based on the adjoint solution are provided in the **Design Tool** dialog box. The **Design Tool** uses computed adjoint sensitivity data to find an optimal design change satisfying one or more goals or constraints, and to predict the resulting changes in observable values. Among the goals that can be specified in the **Design Tool** are:

- minimization, maximization, or targeted changes of single or multiple observable values
- prescribed forms of deformation for selected geometry
- general constraints on deformations including restrictions on direction of motion, symmetry, and boundary conditions.
- equality and/or inequality constraints on geometry (fixed walls, bounding boxes, and so on).

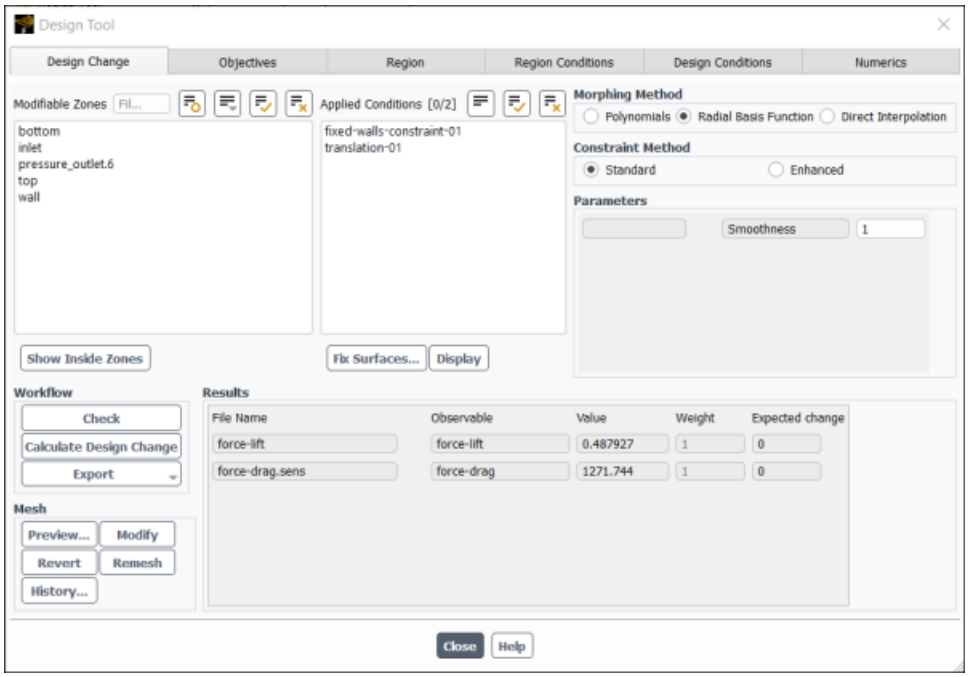
The general procedure to use the Design Tool is as follows.

1. Open the **Design Tool** by clicking **Design Tool...** in the **Design** ribbon tab (**Gradient-Based** group box).

   Design → **Gradient-Based** → **Design Tool...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 48.13: Design Tool Dialog Box



2. Select the appropriate **Morphing Method** and **Constraint Method** for your problem.

   If your problem contains some design constraints such as fixed zones and bounded-by-surface conditions, the **Enhanced Constraint Method** with the **Polynomials Morphing Method** or **Radial Basis Function** is recommended. The **Enhanced** constraint method can provide a better balance during mesh morphing compared to the **Standard** constraint method in terms of efficiency and accuracy when handling design conditions. Additionally, the **Enhanced** constraint method can produce a better quality mesh than the **Standard** constraint method and setup of the design tool is greatly simplified.

   Note:

   The **Enhanced Constraint Method** is only available for the **Radial Basis Function** and **Polynomials** morphing methods.

   Prior to using the **Enhanced** constraint method, you must ensure that there are no limitations when using this method for your problem as outlined in Smoothing and Mesh Morphing (p. 4506).

   When **Standard** is selected for the **Constraint Method**:

   - **Radial Basis Function** is generally recommended for the **Morphing Method** if you prefer mesh quality and have some design conditions.
   - If no design conditions are present, including no zones intended to stay fixed in the morphing region, **Polynomials** is recommended due to its computational speed and good mesh quality.
   - **Direct Interpolation** is recommended if you have many design conditions.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For details on how they calculate the displacement of the mesh nodes and their relative merits, see Smoothing and Mesh Morphing (p. 4506).

3. Select those boundaries that are free to be deformed in the **Modifiable Zones** field on the **Design Change** tab. Boundaries that are not selected in this box that intersect the control volume will be constrained not to move. This is useful if there are walls which, for design reasons, must not move.

   Note:

   - If you have a two-sided wall that you want to deform, it is recommended that you select both the wall and the corresponding wall-shadow.
   - When using the **Direct Interpolation** morphing method, you must ensure that none of the zones involved in a non-conformal mesh interface are selected.

4. Define the region that will be modified in the **Region** tab. (Defining the Region for the Design Change (p. 4556))

5. Define conditions on the deformation region such as number of control points, symmetry, directional invariance, and boundary continuity. (Defining Region Conditions (p. 4558))

6. Export the sensitivities from the adjoint solution for each observable of interest. (Defining Observable Objectives (p. 4559))

7. Import the sensitivity data for each observable that will participate in the design change and specify the optimization goals for the observables. (Defining Observable Objectives (p. 4559))

   Note:

   Export and Import of sensitivity data is only required for multi-objective analysis.

8. Define constraining and/or deformation conditions on the design change, such as fixed boundaries, prescribed forms of boundary deformation (through a profile, translation, rotation, scaling, or rigid body motion), and bounding surfaces / planes. (Defining Conditions for the Deformation (p. 4560))

9. (optional) Adjust the optimal design calculation numerics if required. (Design Tool Numerics (p. 4570))

10. Specify the design conditions to be applied (and whether they are strictly enforced), the weights for the various objectives, and the deformation parameters; then compute the optimal design change and modify the mesh. (Shape Modification (p. 4575))

Details of these steps are elaborated in the following sections:

48.2.5.1. Defining the Region for the Design Change

48.2.5.2. Defining Region Conditions

48.2.5.3. Defining Observable Objectives

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

48.2.5.4. Defining Conditions for the Deformation

48.2.5.5. Design Tool Numerics

48.2.5.6. Shape Modification

Note that if you are setting up the **Design Tool** in preparation to use the **Gradient-Based Optimizer**, you will need to perform most—but not necessarily all—of the previous steps; for details, see Using the Gradient-Based Optimizer (p. 4583).

48.2.5.1. Defining the Region for the Design Change

To define the region of space where a design change is to be applied, perform the following steps:

1. Click the **Region** tab of the **Design Tool** dialog box.

2. Make a selection from the **Region Geometry** drop-down menu to specify whether you will define the space as either a **Cartesian** or a **Cylindrical** region type.

3. The **Auto Display** option is enabled by default and automatically displays the morphing region in the graphics window when **Region Extent** settings are changed or when **Symmetric** options are changed within the **Region Conditions** tab. When **Auto Display** is enabled it may be computationally expensive for large cases. When the **Auto Display** option is disabled, changes to the morphing region will not be displayed in the graphics window until you click the **Update Region** button.

4. The **Show Bounding Region** option is enabled by default, so that you can view the region in the graphics window as it is defined.

5. The simplest way to set up the bounding region is to click the **Get Bounds...** button. In the **Bounding Region Definition** dialog box that opens, you can select the **Zones to Be Bounded** and click **OK** to define a region that encompasses those zones. The following options are enabled in the **Bounding Region Definition** dialog box by default:

   - The **Comfortable Region** option specifies that the region is slightly larger than selected zones. You can disable this option if you want the region to exactly adhere to the extents of the selected zones.

   - For the **Cylindrical** region type, the **Automatic Coordinate** option defines the origin and axis of the region to match those defined for the selected zones (if possible). This is helpful when a selected zone has been defined as rotationally periodic, or is adjacent to a cell zone that has the **Frame Motion** option enabled as part of a rotating reference frame problem.

   If necessary, you can then click the **Larger Region** and/or **Smaller Region** buttons to adjust the size of the region.

6. If you need more precise control over the definition of the region, you can enter values directly for the settings.

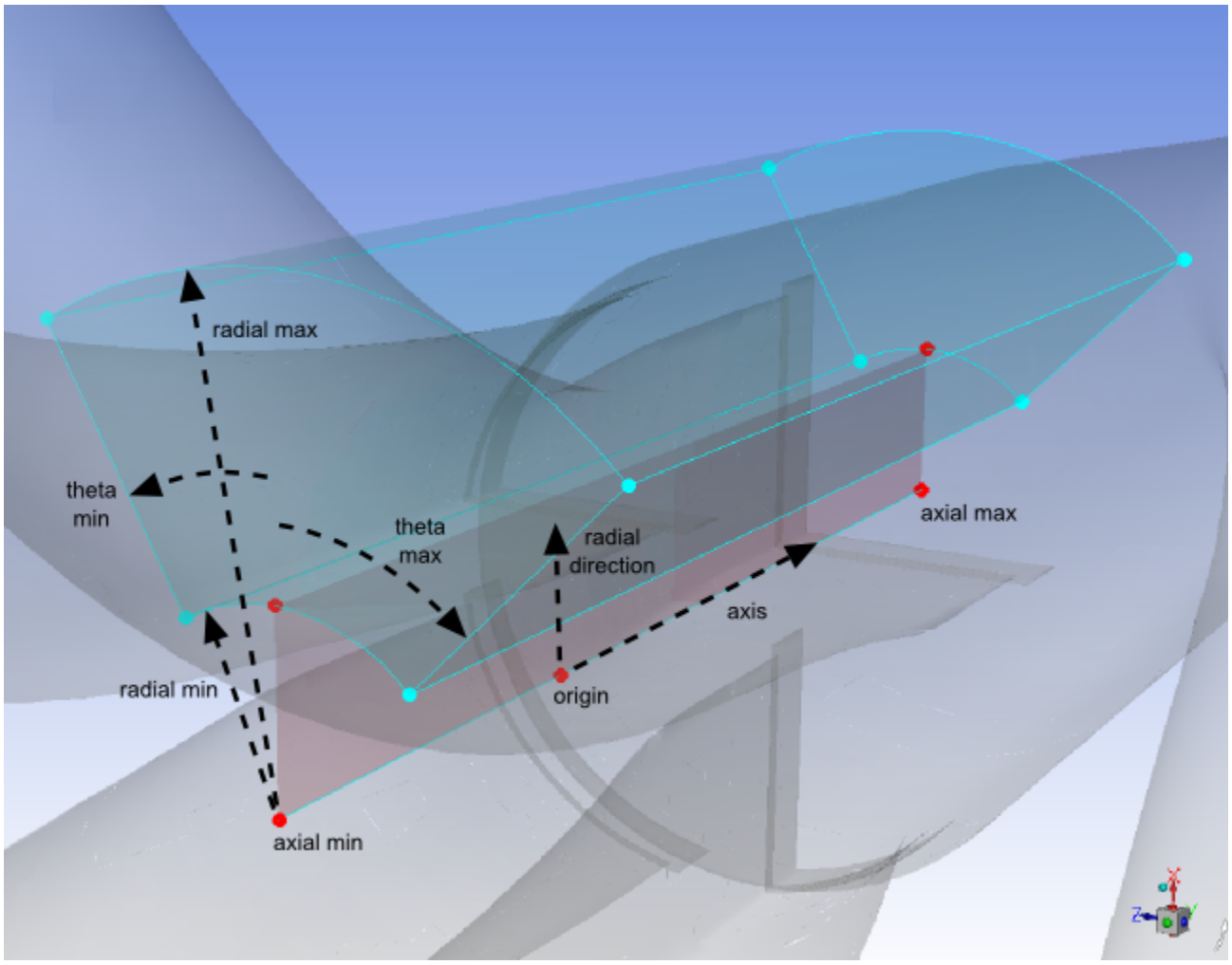
   For the **Cartesian** region type, you can specify the **X Min**, **X Max** , **Y Min**, and **Y Max** (and **Z Min** and **Z Max** in 3D) fields.

   For the **Cylindrical** region type, you can specify the angular, radial, and (for 3D) axial coordinate ranges; this will require you to define a **Coordinate System**, and then enter the values of the

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Region Extent relative to that system. The fields you will define are illustrated in Figure 48.14: A Cylindrical Region (p. 4557).

Figure 48.14: A Cylindrical Region



Note the following for **Cylindrical** regions:

- For 2D cases, the axis is defined implicitly as the Z-axis (as determined by the right-hand rule, where in a front view the X-axis is horizontal and the Y-axis is vertical).
- For the angular range (defined by **Theta Min** and **Theta Max**), the positive direction is determined by the right-hand rule about the axis, and the total range must not exceed 360 degrees.
- For 3D cases, the axis and radial direction must not be co-linear.
- Radial symmetry is not possible.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

After you have entered the settings, click **Update Region** to set the values and update the display.

Note:

Symmetry conditions affect the location and size of the region:

- If you plan to enable the **Symmetric** options in the **Region Conditions** tab (as described in Defining Region Conditions (p. 4558)), the region should be positioned with forethought given to the effect on the various boundary zones in the region.
- If you have a boundary zone of type **symmetry** in the region, it is recommended that it either is located on a planar boundary of the region, or is centered within the region in a manner that is consistent with the **Symmetric** options in the **Region Conditions** tab.

48.2.5.2. Defining Region Conditions

To define the region conditions, perform the following steps:

1. Click the **Region Conditions** tab in the **Design Tool** dialog box.
2. Specify the control points and motion conditions for each coordinate direction.
   a. When using the polynomials-based morphing method, select the number of control **Points**.

      The control points are distributed uniformly within each coordinate range (in the X, Y, and Z directions for a Cartesian region, and the Theta, Radial, and Axial directions for a cylindrical region). The spacing between the control points defines the characteristic spatial scale for the deformation operation. The larger the number of control points, the smaller the spacing and hence, the smaller the spatial scale on which changes can be made. 20 to 40 control points for each coordinate direction is typical.

      Note that the control points are displayed in red along representative directions of the region in the graphics window, in order to allow you to visually determine if the number you have specified is appropriate.

   b. If you want to prevent deformation in the given coordinate direction, disable **Motion Enabled**.

   c. When using the polynomials-based morphing method, you can specify that the X, Y, and Z motion is uniform with respect to the different coordinate directions by enabling the **Invariant** options. This invariance can be imposed in as many directions as desired. If you want greater control of this invariance (for example, you might only want the X motion to be uniform with respect to a particular direction), you can enable the **Show More** option at the top of the tab, and additional options will be available in a group box below the **Invariant** option.

   d. You can ensure that the deformation is symmetrical relative to the different coordinate directions by enabling the **Symmetric** options. Symmetry can be imposed in as many directions as desired and when enabled, the symmetry planes will be displayed in the

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

graphics window. You should ensure they reflect your intent for the design change, modifying the extents of the deformation region as necessary.

e. When using the radial basis function or direct interpolation morphing methods, periodic morphing can be enforced in the theta direction for cylindrical region conditions by entering the desired number of repeats under **Periodic** . It is recommended that the morphing region be `360` degrees in the theta direction so that no continuity condition needs to be enforced. Note that periodic morphing is distinct from periodic boundary conditions and periodic repeats. For periodic morphing, when the desired number of repeats is `0` or `1`, no repeats will be applied.

3. The **Region Boundary Continuity** settings control the order of continuity of the deformation process at the boundaries of the deformation region. This is of particular relevance when the boundary of the deformation region intersects a wall that will be deformed. When using the polynomials-based morphing method, the **Continuity Order** controls the smoothness of the transition from the undeformed wall lying outside the deformation region to the deformed wall within. A value of 1 ensures first-order continuity at the transition, that is, a continuous first derivative of the deformation operation. A value of 2 ensures second-order continuity, and so on.

   By default, the continuity **Definition** is **Uniform** and is enforced at all deformation region boundaries. Alternatively, you can choose **By Boundary** to use different continuity conditions at each boundary. For each boundary (**X-Min** , **X-Max**, and so on) you can specify what order continuity to enforce for each motion direction. You can also specify **In-Plane Motion Only** on that boundary; note that this option may be useful if a deformation region boundary coincides with a boundary zone that is of type **symmetry**.

   The continuity settings will not affect the transition between fixed and movable boundaries within the deformation region, only mesh nodes in the transition zone from inside to outside the deformation region.

   Note that when using the radial basis function or direct interpolation morphing method, you do not have the options described above. However, the **Apply Continuity** option is enabled by default and is recommended to ensure that the nodes at the boundary of the region are fixed.

48.2.5.3. Defining Observable Objectives

The Design Tool computes the optimal mesh deformation to satisfy multiple simultaneous design goals. Among these can be goals for multiple observables, possibly at multiple flow conditions. Once you have obtained the adjoint solutions and displacement sensitivities for the various observables of interest, you can export the current sensitivity data. Additionally, you can import previously saved sensitivity data and specify the design objectives for each.

1. Click the **Objectives** tab in the **Design Tool** dialog box.

2. To export the current sensitivity data in the morphing region, click the **Manage Data...** button and then click the **Export...** button in the **Manage Sensitivity Data** dialog box that opens. The exported file contains the sensitivities of the observable for the mesh nodes within the specified deformation region. The deformation regions for the various observables can be dif-

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ferent, in principle. However, it is advisable that there is as much overlap as possible for observables that will be used together in a multi-objective design process.

3. To import previously saved sensitivity data, click **Manage Data...**. Then click the **Import...** button in the **Manage Sensitivity Data** dialog box that opens to load previously saved sensitivity files. Close the **Manage Sensitivity Data** dialog box when done.

4. If you want to include computed sensitivity data in the current session that you have not exported yet enable **Include Current Data**.

5. For each observable objective you can specify the following.

   Objective

   Determines how Fluent will attempt to change the value of the observable through the design change. You can choose to specify a target change in value, which is the recommended approach, or you can choose to increase or decrease the value of the observable optimally. You can also select **None** in which case the observable will not be considered in computing the optimal design change, but the predicted change in observable value will still be reported.

   Target/Reference Change

   When the chosen **Objective** is **Target Change in Value**, this field corresponds to the desired change in the value. When the chosen **Objective** is **Maximize** or **Minimize**, the magnitude of this field is a reference weighting for the observable. This can be used to normalize the objectives in cases where you have multiple observables of vastly differing magnitudes, thereby allowing the values entered for **Weight** in the **Design Change** tab to all be of similar scale. You can check **As Percentage** to specify the **Target/Reference Change** as a percentage of the value of the observable.

48.2.5.4. Defining Conditions for the Deformation

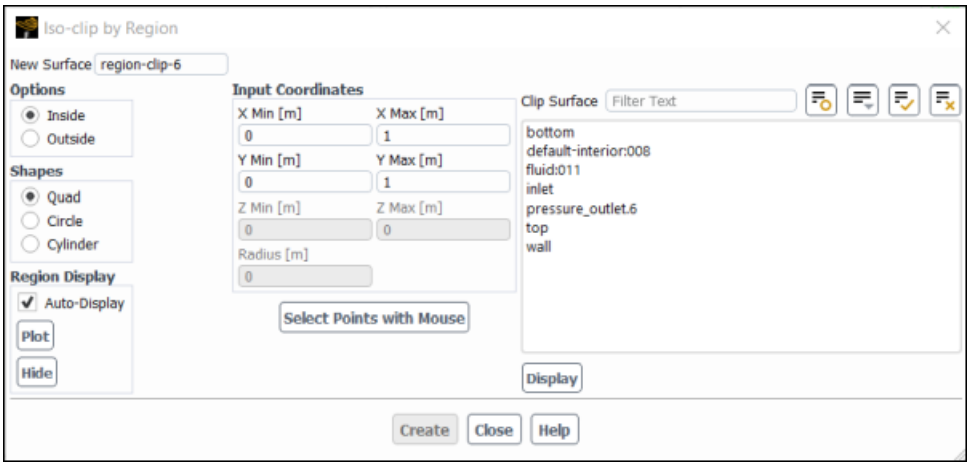
In the **Design Conditions** tab you can create new design conditions to be applied on the mesh deformation, or manage existing design conditions. New surfaces can also be created for defining new design conditions.

You can click the **New Surface** button to create a new point (**Point...**), clip-surface (**Iso-Clip...**), clip-surface by region (**Iso-Clip by Region...**), or expression volume (**Expression-Volume...**) for a design condition.

Selecting **Iso-Clip by Region...** will open the Iso-Clip by Region dialog box. Note that the **Iso-Clip by Region...** surface can only be created from within the design tool.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 48.15: Iso-clip by Region dialog box



Within the **Iso-clip by Region** dialog box, you can define settings to clip a surface within a specific region, as follows:

1. Select the surfaces to be included within the custom region from the **Clip Surface** list. After selecting the desired surfaces, you can click the **Display** button to display them in the graphics window.

2. Within the **Shapes** group box, you can select the desired shape of the custom region. Note that for 2D cases, **Quad** and **Circle** will be listed for the first and second shape options, respectively. However, for 3D cases, the first and second shape options will be listed as **Hex** and **Sphere**, respectively.

3. Within the Region Display group box, turning off the **Auto-Display** option will disable any changes made to the region from displaying in the graphics window. Clicking the **Plot** button will display the current region in the graphics window, while clicking the **Hide** button will remove the display of the current region in the graphics window.

4. Within the Input Coordinates group box, define the region. Note that you can click the **Select Points With Mouse** button to define the region by selecting points within the graphics window, or enter them manually as follows:

   - For the **Quad** shape type, you can specify the **X Min**, **X Max** , **Y Min**, and **Y Max** (and **Z Min** and **Z Max** for the **Hex** shape type) fields.

   - For both the **Circle** and **Sphere** shape types specify the origin for the X and Y axes (**X Center** and **Y Center**), then specify the circle/sphere **Radius**. Additionally, for the **Sphere** shape type, specify the Z-axis origin (**Z Center**), then define the maximum coordinate points for the X, Y, and Z axes (**X Max**, **Y Max**, and **Z Max**).

   - For the **Cylindrical** shape type, specify the axial coordinate ranges for the X-axis (**X-Axis Min** and **X-Axis Max**), Y-axis (**Y-Axis Min** and **Y-Axis Max**), and, for 3D cases, the Z-axis (**Z-Axis Min** and **Z-Axis Max**). Then specify the cylinder **Radius**.

5. Within the **Options** group box, choose whether to include only the portion of the selected clip surfaces located inside the specified region, or located outside the specified region, by enabling **Inside** or **Outside**, respectively.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. Click **Create** to save the Iso-Clip by Region.

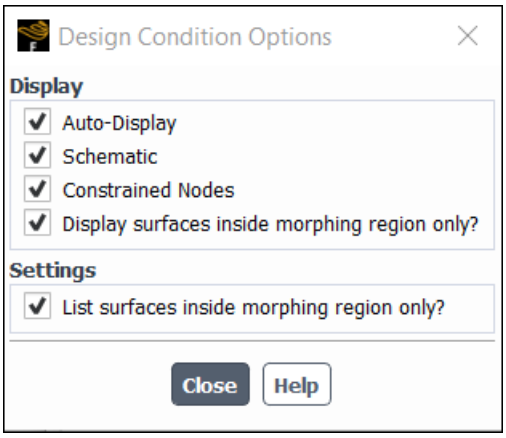
In the **Design Conditions** tab, you can create new conditions (design conditions) on the mesh morphing, such as fixing portions of wall surfaces, prescribing the deformation of surfaces, or setting up bounding planes / surfaces. To create a condition, click **Create...**, select the type of condition in the **Create New Condition** dialog box (you can choose between **fixed-walls**, **bounded-by-plane**, **bounded-by-surfaces**, **prescribed-profile**, **rotation**, **translation**, **scaling**, **rigid-body**, and **compound**), and provide a unique **Name**. You can then define the condition by selecting it in the **Defined Conditions** list on the left, defining the settings (as described in the steps that follow), and clicking **Apply**. If necessary, you can **Rename** a selected condition using the **Rename Constraint** dialog box that opens. Once created, the design conditions can be individually included or excluded from the design change in the **Design Change** tab.

At any time you can click the **Display** button so you can view the surfaces of the selected design condition in the graphics window. The type of design condition is indicated by the color of the surface. Clicking the **Options...** button opens the **Design Condition Options** dialog box, where you can set the following additional options:

- The **Auto-Display** option specifies that the graphics window automatically updates whenever a design condition is created or selected, as well as when the **Apply** button is clicked.
- The **Schematic** option specifies that settings for the design conditions are also displayed in the graphics window, including origins, axes, planes, and normals.
- When using the **Polynomials** morphing method with the **Standard Constraint Method**, design conditions are not applied to every node, but instead to a subset of nodes distributed throughout the selected surfaces; this is to avoid overly constraining the zone. The **Constrained Nodes** option in the **Design Condition Display Options** dialog box allows you to highlight the nodes of this subset. This can help you determine if the number of control **Points** defined in the **Region Conditions** tab / size of the region is appropriate for the surfaces involved in your design conditions; if the density of constrained nodes is not sufficient, you should either increase the number of controls points or reduce the size of your region.
- When the **Display surfaces inside the morphing region only?** option is enabled, clicking the **Display** button within the **Design Conditions** tab will only display the surfaces selected in the **Surfaces** list that are contained within the morphing region.
- Enabling the **List surfaces inside morphing region only?** option will automatically hide any surfaces listed in the **Surfaces** list that are not contained within the specified morphing region. This option can be useful for filtering out any unrelated surfaces. However, note that this option may cause lagging within the GUI, due to the additional operations required to check if surfaces exist within the morphing region.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 48.16: The Design Condition Options Dialog Box



Note that the adjoint solver's prediction range is reduced when you include design conditions—especially **prescribed-profile** , **rotation**, **translation** , **scaling**, and/or **rigid-body** conditions—so you should attempt smaller deformations (for example, when setting the **Target/Reference Change** in the **Objectives** tab).

Important:

A given surface cannot have multiple conditions of type **prescribed-profile**, **rotation** , **translation**, **scaling**, or **rigid-body**.

Fixed Walls

The **fixed-walls** constraint can be applied to both boundary zones and clip-surfaces of walls in the deformation region. This offers a fine-grained control over parts of the geometry that should not move. Whereas deselecting a zone in the **Modifiable Zones** list on the **Design Change** tab will fix an entire wall zone, selecting a clip-surface from in the **Fixed Walls Condition** list will fix only the portion of the wall that forms the clip-surface.

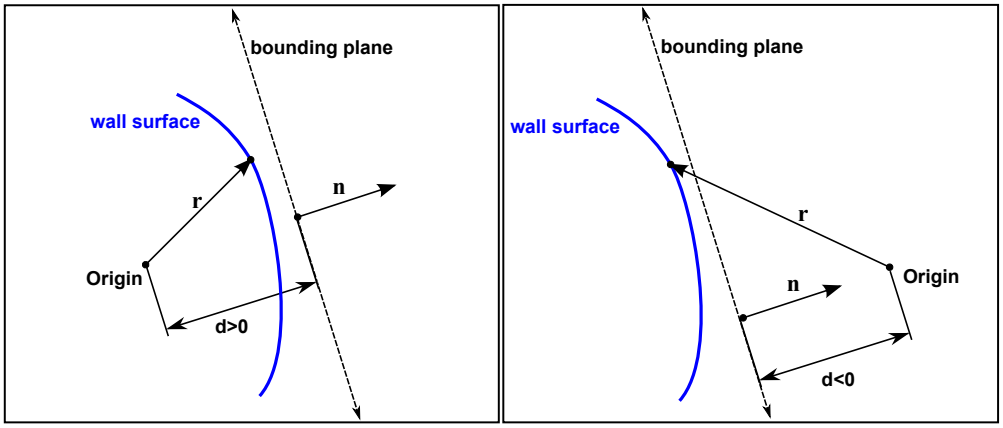
Any clip surfaces you have defined will appear in the **Surfaces** list. You can assign either a boundary zone or clip surfaces to a fixed walls constraint by selecting them in the list and clicking **Apply** .

Bounding Planes

The **bounded-by-plane** condition allows you to define one or more bounding planes that constrain the geometry deformation in such a way that the geometry cannot cross the given planes, thereby respecting packaging or form-factor constraints. The bounding plane is characterized by a vector normal to the plane and a distance measured from the global coordinate system origin to the plane, along the normal direction. The motion of the selected wall surface(s) will be constrained such that all points on the surface(s) lie on the side of the bounding plane that is opposite to the normal direction.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 48.17: Specifying a Bounding Plane for Design Changes



In Figure 48.17: Specifying a Bounding Plane for Design Changes (p. 4564) the bounding plane normal vector is denoted by $\mathbf{n}$ and the distance is denoted by d. $\mathbf{r}$ is the position vector of a point on the wall surface subject to the bounded-by-plane condition. Note that a positive value for the distance yields a bounding plane that is oriented with its normal pointing away from the origin, while a negative value for the distance yields a bounding plane oriented with its normal direction pointing towards the origin. The constraint equation can be expressed as:

$$\mathbf{r} \cdot \frac{\mathbf{n}}{|\mathbf{n}|} \leq d$$

1. Select the surfaces (or clip-surfaces) that are subject to the bounding plane condition in the **Bounded Surfaces** list.
2. Enter the components of the bounding plane normal direction.
3. Enter the distance of the plane from the origin (along the normal direction).
4. Click **Apply**.

Bounding Surfaces

The **bounded-by-surfaces** condition allows you to import one or more bounding surfaces that constrain the geometry deformation in such a way that the geometry cannot cross the nearest facet of the bounding surfaces, thereby respecting packaging or form-factor constraints; alternatively, you can prescribe that the geometry deforms to conform to the bounding surfaces. The bounding surfaces can be read from an `.stl`, `.msh`, or `.cas` file. The deformation of the selected bounded surfaces will be constrained such that all points on the surfaces lie on the sides of the bounding surfaces that have a positive orientation.

1. Select the surfaces (or clip-surfaces) that are subject to the bounded-by-surfaces condition in the **Bounded Surfaces** list.
2. Use the **Read...** button to import a `.stl`, `.msh`, or `.cas` file that contains the appropriate bounding surface(s). Note that the file you read must have the same dimensionality (2D or 3D) as the working case file.
3. Select the **Imported Surfaces** that will constrain the bounded surfaces.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

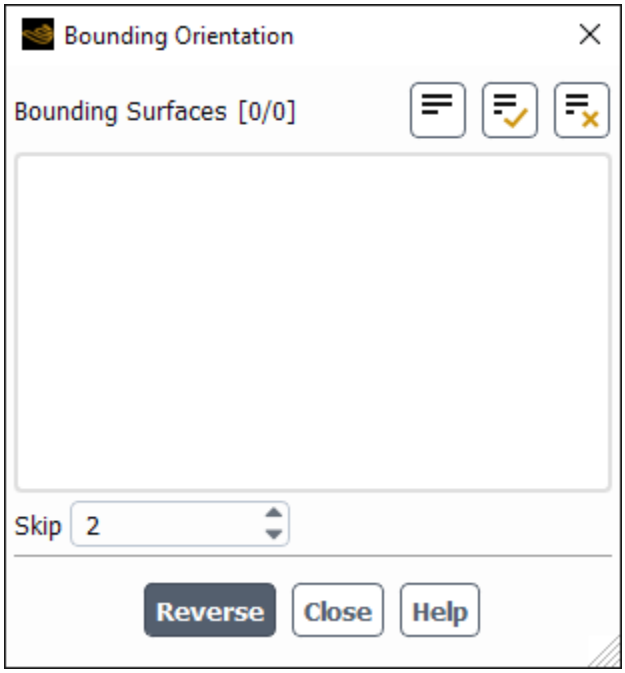
4. To visually confirm that the selected **Bounded Surfaces** and **Imported Surfaces** are appropriate, click the **Display** button. The imported surfaces will be colored green by default.

   Note:

   When examining the surfaces, it may be helpful to revise display properties of particular surfaces (for example, the color, the level of transparency, and the visibility of the mesh edges or faces). This can be done using the **Scene Description** dialog box, as described in Advanced Scene Composition (p. 4042). Note that you can even revise the display properties of the imported surfaces, the names of which will be displayed with the prefix **bounding-**.

5. When constraining the **Bounded Surfaces**, you should ensure that they are constrained on the correct side of the **Imported Surfaces** (that is, the side that has a positive orientation) by clicking the **Orientation...** button. The selected **Imported Surfaces** will be displayed, with their orientation indicated by arrows. In the dialog box that opens (Figure 48.18: The Bounding Orientation Dialog Box (p. 4565)), you can select individual **Bounding Surfaces** and **Reverse** their orientation as necessary; you can also increase the **Skip** value to display fewer orientation arrows, for cases where the arrow density obscures the orientation. Note that the orientation you define for a particular bounding surface will be used globally for all **bounded-by-surfaces** conditions.

   Figure 48.18: The Bounding Orientation Dialog Box



   When reviewing the orientation of a bounding surface, you should verify that the facets have a uniform orientation, that is, all of the arrows are on the same side of the surface. Bounding surfaces with facets that have a mixed orientation can affect the convergence and/or accuracy of the calculation.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. You have the option of specifying a **Bounding Offset** that is enforced between the deforming and bounding surfaces to ensure that the two surfaces are never coincident. The value you enter must be relatively small to avoid unexpected bounding.

7. To prescribe that the **Bounded Surfaces** deforms so as to conform to the **Imported Surfaces**, enable the **Fit to the imported surfaces?** option. This is only recommended when the bounded and imported surfaces are very similar in shape and separated by a small distance (on the order of the local cell dimensions). Note that when the **Fit to the imported surfaces?** option is enabled, any previously defined **Bounding Offset** is ignored.

8. When **Fit to the imported surfaces?** is enabled, define the **Scale Factor** to scale the displacement. You have the following options:

   - Enter a set value. A value of 0 will retain the shape of the **Bounded Surfaces**, while a value of 1 will fit the bounded surfaces to the **Imported Surfaces**.

   - Use the drop-down menu to define the scale factor using an expression or input parameter. Input parameters can be created for geometry parameterization (see Geometry Parameterization and Exploration (p. 4614) for details), or as part of a parametric study managed by Workbench (see Creating a New Parameter (p. 1311) for details). Additionally, for details on defining an expression see Directly Applied Expressions (p. 1050).

9. Click **Apply** in the **Design Conditions** tab of the **Design Tool** dialog box to save your settings.

10. If at any point you would like to delete any of the **Imported Surfaces**, select them and click the **Delete** button below the list. You can also delete them using the Surface Meshes Dialog Box (p. 5624).

Note:

When using the **bounded-by-surfaces** condition, it is recommended that you apply preconditioning on the advancement of the process for updating the freeform displacements (by changing the **Preconditioning** field to a positive value in the **Numerics** tab of the **Design Tool** dialog box, as described in Design Tool Numerics (p. 4570)), in order to encourage the stability of the solution. A value on the order of 1 or 10 is recommended.

Prescribed Profiles

You can use the **prescribed-profile** condition to assign a motion profile to a wall zone or zones using either a vector expression or with a `DEFINE_GRID_MOTION` UDF. When defining a **prescribed-profile** using an expression, note the following:

- The expression must be a vector and be a function of `x`, `y`, and `z`. For example the expression `vector(0.5*(x-1[m]), 0[m], 0[m])` will scale the wall zone along the x-axis with respect to the origin `(1[m], 0[m], 0[m])`.

- Vector expressions containing previously defined input parameters are supported. For example the expression `vector(parameter_1, 0.002[m], 0.003[m])` will apply a translation to the wall zone of the `parameter_1` value in meters along the x-axis, `0.002` meters along the y-axis, and `0.003` meters along the z-axis.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The deformation profile will provide the displacement field after the deformation, rather than the new nodal position.

1. To define the **prescribed-profile** condition using a `DEFINE_GRID_MOTION` UDF:

   a. Compile your DEFINE_GRID_MOTION UDF in Fluent. (Refer to `DEFINE_GRID_MOTION` in the *Fluent Customization Manual*).

   b. Select your UDF in the **Deformation profile (Vector Expression/UDF)** drop-down list.

      Note that when a `DEFINE_GRID_MOTION` UDF is used for the **prescribed-profile** condition, the deformation profile will provide the new nodal location after deformation, rather than the displacement field.

2. To define the **prescribed-profile** condition using a vector expression:

   a. Select the desired expression from the **Deformation profile (Vector Expression/UDF)** drop-down list.

3. Select the surfaces (or clip-surfaces) from the **Surfaces** list that are subject to the **prescribed-profile** condition.

4. Define the **Scale Factor** to be applied to the deformation profile. You have the following options:

   - Enter a set value.

   - Use the drop-down menu to define the scale factor using an expression or input parameter. Input parameters can be created for geometry parameterization (see Geometry Parameterization and Exploration (p. 4614) for details), or as part of a parametric study managed by Workbench (see Creating a New Parameter (p. 1311) for details). Additionally, for details on defining an expression see Directly Applied Expressions (p. 1050).

   - Disable the **Prescribed** option so that the scale factor is a free parameter. This can be helpful when you are trying to optimize a design, because the Design Tool will determine a scale factor that satisfies your goals, and so no input is required from you. The value will be determined when you calculate the design change, and will be printed in the console; the printed value can be useful if you would like to recreate the design change in another software program (rather than exporting it as an `.stl` file).

5. Click **Apply**.

Rotation

The **rotation** condition allows you to specify the surfaces to undergo rotation.

1. Select the surfaces (or clip-surfaces) you want to rotate in the **Surfaces** list.

2. Define the rotation **Angle**. You have the following options:

   - Enter a set value in degrees.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Use the drop-down menu to define the angle using an expression or input parameter. Input parameters can be created for geometry parameterization (see Geometry Parameterization and Exploration (p. 4614) for details), or as part of a parametric study managed by Workbench (see Creating a New Parameter (p. 1311) for details). Additionally, for details on defining an expression see Directly Applied Expressions (p. 1050).
- Disable the **Prescribed** option so that the rotation is a free parameter.This can be helpful when you are trying to optimize a design, because the Design Tool will determine an angle that satisfies your goals, and so no input is required from you. The value will be determined when you calculate the design change, and will be printed in the console; the printed value can be useful if you would like to recreate the design change in another software program (rather than exporting it as an `.stl` file).

3. Define the **Origin** about which you want the surfaces to rotate. Note that you can click the **Get Center** button to set the origin fields to the coordinates of the center of the selected surfaces.
4. For 3D cases, define the **Axis** of rotation. For 2D cases, this is automatically defined as the positive Z axis.
5. Click **Apply**.

Translation

The **translation** condition allows you to specify that surfaces undergo translation.

1. Select the surfaces (or clip-surfaces) you want to translate in the **Surfaces** list.
2. Define the **Displacement** for each coordinate axis. For each field you have the following options:
   - Enter a set value.
   - Use the drop-down menu to define the displacement using an expression or input parameter. Input parameters can be created for geometry parameterization (see Geometry Parameterization and Exploration (p. 4614) for details), or as part of a parametric study managed by Workbench (see Creating a New Parameter (p. 1311) for details). Additionally, for details on defining an expression see Directly Applied Expressions (p. 1050).
   - Disable the **Prescribed** option so that the translation is a free parameter. This can be helpful when you are trying to optimize a design, because the Design Tool will determine a displacement that satisfies your goals, and so no input is required from you. The value will be determined when you calculate the design change, and will be printed in the console; the printed value can be useful if you would like to recreate the design change in another software program (rather than exporting it as an `.stl` file).
3. Click **Apply**.

Scaling

The **scaling** condition allows you to specify that surfaces are scaled along one or more axes.

1. Select the surfaces (or clip-surfaces) you want to scale in the **Surfaces** list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Select the scaling **Type** and define the associated settings:

   - **Radial**

     This type scales the surfaces radially about the **Origin**, using a single scaling **Factor** along each of the coordinate axes. The size of the surfaces will change, though not the shape.

   - **In-Plane**

     This type is available for 3D cases, and scales the surfaces about the **Origin** using a single scaling **Factor** along the two axes of a plane (as defined by the **Normal Direction**).

   - **Axial**

     This type scales the surfaces about the **Origin** using a scaling **Factor** along a single **Axis**.

   - **General**

     This type scales the surfaces about the **Origin** using a separate scaling **Factor** for each of the defined axes. For 2D cases, you define one **Axis**, and the second axis is automatically defined as being perpendicular; for 3D cases, you define **Axis 1** and **Axis 2** (which should not be parallel), and the third axis is defined automatically as the cross product. Note that the surfaces will be scaled along each axis sequentially, so the order will be relevant for non-orthogonal axes.

   For all types, the scaling **Factor** must be positive. A value of 1 results in no change, while a value greater or less than 1 results in stretching or shrinking, respectively. You have the following options:

   - Enter a set value.
   - Use the drop-down menu to define the factor using an expression or input parameter. Input parameters can be created for geometry parameterization (see Geometry Parameterization and Exploration (p. 4614) for details), or as part of a parametric study managed by Workbench (see Creating a New Parameter (p. 1311) for details). Additionally, for details on defining an expression see Directly Applied Expressions (p. 1050).
   - Disable the **Prescribed** option so that the factor is a free parameter. This can be helpful when you are trying to optimize a design, because the Design Tool will determine a factor that satisfies your goals, and so no input is required from you. The value will be determined when you calculate the design change, and will be printed in the console; the printed value can be useful if you would like to recreate the design change in another software program (rather than exporting it as an `.stl` file).

   When you are defining the **Origin** for any of the types, note that you can click the **Get Center** button to set the origin fields to the coordinates of the center of the selected surfaces.

3. Click **Apply**.

Rigid Body Deformation

The **rigid-body** condition allows you to specify that surfaces undergo rigid body motion, in which the rotations and translations are free parameters. The Design Tool will determine rotations and translations that satisfy your goals when you calculate the design change, and will print them in

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the console. The printed values can be useful if you would like to recreate the design change in another software program (rather than exporting it as an `.stl` file); note that you must apply the deformations in the same order as they are printed to obtain the same results.

1. Select the surfaces (or clip-surfaces) you want to undergo rigid body motion in the **Surfaces** list.

2. Click **Apply**.

Compound

The **Compound** condition allows you to combine a number of previously defined **Prescribed** design conditions into a single design condition. Note that when using the **Design Tool**, the compound design condition can only be used with the **Enhanced Constraint Method** for the morphing method.

1. To add design conditions to the **Design Conditions** list, click the **Add...** button to open the **Add Design Conditions** dialog box. Only **Prescribed** design conditions can be used for the compound condition and all design conditions must be defined on same surface.

2. After selecting the desired design conditions click **OK** to close the **Add Design Conditions** dialog box.

3. Select the design conditions you want to include in the compound condition from the **Design Conditions** list.

4. From the **Method** drop-down list, select the method for compounding the design conditions as **superposed** or **sequenced**. When **superposed** is selected, the prescribed motions for all design conditions are linearly superposed based on the deformed geometry. When **sequenced** is selected, the deformation will occur in a sequenced manner such that the prescribed motion of a given design condition is based on the geometry deformation caused by the prescribed motions of previous design conditions. Therefore, the order of the design conditions within the **Design Conditions** list only matters when using the **sequenced** method.

5. When using **sequenced** for the compound method, you can change the order in which the constraint equations are solved for a specific design condition by selecting it from the **Design Conditions** list and clicking the **Up** or **Down** buttons.

6. Click **Apply**.

48.2.5.5. Design Tool Numerics

Smoothness

As described in Smoothing and Mesh Morphing (p. 4506), there are three morphing methods available for the adjoint solver. The following describes how smoothing is performed in each:

- Polynomials-based

  In this approach, the mesh morphing is controlled by the summation of two smoothing bases: the Bernstein polynomial basis which control the large scale smooth deformation and the B-spline basis which controls fine-scale deformation. The choice of smoothness determines the relative weighting of these bases and therefore the spatial scale of the design change. A large value of

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

smoothness biases the design change to one in which optimal displacements vary in space in a manner dominated by the Bernstein polynomial basis. That is, they have a high degree of smoothness. A smaller value of smoothness leads to a recommended design change with smaller spatial scales.

The ability to vary the smoothness parameter is particularly important when prescribed forms of deformation are applied. Deformation at smaller spatial scales relieves the stiffness that would otherwise cause numerical challenges as well as lead to poor quality of the computed design and mesh.

- Direct Interpolation

  In this method, the surface mesh is smoothed using a Laplacian smoothing approach. Unlike the polynomials-based approach, the smoothness input does not effect the convergence of the design conditions, but it has a strong impact of the smoothness of the displacement on the surfaces undergoing free deformation. This is an important parameter in the direct interpolation method.

- Radial Basis Function

  In this method, mesh morphing is controlled by kernel functions. The kernel function has a strong effect on the stiffness of the equation being solved and different kernels provide different smoothness of deformation. Two kernel functions are available under the **Numerics** tab of the **Design Tool** :

  Inverse MultiQuadric Method:

$$\phi=\frac{1}{\sqrt{r^2+C^2}} \tag{48.54}$$

  Where C is determined by the smoothness value, the inverse multiquadric method provides a balance between smoothness of deformation and speed of convergence therefore it is the default kernel for the radial basis function. Larger smoothness values will produce smoother mesh deformation and require more iterations to converge.

  Bi-Harmonic Method:

$$\phi=r \tag{48.55}$$

  The bi-harmonic method has energy minimization properties that yield very smooth mesh deformation.

Note:

– **Smoothness** settings have no affect when using the bi-harmonic method.

– The bi-harmonic method will produce a better mesh quality than the inverse multiquadric method with the same level of deformation.

After enabling the **Radial Basis Function** in the **Design Change** tab, a **Smoothness** input will become available and Fluent will automatically define the **Smoothness** with a default value of 1. A

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

reasonable range for the smoothness field is between 10^{-3} and 10^{3}, which is suitable for a variety of cases.

Calculation Numerics

Note:

If no **Design Conditions** are imposed, and if there are no walls that are treated as fixed which intersect the region being modified, the numerics settings described below will not be used.

When there are no **Design Conditions**, including no zones intended to stay fixed in the morphing region, no adjustment of the detailed **Numerics** will be required for the **Design Tool** to compute an optimal solution.

If there are **Design Conditions** imposed, including some zones intended to stay fixed in the morphing region, it is generally not necessary to adjust the **Numerics** settings unless you are using the **Polynomials** morphing method. However, if the **Design Tool** calculation does not converge well or the **Design Conditions** are not well satisfied, the **Numerics** settings will need to be adjusted. For example, if you encounter geometrical changes on unselected zones in the **Modifiable Zones** list of the **Design Change** tab, you may need to reduce the constraint tolerance to better satisfy the **Design Conditions**.

Note:

In the following list of numerics settings, the **Freeform Motions** will be different depending on the specific morphing method and constraint method being used.

When using the **Direct Interpolation** morphing method the following settings are available:

Freeform Motions

Max. Main Iterations

Maximum number of outer loop iterations for nonlinear problems.

When using the **Polynomials** morphing method with the **Standard** constraint method, the following settings are available:

Prescribed Motions

Prescribed motions settings apply to cases that involve the **prescribed-profile**, **rotation**, **scaling**, **translation**, and/or **rigid-body** design conditions.

Max. Iterations

The maximum number of inner iterations that will be performed in an effort to compute displacements from prescribed changes.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Constraint Relaxation

The under-relaxation factor that is applied to the field to converge the prescribed displacements. This takes a value between 0 and 1.

Preconditioning

A value of 0 yields the most aggressive advancement, while a larger value encourages more stable, albeit slower, advancement of the process for updating the prescribed displacements. A value of order 1 is typical in the event that preconditioning needs to be applied.

Freeform Motions

Max. Iterations

The maximum number of inner iterations that will be performed in an effort to compute freeform displacements that satisfy the applied **Design Conditions**.

Constraint Relaxation

The under-relaxation factor that is applied to the field to converge the constrained freeform displacements. This takes a value between 0 and 1.

Preconditioning

A value of 0 means that no preconditioning is applied, while a larger value encourages more stable, albeit slower, advancement of the process for updating the freeform displacements. A value of order 1 is typical in the event that preconditioning needs to be applied.

Parameter Relaxation

For cases that include a **rigid-body** condition and/or a design condition with a free parameter (for example, the **rotation** condition with **Prescribed** option disabled for the **Angle**), there are additional parameters that are implicit in the calculation. The values of these parameters are computed as part of the solution process. The value of the parameter relaxation defines the rate at which these parameters are corrected as the calculation progresses. This takes a value between 0 and 1.

Tolerances

Constraints

The relative tolerance for convergence of the residuals associated with the design conditions themselves, or fixed zone conditions.

Parameters

The tolerance for convergence of the parameters associated with the design conditions.

Using a value of 1 for **Constraint Relaxation** and a value of 0 for **Preconditioning** yields the most aggressive behavior. This can result in the fewest number of iterations to converge a constrained problem, at the expense of some added numerical effort and memory in preparing the preconditioning. For 2D and small 3D problems the added cost in time and memory is typically small. For larger problems involving multiple constraints a non-zero **Preconditioning** and **Constraint Relax-**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ation less than 1 may be preferable. The execution time to set up the preconditioning will be less, as will the memory usage. Calculation stability can also be improved. The number of iterations required to converge the problem will increase. However, this can prove to be an effective strategy that results in an overall shorter time needed to compute the design change.

When using the **Radial Basis Function** morphing method with either the **Enhanced** or **Standard** constraint method the following setting is available.

Freeform Motions

Kernel Type

The basis function of the radial basis function, which has a strong effect on the speed of convergence, mesh quality, and smoothness of deformation. For more information on the different kernel types, see Equation 48.54 (p. 4571) and Equation 48.55 (p. 4571) .

When using the **Radial Basis Function** morphing method with the **Standard** constraint method or when **Improve Smoothness by Solving Primary Morpher** is enabled with the **Enhanced** constraint method, the following settings are available:

Freeform Motions

Max. Main Iterations

Maximum number of outer loop iterations for nonlinear problems.

Tolerance

Determines how well the **Design Condition** (for example, the bounded-by-plane/surfaces condition) is satisfied when the solver converges. A small tolerance gives high constraint satisfaction and more sub iterations. The residuals, including max and averaged residuals, offset from the constraint and are printed to the console.

Max. Sub-iterations

The maximum number of sub-iterations in each outer loop iteration of the **Design Tool** calculation.

Number of Modes

This specifies the number of modes in the Krylov subspace used to approximate the solution. Increasing the number of modes will increase the convergence and reduce the final residual. Note that increasing the number of modes also requires more memory and computation time for each sub-iteration. The convergence rate may increase sharply when the number of modes is larger than a certain threshold.

Note:

The linear radial basis function equation uses the same linear solver as that of the residual minimization scheme. For more information refer to Residual Minimization Scheme (p. 4535).

When using the **Polynomials** or **Radial Basis Function** morphing methods with the **Enhanced** constraint method, the following settings are available:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Freeform Motions

Freeform motions settings for primary morpher with the **Enhanced** constraint method.

Improve Smoothness by Masking Shape Sensitivity

When enabled, this option will mask the shape sensitivity along the fixed surfaces to reduce the freeform deformation. This improves the deformation smoothness by reducing the need for correction during the secondary morphing.

Improve Smoothness by Solving Primary Morpher

When enabled, this option specifies the standard morphing method (primary morpher) to calculate an initial approximation for the constraint equations. The radial basis function (secondary morpher) then enforces the design constraint by refining the approximation of the constraint equations. This option improves the smoothness of the deformation by reducing the computational load on the secondary morpher.

Constraint Settings

Enhanced constraint method settings for the secondary morpher.

Improve Smoothness with 2nd Order Local Scheme

When enabled, the second order radial basis function will be used to provide additional smoothness of the deformation. Note that enabling this option may increase the computation time by a factor of 2 - 4 and increase memory by a factor of 3.

Improve Solvability/Smoothness with More Local Control Points

When enabled, more local control points will be used to improve the smoothness of the deformation. Enabling this option may increase memory cost and computational time.

Improve Smoothness with More Global Control Points

When enabled, more global control points will be used to improve the smoothness of the deformation. Enabling this option may increase the computational time.

Tolerances

Tolerance settings for **Enhanced** constraint method.

Constraints

The absolute tolerance for convergence of the residuals associated with the design conditions, including fixed zone conditions in meters. Tighter tolerance may increase both the computational time and memory cost.

48.2.5.6. Shape Modification

The procedure for computing the optimal design change and deforming the mesh is descibed below. Note that much of the procedure for performing shape modification in the **Design Tool** is also relevant when using the **Parameterize and Explore** dialog box for geometry parameterization, as outlined in Geometry Parameterization and Exploration (p. 4614).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click the **Design Change** tab.

2. **Note:**

   This step is not applicable if you are defining **Design Change** settings within the **Parameterize and Explore** dialog box for geometry parameterization.

   Select those boundaries that are free to be partially constrained or deformed in the **Modifiable Zones** field. Boundaries that are not selected in this box that intersect the control volume will be constrained not to move. This is useful if there are walls that, for design reasons, must remain fixed and not move.

   For easier selection, you can click the **Show Inside Zones** button, which will only show zones in the **Modifiable Zones** list that exist in the specified morphing regions. To show all available zones, you can click the **Show All Zones** button.

   Note:

   - If you are defining **Design Change** settings for geometry parameterization within the **Parameterize and Explore** dialog box, the **Show Inside Zones** button, **Show All Zones** button, and**Modifiable Zones** list are not applicable.
   - If you have a two-sided wall that you want to deform, it is recommended that you select both the wall and the corresponding wall-shadow.
   - When using the **Direct Interpolation** morphing method, you must ensure that none of the zones involved in a non-conformal mesh interface are selected.

3. Select the constraining and/or deformation conditions (Defining Conditions for the Deformation (p. 4560)) to be applied to the design change in the list of **Applied Conditions**. Note that a given surface cannot have multiple conditions of type **prescribed-profile**, **rotation**, **translation**, **scaling**, or **rigid-body**.

   You can click the **Display** button under the **Applied Conditions** list, in order to display the applied conditions in the graphics window, each type in a unique color. This includes the conditions selected in the list, as well as the fixed conditions applied to zones that are not selected in the **Modifiable Zones** list. This makes it easy to review your setup, and identify any overlapping conditions (recognizable by their two-tone cells).

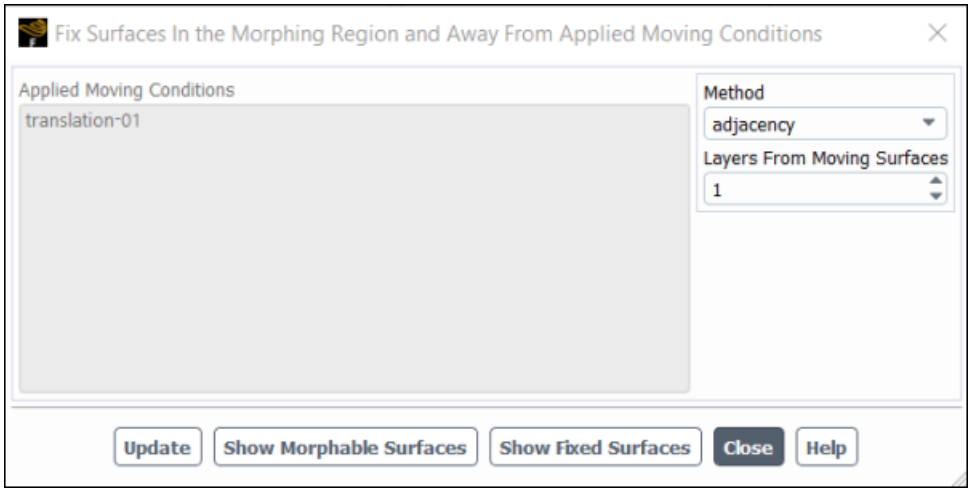
   Note:

   For geometry parameterization, the **Modifiable Zones** list is not available within the **Parameterize and Explore** dialog box.

   If your problem only has design conditions with moving surfaces selected in the list of **Applied Conditions**, you can click the **Fix Surfaces...** button under the **Applied Conditions** list to create a fixed condition away from any moving surfaces. When creating a fixed condition, you should ensure that there is some space between moving surfaces and fixed surfaces to avoid unphysical or abrupt geometrical changes (which cause poor mesh quality).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 48.19: Fix Surfaces in the Morphing Region and Away From Applied Moving Conditions Dialog Box



a. Select the method for creating the fixed condition as **none**, **adjacency**, or **distance**.

   - When **none** is selected, the fixed condition will be disabled.
   - The **adjacency** method will automatically fix any surfaces that are not adjacent to a zone containing moving surfaces.

     When **adjacency** is selected for the method, you must define the **Layers From Moving Surfaces**. This value represents the the number of zones that will be used to create a transition region for passive mesh morphing in zones adjacent to the moving surface. For example, if you set the **Layers From Moving Surfaces** value to 1, only the zones immediately adjacent to the moving surface are allowed to be morphed, and all other zones will be fixed. If you set the value to 2, zones that are adjacent to the first layer are also allowed to be morphed, and so on.
   - The **distance** method allows you to define a specific distance from a moving surface that will undergo passive mesh morphing. You can set the distance by entering the distance value for **Distance From Moving Surfaces**.

b. After specifing the **Method** and method settings, you can click the **Show Morphable Surfaces** button to display the deformable surfaces. Within the graphics window, deformable surfaces will be shown in blue, while moving surfaces are displayed in green.

c. You can click the **Show Fixed Surfaces** button to display the surfaces defined as fixed based on the distance specified away from moving surfaces. Fixed surfaces will be displayed in grey, while moving surfaces are displayed in green within the graphics window.

d. If the surfaces displayed for the fixed condition are satisfactory, you can click the **Update** button to save the fixed condition.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. **Note:**

   This step is not applicable if you are defining **Design Change** settings within the **Parameterize and Explore** dialog box for geometry parameterization.

   For each observable, specify its **Weight** in the optimization calculation. The weights are applied to the **Target/Reference Change** you specified in the **Objectives** tab. That is, if you have two observables that are each given a **Weight** equal to 1, Fluent will attempt to find an optimal design change that alters the observable values in proportion to their respective values for **Target/Reference Change.**

   Note:

   Weights are only required if there are two or more objectives that do not have explicit target changes specified.

5. **Note:**

   This step is not applicable if you are defining **Design Change** settings within the **Parameterize and Explore** dialog box for geometry parameterization.

   Specify the deformation parameters.

   a. When using the **Polynomials** morphing method with the **Standard** constraint method, choose a **Freeform Scaling Scheme** and specify the **Freeform Scaling Factor**. This setting controls how the overall scaling factor for the design change magnitude is interpreted.

      Control-Point Spacing

      The magnitude of the freeform deformation will be based on the control-point spacing. That is, a **Freeform Scale Factor** of `1.0` leads a to maximum control point movement on the order of one sub-cell of the control volume. The sub-cell size is determined by a uniform sub-division of the morphed region based on the number of control points specified (Defining Region Conditions (p. 4558)). A value of `1.0` is a reasonable default to preserve mesh quality during the morphing process.

      Objective Reference Change

      The magnitude of the freeform deformation will be based on the **Target/Reference Change** value in the **Objectives** tab. That is, for a single observable optimization with **Freeform Scale Factor** of `1.0`, the design change computed will yield an expected change in the observable equal to the **Target/Reference Change** for the observable.

   b. If you have disabled **Auto-Select Smoothness** in the **Numerics** tab, you can also specify a value for **Smoothness** of the deformed geometry. A lower value will allow locally sharper changes in the geometry, potentially at the expense of manufacturability. For details about this setting, see Design Tool Numerics (p. 4570).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

6. If you used an input parameter as part of the setup of a deformation condition in the **Design Conditions** tab, the initial value will be displayed and is available for modification.

7. Click the **Check** button to verify that your constraining and/or deformation conditions are set up in such a way as to produce good results. A report will be printed in the console listing conflicts between multiple constraints / deformations applied on a single node (including the **Design Conditions** and the continuity constraint applied along the boundary of the morphing region). The conflicts are categorized as being either of type `Fatal` (which indicates the solution is unlikely to converge or be desirable) or `Possible` (which may be allowable, depending on the details of the applied conditions).

8. Click **Calculate Design Change** to compute the optimal design change. The **Expected change** for each observable will be reported in the **Design Tool** dialog box.

9. (optional) You can export the expected changes or displacements by clicking the **Export...** button then selecting one of the following from the drop-down list:

   Note:

   The **Export Expected Changes...** option is not available if you are using the **Parameterize and Explore** dialog box for geometry parameterization.

   - Click **Export Expected Changes...** to write to file the adjustment in the observable that is expected due to the change defined by the current settings.
   - Click **Export Displacements...** to write to file the optimal surface displacement field (overwriting any pre-existing file of the same name). An example of the format of the text file that is written (for a 3D mesh) is as follows:

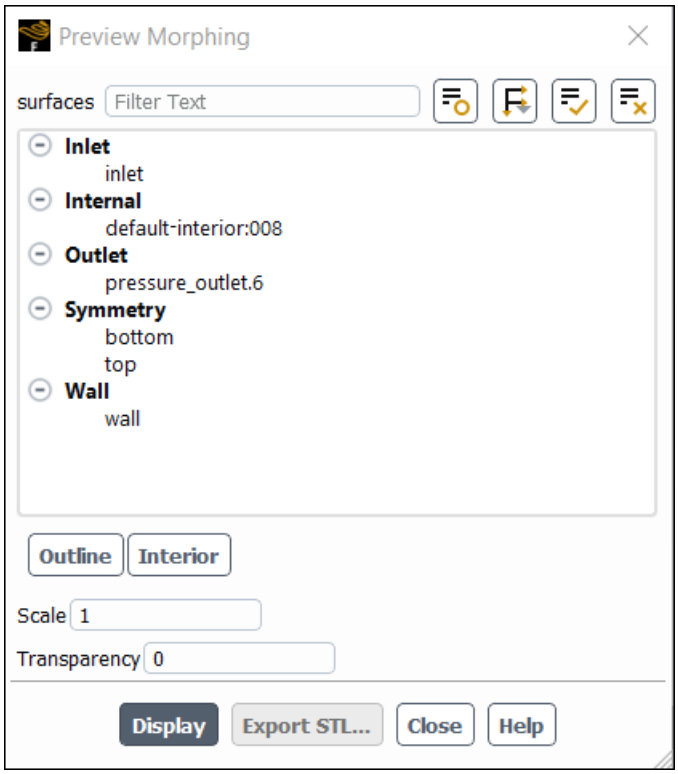
     ```
     Line 1:
     n
     Lines 2 to n+1:
     x y z dx dy dz
     ```

     This displacement field is for the calculated design change for the current mesh. If you plan to repeatedly create a series of iterations of a design (to gradually deform the mesh), you can also compare exported displacements to a reference mesh, set at an appropriate iteration. This is done using the **Mesh History** dialog box, as described in a later step.

10. The planned change to the geometry can be displayed in the graphics window by clicking the **Preview...** button and then using the **Preview Morphing** dialog box that opens to display the surfaces of interest. Note that you can use the buttons to easily select all the **Outline** boundaries and/or the **Interior** surfaces, and you can increase the **Scale** to accentuate the morphing preview or exported `.stl`. When you click the **Display** button, the selected surfaces are displayed undeformed in white and deformed in green.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 48.20: The Preview Morphing Dialog Box

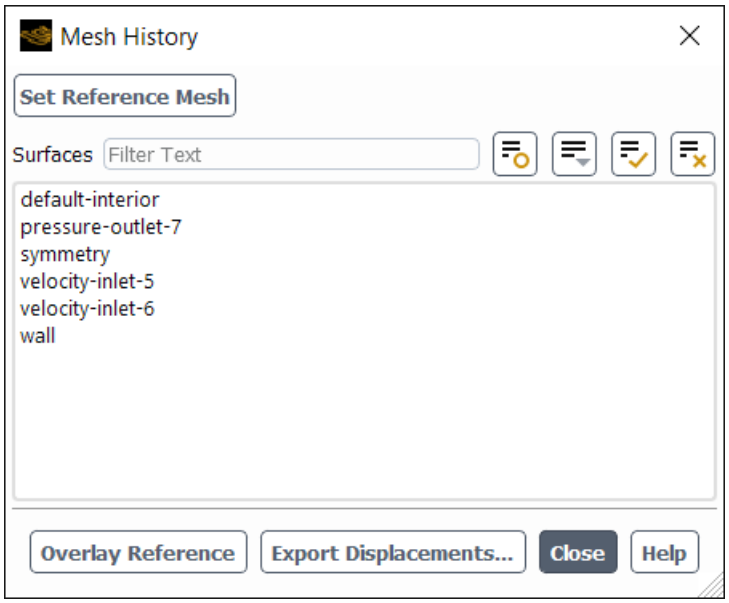


Note that for 3D cases you can also use the **Export STL...** button to export the previewed mesh as an `.stl` file. This is useful for large cases, as it means you do not have to actually morph the mesh in order to have the previewed geometry available for use in other software packages.

11. If you plan to repeatedly create a series of iterations of a design (to gradually deform the mesh), you can set the current mesh to be a reference mesh, and then at a later design iteration you can export the displacement field of the design compared to this reference mesh. This is different than using the **Export Displacements...** button in the **Design Export** dialog box, as that button does not allow you to see the total displacements over multiple instances of mesh modification. To start this process, click the **History...** button to open the **Mesh History** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 48.21: The Mesh History Dialog Box



Click the **Set Reference Mesh** button to specify the current mesh as the one used for comparisons in the future. After it has been set, this button changes to **Reset Reference Mesh**, which can be used if you decide that the mesh from a later design iteration should instead be used as the reference mesh.

After you have modified the mesh (as described in the next step) one or more times, you can return to this dialog box; then, for the selected **Surfaces**, you can do the following:

- Click the **Overlay Reference** button to visualize the displacements in the graphics window. The reference mesh is displayed in green and the modified mesh is displayed in white.
- Click **Export Displacements...** to write to file the surface displacement field (overwriting any pre-existing file of the same name). An example of the format of the text file that is written (for a 3D mesh) is as follows:

```
Line 1:
n
Lines 2 to n+1:
x y z dx dy dz
```

Note:

The following limitations apply to the use of the **Mesh History** dialog box:

- The reference mesh can only be saved in a CFF case file (which is the default format).
- Any operation that changes the node count (such as remeshing or adaption) is not supported.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

12. Click the **Modify** button in the **Mesh** group box to deform the boundary and interior mesh with the optimal design change. You should then examine the modified shape and mesh to determine if the results are satisfactory, using the usual commands for viewing the mesh in Ansys Fluent. When using the **Direct Interpolation** morphing method, it is recommended that you check the resulting mesh quality and verify that no boundary zones cross / penetrate themselves or other boundary zones. Note that the **Direct Interpolation** morphing method is not applicable if you are using the **Parameterize and Explore** dialog box for geometry parameterization.

    If you determine that the change is unsatisfactory, you can click **Revert** to undo the mesh changes.

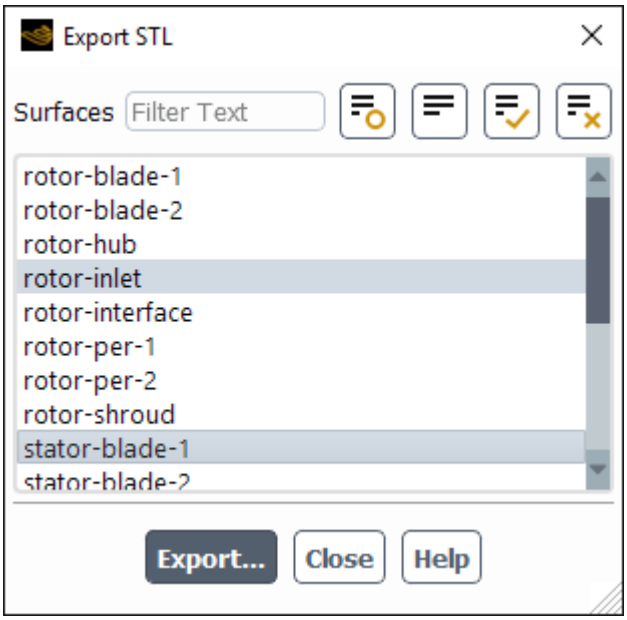
13. After modifying the mesh you can click the **Remesh** button to remesh the modified geometry, which can improve mesh quality and fix negative cell volumes. Remeshing can lead to a better design, especially for large geometry changes after modifying the mesh in the previous step. Remeshing is accomplished using the unified remeshing method in Fluent and uses the unified remeshing settings defined within the **Dynamic Mesh** task page (see Advanced Remeshing Settings Dialog Box (p. 5120)). Upon clicking the **Remesh** button, deforming dynamic mesh zones are automatically created in the morphing region and when remeshing is complete, the dynamic mesh zones are deleted. Only tetrahedral and prism meshes for 3D cases and triangle meshes for 2D cases can be remeshed and all other types of meshes will not be affected by remeshing. Once the modified geometry has been remeshed, the new mesh can not be reverted and any previously defined reference meshes within the **Mesh History** dialog box will not be accessible due to changes in the mesh connection.

After the mesh has been modified according to the control settings, the original flow calculation can be restarted and converged with the modified geometry. Upon successful convergence, the observable value should now have changed by an amount similar to that reported when the **Export Expected Changes...** button was clicked prior to deforming the geometry. Note that observable values and the **Export Expected Changes...** button is not relevant when using the **Parameterize and Explore** dialog box for geometry parameterization.

For 3D cases, you have the option of exporting the boundary surfaces and/or isosurfaces from your modified geometry as an `.stl` file, for use in other software packages. Click the **Export...** button near the bottom of the **Design Change** tab and then click the **Export STL...** button in the **Export** dialog box that opens. Finally, select the **Surfaces** of interest in the dialog box that opens (Figure 48.22: The Export STL Dialog Box (p. 4583)) and **Export...** them.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 48.22: The Export STL Dialog Box



48.2.6. Using the Gradient-Based Optimizer

The adjoint solver provides a **Gradient-Based Optimizer** that can automatically create a series of iterations of a design for both shape optimization and turbulence model optimization. For shape optimization, the mesh will gradually deform to an optimal shape in order to meet multiple objectives at multiple operating conditions. For turbulence model optimization, the adjoint-method and machine learning are combined to tune turbulence model parameters (e.g **GEKO** coefficients), with the aim of reducing the difference between the flow solution of a RANS (e.g **GEKO**) simulation and the flow solution of a high-fidelity simulation or results from an experiment. Using a neural network, the gradient-based optimizer will build an augmented turbulence model by correlating flow features with turbulence model parameters.

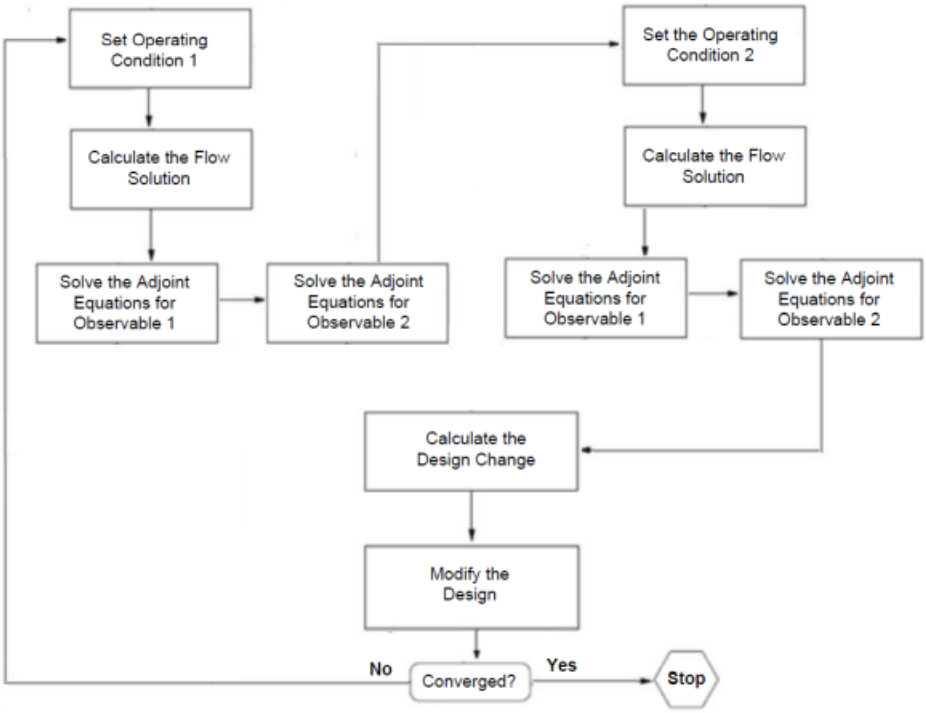
The workflow of the optimizer is shown in Figure 48.23: Gradient-Based Optimizer Workflow (p. 4584). Note that this figure represents the optimizer workflow when two operating conditions and two observables have been defined. After the flow solution is calculated for a specified operating condition, the adjoint equations are solved for each individual observable in sequence. For example, if three observables and four operating conditions are defined, the optimizer worklow will resemble the following procedure:

1. Flow solution is calculated for operating condition 1
2. Adjoint equations are solved for observable 1
3. Adjoint equations are solved for observable 2
4. Adjoint equations are solved for observable 3

After the adjoint equations are solved for observable 3 for the first operating condition, the procedure will repeat for the next operating condition. When the flow solution and adjoint equations have been solved for all operating conditions and all observables, the design change will then be calculated followed by the design modification.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 48.23: Gradient-Based Optimizer Workflow



To prepare to use the **Gradient-Based Optimizer**, you will perform most of the usual steps outlined in Using the Adjoint Solver (p. 4509) with the following exceptions:

- It is not necessary to start with a converged flow solution; simply initializing the case file is sufficient.
- It is not necessary to select a single observable in the **Adjoint Observables** dialog box, as you will be able to select multiple observables in a separate dialog box.
- It is not necessary to initialize the adjoint solution and iterate to convergence using the **Run Adjoint Calculation** dialog box, as these actions will be performed in the **Gradient-Based Optimizer** dialog box.
- For shape optimization, when setting up the **Design Tool** :
  - It is not necessary to export / import sensitivity data, as this will be handled automatically during optimization.
  - It is not necessary to specify the objectives for the observables in the **Objectives** tab of the **Design Tool** dialog box, as this will be done in the **Gradient-Based Optimizer** dialog box.
  - In the **Design Change** tab of the **Design Tool** dialog box, it is not necessary to take any action in the **Workflow** , **Mesh**, and **Results** group boxes.

Note:

Setting up the **Design Tool** is only necessary for shape optimization. When using the **Gradient-Based Optimizer** for turbulence model optimization, optimization settings will be specified within the **Turbulence Model Design Tool** group box as outlined in Using the Turbulence Model Design Tool (p. 4599).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

While the above steps are not necessary, you may yet want to perform them in order to verify that your settings are properly defined (for example, to ensure your design conditions are not conflicting for shape optimization) prior to using the optimizer.

For turbulence model optimization, the following additional steps are required prior to using the **Gradient-Based Optimizer** :

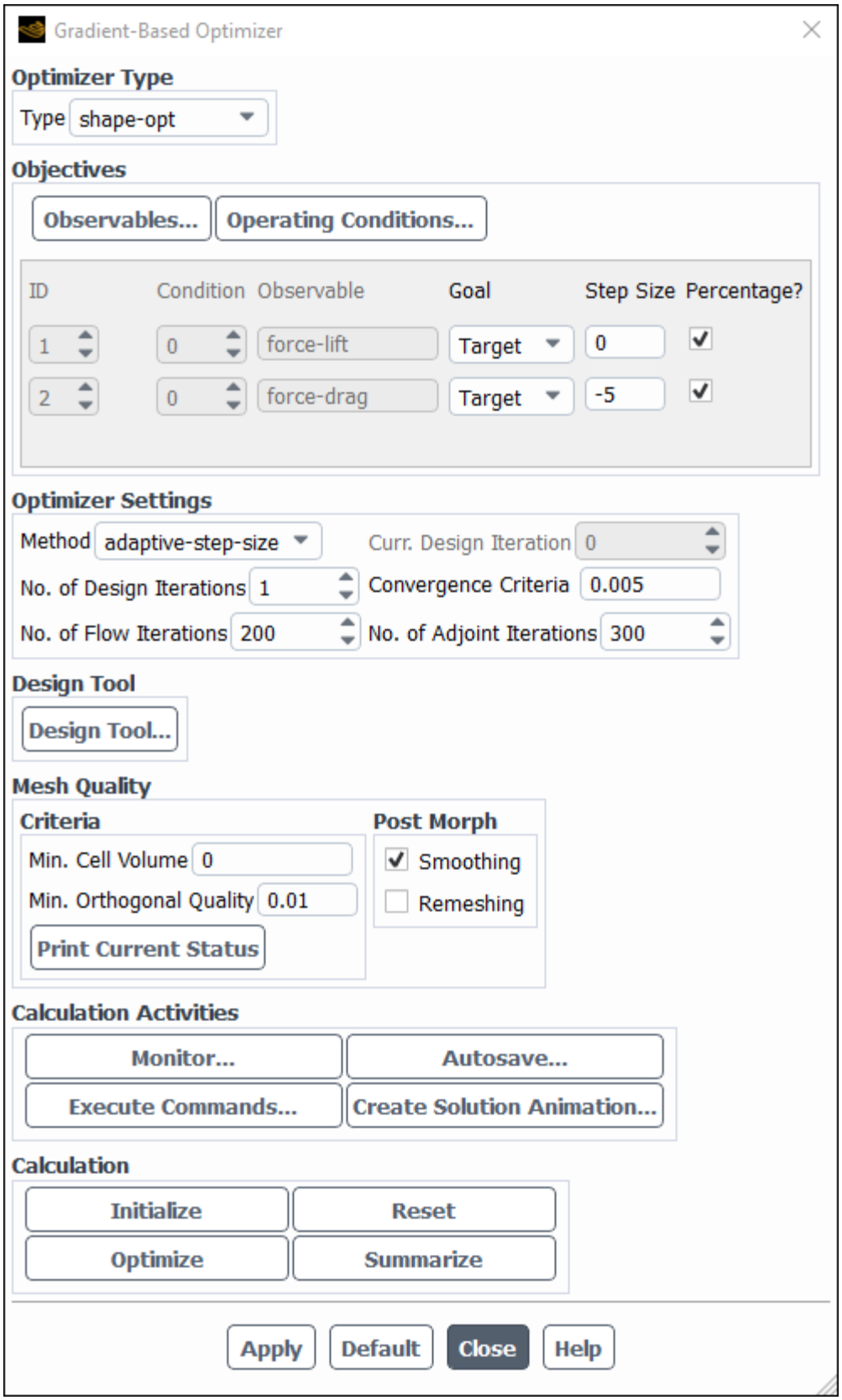
- The **GEKO** turbulence model must be selected within the **Viscous Model** as outlined in Setting up the Generalized k-ω (GEKO) Model (p. 2090).
- **Adjoint Turbulence** must be enabled within the **Adjoint Solution Methods** dialog box as outlined in Using the Adjoint Solution Methods Dialog Box (p. 4526).
- Adjoint **Observables** specific to turbulence model optimization should be defined as outlined in Defining Observables for Turbulence Model Optimization (p. 4525).
- Prior to configuring settings within the **Turbulence Model Design Tool** group box as outlined in Using the Turbulence Model Design Tool (p. 4599), it is recommended to plot sensitivity contours for the **GEKO** coefficients of interest (e.g **Contours** of **Sensitivities...** to **Sensitivities to GEKO CSEP**) to observe their influence on your RANS simulation. Selecting **Design Variables** for **GEKO** parameters with large sensitivity magnitudes will increase the controllability of tuning during optimization.

To start, you must define the settings for the flow solution and initialize it; if you want to optimize the design for multiple operating conditions, as part of this step you should create input parameters for the cell zone and boundary condition settings you would like to vary. Next, you must define the adjoint observables, methods, solver controls, monitors, and **Design Tool** (for shape optimization) or **Turbulence Model Design Tool** (for turbulence model optimization), noting the exceptions listed previously. Then you can open the **Gradient-Based Optimizer** dialog box by clicking **Gradient-Based Optimizer...** in the **Design** ribbon tab (**Gradient-Based** group box).

Design → **Gradient-Based** → **Gradient-Based Optimizer...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 48.24: Gradient-Based Optimizer Dialog Box



The general procedure to use the **Gradient-Based Optimizer** is as follows:

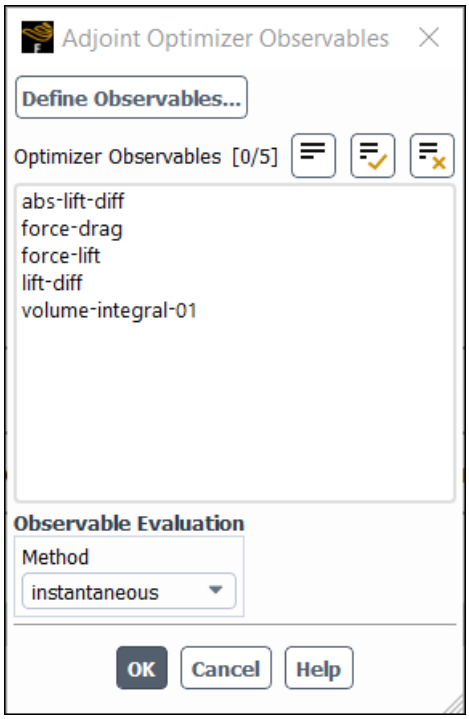
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Select the **Optimizer Type** from the **Type** drop-down list.

   - For shape optimization, select **shape-opt** .
   - For turbulence model optimization, select **turb-model-opt**.

2. Click the **Observables...** button to open the **Adjoint Optimizer Observables** dialog box. You can click the **Define Observables...** button to create and/or edit observables (as described in Defining Observables (p. 4515)).

   Figure 48.25: Adjoint Optimizer Observables Dialog Box



   1. For shape optimization, you can select all of the relevant observables for your optimization problem in the **Adjoint Optimizer Observables** dialog box.
   2. For turbulence model optimization, refer to Defining Observables for Turbulence Model Optimization (p. 4525) to create and edit observables specific to turbulence model optimization. After defining the observables for your optimization problem, you can select all of the relevant observables in the **Adjoint Optimizer Observables** dialog box.
   3. Select the **Observable Evaluation** method from the **Method** drop-down list. You have the following options:

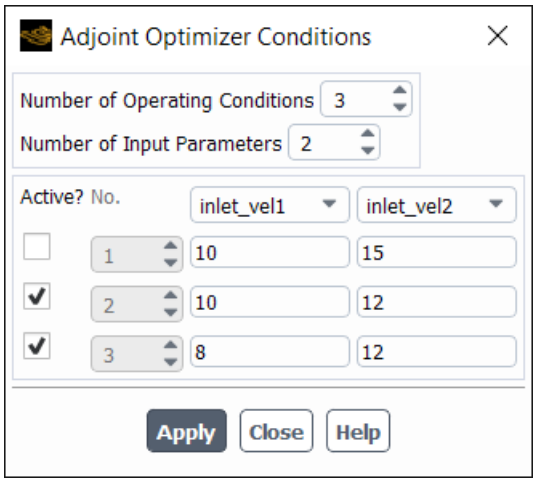
      - When **instantaneous** is selected, the observable value will be evaluated as it's current instantaneous value at the end of the flow solver.
      - When **averaged** is selected, the observable will be evaluated as it's average value over a specified **Sampling Interval**. You can also set the **Start after Iteration** value to begin averaging the observable after a certain interval.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

4. After you have selected the desired observables for your optimization problem and specified the **Method** for **Observable Evaluation**, click **OK** to close the **Adjoint Optimizer Observables** dialog box.

3. Click the **Operating Conditions...** button to open the **Adjoint Optimizer Conditions** dialog box.

Figure 48.26: Adjoint Optimizer Conditions Dialog Box



Define the following:

Number of Operating Conditions

This defines the number of possible operating conditions. Each operating condition is a set of simulation properties for which a flow solution will be calculated. As you increase this number, more rows will be added to the group box below, each with a unique **No.**. The **Active?** option must be enabled to calculate the flow for that operating condition.

Number of Input Parameters

This defines the number of input parameters you would like to define for each operating condition. As you increase this number, more columns will be added to the group box below. You can select an input parameter from one of the top drop-down lists, and then define its value for each operating condition in the number-entry boxes below it.

When you have defined all the operating conditions, click **Apply**, and close the **Adjoint Optimizer Conditions** dialog box.

4. Fill out the following columns for the table in the **Objectives** group box. Each row represents a unique combination of the active operating conditions and the selected observables defined in the previous steps.

Goal

This determines how Fluent will attempt to change the value of the observable through the design change. You can specify a **Target** change in value; otherwise you can select **None**, in which case the observable will not be considered in computing the optimal design change, and the predicted change in observable value will not be reported.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Step Size

When the **Goal** is a **Target** change in the observable for each design iteration, this field corresponds to the desired or initial change in the observable for each design iteration.

Percentage?

Enabling this option specifies that the **Step Size** is a percentage of the value of the observable.

A target change of 5–10% is typically a good starting point, depending on the problem.

5. The **Optimizer Settings** group box allows you to view / set the following:

Method

Specifies step size behavior during optimization. The optimization methods are as follows:

fixed-step-size

The fixed step size method will advance the optimization based only on the user-specified **Step Size**, regardless of whether the observable values improve or worsen. For shape optimimization, the **Step Size** will only be reduced if the mesh quality requirement is not met, which is also true for the **adaptive-step-size** and **line-search** methods.

adaptive-step-size

When using this method, Fluent automatically detects when a design iteration produces non-optimal results (compared to previous flow solutions), and reattempts the design using **Objectives** settings that are less aggressive. The process is as follows: the solution calculated for the initial design is considered optimal and written to disk; then the following two steps are repeated.

1. The design is revised based on the current **Objectives** settings.
2. The flow solution is calculated, and the results are compared to the optimal design solution:
   - If the values of all of the observables for all of the operating conditions improve or remain the same, the revised design solution is deemed the new optimal (and the files are overwritten), and the process returns to step 1.
   - If the value of any of the observables for any of the operating conditions worsens, the optimal design solution is restored, the **Objectives** settings are revised such that every **Step Size** is reduced by 50% going forward, a new design is generated, and the process returns to step 2.

line-search

Note:

The **line-search** option only supports optimization of a single **Observable**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The line search method aims to find the optimal design change along the direction of the gradient to improve the value of the observable during each design iteration. During each design iteration, the flow solution, adjoint solution, and gradients are first computed. The case and data files are then written to disk as the baseline for the line search. The line-search then performs sub-iterations to find the optimal step size needed to improve the observable value. For each sub-iteration, the following steps are performed:

1. The case and data files are read and the step size is adjusted.
2. The design change is calculated.
3. The design is modified based on the design calculation.
4. The flow solution is calculated for the modified design.
5. The observable value is recorded.

Once the optimal step size is found, it is then used for the next design iteration and the above process is repeated.

Note that when the **line-search** method is selected, one design iteration will consist of one adjoint calculation and multiple flow calculations (typically in the range of 5-10). Alternatively, when the **fixed-step-size** or **adaptive-step-size** methods are selected, one design iteration will consist of one flow calculation and one adjoint calculation.

For shape optimization, if the adjoint convergence is poor or if the shape sensitivity distribution is not irregular, the **line-search** method may produce a much better solution compared to the **adaptive-step-size** and **fixed-step-size** methods.

When using the **adaptive-step-size** or **line-search** methods, note that:

- The naming convention for the optimal design solution files is provided in a later step that describes how to set up autosaving.
- You must ensure that you have sufficient disk space for the optimal design solution files.
- You must not remove or rename the optimal design solution files during optimization.
- You must ensure that the optimal design solution files are in the correct folder if you stop the optimization and then continue it.

If a problem occurs during optimization, you can either:

- initialize the optimizer again: the current design solution files will be saved as the optimal if optimization continues, and the **Curr. Design Iteration** will be reset to 0 and the optimization history in the monitor plot will be lost.
- reset it: the current design solution files will be saved as the optimal if optimization continues, but the **Curr. Design Iteration** will not be reset to 0 and the optimization history in the monitor plot will not be lost.

If you stop the optimizer, modify some settings, and then continue the calculation, note that your changed settings may be overwritten if the optimal design solution is restored during the first iteration.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Curr. Design Iteration

This displays the current design iteration.

No. of Design Iterations

This specifies the maximum number of design iterations that the optimizer will perform. A value in the range of 10–20 is typically appropriate, depending on the problem and settings. When using **line-search** as the optimization method, a value in the range of 4–10 is recommended.

Convergence Criteria

This defines the criteria by which the design is considered converged. For every observable at every operating condition, the following are calculated during various points in the design iteration:

- the expected change / initial value (when the design change is calculated)
- the actual change / initial value (when the flow solution is calculated)

When either of the above are less than your specified **Convergence Criteria** for all observables at all operating conditions, the optimizer will stop iterating. If you want to continue the optimization, you can either reduce the convergence criteria or initialize/reset the optimizer.

No. of Flow Iterations

This specifies the maximum number of iterations performed during the calculation of the flow solution. A starting value of 300 is typically appropriate, depending on the problem and settings.

No. of Adjoint Iterations

This specifies the maximum number of iterations performed during the adjoint calculation. A starting value of 500 is typically appropriate, depending on the problem and settings. When using the blending stabilization strategy, the adjoint solver will stop early when the max iteration of the 2nd scheme has been reached. See Stabilization Strategies, Schemes, and Settings (p. 4533) for details.

6. For turbulence model optimization:

   1. Click the **Design Region...** button to open the **Optimizer Design Region** dialog box and specify the desired **Design Zone** (see Defining the Design Region (p. 4603)).

   2. Click the **Design Variables...** button to open the **Optimizer Design Variable Settings** dialog box and specify the settings for your optimization problem. For detailed instructions on specifying these settings see Defining Optimizer Design Variable Settings (p. 4604).

7. For shape optimization:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1. Click the **Design Tool...** button if you need to set up or edit the settings in the **Design Tool** dialog box. Note that you do not need to perform all of the steps that are necessary when not using the **Gradient-Based Optimizer**, as described at the beginning of this section.

2. Specify the settings in the **Mesh Quality** group box.

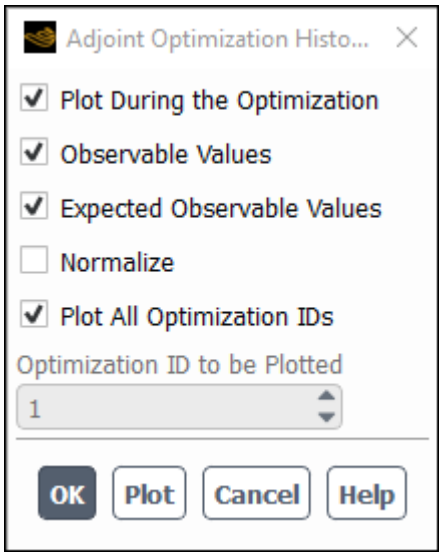
   By default, the cell volume and orthogonal quality (as defined in Mesh Quality (p. 1146)) is computed at the start of every design iteration, before the flow solution is calculated; if the mesh violates your specified **Min. Cell Volume** and/or **Min. Orthogonal Quality**, the optimizer stops. Such a check is also performed immediately after each design change is calculated; if this check fails, Fluent reattempts the design using **Objectives** settings that are less aggressive, in the same manner as described previously for the **adaptive-step-size** option (regardless of whether that option is selected). This helps to ensure that your solution accuracy is not compromised by poor quality meshes. When the direct interpolation method is used, the mesh quality may deteriorate quickly at local regions, therefore the value will be reduced quickly. You may continue the optimization by readjusting the value, accordingly. When setting these minimums, you can click the **Print Current Status** button and enter values that are sufficiently lower then those printed in the console, so as not to reject a majority of the meshes generated during the optimization process. Note that it is recommended that you set the **Min. Orthogonal Quality** to a value greater than 0, in order to avoid left-handed faces.

   The **Smoothing** option within the **Post Morph** group box is enabled by default. This will smooth the mesh at the end of the design iteration (after the mesh has morphed). You can enable the **Remeshing** option to remesh the geometry at the end of the design iteration (after the mesh has morphed) which can improve the mesh or fix negative cell volumes. Remeshing can lead to a better design, especially for large geometry changes after modifying the mesh. Only tetrahedral and prism meshes for 3D cases and triangle meshes for 2D cases can be remeshed and all other types of meshes will not be affected by remeshing. Additionally, any other remeshing limitations outlined in the **Remeshing** step of Shape Modification (p. 4575) will also apply to shape optimization.

8. You can set up a monitor of the optimization history by clicking the **Monitor...** button in the **Calculation Activities** group box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 48.27: Adjoint Optimization History Monitor Dialog Box



The **Adjoint Optimization History Monitor** dialog box has the following controls:

Plot During the Optimization

This option specifies whether plotting is performed for each design iteration.

Observable Values

This option enables the plotting of observable values from the flow calculation. The data is represented by solid lines.

Expected Observable Values

This option enables the plotting of expected observable values for the next design iteration. It is calculated by taking the current observable value and adding the expected change due to the mesh morphing (for shape optimization) or design variable tuning (for turbulence model optimization) in the current design iteration. The data is represented by dashed lines. If dashed and solid lines of the same color do not match well, that is a sign that the sensitivity is not accurate for that observable, and so you may need revise your setup (improve the convergence of the flow and/or adjoint solvers, reduce optimization values, and so on).

Normalize

Enabling this option specifies that the plotted values are normalized based on the values at the first design iteration. This can be helpful for multi-objective optimization when the values for the observables are quite different.

Plot All Optimization IDs

This option enables the plotting of data for all optimization IDs, that is, all of the rows of the table in the **Objectives** group box; otherwise, only a single optimization ID will be plotted.

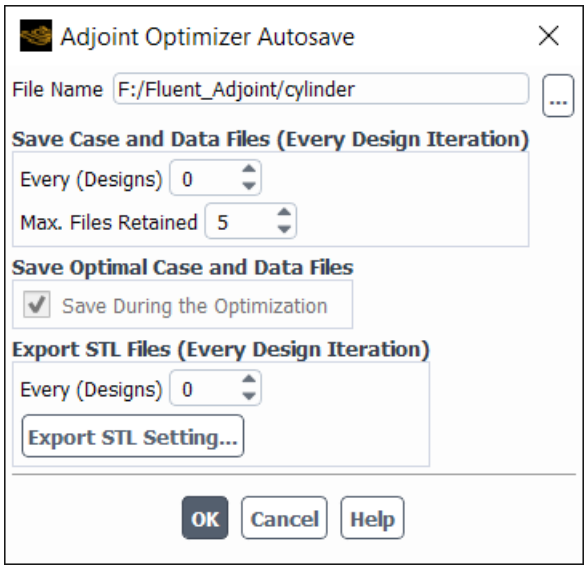
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Optimization ID to be Plotted

This field specifies the individual optimization ID to be plotted, and is only available when the **Plot All Optimization IDs** option is disabled. The number you enter corresponds to the **ID** in the **Objectives** group box.

9. You can set up the automatic saving of files during optimization by clicking the **Autosave....** button in the **Calculation Activities** group box. All of the files will be saved after the adjoint design calculation but before the morphing of the mesh (for shape optimization).

Figure 48.28: Adjoint Optimizer Autosave Dialog Box



The **Adjoint Optimizer Autosave** dialog box has the following controls:

File Name

This field specifies the root name for the files that are saved. It is suggested that you do not include an extension for the file name. If you include a folder, you must make sure that it is accessible.

Save Case and Data Files (Every Design Iteration)

This group box allows you to specify settings for the autosaving of case and data files. **Every (Designs)** sets the frequency, in design iterations. The **Max. Files Retained** sets the limit after which the earliest existing files are overwritten by the latest.

The files are saved with the following names:

- <*auto_name*> `-design-` <*des_iter*> `.cas.h5`
- <*auto_name*> `-design-` <*des_iter*> `.dat.h5`
- <*auto_name*> `-id-` <*optimization_id*> `.sens` (only when you have defined multiple operating conditions and/or objectives)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

where <*auto_name*> is the **File Name** , <*des_iter*> is a three-digit number corresponding to the design iteration, and <*optimization_id*> is a two-digit number corresponding to the **ID** in the **Objectives** group box.

Save Optimal Case and Data Files

This group box provides the **Save During the Optimization** option; when enabled, a single set of optimal design solution files is saved and rewritten as necessary (as described previously in the step describing the **adaptive-step-size** option). The files are saved with the following names:

- <*auto_name*> `-design-optimal.cas.h5`
- <*auto_name*> `-design-optimal.dat.h5`
- <*auto_name*> `-id-` <*optimization_id*> `-optimal.sens` (only when you have defined multiple operating conditions and/or objectives)

where <*auto_name*> is the **File Name** and <*optimization_id*> is a two-digit number corresponding to the **ID** in the **Objectives** group box.

If you move the `.cas.h5` / `.dat.h5` files to a different location, make sure to move the optimal `.cas.h5` / `.dat.h5` and `.sens` files (only when you have defined multiple operating conditions and/or objectives) along with it, and set up the proper **File Name** for the new location in order to continue the optimization. The optimal files are needed to restore the optimization. Otherwise, you can reset the optimizer and continue the optimization.

Note that the **Save During the Optimization** option can only be disabled if you have disabled the **adaptive-step-size** option.

Export STL Files (Every Design Iteration)

This group box allows you to set up the automatic exporting of `.stl` files (for 3D cases only), for use in other software packages. **Every (Designs)** sets the frequency, in design iterations. The **Export STL Setting...** button opens the **Export STL** dialog box, where you can select **Surfaces** to specify the boundary surfaces / isosurfaces of interest.

The files are saved as <*auto_name*> `-design-` <*des_iter*> `.stl`, where <*auto_name*> is the **File Name** and <*des_iter*> is a three-digit number corresponding to the design iteration.

10. You can set up commands to be executed during optimization (such as postprocessing commands that export fields) by clicking the **Execute Commands...** button in the **Calculation Activities** group box and using the Execute Commands Dialog Box (p. 5201) that opens. For details, see Executing Commands During the Calculation (p. 3700). Note that once you have opened this dialog box from **Gradient-Based Optimizer** dialog box, the following selections become available in the **When** drop-down list:

Adjoint Iteration

This specifies that execution of the command is based on the adjoint iteration. It is executed immediately before each adjoint advancement step.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Design Iteration

This specifies that execution of the command is based on the design iteration when the keyword `;@` is not entered after the command. The command will be executed after the design calculation and before any animation recording, both of which are before the morphing of the mesh.

Commands can also be set to execute before or after specific calculations in the optimization process (see Figure 48.23: Gradient-Based Optimizer Workflow (p. 4584) for details). The following keywords can be entered after typing in a command within the **Execute Commands** dialog box:

- `;@before-optimization`

  Specifies the command to execute immediately before optimization begins

- `;@after-optimization`

  Specifies the command to execute immediately after optimization is complete

- `;@before-flow`

  Specifies the command to execute immediately before the flow calculation.

- `;@after-flow`

  Specifies the command to execute immediately after solving the flow calculation.

- `;@before-adjoint`

  Specifies the command to execute before the adjoint calculation.

- `;@after-adjoint`

  Specifies the command to execute immediately after the adjoint calculation.

- `;@before-design-calculation`

  Specifies the command to execute immediately before calculating the design change.

- `;@after-design-calculation`

  Specifies the command to execute immediately after calculating the design change.

- `;@before-modify`

  Specifies the command to execute immediately before the design modification.

- `;@after-modify`

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Specifies the command to execute immediately after the design modification.

Note:

If the keywords are not entered exactly as listed above, then the command will not be executed. Additionally, it is recommened to leave a space between the command and the `;@` keyword.

For example, the execute command `adjoint/observable/write yes out/adjoint-obs.monset ;@after-flow` will write the adjoint observable values to file immediately after the flow solution is calculated.

Note that unexpected behavior may result if the executed command changes settings for the adjoint calculation and/or gradient-based optimizer.

11. You can set up animations to be recorded during optimization by clicking the **Solution Animation...** button in the **Calculation Activities** group box and using the Animation Definition Dialog Box (p. 5398) that opens. For details, see Creating an Animation Definition (p. 3714). Note that once you have opened this dialog box from **Gradient-Based Optimizer** dialog box, the following selection becomes available in the drop-down list next to the **Record after every** field:

    design-iteration

    This specifies that the recording is based on the design iteration. It is recorded after the design calculation and the execution of any commands, and before the morphing of the mesh (for shape optimization).

    For details on playing, saving, and/or reading the resulting animation, see Animating the Solution (p. 3714).

12. For shape optimization:

    1. Click the **Initialize** button in the **Calculation** group box of the **Gradient-Based Optimizer** dialog box.

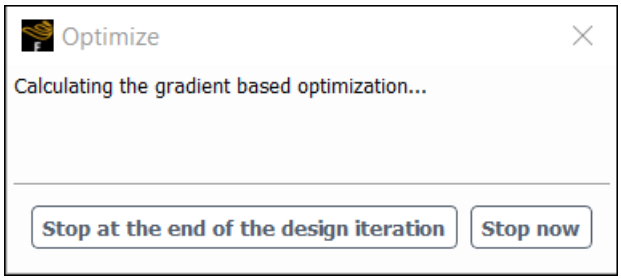
13. For turbulence model optimization:

    1. You can click the **Initialize** button in the **Calculation** group box of the **Gradient-Based Optimizer** dialog box if you want to clear the optimization history, or switch between shape optimization and turbulence model optimization.

14. Click the **Optimize** button to start the design iterations. After clicking the **Optimize** button the **Optimize** dialog box will open, which can be used to stop the optimization at the end of the current design iteration or stop the optimization immediately by clicking the **Stop at the end of the design iteration** button or the **Stop now** button, respectively.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 48.29: Optimize Dialog Box



The optimizer will proceed until one of the following conditions is met:

- the **Convergence Criteria** is met for all of the observables
- the **No. of Design Iterations** is reached for this optimizer run
- the mesh quality requirements cannot be met (for shape optimization)
- you interrupt the calculation
- an error occurs

If you want to continue to optimize the design with revised settings for the observables, operating conditions, objectives, and/or the **adaptive-step-size** option, it is recommended that you either **Initialize** the optimizer again or **Reset** it: the current design solution files will be saved as the optimal if optimization continues, and (for **Initialize**) the **Curr. Design Iteration** will be reset to 0 and the optimization history in the monitor plot will be lost.

When the optimization is complete, you can click the **Summarize** button to print a report of the results for each design iteration in the console, including the observable values, the expected changes, whether the flow and adjoint calculations converged, and the mesh quality metrics.

For turbulence model optimization, after optimizing the design variables in offline mode (e.g Field Inversion), continue to the **Offline training** procedure outlined in Offline Mode Optimization (p. 4602). For online mode optimization, continue to Managing Trained Turbulence Models (p. 4613) if you would like to export the neural network turbulence model.

When using the **Gradient-Based Optimizer**, note the following:

- It is a good practice to initially run the optimizer for one design iteration, in order to check if the settings are proper.
- The resulting design will be highly affected by whether or not the flow solver and/or adjoint solver reach convergence. Auto-adjust controls and blend stabilization strategies are recommended for convergence of the adjoint solver.
- You may want to perform a first round of optimization using lower accuracy settings, and then switch to higher accuracy settings as you approach the optimal solution. For the first round you could consider using the following:
  - settings (for example, in the **Solution Method** task page) that help flow convergence and/or reduce the residual oscillation

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

– a coarser mesh (for shape optimization)

- If you change the settings during the optimization (which can change an observable value), you should make sure to initialize or reset, so that the optimizer considers the current solution to be optimal and starts the optimization from the current solution; otherwise, the changed observable value may be worse than in previous calculations, and if the **adaptive-step-size** option is enabled, Fluent will restore the saved optimal solution and continue, which will undo your settings changes.

48.2.6.1. Using the Turbulence Model Design Tool

Adjoint-based turbulence model optimization is supported by the adjoint solver within the **Gradient-Based Optimizer**. Turbulence model optimization utilizes the adjoint method and machine learning to build an augmented turbulence model that correlates flow features with turbulence model parameters (e.g **CSEP** or **CMIX**). The goal of turbulence model optimization is to reduce the difference between the computed results of a **GEKO** simulation and the results computed by a high-fidelity simulation or experimental data.

Prior to using the **Gradient-Based Optimizer** for turbulence model optimization you must perform the following:

1. Enable the **GEKO** turbulence model within the **Viscous Model** (see Setting up the Generalized k-ω (GEKO) Model (p. 2090)).

2. Enable **Adjoint Turbulence** within the **Adjoint Solution Methods** dialog box (see Using the Adjoint Solution Methods Dialog Box (p. 4526)).

3. Create and edit the adjoint **Observables** specific to turbulence model optimization as outlined in Defining Observables for Turbulence Model Optimization (p. 4525).

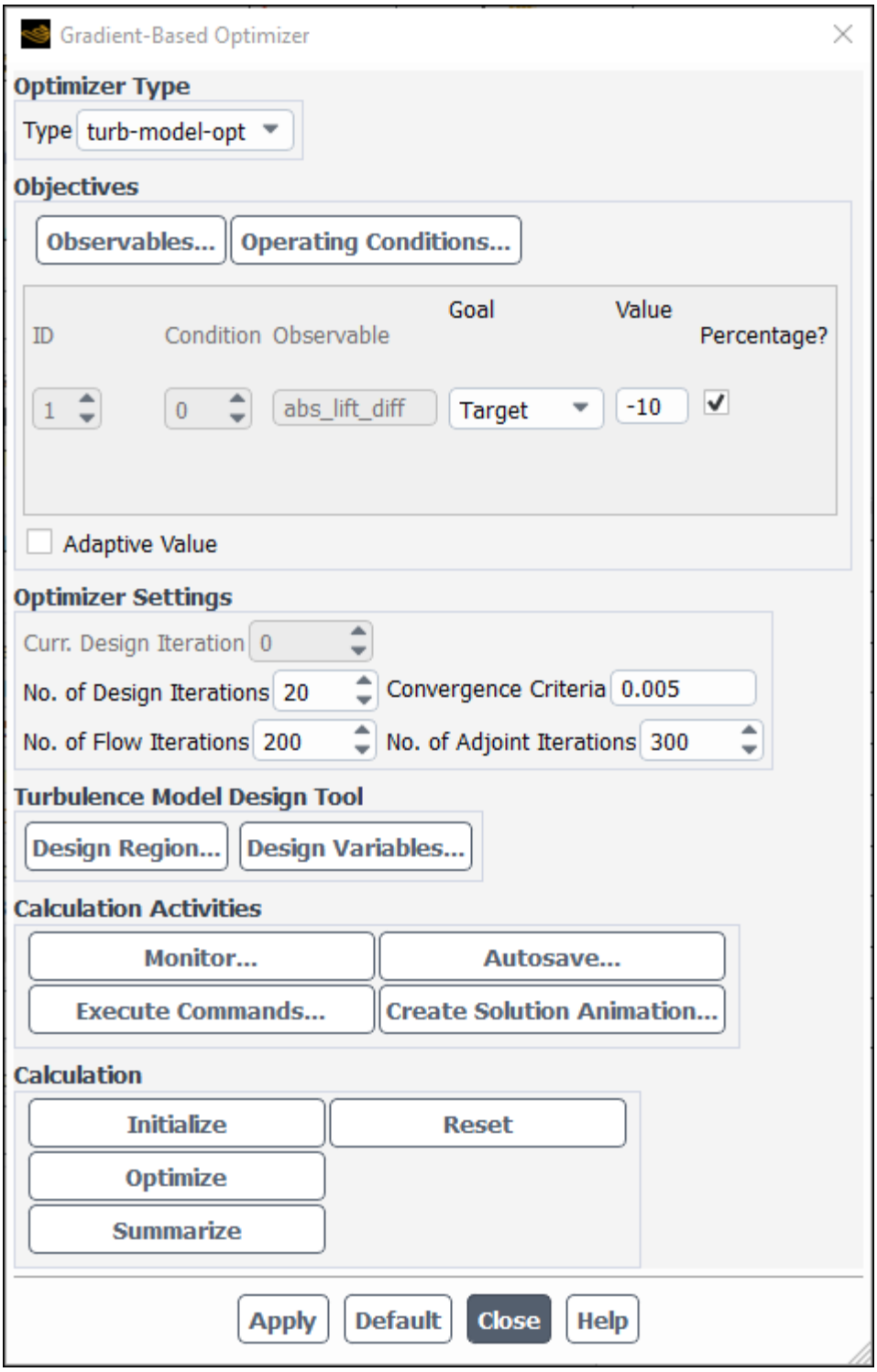
4. Specify all relevant settings for your optimization problem within the **Gradient-Based Optimizer** dialog box as outlined in Using the Gradient-Based Optimizer (p. 4583).

5. Prior to defining optimization settings outlined in Defining Optimizer Design Variable Settings (p. 4604), it is recommended to plot sensitivity contours for the **GEKO** coefficients of interest (e.g **Contours** of **Sensitivities...** to **Sensitivities to GEKO CSEP**) to observe their influence on your RANS simulation (e.g **GEKO** simulation). Selecting **Design Variables** for **GEKO** parameters with large sensitivity magnitudes will increase the controllability of tuning during optimization.

Design → **Gradient-Based** → **Gradient-Based Optimizer...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 48.30: Gradient-Based Optimizer Dialog Box



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

48.2.6.1.1. Turbulence Model Optimization Strategies

Turbulence model optimization can be performed using one of two strategies referred to as offline mode optimization and online mode optimization.

Note:

Online mode optimization is currently the recommended optimization method.

In online mode, the **Design Variables** are a specific function of flow **Input Features** in the form of a **Neural Network**. The neural network model has a topology defined within **Optimizer Design Variable Settings** and during optimization the design variables are optimized directly. After optimization, the correlation between the optimized design variables and flow input features are known and do not need to be learned through **Offline Training**. Refer to Online Mode Optimization (p. 4602) for the online mode optimization workflow.

In offline mode, the **Design Variables** are represented as varying coefficient fields and are not a function of flow **Input Features** .To compute the optimal design variable fields, field inversion is performed using gradient-based optimization. When optimization is complete, the optimized solution is generalized through **Offline Training** which uses machine learning to create the correlation between the optimized **Design Variable** field and flow **Input Features** . Refer to Offline Mode Optimization (p. 4602) for the offline mode optimization workflow.

When deciding whether to use offline mode or online mode optimization, consider the following advantages and disadvantages of both methods:

- For online mode optimization it is not necessary to define **Offline Training** settings or train the **Neural Network** after optimization, which simplifies the workflow. Optimization is model consistent as well because the neural network model is integrated within the entire optimization process. Additionally if the optimization results are acceptable, the optimized model will be numerically stable and demonstrate improved performance when compared to the original flow solution.

  The disadvantage of online mode is that **Neural Network** settings must be defined prior to optimization, and therefore training settings (which are defined in **Offline Training**) can not be customized after optimization.

- For offline mode optimization, the **Neural Network** model is not used during optimization (field inversion) and there are no pre-defined neural network settings that could restrict the optimizer from solving for the optimal **Design Variables**. This allows for more extensive optimization and the flexibility for the optimizer to solve for all possible solutions. Settings for **Offline Training** are specified after the design variables have been optimized, and various training settings can be explored and customized after optimization.

  The disadvantage of offline mode is that the **Models** being used for optimization and training are not consistent, which can lead to some optimization information being lost during **Offline Training** . When training is complete and when the trained turbulence model is hooked to the flow solver, the results may deteriorate and the trained model may be numerically unstable.

Offline and online mode optimization methods can also be used together to find the appropriate **Neural Network** settings prior to training the model. This can be done by initially performing offline mode optimization (including **Offline Training**) as a pre-training step to find the appro-

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

priate neural network **Model** settings (e.g flow **Input Features** and **Neural Network Topology**) for your optimization problem. Once the appropriate neural network settings have been configured, you can perform the online mode optimization workflow using the appropriate neural network settings.

48.2.6.1.1.1. Online Mode Optimization

For performing turbulence model optimization in online mode, the general procedure is as follows:

1. Online Mode Optimization:
    a. Define the **Design Region** (see Defining the Design Region (p. 4603)).
    b. See Defining Design Variables for Optimization (p. 4605) for information on defining the following:
        i. Select the design variables to be optimized.
        ii. (optional) Initialize the design variable field and set the design variable **Limits**.
    c. Select **Neural Network** from the **Type** drop-down list for the optimization **Model**, which will specify the design variables as a function of flow **Input Features** (see Defining the Model for Design Variables (p. 4606)).
    d. Configure the neural network settings (see Defining Neural Network Model Settings (p. 4607)).
    e. Click the **Apply and Hook Model** button so that `adjoint-model` updates the **GEKO** model settings during optimization (see Defining the Model for Design Variables (p. 4606)).
    f. Click the **Close** button to close the **Optimizer Design Variable Settings** dialog box.
    g. Within the **Gradient-Based Optimizer** dialog box, perform the optimization procedure as outlined in Using the Gradient-Based Optimizer (p. 4583).
    h. (optional) Export the trained turbulence model to be applied on a similar case (see Managing Trained Turbulence Models (p. 4613)).

48.2.6.1.1.2. Offline Mode Optimization

For performing turbulence model optimization in offline mode, the general procedure is as follows:

1. Field Inversion:
    a. Define the **Design Region** (see Defining the Design Region (p. 4603)).
    b. See Defining Design Variables for Optimization (p. 4605) for information on defining the following:
        i. Select the design variables to be optimized.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ii. (optional) Initialize the design variable field and set the design variable **Limits**.

c. See Defining the Model for Design Variables (p. 4606) for information on defining the following:

i. Use the **None** design function **Type** for the optimization **Model** which will specify the design variables as a varying coefficient field to be optimized using field inversion.

ii. Click the **Apply and Hook Model** button so that `adjoint-model` updates the **GEKO** model settings during optimization.

d. Click the **Close** button to close the **Optimizer Design Variable Settings** dialog box.

e. Within the **Gradient-Based Optimizer** dialog box, perform the optimization procedure outlined in Using the Gradient-Based Optimizer (p. 4583).

f. When optimization is complete, verify the results by plotting sensitivity contours for the optimized design variable field (e.g **Contours** of **Sensitivities...** to **Adjoint GEKO CSEP**). In general, the change to the GEKO coefficients will be very local near the wall boundary and wake region.

2. Offline Training:

a. After checking the distribution of the optimized design variable field, click the **Design Variables...** button within the **Gradient-Based Optimizer** to reopen the **Optimizer Design Variable Settings** dialog box.

b. Select **Neural Network** from the **Type** drop-down list for the optimization **Model**, which will specify the design variables as a function of flow **Input Features** (see Defining the Model for Design Variables (p. 4606)).

c. Specify the neural network **Settings** (see Defining Neural Network Model Settings (p. 4607)).

d. Perform the procedure for training the neural network turbulence model outlined in Defining Offline Training Settings (p. 4611), which includes:

- Configuring **Offline training** settings.
- (optional) Managing training data used for training the model.
- Training the neural network model.

e. Verify the training results and apply the trained trained turbulence model (see Verifying the Trained Turbulence Model (p. 4613)).

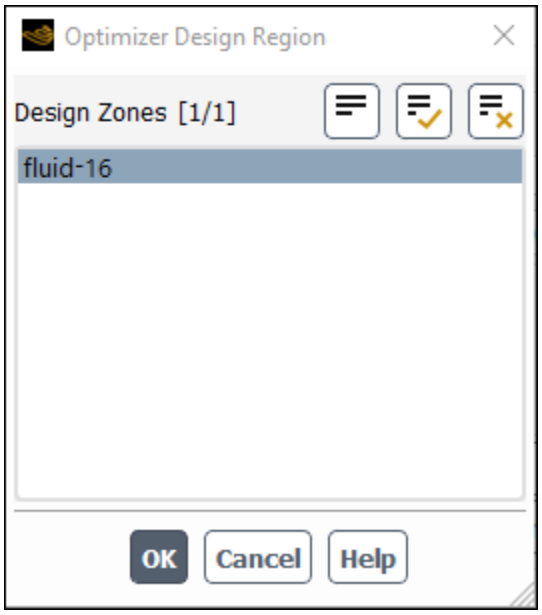
f. (optional) Export the trained turbulence model to be applied on a similar case (see Managing Trained Turbulence Models (p. 4613)).

48.2.6.1.2. Defining the Design Region

Prior to defining **Optimizer Design Variable Settings** , click the **Design Region...** button within the **Turbulence Model Design Tool** group box to open the **Optimizer Design Region** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 48.31: Optimizer Design Region Dialog Box



1. Specify the desired **Design Zones** for your optimization problem.

   In general, regions near walls and the wake region are of interest.

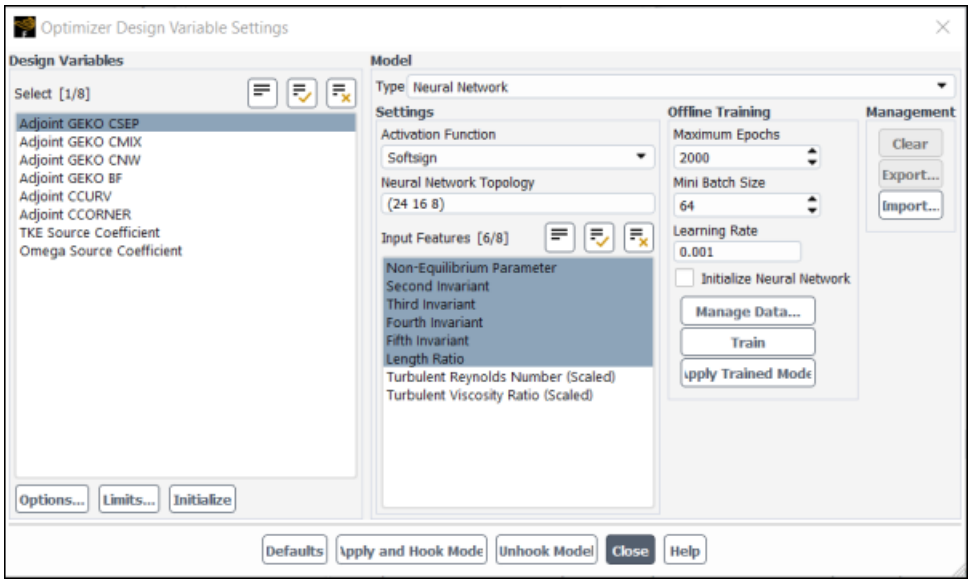
2. After selecting the **Design Zones**, click **OK** to close the **Optimizer Design Region** dialog box.

48.2.6.1.3. Defining Optimizer Design Variable Settings

This section defines the various **Optimizer Design Variable Settings** and provides instruction on how to define them for both offline and online mode optimization strategies.

1. Click the **Design Variables...** button within the **Gradient-Based Optimizer** dialog box to open the **Optimizer Design Variable Settings** dialog box.

Figure 48.32: Optimizer Design Variable Settings Dialog Box



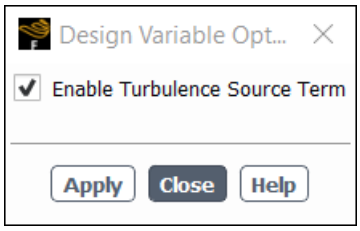
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

48.2.6.1.3.1. Defining Design Variables for Optimization

When choosing **Design Variables** to optimize, consider the following:

1. You can click the **Options...** button to open the **Design Variable Options** dialog box which allows you to turn on the **Enable Turbulence Source Term** option.

   Figure 48.33: Design Variable Options Dialog Box

   This option will enable the turbulence source coefficient design variables (TKE and Omega). When enabled, $-S_k*k$ and $-S_\omega*\omega$ will be added as source terms to the transport equations for turbulence kinetic energy (k) and specific dissipation rate (ω).

   Where,

   S_k is the **TKE Source Coefficient**

   And,

   S_ω is the **Omega Source Coefficient**

   You can plot contours of **Machine Learning Tke Coefficient** and **Machine Learning Omega Coefficient** (both in the **Turbulence...** category) to observe their influence.

2. Sensitivities for **GEKO** coefficients of interest can be plotted to observe their influence (for example, **Sensitivities to GEKO CSEP** in the **Sensitivities...** category). Selecting **Design Variables** for **GEKO** parameters with large sensitivity magnitudes will increase the controllability of tuning during optimization.

3. In general, selecting **TKE Source Coefficient** as the design variable is recommended for the following reasons:

   - More tuning flexibility compared to **GEKO** coefficients such as **CMIX**, which can be deactivated by the blending function.
   - Works independently along with **GEKO** coefficients with the highest sensitivities.

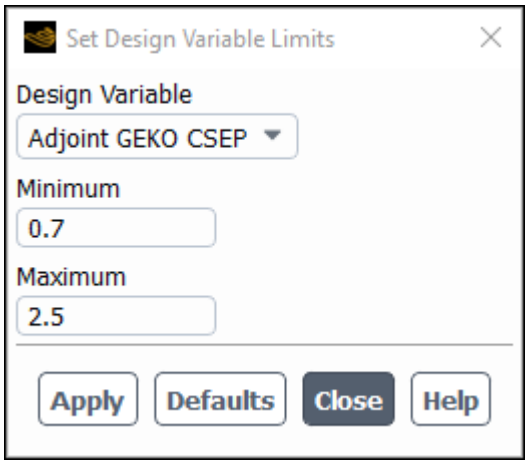
   If you choose not use **TKE Source Coefficient**, generally **CSEP** and **CMIX** are the **GEKO** coefficients of interest. However, note that when optimizing **CMIX** for a bluff body problem, you may need to fix the blending function as the built-in blending function may deactivate **CMIX** in the wake region

4. Refer to the definitions of the **GEKO** coefficients in section Generalized k-ω (GEKO) Model of the theory guide to see if they are physically relevant.

You can click the **Limits...** button to open the **Set Design Variable Limits** dialog box and specify the upper and lower bounds of the design variables during optimization.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 48.34: Set Design Variable Limits Dialog Box



1. For bluff body problems and when **Adjoint GEKO CMIX** is selected from the list of **Design Variables**, you may need to increase the range so that **CMIX** can mimic the mixing effect of large unsteady vortex shedding. Too large of a limit however may break the calibration of basic cases.

2. After specifying the **Design Variable Limits**, click **Close** to close the **Set Design Variable Limits** dialog box.

You can click the **Initialize** button to initialize the field for the selected **Design Variables** with the default Fluent values.

- Note that when the blending function is selected from the list of **Design Variables** , **Adjoint GEKO BF** will be initialized with the blending-function that is built into the **GEKO** settings by default. The built-in blending function is dependent on the flow state and therefore it is necessary to converge the flow solver prior to initializing **Adjoint GEKO BF** .

48.2.6.1.3.2. Defining the Model for Design Variables

From the **Type** drop-down list of the **Model** group box, you can select the design function that will be used for optimization.

- For optimizing the **Design Variables** in offline mode, choose **None** for the design function. This will specify the **Design Variables** as a field that will be optimized using gradient-based optimization (Field Inversion).

  – When using **None** as the optimization **Model** you can enable the **Smooth Sensitivities** option which generally helps to improve optimization by adding regularization to the updated sensitivity field.

- For offline training and online mode optimization, select **Neural Network** for the design function **Type**. This will specify the **Design Variables** as a function of flow **Input Features** in the form of a neural network.

During optimization for both optimization methods, the `adjoint-model` *is automatically hooked to the relevant* ***GEKO*** *coefficients within the* ***Viscous Model****. The* `adjoint-model` *can be unhooked at anytime to use the default* ***GEKO*** *settings by clicking the* ***Unhook Model*** *button. Alternatively,*

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

*you can click the **Apply and Hook Model** button to apply the `adjoint-model` to the **GEKO** coefficients and overwrite the default **GEKO** coefficients using the trained model during optimization.*

48.2.6.1.3.2.1. Defining Neural Network Model Settings

The following **Neural Network** settings should be defined when using **Offline Training** or online mode optimization:

Activation Function

This specifies the nonlinear activation function of nodes within the hidden layers of the neural network during optimization.

Note:

Although various **Activation Functions** are provided for the **Neural Network**, the **Activation Function** does not appear to have a significant affect on optimization or training results.

---

tanh

$$f(x)=\frac{e^{x}-e^{-x}}{e^{x}+e^{-x}}$$

The **tanh** activation function is a sigmoidal hyperbolic tangent function with an output range of `-1` to `1`.

Softsign

$$f(x)=\frac{x}{1+|x|}$$

The **Softsign** activation function has an output range of `-1` to `1` and is similar to the **tanh** function, where tanh grows exponentially and Softsign grows polynomially. Softsign has the advantage over tanh in that Softsign is less likely to become saturated when there are large or small inputs to the function. The Softsign function also has a zero-centered range which increases the learning effectiveness. However when the input is zero, nodes will become inactive which can cause gradient saturation.

ReLu

$$f(x)=\max(0,x)$$

The **ReLu** (Rectified Linear Unit) activation function is an unbounded activation function that is linear in the positive dimension, but zero in the negative dimension.

The advantages of using the ReLu activation function include:

- The activation of nodes in the network is minimal. In a randomly initialized network, only about 50% of hidden nodes are activated (have a non-zero output).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- The gradient propagation is improved when compared to bounded sigmoidal activation functions (e.g **tanh** and **Softsign**) that can saturate in both directions causing gradient propagation to stop completely throughout all nodes in the network.
- Computation of the **ReLu** activation function is more efficient compared to sigmoidal activation functions.

The disadvantage of using the ReLu activation function is that nodes in the network can be pushed into an inactive state for all inputs which stops the propagation of the gradient through the network. This is known as the vanishing gradient gradient problem and usually arises when the **Learning Rate** is set too high. This problem can be avoided by decreasing the **Learning Rate**, or by using the **Leaky-ReLU** activation function. However note that the neural network performance is reduced when using the **Leaky-ReLU** activation function.

Leaky-ReLu

$$f(x)=\max(0.01x,x)$$

The **Leaky-ReLU** activation function is a linear variant of the **ReLu** activation function that outputs small positive gradient values at inactive nodes. The Leaky-ReLu function can be used as an alternative to the ReLu activation function if there is no gradient flowing backward through the nodes or if a large number of nodes in the network are inactive when using the ReLu activation function.

Neural Network Topology

Defines the number of hidden layers in the neural network and the number of nodes in each hidden layer. By default, the neural network has three hidden layers consisting of 24, 16, and 8 nodes for the first, second, and third layer, respectively. The input layer consists of the selected **Input Features** and the output layer outputs the **Design Variables**. A large neural network topology will be able to build a better correlation between **Input Features** (input) and **Design Variables** (output). However, large neural network topologies can be computationally expensive and may not be easy to generalize for use in other cases. A neural network topology that is too small may have more difficulty correlating the **Input Features** (input) and **Design Variables** (output).

Input Features

Selecting appropriate flow **Input Features** for your specific optimization problem is crucial for creating a succesful neural network model. By default, eight flow input features are available under the **Input Features** list, which have been carefully chosen based on the following physical principles:

- Galilean invariance
- Rotational invariance
- Non-dimensional
- Scaled

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The following equations define the strain and rotation terms used to derive the invariant input features:

- Strain rate tensor

$$S_{ij}=\frac{1}{2}\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right) \tag{48.56}$$

- Rate of rotation tensor

$$\Omega_{ij}=\frac{1}{2}\left(\frac{\partial u_i}{\partial x_j}-\frac{\partial u_j}{\partial x_i}\right) \tag{48.57}$$

- Strain rate magnitude

$$s^2=\frac{\partial u_i}{\partial x_j}\frac{\partial u_i}{\partial x_j}=S_{ij}S_{ji}-\Omega_{ij}\Omega_{ji} \tag{48.58}$$

With,

$$S'_{ij}=\frac{S_{ij}}{s} \tag{48.59}$$

$$\Omega'_{ij}=\frac{\Omega_{ij}}{s} \tag{48.60}$$

The default flow input features are defined as follows:

Note:

The **Turbulent Reynolds Number (Scaled)** and **Turbulent Viscosity Ratio (Scaled)** input features are only appropriate for low Reynolds number flows or for differentiating the boundary layer region with other regions.

Non-Equilibrium Parameter

$$\phi_1=\frac{s}{0.3\omega}-1 \tag{48.61}$$

Where ω is the specific dissipation rate.

Output range: [-1,3]

Second Invariant

$$\phi_2=II'_s \tag{48.62}$$

Where,

$$II'_s=S'_{ij}S'_{ji} \tag{48.63}$$

And,

$$II'_\Omega=\Omega'_{ij}\Omega'_{ji} \tag{48.64}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Due to non-dimensionalization,

$$II'_S - II'_\Omega = 1 \tag{48.65}$$

Output range: [0,1]

Third Invariant

$$\phi_3 = III'_S \tag{48.66}$$

Where,

$$III'_S = S'_{ij} S'_{jk} S'_{ki} \tag{48.67}$$

Output range: [0,1]

Fourth Invariant

$$\phi_4 = IV' \tag{48.68}$$

Where,

$$IV' = S'_{ij} \Omega'_{jk} \Omega'_{ki} \tag{48.69}$$

Output range: [0,1]

Fifth Invariant

$$\phi_5 = V' \tag{48.70}$$

Where,

$$V' = S'_{ij} S'_{jk} \Omega'_{kl} \Omega'_{li} - \frac{1}{2} II'_S II'_\Omega \tag{48.71}$$

Output range: [0,1]

Length Ratio

$$\phi_6 = \frac{1}{20} \frac{\sqrt{k}}{0.09 \omega y} \tag{48.72}$$

Where k and y are the turbulence kinetic energy and position in the y-direction respectively.

Output range: [0,2]

Turbulent Reynolds Number (Scaled)

$$\phi_7 = \mathrm{Re}_{y'} = \frac{1}{200} \frac{\sqrt{k}\, y}{\nu} \tag{48.73}$$

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Where ν is the molecular viscosity.

Output range: [0,2]

Turbulent Viscosity Ratio (Scaled)

$$\phi_8 = \frac{1}{100}\frac{\nu_t}{\nu} \tag{48.74}$$

Where $\frac{\nu_t}{\nu}$ is the turbulent viscosity ratio.

Output range: [0,2]

Flow input features that are not contained in the above list can be created by defining an **Expression**, and up to six expressions are allowed.

When optimization is complete, all input features can be plotted as **Contours** of **Sensitivities...** to the selected features (e.g **Contours** of **Sensitivities...** to **Non-Equilibrium Parameter**).

For specific cases and when performing online mode optimization, you can deselect all **Input Features** and enter ( ) for the **Neural Network Topology**, to use no hidden layers. This will specify the design variables as a constant field and the neural network model will be reduced to optimize the design variables as a global constant.

48.2.6.1.3.2.2. Defining Offline Training Settings

During training, the neural network will learn the correlation between the optimal **Design Variable** field and flow **Input Features** to produce a generalized optimal solution. Note that offline training should only be performed after performing offline mode optimization (Field Inversion). The procedure for **Offline training** is as follows:

1. Specify the **Neural Network** settings as outlined in Defining Neural Network Model Settings (p. 4607) .
2. Define the following **Offline Training** settings:

   Maximum Epochs

   Defines the maximum number of cycles ran through the full training data set.

   Mini Batch Size

   Specifies the number of data samples (number of cells) used for correcting the neural network each time during the training process. The mini batch size should not be too large so the gradient can have some noise to escape from the local minimum or saddle points. The mini batch size should also not be too small as to approximate the full neural network gradient. A small mini batch size can be computationally expensive as well. A mini batch size between `10` to `1000` should be appropriate for most cases.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Learning Rate

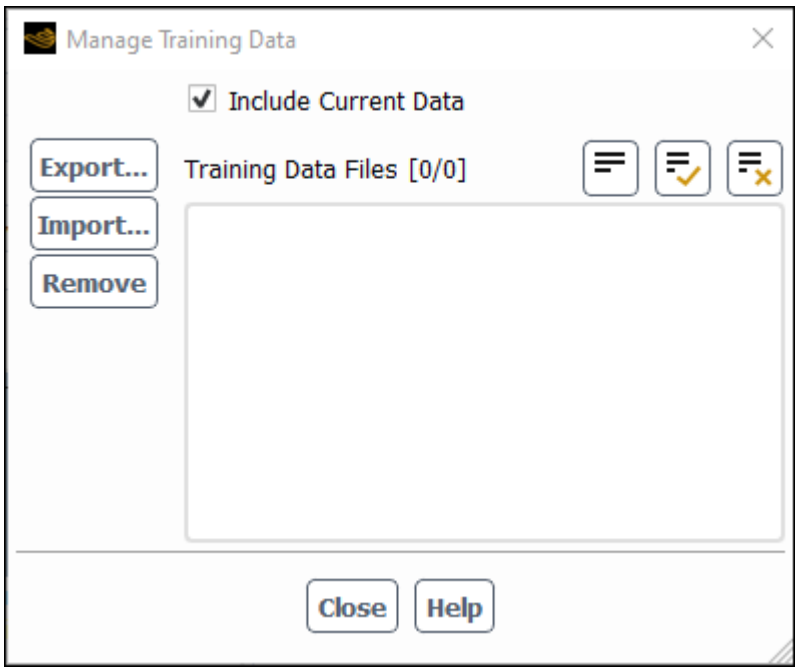
Defines the amount the neural network model adjusts in response to error after each iteration. A large learning rate value will increase the speed of the learning process, however it may lead to oscillation and divergence.

Initialize Neural Network

When enabled, this will initialize the neural network model with random values during **Offline Training**. When disabled, training will continue on the existing model if there is any present.

3. Click the **Manage Data...** button to open the **Manage Training Data** dialog box.

Figure 48.35: Manage Training Data Dialog Box



The **Manage Training Data** dialog box allows you to use training data (which contain selected flow **Input Features** and **Design Variables**) from a different case or multiple different cases to train the neural network model for your current case. Training data can be exported as a `.tdat` file from your current case by clicking the **Export...** button. Alternatively, you can import multiple `.tdat` files from different cases by clicking the **Import...** button. If **Include Current Data** is enabled, training will be based on any existing training data from your current case as well as any imported training data that is selected from the **Training Data Files** list. If **Include Current Data** is disabled, then training will be based only on imported training data.

4. Clicking the **Train** button will begin training the **Neural Network** model. Training is parallelized and training information is printed to the console. Statistics about the training data (e.g mean, covariances, and correlations) will be printed to the console as well. If the value for the flow input feature correlation printed to the console is close to `1` or `-1` then the relevant flow input features are redundant and can be removed. Additionally during training, the optimization history will be plotted in the graphics window.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

48.2.6.1.3.3. Verifying the Trained Turbulence Model

The following outlines the procedure for verifying the trained turbulence model created during **Offline training** :

1. When training is complete, you can compare the training results to the original sensitivity field by plotting sensitivity contours for the trained **Design Variables** (e.g **Contours** of **Sensitivities...** to **Adjoint GEKO CSEP (Trained)**). If the difference between the original and trained sensitivity field distributions are large, then you should continue training the model.

2. If the results produced from **Offline training** are acceptable, click the **Apply Trained Model** button to apply the augmented turbulence model to the Fluent turbulence model and update the **Design Variables**. Once the trained **Design Variables** have been applied they will no longer be accesible and the neural network model will be used to overwrite the **GEKO** model coefficients during the flow calculation.

3. Rerun the flow solution using the augmented turbulence model and analyze it's improvement from the original solution. Note that when the neural network model is used as the turbulence model during the flow calculation, the console will report that the turbulence model coefficients (e.g **GEKO** coefficients) are being updated using the neural network model.

48.2.6.1.3.4. Managing Trained Turbulence Models

Within the **Management** group box you can export (from your current case) or import (from a different case), a trained turbulence model as a scheme file (`.scm`).

If the flow solution produced using a trained turbulence model (created on your current case) is satisfactory, the trained model can be exported as a scheme file by clicking the **Export...** button to be used on a different case with similar flow physics.

To use a turbulence model (trained on a different case with similar flow physics) on your current case, you can import the scheme file of the trained model by clicking the **Import...** button.

- Once the trained turbulence model has been imported into your current case, you can simply click the **Apply Trained Model** button, then run the flow solver using the trained turbulence model.

- For **Offline training**, trained models from a different case (deployment case) can be imported to continue training the neural network of your current case (training case).

  Note that data driven turbulence model tuning can be viewed as an advanced interpolation from the training condition to the deployment condition. Therefore the flow conditions and physics of the deployment case should be similar to the training case. If the deployment case and training case have different flow conditions and physics, the trained turbulence model may not be applicable, and the performance can deteriorate.

Additionally, you can click the **Clear** button to delete all the existing information for your current model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

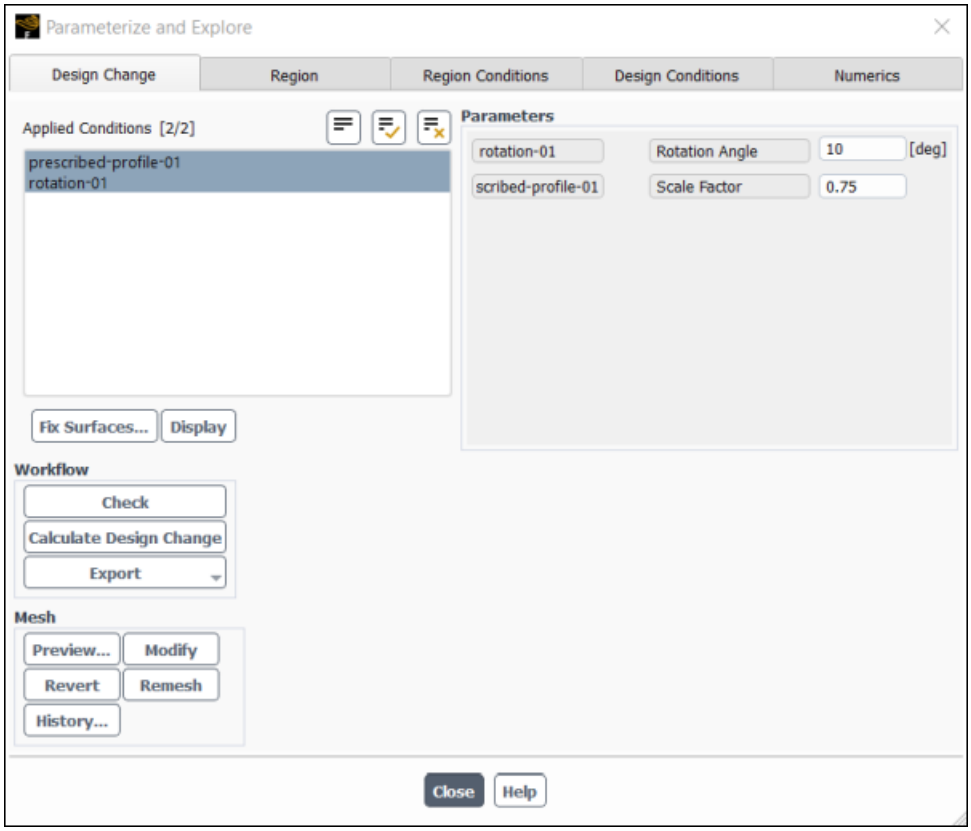
48.3. Geometry Parameterization and Exploration

Geometry parameterization and exploration is available for exploring geometry designs for various design conditions, by means of mesh morphing. When design conditions are defined using input parameters, geometry parameterization and exploration can be used for a parametric study and for performing parametric analysis.

Geometry parameterization can be accessed by clicking **Parameterize and Explore...** in the **Design** ribbon tab (**Geometry** group box).

Design → Geometry → Parameterize and Explore...

Figure 48.36: Parameterize and Explore Dialog Box



The graphical user interface (GUI) of the **Parameterize and Explore** dialog box is very similar to that of the **Design Tool**, but with more simplistic settings. While geometry parameterization is intended for exploring a design change based only on specified design conditions, the goal of the design tool is to compute the optimal freeform deformation using shape sensitivity and design constraints. Additionally, the adjoint solver is not required for geometry parameterization. Therefore only the design change, region, region conditions, design conditions, and numerics settings of the design tool are relevant for geometry parameterization.

Much of the procedure for geometry parameterization is the same as that of the design tool and therefore, much of the procedure below references sections in the design tool for defining the relevant settings. However, many settings described within Modifying the Geometry Using the Design Tool (p. 4553) are not applicable to geometry parameterization.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The general workflow for performing geometry parameterization is as follows:

1. Define the morphing region by clicking the **Region** tab of the **Parameterize and Explore** dialog box.

   Instruction on defining the morphing region within the design tool is the same for geometry parameterization, and is outlined in Defining the Region for the Design Change (p. 4556). Note that only the mesh inside the morphing region will be changed.

2. Define the region conditions by clicking the **Region Conditions** tab of the **Parameterize and Explore** dialog box.

   Specify the region conditions for each coordinate direction:

   a. You can ensure that the deformation is symmetrical relative to the different coordinate directions by enabling the **Symmetric** options. Symmetry can be imposed in as many directions as desired and when enabled, the symmetry planes will be displayed in the graphics window. You should ensure they reflect your intent for the design change, modifying the extents of the deformation region as necessary.

   b. When the **Symmetric** option is enabled, the **Custom Plane?** option becomes available which, when enabled, allows you to enter a custom value for the plane of symmetry.

   c. Periodic morphing can be enforced in the theta direction for cylindrical region conditions by entering the desired number of repeats under **External Periodic**. Note that periodic morphing is distinct from periodic boundary conditions and periodic repeats. For periodic morphing, when the desired number of repeats is `0` or `1`, no repeats will be applied.

   Note:

   Symmetric and **External Periodic** conditions behave differently than the design tool's symmetric and periodic region conditions outlined in Defining Region Conditions (p. 4558).

   d. The **Apply Continuity** option is enabled by default and is recommended to ensure that the nodes at the boundary of the region are fixed.

3. Define the design conditions by clicking the **Design Conditions** tab of the **Parameterize and Explore** dialog box.

   Instruction on defining design conditions for geometry parameterization is the same as that of the design tool design tool and is outlined in Defining Conditions for the Deformation (p. 4560).

   a. Create the desired design conditions for your parametric analysis, defining them as a constant value or defined using an input parameter. The following design conditions can be defined for geometry parameterization:

      - **translation**
      - **scaling**
      - **rotation**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **prescribed-profile**
- **compound**
- **bounded-by-plane**
- **bounded-by-surface** (including fit to surface)

By default, all surfaces inside the morphing region (defined within the **Region** tab) may be modified unless specified by a design condition. Therefore it is important to define the **fixed-walls** constraint to fix any surfaces that should not be modified. Additionally, unlike the **Design Tool** where only the mesh inside the morphing region is morphed, for geometry parameterization, the mesh within a **Symmetric** or **External Periodic** region (defined within the **Region Conditions** tab) will also be morphed.

If you have defined a design condition with an input parameter, the parameter value will appear in the **Parameters** group box of the **Design Change** tab for easy modification and experimental trials. For example, if the **Angle** for a **rotation** condition is defined with an input parameter, the **Rotation Angle** parameter value will be shown in the **Design Change** tab.

4. (optional) Click the **Numerics** tab of the **Parameterize and Explore** dialog box.

   Adjusting the numerics settings for the design change is optional and the settings generally do not need to be changed. However, numerics settings can be adjusted as follows:

   a. Specify the value for the **Constraints**. This value is the absolute tolerance for convergence of the residuals associated with the design conditions, including fixed zone conditions in meters. Tighter tolerance may increase both the computational time and memory cost.

   b. **Increase Local Smoothness** is disabled by default, and enabling it will use more local control points to improve the smoothness of the deformation.

   c. **Increase Global Smoothness** is disabled by default, and enabling it will use more global control points to improve the smoothness of the deformation.

5. Calculate the design change and modify the mesh by clicking the **Design Change** tab of the **Parameterize and Explore** dialog box.

   After creating one or multiple design conditions for your parametric analysis, they will become available in the **Applied Conditions** list of the **Design Change** tab.

   Much of the procedure for calculating the design change and modifying the geometry within the **Design Tool** is the same for geometry parameterization, and is outlined in Shape Modification (p. 4575). However, some steps are not applicable for geometry parameterization, such as morphing and constraint methods and observable settings.

48.4. The Mesh Morpher/Optimizer

The mesh morpher/optimizer offers a variety of non-gradient-based optimization algorithms to optimize the geometric shape of a system [72] (p. 5717). The optimization can be performed for a wide variety of physics and quantities of interest to be optimized.

For additional information, see the following sections:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

48.4.1. Limitations
48.4.2. The Optimization Process
48.4.3. Optimizers

48.4.1. Limitations

Note the following limitations of the Ansys Fluent mesh morpher/optimizer:

- The mesh morpher/optimizer should not be used with dynamic or sliding mesh problems.
- Any geometrically evaluated parameters required for flow iterations (for example, view factors for the surface to surface (S2S) radiation model) will need to be recomputed at the very least when the optimization is complete. If the objective function is affected by one of these geometrically evaluated parameters, then such parameters will need to be recomputed after each design change; for example, the view factors need to be recomputed after each design change when the objective function is a function of temperature.
- Arbitrarily shaped deformation regions are not supported.

48.4.2. The Optimization Process

All optimization problems require that you identify parameters that can be modified in order to reach the optimized solution. In the case of the mesh morpher/optimizer, it is the geometry that must be parameterized. Geometric parameterization for general shapes used in CFD can be very complicated, due to the large variety of shapes available in engineering applications. In order to minimize such complications in your Ansys Fluent simulation, the problem of shape parameterization is reduced to a problem of the parameterization of changes in the geometry.

The next essential requirement for mesh morphing is a tool that can smoothly alter the shape, irrespective of the underlying mesh topology. In Ansys Fluent, designated deformation regions are manipulated via displacements applied to a set of control points. The mesh region that is to be deformed is defined by a "box" (that is, a rectangle for 2D cases and a rectangular hexahedron for 3D), and the control points must be located within the box. The displacements of the control points are the result of user-defined motions (each of which involves a parameter value and other directional settings) and these displacements are then applied to the mesh as a smooth deformation by either interpolating the displacement based on radial basis functions or using the tensor product of Bernstein polynomials.

The next requirement is to have an optimizer that is robust enough to handle a wide range of problems. By coupling such optimizers with your CFD analysis, you can greatly improve your design with minimal intervention. You can use built-in optimizers to vary the parameter values along the prescribed directions and within defined bounds, in order to satisfy a condition specified by an objective function. The mesh morpher/optimizer provides you with access to six optimizers that are not based on gradients. Otherwise, you can manually specify the deformation (that is, define both the parameter values and the directions) and analyze the results; you also have the option of using Design Exploration in Ansys Workbench to easily explore the impact of a variety of parameter values.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

48.4.3. Optimizers

The built-in optimizers used as part of the mesh morpher/optimizer capability in Ansys Fluent use direct search methods for optimization. Direct search methods are zeroth order, as they only use the objective function values for optimization. The direct search methods do not use the derivatives.

The following is a list of advantages of direct search methods:

- Direct search methods do not require derivatives for optimization.
- Direct search methods are robust for problems with discontinuities and in situations where the derivative computation is not possible or unreliable.

The following is a list of disadvantages of direct search methods:

- Convergence proof is not clearly defined.
- The rate of convergence can be very slow.

General explanations of the six different built-in optimizers are provided in the following sections:

48.4.3.1. The Compass Optimizer
48.4.3.2. The NEWUOA Optimizer
48.4.3.3. The Simplex Optimizer
48.4.3.4. The Torczon Optimizer
48.4.3.5. The Powell Optimizer
48.4.3.6. The Rosenbrock Optimizer

48.4.3.1. The Compass Optimizer

In the Compass optimizer [80] (p. 5717), the parameters are adjusted one by one until the objective function is minimized. This optimizer starts with a given value and then evaluates the function value in all the basic directions. The direction here refers to the positive and negative increments to the initial parameter values. If there is a reduction in the function value, then that point becomes an improved point. If there is no improvement in the function values, then the step length is reduced by half and the search is repeated in all directions. The algorithm terminates when the step size falls below a certain tolerance.

The Compass optimizer initially makes rapid progress towards the solution. While this method might quickly approach the minimum value of an objective function, it may be slow to detect this fact. It may also converge very slowly if the level sets of the objective function are extremely elongated.

48.4.3.2. The NEWUOA Optimizer

The NEW Unconstrained Optimization Algorithm (NEWUOA) optimizer [124] (p. 5720) attempts to find the least value of a function $F(x)$, where x is a parameter vector of n dimensions. At the start of every iterative step, a quadratic model approximation ($Q \sim F$) is constructed and the minimization is performed on a trust region (that is, a region around the parameter that is limited by an initial parameter variation). The perturbation to the parameter value needed to obtain the lowest value of Q is evaluated during the iteration, and correspondingly, the new least value of $F(x)$ is obtained.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The iterative process continues until the trust region is reduced to the optimizer convergence criterion, and the optimizer exits with the least value of the objective function. This optimization algorithm can be applied to any setup case of the mesh morpher/optimizer.

The NEWUOA optimizer provides the least value of the objective function in a manner that is highly accurate and robust. The main advantage of this optimizer is that it is very fast; hence, this optimizer is recommended for problems that have a large number of parameters.

48.4.3.3. The Simplex Optimizer

The Simplex optimizer is also referred to as the Downhill Simplex optimizer [98] (p. 5718), [72] (p. 5717) and the Nelder-Mead method [105] (p. 5719). It is based on the idea of geometric simplexes; for example, a 2D simplex is a triangle, and a 3D simplex is a tetrahedron.

For optimization purposes, Ansys Fluent requires that simplexes are regular polyhedra (that is, not degenerate polyhedra with collapsed sides). Each vertex of the geometric simplex represents one function evaluation (which in this case is one CFD run), and the number of vertices corresponds to the number of parameters. For the free-form deformation method that is used by Ansys Fluent to parameterize changes in shapes, the number of active control points will determine the number of vertices of the geometric simplex.

Minimization of the objective function is performed based on a set of the rules about the "quality" of each vertex. The vertex quality is the value of the function evaluated for each position of the control point. A set of geometric operations such as reflection, expansion, contraction, and shrinking are performed in order to find the region in which to look for the minimum of the function. Because optimization here is formulated as a minimization problem, the simplex optimizer algorithm seeks the "worst" vertex, that is, the vertex that has the largest value when the corresponding parameter is evaluated. By performing the reflection around the center of the gravity, the new value of the function is obtained after performing the CFD run. Similarly, the operations of expansion, contraction, and shrinking are used to obtain the minimum of the function.

The Simplex optimizer is known to work well, but it suffers from the large number of function evaluations. It also requires smooth objective functions for convergence.

48.4.3.4. The Torczon Optimizer

The Torczon optimizer [165] (p. 5722) is a slightly modified version of the simplex optimizer described previously. Given an initial vertex, this optimizer tries to find a better vertex that has a function value that is strictly less than the function value at the previous best vertex. There are three possible trial steps: the rotation step, the expansion step, and the contraction step. The algorithm always computes the rotation step and then tests to see if a new best vertex has been identified. If it has, then the expansion step is computed. Otherwise, the algorithm computes and automatically accepts the contraction step.

48.4.3.5. The Powell Optimizer

For the Powell optimizer, the method used is based on the number of dimensions of the problem. For optimization problems of single dimension (that is, problems with a single parameter), the golden section search algorithm [125] (p. 5720) is used to find the optimal value for the objective function. In this algorithm, the optimal value is found by reducing the size of the bracketing triplet, until the size of the bracket (that is, the distance between the outer points of the triplet) reaches

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

a certain tolerance level. In all the cases, the middle point of the new triplet is determined to be the best value obtained so far.

For multidimensional problems (that is, problems with multiple parameters), the minimum value and the largest decrease is found using the golden search algorithm for each dimension, in order to find the conjugate directions. A conjugate direction is a direction that when searched will not alter the minimum value attained by the previous movement in another direction—that is, a direction in which the gradient is perpendicular to the first direction. After finding the N linearly independent, mutually conjugate directions, one pass will find the exact minimum value. For functions that are not quadratic, repeated cycles of N line minimizations will converge to minimum.

48.4.3.6. The Rosenbrock Optimizer

In the Rosenbrock optimizer [132] (p. 5720), after the initial direction is found, multiple steps are taken in that direction until the least value is attained. The process starts with an arbitrary length e. If this initial step succeeds (that is, the new value of the function is less than or equal to the old value), the length e is multiplied by α, where α is more than 1. If the steps fails, the length e is multiplied by β, where β is between -1 and 0. The direction or the parameter that must be modified is determined by advancing all the parameters by the step length e and then selecting the best among those that yield a function value that is less than the previous value. After that point is accepted as the best point, the process is repeated. These steps continue to repeat until e becomes so small so that any further change in the value of e does not significantly reduce the value of the function.

48.5. Using the Mesh Morpher/Optimizer

The procedure for setting up and using the mesh morpher/optimizer for shape optimization is as follows:

1. Read the case into Ansys Fluent.

   File → **Read** → **Case...**

2. If you want to use one of the built-in optimizers rather than specifying the deformation manually or using Design Exploration in Ansys Workbench to explore multiple deformation scenarios, you will need to provide an objective function in one of three ways: either as a user-defined function (UDF), a Scheme source file, or a customized function that is based on output parameters (that is, values from flux, force, surface integral, or volume integral reports). The goal of the optimizer is to deform the mesh in such a way that this objective function is minimized. Ansys Fluent will run the solution for a given design stage until convergence is reached, and then check to see if the objective function is satisfied. If the objective function is not satisfied, Ansys Fluent will proceed to the next design, and so on, until convergence is achieved from the point of view of the optimizer. You can specify the optimizer convergence criteria in step 12.d.iii.

   Note that if you plan to use the **newuoa** optimizer, you must define the objective function as a UDF.

   a. If you want to provide the objective function as a user-defined function (UDF), perform the steps that follow. For more information about UDFs, see the separate Fluent Customization Manual.

      i. Write a UDF using the `DEFINE_ON_DEMAND` macro to define the objective function. The function name must be `objective_function`. At the end of the objective function, the

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

rpvar `morpher/objective_function` must be set to the `(current - target)` value. It is this value that the optimizer will attempt to minimize.

ii. Compile the UDF using the **Compiled UDFs** dialog box. Make sure `libudf` appears as the **Library Name**.

User Defined → **User Defined** → **Functions** → **Compiled...**

b. If you want to provide the objective function as a Scheme source file, perform the following steps:

i. Write a Scheme source file to define the objective function. The procedure name must be `objective-function`. At the end of the objective function, the rpvar `morpher/objective-function` must be set to the `(current - target)` value. It is this value that the optimizer will attempt to minimize.

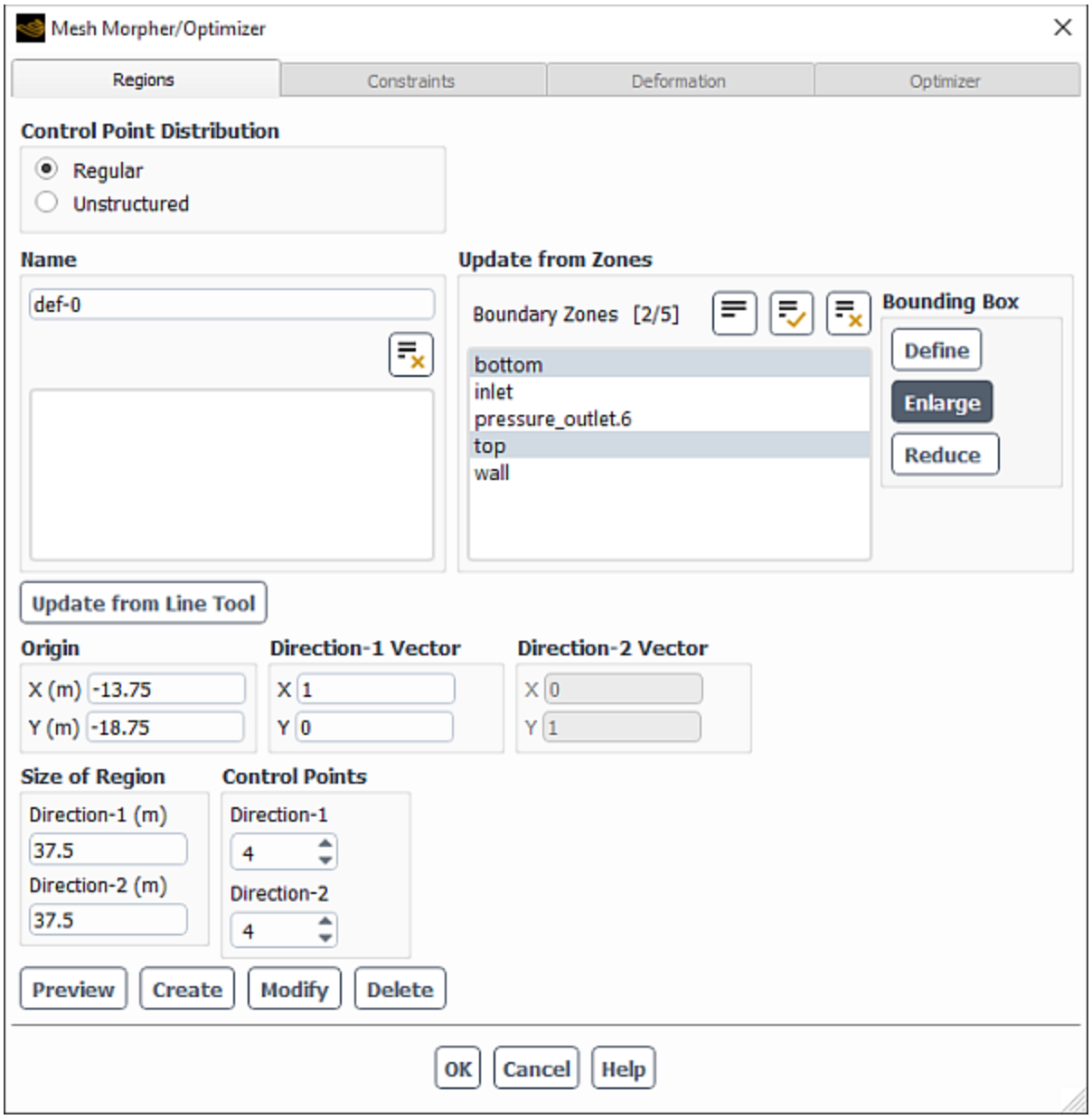
ii. Load the Scheme source file (see Reading Scheme Source Files (p. 972) for details).

3. Open the **Mesh Morpher/Optimizer** dialog box by clicking **Optimizer...** in the **Design** ribbon tab (**Parameter-Based** group box).

The first time you click **Optimizer...** in a session, a question dialog box will ask you whether you want to enable the mesh morpher/optimizer. Click **Yes** to load the libraries and open the **Mesh Morpher/Optimizer** dialog box (Figure 48.37: The Regions Tab of the Mesh Morpher/Optimizer Dialog Box (p. 4622)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 48.37: The Regions Tab of the Mesh Morpher/Optimizer Dialog Box



4. Specify the manner in which you would like to define the locations of the control points, by making a selection from the **Control Point Distribution** list in the **Regions** tab of the **Mesh Morpher/Optimizer** dialog box (Figure 48.37: The Regions Tab of the Mesh Morpher/Optimizer Dialog Box (p. 4622) and Figure 48.38: The Regions Tab of the Mesh Morpher/Optimizer Dialog Box for an Unstructured Distribution (p. 4624)). Your selection here will also determine the kinds of motions you can apply to the control points (in the **Deformation** tab). You have the following choices:

   - **Regular**

     For this option, the control points are spread in a regular distribution throughout the entire deformation region; you only have to define the number of control points along each of the defining direction vectors. Note that as part of this option, the motion of groups of control

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

points will have to be defined as a translation, as the ability to define rotational or radial motions is not available.

Locating control points in this manner is easy and simple, though this comes at the cost of being able to precisely define control points on specific mesh nodes and boundaries. This option can be a good choice in cases where you are able to allow the entire mesh within the region to be deformed—for example, internal flows within a pipe or duct.

Note that as part of this option, the mesh deformation is based on Bernstein polynomials. While such a direct morphing approach is computationally less expensive than using radial basis functions, it does not provide you with precise control of a particular mesh node.

- **Unstructured**

  For this option, you will define the locations of the control point by right-clicking with the mouse on boundary zones, distributing an approximate number throughout a zone, or by entering coordinates. Note that as part of this option, the motion of groups of control points can be defined as a translation, as a rotation about a point / axis, or radially about a point / axis.

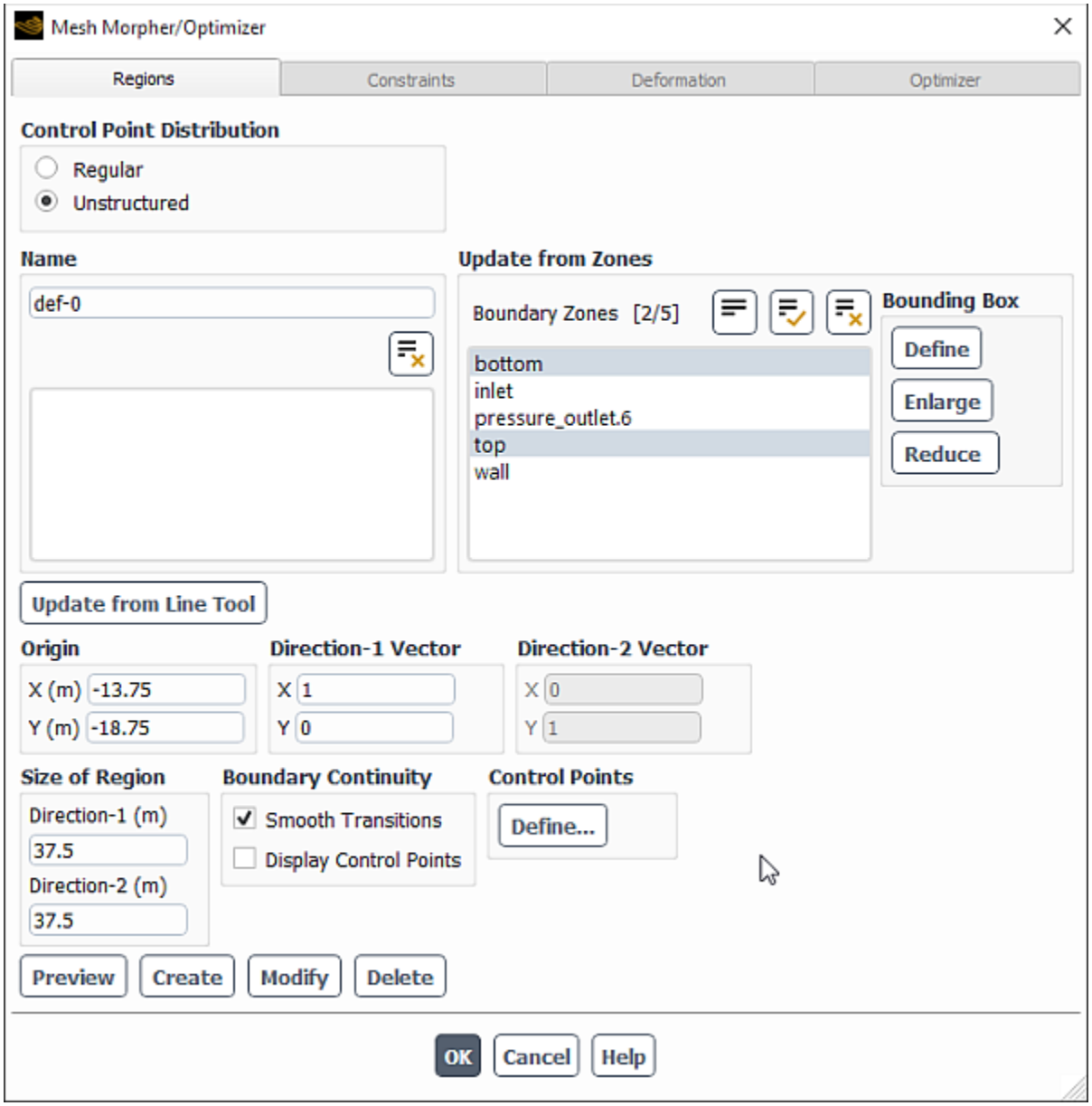
  This option provides great precision with regard to the location of the control points; this allows you to morph particular nodes and regions of the mesh, as well as to preserve features within the deformation region (as you can make nodes stationary by applying a parameter value of zero). This option also allows you to easily define twisting and axisymmetrical motions. This may be a good choice for external flows and aerodynamic applications.

  Note that as part of this option, the mesh deformation is based on radial basis functions. While this is computationally more expensive than using Bernstein polynomials (because it involves the solution of a system of equations), it allows you to have precise control of a particular mesh node if you have created a control point with the same coordinates.

Note that all of the deformation regions that you define in the next step must use the same **Control Point Distribution** ; if you change this selection at any point, all of the settings in the **Regions** tab will be deleted.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 48.38: The Regions Tab of the Mesh Morpher/Optimizer Dialog Box for an Unstructured Distribution



5. Define the region(s) of the domain where the mesh will be deformed in order to optimize the shape, by performing the following steps. Each deformation region will be defined as a "box", that is, a rectangle for 2D cases and a rectangular hexahedron for 3D cases.

Important:

It is recommended that you define each deformation region to be bigger than the area of interest, in order to maintain proper continuity between deforming and non-deforming regions.

---

a. In the **Regions** tab, enter a name for a deformation region in the text-entry box at the top of the **Name** group box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

b. You have the option of using the boundary zones to create a bounding box for the deformation region; this box can represent the final scope of the region, or it can act as a starting point that is further refined by manually editing the origin, direction vectors, and size of the region (as described in the steps that follow). Perform the following steps in the **Update from Zones** group box:

   i. Select the zones from the **Boundary Zones** list that best represent the extents of the deformation region you would like to create.

   ii. Click the **Define** button in the **Bounding Box** group box to update the values in the **Origin** , **Direction-1 Vector** , **Direction-2 Vector** (for 3D cases), and **Size of Region** group boxes. Note that the bounding box will be previewed in green in the graphics window when you click any of the **Bounding Box** buttons.

   iii. You can increase or decrease the size of the bounding box by clicking the **Enlarge** and **Reduce** buttons, respectively. Note that you have the option of setting the scaling factors associated with these buttons via the following text commands:

   `define` → `mesh-morpher-optimizer` → `region` → `scaling-enlarge`

   `define` → `mesh-morpher-optimizer` → `region` → `scaling-reduce`

c. You have the option of using the line tool (for 2D cases) or the plane tool (for 3D cases) to define the direction vectors of the deformation region; the values defined can represent the final components of the vectors, or they can act as a starting point that is further refined by manually editing the direction vectors (as described in the steps that follow). Set up the line tool or the plane tool (as described in Using the Line Tool (p. 3828) and Using the Plane Tool (p. 3833), respectively), and then click the **Update from Line Tool** or **Update from Plane Tool** button to update the values in the **Direction-1 Vector** and (for 3D cases) **Direction-2 Vector** group boxes.

   Open the Line/Rake Surface Dialog Box (p. 5514) or the Plane Surface Dialog Box (p. 5580) to make the line or plane tool appear.

d. Define an origin for the deformation region by entering the Cartesian coordinates of a point in the **X** , **Y**, and (for 3D) **Z** number-entry boxes in the **Origin** group box. Note that as you enter values, you can view the resulting bounding box (in green) by clicking the **Preview** button.

e. Define the first direction vector of the deformation region relative to the **Origin** coordinates, by entering values in the **X**, **Y**, and (for 3D) **Z** number-entry boxes in the **Direction-1 Vector** group box.

   For 2D cases, Ansys Fluent will automatically define the second direction vector of the deformation region to be perpendicular to the **Direction-1 Vector**, and will display the components in the uneditable **Direction-2 Vector** group box.

f. For 3D cases, define the second direction vector of the deformation region relative to the **Origin** coordinates, by entering values in the **X**, **Y** , and **Z** number-entry boxes in the **Direction-2 Vector** group box. If the vector you define is not perpendicular to the **Direction-1 Vector**, Ansys Fluent will automatically redefine the second vector to be the projection of the **Direction-2 Vector** you entered onto a plane that is perpendicular to the **Direction-1 Vector**. Based on this definition, the **Direction-2 Vector** you define cannot be colinear with the **Direction-1 Vector**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The third direction vector of the deformation region will be automatically defined as the cross product of the first and second direction vectors.

g. Define the overall dimensions of the deformation region by entering length values for **Direction-1** , **Direction-2**, and (for 3D cases) **Direction-3** in the **Size of Region** group box.

h. Create control points within the deformation region.

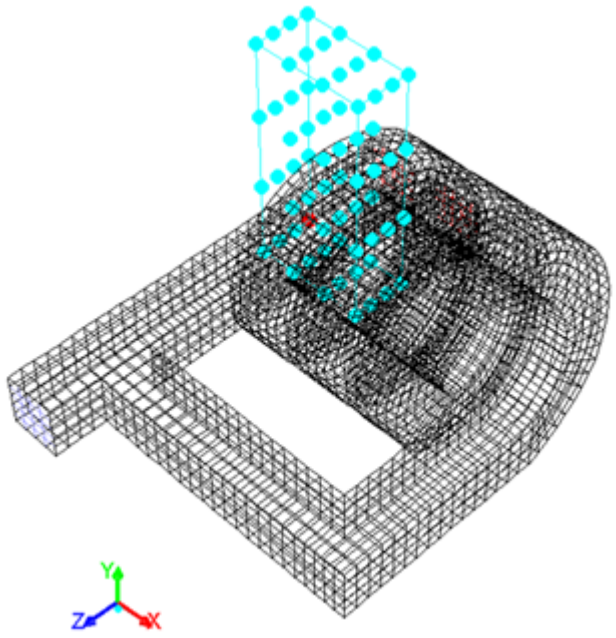
   i. If you selected **Regular** for the **Control Point Distribution**, define the number of control points you want along each direction vector of the deformation region by entering values for **Direction-1** , **Direction-2**, and (for 3D cases) **Direction-3** in the **Control Points** group box. The total number of control points for the region will be the product of the numbers you enter. Increasing the number of control points allows you greater control of the deformation, but also increases the computational expense.

Important:

When you define multiple deformation regions, you must ensure that all the deformation regions have the same total number of control points.

Save the settings you have created for the deformation region by clicking the **Create** button. The name of the deformation region will be added (and selected) in the selection list at the bottom of the **Name** group box, and the control points and bounding box of the deformation region will be displayed in blue in the graphics window (see Figure 48.39: Displaying the Control Points for a Regular Distribution (p. 4626)). If you do not want the control points and bounding box displayed in the graphics window, you can deselect the item by clicking the button located above the right side of the **Name** selection list.

Figure 48.39: Displaying the Control Points for a Regular Distribution



A new default name will be automatically entered in the text-entry box at the top of the **Name** group box, in preparation for any deformation regions you want to create in the future.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

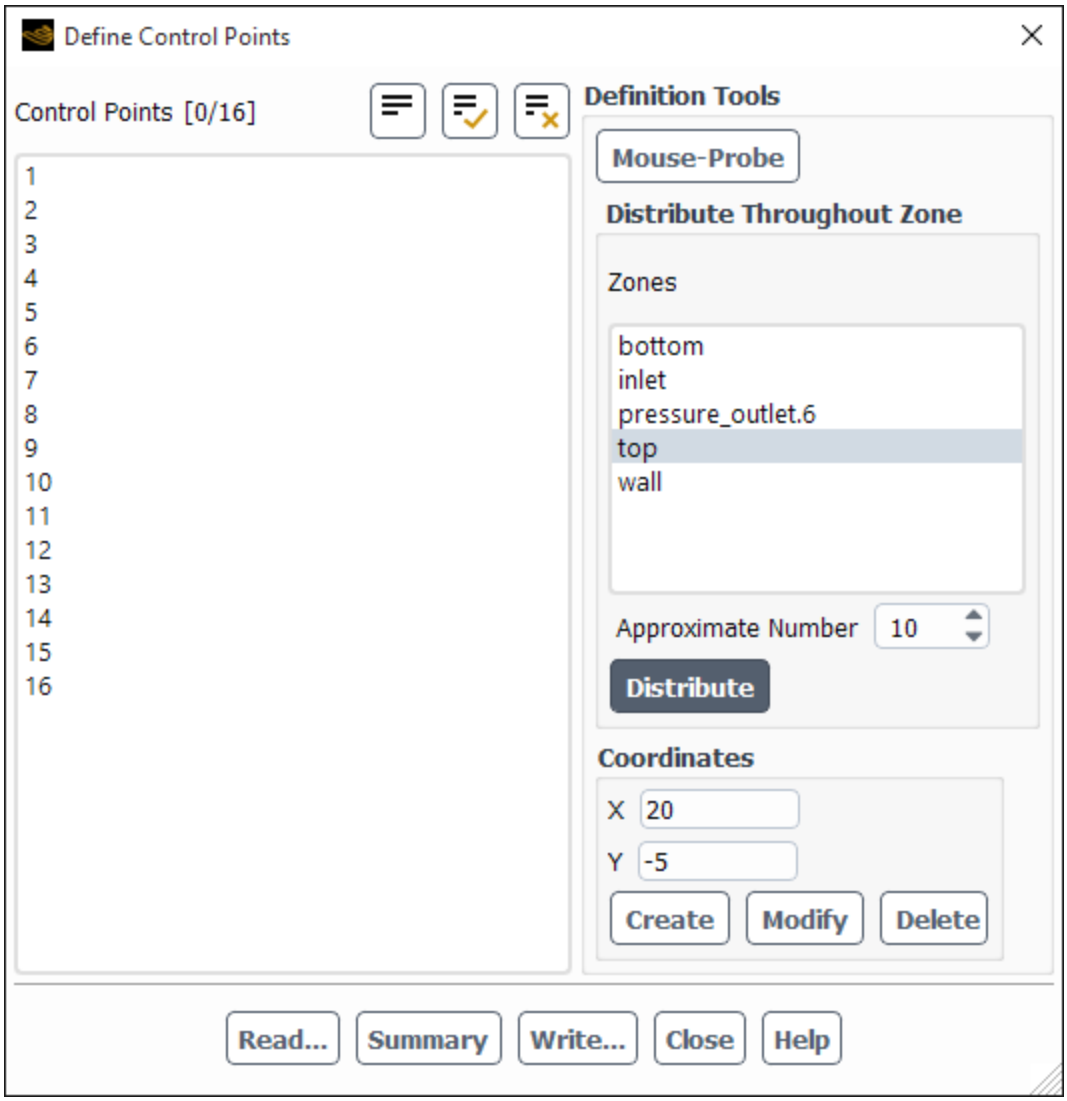
ii. If you selected **Unstructured** for the **Control Point Distribution**, click the **Create** button to save the settings for the deformation region, as you need an existing region to be able to create control points. The name of the deformation region will be added (and selected) in the selection list at the bottom of the **Name** group box, and the bounding box will be displayed in blue in the graphics window (see Figure 48.41: Displaying the Control Points for an Unstructured Distribution (p. 4630)). If at any point you do not want the bounding box displayed in the graphics window, you can deselect the item by clicking the button located above the right side of the **Name** selection list.

Next, examine the settings in the **Boundary Continuity** group box. By default, the mesh morpher/optimizer is set to provide smooth mesh transitions wherever the mesh intersects with the limits of the deformation region (that is, the sides of the bounding box). This is accomplished through the creation of stationary control points at those intersections; by default, these control points are hidden, though you can view them by enabling the **Display Control Points** option. If you would like to allow more abrupt mesh transitions at the boundaries of the region, you can disable the **Smooth Transitions** option.

The next step is to create control points. Click the **Define...** button in the **Control Points** group box to open the **Define Control Points** dialog box (Figure 48.40: The Define Control Points Dialog Box (p. 4628)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 48.40: The Define Control Points Dialog Box



Use the **Define Control Points** dialog box to create control points within the region. You have the following options:

- Click the **Mouse-Probe** button and then probe (using the right mouse button, by default) within the deformation region. For 3D cases, if you probe on a displayed boundary, the control point will be located at that point on that boundary. Note that it may be helpful to revise what surfaces are displayed using the Mesh Display Dialog Box (p. 4688).

- Make a selection from the **Zones** list, enter an **Approximate Number** of control points that you want on that zone, and click the **Distribute** button. This method uses Binary Space Partitioning (BSP) to agglomerate the cell faces of the zone within the bounding box based on the intended number of control points, and then a control point is assigned to the node that is closest to the centroid of each agglomeration. While this method is convenient when you want a large number of control points or want to morph entire boundary zones or surfaces, there are some limitations to BSP: the number of control points may exceed the value you enter, and the control points will not be as evenly distributed on curved surfaces when compared to flat surfaces.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Enter **X**, **Y** , and **Z** coordinates and click the **Create** button. This method allows you to locate a control point directly on a mesh node, if you have first obtained the vertex position (which can be displayed in the console if you select **long description** from the **Probe** drop-down list in the **View** tab (**Mouse** group box) and then probe near the node with the mouse).

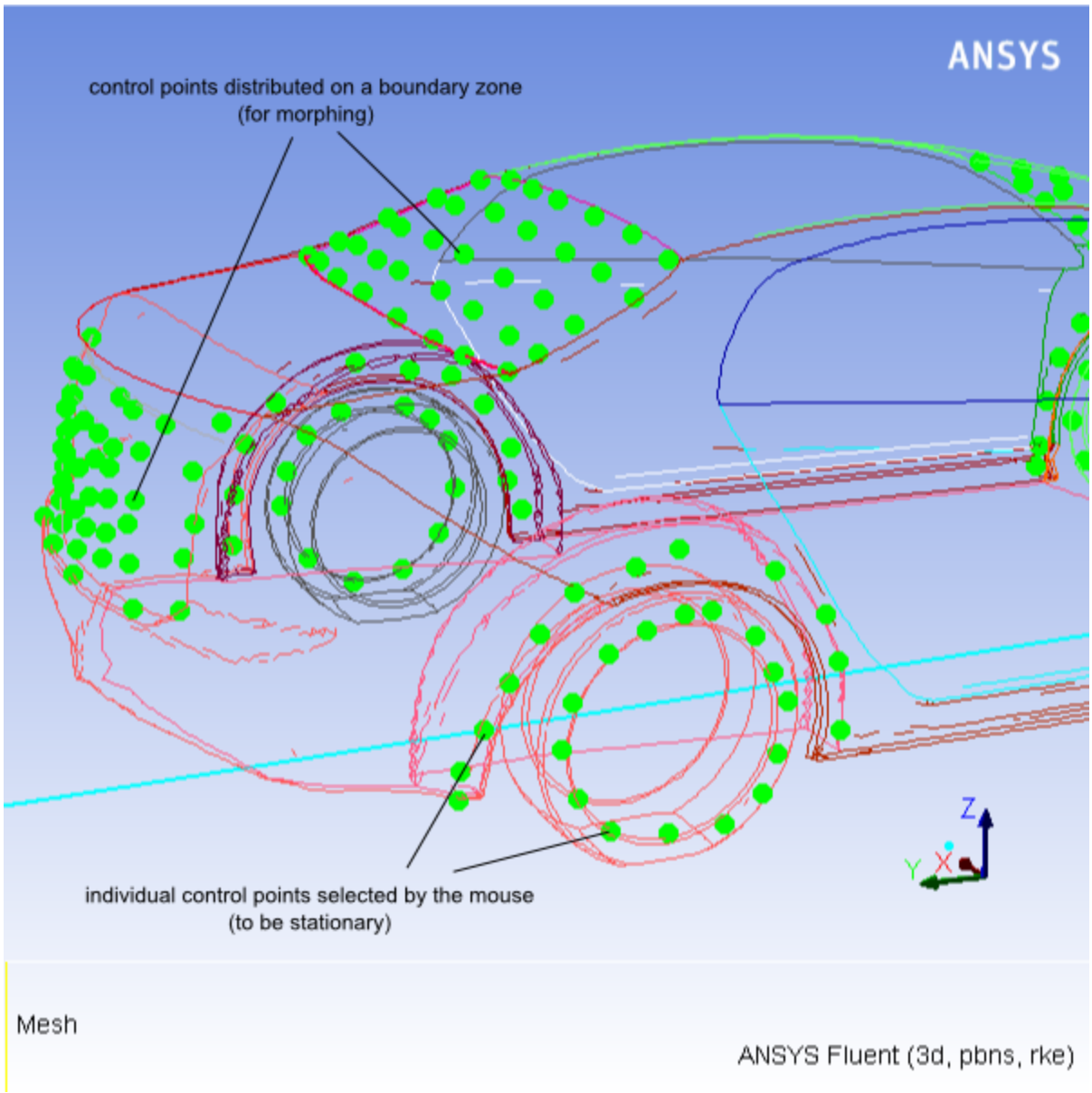
- If you have previously generated an ASCII text file that contains control point definitions, you can read it by clicking **Read...** and using the dialog box that opens. Using a text file can be helpful when you have a large number of points. It is not necessary to create the text file from scratch, as you can create a simplified text file with sample data using the **Write...** button, and then define the bulk of the control point locations using a spreadsheet program. See Mesh Morpher/Optimizer File Formats (p. 5693) for details about the format of the text file.

The control points will be displayed in green in the graphics window, and will turn red when selected in the **Control Points** selection list. When a control point is selected, you also have the ability to **Delete** it, or enter new coordinates and **Modify** it.

When adding control points, remember that it may be helpful to locate them not only in areas that you want to deform, but also on features in the deformation region that you want to preserve (as you can make nodes stationary by applying a parameter value of zero). When you are done adding control points, click **OK** to close the **Define Control Points** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 48.41: Displaying the Control Points for an Unstructured Distribution



A new default name will be automatically entered in the text-entry box at the top of the **Name** group box, in preparation for any deformation regions you want to create in the future.

i. Create any additional deformation regions as necessary by repeating steps 5.a.–h.

Important:

Overlapping deformation regions are not supported.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

j. If you need to modify a deformation region at any point, select it in the **Name** selection list, revise the appropriate settings, and click the **Modify** button. The updated control points and bounding box will be displayed in the graphics window.

> **Important:**
>
> When modifying an existing deformation region, be sure to click **Modify** rather than **Create**. If you click **Create**, you will not modify the deformation region, but will instead create a new one with the modified settings.

k. If you need to delete a deformation region at any point, select it in the **Name** selection list and click the **Delete** button.

6. You have the option of defining constraints on the boundary zones, in order to limit the freedom of particular zones that fall within the deformation region(s) during the morphing of the mesh. The following options are available:

- unconstrained

  This option specifies that the boundary zone is completely free to be deformed according to the assigned parameters. By default, all wall zones are unconstrained.

- fixed

  This option specifies that the boundary zone is fixed and will not be deformed. None of the zones are fixed by default.

> **Important:**
>
> If you specify one or more fixed boundary zones in a deformation region, you must ensure that there is at least one unconstrained boundary zone in the region as well.

- passive

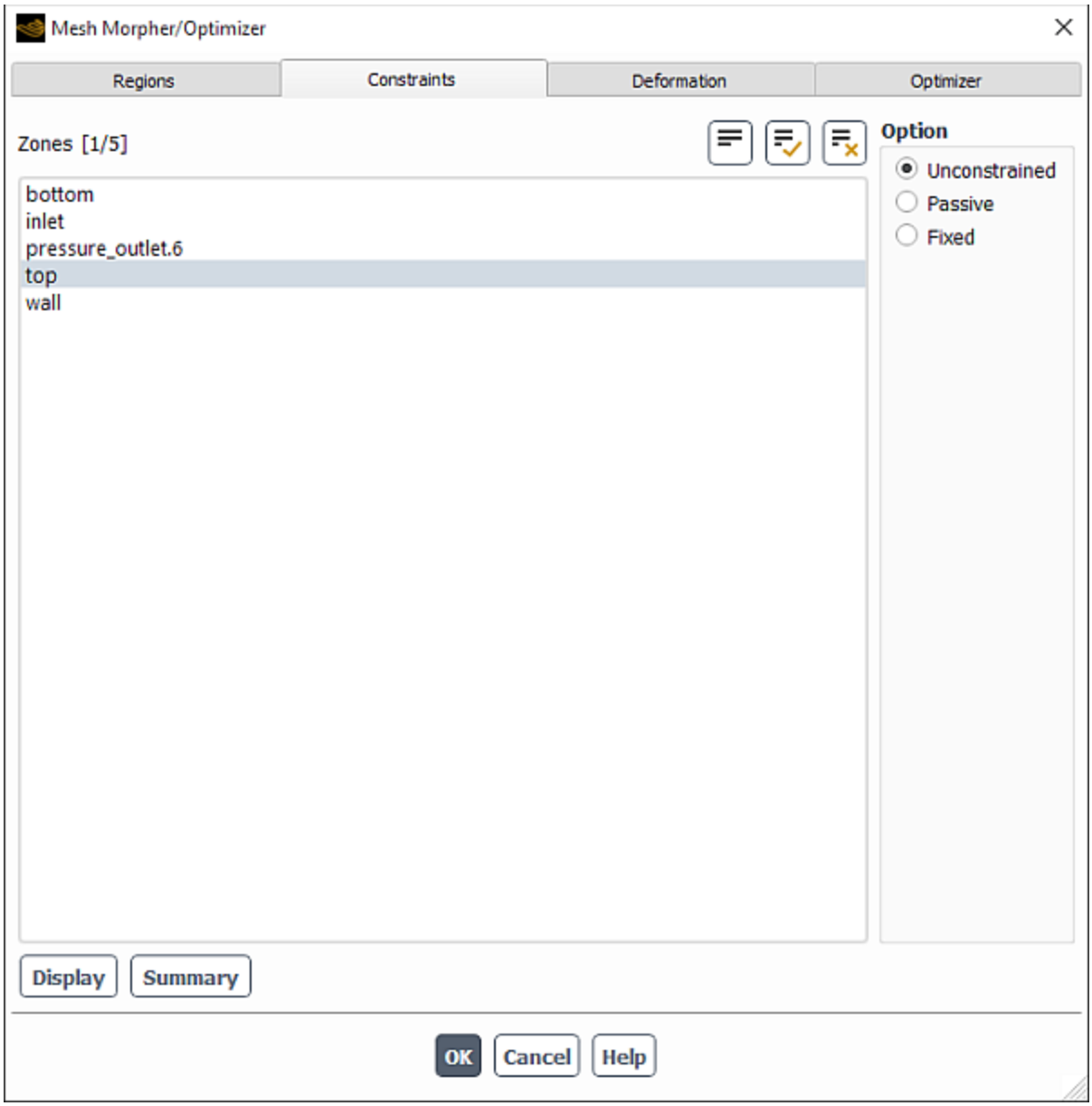
  This option specifies that the nodes of the boundary zone are partially constrained to varying degrees, based on their proximity to adjacent boundary zones that are fixed. The nodes in a passive boundary zone behave in a similar manner to the interior mesh nodes, in order to ensure that there is a smooth transition between fixed and unconstrained boundary zones. By default, all boundary zones that are not walls (for example, inlets, outlets, symmetry, and periodic boundaries) are passive.

To define the constraints on the boundary zones, perform the following steps.

a. Click the **Constraints** tab (Figure 48.42: The Constraints Tab of the Mesh Morpher/Optimizer Dialog Box (p. 4632)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 48.42: The Constraints Tab of the Mesh Morpher/Optimizer Dialog Box



b. To revise the default constraints on the boundary zones, select the zone name(s) from the **Zones** selection list in the **Constraints** group box, and then select either **Unconstrained**, **Passive** , or **Fixed** from the **Option** list.

c. Click **Display** if you want view the boundary zones selected from the **Zones** selection list, in order to verify the zones for which you are defining constraints.

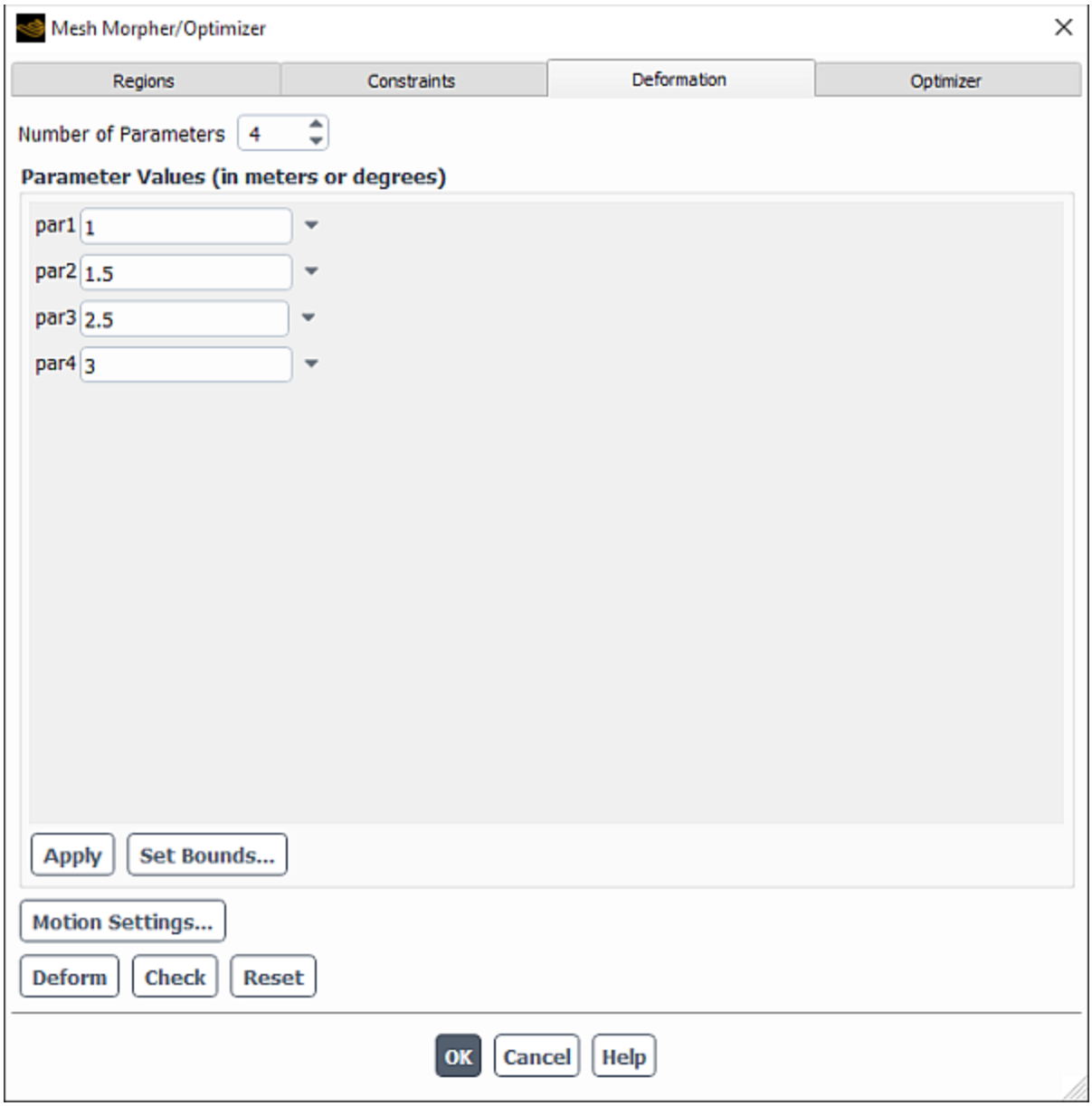
d. Click **Summary** if you would like to print a list in the console that summarizes the constraint definitions for all of the boundary zones.

7. Begin the deformation definition by defining the parameters.

   a. Click the **Deformation** tab (Figure 48.43: The Deformation Tab of the Mesh Morpher/Optimizer Dialog Box (p. 4633)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 48.43: The Deformation Tab of the Mesh Morpher/Optimizer Dialog Box



b. Enter the **Number of Parameters** that will be used to define the deformation. This number may be as low as the maximum number of parameters you will define on a single control point, or as high as the total number of parameters you will set on all of the control points combined.

c. If you want to manually specify the deformation or use Design Exploration in Ansys Workbench to explore multiple deformation scenarios (rather than using the built-in optimizers), define the parameter fields in the **Parameter Values** group box. Note that these fields are only available when **none** or **workbench** is selected from the **Optimizer** drop-down list in the **Optimizer** tab. The value of the parameter defines a magnitude that will then be used along with other direction settings to define motions that produce an overall displacement for a given control point. The value will specify a length in meters for translations and radial motions, and will specify an angle in degrees for rotations; the units here will be used regardless of what units you are using in the case.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For each parameter you have the following options:

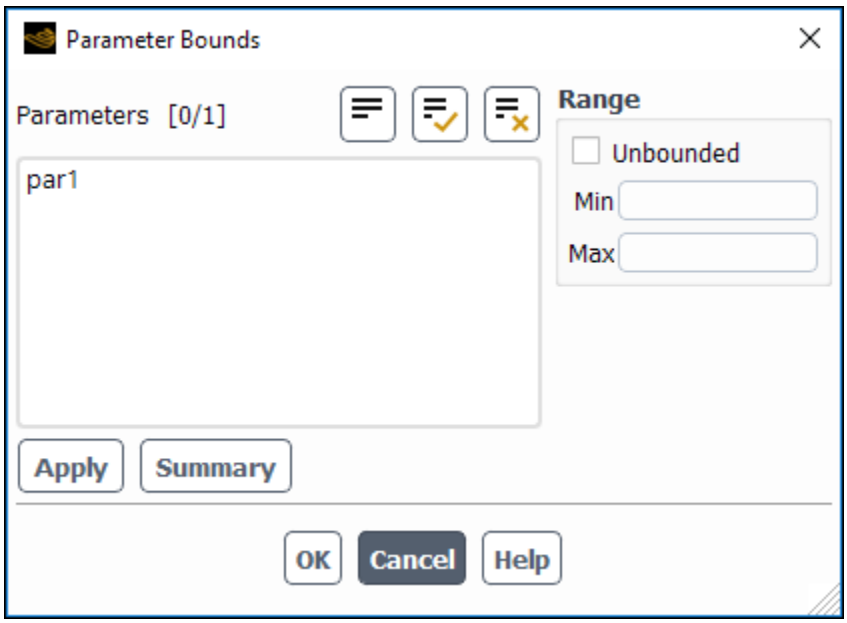
- Enter a constant numeric value. Note that having a parameter with a value of zero allows you to make a control point stationary.
- Use the drop-down menu to define the parameter through an expression or input parameter (for example, as part of a parametric study managed by Workbench); for details, see Directly Applied Expressions (p. 1050) or Creating a New Parameter (p. 1311), respectively.

When you have finished defining the parameter fields, click the **Apply** button in the **Parameter Values** group box to save the definitions.

d. You have the option of limiting how much each parameter is allowed to deform by defining strict minimum and maximum values. This can be useful when you need to keep the parameters within a certain tolerance of the initial design, or when you know in advance that certain designs are impractical. While this option is typically used with the built-in optimizers, it is available for manual deformation, so that you can visualize the effects of the bounds and set them interactively. Note that this option is not available when **workbench** is selected from the **Optimizer** drop-down list in the **Optimizer** tab.

Click the **Set Bounds...** button in the **Parameter Values** group box to open the **Parameter Bounds** dialog box (Figure 48.44: The Parameter Bounds Dialog Box (p. 4634)).

Figure 48.44: The Parameter Bounds Dialog Box



Select the parameters you want to limit from the **Parameters** selection list, disable the **Unbounded** option in the **Range** group box, and then enter appropriate values for **Min** and **Max**.

At any point you can click the **Apply** button to save your bounds, and you can click the **Summary** button to display a summary of the saved bounds in the console. When all of the parameter bounds are defined to your satisfaction, click the **OK** button to close the **Parameter Bounds** dialog box.

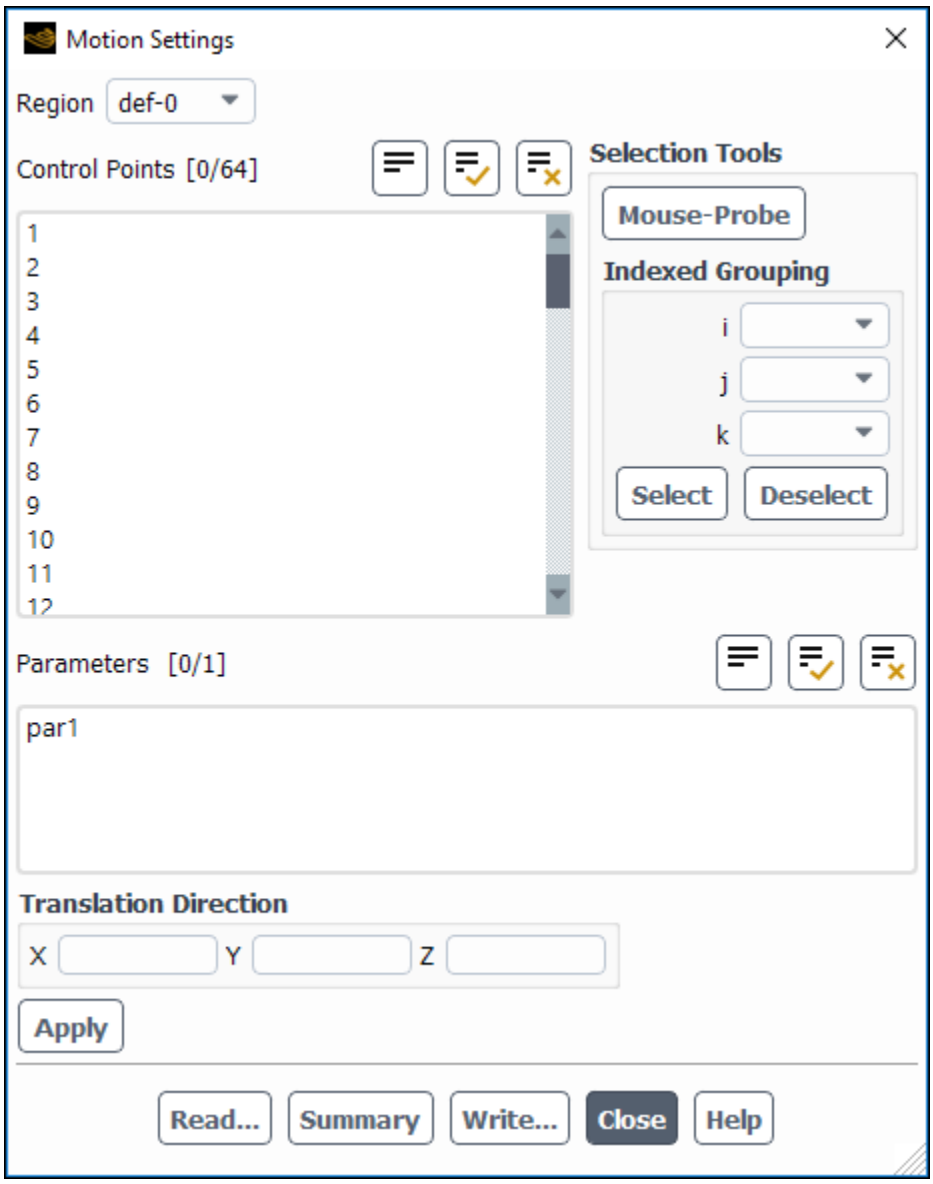
8. If you selected **Regular** for the **Control Point Distribution** in the **Regions** tab, perform the following steps. You will define motion settings for the control points by selecting parameters (which you

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

defined previously) and specifying the translation direction, thus completing the deformation definition.

a. Click the **Motion Settings...** button in the **Deformation** tab to open the **Motion Settings** dialog box (Figure 48.45: The Motion Settings Dialog Box for a Regular Distribution (p. 4635)).

Figure 48.45: The Motion Settings Dialog Box for a Regular Distribution



b. (optional) If you have previously generated an ASCII text file that contains motion settings, you can read it by clicking **Read...** at the bottom of the **Motion Settings** dialog box and using the dialog box that opens; otherwise, you should proceed to steps 8.c.–8.j. and define the settings using the other controls in the **Motion Settings** dialog box.

Using a text file can be helpful when you have a large number of motion settings. It is not necessary to create the text file from scratch, as you can create a simplified text file with sample values using the **Write...** button (as described in step 8.k.), and then add the bulk of the motion

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

settings using a spreadsheet program. See Mesh Morpher/Optimizer File Formats (p. 5693) for details about the format of the text file.

Important:

Immediately after you read a text file, the **Motion Settings** dialog box will not display the applied settings. To begin viewing the applied settings, make a selection from the **Region**, **Control Points**, and **Parameters** lists, and the associated **Translation Direction** will be displayed.

You can then edit the motion settings as necessary (in a manner similar to steps 8.c.–8.g.) and/or proceed to step 8.m.

c. Make a selection from the **Region** drop-down list in the **Motion Settings** dialog box.

d. Specify the control points in the current **Region** to which you want to assign the same motion settings (that is, the same combination of parameters and direction). You can specify these control points using one or more of the following methods:

- Select the ID numbers from the **Control Points** selection list. The selected control points will be highlighted in the graphics window.
- Click the **Mouse-Probe** button in the **Selection Tools** group box and then click the control points in the graphics window with the mouse-probe button (which is the right mouse button, by default). To deselect, select a highlighted control point with the mouse-probe button. As you select the control points, they will also be selected in the **Control Points** selection list.
- Use the **Indexed Grouping** group box to select the control points based on their assigned indices. Each control point has **i** , **j**, and (for 3D cases) **k** index numbers, each of which denote its place in the sequence (starting at the origin of the region) of control points along the direction-1, direction-2, and direction-3 vectors, respectively (as defined in the **Regions** tab of the **Mesh Morpher/Optimizer** dialog box). You can make selections from the index drop-down lists, and then click the **Select** button. In a similar way you can deselect control points using the **Deselect** button. As you select the control points, they will also be selected in the **Control Points** selection list and highlighted in the graphics window.

  For example, consider the case of a 3D region: if you select **2** from the **k** drop-down list, the control points in the second plane perpendicular to the direction-3 vector (that is, the direction-1–direction-2 plane) will be selected. If you select **1** from both the **i** and **j** drop-down lists, the line of control points at the intersection of the first planes perpendicular to the direction-1 and direction-2 vectors will be selected. Making selections from all three drop-down lists will select a single control point.

e. Make selections from the **Parameters** selection list, in order to specify those you want to assign (along with a translation direction) to the selected **Control Points**. Note that the values associated with these parameters (as displayed in the **Parameter Values** group box in the **Deformation** tab of the **Mesh Morpher/Optimizer** dialog box) are irrelevant if you plan to use the built-in optimizers.

f. Define the **Translation Direction** that will be associated with the selected **Parameters** by entering values for the **X**, **Y**, and (for 3D cases) **Z** directions. Note that you do not need to define a unit vector.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If you use a built-in optimizer, these values will provide the direction of the displacement of the control point and the optimizer will determine the overall magnitude of displacement. Alternatively, if you specify the deformation manually or using Design Exploration, the values you enter in the **Translation Direction** group box will be multiplied with the values of the **Parameters** to define the displacement applied to the control points.

g. Click the **Apply** button to save the motion settings.

h. If you want to apply another set of motion settings to the selected **Control Points**, repeat steps 8.e.–8.g. for each additional combination of parameters and direction.

i. Repeat steps 8.d.–8.h. for each additional set of control points in the current **Region** to which you want to assign motion settings.

j. Repeat steps 8.c.–8.i. for each additional region in which you want to apply deformation parameters to control points.

k. If you want to save all of the settings you specified in the **Motion Settings** dialog box as an ASCII text file, click **Write...** and specify a name in the dialog box that opens. You can edit the motion settings in this text file using a spreadsheet program, and then read it in this or a separate case file, as described in step 8.b.

l. You can display a summary of the motion settings in the console by clicking the **Summary** button.

m. Click **OK** to close the **Motion Settings** dialog box.

9. If you selected **Unstructured** for the **Control Point Distribution** in the **Regions** tab, perform the following steps. You will define one or more motions—which consist of a parameter (defined previously), direction settings, and the affected control points—and thus complete the deformation definition.

Important:

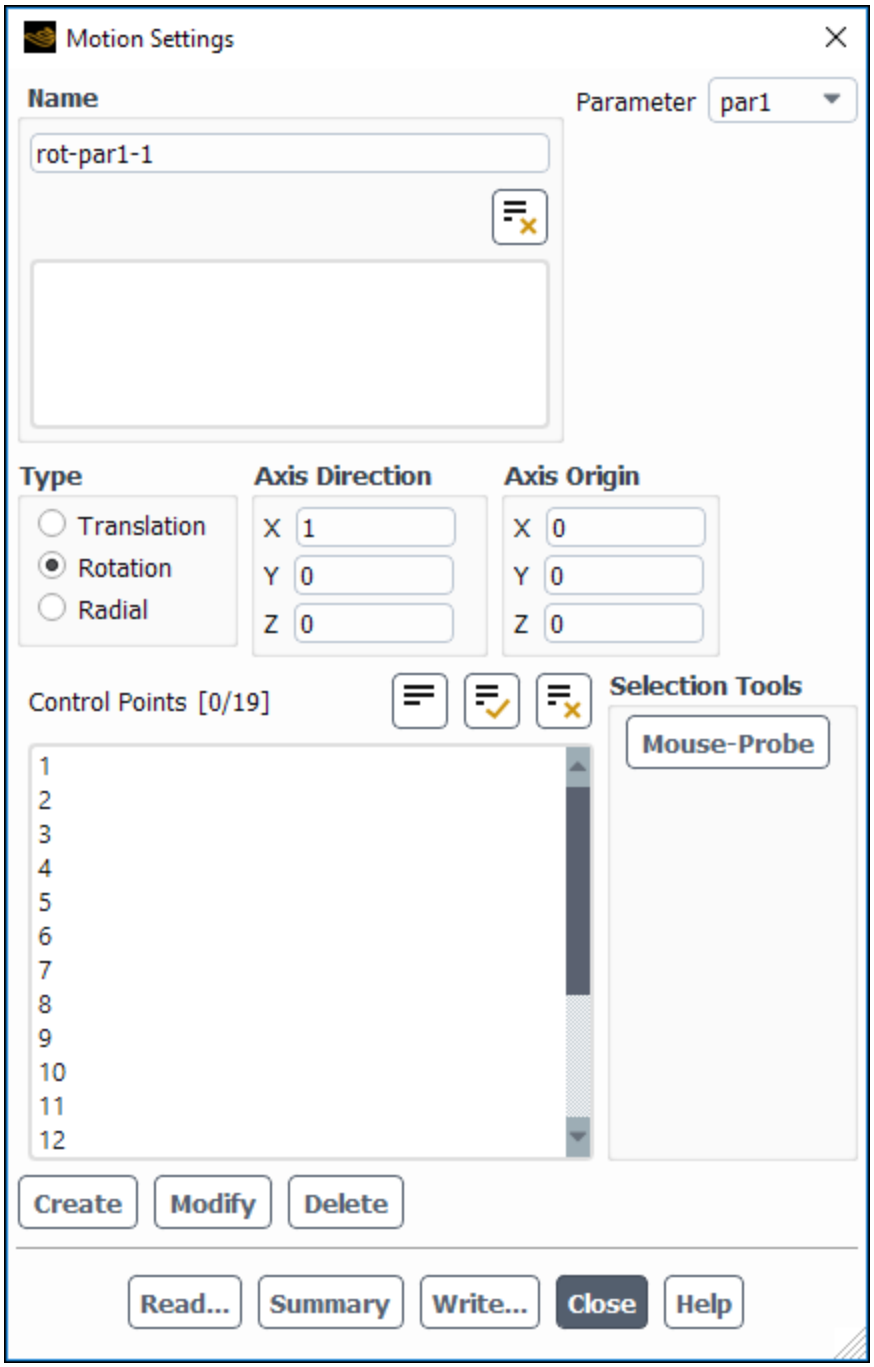
When a single control point is associated with multiple motions, note that the resultant displacement for each individual motion will be calculated from the original position, and then the displacements will be summed. This means that the order in which the motions are created will have no effect on the final location of the control point.

---

a. Click the **Motion Settings...** button in the **Deformation** tab to open the **Motion Settings** dialog box (Figure 48.46: The Motion Settings Dialog Box for an Unstructured Distribution (p. 4638)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 48.46: The Motion Settings Dialog Box for an Unstructured Distribution



b. (optional) If you have previously generated an ASCII text file that contains motion definitions, you can read it by clicking **Read...** at the bottom of the **Motion Settings** dialog box and using the dialog box that opens; otherwise, you should proceed to steps 9.c.–9.k. and define the settings using the other controls in the **Motion Settings** dialog box.

Using a text file can be helpful when you have a large number of motions. It is not necessary to create the text file from scratch, as you can create a simplified text file with sample values using the **Write...** button (as described in step 9.l.), and then add the bulk of the motion definitions

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

using a spreadsheet program. See Mesh Morpher/Optimizer File Formats (p. 5693) for details about the format of the text file.

You can then either add additional motions (in a manner similar to steps 9c.–9.h.) or modify / delete an existing motion (as described in steps 9.i. and 9.j.). When the settings are finalized, you can proceed to step 9.k.

c. To begin defining a new motion, make a selection from the **Parameter** drop-down list.

d. Make a selection from the **Type** list, to indicate the type of motion you would like to define. You can specify that a group of control points undergoes a **Translation**, a **Rotation** about a point (for 2D) or an axis (for 3D), or a **Radial** motion about a point (for 2D) or an axis (for 3D).

e. Define the motion by entering **X** , **Y**, and (for 3D cases) **Z** values for the following: the **Direction** used for translations; or the **Axis Origin** and (for 3D cases) **Axis Direction** used for rotations and radial motions. For **Direction** or **Axis Direction**, note that you do not need to define unit vectors.

If you use a built-in optimizer, the values you enter will define the direction of the displacement of the control point and the optimizer will determine the overall magnitude of displacement. Alternatively, if you specify the deformation manually or using Design Exploration, the magnitude will be the value of the **Parameter** ; translation is a special case, in which the values you enter in the **Direction** group box are multiplied with the value of the **Parameter** to define the displacement applied to the control points.

f. Select the control points to which you want to assign the selected motion. You can either select them in the **Control Points** list, or click the **Mouse-Probe** button and click the control points in the graphics window using the mouse-probe button (which is the right mouse button, by default).

g. A default name will be generated in the top field of the **Name** group box, based on the **Type** and **Parameter** of the motion. You can edit this text to make it more descriptive, as necessary.

h. Click **Create** to create the motion. The name will be added to the selection list in the **Name** group box.

i. Repeat steps 9.c.–9.g. for each additional motion that you would like to create.

j. If you need to modify a motion at any point, select it in the lower selection list in the **Name** group box, revise the appropriate settings, and click the **Modify** button.

Important:

When modifying an existing motion, be sure to click **Modify** rather than **Create**. If you click **Create**, you will not modify the motion, but will instead create a new one with the modified settings.

---

k. If you need to delete a deformation region at any point, select it in the lower selection list in the **Name** group box and click the **Delete** button.

l. You can display a summary of the motion definitions in the console by clicking the **Summary** button.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

m. If you want to save all of the settings you specified in the **Motion Settings** dialog box as an ASCII text file, click **Write...** and specify a name in the dialog box that opens. You can edit the motion settings in this text file using a spreadsheet program, and then read it in this or a separate case file, as described in step 9.b.

10. If you want to manually specify the deformation (that is, **none** is selected from the **Optimizer** drop-down list in the **Optimizer** tab of the **Mesh Morpher/Optimizer** dialog box), perform the following steps:

    a. Click the **Deform** button to apply the current settings (in the **Parameter Values** group box and **Motion Settings** dialog box) and to display the deformed mesh in the graphics window. The **Deform** button allows you to manually specify the deformation.

    b. You can click the **Check** button to print out a mesh check report in the console for the currently displayed mesh. The mesh check report is the same as that produced by the **Check** button in the **General** task page, as described in Mesh Check Report (p. 1246).

    Important:

    If you save parameter settings but do not click the **Deform** button, the mesh check report will not account for the mesh that is produced from those settings.

    c. Click the **Reset** button if you want to revert to the original mesh without the deformations that result from the **Deform** button.

11. If you plan to use the built-in optimizers, you should decide whether you want to disable the general mesh check that is performed by default immediately after the mesh is deformed in every design stage. The mesh check that is performed is the same as that initiated by the **Check** button in the **General** task page (see Checking the Mesh (p. 1245) for details). If any errors are discovered, the mesh is rejected and the next design stage is attempted.

    Disabling the general mesh check allows you to repair the mesh, so that an accurate solution can be calculated for it. Note that you can set up the mesh repair by entering the appropriate text commands (as described in Repairing Meshes (p. 1247)) in the **Initial Commands** text-entry box in the **Optimizer** tab.

    You can disable the general mesh check by using the following text command:

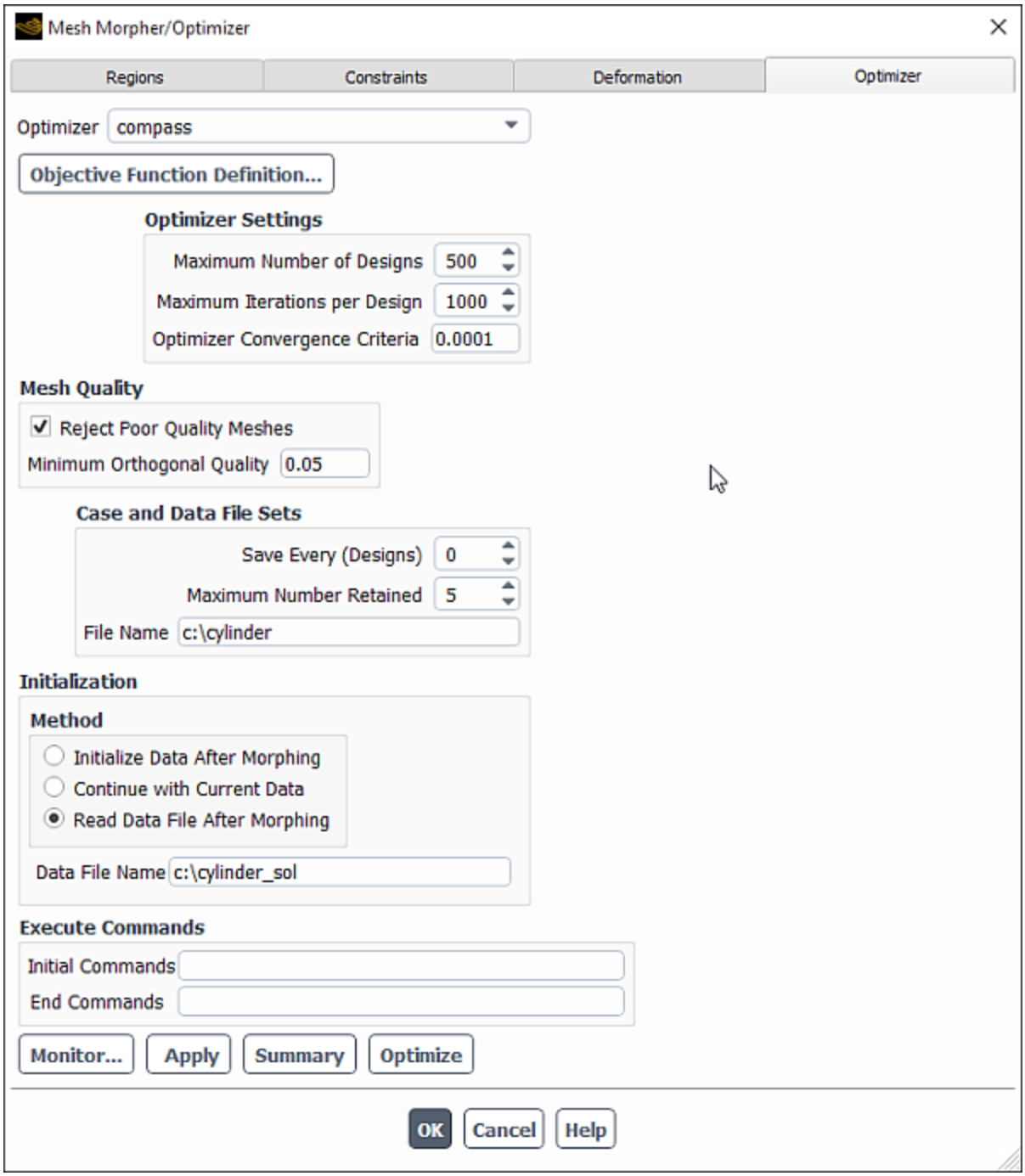
    `define → mesh-morpher-optimizer → optimizer-parameters → disable-mesh-check`

12. If you want to use the built-in optimizers, define the optimizer settings and initiate the optimization process.

    a. Click the **Optimizer** tab (Figure 48.47: The Optimizer Tab of the Mesh Morpher/Optimizer Dialog Box (p. 4641)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 48.47: The Optimizer Tab of the Mesh Morpher/Optimizer Dialog Box

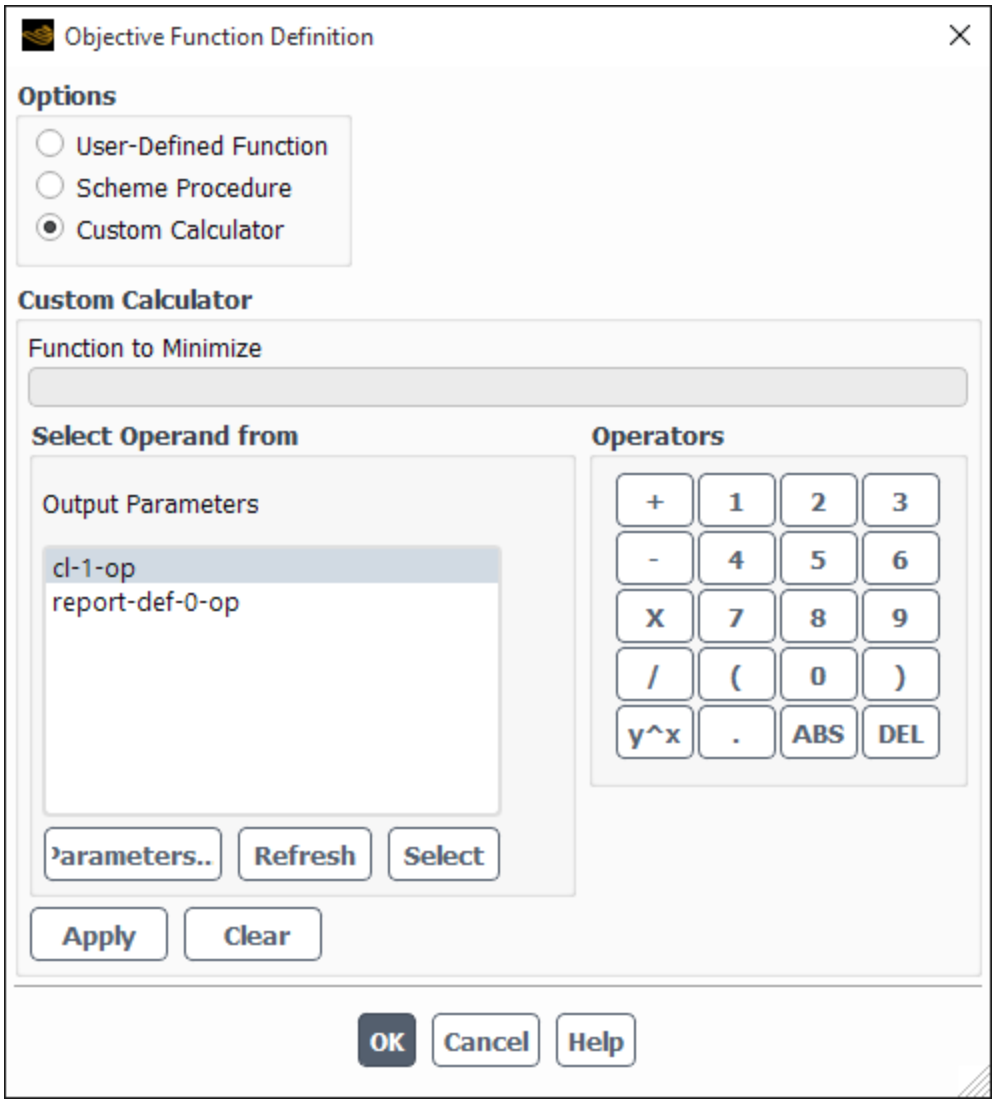


b. Select an optimizer from the **Optimizer** drop-down list. The available optimizers are not based on gradients, and include the following: **compass** , **newuoa**, **powell** , **rosenbrock**, **simplex**, and **torczon**. For more information about how these optimizers function, see Optimizers (p. 4618) .

c. Click the **Objective Function Definition...** button to open the **Objective Function Definition** dialog box (Figure 48.48: The Objective Function Definition Dialog Box (p. 4642)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 48.48: The Objective Function Definition Dialog Box



i. Make a selection in the **Options** group box to specify whether the objective function that will be minimized during the optimization process is a **User-Defined Function**, a **Scheme Procedure**, or a customized function of output parameters as defined by the **Custom Calculator**. Your selection should correspond with the actions you took in step 2.

ii. If you selected **Custom Calculator** in the previous step, define the objective function via the GUI controls in the **Custom Calculator** group box. As you click the buttons in this group box, text and symbols will appear in the **Function to Minimize** text box. You *cannot* edit the contents of this box directly; if you want to delete part or all of the function, use the **DEL** or **Clear** button, respectively.

    You can include output parameters in the definition of the function by making a selection from the **Output Parameters** list and clicking **Select**. If you want to add a new output parameter to the selection list or check the definition of an existing one, you can click the **Parameters...** button and use the **Parameters** dialog box (as described in Creating Output Parameters (p. 4133)); be sure to click the **Refresh** button when you return to the **Objective Function Definition** dialog box, so that the **Output Parameters** list is updated. You can also use **Operators** to manipulate the output parameters in the function.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

You can click the **Apply** button at any point to save your function. When it is fully defined, click the **OK** button to save your settings and close the **Objective Function Definition** dialog box.

d. Define the settings in the **Optimizer Settings** group box.

   i. Enter a value for **Maximum Number of Designs**, to specify the maximum number of design stages the optimizer will undergo to reach the specified objective function. Note that this number is not the number displayed in the console during the optimization process. Each optimizer undergoes a certain number of design modifications (called "runs") before it converges to a design for a particular design stage. The Run number displayed in the console refers to the actual design modifications (including the inner loop) irrespective of the optimizer, whereas the **Maximum Number of Designs** value is the maximum number of converged designs provided by the optimizer.

   ii. Enter the **Maximum Iterations per Design** , to specify the maximum number of iterations Ansys Fluent performs for each design change.

   iii. Enter a number for **Optimizer Convergence Criteria**, to specify the convergence criteria for the optimizer.

   iv. If you selected the **newuoa** optimizer, enter a number for the **Initial Parameter Variation** to specify how much the parameters will be allowed to vary during the initial calculations. This value is not intended to define a strict minimum / maximum for the parameters (as is defined by the bounds specified using the **Parameter Bounds** dialog box), but should only be large enough to allow the optimizer to capture the minima. A better estimation (that is, a smaller value) allows the optimizer to reach convergence faster.

e. By default, the orthogonal quality (as defined in Mesh Quality (p. 1146)) of the initial and every subsequent mesh is computed, before the related solution calculation is initiated. If the orthogonal quality for any cell is less than a specified value, the mesh is rejected and Fluent proceeds to the next design stage (therefore helping to ensure that your solution accuracy is not compromised by poor quality meshes). These actions are governed by the settings in the **Mesh Quality** group box.

   If you do not want the meshes rejected based on their orthogonal quality, you can disable the **Reject Poor Quality Meshes** option; otherwise, specify the minimum orthogonal quality that must be maintained by every cell by entering a value from `0` – `1` (where `0` represents the worst quality) for **Minimum Orthogonal Quality**. When determining an appropriate minimum value, you can check the orthogonal quality of the initial mesh (by using the **Report Quality** button in the **General** task page) and then select a value that is sufficiently lower, so as not to reject a majority of the meshes generated during the optimization process.

f. You have the option of saving intermediate case and data files during the optimization run, so that you can restart an interrupted solution in the same or a different Fluent session without increasing the overall number of design iterations needed to reach convergence. To enable the saving of such intermediate files, define the settings in the **Case and Data File Set** group box.

   i. Specify the frequency (in number of design iterations) that you want the intermediate case and data files saved by entering a value for **Save Every**. The default value is **0**, which specifies that no intermediate files will be saved. Note that if you selected the **newuoa** optimizer, the frequency at which files are saved may not exactly correspond to the number you enter, but should be reasonably close.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ii. Specify the maximum number of intermediate files sets you want to retain by entering a value for **Maximum Number Retained**. After the maximum limit of file sets has been saved, Ansys Fluent begins overwriting the earliest existing intermediate file set. Lower values should be used if you have limited disk space or you are concerned about saving unnecessary files.

iii. Specify a root name for the intermediate files in the **File Name** text box. When the files are saved, the design iteration will be appended to this root name to indicate the point at which it was saved during the optimization run. An extension will also be automatically added to the root name (`.cas.h5` or `.dat.h5`, by default). You can include a folder path in the file-name if you want the files saved outside of the working folder.

g. Specify how the solution variables should be treated after the mesh is deformed by making a selection from the **Method** list in the **Initialization** group box:

- Select **Initialize Data After Morphing** to specify that the solution variables should be initialized to the values specified in the **Solution Initialization** task page after deformation.

- Select **Continue with Current Data** to specify that the solution variables remain the values obtained in the previous design stage. This selection will reduce the number of iterations needed to reach convergence compared to the **Initialize Data After Morphing** selection. Note that if the solver diverges during an intermediate design stage, the solution variables will be initialized and the solution will be attempted again.

- Select **Read Data File After Morphing** to specify that the solution variables are set to the values obtained from the data file specified in the **Data File Name** text-entry box. This selection will reduce the number of iterations needed to reach convergence compared to **Initialize Data After Morphing** selection or (in cases where the solution diverges for an intermediate design stage) the **Continue with Current Data** selection. Note that this selection is not available when **newuoa** is selected from the **Optimizer** drop-down list.

h. If you did not select the **newuoa** optimizer, you have the option of specifying commands that will be executed during the optimization runs of the mesh morpher/optimizer, via the text-entry boxes in the **Execute Commands** group box. A command can be a text command or the name of a command macro you have defined (or will define), as described in Defining Macros (p. 3703). You can also enter a series of text commands and/or macros, as long as they are separated by a semi-colon (`;`).

During optimization runs, deformation occurs as part of every design iteration. You decide the specific point during the design iteration that the command is executed:

- If you want a command to be executed after the design has been modified, but before Ansys Fluent has started to run the calculation for that design stage, enter the command in the **Initial Commands** text-entry box. There are no restrictions on the commands that can be entered: you can enter commands that read saved data, perform FMG initialization, execute an entirely independent on-demand UDF, or even call a Scheme routine.

- If you want a command to be executed after the solution has run and converged for a design stage, enter the command in the **End Commands** text-entry box. There are no restrictions on the commands that can be entered. Examples are commands for postpro-

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

cessing solution variables, monitoring contours and vectors of different variables, taking snapshots of each design change, and so on, at every design stage.

Important:

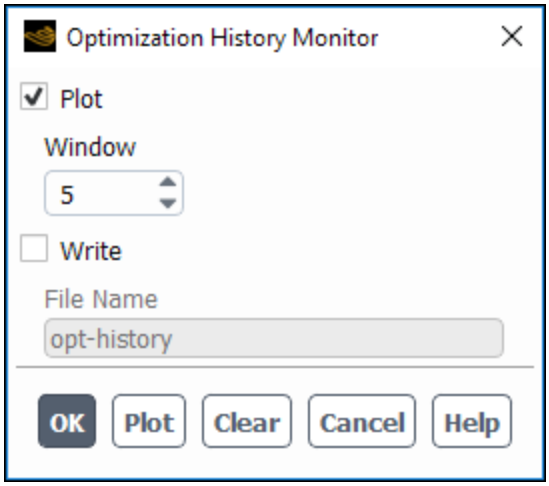
If the command to be executed involves saving a file, see Saving Files During the Calculation (p. 3705) for important general information.

As noted in Automatic Numbering of Files (p. 952), the special character, `%i`, can be used to create unique filenames by including the iteration number. However, by default, the solution will be initialized after each mesh deformation and the iteration count restarted. Therefore, the resulting filename cannot be used to identify which design stage produced it (that is, the first design stage may converge at 50 iterations, and then the second design stage may converge at 43 iterations). The time that the file was created is a better way to identify where in the series of design stages the file was generated. In addition, some files may be overwritten if two design stages converge in the same number of iterations. Alternatively, if you have selected **Continue with Current Data** under **Initialization** when setting up the optimizer, then the iteration count will not reset and no duplicate filenames will occur.

---

i. If you did not select the **newuoa** optimizer, you can plot and/or record the optimization history (that is, how the value of the objective function varies with each design stage). Click the **Monitor...** button to open the **Optimization History Monitor** dialog box (Figure 48.49: The Optimization History Monitor Dialog Box (p. 4645)) and perform the steps that follow.

Figure 48.49: The Optimization History Monitor Dialog Box



i. Enable the **Plot** option if you want to display a plot of the optimization history, which will automatically be plotted in an appropriate graphics window.

ii. Enable the **Write** option if you want to save the optimization history data in a file, and specify the **File Name**.

Note that you can display a plot of the optimization history data generated during the last calculation, even if the **Plot** or the **Write** options were not enabled during the calculation, as long

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

as you have not discarded the data using the **Clear** button. Simply click the **Plot** button and the plot will be displayed in the active graphics window.

j. Click the **Apply** button to save your optimizer settings.

k. Click the **Summary** button to display a summary of the mesh morpher/optimizer settings in the console.

l. Click the **Optimize** button to initiate the optimization process. Information about each run (that is, each design modification) that is generated as part of the production of a design stage will be displayed in the console, and the deformed mesh for each run will be updated in the graphics window.

If you set up the saving of intermediate case and data files in step 12.f and your solution is interrupted during the optimization run, you can restart the calculation using the latest intermediate files. Note that you should not revise the deformation regions or constraint settings for the restarted mesh.

13. If you want to use Design Exploration in Ansys Workbench to explore multiple deformation scenarios, define the optimizer settings.

a. Click the **Optimizer** tab (Figure 48.47: The Optimizer Tab of the Mesh Morpher/Optimizer Dialog Box (p. 4641)).

b. Select **workbench** from the **Optimizer** drop-down list. Note that **workbench** is only available if you have launched your Ansys Fluent session from Ansys Workbench.

c. If necessary, revise the default settings in the **Mesh Quality** group box, as described in step 12.e. Note that the design point associated with a poor quality mesh will not be updated in Workbench.

14. Click **OK** to close the **Mesh Morpher/Optimization** dialog box.

15. If you want to use Design Exploration in Ansys Workbench to explore multiple deformation scenarios, save your case file and close your Ansys Fluent session. You can then proceed to Ansys Workbench and define the design points that will provide the multiple values for the input parameters you created / assigned in step 7.c. For more information about using design points to explore Ansys Fluent results, see Working With Input and Output Parameters in Workbench in the Ansys Fluent in Workbench User's Guide.

16. If you manually deformed the mesh or used the built-in optimizers, run your case file with the deformed mesh.

17. If you want to rerun a case file in which the mesh has been deformed by the **Optimize** button in the **Optimizer** tab, you should not revise the deformation regions or constraint settings for the deformed mesh.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 49: Performing System Coupling Simulations Using Fluent

You can use Ansys System Coupling to perform coupled simulations that involve multiple physics solvers, coupling active co-simulation participants and/or importing static data from an external data source. For example, you can run Mechanical and Fluent) in a single analysis and/or import data from an output file into an Fluent analysis. Once the physics and coupling setups are complete, the coupled analysis is executed and managed by System Coupling.

System Coupling can be used in the following contexts:

- **System Coupling's user interfaces**: System Coupling is run from its graphical user interface (GUI) or its command-line interface (CLI).
- **System Coupling in Workbench**: System Coupling is run from the Workbench interface.

Note:

For System Coupling cases using Fluent, it is a best practice to run Fluent in double precision.

Additional information can be found in the following sections:

49.1. Supported Capabilities and Limitations
49.2. Performing System Coupling in the GUI or CLI
49.3. Performing System Coupling in Ansys Workbench
49.4. Variables Available for System Coupling
49.5. System Coupling Related Settings in Fluent
49.6. FSI Setup Recommendations for Fluent-Mechanical Couplings
49.7. How Fluent's Execution is Affected by System Couplings
49.8. Restarting Fluent Analyses as Part of System Couplings
49.9. System Coupling case with Fluent using Patched Data
49.10. Running Fluent as a Participant from System Coupling's GUI or CLI
49.11. Troubleshooting Two-Way Coupled Analysis Problems
49.12. Product Licensing Considerations when using System Coupling

49.1. Supported Capabilities and Limitations

Ansys Fluent supports the following capabilities when used in a System Coupling analysis:

- Force and displacement data transfers on wall boundaries (including walls in sliding mesh zones), porous jump boundaries, and porous zone interfaces:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - Input of incremental displacements data from System Coupling allows moving and deforming mesh specification on boundary wall regions (including walls in sliding mesh zones), porous jump boundaries, and porous zone interfaces (via a zero-thickness porous jump boundary). Note the additional **System Coupling** option in the **Dynamic Mesh Zones** dialog box. For more information, see Specifying the Motion of Dynamic Zones (p. 1904).

  - Output of force data on boundary wall regions (including walls in sliding mesh zones), porous jump boundaries, and porous zone interfaces (via a zero-thickness porous jump boundary). Fluent can serve the total force variable (viscous and normal) on all wall boundaries (with or without enabling the **System Coupling** option in the **Dynamic Mesh Zones** dialog box). For details about the forces transferred from the wall and porous jump boundaries, see Variables Available for System Coupling (p. 4654).

- Thermal data transfers on wall boundaries and cell zones:

  - Input of temperature data, heat flow data, and heat transfer coefficient data from System Coupling on all wall boundaries. Note the additional **via System Coupling** option on the **Thermal** tab of the **Boundary Conditions** dialog box.

    Input of surface heat rates from System Coupling (via Maxwell) on coupled walls adjacent to a solid zone.

  - Input of heat rate data from System Coupling on cell zones.

  - Output of temperature data, heat flow data, and heat transfer coefficient data to System Coupling on all wall boundaries (with or without enabling the **System Coupling** option in the **Dynamic Mesh Zones** dialog box).

  - Output of temperature data to System Coupling from cell zones.

- Electric arc propagation on fluid volumes between Fluent and Ansys Maxwell.

  - Input of Lorentz force, temperature, and heat flow data on cell zones from System Coupling.

  - Output of electrical conductivity, temperature, and heat flow data to System Coupling from cell zones.

- Time-dependent motion and excitation on coupled and non-coupled bodies :

  - Supported for System Coupling's CLI and GUI (not for System Coupling in Workbench).

  - Motion can be defined on coupled bodies.

    → Motion must be defined with constant velocity in Fluent and consistent/matching displacement in Maxwell (variable velocity not supported.)

    → Currently supported only for Maxwell/Fluent electromagnetic-thermal analyses.

  - Motion can be specified for rigid bodies and cell zones.

    → Can set solid cell zone movement as rigid body motion using total displacement (vs. System Coupling time) relative to the original cell zone location.

  - Motion can be set using a profile.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

→ This motion corresponds to the displacement at the end of the current coupling step.

→ Fluent's motion should be set to be consistent with Maxwell's.

- For command-line runs, active surface thermal coupling between two Fluent applications, one running in transient and one running in steady state.
  - For such mixed type of simulations, Fluent has been extended to support time averaged (HTC or T_{ref}) data transfers from transient to steady state Fluent.
  - Within a coupling step, a new sub-stepping option is introduced in Fluent which uses a smaller step size than the specified time step size in System Coupling. This sub stepping option will be used to accumulate the unsteady thermal data of HTC or T_{ref}. This accumulated thermal data will be used to compute time averaged data according to the following formulation:

    $$\sum(\phi_i * \Delta t) \Big/ \sum \Delta t$$

    where Δt is the physical time step size and ϕ_i is the thermal scalar value (HTC or T_{ref}).
  - Also, the Fluent coupling code has been extended to compute HTC based on Wall Functions and Y plus. By default, Fluent coupling code uses a wall adjacent based HTC method.

    All these HTC calculation methods and sub-stepping options can be accessed by TUI commands at this location:

    ```
    /define/models/system-coupling-settings/htc-calculation-settings>

    htc-calculation-method/    unsteady-statistics/
    ```
  - Fluent supports only these two quantities (HTC and T_{ref}) for time averaged data transfers and uses an explicit coupling approach (one coupling iteration per coupling step). Apart from time averaged thermal data of HTC and T_{ref}, Fluent also supports instantaneous values of HTC based on Wall Functions or Y plus for data transfers.
- Data transfers from wall boundaries that are in Fluent sliding mesh zone or rigid body zone.
- The use of triangular and quadrilateral faced interface cell types.
- Full support for local and distributed parallel solver execution.
- Custom solver input and restart files (when using System Coupling's GUI or CLI).
- Convergence data for all equations (for example, continuity, momentum, and energy convergence data) is shared with the system coupling service at run time. Monitor data is shared if the monitor point has an iteration frequency set. If the monitor point has a time step frequency set, monitor data will not be shared with System Coupling.

  In Workbench, this data from Fluent can be charted in conjunction with other data in the system coupling chart at all available intervals. Intermediate data points are not artificially created in the system coupling chart.

  Monitor data can also be plotted in Fluent Solution Monitoring.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note the following limitations when using Fluent in a System Coupling analysis:

- Using System Coupling with the Remote Solver Manager (RSM) in Fluent is not supported. If attempted, you will receive the following message:

```
Solution updates for Fluent systems participating in System Couplings
must run in the foreground. This change has been automatically applied.
```

- It is recommended to use Unified remeshing for all system coupling cases with dynamic mesh remeshing (except 2.5D, which is not supported by unified remeshing). *Cases coupling Fluent with Ansys Structural in Workbench may experience errors if using Methods-Based remeshing.*

- When using a wall and wall-shadow pair in Fluent as System Coupling interfaces, you'll need to create a duplicate surface in Mechanical. These will be paired with two sets of shell elements in Mechanical. For more information, see Coupling Thin Surfaces in CFX in the *Mechanical User's Guide*.

- Fluent must use a 3D mesh, with data transfer regions consisting of element faces within the 3D mesh. Data Transfer regions cannot exist in 2D meshes (where the data transfer region would be a line/curve).

- Data transfer regions on the coupling interface cannot contain hanging nodes (which result from hanging node adaption and may be in hexcore or CutCell meshes if you have disabled their conversion when reading the file).

- For FSI, the mesh topology on a System Coupling boundary in Fluent must remain fixed for the duration of the analysis. This means that dynamic remeshing cannot be used on the System Coupling boundary, but interior nodes away from the System Coupling boundary can still be remeshed during the solution.

- To transfer forces on wall and wall-shadow pair, you need two surfaces in Mechanical to match the two surfaces in Fluent. In this case, you cannot use a single set of shell elements in Mechanical. An example of a wall and wall-shadow pair is two different fluids on either side of a zero-thickness internal wall in Fluent.

- The non-iterative time-advancement (NITA) scheme in Fluent cannot be used in a System Coupling analysis.

- When executing design points for System Coupling cases where Fluent is a participant, **Program Controlled** should not be used for Fluent Initialization Method.

- Shell conduction is not available when the Fluent region is set up to receive thermal data via System Coupling.

49.2. Performing System Coupling in the GUI or CLI

System Coupling can be run in either of its user interfaces: its graphical user interface (GUI) or its command-line interface (CLI). Both contexts provide enhanced control over coupled simulation processes, including automatic starts and restarts for participants, the ability to manipulate the System Coupling data model, and an interactive solution workflow. In these contexts, you set up participant physics in the participant's user interface, but perform the coupled analysis — starting the coupling service, loading participants, specifying values for coupling-related analysis settings, and automatically starting

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

participants — in the System Coupling interface. Alternatively, if a coupled analysis setup was exported from Workbench, you can open and execute it in System Coupling's GUI or CLI.

When the coupled analysis is created in System Coupling's GUI or CLI, the coupling workflow requires a System Coupling participant setup file (`.scp`) written by Fluent, as described in Generating a System Coupling Setup File (p. 4651). When a coupled analysis setup has been exported from Workbench, however, manual generation of the `.scp` file is not necessary

For more information, see Using System Coupling's User Interfaces.

49.2.1. Generating a System Coupling Setup File

To set up a coupled analysis in System Coupling's GUI or CLI, you must generate a System Coupling participant setup (`.scp`) file.

You can write a `.scp` file at any time even if you have not specified any System Coupling information by selecting the **Write SCP File...** ribbon tab item to open the **Select File** dialog box.

File → **Export** → **System Coupling...** → **Write SCP File...**

Enter the name of the System Coupling participant setup file and click **OK**.

You also have the option of automatically writing all available surface and volume information that could be used in a System Coupling run any time you write a case file.

You can enable this at any time by selecting the **Auto-write SCP File...** ribbon tab item.

File → **Export** → **System Coupling...** → **Auto-write SCP File...**

On the dialog box that appears, select **Auto-write SCP File with Case File** and click **OK**.

Afterwards, any time you write a case file, a `.scp` file will also be written.

Important:

From Ansys 2020 R1 onward, element data can be used for mapping surface conservative quantities such as surface forces, heat flows, and volumetric heat rates using the following command:

```
/define/models/system-coupling-settings/use-face-or-element-based-
data-transfer?
```

This enhancement mitigates common mapping issues such as edge effects and profile smearing, resulting in improved mapping accuracy and results.

For cases using `.scp` files created before Ansys 2020 R1, it is recommended that you generate a new `.scp`, and use this updated file for the cosimulation run.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

49.2.2. Using Custom Input Files in System Coupling's User Interfaces

In some cases, setup changes are desired or are required to avoid failure of the coupled analysis. When you are running a coupled analysis in System Coupling's GUI or CLI, settings defined under the participant's Execution Control | Fluent Input object allow you to define custom input files. You can use these custom inputs to make changes to Fluent's setup prior to starting an initial or restarted solution.

For more information, see:

49.2.2.1. Setting the Fluent Input Option

49.2.2.2. Generating a Fluent Journal Script

49.2.2.1. Setting the Fluent Input Option

The **Fluent Input | Option** setting defines the type of input files to be used for coupling runs, which in turn determines if/how the inputs may be customized.

The table below provides information on how different Option values affect the input files are used for coupled analysis runs.

Table 49.1: Using the Fluent Input | Option setting to define custom input files

Option setting value	Settings defining input files[1] (p. 4653)	System Coupling's behavior		Comments
		For initial runs:	For restart runs:	
Initial Case File (default)	The input file defined is defined by the Case File setting.	Auto-generates a Fluent journal script based on the specified input file.	Uses the case and data files created by Fluent during the previous run.	A journal script may be generated and edited for further customization. This is the best option for general workflows.
Initial Case and Data File	The input files defined are defined by the Case File and Data File settings.	Auto-generates a Fluent journal script based on the specified input files.	Uses the case and data files created by Fluent during the previous run.	Input files can be edited. A journal script may be generated and edited for further customization.
Journal File[2] (p. 4653)	The input script is defined by the Journal File setting.	Uses the specified Fluent journal script.	Uses the specified Fluent journal script.	Input files can be edited. Additional commands can be added to the

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Option setting value	Settings defining input files[1] (p. 4653)	System Coupling's behavior		Comments
		For initial runs:	For restart runs:	
				Fluent journal script. This option is designed for expert users.
1: When specifying any input file, ensure that the file is valid. System Coupling does not validate input files before the solution is started/restarted. **2:** For multiple restarts, ensure that you keep the case and data file values up to date.				

49.2.2.2. Generating a Fluent Journal Script

If the Fluent Input | Option setting is not set to ***`JournalFile`***, then you can use the `GenerateInputFile()` command to generate a Fluent journal script on demand, as shown below:

```
>>> GenerateInputFile(ParticipantName = 'FLUENT-2', FileName = 'FluentRestart.jou')
```

When issued, this command generates a journal script that respects the participant's **Fluent Input** settings and which may be set as the value of the Journal File.

In this way, you can both edit the input file(s) to be used for the coupling run and add further customizations by adding commands to the generated journal script.

49.3. Performing System Coupling in Ansys Workbench

You can perform system coupling in Workbench by connecting a **System Coupling** component system to your Mechanical system and to your Ansys Fluent fluid flow analysis system.

The following is the list of supported coupling participants:

- **Fluent**
- **Static Structural**
- **Transient Structural**
- **Steady-State Thermal**
- **Transient Thermal**
- **External Data**

Connecting the **Setup** cell from an Ansys Fluent analysis system to the **Setup** cell for the **System Coupling** component system signals the latter system that the Fluent solver will act as a co-simulation

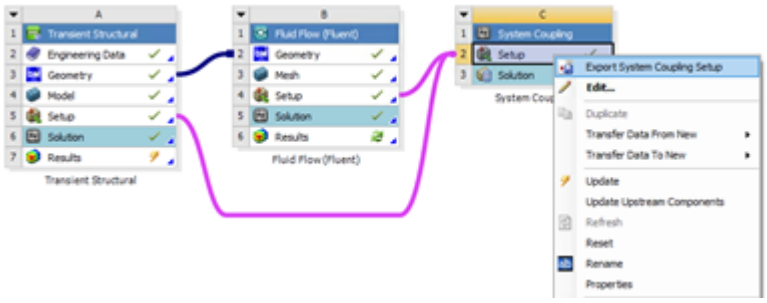
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

participant in a coupled analysis. Most of the coupling related settings for your analysis are made through the **System Coupling** system's **Setup** user interface. The few coupling related settings that are required in the Fluent setup are described below, in the section System Coupling Related Settings in Fluent (p. 4658).

Once the coupling setup is complete, you can either run the analysis in Workbench or export it for execution in one of System Coupling's user interfaces.

- When the coupled analysis is run from Workbench, it is executed by updating the System Coupling system's **Solution** cell, rather than the same cell in the connected co-simulation participant systems.
- Should you need to perform a co-simulation run outside of Workbench, you can export the relevant input files (`.cas`, `.dat`, `.scp`) to a specified directory. To do this, right-click the System Coupling **Setup** cell and select **Export System Coupling Setup**.

Figure 49.1: Exporting System Coupling Files from Workbench



For more information, see Using System Coupling in Workbench and Supported Coupling Participants for System Coupling in Workbench.

Note:

In order to have changes made in Fluent systems propagated to downstream **System Coupling** systems, save your project, synchronize with Workbench, or close Fluent. This will ensure that the information passed to System Coupling matches settings in Fluent. If this is not done, the analysis may be allowed to run but the following Workbench state error will be issued at the end of the run:

```
Update of the Solution component in System Coupling failed: Update
of the Solution component in System Coupling did not mark container
as updated (final state is OutOfDate).
```

49.4. Variables Available for System Coupling

The following variables will be available on all boundary wall regions.

Table 49.2: Variables On Boundary Wall Regions

Display Name / Internal Name	Transfer Direction	Quantity Type
force / `force`	Output	Force

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Display Name / Internal Name	Transfer Direction	Quantity Type
displacement / `displacement`	Input	Length
temperature / `temperature`	Input and Output	Temperature
heat rate / `heatrate`	Input	Heat Rate
heat flow / `heatflow`	Input and Output	Heat Rate
heat transfer coefficient / `heat-transfer-coefficient`	Input and Output	Heat Transfer Coefficient
near wall temperature / `near-wall-temperature`	Output	Temperature

The following variables will be available on all cell zone regions.

Table 49.3: Variables On Cell Zone Regions

Display Name / Internal Name	Transfer Direction	Quantity Type
heat rate / `heatrate`	Input	Heat Rate
temperature / `temperature`	Input and Output	Temperature
Lorentz force / `lorentz-force`	Input	Force
electrical conductivity / `elec-trical-conductivity`	Output	Electrical conductivity

The following variables will be available on all porous jump boundaries.

Table 49.4: Variables On Porous Jump Boundary

Display Name / Internal Name	Transfer Direction	Quantity Type
force / `force`	Output	Force
displacement / `displacement`	Input	Length

*Represents the force vector $\vec{F}$ (F_x, F_y, F_z) and the displacements vector $\vec{d}$ (Δd_x, Δd_y, Δd_z) respectively.

49.4.1. Force transferred to System Coupling from a Wall Boundary

The force vector sent from each wall boundary in Fluent to System Coupling is given by:

$$F=(p+p_{raise}-p_{ref})A+F_{viscous}+F_{particle} \tag{49.1}$$

where

- p is the gauge pressure in Fluent.
- p_{raise} is the pressure rise due to **Floating Operating Pressure** if enabled (for details, see Floating Operating Pressure (p. 1763)).
- p_{ref} is the **Reference Pressure**, which differs from the **Operating Pressure** (for details, see Reference Values (p. 4158)). If the **Reference Pressure** is set to zero, then gauge pressure forces will be sent to System Coupling. If it is set to the negative of the operating pressure, then absolute pressure forces will be sent to System Coupling.
- A is the face area vector.
- $F_{viscous}$ is the force vector due to viscous forces.
- $F_{particle}$ is the force vector due to particle forces.

Note:

If buoyancy is active, the hydrostatic pressure force due to a nonzero operating density is excluded from the force calculation. To include this contribution, the operating density must be set to 0.

For more information on force calculations, see Computing Forces, Moments, and the Center of Pressure in the *Fluent Theory Guide*.

49.4.2. Force transferred to System Coupling from a Porous Jump Boundary

When a porous jump boundary is selected as a source region for a System Coupling analysis, the forces transferred from this porous jump boundary are calculated differently depending on if the porous jump is between fluid regions, or between a fluid region and a porous zone.

- When the porous jump separates two fluid zones, or forms an interior region in a fluid zone:

  The force transferred to System Coupling is the net force across the jump. The thickness and coefficient parameters defined for the porous jump are used to calculate the pressure drop across the jump. This pressure drop is converted to a force vector that is transferred to System Coupling. This force does not include any viscous forces.

- When the porous jump separates a porous zone and a fluid zone:

  This case will occur if the upstream or downstream fluid-porous interface is changed to a porous jump. The porous jump is used to allow a connection between the upstream and downstream porous zone interface and System Coupling, since the porous zone interface regions cannot connect directly to System Coupling.

  The face pressure on the porous jump is converted to a force vector and transferred to System Coupling. A positive face pressure will produce a force vector that always points into the porous zone. The force calculated does not include any viscous forces.

  It is best to assign zero-thickness to the porous jump so that there is no pressure drop across the jump. Note that volumetric forces in the porous zone are not passed to System Coupling.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

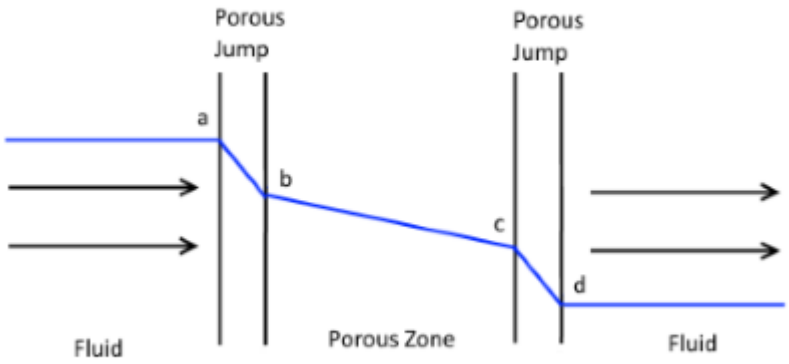
This case is intended to model thin porous zones where the deformation of the zone can be approximated by considering only the fluid pressure on the upstream and downstream porous interfaces.

If a nonzero thickness is assigned to a porous jump that is adjacent to a porous zone, then there will be a pressure drop across the porous jump. The force transferred to system coupling is calculated using the face pressure on the upstream side of the porous jump, before the pressure drop across the jump has been applied. See the image below.

– Force transferred on the upstream side of the porous zone (a) will be consistent with the upstream fluid pressure (a).

– Force transferred on the downstream side of the porous zone (c) will be inconsistent with the downstream fluid pressure (d). On the downstream side of the porous zone, the force transferred to system coupling is consistent with the exit pressure from the porous zone, before the pressure drop across the porous jump has been applied. The downstream fluid pressure is calculated after the pressure drop is applied.

Figure 49.2: Force transferred to System Coupling when Porous Jump Thickness is Non-Zero



For information about the porous jump boundary condition, see Porous Jump Boundary Conditions (p. 1519).

49.4.3. Displacement transferred from System Coupling

The following applies in a coupled analysis where displacement is received by Fluent.

- The displacement data received by Fluent represents the incremental displacement for the current time step (the incremental displacement since the end of the previous time step).
- The displacement variable is only available on walls that have the **System Coupling** moving and deforming mesh (MDM) option selected, and that qualify as valid coupling regions.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- In a general coupled analysis, when Fluent solves before or simultaneously to the solver sending the displacement (such as Mechanical), then during the first coupling iteration of each coupling step the displacement received is 0 [m].

49.4.4. Displacement transferred from System Coupling to a Sliding Mesh Zone

When transferring data to or from a wall boundary in a sliding mesh zone, you must ensure that the coupled participant (such as Mechanical) does not rotate its mesh. One way to do this in Mechanical is to use a Rotational Velocity.

For more information about the Rotational Velocity boundary condition, see Rotational Velocity in the *Mechanical User's Guide*. For information about sliding meshes in Fluent, see Using Sliding Meshes (p. 1804).

49.4.5. Absolute Pressure Example

Force transferred from a Fluent wall boundary in a System Coupling analysis is the gauge pressure minus the specified Reference Pressure. You can change the force transferred by changing the Reference Pressure value. The following example of a pressurized sub-sea pipe is a demonstration of how to transfer the absolute pressure in a System Coupling analysis by changing the Reference Pressure.

Consider flow in a pressurized sub-sea pipe. The pipe is pressurized to 110 bar absolute and the exterior water pressure is 100 bar. The exterior sea water is not modeled. The internal surface of the pipe forms an FSI interface to a Mechanical model of the pipe.

In Fluent the Operating Pressure would typically be set to 110 bar. By default the forces passed to System Coupling would only include the local pressure changes relative to 110 bar, which clearly does not represent the real physics. To include the absolute pressure in the forces sent to System Coupling, the Reference Pressure in Fluent should be set to -110 bar. The Mechanical model would then need a pressure of 100 bar applied to the exterior surfaces of the pipe.

Although this approach is valid, it is not ideal because the correct structural solution is only obtained if 100 bar of the interior pressure cancels out with the 100 bar exterior pressure. Relying on two large forces to exactly cancel out is prone to discretization (mesh resolution) errors. Minimizing the forces sent across the FSI interface is also good practice. Therefore the best approach is to use an Operating Pressure of 110 bar and a Reference Pressure of -10 bar in Fluent. The forces Fluent sends to System Coupling are then based on the gauge pressure minus -10 bar, that is, the gauge pressure plus 10 bar. This accounts for the pressure difference between the interior and exterior of the pipe, so there is no need to apply an external pressure in Mechanical. We are no longer relying on two large forces cancelling out and we have minimized the forces sent across the FSI interface.

49.5. System Coupling Related Settings in Fluent

- Double-Precision Solver

  System Coupling cases may benefit from the use of Fluent's double-precision solver. For coupling cases with moving and deforming meshes (MDM), it is a best practice to use Fluent's double-precision solver. For more information about Fluent's single- and double-precision solvers, see Single-Precision and Double-Precision Solvers in the *Fluent User's Guide* (p. 185).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

  - In the Fluent Launcher, select **Double Precision** to use Fluent's double-precision solver.

- Dynamic Mesh

  - The **System Coupling** option must be selected on the desired moving and deforming wall boundaries in order to obtain displacements from other co-simulation participants taking part in the coupled analysis.

  - Make sure the appropriate solution stabilization options are selected in the **Solver Options** tab for the **System Coupling** boundary zone (and possibly other boundary zones nearby). Solution stabilization should be activated if you find that the Fluent solution is particularly sensitive to the data obtained through system couplings. For example, it is often beneficial for FSI problems in which the fluid is incompressible.

- Heat Transfer

  - The **via System Coupling** option must be selected on the desired wall boundaries in order to obtain heat transfer data (temperature, heat flow, or heat transfer coefficient) from other participants taking part in the coupled analysis.

  - The Fluent coupling code has been extended to compute HTC based on Wall Functions and Y plus. By default, Fluent coupling code uses a wall adjacent based HTC method.

    All these HTC calculation methods and sub-stepping options can be accessed by TUI commands at this location:

    ```
    /define/models/system-coupling-settings/htc-calculation-settings>

    htc-calculation-method/    unsteady-statistics/
    ```

  - Two-way thermal coupling cases may experience slow convergence when the near wall temperature value used in the heat transfer coefficient calculation is close to the wall temperature and is not a good representation of the free stream temperature. This typically occurs when the first node of the mesh is well within the boundary layer.

    Convergence can be improved for such cases by allowing Ansys Fluent to send System Coupling a heat transfer coefficient value based on a constant, user-specified reference temperature. Note that this option results in the heat transfer coefficient having the same definition as the field variable **Surface Heat Transfer Coef.**.

    Note:

    Basing the heat transfer coefficient value on a constant reference temperature may not be suitable when there is a strong variation in the free stream temperature along the coupled wall.

    Specify that the heat transfer coefficient is based on a reference temperature by using the following text command:

    ```
    /define/models/system-coupling-settings/htc-calculation-settings/htc-calculation-methods/use-tref-in-htc-calculation?
    ```

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Define the reference temperature in the **Temperature** field of the **Reference Values** task page. It is recommended that you use a value that is close to the fluid free stream temperature.

Setting Up Physics → **Solver** → **Reference Values...**

- Custom Data Transfer Variables via User-Defined Memory (UDM)

  You can create user-defined data transfer variables using **Zone-Based Memory Allocation**. For details, see User-Defined Memory Storage in the *Fluent Customization Manual*.

  Once created you can enable the UDM for use as a System Coupling variable using the following text command:

  ```
  define/models/system-coupling-settings/user-defined-coupling-variables-via-udm
  ```

- Run Calculation

  – When running as part of a transient coupled analysis, the step size for and duration of the analysis are controlled by System Coupling.

  → The **Time Step Size** specified in the Fluent setup is currently ignored; the Fluent solution will be advanced using the time step size specified as part of the System Coupling setup.

  → The **Number of Time Steps** is also ignored.

  – The specified **Max Iterations/Time Step** corresponds to the maximum number of nonlinear solver iterations performed per coupling iteration.

For steady-state, system-coupled Fluent analyses, the number of iterations specified in Fluent is equal to the maximum number of solver iterations solved per coupling iteration in a coupling step. Prior to ANSYS FLUENT 14.5, the number of iterations specified in Fluent was equal to the solver iterations solved in a coupling step and was divided equally between the coupling iterations in a coupling step. You can recover the previous behavior by using the following command: `(rpsetvar 'sc/steady/default-iteration-method? #f)`.

For cases that use a pseudo time method, refer to Performing Calculations with a Pseudo Time Method (p. 3673) in the Fluent User's Guide (p. 1).

For steady-state dynamic mesh applications, refer to Steady-State Dynamic Mesh Applications (p. 1924) in the Fluent User's Guide (p. 1).

49.6. FSI Setup Recommendations for Fluent-Mechanical Couplings

The following are best practices for FSI (fluid-structure interaction) cases involving Fluent and Mechanical:

49.6.1. Using Contact Detection for Fluent-Mechanical FSI Problems

49.6.2. Recommendations for Dynamic Mesh Settings for Fluent-Mechanical FSI

49.6.3. Pathologies & Candidate Resolutions for Fluent-Mechanical FSI

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

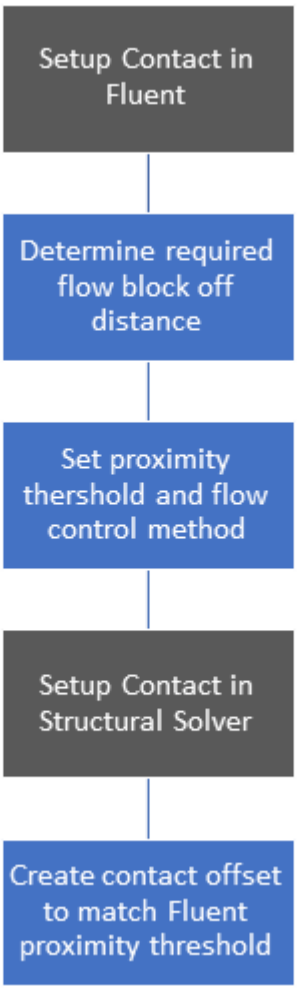
49.6.1. Using Contact Detection for Fluent-Mechanical FSI Problems

In general terms, contact detection determines where the computed mesh motion will result in contact of a moving surface with other surrounding surfaces. However, the Mechanical and Fluent solvers each define it a bit differently. For Mechanical, it refers to surfaces/bodies coming into physical contact, while for Fluent, it refers to what Fluent does when two surfaces come into close proximity (that is, flow blockage).

Contact detection is often used in FSI simulations requiring flow blockage – that is, cases for which you need to restrict the flow or limit the movement between boundaries – such as valve applications with highly flexible flow-control geometry. Using the same geometry, you can apply an offset/proximity threshold to both the fluid and structural sides of the analysis. The proximity threshold applied in Fluent (as described in Contact Detection Settings (p. 1890)) specifies a tolerance within which cells are flagged for the application of flow resistance, while the offset specified in Mechanical (as described in Geometric Modification in the *Mechanical User's Guide*) restricts the moving boundary by a certain distance from a surrounding surface, so that a flow restriction can be applied in that cavity. This also prevents the fluid mesh from folding if the mesh is pinched between the moving boundary and a surrounding surface. The offset applied to the structural side is intended to mirror the movement restriction applied on the fluid side, so that both bodies are aligned during coupling. This simplifies the process because no modification to the geometry or mesh is required.

The recommended sequence for setting up contact detection in a coupled FSI problem between a Fluent-Mechanical coupling is summarized below:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



1. Set up contact detection in the Fluent solver.

   a. Identify the distance that will be required to block off the flow. (To do this, you may need to run one-way FSI analysis to test how close the moving boundary can get to surrounding surfaces without causing the mesh to fold.)

   b. Using the obtained distance, set the contact detection tolerance — that is, a proximity threshold to detect when the moving boundary is within a specified tolerance of the surrounding surface.

   c. Set up contact marks or contact zones to block or restrict the flow between the offset, using the flow control settings provided by the Fluent solver.

   d. Choose a flow control method. Ansys recommends that you use contact numerics. Contact numerics mark cells within the proximity threshold set between two or more faces and blocks the flow in the marked cells.

   For more information, see Contact Detection Settings (p. 1890).

2. Set up contact detection in the Mechanical solver.

   In the structural solver, configure contact detection settings to match those of the Fluent solver.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For more information, see Geometric Modification in the *Mechanical User's Guide*.

When trying to determine the appropriate contact detection distance and/or if contact is established in Fluent, it can be helpful to turn on contact marks rendering, as follows:

1. In the Fluent TUI, specify that contact cells should be rendered by entering the following text command:

   ```
   /define/dynamic-mesh/controls/contact-parameters/render-contact-cells? yes
   ```

2. Verify the contact markings in the **Contours** dialog box, as follows:

   a. Under **Contours of** , select **Cell Info** and **Contact Markings**.

   b. Under **Options**, clear the **Node Values** check box.

In addition, it can be useful to set the verbosity for contact detection so as to determine when contact is established in the transcript. In the Fluent TUI, set verbosity by entering the following command:

```
/define/dynamic-mesh/controls/contact-parameters/verbosity 1
```

49.6.2. Recommendations for Dynamic Mesh Settings for Fluent-Mechanical FSI

Smoothing

- Diffusion-based smoothing is generally recommended, using the boundary distance diffusion function.
- A diffusion parameter (α) = 1.5 is a good starting point, as it works well for absorbing the motion with the far field mesh.
- If the boundary layer mesh deforms, increasing the diffusion parameter can further preserve the mesh close to the moving boundary.
- If remeshing is not required **Smoothing From Reference Position** should be selected, specifically for periodic motion.
- Although the first bullet point generally recommends diffusion-based smoothing, there are exceptions. As an example, consider the intersection between an FSI boundary and a non-FSI boundary, such as a flow outlet.

  If diffusion-based smoothing is used, then the nodes on the non-FSI boundary will remain on its original surface. If the FSI boundary motion has a large tangential component (for example, displacement caused by thermal stresses), this can lead to a mesh wiggle on the non-FSI boundary:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 49.3: Undeformed mesh

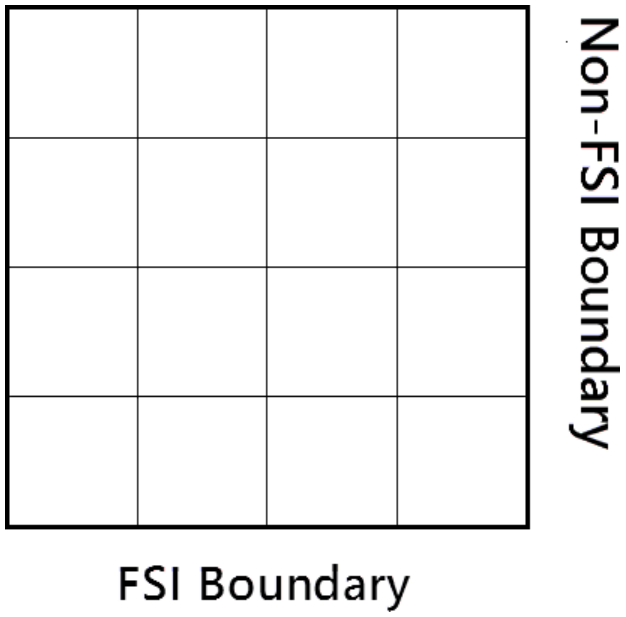
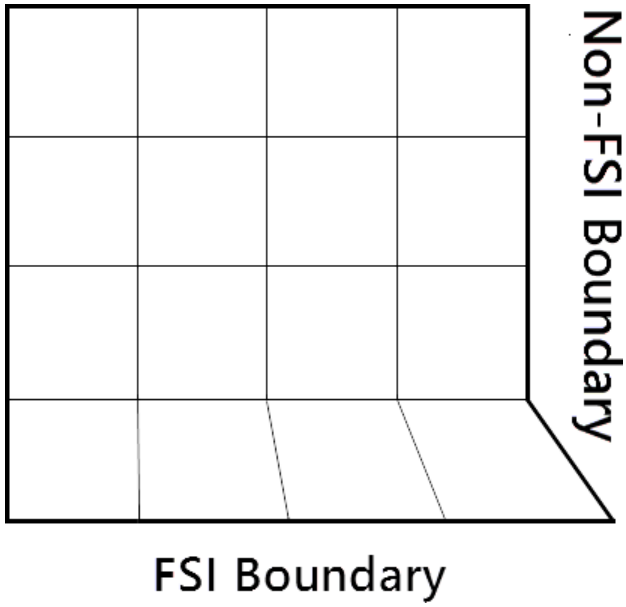


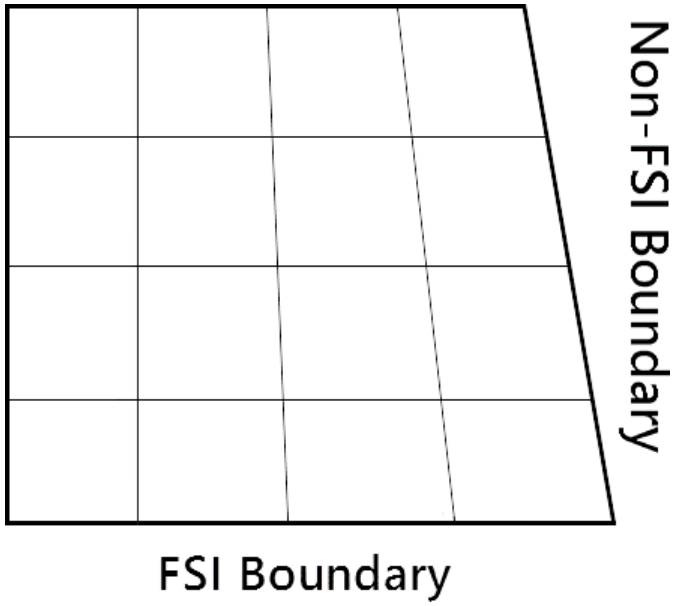
Figure 49.4: Deformed mesh with diffusion smoothing



Better smoothing behavior will result by using linearly elastic smoothing or radial basis function smoothing, and setting the geometry definition for the non-FSI boundary to **unspecified**:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 49.5: Deformed mesh with linearly elastic smoothing and non-FSI boundary set to Unspecified



Remeshing

- Recommended starting point for the remeshing minimum (Lmin) and maximum length scales (Lmax) are 0.4Lmin and 1.4Lmax.
  - Global length scales can be set through: Dynamic Mesh > Settings > Remeshing > Parameters.
  - Global length scales of the mesh can be found in Mesh Scale Info.
  - Local length scales can be found for each Dynamic Mesh Zone under the Meshing Options tab
- Remeshing and smoothing should be enabled locally for all Dynamic Mesh Zones that are of type **Deforming**, this allows remeshing and smoothing to be applied to faces adjacent to the System Coupling Region.
  - In addition, both **Region Face** and **Local Cell** should be selected for remeshing methods. **Region Face** allows remeshing based on length scales while **Local Cell** allows for remeshing based on skewness. It is important to note that remeshing will not occur if **Local Cell** is not selected.

It is also important to remember that there is no direct control over skewness and size criteria of the remeshed cells. Therefore, you can only indirectly control these criteria of the deformed mesh. For that reason, if the remeshed cells are not of adequate quality, you can further tighten the related criteria.

49.6.3. Pathologies & Candidate Resolutions for Fluent-Mechanical FSI

Troubleshooting information is available on the following topics:

49.6.3.1. Mesh Folds within the First Coupling Steps
49.6.3.2. Deformed Prism Layers
49.6.3.3. Interior Elements have High Skewness or Are Too Large/small

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

49.6.3.4. Divergence if Flow Block-Off is Established at the Beginning of a Run

49.6.3.1. Mesh Folds within the First Coupling Steps

1. Identify where the mesh folding is occurring.

   Open case file for the folded mesh and examine the Cell Volume contour for the domain. You can highlight the folded cells by setting the maximum range to 0, so that only cells with a negative volume are displayed. This can give an indication of what might be causing the mesh folding — that is, the time step size might be too large, or the dynamic zones may be set up incorrectly. For information on how to address cases where the prism layer is folding, see Deformed Prism Layers (p. 4666).

2. Rerun the case with smoothing and remeshing verbosity turned on. This may provide clues as to what is causing mesh folding.

   a. To turn on verbosity for smoothing, enter the following command into the Fluent TUI:

   ```
   /define/dynamic-mesh/controls/smoothing-parameters/verbosity 1
   ```

   b. To turn on verbosity for remeshing, enter the following command into the Fluent TUI:

   ```
   (rpsetvar 'dynamesh/remesh/verbosity 1)
   ```

Note:

If remeshing is not triggered, verify that the **Dynamic Mesh Zones** and length scales are set up correctly

1. Open the case file for the folded mesh.
2. Check the minimum and maximum length scales for the mesh and compare these to the remeshing values.
3. If needed:
   - Tighten the length scales and skewness factors.
   - Increase the **Size Remeshing Interval** value.

49.6.3.2. Deformed Prism Layers

A deformed prism layer can quickly cause a case to fail due to the mesh folding. Fluent currently does not support the remeshing of prism layers. If a case exhibits deformation in the boundary layer due to large movement/deformation of the FSI regions, there are three main recommendations that depending on the case can resolve this pathology:

1. Increase the **Diffusion Parameter** to increase the mesh stiffness around the boundaries and cause regions away from the moving boundary to absorb more of the motion.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. If the prism layer mesh around the moving body is uniform (i.e. constant number of prism mesh layers around the geometry), then Ansys recommends that you split the boundary layer mesh into a separate Cell Zone.

3. If the prism layer is not uniform (i.e., stair-stepping due to small gaps) and/or contact detection is required in Fluent, then the use of overset meshing is recommended. Overset meshing is also recommended for complex topologies.

49.6.3.2.1. Using Boundary Layer Smoothing and Region Face Remeshing

For cases that exhibit prism layer deformation/shearing (i.e., due to translational movement between two bodies in close proximity as shown in Figure 49.6: Skewed prism cells due to translational motion between two bodies in close proximity (p. 4667) below), boundary Boundary Layer Smoothing Method (p. 1834) can be implemented along with **Region Face Meshing**. Boundary Layer Smoothing preserves the boundary layer by applying the displacement vectors on the coupled face zone to the nodes of each cell in the boundary layer zone. This requires that the boundary layer mesh is split into a separate zone. As an example, we will go through the procedure of splitting the boundary layer mesh for the geometry shown below (Figure 49.7: Example geometry (p. 4668)). As the geometry includes a prism layer mesh on the outside walls as well as the inside walls, they will need to be separated individually. It is important to note that Boundary Layer Smoothing is only applicable when using Spring-based Smoothing.

Figure 49.6: Skewed prism cells due to translational motion between two bodies in close proximity

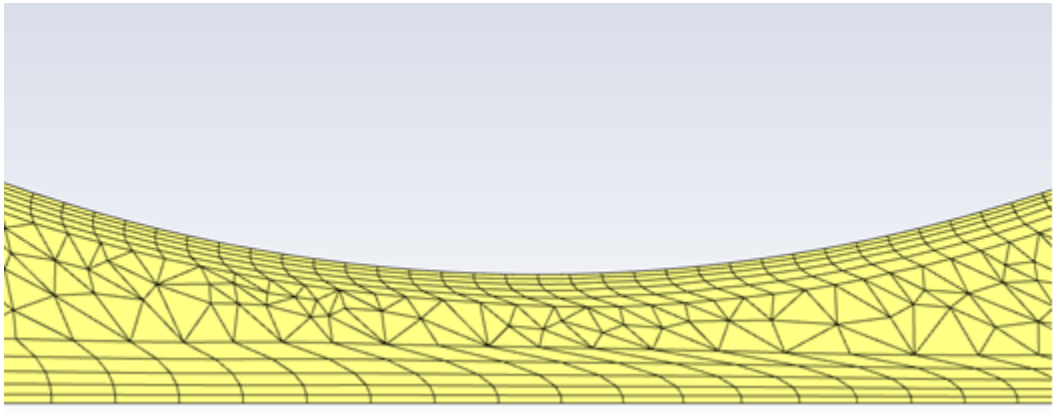
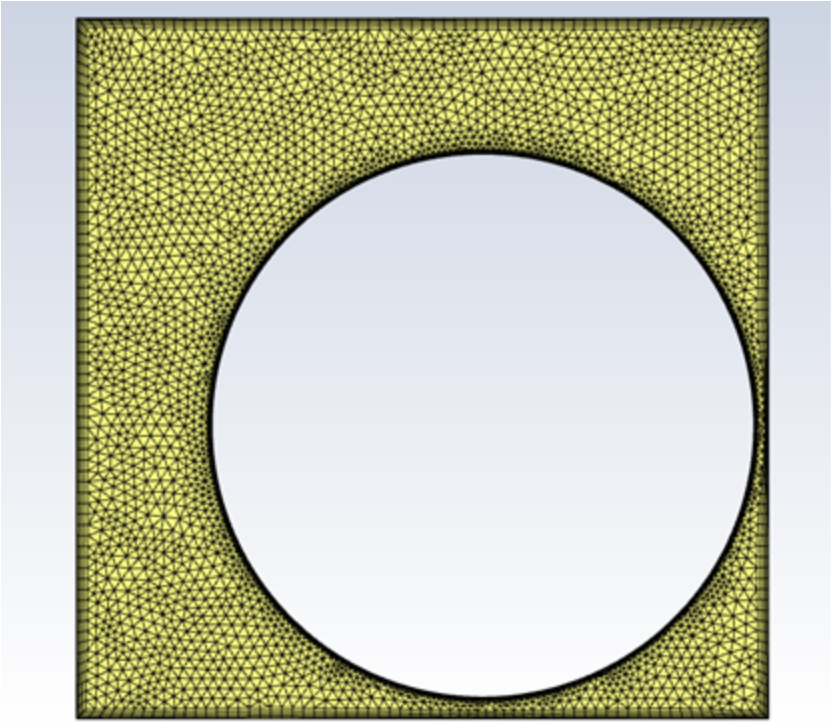


Figure 49.7: Example geometry



Split the prism layer cap to a separate cell zone condition to allow Region Face remeshing to be invoked:

1. Create a Boundary Cell Register with the required prism elements.

   a. Right-click **Solution**>**Cell Registers**>**New**>**Boundary**

   b. Use Cell Distance and highlight the boundary zone were the prism layers are located. The number of cells is set based on the number of prism layers.

2. Use the Boundary Cell Register to separate the prism layers into a separate cell zone.

   a. Click **Zones** > **Separate** > **Cells**.

   b. Select the register with the prism cells and the zone that you'd like to split.

3. Verify that the correct cell zones were created based on the cell registers and zones.

   In this case we have split our single cell zone into three separate cell zones. The first cell zone contains the inner boundary layer mesh (Figure 49.8: Cell Zone containing the inner boundary layer mesh (p. 4669)), the second cell zone contains the outer boundary layer mesh (Figure 49.9: Cell Zone containing the outer boundary layer mesh (p. 4669)) and finally the third cell zone contains the interior mesh (Figure 49.8: Cell Zone containing the inner boundary layer mesh (p. 4669)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 49.8: Cell Zone containing the inner boundary layer mesh

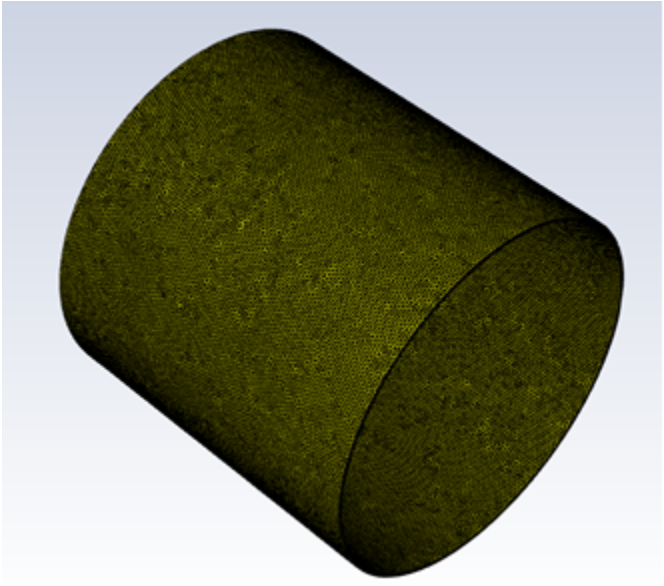
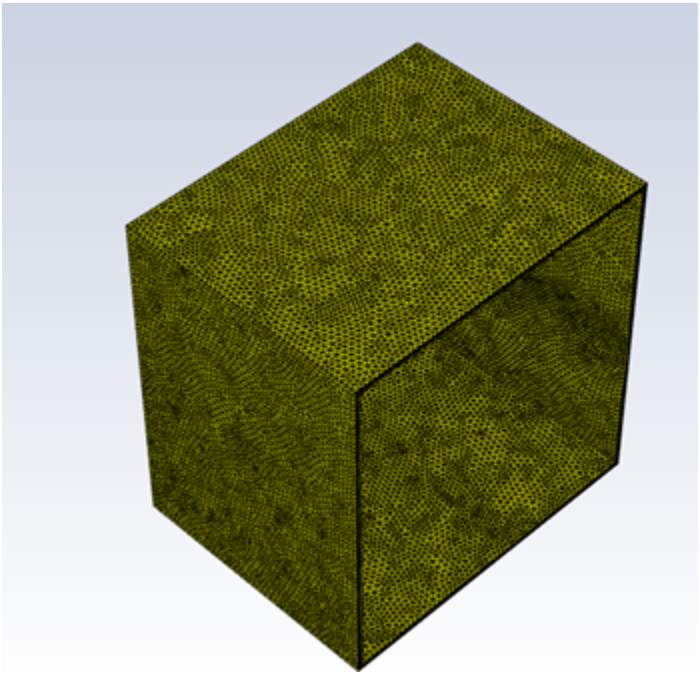
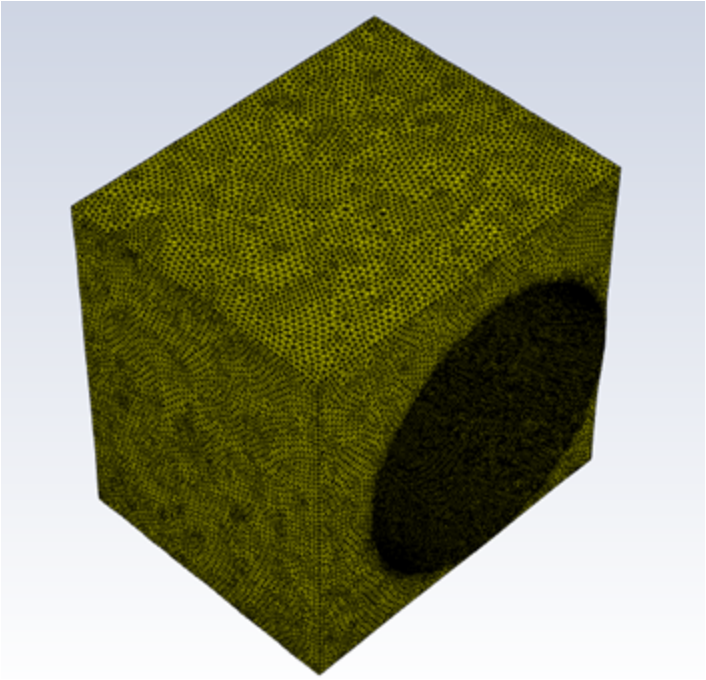


Figure 49.9: Cell Zone containing the outer boundary layer mesh



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 49.10: Cell Zone containing the interior mesh



4. For the Dynamic Meshing settings, select Spring based smoothing as all other smoothing methods do not support Boundary Layer Smoothing (**Deform Adjacent Boundary Layer with Zone**) will not be available.

5. Turn on **Region Face** remeshing.

6. Turn off smoothing for the symmetry/wall regions. This prevents cells adjacent to moving wall from getting too stretched. Smoothing would still be conducted in the interior cells.

7. In the Dynamic Mesh Zones, set the moving prism layer cell zone as a **rigid body:**

   a. By default, Rigid Body Dynamic Mesh Zones require a UDF to specify the mesh motion. Since the motion is provided by System Coupling, assign a “dummy UDF."

   b. The dummy UDF is an empty UDF, as follows:

```
#include "udf.h"


DEFINE_GRID_MOTION(dummy_motion, domain, dt, time, dtime)
{
    return;
}
```

8. For the boundary layer mesh to move with the Coupled region, **Deform Adjacent Boundary Layer with Zone** must be turned on:

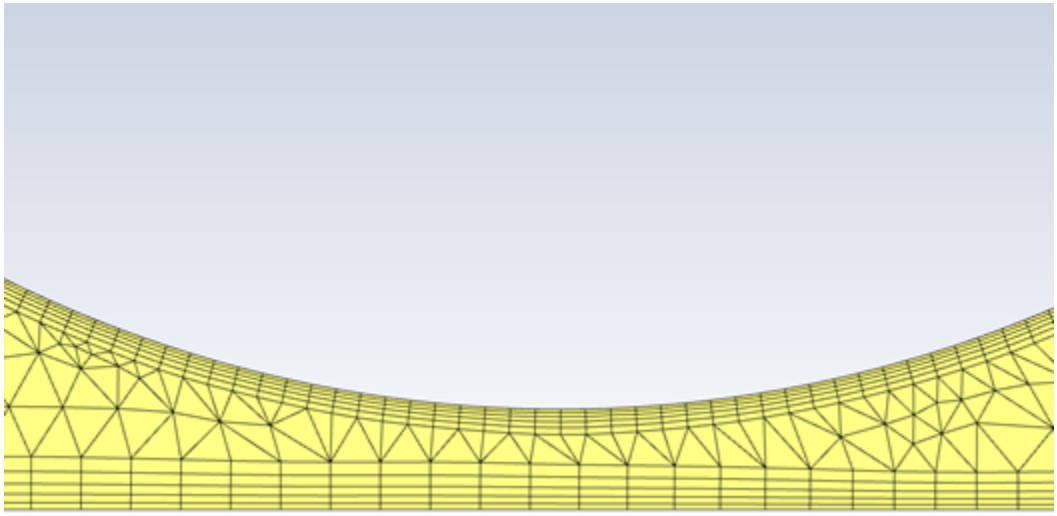
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Setting the **Cell Height** in the **Adjacent Zone** is not required, since the prism layer mesh will not be remeshed.

9. Set the Cell Zone containing the interior mesh as **Deforming**, this will allow the tri/tet faces and cells (Figure 49.11: Prism layer mesh quality maintained for large deformations (p. 4671)) to be remeshed.

Running the case again, we can see how the prism layer mesh is maintained using Boundary Layer Smoothing and only the interior cells are remeshed.

Figure 49.11: Prism layer mesh quality maintained for large deformations

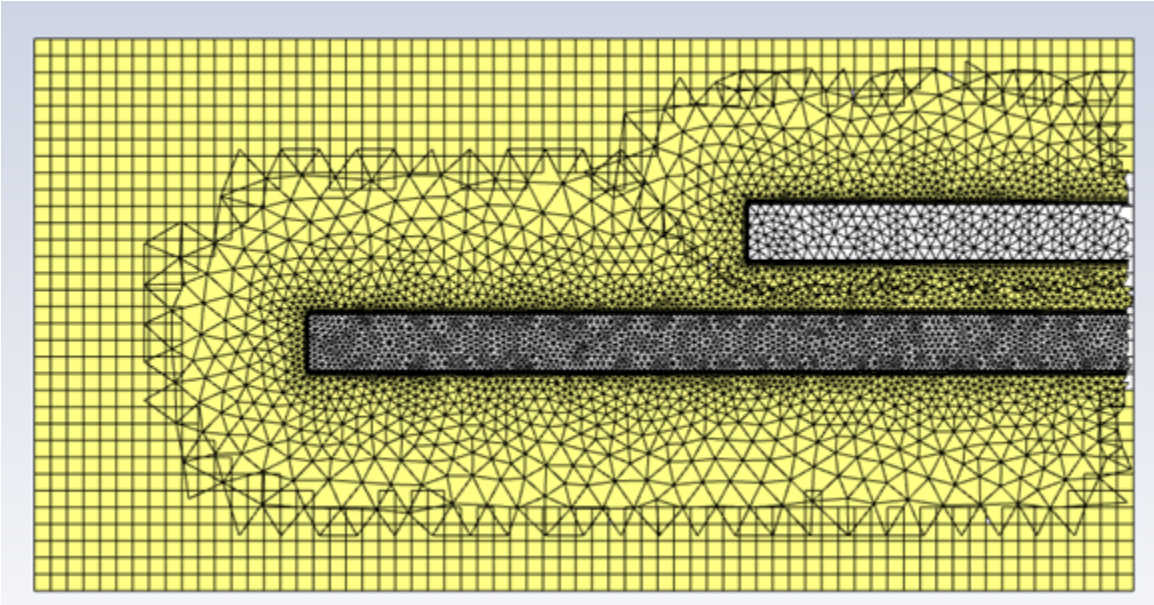


49.6.3.2.2. Overset Meshes

As previously mentioned, for complex topologies and/or if the prism layer is not uniform (i.e., stair-stepping due to small gaps), Ansys recommends the use of overset meshes. By having a separate mesh for the background and the components, the remeshing and smoothing parameters can be controlled for each component mesh. For example, the overset mesh below (Figure 49.12: Overset mesh generated using three separate meshes (p. 4672)) is created using three separate meshes. A structured hexahedral mesh is used for the background, while a tetrahedral mesh is used for the two geometries.

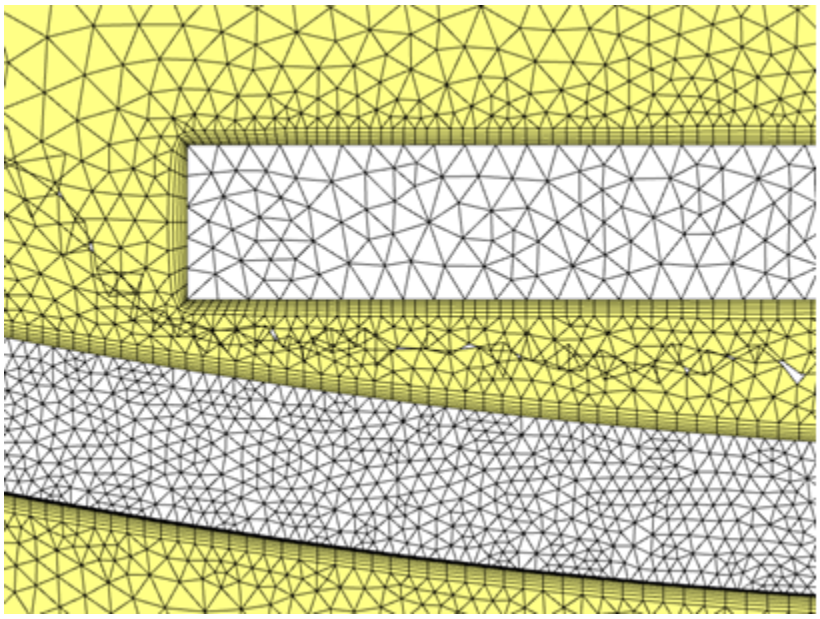
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 49.12: Overset mesh generated using three separate meshes



The leading edge of the long rectangle rotates clockwise, creating a small gap between the short rectangle. If a single mesh was used for this case, the prism layer mesh around the two components deforms and causes the mesh to fold. This occurs as the prism layer mesh starts to absorb the deformation as the gap between the two geometries decreases. By having the components separately meshed, each components mesh absorbs the deformation in its far field, before mapping it back to the overset mesh, maintaining the prism layer mesh quality (Figure 49.13: Prism layer mesh quality is maintained (p. 4672)).

Figure 49.13: Prism layer mesh quality is maintained



The general workflow to setup an overset mesh case is as follows:

1. Split and remesh the required geometries.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Revise the time step size:

   - Consider the three time scales. Note that the smaller timescale is dominant.

     – Simulation

     – Mesh deformation

     – Overset meshing

   - The ideal time step size should be chosen such that the relative mesh motion does not exceed the length of the smallest cell at the overset interface for a given time step. The mesh length used for calculating the time step size should be restricted to cells in the overset overlap region.

   - It is important to not have large variations in mesh resolution in the motion path that could trigger large changes in the overset interface as the motion proceeds. The use of grid priorities can minimize large variations in the overset interface.

3. Choose the Overlap Minimization required:

   - A cell-volume-based method is recommended if the component mesh resolution is fine near walls and increases gradually away from walls and becomes similar in size to or larger than the background mesh.

     Enter the following command into the Fluent TUI:

     ```
     define/overset-interfaces/options/donor-priority-method 0
     ```

   - A boundary-distance-based method is recommended when overlapping meshes have uniform and nearly identical resolutions. Therefore, it is most suitable when two walls have small distance in between them.

     Enter the following command into the Fluent TUI:

     ```
     define/overset-interfaces/options/donor-priority-method 1
     ```

4. Create Overset interface and initialize solution:

   - The overset mesh is created during initialization; it can be viewed through **Results** > **Graphics** > **Mesh** by checking **Overset** under **Options**

Refer to Diagnosing Overset Interface Issues (p. 1216) for more information.

49.6.3.3. Interior Elements have High Skewness or Are Too Large/small

For cases where interior cells are skewed due to translational and/or rotational movement (Figure 49.6: Skewed prism cells due to translational motion between two bodies in close proximity (p. 4667)) or remeshing resulting in elements that are too large or small, there are two main recommendations that depending on the case can resolve this pathology:

1. Use tighter criteria for the **minimum and maximum length scales** as well as **skewness**. In addition, decrease the **Size Remeshing Interval** to 1 so that remeshing based on size is triggered at every time step.

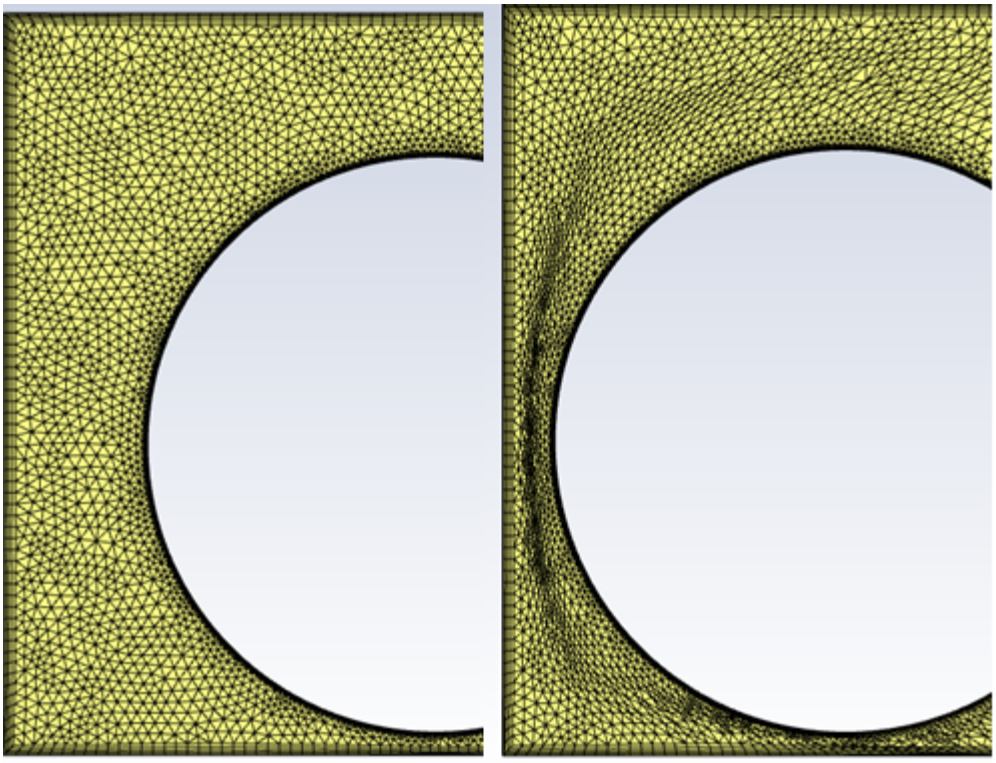
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. If the previous step did not resolve this, then enabling **Region Face Remeshing. Region Face Remeshing** will not trigger if a prism layer mesh is located in the same cell zone were remeshing needs to occur. Thus, you must split the prism layer mesh into a separate cell zone. Refer to the **Deformed Prism Layers** section for detailed instructions.

For the case shown below, setting tighter criteria for the minimum and maximum length scales to remesh every time step was not sufficient to produce a high-quality mesh in the gap (Figure 49.14: Original Mesh (left) and deformed mesh (right) with smoothing and remeshing enabled (p. 4674)). Since there is no direct control over skewness and size criteria of the remeshed cells, you can only indirectly control these criteria of the deformed mesh.

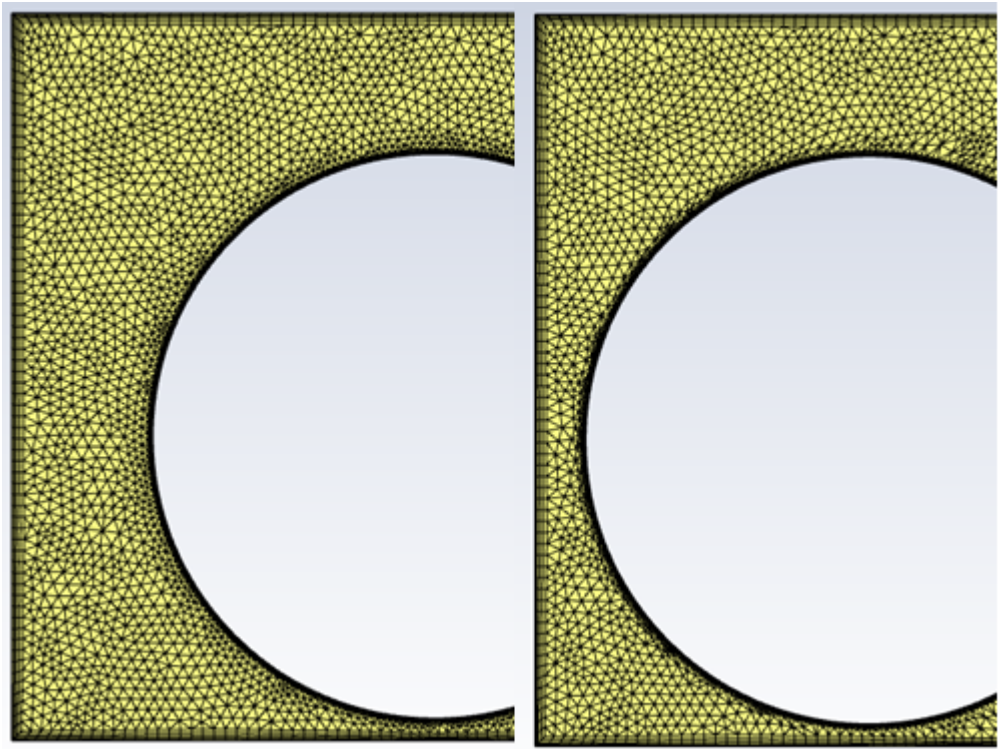
By using **Region Face Remeshing**, this allows more control over the size of the remeshed cells (Figure 49.15: Original Mesh (left) and deformed mesh (right) using smoothing and Region Face Remeshing (p. 4675)). This is done by splitting or deleting the cells directly adjacent to the moving boundary based on the maximum and minimum length scales only. But as previously mentioned, the prism layer mesh has to be in a separate cell zone and treated as a rigid body.

Figure 49.14: Original Mesh (left) and deformed mesh (right) with smoothing and remeshing enabled



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 49.15: Original Mesh (left) and deformed mesh (right) using smoothing and Region Face Remeshing



Note:

- Overset meshes are not supported by CFD-Post. So, post processing must be handled directly in Fluent.
- For overset meshes, volume integrals are taken over all solve cells in the domain. This results in double counting where solve cells overlap, leading to errors in volume integrals on overset meshes.

49.6.3.4. Divergence if Flow Block-Off is Established at the Beginning of a Run

For cases where divergence in Fluent occurs because flow is blocked off at the beginning of the run (for example, a valve starting at closed position with contact). The following list offers recommendations for working with these types of cases:

- Initialize the flow with zero velocity. This also prevents significant movement of the FSI region in the first coupling step, which can fold the mesh.
- Ramping the inlet boundary condition can help with robustness. For details, see Velocity Inlet Boundary Conditions (p. 1411) or Pressure Inlet Boundary Conditions (p. 1401)
- If the fluid is a gas such as air, using the ideal gas model (rather than treating it as incompressible) can aid with solution stability. For details, see Ideal Gas Law for Compressible Flows (p. 1631).

Note that these recommendations have an effect on the solution, as well as convergence behavior.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

49.7. How Fluent's Execution is Affected by System Couplings

Ansys Fluent's execution is modified slightly to allow System Coupling to manage the evolution of the coupled analysis. For more information, see Coupled Analysis Management.

49.8. Restarting Fluent Analyses as Part of System Couplings

Go to the section Stops and Restarts of System Coupling Analyses for general information on restarting a coupled analysis and links to context-specific information.

- For **System Coupling in Workbench**: Restarting a Coupled Analysis in Workbench.
- For **System Coupling's GUI or CLI**: Restarting a Coupled Analysis.

To restart your coupled analysis, you will also need restart information specific to the participants connected to System Coupling.

For details on other participant systems connected to System Coupling, see the following lists of supported systems and references to their corresponding documentation regarding restarts:

- Supported Coupling Participants
- Supported Coupling Participants for System Coupling in Workbench

49.8.1. Generating Fluent Restart Files
49.8.2. Specify a Restart Point in Fluent
49.8.3. Making Setup Changes Before Restarting
49.8.4. Recovering the Fluent Restart Point after a Workbench Crash

49.8.1. Generating Fluent Restart Files

Restarts of a system coupling analysis requires corresponding restart points to exist in the coupling service and in each of the solvers participating in the analysis.

Fluent will automatically generate the corresponding case and data (cas/dat) files based upon requests received from System Coupling. Note that these files are generated in addition to the restart points (or results) manually requested via the Autosave Every field in the Calculation Activities task page within Fluent.

49.8.2. Specify a Restart Point in Fluent

When the Fluent solution is updated (or restarted), the case and data files (`.cas.h5` and `.dat.h5`) corresponding to the current restart point will be used, if it exists. The default restart point is taken from the last generated case and data files. To specify a different restart point, perform the following steps:

1. In the **Project Schematic**, double-click the **Solution** cell to open the Fluent interface. Do not double-click the **Setup** cell, as this will restore the original mesh and settings.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. Select the **File/Solution Files...** ribbon tab item to open the **Solution Files** dialog box with a list of all available restart points.

3. Select the desired restart point. By default, the latest saved restart point is selected. Only one restart point can be selected, so to be able to select a different restart point, first click the highlighted restart point to deselect it.

   Make sure that you select the restart point that corresponds to the restart points selected for the System Coupling service and the other coupling participants.

4. Click **Read** to read-in the data for the selected restart point.

5. Once the data is read, click **Close** to close the **Solution Files** dialog box.

6. Close Fluent by selecting the **File/Close Fluent** ribbon tab item.

49.8.3. Making Setup Changes Before Restarting

In some cases, Fluent setup changes are required to avoid failure of the coupled analysis (for example, changes to iteration controls, under-relaxation factors, and so on). You may make these changes either by modifying settings in Fluent or (if working in System Coupling's GUI or CLI) by using a custom restart file.

Modifying Settings in Fluent

Open Fluent and make the required changes to the setup. The settings that are stored in the case file corresponding to the current restart point will be used for Fluent restarts.

Close Fluent by selecting the **File/Close Fluent** ribbon tab item. If you are presented with a notification that the modified mesh and settings will be used for the current run, choose **OK** to accept and close the dialog box.

Using a Custom Restart File

Custom restart files are controlled by settings defined under Fluent's Execution Control | Fluent Input object. By editing these files, you can make changes to Fluent's setup before restarting the solution.

For more information, see Using Custom Input Files in System Coupling's User Interfaces (p. 4652).

49.8.4. Recovering the Fluent Restart Point after a Workbench Crash

Workbench or one of the components may crash such that restart files are available but they are not recognized or populated in the Workbench project. If this is the case, you can recover your project and restart your analysis.

See Restarting a Coupled Analysis in Workbench for the steps needed to recover a coupled analysis after a Workbench crash. You will also need the information below about Fluent, as well as information specific to the other participant systems connected to System Coupling. For other participant systems connected to your **System Coupling** system, see Supported Coupling Participants for System Coupling in Workbench for a list of supported systems and references to their corresponding documentation regarding restarts.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The usual project directory (*ProjectName*_files) contains the latest Fluent and System Coupling results and restart points (these solvers use the live project instead of running in a temporary directory).

Note that the .backup directory contains the original version of any files that have been modified since the last save. These files are useful to recover the last saved state, but they are not useful for restarting your analysis.

To recover Fluent's restart point after a Workbench crash:

1. Double-click the Fluent **Solution** cell to open Fluent interface.
2. Select the **File/Solution Files...** ribbon tab item to open the **Solution Files** dialog box with a list of the restart points.
3. The **Solution Files** dialog box will not be populated with the data files written before Workbench crashed, so the correct restart point must be recovered. Select **Recover Missing Solution...** to recover the latest case and data files, and to populate the corresponding restart points in the **Solution Files at** list.
4. Now that the solution files list is populated, select the desired restart point.

   By default, the latest saved restart point is selected. Only one restart point can be selected, so to be able to select a different restart point, first click the highlighted restart point to deselect it.

   Make sure that you select the restart point that corresponds to the restart points selected for System Coupling and the other coupling participants.
5. Click **Read** to read-in the data for the selected restart point.
6. Once the data is read, click **Close** to close the **Solution Files** dialog box.
7. Close Fluent by selecting the **File/Close Fluent** ribbon tab item.

49.9. System Coupling case with Fluent using Patched Data

When generated data is cleared from a System Coupling case before a rerun, the data that was specified using Fluent's patch option is also cleared. For example, in the Multiphase volume of fluid (VOF) cases, the patched data that is cleared is the volume fraction information. Fluent needs this patched data to be able to solve accurately. See Patching Values in Selected Cells (p. 3630) for more information about patched data.

For a System Coupling case using patched data, follow the steps below. Note that this workaround will not work with Design Point updates.

1. Before setting up your case in Fluent, tell Fluent to generate the *.ip file so that if needed, you will have access to patched data. To do this:

   a. In Workbench, select **Tools**>**Options**.

   b. On the left side of the dialog box that opens, select **Fluent**, and then check **Enable Generation of Interpolation File**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2. During Fluent setup, an `*.ip` file will now be written after initialization of your Fluent system.

3. If you want to do a rerun in Workbench, after completing a System Coupling run, you need to **Clear Generated Data** on the System Coupling system's **Solution** cell. For the cases that use patched data, you need to then have Fluent read the `*.ip` file that was generated in your initial setup so that it has the correct data. To do this:

   a. From the **Project Schematic**, open Fluent by double-clicking Fluent's **Setup** cell.

   b. Select the **File/Interpolate...** ribbon tab item. In the dialog box that appears, select **Read and Interpolate**, then click **Read...** .

   c. Browse to the `*.ip` file, and then click **OK** to add have Fluent read this file. In the **Interpolate Data** dialog box, select **Close**.

   d. You do not need to initialize Fluent after reading the `*.ip` file. Close Fluent and continue with the rerun of your System Coupling case.

49.10. Running Fluent as a Participant from System Coupling's GUI or CLI

System Coupling analyses can be run using either of System Coupling's user interfaces. To run Ansys Fluent as a coupling participant:

For coupled analyses set up in a System Coupling interface:

- To set up a coupled analysis in System Coupling's GUI or CLI,, perform the steps outlined in Preparing for a Coupled Analysis, Creating a Coupled Analysis, and Modifying Coupled Analysis Settings.
- To run the coupled analysis in System Coupling's GUI or CLI, perform the steps outlined in Running a Coupled Analysis.

For coupled analyses set up using System Coupling in Workbench:

System Coupling analyses that are set up in Workbench can be exported and executed in one of System Coupling's user interfaces.

- To set up a coupled analysis in Workbench, follow the steps outlined in Setting Up a Coupled Analysis in Workbench in the *System Coupling User's Guide*.
- To export the coupled analysis setup, follow the steps outlined in Advanced Tasks for System Coupling in Workbench in the *System Coupling User's Guide*.
- To execute the exported setup in System Coupling's GUI or CLI, follow the steps outlined in Running an Exported System Coupling Setup in the *System Coupling User's Guide*.

For information on how to restart an analysis set up and run from one of System Coupling's user interfaces, see Restarting a Coupled Analysis.

For details on alternate workflows and capabilities, see Common Coupled Analysis Tasks and Running an Exported System Coupling Setup.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

49.11. Troubleshooting Two-Way Coupled Analysis Problems

The following files, found in the Fluent run directory (FFF/Fluent under a Workbench design point directory), may prove useful in troubleshooting coupled analysis problems:

- `cortexerror.log`: This file contains a historical summary of all of the errors that have occurred during all runs executed in the same run directory. With this in mind, ensure that you are considering messages corresponding to the most recent run. An example message is: "Update-Dynamic-Mesh failed. Negative cell volume detected." This indicates that at the date and time noted, the mesh unexpectedly failed.
- `solution 1.log`: This file includes information regarding the Fluent boundary conditions at which data from System Couplings was/is used. This file is never appended, and so it only contains information from the current run.
- `solution.trn` (or other transcript file): This file contains a complete summary of the current/latest run's evolution. This is one of the most useful files to determine why the coupled analysis failed. To generate extensive debug output during the analysis, enter the following command when completing the Fluent problem setup:

  ```
  (rpsetvar 'sc/verbosity 1)
  ```

Provide all of these files when submitting a request for service to ANSYS personnel.

49.12. Product Licensing Considerations when using System Coupling

A distinct license is required for each coupling participant product, but no additional licenses are required for the System Coupling infrastructure itself. For more information on applicable licenses, see Coupled Analysis Licensing Requirements in the *System Coupling User's Guide*.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 50: Customizing Fluent

You can use Ansys ACT to create simulation wizards that can be run from either stand-alone Fluent or Fluent in Workbench. For general ACT usage information, see the *ACT Developer's Guide*. For information on ACT usage specific to Fluent, see *ACT Customization Guide for Fluent*.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 51: Task Page Reference Guide

This reference guide provides information about the task pages in Fluent.

51.1. Meshing Task Page
51.2. Setup Task Page
51.3. General Task Page
51.4. Models Task Page
51.5. Materials Task Page
51.6. Cell Zone Conditions Task Page
51.7. Boundary Conditions Task Page
51.8. Overset Interfaces Task Page
51.9. Dynamic Mesh Task Page
51.10. Reference Values Task Page
51.11. Solution Task Page
51.12. Solution Methods Task Page
51.13. Solution Controls Task Page
51.14. Solution Initialization Task Page
51.15. Calculation Activities Task Page
51.16. Run Calculation Task Page
51.17. Results Task Page
51.18. Graphics and Animations Task Page
51.19. Plots Task Page
51.20. Reports Task Page
51.21. Parameters and Customization Task Page

51.1. Meshing Task Page

The **Meshing** task page introduces you to the most common meshing tasks involved in solving your CFD simulation using Ansys Fluent.

51.2. Setup Task Page

The **Setup** task page introduces you to the main tasks involved in setting up your CFD simulation using Ansys Fluent.

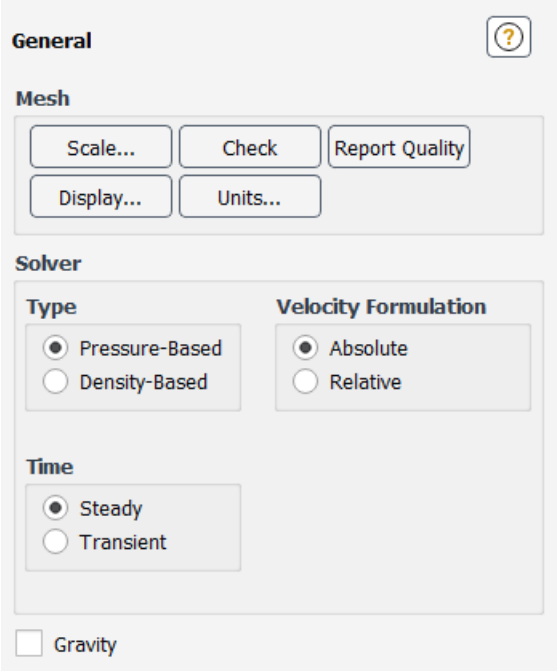
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

51.3. General Task Page

The **General** task page allows you to set various generic problem settings, such as those related to the mesh or the solver. Note that these settings become available only after you read in a case or mesh file. See Reading Mesh Files (p. 953) and Reading and Writing Case Files (p. 956) for details.

Important:

This feature offers reduced functionality when running Fluent under the **Pro** capability level.



Controls

Mesh

contains controls relating to mesh settings.

Scale...

opens the Scale Mesh Dialog Box (p. 4686).

Check

verifies the validity of the mesh. The check provides volume statistics, mesh topology and periodic boundary information, verification of simplex counters, and verification of node position with reference to the x axis for axisymmetric cases. See Checking the Mesh (p. 1245) for details.

It is recommended that you check the mesh right after reading it into the solver, in order to detect any mesh trouble before you get started with the problem setup.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Report Quality

displays various quantities related to the quality of the mesh in the console, such as `Minimum Orthogonal Quality` and `Maximum Aspect Ratio`. See Mesh Quality (p. 1146) for details.

Display...

opens the Mesh Display Dialog Box (p. 4688).

Solver

contains controls relating to solver settings.

Type

contains the solution methods available for computing a solution for your model. See Using the Solver (p. 3571) for details.

Pressure-Based

enables the pressure-based Navier-Stokes solution algorithm (the default).

Density-Based

enables the density-based Navier-Stokes coupled solution algorithm.

Velocity Formulation

specifies the velocity formulation to be used in the calculation. See Choosing the Relative or Absolute Velocity Formulation (p. 1775) for details.

Absolute

enables the use of the absolute velocity formulation. This is the default setting.

Relative

enables the use of the relative velocity formulation. This option is available only with the **Pressure-Based** solver.

Time

contains options related to time dependence.

Steady

specifies that a steady flow is being solved.

Transient

enables a time-dependent solution. See Performing Time-Dependent Calculations (p. 3643) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

2D Space

contains options available only when solving two-dimensional problems.

Planar

indicates that the problem is two-dimensional. (This option is available only when you start the 2D version of the solver.)

Axisymmetric

indicates that the domain is axisymmetric about the x axis. When **Axisymmetric** is enabled, the 2D axisymmetric form of the governing equations is solved instead of the 2D Cartesian form. (This option is available only when you start the 2D version of the solver.) Be sure to change the zone type of the axis of rotation to **axis**, using the Boundary Conditions Task Page (p. 4991), as described in Changing Cell and Boundary Zone Types (p. 1301).

Axisymmetric Swirl

specifies that the swirl component (circumferential component) of velocity is to be included in your axisymmetric model. You should select this option if you are solving swirling flow in an axisymmetric geometry (see Swirling and Rotating Flows (p. 1753) for more information).

Gravity

enables the specification of gravity.

Gravitational Acceleration

sets the x, y, and z components of the gravitational acceleration vector. (The z component is available only in 3D solvers.) See Buoyancy-Driven Flows and Natural Convection (p. 1387) for details about buoyancy-driven flows. This option appears only when **Gravity** is enabled.

Units...

opens the Set Units Dialog Box (p. 4693).

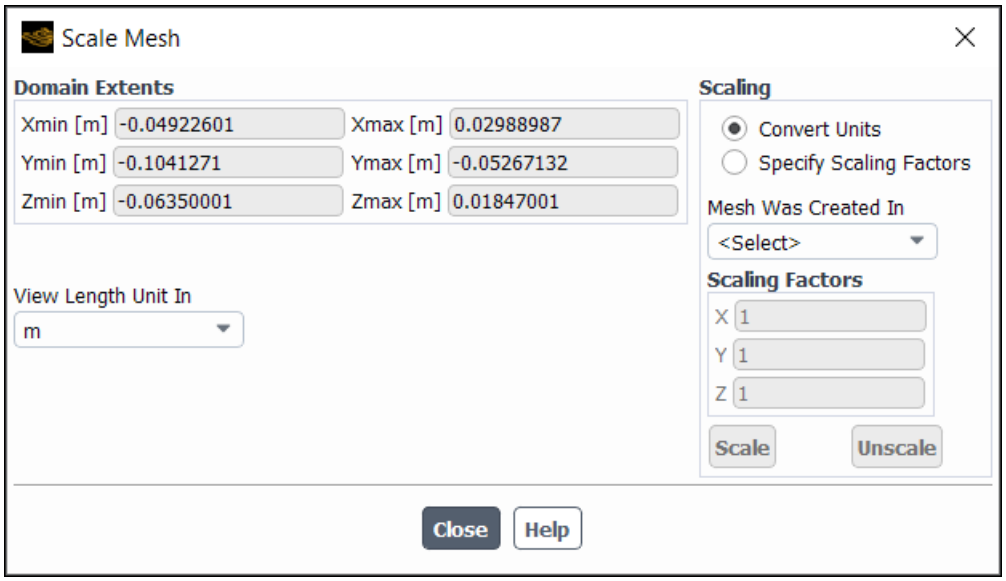
For additional information, see the following sections:

51.3.1. Scale Mesh Dialog Box
51.3.2. Mesh Display Dialog Box
51.3.3. Set Units Dialog Box
51.3.4. Define Unit Dialog Box
51.3.5. Mesh Colors Dialog Box

51.3.1. Scale Mesh Dialog Box

The **Scale Mesh** dialog box allows you to convert the mesh from various units of measurement to SI or to apply custom scale factors to the individual coordinates of the mesh. See Scaling the Mesh (p. 1285) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Domain Extents

displays the Cartesian coordinate extremes of the nodes in the mesh. (These values are not editable; they are purely informational.)

Xmin, Ymin, Zmin

shows the minimum values of Cartesian coordinates in the mesh.

Xmax, Ymax, Zmax

shows the maximum values of Cartesian coordinates in the mesh.

Scaling

contains controls for converting units and setting the scale factors automatically.

Convert Units

allows you to use the conversion factors provided by Ansys Fluent. Then indicate the units used when creating the mesh by selecting the appropriate abbreviation for meters, centimeters, millimeters, inches, or feet from the **Mesh Was Created In** drop-down list. The **Scaling Factors** will automatically be set to the correct values (for example, 0.0254 meters/inch).

Specify Scaling Factors

allows you to manually specify a scale factor in each of the Cartesian coordinate directions.

Mesh Was Created In

contains a list of common units of length. The **Scaling Factors** will automatically be set based on your selection. The units include common SI and British units such as centimeters (cm), millimeters (mm), inches (in), and feet (ft).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Scaling Factors

contains the factors applied to the mesh in each of the Cartesian coordinate directions. You can enter values manually, or use the **Mesh Was Created In** list to set scale factors automatically.

X

is the scale factor in the x direction.

Y

is the scale factor in the y direction.

Z

is the scale factor in the z direction (appears only in 3D).

Scale

multiplies each of the node coordinates by the specified scale factors.

Unscale

divides each of the node coordinates by the specified scale factors.

View Length Unit In

contains a list of common units of length. The **Domain Extents** will automatically be set based on your selection. The units include common SI and British units such as centimeters (cm), millimeters (mm), inches (in), and feet (ft). The units of length in the Set Units Dialog Box (p. 4693) will change each time you change your selection in the **View Length Unit In** drop-down list.

51.3.2. Mesh Display Dialog Box

The **Mesh Display** dialog box controls the display of zone, surface, and partition boundary meshes. See Displaying the Mesh (p. 3872) for details about the items below.

*You can also access the **Mesh Display** dialog box through the **Results** ribbon tab.*

Results → **Graphics** → **Mesh** → **New...**

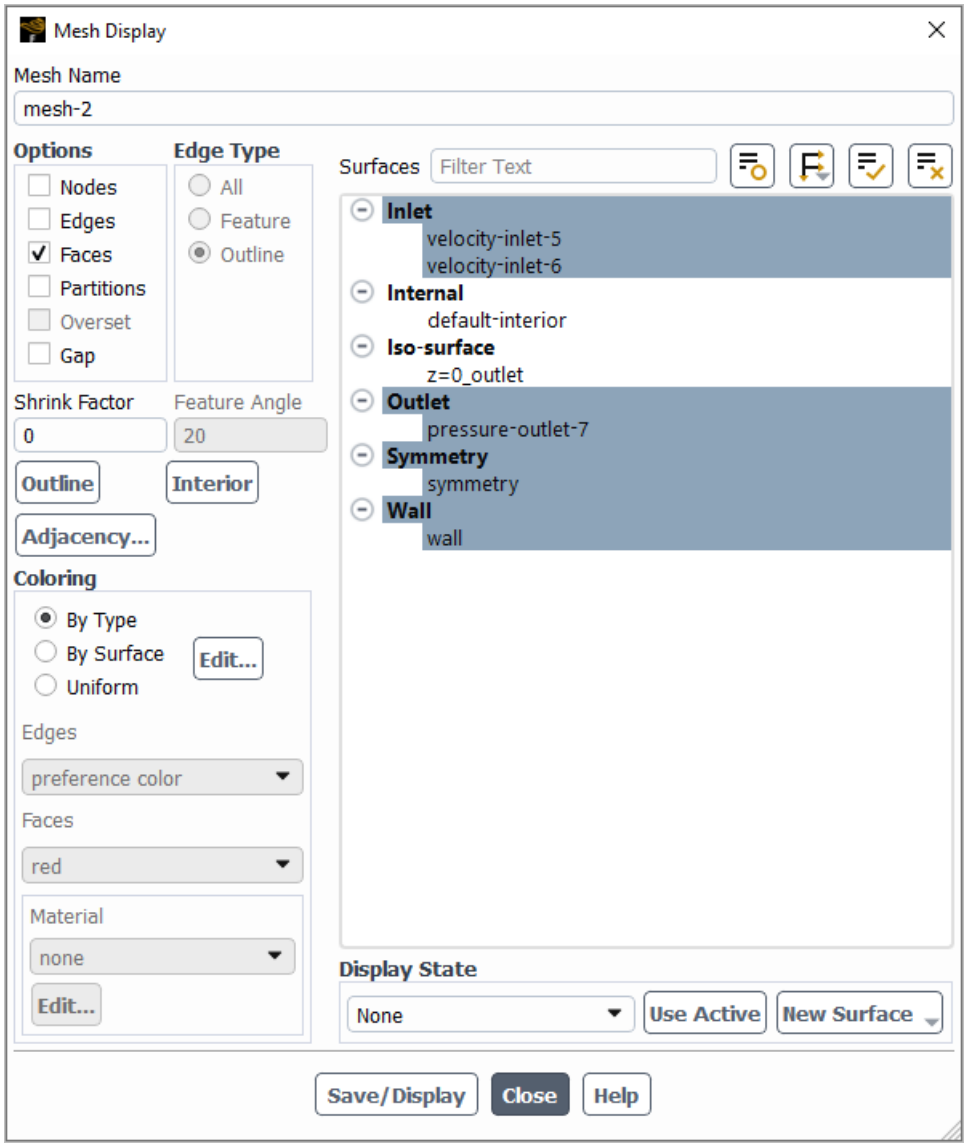
Note:

If you click the mesh display toolbar button ( ), it opens the "global"/non-persistent version of the **Mesh Display** dialog box. This version of the dialog box does not have a **Name** field and it is not saved with the case file.

You are encouraged to use the persistent version that is accessed by clicking **New...** in the **Results** ribbon tab if you anticipate wanting to display the same areas of the mesh in a

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

future Fluent session or if you want to reuse the mesh display for other purposes, such as displaying a scene (Generating a Scene (p. 3935)).



Controls

Mesh Name

is the name for a mesh display definition. You can specify a name or use the default name **mesh-*id*** . This control appears only for mesh display definitions.

Options

contains the rendering options described below. To see the effects of your selection you must click the **Display** button.

Nodes

enables the display of nodes on the selected surfaces.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Edges

enables the display of mesh edges on the selected surfaces.

Faces

enables the display of mesh faces (filled meshes) on the selected surfaces.

Partitions

enables the display of partition boundaries.

Overset

when enabled, displays only the solve cells for cases with overset interfaces. Receptor cells are also displayed if the text command `define/overset-interfaces/options/render-receptor-cells?` is set to `yes`.

Gap

enables the display of only the cells in which flow is allowed by the gap model (that is, those that are not marked for blockage in the gap region). This option is only available when you have created a gap definition, which can be done by using the **Gap Model** dialog box. For details about the gap model, see Controlling Flow in Narrow Gaps for Valves and Pumps (p. 1234).

Shrink Factor

specifies the amount to shrink faces and cells. See Shrinking Faces and Cells in the Display (p. 3894) for details.

Edge Type

controls the display of edges. (These items will not appear if the **Edges** option is turned off.)

All

enables the display of all mesh edges.

Feature

enables feature lines in an outline display. See Adding Features to an Outline Display (p. 3893) for details.

Outline

enables the display of the mesh outline.

Feature Angle

controls the amount of detail added to a feature outline display. See Adding Features to an Outline Display (p. 3893) for details. (This item will be available only if the **Feature** edge type is enabled.)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Transparency

(global mesh display only, accessed by clicking the mesh display toolbar button) controls the transparency of the mesh display.

Surfaces

contains a list from which you can select the surfaces for which the mesh is to be drawn.

New Surface

is a drop-down list button that contains a list of surface options:

Point

opens the Point Surface Dialog Box (p. 5582).

Line/Rake

opens the Line/Rake Surface Dialog Box (p. 5514).

Plane

opens the Plane Surface Dialog Box (p. 5580).

Quadric

opens the Quadric Surface Dialog Box (p. 5584).

Iso-Surface

opens the Iso-Surface Dialog Box (p. 5501).

Iso-Clip

opens the Iso-Clip Dialog Box (p. 5500).

Structural Point

opens the Structural Point Surface Dialog Box (p. 5623).

Outline

selects all "outline" boundaries in the **Surfaces** list. If all outline boundaries are already selected, it deselects them.

Interior

selects all "interior" surfaces in the **Surfaces** list. If all interior surfaces are already selected, it deselects them.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Adjacency

opens the Adjacency Dialog Box (p. 5390) from which you can identify and display face zones that are adjacent to selected cell zones.

Coloring

By Type

nodes, edges, and faces are automatically colored by type. The colors for each type are determined in the Zone Type Color and Material Assignment Dialog Box (p. 5664).

Edit...

opens Zone Type Color and Material Assignment Dialog Box (p. 5664) if **By Type** is selected or Surface Rendering Properties Dialog Box (p. 5625) if **By Surface** is selected.

By Surface

nodes, edges, and faces are colored by zone ID by default. However, you can assign colors and/or materials to surfaces so they are rendered consistently anytime the **By Surface** option is selected within a **Mesh Display** graphics object. Click **Edit...** when **By Surface** is selected to open the Surface Rendering Properties Dialog Box (p. 5625) and make these assignments.

*Note that if you do not like the default colors assigned with this method, you can use **Ctrl**+**Shift**+**O** to randomize the mesh colors. Ensure the graphics window is in focus by clicking in it before using the hotkey combination (otherwise it won't work).*

Uniform

allows you to manually specify the color of edges, and faces, if enabled under **Options**.

Edges

specify the color for the edges.

Faces

specify the color for the faces.

Material

specify the material type for the selected surfaces.

Edit...

opens the Material Editor Dialog Box (p. 5521). *Refer to Creating Custom Realistic Materials (p. 3889) for additional information on using the **Material Editor** dialog box.*

Colors...

(available only for the "global"/non-persistent dialog box access by clicking the mesh display icon) opens the Mesh Colors Dialog Box (p. 4695).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Display

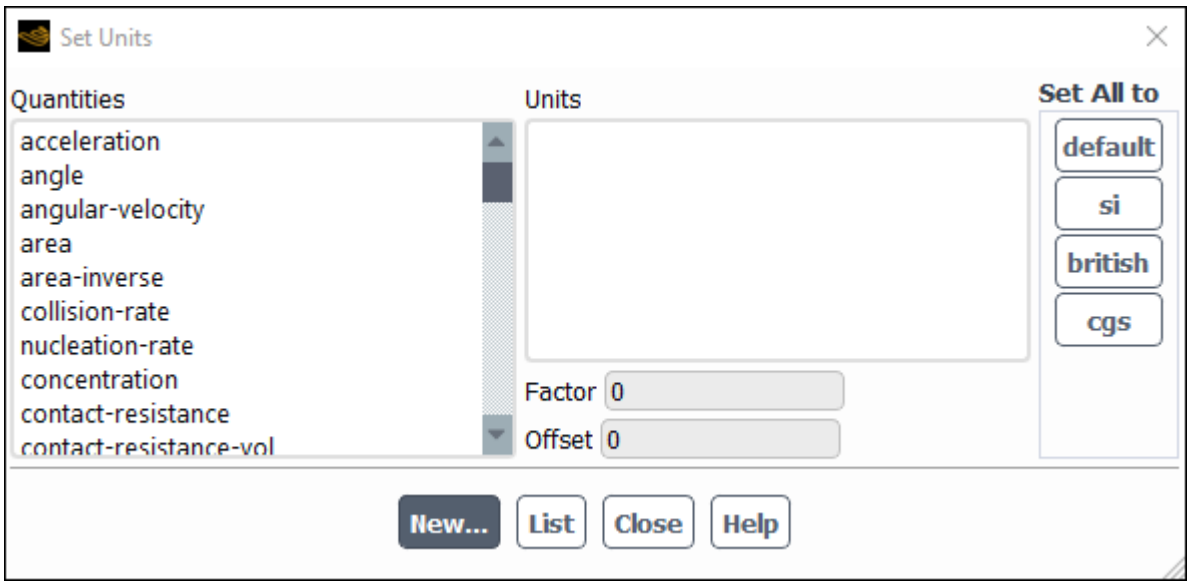
(available only for the "global"/non-persistent dialog box access by clicking the mesh display icon) displays the defined mesh plot.

Save/Display

displays the mesh in the active graphics window and saves the mesh display definition. This button appears only for mesh display definitions and replaces the **Display** button.

51.3.3. Set Units Dialog Box

The **Set Units** dialog box allows you to set the units system for any quantity used in Ansys Fluent. All quantities may be set to a standard system, such as SI or British, or the units of individual quantities may be set. See Unit Systems (p. 1033) for details about the items below.



Controls

Quantities

displays the list of all quantities used by Ansys Fluent for input and output.

Units

lists the units appropriate for the currently selected quantity. Selecting an item in the **Units** list causes that unit to be used for the currently selected quantity.

Factor

displays the conversion factor from the currently selected units to SI.

Offset

displays the conversion offset from the currently selected units to SI.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Set All to

contains standard sets of units that are applied to all quantities when selected.

default

is similar to **si**, but uses degrees instead of radians for angles.

si

selects the System International (SI) standard for all units.

british

selects the English Engineering standard for all units.

cgs

selects the CGS (centimeter-gram-second) standard for all units

New...

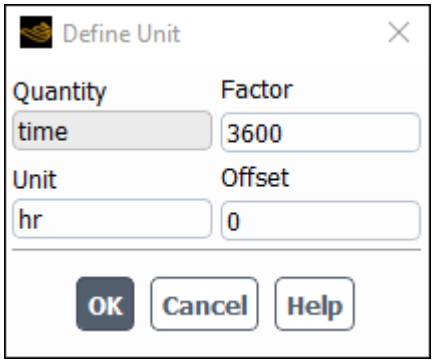
opens the Define Unit Dialog Box (p. 4694), in which you can specify a customized unit for a particular quantity.

List

displays the current units for all quantities in the console.

51.3.4. Define Unit Dialog Box

The **Define Unit** dialog box allows you to customize the units for a particular quantity. It is opened from the Set Units Dialog Box (p. 4693). For details about using this dialog box, see Defining a New Unit (p. 1035).



Controls

Quantity

shows the name of the quantity for which you are defining a new unit. (You cannot edit this field; the quantity is selected in the Set Units Dialog Box (p. 4693).)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Unit

sets the name for the new unit.

Factor

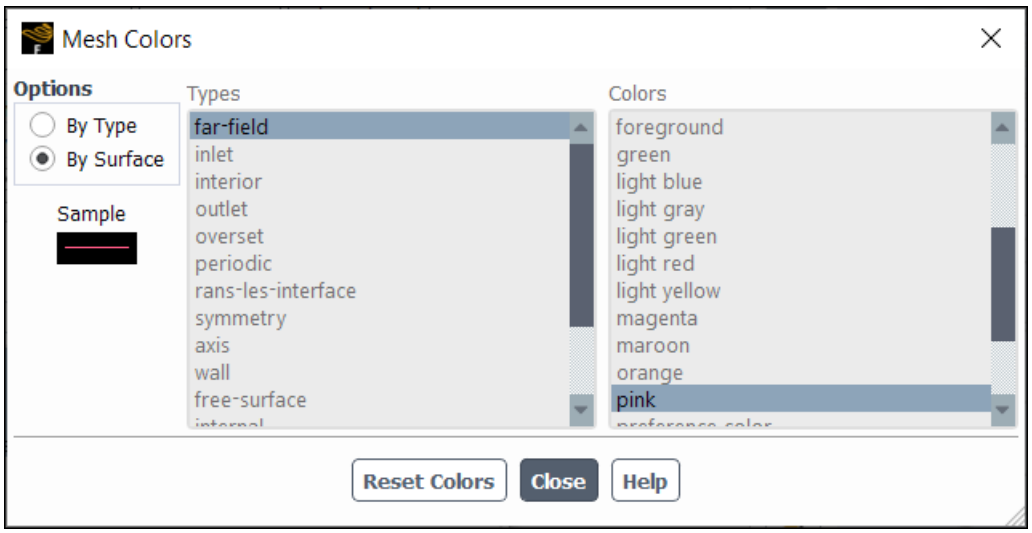
sets the conversion factor from the currently selected units to SI.

Offset

sets the conversion offset from the currently selected units to SI.

51.3.5. Mesh Colors Dialog Box

The **Mesh Colors** dialog box allows you to control the colors that are used to draw meshes. It is opened from the "global"/non-persistent version of the Mesh Display Dialog Box (p. 4688) accessed by clicking the mesh display icon . See Modifying the Mesh Colors (p. 3877) for details about the items below.



Controls

Options

contains options to select the method by which to set the colors.

Color by Type

sets the color based on the type of zone.

Color by Surface

sets the color by the zone ID. Note that if you do not like the colors assigned with this method, you can use **Ctrl**+**Shift**+**O** to randomize the mesh colors. Ensure the graphics window is in focus by clicking in it before using the hotkey combination (otherwise it won't work).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Types

contains a selectable list of zone types. You can select the zone type for which you want to set the color.

Colors

contains a list from which you can select a color for the selected type.

Sample

displays a sample of the currently selected color.

Reset Colors

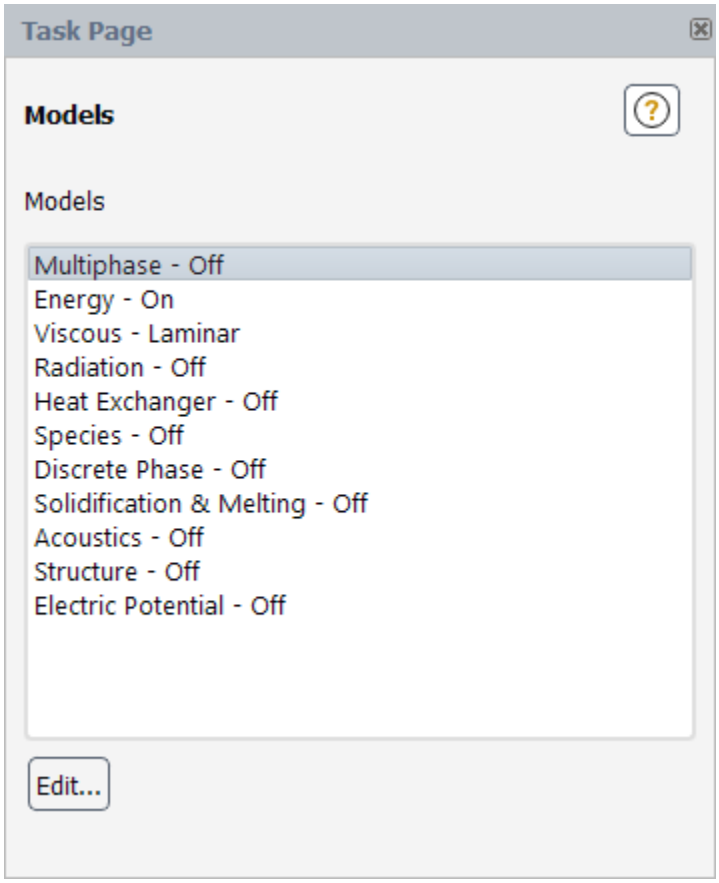
resets the mesh colors to the default selections.

51.4. Models Task Page

The **Models** task page allows you to set various generic model settings.

Important:

This feature offers reduced functionality when running Fluent under the **Pro** capability level.



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Controls

Models

contains a listing of the various models available in Ansys Fluent.

You can double-click an item in the **Models** list to open the corresponding dialog box, or you can select the item in the list and click the **Edit...** button.

Multiphase

- selecting this item and clicking the **Edit...** button opens the Multiphase Model Dialog Box (p. 4700).

Energy

- selecting this item and clicking the **Edit...** button opens the Energy Dialog Box (p. 4714).

Viscous

- selecting this item and clicking the **Edit...** button opens the Viscous Model Dialog Box (p. 4715).

Radiation

- selecting this item and clicking the **Edit...** button opens the Radiation Model Dialog Box (p. 4736).

Heat Exchanger

- selecting this item and clicking the **Edit...** button opens the Heat Exchanger Model Dialog Box (p. 4748).

Species

- selecting this item and clicking the **Edit...** button opens the Species Model Dialog Box (p. 4768).

The following models are made available, depending on your setup of the **Species** dialog box.

- **Spark Ignition** - selecting this item and clicking the **Edit...** button opens the Spark Ignition Dialog Box (p. 4809). Note that spark ignition is only available for transient calculations.
- **Autoignition** - selecting this item and clicking the **Edit...** button opens the Autoignition Model Dialog Box (p. 4811). Note that autoignition is only available for transient calculations.
- **Inert** - selecting this item and clicking the **Edit...** button opens the Inert Dialog Box (p. 4814). Note that the inert model is only available when the non-premixed or partially premixed model is selected in the **Species Model** dialog box, or when a PDF file is read.
- **NOx** - selecting this item and clicking the **Edit...** button opens the NOx Model Dialog Box (p. 4816). Note that the NOx model is not compatible with premixed combustion.
- **Soot** - selecting this item and clicking the **Edit...** button opens the Soot Model Dialog Box (p. 4825). Note that none of the soot models are compatible with premixed combustion.
- **Decoupled Detailed Chemistry** - selecting this item and clicking the **Edit...** button opens the Decoupled Detailed Chemistry Dialog Box (p. 4840).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Reacting Channel Model** - selecting this item and clicking the **Edit...** button opens the Reacting Channel Model Dialog Box (p. 4841).

Discrete Phase

- selecting this item and clicking the **Edit...** button opens the Discrete Phase Model Dialog Box (p. 4845).

Solidification & Melting

- selecting this item and clicking the **Edit...** button opens the Solidification and Melting Dialog Box (p. 4858).

Acoustics

- selecting this item and clicking the **Edit...** button opens the Acoustics Model Dialog Box (p. 4860).

Structure

- selecting this item and clicking the **Edit...** button opens the Structural Model Dialog Box (p. 4869).

Eulerian Wall Film

- selecting this item and clicking the **Edit...** button opens the Eulerian Wall Film Dialog Box (p. 4870).

Electric Potential

- selecting this item and clicking the **Edit...** button opens the Potential/Electrochemistry Dialog Box (p. 4881).

Edit...

opens the dialog box corresponding to the selected item in the **Models** list.

For additional information, see the following sections:

51.4.1. Multiphase Model Dialog Box
51.4.2. Energy Dialog Box
51.4.3. Viscous Model Dialog Box
51.4.4. Radiation Model Dialog Box
51.4.5. View Factors and Clustering Dialog Box
51.4.6. Participating Boundary Zones Dialog Box
51.4.7. Solar Calculator Dialog Box
51.4.8. Heat Exchanger Model Dialog Box
51.4.9. Dual Cell Heat Exchanger Dialog Box
51.4.10. Set Dual Cell Heat Exchanger Dialog Box
51.4.11. Heat Transfer Data Table Dialog Box
51.4.12. NTU Table Dialog Box
51.4.13. Copy From Dialog Box
51.4.14. Ungrouped Macro Heat Exchanger Dialog Box

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

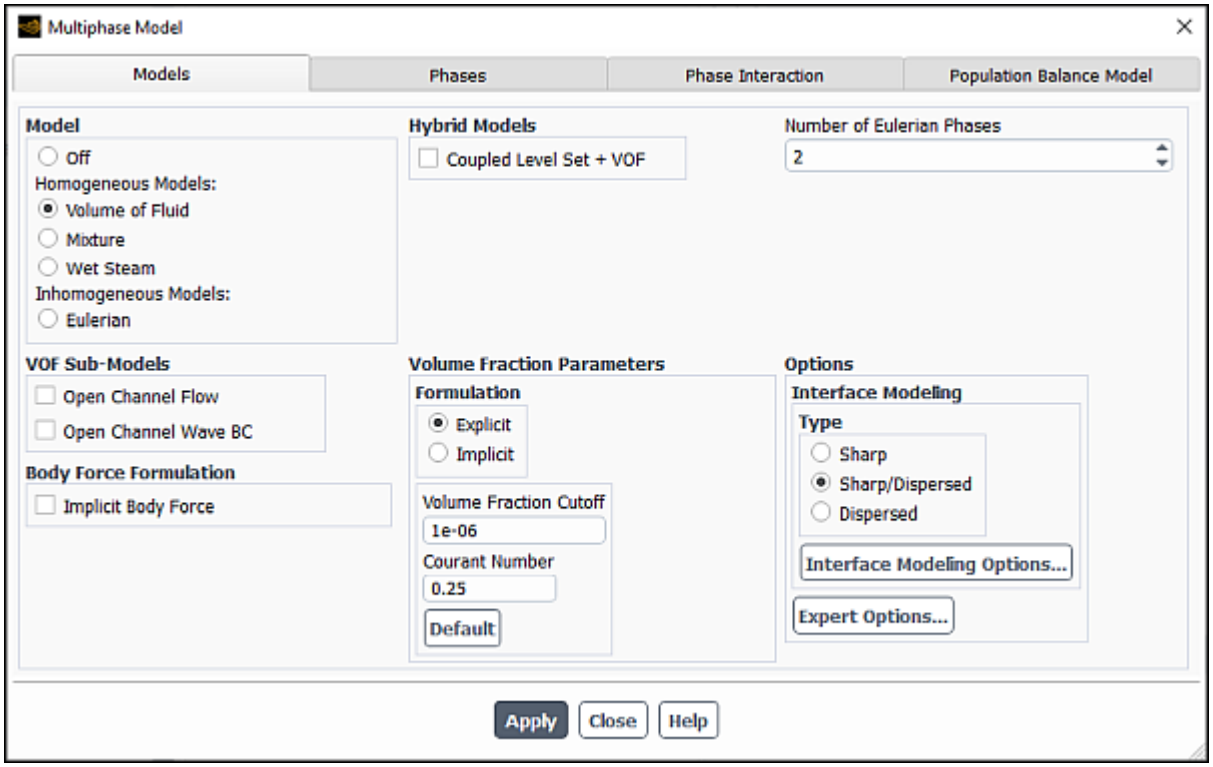
51.4.15. Velocity Effectiveness Curve Dialog Box
51.4.16. Core Porosity Model Dialog Box
51.4.17. Macro Heat Exchanger Group Dialog Box
51.4.18. Species Model Dialog Box
51.4.19. Coal Calculator Dialog Box
51.4.20. Integration Parameters Dialog Box
51.4.21. Flamelet 3D Surfaces Dialog Box
51.4.22. Flamelet 2D Curves Dialog Box
51.4.23. Unsteady Flamelet Parameters Dialog Box
51.4.24. Flamelet Fluid Zones Dialog Box
51.4.25. Select Transported Scalars Dialog Box
51.4.26. Distribution of Points Dialog Box
51.4.27. PDF Table Dialog Box
51.4.28. Spark Ignition Dialog Box
51.4.29. Set Spark Ignition Dialog Box
51.4.30. Autoignition Model Dialog Box
51.4.31. Inert Dialog Box
51.4.32. NOx Model Dialog Box
51.4.33. Soot Model Dialog Box
51.4.34. Sticking Coefficients Dialog Box
51.4.35. Mechanism Dialog Box
51.4.36. Reactor Network Dialog Box
51.4.37. Decoupled Detailed Chemistry Dialog Box
51.4.38. Reacting Channel Model Dialog Box
51.4.39. Reacting Channel 2D Curves Dialog Box
51.4.40. Discrete Phase Model Dialog Box
51.4.41. DEM Collisions Dialog Box
51.4.42. Create Collision Partner Dialog Box
51.4.43. Copy Collision Partner Dialog Box
51.4.44. Rename Collision Partner Dialog Box
51.4.45. DEM Collision Settings Dialog Box
51.4.46. Solidification and Melting Dialog Box
51.4.47. Acoustics Model Dialog Box
51.4.48. Acoustic Sources Dialog Box
51.4.49. Acoustic Receivers Dialog Box
51.4.50. Basic Shapes Dialog Box
51.4.51. Integration Surface Dialog Box
51.4.52. Interior Cell Zone Selection Dialog Box
51.4.53. Structural Model Dialog Box

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

51.4.54. Eulerian Wall Film Dialog Box
51.4.55. Battery Model Dialog Box
51.4.56. Standalone Echem Model Dialog Box
51.4.57. Potential/Electrochemistry Dialog Box

51.4.1. Multiphase Model Dialog Box

The **Multiphase Model** dialog box allows you to set parameters for modeling multiphase flow. See Enabling the Multiphase Model (p. 2918) – Using the Singhal et al. Expert Cavitation Model (p. 3056) for details.



Controls

Apply

saves the settings specified in the current tab.

> **Important:**
>
> When setting up your case, if you have made changes in the current tab, you should click the **Apply** button to make them effective before moving to the next tab. Otherwise, the relevant models may not be available in the other tabs, and your settings may be lost.

Models

tab allows you to select a multiphase model and adjust model controls.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Model

allows you to select one of four multiphase models.

Off

disables the calculation of multiphase flow.

Volume of Fluid

enables the VOF model described in Volume of Fluid (VOF) Model Theory in the Theory Guide. See Setting Up the VOF Model (p. 2977) for details about using the model. This is available only with the pressure-based solver.

Mixture

enables the mixture model described in Mixture Model Theory in the Theory Guide. See Setting Up the Mixture Model (p. 3038) for details about using the model. This is available only with the pressure-based solver.

Eulerian

enables the Eulerian model described in Eulerian Model Theory in the Theory Guide. See Setting Up the Eulerian Model (p. 3065) for details about using the model. This is available only with the pressure-based solver.

Wet Steam

enables the wet steam model described in Wet Steam Model Theory in the Theory Guide. See Setting Up the Wet Steam Model (p. 3173) for details about using the model.

Important:

When General Turbo Interfaces (GTI) are used, available models reduce to Wet Steam only.

For details on the different GTI types, see Blade Row Interaction Modeling (p. 1956).

Number of Eulerian Phases

allows you to specify the number of phases for the multiphase calculation. You can specify up to 20 phases.

Coupled Level Set + VOF

allows you to apply an interface tracking method that couples the level set method with the VOF formulation.

Volume Fraction Parameters

contains parameters related to the VOF and Eulerian model. This section of the dialog box will appear only when **Volume of Fluid** or **Eulerian** is the selected **Model**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Formulation

allows you to select the desired volume tracking formulation.

Explicit

enables the Euler explicit formulation, described in The Explicit Formulation in the Theory Guide. See Choosing Volume Fraction Formulation (p. 2921) for more information.

Implicit

enables the implicit formulation, described in The Implicit Formulation in the Theory Guide. See Choosing Volume Fraction Formulation (p. 2921) for more information.

Volume Fraction Cutoff

specifies a cutoff limit for the volume fraction values. The value that you provide is used as the lower cutoff for the volume fraction. All volume fraction values in the domain below this cutoff value are set to zero. The upper cutoff is calculated as (1.0 - lower cutoff). All volume fraction values above the upper cutoff value are set to 1.0. The default value is 1e-6.

Important:

The **Volume Fraction Cutoff** value can be specified when using the **Explicit** formulation for volume fraction.

Courant Number

specifies the maximum Courant number allowed near the free surface. This item will not appear if the **Implicit Scheme** is selected. See Setting Time-Dependent Parameters for the Explicit Volume Fraction Formulation (p. 3017) for details.

Model Parameters

contains options related to the **Mixture** model.

Slip Velocity

enables/disables the calculation of slip velocities for the secondary phases as described in Relative (Slip) Velocity and the Drift Velocity in the Theory Guide. See also Solving a Homogeneous Multiphase Flow (p. 2929).

Flow Regime Modeling

enables/disables the flow regime modeling as described in Using the Flow Regime Modeling (p. 3048). See also Flow Regime Modeling in the *Fluent Theory Guide*. This option is available only when **Slip Velocity** is enabled.

Regime Transition Modeling

contains options for modeling flow regime transition. This group box is available only with the Eulerian multiphase model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Algebraic Interfacial Area Density (AIAD)

allows you to enable the algebraic interfacial area density model and sets its parameters (see Using the Algebraic Interfacial Area Density (AIAD) Model (p. 3097) for details).

Generalized Two Phase Flow (GENTOP)

allows you to enable the GENTOP model and set its parameters (see Steps for Using the GENTOP Model (p. 3105) for details).

Hybrid Models

contains options related to the **Eulerian** model.

Dense Discrete Phase Model

allows you to include the Dense Discrete Phase model (see Including the Dense Discrete Phase Model (p. 3107) for details). Enabling this model automatically enables the DPM model.

Multi-Fluid VOF Model

allows you to include the multi-fluid VOF model (see Including the Multi-Fluid VOF Model (p. 3129) for details). The multi-fluid VOF model allows the modeling of interface sharpening schemes and free surface flow.

Sub-Models

is a group box that contains options for the boiling model.

Boiling Model

when enabled, includes the boiling effects in your simulation (see Including the Boiling Model (p. 3113) for details).

Boiling Options...

opens the **Boiling Model** dialog box (see Including the Boiling Model (p. 3113) for details). This item is available only when the **Boiling Model** is selected.

Number of Discrete Phase

allows you to specify the number of discrete phases when the **Dense Discrete Phase Model** option is enabled.

Body Force Formulation

contains an additional option for body force calculations.

Implicit Body Force

enables the implicit body force treatment described in Including Body Forces (p. 2933).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

VOF Sub-Models

contains additional sub-models that can be applied when using the VOF model.

Open Channel Flow

enables the model to study the effects of open channel flow. See Open Channel Flow in the Theory Guide and Modeling Open Channel Flows (p. 2979) for details.

Open Channel Wave BC

enables the model to set specific parameters for a particular boundary for open channel wave boundaries. This option is not available for axisymmetric cases. See Open Channel Wave Boundary Conditions in the Theory Guide and Modeling Open Channel Wave Boundary Conditions (p. 2990) for details.

Options

contains additional modeling options.

Interface Modeling

contains settings for how to model interfaces for the VOF model, Mixture multiphase model, and Eulerian with Multi-Fluid VOF enabled. See Interface Modeling Type (p. 2922) for details.

Type

allows you to select the type of interface modeling to be performed. You can choose between **Sharp**, **Dispersed**, or a hybrid **Sharp/Dispersed** when both types of interfaces may be present, or when the interface is mildly sharp. See Interface Modeling Type (p. 2922) for more details.

Interfacial Anti-Diffusion

enables an anti-diffusion treatment at the interface when using the **Sharp** interface modeling type. This can be used to reduce the effects of numerical diffusion that may occur with coarse meshes, high-aspect ratio cells, or large jumps in cell volume near the interface.

Interface Modeling Options...

opens the **Interface Modeling Options** dialog box from which you can enable **Zonal Discretization** and/or **Phase Localized Discretization**.

Expert Options...

opens the **Expert Options** dialog box which contains settings for VOF time advancement when using the Explicit formulation. Here you can set the **Sub-Time Step Calculation Method** and enable **Solve VOF Every Iteration**. See Expert Options (p. 2926) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Phases

tab allows you to define the phases for your simulation. See Steps for Setting Cell Zone Conditions (p. 2963) for details.

Phases

contains a list of all of the phases in the problem from which you can select the phase you want to define or modify. A phase can be a **Primary Phase** or a **Secondary Phase**. You cannot change a phase from primary to secondary, or vice versa. Instead, you can redefine the properties of the primary phase to reflect the new phase designated as primary, and redefine the secondary phases accordingly as well.

Add Phase

adds a new phase in your simulation.

Delete Phase

deletes the selected phase.

Phase Setup

is a group box where you can define the properties of the selected primary or secondary phase.

Name

specifies the name of the phase.

ID

displays the ID number of the phase. You will need this number only if you are writing a user-defined function. See the separate Fluent Customization Manual for details about writing user-defined functions for multiphase applications.

Phase Material

contains a drop-down list of available materials, from which you can select the appropriate one for this phase. This control is not available for the DDPM.

Edit...

opens the Edit Material Dialog Box (p. 4957) for the selected **Phase Material**, where you can modify its properties.

Granular

(secondary phase) indicates whether or not this is a solid phase. This item appears only for the Eulerian model.

Packed Bed

(secondary phase only) indicates whether or not the granular phase is a packed bed. This option appears only if **Granular** is selected.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Interfacial Area Concentration

(secondary phase only) is used to predict mass, momentum and energy transfers through the interface between the phases. See Interfacial Area Concentration in the Theory Guide for details. This option is not available if **Granular** is selected.

Granular Temperature Model

(secondary phase only) lists the granular temperature models.

Phase Property

enables phase property model for granular temperature.

Partial Differential Equation

enables partial differential equation model for granular temperature. See Granular Temperature in the Theory Guide for details.

Volume Fraction Approaching Packing Limit

prevents the unlimited accumulation of particles, which are operating at packing limit conditions. This options is available only for the DDPM.

Transition Factor

is specified as either a **constant** or a **user-defined** function. The default value is assumed to be 0.75, which corresponds to the closest sphere packing for monosized spheres (a factor of 4/3). In other words, the transition criterion is based on the local particle volume fraction of the given discrete phase and is specified as a factor multiplied by the maximum packing limit (also a user specified value). This control appears only when **Volume Fraction Approaching Packing Limit** is selected.

Diameter

(secondary phase only) specifies the diameter of the particles. You can select **constant** in the drop-down list and specify a constant value, or select **user-defined** to use a user-defined function. See the separate Fluent Customization Manual for details about user-defined functions. The **Diameter** appears for both the mixture model and the Eulerian model. This control is not available for the DDPM.

Granular Properties

(secondary phase only) contains a list of granular phase-specific properties. This section of the dialog box will not appear for the VOF model. This list is only if **Granular** is selected.

Granular Viscosity

specifies the kinetic part of the granular viscosity of the particles ($\mu_{s,kin}$ in Equation 14.362 in the Theory Guide). You can select **constant** (the default) in the drop-down list and specify a constant value, select **syamlal-obrien** to compute the value using Equation 14.364 in the Theory Guide, select **gidaspow** to compute the value using Equation 14.365 in the Theory Guide, or select **user-defined** to use a user-defined function. Note the following:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- If you select **filtered** for **Drag Coefficient**, then **Granular Viscosity** is set automatically to **filtered**. The granular viscosity will be calculated as described in The Filtered Two-Fluid Model in the *Fluent Theory Guide*.
- If you select **user-defined**, your user-defined function must include both the kinetic portion and the collisional portion of the viscosity in the value it returns.

Granular Bulk Viscosity

specifies the solids bulk viscosity (λ_q in Equation 14.195 in the Theory Guide). You can select **constant** (the default) in the drop-down list and specify a constant value, select **lun-et-al** to compute the value using Equation 14.366 in the Theory Guide, or select **user-defined** to use a user-defined function. Note that if you select **filtered** for **Drag Coefficient**, then **Granular Bulk Viscosity** is set to zero automatically.

Frictional Viscosity

specifies a shear viscosity based on the viscous-plastic flow ($\mu_{friction}$ in Equation 14.362 in the Theory Guide). By default, the frictional viscosity is neglected, as indicated by the default selection of **none** in the drop-down list. If you want to include the frictional viscosity, you can select **constant** and specify a constant value, select **schaeffer** to compute the value using Equation 14.367 in the Theory Guide, or select **user-defined** to use a user-defined function.

Angle Of Internal Friction

specifies a constant value for the angle φ used in Schaeffer's expression for frictional viscosity (Equation 14.367 in the Theory Guide). This parameter is relevant only if you have selected **schaeffer** or **user-defined** for the **Frictional Viscosity**.

Frictional Pressure

specifies the pressure gradient term, $\nabla P_{friction}$, in the granular-phase momentum equation. Choose **none** to exclude frictional pressure from your calculation, **johnson-et-al** to apply Equation 14.371 in the Theory Guide, **syamlal-et-al** to apply Equation 14.268 in the Theory Guide, **based-ktgf**, where the frictional pressure is defined by the kinetic theory [38] (p. 5715). The solids pressure tends to a large value near the packing limit, depending on the model selected for the radial distribution function. You must hook a user-defined function when selecting the **user-defined** option. See the separate Fluent Customization Manual for information on hooking a UDF.

Frictional Modulus

can be set as **derived**, or as a **user-defined** function. This is defined as Equation 26.17 (p. 3041).

Friction Packing Limit

specifies a threshold volume fraction at which the frictional regime becomes dominant. The default value is 0.61.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Granular Conductivity

specifies the solids conductivity. You can select **syamlal-obrien**, **gidaspow**, **constant** or **user-defined**.

Granular Temperature

specifies temperature for the solids phase and is proportional to the kinetic energy of the random motion of the particles. You can choose the **algebraic**, **constant**, **dpm-averaged**,or **user-defined** option. **dpm-averaged** is available only when using the Dense Discrete Phase Model (DDPM). Note that if you select **filtered** for **Drag Coefficient**, then **Granular Temperature** is set to zero automatically.

Solids Pressure

specifies the pressure gradient term, ∇p_s, in the granular-phase momentum equation. Choose either the **lun-et-al**, the **syamlal-obrien**, the **ma-ahmadi**, or the **user-defined** option. Note that if you select **filtered** for **Drag Coefficient**, then **Solids Pressure** is set automatically to **filtered**. The solid pressure will be calculated as described in The Filtered Two-Fluid Model in the *Fluent Theory Guide*.

Radial Distribution

specifies a correction factor that modifies the probability of collisions between grains when the solid granular phase becomes dense. Choose either the **lun-et-al**, the **syamlal-obrien**, the **ma-ahmadi**, the **arastapour**, or a **user-defined** option. Note that if **filtered** is selected for **Drag Coefficient**, then **Radial Distribution** is set automatically to **filtered**. In the simulation, the radial distribution will be set to zero since the filtered two-fluid model is not based on kinetic theory.

Elasticity Modulus

is defined as

$$G=\frac{\partial P_s}{\partial \alpha_s} \tag{51.1}$$

with $G \geq 0$.

Choose either the **derived** or **user-defined** options.

Packing Limit

specifies the maximum volume fraction for the granular phase. For monodispersed spheres the packing limit is about 0.63, which is the default value in Ansys Fluent. In polydispersed cases, however, smaller spheres can fill the small gaps between larger spheres, so you may need to increase the maximum packing limit.

Phase State

is a group box where you can select either **Liquid** or **Gas** phase state. This item is available only when **Flow Regime Modeling** is selected in the **Models** tab (**Model Parameters** group box).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Phase Morphology

is a group box where you can select from the following phase morphology options:

- **Continuous**
- **Dispersed** (secondary phase only)
- **Hybrid**

This item is available only when **Flow Regime Modeling** is selected in the **Models** tab (**Model Parameters** group box).

Interfacial Area Concentration Properties

(secondary phase only) contains a list of phase-specific properties. This list is available only if **Interfacial Area Concentration** is selected. This section of the dialog box will not appear for the VOF, mixture, or DDPM multiphase models.

Min/Max Diameter

are the limits of the bubble diameters.

Breakage Kernel

allows you to specify the breakage kernel. You can select **none**, **constant**, **hibiki-ishii**, **ishii-kim**, **yao-morel**, or **user-defined**. The three options, **hibiki-ishii**, **ishii-kim**, and **yao-morel** are described in detail in Interfacial Area Concentration in the Theory Guide.

Coalescence Kernel

allows you to specify the coalescence kernel. You can select **none**, **constant**, **hibiki-ishii**, **ishii-kim**, **yao-morel**, or **user-defined**. The three options, **hibiki-ishii**, **ishii-kim**, and **yao-morel** are described in detail in Interfacial Area Concentration in the Theory Guide.

Nucleation Rate

is a source term for the interfacial area concentration that models the rate of formation of the dispersed phase. You can choose from **constant** or **user-defined**. If the **Boiling Model** option is enabled when using the Eulerian multiphase model, you can also select **yao-morel**. The **yao-morel** option is described in Yao-Morel Model in the Theory Guide.

Surface Tension

specifies the attractive forces between the interfaces.

Critical Weber Number

will need to be specified if you selected **yao-morel** for the **Breakage Kernel**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Dissipation Function

gives you the option to choose the formula which calculates the dissipation rate used in the **hibiki-ishii** and **ishii-kim** models. You can choose **constant**, **wu-ishii-kim**, **fluent-ke**, and **user-defined** for the dissipation function.

Hydraulic Diameter

is the value used in Equation 26.19 (p. 3045). This is available when the **wu-ishii-kim** formulation is selected as the **Dissipation Function**. .

Phase Interaction

tab allows you to define the interaction between phases.

Interaction Domain ID

displays the ID number of the interaction domain. You will need this number only if you are writing a user-defined function. See the separate Fluent Customization Manual for details about writing user-defined functions for multiphase applications.

Forces

tab allows you to specify forces acting between the phases.

Phase Pairs

is a list of all of the phase pairs in the problem from which you can select the pair and define the forces between the phases.

Global Options

contains controls that apply along phase interface in each pair of phases.

Shaver-Podowski Lift Correlation

(Eulerian Multiphase model only) when enabled, improves the accuracy of the prediction of the void peak near the wall and overall robustness for cases where lift force, turbulent dispersion, and wall lubrication are included. See Including the Lift Correlation (p. 3081) for more information.

Virtual Mass Implicit Options

specifies what form of the implicit method to use. **Default** models the entire virtual mass force while **Option 2** and **Option 3** model truncated expressions which may further improve convergence. This option appears only when the **Virtual Mass Coefficient** is set to a nonzero value in the **Force Setup** group box.

Surface Tension Force Modeling

includes the effects of surface tension along the fluid-fluid interface.

Model

contains two surface tension models from which to choose.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Continuum Surface Force

adds the surface tension to the VOF calculation, which results in a source term in the momentum equation. This method is available only for the VOF and Eulerian models.

Continuum Surface Stress

is an alternative way to modeling surface tension in a conservative manner compared to the continuum surface force method. This method is available only for the VOF and Eulerian models.

Adhesion Options

contains options to include wall and jump adhesion.

Wall Adhesion

enables the specification of a wall adhesion angle. (The angle itself, as defined in Figure 26.21: Measuring the Contact Angle (p. 2960), will be specified in the Wall Dialog Box (p. 5085).)

Jump Adhesion

enables the treatment of the contact angle specification at the porous jump boundary. (The angle itself will be specified in the Porous Jump Dialog Box (p. 5042).)

Force Setup

contains a list of forces specified for the selected pair of phases. By default, the forces in the **Force Setup** group box are set to **none**. Depending on your case, the following forces can be defined:

- **Drag Coefficient**: You can select the appropriate drag function from the **Coefficient** drop-down list and drag factor from the **Modification** drop-down list. See Specifying the Drag Function (p. 3075) and Drag Modification (p. 3079) for information about the available options. This input is available only for the Eulerian and mixture models only.
- **Lift Coefficient**: Specifies the lift function for each pair of phases. See Defining the Phases for the Eulerian Model (p. 3066) for information about the available options. This input is available only for the Eulerian model.
- **Wall Lubrication**: Specifies the wall lubrication model for each primary-secondary phase pair. See Including the Wall Lubrication Force (p. 3081) for information about the available options. This input is available only for Eulerian two-phase flows with a non-granular phase.
- **Turbulent Dispersion**: Specifies the turbulent dispersion model for each primary-secondary phase pair. See Including the Turbulent Dispersion Force (p. 3085) for information about the available options. This input is available only for the Eulerian model.
- **Turbulence Interaction**: Specifies the turbulence interaction model for each primary-secondary phase pair. See Including Turbulence Interaction Source Terms (p. 3091) for

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

information about the available options. This input is available only for the Eulerian model.

- **Virtual Mass Coefficient**: Specifies the virtual mass coefficient for each pair of phases. See Including the Virtual Mass Force (p. 3088) for details. This input is available only for the Eulerian model.
- **Slip Velocity**: Specifies the slip velocity function for each secondary phase with respect to the primary phase. See Defining the Phases for the Mixture Model (p. 3038) for information about the available options. This input is available only for the mixture multiphase model.
- **Surface Tension Coefficients**: Specify the surface tension coefficient, σ in Equation 14.23 and Equation 12.58 in the Theory Guide, for each pair of phases. See Defining the Phases for the VOF Model (p. 3009) for details. This input is available only for the Eulerian model.
- **Restitution Coefficient**: Specifies the restitution coefficient for collisions between each pair of granular phases, and for collisions between particles of the same granular phase. It is relevant only if two or more granular phases are involved. See Defining the Phases for the Eulerian Model (p. 3066) for information about the available options.

Heat, Mass, Reactions

tab contains the following tabs:

Heat

tab displays the **Heat Transfer Coefficient** inputs. See Including Heat Transfer Effects (p. 3093) for information about the available options. This tab is active only for the Eulerian model when the energy equation is active.

Heat Transfer Coefficient

specifies the heat transfer coefficient function between each pair of phases. See Including Heat Transfer Effects (p. 3093) for information about the available options.

Mass

tab displays the **Mass Transfer Function** inputs.

Number of Mass Transfer Mechanisms

specifies the number of mass transfer mechanisms in your simulation. See Including Mass Transfer Effects (p. 2940) for information about the available options.

Reactions

tab allows you to define multiple heterogeneous reactions and stoichiometry.

Total Number of Heterogeneous Reactions

specifies the total number of reactions (volumetric reactions, wall surface reactions, and particle reactions). See Specifying Heterogeneous Reactions (p. 2937) for information about the available options.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Heterogeneous Stiff Chemistry Solver

is used in inter-phase reaction mechanisms containing numerically stiff reactions. This option can improve convergence and is available for transient Eulerian multiphase simulations.

Reaction Name

allows you to enter a name for the reaction.

ID

enables you to set the reaction ID for each reaction.

Number of Reactants

allows you to specify the number of reactants that are involved in the reaction.

Number of Products

allows you to specify the number of products that are involved in the reaction.

Phase

drop-down list allows you to select the phase that is involved in the reaction.

Species

drop-down list allows you to select the species.

Stoich. Coefficient

allows you to set the stoichiometric coefficient.

Reaction Rate Function

allows you to choose rate exponents for an Arrhenius-type reaction, a user-defined function, or a population balance mechanism for the reaction rate.

Interfacial Area

tab displays the **Interfacial Area** settings to predict interfacial area between the phases. This tab appears only for the Mixture multiphase model with interphase mass transfer and the Eulerian multiphase model. See Defining the Algebraic Interfacial Area Concentration (p. 3045) and Using an Algebraic Interfacial Area Model (p. 3096) for information about the available options.

Discretization

tab allows you to use the diffusive and anti-diffusive discretization procedure across the distinct interfaces.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Phase Localized Compressive Scheme

enables the compressive discretization scheme in Ansys Fluent, where the degree of diffusion/sharpness is controlled through the value of the slope limiters. This item will appear only for the VOF model and for the Eulerian model with **Multi-Fluid VOF Model** enabled.

Slope Limiters

are values equating to specific discretization schemes. For each pair of phases, you can enter a value of 0, 1, or 2, or any value between 0 and 2. Depending on the value you use, first order upwind, second order upwind, compressive, or the blended scheme will be applied. Refer to Table 26.11: Slope Limiter Discretization Scheme (p. 3017) to equate each value of the slope limiter with a discretization scheme. For more information, refer to Discretizing Using the Phase Localized Compressive Scheme (p. 3015).

Model Transition

displays the **VOF-to-DPM Model Transition** inputs. This tab is available only when a DPM injection is set as described in Setting up the VOF-to-DPM Model Transition (p. 3021).

Number of Model Transition Mechanisms

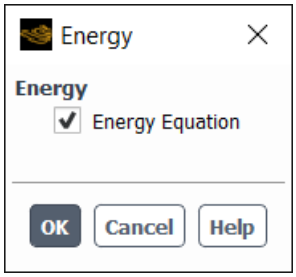
specifies the number of VOF-to-DPM model transition mechanisms in your simulation. See Setting up the VOF-to-DPM Model Transition (p. 3021) for information about the available options.

Population Balance

tab contains controls for setting population balance simulations. This tab is displayed for the mixture and Eulerian multiphase models. See Population Balance Model (p. 3131) for details.

51.4.2. Energy Dialog Box

The **Energy** dialog box allows you to set parameters related to energy or heat transfer in your model.



Controls

Energy

contains inputs related to the modeling of energy.

Energy Equation

enables/disables the calculation of energy in the model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Energy Modes

contains inputs related to the alternative energy modeling modes.

Two-Temperature Model

enables/disables the Two-Temperature Model. Only available for the density-based solver.

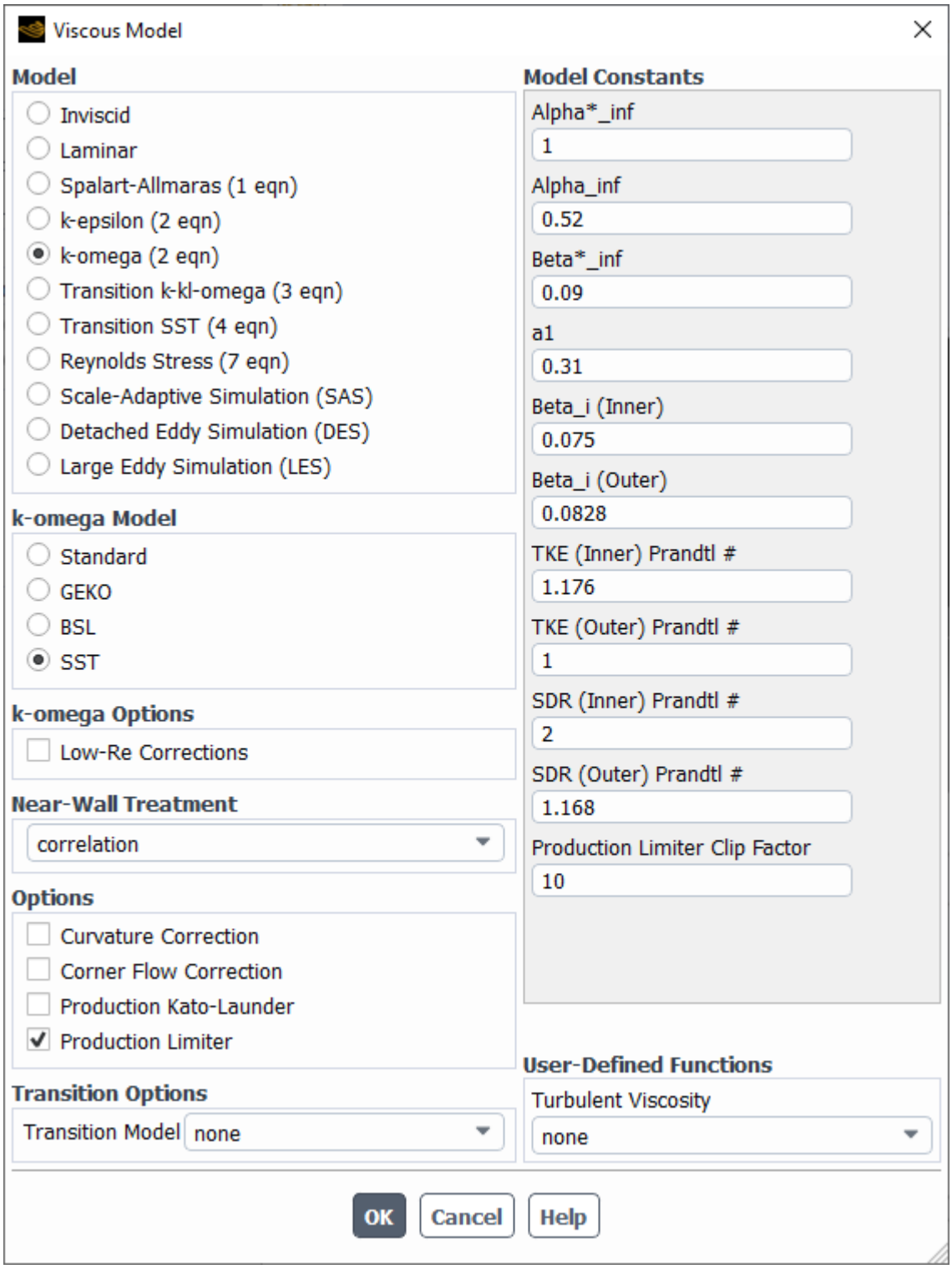
51.4.3. Viscous Model Dialog Box

The **Viscous Model** dialog box allows you to set parameters for inviscid, laminar, and turbulent flow. See Steps in Using a Turbulence Model (p. 2081) for details about using this dialog box.

Important:

This feature offers reduced functionality when running Fluent under the **Pro** capability level.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Model

contains options for specifying the viscous model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Inviscid

specifies inviscid flow.

Laminar

specifies laminar flow.

Spalart-Allmaras

specifies turbulent flow to be calculated using the Spalart-Allmaras model. (See Spalart-Allmaras Model in the Theory Guide for background about this model. See Setting Up the Spalart-Allmaras Model (p. 2084) for details about using this model.)

k-epsilon

specifies turbulent flow to be calculated using one of three k-ε models. (See Standard, RNG, and Realizable k-ε Models in the Theory Guide for background about this model. See Setting Up the k-ε Model (p. 2085) for details about using this model.)

k-omega

specifies turbulent flow to be calculated using one of two k-ω models. (See Standard, BSL, and SST k-ω Models in the Theory Guide for background about these models. See Setting Up the k-ω Model (p. 2089) for details about using this model.)

Transition k-kl-omega

specifies turbulent flow to be calculated using the Transition k-kl-ω model. (See k-kl-ω Transition Model in the Theory Guide for background about this model. See Setting Up the Transition k-kl-ω Model (p. 2100) for details about using this model.)

Transition SST

specifies turbulent flow to be calculated using the Transition SST model. (See Transition SST Model in the Theory Guide for background about this model. See Setting Up the Transition SST Model (p. 2101) for details about using this model.)

Reynolds Stress

specifies turbulent flow to be calculated using the RSM. (See Reynolds Stress Model (RSM) in the Theory Guide for background about this model. See Setting Up the Reynolds Stress Model (p. 2105) for details about using this model.)

Scale-Adaptive Simulation (SAS)

specifies turbulent flow to be calculated using the SAS model in combination with the SST k-ω model. (See Scale-Adaptive Simulation (SAS) Model in the Theory Guide for background about this model. See Setting Up Scale-Adaptive Simulation (SAS) Modeling (p. 2109) for details about using this model.)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Detached Eddy Simulation

specifies turbulent flow to be calculated using the DES model. (See Detached Eddy Simulation (DES) in the Theory Guide for background about this model. See Setting Up the Detached Eddy Simulation Model (p. 2112) for details about using this model.)

Large Eddy Simulation

(3D only) specifies turbulent flow to be calculated using the LES model. (See Large Eddy Simulation (LES) Model in the Theory Guide for background about this model. See Setting Up the Large Eddy Simulation Model (p. 2122) for details about using this model.)

Spalart-Allmaras Production

contains options for the Spalart-Allmaras model. This portion of the dialog box will appear only if **Spalart-Allmaras** is selected as the **Model**.

Vorticity-Based

selects the vorticity-based calculation of the deformation tensor S (see Equation 4.22 in the Theory Guide).

Strain/Vorticity-Based

selects the strain/vorticity-based calculation of the deformation tensor S (see Equation 4.24 in the Theory Guide).

k-epsilon Model

contains options for specifying which of the k-ε models is to be used. This portion of the dialog box will appear only if **k-epsilon** is selected as the **Model**.

Standard

selects the standard k-ε model, described in Standard k-ε Model in the Theory Guide and Setting Up the k-ε Model (p. 2085).

RNG

selects the RNG k-ε model, described in RNG k-ε Model. in the Theory Guide and Setting Up the k-ε Model (p. 2085).

Realizable

selects the realizable k-ε model, described in Realizable k-ε Model in the Theory Guide and Setting Up the k-ε Model (p. 2085).

RNG Options

specifies parameters that affect the solution of problems solved with the RNG k-ε model. This portion of the dialog box will appear only if **RNG** is selected as the **k-epsilon Model**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Differential Viscosity Model

specifies whether or not the low-Reynolds-number RNG modifications to turbulent viscosity should be included. By default, this option is turned off. It is likely to have an effect only when the near-wall regions in the domain are well resolved in terms of mesh density. See Differential Viscosity Modification (p. 2135) for details.

Swirl Dominated Flow

specifies whether or not the RNG modification to turbulent viscosity for swirling flows should be included. This option is available only in 3D and 2D axisymmetric swirl solvers, and it can yield improved predictions when solving flows with significant swirl. See Swirl Modification (p. 2136) for details.

k-omega Model

contains options for specifying which of the k-ω models is to be used. This portion of the dialog box will appear only if **k-omega** is selected as the **Model**.

Standard

selects the standard k-ω model, described in Standard k-ω Model in the Theory Guide and Setting Up the k-ω Model (p. 2089).

GEKO

selects the generalized k-ω model, described in Generalized k-ω (GEKO) Model in the Theory Guide and Setting up the Generalized k-ω (GEKO) Model (p. 2090).

BSL

selects the baseline (BSL) k-ω model, described in Baseline (BSL) k-ω Model in the Theory Guide and Setting Up the k-ω Model (p. 2089).

SST

selects the shear-stress transport (SST) k-ε model, described in Shear-Stress Transport (SST) k-ω Model in the Theory Guide and Setting Up the k-ω Model (p. 2089).

k-omega Options

specifies parameters that affect the solution of problems solved with the k-ω models. This portion of the dialog box will appear only if **k-omega** is selected as the **Model**.

Low-Re Corrections

specifies whether corrections that improve the accuracy in predicting low Reynolds number flows should be included. This option is available only for the k-ω models and the stress-omega RSM model. See Low-Re Corrections (p. 2136) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Shear Flow Corrections

specifies whether corrections that improve the accuracy in predicting free shear flows should be included. This option is available only for the standard k-ω model and the stress-omega RSM model. See Shear Flow Corrections (p. 2136) for details.

Transition SST Options

allows you to include the **Roughness Correlation** of rough walls as described in Transition SST and Rough Walls in the Theory Guide.

Roughness Correlation

when enabled allows you to specify the **Geometric Roughness Height** as a constant value.

Reynolds-Stress Model

specifies the various Reynolds stress models (RSM).

Linear Pressure-Strain

enables the linear pressure-strain model. See Linear Pressure-Strain Model in the Theory Guide for details.

Quadratic Pressure-Strain

enables the quadratic pressure-strain model for superior performance in a range of basic shear flows, including plane strain, rotating plane shear, and axisymmetric expansion/contraction. See Quadratic Pressure-Strain Model in the Theory Guide for details. Note that this option cannot be used with the **Wall Reflection Effects** option or the **Enhanced Wall Treatment**.

Stress-Omega

enables a stress-transport model that is based on the omega equations and LRR model [173] (p. 5723). This model is ideal for modeling flows over curved surfaces and swirling flows. See Stress-Omega Model in the Theory Guide for details.

Stress-BSL

enables a linear model for the pressure-strain term like the **Stress-Omega** model, but solves the scale equation from the baseline (BSL) k-ω model, and thus removes the free-stream sensitivity observed with the **Stress-Omega** model. Like the **Stress-Omega** model, it is recommended for modeling flows over curved surfaces and swirling flows. See Stress-BSL Model in the Theory Guide for details.

Reynolds-Stress Options

specifies parameters that affect the solution of problems solved with the Reynolds stress model. This portion of the dialog box will appear only if **Reynolds Stress** is selected as the **Model**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Wall BC from k Equation

enables the explicit setting of boundary conditions for the Reynolds stresses near the walls, using the values computed with Equation 4.260 in the Theory Guide. See Solving the k Equation to Obtain Wall Boundary Conditions (p. 2137) for details. This option is on by default.

Wall Reflection Effects

enables the calculation of the component of the pressure strain term responsible for the re-distribution of normal stresses near the wall. See Including the Wall Reflection Term (p. 2137) for details. Note that this option is not available if you have enabled the **Quadratic Pressure-Strain Model**.

RANS Model

contains options for the subgrid-scale model used by the **Detached Eddy Simulation Model**. This portion of the dialog box will appear only if **Detached Eddy Simulation Model** is selected as the **Model**.

Spalart-Allmaras

enables the Spalart-Allmaras RANS model. See Detached Eddy Simulation (DES) in the Theory Guide for details.

Realizable k-epsilon

enables the Realizable k-ε RANS model. See Detached Eddy Simulation (DES) in the Theory Guide for details.

SST k-omega

enables the SST k-ω RANS Model. See Detached Eddy Simulation (DES) in the Theory Guide for details.

DES Options

contain the option to include a delayed Detached Eddy Simulation.

Delayed DES

is useful for RANS meshes with high aspect ratios in the boundary layer. This option preserves the RANS model throughout the boundary layer. (See Delayed Detached Eddy Simulation (DDES) (p. 2135) for details.)

Subgrid-Scale Model

contains options for the subgrid-scale model used by the LES model. This portion of the dialog box will appear only if **Large Eddy Simulation** is selected as the **Model**.

Smagorinsky-Lilly

selects the Smagorinsky-Lilly subgrid-scale model described in Subgrid-Scale Models in the Theory Guide.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

WALE

selects the Wall-Adapting local Eddy-Viscosity model described in Wall-Adapting Local Eddy-Viscosity (WALE) Model in the Theory Guide.

WMLES

selects the algebraic Wall-Modeled LES model described in Algebraic Wall-Modeled LES Model (WMLES) in the Theory Guide.

WMLES S-Omega

selects the algebraic Wall-Modeled LES S-Ω model described in Algebraic WMLES S-Omega Model Formulation in the Theory Guide.

Kinetic-Energy Transport

selects the dynamic kinetic energy subgrid-scale model described in Dynamic Kinetic Energy Subgrid-Scale Model in the Theory Guide.

LES Model Options

contains options for the Large Eddy Simulation model. This portion of the dialog box will appear only if **Large Eddy Simulation** is selected as the **Model**.

Dynamic Stress

enables the dynamic stress model. It is available for selection when the **Smagorinsky-Lilly** is selected, and is enabled (and cannot be disabled) when the **Kinetic-Energy Transport** is selected.

Dynamic Energy Flux

enables the dynamic energy flux model. It is available when the LES model option **Dynamic Stress** is enabled.

Dynamic Scalar Flux

enables the dynamic computation of turbulent Sc (σ_t in Equation 8.5 in the Theory Guide). See Definition of the Mixture Fraction in the Theory Guide for details. It is available when the LES model option **Dynamic Stress** is enabled.

Dynamic Fvar

enables the dynamic mixture fraction variance model. It is available when **Non-Premixed Combustion** or **Partially Premixed Combustion** is selected in the Species Model Dialog Box (p. 4768). See The Non-Premixed Model for LES in the Theory Guide for details.

Near-Wall Treatment

specifies the near-wall treatment to be used for modeling turbulence. See Near-Wall Treatments for Wall-Bounded Turbulent Flows in the Theory Guide for details about the available methods. This portion of the dialog box will appear if **k-epsilon** or **Reynolds Stress** is selected as the **Model**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Standard Wall Functions

enables the use of standard wall functions (described in Standard Wall Functions in the Theory Guide).

Scalable Wall Functions

enables the use of scalable wall functions (described in Scalable Wall Functions in the Theory Guide).

Non-Equilibrium Wall Functions

enables the use of non-equilibrium wall functions (described in Non-Equilibrium Wall Functions in the Theory Guide).

Enhanced Wall Treatment

enables the use of the enhanced wall treatment (described in Enhanced Wall Treatment ε-Equation (EWT-ε) in the Theory Guide). Note that this option will not appear if you have enabled the **Quadratic Pressure-Strain Model** under **Reynolds-Stress Options**.

Menter-Lechner

enables the use of the Menter-Lechner near-wall treatment (described in Menter-Lechner ε-Equation (ML-ε) in the Theory Guide). Note that this option is only available when **k-epsilon** is selected for the **Model**.

User-Defined Wall Functions

enables you to hook a user-defined function, used to define the **Law of the Wall**. See User-Defined Wall Functions in the Theory Guide for more information.

Enhanced Wall Treatment Options

allows you to include pressure gradient or thermal effects in the calculation. See Near-Wall Treatments for Wall-Bounded Turbulent Flows in the Theory Guide.

Pressure Gradient Effects

enables the effect of pressure gradient.

Thermal Effects

enables thermal effects in the calculation. This option appears only if the energy equation is enabled.

Options

contains general options for viscous modeling.

Viscous Heating

(if enabled) includes the viscous dissipation terms in the energy equation. This option is recommended when you are solving a compressible flow. Note that this option is always turned on when one of the density-based solvers is used; you will not be able to turn it off.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Low-Pressure Boundary Slip

includes slip boundary conditions for velocity and temperature for modeling fluid flow at very low pressures as in semiconductor fabrication devices. See Slip Boundary Formulation for Low-Pressure Gas Systems in the Theory Guide. This option is available only for laminar flows.

Buoyancy Effects

enables the inclusion of buoyancy effects. See Including Buoyancy Effects on Turbulence (p. 2129) for details. Only available for non-isothermal flow and when a nonzero gravitational acceleration has been specified in the Operating Conditions Dialog Box (p. 4978).

Curvature Correction

when enabled, modifies the turbulence production term to sensitize the standard eddy-viscosity models to the effects of streamline curvature and system rotation. This is available for the **Spalart-Allmaras**, **k-epsilon**, **k-omega**, **Transition SST**, **Scale-Adaptive Simulation**, and **Detached Eddy Simulation** with the **SST k-omega** model.

Curvature Correction Options

Includes options related to curvature corrections.

CCURV

specifies a coefficient used in the curvature correction term to influence the strength of the curvature correction if needed for a specific flow. Can be a constant or specified via UDF.

Corner Flow Correction

when enabled, accounts for secondary flows found in rectangular channels, pipes of non-circular cross-section, and wing-body-junction type geometries in the plane normal to the main flow direction into the corner along the bisector.

Corner Flow Correction Options

Includes options related to corner flow corrections.

CCORNER

Sets the strength of the quadratic term of the corner flow correction. The default value is 1. Can be a constant or specified via UDF.

Compressibility Effects

when enabled, can improve the prediction of free shear layers at high Mach numbers. This is available when the compressible form of the ideal gas law or the real-gas model is activated for **k-epsilon**, **k-omega**, **Transition k-kl-omega**, **Reynolds Stress**, **Scale-Adaptive Simulation**, and **Detached Eddy Simulation** with the **Realizable k-epsilon** model. For details, see Model Enhancements (p. 2071).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mixture Drift Force

includes the effect of turbulent drift velocity when using the **Mixture** model. See Modeling Turbulence (p. 3089). Note that this option is only available when using the **Mixture** model with **Slip Velocity** enabled in the **Multiphase Model** dialog box. See Including Mixture Drift Force (p. 3048).

Production Kato-Launder

when enabled, the Kato-Launder modification for the production term limits the production term in the turbulence equation. For details see, Production Limiters for Two-Equation Models in the *Fluent Theory Guide*.

Production Limiter

when enabled, limits the production term in the turbulence equation. For details, see Production Limiters for Two-Equation Models in the *Fluent Theory Guide*.

Transition Options

contains options for transition modeling.

Transition Model

selects the transition model. You can enable the gamma-transport-eqn model (also known as the Intermittency Transition model). For details, see Intermittency Transition Model in the Theory Guide.

Include Crossflow Transition

includes the effects of crossflow instability. For details, see Transport Equations for the Intermittency Transition Model in the Theory Guide.

Turbulence Multiphase Model

contains options for multiphase turbulence models. This portion of the dialog box will appear if **Eulerian** is selected as the **Model** in the Multiphase Model Dialog Box (p. 4700).

Mixture

specifies the (default) mixture turbulence model.

Dispersed

specifies the dispersed turbulence model.

Per Phase

specifies the calculation of a set of turbulence equations for each phase.

See Turbulence Models in the Theory Guide for details about the available multiphase turbulence models.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

GEKO Options

contains options for the GEKO model.

Wall Distance Free

Selects the wall distance free version of the GEKO model.

CSEP

Specifies C_{SEP} - parameter to optimize flow separation from smooth surfaces.

CNW

Specifies C_{NW} - parameter to optimize flow in non-equilibrium near wall regions (such as heat transfer or C_f).

CMIX

Specifies C_{MIX} - parameter to optimize strength of mixing in free shear flows.

CJET

Specifies C_{JET} - parameter to optimize free shear layer mixing (optimize free jets independent of mixing layer).

Blending Function

Specifies the blending function, F_{Blend}, which deactivates these parameters inside boundary layers.

See Generalized k-ω (GEKO) Model in the *Fluent Theory Guide* for details about the GEKO model options.

Model Constants

contains constants used in the equations for turbulence. See Spalart-Allmaras Model, Standard k-ε Model, RNG k-ε Model, k-kl-ω Transition Model, Transition SST Model, Realizable k-ε Model, Reynolds Stress Model (RSM), Standard k-ω Model, Shear-Stress Transport (SST) k-ω Model, and Large Eddy Simulation (LES) Model in the Theory Guide for details about these constants.

Cb1

(only for the Spalart-Allmaras model) is the constant C_{b1} in Equation 4.19 in the Theory Guide.

Cb2

(only for the Spalart-Allmaras model) is the constant C_{b2} in Equation 4.15 in the Theory Guide.

Cv1

(only for the Spalart-Allmaras model) is the constant C_{v1} in Equation 4.17 in the Theory Guide.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cw2

(only for the Spalart-Allmaras model) is the constant C_{w2} in Equation 4.28 in the Theory Guide.

Cw3*

(* only for the Spalart-Allmaras model) is the constant C_{w3} in Equation 4.27 in the Theory Guide.

Cprod

(only for the Spalart-Allmaras model when the **Strain/Vorticity-Based Production** option is used) is the constant C_{prod} in Equation 4.24 in the Theory Guide.

Cmu

(only for the standard or RNG k-ε model, the RSM, or the k-kl-ω Transition model) is the constant C_μ that is used to compute μ_t.

C1-Epsilon

(only for the standard or RNG k-ε model or the RSM) is the constant $C_{1\varepsilon}$ used in the transport equation for ε.

C2-Epsilon

(only for the standard, RNG, or realizable k-ε model or the RSM) is the constant $C_{2\varepsilon}$ used in the transport equation for ε.

C3-Epsilon

(only for the dispersed or per-phase k-ε multiphase models) is the constant $C_{3\varepsilon}$ in Equation 14.412 in the Theory Guide.

C-lambda

(only for the k-kl-ω Transition model) is the constant C_λ in the definition of the effective length,

CR

(only for the k-kl-ω Transition model) is the constant C_R used in the definition of R, where R represents the averaged effect of the breakdown of streamwise fluctuations into turbulence during bypass transition

ANAT

(only for the k-kl-ω Transition model) is the constant A_{NAT}

ATS

(only for the k-kl-ω Transition model) is the constant A_{TS}

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CNAT, crit

(only for the k-kl-ω Transition model) is the constant $C_{NAT,crit}$

CTS, crit

(only for the k-kl-ω Transition model) is the constant $C_{TS,crit}$

CRNAT

(only for the k-kl-ω Transition model) is the constant $C_{R,NAT}$

Anu

(only for the k-kl-ω Transition model) is the constant A_ν

CINT

(only for the k-kl-ω Transition model) is the constant C_{INT}

Cw1

(only for the k-kl-ω Transition model) is the constant $C_{\omega 1}$

Cw3**

(**only for the k-kl-ω Transition model) is the constant $C_{\omega 3}$

Calpha-teta

(only for the k-kl-ω Transition model) is the constant $C_{\alpha,\theta}$

Ctaul

(only for the k-kl-ω Transition model) is the constant $C_{\tau,1}$

SDR Prandtl Number

(only for the standard k-ω model and the k-kl-ω Transition model) is the effective "Prandtl" number for the transport of the specific dissipation rate, σ_ω.

Ca1

(only for the Transition SST model)

Ca2

(only for the Transition SST model)

Ce1

(only for the Transition SST model)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ce2

(only for the Transition SST model)

C_thetat

(only for the Transition SST model)

C_s1

(only for the Transition SST model)

Intermit. Prandtl #)

(only for the Transition SST model)

Re_theta. Prandtl #)

(only for the Transition SST model)

C Shielded DES

(only for the SDES or SBES model) is a constant that serves the same purpose as C_{d1} in the DDES turbulence model (see Equation 4.284 in the Theory Guide).

Csdes

(only for the SDES or SBES model) is C_{SDES}, as described in Shielded Detached Eddy Simulation (SDES) in the Theory Guide.

Cwale

(only for the WALE subgrid-scale model of the LES or SBES model) is C_w, as described in Wall-Adapting Local Eddy-Viscosity (WALE) Model in the Theory Guide.

CREAL (GEKO)

(only for the GEKO model) is C_{REAL}, as described in Generalized k-ω (GEKO) Model in the *Fluent Theory Guide*.

CNW_SUB (GEKO)

(only for the GEKO model) is C_{NW_SUB}, as described in Generalized k-ω (GEKO) Model in the *Fluent Theory Guide*.

CJET_AUX (GEKO)

(only for the GEKO model) is C_{JET_AUX}, as described in Generalized k-ω (GEKO) Model in the *Fluent Theory Guide*.

CBF_TUR (GEKO)

(only for the GEKO model) is C_{BF_TUR}, as described in Generalized k-ω (GEKO) Model in the *Fluent Theory Guide*.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CBF_LAM (GEKO)

(only for the GEKO model) is C_{BF_LAM}, as described in Generalized k-ω (GEKO) Model in the *Fluent Theory Guide*.

Swirl Factor

sets the value of α_s in Equation 4.46 in the Theory Guide. This item appears for the RNG k-ε model when the **Swirl Dominated Flow** option is turned on.

Alpha*_inf

(only for the standard, BSL, or SST k-ω model, and the Transition SST model) is the constant α^*_∞ in Equation 4.75 in the Theory Guide.

Alpha_inf

(only for the standard, BSL, or SST k-ω model, and the Transition SST model) is the constant α_∞ in Equation 4.83 in the Theory Guide.

Alpha_0

(only for the standard, BSL, or SST k-ω model with the **Low-Re Corrections** option enabled) is the constant α_0 in Equation 4.83 in the Theory Guide.

Beta*_inf

(only for the standard, BSL, or SST k-ω model, and the Transition SST model) is the constant β^*_∞ in Equation 4.88 in the Theory Guide.

Beta_i

(only for the standard k-ω model) is the constant β_i in Equation 4.96 in the Theory Guide.

R_beta

(only for the standard, BSL, or SST k-ω model) is the constant R_β in Equation 4.88 in the Theory Guide.

R_k

(only for the standard, BSL, or SST k-ω model with the **Low-Re Corrections** option enabled) is the constant R_k in Equation 4.75 in the Theory Guide.

R_w

(only for the standard, BSL, or SST k-ω model with the **Low-Re Corrections** option enabled) is the constant R_ω in Equation 4.83 in the Theory Guide.

Zeta*

(only for the standard, BSL, or SST k-ω model when the **Compressibility Effects** option is enabled) is the constant ζ^* in Equation 4.87 in the Theory Guide.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mt0

(only for the standard, BSL, or SST k-ω model when the **Compressibility Effects** option is enabled) is the constant M_{t0} in Equation 4.97 in the Theory Guide.

a1

(only for the SST k-ω model, and the Transition SST model) is the constant a_1 in Equation 4.118 in the Theory Guide.

Beta_i (Inner)

(only for the BSL or SST k-ω model, and the Transition SST model) is the constant $\beta_{i,1}$ in Model Constants in the Theory Guide.

Beta_i (Outer)

(only for the BSL or SST k-ω model, and the Transition SST model) is the constant $\beta_{i,2}$ in Model Constants in the Theory Guide.

Cs

is a constant that depends on the turbulence model. For LES, it is the Smagorinsky constant C_s in Equation 4.307 in the Theory Guide. For SAS, it is the high wave number damping coefficient C_S in Equation 4.270 in the Theory Guide.

C1-PS

(only for RSM) is the constant C_1 in Equation 4.231 in the Theory Guide.

C2-PS

(only for RSM) is the constant C_2 in Equation 4.232 in the Theory Guide.

C1′-PS

(only for RSM) is the constant $C_1^{'}$ in Equation 4.233 in the Theory Guide.

C2′-PS

(only for RSM) is the constant $C_2^{'}$ in Equation 4.233 in the Theory Guide.

C1-SSG-PS

(only for RSM with the **Quadratic Pressure-Strain Model**) is the constant C_1 in Equation 4.242 in the Theory Guide.

C1′-SSG-PS

(only for RSM with the **Quadratic Pressure-Strain Model**) is the constant C_1^* in Equation 4.242 in the Theory Guide.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

C2-SSG-PS

(only for RSM with the **Quadratic Pressure-Strain Model**) is the constant C_2 in Equation 4.242 in the Theory Guide.

C3-SSG-PS

(only for RSM with the **Quadratic Pressure-Strain Model**) is the constant C_3 in Equation 4.242 in the Theory Guide.

C3'-SSG-PS

(only for RSM with the **Quadratic Pressure-Strain Model**) is the constant C_3^* in Equation 4.242 in the Theory Guide.

C4-SSG-PS

(only for RSM with the **Quadratic Pressure-Strain Model**) is the constant C_4 in Equation 4.242 in the Theory Guide.

C5-SSG-PS

(only for RSM with the **Quadratic Pressure-Strain Model**) is the constant C_5 in Equation 4.242 in the Theory Guide.

Prandtl Number

(only for the Spalart-Allmaras model) is the constant $\sigma_{\tilde{\nu}}$ in Equation 4.15 in the Theory Guide.

TKE Prandtl Number

(only for the standard or realizable k-ε model, the standard k-ω model, the k-kl-ω Transition model, or the RSM) is the effective "Prandtl" number for transport of turbulence kinetic energy σ_k. This effective Prandtl number defines the ratio of the momentum diffusivity to the diffusivity of turbulence kinetic energy via turbulent transport.

TKE (Inner) Prandtl #

(only for the BSL or SST k-ω model, and the Transition SST model) is the effective "Prandtl" number for the transport of turbulence kinetic energy, $\sigma_{k,1}$, inside the near-wall region. See Modeling the Turbulent Viscosity in the Theory Guide for details.

TKE (Outer) Prandtl #

(only for the BSL or SST k-ω model, and the Transition SST model) is the effective "Prandtl" number for the transport of turbulence kinetic energy, $\sigma_{k,2}$, outside the near-wall region. See Modeling the Turbulent Viscosity in the Theory Guide for details.

TDR Prandtl Number

is the effective "Prandtl" number for transport of the turbulent dissipation rate, σ_ε, for the standard or realizable k-ε model or the RSM. This effective Prandtl number defines the ratio of the momentum diffusivity to the diffusivity of turbulence dissipation via turbulent transport.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

SDR (Inner) Prandtl #

(only for the BSL or SST k-ω model, and the Transition SST model) is the effective "Prandtl" number for the transport of the specific dissipation rate, $\sigma_{\omega,1}$, inside the near-wall region. See Modeling the Turbulent Viscosity in the Theory Guide for details.

SDR (Outer) Prandtl #

(only for the BSL or SST k-ω model, and the Transition SST model) is the effective "Prandtl" number for the transport of the specific dissipation rate, $\sigma_{\omega,2}$, outside the near-wall region. See Modeling the Turbulent Viscosity in the Theory Guide for details.

Dispersion Prandtl Number

(only for the k-ε multiphase models) is the effective "Prandtl" number for the dispersed phase, σ_{pq}. See Turbulence Models in the Theory Guide for details.

Energy Prandtl Number

(for any turbulence model except the RNG k-ε model) is the turbulent Prandtl number for energy, Pr_t, in Equation 4.250 in the Theory Guide. (This item will not appear for the LES model if the **Dynamic Energy Flux** option has been enabled, or for premixed or partially premixed combustion models.) For non-premixed and partially premixed combustion, the energy Prandtl Number is set equal to the PDF Schmidt Number.

Wall Prandtl Number

(for all turbulence models) is the turbulent Prandtl number at the wall, Pr_t in Equation 4.343 in the Theory Guide. (This item will not appear for adiabatic premixed combustion or partially premixed combustion models.)

Turbulent Schmidt Number

(for turbulent species transport calculations using any turbulence model except the RNG k-ε model) is the turbulent Schmidt number, Sc_t, in Equation 7.4 in the Theory Guide. (This item will not appear for the LES model if the **Dynamic Scalar Flux** option has been enabled.)

PDF Schmidt Number

(for non-premixed or partially premixed combustion calculations using any turbulence model) is the model constant σ_t in Equation 8.5 in the Theory Guide.

Production Limiter Clip Factor

This coefficient is used by the **Production Limiter**. For details, see Equation 4.436 in the *Fluent Theory Guide*.

User-Defined Transition Correlations

(only for the Transition SST model) allows you to select the user-defined correlations for **F_length**, **Re_thetac**, **Re_thetat**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

User-Defined Functions

allows you to select the user-defined functions for various constants.

Turbulent Viscosity

appears for Spalart Allmaras, k-ε, k-ω, DES, SDES with BSL / SST models, and SBES with BSL / SST. You can select the user-defined functions for turbulent viscosity in the drop-down list.

Prandtl Numbers, Prandtl and Schmidt Numbers

contains a list of relevant Prandtl and Schmidt numbers for which you can select user-defined functions.

TKE Prandtl Number

allows you to select a user-defined function to define the TKE Prandtl number for the standard and realizable k-ε models and the standard k-ω model.

TDR Prandtl Number

allows you to select a user-defined function to define the TDR Prandtl number for the standard and realizable k-ε models.

Energy Prandtl Number

allows you to select a user-defined function to define the turbulent Prandtl number for energy, when energy is enabled for all turbulence models except the RNG k-ε model. (This item will not appear for the LES model if the **Dynamic Energy Flux** option has been enabled.)

Wall Prandtl Number

allows you to select a user-defined function to define the turbulent Prandtl number at the wall for all turbulence models when energy is enabled.

Turbulent Schmidt Number

allows you to select a user-defined function to define the turbulent Schmidt number when species transport is enabled for all turbulence models except the RNG k-ε model. (This item will not appear for the LES model if the **Dynamic Scalar Flux** option has been enabled.)

SDR Prandtl Number

allows you to select a user-defined function to define the SDR Prandtl number for the standard k-ω model.

Subgrid-Scale Turbulent Viscosity

allows you to select a user-defined function for the subgrid-scale turbulent viscosity for the LES model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Scale-Resolving Simulation Options

allows you to combine two turbulence models, which are applied on the appropriate regions of the flow domain. For additional information, see Scale-Resolving Simulation (SRS) Models (p. 2073).

Scale-Adaptive Simulation (SAS)

allows you to apply SAS in combination with the following ω-based URANS models: the Standard or BSL k-ω model, the Transition SST model, and the ω-based Reynolds stress models. For additional information, see Setting Up Scale-Adaptive Simulation (SAS) Modeling (p. 2109).

Detached Eddy Simulation (DES)

allows you to apply DES in combination with the BSL k-ω model and Transition SST model. For additional information, see Detached Eddy Simulation (DES) (p. 2075).

Stress Blending (SBES) / Shielded DES

allows you to apply SBES or SDES in combination with the BSL k-ω model, SST k-ω model, and Transition SST model. See Stress-Blended Eddy Simulation (SBES) and Shielded Detached Eddy Simulation (SDES) in the Theory Guide for details.

Shielding Functions

allows you to select the shielding functions (**SST F1 Function**, **SST F2 Function**, **DDES**, and **IDDES**) for the Detached Eddy Simulation model with the BSL k-ω model, SST k-ω model, and Transition SST model. (see Shielding Functions for the BSL / SST / Transition SST Detached Eddy Simulation Model (p. 2143) for details).

SBES / SDES Options

allows you to define the following settings as part of the SBES or SDES model:

Hybrid Model

specifies that the hybrid RANS-LES model used is either **SDES** (Shielded DES model), **SBES** (Stress-Blended Eddy Simulation model), or **SBES with User-Defined Function**. See Including the SDES or SBES Model with RANS Models (p. 2140) for details.

Subgrid-Scale Model

contains options for the subgrid-scale model used in the LES region of your Stress-Blended Eddy Simulation. This portion of the dialog box will appear only if **SBES** or **SBES with User-Defined Function** is selected as the **Hybrid Model**.

Smagorinsky-Lilly

selects the Smagorinsky-Lilly subgrid-scale model described in Subgrid-Scale Models in the Theory Guide.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Dynamic Smagorinsky

selects the Smagorinsky-Lilly subgrid-scale model (described in Subgrid-Scale Models in the Theory Guide) with the dynamic stress model enabled.

WALE

selects the Wall-Adapting local Eddy-Viscosity model described in Wall-Adapting Local Eddy-Viscosity (WALE) Model in the Theory Guide.

WMLES S-Omega

selects the algebraic Wall-Modeled LES S-Ω model described in Algebraic WMLES S-Omega Model Formulation in the Theory Guide.

Update Interval k-omega

Sets the number of time steps between updates of the k-ω part of the SBES model.

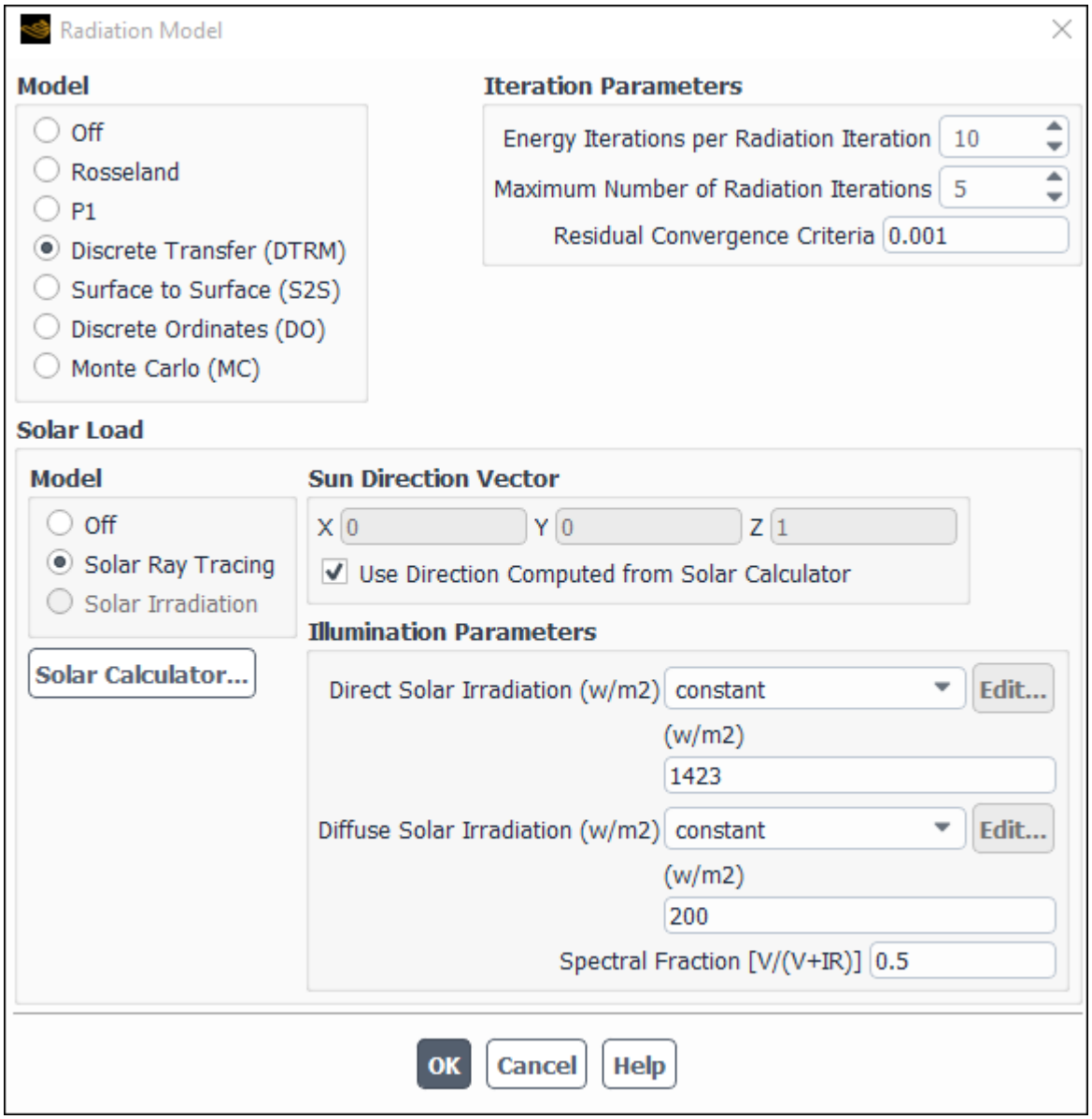
Turbulence Damping Options

is a group box where you can enable **Turbulence Damping** to include the turbulence damping effects in multiphase simulations, which is required for better resolution of velocity gradients in the vicinity of fluid-fluid interface. When this option is enabled, you will need to specify **Damping Factor** (default = 10). This item is available for the VOF and Mixture models and for the Eulerian multiphase model with the **Multi-Fluid VOF** model selected. See Turbulence Damping (p. 2136) for details.

51.4.4. Radiation Model Dialog Box

The **Radiation Model** dialog box allows you to select a model for radiation heat transfer and set the associated parameters. See Using the Radiation Models (p. 2193) – Defining Non-Gray Radiation for the DO Model (p. 2212) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Model

indicates which model, if any, is used to calculate radiation heat transfer. See Modeling Radiation (p. 2192) for details about modeling radiation heat transfer.

Off

disables the calculation of radiation heat transfer.

Rosseland

enables the Rosseland radiation model.

P1

enables the P-1 radiation model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Discrete Transfer (DTRM)

enables the discrete transfer radiation model (DTRM).

Surface to Surface (S2S)

enables the surface-to-surface (S2S) radiation model.

Discrete Ordinates (DO)

enables the discrete ordinates (DO) radiation model.

Monte Carlo (MC)

enables the Monte Carlo (MC) radiation model.

DO/Energy Coupling

allows you to couple the energy and DO intensity equations at each cell, solving them simultaneously. See Defining Non-Gray Radiation for the DO Model (p. 2212) for details.

Iteration Parameters

contains parameters related to the DTRM, S2S, and the DO models. This portion of the dialog box will appear only if **Discrete Transfer (DTRM)**, **Surface to Surface (S2S)**, or **Discrete Ordinates (DO)** is selected as the **Model**.

Energy Iterations per Radiation Iteration

controls the frequency with which the radiation terms are updated as the continuous phase solution proceeds. The **Energy Iterations per Radiation Iteration** parameter is set to 10 by default. This implies that the radiation calculation is performed once every 10 iterations of the solution process. Increasing the number can speed the calculation process, but may slow overall convergence.

Maximum Number of Radiation Iterations

controls the maximum number of iterations of the radiation calculation during each global iteration. The default setting of 5 means that the radiosity will be updated up to 5 times. The actual number of iterations will be less if the residual convergence criterion is exceeded at any point during these iterations.

This item appears only when **Discrete Transfer** or **Surface to Surface** is selected as the **Model**.

Residual Convergence Criteria

determines when the radiation intensity update is converged. It is defined as the maximum normalized change in the surface intensity from one radiation iteration to the next (see Equation 16.4 (p. 2242)).

This item appears only when **Discrete Transfer** or **Surface to Surface** is selected as the **Model**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Target Number of Histories

determines the number of photon histories to be tracked for the Monte Carlo simulation. Too small of a number will produce "speckled" results. Increasing the target number of histories produces a smoother and more accurate solution, but at the expense of higher computation effort.

This item appears only when **Monte Carlo** is selected.

View Factors and Clustering

contains parameters related to the S2S model. This portion of the dialog box will appear only if **Surface to Surface** is selected as the **Model**. See Setting Up the S2S Model (p. 2199) for more information about the use of these parameters.

Settings...

opens the View Factors and Clustering Dialog Box (p. 4741), in which you can set parameters related to surface clusters and view factors.

Compute/Write/Read...

allows you to compute the view factors, write the computed view factors to a file, and read the file into Ansys Fluent. When you click **Compute/Write/Read...**, The Select File Dialog Box (p. 931) will open so that you can specify a name for the file where Ansys Fluent should save the view factors after computing them, and indicate whether the files should be saved in binary format.

Read Existing File...

opens The Select File Dialog Box (p. 931), in which you can specify the file from which Ansys Fluent should read view factors.

Angular Discretization

contains parameters for angular discretization and pixelation for the DO model. This portion of the dialog box will appear only if **Discrete Ordinates** is selected as the **Model**. See Setting Up the DO Model (p. 2211) for more information about the use of these parameters.

Theta Divisions, Phi Divisions

define the number of control angles used to discretize each octant of the angular space (see Figure 5.3: Angular Coordinate System in the Theory Guide).

Theta Pixels, Phi Pixels

are used to control the pixelation that accounts for any control volume overhang (see Figure 5.7: Pixelation of Control Angle in the Theory Guide and the figures and discussion preceding it).

Radiation Mesh Options

This portion of the dialog box will appear only if the **Monte Carlo** model is selected.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Target Cells Per Volume Cluster

Coarsens the radiation mesh used by the MC model. See Setting Up the MC Model (p. 2214) for more information about the use of this parameter.

Non-Gray Model

contains parameters related to the non-gray P-1, DO, or MC model. This portion of the dialog box will appear only if **P1**, **Discrete Ordinates**, or **Monte Carlo** is selected as the **Model**. See Setting Up the P-1 Model with Non-Gray Radiation (p. 2194), Defining Non-Gray Radiation for the DO Model (p. 2212), or Setting Up the MC Model (p. 2214) for more information about the use of these parameters.

Number of Bands

specifies the number of bands for the non-gray radiation.

Wavelength Intervals (n=1)

contains inputs that define each wavelength band. (It appears only when the **Number of Bands** is nonzero.) For each band, you can specify a **Name**, as well as the **Start** and **End** wavelength of the band in μm. Note that the wavelength bands are specified for vacuum (n=1). Ansys Fluent will automatically account for the refractive index in setting band limits for media with n different from unity.

Solar Load

contains parameters related to solar load model that can be used to calculate radiation effects from the sun's rays that enter a computational domain. It is available only for the 3D version of Ansys Fluent. See Solar Load Model (p. 2247) for more information about the use of these parameters.

Model

indicates which model is used to calculate radiation effects.

Off

disables the calculation of solar radiations.

Solar Ray Tracing

enables the solar ray tracing algorithm

DO Irradiation

enables the DO irradiation option. Select **Discrete Ordinates** under **Model** before selecting this option.

Solar Calculator

opens the Solar Calculator Dialog Box (p. 4747).

Sun Direction Vector

contains the components of sun direction vector.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

X, Y, Z

are the components of the sun direction vector.

Use Direction Computed from Solar Calculator

enables the use of direction vector computed from the solar calculator.

Illumination Parameters

contains illumination options.

Direct Solar Irradiation

is the amount of energy per unit area due to direct solar irradiation.

Diffuse Solar Irradiation

is the amount of energy per unit area due to diffuse solar irradiation.

Spectral Fraction

is the fraction of incident solar radiation in the visible part of the solar radiation spectrum. The spectral fraction is not used for DO irradiation since the DO implementation is intended only for a single band. This parameter is available only for **Solar Ray Tracing**.

Update Parameters

contains update parameters for transient simulations.

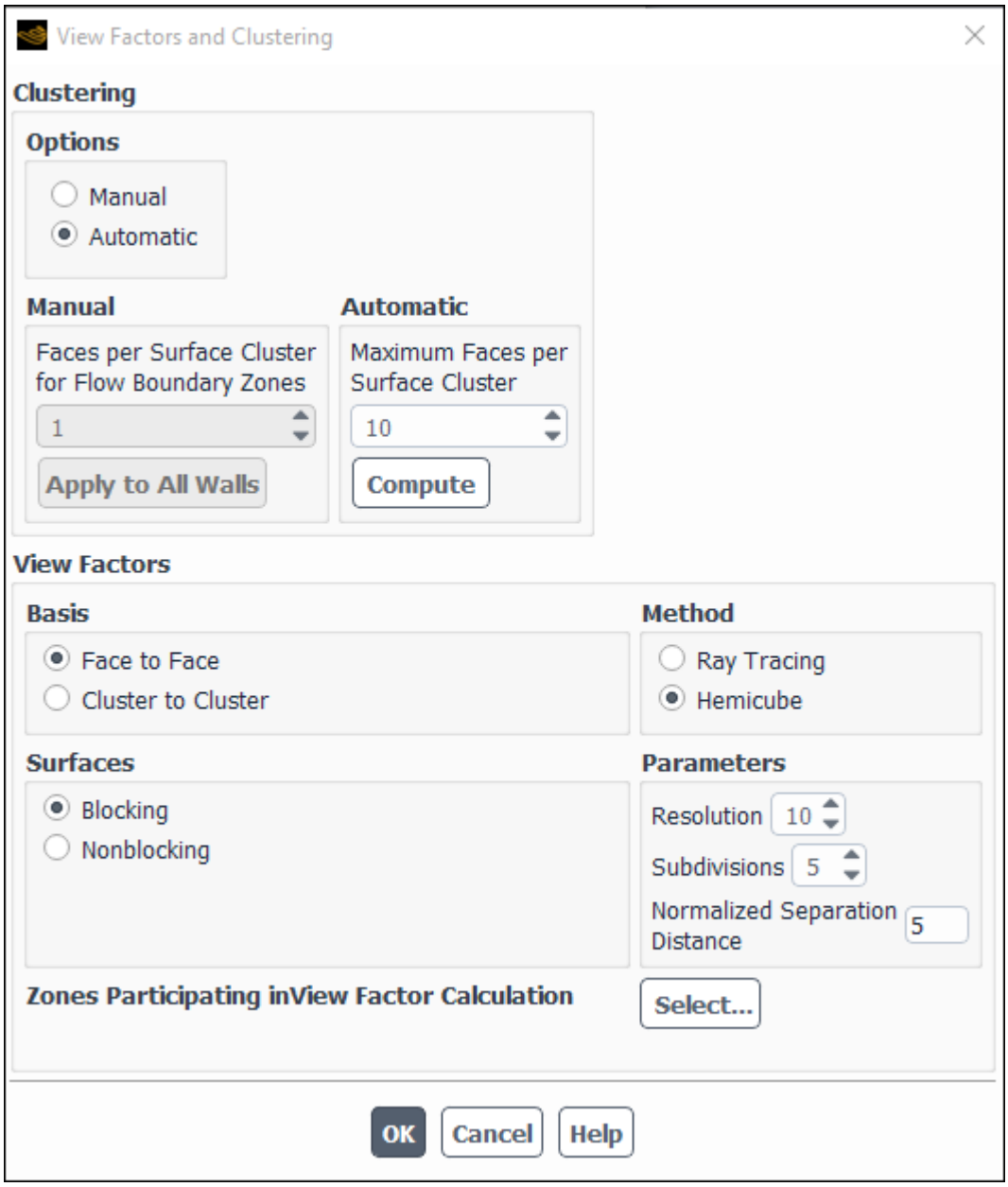
Time Steps per Solar Load Update

specifies the number of time steps that will direct the Ansys Fluent solver to update the solar load data for the specified flow-time intervals in the unsteady solution process.

51.4.5. View Factors and Clustering Dialog Box

The **View Factors and Clustering** dialog box allows you to set parameters related to surface clusters and view factors for the surface-to-surface radiation model. See Setting Up the S2S Model (p. 2199) for information about using this dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Clustering

contains the methods and settings for creating surface clusters. See Forming Surface Clusters (p. 2201) for information about surface cluster settings.

Options

gives you a choice of forming clusters either manually or automatically.

Manual

allows you to form clusters manually.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Automatic

allows Ansys Fluent to form the cluster automatically.

Manual

(available only when the **Manual** option is enabled) allows you to set the faces per surface cluster (FPSC) value for walls and inlet and outlet boundaries.

Faces per Surface Cluster for Flow Boundary Zones

specifies the number of faces in each surface cluster for inlet and outlet boundaries (that is, exhaust fan, inlet vent, intake fan, outlet vent, mass-flow inlet, mass-flow outlet, pressure far-field, pressure inlet, pressure outlet, outflow, and velocity inlet boundaries). This FPSC value can also be applied to all walls that are adjacent to fluid zones by clicking the **Apply to All Walls** button. The default is set to `1`. See Clustering in the Theory Guide for details about clustering.

Apply to All Walls

applies the value specified in the **Faces per Surface Cluster for Flow Boundary Zones** field to all walls that are adjacent to fluid zones.

Automatic

(available only when the **Automatic** option is enabled) allows you to assign different faces per surface cluster (FPSC) values to the walls automatically, based on the distance of the walls from and the FPSC values of the walls that are defined as critical.

Maximum Faces per Surface Cluster

specifies the maximum number of faces per surface cluster automatically assigned to non-critical wall zones adjacent to fluid zones. The default is set to `10`.

Compute

results in Ansys Fluent automatically calculating and updating the face per surface cluster values in the boundary conditions dialog box for non-critical wall zones adjacent to fluid zones, without computing the clusters.

View Factors

contains settings for computing the view factors. See Setting Up the View Factor Calculation (p. 2204) for information about view factor calculation settings.

Basis

specifies how surfaces are defined for the calculation of view factors. See Selecting the Basis for Computing View Factors (p. 2204) for more information.

Face to Face

specifies that the surfaces used to calculate the view factors are the boundary faces.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cluster to Cluster

specifies that the surfaces used to calculate the view factors are the clusters defined by the settings in the **Clustering** group box. The cluster to cluster basis is only available for 3D cases.

Surfaces

specifies the geometric orientation of surface pairs with respect to each other when using the hemicube method.

Blocking

specifies that the view factor calculation accounts for surfaces that block the views between the surfaces under consideration.

Nonblocking

specifies that the view factor calculation does not account for surfaces that block the views between the surfaces under consideration.

Method

specifies the method for computing the view factors. See Selecting the Method for Computing View Factors (p. 2205) for information about choosing a method.

Hemicube

specifies the use of the hemicube method for computing the view factors. The hemicube method is available only for 3D and axisymmetric cases, and should not be used if any of the zones are defined as periodic or symmetry boundaries.

Ray Tracing

specifies the use of the ray tracing method for computing the view factors. For 2D cases, the number of rays used with this method is two times the value set for **Resolution** in the **Parameters** group box; for 3D cases, the number of rays is three times the square of the **Resolution** value.

Parameters

contains settings related to the hemicube and ray tracing methods for computing the view factors. All of these inputs are available when **Hemicube** is selected under **Method**, whereas only **Resolution** is available when **Ray Tracing** is selected. See Selecting the Method for Computing View Factors (p. 2205) for information about setting the method parameters.

Resolution

specifies the resolution of the hemicube. The default value is set to `10`. You can increase the value to reduce aliasing effects that can lead to overestimated or underestimated view factors.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Subdivisions

specifies the number of subfaces into which each face is divided. The default value is set to 5. This parameter is only available when the hemicube method is used in conjunction with the face to face basis.

Normalized Separation Distance

specifies the ratio of the minimum face separation to the effective diameter of the face. The default value is set to 5. This parameter is only available for the hemicube method.

Zones Participating in View Factor Calculation

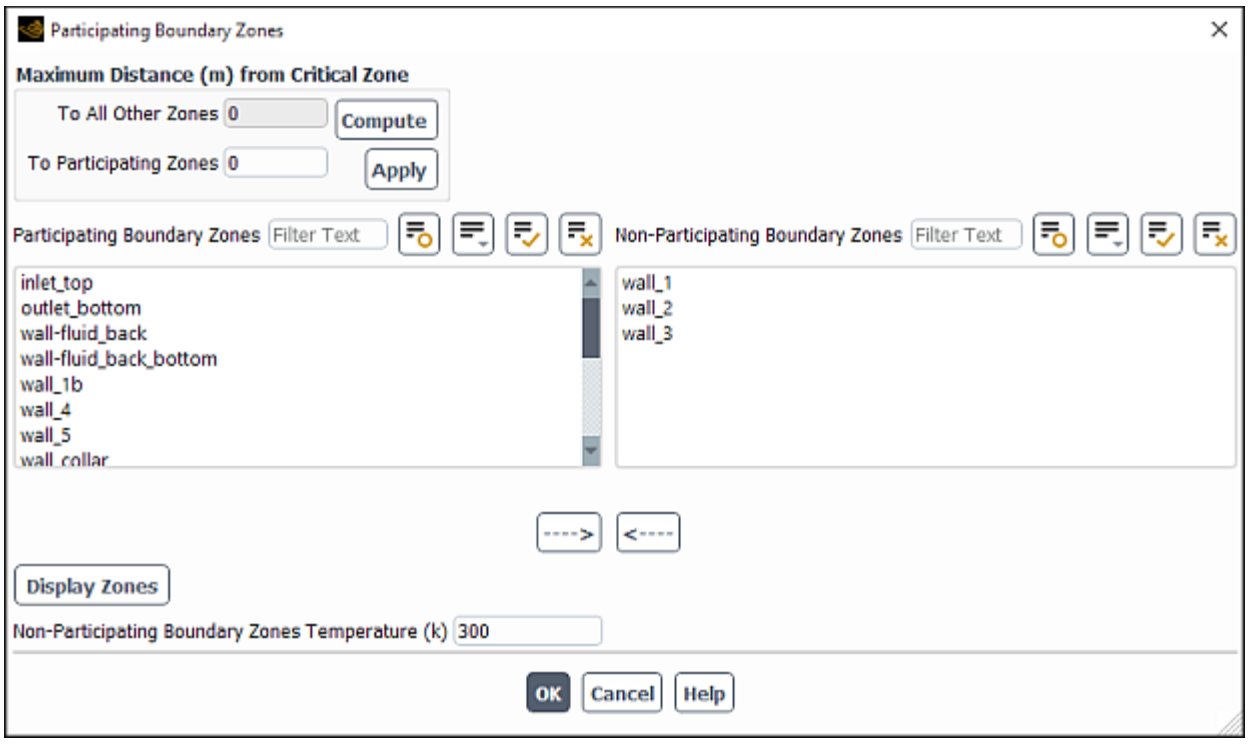
allows you to define which boundary zones participate in the view factor calculation. See Specifying Boundary Zone Participation (p. 2206) for more information.

Select...

opens the Participating Boundary Zones Dialog Box (p. 4745), where you can define which boundary zones participate in the view factor calculation.

51.4.6. Participating Boundary Zones Dialog Box

The **Participating Boundary Zones** dialog box allows you to define which boundary zones participate in the view factor calculation, to display zones in the graphics window, and set the temperature of zones that do not participate in the view factor calculation. See Specifying Boundary Zone Participation (p. 2206) for details. This dialog box opens when you click the **Select...** button next to the **Zones Participating in View Factor Calculation** label in the **View Factors and Clustering** dialog box.



Controls

Maximum Distance from Critical Zone

allows you to view the maximum distance between critical zones and other zones, and set all boundary zones that are located beyond a certain distance from a critical zone as not participating in the view factor calculation.

To All Other Zones

displays the maximum of the distances between the centroids of critical and all other wall, inlet, and exit zones when the **Compute** button is clicked. This field is not editable. This is only available when using the **Automatic** option for clustering.

Compute

calculates the distances between the centroids of critical zones and all the other wall, inlet, and exit zones and displays the maximum value in the **To All Other Zones** field and the **To Participating Zones** text-entry box. It requires the definition of the critical zone.

To Participating Zones

displays the maximum of the distances between the centroids of critical and all other wall, inlet, and exit zones when the **Compute** button is clicked. Unlike the **To All Other Zones** field, you can edit this text-entry box. You can specify the maximum distance allowed between the centroids of critical zones and all other wall, inlet, or exit zones that participate in the view factor calculation; when you click the **Apply** button, all of the zones will be marked as either participating or not participating, according to the distance criteria you specified. Note that this field is only available when using the **Automatic** option for clustering.

Apply

marks all of the wall, inlet, and exit zones as either participating or not participating in the view factor calculation, depending on whether the distance from their centroid to the centroid of a critical zone is equal to or less than value entered for **To Participating Zones**. The **Participating Boundary Zones** and **Non-Participating Boundary Zones** lists will be updated accordingly.

Participating Boundary Zones

shows all the zones that are participating in the view factor calculation. You can select a zone and click the arrow button that points to the right to move the zone to the **Non-Participating Boundary Zones** list.

Non-Participating Boundary Zones

shows all the zones that are not participating in the view factor calculation. You can select a zone and click the arrow button that points to the left to move the zone to the **Participating Boundary Zones** list.

Display Zones

displays any zones selected in the **Participating Boundary Zones** and **Non-Participating Boundary Zones** lists in the graphics window.

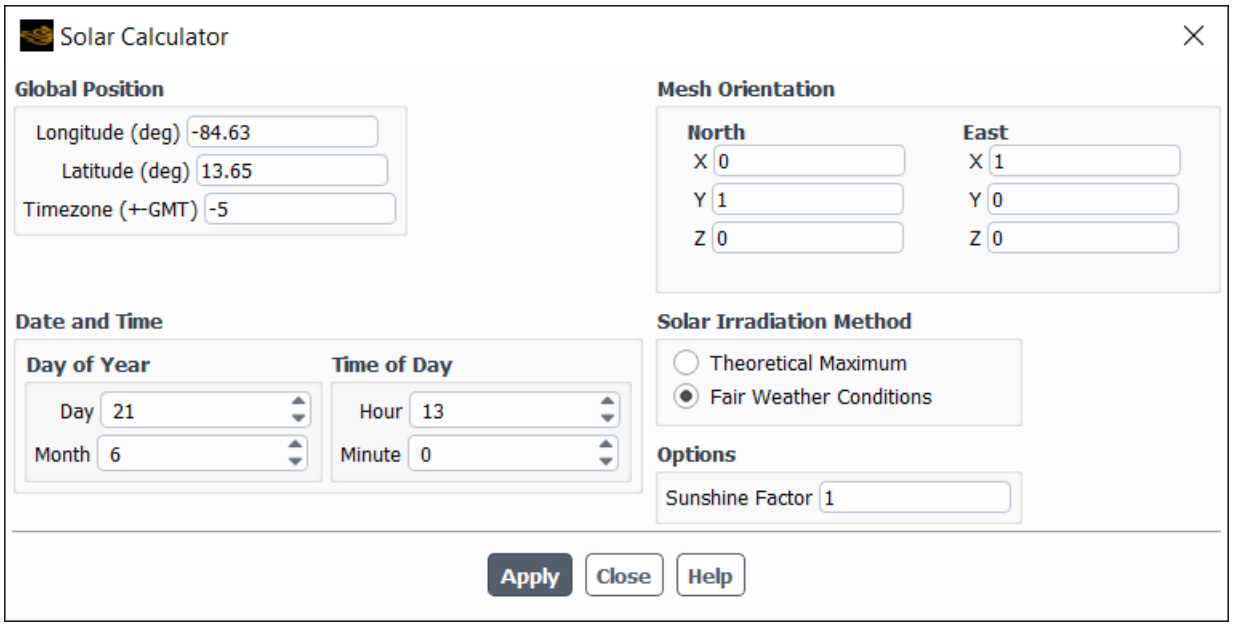
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Non-Participating Boundary Zones Temperature

allows you to specify the temperature of the zones that do not participate in the view factor calculation.

51.4.7. Solar Calculator Dialog Box

The **Solar Calculator** dialog box allows you to set parameters related to the calculation of solar load models. See Solar Load Model (p. 2247) for details. This dialog box opens when you click the **Solar Calculator...** button in the **Radiation Model** dialog box.



Controls

Global Position

contains the parameters to define the position of solar radiation.

Longitude

specifies the longitude of the desired location in degrees. Values may range from -180 to 180, where negative values indicate the Western hemisphere and positive values indicate the Eastern hemisphere.

Latitude

specifies the latitude of the desired location in degrees. Values may range from -90 (South pole) to 90 (North pole) with 0 defined as the equator.

Timezone

specifies the local time zone of the desired location in hours relative to Greenwich Mean Time (+-GMT). This integer value can range from 12 to -12.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Starting Date and Time

contains parameters to specify date and time.

Day of Year

contains parameters to specify day and month.

Time of Day

contains parameters to specify hour and minutes.

Mesh Orientation

specifies the orientation as the vectors for North and East in the CFD grid system of coordinates.

Solar Irradiation Method

contains parameters to choose the solar irradiation method.

Theoretical Maximum

enables theoretically maximum solar irradiation method.

Fair Weather Conditions

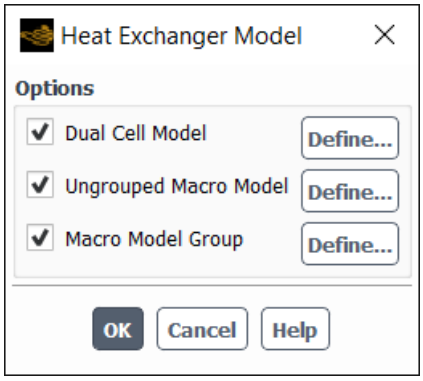
enables fair weather condition solar irradiation method.

Sunshine Factor

is a linear reduction factor for the computed incident load that allows for cloud cover to be accounted for, if appropriate.

51.4.8. Heat Exchanger Model Dialog Box

The **Heat Exchanger Model** dialog box allows you to define a heat exchanger as part of your model. See Modeling Heat Exchangers (p. 2279) for details about using the items below.



Controls

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Options

contains options related to the heat exchanger model. See Choosing a Heat Exchanger Model (p. 2280) for details about the differences between the models associated with these options.

Dual Cell Model

allows you to enable or disable the Dual Cell heat exchanger model. When this option is enabled, the corresponding **Define...** button opens the Dual Cell Heat Exchanger Dialog Box (p. 4749).

Ungrouped Macro Model

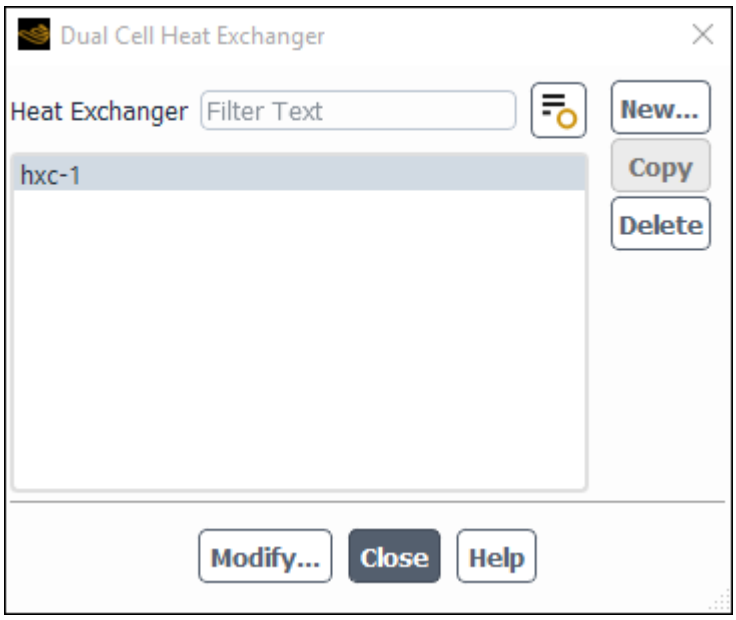
allows you to enable or disable the Ungrouped Macro heat exchanger model. When this option is enabled, the corresponding **Define...** button opens the Ungrouped Macro Heat Exchanger Dialog Box (p. 4756).

Macro Model Group

allows you to enable or disable the Macro heat exchanger group model. When this option is enabled, the corresponding **Define...** button opens the Macro Heat Exchanger Group Dialog Box (p. 4763).

51.4.9. Dual Cell Heat Exchanger Dialog Box

The **Dual Cell Heat Exchanger** dialog box allows you to use the NTU method for heat transfer calculations. This model allows the solution of auxiliary flow on a separate mesh (other than the primary fluid mesh). See Using the Dual Cell Heat Exchanger Model (p. 2283) for information about using this dialog box.



Controls

Heat Exchanger

contains a list of predefined heat exchangers.

New...

opens the Set Dual Cell Heat Exchanger Dialog Box (p. 4750).

Copy

opens the Copy From Dialog Box (p. 4756), which allows you to copy the currently selected heat exchanger.

Delete

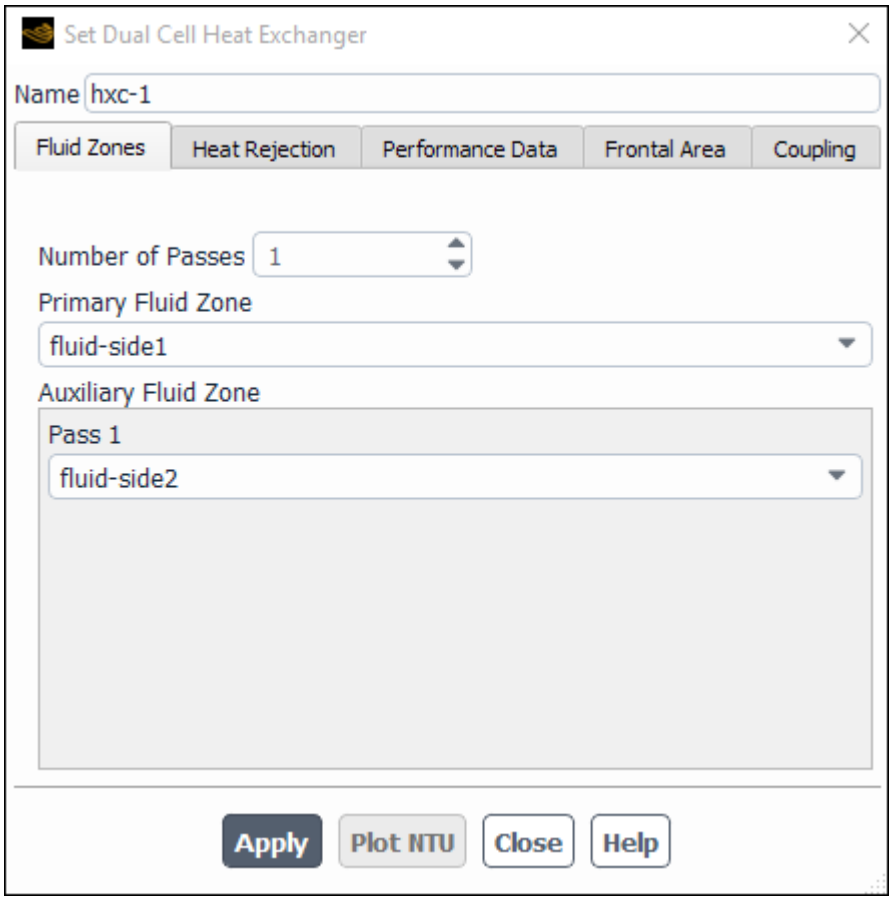
removes the currently selected heat exchanger.

Modify...

opens the Set Dual Cell Heat Exchanger Dialog Box (p. 4750).

51.4.10. Set Dual Cell Heat Exchanger Dialog Box

The **Set Dual Cell Heat Exchanger** dialog box allows you to define the heat exchanger parameters. See Using the Dual Cell Heat Exchanger Model (p. 2283) for details about using this dialog box.



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Controls

Name

allows you to specify a name for the dual cell heat exchanger.

Fluid Zones

allows you to specify the fluid zone parameters for the heat exchanger.

Number of Passes

specifies the number of passes for the heat exchanger.

Primary Fluid Zone

allows you to specify the fluid of the primary fluid zone for the heat exchanger.

Auxiliary Fluid Zone

allows you to specify the fluid for the auxiliary fluid zone, per pass.

Important:

The selected zones must be overlapping in physical space.

Heat Rejection

contains parameters specific to the rejection of heat in the heat exchanger.

Options

allows you to specify how the heat rejection in the heat exchanger is computed.

Fixed Heat Rejection

allows you to specify heat rejection parameters.

Fixed Inlet Temperature

allows you to use total heat rejection as the desired output.

Heat Rejection Targeted

allows you to specify the heat rejection desired from the heat exchanger (available only when the **Fixed Heat Rejection** option is enabled).

Inlet Zone for Temperature Updates

allows Ansys Fluent to change the temperature of the specified inlet zone in order to match the targeted heat rejection (available only when the **Fixed Heat Rejection** option is enabled).

Temperature Update Under-Relaxation

controls convergence (available only when the **Fixed Heat Rejection** option is enabled).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Iteration Interval Between Temperature Updates

controls divergence (available only when the **Fixed Heat Rejection** option is enabled).

Performance Data

contains parameters for specifying the heat exchanger's performance data.

Options

allows you to choose between using raw performance data or NTU performance data.

Raw Data

allows you to specify raw performance data for the heat exchanger.

NTU Data

allows you to specify NTU performance data for the heat exchanger.

Heat Exchanger Performance Data

contains parameters concerning the heat exchanger's performance data.

NTU Table

opens the NTU Table Dialog Box (p. 4755) (available only when the **NTU Data** option is enabled).

Heat Transfer Table

opens the Heat Transfer Data Table Dialog Box (p. 4753) (available only when the **Raw Data** option is enabled).

Effectiveness-NTU Relation

computes the NTU values from the heat transfer data. Choose **cross-flow-unmixed**, **parallel-flow**, or **counter-flow**, all of which are described in NTU Relations (available only when the **Raw Data** option is enabled).

Reference Inlet Temperature

allows you to specify the inlet reference temperature for the primary and the auxiliary fluids. (Available only when the **Raw Data** option is enabled.)

Frontal Area

allows you to specify the frontal area for the heat exchanger.

Primary Fluid

allows you to specify a value for the **Core Frontal Area** for the primary fluid, or to compute the value from a surface zone using the **Compute From** drop-down list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Auxiliary Fluid

allows you to specify a value for the **Core Frontal Area** for the auxiliary fluid, or to compute the value from a surface zone using the **Compute From** drop-down list.

Coupling

specifies parameters when you want to couple the heat exchanger passes.

Temperature

specifies, by default, the mass-weighted average for the temperature of the outlet of Pass 1 to the inlet of Pass 2. Similarly, the mass-weighted-average temperature of the outlet of Pass 2 will be applied at the inlet zone of Pass 3, and so on. (Available only when multiple passes are specified in the **Fluid Zones** tab.)

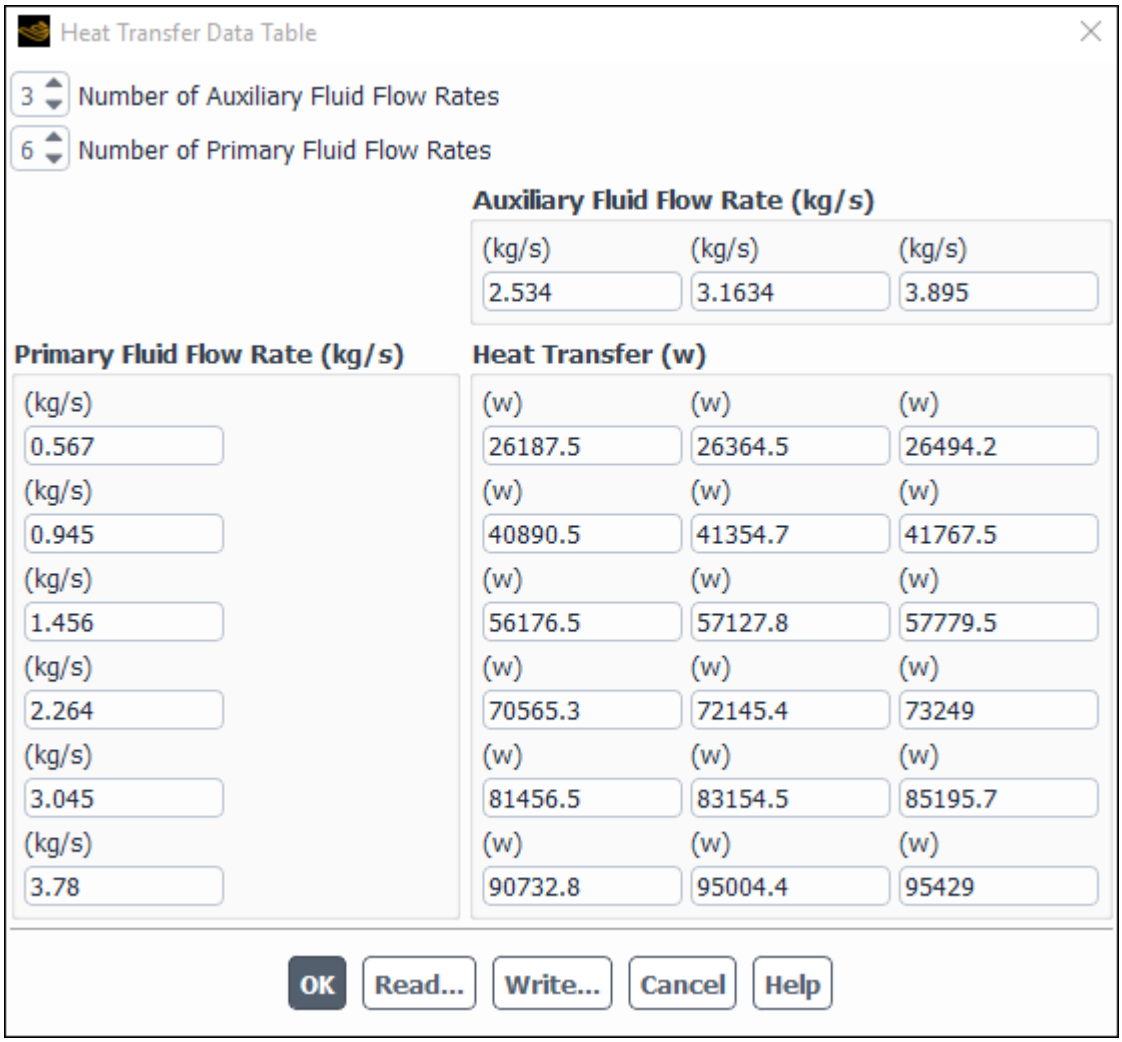
Plot NTU

plots the performance data curve for the selected heat exchanger. The performance data is supplied through the **Performance Data** tab.

51.4.11. Heat Transfer Data Table Dialog Box

The **Heat Transfer Data Table** dialog box contains information on the number of fluid flow rates and heat transfer data for the primary and auxiliary fluids. See Using the Ungrouped Macro Heat Exchanger Model (p. 2294) for details about using this dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Number of Auxiliary Fluid Flow Rates

sets the number of auxiliary fluid flow rates.

Number of Primary Fluid Flow Rates

sets the number of primary fluid flow rates.

Auxiliary Fluid Flow Rate

sets fluid flow rates for the auxiliary fluid.

Primary Fluid Flow Rate

sets fluid flow rates for the primary fluid.

Heat Transfer

sets the heat transfer for the corresponding primary and auxiliary fluid flow rates.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Read...

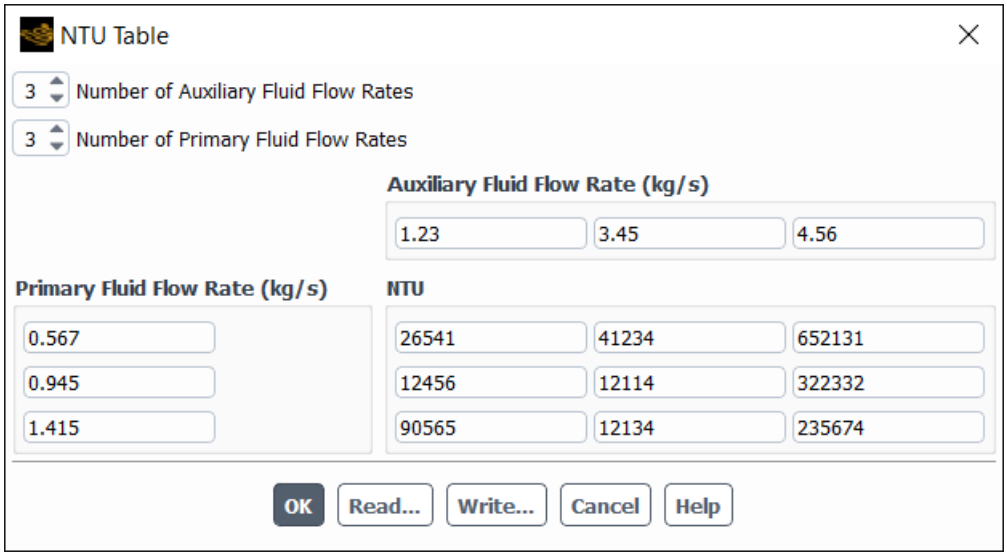
allows you to read in a file containing heat transfer data.

Write...

allows you to write a file containing heat transfer data.

51.4.12. NTU Table Dialog Box

The **NTU Table** dialog box contains information on the number of fluid flow rates and NTU data for the primary and auxiliary fluids. See Using the Ungrouped Macro Heat Exchanger Model (p. 2294) for details about using this dialog box.



Controls

Number of Auxiliary Fluid Flow Rates

sets the number of auxiliary fluid flow rates.

Number of Primary Fluid Flow Rates

sets the number of primary fluid flow rates.

Auxiliary Fluid Flow Rate

sets fluid flow rates for the auxiliary fluid.

Primary Fluid Flow Rate

sets fluid flow rates for the primary fluid.

NTU

sets the NTU values for the corresponding primary and auxiliary fluid flow rates.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Read...

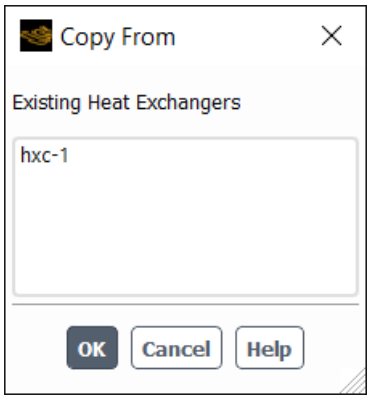
allows you to read in a file containing NTU data.

Write...

allows you to write a file containing NTU data.

51.4.13. Copy From Dialog Box

The **Copy From** dialog box allows you to copy the setup of one heat exchanger to another.



Controls

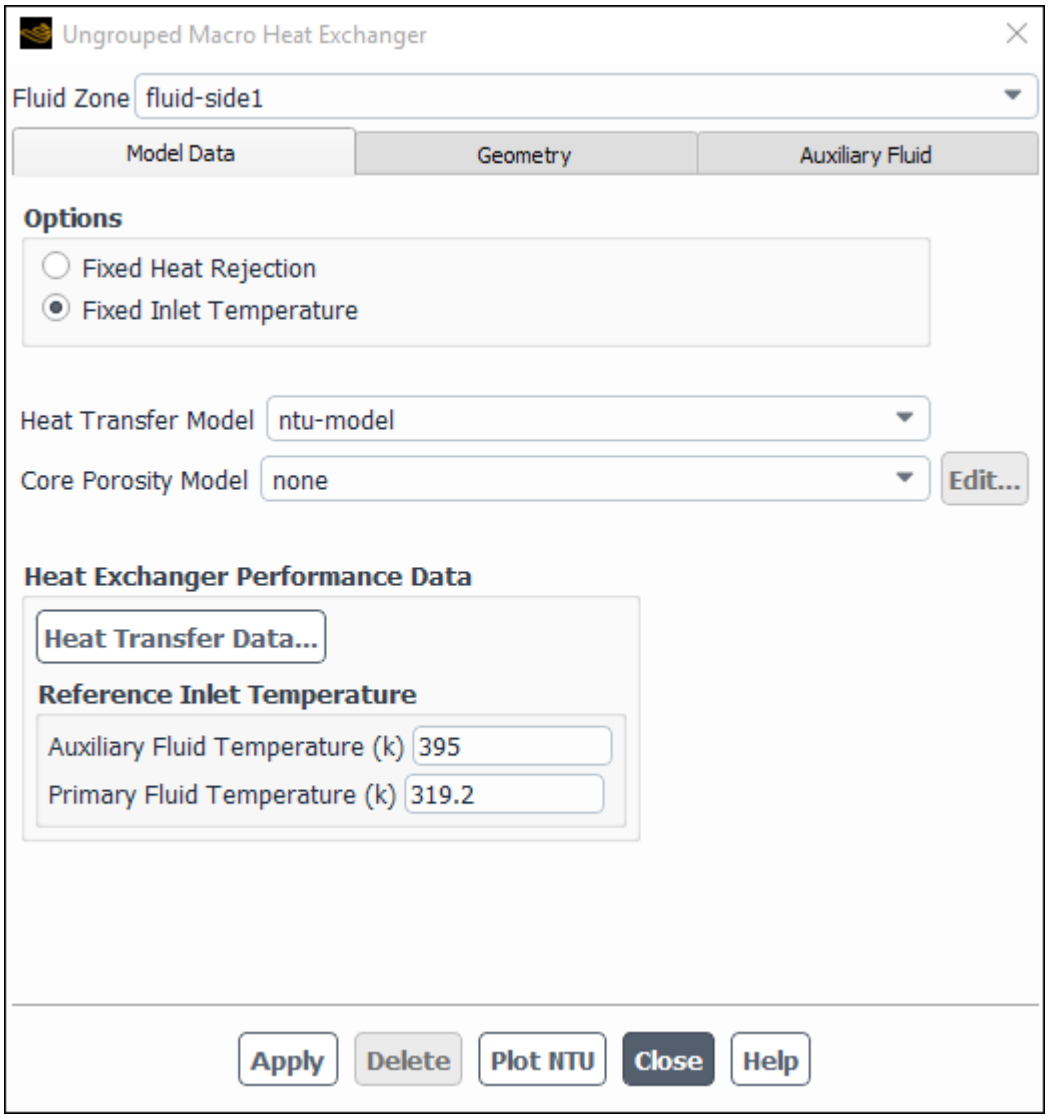
Existing Heat Exchangers

contains a list of heat exchangers, from which you can copy the settings from one heat exchanger to another.

51.4.14. Ungrouped Macro Heat Exchanger Dialog Box

The **Ungrouped Macro Heat Exchanger** dialog box allows you to set up the ungrouped macro heat exchanger model. See Using the Ungrouped Macro Heat Exchanger Model (p. 2294) for details about using this dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Fluid Zone

specifies the zone that represents the heat exchanger.

Model Data

contains the parameters related to the heat exchanger model.

Options

allows you to choose one of the following settings:

Fixed Heat Rejection

specifies that Ansys Fluent should compute the auxiliary fluid inlet temperature for a specified heat rejection.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fixed Inlet Temperature

specifies that Ansys Fluent should compute the total heat rejection of the core for a given inlet auxiliary temperature.

Heat Transfer Model

allows you to specify either the **ntu-model** or the **simple-effectiveness-model** for heat transfer. See Choosing a Heat Exchanger Model (p. 2280) for information on the differences between these models.

Core Porosity Model

contains a drop-down list of all available core porosity models.

Edit...

opens the Core Porosity Model Dialog Box (p. 4761).

Heat Exchanger Performance Data

contains the parameters for heat transfer.

Heat Transfer Data...

opens the Heat Transfer Data Table Dialog Box (p. 4753). This item will appear for the NTU model only.

Auxiliary Fluid Temperature

specifies the auxiliary fluid temperature. This item will appear only for the NTU model.

Primary Fluid Temperature

specifies the gas stream temperature. This item will appear only for the NTU model.

Velocity Effectiveness Curve...

opens the Velocity Effectiveness Curve Dialog Box (p. 4761) in which you can define the effectiveness of the heat exchanger core (ε in Equation 6.11 in the Theory Guide). This item will appear for simple effectiveness model only.

Geometry

contains parameters to define the macro grid.

Width

sets the width of the heat exchanger core. The **Width** is measured in the pass-to-pass direction.

Height

sets the height of the heat exchanger core. The **Height** is measured in the auxiliary fluid inlet direction.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Depth

sets the depth of the heat exchanger core.

Number of Passes

specifies the number of passes for the macro grid. (See Figure 16.76: 1x4x3 Macros (p. 2301).)

Number of Rows/Pass, Number of Columns/Pass

specify the number of macro rows and columns per pass in the macro grid. (See Figure 16.76: 1x4x3 Macros (p. 2301).)

View Passes

displays the macro grid. (This button becomes available after you click **Apply**.) The path of the auxiliary fluid is color-coded in the display: macro 0 is red and macro $n-1$ is blue.

Draw Mesh

toggles between displaying and not displaying the mesh when the macro mesh is displayed (using the **View Passes** button). The Mesh Display Dialog Box (p. 4688) opens when **Draw Mesh** is selected.

Update from Plane Tool

updates the **Auxiliary Fluid Inlet Direction** and **Pass-to-Pass Direction** from the plane tool orientation. The **Width**, **Height**, and **Depth** will also be updated. See Using the Plane Tool (p. 3833) for information about using the plane tool.

Open the Plane Surface Dialog Box (p. 5580) to make the plane tool appear.

Auxiliary Fluid Inlet Direction (height)

specifies the direction in which the auxiliary fluid enters the heat exchanger. See Figure 16.76: 1x4x3 Macros (p. 2301).

Pass-to-Pass Direction (width)

specifies the direction in which the auxiliary fluid moves at the end of each pass through the heat exchanger. See Figure 16.76: 1x4x3 Macros (p. 2301).

Auxiliary Fluid

contains the option to specify the auxiliary fluid properties.

Auxiliary Fluid Properties Method

contains options for specifying auxiliary fluid properties.

constant-specific-heat

allows you to specific a constant value for the auxiliary fluid specific heat.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

user-defined-enthalpy

allows you to specify a user-defined function for the auxiliary fluid enthalpy.

Auxiliary Fluid Specific Heat

specifies the value of $c_{p,auxiliary}$ in Equation 6.17 in the Theory Guide. This value is specified only if **constant-specific-heat** is selected.

Auxiliary Fluid Enthalpy UDF

allows you to specify a user-defined function for the auxiliary fluid enthalpy (see Equation 6.17 in the Theory Guide). This option is available only when **user-defined-enthalpy** is selected.

Auxiliary Fluid Flow Rate

sets the flow rate of the auxiliary fluid ($\dot{m}$ in Equation 6.16 in the Theory Guide).

Heat Rejection

sets the total heat rejection (q_{total} in Equation 6.15 in the Theory Guide). This value is specified only if **Fixed Heat Rejection** is selected.

Initial Temperature

sets an initial guess for the inlet temperature (T_{in} in Equation 6.11 and Equation 6.16 in the Theory Guide). This value is specified only if **Fixed Heat Rejection** is selected.

Inlet Temperature

sets the auxiliary fluid initial temperature (T_{in} in Equation 6.11 and Equation 6.16 in the Theory Guide). This value is specified only if **Fixed Inlet Temperature** is selected in the **Model Data** tab.

Inlet Pressure

sets the auxiliary fluid inlet pressure. This value is specified only if **user-defined-enthalpy** is selected.

Inlet Quality

specifies the value of x in Equation 6.20 in the Theory Guide. This value is specified only if **user-defined-enthalpy** is selected.

Pressure Drop

specifies the value of p_{in} in Equation 6.21 in the Theory Guide. This value is specified only if **user-defined-enthalpy** is selected.

Apply

saves all the settings for the heat exchanger specified in the **Fluid Zone** list.

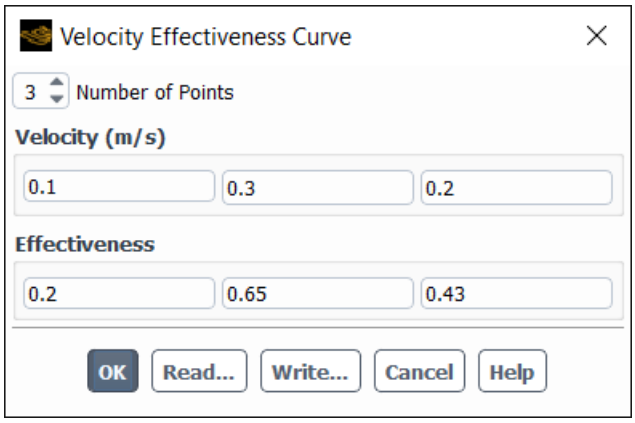
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Delete

deletes the heat exchanger specified in the **Fluid Zone** list.

51.4.15. Velocity Effectiveness Curve Dialog Box

The **Velocity Effectiveness Curve** dialog box allows you to define effectiveness curve. It is opened by clicking **Velocity Effectiveness Curve...** in the Ungrouped Macro Heat Exchanger Dialog Box (p. 4756).



Controls

Number of Points

specifies the number of data pairs in the effectiveness profile. The default value of 1 indicates a constant effectiveness.

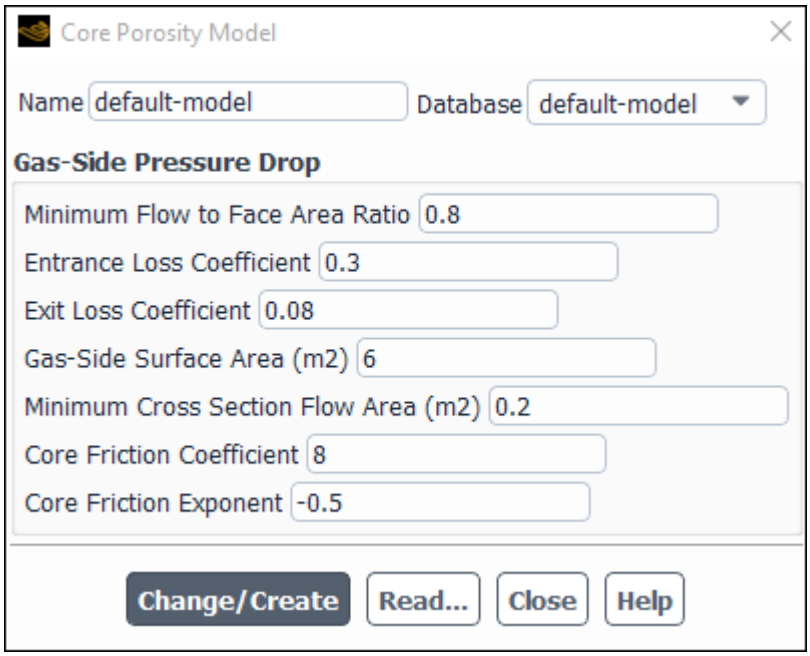
Velocity, Effectiveness

specify the data pairs for the effectiveness profile. These items are available only if the simple effective model has been selected. Note that the **Effectiveness** values must be within the range of 0–1.

51.4.16. Core Porosity Model Dialog Box

The **Core Porosity Model** dialog box allows you to modify or define a heat exchanger core model. This dialog box opens when you click **Edit...** in the Ungrouped Macro Heat Exchanger Dialog Box (p. 4756).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name

specifies the name of a new heat exchanger core model.

Database

contains a drop-down list of all heat exchanger core models that are currently available.

Gas-Side Pressure Drop

contains parameters that define the air-side pressure drop.

Minimum Flow to Face Area Ratio

sets the value of σ in Equation 6.2 in the Theory Guide.

Entrance Loss Coefficient

sets the value of K_c in Equation 6.2 in the Theory Guide.

Exit Loss Coefficient

sets the value of K_e in Equation 6.2 in the Theory Guide.

Gas Side Surface Area

sets the value of A in Equation 6.2 in the Theory Guide.

Minimum Cross Section Flow Area

sets the value of A_c in Equation 6.2 in the Theory Guide.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Core Friction Coefficient

sets the value of a in Equation 6.3 in the Theory Guide.

Core Friction Exponent

sets the value of b in Equation 6.3 in the Theory Guide.

Change/Create

saves the settings in the dialog box and adds the new model to the **Database** list.

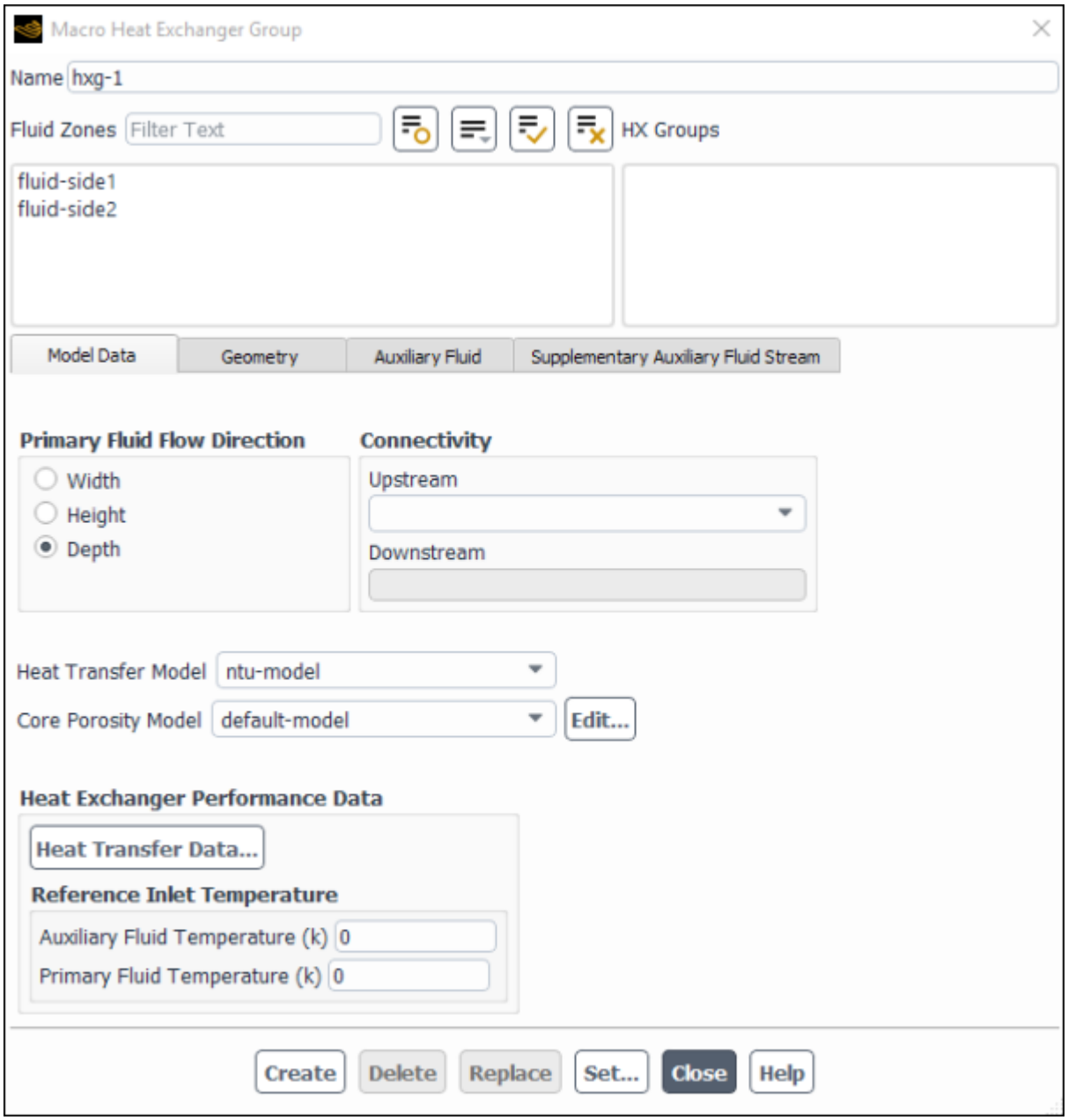
Read...

opens The Select File Dialog Box (p. 931), in which you can select an external file containing a predefined heat exchanger core model.

51.4.17. Macro Heat Exchanger Group Dialog Box

The **Macro Heat Exchanger Group** dialog box allows you to modify or define a heat exchanger core group.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name

specifies the name of a new heat exchanger group.

Fluid Zones

contains a list of all the fluid zones.

HX Groups

contains list of the heat exchanger groups.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Model Data

contains all the parameters to be specified for the model.

Primary Fluid Flow Direction

gives you a choice of gas flow direction.

Width, Height, Depth

specifies the width, height and the depth of the gas flow direction.

Connectivity

allows you to define the upstream and downstream connections.

Upstream

specifies the upstream heat exchanger group.

Downstream

specifies the downstream heat exchanger group.

Heat Transfer Model

allows you to select either the **simple-effectiveness-model** or the **ntu-model**. See Choosing a Heat Exchanger Model (p. 2280) for information on the differences between these models.

Core Porosity Model

specifies whether default values are chosen for the core porosity model.

Edit...

opens the Core Porosity Model Dialog Box (p. 4761) for the definition of a new core porosity model.

Heat Exchanger Performance Data

contains the parameters for heat transfer.

Heat Transfer Data...

opens the Heat Transfer Data Table Dialog Box (p. 4753). This dialog box allows you to define the heat transfer for different primary and auxiliary fluid flow rates. This item will appear for the NTU model only.

Velocity Effectiveness Curve...

opens the Velocity Effectiveness Curve Dialog Box (p. 4761) in which you can define the effectiveness of the heat exchanger core (ε in Equation 6.11 in the Theory Guide). This item will appear for simple effectiveness model only.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Auxiliary Fluid Temperature

specifies the auxiliary fluid temperature. This item will appear for the NTU model only.

Primary Fluid Temperature

specifies the gas stream fluid temperature. This item will appear for the NTU model only.

Geometry

contains parameters to define the macro grid.

Width, Height, Depth

specifies the width, height and the depth of the heat exchanger.

Number of Passes

specifies the number of passes.

Number of Rows/Pass

specifies the number of rows per pass.

Number of Columns/Pass

specifies the number of columns per pass.

View Passes

displays the macro grid. (This button becomes available after you click **Set**.) The path of the auxiliary fluid is color-coded in the display: macro 0 is red and macro $n-1$ is blue.

Draw Mesh

toggles between displaying and not displaying the mesh when the macro mesh is displayed (using the **View Passes** button). The Mesh Display Dialog Box (p. 4688) opens when **Draw Mesh** is selected.

Update from Plane Tool

updates the **Auxiliary Fluid Inlet Direction** and **Pass-to-Pass Direction** from the plane tool orientation. The **Width**, **Height**, and **Depth** will also be updated. See Using the Plane Tool (p. 3833) for information about using the plane tool.

Auxiliary Fluid Inlet Direction

specifies the direction in which the auxiliary fluid enters the heat exchanger. See Figure 16.76: 1x4x3 Macros (p. 2301).

Pass-to-Pass Direction

specifies the direction in which the auxiliary fluid moves at the end of each pass through the heat exchanger. See Figure 16.76: 1x4x3 Macros (p. 2301).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Auxiliary Fluid

contains inputs to specify the properties of the auxiliary fluid.

Properties Method

specifies the method to specify the auxiliary fluid properties. You can choose from **constant-specific-heat** and **user-defined-enthalpy**.

Specific Heat

sets the specific heat of the auxiliary fluid (c_p in Equation 6.16 you choose **constant-specific-heat**.

Enthalpy UDF

sets the enthalpy as defined by the user-defined function selected from the drop-down list.

Auxiliary Fluid Flow Rate

sets the flow rate of the auxiliary fluid ($\dot{m}$ in Equation 6.16 in the Theory Guide).

Initial Temperature

sets an initial guess for the inlet temperature (T_{in} in Equation 6.11 and Equation 6.16 in the Theory Guide). This value is specified only if **Fixed Heat Rejection** is selected.

Inlet Pressure

sets the auxiliary fluid inlet pressure. This value is specified only if **user-defined-enthalpy** is selected.

Inlet Quality

specifies the value of x in Equation 6.20 in the Theory Guide.

Pressure Drop

specifies the value of p_{in} in Equation 6.21 in the Theory Guide. This value is specified only if **user-defined-enthalpy** is selected.

Supplementary Auxiliary Fluid Stream

specifies properties of the supplementary auxiliary stream.

Supplementary Mass Flow Rate

specifies the supplementary fluid flow rate as **constant**, **polynomial** or **piecewise-linear**.

Supplementary Flow Temperature

specifies the supplementary fluid temperature as **constant**, **polynomial** or **piecewise-linear**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Supplementary Flow Quality

specifies the supplementary fluid quality.

Create

saves all the settings in the dialog box.

Delete

deletes the group that is selected in the **HX Groups** list.

Replace

changes the parameters of the already existing group that is selected in the **HX Groups** list.

Set...

opens the Ungrouped Macro Heat Exchanger Dialog Box (p. 4756), in which you can define a new heat exchanger core model or read one from an external file.

51.4.18. Species Model Dialog Box

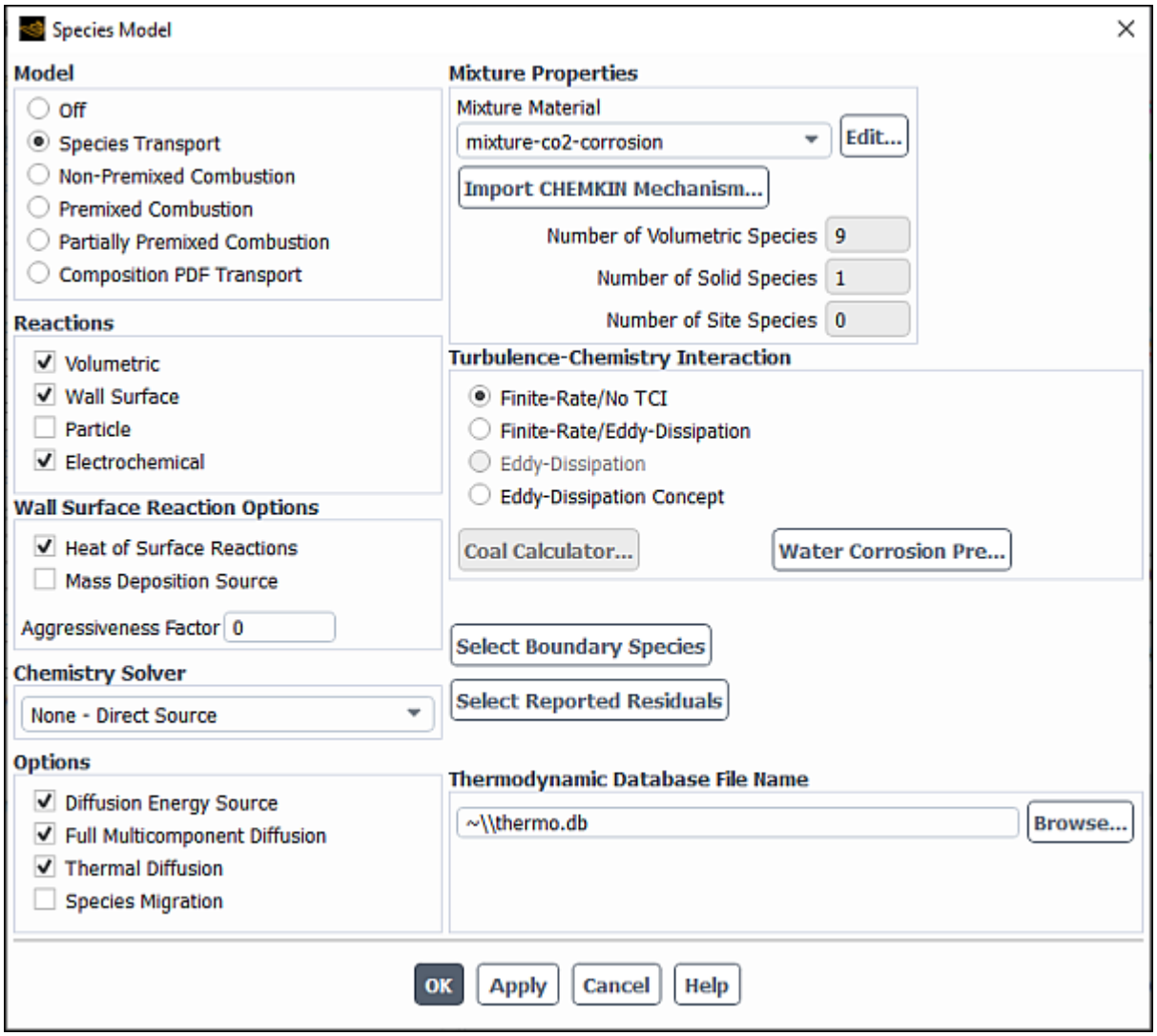
Important:

This feature offers reduced functionality when running Fluent under the **Pro** capability level.

The **Species Model** dialog box allows you to set parameters related to the calculation of species transport and combustion. For details about the items below, see the following sections:

- Enabling Species Transport and Reactions and Choosing the Mixture Material (p. 2359)
- Manual Inputs for Wall Surface Reactions (p. 2403)
- Particle Reactions (p. 2406)
- Using the Premixed Combustion Model (p. 2536)
- Using the Partially Premixed Combustion Model (p. 2543)
- Steps for Using the Composition PDF Transport Model (p. 2442)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Model

indicates which model, if any, is used to calculate species transport/combustion.

Off

disables species calculations.

Species Transport

enables the calculation of multi-species transport (either non-reacting or reacting, depending on the selection for **Reactions**). See Modeling Species Transport and Finite-Rate Chemistry (p. 2355) for details.

Non-Premixed Combustion

enables the calculation of turbulent reacting flow using the non-premixed combustion model. See Modeling Non-Premixed Combustion (p. 2471) for details. This option is available only for turbulent flows using the pressure-based solver.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Premixed Combustion

enables the premixed turbulent combustion model. See Modeling Premixed Combustion (p. 2536) for details. This option is available only for turbulent flows using the pressure-based solver.

Partially Premixed Combustion

enables the partially premixed turbulent combustion model. See Modeling Partially Premixed Combustion (p. 2543) for details. This option is available only for turbulent flows using the pressure-based solver.

Composition PDF Transport

enables the composition PDF transport model. See Modeling a Composition PDF Transport Problem (p. 2442) for details. This option is available only for turbulent flows using the pressure-based solver.

Reactions

contains options related to the modeling of reacting flow. (This section of the dialog box appears only when **Species Transport** or **Composition PDF Transport** is the specified **Model**.)

Volumetric

enables the calculation of reacting flow using the finite-rate formulation. See Volumetric Reactions (p. 2356) for details.

Wall Surface

enables the calculation of wall surface reactions. See Wall Surface Reactions and Chemical Vapor Deposition (p. 2400) for details. This item will appear only if **Volumetric** is enabled.

Particle

enables the calculation of particle reactions. See Particle Reactions (p. 2406) for details. This item will appear only if **Volumetric** is enabled.

Electrochemical

enables the calculation of electrochemical reactions. See Electrochemical Reactions (p. 2414) for details. This item will appear only if **Volumetric** is enabled.

Integration Parameters...

is a command button that opens the Integration Parameters Dialog Box (p. 4795). This button appears for the species transport model, when **Volumetric** is enabled under **Reactions** and **Stiff Chemistry Solver** is enabled under **Options** or when **Eddy-Dissipation Concept** is enabled under **Turbulence-Chemistry Interaction**.

Wall Surface Reaction Options

contains additional options for wall surface reactions. This portion of the dialog box appears only if **Wall Surface** is enabled under **Reactions**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Heat of Surface Reactions

(if enabled) includes the heat release due to surface reactions in the energy equation. You must remember to set appropriate formation enthalpies (standard state enthalpies) if you enable this option.

Mass Deposition Source

when enabled, includes the effect of surface mass transfer in the continuity equation. This option is always enabled when the density-based solver is used. With the density-based solver, the effect of surface mass transfer in the continuity equation is always included.

Aggressiveness Factor

is a numerical factor that controls the robustness and the convergence speed. This value ranges between 0 and 1, where 0 is the most robust, but results in the slowest convergence. The default value for the **Aggressiveness Factor** is 0.5. This parameter appears only when the pressure-based solver is used.

Chemistry Solver

is a drop-down list in which you can select the solver for the chemically reactive flow simulation. This item appears only for **Species Transport** and **Composition PDF Transport** models involving **Volumetric** reactions.

Stiff Chemistry Solver

enables efficient integration of stiff chemical kinetics. This item appears only when **Finite-Rate/No TCI** or **Eddy-Dissipation Concept** is selected in the **Turbulence-Chemistry Interaction** group box. See Solution of Stiff Chemistry Systems (p. 2397) for details.

CHEMKIN-CFD Solver

enables the integration of chemical kinetics using the Ansys CHEMKIN-CFD Solver, designed for large, stiff chemistry mechanisms. This item appears only when **Finite-Rate/No TCI** or **Eddy-Dissipation Concept** is selected in the **Turbulence-Chemistry Interaction** group box.

Relax to Chemical Equilibrium

enables efficient modeling of chemical kinetics effects based only on equilibrium calculation and assumptions about characteristic timescales for approaching equilibrium. This item is not available for the **Eddy Dissipation Concept TCI** option. (See The Relaxation to Chemical Equilibrium Model for details.

None - Direct Source

makes explicit use of chemistry source terms in the species transport equations, without a stiff-chemistry solver (not recommended for stiff or complex chemistry).

Options

contains additional options for the **Species Transport** and **Composition PDF Transport** models only. This group box will not appear in the dialog box for the other models.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Diffusion Energy Source

(if enabled) includes the effect of enthalpy transport due to species diffusion in the energy equation.

Full Multicomponent Diffusion

enables the full multicomponent diffusion model. See Full Multicomponent Diffusion in the *Fluent Theory Guide* for details.

Thermal Diffusion

enables the thermal diffusion model. See Thermal Diffusion Coefficients in the *Fluent Theory Guide* for details.

Liquid Micro-Mixing

is used to model liquid reactions. When the **Liquid Micro-Mixing** model is invoked, Ansys Fluent uses the **volume-weighted-mixing-law** formula to calculate the density.

Species Migration

(if enabled) includes the ion species migration term in the species transport equation. This item appears only when **Electrochemical** reactions are enabled.

Thickened Flame Model

enables the modeling of laminar flames. This application is typically used as an LES combustion model for turbulent premixed and partially-premixed flames.

Include Temperature Fluctuations

enables the calculation of the multi-mode energy equation. This option is available when the composition PDF transport model is selected.

Mixture Properties

contains controls and information about the mixture being modeled. This section of the dialog box will not appear if **Premixed Combustion** is the selected under **Model**.

Mixture Material

contains a drop-down list of available mixture materials. When you first enable the **Species Transport** model, you can choose from all of the mixture materials defined in the database, or you can choose a "template" and define your own material. (Click **View...** to open the Fluent Database Materials Dialog Box (p. 4897) and check the properties of the mixture material selected in the list.) See Enabling Species Transport and Reactions and Choosing the Mixture Material (p. 2359) for details.

When you use the **Non-Premixed Combustion** or **Partially Premixed Combustion** model, this list will be inactive. The mixture material for a non-premixed or partially premixed combustion calculation will be determined from the content of the PDF file generated in Ansys Fluent using the **PDF Options** parameters.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Number of Volumetric Species

specifies the number of gas-phase species in the selected **Mixture Material**. This is an informational display only; you cannot edit this value.

Number of Solid Species

specifies the number of solid species defined in the selected **Mixture Material**. This is an informational display only; you cannot edit this value. (This list will appear only for **Species Transport** models involving **Wall Surface** reactions.)

Number of Site Species

specifies the number of site species defined in the selected **Mixture Material**. This is an informational display only; you cannot edit this value. (This list will appear only for **Species Transport** models involving **Wall Surface** reactions.)

Turbulence-Chemistry Interaction

indicates which model is to be used for turbulence-chemistry interaction when the **Species Transport** model with **Volumetric** reactions is used.

Finite-Rate/No TCI

computes only the Arrhenius rate (see Equation 7.21 in the Theory Guide) and neglects turbulence-chemistry interaction.

Finite-Rate/Eddy-Dissipation

(for turbulent flows) computes both the Arrhenius rate and the mixing rate and uses the smaller of the two.

Eddy-Dissipation

(for turbulent flows) computes only the mixing rate (see Equation 7.39 and Equation 7.40 in the Theory Guide).

Eddy-Dissipation Concept

(for turbulent flows) models turbulence-chemistry interaction with detailed chemical mechanisms (see Equation 7.39 and Equation 7.40 in the Theory Guide).

Coal Calculator...

opens the Coal Calculator Dialog Box (p. 4791).

Water Corrosion Pre...

opens the **Water Chemistry and Corrosion Mechanism** dialog box. This item is available only for the Species Transport model with electrochemical reactions. See Procedure for Setting Corrosion Simulations using the Water Corrosion Pre Tool (p. 2424) for additional information on using the **Water Chemistry and Corrosion Mechanism** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select Boundary Species...

opens the **Select Boundary Species** dialog box (see Figure 19.3: The Select Boundary Species Dialog Box (p. 2367)).

Select Reported Residuals

opens the **Select Residual Monitored Species** dialog box (see Figure 19.4: The Select Residual Monitored Species (p. 2368)).

Options

contains parameters related to the laminar finite-rate or the eddy-dissipation concept model. This section of the dialog box will appear when **Finite-Rate/No TCI** or the **Eddy-Dissipation Concept** is selected for **Turbulence-Chemistry Interaction**.

EDC Model

(**Eddy-Dissipation Concept** only) contains the following submodels:

- **Constant Coefficients** (default): Standard EDC model (see The Standard EDC Model in the *Fluent Theory Guide* for details)
- **Partially Stirred Reactor**: Partially stirred reactor EDC model (see The Partially Stirred Reactor EDC Model in the *Fluent Theory Guide* for details)

Flow Iterations Per Chemistry Update

(steady-state only) specifies how often Ansys Fluent will update the chemistry during the calculation. Increasing the number can reduce the computational expense of the chemistry calculations. This option is not available when the **None - Direct Source** option is selected for **Chemistry Solver**.

Aggressiveness Factor

(steady-state only) is a numerical factor that controls the robustness and the convergence speed. This value ranges between 0 and 1, where 0 is the most robust, but results in the slowest convergence. The default value for the **Aggressiveness Factor** is 0.5. This option is available only when **Stiff Chemistry Solver** or **CHEMKIN CFD Solver** is selected for **Chemistry Solver**.

Temperature Threshold

is the threshold for cell temperature below which the chemistry reaction rate in the cell will be set to zero. This may improve the computational time without sacrificing accuracy. The default value is 200 K. This option is available only when **Stiff Chemistry Solver** or **CHEMKIN CFD Solver** is selected for **Chemistry Solver**.

Volume Fraction Constant

specifies the value of C_ξ in Equation 7.44 in the Theory Guide. This item appears only for the **Constant Coefficients** EDC model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Time Scale Constant

specifies the value of C_τ in Equation 7.45 in the Theory Guide. This item appears only for the **Constant Coefficients** EDC model.

Mixing Model

(**Partially Stirred Reactor** EDC model only) is a group box where you can select how C_{mix} in Equation 7.50 in the *Fluent Theory Guide* is computed:

- **Constant Cmix**: Sets C_{mix} to a constant value specified in **Mixing Constant**
- **Dynamic Cmix** (default): Computes C_{mix} by Equation 7.51 in the *Fluent Theory Guide* with the fractal dimension D in Equation 7.52 in the *Fluent Theory Guide* specified in **Fractal Dimension**

Thickened Flame Model Options

contains parameters related to the thickened flame model. This group box is displayed only if the **Thickened Flame Model** is enabled in the **Options** group box.

Efficiency Function

allows you to select the efficiency function. You can choose between **none**, **Colin** (default), or **Charlette**. See The Thickened Flame Model in the *Fluent Theory Guide* for more information about these options.

Number of Grid Points in Flame

by default are 8 grid points.

Integral Length Scale

is the representative of the largest eddy sizes, l_t in Equation 7.57 in the *Fluent Theory Guide* (typically, 1/4 to 1/2 of a characteristics dimension, such as a burner diameter, an inlet diameter, or a size of a bluff body). This item is available only for simulations with the Large Eddy Simulation (LES) turbulence model when **Colin** is used as the efficiency function.

PDF Options

contains options related for the non-premixed combustion model. (This section will appear only if **Non-Premixed Combustion** or **Partially-Premixed Combustion** is the selected **Model**.)

Inlet Diffusion

includes the diffusion flux of species at inlet.

Compressibility Effects

can be enabled to account for cases where substantial pressure changes occur in time and/or space when modeling a non-adiabatic system. See Specifying the Operating Pressure for the System (p. 2478) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Liquid Micro-Mixing

is used to model liquid reactions. When the **Liquid Micro-Mixing** model is invoked, Ansys Fluent uses the **volume-weighted-mixing-law** formula to calculate the density.

Probability Density Function

is a drop-down list that allows you to choose between the shape of the assumed probability density function (PDF):

- **double delta**: as given by Equation 8.21 in the *Fluent Theory Guide*.
- **beta**: as given by Equation 8.22 in the *Fluent Theory Guide*.

This control appears after you generate or read a PDF table into Ansys Fluent.

Chemistry

tab contains the parameters to define problems using the chemistry model. See Setting Up the Equilibrium Chemistry Model (p. 2476) for details.

State Relation

Chemical Equilibrium

enables the equilibrium chemistry model. See Setting Up the Equilibrium Chemistry Model (p. 2476) for details.

Steady Diffusion Flamelet

enables the steady diffusion flamelet model. See The Diffusion Flamelet Models Theory in the Theory Guide for details.

Unsteady Diffusion Flamelet

enables the Eulerian unsteady diffusion flamelet model.

Diesel Unsteady Flamelet

enables the diesel unsteady laminar flamelet model. See Using the Diesel Unsteady Laminar Flamelet Model (p. 2484) for details.

Flamelet Generated Manifold

enables the flamelet generated manifold (FGM) model. This is available when the **Partially Premixed Combustion** model is selected. See Partially Premixed Combustion in the Theory Guide for details.

Energy Treatment

Adiabatic

enables adiabatic modeling options for the problem.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Non-Adiabatic

enables non-adiabatic modeling options for the problem. See Non-Adiabatic Extensions of the Non-Premixed Model in the Theory Guide for details.

Coal Calculator

opens the Coal Calculator Dialog Box (p. 4791).

Stream Options

contains the parameters for the equilibrium chemistry model or the steady diffusion flamelet model.

Secondary Stream

includes the secondary inlet stream in the model.

Empirical Fuel Stream

enables parameters to define fuel stream empirically. This option is available only with the full equilibrium chemistry model.

Empirical Secondary Stream

enables parameters to define secondary stream empirically. This option is available only with the full equilibrium chemistry model.

Model Settings

contains a list of parameter settings.

Operating Pressure

specifies the system operating pressure used to calculate the density using the ideal gas law. See Specifying the Operating Pressure for the System (p. 2478) for details.

Fuel Stream Rich Flammability Limit

specifies the rich flammability limit for fuel stream when the equilibrium chemistry option is used. You will not set these if you have used the empirical definition option for fuel composition. See Enabling the Rich Flammability Limit (RFL) Option (p. 2480) for details.

Secondary Stream Flammability Limit

specifies the rich flammability limit for secondary stream when the equilibrium chemistry option is used. You will not set these if you have used the empirical definition option for fuel composition. See Enabling the Rich Flammability Limit (RFL) Option (p. 2480) for details.

Empirical Fuel Lower Calorific Value

specifies the lower calorific value of fuel stream.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Empirical Fuel Specific Heat

specifies the specific heat value of fuel stream.

Empirical Fuel Molecular Weight

specifies the molecular weight of the fuel stream.

Empirical Secondary Lower Calorific Value

specifies the lower calorific value of secondary stream.

Empirical Secondary Specific Heat

specifies the specific heat value of secondary stream.

Empirical Secondary Molecular Weight

specifies the molecular weight of the secondary stream.

Options

contains options related to the steady flamelet model.

Create Flamelet

enables the **Import CHEMKIN Mechanism...** button that opens the Import CHEMKIN Format Mechanism Dialog Box (p. 5358) where you can import the CHEMKIN mechanism and thermodynamic data, to create a flamelet file. This option is available for the steady flamelet model. See Setting Up the Steady and Unsteady Diffusion Flamelet Models (p. 2481) for details.

Import Flamelet

enables the **Import Flamelet File...** button that opens The Select File Dialog Box (p. 931) where you can select the existing flamelet in Ansys Fluent. You can also set the file type parameters to import the existing flamelet in Ansys Fluent. See Setting Up the Steady and Unsteady Diffusion Flamelet Models (p. 2481) for details. This option is available for the steady flamelet model.

Flamelet Type

gives you the option of creating a **Premixed Flamelet** or a **Diffusion Flamelet**. This is available when the **Flamelet Generated Manifold** model is selected.

Flamelet Solution Method

allows you to solve the premixed FGM either in the **CHEMKIN Physical Space** or in the **Progress Variable Space**. This is available when the **Flamelet Generated Manifold** state reaction is selected and **Premixed Flamelet** is selected as the **Flamelet Type**.

File Type

contains the toggle buttons for two flamelet file types.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Standard

enables the import of an ASCII format standard flamelet file.

CFX-RIF

enables the import of an ASCII format CFX-RIF flamelet file.

Flamelet Property File Name

opens The Select File Dialog Box (p. 931) in which you can save the existing flamelet in Ansys Fluent to use when running an existing case.

Thermodynamic Database File Name

specifies a path for the thermodynamic database file to be read.

Boundary

tab contains the list of boundary species and related parameters. See Defining the Stream Compositions (p. 2487) for details.

Species

contains the list of the species used in the problem.

Fuel

specifies the fuel species.

Oxid

specifies the oxidizing species.

Second

specifies the secondary species.

Boundary Species

allows you to specify the species you want to add or remove from the model. You can type the species chemical formula in the field below it.

Add

adds the species in the model.

Remove

removes the species from the model.

List Available Species

prints a list of all species in the thermodynamic database file (`thermo.db`) in the console window.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Temperature

specifies the temperature of different streams that you have defined.

Fuel

is the temperature of the fuel inlet in the model.

Oxid

is the temperature of the oxidizer inlet in the model.

Second

is the temperature of the secondary stream inlet in the model.

Specify Species in

allows you to define the unit of species concentration.

Mass Fraction

allows you to specify the species in terms of mass fraction.

Mole Fraction

allows you to specify the species in terms of mole fraction.

Control

tab contains the parameters for exclusion and inclusion of equilibrium species. This is available only for equilibrium chemistry model. See Forcing the Exclusion and Inclusion of Equilibrium Species (p. 2499) for details.

Species Excluded from Equilibrium

lists the species excluded from equilibrium calculation.

Species

lists the slow-forming species that are zeroed in the initial flamelet profile.

Add

allows you to add equilibrium species.

Remove

allows you to remove equilibrium species.

List Available Species

prints a list of all species in the thermodynamic database file in the console window.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Flamelet Controls

allows you to adjust the controls for the flamelet solution. Note that the **Create Flamelet** option in the **Chemistry** tab must be selected for the **Steady Diffusion Flamelet** or **Flamelet Generated Manifold** models for these controls to be available.

Initial Fourier Number

sets the size for the first time step for the solution.

Fourier Number Multiplier

increases the time step size at subsequent times. The size for every time step after the first is multiplied by this value.

Relative Error Tolerance and Absolute Error Tolerance

specifies the local error controls during numerical integration.

Flamelet Convergence Tolerance

specifies the maximum absolute change in species fraction or temperature at any discrete mixture-fraction.

Maximum Integration Time

specifies the maximum total elapsed time for flamelet calculation. Ansys Fluent will stop the flamelet calculation after the total elapsed time has exceeded this value.

Set Progress Variable...

Opens the **Progress Variable Definition** dialog box. See Flamelet Generated Manifold (p. 2546) for details.

Flamelet

tab allows you to adjust the controls for the flamelet solution. This tab is available when chemistry model is not equilibrium. See Defining the Flamelet Controls (p. 2500) for details.

Flamelet Parameters

consist of the controls for the flamelet solution.

Note:

The parameters may vary slightly if you selected the **Flamelet Generated Manifold** model (see Flamelet Generated Manifold (p. 2546) for those specific parameters).

Number of Grid Points in Flamelet

specifies the number of mixture fraction grid points distributed between the oxidizer ($f=0$) and the fuel ($f=1$).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Maximum Number of Flamelets

specifies the maximum number of laminar flamelet profiles to be calculated.

Number of Grid Points in Enthalpy Space

specifies the number of enthalpy levels for which the flamelets will be generated. Ansys Fluent automatically distributes the enthalpy levels based on the reference fuel and oxidizer temperatures. A higher number of points provides the increased resolution in enthalpy space, but requires more computation time because each additional point in enthalpy space will generate a set of flamelets for entire mixture fraction space. This item is available when **Nonadiabatic Flamelets** is selected.

Nonadiabatic Flamelets

enables generation of the nonadiabatic flamelets. For more information, see Using the Heat Loss Modeling Capability for Nonadiabatic FGM (p. 2552).

Fully Premixed FGM

is an option to generate nonadiabatic flamelets only for one mixture fraction specified in the **Mixture Fraction Value for Flamelet Generation** entry field. This is useful for fully premixed CFD problems when the mixture fraction is constant, and flamelets are required only for a single mixture fraction. When this option is enabled, the **Number of Grid Points in Mixture Fraction Space** is automatically set to **1**. Since the number of mixture fraction space is only **1**, this option is not only fast to generate the manifold, but it also produces flamelet and PDF files that are greatly reduced size. This item is available when **Nonadiabatic Flamelets** is selected.

Initial Scalar Dissipation

is the scalar dissipation of the first flamelet in the library.

Scalar Dissipation Multiplier

specifies the ratio of the scalar dissipation step in which successive flamelets are generated when the scalar dissipation is less than 1 s^{-1}. This corresponds to $\Delta\chi$ for < 1 in Equation 8.53 in the Theory Guide. This option is available with the Steady Diffusion Flamelet model and Diffusion FGM only.

Scalar Dissipation Step

specifies the interval between scalar dissipation values (in s^{-1}) for which multiple flamelets will be calculated. This corresponds to $\Delta\chi$ for ≥ 1 in Equation 8.53 in the Theory Guide. This option is available with the Steady Diffusion Flamelet model and Diffusion FGM only.

User Defined Flamelet Parameters

enables you to hook a user-defined function for scalar dissipation and mean mixture fraction (or progress variable) grid discretization

Unsteady Flamelet Parameters

consist of the controls for the unsteady flamelet solution.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Number of Grid Points in Flamelet

specifies the number of mixture fraction grid points distributed between the oxidizer ($f=0$) and the fuel ($f=1$).

Mixture Fraction Lower Limit for Initial Probability

is the limit at which the unsteady flamelet model temporally convects and diffuses a marker probability equation through a steady-state Ansys Fluent flow-field.

Maximum Scalar Dissipation

is where flamelets extinguish at large scalar dissipation (mixing) rates.

Courant Number

is the number at which Ansys Fluent automatically selects the time step for the probability equation based on this convective Courant number.

Number of Flamelets

specifies the number of unsteady laminar flamelets that Ansys Fluent will automatically generate during the simulation.

Include Equilibrium Flamelet

determines whether the equilibrium assumption is used to compute the progress variable (Equation 8.107 in the *Fluent Theory Guide*). If this option is selected, Ansys Fluent will compute the progress variable using equilibrium. Otherwise, the progress variable will be computed using the solution of flamelet generated with initial scalar dissipation as a denominator in Equation 8.107 in the *Fluent Theory Guide* This option is available only with the diffusion FGM model.

Automated Grid Refinement

employs an adaptive algorithm, which inserts grid points so that the change of values, as well as the change of slopes, between successive grid points is less than user-specified tolerances. Once this option enabled, you can specify the following parameters:

- **Initial Number of Grid Points in Flamelet**: calculates a steady solution on a coarse grid.
- **Maximum Number of Grid Points in Flamelet**
- **Maximum Change in Value Ratio**: is ε_v in Equation 8.28 in the Theory Guide.
- **Maximum Change in Slope Ratio**: is ε_s in Equation 8.29 in the Theory Guide.
- **Refine flamelet based on**: allows you to select either the **Stoichiometric mixture fraction** or **Specified mixture fraction**. For **Specified mixture fraction**, you must also specify a mixture fraction level at which the flamelet is solved during the grid refinement. This option is available for the premixed FGM model only.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Set Flamelet Parameters

opens Unsteady Flamelet Parameters Dialog Box (p. 4801).

Calculate Flamelets

begins the laminar flamelet calculation.

Display Flamelets...

opens the Flamelet 3D Surfaces Dialog Box (p. 4797) from which you can display 2D plots and 3D surfaces showing the variation of species fraction or temperature with the mean mixture fraction or scalar dissipation.

Initialize Unsteady Flamelet Probability

initializes the unsteady flamelet and its probability marker equation.

Display Unsteady Flamelet...

opens the Flamelet 2D Curves Dialog Box (p. 4800) from which you can display 2D plots of the different variables.

Table

tab contains parameters to create the look-up table. See Calculating the Look-Up Tables (p. 2509) for details of the items listed below.

Note:

If you selected the **Flamelet Generated Manifold** model, the parameters will be different. See Calculating the Look-Up Tables (p. 2554) for those specific parameters.

Table Parameters

consists of the controls for the lookup table. A different set of parameters for you to enter will be displayed if **Automated Grid Refinement** is enabled or disabled.

Initial Number of Grid Points

specifies the number of grid points for the resolution of the mean mixture fraction, mixture fraction variance, and mean enthalpy (for non-adiabatic systems).

Maximum Number of Grid Points

specifies the maximum number of grid points used for tabulation. The grid refinement procedure will stop inserting the points when either the change in value and slope between successive points is within tolerance or the maximum number of grid points are generated.

Maximum Change in Value Ratio

specifies the maximum allowable change in value of table variables between successive grid points as specified by Equation 8.28 in the Theory Guide.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Maximum Change in Slope Ratio

specifies the maximum change in the slope of table variables between successive grid points as specified by Equation 8.29 in the Theory Guide.

Number of Mean Mixture Fraction Points

is the number of discrete values of $\overline{f}$ at which the look-up tables will be computed.

Number of Secondary Mixture Fraction Points

is the number of discrete values of p_{sec} at which the look-up tables will be computed. This option is available only when a secondary stream has been defined.

Number of Mixture Fraction Variance Points

is the number of discrete values of $\overline{f'^2}$ at which the look-up tables will be computed. This option is available only when no secondary stream has been defined.

Maximum Number of Species

is the maximum number of species that will be included in the look-up tables.

Number of Mean Enthalpy Points

is the number of discrete values of enthalpy at which the three-dimensional look-up tables will be computed. This input is required only if you are modeling a non-adiabatic system.

Minimum Temperature

is used to determine the lowest temperature for which the look-up tables are generated (see Figure 8.10: Visual Representation of a Look-Up Table for the Scalar as a Function of Mean Mixture Fraction and Mixture Fraction Variance and Normalized Heat Loss/Gain in Non-Adiabatic Single-Mixture-Fraction Systems in the Theory Guide). This option is available only if you are modeling a non-adiabatic system.

Automated Grid Refinement

is an adaptive algorithm that inserts grid points in all table dimensions so that changes in the values of tabulated variables (such as mean temperature, density and species mass fractions) between successive grid points, as well as changes in their slopes, are less than a user specified tolerance.

Include Equilibrium Flamelet

specifies that an equilibrium flamelet (that is, $\chi=0$) will be generated in Ansys Fluent and appended to the flamelet library before the PDF table is calculated. This option is available only with the steady diffusion flamelet model.

FGM Scalar Transport

opens the Select Transported Scalars Dialog Box (p. 4803).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Calculate PDF Table

generates the look-up table.

Display PDF Table

opens the PDF Table Dialog Box (p. 4805) where you can display 2D plots and 3D surfaces showing the variation of species mole fraction, density, or temperature with the mean mixture fraction, mixture fraction variance, or enthalpy.

Premix

tab contains options and parameters for the **Turbulent Flame Speed Model**, the **Variance Settings**, and the **Turbulence-Chemistry Interaction**.

Note:

Note that the **Turbulent Flame Speed Model** controls are available for both premixed and partially premixed combustion models. However, for the partially premixed combustion model, they are organized in the **Premix** tab.

Turbulent Flame Speed Model

allows you to select the **Flame Speed Model** and specify its parameters. This section will appear only if **Premixed Combustion** or **Partially Premixed Combustion** is the selected **Model** and if the **C Equation** or **G Equation** premixed model is chosen.

Flame Speed Model

is a drop-down list in which you can select the turbulent flame speed model. You can choose between **zimont** (see Zimont Turbulent Flame Speed Closure Model in the *Fluent Theory Guide*) and **peters** (see Peters Flame Speed Model in the *Fluent Theory Guide*).

Turbulent Length Scale Constant

(**zimont** model) specifies the value of C_D in Equation 8.79 in the Theory Guide.

Turbulent Flame Speed Constant

(**zimont** model) specifies the value of A in Equation 8.77 in the Theory Guide.

Stretch Factor Coefficient

(**zimont** model) specifies the value of μ_{str} in Equation 8.85 in the Theory Guide.

Wall Damping Coefficient

specifies the value of α_w in Equation 8.88 (premixed combustion model) or in Equation 8.114 in the *Fluent Theory Guide* (partially premixed combustion model).

Turbulent Schmidt Number

specifies the value of Sc_t in Equation 8.70 in the Theory Guide.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Turbulent Length Scale Constant (RANS)

(**zimont** model) is the modeling constant in the RANS region (C_D in Equation 8.79 in the Theory Guide). This option is available for the Partially Premixed Combustion model with the SBES model only.

Turbulent Flame Speed Constant (RANS)

(**zimont** model) is the modeling constant in the RANS region (A in Equation 8.77 in the Theory Guide). This option is available for the Partially Premixed Combustion model with the SBES model only.

Ewald Corrector

(**peters** model) is enabled by default and described in Peters Flame Speed Model.

Blint Modifier

(**peters** model) enables/disables Blint's correlation for laminar flame thickness δ (used in Equation 8.90 thru Equation 8.92 in the *Fluent Theory Guide*). See Peters Flame Speed Model in the *Fluent Theory Guide* for more information.

G Equation Settings

allows you to select either the **transport equation** or **algebraic** option for the calculation of the flame distance variance. Consult Peters Flame Speed Model in the Theory guide for the variance transport and algebraic equation expressions (Equation 8.74 and Equation 8.75).

Flame Curvature Source

includes the curvature source term in the G-Equation, which is the last term in Equation 8.72.

Turbulence-Chemistry Interaction

(partially-premixed combustion FGM model only) allows you to choose between **Finite-Rate** and **Turbulent Flame Speed**. For more information about these models, see FGM Turbulent Closure in the Theory Guide.

Variance Settings

is available for the **G Equation** premixed or partially premixed model and **C Equation** partially premixed FGM model.

Variance Method

is a drop-down list that allows you to choose between **transport equation** and **algebraic**. If the SBES model is enabled, you can also choose **hybrid**. Consult FGM Turbulent Closure in the *Fluent Theory Guide* for the **C Equation** model (Equation 8.116 and Equation 8.92) and G-Equation Model Theory in the *Fluent Theory Guide* for the **G-Equation** model (Equation 8.74 and Equation 8.75). It is recommended that you use the **transport equation** option for RANS, the **algebraic** option for LES, and the **hybrid** for SBES.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Properties

tab contains parameters needed to modify the piecewise-linear points. See Modifying the Unburnt Mixture Property Polynomials (p. 2564) for the details.

Partially Premixed Mixture Properties

contains the list of properties that you can modify. See Modifying the Unburnt Mixture Property Polynomials (p. 2564) for details

Non-Adiabatic Laminar Flame Speed

when enabled, includes the non-adiabatic effects on the laminar flame speed by tabulating the laminar speeds in the PDF table. See Laminar Flame Speed in the Theory Guide.

Strained Laminar Flame Speed

when enabled, includes the strain rate effects on the laminar flame speed. See Strained Laminar Flame Speed in the *Fluent Theory Guide* for more information. This item is available only for the Flamelet Generated Manifold (FGM) model with the turbulent flame speed model after you import a transport data file along with mechanism and thermodynamic data files.

Number of Heat Loss Points

specifies the number of the heat loss points in the strained flame speed calculation. The default value of 1 corresponds to adiabatic strained flame speed. This item is available only when **Strained Laminar Flame Speed** is enabled.

Compute Strained Laminar Flame Speed Table

generates strained flamelets for different mixture fraction points and strain rates. This item is available only when **Strained Laminar Flame Speed** is enabled.

Display Strained Flamelets...

opens the Flamelet 2D Curves Dialog Box (p. 4800). See Modeling Strained Laminar Flame Speed (p. 2567) for more information. This item is available only when **Strained Laminar Flame Speed** is enabled.

Recalculate Properties

will calculate the partially premixed properties.

Premixed Combustion Model Options

is a group box that contains options for the premixed combustion model. (This section will appear only if **Premixed Combustion** is the selected **Model**.)

Adiabatic

enables the adiabatic premixed combustion model, which calculates temperature using Equation 8.97 in the Theory Guide.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Non-Adiabatic

enables the non-adiabatic premixed combustion model, which calculates temperature using Equation 8.98 in the Theory Guide.

Premixed Model

is a group box that contains options for choosing a premixed model.

C Equation

allows you to choose the C equation as described in C-Equation Model Theory.

G Equation

allows you to choose the G equation as described in G-Equation Model Theory.

PDF Transport Options

contains options for the Composition PDF Transport combustion model. (This section will appear only if **Composition PDF Transport** is the selected **Model**.)

Lagrangian

solves the composition PDF transport equation by stochastically tracking Lagrangian particles through the domain.

Eulerian

assumes a shape for the PDF, allowing Eulerian transport equations to be derived.

Mixing (tab)

tab contains the mixing models.

Mixing Model

contains options to specify the method for modeling molecular diffusion. (This section will appear only if **Composition PDF Transport** is the selected **Model**.) See Particle Mixing in the Theory Guide for details.

Modified Curl

enables the modified curl model for molecular diffusion.

IEM

enables the IEM model for molecular diffusion.

EMST

enables the EMST mixing model for molecular diffusion.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mixing Constant

specifies the value of the mixing constant C_φ in Equation 7.153 and Equation 7.155 in the Theory Guide.

Boundary (tab)

tab allows you to define the fuel and oxidizer compositions. This is only available if you select **Eulerian** as the **PDF Transport Option**.

Species

consists of the fuel species and the oxidizer.

Fuel

is the mole or mass fraction of the fuel stream. The sum of mass or mole fractions of all species in the fuel stream should be 1.

Oxidizer

is the mole or mass fraction of the oxidizer stream. The sum of mass or mole fractions of all species in the fuel oxidizer stream should be 1.

Specify Species in

specifies the species as a **Mass Fraction** or **Mole Fraction**.

Control (tab)

tab contains Lagrangian PDF transport parameters.

PDF Transport Parameters

allows you to set the **Particles Per Cell**.

Particles Per Cell

sets the number of PDF particles per cell. Higher values of this parameter will reduce statistical error, but increase computational time.

Local Time Stepping

toggles the calculation of local time steps. If this option is disabled, then you will need to specify the **Time Step** directly (see Equation 7.151 in the Theory Guide). This option is available for steady-state simulations.

Convection #

specifies the particle convection number (see Δt_{conv} in Equation 7.151 in the Theory Guide).

Diffusion #

specifies the particle diffusion number (see Δt_{diff} in Equation 7.151 in the Theory Guide).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

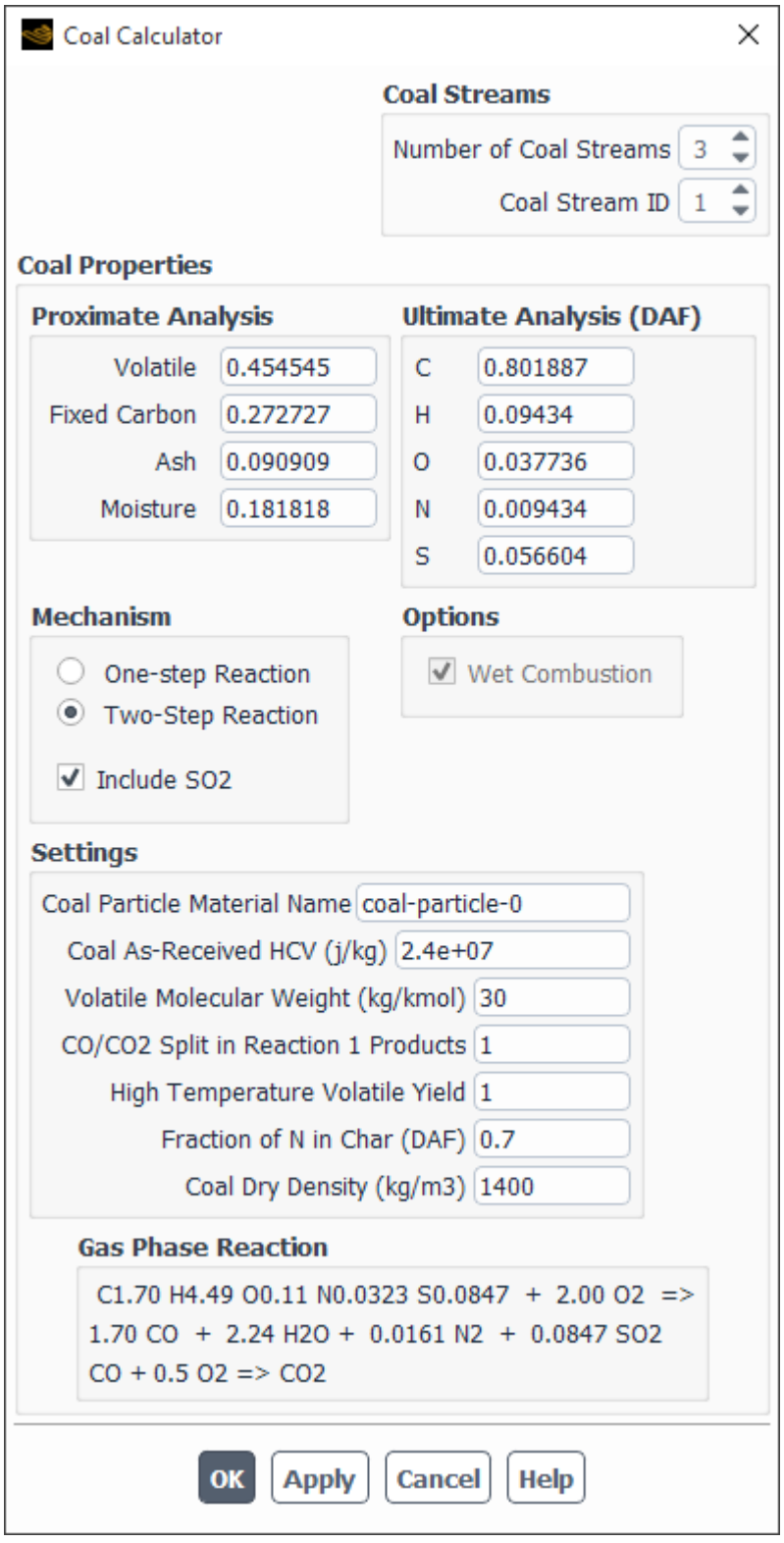
Mixing #

specifies the particle mixing number (see Δt_{mix} in Equation 7.151 in the Theory Guide).

51.4.19. Coal Calculator Dialog Box

The **Coal Calculator** dialog box (opened by clicking **Coal Calculator...** in the **Species Model** dialog box) automates the calculations described in Setting up Coal Simulations with the Coal Calculator Dialog Box (p. 2391) and Additional Coal Modeling Inputs in Ansys Fluent (p. 2495).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Coal Streams

allows you to set up to three coal streams (Species Transport model only).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Number of Coal Streams

specifies the total number of coal streams.

ID

sets the ID of the coal stream for which you want to specify **Coal Properties**.

Coal Properties

contains inputs for the current coal stream.

Proximate Analysis

is the mass fraction of **Volatile**, **Fixed Carbon**, **Ash**, and **Moisture** in the coal.

Volatile

is the fraction of the volatile component.

Fixed Carbon

is calculated as one minus the sum of the actual **Volatile**, **Ash**, and **Moisture** fractions.

Ash

is the fraction of ash.

Moisture

is the moisture fraction in the coal.

Ultimate Analysis (DAF)

is the mass fraction of atomic **C, H, O, N** and optionally **S**, in the Dry-Ash-Free (DAF) coal.

Mechanism

allows you to set the mechanisms.

Secondary Stream

when enabled, allows you to set the two mixture fraction model with the primary stream representing char as C <s> , and an empirical secondary stream representing the volatiles. This is available when using the non-premixed combustion model.

One-step Reaction

is defined in Equation 19.6 (p. 2393).

Two-step Reaction

involves oxidation of volatiles to CO in the first reaction and oxidation of CO to CO_2 in the second reaction, as described in Equation 19.7 (p. 2393).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Include SO2

when enabled, allows you to specify the atomic mass fraction of sulphur, **S**, which appears under **Ultimate Analysis**.

Options

Wet Combustion

when enabled will enable the DPM **Wet Combustion** option by default in all injections created after the **OK** button is clicked in the **Coal Calculator** dialog box.

Settings

is where you will specify the values used in the calculation.

Coal Particle Material Name

is the name of the DPM combusting particle material. The default name is *coal-particle*.

Coal As-Received HCV

is the higher caloric value of the coal.

Volatile Molecular Weight

is the molecular weight of pure volatiles.

CO/CO2 Split in Reaction 1 Products

can be used to specify the molar fraction of CO to CO_2 in the first reaction of Equation 19.7 (p. 2393). The default value of 1 implies that all carbon is reacted to CO, with no CO_2 produced.

High Temperature Volatile Yield

is where the enhanced devolatization at higher temperatures can cause the volatile yield to exceed the proximate analysis fraction.

Fraction of N in Char (DAF)

is used in calculating the split of atomic nitrogen for the Fuel NOx model.

Coal Dry Density

is used to calculate the **Volume Fraction** of liquid-water for the **Wet Combustion** option in the **Injections** dialog box.

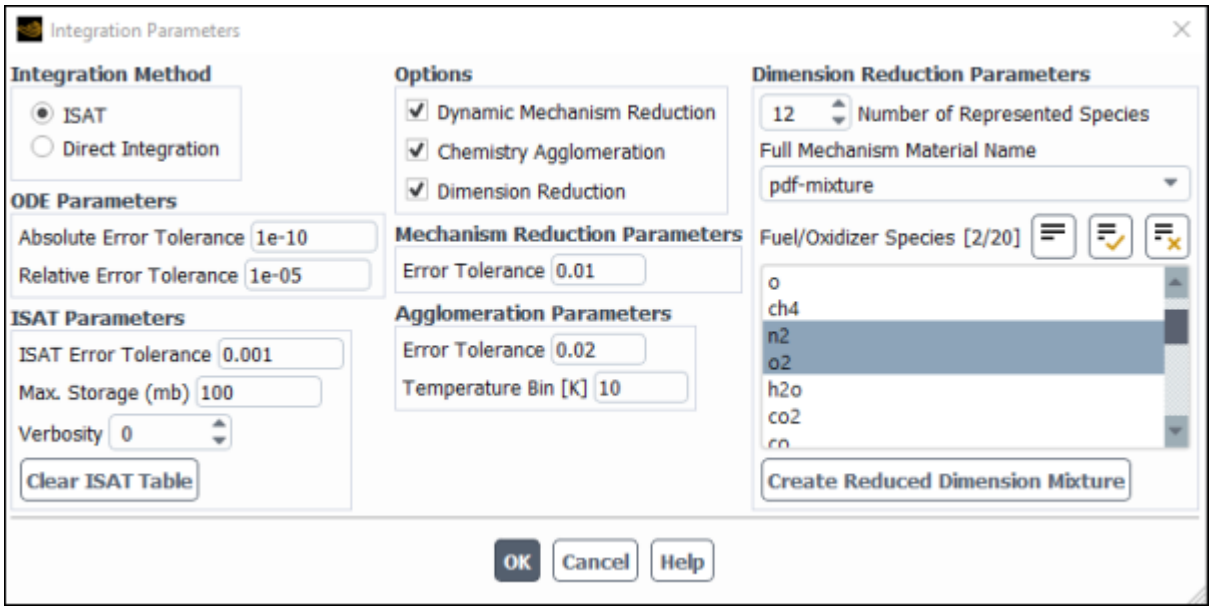
Gas Phase Reaction

lists the reaction based on your entries for the proximate and ultimate analyses.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

51.4.20. Integration Parameters Dialog Box

The **Integration Parameters** dialog box (opened by clicking **Integration Parameters...** in the **Species Model** dialog box) allows you to set the parameters for the integration of the chemical source term S in Equation 7.157 in the Theory Guide. See Using ISAT (p. 2457) for details.



Integration Method

contains options to choose the method for integration.

ISAT

enables the ISAT option and expands the dialog box to include inputs for **ISAT Parameters**.

Direct Integration

enables the direct integration method to integrate the chemical source term in the calculation.

ODE Parameters

contains options to specify the error tolerances.

Absolute Error Tolerance

specifies the absolute error tolerance.

Relative Error Tolerance

specifies the relative error tolerance.

ISAT Parameters

contains inputs required for ISAT integration method.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ISAT Error Tolerance

controls the numerical error in ISAT liner interpolation. Decrease this value to get accurate minor species and pollutant predictions.

Max. Storage

is the maximum RAM used by the ISAT table, and has a default value is 100 MB.

Verbosity

specifies the level of detail at which you can monitor the ISAT performance.

Clear ISAT Table

purges the ISAT table.

Options

contains options to choose the method for chemistry acceleration.

Dynamic Mechanism Reduction

accelerates chemistry by reducing the chemical kinetics mechanism on-the-fly to include only important species and reactions, with a corresponding decrease in accuracy. This method is available for single-phase flow simulations with **Stiff Chemistry Solver** only.

Chemistry Agglomeration

when enabled, provides additional run-time improvement, with a corresponding decrease in accuracy. This method is available for **Stiff Chemistry Solver** (single-phase flow) and **Relax to Chemical Equilibrium** only.

Dimension Reduction

is a chemistry acceleration method in addition to ISAT storage-retrieval and Cell Agglomeration, providing faster chemistry calculations with a corresponding loss of accuracy. This method is available for single-phase flow simulations with **Stiff Chemistry Solver** only.

Dynamic Cell Clustering

enables a computationally efficient dynamic cell clustering method (DCC) that groups computational cells of high similarity into clusters using two DCC parameters:

- **Max. Temperature Dispersion** (default =10 K)
- **Max. Equiv. Ratio Dispersion** (default =0.05)
- **Max. Clusterization** (default = 10)
- **Min. Clusterization** (default = 0)
- **Reactants Threshold (mass fraction)** (default = 1e-09)

This option is enabled by default and recommended. It is available only with the **CHEMKIN-CFD Solver**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Dynamic Adaptive Chemistry

accelerates chemistry by reducing the chemical kinetics mechanism on-the-fly to locally valid smaller mechanisms, with a corresponding decrease in accuracy. The level of accuracy is controlled by **DAC Error Tolerance**. You can select the initial target species to be tracked by the DAC algorithm in the **Select DAC Target Species** dialog box that opens when you click **Select Target Species**. See Using Dynamic Adaptive Chemistry with Ansys Fluent CHEMKIN-CFD Solver (p. 2468) for more details.

This option is available only with the **CHEMKIN-CFD Solver**.

Agglomeration Parameters

contains settings for the agglomeration chemistry method.

Error Tolerance

determines the size of the clusters for species. By default, the value is 0.05.

Temperature Bin

specifies the maximum bin size for temperature. The default value is 100 K.

Dimension Reduction Parameters

contains settings to accelerate the chemistry.

Number of Represented Species

must be greater than 10 and less than the number of species in the full mechanism. The **Number of Represented Species** must also be less than 50 minus the number of unrepresented elements (the number of chemical elements in the unrepresented species).

Full Mechanism Material Name

is typically the name of the CHEMKIN mechanism that you imported.

Fuel/Oxidizer Species

is where the boundary and initial fuel and oxidizer, as well as product species, are set as represented species.

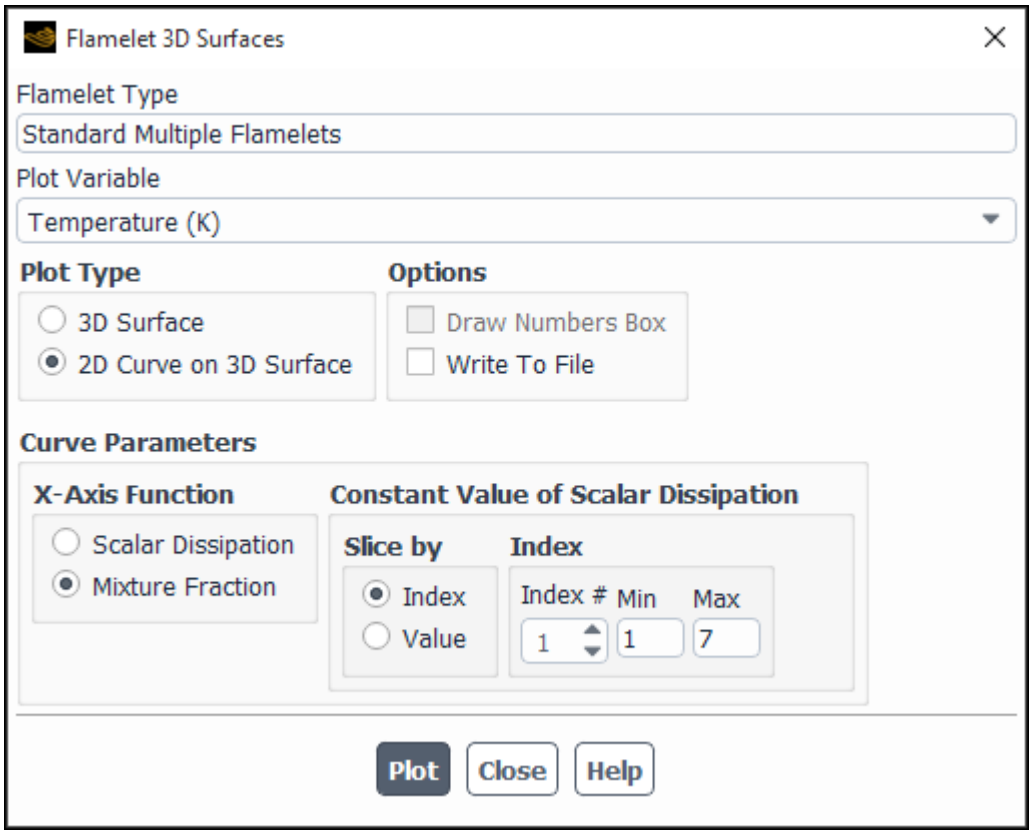
Create Reduced Dimension Mixture

creates a new mixture material called `reduced-dimension-mixture`, which contains the represented species as well as proxy 'species' for the unrepresented elements.

51.4.21. Flamelet 3D Surfaces Dialog Box

The **Flamelet 3D Surfaces** dialog box allows you to display 2D plots and 3D surfaces showing the variation of species fraction or temperature with the mean mixture fraction or scalar dissipation. See Postprocessing the Flamelet Data (p. 2506) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Flamelet Type

indicates the type of flamelets whose surfaces will be displayed.

Plot Variable

enables you to choose temperature or species fraction as the variable to be plotted.

Plot Type

consists of options for plot type.

3D Surface

enables plotting on 3D surfaces.

2D curve on 3D surface

enables plotting of a 2D curve on a 3D surface.

Options

consists of the following parameters:

Draw Numbers Box

enables the display of a wireframe box with the numerical limits in each coordinate direction.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Write To File

enables saving the plot data to a file.

Curve Parameters

consists of controls related to plot display.

X-Axis Function

consists of the function against which the plot variable will be displayed.

Scalar Dissipation

enables display of plot variable against the scalar dissipation function.

Mixture Fraction

enables display of plot variable against the mixture fraction.

Constant Value of Scalar Dissipation

consists of the controls to specify the type of discretization (that is, how the flamelet data will be sliced) for the variable that is being held constant.

Slice by

consists of the controls to specify discretization.

Index

enables you to specify discretization index of the variable that is being held constant.

Value

enables you to specify the numerical value of the variable that is being held constant.

Index

consists of the controls that are displayed when you enable **Index** under **Slice by**.

Index #

displays the index number.

Min

displays the minimum of the range of integer values that you are allowed to choose from.

Max

displays the maximum of the range of integer values that you are allowed to choose from.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Value

consists of the controls that are displayed when you enable **Value** under **Slice by**.

Value

enables you to enter the numerical value of the variable that is being held constant.

Min

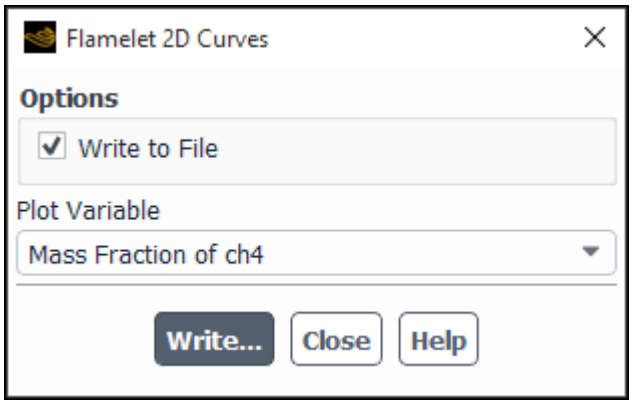
displays the minimum of the range of integer values that you are allowed to choose from.

Max

displays the minimum of the range of integer values that you are allowed to choose from.

51.4.22. Flamelet 2D Curves Dialog Box

The **Flamelet 2D Curves** dialog box allows you to display or write 2D curves of the unsteady flamelets or strained flamelets. See Postprocessing the Flamelet Data (p. 2506) or Modeling Strained Laminar Flame Speed (p. 2567), respectively, for details.



Write to File

when enabled, write 2D curves to a file that you specify. When this option is selected, the **Plot** button will change to **Write...**. Clicking **Write...** opens the **Select File** dialog box (The Select File Dialog Box (p. 931)) where you can specify a name and save a file containing the plot data.

Plot Variable

(non-premixed combustion flamelet model only) consists of a drop-down list of variables that you can plot or write.

Flamelet Variable

(partially-premixed combustion FGM only) displays the **Flame Speed** flamelet variable that you can plot or write to file.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Curve Parameters

(partially-premixed combustion FGM model only) consists of controls related to plot display.

X-Axis Function

consists of the function against which the plot variable will be displayed.

Strain Rate

enables display of plot variable against the strain rate function.

Mixture Fraction

enables display of plot variable against the mixture fraction.

Slice by

consists of the controls to specify discretization.

Index

enables you to specify discretization index of the variable that is being held constant.

Value

enables you to specify the numerical value of the variable that is being held constant.

Index

consists of the controls that are displayed when you enable **Index** under **Slice by**.

Index #

displays the index number.

Min

displays the minimum of the range of integer values that you are allowed to choose from.

Max

displays the maximum of the range of integer values that you are allowed to choose from.

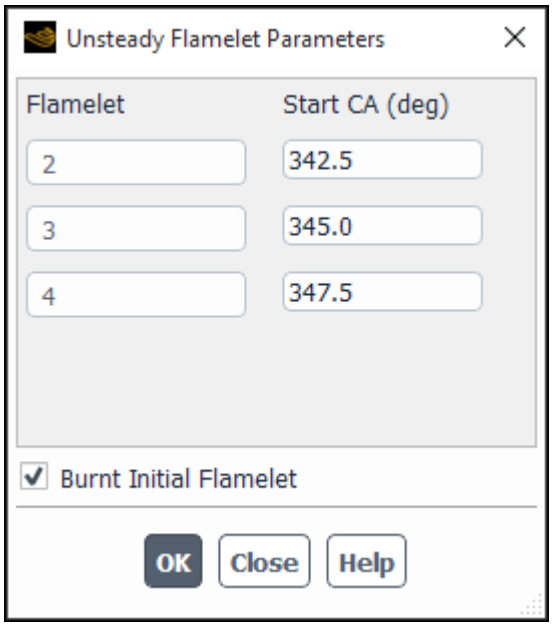
Plot

displays a plot of 2D curves of the flamelets in the active graphics window. When the **Write to File** option is selected, the **Plot** button changes to a **Write...**.

51.4.23. Unsteady Flamelet Parameters Dialog Box

The **Unsteady Flamelet Parameters** dialog box (opened by clicking **Set Flamelet Parameters** in the **Flamelet** tab of the **Species Model** dialog box) allows you to specify the initiation time of the unsteady flamelets. See Using the Diesel Unsteady Laminar Flamelet Model (p. 2484) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Start CA (deg)

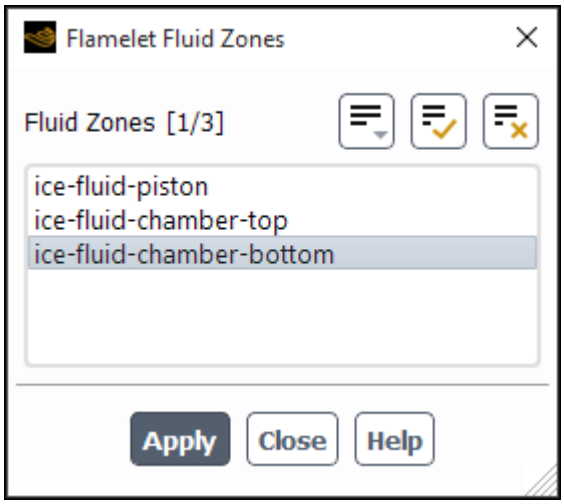
is the time at which each unsteady laminar flamelet will be generated during your simulation. The time is specified in terms of seconds or degrees of crank angle if the dynamic mesh is enabled.

Burnt Initial Flamelet

if enabled, sets the initial flamelet condition to a chemical equilibrium burnt state. Otherwise, the initial flamelet condition is set to unburnt state (default).

51.4.24. Flamelet Fluid Zones Dialog Box

The **Flamelet Fluid Zones** dialog box allows you to select the fluid zones for computing zone-averaged pressure and scalar dissipation. The dialog box is opened by clicking **Set Flamelet Fluid Zones** in the **Flamelet** tab of the **Species Model** dialog box. See Using the Diesel Unsteady Laminar Flamelet Model (p. 2484) for details.



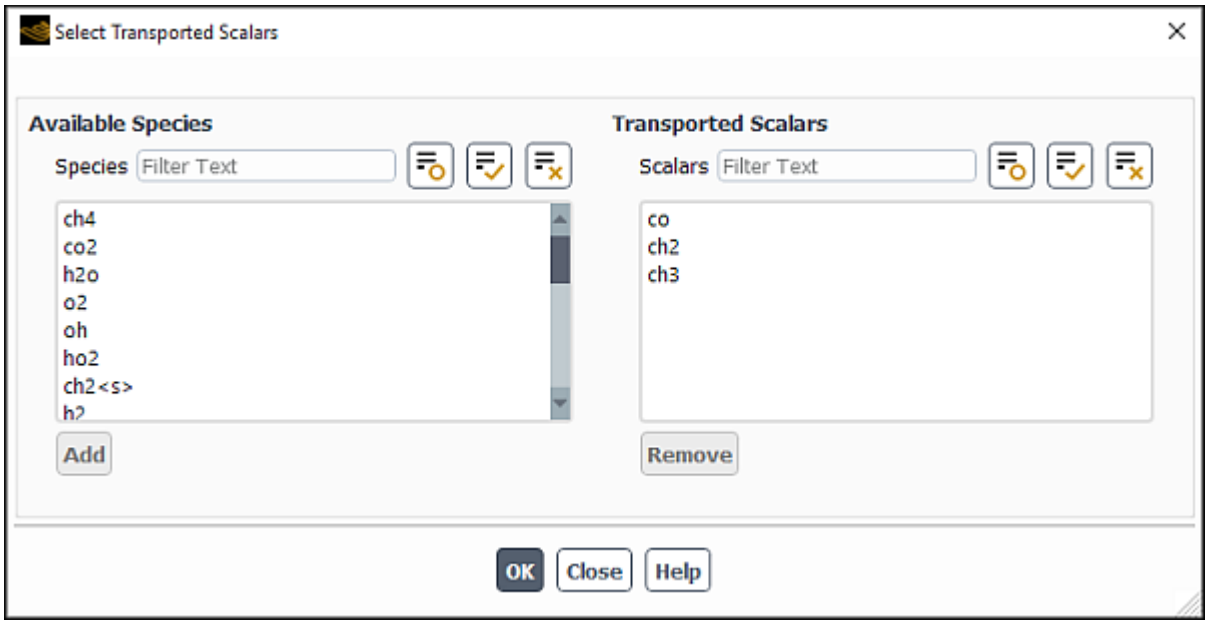
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fluid Zones

is a selectable list of the fluid zones over which the average pressure and scalar dissipation are computed.

51.4.25. Select Transported Scalars Dialog Box

The **Select Transported Scalars** dialog box (opened by clicking **FGM Scalar Transport** in the **Table** tab of the **Species Model** dialog box) allows you to select the species for which transport equations will be solved. See Calculating the Look-Up Tables (p. 2554) for more information about the items below.



Controls

Available Species

is a list of species available in the FGM flamelet table (either generated or imported). To add a species to the **Transported Scalars** list, select it in the **Available Species** multiple-selection list and click **Add**.

Transported Scalars

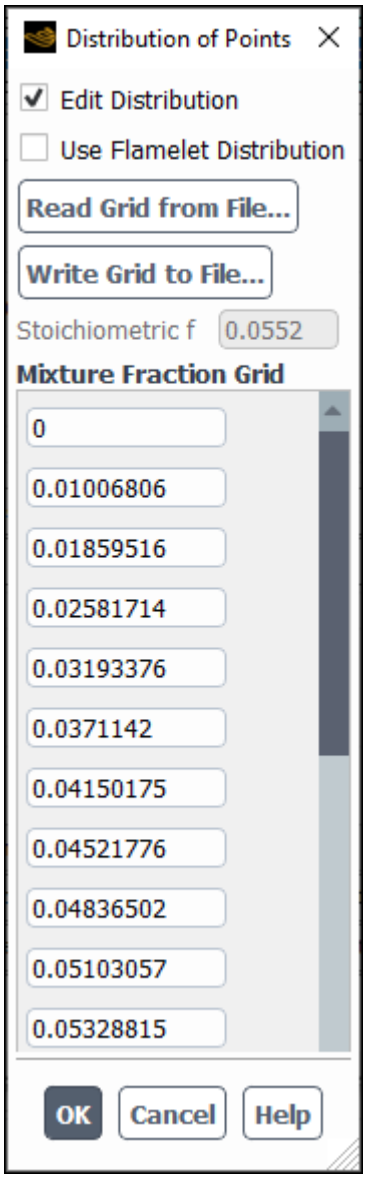
is a list of selected species for which the transport equations will be solved, as described in Scalar Transport with FGM Closure in the *Fluent Theory Guide*. To remove a species from the **Transported Scalars** list back to the **Available Species** multiple-selection list, select it and click **Remove**.

51.4.26. Distribution of Points Dialog Box

The **Distribution of Points** dialog box (opened by clicking **Edit...** next to the chosen parameter either in the **Flamelet** tab or in the **Table** tab of the **Species Model** dialog box) allows you to edit the grid distribution for a chosen parameter (only for partially premixed combustion cases with FGM). See Editing the Flamelet Grid Distribution (p. 2549) for more information about the items below. See also Premixed Flamelet Generated Manifolds (p. 2547), Diffusion Flamelet Generated Manifolds (p. 2551), and Calculating the Look-Up Tables (p. 2554) for additional information.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that the default grid distributions for flamelet and table parameters are sufficient for most of the cases and should not be modified. However, in very specific cases, changing the grid distribution may improve accuracy.



Controls

Edit Distribution

enables you to edit the points of the grid distribution for the chosen parameter.

Use Flamelet Distribution

allows you to use the points of the grid distribution of the FGM flamelet. This item is available only for the mixture fraction table parameter.

Read Grid from File...

allows you to import a previously saved distribution point file into your case.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Write Grid to File...

allows you to export the grid point data to an ASCII file in the XY plot file format.

Stoichiometric f

displays the value of the stoichiometric mixture fraction. This is an information box; you cannot edit this value.

***Parameter* Grid**

displays the values of the points of the grid distribution. (The name of the chosen parameter will replace *Parameter* in ***Parameter* Grid**).

51.4.27. PDF Table Dialog Box

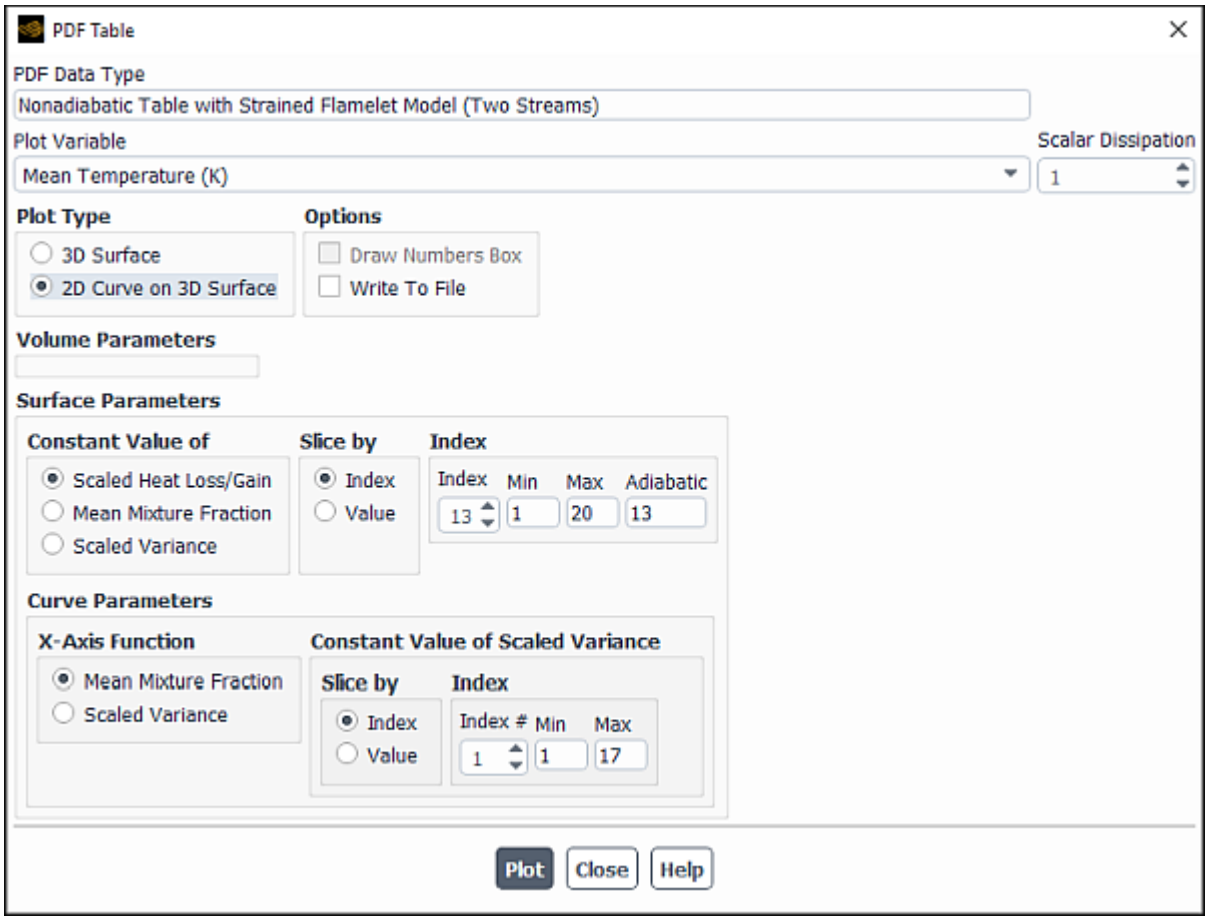
You can display 2D plots and 3D surfaces showing the variation of species mole fraction, density, or temperature with the mean mixture fraction, mixture fraction variance, or enthalpy. For the Flamelet Generated Manifold (FGM) model, additional variables are available for generating plots for PDF lookup tables. See Postprocessing the Look-Up Table Data (p. 2515) for details about the items below.

The **PDF Table** dialog box can be accessed in one of two ways:

- by clicking the **Display PDF Table...** button in the **Table** tab of the **Species Model** dialog box (as shown in Figure 20.23: The Species Model Dialog Box (Table) Tab Excluding Automated Grid Refinement (p. 2511))
- by using the following path:

  Postprocessing → **Model Specific** → **PDF Table...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

PDF Data Type

describes the system that you are displaying.

Plot Variable

is a drop-down list from which you can select temperature, density, or species mass fraction as the variable to be plotted. For the FMG model, you can also select mean rate of reaction progress.

Enthalpy Slice, Adiabatic = *slice number*

(FGM model only) specifies the value of the enthalpy slice level. By default, the enthalpy level is fixed at the adiabatic level displayed in the label as **Adiabatic = *slice number*** . Enthalpy slice numbers below the adiabatic slice number are associated with heat loss, while enthalpy slice numbers above the adiabatic slice number are associated with enthalpy gain.

Scalar Dissipation

(multiple flamelets only) specifies the value of the **Scalar Dissipation**.

Plot Type

gives you the choice of plotting a 3D surface or a slice of a 3D surface.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

3D Surface

displays a 3D plot of the variation of species mole fraction, density, or temperature with the mean mixture fraction, mixture fraction variance, or enthalpy.

2D Curve on 3D Surface

consists of a 2D curve that is a slice of a 3D surface.

Options

contains options specific to the display of 3D surfaces or 2D curves on 3D surfaces.

Draw Numbers Box

enabling this option displays a wireframe box with the numerical limits in each coordinate direction. This option is available only when **3D Surface** is selected.

Write To File

specifies whether you want to write the plot data to a file. This option is available only when **2D Curve on 3D Surface** is selected. The **Plot** button changes to a **Write** button when this option is enabled.

Volume Parameters

(FGM model only) contains settings for selecting and fixing a discrete independent variable.

Constant Value of

contains the discrete independent variables that can be held constant in the lookup table. The choices are **Mean Reaction Progress**, **Scaled Reaction Progress Variance**, **Mean Mixture Fraction**, and **Scaled Variance**.

Slice by

allows you to select whether the 3D array of data points available in the look-up table will be sliced by **Index** or **Value**.

Index/Value

contains index/values and their ranges.

Index/Value

allows you to specify the discretization index or numerical value of the variable that is being held constant.

Min/Max

are the range of integer values that you are allowed to choose from or display.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Surface Parameters

contains settings where discrete independent variables are held constant and where curve parameters are defined.

Constant Value of

contains the discrete independent variables that can be held constant in the lookup table. The choices are **Scaled Heat Loss/Gain**, **Mean Mixture Fraction**, and **Scaled Variance**. For a two-mixture-fraction case, the **Scaled Heat Loss/Gain** is the only available option. For the FGM model, this group box contains the discrete independent variables that have not been fixed in the **Volume Parameters** group box.

Slice by

allows you to select whether the 3D array of data points available in the look-up table will be sliced by **Index** or **Value**.

Index/Value

contains index/values and their ranges.

Index/Value

allows you to specify the discretization index or numerical value of the variable that is being held constant.

Min/Max

are the range of integer values that you are allowed to choose from or display.

Adiabatic

is the enthalpy slice index corresponding to the adiabatic case for which the enthalpy (**Scaled Heat Loss/Gain**) is held constant.

Curve Parameters

allows you to specify the **X-Axis Function** against which the plot variable will be displayed when **2D Curve on 3D Surface** is selected.

X-Axis Function

contains the remaining discrete independent parameters that have not been fixed yet.

Constant Value of *`Independent Parameter`*

allows you to specify the type of discretization for the variable that is being held constant using the controls in the **Slice by** and **Index/Value** group boxes. These items are similar to those described above.

Display

displays the plot variable of the 3D surface.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Plot

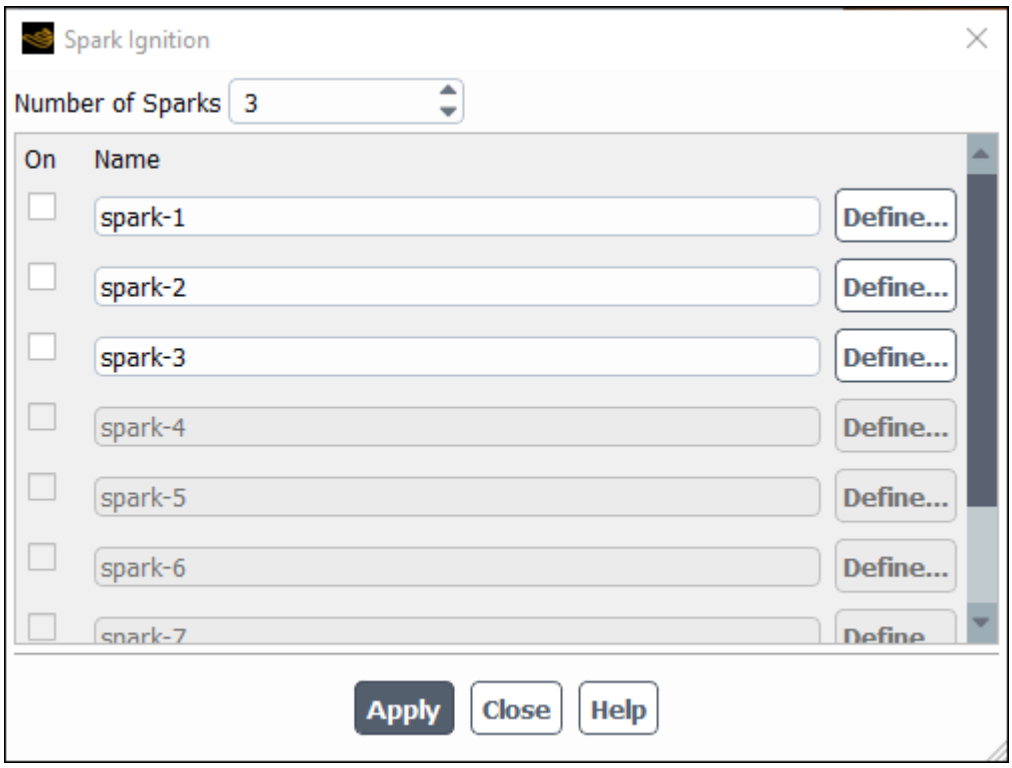
plots the plot variable for the 2D curve on 3D surface.

Write

opens The Select File Dialog Box (p. 931) where you will specify a name for the file containing plot data. This button appears when **Write To File** is enabled for the **2D Curve on 3D Surface** plot type.

51.4.28. Spark Ignition Dialog Box

The **Spark Ignition** dialog box allows you to define multiple sparks (see Using the Spark Model (p. 2571) for details).



Controls

Number of Sparks

is the quantity of sparks you would like to include in your simulation. You can define up to 16 sparks.

On

if enabled, turns on those sparks that will be included in the simulation.

Name

is the name of the spark. You can specify a name, or use the default name.

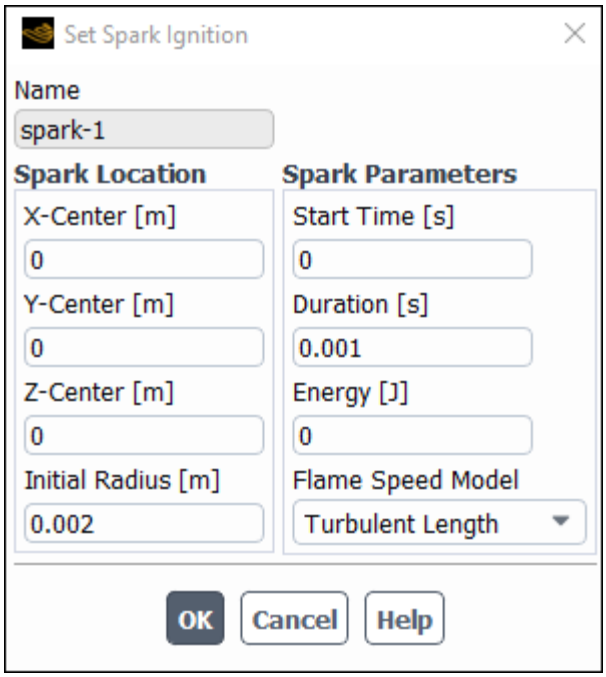
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Define...

opens the Set Spark Ignition Dialog Box (p. 4810).

51.4.29. Set Spark Ignition Dialog Box

The **Set Spark Ignition** dialog box allows you to set the parameters related to the spark ignition model (see Using the Spark Model (p. 2571) for details).



Controls

Name

displays the name of the spark being defined.

Spark Location

contains the parameters needed to define the location and size of the spark.

X, Y, and Z-Center

specifies x, y, and z coordinates of the spark center.

Initial Radius

specifies the initial spark radius.

Spark Parameters

contains the parameters needed to define the spark.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Start Time

is the time of spark ignition initialization in seconds. When the in-cylinder model is turned on, this control is replaced by the **Start Crank Angle** control.

Start Crank Angle

is the time of spark ignition initialization in crank angle degrees. When the in-cylinder model is turned off, this control is replaced by the **Start Time** option.

Duration

is the duration of the spark ignition in seconds.

Energy

contains the total energy input by the spark. The default energy value is 0. A positive value for energy will cause the temperature of the spark kernel to rise above the combustion process temperature.

Flame Speed Model

allows you to select the turbulent flame speed model for controlling the rate at which the flame front moves.

Turbulent Curvature

includes the effect of flame curvature as specified in Turbulent Curvature in the *Fluent Theory Guide*.

Turbulent Length

neglects the effects of flame curvature on the flame speed as described in Turbulent Length in the *Fluent Theory Guide*.

Herweg-Maly

calculates the turbulent flame speed using Herweg-Maly model. See Herweg-Maly in the *Fluent Theory Guide* for details.

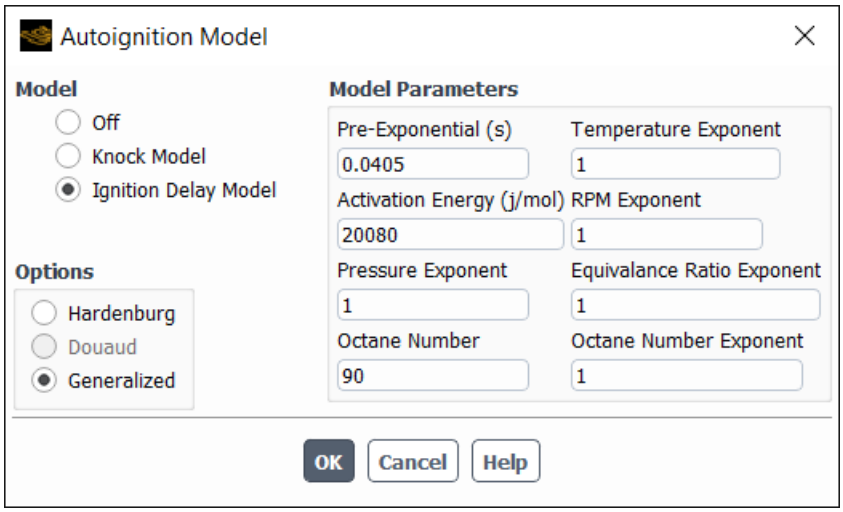
Laminar

specifies the turbulent flame speed as the laminar flame speed. This option can be used to apply user-defined function (UDF) for the turbulent flame speed definition.

51.4.30. Autoignition Model Dialog Box

The **Autoignition Model** dialog box allows you to set the parameters related to the **Knock Model** or the **Ignition Delay Model**. See Modeling Engine Ignition (p. 2571) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Model

contains options to disable or enable models.

Off

disables the model.

Knock Model

enables the knock model. With the **Premixed Combustion** or **Partially Premixed Combustion** models selected, only the **Knock Model** can be turned on.

Ignition Delay Model

enables the ignition delay model.

Options

contains two correlation options that exist with each model.

Douaud

option is used for knock in spark ignition engines. The modeling parameters that are specified for this option are the **Pre Exponential**, **Pressure Exponent**, **Activation Temperature**, **Octane Number**, and **Octane Exponent** (Equation 10.8 in the Theory Guide).

Generalized

enables generalized correlation described by Equation 10.9 in the Theory Guide. It requires the same parameters as in the ignition delay model.

Hardenburg

enables Hardenburg correlation, which is used for heavy-duty diesel engines. This option is enabled only for the ignition delay model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Model Parameters

contains parameters related to the selected model. See Using the Autoignition Models (p. 2574) for the details about the parameters in this dialog box.

Pre-Exponential

see Equation 10.9 and Equation 10.8 in the Theory Guide.

Activation Temperature

see Equation 10.8 in the Theory Guide.

Pressure Exponent

see Equation 10.9 and Equation 10.8 in the Theory Guide.

Octane Number

see Equation 10.9 and Equation 10.8 in the Theory Guide.

Octane Number Exponent

see Equation 10.9 and Equation 10.8 in the Theory Guide.

Activation Energy

see Equation 10.9 and Equation 10.11 in the Theory Guide.

Temperature Exponent

see Equation 10.9 in the Theory Guide.

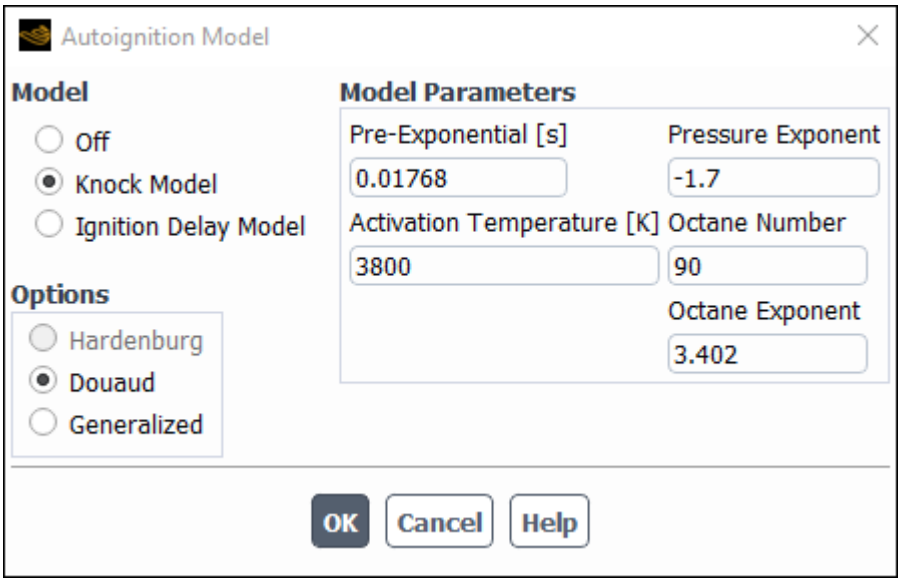
RPM Exponent

see Equation 10.9 in the Theory Guide.

Equivalence Ratio Exponent

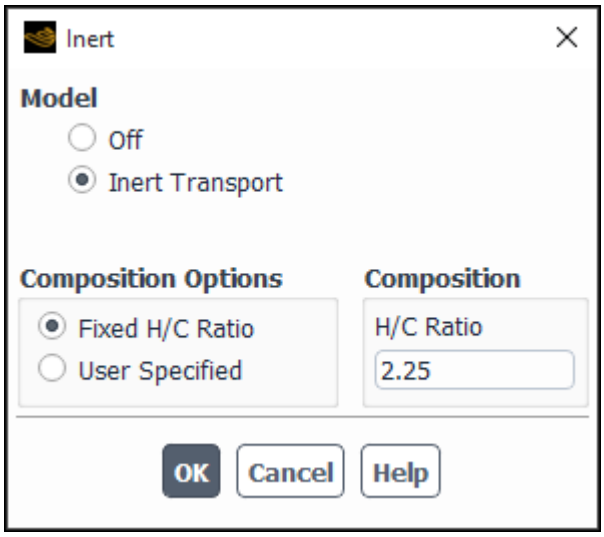
see Equation 10.9 in the Theory Guide.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



51.4.31. Inert Dialog Box

The **Inert** dialog box allows you to set the parameters related to the inert model. For details, see Setting Up the Inert Model (p. 2521).



Controls

Model

allows you to enable or disable the inert model.

Off

disables the model.

Inert Transport

enables the inert model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Composition Options

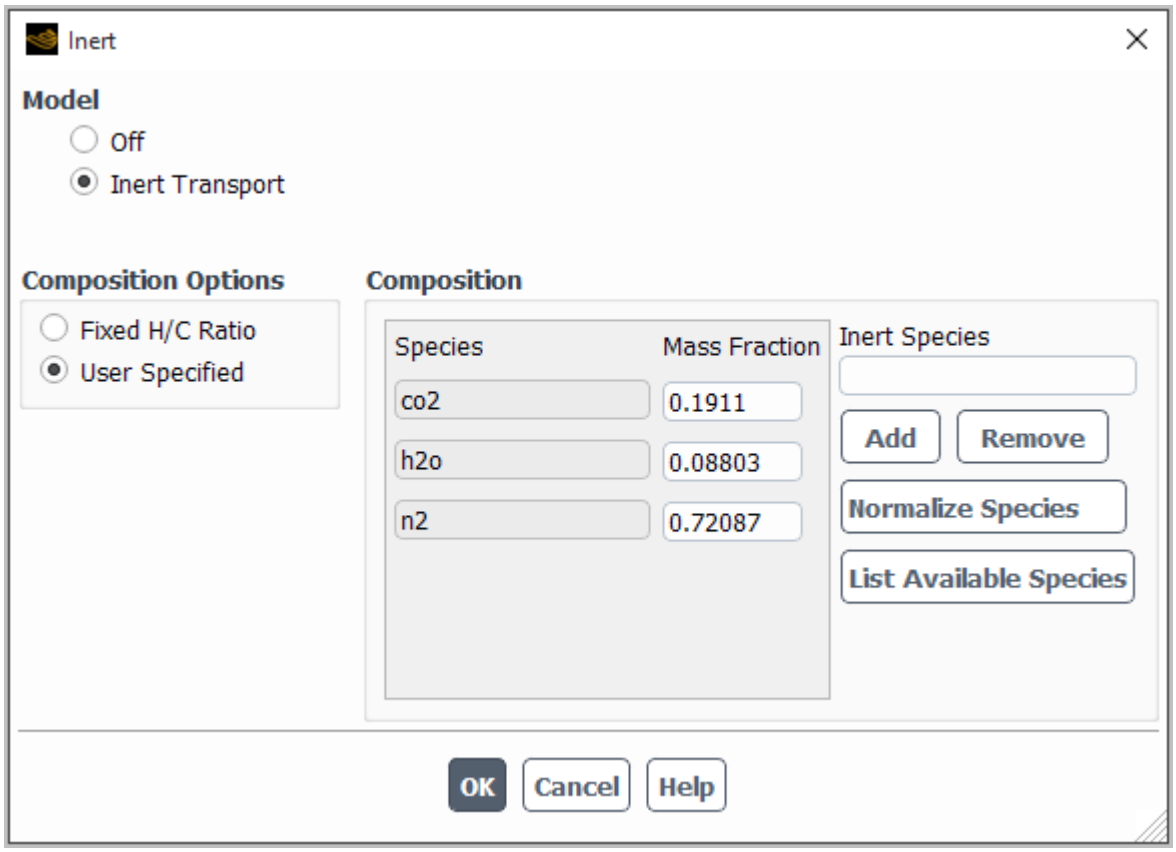
allows you to select a fixed H/C ratio or to specify the composition.

Fixed H/C Ratio

allows you to specify a fixed ratio of hydrogen to carbon in the **H/C Ratio** field.

User Specified

allows you to specify an arbitrary composition for the inert stream.



Composition

allows you to set the H/C ratio or the mass fraction.

When **Fixed H/C Ratio** is selected under **Composition Options**, the following option(s) are available:

H/C Ratio

specifies the fixed ratio of hydrogen to carbon when the **Fixed H/C Ratio** option is enabled.

When **User Specified** is selected under **Composition Options**, the following option(s) are available:

Species

lists the inert species name.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mass Fraction

displays the mass fraction of the corresponding species.

Inert Species

allows you to specify the name of the inert species.

Add

adds the specified inert species to the species list.

Remove

removes the specified inert species from the species list.

Normalize Species

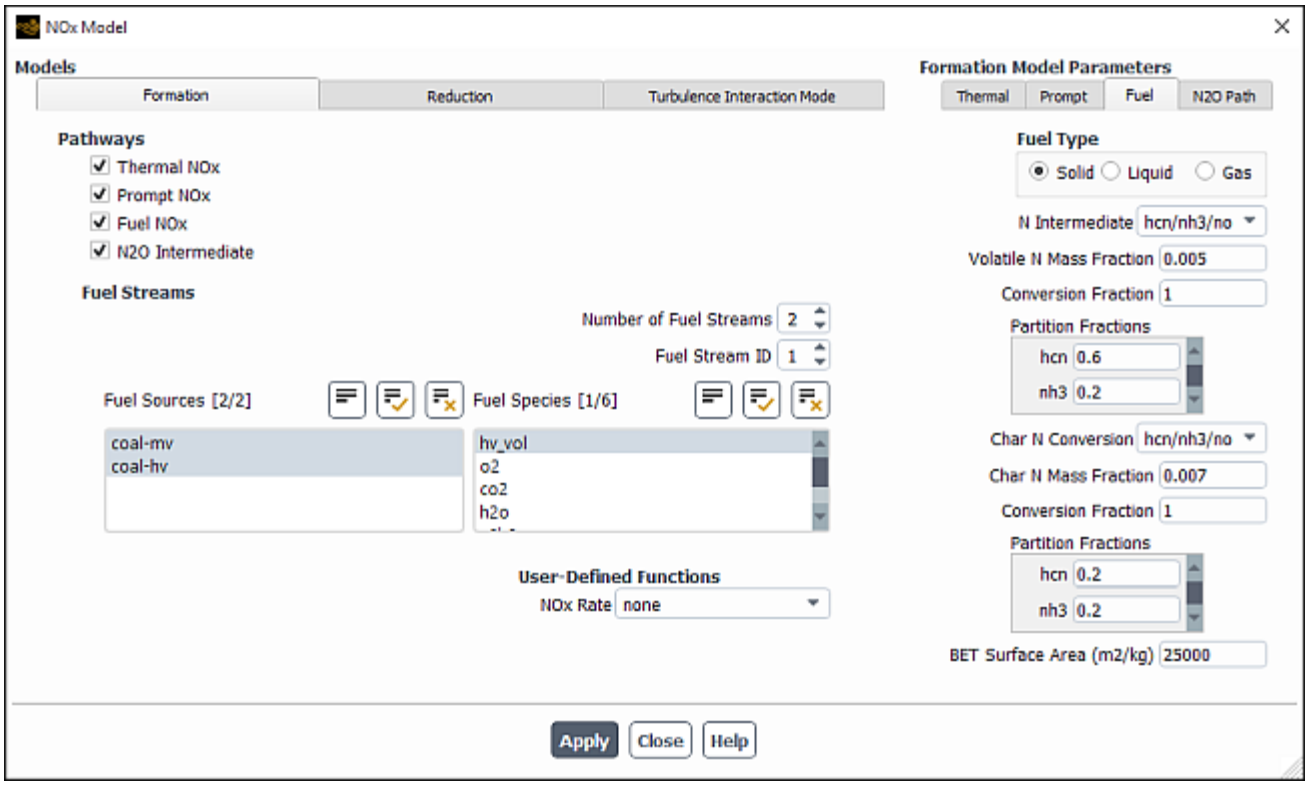
makes sure the species mass fractions add up to 1.

List Available Species

lists all species in the thermodynamic database file (`thermo.db`) in the console window,

51.4.32. NOx Model Dialog Box

The **NOx Model** dialog box allows you to set parameters related to the NOx postprocessor. See Using the NOx Model (p. 2585) for details about the items below.



Controls

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Models

contains tabs for defining the models used to calculate the NOx production.

Formation

contains the parameters to define the NOx model formation.

Pathways

contains toggle buttons for enabling the NOx models to be used for the calculation of NO and HCN concentrations.

Thermal NOx

enables calculation of thermal NOx.

Prompt NOx

enables the calculation of prompt NOx.

Fuel NOx

enables the calculation of fuel NOx. When using the non-premixed combustion model, the **Fuel NOx** option is only available if the DPM model is also enabled.

N2O Intermediate

enables the formation of NOx through an N_2O intermediate. This option will only appear if one of the previously listed NOx models is enabled.

Fuel Streams

allows you to define multiple fuel streams for prompt NOx and fuel NOx formation.

Number of Fuel Streams

sets the number of fuel streams. You are allowed up to three fuel streams.

Fuel Stream ID

specifies the fuel stream you are defining in the **PDF Stream** drop-down list, the **Fuel Species** selection list, the **Prompt** tab, and the **Fuel** tab.

PDF Stream

specifies the PDF stream species associated with a particular **Fuel Stream ID**, when calculating fuel NOx formation in conjunction with the non-premixed combustion model. You can select either the **primary** or **secondary** fuel streams, as defined in the PDF table.

Fuel Species

is a list containing all of the defined species, which allows you to specify the species that is the fuel associated with a particular **Fuel Stream ID**. You cannot select more

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

than 5 fuel species for each fuel stream, and the total number of fuel species selected for all the fuel streams combined cannot exceed 10. Note that when the non-premixed combustion model is enabled, your selection in the **Fuel Species** list only applies to prompt NOx calculations.

Fuel Sources

is a list containing all of the fuel N sources. This list is available when **Fuel NOx** is enabled and your case contains injections with different DPM materials defined, such as combusting particles and droplets.

User-Defined Functions

contains the **NOx Rate** drop-down list, which allows you to use a user-defined function (UDF) to contribute to the rate of NOx production. See the separate Fluent Customization Manual for details. Note that you may also use a UDF to specify custom values for the maximum limit (T_{max}) that is used for the integration of the temperature PDF (when temperature is accounted for in the turbulence interaction modeling).

Reduction

allows you to specify the reduction methods.

Methods

contains the list of reduction methods.

Reburn

enables the calculation of NOx reburning effects.

SNCR

enables the calculation of NOx reduction by the SNCR method.

Turbulence Interaction Mode

contains parameters related to the effect of turbulent fluctuations on the NOx formation. See NOx Formation in Turbulent Flows in the Theory Guide for details.

PDF Mode

is a drop-down list containing options that take into account turbulent fluctuations when you compute the specified NOx formation. See Setting Turbulence Parameters (p. 2598) for details.

none

specifies the use of laminar NOx rate calculations, so that the effects of turbulence are ignored.

temperature

includes fluctuations of temperature.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

temperature/species

includes fluctuations of the temperature and the mass fraction of the species selected in the **Species** drop-down list (which appears when you select this option).

mixture fraction

includes fluctuations of the mixture fraction(s). This is available for non-premixed combustion calculations only.

PDF Type

allows you to specify the shape of the PDF.

beta

models the PDF using Equation 9.108 in the Theory Guide.

gaussian

models the PDF using Equation 9.111 in the Theory Guide.

PDF Points

controls the number of points at which the beta function in Equation 9.105 or Equation 9.106 in the Theory Guide will be integrated. The default value of 10, which indicates that the beta function will be integrated at 10 points on a histogram basis. The default value should yield an accurate solution with a reasonable computation time. Increasing this value may improve accuracy, but will also increase the computation time. This field is only available when **temperature** or **temperature/species** is selected from the **PDF Mode** drop-down list.

Temperature Variance

allows you to specify the form of the transport equation that is solved to calculate the temperature variance.

algebraic

is an approximate form of the transport equation (see Equation 9.114 in the Theory Guide).

transported

solves Equation 9.113 in the Theory Guide.

Tmax Option

provides various options for determining the maximum limit(s) for the integration of the PDF used to calculate the temperature.

global-tmax

sets the limit as the maximum temperature in the flow field.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

local-tmax-factor

yields cell-based maximum temperature limits by multiplying the local cell mean temperature by the value entered in **Tmax Factor**.

specified-tmax

sets the limit for each cell to be the value entered in **Tmax**.

user-defined

allows you to hook a user-defined function that specifies custom values for the maximum limit (T_{max}), which is used for the integration of the temperature PDF. This option is only available if you have already compiled a user-defined function and selected it in the **Formation** tab.

Species

is a drop-down list that appears when **temperature/species** is selected from the **PDF Mode** drop-down list. Here you will select the species whose mass fraction fluctuations will be factored into the NOx rate calculations.

Formation Model Parameters

contains the tabs used to define the NOx pathways.

Thermal

contains parameters for modeling thermal NOx. (The contents of this tab will appear only if **Thermal NOx** is enabled in the **Formation** tab.)

[O] Model

is a drop-down list in which you can select the method to be used for calculation of thermal NOx. To choose the equilibrium method, select **equilibrium**. To choose the partial equilibrium method, select **partial-equilibrium**. To choose the predicted O concentration method, select **instantaneous**. See Method 1: Equilibrium Approach, Method 2: Partial Equilibrium Approach, and Method 3: Predicted O Approach in the Theory Guide for details.

[OH] Model

is a drop-down list in which you can select the method to be used for calculation of thermal NOx. To exclude OH, select **none**. To choose the partial equilibrium method, select **partial-equilibrium**. To choose the predicted OH concentration method, select **instantaneous**. See Method 1: Exclusion of OH Approach, Method 2: Partial Equilibrium Approach, and Method 3: Predicted OH Approach in the Theory Guide for details.

UDF Rate

provides options for the treatment of the NOx production specified by the user-defined function selected in the **Formation** tab.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Replace Fluent Rate

replaces Ansys Fluent's thermal NOx rate calculations with the custom NOx rate produced by your user-defined function.

Add to Fluent Rate

adds the custom NOx rate produced by your user-defined function to Ansys Fluent's thermal NOx rate calculations.

Prompt

contains parameters for modeling prompt NOx. (The contents of this tab will appear only if **Prompt NOx** is enabled in the **Formation** tab.) The settings made in this tab will be associated with a particular fuel stream, specified in the **Fuel Stream ID** field in the **Formation** tab.

Fuel Carbon Number

specifies the number of carbon atoms per fuel molecule.

Equivalence Ratio

is the ratio of the actual fuel/air ratio to the stoichiometric fuel/air ratio.

UDF Rate

provides options for the treatment of the NOx production specified by the user-defined function selected in the **Formation** tab.

Replace Fluent Rate

replaces Ansys Fluent's prompt NOx rate calculations with the custom NOx rate produced by your user-defined function.

Add to Fluent Rate

adds the custom NOx rate produced by your user-defined function to Ansys Fluent's prompt NOx rate calculations.

Fuel

contains parameters for modeling fuel NOx. (The contents of this tab will appear only if **Fuel NOx** is enabled in the **Formation** tab.) The settings made in this tab will be associated with a particular fuel stream, specified in the **Fuel Stream ID** field in the **Formation** tab.

Fuel Type

specifies the type of fuel NOx to be calculated.

Solid

enables the calculation of solid fuel NOx.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Liquid

enables the calculation of liquid fuel NOx.

Gas

enables the calculation of gas fuel NOx.

N Intermediate

allows you to specify any one of the **hcn**, **nh3**, or **hcn/nh3/no** as the intermediate species. See Fuel NOx Formation in the Theory Guide for details.

Volatile N Mass Fraction

specifies the mass fraction of nitrogen in the volatiles. This parameter appears only for **Solid** fuel NOx calculations.

Fuel N Mass Fraction

specifies the mass fraction of nitrogen in the fuel. This parameter appears only for **Gas** or **Liquid** fuel NOx calculations.

Conversion Fraction

specifies the overall mass fraction of fuel N (for gas and liquid fuels), or volatile N or char N (for solid fuels), that will be converted to intermediate species and/or product NO.

Partition Fractions

specifies the mass fraction of the converted fuel N (for gas and liquid fuels), or volatile N or char N (for solid fuels), that will become **hcn** and **nh3**. The fraction that will become NO will be calculated by the remainder. This option will appear only if you have selected **hcn/nh3/no** for the **N Intermediate** or **Char N Conversion** drop-down lists.

Char N Conversion

is a drop-down list in which you can select **no**, **hcn**, **nh3**, or **hcn/nh3/no** as the species to which the char N is converted (when you are calculating solid fuel NOx). This parameter appears only for **Solid** fuel NOx calculations. See Setting Solid (Coal) Fuel NOx Parameters (p. 2593) for details.

Char N Mass Fraction

specifies the mass fraction of nitrogen in the char. This parameter appears only for **Solid** fuel NOx calculations.

BET Surface Area

sets the BET internal pore surface area (see BET Surface Area in the Theory Guide for details) of the particles. This parameter appears only for **Solid** fuel NOx calculations.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UDF Rate

provides options for the treatment of the NOx production specified by the user-defined function selected in the **Formation** tab.

Replace Fluent Rate

replaces Ansys Fluent's fuel NOx rate calculations with the custom NOx rate produced by your user-defined function.

Add to Fluent Rate

adds the custom NOx rate produced by your user-defined function to Ansys Fluent's fuel NOx rate calculations.

N2O Path

contains the method to be used for formation of NO through an N_2O intermediate. (The contents of this tab will appear only if **N20 Intermediate** is enabled in the **Formation** tab.)

N2O Model

contains the drop-down list of available N_2O models.

quasi-steady

enables the quasi-steady-state method of calculation (The transport equation for the species N_2O will not be solved).

transported-simple

enables the transported simple method of calculation (The pollutant species N_2O is added in the species list and it's mass fraction will be calculated using the transport equations).

UDF Rate

provides options for the treatment of the NOx production specified by the user-defined function selected in the **Formation** tab.

Replace Ansys Fluent Rate

replaces the NOx rate calculated by Ansys Fluent using N_2O intermediates with the custom NOx rate produced by your user-defined function.

Add to Ansys Fluent Rate

adds the custom NOx rate produced by your user-defined function to the NOx rate calculated by Ansys Fluent using N_2O intermediates.

Reduction Method Parameters

contains tabs that allow you to define the methods of reduction. (These tabs are do not appear unless a reduction method has been enabled in the **Reduction** tab.)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reburn

allows you to define the NOx reduction when **Reburn** is enabled in the **Reduction** tab.

Reburn Model

contains the drop-down list of reburn methods.

instantaneous[CH]

enables the instantaneous method in the **Reburn Model**. When you choose this method a warning to include CH, CH_2, and CH_3 will be displayed.

partial-equilibrium

activates partial method in the **Reburn Model**.

Reburn Fuel Species

contains reburn fuel species drop-down list.

Equivalent Fuel Type

contains equivalent fuel type drop-down list.

SNCR

allows you to define the NOx reduction when **SNCR** is enabled in the **Reduction** tab.

Injection Method

contains the parameters for NOx reduction by SNCR method.

gaseous

includes ammonia or urea as a gas-phase pollutant species from the injection locations.

liquid

includes ammonia or urea as a liquid-phase pollutant species from the injection locations.

Reagent Species

allows you to specify the reagent species as either ammonia (**nh3**) or urea (**co<nh2>2**)

Reagent Fraction in Stream

allows you to specify the mass fraction of the reagent in the reagent stream. The remaining mass fraction is assumed to be water. If you enabled a secondary stream in your PDF calculation, by default the secondary stream will act as the reagent stream. Note that the **Reagent Fraction in Stream** field is only available when using the non-premixed combustion model with a liquid-phase reagent injection.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Urea Decomposition

allows you to specify the decomposition model to use when the selected **Reagent Species** is **co<nh2>2**.

rate-limiting

specifies that the source terms be calculated according to the rates given in Table 9.3: Two-Step Urea Breakdown Process.

user-specified

allows you to specify the molar conversion fraction for ammonia, assuming that the rest of the urea is converted to HNCO.

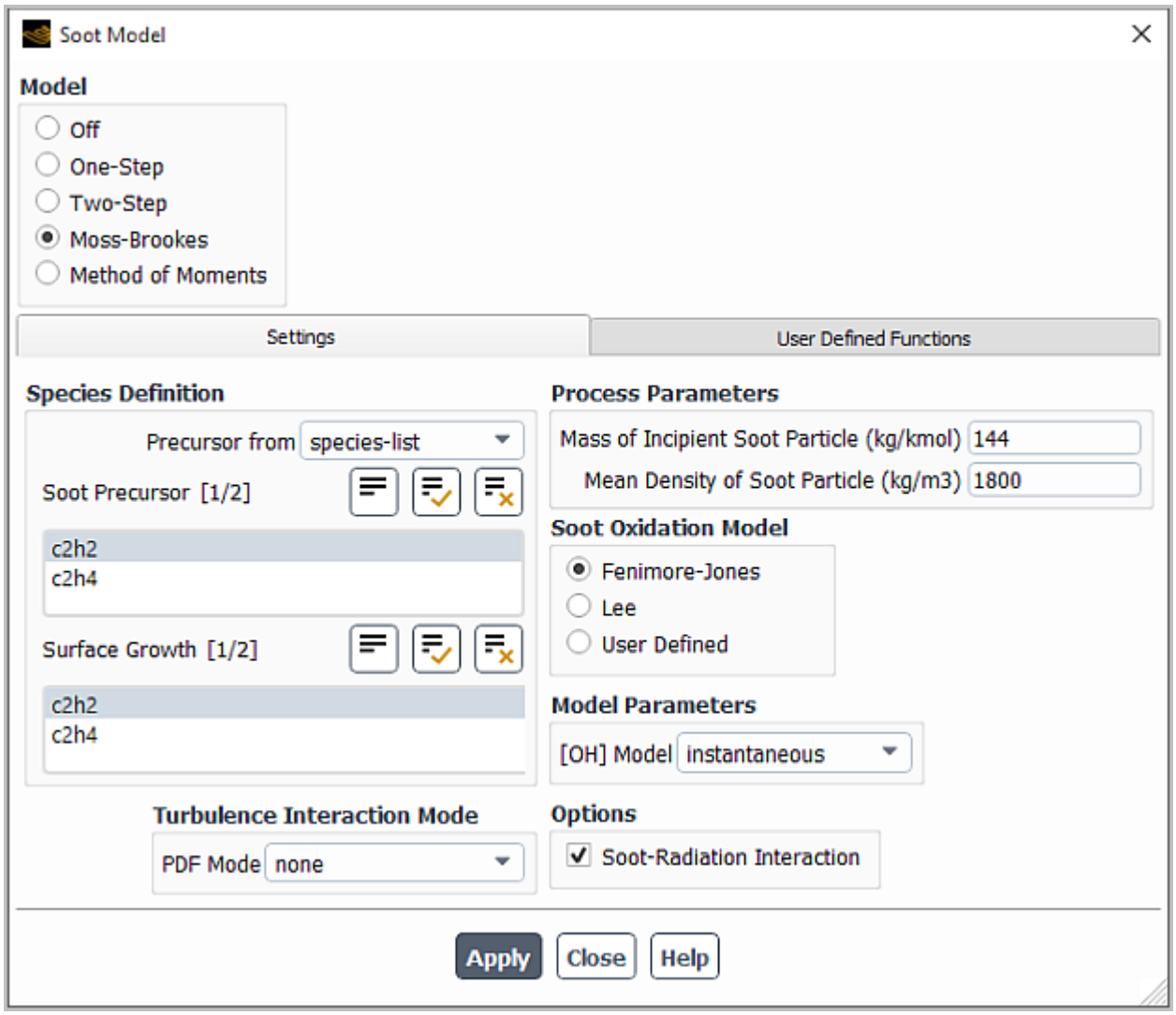
NH3 Conversion

is the mole fraction of NH_3 in the mixture of NH_3 and HNCO instantly created from the reagent injection. The **NH3 Conversion** field only appears when **user-specified** is selected for **Urea Decomposition**.

51.4.33. Soot Model Dialog Box

The **Soot Model** dialog box allows you to set parameters related to the soot model. See Using the Soot Models (p. 2603) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Model

specifies which model should be used for computing soot formation.

Off

disables the calculation of soot formation.

One-Step

enables the one-step soot model described in The One-Step Soot Formation Model in the Theory Guide.

Two-Step

enables the two-step soot model described in The Two-Step Soot Formation Model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Moss-Brookes

enables the Moss-Brookes soot model described in The Moss-Brookes Model in the Theory Guide.

Moss-Brookes-Hall

enables the Moss-Brookes-Hall soot model described in The Moss-Brookes-Hall Model in the Theory Guide. This option is only available when C_2H_2, C_6H_6, C_6H_5, and H_2 are present in the gas phase species list.

Method of Moment

enables the Method of Moment soot model described in Setting Up the Method of Moments Soot Model (p. 2616).

Species Definition

contains inputs for specifying the chemical species for your model.

Fuel

is a drop-down list containing all of the defined species. Here you will select the species that is the fuel for the **One-Step** and **Two-Step** models, as well as the **Moss-Brookes** model when a precursor species is not identified in the defined species list.

Oxidant

is a drop-down list containing all of the defined species. Here you will select the species that is the oxidizer for the **One-Step** and **Two-Step** models.

Precursor from

allows you to select from a list of species, enter the correlation values of species, or enables you to hook a user-defined function, used to define the user defined precursor species. This selection is available when using the **Moss-Brookes** and **Moss-Brookes-Hall** models.

User Defined Precursor

allows you to hook a user-defined function to specify the user defined soot precursor. This selection is available when using the **Moss-Brookes** and **Moss-Brookes-Hall** models.

Soot Precursor

is a selection list containing all of the possible precursor species found via a query of the defined species list. By default, Ansys Fluent only considers c2h2, c6h6, and c2h4 as possible precursor species. For information about including other species in the possible precursor species search, contact your Ansys Fluent support engineer. From this list you will select the species that are the soot precursor species for the **Moss-Brookes**, **Moss-Brookes-Hall**, or **Method of Moments** models.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Surface Growth

is a selection list containing all of the possible surface growth species, as explained previously for the **Soot Precursor** selection list. Here you will select the species that are the surface growth species for the **Moss-Brookes** and **Moss-Brookes-Hall** models.

Fuel Carbon Number

is the number of carbon atoms in the species selected in the **Fuel** drop-down list. This field appears only for the **Moss-Brookes** and **Moss-Brookes-Hall** model, when **user-correlation** is selected in the **Precursor from** drop-down list.

Fuel Hydrogen Number

is the number of hydrogen atoms in the species selected in the **Fuel** drop-down list. This field appears only for the **Moss-Brookes** and **Moss-Brookes-Hall** model, when **user-correlation** is selected in the **Precursor from** drop-down list.

Molecular Weight of Precursor

is the molecular weight of the precursor species. This field appears only for the **Moss-Brookes** and **Moss-Brookes-Hall** model, when **user-correlation** is selected in the **Precursor from** drop-down list. The default value is the weight of acetylene.

Precursor Correlation

is a drop-down list you can use to define a laminar diffusion profile that relates mixture fraction to precursor mass fraction. This field appears only for the **Moss-Brookes** and **Moss-Brookes-Hall** model, when **user-correlation** is selected in the **Precursor from** drop-down list.

piecewise-polynomial

specifies that the precursor mass fraction is a piecewise-polynomial function of mixture fraction. The default values used by Ansys Fluent correspond to a methane diffusion flame simulation, in which both the air and fuel initial temperatures are set to 290 K, and acetylene is assumed as the soot precursor. These values can be revised via the **Edit...** button.

constant

specifies that the precursor mass fraction is a constant function of mixture fraction, the value of which is specified in the field below the **Precursor Correlation** drop-down list.

Edit...

opens the Piecewise-Polynomial Profile Dialog Box (p. 4909) when **piecewise-polynomial** is selected from the **Precursor Correlation** drop-down list, thus allowing you to revise the default values.

Sticking Coefficients...

(only for the method of moments soot model) opens Sticking Coefficients Dialog Box (p. 4835) where you can provide the correction factors to be used for calculating the nucleation rates. This button is available only when **User Precursor** is the selected soot nucleation model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Turbulence Interaction Mode

contains inputs that specify how turbulent fluctuations are accounted for in the soot formation calculations for the **Moss-Brookes** and **Moss-Brookes-Hall** models. For further details on these inputs, see Setting Up the Moss-Brookes Model and the Hall Extension (p. 2608).

PDF Mode

is a drop-down list that contains the options for addressing turbulent fluctuations in the soot rate calculations. Note that **mixture fraction** is the most accurate option, and should be used if it is available.

none

specifies the use of laminar soot rate calculations, so that the effects of turbulence are ignored.

temperature

specifies that the soot rate calculations include the effect of temperature fluctuations.

temperature/species

specifies that the soot rate calculations include the effect of fluctuations of temperature, as well as fluctuations of the mass fraction of the species selected in the **Species** drop-down list (which appears when you select this option).

mixture fraction

is the most accurate option, specifying that the soot rate calculations include the effect of fluctuations of mixture fraction(s). Note that this option is not available if you are using the eddy-dissipation model.

PDF Type

allows you to specify the shape of the PDF.

beta

models the PDF using Equation 9.108 in the Theory Guide.

gaussian

models the PDF using Equation 9.111 in the Theory Guide.

PDF Points

controls the number of points at which the beta function will be integrated on a histogram basis. Increasing this number may improve accuracy, but will also increase compute time. This field is only available when **temperature** or **temperature/species** is selected from the **PDF Mode** drop-down list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Temperature Variance

allows you to specify the form of transport equation that is solved to calculate the temperature variance.

algebraic

is an approximate form of the transport equation (see Equation 9.114 in the Theory Guide).

transported

solves Equation 9.113 in the Theory Guide.

Tmax Option

provides various options for determining the maximum limit(s) for the integration of the PDF used to calculate the temperature.

global-tmax

sets the limit as the maximum temperature in the flow field.

local-tmax-factor

yields cell-based maximum temperature limits by multiplying the local cell mean temperature by the value entered in **Tmax Factor**.

specified-tmax

sets the limit for each cell to be the value entered in **Tmax**.

user-defined

allows you to hook a user-defined function that specifies custom values for the maximum limit (T_{max}), which is used for the integration of the temperature PDF. This option is only available if you have already compiled a user-defined function.

Species

is a drop-down list which appears when **temperature/species** is selected from the **PDF Mode** drop-down list. Here you will select the species whose mass fraction fluctuations will be factored into the soot rate calculations.

Process Parameters

contains parameters that control the combustion process modeling.

Mean Diameter of Soot Particle

is the assumed average diameter of the soot particles in the combustion system. For the **Two-Step** model, it is used to compute the soot particle mass m_p in Equation 9.123 in the Theory Guide.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mean Density of Soot Particle

is the assumed average density of the soot particles in the combustion system. For the **Two-Step** model, it is used to compute the soot particle mass m_p in Equation 9.123 in the Theory Guide. For the **Moss-Brookes** and **Moss-Brookes-Hall** models, it is ρ_{soot} in Equation 9.130 and ρ in Equation 9.133 in the Theory Guide. The default value supplied by Ansys Fluent is 1800 kg/m^3 (as was used in the work of Brookes and Moss [28] (p. 5714)).

Normalization Parameter for Moments

(only for the method of moments soot model) is used to solve the normalized soot moment transport equations to minimize the numerical errors. The default value of this input is sufficient and you may not need to change this.

Stoichiometry for Soot Combustion

is the mass stoichiometry ν_{soot} in Equation 9.120 in the Theory Guide, which computes the soot combustion rate in the **One-Step** and **Two-Step** models. The default value supplied by Ansys Fluent (2.6667) assumes that the soot is pure carbon and that the oxidizer is O_2.

Stoichiometry for Fuel Combustion

is the mass stoichiometry ν_{fuel} in Equation 9.120 in the Theory Guide, which computes the soot combustion rate in the **One-Step** and **Two-Step** models. The default value supplied by Ansys Fluent (3.6363) is for combustion of propane (C_3H_8) by oxygen (O_2).

Mass of Incipient Soot Particle

is M_P in Equation 9.131 and Equation 9.133, which is used in the **Moss-Brookes** and **Moss-Brookes-Hall** model computations. The default value supplied by Ansys Fluent (144 kg/mol) is the mass of 12 carbon atoms. Note that for the original implementation of the Hall extension, the model assumed this mass to be 100 carbon atoms.

Soot Oxidation Model

contains model options that determine the form of the soot oxidation term in the calculations of the **Moss-Brookes** and **Moss-Brookes-Hall** models.

Fenimore-Jones

takes into account the soot oxidation due to the hydroxyl radical.

Lee

takes into account the soot oxidation due to the hydroxyl radical and molecular oxygen.

User Defined

enables you to select a user-defined function for soot oxidation rate. This selection is available when using the **Moss-Brookes** and **Moss-Brookes-Hall** models.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Number of Moments

(only for the method of moments soot model) specifies the number of moments to be used in the Method of Moments soot model. Ansys Fluent will solve an equal number of moment transport equations. The default value of three moments works reasonably well for wide range of problems. You can specify up to six moments.

Soot Nucleation Model

(only for the method of moments soot model) contains model options related to soot nucleation modeling.

Use Precursor

enables you to define the precursor species. The nucleation rates are calculated considering coagulation between precursor species using kinetic theory.

Specify Mechanism

enables you to specify nucleation in terms of reactions.

User Defined Oxidation Rate

allows you to hook the user-defined function to specify the soot oxidation rate. This selection is available when using the **Moss-Brookes** and **Moss-Brookes-Hall** models.

Model Parameters

contains parameters that control the soot formation model.

Soot Formation Constant

is the parameter C_s in Equation 9.117 in the Theory Guide. This item appears only for the **One-Step** soot model.

Equivalence Ratio Exponent

is the exponent r in Equation 9.117 in the Theory Guide. This item appears only for the **One-Step** soot model.

Equivalence Ratio Minimum

and **Equivalence Ratio Maximum** are the minimum and maximum values of the fuel equivalence ratio φ in Equation 9.117 in the Theory Guide. Equation 9.117 will be solved only if **Equivalence Ratio Minimum** $<\ \varphi\ <$ **Equivalence Ratio Maximum**; if φ is outside of this range, there is no soot formation. This item appears only for the **One-Step** soot model.

Activation Temperature of Soot Formation Rate

is the term $E/\ R$ in Equation 9.117 in the Theory Guide. This item appears only for the **One-Step** soot model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Magnussen Constant for Soot Combustion

is the constant A used in the rate expressions governing the soot combustion rate (Equation 9.119 and Equation 9.120) in the Theory Guide. This item appears only for the **One-Step** and the **Two-Step** soot models. For the **Two-Step** model, this input will be called **Magnussen Constant for Soot and Nuclei Combustion**.

Limiting Nuclei Formation Rate

is the limiting value of the kinetic nuclei formation rate η_0 in Equation 9.126 in the Theory Guide. Below this limiting value, the branching and termination term, $(f-g)$ in Equation 9.125 in the Theory Guide, is not included. This item appears only for the **Two-Step** soot model.

Nuclei Branching-Termination Coefficient

is the term $\left(f-g\right)$ in Equation 9.125 in the Theory Guide. This item appears only for the **Two-Step** soot model.

Nuclei Coefficient of Linear Termination on Soot

is the term g_0 in Equation 9.125 in the Theory Guide. This item appears only for the **Two-Step** soot model.

Pre-Exponential Constant of Nuclei Formation

is the pre-exponential term a_0 in the kinetic nuclei formation term, Equation 9.126 in the Theory Guide. This item appears only for the **Two-Step** soot model.

Activation Temperature of Nuclei Formation Rate

is the term E/R in the kinetic nuclei formation term, Equation 9.126 in the Theory Guide. This item appears only for the **Two-Step** soot model.

Alpha for Soot Formation Rate

is α, the constant in the soot formation rate equation, Equation 9.123 **Two-Step** soot model.

Beta for Soot Formation Rate

is β, the constant in the soot formation rate equation, Equation 9.123 in the Theory Guide. This item appears only for the **Two-Step** soot model.

Magnussen Constant for Soot and Nuclei Combustion

is the constant A used in the rate expressions governing the soot combustion rate (Equation 9.119 and Equation 9.120 in the Theory Guide). This item appears only for the **One-Step** and the **Two-Step** soot models.

[OH] Model

is a drop-down list that allows you to specify the method by which the OH radical concentration is calculated, that is, **instantaneous** or **partial-equilibrium**. This list appears only for the **Moss-Brookes**, **Moss-Brookes-Hall**, and **Method of Moments** soot models.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

[O] Model

is a drop-down list that specifies the method by which the O radical concentration is calculated, that is, **equilibrium**, **partial-equilibrium**, or **instantaneous**. This list appears only for the **Moss-Brookes** and the **Moss-Brookes-Hall** soot models, when you have selected **partial-equilibrium** for the **[OH] Model**.

Important:

To use the concentration of OH or O predicted by the combustion model, select **instantaneous** for **[OH] Model** or **[O] Model**.

User Defined Functions

contains the **User Defined Soot MOM Rates** drop-down list, which allows you to use a user-defined function (UDF) to specify soot nucleation, surface growth, and oxidation rates for soot moment equations. This item is available only for the Method of Moments soot model. See Hooking `DEFINE_SOOT_MOM_RATES` UDFs in the *Fluent Customization Manual* for details.

Options

contains an option for modeling the effect of soot on a variable radiation absorption coefficient. This group box will appear only when one of the radiation models in the Radiation Model Dialog Box (p. 4736) is active.

Soot-Radiation Interaction

enables the soot-radiation interaction model described in The Effect of Soot on the Absorption Coefficient in the Theory Guide.

Soot Surface Growth Options

(only for the method of moments soot model) contains a parameter for modeling the soot surface growth.

Site Density

is the number of sites/cm^2 available on the soot surface for surface growth and oxidation reactions with the method of moment soot model.

Enable Aggregation Model

(only for the method of moments soot model) enables the modeling of soot aggregates.

Critical diameter

is the value of the soot particle diameter above which the coagulation regime is switched from coalescent to aggregate.

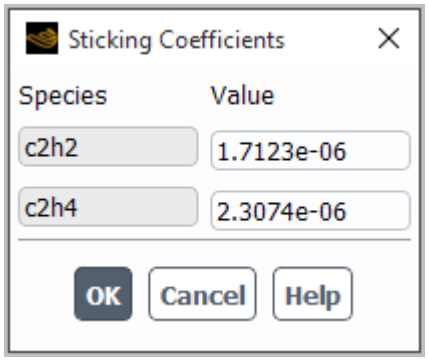
Fractal dimension

is the parameter that describes the aggregate nature. For soot particles, a value between 1.7 and 2.0 is found to be reasonably good.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

51.4.34. Sticking Coefficients Dialog Box

The **Sticking Coefficients** dialog box allows you to set sticking coefficients for the soot model with method of moments. It is opened from the Soot Model Dialog Box (p. 4825). See Setting Up the Method of Moments Soot Model (p. 2616) for details about using this dialog box.



Controls

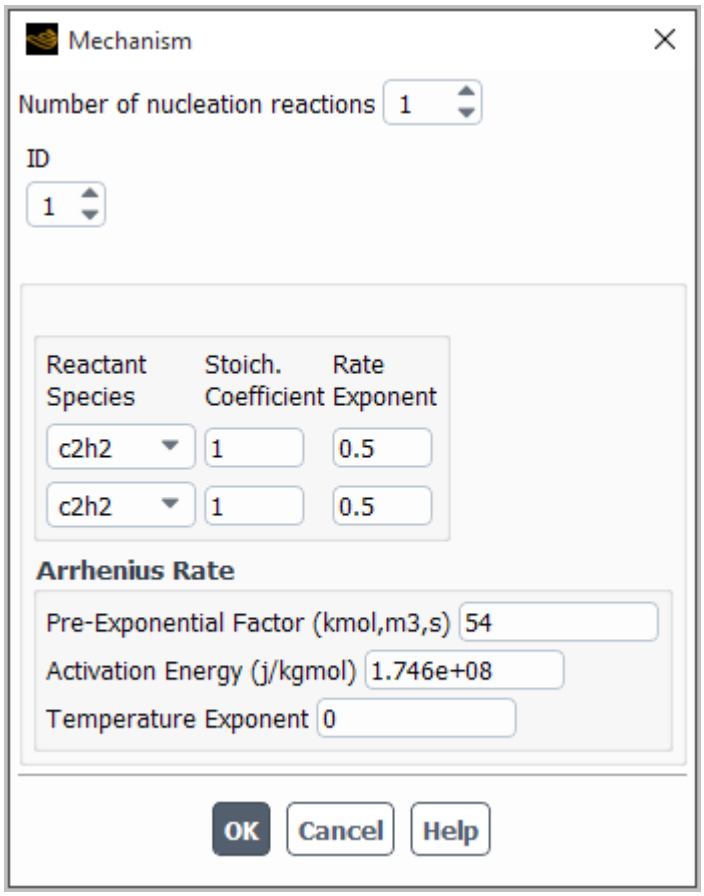
Species

shows the name of the species for which you specify the **Value**.

51.4.35. Mechanism Dialog Box

The **Mechanism** dialog box allows you to set parameters related to nucleation reactions for the soot model with method of moments. It is opened from the Soot Model Dialog Box (p. 4825). See Setting Up the Method of Moments Soot Model (p. 2616) for details about using this dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Number of nucleation reactions

allows you to specify nucleation process as with a single or multi-step reaction mechanism. For each reaction, you must specify all appropriate reaction parameters.

ID

displays the ID of the current nucleation reaction setup.

Reactant Species

is a drop-down list that enables you to select the species for the current nucleation reaction.

Stoich. Coefficient

specifies the stoichiometric coefficient of the reactant species in the nucleation reaction.

Rate Exponent

specifies the rate constant for the reactant species in the current nucleation reaction.

Arrhenius Rate

contains the inputs for the Arrhenius rate parameters related to the current nucleation reaction.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Pre-exponential Factor

is the constant A_r in Equation 7.23 in the Theory Guide. The units of A_r must be specified such that the units of the molar reaction rate, $R_{i,r}$ (Equation 7.19 in the Theory Guide, are moles/volume-time (for example, kmol/m^3-s) and the units of the volumetric reaction rate, R_i in Equation 7.19 in the Theory Guide, are mass/volume-time (for example, kg/m^3-s).

Important:

It is important to note that if you have selected the British units system, the Arrhenius factor should still be specified in SI units. This is because Ansys Fluent applies no conversion factor to the value you enter for A_r (the conversion factor is 1.0) when you work in British units, as the correct conversion factor depends on your values for $\nu_{j,r}$, β_r, and so on.

Activation Energy

is the constant E_r in the forward rate constant expression, Equation 7.23 in the Theory Guide).

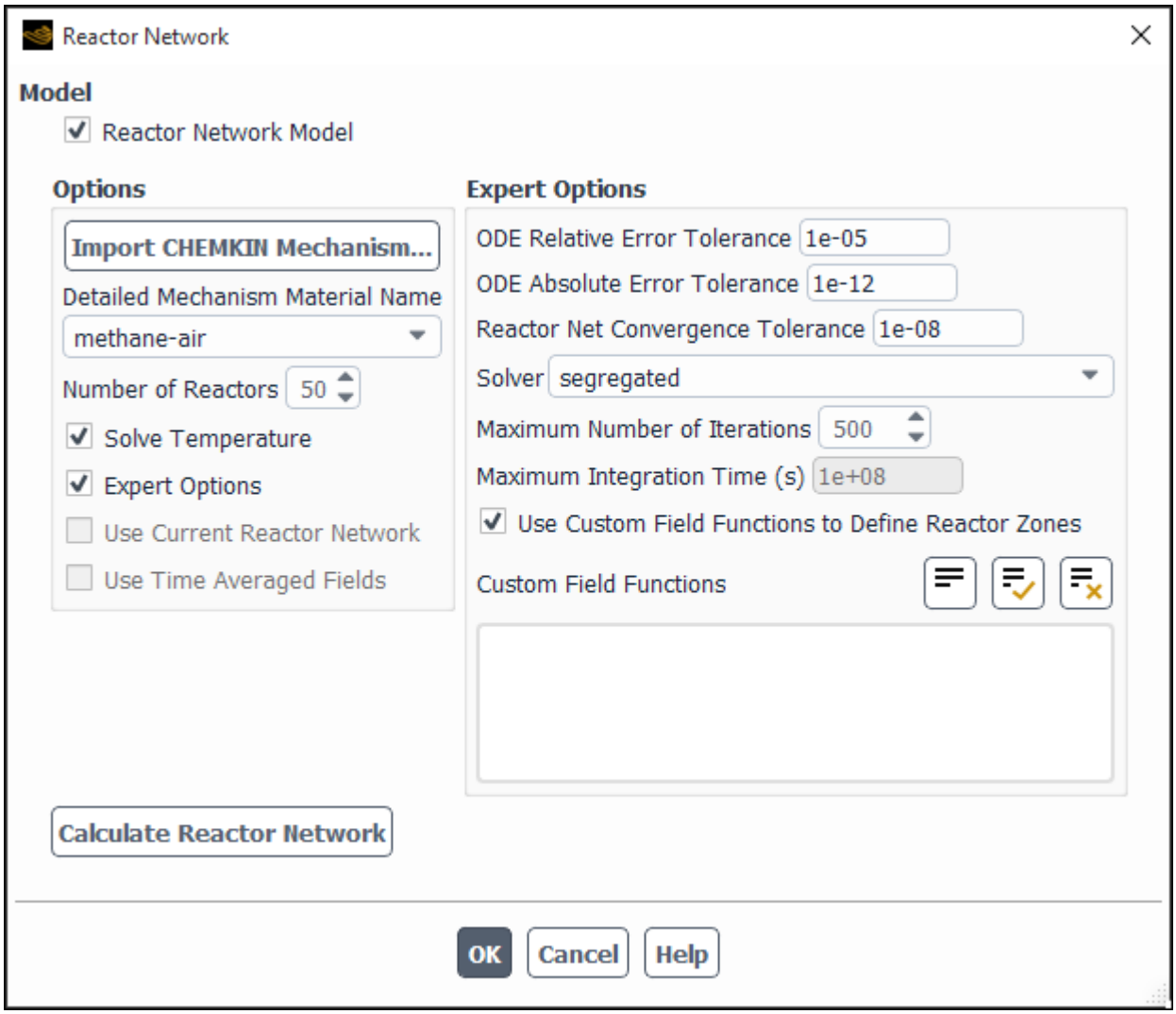
Temperature Exponent

is the value for the constant β_r in Equation 7.23 in the Theory Guide.

51.4.36. Reactor Network Dialog Box

The **Reactor Network** dialog box allows you to set parameters related to the reactor network model and calculate the reactor network for species transport/combustion. See Reactor Network Model (p. 2437) for details about using this dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Model

contains the option for enabling (or disabling) the reactor network model.

Reactor Network Model

enables/disables the modeling of the reactor network.

Options

contains the options for reactor network model. This portion of the dialog box appears only if **Reactor Network Model** is selected.

Import CHEMKIN Mechanism

allows you to import a detailed chemical mechanism in CHEMKIN format.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Detailed Mechanism Material Name

is a drop-down list of available mixture materials from which you can select a mixture material for your reactor network simulation.

Number of Reactors

sets the number of reactors in which cells are grouped based on specified criteria.

Solve Temperature

specifies whether or not the reactor network algorithm solves the temperature. By default, this option is selected.

Expert Option

when selected, displays the advanced options related to the solver settings.

Use Current Reactor Network

allows you to perform additional reactor network simulations with the existing network of connected reactors. This option can be enabled once the Fluent solver agglomerates cells and obtains the reactor-network solution.

Use Time Averaged Fields

enables you to select the time-averaged composition fields to be used for clustering reactors and the time-averaged velocity fields for calculating reactor mass flux matrix. This option is available only for unsteady simulations with enabled **Data Sampling For Unsteady Statistics** option.

Expert Options

contains advanced options for ODE solver and for reactor cell clustering control. This portion of the dialog box appears only if **Expert Options** is selected.

ODE Relative Error Tolerance

specifies relative error control for ODE integrator.

ODE Absolute Error Tolerance

specifies absolute error control for ODE integrator.

Reactor Network Convergence Tolerance

allows you to specify tolerance to control the accuracy of the reactor network calculations.

Solver

allows you to select the solver for your simulation. The default segregated solver typically converges faster than the coupled solver. However, if residuals stall above an acceptable tolerance, the coupled solver should be used.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Maximum Number of Iterations

(segregated solver only) sets the maximum number of solver iterations.

Maximum Integration Time

(coupled solver only) sets the maximum elapsed run time for ODE solver. Ansys Fluent terminates the run at this time if the residuals fail to converge.

Use Custom Field Functions to Define Reactor Zones

enables the use of custom field functions (Custom Field Functions (p. 4285)) for reactor network cell clustering.

Custom Field Functions

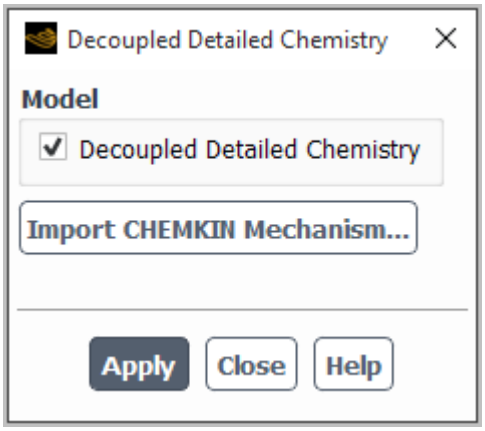
contains the list of available user-defined field functions from which you can select the functions for reactor network cell clustering. This selection list will not appear if the **Use Custom Field Functions to Define Reactor Zones** option is not selected.

Calculate Reactor Network

calculates a chemically reactive flow using reactor network model.

51.4.37. Decoupled Detailed Chemistry Dialog Box

The **Decoupled Detailed Chemistry** dialog box allows you to postprocess slowly-forming, trace pollutant species on a steady-state flow field using detailed chemical kinetic mechanisms.



Controls

Model

includes the option to enable **Decoupled Detailed Chemistry**.

Import CHEMKIN Mechanism...

opens the Import CHEMKIN Format Mechanism Dialog Box (p. 5358).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Integration Parameters...

opens the Integration Parameters Dialog Box (p. 4795).

Original Species

are the species that are in the original case setup.

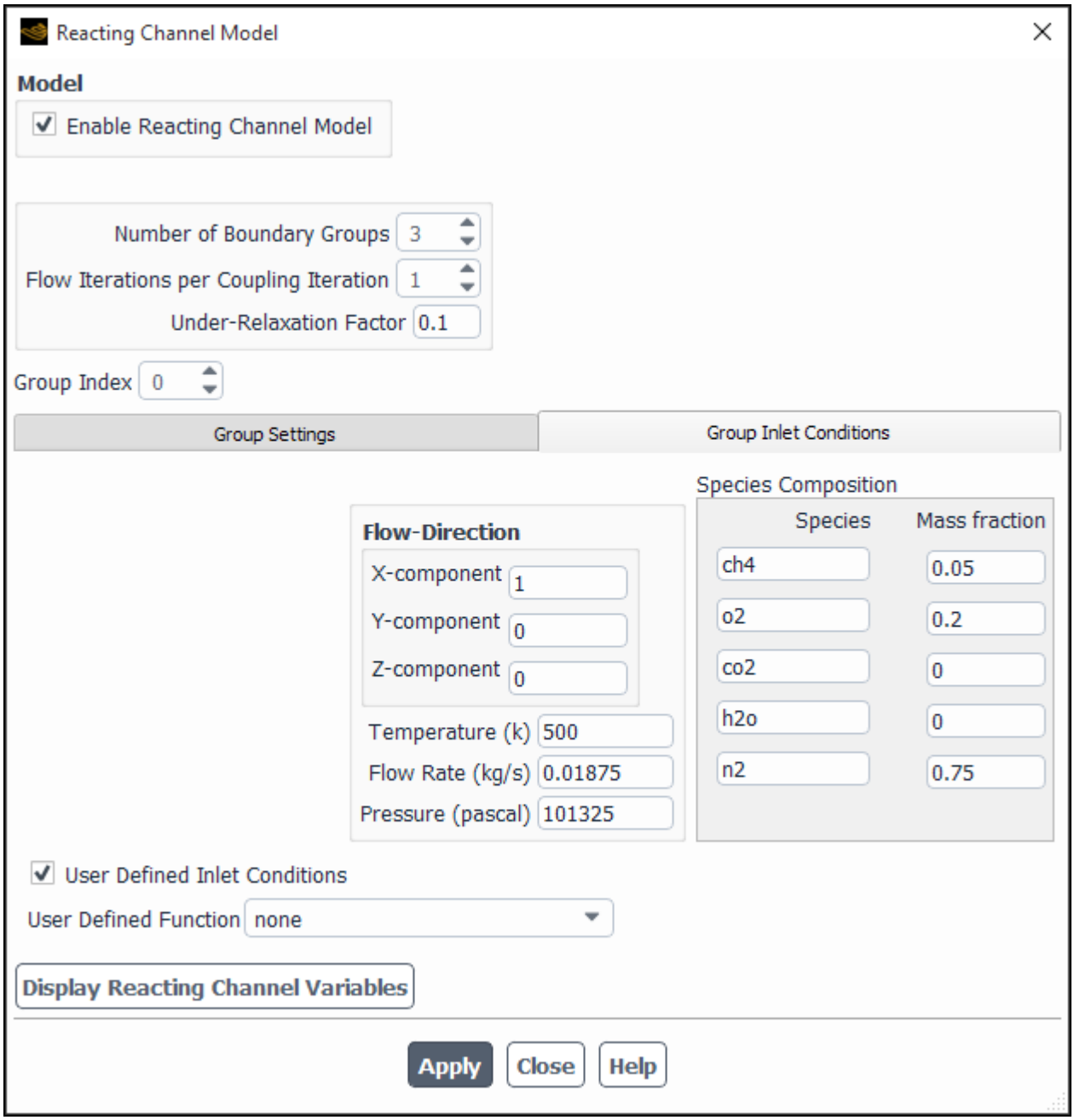
Pollutant Species

are typically slowly forming (far from chemical equilibrium), and occur at miniscule mass fractions.

51.4.38. Reacting Channel Model Dialog Box

The **Reacting Channel Model** dialog box allows you to solve reacting flow in shell and tube heat exchangers with long and thin channels.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Model

includes the option to **Enable Reacting Channel Model**.

Number of Boundary Groups

allows you to group together channels with common flow direction, mixture materials, inlet compositions, temperature, pressure, and mass flow rate in cases where you have multiple channels.

Flow Iterations per Coupling Iteration

is the number of outer flow iterations for each channel flow iteration.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Under-Relaxation Factor

is used to update the heat flux from the reacting channel. See Equation 7.145 in the Theory Guide.

Group Index

is used to identify the boundary group.

Group Settings

contains **Group Settings** that you will set.

Channel Walls in Group

are wall boundary zones that correspond to the reacting channels in the group.

Material

is the group material.

Import CHEMKIN Mechanism...

opens the Import CHEMKIN Format Mechanism Dialog Box (p. 5358).

Model Options

allows you to enable **Surface Reactions** and **Porous Medium** models and set related parameters.

Group Inlet Conditions

is where you specify the **Temperature**, the **Flow Rate**, the **Pressure**, and other inlet conditions of the reacting channel group.

Flow Direction

is the **X-**, **Y-**, and **Z-component** of the flow at the inlets of the channels in the current group.

Species Composition

is where you will specify the inlet mass fractions of the group.

User Defined Inlet Conditions

enables you to hook a user-defined function to specify the reacting channel inlet conditions. When this option is selected, the **User Defined Function** drop-down list becomes visible.

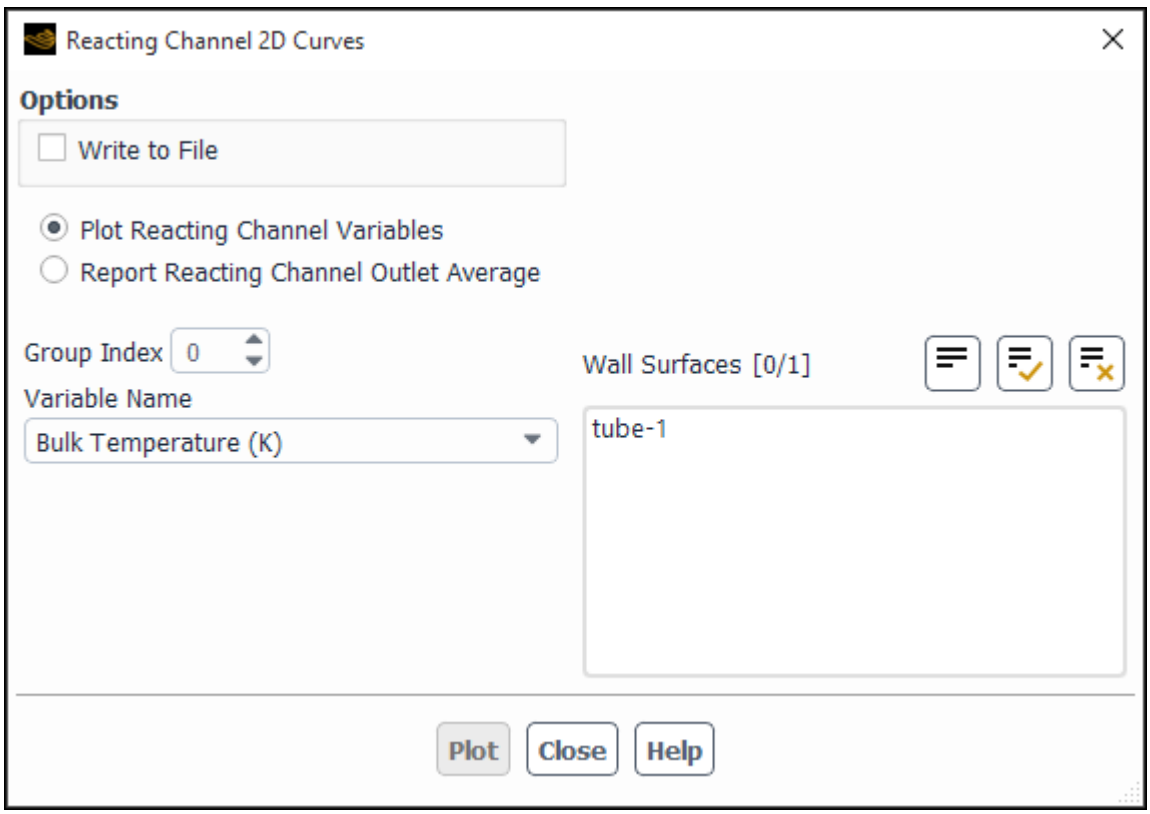
Display Reacting Channel Variables

opens the Reacting Channel 2D Curves Dialog Box (p. 4844) for postprocessing reacting channel variables.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

51.4.39. Reacting Channel 2D Curves Dialog Box

The **Reacting Channel 2D Curves** dialog box allows you to display or write 2D curves of the reacting channel.



Controls

Options

gives you the option to **Plot**, **Report**, or **Write to File** channel variables.

Group Index

is used to identify the reacting channel boundary group.

Plot Reacting Channel Variables

enables the plotting of reacting channel variables.

Report Reacting Channel Outlet Average

enables the reporting of reacting channel outlet average.

Variable Name

specifies the channel variable for plotting.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Variable Names

contains a selectable list of channel variables for reporting.

Wall Surfaces

contains a selectable list of wall surfaces on which you can plot, report or write the selected channel variables.

Plot

generates an X-Y plot. The **Plot** button is available when **Plot Reacting Channel Variables** is selected. When the **Write to File** option is enabled, the **Plot** button changes to the **Write...** button.

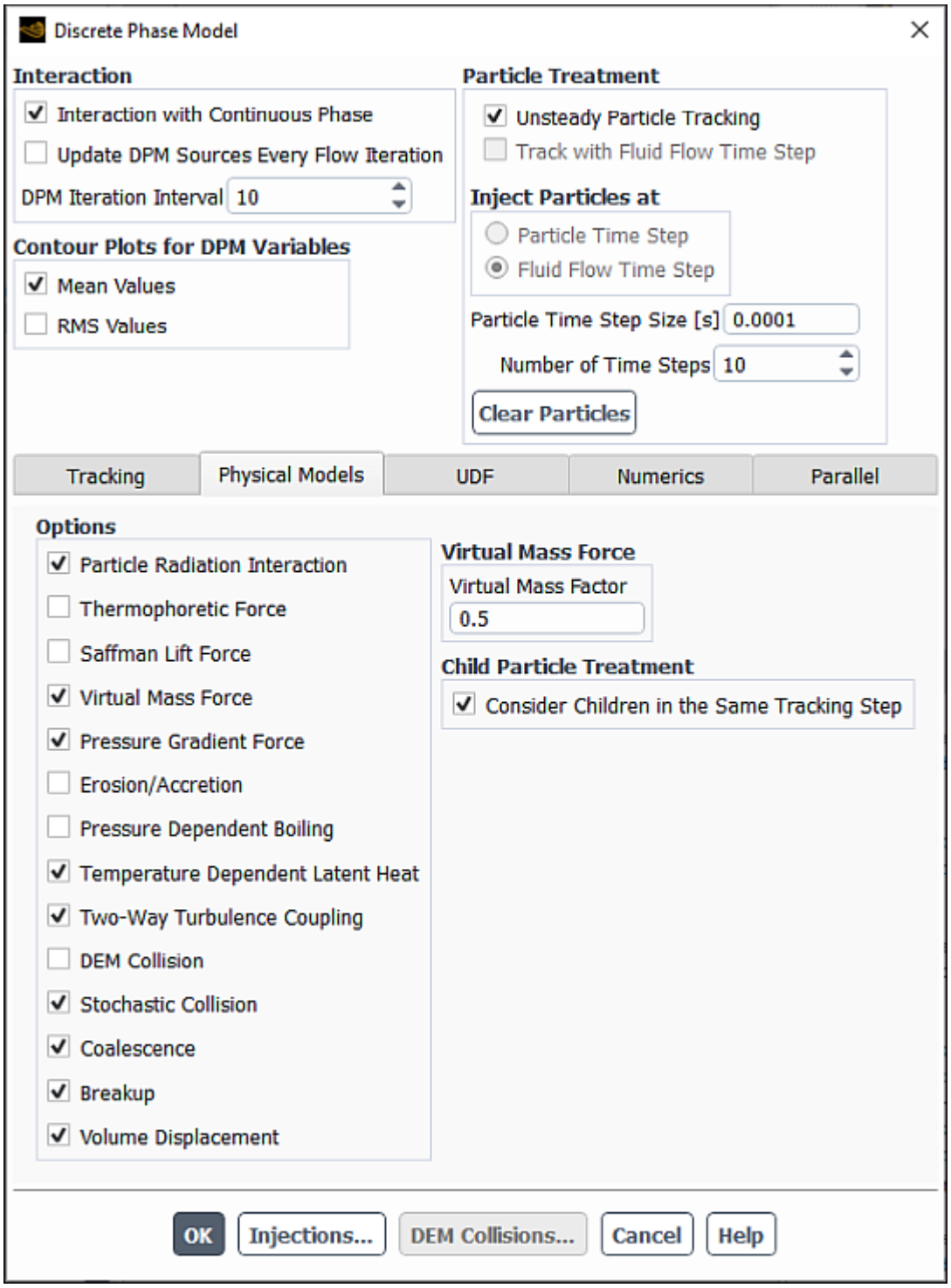
Report

generates the report of the channel variables. The **Report** button is available when **Report Reacting Channel Outlet Average** is selected. When the **Write to File** option is enabled, the **Report** button changes to the **Write...** button.

51.4.40. Discrete Phase Model Dialog Box

The **Discrete Phase Model** dialog box allows you to set parameters related to the calculation of a discrete phase of particles. See Modeling Discrete Phase (p. 2671) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Interaction

contains parameters used for performing coupled calculations of the continuous and discrete phase flow. See Procedures for a Coupled Two-Phase Flow (p. 2829) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Interaction with Continuous Phase

enables a coupled calculation of the discrete phase and the continuous phase.

Update DPM Sources Every Flow Iteration

when enabled, particle source terms will be updated according to Equation 24.14 (p. 2830) at every flow iteration. For unsteady simulations, it is the default and is recommended.

DPM Iteration Interval

allows you to control the frequency at which the particles are tracked and the DPM sources are updated.

Contour Plots for DPM Variables

contains options for enabling cell-averaged discrete phase variables for postprocessing.

Mean Values

enables additional cell-averaged discrete phase variables for postprocessing.

RMS Values

enables additional RMS values for several discrete phase variables for postprocessing.

Particle Treatment

contains options for choosing to treat the particles in an unsteady or a steady fashion.

Unsteady Particle Tracking

enables unsteady tracking of particles.

Track with Fluid Flow Time Step

enables the use of fluid flow time steps to inject the particles.

Inject Particles at

contains parameters to decide when to inject the particles for a new time step.

Particle Time Step

enables injection of particles for every particle time step.

Fluid Flow Time Step

enables injection of particles for every fluid flow time step.

In any case, the particles will always be tracked in such a way that they coincide with the flow time of the continuous flow solver, as long as the **Max. Number of Steps** (in the **Tracking** tab) used to compute a single trajectory is sufficient.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Particle Time Step Size

specifies particle time step size for the calculation.

Number of Time Steps

allows you to specify the number of time steps for the calculation.

Clear Particles

clears the particles that are currently in the domain.

Tracking

contains settings to control the time integration of the particle trajectory equations and particle tracking.

Tracking Parameters

contains parameters that control the tracking of particle trajectories. For steady-state particle tracking, one simple rule of thumb to follow when setting the two parameters below is that if you want the particles to advance through a domain of length D, the **Length Scale** times the number of **Max. Number of Steps** should be approximately equal to D. See Integration of Particle Equation of Motion in the Theory Guide for details about the items below.

Max. Number of Steps

is the maximum number of time steps used to compute a single particle trajectory via integration of Equation 12.1 in the Theory Guide and Equation 12.35. The default value is 50,000 for steady-state particle tracking and 500 for unsteady particle tracking.

Specify Length Scale

when enabled allows you to specify the length scale.

Length Scale

controls the integration time step size used to integrate the equations of motion for the particle. It appears only when the **Specify Length Scale** option is enabled.

Step Length Factor

specifies the value of λ in Equation 24.3 (p. 2685).

Tracking Option

High-Res Tracking

is a particle tracking algorithm that tracks particles through cells decomposed into tetrahedra (subtets). See Tracking Settings for the Discrete Phase Model (p. 2683) for more information about this method.

Physical Models

contains optional discrete phase models and their relevant parameters.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Options

contains additional models that can be included in the calculation. See Physical Models for the Discrete Phase Model (p. 2695) for more information.

Particle Radiation Interaction

includes the effect of radiation heat transfer to the particles (Equation 5.73 in the Theory Guide). You will also need to define additional properties for the particle materials (emissivity and scattering factor), as described in Description of the Properties (p. 2817).

This item appears only if the P-1 or discrete ordinates model is selected in the Radiation Model Dialog Box (p. 4736).

Thermophoretic Force

enables the inclusion of a thermophoretic force on the particles as an additional force term. See Thermophoretic Force in the Theory Guide for details.

Saffman Lift Force

enables the inclusion of Saffman's lift force (lift due to shear) as an additional force term. See Saffman's Lift Force in the Theory Guide for details.

Virtual Mass Force

enables the inclusion of virtual mass forces in the particle force balance. See Including the Virtual Mass Force and Pressure Gradient Effects on Particles (p. 2697) for details.

Pressure Gradient Force

enables the inclusion of pressure gradient effects in the particle force balance. See Including the Virtual Mass Force and Pressure Gradient Effects on Particles (p. 2697) for details.

Erosion/Accretion

enables the monitoring of erosion/accretion rates at wall boundaries. See Monitoring Erosion/Accretion of Particles at Walls (p. 2697) for details.

Pressure Dependent Boiling

allows you to switch from droplet vaporization (Law 2) to boiling (Law 3), as described in Considering Pressure Dependence in Boiling (p. 2698).

Temperature Dependent Latent Heat

allows you to include the droplet temperature effects on the latent heat, as described in Equation 12.92 in the Theory Guide.

Two-Way Turbulence Coupling

enables the effect of change in turbulent quantities due to particle damping and turbulence eddies. See Including the Effect of Particles on Turbulent Quantities (p. 2700) for more information.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DEM Collision

allows you to model DEM collision as described in Modeling Collision Using the DEM Model (p. 2701).

Stochastic Collision

enables the effect of droplet collisions, as described in Collision and Droplet Coalescence Model Theory in the Theory Guide.

Coalescence

enables the effect of droplet coalescence, as described in Collision and Droplet Coalescence Model Theory in the Theory Guide. This option is available when the **Stochastic Collision** option is enabled.

Breakup

enables breakup for all suitable injections. You can select the breakup model and parameters, or disable breakup, for each injection in the Set Injection Properties Dialog Box (p. 5608).

Volume Displacement

enables the volume displacement of particles, as described in Including the Volume Displacement of Particles (p. 2707).

Child Particle Treatment

contains an option to **Consider Children in the Same Tracking Step**. This control appears only when either **Breakup** is enabled under **Options** or **wall-film** is selected for **Discrete Phase BC Type** in the **DPM** tab of the **Wall** dialog box.

DEM Collision Model

contains model settings that allow you to continue your setup of the model.

Adaptive Collision Mesh Width

is enabled by default. This adjusts the width of the collision mesh to the largest parcel diameter multiplied by the **Edge Scale Factor**

Edge Scale Factor

is the factor by which the width of the collision mesh is adjusted to the largest parcel diameter.

Static Collision Mesh Width

is the fixed width of the collision mesh. This appears when the **Adaptive Collision Mesh Width** is disabled.

Maximum Particle Velocity

limits the maximum particle velocity to a physically plausible range.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DEM Collisions...

opens the DEM Collisions Dialog Box (p. 4855). This button is available at the bottom of the **Discrete Phase Model** dialog box when the **DEM Collision** model is enabled.

UDF

contains parameters that can be used to customize the discrete phase model using a user-defined function. See the separate Fluent Customization Manual for details about user-defined functions.

User-Defined Functions

lists will show available user-defined functions that can be selected to customize the discrete phase model.

Body Force

contains a drop-down list of user-defined functions available for including additional body forces.

Erosion/Accretion

contains a drop-down list of user-defined functions available for incorporating non-standard erosion rate definitions. This item will appear only when the **Erosion/Accretion** option has been enabled.

Scalar Update

contains a drop-down list of user-defined functions available for calculating or integrating scalar values along the particle trajectory.

Source

contains a drop-down list of user-defined functions available for modifying interphase exchange terms.

Spray Collide Function

contains a drop-down list of user-defined functions available for modifying spray collide function.

DPM Timestep

contains a drop-down list of user-defined functions available for modifying DPM time step.

Impingement Model

contains a drop-down list of user-defined functions available for modifying the impingement model. If selected, the **Impingement Model** UDF will overwrite the impingement/splashing model selected for the Lagrangian or Eulerian wall film boundary condition. This item is available only when either **wall-film** is selected for **Discrete Phase BC Type** in the **DPM** tab of the **Wall** dialog box or the **Eulerian Wall Film** model is enabled.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Film Regime

contains a drop-down list of user-defined functions available for setting the film regime of particle impingement. This item is available only when either **wall-film** is selected for **Discrete Phase BC Type** in the **DPM** tab of the **Wall** dialog box or the **Eulerian Wall Film** model is enabled.

Splashing Distribution

contains a drop-down list of user-defined functions available for modifying distribution of splashed particle variables. If selected, the **Splashing Distribution** UDF will overwrite the impingement/splashing model selected for the Lagrangian or Eulerian wall film boundary condition. This item is available only when either **wall-film** is selected for **Discrete Phase BC Type** in the **DPM** tab of the **Wall** dialog box or the **Eulerian Wall Film** model is enabled.

User Variables

lists the user input variables.

Number of Scalars

sets the number of scalar values used in the calculations with user-defined functions.

Numerics

tab gives you control over the numerical schemes for particle tracking as well as solutions of heat and mass equations. See Numerics of the Discrete Phase Model (p. 2710) for details.

Tracking Options

contains parameters of solutions of heat and mass equations.

Accuracy Control

enables the solution of equations of motion within a specified tolerance.

Tolerance

is the maximum relative error that has to be achieved by the tracking procedure.

Max. Refinements

is the maximum number of step size refinements in one single integration step. If this number is exceeded the integration will be conducted with the last refined integration step size.

Track in Absolute Frame

enables tracking the particles in the absolute reference frame.

Tracking Scheme Selection

contains parameters for selection of numerical schemes.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Automated

enables a mechanism to switch in an automated fashion between numerically stable lower order schemes and higher order schemes, which are stable only in a limited range.

High Order Scheme

can be chosen from the group consisting of **trapezoidal** and **runge-kutta** scheme.

Low Order Scheme

consists of **implicit** and the exponential **analytic** integration scheme.

Tracking Scheme

allows you to choose any of the tracking schemes. You also can combine each of the tracking schemes with Accuracy Control. It is selectable only if **Automated** is switched off.

Coupled Heat-Mass Solution

is a group box where you can enable the solution of the corresponding equations using a coupled ODE solver with error tolerance control for the **Droplet**, **Combusting**, or **Multicomponent** particles. See Including Coupled Heat-Mass Solution Effects on the Particles (p. 2713) for details.

Vaporizing Limiting Factors

for the **Mass** and **Heat** are set at the default values of 0.3 and 0.1, respectively. See Including Coupled Heat-Mass Solution Effects on the Particles (p. 2713) for details.

Averaging

contains settings for node based averaging of DPM quantities. See Node Based Averaging of Particle Data (p. 2714) for details.

Enable Node Based Averaging

enables node based averaging and displays additional related options. For non-DDPM cases, this option is not available when the **Linearize Source Terms** is selected.

Average DPM Source Terms

enables averaging of source terms for the discrete phase.

Average in Each Integration Time Step

enables averages during each integration time step.

Average DDPM Variables

enables averaging of DDPM variables when the DDPM model is active.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Kernel Settings

contains settings for the kernel used by the node based averaging algorithm.

Averaging Kernel

selects the kernel to use for averaging

Gaussian Factor

sets the constant for the Gaussian distribution when **gaussian** is selected for **Averaging Kernel**.

Source Terms

Linearize Source Terms

enables the linearization of source terms for the discrete phase. For non-DDPM cases, this option is not available when the **Enable Node Based Averaging** is selected.

Enhanced Numerics

when enabled, reduces excessive accumulation of particles in cells. This item is available only when the DDPM model is used.

Parallel

contains parameters that control the compute nodes for performing discrete phase calculations in parallel. See Parallel Processing for the Discrete Phase Model (p. 2888) for details.

Methods

allows you to select the mode for parallel processing.

Message Passing

enables cluster computing and also works on shared-memory machines. With this option, the compute node processes themselves perform the particle work on their local partitions and particle migration to other compute nodes is implemented using message passing primitives. No special requirements are placed on the host machine.

Shared Memory

allows you to specify parameters for performing the calculations on shared-memory multiprocessor machines.

Hybrid

allows you to combine **Message Passing** (as previously described) and **OpenMP** for a dynamic load balancing without migration of cells. This option enables multicore cluster computing, and also works on shared-memory machines. When using the **Hybrid** option, you can control the maximum number of threads on each machine via the **Thread Control** dialog box (see Controlling the Threads (p. 4345) for details).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Shared Memory Options

contains settings for the **Shared Memory** option. This group box is only available on Linux, when the **Shared Memory** is selected from the **Methods** list.

Workpile Algorithm

specifies that the discrete phase calculations are to be performed on a shared-memory multiprocessor machine.

Number of Threads

specifies the number of threads to be used in performing the particle calculations. By default, this parameter is equal to the number of compute nodes you specified for the parallel solver. (This item appears only if **Workpile Algorithm** is enabled.)

Hybrid Options

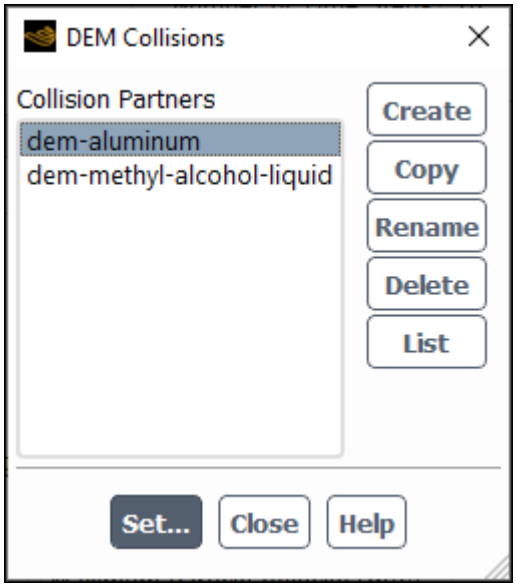
contains settings for the **Hybrid** option. This group box is only available if **Hybrid** is selected from the **Methods** list.

Use DPM Domain

enables the use of a separate computational domain for particle tracking, which improves load balancing and scalability at the expense of some additional memory overhead.

51.4.41. DEM Collisions Dialog Box

The **DEM Collisions** dialog box allows you to manage the collision partners.



Controls

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Collision Partners

contains a list from which you can select one or more collision partners in order to set, create, copy, rename, delete, or list.

Create

creates a new collision partner and opens the Create Collision Partner Dialog Box (p. 4856), in which you can enter the collision partner name.

Copy

creates a new injection with the same properties as the selected collision partner and opens the Copy Collision Partner Dialog Box (p. 4856).

Rename

allows you to change the name of the collision partner. It opens the Copy Collision Partner Dialog Box (p. 4856).

Delete

deletes the collision partner selected in the **Collision Partners** list.

List

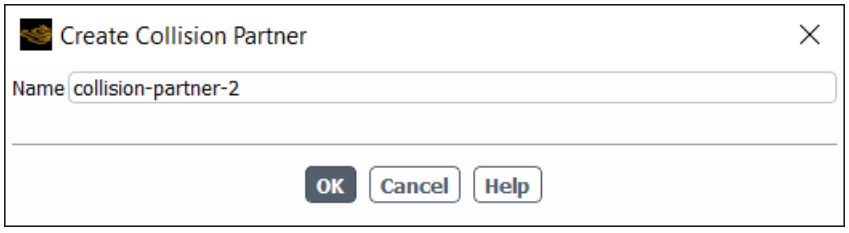
lists the collision partners with their corresponding laws.

Set...

opens the DEM Collision Settings Dialog Box (p. 4857) for the collision partner selected in the **Collision Partners** list.

51.4.42. Create Collision Partner Dialog Box

The **Create Collision Partner** dialog box allows you to specify the name of the collision partner.



Controls

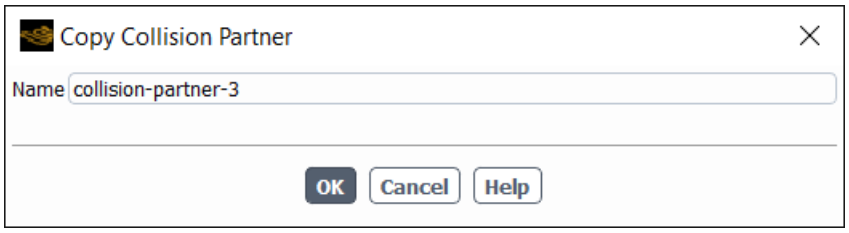
Name

is the name of the collision partner being created.

51.4.43. Copy Collision Partner Dialog Box

The **Copy Collision Partner** dialog box allows you to copy the selected collision partner.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



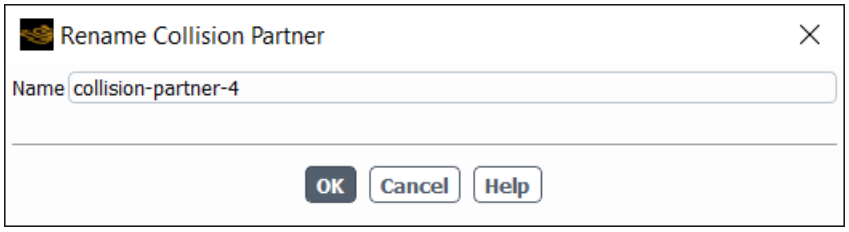
Controls

Name

is the name of the new collision partner copied from the selected partner.

51.4.44. Rename Collision Partner Dialog Box

The **Rename Collision Partner** dialog box allows you to rename the selected collision partner.



Controls

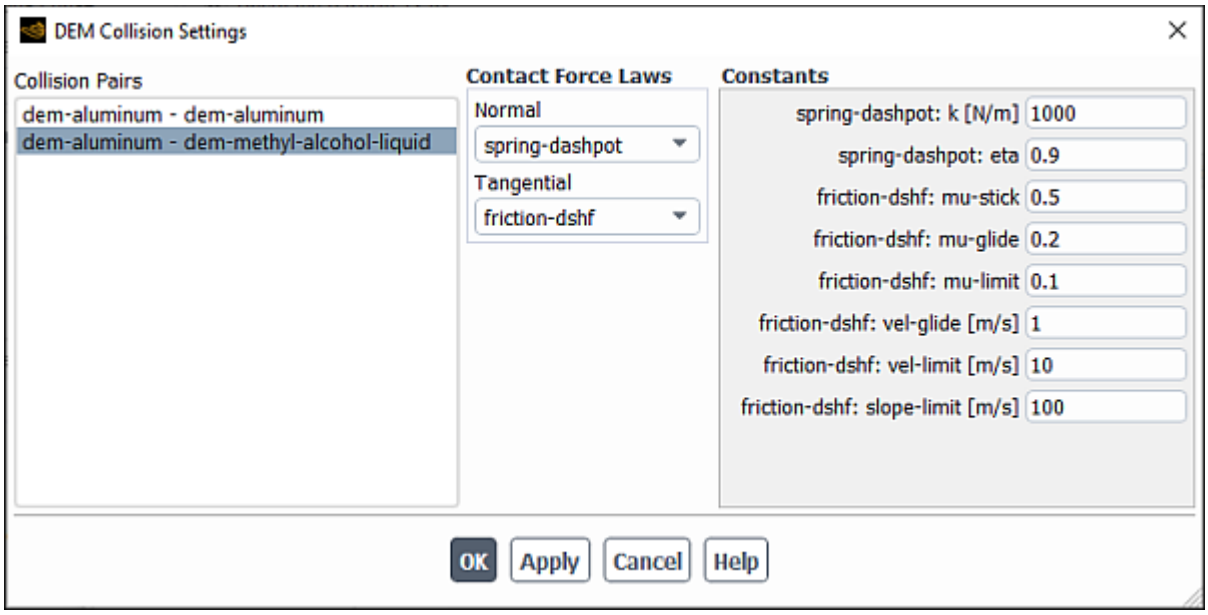
Name

is the new name of the collision partner selected in the list.

51.4.45. DEM Collision Settings Dialog Box

The **DEM Collision Settings** dialog box allows you to specify the collision laws of the collision partners.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Collision Pair

contains the list of created collision partners.

Contact Force Laws

is where you will define the collisions laws.

Normal

allows you to choose between **spring**, **spring-dashpot**, **hertzian**, and **hertzian-dashpot**. You can also choose **none** if you do not want to include a contact force.

Tangential

allows you to choose between the **friction-dhsf** and **friction-dshf-rolling** force laws. You can also exclude a force law, in which case you would select **none**.

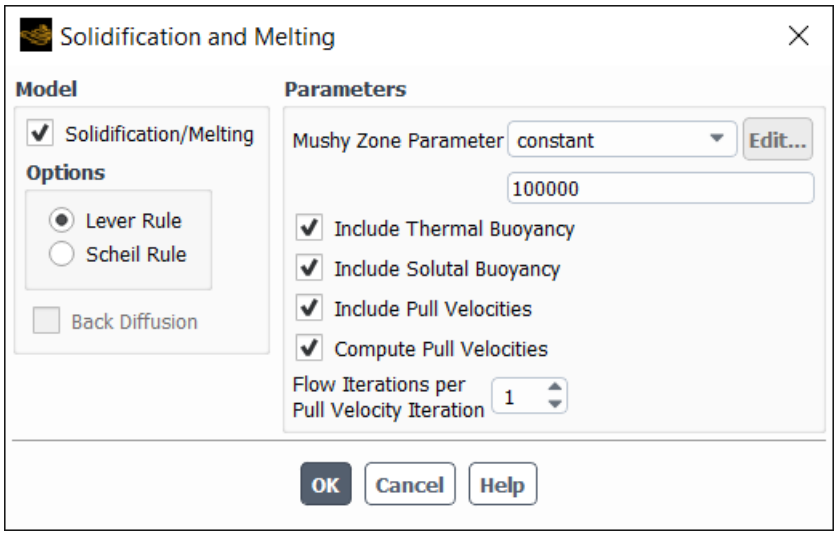
Constants

contains the list of contact force constants. Depending on the contact force law in effect, different constants will be visible.

51.4.46. Solidification and Melting Dialog Box

The **Solidification and Melting** dialog box allows you to set parameters related to the solidification/melting model. See Modeling Solidification and Melting (p. 3219) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Model

contains the option for turning on the model.

Solidification/Melting

enables/disables the modeling of solidification and/or melting.

Options

gives you the option of selecting one of two model rules.

Lever Rule

allows you to use the Lever rule (Equation 15.14 in the Theory Guide).

Scheil Rule

allows you to use the Scheil rule (Equation 15.18 in the Theory Guide). If you select **Scheil Rule**, then you can enable **Back Diffusion**. Enter either a **constant** or a user-defined function to specify the value of the **Back Diffusion Parameter**, used in Equation 15.19 to Equation 15.21 in the Theory Guide. Refer to `DEFINE_SOLIDIFICATION_PARAMS` of the Fluent Customization Manual for detailed information about the user-defined function.

Parameters

contains parameters related to the solidification/melting model.

Mushy Zone Constant

sets the value of A_{mush} in Equation 15.6 in the Theory Guide.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Include Thermal Buoyancy

includes the gravitational force due to the variation of density with temperature. Refer to Thermal and Solutal Buoyancy in the Theory Guide for more information.

Note:

This option is available only when solidification is modeled with species transport.

Include Solutal Buoyancy

includes the gravitational force due to the variation of density with the change in the species composition of the melt. Refer to Thermal and Solutal Buoyancy in the Theory Guide for more information.

Note:

This option is available only when solidification is modeled with species transport.

Include Pull Velocities

includes pull velocities in the model, as described in Momentum Equations and Pull Velocity for Continuous Casting in the Theory Guide.

Compute Pull Velocities

enables the calculation of pull velocities based on the specified velocity boundary conditions, as described in Pull Velocity for Continuous Casting in the Theory Guide. This option appears when **Include Pull Velocities** is turned on.

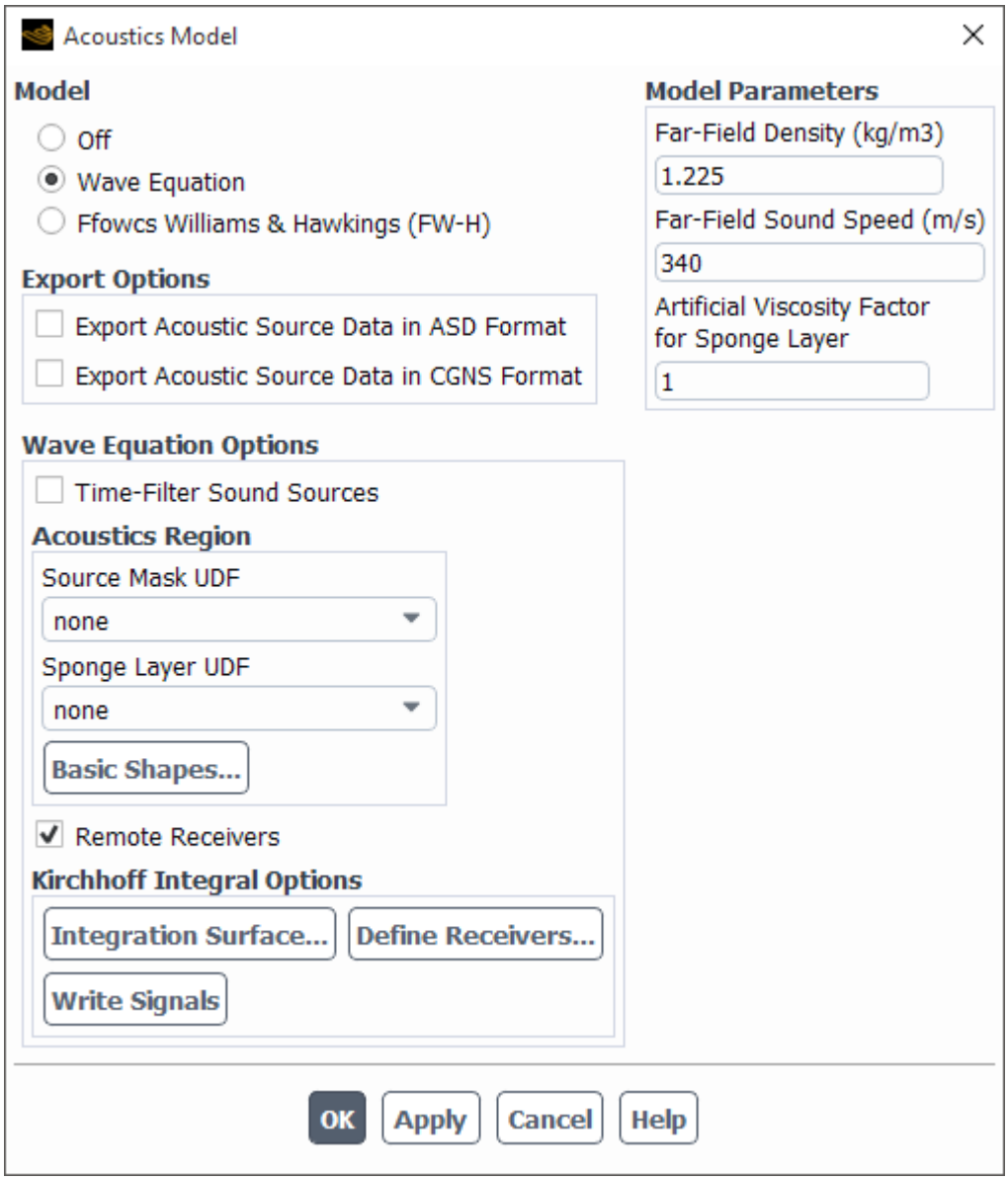
Flow Iterations per Pull Velocity Iteration

specifies the number of times the pull velocity equations will be solved after each iteration of the solver. This option appears when **Compute Pull Velocities** is turned on.

51.4.47. Acoustics Model Dialog Box

The **Acoustics Models** dialog box allows you to set parameters related to the acoustics model. See Predicting Aerodynamically Generated Noise (p. 2625) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Model

contains the option for turning on the acoustics model.

Off

disables the acoustics model.

Wave Equation

enables the wave equation model. See Using the Acoustics Wave Equation Model (p. 2653) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ffowcs-Williams & Hawkings

enables the Ffowcs Williams and Hawkings (FW-H) acoustics model. See Postprocessing the FW-H Acoustics Model Data (p. 2641) for details.

Broadband Noise Sources

enables the broadband noise acoustics model. See Using the Broadband Noise Source Models (p. 2664) for details.

Export Options

contains the parameters related to exporting acoustics data.

Export Acoustic Source Data in ASD Format

enables/disables the saving of source data files in ASD format.

Export Acoustic Source Data in CGNS Format

enables/disables the saving of source data files in CGNS format.

Compute Acoustics Signals Simultaneously

enables "on-the-fly" calculation of sound. When this option is chosen, the Ansys Fluent console window will print a message at the end of each time step indicating that the sound pressure signals have been computed (for example, `Extracting sound signals at x receiver locations...`, where `x` is the number of receivers you specified). Enabling this option instructs Ansys Fluent to compute sound pressure signals at the end of each time step, which will slightly increase the computation time.

Wave Equation Options

contains options for the wave equation.

Time-Filter Sounds Sources

enables/disables the filtering of sounds sources.

Source Mask UDF

Specifies a UDF for the source mask.

Sponge Layer UDF

Specifies a UDF for the sponge layer.

Basic Shapes...

opens the Basic Shapes Dialog Box (p. 4867), which allows you to specify shapes for source mask and sponge layer geometry.

Remote Receivers

enables/disables the Kirchhoff integral model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Integration Surface...

selects the integration surface for the Kirchhoff model.

Define Receivers...

opens the Acoustic Receivers Dialog Box (p. 4866), which allows you to specify the location of the receivers.

Write Signals

writes the computed receiver signals to the ASCII files.

Ffowcs-Williams & Hawkings Options

contains the **Convective Effects** option, which should be enabled for all cases dealing with external flows around bodies. When this option is enabled, you will need to specify the proper **Free Stream Velocity** and **Free Stream Direction**.

Define Sources...

opens the Acoustic Sources Dialog Box (p. 4864), which allows you to specify the acoustics sources.

Define Receivers...

opens the Acoustic Receivers Dialog Box (p. 4866), which allows you to specify the location of the receivers.

Model Constants

contains parameters related to the acoustics model.

Far-Field Density

sets the value of the far-field fluid density (ρ_0 in Equation 11.1 in the Theory Guide).

Far-Field Sound Speed

sets the speed of the sound at the far-field (a_0 in Equation 11.5 and Equation 11.6 in the Theory Guide).

Artificial Viscosity Factor for Sponge Layer

Set the Artificial Viscosity Factor for Sponge Layer.

Free Stream Velocity

is available when the **Convective Effects** option for the **Ffowcs-Williams & Hawkings** model is enabled.

Free Stream Direction

is available when the **Convective Effects** option for the **Ffowcs-Williams & Hawkings** model is enabled.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reference Acoustic Pressure

is used to calculate the sound pressure level in dB.

Reference Acoustic Power

is used to non-dimensionalize the acoustic power, giving the sound pressure level in dB (see Fast Fourier Transform (FFT) Postprocessing (p. 4078)). This parameter is equal to 4×10^{-10} W for airborne sound and 10^{-12} W for underwater sound.

Reference Acoustic Intensity

is defined as $P_{ref}^2 / \left(\rho_0 a_0\right)$, although this quantity is not currently used by Ansys Fluent.

Source Correlation Length

is required when sound is to be computed using a 2D flow result. The FW-H integrals will be evaluated over this length in the depthwise direction using the identical source data.

Number of Realizations

is the number of times the noise source terms will be computed through the generation of stochastic turbulent velocity field.

Number of Fourier Modes

is the number of terms in the Fourier summation from which the turbulent velocity field and its derivatives are computed.

Number of Time Steps Per Revolution

is available only for steady-state cases that have a single moving reference frame. Here you will specify the number of equivalent time steps that it will take for the rotating zone to complete one revolution.

Number of Revolutions

is available only for steady-state cases that have a single moving reference frame. Here you will specify the number of revolutions that will be simulated in the model.

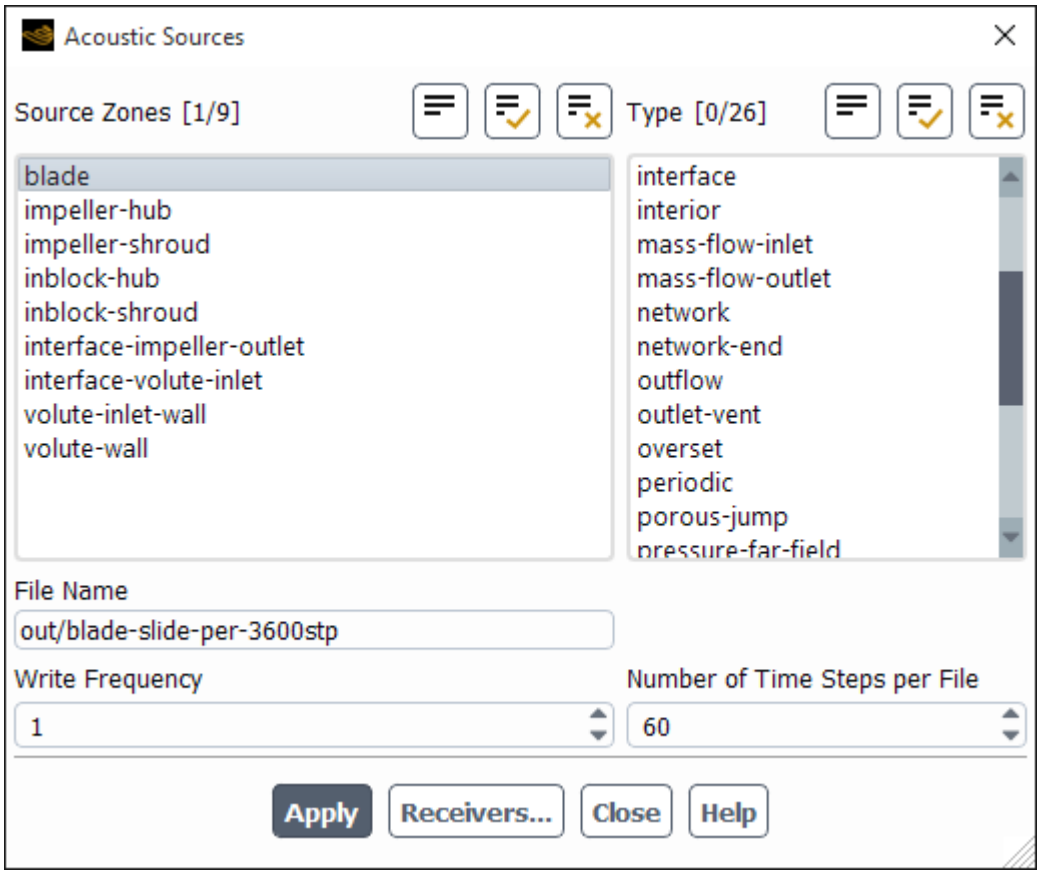
Apply

saves the acoustics model settings.

51.4.48. Acoustic Sources Dialog Box

The **Acoustic Sources** dialog box allows you to specify the acoustics sources. See Using the Ffowcs Williams and Hawkings Acoustics Model (p. 2628) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Source Zones

contains a list of zones from which you can select one or more emission (source) surfaces.

Type

contains a list of all possible types of zones from which you can select one or more types of the available zones. If you select an available type of zone, the relevant zones will then be selected in the **Source Zones** list. If you specify any valid **interior** zones as source surfaces, the Interior Cell Zone Selection Dialog Box (p. 4869) will appear.

File Name

is used to give the names of the source data files (for example, `acoustic_example_xxxx.asd`, where `xxxx` is the global time-step index of the transient solution) and an index file (for example, `acoustic_example.index`) that will store the information associated with the source data.

Write Frequency

allows you to control how often the source data will be written. This will enable you to save disk space if the time-step size used in the transient flow simulation is smaller than necessary to resolve the sound frequency you are attempting to predict.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Number of Time Steps per File

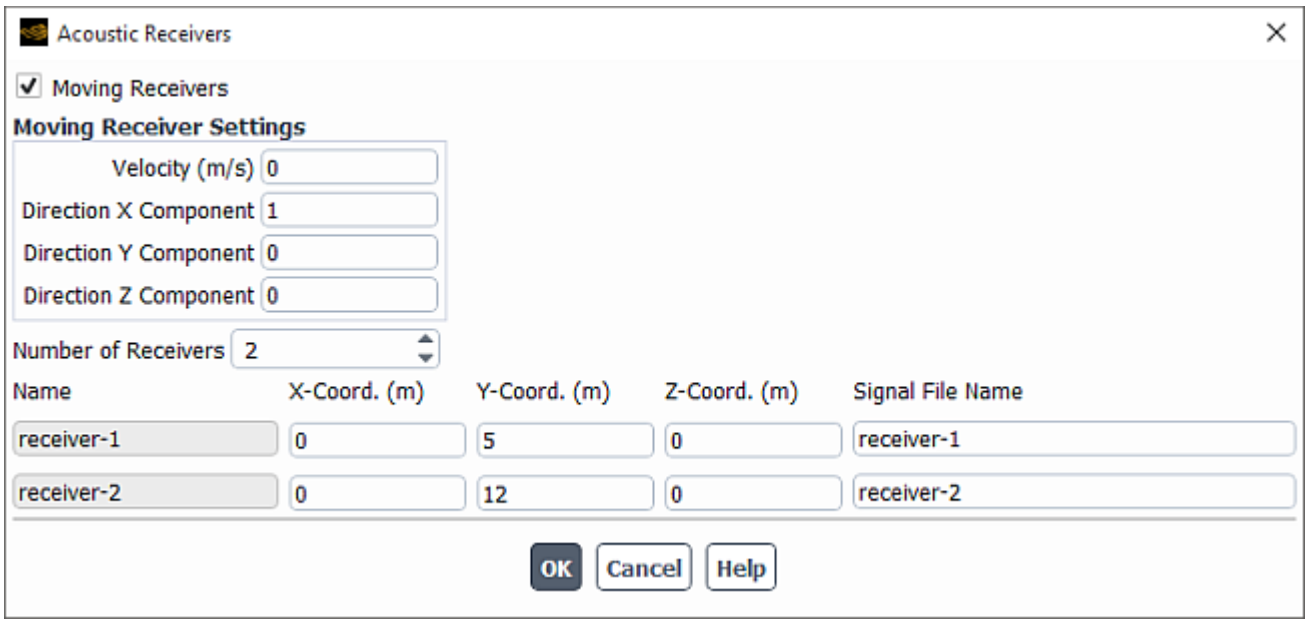
allows you to write the source data to multiple files, each containing the source data for a number of time steps specified by you.

Receivers...

opens the Acoustic Receivers Dialog Box (p. 4866), which allows you to specify the location of the receivers.

51.4.49. Acoustic Receivers Dialog Box

The **Acoustic Receivers** dialog box allows you to specify the location of the acoustic receivers. See Using the Ffowcs Williams and Hawkings Acoustics Model (p. 2628) for details about the items below.



Controls

Moving Receivers

allows you to specify the locations of the moving receivers.

Moving Receiver Settings

allows you to specify the **Velocity** and the **Direction** of the moving receivers.

Number of Receivers

allows you to specify the total number of receivers for which you want to compute sound.

Name

shows the name of the receiver. If you edit the field, the new name will take effect after you click the **OK** button.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

X-Coord., Y-Coord., Z-Coord.

specifies the coordinates for each receiver. Note that because Ansys Fluent's acoustics model is ideally suited for far-field noise prediction, the receiver locations you define should be at a reasonable distance from the sources of sound (that is, the selected acoustic surface), and can fall outside of the computational domain if needed.

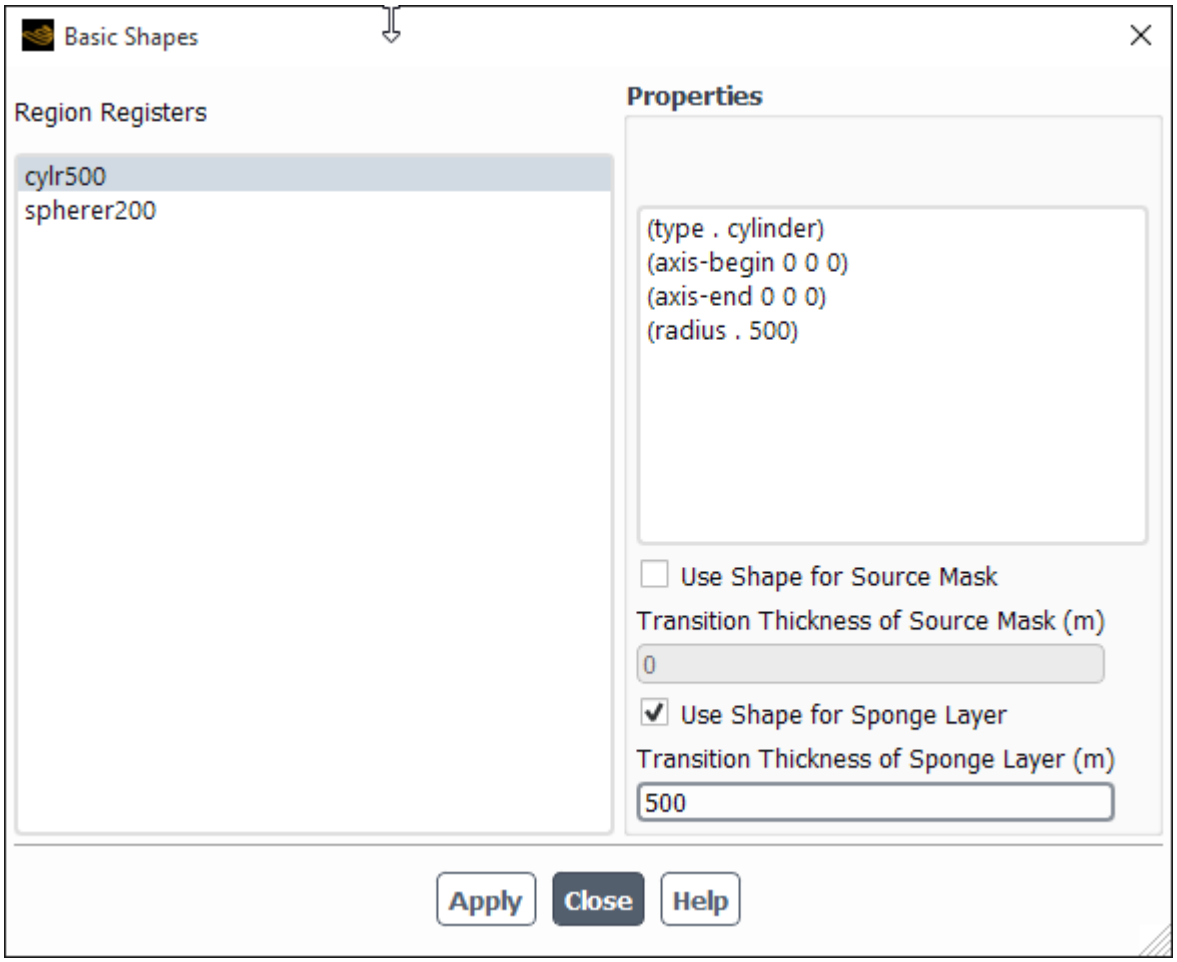
Signal File Name

specifies the name of the file used to store sound pressure signals for the corresponding receivers. By default, the files will be named `receiver-1.dat.h5`, `receiver-2.dat.h5`, and so on.

51.4.50. Basic Shapes Dialog Box

The Basic Shapes dialog box allows you to select shapes for use as source mask and sponge layer geometry.

Figure 51.1: The Basic Shapes Dialog Box



Controls

Region Registers

contains a list of all registers which have been created for use as geometric shapes.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Properties

displays information about the selected region register.

Use Shape for Source Mask

enables/disables the use of the selected region register as a source mask.

Transition Thickness of Source Mask

Specifies transition thickness of source mask.

Use Shape for Sponge Layer

enables/disables the use of the selected region register as a sponge layer.

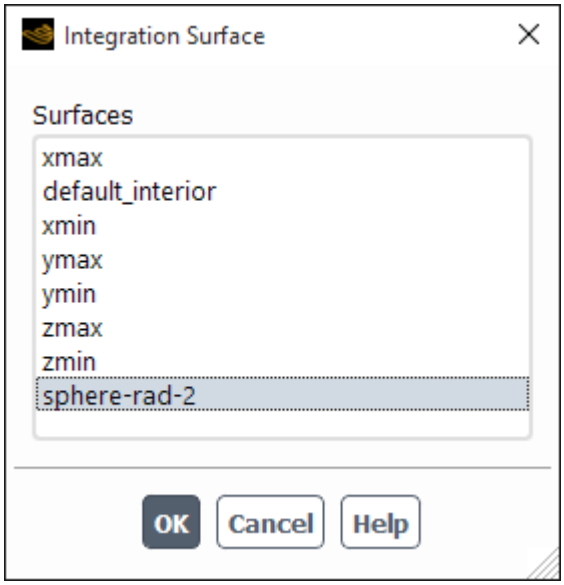
Transition Thickness of Sponge Layer

Specifies transition thickness of sponge layer.

51.4.51. Integration Surface Dialog Box

The **Integration Surface Dialog Box** displays a list of all currently available surfaces. With the current implementation only one surface can be selected in the list and used for the integration..

Figure 51.2: Integration Surface Dialog Box



Controls

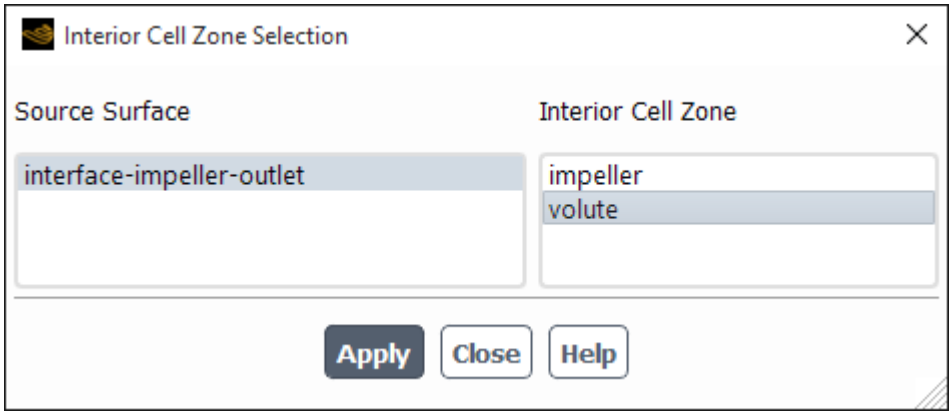
Surfaces

contains a list of all surfaces that can be used as an integration surface.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

51.4.52. Interior Cell Zone Selection Dialog Box

The **Interior Cell Zone Selection** dialog box allows you to specify the acoustics sources. See Specifying Source Surfaces (p. 2635) for details about the items below.



Controls

Source Surface

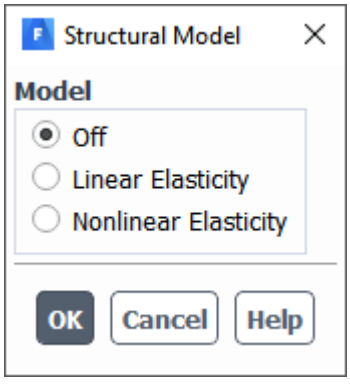
displays the selected interior surface.

Interior Cell Zone

contains a list of two interior cell zones, from which you will select the zone that is occupied by the quadrupole sources.

51.4.53. Structural Model Dialog Box

The **Structural Model** dialog box allows you to enable structural model calculations as part of an intrinsic fluid-structure interaction (FSI) problem. For details, see Modeling Fluid-Structure Interaction (FSI) Within Fluent (p. 3227).



Controls

Model

indicates which model, if any, is used for the structural model calculations.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Off

disables structural model calculations.

Linear Elasticity

enables structural calculations for the solid cell zone such that the internal load is linearly proportional to the nodal displacement, and the structural stiffness matrix remains constant. This model is appropriate when the stress loading does not exceed the yield strength of the solid material.

Nonlinear Elasticity

enables you to model geometrically nonlinear elasticity, in order to simulate large deformation in solid cell zones.

Options

allows you access to fluid-structural interaction options.

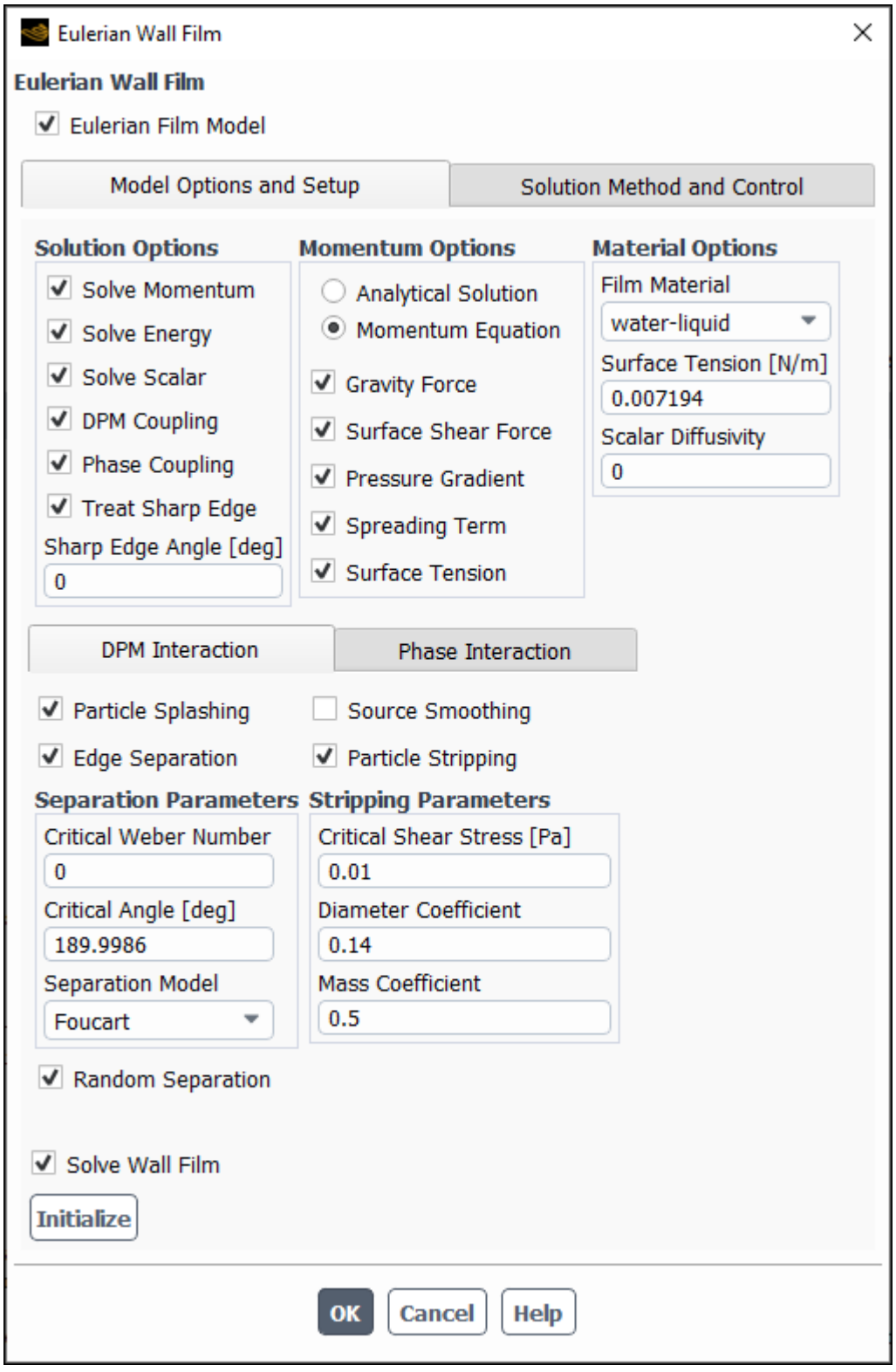
Thermal Effects

enables you to simulate the effects of thermal loads on solid structure deformation.

51.4.54. Eulerian Wall Film Dialog Box

The **Eulerian Wall Film** dialog box allows you to set various model and solution method controls for the Eulerian Wall Film model. For more information, see Modeling Eulerian Wall Films (p. 3239).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Eulerian Wall Film

enables/disables the use of the Eulerian Wall Film model in the calculations.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Model Options and Setup

contains controls for specific solution, discrete phase model (DPM), and material options for the Eulerian Wall Film model.

Solution Options

includes options for enabling/disabling the momentum equation and DPM calculations.

Solve Momentum

specifies whether film velocity is solved for the wall film. When the option is disabled, film velocity is fixed at the existing values. When the option is enabled, you can choose to solve film velocity using options in the **Momentum Options** group box.

Solve Energy

allows the energy equation (see Equation 17.3 in the *Fluent Theory Guide*) to be solved.

Solve Scalar

allows the passive scalar equation (see Equation 17.38 in the *Fluent Theory Guide*) to be solved. Once you enable this option, you can set **Scalar Diffusivity** in the **Material Options** group box.

DPM Coupling

enables the interaction of discrete phase particles with the film model (see DPM Collection in the *Fluent Theory Guide*). When this option is enabled, you can adjust the DPM coupling settings in the **DPM Interaction** tab. This option is only available when the DPM model is enabled.

Phase Coupling

allows you to account for phase changes between the film material (liquid) and the gas species (vapor) (see Coupling of Wall Film with Mixture Species Transport in the *Fluent Theory Guide*) and compute the secondary phase collection efficiency on a wall surface. When this option is enabled, you can adjust the phase coupling settings in the **Phase Interaction** tab. This option is only available when the **Eulerian** or **Mixture** (with **Slip Velocity**) multiphase model or the species model is enabled.

Treat Sharp Edge

allows you to set the **Sharp Edge Angle** when handling the effects of sharp edges alongside the wall film.

Momentum Options

is a group box where you can select from the following solution methods. This item is available when the **Solve Momentum** option is enabled.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Analytical Solution

applies film velocity computed from the external flow shear and gravity as described in Obtaining Film Velocity Without Solving the Momentum Equations in the *Fluent Theory Guide*.

Momentum Equation

enables the solution of the film momentum equation (see Equation 17.2). You can select the following terms for inclusion in the calculation:

Gravity Force

is the second term on the right hand side of Equation 17.2, and is responsible for accelerating the film in the direction of gravity component that is parallel to the wall.

Surface Shear Force

is the third term on the right hand side of Equation 17.2, and is responsible for accelerating the film in the direction of the external flow.

Pressure Gradient

is the first term on the right hand side of Equation 17.2, and is the term accelerating the film in the direction opposing the gradient in external pressure. For example, if a film on the surface of a wing is being modeled and there is a high pressure region at the leading edge with low pressure on the top of the wing, this term will tend to move a uniform film towards the low pressure region on the top of the wing.

Spreading Term

is the P_h term on the right hand side of Equation 17.2, and is responsible for spreading. This term will accelerate the flow in the direction opposing the gradient in height, moving the film towards regions of lower thickness. This term becomes available only when both **Pressure Gradient** and **Gravity Term** are selected.

Surface Tension

is the P_σ term on the right hand side of Equation 17.2. This term becomes available only when **Pressure Gradient** is selected.

Material Options

allows you to apply material properties to the wall film model.

Film Material

is the material assigned to the wall film.

Surface Tension

is the surface tension for the designated wall film material. This item is available only when the **Solve Momentum** option is enabled in the **Solution Options** group box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Scalar Diffusivity

is the diffusivity of the scalar being modeled for the designated wall film material. This item is available only when the **Solve Momentum** option is enabled in the **Solution Options** group box.

DPM Interaction

tab defines the interaction between the discrete phase and the film (see DPM Collection in the *Fluent Theory Guide*). this tab is available when **DPM Coupling** is enabled in the **Solution Options** group box.

Particle Splashing

allows you to account for splashing of the wall film particles. When this option is enabled, you can specify splashing options in the **Wall Film** tab of the **Wall** dialog box. For details, see Setting Eulerian Wall Film Boundary, Initial, and Source Term Conditions (p. 3252).

Source Smoothing

When enabled, smoothing is applied to the discrete phase impingement source terms to reduce the impact of spotty source distributions.

Edge Separation

allows you to specify the following **Separation Parameters**:

Critical Weber Number

is the critical value for the Weber number.

Critical Angle

is the critical angle, θ, that separation occurs.

Separation Model

allows you to specify one of three different models to calculate the number and diameter of the shed particle stream at an edge once separation occurs. Available models include: **Foucart** (the default, see Foucart Separation in the Theory Guide (p. 1)), **O'Rourke** (see O'Rourke Separation), and **Friedrich** (see Friedrich Separation).

Random Separation

allows the random selection of the locations at which the newly spawn particles are injected (due to film separation) along the edge at which the separation takes place.

Particle Stripping

allows you to set the below DPM **Stripping Parameters**. For details about these settings, see Film Separation in the *Fluent Theory Guide*.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Critical Shear Stress

is the critical value of the shear stress on the face where liquid film exists, which, when exceeded, causes mass to be taken from the film.

Diameter Coefficient

is the value of F used in Equation 17.20 in the Theory Guide (p. 1).

Mass Coefficient

is the value of C used in Equation 17.21 in the Theory Guide (p. 1).

Phase Interaction

tab allows you to set secondary phase accretion options (see Secondary Phase Accretion in the Theory Guide (p. 1)). The tab is available when **Phase Coupling** is enabled in the **Solution Options** group box.

Phase Change

allows you to account for phase changes between the film material (liquid) and the gas species (vapor). When this option is selected, you can specify the phase change parameters on the selected film wall boundaries in the **Wall Film** tab of the Wall Dialog Box (p. 5085). (See Coupling of Wall Film with Mixture Species Transport in the *Fluent Theory Guide*). This option is only available when the species model is enabled.

Film Vapor Material

is the material assigned to the wall film vapor.

Phase Accretion

(available when the Eulerian Multiphase or Mixture (with Slip Velocity) model is enabled) allows for the interaction of the wall film with a secondary phase in a multiphase flow. Once enabled, you can set the **Phase Accretion Parameters** (see Secondary Phase Accretion in the Theory Guide (p. 1)).

Phase Concentration

is the value of C_d used in Equation 17.24 in the Theory Guide (p. 1).

Phase Velocity

is the value of V_d used in Equation 17.24 in the Theory Guide (p. 1).

Solve Wall Film

allows you to skip the wall film solution during the gas phase solution, but keep the variables and setup active.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Initialize

allows you to initialize the wall film variables and prepare the solver for the solution procedure. The film cannot be solved without first initializing the wall film model.

Solution Method and Control

contains controls for temporal or spatial discretization as well as wall film thickness for the Eulerian Wall Film model.

Discretization

allows you to set the temporal or spatial discretization methods.

Time

allows you to specify the temporal discretization methods. Available options include: **First Order Explicit**, **High Order Explicit**, **First Order Implicit**, or **Second Order Implicit**.

Continuity

allows you to specify the spatial discretization methods for continuity. Available options include: **First Order Upwind**, or **Second Order Upwind**.

Momentum

allows you to specify the spatial discretization methods for momentum. Available options include: **First Order Upwind**, or **Second Order Upwind**.

Energy

allows you to specify the spatial discretization methods for energy. Available options include: **First Order Upwind**, or **Second Order Upwind**.

Thickness Control

allows you to specify **Maximum Thickness**.

Continuity and Momentum Coupling

is a group box where you can specify the following coupling solution inputs. This item appears only when **Solve Momentum** is selected in the **Model Options and Setup** tab. For more information about these options, see Coupled Solution Approach in the *Fluent Theory Guide*.

Coupled Solution

enables the coupled solution approach.

Curvature Smoothing

applies smoothing to the film variables. When this option is enabled, you can specify **Smoothing Level** and **Smoothing Factor**. This item appears only when **Surface Tension** is selected in the **Model Options and Setup** tab.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Time Marching and Time Step Control

allows you to set the number of sub-steps that the film model will take between time steps from the main solver. For example, if the flow time step is 0.001 seconds and the number of time steps is set to 10 in the film model, the film model will take 10 time steps of 0.0001 so that the film time and the flow time match at the end of the solution step. For implicit time stepping, the film model will take 5 additional sub iterations per implicit step, stopping the sub-step when the film residual drops below the value set in the **Sub-Iteration Stop** option.

Adaptive Time Stepping

(steady-state flow) enables the adaptive time stepping method to determine the film time step size (Δt) using the maximum Courant number and the initial time step (see Steady Flow in the Theory Guide (p. 1)).

Max. Courant Number

is the maximum value of the Courant number used to compute the film time step.

Initial Time Step

is an initial value for the time step used to compute the film time step.

Time-Step

(steady-state flow; default) is the user-specified size of the time step, Δt (see Steady Flow in the Theory Guide (p. 1)).

Per Flow Iterations

(transient flow) specifies the flow iteration interval at which the flow solution is called within each flow time step. The film solution is always called at the first flow iteration.

Sub-Time Steps

(transient flow) is N_{film} used in Equation 17.52 in the Theory Guide (p. 1).

Sub-Time Report Interval

(transient flow) specifies the number of iterations between the wall film solution reports.

Sub-Iterations

(available when either **First Order Implicit** or **Second Order Implicit** are enabled for **Time** under **Discretization**) is the number of additional sub iterations per implicit step.

Sub-Iteration Stop

(available when either **First Order Implicit** or **Second Order Implicit** are enabled for **Time** under **Discretization**) is the point at which the sub iterations stop (when the film residual drops below this value).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sub-Iter Report Interval

(implicit time discretization calculation) specifies the number of sub-iterations between the wall film solution reports.

DPM Control

(available when **DPM Coupling** is enabled in the **Model Options and Setup** tab) allows you to specify DPM-specific discretization.

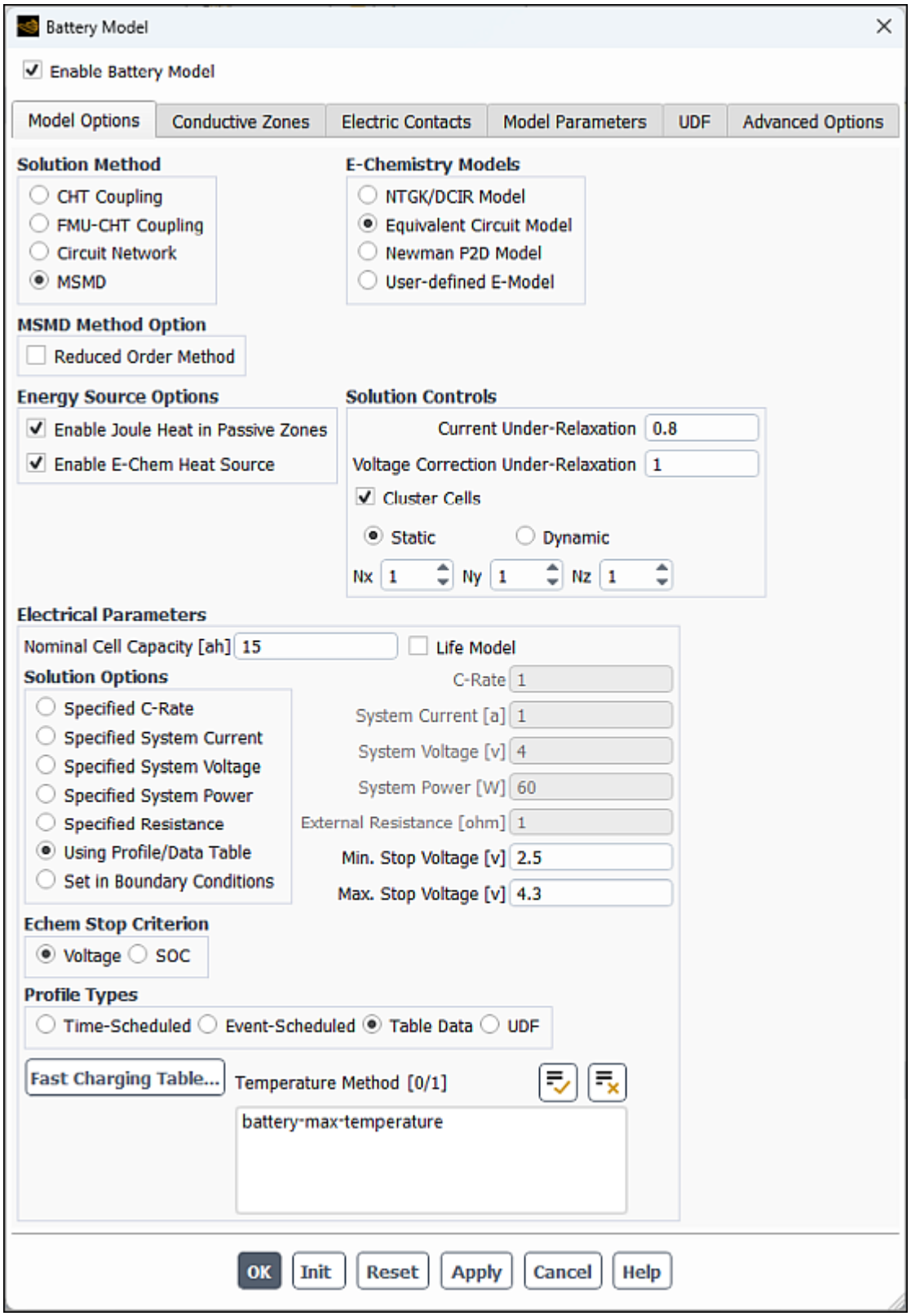
Film Steps per DPM Step

allows you to set how often the DPM phase is calculated for the film.

51.4.55. Battery Model Dialog Box

The **Battery Model** dialog box allows to set parameters related to thermal and electrochemical behavior of a single battery cell or a battery pack.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Enable Battery Model

enables/disables the battery modeling.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Model Options

tab allows you to select the solution method and E-chemistry model, and specify general battery model settings. See Specifying Battery Model Options (p. 3303) for detailed information about available models and settings.

Conductive Zones

tab allows you to select zones for the active, tab and busbar components. See Specifying Conductive Zones (p. 3317) for detailed information about available settings.

Electric Contacts

tab allows you to define the contact surface and external connectors. See Specifying Electric Contacts (p. 3318) for detailed information about available settings.

Model Parameters

tab allows you to specify the parameters to be used for solving the model equations. See Specifying Battery Model Parameters (p. 3321) for detailed information about available controls.

UDF

tab allows you to hook your user-defined functions. See Hooking User-Defined Functions (p. 3350) for detailed information about this tab.

Advanced Options

tab allows you to quickly check the selected electrochemical submodel behavior in a standalone mode, and also to use various advanced capabilities. See Specifying Advanced Options (p. 3351) for more information.

Init

initializes the battery model only. See Initializing the Battery Model (p. 3395) for more information.

Reset

restores the last saved settings for all tabs.

Apply

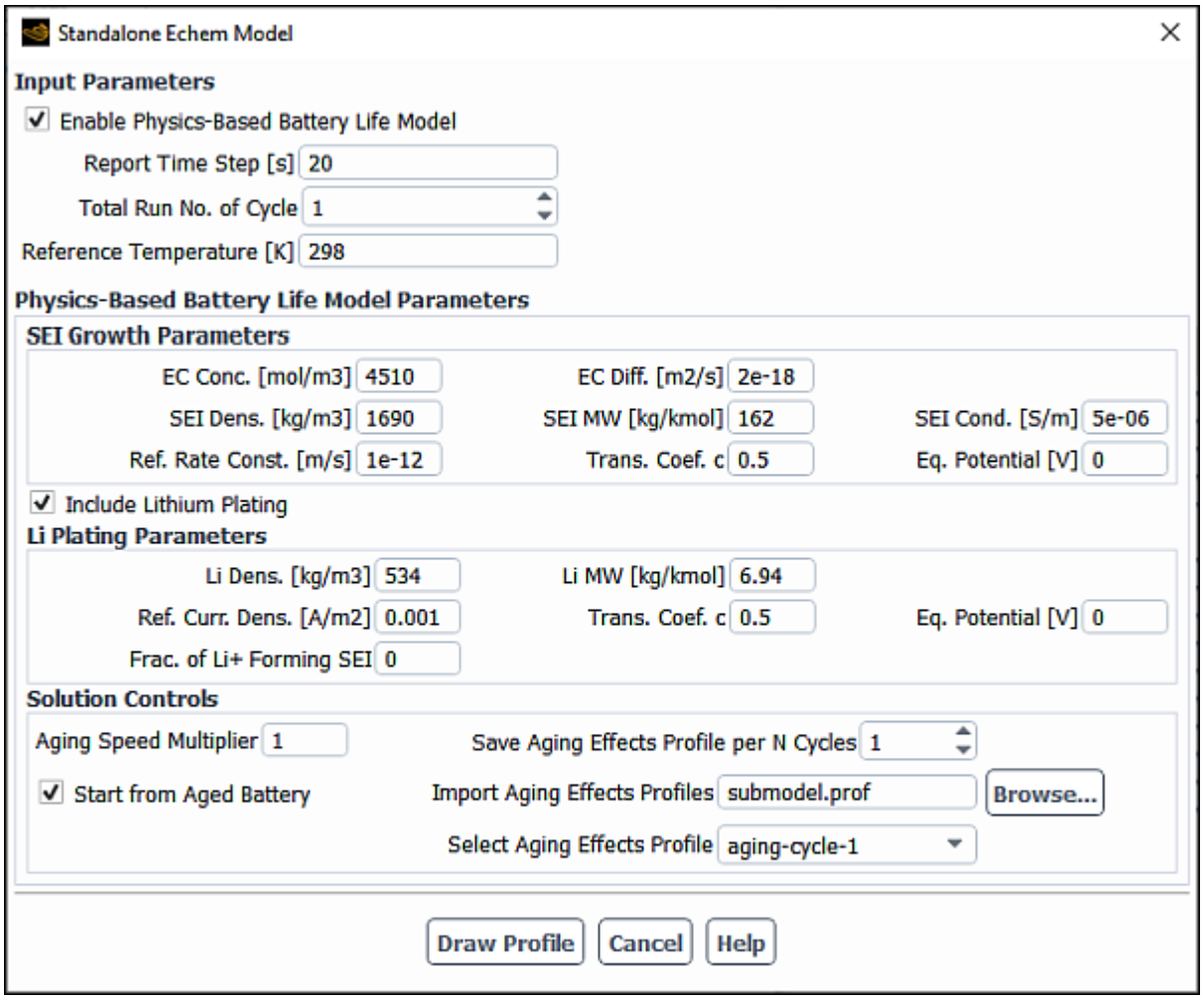
saves the specified values and options for the selected model.

51.4.56. Standalone Echem Model Dialog Box

The **Standalone Echem Model** dialog box enables you to:

- Perform a quick analysis of your electrochemical model by simulating it in a standalone mode
- Predict the battery aging process during a large number of electric load cycles in the standalone mode

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



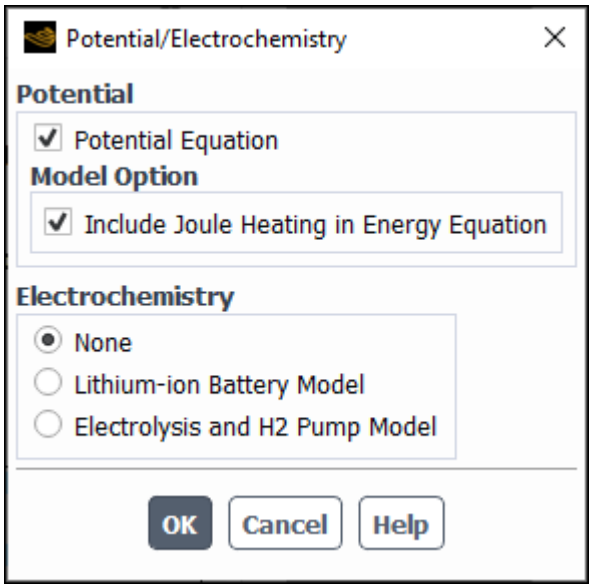
For a full description of available controls, see:

- Running the Standalone Echem Model (p. 3352)
- Simulating the Aging Process of a Battery (Standalone Mode) (p. 3356)

51.4.57. Potential/Electrochemistry Dialog Box

The **Potential/Electrochemistry** dialog box allows you to set parameters related to the electric potential field or electrochemistry models (Li-ion battery model or electrolysis and H2 pump model) in your simulation. See Setting Up the Electric Potential Model (p. 3266), Setting Up the Lithium-ion Battery Model (p. 3268), and Setting the Electrolysis and H2 Pump Model (p. 3275) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Potential

contains controls related to modeling of the electric potential field.

Potential Equation

enables/disables the simulation of the electric potential field in the model.

Model Option

allows you to enable/disable the **Include Joule Heating in Energy Equation** option. This option appears only if the energy equation is enabled.

Electrochemistry

contains the following electrochemistry options:

None

no electrochemistry is involved in the simulation

Lithium-ion Battery Model

allows you to enables the Lithium-ion Battery model and specify related parameters as described in Simulating the Lithium-ion Battery (p. 3267). This option is available only when the **Potential Equation** is enabled.

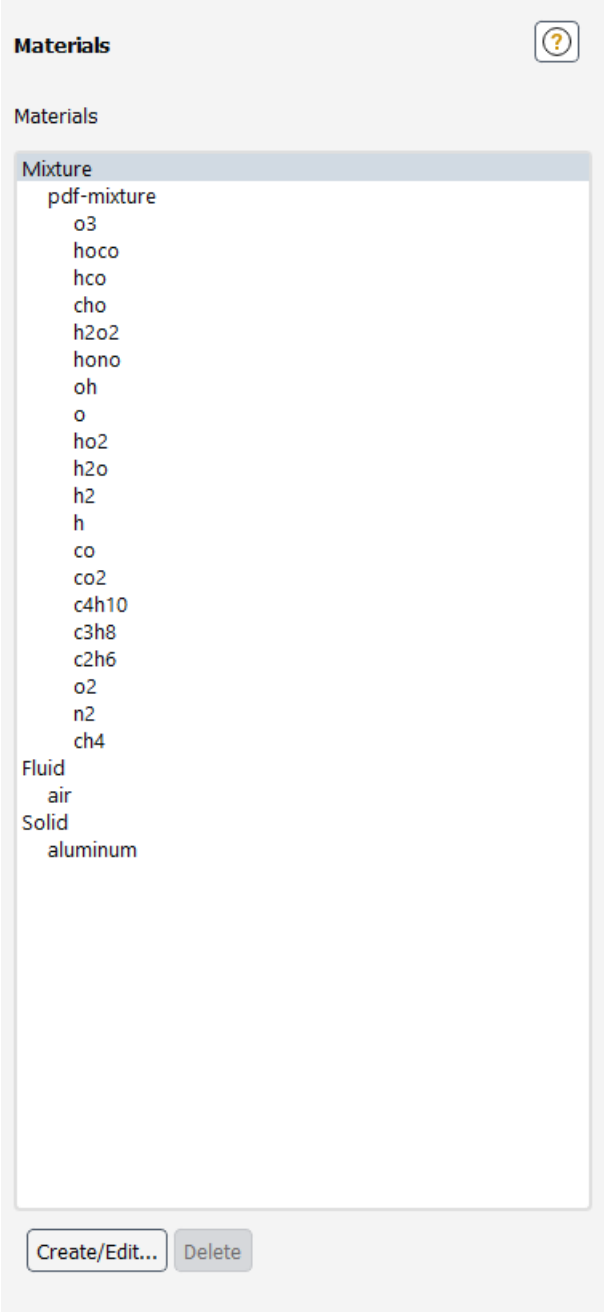
Electrolysis and H2 Pump Model

allows you to enables the electrolysis and H2 pump model and specify related parameters as described in Setting the Electrolysis and H2 Pump Model (p. 3275). This option is available only when the **Potential Equation** is enabled.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

51.5. Materials Task Page

The **Materials** task page allows you to set properties for any fluid or solid (or mixture, if applicable) materials in your Ansys Fluent simulation.



Controls

Materials

contains a listing of available fluid or solid (or mixture, if applicable) materials.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Create/Edit...

displays the Create/Edit Materials Dialog Box (p. 4885) for the selected item in the **Materials** list.

Delete

removes the selected material from the **Materials** list,

For additional information, see the following sections:

51.5.1. Create/Edit Materials Dialog Box
51.5.2. Fluent Database Materials Dialog Box
51.5.3. GRANTA MDS Materials Dialog Box
51.5.4. Open Database Dialog Box
51.5.5. User-Defined Database Materials Dialog Box
51.5.6. Copy Case Material Dialog Box
51.5.7. Material Properties Dialog Box
51.5.8. Edit Property Methods Dialog Box
51.5.9. New Material Name Dialog Box
51.5.10. Polynomial Profile Dialog Box
51.5.11. Piecewise-Linear Profile Dialog Box
51.5.12. Piecewise-Polynomial Profile Dialog Box
51.5.13. NASA-9-Coefficient Piecewise-Polynomial Profile Dialog Box
51.5.14. Model Options Dialog Box
51.5.15. Compressible Liquid Dialog Box
51.5.16. User-Defined Functions Dialog Box
51.5.17. Sutherland Law Dialog Box
51.5.18. Power Law Dialog Box
51.5.19. Non-Newtonian Power Law Dialog Box
51.5.20. Carreau Model Dialog Box
51.5.21. Cross Model Dialog Box
51.5.22. Herschel-Bulkley Dialog Box
51.5.23. Biaxial Conductivity Dialog Box
51.5.24. Cylindrical Orthotropic Conductivity Dialog Box
51.5.25. Orthotropic Conductivity Dialog Box
51.5.26. Anisotropic Conduction - Principal Components Dialog Box
51.5.27. Anisotropic Conductivity Dialog Box
51.5.28. Species Dialog Box
51.5.29. Reactions Dialog Box
51.5.30. Backward Reaction Parameters Dialog Box
51.5.31. Third-Body Efficiency Dialog Box
51.5.32. Pressure-Dependent Reaction Dialog Box

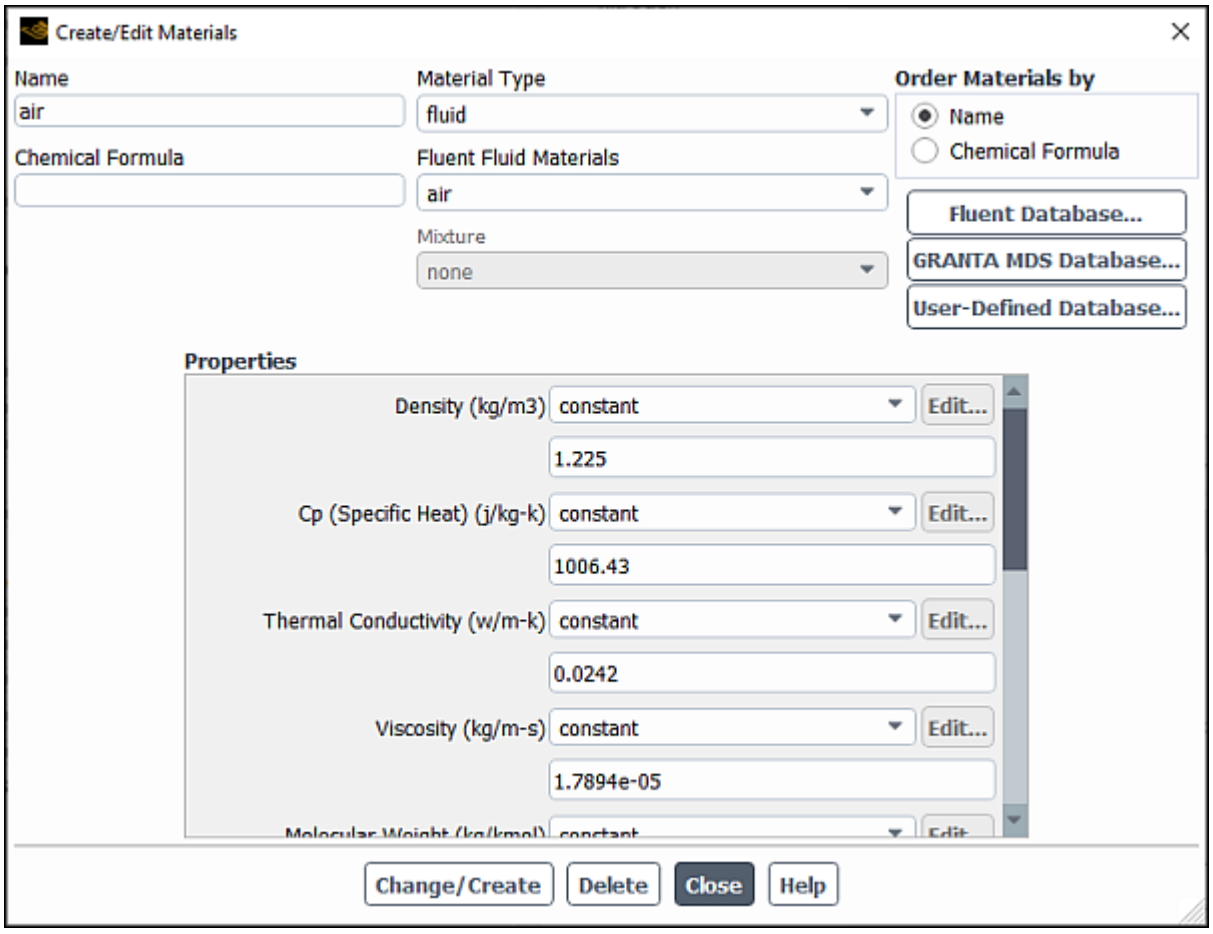
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

51.5.33. Coverage-Dependent Reaction Dialog Box
51.5.34. Reference Molar Concentrations Dialog Box
51.5.35. Reaction Mechanisms Dialog Box
51.5.36. Site Parameters Dialog Box
51.5.37. Mass Diffusion Coefficients Dialog Box
51.5.38. Thermal Diffusion Coefficients Dialog Box
51.5.39. UDS Diffusion Coefficients Dialog Box
51.5.40. WSGGM User Specified Dialog Box
51.5.41. Gray-Band Absorption Coefficient Dialog Box
51.5.42. Delta-Eddington Scattering Function Dialog Box
51.5.43. Gray-Band Refractive Index Dialog Box
51.5.44. Single Rate Model Dialog Box
51.5.45. Secondary Rate Model Dialog Box
51.5.46. Two Competing Rates Model Dialog Box
51.5.47. CPD Model Dialog Box
51.5.48. Kinetics/Diffusion-Limited Combustion Model Dialog Box
51.5.49. Intrinsic Combustion Model Dialog Box
51.5.50. Multiple Surface Reactions Dialog Box
51.5.51. Edit Material Dialog Box

51.5.1. Create/Edit Materials Dialog Box

The **Create/Edit Materials** dialog box is used to create and modify materials. Materials can be downloaded from the global database or defined locally. See Physical Properties (p. 1597) for details about defining material properties. Using the **Create/Edit Materials** Dialog Box (p. 1599) describes how to use the dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name

shows the name of the material. If you edit this field, the new name will take effect when you click **Change/Create**.

Chemical Formula

displays the chemical formula for the material. You should generally not edit this field unless you are creating a material from scratch.

Material Type

is a drop-down list containing all of the available material types. By default, **fluid** and **solid** will be the only choices. If you are modeling species transport/combustion, **mixture** will also be available. For problems in which you have defined discrete-phase injections, **inert-particle**, **droplet-particle**, and/or **combusting-particle** will also appear.

Fluent Fluid Materials

allows you to choose the fluid material for which you want to modify properties. This option is available when **fluid** is selected in the **Material Type** drop-down list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fluent Solid Materials

allows you to choose the solid material for which you want to modify properties. This option is available when **solid** is selected in the **Material Type** drop-down list.

Fluent Mixture Materials

allows you to choose the mixture material for which you want to modify properties. This option is available when **mixture** is selected in the **Material Type** drop-down list.

Important:

A material may co-exist as a pure substance fluid as well as a species in one or multiple mixture materials. The pure fluid and the species for each mixture are separate objects and may have independent property settings. *Therefore, editing the properties of the pure fluid will not modify the same properties for that fluid included in the mixture. If you intend to modify some material properties, you can make a copy of the pure fluid, and re-name it. Using the copy in the mixture makes it easier to understand which version of the fluid is being modified.*

Fluent Droplet Particle Materials

allows you to choose the mixture parent when editing species. Choose **none** to edit the pure substance material. *This option is available when **droplet-particle** is selected in the **Material Type** drop-down list.*

Order Materials by

allows you to order the materials in the **Materials** list alphabetically by **Name** or alphabetically by **Chemical Formula**.

GRANTA MDS Database...

opens the GRANTA MDS Materials Dialog Box (p. 4899), from where you can copy materials from the GRANTA MDS database into the current solver.

Database...

opens the Fluent Database Materials Dialog Box (p. 4897), from where you can copy materials from the global database into the current solver.

User-Defined Database...

opens the Open Database Dialog Box (p. 4901), where you can specify the user-defined database to be used.

Properties

contains input fields for the material properties that are required for the active physical models.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Density

sets the material density. You may set a constant value, or select one of the other methods from the drop-down list above the real number field. See Density (p. 1624) for instructions on setting density.

Cp (Specific Heat)

sets the constant-pressure specific heat of the material. You may set a constant value, or select one of the other methods from the drop-down list above the real number field. See Specific Heat Capacity (p. 1663) for instructions on setting specific heat.

Thermal Conductivity

sets the thermal conductivity of the material. You may set a constant value, or select one of the other methods from the drop-down list above the real number field. See Thermal Conductivity (p. 1644) for instructions on setting thermal conductivity.

Viscosity

sets the viscosity of the material. You may set a constant value, or select one of the other methods from the drop-down list above the real number field. See Viscosity (p. 1634) for instructions on setting viscosity.

Molecular Weight

sets the molecular weight of the material. It is used to derive the gas constant of the material.

Standard State Enthalpy

specifies the formation enthalpy of a fluid material for a reacting flow. See Standard State Enthalpies (p. 1675) for details.

Standard State Entropy

specifies the standard state entropy of a fluid material for a reacting flow. This input is used only if the fluid material is involved in a reversible reaction. See Standard State Entropies (p. 1676) for details.

Reference Temperature

specifies the reference temperature for the **Heat of Formation**.

L-J Characteristic Length

specifies the kinetic theory parameter σ for a fluid material. See Kinetic Theory Parameters (p. 1676) for details.

L-J Energy Parameter

specifies the kinetic theory parameter ε / k for a fluid material. See Kinetic Theory Parameters (p. 1676) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Absorption Coefficient

specifies the absorption coefficient a for radiation heat transfer. See Radiation Properties (p. 1665) for details. If you choose the **wsggm-user-specified** option from the drop-down list next to **Absorption Coefficient**, the WSGGM User Specified Dialog Box (p. 4947) will open. If you choose the **user-defined-wsggm** option from the drop-down list next to **Absorption Coefficient**, the User-Defined Functions Dialog Box (p. 4915) will open.

Scattering Coefficient

specifies the scattering coefficient σ_s for radiation heat transfer (only for the P-1, Rosseland, DO, or MC radiation models). See Radiation Properties (p. 1665) for details.

Scattering Phase Function

specifies an **isotropic** (by default) or **linear-anisotropic** scattering function. If you are using the DO or MC models, **delta-eddington** and **user-defined** scattering functions are also available. See Radiation Properties (p. 1665) for details. If you choose **delta-eddington**, the Delta-Eddington Scattering Function Dialog Box (p. 4949) will open.

Refractive Index

specifies the refractive index for the material. It is used only when semi-transparent media are modeled with the DO or MC radiation models.

Mixture Species

specifies the names of the species that make up a mixture material. To check or modify these names, click the **Edit...** button to open the Species Dialog Box (p. 4929). This property appears only for mixture materials.

Reaction

displays the reaction mechanism being used when you are modeling finite-rate reactions. **finite-rate** appears if **LaminarFinite-Rate** or **Eddy-Dissipation Concept** is selected in the Species Model Dialog Box (p. 4768), **eddy-dissipation** appears if **Eddy-Dissipation** is selected, and **finite-rate/eddy-dissipation** appears if **Finite-Rate/Eddy-Dissipation** is selected.

Click **Edit...** to open the Reactions Dialog Box (p. 4930).

Mechanism

allows you to enable different reactions selectively in different geometrical zones. Click the **Edit** button to open the Reaction Mechanisms Dialog Box (p. 4941). See Defining Zone-Based Reaction Mechanisms (p. 2387) for details.

Mass Diffusivity

contains a drop-down list of available methods for specifying the diffusion coefficients for the species in a mixture material. If you select **constant-dilute-appx**, you will enter a constant value in the field below. If you select **dilute-approx** or **multicomponent**, the Mass Diffusion Coefficients Dialog Box (p. 4944) will open, and you can specify the coefficients there. If you select **kinetic-theory**, you will need to specify the kinetic theory parameters for the individual

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

fluid materials (species) that make up the mixture. See Mass Diffusion Coefficients (p. 1668) for details about specifying mass diffusivity.

Thermal Diffusion Coefficient

contains a drop-down list of available methods for specifying the thermal diffusion coefficients for the species in a mixture material. If you select **kinetic-theory**, you will need to specify the kinetic theory materials (species) that make up the mixture. If you select **specified**, the Thermal Diffusion Coefficients Dialog Box (p. 4945) will open, and you can specify the coefficients there. See Thermal Diffusion Coefficient Inputs (p. 1674) for details about specifying thermal diffusion coefficients.

Density of Unburnt Reactants

sets the density (ρ_u in Equation 8.100 in the Theory Guide) of the unburnt products.

Temperature of Unburnt Reactants

sets the temperature (T_u in Equation 8.100 in the Theory Guide) of the unburnt products.

Adiabatic Temperature of Burnt Products

(only for adiabatic premixed combustion models) specifies the value of the burnt products under adiabatic conditions, T_{ad} in Equation 8.97 in the Theory Guide.

Unburnt Thermal Diffusivity

specifies the thermal diffusivity (α in Equation 8.77 in the Theory Guide) for use with the premixed combustion model. See Modeling Premixed Combustion (p. 2536) for details.

Laminar Flame Speed

specifies the value of U_l in Equation 8.77 in the Theory Guide.

Laminar Flame Thickness

specifies the thickness of the flame for which you have a choice of **constant**, **user-defined**, or **diffusivity-over-flame-speed**.

Critical Rate of Strain

specifies the value of g_{cr} in Equation 8.86 in the Theory Guide.

Heat of Combustion

(only for non-adiabatic premixed combustion models) specifies the value of H_{comb} in Equation 8.99 in the Theory Guide.

Unburnt Fuel Mass Fraction

(only for non-adiabatic premixed combustion models) specifies the value of Y_{fuel} in Equation 8.99 in the Theory Guide.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Thermal Expansion Coefficient

specifies the thermal expansion coefficient (β in Equation 7.67 (p. 1388)) for use with the Boussinesq approximation. See Steps in Solving Buoyancy-Driven Flow Problems (p. 1388) for details.

Droplet Surface Tension

specifies the value of the droplet surface tension (σ in Equation 12.371 in the Theory Guide).

Latent Heat

is the latent heat of vaporization, h_{fg}, required for phase change from an evaporating liquid droplet or for the evolution of volatiles from a combusting particle. See Setting Material Properties for the Discrete Phase (p. 2811) for details.

Latent Heat at NBP

is the latent heat of vaporization at the normal boiling point, available for droplet-particles. See Using the Cubic Equation of State Models with the Lagrangian Dispersed Phase Models (p. 1698) for details.

Normal Boiling Point (NBP)

is given by Equation 8.104 (p. 1699). See Using the Cubic Equation of State Models with the Lagrangian Dispersed Phase Models (p. 1698) for details.

Thermophoretic Coefficient

specifies the thermophoretic coefficient ($D_{T,p}$ in Equation 12.11 in the Theory Guide), and appears when the thermophoretic force is included in the discrete phase calculation.

Vaporization Temperature

is the temperature, T_{vap}, at which the calculation of vaporization from a liquid droplet or devolatilization from a combusting particle is initiated by Ansys Fluent. See Setting Material Properties for the Discrete Phase (p. 2811) for details.

Boiling Point

is the temperature, T_{bp}, at which the calculation of the boiling rate equation is initiated by Ansys Fluent. See Setting Material Properties for the Discrete Phase (p. 2811) for details.

Vapor-Particle-Equilibrium

is the selected approach for the calculation of the vapor concentration of the components at the surface. This can be Raoult's law (Equation 12.176 in the Theory Guide), the Peng-Robinson real gas model (Equation 12.184 in the Theory Guide), or a user-defined function that provides this value.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reaction Model

allows you to select the reaction model for the particle mixture material. You can choose from **none** and **multicomponent-reactions**. This item appears only for cases with reacting multicomponent particles. See Multicomponent Particles with Chemical Reactions (p. 2409) for details.

Volatile Component Fraction

(f_{v0}) is the fraction of a droplet particle that may vaporize via Laws 2 and/or 3 (Droplet Vaporization (Law 2) in the Theory Guide). For combusting particles, it is the fraction of volatiles that may be evolved via Law 4 (Devolatilization (Law 4) in the Theory Guide). See Setting Material Properties for the Discrete Phase (p. 2811) for details.

Binary Diffusivity

is the mass diffusion coefficient, $D_{i,m}$, used in the vaporization law, Law 2. This input is also used to define the mass diffusion of the oxidizing species to the surface of a combusting particle, $D_{i,m}$. See Setting Material Properties for the Discrete Phase (p. 2811) for details.

Diffusivity Reference Pressure

is the reference pressure P_{ref} for the pressure dependent binary diffusivity in Equation 12.91 in the *Fluent Theory Guide*.

Saturation Vapor Pressure

is the saturated vapor pressure, P_{sat}, defined as a function of temperature, which is used in the vaporization law, Law 2. See Setting Material Properties for the Discrete Phase (p. 2811) for details.

Heat of Pyrolysis

is the heat of the instantaneous pyrolysis reaction, h_{pyrol}, that the evaporating/boiling species may undergo when released to the continuous phase. The heat of pyrolysis should be entered as a positive number for exothermic reaction and as a negative number for endothermic reaction. The default value of zero implies that the heat of pyrolysis is not considered. See Setting Material Properties for the Discrete Phase (p. 2811) for details.

Composition Averaging Coefficient

is the coefficient α_Y in Equation 12.186 in the *Fluent Theory Guide*. To assume bulk gas composition for the physical properties in the particle vaporization and heating rates equations, select `none` from the **Composition Averaging Coefficient** drop-down in the **Create/Edit Materials** dialog box, under **Properties**. To enable property averaging, select `constant` and enter a value between 0 and 1 for the **Composition Averaging Coefficient**. $\alpha_Y = 1$ corresponds to bulk gas mixture. For $\alpha_Y = 0$, the composition reverts to the particle surface composition.

Important:

When you define the **Composition Averaging Coefficient** with a `constant` value other than 1, you must also specify **Specific Heat**, **Viscosity** and **Thermal Conduct-**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ivity for the evaporating species (in the **Create/Edit Materials** dialog box, under **Properties**, for the corresponding fluid material of the continuous phase mixture).

The default and recommended value for the averaging coefficient is 1/3. However, to improve the simulation results, you must provide accurate temperature-dependent data for the vapor material.

Temperature Averaging Coefficient

is the coefficient α_T in Equation 12.185 in the *Fluent Theory Guide*. To assume bulk gas temperature for the physical properties in the particle vaporization and heating rates equations, select `none` from the **Temperature Averaging Coefficient** drop-down in the **Create/Edit Materials** dialog box, under **Properties**. To enable property averaging, select `constant` and enter a value between 0 and 1 for the **Temperature Averaging Coefficient**. α_T= 1 corresponds to bulk gas conditions. For α_T= 0, the temperature reverts to the particle surface temperature.

Important:

When you define the **Temperature Averaging Coefficient** with a `constant` value other than 1, you must also specify **Specific Heat**, **Viscosity** and **Thermal Conductivity** for the evaporating species (in the **Create/Edit Materials** dialog box, under **Properties**, for the corresponding fluid material of the continuous phase mixture).

The default and recommended value for the averaging coefficient is 1/3. However, to improve the simulation results, you must provide accurate temperature-dependent data for the evaporating vapor material.

Vaporization/Boiling Model

defines which vaporization/boiling model is used for pure droplets (Law 2) and for multicomponent droplets (Law 7).

If you want to use the default diffusion-controlled model, retain the selection of **diffusion-controlled** from the drop-down list to the right of **Vaporization/Boiling Model**. This will apply Equation 12.80 in the Theory Guide.

To use the convection/diffusion-controlled model for vaporization select **convection/diffusion-controlled** from the drop-down list. Equation 12.85 in the Theory Guide will be applied for the calculation of the vaporization rate, and Equation 12.97 in the Theory Guide will be applied in the particle heat transfer calculations This model is recommended when evaporation rates are high. For slowly evaporating droplets both models are expected to give similar results.

You can select options for the vaporization/boiling model in the Model Options Dialog Box (p. 4913) that opens when you click **Edit...** next to **Vaporization/Boiling Model**.

Degrees of Freedom

specifies the kinetic theory parameter f, which is the number of nodes of energy storage. This parameter is required only if you are defining specific heat via kinetic theory. See Kinetic Theory Parameters (p. 1676) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Particle Emissivity

is the emissivity of particles in your model, ε_p, used to compute radiation heat transfer to the particles when the P-1 or DO radiation model is active and particle radiation interaction is enabled in the Discrete Phase Model Dialog Box (p. 4845). See Setting Material Properties for the Discrete Phase (p. 2811) for details.

Particle Scattering Factor

is the scattering factor, f, due to particles in the P-1 or DO radiation model. Note that this property will appear only if particle radiation interaction is enabled in the Discrete Phase Model Dialog Box (p. 4845). See Setting Material Properties for the Discrete Phase (p. 2811) for details.

Swelling Coefficient

is the coefficient, C_{sw}, which governs the swelling of the coal particle during the devolatilization law, Law 4. A swelling coefficient of unity (the default) implies that the coal particle stays at constant diameter during the devolatilization process. See Setting Material Properties for the Discrete Phase (p. 2811) for details.

Burnout Stoichiometric Ratio

is the stoichiometric requirement, S_b, for the burnout reaction, in terms of mass of oxidant per mass of char in the particle. See Setting Material Properties for the Discrete Phase (p. 2811) for details.

Combustible Fraction

is the mass fraction of char, f_{comb}, in the coal particle (the fraction of the initial combusting particle that will react in the surface reaction) Law 5. See Setting Material Properties for the Discrete Phase (p. 2811) for details.

React. Heat Fraction Absorbed by Solid

is the parameter f_h, which controls the distribution of the heat of reaction between the particle and the continuous phase. The default value of zero implies that the entire heat of reaction is released to the continuous phase. See Setting Material Properties for the Discrete Phase (p. 2811) for details.

Heat of Reaction for Burnout

is the heat released by the surface char combustion reaction, Law 5. This parameter is specified in terms of heat release (for example, Joules) per unit mass of char consumed in the surface reaction. See Setting Material Properties for the Discrete Phase (p. 2811) for details.

Devolatilization Model

defines which version of the devolatilization model, Law 4, is being used. If you want to use the default constant rate devolatilization model, retain the selection of **constant** in the drop-down list to the right of **Devolatilization Model** and enter the rate constant A_0 in the field below the list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select **single-rate**, **two-competing-rates**, or **cpd-model** in the drop-down list to choose one of the optional devolatilization models (the single kinetic rate model, two kinetic rates model, or CPD model, as described in Devolatilization (Law 4) in the Theory Guide).

When the single kinetic rate model (**single-rate**) is selected, the Single Rate Model Dialog Box (p. 4950) will appear; when the two competing rates model (**two-competing-rates**) is selected, the Two Competing Rates Model Dialog Box (p. 4952) will appear; and when the CPD model (**cpd-model**) is selected, the CPD Model Dialog Box (p. 4954) will appear.

See Setting Material Properties for the Discrete Phase (p. 2811) for details.

Combustion Model

defines which version of the surface char combustion law (Law 5) is being used. If you want to use the default diffusion-limited rate model, retain the selection of **diffusion-limited** in the drop-down list. No additional inputs are necessary, because the binary diffusivity defined above will be used in Equation 12.146 in the Theory Guide.

To use the kinetics/diffusion-limited rate model for the surface combustion model, select **kinetics/diffusion-limited** in the drop-down list and enter the parameters in the resulting Kinetics/Diffusion-Limited Combustion Model Dialog Box (p. 4955).

To use the intrinsic model for the surface combustion model, select **intrinsic-model** in the drop-down list and enter the parameters in the resulting Intrinsic Combustion Model Dialog Box (p. 4955).

To use the multiple surface reactions model, select **multiple-surface-reactions** in the drop-down list.

See Setting Material Properties for the Discrete Phase (p. 2811) for details.

Pure Solvent Melting Heat

specifies the latent heat for the melting and solidification model (L in Equation 15.3 in the Theory Guide). Note that for solidification to occur, **Pure Solvent Melting Heat** must be set to a positive non-zero value.

Solidus Temperature

specifies the solidus temperature for the melting and solidification model ($T_{solidus}$ in Equation 15.3 in the Theory Guide). If you are solving for species transport, the solidus temperature is $T_{solidus}$ in Equation 15.8, and you specify the method by which it is calculated: either according to the **mixing-law** (which is based on the parameters of the solutes) or a **user-defined** function.

Liquidus Temperature

specifies the liquidus temperature for the melting and solidification model ($T_{liquidus}$ in Equation 15.3 in the Theory Guide). If you are solving for species transport, the liquidus temperature is T_{liquidus} in Equation 15.9, and you specify the method by which it is calculated: either according to the **mixing-law** (which is based on the parameters of the solutes) or a **user-defined** function.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Pure Solvent Melting Temperature

specifies the melting temperature of pure solvent (T_{melt} in Equation 15.8 and Equation 15.9 in the Theory Guide) for the melting and solidification model when species transport has also been enabled. The solvent is the last species listed under of the mixture material.

Eutectic Temperature

is the lowest alloy melting temperature, which depends on the relative proportions of the mixture composition of the Eutectic species mass fractions.

Slope of Liquidus Line

specifies the slope of the liquidus surface with respect to the concentration of the solute (m_i in Equation 15.8 and Equation 15.9 in the Theory Guide). It is not necessary to specify this value for the solvent. Note that this option is available only for the melting and solidification model when species transport has also been enabled.

Partition Coefficient

specifies the partition coefficient with respect to the concentration of the solute fluid (K_i in Equation 15.8 and Equation 15.9 in the Theory Guide). It is not necessary to specify this value for the solvent. Note that this option is available only for the melting and solidification model when species transport has also been enabled.

Eutectic Mass Fraction

is the mass fraction of a solute in an alloy at which the melting temperature of the alloy is the lowest possible value ($Y_{i,\mathrm{Eut}}$ in Equation 15.10 in the Theory Guide). It is not necessary to specify this value for the solvent. Note that this option is available only for the melting and solidification model when species transport has also been enabled.

Solutal Expansion Coefficient

allows you to specify the coefficient in Equation 15.24 in the Theory Guide for all the species except the last one in the mixture. Note that this option is available only for the melting and solidification model when the **Include Solutal Buoyancy** option is enabled.

Diffusion in Solid

specifies the rate of diffusion in the solid. Note that this option is available only for the melting and solidification model when the Lever rule is selected and the species transport is enabled.

UDS Diffusivity

specifies the diffusion coefficient for a user-defined scalar. This material property is available in the Create/Edit Materials Dialog Box (p. 4885) when you specify one or more user-defined scalars in the User-Defined Scalars Dialog Box (p. 5654). If you select **defined-per-uds**, you will need to specify the diffusion coefficient for each user-defined scalar transport equation in the UDS Diffusion Coefficients Dialog Box (p. 4946).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Youngs Modulus

specifies the Young's modulus. This material property is only available for solid materials when you have selected a model in the Structural Model Dialog Box (p. 4869).

Poisson Ratio

specifies the Poisson's ratio. This material property is only available for solid materials when you have selected a model in the Structural Model Dialog Box (p. 4869).

Electrical Conductivity

sets the material electrical conductivity. You may set a constant value, or select one of the other methods from the drop-down list above the real number field. This material property only becomes available if the electric potential model is enabled.

Charge Number

specifies the charge number for a fluid material for a flow involving electrochemical reactions. The default setting of 0 means that the fluid species is neutral. This material property only becomes available if the electric potential model is enabled.

When you are viewing the database, additional properties may be displayed. However, after you copy the material to the local area, only the properties with relevance to the current problem will be displayed.

Change/Create

changes the properties of a locally stored material or creates a new one in the local area. If no material with the specified **Name** exists locally, Ansys Fluent will create it. If you have modified the material without changing its name, Ansys Fluent will simply update the material with your modifications. If you have assigned a new name to the material and a material with this name already exists locally, an error will be indicated; you must then specify a different name or delete the existing material with that name before trying to save the new material.

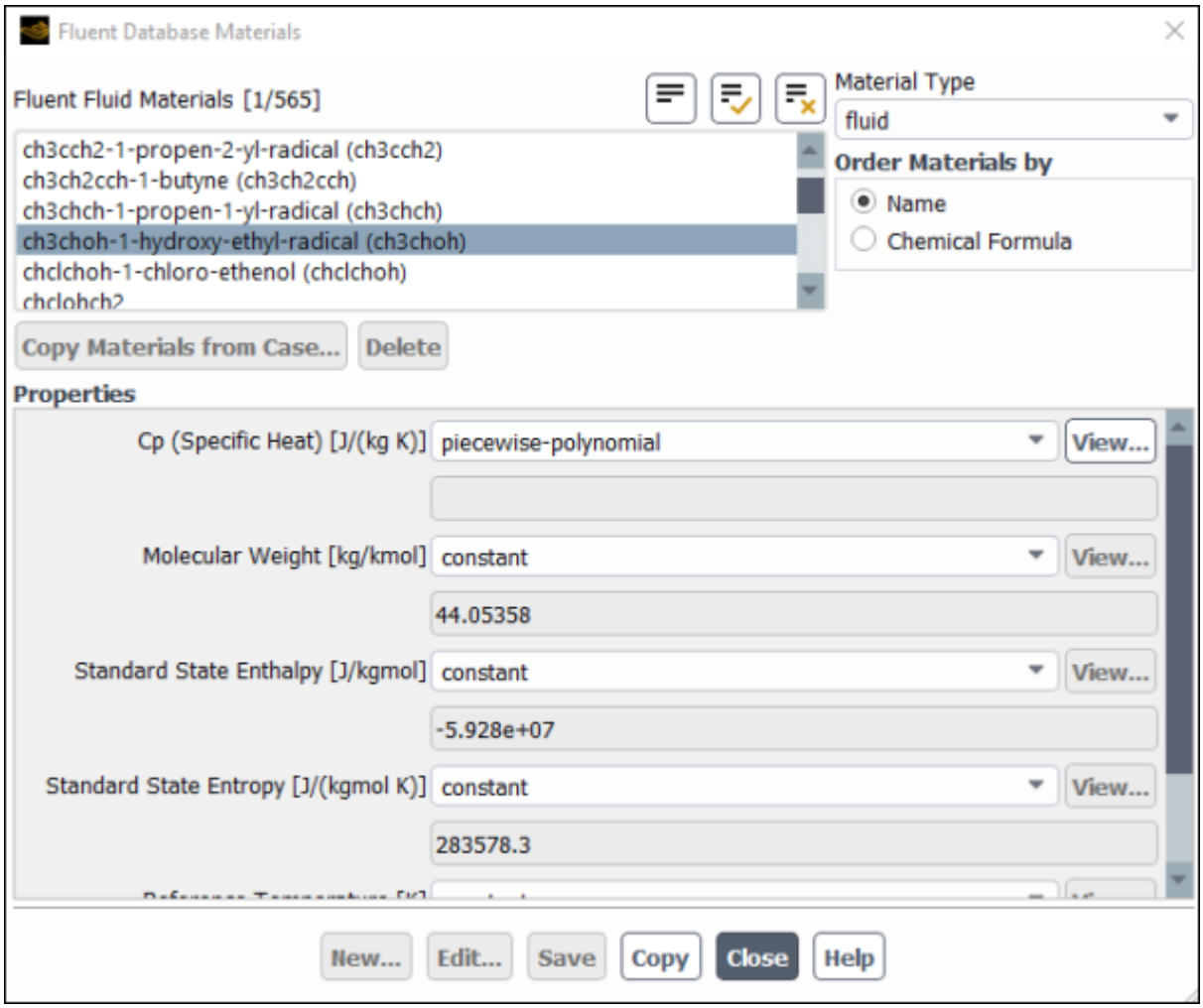
Delete

deletes the currently selected material from the local materials list. It has no effect on the global database.

51.5.2. Fluent Database Materials Dialog Box

The **Fluent Database Materials** dialog box is opened by clicking the **Fluent Database...** button in the Create/Edit Materials Dialog Box (p. 4885). In this dialog box you can view the global (site-wide) material properties database and copy materials list in the solver. See Copying Materials from the Ansys Fluent Database (p. 1602) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Fluent Fluid Materials

contains a list of all materials of the selected **Material Type** that are defined in the database. The name of this list will change depending on the selected material type (for example, **fluid**, **solid**, and so on). You can select one or more of these materials to be copied to the solver.

Material Type

is a drop-down list containing all of the available material types. By default, **fluid** and **solid** will be the only choices. If you are modeling species transport/combustion, **mixture** will also be available. For problems in which you have defined discrete-phase injections, **inert-particle**, **droplet-particle**, and/or **combusting-particle** will also appear.

Order Materials by

allows you to order the materials in the **Materials** list alphabetically by **Name** or alphabetically by **Chemical Formula**.

Copy Materials from Case...

opens the Copy Case Material Dialog Box (p. 4903).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Delete

deletes the selected materials from the database.

Properties

contains fields for the material properties that are defined for the selected material. These fields are for informational purposes only; they cannot be edited.

When you are viewing the database, not all properties displayed are relevant to your Ansys Fluent solution. After you copy the material, only properties with relevance to the current physical models will be displayed.

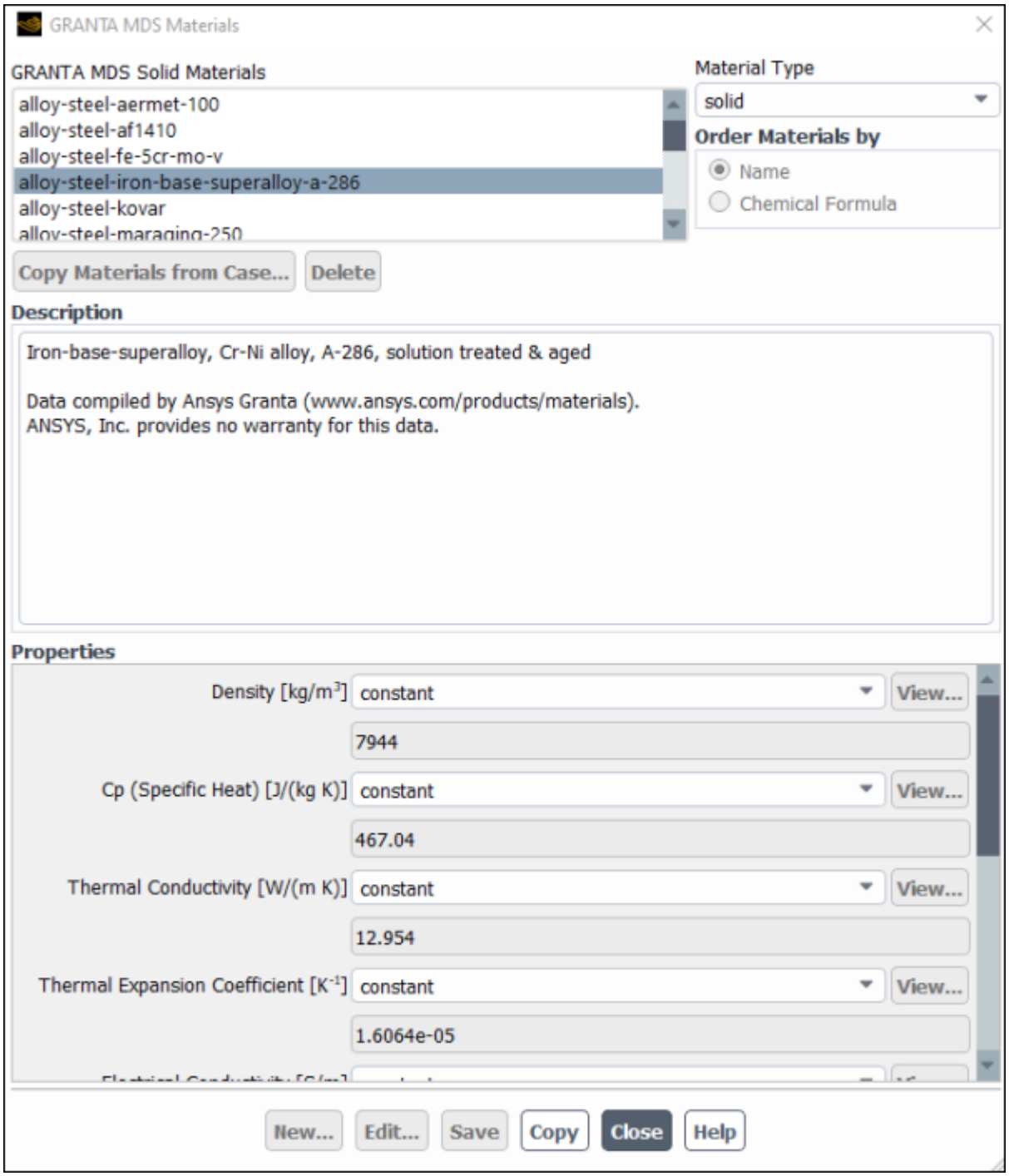
Copy

copies the current material from the global database to the local materials list in the solver.

51.5.3. GRANTA MDS Materials Dialog Box

The **GRANTA MDS Materials** dialog box is opened by clicking the **GRANTA MDS Database...** button in the Create/Edit Materials Dialog Box (p. 4885). In this dialog box you can view the global (site-wide) GRANTA MDS material properties database and copy materials list in the solver (available with a separate GRANTA licence). See Copying Materials from the Ansys GRANTA MDS Database (p. 1604) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

GRANTA MDS Solid Materials

contains a list of all materials of the selected **Material Type** that are defined in the database. You can select one of these materials to be copied to the solver. Note that multi-selection is not supported.

Material Type

is a drop-down list containing all of the available material types.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Description

displays a description of the selected material.

Properties

contains fields for the material properties that are defined for the selected material. These fields are for informational purposes only; they cannot be edited.

When you are viewing the database, not all properties displayed are relevant to your Ansys Fluent solution. After you copy the material, only properties with relevance to the current physical models will be displayed.

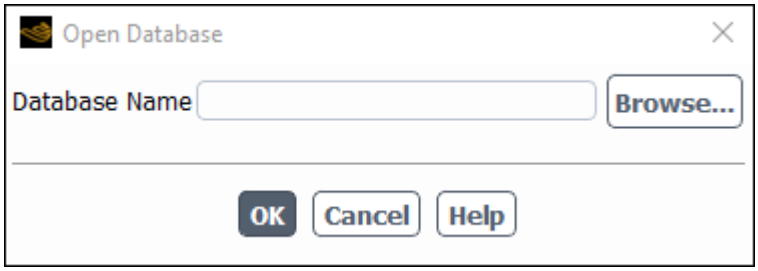
A license is required for viewing properties in the **GRANTA MDS Materials** dialog box. When no license is available, only the **Description** group box is displayed.

Copy

copies the current material from the global database to the local materials list in the solver (a GRANTA license is required).

51.5.4. Open Database Dialog Box

The **Open Database** dialog box is opened by clicking the **User-Defined Database** button in the Create/Edit Materials Dialog Box (p. 4885).



Controls

Browse...

opens The Select File Dialog Box (p. 931) where you can select the user-defined database to be used in the current solver session.

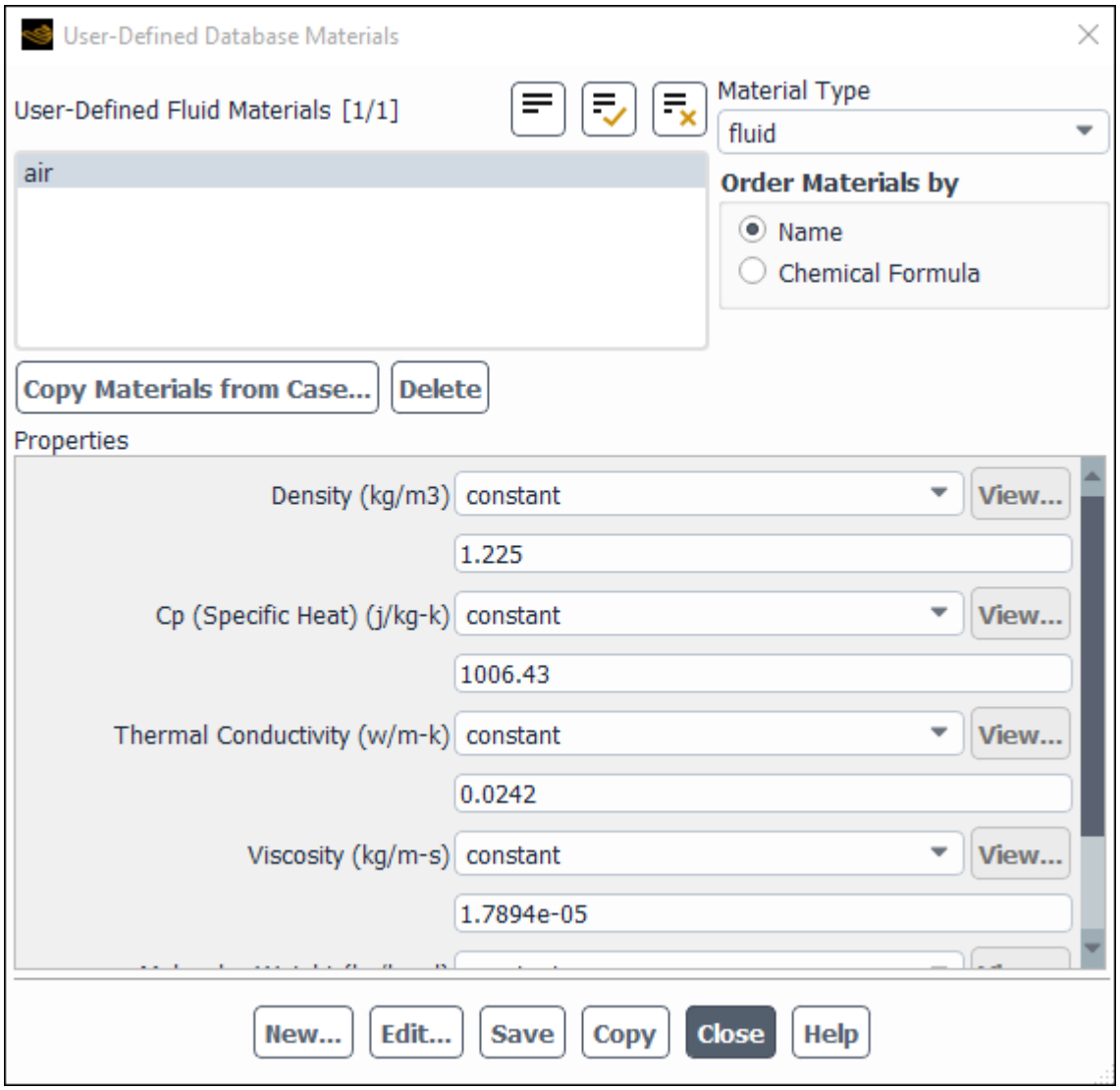
Database Name

allows you to enter the path and name of a new database. If you select an existing database, this field displays the path and name of the selected database.

51.5.5. User-Defined Database Materials Dialog Box

The **User-Defined Database Materials** dialog box is opened by clicking **OK** in the Open Database Dialog Box (p. 4901). In this dialog box you can view the user-defined material properties database and copy materials list in the solver. See Viewing Materials in a User-Defined Database (p. 1609) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

User-Defined Materials

contains a list of all materials of the selected **Material Type** that are defined in the database. The name of this list will change depending on the selected material type (for example, **User-Defined Fluid Materials**, **User-Defined Solid Materials**, and so on). You can select one or more of these materials to copy to your local list or edit their properties.

Material Type

is a drop-down list containing all of the available material types.

Order Materials By

allows you to order the materials in the **User-Defined Materials** list alphabetically by **Name** or alphabetically by **Chemical Formula**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Copy Materials from Case...

opens the Copy Case Material Dialog Box (p. 4903).

Delete

deletes the selected materials from the database.

Properties

lists the properties and values of the selected material.

New...

opens a blank Material Properties Dialog Box (p. 4904) where you can define a new material.

Edit...

opens the Material Properties Dialog Box (p. 4904) displaying the properties of the selected material.

Save

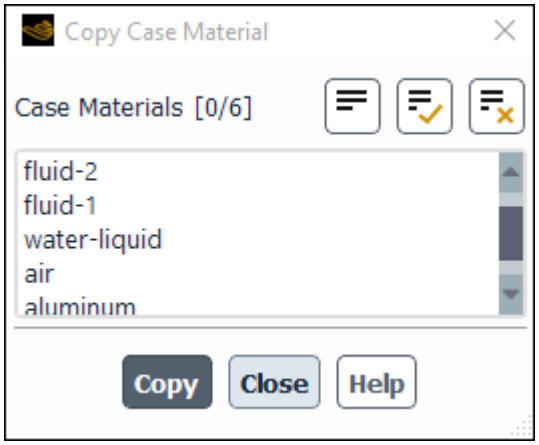
saves the information to the selected database.

Copy

copies the selected material to your local material list. If the material already exists the New Material Name Dialog Box (p. 4906) opens

51.5.6. Copy Case Material Dialog Box

This dialog box is opened by clicking the **Copy Materials from Case...** button in the User-Defined Database Materials Dialog Box (p. 4901).



Controls

Case Materials

lists all the materials present in your local materials list.

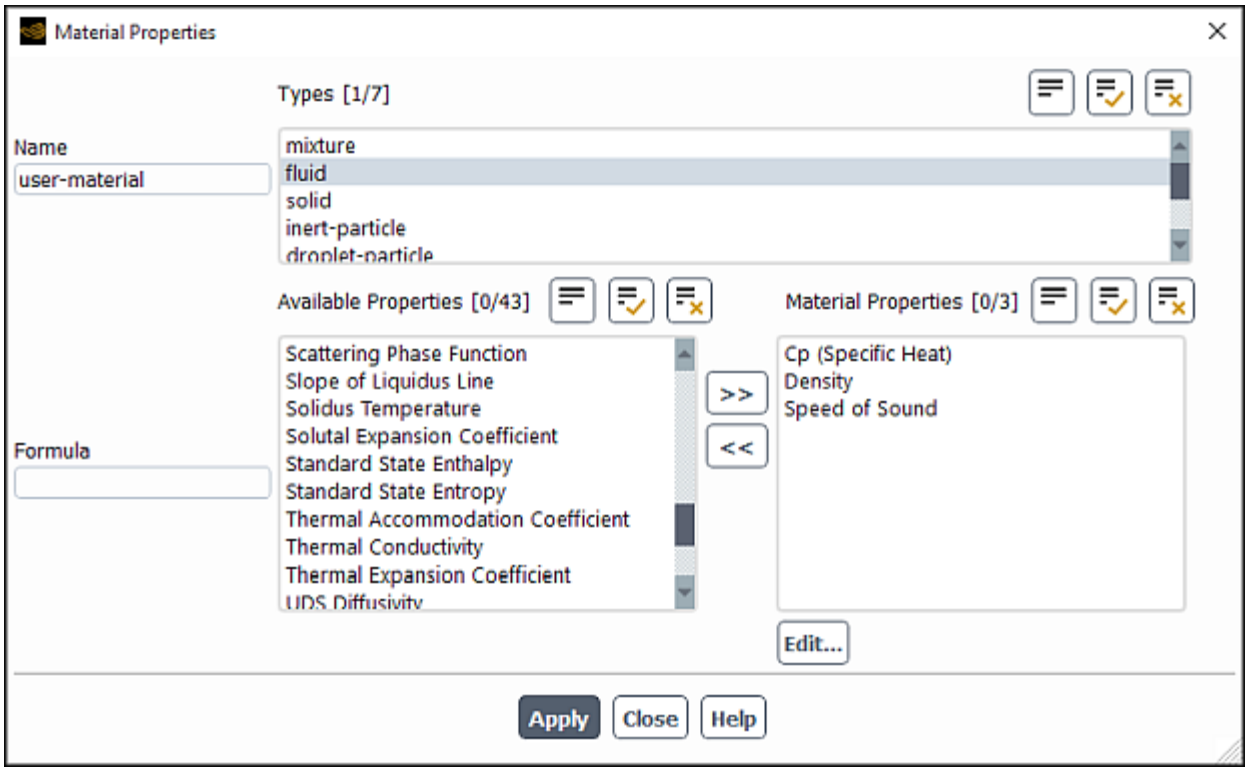
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Copy

copies the selected materials to the user-defined database.

51.5.7. Material Properties Dialog Box

This dialog box is opened by clicking **New...** button in the User-Defined Database Materials Dialog Box (p. 4901). See Creating a New Materials Database and Materials (p. 1612) for details.



Controls

Name

specifies the name of the material that you are creating.

Formula

(optional) specifies the chemical formula of the material that you are creating.

Types

allows you to select material type from fluid, solid, inter-particle, droplet-particle, combusting-particle, and mixture materials.

Available Properties

lists properties applicable to the selected material type.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Material Properties

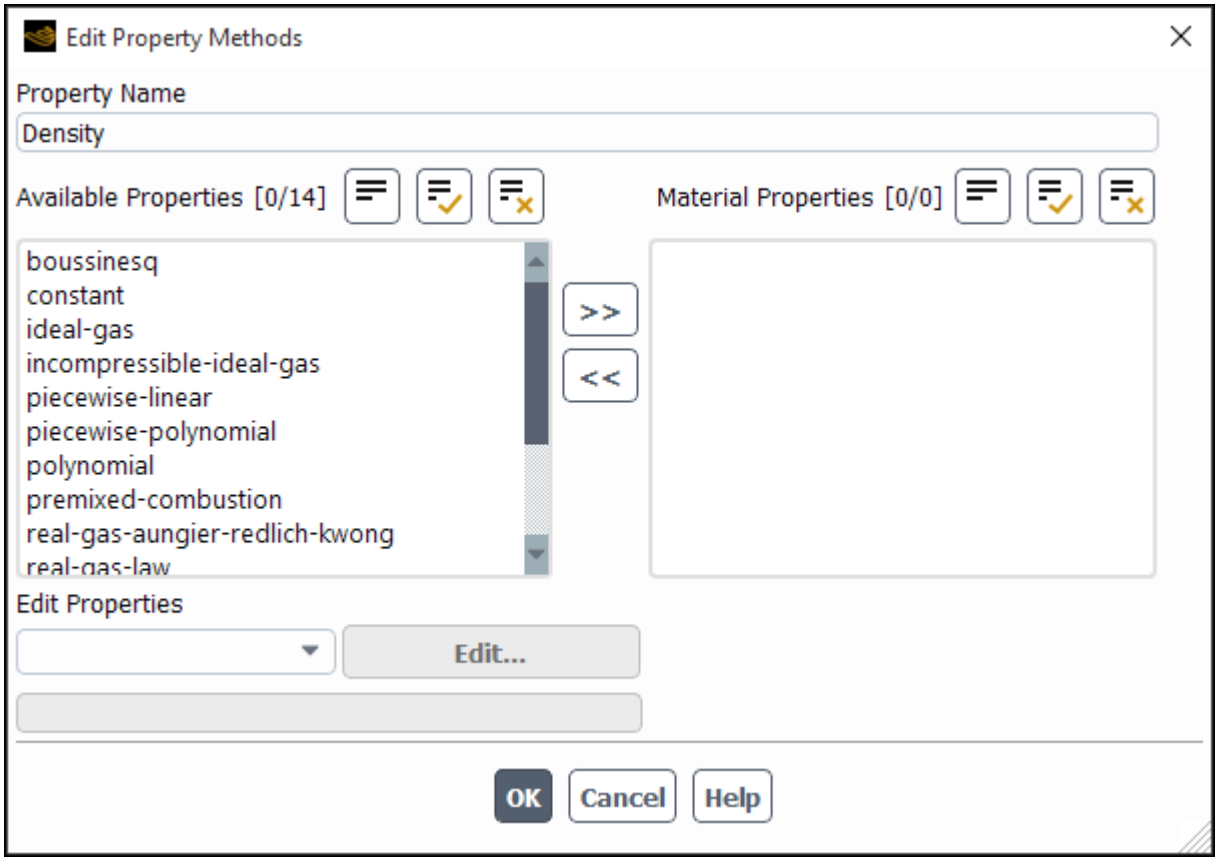
lists properties that you have selected from **Available Properties** list.

Edit...

opens Edit Property Methods Dialog Box (p. 4905) where you can edit the parameters that define a property.

51.5.8. Edit Property Methods Dialog Box

This dialog box is opened by clicking **Edit...** button in the Material Properties Dialog Box (p. 4904). See Creating a New Materials Database and Materials (p. 1612) for details.



Controls

Property Name

specifies the name of the property that you want to edit.

Available Properties

specifies the methods that can be used to define the selected property.

Material Properties

lists properties that you have selected from **Available Properties** list.

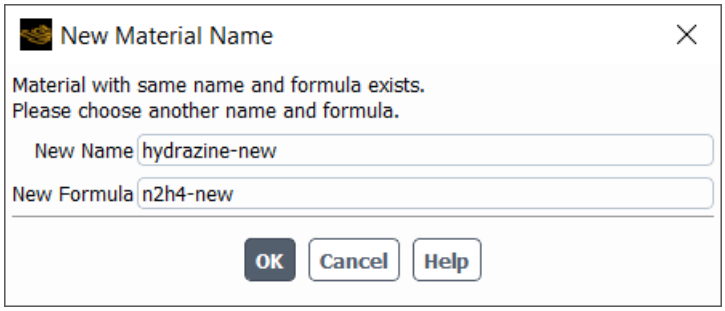
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Edit Properties

allows you to select the property that you want to edit.

51.5.9. New Material Name Dialog Box

This dialog box is opened by clicking the **Copy** button in the User-Defined Database Materials Dialog Box (p. 4901) when a material is already defined with the same name in the Create/Edit Materials Dialog Box (p. 4885).



Controls

New Name

allows you to enter the new name for the material you need to copy.

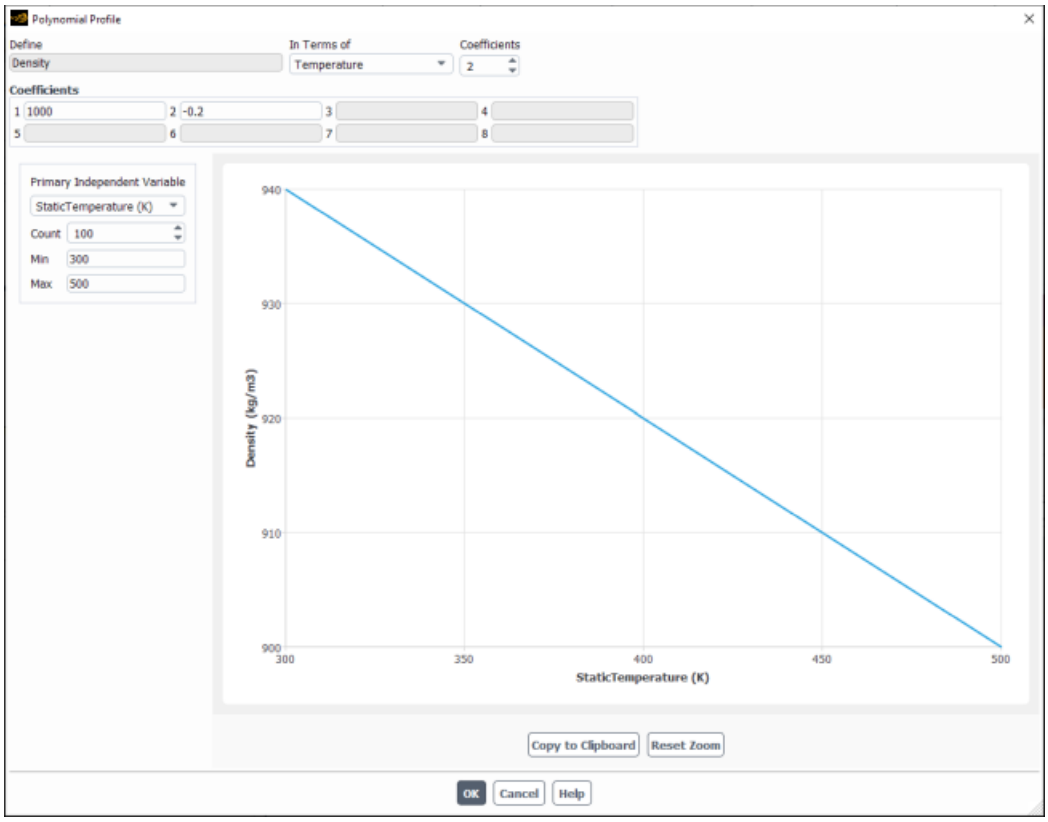
New Formula

allows you to enter the new formula for the material you need to copy.

51.5.10. Polynomial Profile Dialog Box

The **Polynomial Profile** dialog box allows you to define a physical property as a polynomial function of temperature. This dialog box will open when you select **polynomial** in the drop-down list next to a physical property in the Create/Edit Materials Dialog Box (p. 4885). See Inputs for Polynomial Functions (p. 1617) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Define

shows the property that is being defined as a function of temperature.

In Terms of

shows the independent variable (**Temperature**). The property shown in **Define** will be defined as a polynomial function of temperature.

Coefficients

is an integer number entry that indicates the number of coefficients to be defined. You can define up to 8 coefficients.

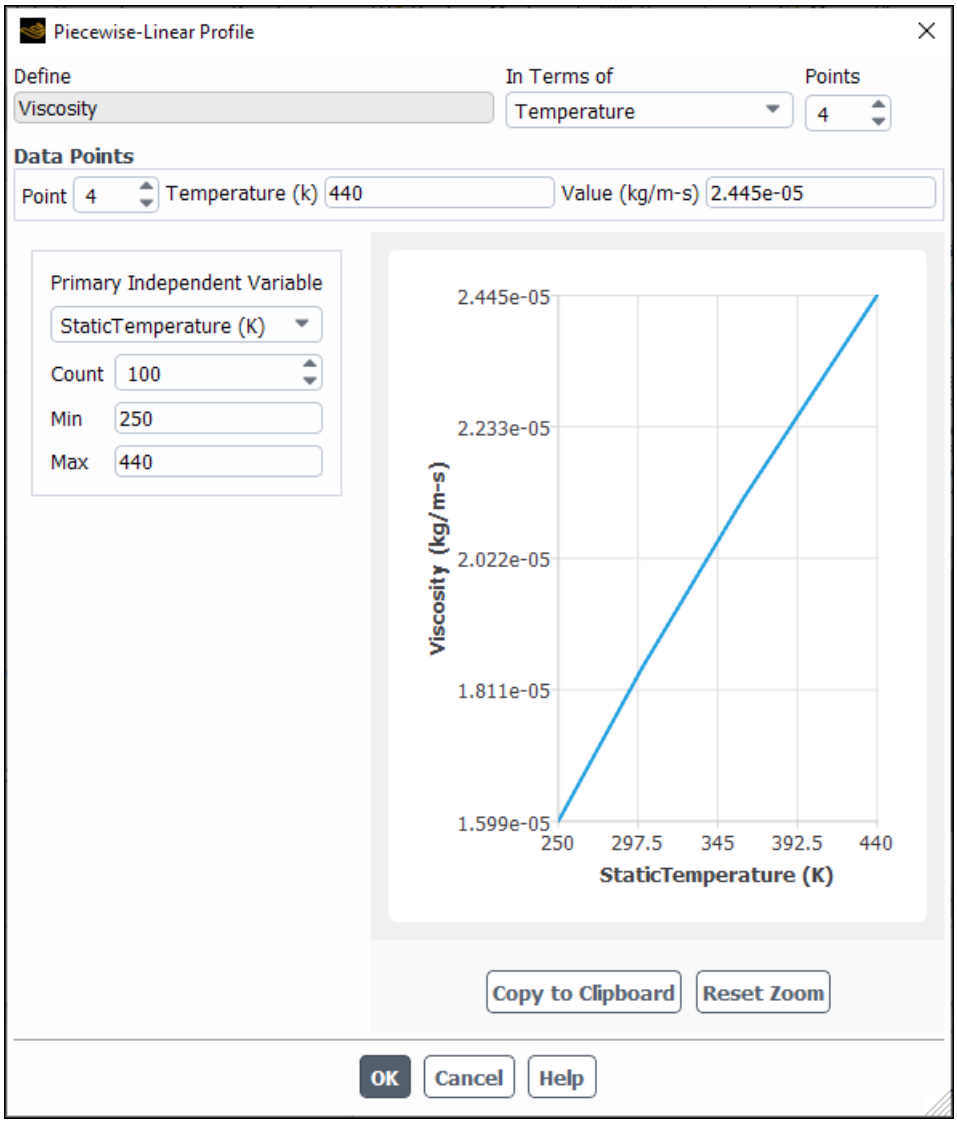
Coefficient Values

contains real number entries for the number of coefficients set in the **Coefficients** integer number entry above. The number of entries that are editable will be the same as the number of coefficients you requested.

51.5.11. Piecewise-Linear Profile Dialog Box

The **Piecewise-Linear Profile** dialog box allows you to define a physical property as a piecewise-linear function of temperature. This dialog box will open when you select **piecewise-linear** in the drop-down list next to a physical property in the Create/Edit Materials Dialog Box (p. 4885). See Inputs for Piecewise-Linear Functions (p. 1618) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Define

shows the property that is being defined as a function of temperature.

In Terms of

shows the independent variable (**Temperature**). The property shown in **Define** will be defined as a piecewise-linear function of temperature.

Points

indicates the number of data pairs that will define the piecewise distribution. You can define up to 50 pairs.

Data Points

contains entries for defining the data pairs.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Point

indicates the point for which the data pair (**Temperature**, **Value**) is being defined.

Temperature

is the independent variable.

Value

is the dependent variable (the property). In the example dialog box above, **Viscosity** is the variable being defined, as shown in the **Define** field.

Primary Independent Variable

allows you to select which variable is the independent variable.

Count

allows you to specify the number of plot points.

Min

sets the minimum value for the independent variable.

Max

sets the maximum value for the independent variable.

Copy to Clipboard

copies the current plot onto your clipboard.

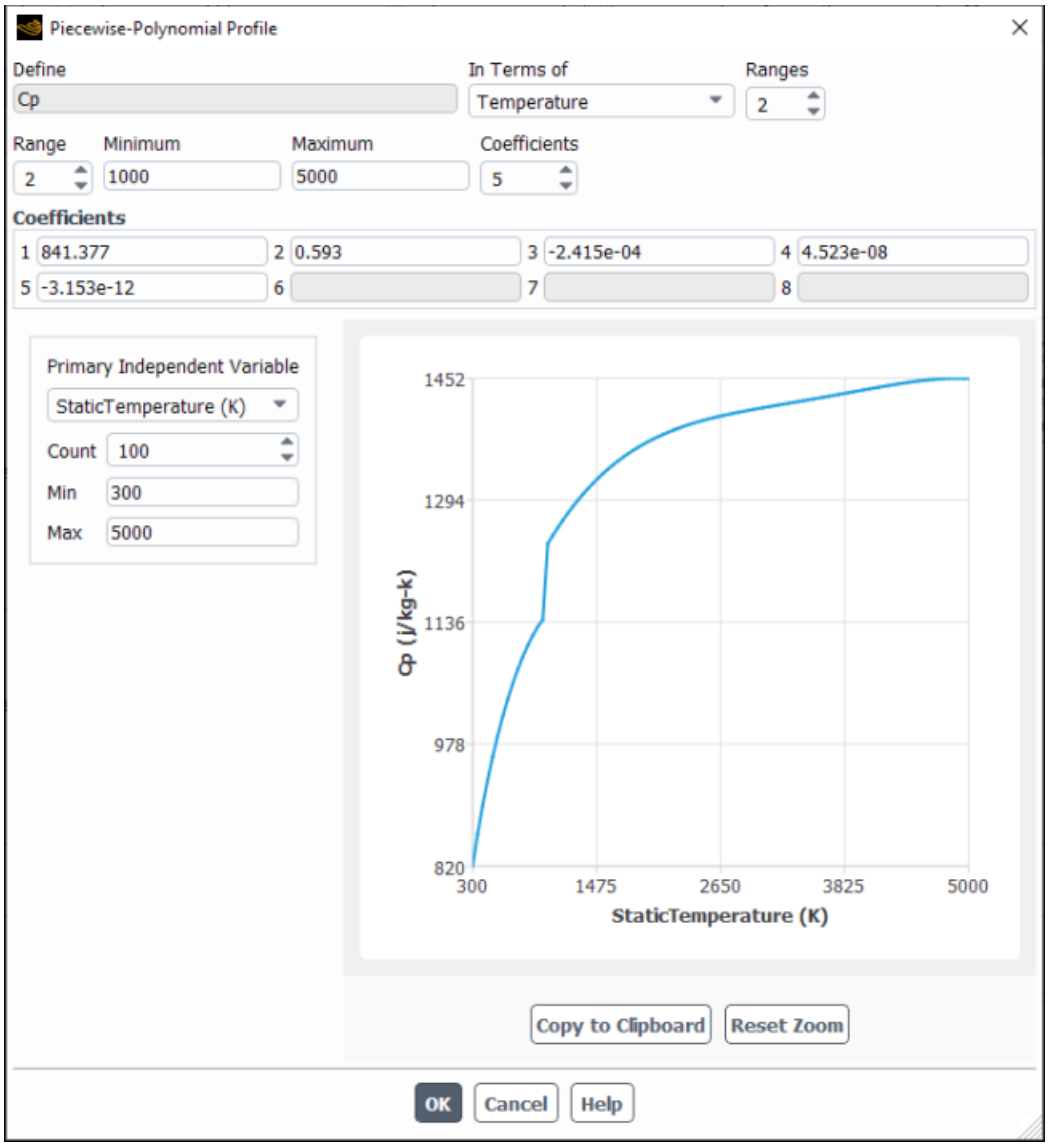
Reset Zoom

resets to the default zoom. *You can click and drag in the plot to change the zoom level.*

51.5.12. Piecewise-Polynomial Profile Dialog Box

The **Piecewise-Polynomial Profile** dialog box allows you to define a physical property as a piecewise-polynomial function of temperature. This dialog box will open when you select **piecewise-polynomial** in the drop-down list next to a physical property in the Create/Edit Materials Dialog Box (p. 4885). See Inputs for Piecewise-Polynomial Functions (p. 1620) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Define

shows the property that is being defined as a function of temperature.

In Terms of

shows the independent variable (**Temperature**). The property shown in **Define** will be defined as a polynomial function of temperature.

Ranges

sets the number of temperature ranges for which you will define polynomial functions. You can define up to 3 ranges.

Range

indicates the temperature range for which you are defining the polynomial function.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Minimum, Maximum

set the minimum and maximum temperatures for the specified **Range**.

Coefficients

is an integer number entry that indicates the number of coefficients to be defined for the specified **Range**. You can define up to 8 coefficients.

Coefficient Values

contains real number entries for the number of coefficients set in the **Coefficients** integer number entry above. The number of entries that are editable will be the same as the number of coefficients you requested for the specified **Range**.

Primary Independent Variable

allows you to select which variable is the independent variable.

Count

allows you to specify the number of plot points.

Min

sets the minimum value for the independent variable.

Max

sets the maximum value for the independent variable.

Copy to Clipboard

copies the current plot onto your clipboard.

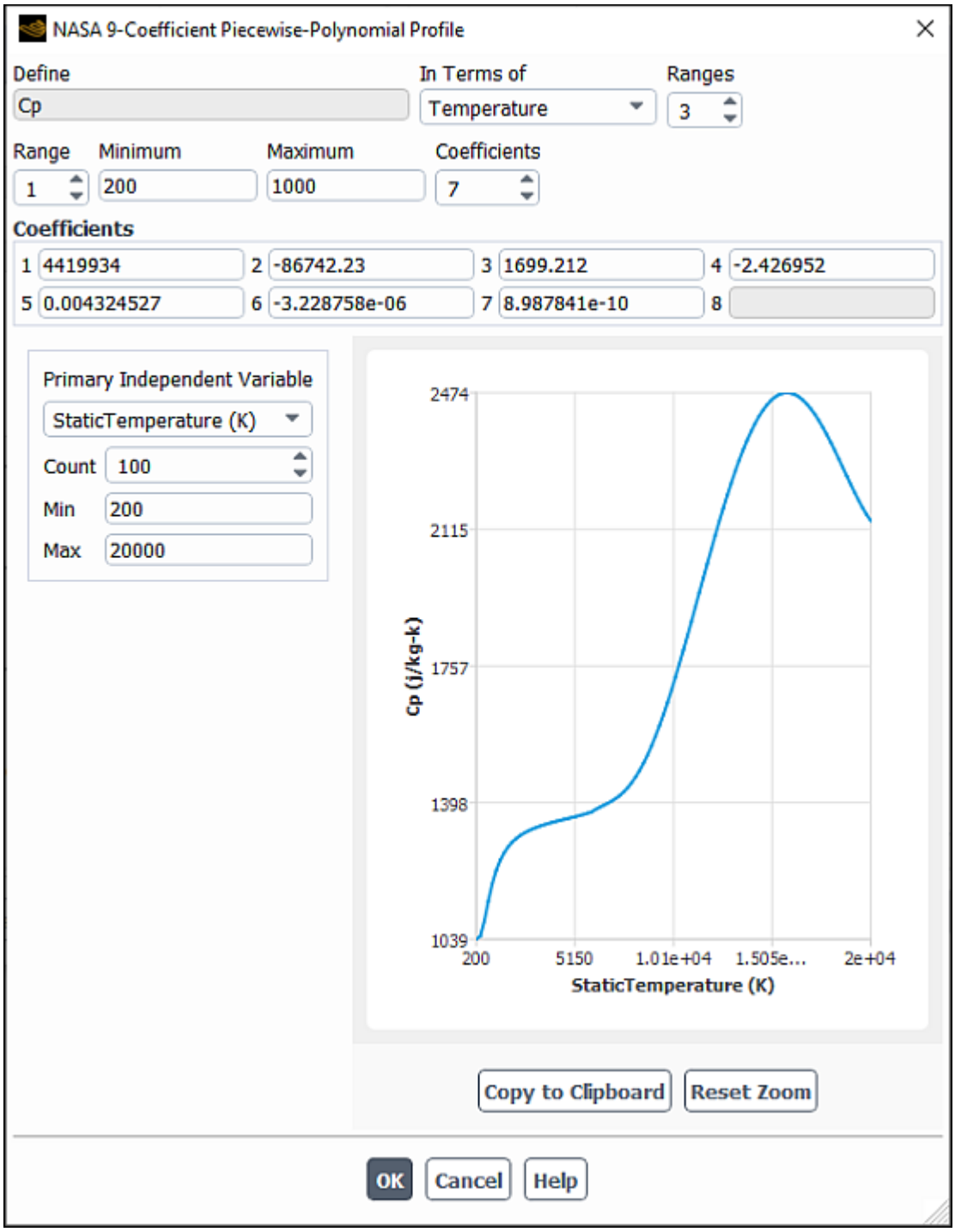
Reset Zoom

resets to the default zoom. *You can click and drag in the plot to change the zoom level.*

51.5.13. NASA-9-Coefficient Piecewise-Polynomial Profile Dialog Box

The **NASA-9-Coefficient Piecewise-Polynomial Profile** dialog box allows you to define specific heat as a piecewise-polynomial function of temperature. The profile can be used for cases with extremely high temperatures (such as hypersonic flows). This dialog box will open when you select **nasa-9-piecewise-polynomial** from the **Cp (Specific Heat)** drop-down list in the Create/Edit Materials Dialog Box (p. 4885). See Inputs for NASA-9-Piecewise-Polynomial Functions (p. 1622) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Define

shows the property (**Cp**) that is being defined as a function of temperature.

In Terms of

shows the independent variable (**Temperature**). The specific heat (**Cp**) will be defined as a function of temperature.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Ranges

sets the number of temperature ranges for which you will define polynomial functions. You can define up to 8 ranges.

Range

indicates the temperature range for which you are defining the polynomial function.

Minimum, Maximum

set the minimum and maximum temperatures for the specified **Range**.

Coefficients

is an integer number entry that indicates the number of coefficients to be defined for the specified **Range**. You can define up to 8 coefficients.

Coefficient Values

is a group box that contains eight real number entry boxes. You can specify values for coefficients **1**, **2**, **3**, and so on, that are used in Equation 8.4 (p. 1616). The number of editable boxes is the same as the integer number you entered in **Coefficients** for the specified **Range**.

Primary Independent Variable

is a drop-down box for selecting the independent variable. For the **nasa-9-piecewise-polynomial** profile, **Static Temperature** is the only available option.

Count

allows you to specify the number of plot points.

Min

sets the minimum value for the independent variable.

Max

sets the maximum value for the independent variable.

Copy to Clipboard

copies the current plot onto your clipboard.

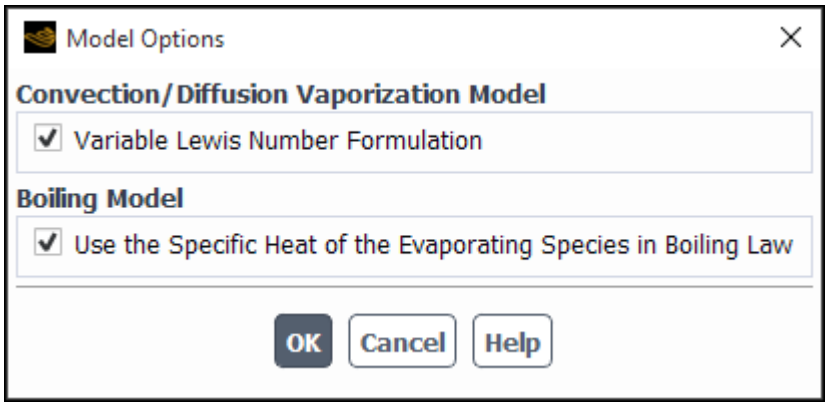
Reset Zoom

resets to the default zoom. *You can click and drag in the plot to change the zoom level.*

51.5.14. Model Options Dialog Box

The **Model Options** dialog box allows you to select boiling model options and to enable the calculation of the Spalding heat number for the convection/diffusion vaporization model. This dialog box opens when you click **Edit...** next to **Vaporization Model** in the Create/Edit Materials Dialog Box (p. 4885).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Convection/Diffusion Vaporization Model

allows you to enable **Variable Lewis Number Formulation** if you want to compute the Spalding heat number from Equation 12.96 in the *Fluent Theory Guide*. This group box is available only when you select **convection/diffusion-controlled** for **Vaporization/Boiling Model**.

Boiling Model

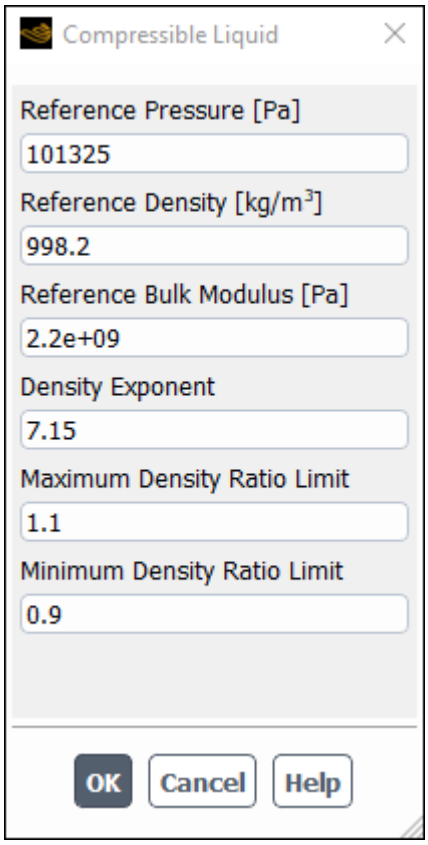
is where you can select the **Use the Specific Heat of the Evaporating Species in Boiling Law** option. This option sets the heat capacity of the gas ($c_{p,\infty}$ in Equation 12.101 in the *Fluent Theory Guide*) to the specific heat of the evaporating species selected in the **Set Injection Properties** dialog box. For the convection/diffusion controlled model with the **Variable Lewis Number Formulation**, this option is selected by default. This ensures consistency with the convection/diffusion controlled model expression for B_T in Equation 12.95 in the *Fluent Theory Guide*.

For more information, see Description of the Properties (p. 2817).

51.5.15. Compressible Liquid Dialog Box

The **Compressible Liquid** dialog box allows you to model liquid compressibility for high pressure applications using the Tait equation of state. This dialog box will open when you select **compressible-liquid** in the drop-down list next to one of the **Properties** in the Create/Edit Materials Dialog Box (p. 4885). See Compressible Liquid Density Method (p. 1626) for details about using the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Reference Pressure

sets the reference pressure.

Reference Density

sets the reference density.

Reference Bulk Modulus

sets the reference bulk modulus.

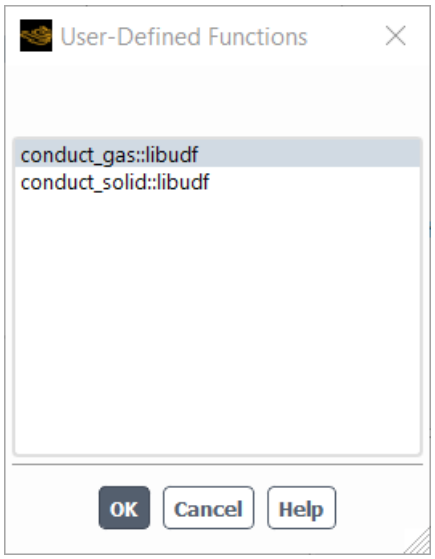
Density Exponent

sets the density exponent.

51.5.16. User-Defined Functions Dialog Box

The **User-Defined Functions** dialog box allows you to choose which user-defined function is to be used to define a material property. This dialog box will open when you select **user-defined** in the drop-down list next to one of the **Properties** in the Create/Edit Materials Dialog Box (p. 4885). See the separate Fluent Customization Manual for details about user-defined functions.

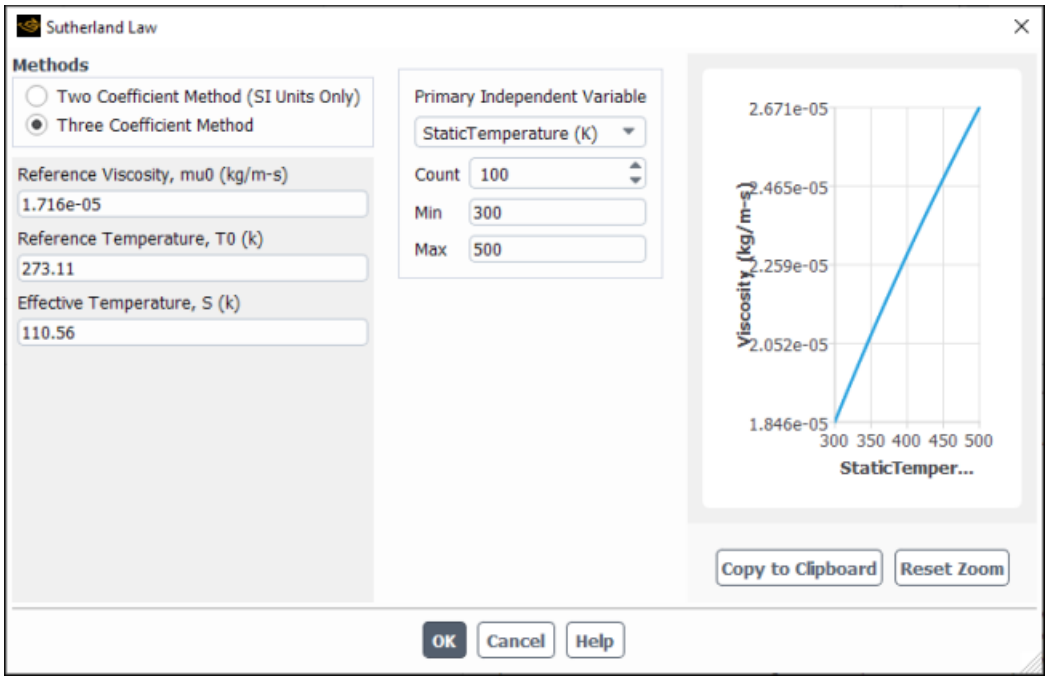
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



The list will contain all available user-defined functions.

51.5.17. Sutherland Law Dialog Box

The **Sutherland Law** dialog box allows you to set the coefficients for Sutherland's law for viscosity. This dialog box will open when you select **sutherland** in the drop-down list next to **Viscosity** in the Create/Edit Materials Dialog Box (p. 4885). See Sutherland Viscosity Law (p. 1636) for details about the items below.



Controls

Methods

contains options for selecting the **Two Coefficient Method** or the **Three Coefficient Method**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

C1, C2

set the coefficients C_1 and C_2 in Equation 8.24 (p. 1636) in SI units. These inputs will appear if you select the **Two Coefficient Method**.

Reference Viscosity

sets the reference viscosity μ_0 in Equation 8.25 (p. 1636). This input will appear if you select the **Three Coefficient Method**.

Reference Temperature

sets the reference temperature T_0 in Equation 8.25 (p. 1636). This input will appear if you select the **Three Coefficient Method**.

Effective Temperature

sets the effective temperature S in Equation 8.25 (p. 1636). This input will appear if you select the **Three Coefficient Method**.

Primary Independent Variable

allows you to select which variable is the independent variable.

Count

allows you to specify the number of plot points.

Min

sets the minimum value for the independent variable.

Max

sets the maximum value for the independent variable.

Copy to Clipboard

copies the current plot onto your clipboard.

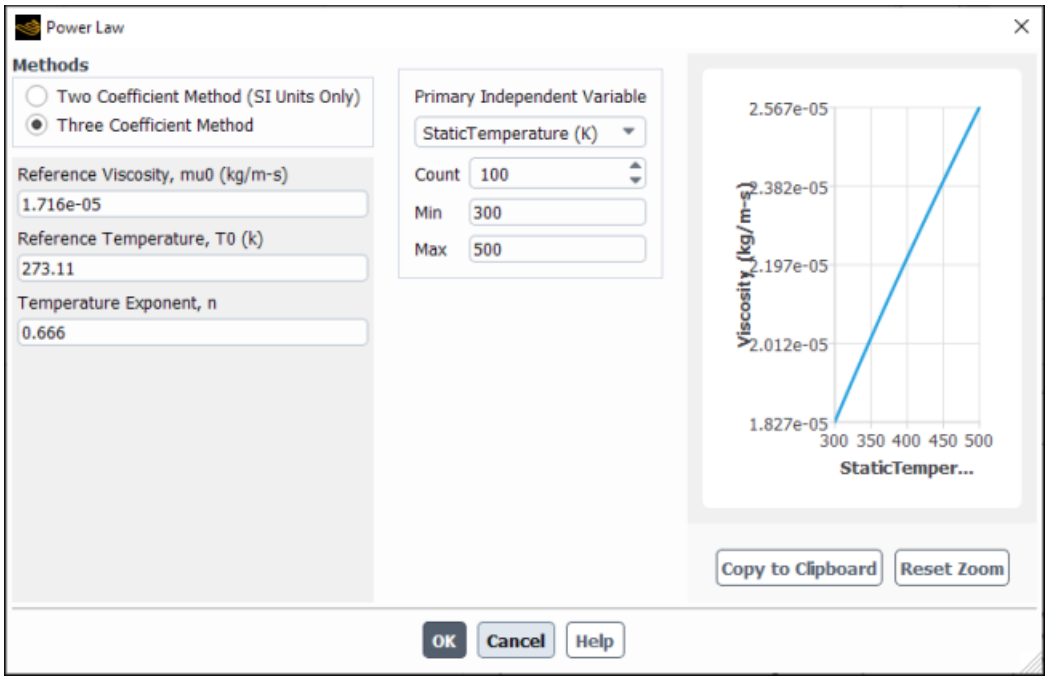
Reset Zoom

resets to the default zoom. *You can click and drag in the plot to change the zoom level.*

51.5.18. Power Law Dialog Box

The **Power Law** dialog box allows you to set the coefficients for the power law for viscosity. This dialog box will open when you select **power-law** in the drop-down list next to **Viscosity** in the Create/Edit Materials Dialog Box (p. 4885). See Power-Law Viscosity Law (p. 1637) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Methods

contains options for selecting the **Two Coefficient Method** or the **Three Coefficient Method**.

B

sets the coefficient B in Equation 8.26 (p. 1637) in SI units. This input will appear if you select the **Two Coefficient Method**.

Reference Viscosity

sets the reference viscosity μ_0 in Equation 8.27 (p. 1637). This input will appear if you select the **Three Coefficient Method**.

Reference Temperature

sets the reference temperature T_0 in Equation 8.27 (p. 1637). This input will appear if you select the **Three Coefficient Method**.

Temperature Exponent

sets the temperature exponent n in Equation 8.26 (p. 1637) or Equation 8.27 (p. 1637), depending on your **Method** selection. If you are using the **Two Coefficient Method**, this input must be in SI units.

Primary Independent Variable

allows you to select which variable is the independent variable.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Count

allows you to specify the number of plot points.

Min

sets the minimum value for the independent variable.

Max

sets the maximum value for the independent variable.

Copy to Clipboard

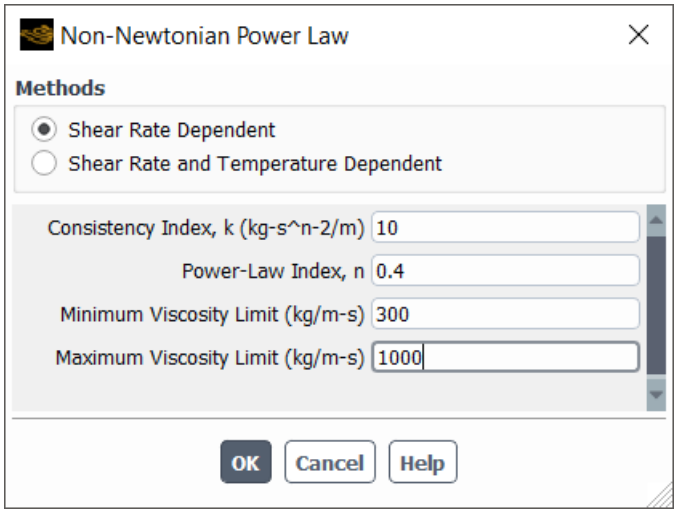
copies the current plot onto your clipboard.

Reset Zoom

resets to the default zoom. *You can click and drag in the plot to change the zoom level.*

51.5.19. Non-Newtonian Power Law Dialog Box

The **Non-Newtonian Power Law** dialog box allows you to set the parameters for the non-Newtonian power law for viscosity. This dialog box will open when you select **non-newtonian-power-law** in the drop-down list next to **Viscosity** in the Create/Edit Materials Dialog Box (p. 4885). See Power Law for Non-Newtonian Viscosity (p. 1641) for details about the items below.



Controls

Methods

allows you to select the type of dependency on the viscosity.

Shear Rate Dependent

is where the viscosity is dependent on the shear rate.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Shear Rate and Temperature Dependent

is where the viscosity is dependent on the shear rate and the temperature.

Consistency Index

sets the consistency index k in Equation 8.39 (p. 1641).

Power-Law Index

sets the power-law index n in Equation 8.39 (p. 1641).

Minimum Viscosity Limit, Maximum Viscosity Limit

set the minimum and maximum viscosity limits.

Reference Temperature

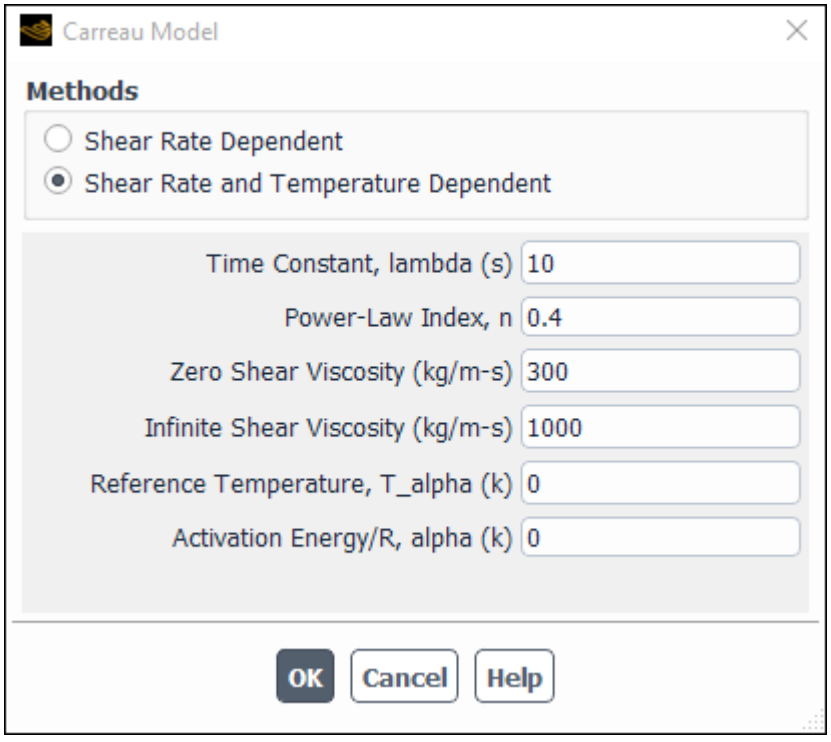
sets the reference temperature.

Activation Energy/R, alpha

is the ratio of the activation energy to the thermodynamic constant α in Equation 8.38 (p. 1640).

51.5.20. Carreau Model Dialog Box

The **Carreau Model** dialog box allows you to set the parameters for the non-Newtonian Carreau model for viscosity. This dialog box will open when you select **carreau** in the drop-down list next to **Viscosity** in the Create/Edit Materials Dialog Box (p. 4885). See The Carreau Model for Pseudo-Plastics (p. 1641) for details about the items below.



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Controls

Methods

allows you to select the type of dependency on the viscosity.

Shear Rate Dependent

is where the viscosity is dependent on the shear rate.

Shear Rate and Temperature Dependent

is where the viscosity is dependent on the shear rate and the temperature.

Time Constant, lambda

sets the time constant λ in Equation 8.40 (p. 1641).

Power-Law Index

sets the power-law index n in Equation 8.40 (p. 1641).

Zero Shear Viscosity, Infinite Shear Viscosity

set the zero and infinite shear viscosity limits η_0 and η_∞ in Equation 8.40 (p. 1641).

Reference Temperature, T_alpha

sets the reference temperature T_0 in Equation 8.40 (p. 1641).

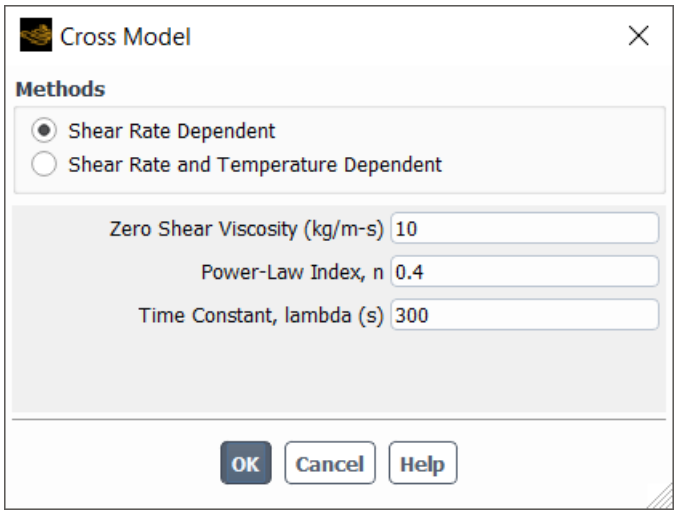
Activation Energy/R, alpha

is the ratio of the activation energy to the thermodynamic constant α in Equation 8.38 (p. 1640).

51.5.21. Cross Model Dialog Box

The **Cross Model** dialog box allows you to set the parameters for the non-Newtonian Cross model for viscosity. This dialog box will open when you select **cross** in the drop-down list next to **Viscosity** in the Create/Edit Materials Dialog Box (p. 4885). See Cross Model (p. 1642) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Methods

allows you to select the type of dependency on the viscosity.

Shear Rate Dependent

is where the viscosity is dependent on the shear rate.

Shear Rate and Temperature Dependent

is where the viscosity is dependent on the shear rate and the temperature.

Zero Shear Viscosity

sets the zero shear viscosity limit η_0 in Equation 8.41 (p. 1642).

Power-Law Index

sets the power-law index n in Equation 8.41 (p. 1642).

Time Constant

sets the time constant λ in Equation 8.41 (p. 1642).

Reference Temperature, T_alpha

sets the reference temperature T_0 in Equation 8.40 (p. 1641).

Activation Energy/R, alpha

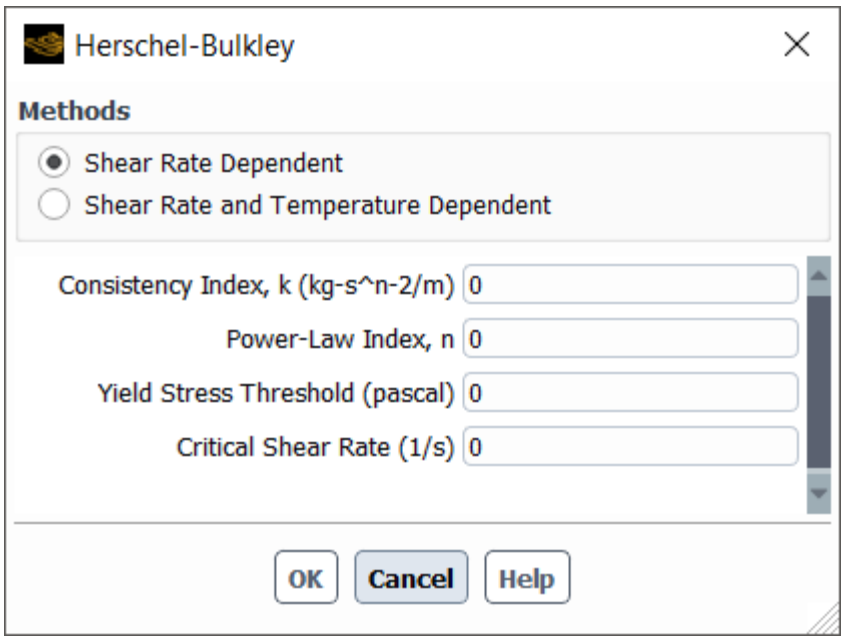
is the ratio of the activation energy to the thermodynamic constant α in Equation 8.38 (p. 1640).

51.5.22. Herschel-Bulkley Dialog Box

The **Herschel-Bulkley** dialog box allows you to set the parameters for the non-Newtonian Herschel-Bulkley model for viscosity. This dialog box will open when you select **herschel-bulkley** in the drop-

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

down list next to **Viscosity** in the Create/Edit Materials Dialog Box (p. 4885). See Herschel-Bulkley Model for Bingham Plastics (p. 1643) for details about the items below.



Controls

Methods

allows you to select the type of dependency on the viscosity.

Shear Rate Dependent

is where the viscosity is dependent on the shear rate.

Shear Rate and Temperature Dependent

is where the viscosity is dependent on the shear rate and the temperature.

Consistency Index

sets the consistency index k in Equation 8.43 (p. 1643).

Power-Law Index

sets the power-law index n in Equation 8.43 (p. 1643).

Yield Stress Threshold

sets the yield stress threshold τ_0 in Equation 8.43 (p. 1643).

Critical Shear Rate

set the critical shear rate $\dot{\gamma}_c$ in Equation 8.43 (p. 1643).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reference Temperature, T_alpha

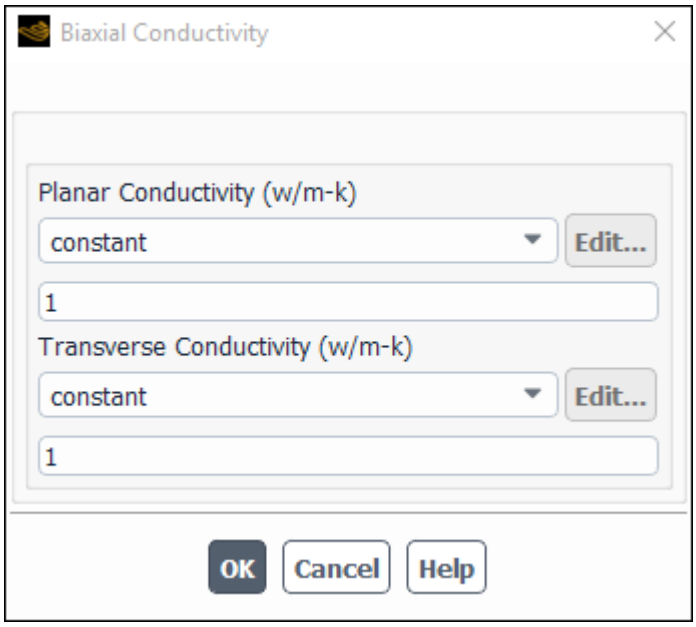
sets the reference temperature T_α in Equation 8.38 (p. 1640).

Activation Energy/R, alpha

is the ratio of the activation energy to the thermodynamic constant α in Equation 8.38 (p. 1640).

51.5.23. Biaxial Conductivity Dialog Box

The **Biaxial Conductivity** dialog box allows you to define a biaxial orthotropic thermal conductivity, which is applicable to solid materials used for the wall shell conduction model. This dialog box will open when you select **biaxial** in the drop-down list next to **Thermal Conductivity** in the Create/Edit Materials Dialog Box (p. 4885). See Biaxial Thermal Conductivity (p. 1650) for details about the items below.



Controls

Planar Conductivity

specifies the conductivity within the shell (or solid) region.

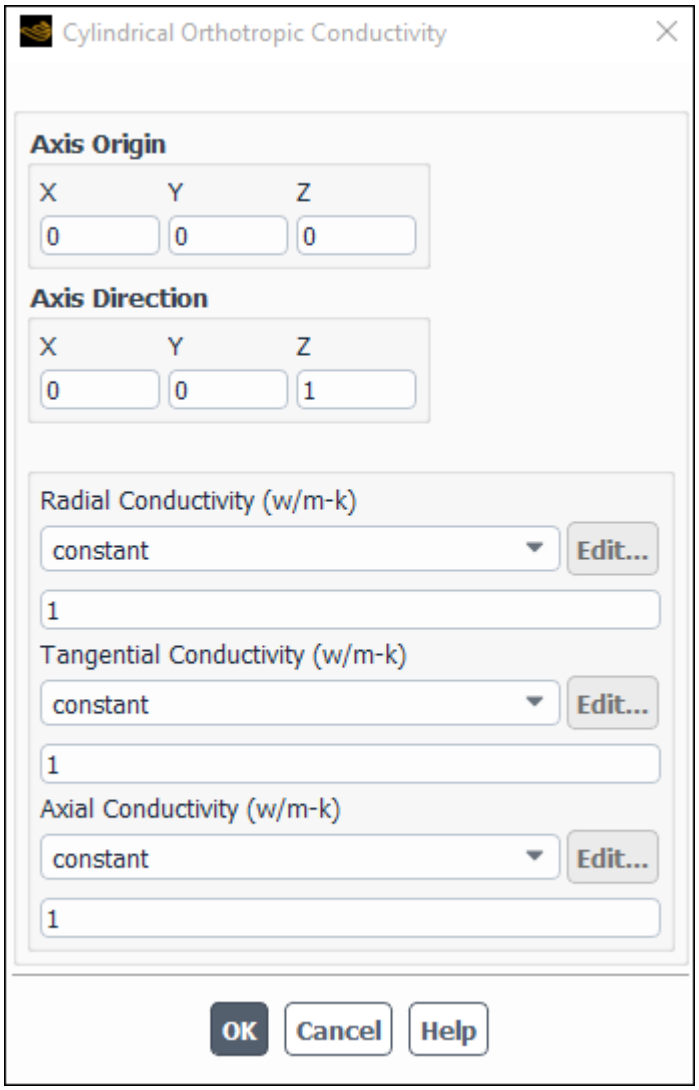
Transverse Conductivity

specifies the conductivity normal to the surface of the solid region.

51.5.24. Cylindrical Orthotropic Conductivity Dialog Box

The **Cylindrical Orthotropic Conductivity** dialog box allows you to define an orthotropic thermal conductivity in cylindrical coordinates. This dialog box will open when you select **cyl-orthotropic** in the drop-down list next to **Thermal Conductivity** in the Create/Edit Materials Dialog Box (p. 4885). See Cylindrical Orthotropic Thermal Conductivity (p. 1652) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Axis Origin

allows you to specify the origin of the cylindrical coordinate system.

X, Y, Z

specify the X, Y, and (for 3D cases) Z coordinates.

Axis Direction

(3D only) allows you to specify the direction of the axis.

X, Y, Z

specify 1 against the direction of the axis.

Radial Conductivity

specifies the conductivity in the radial direction.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tangential Conductivity

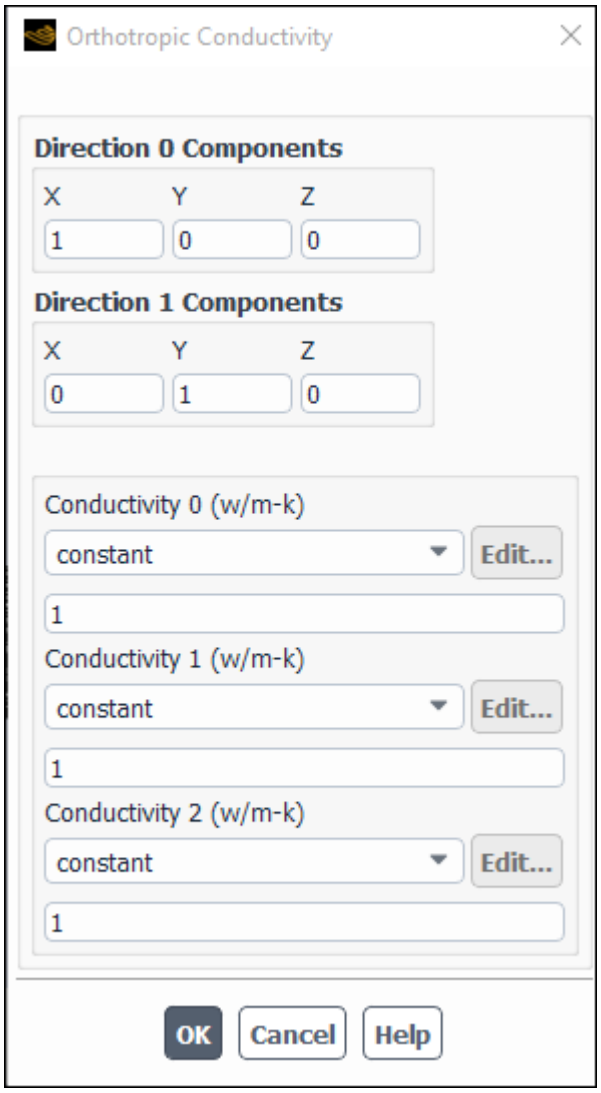
specifies the conductivity in the tangential direction.

Axial Conductivity

(3D only) specifies the conductivity in the axial direction.

51.5.25. Orthotropic Conductivity Dialog Box

The **Orthotropic Conductivity** dialog box allows you to define an orthotropic thermal conductivity for a solid material, using principal directions that do not have to aligned with the global coordinate system of the simulation. This dialog box will open when you select **orthotropic** in the drop-down list next to **Thermal Conductivity** in the Create/Edit Materials Dialog Box (p. 4885). See Orthotropic Thermal Conductivity (p. 1650) for details about the items below.



Controls

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Direction 0 Components, Direction 1 Components

specify the directions $\hat{e}_\xi$ and $\hat{e}_\eta$ in Equation 8.54 (p. 1650) as **X,Y,Z** vectors. For 2D cases, only **Direction 0 Components** will appear.

Conductivity 0, Conductivity 1, Conductivity 2

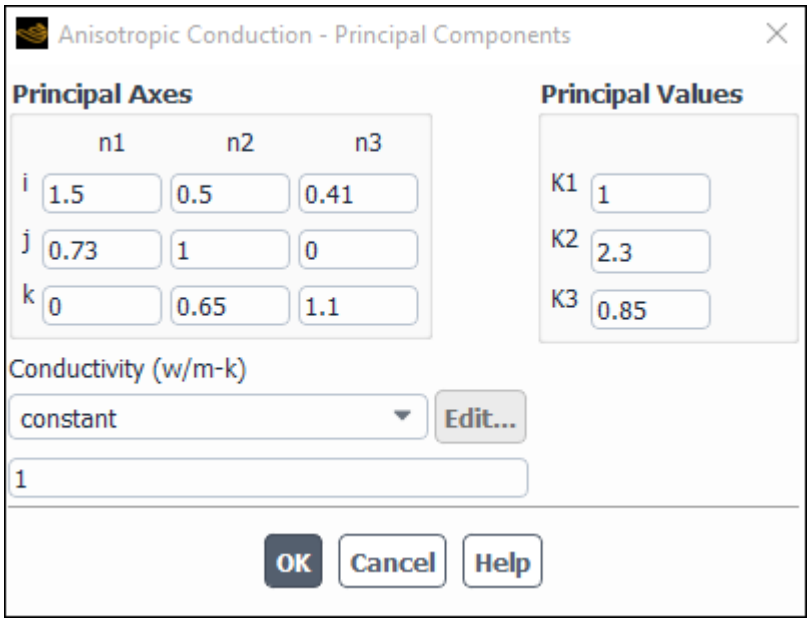
specify (k_ξ), (k_η), and (k_ζ) in Equation 8.54 (p. 1650) as **constant**, **polynomial**, **piecewise-linear**, **piecewise-polynomial**, functions of temperature, **user-defined**, or **expression**. For 2D cases, only **Conductivity 0** and **Conductivity 1** will appear.

Edit...

opens the appropriate dialog box so you can enter a temperature-dependent conductivity. (This button will be unavailable if you specify a **constant** conductivity.)

51.5.26. Anisotropic Conduction - Principal Components Dialog Box

The **Anisotropic Conduction - Principal Components** dialog box allows you to define an anisotropic thermal conductivity using the components of the principal axes (which do not have to be orthogonal and/or aligned with the global coordinate system) and the associated principal values. This dialog box will open when you select **principal-axes-values** in the drop-down list next to **Thermal Conductivity** in the Create/Edit Materials Dialog Box (p. 4885). See Principal Axes and Principal Values (p. 1654) for details about the items that follow.



Controls

Principal Axes

specifies the $\hat{i}$, $\hat{j}$, and $\hat{k}$ components of the $\vec{n}^1$, $\vec{n}^2$, and $\vec{n}^3$ vectors (in Equation 8.55 (p. 1654)) that define the principal axes of the anisotropic material. Note that these vectors must be linearly independent.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Principal Values

specifies the principal values (K_1, K_2, and K_3 in Equation 8.58 (p. 1655)) associated with the principal axes.

Conductivity

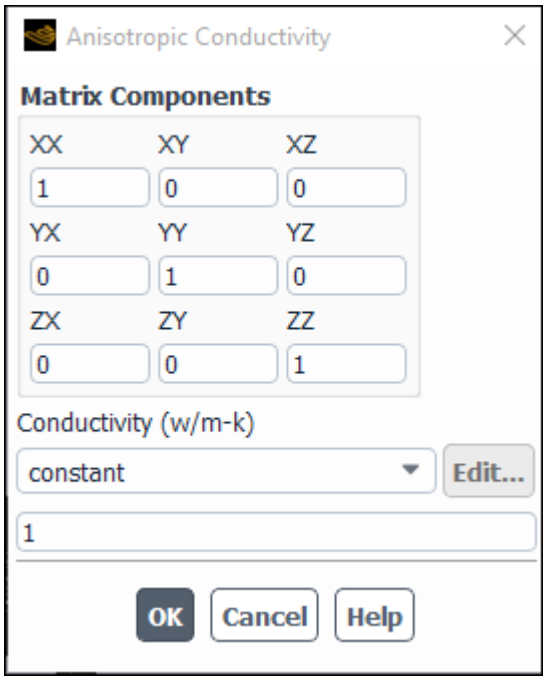
specifies the value of k (in Equation 8.58 (p. 1655)) as a **constant**, function of temperature (**polynomial**, **piecewise-linear**, **piecewise-polynomial**), or **user-defined** function.

Edit...

opens the appropriate dialog box for the definition of a temperature-dependent or user-defined conductivity. (This button will be unavailable if you specify a **constant** conductivity.)

51.5.27. Anisotropic Conductivity Dialog Box

The **Anisotropic Conductivity** dialog box allows you to define a general anisotropic thermal conductivity. This dialog box will open when you select **anisotropic** in the drop-down list next to **Thermal Conductivity** in the Create/Edit Materials Dialog Box (p. 4885). See Anisotropic Thermal Conductivity (p. 1648) for details about the items below.



Controls

Matrix Components

specify the components of the matrix $\hat{e}_{ij}$ in Equation 8.53 (p. 1649).

Conductivity

specifies the value of k in Equation 8.53 (p. 1649) as a **constant**, function of temperature (**polynomial**, **piecewise-linear**, **piecewise-polynomial**), or **user-defined** function.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

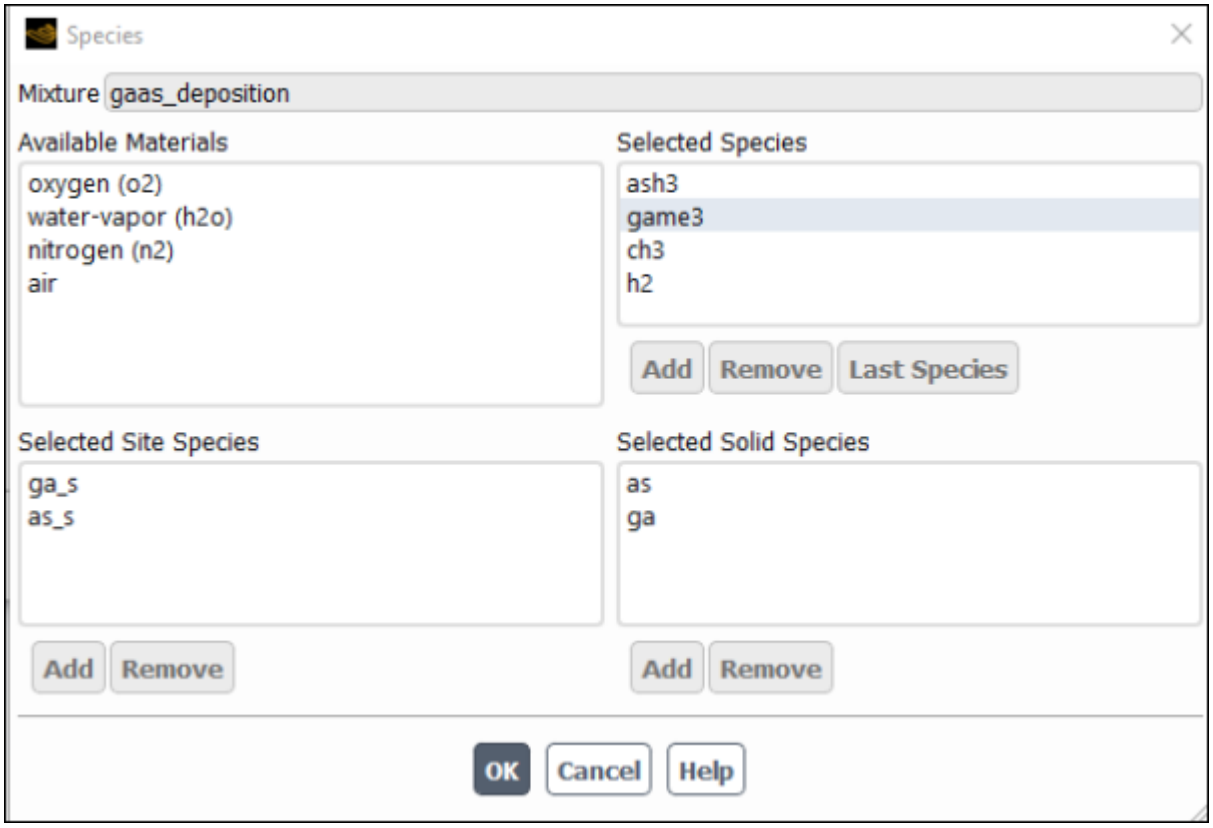
Edit...

opens the appropriate dialog box so you can enter a temperature-dependent or user-defined conductivity. (This button will be unavailable if you specify a **constant** conductivity.)

51.5.28. Species Dialog Box

The **Species** dialog box (opened by clicking the **Edit...** button next to **Mixture Species** in the Create/Edit Materials Dialog Box (p. 4885))) allows you to define the species that make up a mixture material. See Defining the Species in the Mixture (p. 2375) for details about the items below.

(Note that the **Species** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.)



Controls

Mixture

shows the name of the mixture material for which you are defining the species. This field is not editable.

Available Materials

is a list of all of the materials to be a component in the mixture material by selecting it and clicking the **Add** button below the **Selected Species** or **Selected Surface Species** list. To add a material to the **Available Materials** list, use the Fluent Database Materials Dialog Box (p. 4897) to copy the fluid material to local storage.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Selected Species

is a list of all the fluid-phase species in the mixture. You can modify the list as follows:

- To add a material to the list, select it in the **Available Materials** list and click the **Add** button below the **Selected Species** list.
- To remove a material, select it in the **Selected Species** list and click **Remove**.
- To move a bulk species to the bottom of the **Selected Species** list, select the species and click **Last Species**. The transport equation will not be solved for the last species.

See Overview of the Species Dialog Box (p. 2376) for more information.

Selected Solid Species

is a list of all the solid species in the mixture. To add a material to the list, select it in the **Available Materials** list and click the **Add** button below the **Selected Solid Species** list. To remove a material, select it in the **Selected Solid Species** list and click **Remove**. See Overview of the Species Dialog Box (p. 2376) for more information.

Selected Site Species

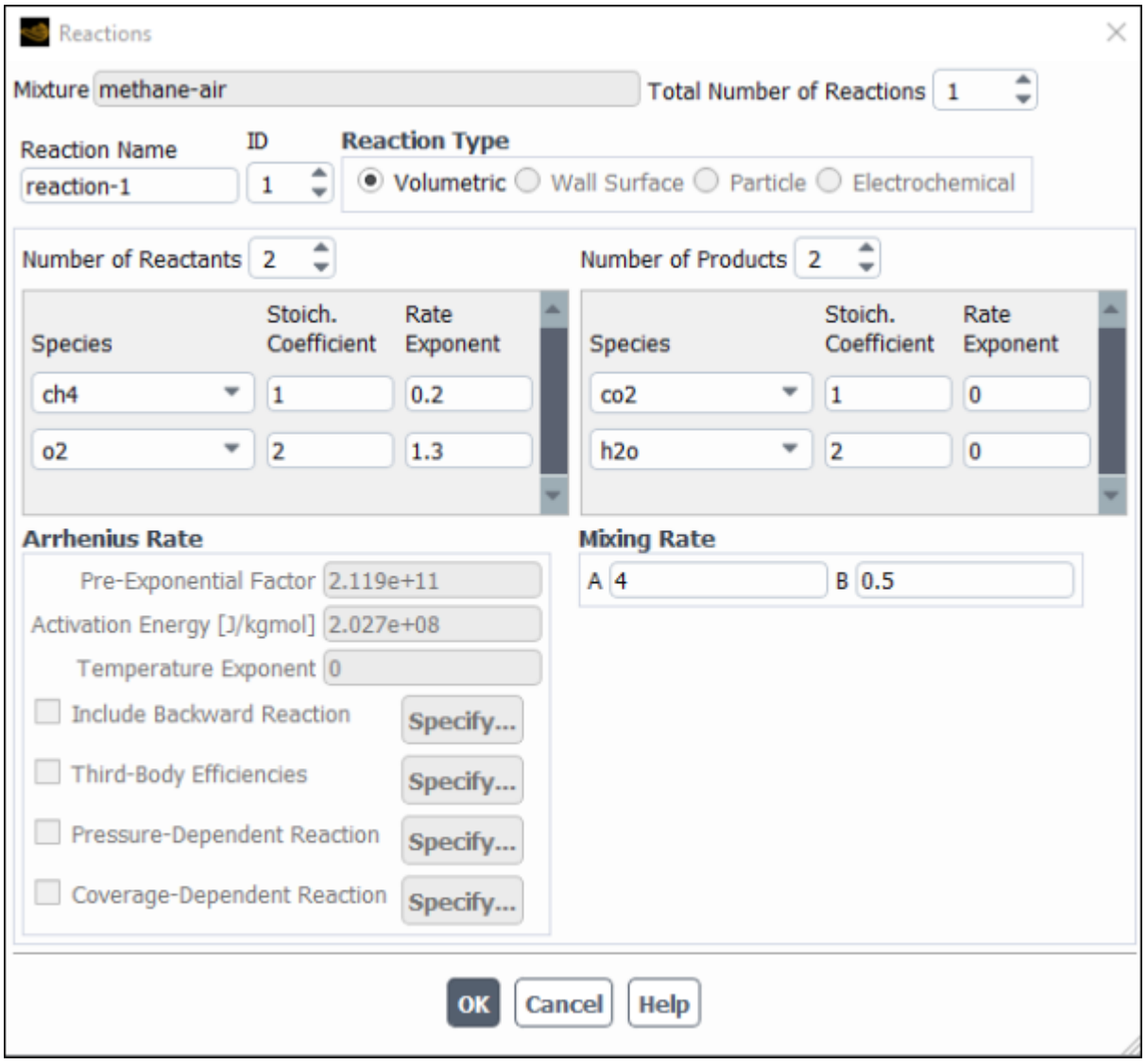
is a list of all the site species in the mixture. To add a material to the list, select it in the **Available Materials** list and click the **Add** button below the **Selected Site Species** list. To remove a material, select it in the **Selected Site Species** list and click **Remove**. See Overview of the Species Dialog Box (p. 2376) for more information.

51.5.29. Reactions Dialog Box

The **Reactions** dialog box (opened by clicking the **Edit...** button next to **Reaction** in the Create/Edit Materials Dialog Box (p. 4885)) allows you to define the reactions for a mixture material. See Defining Reactions (p. 2379) for details about using this dialog box.

(Note that the **Reactions** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Mixture

shows the name of the mixture material for which you are defining the species. This field is not editable.

Total Number of Reactions

sets the total number of reactions (fluid-phase reactions and surface reactions occurring at wall boundaries). Use the arrows to change the value, or type in the value and press RETURN.

Reaction Name

contains the name of the reaction.

ID

sets the number of the reaction you want to define. (Again, if you type in the value be sure to press RETURN.)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reaction Type

contains options that allow you to specify the type of reaction.

Volumetric

indicates that the current reaction is a volumetric reaction.

Wall Surface

indicates that the current reaction is a wall surface reaction.

Particle

indicates that the current reaction is a particle reaction. This item is available only when **Particle** is selected as the **Reaction Type** in the **Species Model** dialog box.

Electrochemical

indicates that the current reaction is an electrochemical reaction. This item is available only when **Electrochemical** is selected as the **Reaction Type** in the **Species Model** dialog box.

Number of Reactants

specifies the number of reactants in the specified reaction.

Species

contains drop-down lists of all species in the mixture. (The number of lists will be equal to the **Number of Reactants**.) Select each reactant in one of these lists.

Stoich. Coefficient

specifies the stoichiometric coefficient of the reactant species in the reaction.

Rate Exponent

specifies the rate constant for the reactant species in the reaction.

Number of Products

indicates the number of products in the specified reaction.

Species

contains drop-down lists of all species in the mixture. (The number of lists will be equal to the **Number of Products**.) Select each product in one of these lists.

Stoich. Coefficient

specifies the stoichiometric coefficient of the product species in the reaction.

Rate Exponent

specifies the rate constant for the product species in the reaction.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Arrhenius Rate

contains inputs related to the Arrhenius rate. (If you have chosen **Eddy-Dissipation** for the **Turbulence-Chemistry Interaction** in the Species Model Dialog Box (p. 4768), these inputs are not required.)

Pre-exponential Factor

is the constant A_r in Equation 7.23 in the Theory Guide. The units of A_r must be specified such that the units of the molar reaction rate, $R_{i,r}$ (Equation 7.19 in the Theory Guide, are moles/volume-time (for example, kmol/m^3-s) and the units of the volumetric reaction rate, R_i in Equation 7.19 in the Theory Guide, are mass/volume-time (for example, kg/m^3-s).

Important:

It is important to note that if you have selected the British units system, the Arrhenius factor should still be specified in SI units. This is because Ansys Fluent applies no conversion factor to the value you enter for A_r (the conversion factor is 1.0) when you work in British units, as the correct conversion factor depends on your values for $\nu_{j,r}$, β_r, and so on.

Activation Energy

is the constant E_r in the forward rate constant expression, Equation 7.23 in the Theory Guide).

Temperature Exponent

is the value for the constant β_r in Equation 7.23 in the Theory Guide.

Include Backward Reaction

specifies that the reaction is reversible. By default, the backward reaction rate constant will be computed from Equation 7.23 in the *Fluent Theory Guide*. You can specify your own backward reaction rate parameters in the Backward Reaction Parameters Dialog Box (p. 4936). In this case, the backward reaction rate constant will be computed from Equation 7.28 in the *Fluent Theory Guide*.

Third-Body Efficiencies

allows you to specify and use third-body efficiencies ($\gamma_{j,r}$ in Equation 7.22 in the Theory Guide). These inputs are optional. (This item is available only if you have selected **Volumetric** for the **Reaction Type**.)

Pressure-Dependent Reaction

enables the modeling of a pressure fall-off reaction. See Inputs for Reaction Definition (p. 2379) **Finite-Rate/No TCI** or **Eddy-Dissipation Concept** for the **Turbulence-Chemistry Interaction** in the Species Model Dialog Box (p. 4768) and have selected **Volumetric** for the **Reaction Type**.)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Coverage-Dependent Reaction

is used when modeling **Wall Surface** reactions with site-balancing and reaction rates depend on site coverages.

Specify...

opens the Backward Reaction Parameters Dialog Box (p. 4936), Third-Body Efficiency Dialog Box (p. 4936), the Pressure-Dependent Reaction Dialog Box (p. 4937), or Coverage-Dependent Reaction Dialog Box (p. 4940) in which you can specify the backward rate parameters for the reversible reaction, third-body efficiencies, pressure-dependent reaction parameters, or coverage parameters.

Mixing Rate

contains inputs related to the mixing rate. (If you have chosen **Finite-Rate/No TCI** or **Eddy-Dissipation Concept** for the **Turbulence-Chemistry Interaction** in the Species Model Dialog Box (p. 4768), these inputs are not required.)

A

is the constant A in the turbulent mixing rate (Equation 7.39 and Equation 7.40 in the Theory Guide) when it is applied to a species that appears as a reactant in this reaction. The default setting of 4.0 is based on the empirically derived values given by Magnussen and Hjertager [94] (p. 5718).

B

is the constant B in the turbulent mixing rate (Equation 7.40 in the Theory Guide) when it is applied to a species that appears as a product in this reaction. The default setting of 0.5 is based on the empirically derived values given by Magnussen and Hjertager [94] (p. 5718).

Particle Reaction

contains inputs related either to a particle surface reaction or to a multicomponent particle reaction for reacting multicomponent particles. For details, see the following sections:

- User Inputs for Particle Surface Reactions (p. 2407)
- Multicomponent Particles with Chemical Reactions (p. 2409)

This group box appears only if you have selected **Particle** for the **Reaction Type**.

Model

(reacting multicomponent particles only) allows you to select between the **kinetics/diffusion**, **surface-kinetics**, **volume-kinetics**, **phase-change**, **thermolysis**, and **user-defined** particle reaction model. See Multicomponent Particles with Chemical Reactions (p. 2409) for details.

Diffusion-Limited Species

is a drop-down list that allows you to select the species for which the concentration gradient between the bulk and the particle surface is the largest when there is more than one gaseous

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

reactant taking part in the particle surface reaction. See User Inputs for Particle Surface Reactions (p. 2407) for details.

Diffusion Rate Constant

is the constant $C_{1,r}$ in Equation 7.104 in the Theory Guide.

Effectiveness Factor

is the constant η_r in Equation 7.102 in the Theory Guide.

Butler-Volmer/Tafel Parameters

contains inputs related to an electrochemical reaction. See User Inputs for Electrochemical Reactions (p. 2414) for details. (This group box appears only if **Electrochemical** is selected for **Reaction Type**.)

Tafel Parameters

indicates whether the Bulter-Volmer equation (Equation 7.118 in the *Fluent Theory Guide*) or the Tafel equation (Equation 7.119 in the *Fluent Theory Guide*) is used in the kinetics calculation.

Anodic Transfer Coefficient

is the value of α_a used in Equation 7.118 in the *Fluent Theory Guide*. This item appears if the **Tafel Parameters** check box is cleared.

Cathodic Transfer Coefficient

is the value of α_c used in Equation 7.118 in the *Fluent Theory Guide*. This item appears if the **Tafel Parameters** check box is cleared.

Anodic Tafel Slope

is the value of $\beta_{a,r}$ in Equation 7.119 in the *Fluent Theory Guide*. This item appears only if the **Tafel Parameters** check box is selected.

Cathodic Tafel Slope

is the value of $\beta_{c,r}$ in Equation 7.119 in the *Fluent Theory Guide*. This item appears only if the **Tafel Parameters** check box is selected.

Exchange Current Density

is the value of $i_{o,r}$ used in Equation 7.118 and Equation 7.119 in the *Fluent Theory Guide*.

Reference Temperature

is a value of T_{ref} used in Equation 7.118 and Equation 7.119 in the *Fluent Theory Guide*.

Activation Energy

is a value of E used in Equation 7.118 and Equation 7.119 in the *Fluent Theory Guide*.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Equilibrium Potential

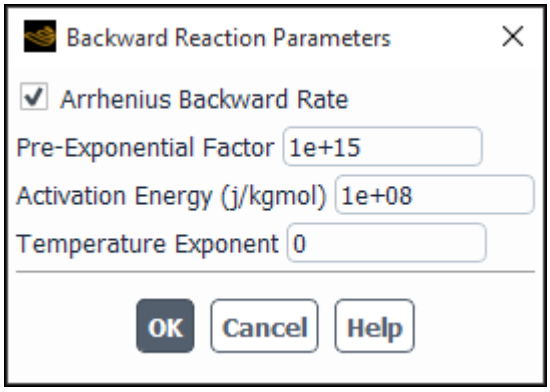
is the value of $E_{eq,r}$ used in Equation 7.120 in the *Fluent Theory Guide*

Specify Reference Molar Concentrations...

opens the Reference Molar Concentrations Dialog Box (p. 4941).

51.5.30. Backward Reaction Parameters Dialog Box

The **Backward Reaction Parameters** dialog box (opened by clicking **Specify...** next to **Include Backward Reaction** in the Reactions Dialog Box (p. 4930)) allows you to specify your custom backward rate parameters for a reversible reaction. See Inputs for Reaction Definition (p. 2379) for details.



Arrhenius Backward Rate

allows you to specify backward rate parameters.

Pre-Exponential Factor

is the constant $A_{b,r}$ in Equation 7.28 in the *Fluent Theory Guide*.

Activation Energy

is the constant $E_{b,r}$ in the backward rate constant expression in Equation 7.28 in the *Fluent Theory Guide*.

Temperature Exponent

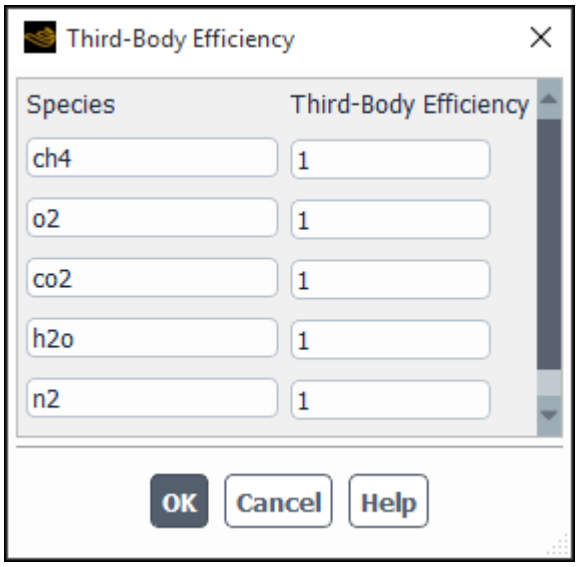
is the constant $\beta_{b,r}$ in Equation 7.28 in the *Fluent Theory Guide*.

51.5.31. Third-Body Efficiency Dialog Box

The **Third-Body Efficiency** dialog box (opened by clicking **Specify...** next to the **Third-Body Efficiencies** button in the Reactions Dialog Box (p. 4930)) allows you to specify the third-body efficiencies for each species in the mixture, to be used in Equation 7.22 in the Theory Guide. See Defining Reactions (p. 2379) for details.

(Note that the **Third-Body Efficiency** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Species

displays the name of each species in the mixture.

Third-body Efficiency

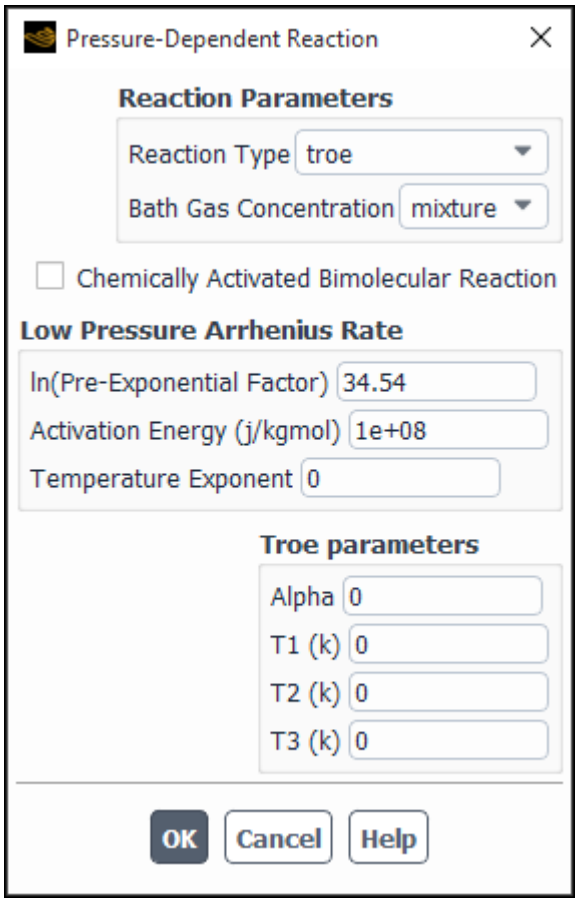
specifies the third-body efficiency for each species.

51.5.32. Pressure-Dependent Reaction Dialog Box

The **Pressure-Dependent Reaction** dialog box (opened by clicking **Specify...** under **Pressure-Dependent Reaction** in the Reactions Dialog Box (p. 4930)) allows you to specify parameters for a pressure fall-off reaction. See Inputs for Reaction Definition (p. 2379) for details.

(Note that the **Pressure-Dependent Reaction** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Reaction Parameters

contains inputs for specifying the type of pressure fall-off reaction and the reaction parameters. See Pressure-Dependent Reactions in the Theory Guide for details.

Reaction Type

contains a drop-down list of the available reaction types: **lindemann**, **troe**, and **sri**. See Pressure-Dependent Reactions in the Theory Guide for details.

Bath Gas Concentration

allows you to specify if the bath gas concentration ($[M]$ in Equation 7.32 in the Theory Guide) is to be defined as the concentration of the **mixture**, or as the concentration of one of the mixture's constituent species.

Chemically Activated Bimolecular Reaction

results in a net rate constant at any pressure being defined as Equation 7.38 in the Theory Guide.

Low Pressure Arrhenius Rate

contains inputs for specifying low-pressure Arrhenius parameters.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

ln(Pre-exponential Factor)

is the natural logarithm of the constant A_{low} in Equation 7.30 in the Theory Guide. The pre-exponential factor A_{low} is often an extremely large number, so you will enter the natural logarithm of this term.

Activation Energy

is the constant E_{low} in Equation 7.30 in the Theory Guide.

Temperature Exponent

is the constant β_{low} in Equation 7.30 in the Theory Guide.

Troe parameters

contains inputs for specifying parameters for the Troe method. See Pressure-Dependent Reactions in the Theory Guide for details. (This section of the dialog box will appear only if you have selected **troe** as the **Reaction Type**.)

Alpha

is the constant α in Equation 7.35 in the Theory Guide.

T1

is the constant T_1 in Equation 7.35 in the Theory Guide.

T2

is the constant T_2 in Equation 7.35 in the Theory Guide.

T3

is the constant T_3 in Equation 7.35 in the Theory Guide.

SRI Parameters

contains inputs for specifying parameters for the SRI method. See Pressure-Dependent Reactions in the Theory Guide for details. (This section of the dialog box will appear only if you have selected **sri** as the **Reaction Type**.)

a

is the constant a in Equation 7.36 in the Theory Guide.

b

is the constant b in Equation 7.36 in the Theory Guide.

c

is the constant c in Equation 7.36 in the Theory Guide.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

d

is the constant d in Equation 7.36 in the Theory Guide.

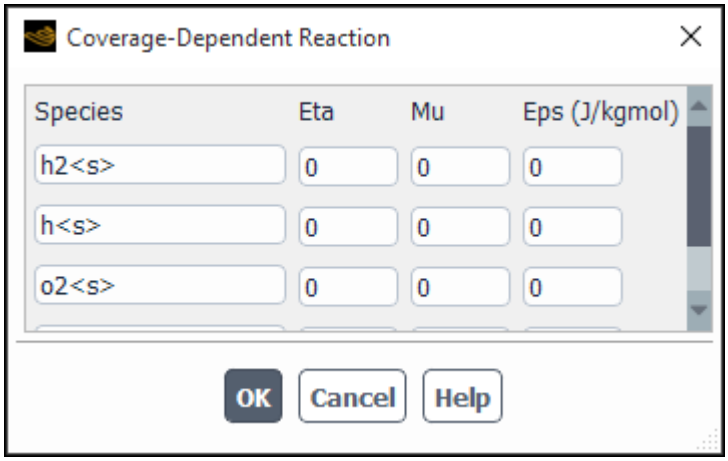
e

is the constant e in Equation 7.36 in the Theory Guide.

51.5.33. Coverage-Dependent Reaction Dialog Box

The **Coverage-Dependent Reaction** dialog box (opened by clicking on **Specify...** under **Coverage-Dependent Reaction** in the Reactions Dialog Box (p. 4930)) allows you to model **Wall Surface** reactions with site-balancing.

(Note that the **Coverage-Dependent Reaction** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.)



Controls

Species

are the site species of the reaction.

Eta

is the surface coverage rate modification of the species and is defined in Equation 7.81 in the Theory Guide.

Mu

is the surface coverage rate modification of the species and is defined in Equation 7.81 in the Theory Guide.

Eps

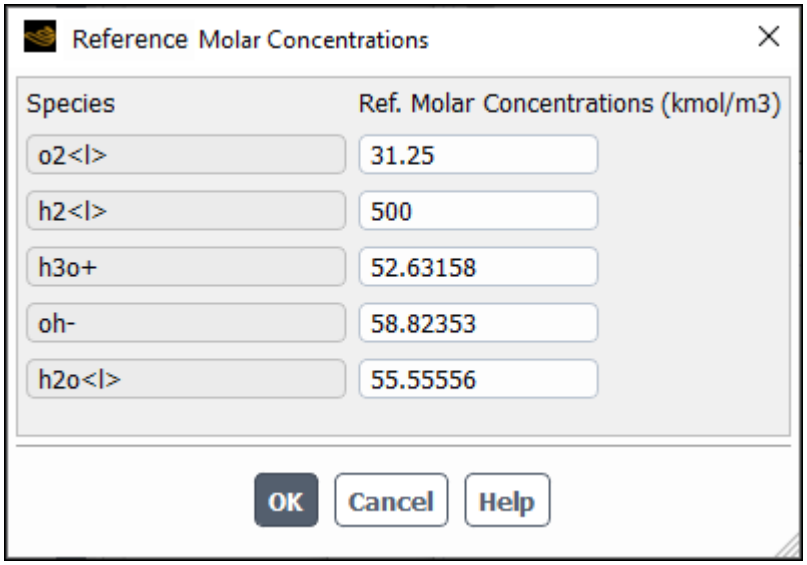
is the surface coverage rate modification of the species and is defined in Equation 7.81 in the Theory Guide.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

51.5.34. Reference Molar Concentrations Dialog Box

The **Reference Molar Concentrations** dialog box (opened by clicking **Specify Reference Molar Concentrations...** under **Butler-Volmer/Tafel Parameters** in the Reactions Dialog Box (p. 4930)) allows you to specify reference species molar concentrations for the electrochemical reaction.

(Note that the **Reference Molar Concentrations** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.)



Controls

Species

are the electrochemical reaction species used in the Butler-Volmer equation (Equation 7.118 in the *Fluent Theory Guide*).

Reference Molar Concentrations

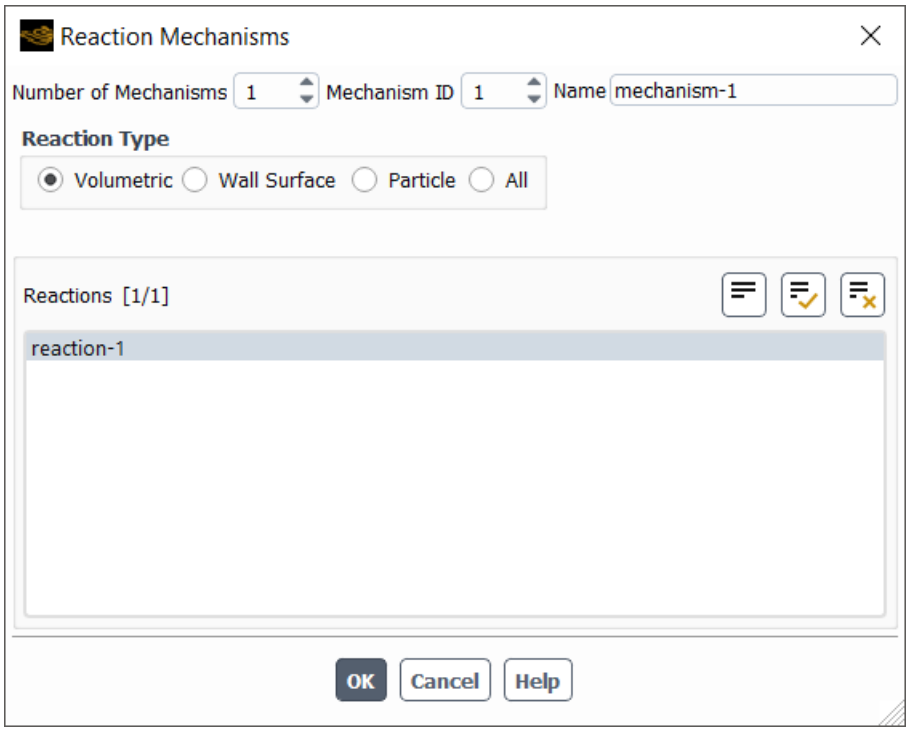
contains inputs for the reference molar concentrations of electrochemical reaction species ($C_{k,r}$ in Equation 7.118).

51.5.35. Reaction Mechanisms Dialog Box

The **Reaction Mechanisms** dialog box (opened by clicking the **Edit...** button next to **Mechanism** in the Create/Edit Materials Dialog Box (p. 4885)) allows you to select the reaction mechanism at a particular zone. See Mixture Materials (p. 2357) for details about these methods and the related inputs.

(Note that the **Reaction Mechanisms** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Number of Mechanisms

specifies the number of mechanisms present.

Mechanism ID

is the ID of the mechanism that you specify.

Name

allows you to enter a name for the mechanism.

Reaction Type

specifies the type of reaction to be displayed for the mechanism.

Volumetric

displays all volumetric reactions under the **Reactions** list.

Wall Surface

displays all wall surface reactions under the **Reactions** list.

Particle

displays all particle reactions under the **Reactions** list. This item is available only when **Particle** is selected as the **Reaction Type** in the **Species Model** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Electrochemical

displays all electrochemical reactions under the **Reactions** list.

All

displays all types of reactions under the **Reactions** list.

Reactions

displays the list of reactions of the category specified under **Reaction Type**.

Number of Sites

specifies the number of sites at which you can specify the reaction.

Site Name

contains the name of the site.

Site Density

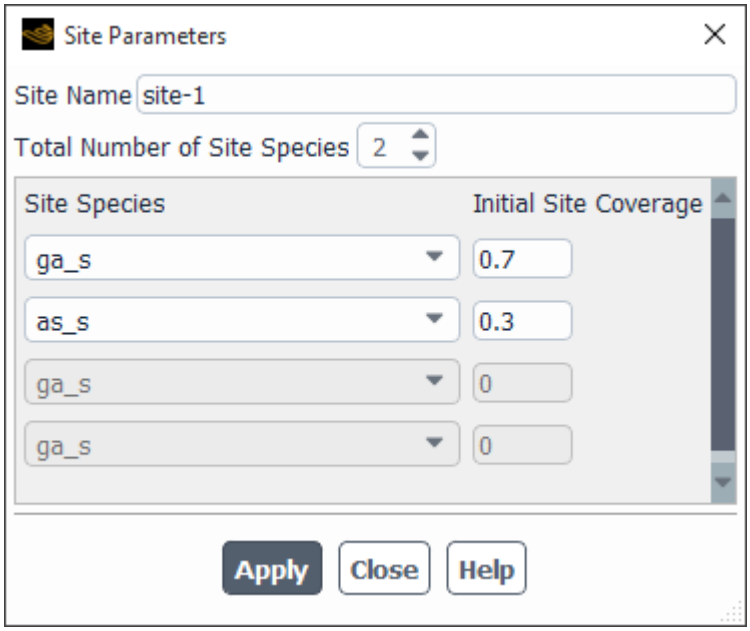
allows you to specify the site density of the species

Define...

opens the Site Parameters Dialog Box (p. 4943).

51.5.36. Site Parameters Dialog Box

The **Site Parameters** dialog box (opened by clicking the **Define...** button next to **Site Density** in the Reaction Mechanisms Dialog Box (p. 4941)) allows you to define the coverage for each site species.



Controls

Site Name

displays the name of site.

Total Number of Site Species

specifies the total number of site species.

Site Species

allows you to select the site species.

Initial Site Coverage

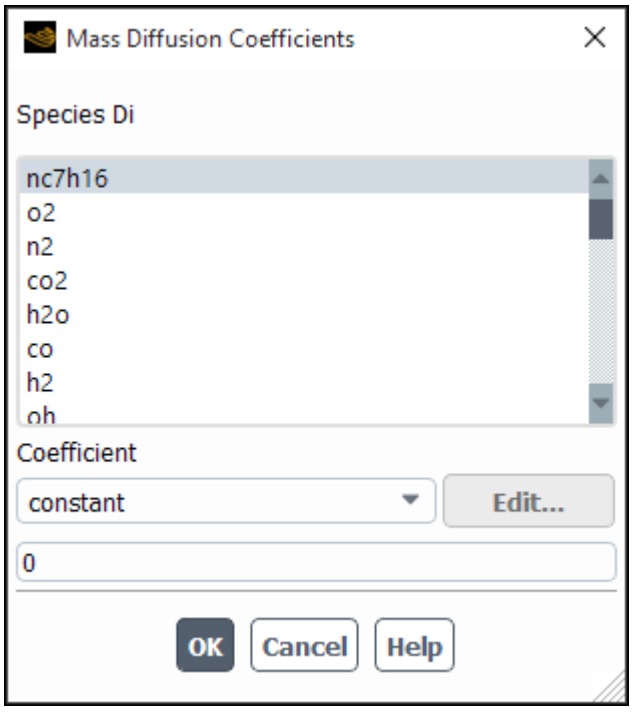
allows you to specify the initial coverage of the site species.

51.5.37. Mass Diffusion Coefficients Dialog Box

The **Mass Diffusion Coefficients** dialog box (opened by clicking the **Edit...** button next to **Mass Diffusivity** in the Create/Edit Materials Dialog Box (p. 4885) allows you to define the diffusion coefficients of the species in the mixture. Its contents will depend on the method you selected for **Mass Diffusivity**. See Mass Diffusion Coefficient Inputs (p. 1669) for details about these methods and the related inputs.

(Note that the **Mass Diffusion Coefficients** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.)

For the **dilute-approx** method:



Controls

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

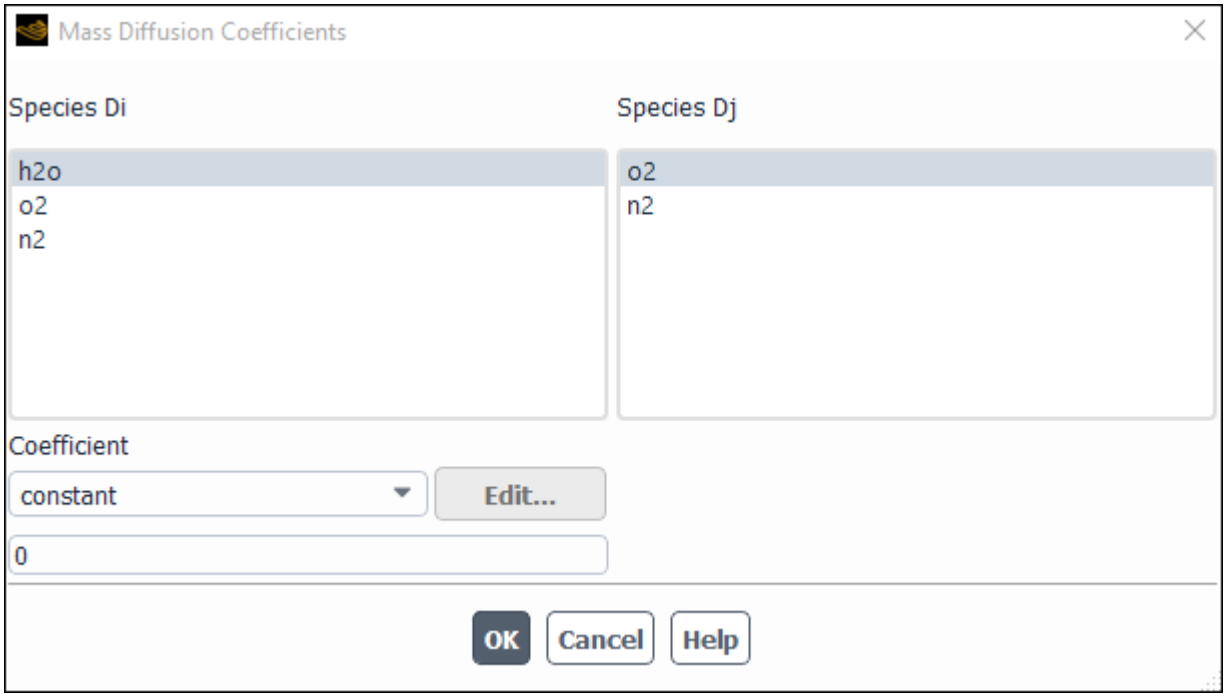
Species Di

contains a selectable list of all species in the mixture, from which you can select each species and specify its diffusion coefficient.

Coefficient

sets the diffusion coefficient for the selected species in the mixture.

For the **multicomponent** method:



Controls

Species Di, Species Dj

contain selectable lists of species in the mixture, from which you can select each pair of species and specify the diffusion coefficient of the selected **Species Di** in the selected **Species Dj**.

Coefficient*

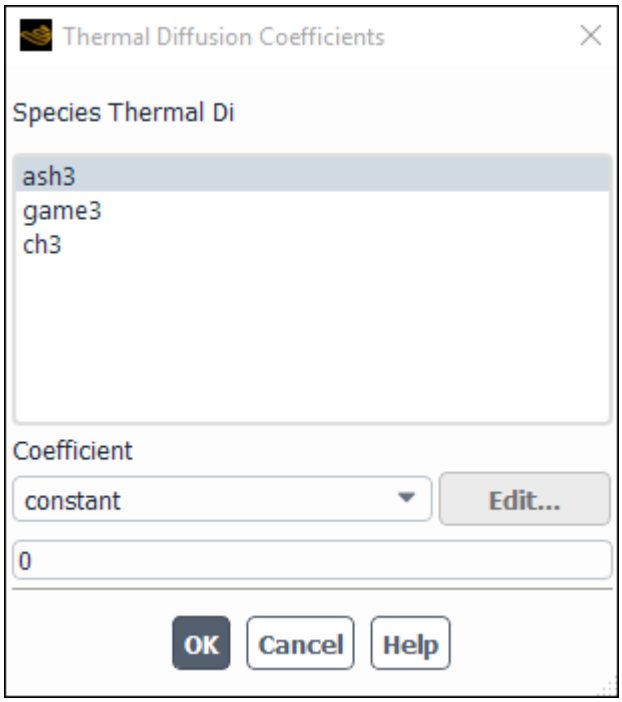
sets the multicomponent diffusion coefficient for **Species Di** in **Species Dj** (which is equivalent to the diffusion coefficient for **Species Dj** in **Species Di**).

51.5.38. Thermal Diffusion Coefficients Dialog Box

The **Thermal Diffusion Coefficients** dialog box (opened by clicking the **Edit...** button next to **Thermal Diffusion Coefficient** in the Create/Edit Materials Dialog Box (p. 4885) allows you to define the thermal diffusion coefficients of the species in the mixture. See Thermal Diffusion Coefficient Inputs (p. 1674) for details about these methods and the related inputs.

(Note that the **Thermal Diffusion Coefficients** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Species Thermal Di

contains a selectable list of all species in the mixture, from which you can select each species and specify its thermal diffusion coefficient.

Coefficient

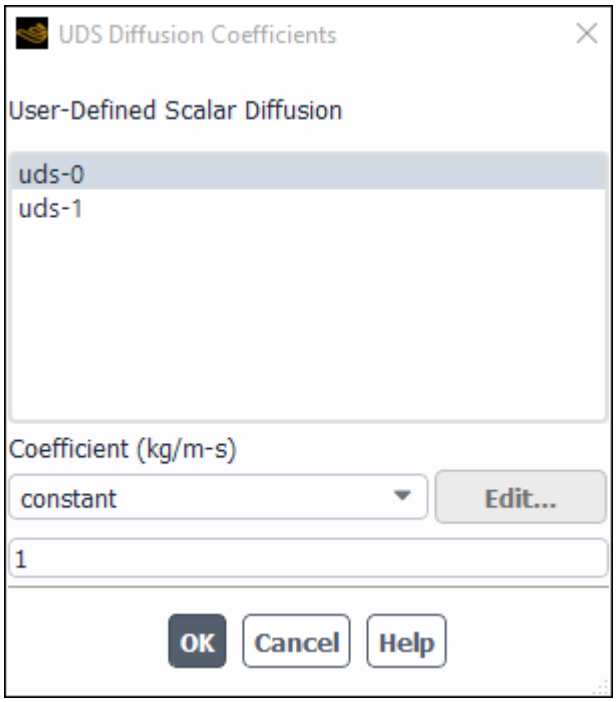
sets the thermal diffusion coefficient for the selected species in the mixture.

51.5.39. UDS Diffusion Coefficients Dialog Box

The **UDS Diffusion Coefficients** dialog box (opened by selecting **uds** and clicking the **Edit...** button next to **UDS Diffusivity** in the Create/Edit Materials Dialog Box (p. 4885)) allows you to define the diffusion coefficients for your user-defined scalar transport equations.

(Note that the **UDS Diffusion Coefficients** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

User-Defined Scalar Diffusion

contains a selectable list of all user-defined scalars, from which you can select each one and specify its diffusion coefficient.

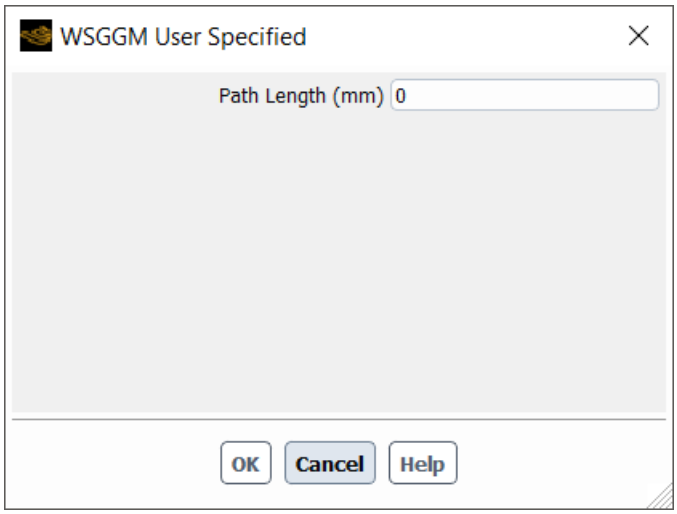
Coefficient

sets the diffusion coefficient for the selected user-defined scalar.

51.5.40. WSGGM User Specified Dialog Box

The **WSGGM User Specified** dialog box opens when you select **wsggm-user-specified** as the input method for a composition-dependent **Absorption Coefficient** in the Create/Edit Materials Dialog Box (p. 4885) and it allows you to define the path length for he weighted-sum-of-gray-gases model. See Inputs for a Composition-Dependent Absorption Coefficient (p. 1666) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



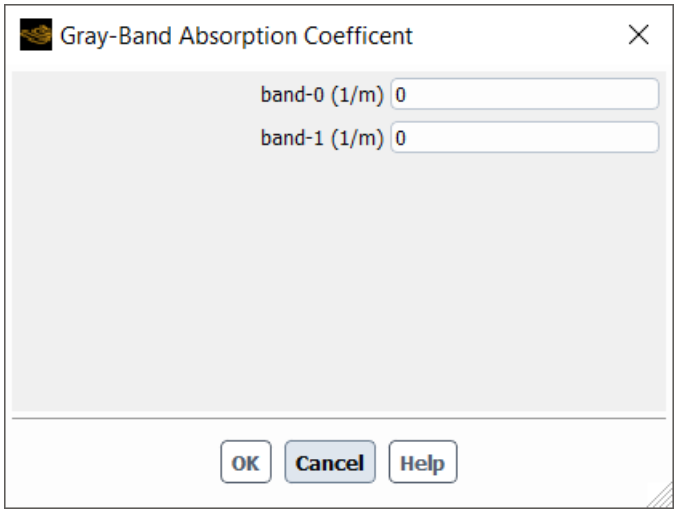
Controls

Path Length

allows you to set the **Path Length** equal to a mean beam length that you have calculated outside of Fluent.

51.5.41. Gray-Band Absorption Coefficient Dialog Box

The **Gray-Band Absorption Coefficient** allows you to specify a different absorption coefficient in each gray band when you are modeling non-gray radiation with the P-1, DO, or MC models (see The P-1 Model Equations, The DO Model Equations, or Monte Carlo (MC) Radiation Model Theory in the Theory Guide and Setting Up the P-1 Model with Non-Gray Radiation (p. 2194), Defining Non-Gray Radiation for the DO Model (p. 2212), or Setting Up the MC Model (p. 2214)). This dialog box will open when you select **gray-band** in the drop-down list next to **Absorption Coefficient** in the Create/Edit Materials Dialog Box (p. 4885).



Controls

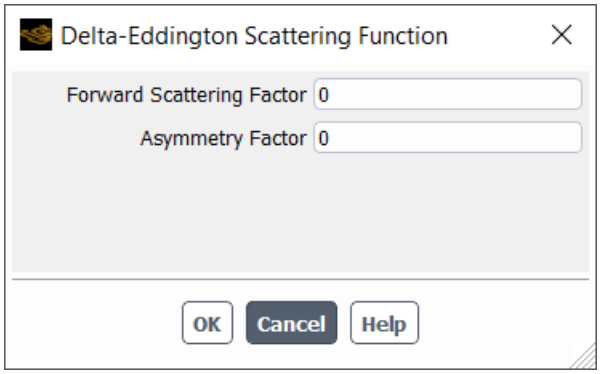
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

band n

specifies the absorption coefficient for the n^{th} gray band.

51.5.42. Delta-Eddington Scattering Function Dialog Box

The **Delta-Eddington Scattering Function** dialog box allows you to define the parameters used in the Delta-Eddington phase function for radiation scattering. This dialog box will open when you select **delta-eddington** in the drop-down list next to **Scattering Phase Function** in the Create/Edit Materials Dialog Box (p. 4885). See Anisotropic Scattering for details about the items below.



Controls

Forward Scattering Factor

specifies the value of f in Equation 5.107 in the Theory Guide.

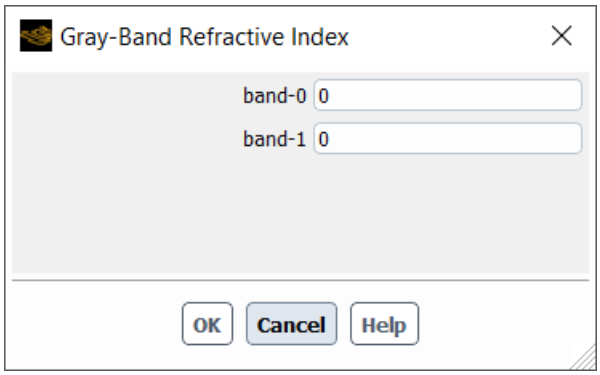
Asymmetry Factor

specifies the value of C in Equation 5.107 in the Theory Guide.

51.5.43. Gray-Band Refractive Index Dialog Box

The **Gray-Band Refractive Index** allows you to specify a different refractive index in each gray band when you are modeling non-gray radiation with the P-1, DO, or MC model (see The P-1 Model Equations, The DO Model Equations, or Monte Carlo (MC) Radiation Model Theory in the Theory Guide and Setting Up the P-1 Model with Non-Gray Radiation (p. 2194), Defining Non-Gray Radiation for the DO Model (p. 2212), or Setting Up the MC Model (p. 2214)). This dialog box will open when you select **refractive-band** in the drop-down list next to **Refractive Index** in the Create/Edit Materials Dialog Box (p. 4885).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

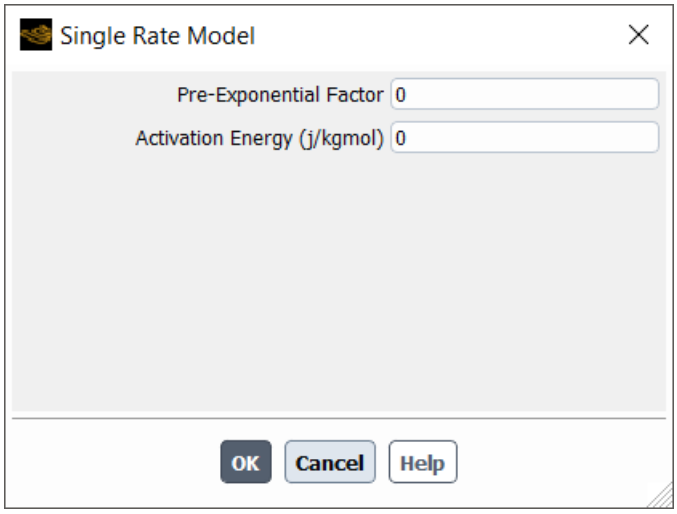
Band n

specifies the refractive index for the n^{th} gray band.

51.5.44. Single Rate Model Dialog Box

The **Single Rate Model** dialog box (which opens when you select **single-rate** from either the **Devolatilization** or **Themolysis Model** drop-down list in the Create/Edit Materials Dialog Box (p. 4885)) allows you to specify the parameters used in either the single kinetic rate devolatilization model or single kinetic rate Thermolysis model. For more details about these models, see Devolatilization (Law 4) and Mass Transfer During Law 2—Thermolysis in the *Fluent Theory Guide*, respectively.

Note that the **Single Rate Model** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.



Controls

Pre-exponential Factor

sets the value of:

- Devolatilization model: A_1 in Equation 12.109 in the Theory Guide for the computation of the kinetic rate.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Thermolysis model: A in Equation 12.87 (for droplet particles) or A_i in Equation 12.164 in the *Fluent Theory Guide* (for multicomponent particles).

Activation Energy

sets the value of

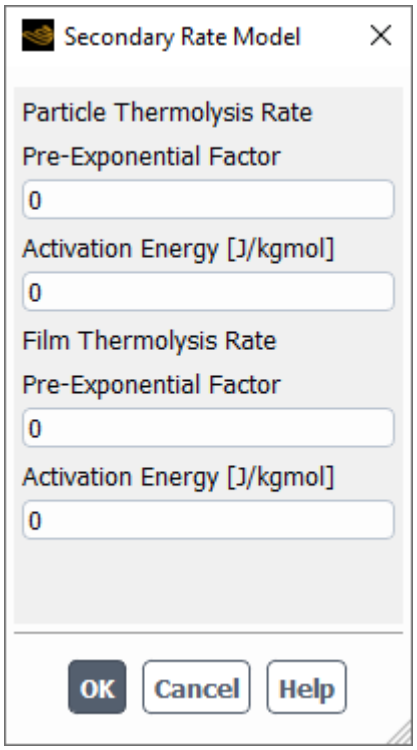
- Devolatilization model: E in Equation 12.109 in the Theory Guide for the computation of the kinetic rate.
- Thermolysis model: E in Equation 12.87 (for droplet particles) or E_i in Equation 12.164 in the *Fluent Theory Guide* (for multicomponent particles).

51.5.45. Secondary Rate Model Dialog Box

The **Secondary Rate Model** dialog box (which opens when you select **secondary-rate** from the **Themolysis Model** drop-down list in the Create/Edit Materials Dialog Box (p. 4885)) allows you to specify the parameters used in the Thermolysis model to calculate either the mass transfer rate from the free particle and the film to the bulk gas phase (for a droplet material) or the vaporization rate of the free particle and the film (for multicomponent material). For more information, see Mass Transfer During Law 2—Thermolysis and Multicomponent Particle Definition (Law 7) in the *Fluent Theory Guide*.

This dialog box is available only when the Lagrangian wall film is used.

Note that the **Secondary Rate Model** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.



Controls

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Particle Thermolysis Rate

is where entries for calculating the particle thermolysis rate can be set.

Pre-exponential Factor

sets the value of $A_{particle}$ to be used in either Equation 12.87 in the *Fluent Theory Guide* for a droplet material or Equation 12.164 in the *Fluent Theory Guide* for a multicomponent material.

Activation Energy

sets the value of $E_{particle}$ to be used in either Equation 12.87 in the *Fluent Theory Guide* for a droplet material or Equation 12.164 in the *Fluent Theory Guide* for a multicomponent material.

Film Thermolysis Rate

is where entries for calculating the film thermolysis rate can be set.

Pre-exponential Factor

sets the value of A_{film} to be used in either Equation 12.87 in the *Fluent Theory Guide* for a droplet material or Equation 12.164 in the *Fluent Theory Guide* for a multicomponent material.

Activation Energy

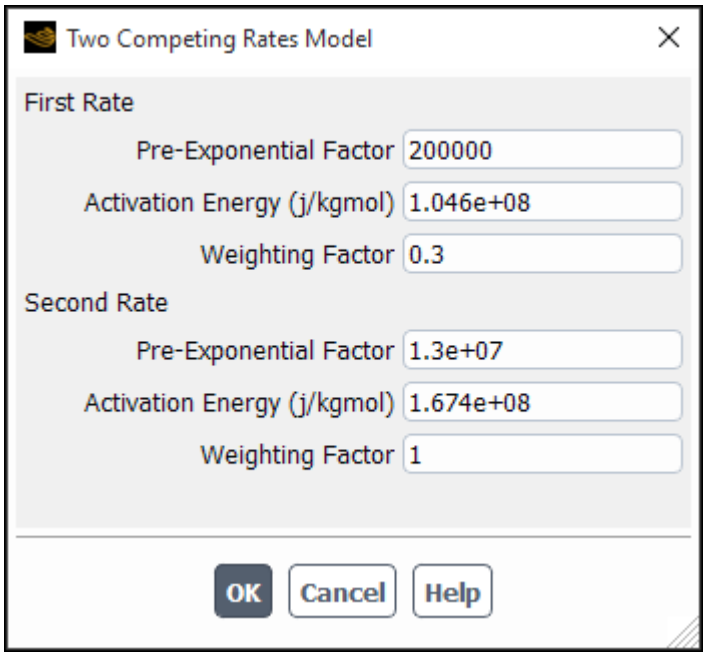
sets the value of E_{film} to be used in either Equation 12.87 in the *Fluent Theory Guide* for a droplet material or Equation 12.164 in the *Fluent Theory Guide* for a multicomponent material.

51.5.46. Two Competing Rates Model Dialog Box

The **Two Competing Rates Model** dialog box (which opens when you select **two-competing-rates** as the **Devolatilization Model** in the Create/Edit Materials Dialog Box (p. 4885) allows you to specify the parameters used for each of the competing rates in the two-competing-rates devolatilization model. See Devolatilization (Law 4) in the Theory Guide for details.

Note that the **Two Competing Rates Model** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

First Rate

sets parameters for the first of the two rates.

Pre-exponential Factor

sets the value of A_1 in Equation 12.111 in the Theory Guide for the computation of the kinetic rate.

Activation Energy

sets the value of E_1 in Equation 12.111 in the Theory Guide for the computation of the kinetic rate.

Weighting Factor

sets the value of α_1 in Equation 12.113 in the Theory Guide.

Second Rate

sets parameters for the second of the two rates.

Pre-exponential Factor

sets the value of A_2 in Equation 12.112 in the Theory Guide for the computation of the kinetic rate.

Activation Energy

sets the value of E_2 in Equation 12.112 in the Theory Guide for the computation of the kinetic rate.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

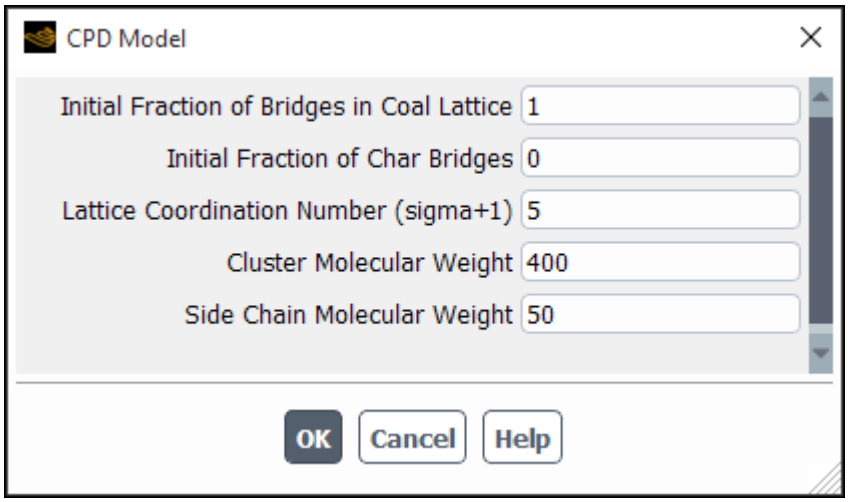
Weighting Factor

sets the value of α_2 in Equation 12.113 in the Theory Guide.

51.5.47. CPD Model Dialog Box

The **CPD Model** dialog box (which opens when you select **cpd-model** as the **Devolatilization Model** in the Create/Edit Materials Dialog Box (p. 4885)) allows you to specify the parameters used in the CPD devolatilization model. See Devolatilization (Law 4) in the Theory Guide for details.

Note that the **CPD Model** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.



Controls

Initial Fraction of Bridges in Coal Lattice

sets the value of p_0 in Equation 12.124 in the Theory Guide.

Initial Fraction of Char Bridges

sets the value of c_0 in Equation 12.123 in the Theory Guide.

Lattice Coordination Number

sets the value of $\sigma+1$ in Equation 12.135 in the Theory Guide.

Cluster Molecular Weight

sets the value of $M_{w,1}$ in Equation 12.135 in the Theory Guide.

Side Chain Molecular Weight

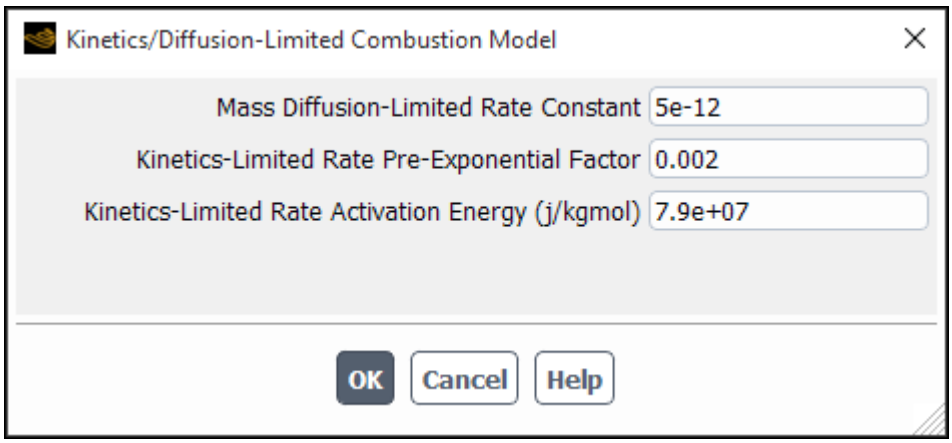
sets the value of $M_{w,\delta}$ in Equation 12.134 in the Theory Guide.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

51.5.48. Kinetics/Diffusion-Limited Combustion Model Dialog Box

The **Kinetics/Diffusion-Limited Combustion Model** dialog box (which opens when you select **kinetics/diffusion-limited** as the **Combustion Model** in the Create/Edit Materials Dialog Box (p. 4885) allows you to specify the parameters used for the kinetics/diffusion-limited rate surface combustion model. See Surface Combustion (Law 5) in the Theory Guide for details.

Note that the **Kinetics/Diffusion-Limited Combustion Model** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.



Controls

Mass Diffusion-Limited Rate Constant

sets the value for C_1 in Equation 12.147 in the Theory Guide.

Kinetics-Limited Rate Pre-exponential Factor

sets the value for C_2 in Equation 12.148 in the Theory Guide.

Kinetics-Limited Rate Activation Energy

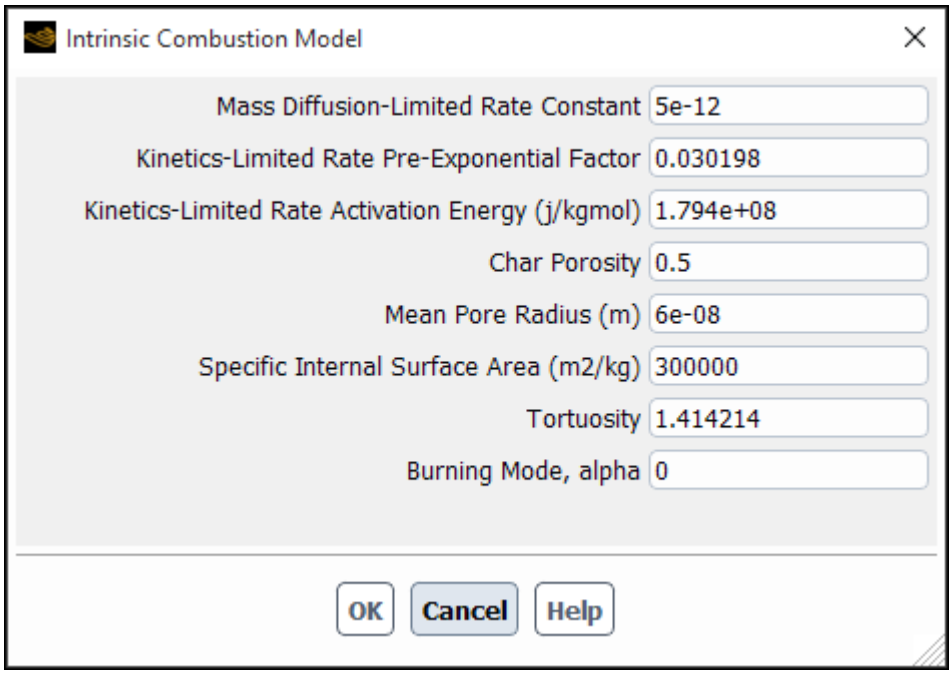
sets the value for E in Equation 12.148 in the Theory Guide.

51.5.49. Intrinsic Combustion Model Dialog Box

The **Intrinsic Combustion Model** dialog box (which opens when you select **intrinsic-model** as the **Combustion Model** in the Create/Edit Materials Dialog Box (p. 4885) allows you to specify the parameters used for the intrinsic surface combustion model. See Surface Combustion (Law 5) in the Theory Guide for details.

Note that the **Intrinsic Combustion Model** dialog box is a modal dialog box, which means that you must tend to it immediately before continuing the property definitions.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Mass Diffusion-Limited Rate Constant

sets the value for C_1 in Equation 12.147 in the Theory Guide.

Kinetics-Limited Rate Pre-exponential Factor

sets the value for A_i in Equation 12.157 in the Theory Guide.

Kinetics-Limited Rate Activation Energy

sets the value for E_i in Equation 12.157 in the Theory Guide.

Char Porosity

sets the value for θ in Equation 12.154 in the Theory Guide.

Mean Pore Radius

sets the value for $\overline{r}_p$ in Equation 12.156 in the Theory Guide.

Specific Internal Surface Area

sets the value for A_g in Equation 12.151 and Equation 12.153 in the Theory Guide.

Tortuosity

sets the value for τ in Equation 12.154 in the Theory Guide.

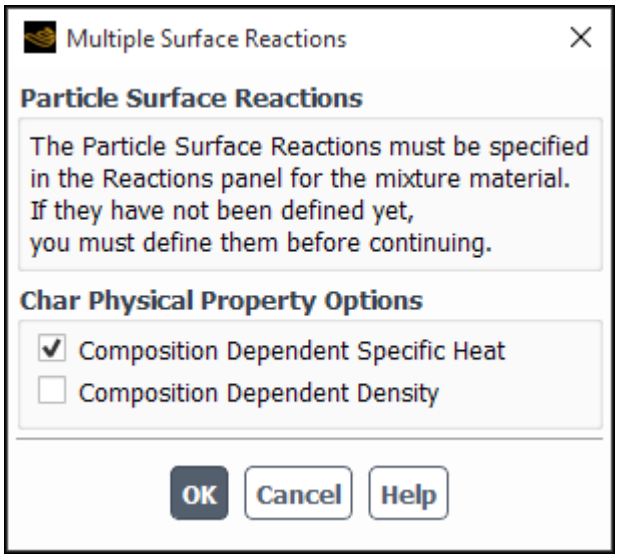
Burning Mode, alpha

sets the value for α in Equation 12.158 in the Theory Guide.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

51.5.50. Multiple Surface Reactions Dialog Box

The **Multiple Surface Reactions** dialog box (which opens when you select **multiple-surface-reactions** as the **Combustion Model** in the Create/Edit Materials Dialog Box (p. 4885)) allows you to enable composition-dependent char properties when using the multiple surface reactions combustion model.



Controls

Composition Dependent Specific Heat

enables calculation of the char specific heat from the particle specific heat values. See **Combustion Model** (p. 2818) for details.

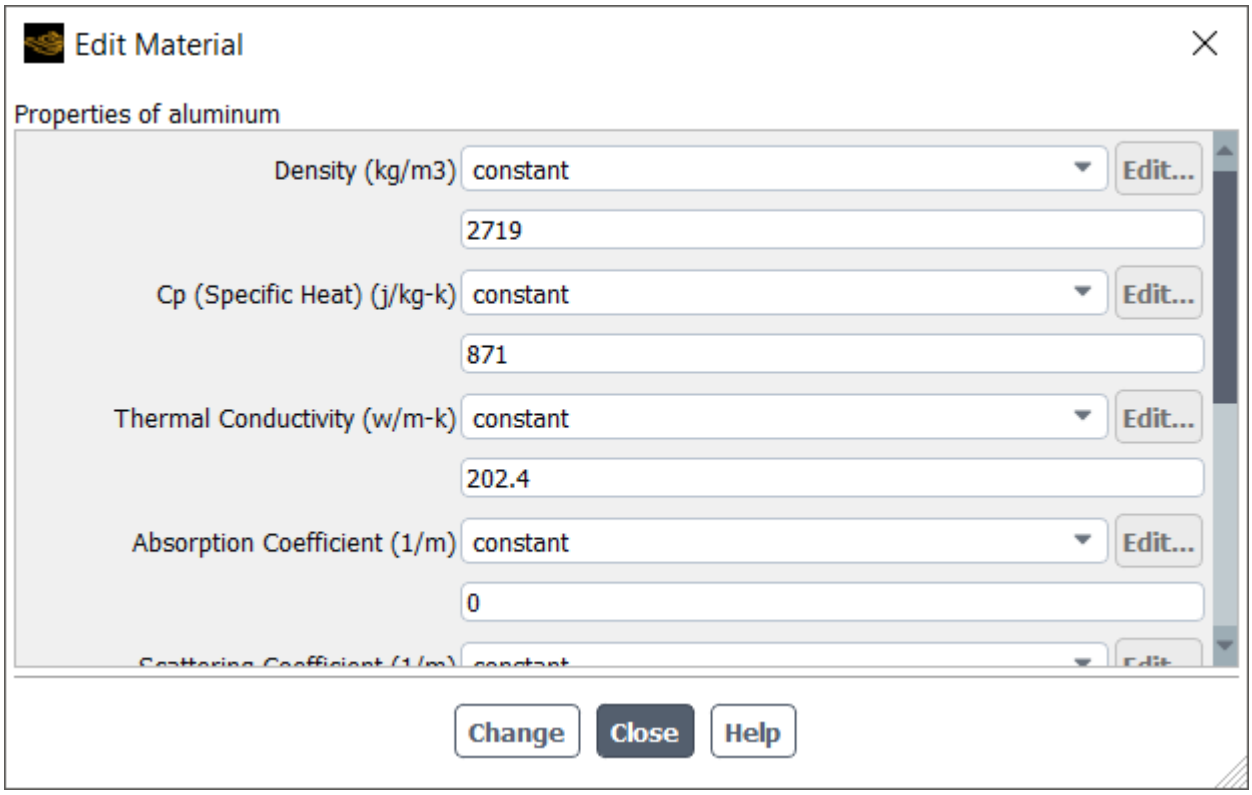
Composition Dependent Density

enables calculation of the char density from the particle density values. See **Combustion Model** (p. 2818) for details.

51.5.51. Edit Material Dialog Box

The **Edit Material** dialog box contains the portion of the Create/Edit Materials Dialog Box (p. 4885) that contains the properties for a specific material. It is opened from the Multiphase Model Dialog Box (p. 4700) (**Phases** tab), Wall Dialog Box (p. 5085), Fluid Dialog Box (p. 4961), or Solid Dialog Box (p. 4973).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Properties of material-n

contains a list of the properties of material-*n*. The items in the list are the same as those in the Create/Edit Materials Dialog Box (p. 4885).

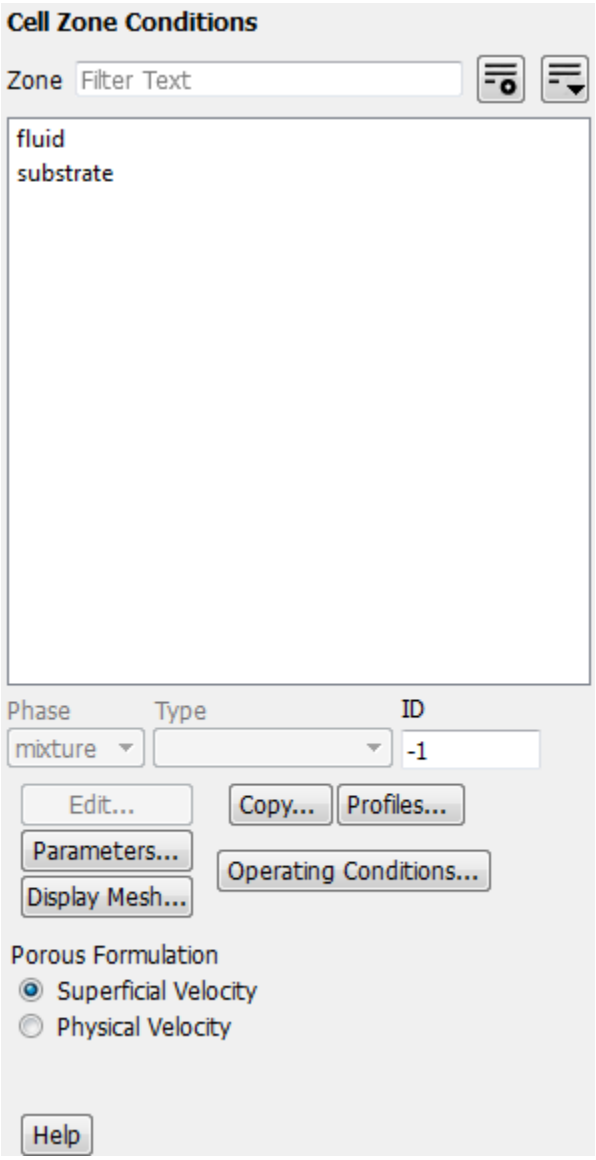
Change

applies any changes you have made to the properties of the material.

51.6. Cell Zone Conditions Task Page

The **Cell Zones Conditions** task page allows you to set the type of a cell zone and display other dialog boxes to set the cell zone condition parameters for each zone. See Cell Zone Conditions (p. 1320) for more information.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Zone

contains a selectable list of available cell zones from which you can select the zone of interest. You can check a zone type by using the mouse probe (see Controlling the Mouse Button Functions (p. 4020)) on the displayed physical mesh. This feature is particularly useful if you are setting up a problem for the first time, or if you have two or more cell zones of the same type and you want to determine the cell zone IDs. To do this you must first display the mesh with the Mesh Display Dialog Box (p. 4688). Then click the boundary zone with the right (select) mouse button. Ansys Fluent will print the cell zone ID and type of that boundary zone in the console window.

Phase

specifies the phase for which conditions at the selected cell **Zone** are being set. This item appears if the VOF, mixture, or Eulerian multiphase model is being used. See Defining Multiphase Cell Zone and Boundary Conditions (p. 2957) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Type

contains a drop-down list of condition types for the selected cell zone. The list contains all possible types to which the cell zone can be changed.

ID

displays the cell zone ID number of the selected cell zone. (This is for informational purposes only; you cannot edit this number.)

Edit...

opens the Fluid Dialog Box (p. 4961) or Solid Dialog Box (p. 4973).

Copy...

opens the Copy Conditions Dialog Box (p. 4977), which allows you to copy conditions from one cell zone to other cell zones of the same type. See Copying Cell Zone and Boundary Conditions (p. 1305) for details.

Profiles...

opens the Profiles Dialog Box (p. 4981).

Parameters...

opens the Parameters Dialog Box (p. 5338).

Operating Conditions...

opens the Operating Conditions Dialog Box (p. 4978).

Display Mesh...

opens the Mesh Display Dialog Box (p. 4688).

Porous Formulation

contains options for setting the velocity in the porous medium simulation. See Defining the Porous Velocity Formulation (p. 1345) for details.

Superficial Velocity

enables the superficial velocity in a porous medium simulation. This is the default method.

Physical Velocity

enables the physical velocity in a porous medium simulation for a more accurate simulation. This option is available only for a pressure-based solver. See Modeling Porous Media Based on Physical Velocity (p. 1340) for details.

For additional information, see the following sections:

51.6.1. Fluid Dialog Box

51.6.2. Solid Dialog Box

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

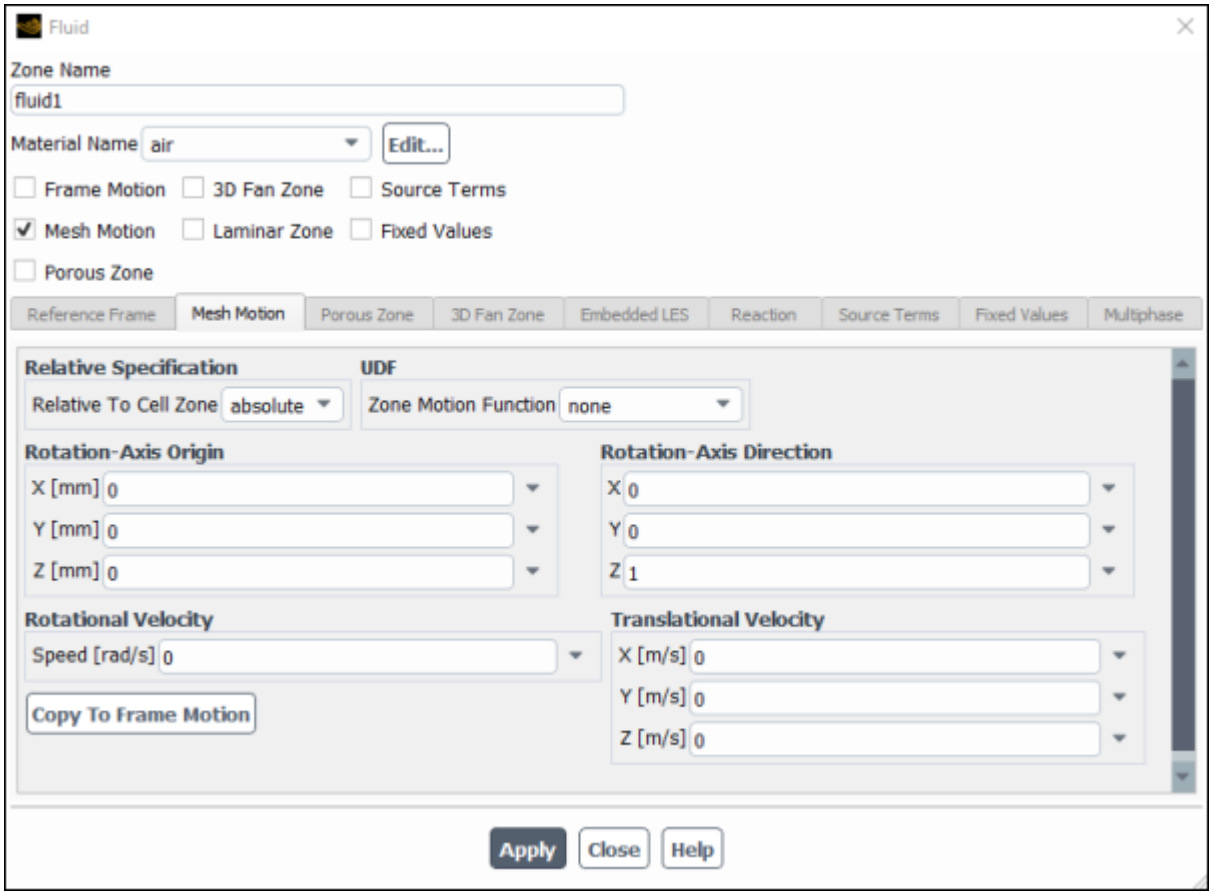
51.6.3. Copy Conditions Dialog Box
51.6.4. Operating Conditions Dialog Box
51.6.5. Select Input Parameter Dialog Box
51.6.6. Profiles Dialog Box
51.6.7. Replicate Profile Dialog Box
51.6.8. Orient Profile Dialog Box
51.6.9. Write Profile Dialog Box
51.6.10. Select Profiles for Writing

51.6.1. Fluid Dialog Box

The **Fluid** dialog box sets the conditions for a fluid cell zone. It is opened from the Cell Zone Conditions Task Page (p. 4958). See Inputs for Fluid Zones (p. 1321) and User Inputs for Porous Media (p. 1343) for details about the items below.

Important:

This feature offers reduced functionality when running Fluent under the **Pro** capability level.



Controls

Zone Name

sets the name of the zone.

Material Name

sets the fluid material. The drop-down list contains the names of all materials that have been loaded into the solver. Materials are defined with the Materials Task Page (p. 4883).

Important:

If you are modeling species transport or multiphase flow, the **Material Name** list will not appear in the **Fluid** dialog box. For species calculations, the mixture material for all fluid zones will be the material you specified in the Species Model Dialog Box (p. 4768). For multiphase flows, the materials are specified when you define the phases, as described in Defining the Phases for the VOF Model (p. 3009).

Phase

(multiphase models only) specifies the phase for which conditions at the selected fluid cell zone are being set.

Frame Motion

enables the moving reference frame model for the cell zone. See Specifying the Rotation Axis (p. 1323) and Defining Zone Motion (p. 1323) for details.

Mesh Motion

enables the sliding mesh model for the cell zone. See Setting Up the Sliding Mesh Problem (p. 1805) for details.

Porous Zone

indicates that the zone is a porous medium. Additional items will appear in the dialog box when this option is enabled. See User Inputs for Porous Media (p. 1343) for details.

3D Fan Zone

allows you to simulate the effect of an axial fan by applying a distributed momentum source in a toroid-shaped fluid volume. This model can calculate not only the axial velocity, but the swirl and radial velocities as well. When you enable this option, the **3D Fan Zone** tab becomes available, so that you can specify the geometry and properties of the fan. The cell zone should have a toroid shape that is sized to match the blade-swept volume of the fan you are simulating (that is, has an inner and outer radius that matches the radius of the fan's hub and blade tips, respectively, as well as a length that matches the thickness of the toroid region swept by the blades in the axial direction). Furthermore, the fluid cell zone should be "interior", that is, have at least two boundary zones of type **interior** that border another fluid cell zone and act as the fan inlet and outlet (the other boundaries can be of type **interior** as well). See 3D Fan Zones (p. 1372) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Laminar Zone

disables the calculation of turbulence production in the fluid zone (appears only for turbulent flow calculations using the Spalart-Allmaras model or one of the k- ε or k- ω models). See Specifying a Laminar Zone (p. 1322) for details.

LES Zone

allows you to model a smaller embedded LES zone within a larger URANS computational domain for turbulent flow calculations. When you turn on this option, the **Embedded LES** tab will be enabled to allow you to specify properties for the embedded LES zone. See Setting Up the Embedded Large Eddy Simulation (ELES) Model (p. 2124) for details.

Source Terms

enables the specification of volumetric sources of mass, momentum, energy, and so on. When you turn on this option, the **Source Terms** tab will be enabled to allow you to specify the values for the desired sources. See Defining Mass, Momentum, Energy, and Other Sources (p. 1382) for details.

Fixed Values

enables the fixing of the value of one or more variables in the fluid zone, rather than computing them during the calculation. See Fixing the Values of Variables (p. 1377) for details.

Important:

You can fix values for velocity components, temperature, and species mass fractions only if you are using the pressure-based solver.

Participates In Radiation

specifies whether or not the fluid zone participates in radiation. This option appears when you are using the DO model for radiation.

Warning:

In general, disabling **Participates in Radiation** for fluid zones is not recommended, as it can produce erroneous results. There are rare cases when it is acceptable: for example, if the domain contains multiple fluid zones, disabling this option for zones where radiation is negligible may save computational time without affecting the results.

Reaction

enables/disables reactions in the porous zone.

Electrolyte

indicates that the zone is an electrolyte zone. By default, all fluid zones are assumed to be electrolyte zones. This option appears when the **Electrochemical** Reaction model is enabled.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reference Frame Tab

lists the parameters that define motion for a moving reference frame.

Rotation-Axis Origin

specifies the origin for the axis of rotation of the fluid zone. See Specifying the Rotation Axis (p. 1323) for details. This item will appear only for 3D and 2D non-axisymmetric models.

Rotation-Axis Direction

specifies the direction vector for the fluid zone's axis of rotation. See Specifying the Rotation Axis (p. 1323) for details. This item will appear only for 3D models.

Rotational Velocity

contains an input field for the rotational **Speed** of the zone.

Translational Velocity

contains inputs for the **X**, **Y**, and **Z** velocities of the zone.

Relative Specification

indicates whether the velocities are absolute velocities (**absolute**) or velocities relative to the motion of each cell zone (**Relative to Cell Zone**). This selection is necessary only if your problem involves moving reference frames or sliding meshes. If there is no zone motion, both options are equivalent.

UDF

allows you to hook the `DEFINE_ZONE_MOTION` user-defined function.

Copy to Mesh Motion

allows you to switch between the MRF and moving mesh models. The variables used for the origin, axis, and velocity components, as well as for the `DEFINE_ZONE_MOTION` user-defined function, will be copied from one model to the other.

Mesh Motion Tab

lists the parameters that define motion for a moving reference frame.

Rotation-Axis Origin

specifies the origin for the axis of rotation of the fluid zone. See Specifying the Rotation Axis (p. 1323) for details. This item will appear only for 3D and 2D non-axisymmetric models.

Rotation-Axis Direction

specifies the direction vector for the fluid zone's axis of rotation. See Specifying the Rotation Axis (p. 1323) for details. This item will appear only for 3D models.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Rotational Velocity

contains an input field for the rotational **Speed** of the zone.

Translational Velocity

contains inputs for the **X**, **Y**, and **Z** velocities of the zone.

Relative Specification

indicates whether the velocities are absolute velocities (**absolute**) or velocities relative to the motion of each cell zone (**Relative to Cell Zone**). This selection is necessary only if your problem involves moving reference frames or sliding meshes. If there is no zone motion, both options are equivalent.

UDF

allows you to hook the `DEFINE_ZONE_MOTION` user-defined function.

Copy to Frame Motion

allows you to switch between the MRF and moving mesh models. The variables used for the origin, axis, and velocity components, as well as for the `DEFINE_ZONE_MOTION` user-defined function, will be copied from one model to the other.

Porous Zone Tab

lists the parameters associated with the porous zone.

Conical

enables the specification of a conical (or cylindrical) porous medium. This item will appear only when the **Porous Zone** option is enabled for a 3D case.

Cone Half Angle

specifies the angle between the cone's axis and its surface (see Figure 7.23: Cone Half Angle (p. 1348)). Set this to 0 to define the porous region using a cylindrical coordinate system. This item will appear only when the **Porous Zone** and **Conical** options are enabled.

Snap to Zone

aligns the plane (or line, in 2D) tool with the zone selected in the drop-down list. The tool is centered at the centroid of the zone, with the tool's axis normal to the zone. If this axis is not the desired cone axis, reposition the tool (as described in Using the Plane Tool (p. 3833)). When you are satisfied with the axis, click the **Update From Plane Tool** (or **Update From Line Tool**) button to update the **Cone Axis Vector** fields.

This item will appear only when the **Porous Zone** and **Conical** options are enabled.

Update From Plane Tool

(**Update From Line Tool** in 2D) updates the **Direction-1 Vector** and (in 3D) the **Direction-2 Vector** from the plane tool orientation. If the **Conical** option is enabled, this button will update the **Cone Axis Vector** and the **Point on Cone Axis**. See Defining the Viscous and Inertial

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Resistance Coefficients (p. 1346) for details. This item will appear only when the **Porous Zone** option is enabled.

Direction-1 Vector, Direction-2 Vector

indicate the directions for which the resistance coefficients are defined. See Defining the Viscous and Inertial Resistance Coefficients (p. 1346) for details. These items will appear only when the **Porous Zone** option is enabled, but the **Conical** option is not. (In 2D, only **Direction-1 Vector** will appear.)

Cone Axis Vector

specifies the **X,Y,Z** vector for the cone's axis.

This item will appear only when the **Porous Zone** and **Conical** options are enabled.

Point on Cone Axis

specifies a point on the cone's axis. This point will be used by Ansys Fluent to transform the resistances to the Cartesian coordinate system.

This item will appear only when the **Porous Zone** and **Conical** options are enabled.

Relative Velocity Resistance Formulation

allows Ansys Fluent to either apply the relative reference frame or the absolute reference frame. This allows for the correct prediction of the source terms.

Viscous Resistance, Inertial Resistance

contain inputs for the viscous resistance coefficient $1/\alpha$ and the inertial resistance coefficient $C2$ in each direction. See Defining the Viscous and Inertial Resistance Coefficients (p. 1346) for details. These items will appear only when the **Porous Zone** option is enabled.

Alternative Formulation

provides better stability to the calculation when your porous medium is anisotropic.

If you have enabled the **Conical** option, **Direction-1** is the cone axis direction, **Direction-2** is the normal to the cone surface (radial (r) direction for a cylinder), and **Direction-3** is the circumferential (θ) direction.

Power Law Model

contains inputs for the **C0** and **C1** coefficients in the power law model for porous media. See Using the Power-Law Model (p. 1354) for details.

Relative Permeability

(Eulerian multiphase model only) is a group box where you can set relative permeability for the mixture. See Specifying the Relative Permeability (p. 1358) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Capillary Pressure

(Eulerian multiphase model only) is a group box where you can set a model for capillary pressure for the mixture. See Capillary Pressure Usage (p. 1367) for details.

Relative Viscosity

is a group box where you can specify a model for relative viscosity. See Specifying the Relative Viscosity (p. 1358) for details.

Fluid Porosity

contains an additional input for the porous medium. See User Inputs for Porous Media (p. 1343) for details.

Porosity

sets the volume fraction of fluid within the porous region.

Heat Transfer Settings

contains heat transfer settings for the porous medium. See Specifying the Heat Transfer Settings (p. 1354) for details.

Thermal Model

specifies whether or not thermal equilibrium is assumed between the medium and the fluid flow.

Equilibrium

specifies that the medium and the fluid flow are in thermal equilibrium in the porous medium.

Non-Equilibrium

specifies that the medium and the fluid flow are not in thermal equilibrium in the porous medium, so that a dual cell approach is enabled.

Solid Material Name

specifies the solid material in the porous region. This drop-down menu is only available when **Equilibrium** is selected from the **Thermal Model** list.

Solid Zone

displays the name of the solid cell zone that is coupled with the porous fluid zone through heat transfer. This field is only displayed when **Non-Equilibrium** is selected from the **Thermal Model** list.

Interfacial Area Density

specifies A_{fs} (as described in Non-Equilibrium Thermal Model Equations (p. 1338)) for the porous region. This text-entry box is only available when **Non-Equilibrium** is selected from the **Thermal Model** list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Heat Transfer Coefficient

specifies h_{fs} (as described in Non-Equilibrium Thermal Model Equations (p. 1338)) for the porous region. This text-entry box is only available when **Non-Equilibrium** is selected from the **Thermal Model** list.

Anisotropic Species Diffusion

allows you to model anisotropic species diffusion in porous media. When this option is selected, you can specify the **Matrix Components** for the anisotropic diffusion matrix in the porous zone. This option is available only with the species transport models.

3D Fan Zone Tab

provides the settings that define a 3D fan zone. See 3D Fan Zones (p. 1372) for details.

Geometry

allows you to define the location and size of the fan.

Inlet Fan Zone

allows you to select a boundary zone of type **interior** that will act as the surface on which the fan will be located.

Hub Radius

defines the radius of the fan hub, that is, the inner radius of the blade-swept volume on which the fan's distributed momentum sources are applied.

Tip Radius

defines the radius of the blade tips of the fan, that is, the outer radius of the blade-swept volume on which the fan's distributed momentum sources are applied.

Thickness

defines the thickness of the toroid region swept by the blades in the axial direction.

Inflection Point

is a ratio that defines the fraction of the fan blade length (starting at the hub) over which the tangential velocity of the fan discharge is increasing with increasing radius and peaks, before tapering off (see Figure 7.30: The Inflection Point Ratio of a Pitched Blade Turbine (p. 1376)).

Fan Origin

defines the **X**, **Y**, and **Z** coordinates of a point on the **Inlet Fan Zone** that is the origin for the fan.

Properties

allows you to define how the fan behaves.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Rotational Direction

allows you to define the fan's rotational direction, using the right-hand rule with respect to the fan direction vector (that is, a vector from the **Fan Origin** that points into the 3D fan zone perpendicular to the **Inlet Fan Zone**).

Operating Angular Velocity

defines the angular velocity of the fan during the simulation.

Limit Flow Rate Through Fan

specifies that the flow rate through the fan adheres to minimum and maximum values, as defined in the **Flow Rate** group box.

Flow Rate

allows you to define the **Maximum** and **Minimum** flow rates through the fan. This group box is available when the **Limit Flow Rate Through Fan** option is enabled.

Tangential Source Term

enables a momentum source in the tangential direction, based on impeller theory.

Radial Source Term

enables a momentum source in the radial direction, based on impeller theory.

Axial Source Term

enables a momentum source in the axial direction, based on the settings in the **Axial Source Term Settings** group box.

Axial Source Term Settings

allows you to define the momentum source in the axial direction. This group box is only available when the **Axial Source Term** option is enabled.

Method

specifies whether a **constant pressure** or a **fan curve** (that is, a curve created from data points that plot pressure rise vs. volumetric flow rate) defines the axial momentum source.

Pressure Jump

defines the pressure jump that is applied across the 3D fan. This setting is only available when **constant pressure** is selected for the **Method**.

Read Fan Curve...

allows you to read a tab-delimited ASCII file that defines the relationship between pressure and flow rate for each 3D fan cell zone (see 3D Fan Curve File Format (p. 5694) for details). This button is only available when **fan curve** is selected for the **Method**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Curve Fitting Method

specifies whether Fluent should apply a **polynomial** or **piecewise-linear** fitting method to construct a curve from the data points in the fan curve file. This drop-down list is only available when **fan curve** is selected for the **Method**.

Order of Polynomial

defines the order of the polynomial curve fitting method. This setting is only available when **polynomial** is selected for the **Curve Fitting Method**.

Initial Flow Rate

specifies your best estimate of the initial flow rate through the fan. This setting is only available when **fan curve** is selected for the **Method**.

Test Angular Velocity

specifies the angular velocity of the fan used in the experiment that generated the fan curve data. This setting is only available when **fan curve** is selected for the **Method**.

Test Temperature

specifies the temperature of the fluid flowing through the fan during the experiment that generated the fan curve data. This setting is only available when **fan curve** is selected for the **Method**.

Reaction Tab

lists the parameters for reactions in the porous zone.

Reaction Mechanism

allows you to specify a defined group, or mechanism, of available reactions. See Defining Zone-Based Reaction Mechanisms (p. 2387) for details about defining reaction mechanisms.

Surface-to-Volume Ratio

specifies the surface area of the pore walls per unit volume ($\frac{A}{V}$), and can be thought of as a measure of catalyst loading. With this value, Ansys Fluent can calculate the total surface area on which the reaction takes place in each cell by multiplying $\frac{A}{V}$ by the volume of the cell.

Source Terms Tab

defines a source of heat, mass, momentum, turbulence, species, or other scalar quantity within the fluid zone.

Fixed Values Tab

allows you to specify which variables in the fluid zone are fixed (as a constant value or as defined by a profile or user-defined function), rather than computed during the calculation. See Fixing the Values of Variables (p. 1377) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Local Coordinate System for Fixed Velocities

enables the specification of fixed cylindrical velocity components instead of Cartesian components. The local coordinate system is defined by the **Rotation-Axis Origin** and **Rotation-Axis Direction**.

This item is available only in 3D, and only when the **Fixed Values** option is on.

Multiphase Tab

allows you to set parameters that are specific to the multiphase models.

Compressive Scheme Slope Limiter

ranges from 0 to 2. For example, a **Compressive Scheme Slope Limiter** of 0 corresponds to first order upwind behavior, a value of 1 corresponds to second order upwind, and a value of 2 applies the compressive scheme. The **Multiphase** tab is available only if **Zonal Discretization** is enabled in the Multiphase Model Dialog Box (p. 4700).

Numeric Beach Treatment

is available when **Open Channel Flow** and/or **Open Channel Wave BC** is enabled in the **Multiphase Model** dialog box (see Numerical Beach Treatment for Open Channels (p. 3005) for more detail).

Numerical Beach

when enabled expands the dialog box where you can specify the numerical beach parameters.

Beach Group ID

represents the cell zones sharing the damping length containing the same input parameters.

Damping Type

allows you to choose between **One Dimensional** and **Two Dimensional** damping.

One Dimensional

is the damping treatment in the flow direction.

Two Dimensional

is the damping treatment in the flow and gravity direction.

Compute From Inlet Boundary

is set to **none** by default. If there are available open channel boundaries (velocity-inlet, pressure-inlet, and mass-flow-inlet), boundary names are added to the drop-down list. If you select a boundary from the list, the **Level Inputs**, **Damping Length Inputs in Flow Direction**, and **Damping Resistance** values will be up-

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

dated in the interface. You have the option to overwrite the updated inputs with values that are more applicable to your simulation.

Level Inputs

is only available for the **Two Dimensional** damping type.

Free Surface Level

is the same definition as for open channel flow, see Modeling Open Channel Flows (p. 2979).

Bottom Level

is the same definition as for open channel flow, see Modeling Open Channel Flows (p. 2979), and is valid only for shallow waves. The bottom level is used for calculating the liquid height.

Flow Direction

is the X, Y, and Z (for 3D) components.

Damping Length Inputs in Flow Direction

are required to calculate the start and end points of the damping length in the flow direction.

Damping Length Specification

is only available if **Open Channel Wave BC** is enabled in the **Multiphase Model** dialog box. There are two options you can choose from **End Point and Wave Lengths** or **End and Start Points**.

End Point

is the end point of the damping zone.

Start Point

is the starting point in the flow direction.

Wave Length

is updated automatically if the boundary is selected from the **Compute From Inlet Boundary** drop-down list.

Number Of Wave Lengths

is set to 2 by default for the calculation of the damping length.

Relative Velocity Resistance Formulation

calculates the source term using relative velocities in the numerical beach zone when using moving/deforming meshes or moving reference frames.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Linear Damping Resistance

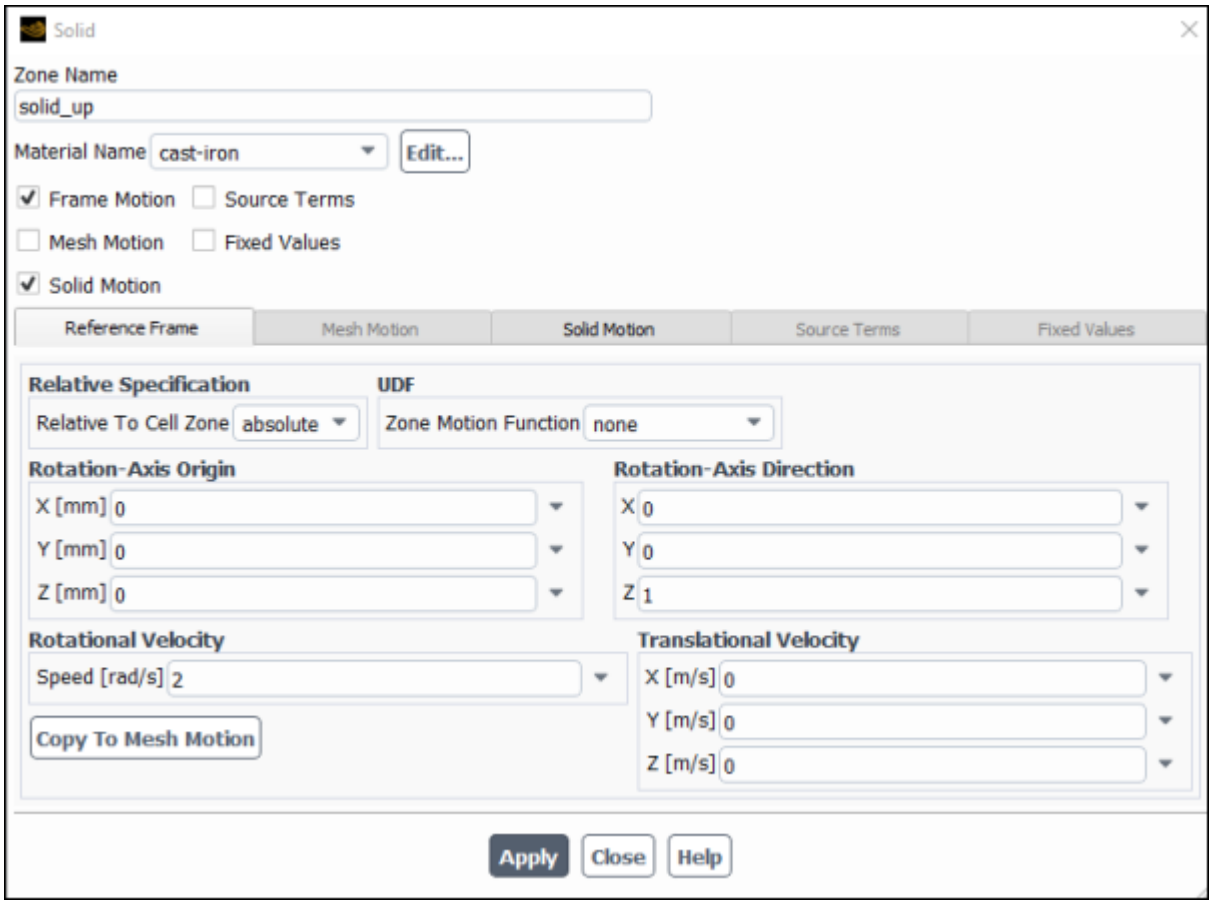
is the resistance per unit time.

Quadratic Damping Resistance

is the resistance per unit length.

51.6.2. Solid Dialog Box

The **Solid** dialog box sets the boundary conditions for a solid cell zone. It is opened from the Cell Zone Conditions Task Page (p. 4958). See Inputs for Solid Zones (p. 1327) for details about the items below.



Controls

Zone Name

sets the name of the zone.

Material Name

selects the material type of the solid. Materials are defined with the Materials Task Page (p. 4883).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Frame Motion

enables the moving reference frame model for the cell zone. This option is suitable when the solid zone is moving, but the motion is not relative to an adjacent solid zone. See Defining Zone Motion (p. 1328) and Modeling Flows with Moving Reference Frames (p. 1769) for details.

Mesh Motion

enables the sliding mesh model for the cell zone. See Defining Zone Motion (p. 1328) and Setting Up the Sliding Mesh Problem (p. 1805) for details.

Solid Motion

enables the solid motion model for the cell zone. This option is suitable when the solid zone is moving relative to an adjacent solid zone. See Defining Zone Motion (p. 1328) for details.

Source Terms

enables the specification of a volumetric source of energy. When you turn on this option, the **Source Term** tab will allow you to specify the value for the energy source. See Defining Mass, Momentum, Energy, and Other Sources (p. 1382) for details.

Fixed Values

enables the fixing of the value of one or more variables in the solid zone, rather than computing them during the calculation. See Fixing the Values of Variables (p. 1377) for details.

Important:

You can fix the value of temperature only if you are using the pressure-based solver.

Participates In Radiation

specifies whether or not the solid zone participates in radiation. This option appears when you are using the DO model for radiation.

Reference Frame Tab

lists the parameters that define motion for a moving reference frame. For further details about the following, see Defining Zone Motion (p. 1328).

Relative Specification

provides the **Relative To Cell Zone** drop-down list, which allows you to indicate whether the velocities are absolute velocities (**absolute**) or are relative to the motion of a selected cell zone.

UDF

provides the **Zone Motion Function** drop-down list, which allows you to hook a `DEFINE_ZONE_MOTION` user-defined function that defines the zone motion.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Rotation-Axis Origin

specifies the origin for the axis of rotation of the solid zone. This item will appear only for 3D and 2D non-axisymmetric models.

Rotation-Axis Direction

specifies the direction vector for the solid zone's axis of rotation. This item will appear only for 3D models.

Rotational Velocity

contains an input field for the rotational **Speed** of the zone.

Translational Velocity

contains inputs for the **X**, **Y**, and **Z** velocities of the zone.

Copy to Mesh Motion

allows you to switch from frame motion to mesh motion. The variables used for the origin, axis, and velocity components, as well as for the `DEFINE_ZONE_MOTION` user-defined function, will be copied from one model to the other.

Mesh Motion Tab

lists the parameters that define motion for a moving reference frame. For further details about the following, see Defining Zone Motion (p. 1328).

Relative Specification

provides the **Relative To Cell Zone** drop-down list, which allows you to indicate whether the velocities are absolute velocities (**absolute**) or are relative to the motion of a selected cell zone.

UDF

provides the **Zone Motion Function** drop-down list, which allows you to hook a `DEFINE_ZONE_MOTION` user-defined function that defines the zone motion.

Rotation-Axis Origin

specifies the origin for the axis of rotation of the solid zone. This item will appear only for 3D and 2D non-axisymmetric models.

Rotation-Axis Direction

specifies the direction vector for the solid zone's axis of rotation. This item will appear only for 3D models.

Rotational Velocity

contains an input field for the rotational **Speed** of the zone.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Translational Velocity

contains inputs for the **X**, **Y**, and **Z** velocities of the zone.

Copy to Frame Motion

allows you to switch from mesh motion to frame motion. The variables used for the origin, axis, and velocity components, as well as for the `DEFINE_ZONE_MOTION` user-defined function, will be copied from one model to the other.

Solid Motion Tab

lists the parameters that define solid motion, when the solid zone is moving relative to an adjacent solid zone. For further details about the following, see Defining Zone Motion (p. 1328).

Relative Specification

provides the **Relative To Cell Zone** drop-down list, which allows you to indicate whether the velocities are absolute velocities (**absolute**) or are relative to the motion of a selected cell zone.

UDF

provides the **Solid Motion Function** drop-down list, which allows you to hook a `DEFINE_ZONE_MOTION` user-defined function that defines the zone motion.

Rotation-Axis Origin

specifies the origin for the axis of rotation of the solid zone. This item will appear only for 3D and 2D non-axisymmetric models.

Rotation-Axis Direction

specifies the direction vector for the solid zone's axis of rotation. This item will appear only for 3D models.

Rotational Velocity

contains an input field for the rotational **Speed** of the zone.

Translational Velocity

contains inputs for the **X**, **Y**, and **Z** velocities of the zone.

Copy to Mesh Motion

allows you to switch from solid motion to mesh motion. The variables used for the origin, axis, and velocity components, as well as for the `DEFINE_ZONE_MOTION` user-defined function, will be copied from one model to the other.

Source Term Tab

lists the parameters for volumetric source of energy.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Energy

displays the total number of energy sources used.

User Scalar n

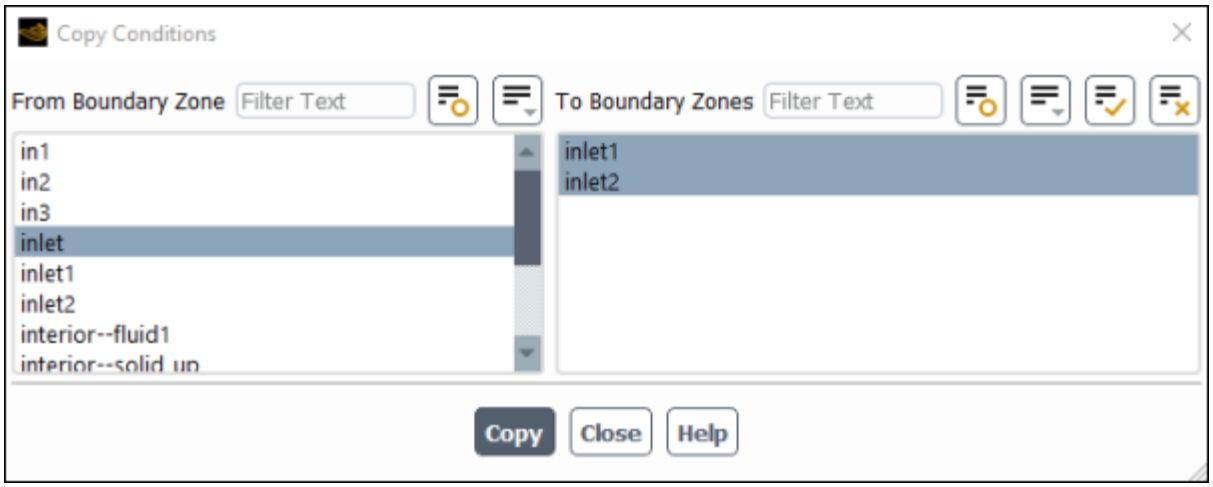
displays the total number of scalars used.

Fixed Values Tab

allows you to specify which variables in the solid zone are fixed (as a constant value or as defined by a profile or user-defined function), rather than computed during the calculation. See Fixing the Values of Variables (p. 1377) for details.

51.6.3. Copy Conditions Dialog Box

The **Copy Conditions** dialog box allows you to copy cell zone and/or boundary conditions from one zone/boundary to other zones/boundaries of the same type. It is opened either from the Cell Zone Conditions Task Page (p. 4958) or from the Boundary Conditions Task Page (p. 4991). See Copying Cell Zone and Boundary Conditions (p. 1305) for details.



Controls

From Zone

specifies the zone that has the conditions you want to copy.

To Zones

specifies the zone or zones to which you want to copy the conditions.

Phase

specifies the phase for which cell zone conditions or boundary conditions are being copied. This item appears if the VOF, mixture, or Eulerian multiphase model is being used. See Steps for Copying Cell Zone and Boundary Conditions (p. 2974) for details.

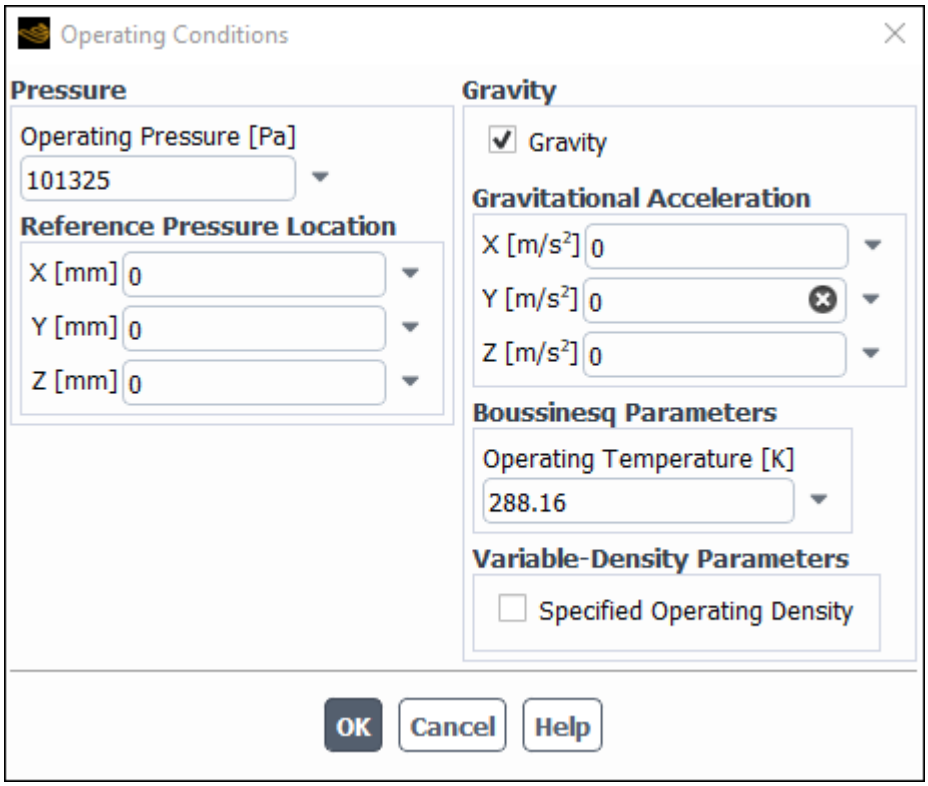
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Copy

copies the cell zone conditions or boundary conditions, setting all of the conditions for the zones selected in the **To Zones** list to be the same as the conditions for the zone selected in the **From Zone** list.

51.6.4. Operating Conditions Dialog Box

The **Operating Conditions** dialog box allows you to set parameters related to operating conditions in your model.



Controls

Pressure

contains items related to the modeling of pressure.

Floating Operating Pressure

specifies the use of a floating operating pressure. See Floating Operating Pressure (p. 1763) for details. This option appears only for time-dependent compressible flows that use the ideal or real gas model.

Operating Pressure

sets the operating pressure for the problem. For all flows, Ansys Fluent uses gauge pressure internally. Any time an absolute pressure is needed, it is generated by adding the operating pressure to the relative pressure. See Operating Pressure (p. 1677) for a detailed description of operating pressure and how to set it.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reference Pressure Location

sets the location of the cell whose pressure value is used to adjust the gauge pressure field for incompressible flows that do not involve any pressure boundaries. See Using a Reference Pressure to Adjust the Gauge Pressure Field (p. 1679) for details.

Note:

The pressure at the reference pressure location is used to correct the gauge pressure. This is not related to the reference pressure quantity in postprocessing.

Real Gas State

allows you to specify the operating conditions in the subcritical regime of your model. Note that if the operating conditions in your model are entirely in the supercritical regime, this setting will have no effect.

Note:

This option appears only when a Cubic Equation of State Real Gas model is chosen.

Vapor

is the default option, and therefore assumes that the real gas state is vapor.

Liquid

assumes that the real gas state is liquid.

Gravity

contains inputs for gravitational acceleration, the Boussinesq model, and variable density.

Gravity

enables the specification of gravity.

Gravitational Acceleration

sets the x, y, and z components of the gravitational acceleration vector. (The z component is available only in 3D solvers.) See Buoyancy-Driven Flows and Natural Convection (p. 1387) for details about buoyancy-driven flows. This option appears only when **Gravity** is enabled.

Boussinesq Parameters

contains inputs related to the Boussinesq model. This option appears only if **Energy** (in the Energy Dialog Box (p. 4714)) and **Gravity** are enabled. See The Boussinesq Model (p. 1388) for more information on the Boussinesq model.

Operating Temperature

sets the operating temperature (T_0 in Equation 7.67 (p. 1388)) for use with the Boussinesq approximation.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Variable-Density Parameters

contains inputs related to the modeling of variable density. This option appears only when **Gravity** is enabled.

Specified Operating Density

enables the specification of operating density. See If you are using the incompressible ideal gas law, check the Operating Density (p. 1391) for details. This item is available for single-phase flows only.

Operating Density Method

specifies which method will be used to calculate the operating density. You can choose **minimum-phase-averaged**, **primary-phase-averaged**, **mixture-averaged**, or **user-input**. For information about these methods, see Modeling Buoyancy-Driven Multiphase Flow (p. 2930).This item is available for buoyancy-driven multiphase flows only.

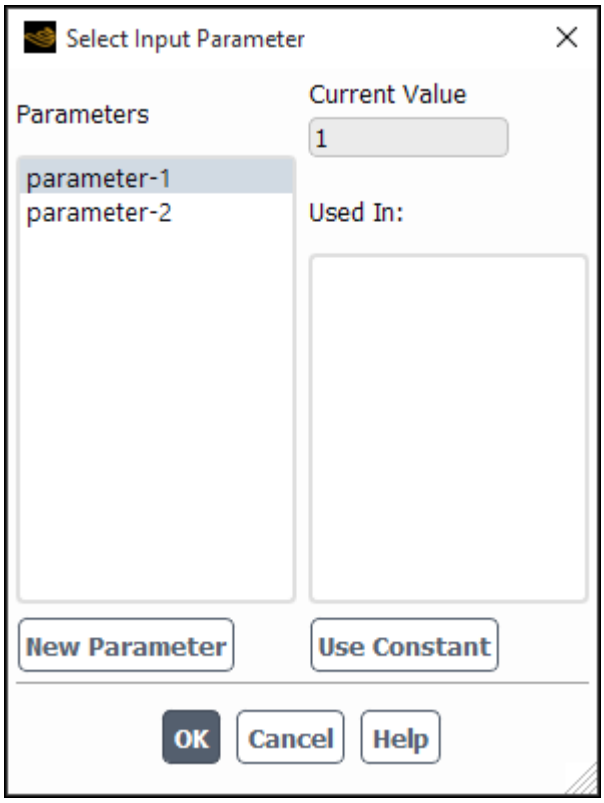
Operating Density

sets the operating density (ρ_0 in Equation 7.68 (p. 1391)). This item is available when either **Specified Operating Density** is enabled for a single-phase flow or the **user-input** operating density method is selected for a multiphase flow.

51.6.5. Select Input Parameter Dialog Box

The **Select Input Parameter** dialog box allows you to choose from a listing of existing input parameters as well as to create and define new input parameters. For more information about parameters, see Defining and Viewing Parameters (p. 1309) in the User's Guide, and see Working With Input and Output Parameters in Workbench in the Ansys Fluent in Workbench User's Guide.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Parameters

contains a list of existing compatible input parameters

Current Value

displays the value of the currently selected parameter.

Used In

lists any variables that are already associated with the currently selected parameter.

Use Constant

allows you to change the associated parameter to a constant (that is, real) value.

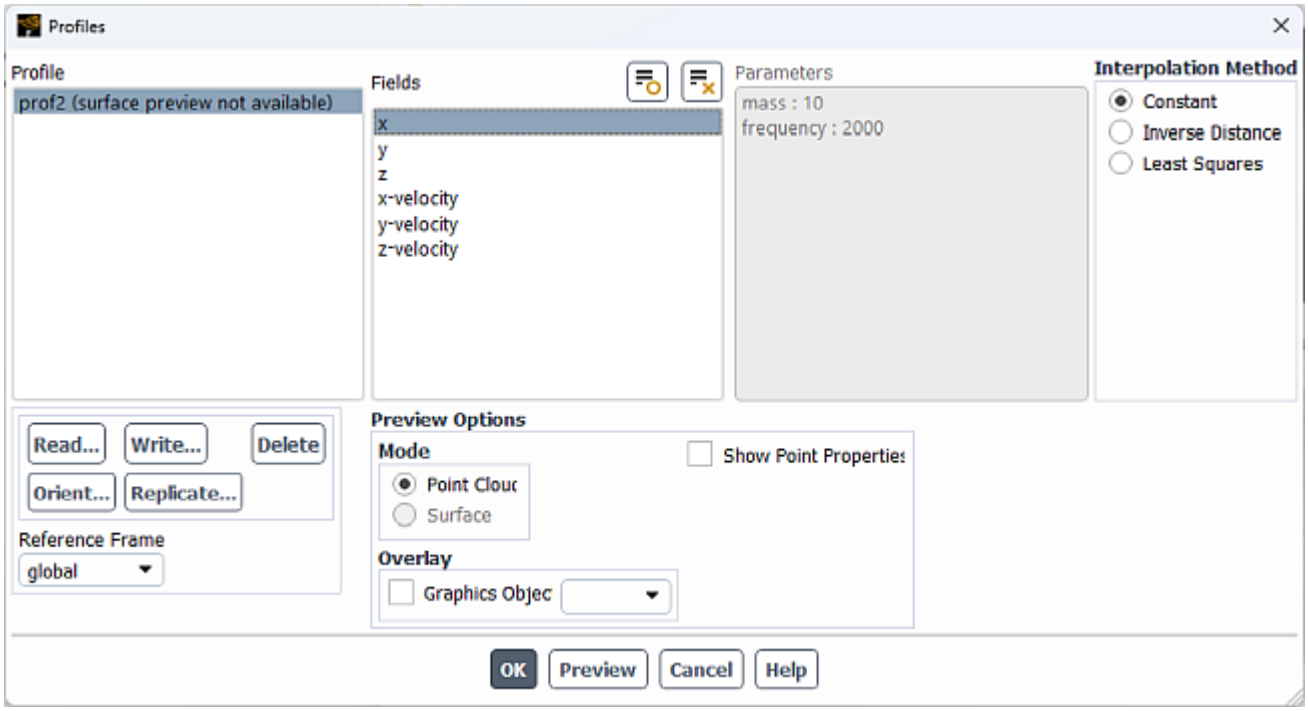
New Parameter

opens the Parameter Expression Dialog Box (p. 5343), in which you can assign names and values to an input parameter.

51.6.6. Profiles Dialog Box

The **Profiles** dialog box allows you to define new profiles by reading cell zone and boundary profile files. You can also examine the existing profile definitions and delete unused profiles. See Profiles (p. 1564) for details about cell zone and boundary profiles.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Profile

contains a selectable list of available profiles. When a profile is selected its available fields are displayed under **Fields**. Parameters, if included in the profile, are displayed under **Parameters**.

Fields

displays the fields available in the selected profile. After the profile file has been read, these fields will also appear in any boundary condition dialog box (for example, the **Velocity Inlet** dialog box) that allows profile specification of a variable. To the right of (or below) the variable in the boundary conditions dialog box, there will be a drop-down list that contains a **constant** and the fields from available profile files. To use a particular profile field, just select it from the list.

Parameters

contains a read-only list of parameters included in the selected CSV profile (when applicable).

Interpolation Method

allows you to select the interpolation method for the profile selected from the **Profile** list. This selection is only available for point profiles, and will only take effect when the **Apply** button is clicked. The choices include the following:

Constant

specifies that Ansys Fluent should use zeroth-order interpolation to assign the point profile values to the nearest cell faces at the boundary. This is the default selection.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Inverse Distance

specifies that Ansys Fluent should assign a value to each cell face at the boundary based on weighted contributions from the values in the point profile file. The weighting factor is inversely proportional to the distance between the profile point and the cell face center.

Least Squares

specifies that Ansys Fluent should assign values to the cell faces at the boundary through a first-order interpolation method that tries to minimizes the sum of the squares of the offsets (residuals) between the profile data points and the cell face centers.

Reference Frame

allows you to attach the profile to a reference frame. If you select a reference frame other than **global**, then the profile will follow the orientation and/or motion of the specified reference frame.

Read...

opens The Select File Dialog Box (p. 931) so that you can read a boundary profile file. If a profile in the file has the same name as an existing profile, the old profile will be overwritten.

Write...

opens the Write Profile Dialog Box (p. 4988), in which you can save profile data.

Delete

deletes the selected profile from memory.

Orient...

opens the Orient Profile Dialog Box (p. 4986), in which you can reorient and scale the profile. This item appears only in 3D.

Replicate...

opens the Replicate Profile Dialog Box (p. 4984), in which you can reorient and scale the profile. This item appears only in 3D.

Preview Options

contains options for previewing profiles as a cloud of points or as a surface.

Point Cloud

preview the profile as a cloud of points.

Surface

preview the profile as a surface. *This option requires that the profile contain node-connectivity data. Profile that do not have this data are indicated as such in the* ***Profile*** *list.*

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Show Point Properties

optionally expands additional settings for how points are displayed when previewing the profile as a point cloud.

Symbol

specifies the symbol used to represent the points in the point cloud preview.

Size

controls the size of the points in the point cloud preview.

Color

specifies the color of the points in the point cloud preview.

Overlay Graphics Object

allows you to overlay any defined mesh or contour plots when previewing a profile as either a surface or a point cloud.

OK

sets the selection made in the **Interpolation Method** list for point profiles in preparation for interpolation. The profile is not actually interpolated until a profile field is selected in a boundary condition dialog box (for example, the **Velocity Inlet** dialog box) and the solution is initialized.

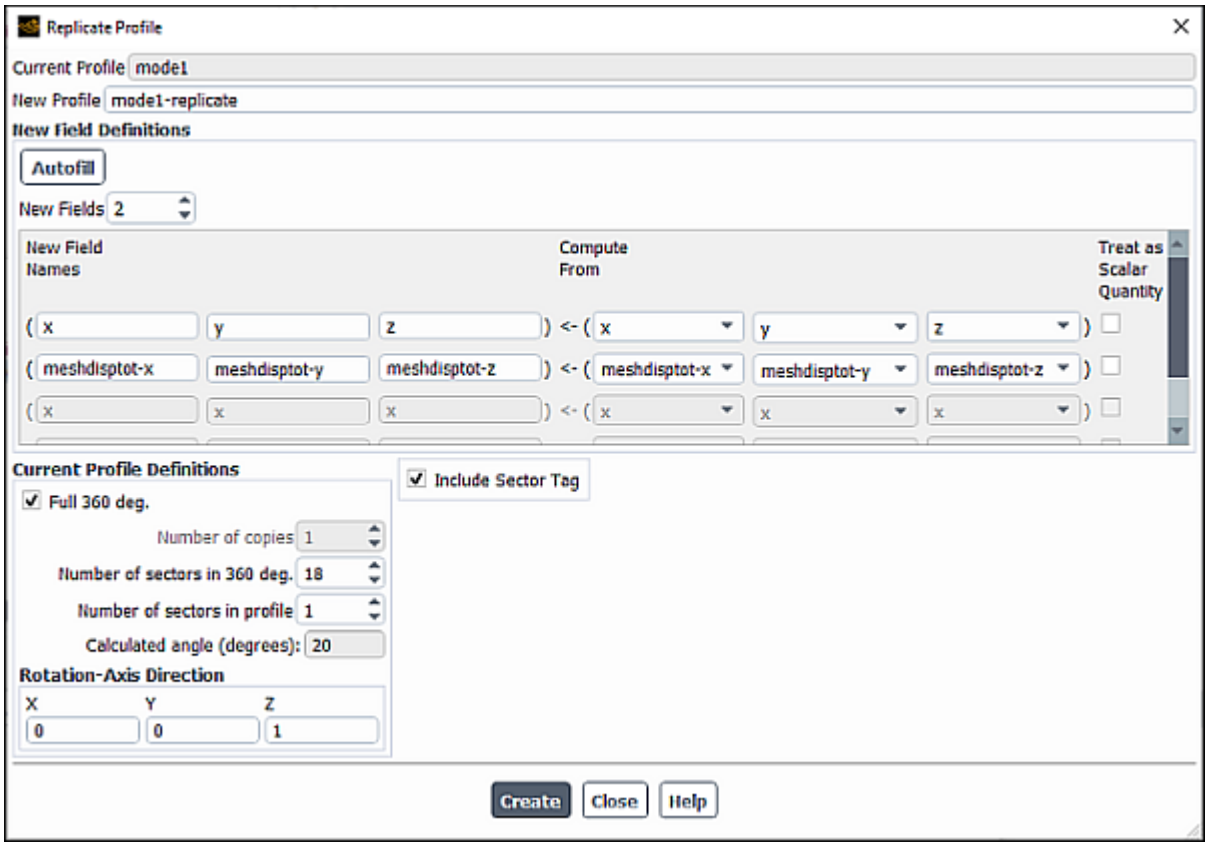
Preview

displays the selected profile as either a point cloud or a surface, depending on which **Mode** is selected.

51.6.7. Replicate Profile Dialog Box

The **Replicate Profile** dialog box allows you to replicate a profile periodically, typically for use as a boundary condition in a turbomachinery case. See Replicating Profiles (p. 1581) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Current profile

shows the name of the currently selected profile in the Profiles Dialog Box (p. 4981). This is the profile on which the new profile will be based.

New Profile

sets the name of the new cell zone or boundary profile.

New Field Definitions

contains inputs and controls for the definition of the vector and scalar fields in the new profile.

New Fields

sets the number of data fields in the new profile.

New Field Names

contains inputs for the names of the data fields in the new profile. For a vector field, all 3 inputs in each row will be active; for a scalar field, only the first will be active.

Compute From...

contains drop-down lists with the names of the fields in the **Current profile**. In these lists, select the fields from which the **New Field Names** will be computed.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Treat as Scalar Quantity

indicates (if on) that the adjacent field is a scalar quantity. If this option is off, the field is a vector quantity.

Current Profile Definitions

contains inputs for the replication of the new profile.

Full 360 deg.

automatically determines the number of copies of the original profile that are needed to make the full 360 degree profile based on the other information you specify

Number of copies

specifies how many copies of the original profile are created.

Number of sectors in 360 deg.

the number of periodic sections, based on the original profile, that are present in 360 degrees.

Number of sectors in profile

the number of periodic sections in the original profile.

Calculated angle

(for display only) the angle occupied by the original profile.

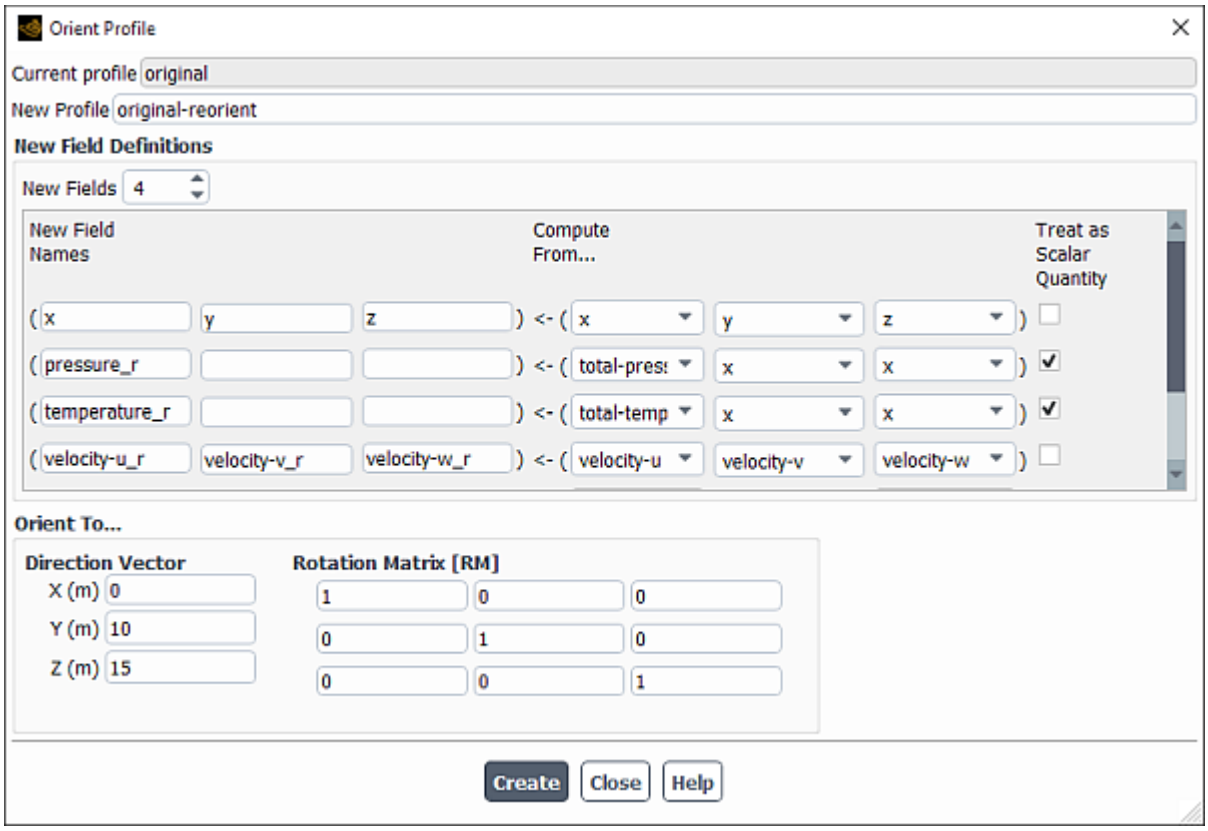
Create

creates a new profile using the information specified in the dialog box.

51.6.8. Orient Profile Dialog Box

The **Orient Profile** dialog box allows you to reorient a profile so that you can apply it to a particular boundary. See Reorienting Profiles (p. 1576) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Current profile

shows the name of the currently selected profile in the Profiles Dialog Box (p. 4981). This is the profile on which the new profile will be based.

New Profile

sets the name of the new cell zone or boundary profile.

New Field Definitions

contains inputs and controls for the definition of the vector and scalar fields in the new profile.

New Fields

sets the number of data fields in the new profile.

New Field Names

contains inputs for the names of the data fields in the new profile. For a vector field, all 3 inputs in each row will be active; for a scalar field, only the first will be active.

Compute From...

contains drop-down lists with the names of the fields in the **Current profile**. In these lists, select the fields from which the **New Field Names** will be computed.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Treat as Scalar Quantity

indicates (if on) that the adjacent field is a scalar quantity. If this option is off, the field is a vector quantity.

Orient To...

contains inputs for the definition of the local coordinate system for the new profile. This coordinate system will determine the orientation of the profile.

Direction Vector

is the vector that translates a cell zone or boundary profile to the new position, and is defined between the centers of the profile fields.

Rotation Matrix [RM

] specifies the rotational matrix RM, which is based on Euler angles (γ, β, and α) that define an orthogonal system $x'y'z'$ as the result of the three successive rotations from the original system xyz. See Reorienting Profiles (p. 1576) for more information.

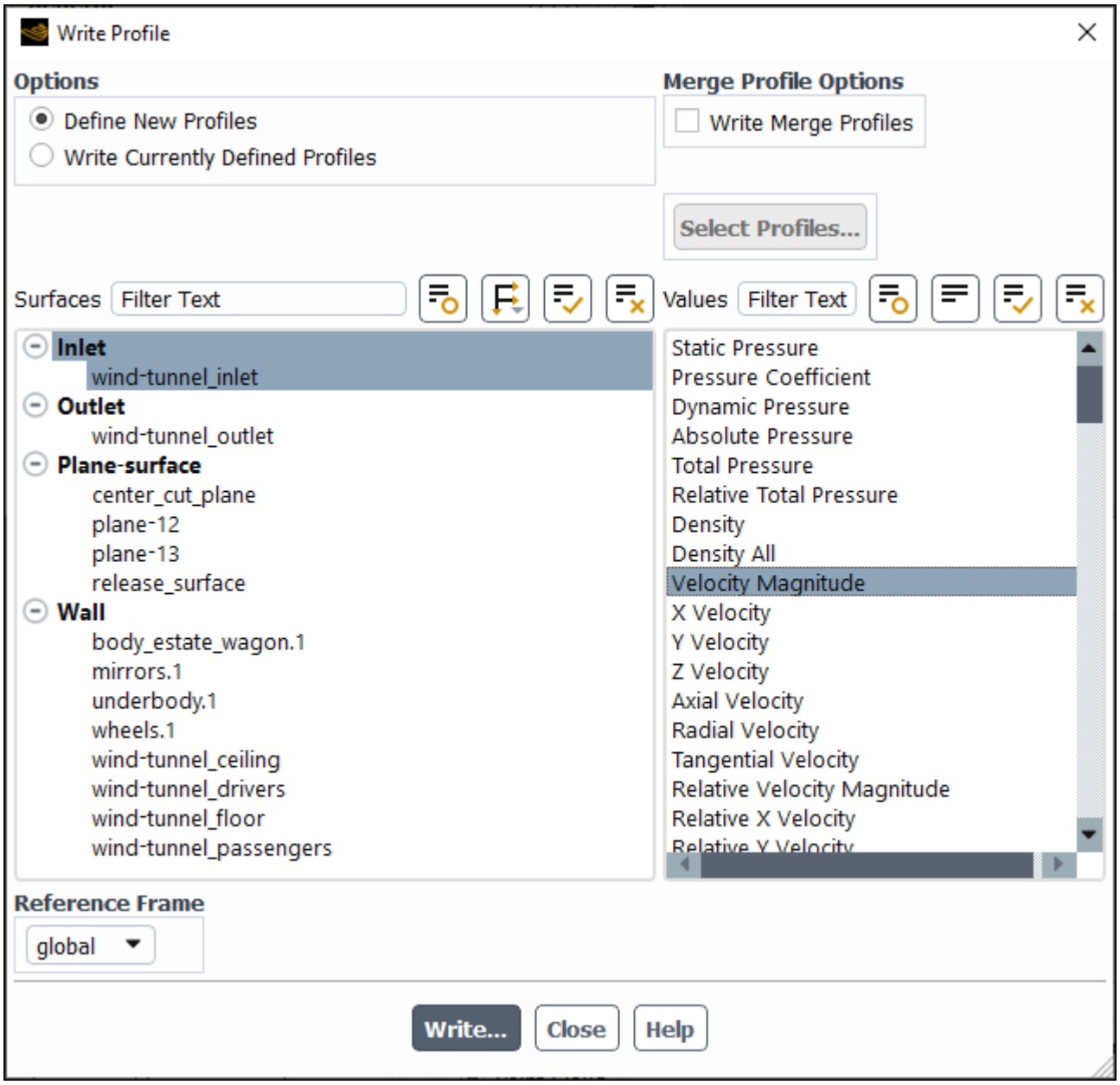
Create

creates a new profile using the information specified in the dialog box.

51.6.9. Write Profile Dialog Box

The **Write Profile** dialog box allows you to create a profile file from the conditions on a specified cell zone or boundary/surface. See Writing Profile Files (p. 964) for details on writing profile files.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options

contains options for writing profiles.

Define New Profiles

enables the creation of a profile file from the conditions on a specified boundary or surface.

Write Currently Defined Profiles

enables the creation of a profile file containing all profiles that are currently defined.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Merge Profile Options

Write Merge Profiles

(only available if two or more surfaces are selected) writes a `.csv` file with the selected surfaces consolidated into one set of data points.

Select Profile...

opens the Select Profiles for Writing (p. 4990), allowing you to specify which profiles are written.

Surfaces

contains a selectable list of surfaces where data can be extracted to write a profile.

Values

contains a selectable list of variables that you use for creating profiles.

Reference Frame

selectable list of reference frames that specifies which reference frame will be used in writing the profile.

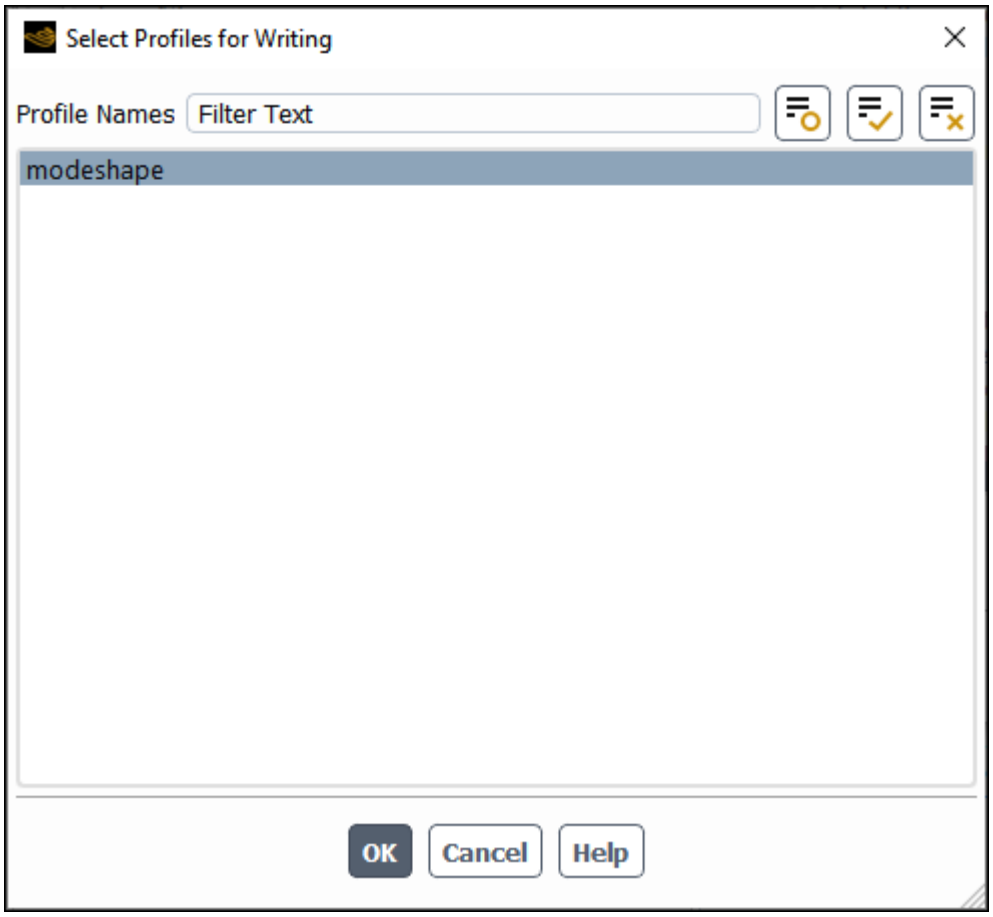
Write...

opens The Select File Dialog Box (p. 931), where you can specify a filename for the profile file.

51.6.10. Select Profiles for Writing

The **Select Profiles for Writing** dialog box allows you to select which of the profiles associated with the current case are written when you click **Write...** in the Write Profile Dialog Box (p. 4988) See Writing Profile Files (p. 964) for details on writing profile files.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

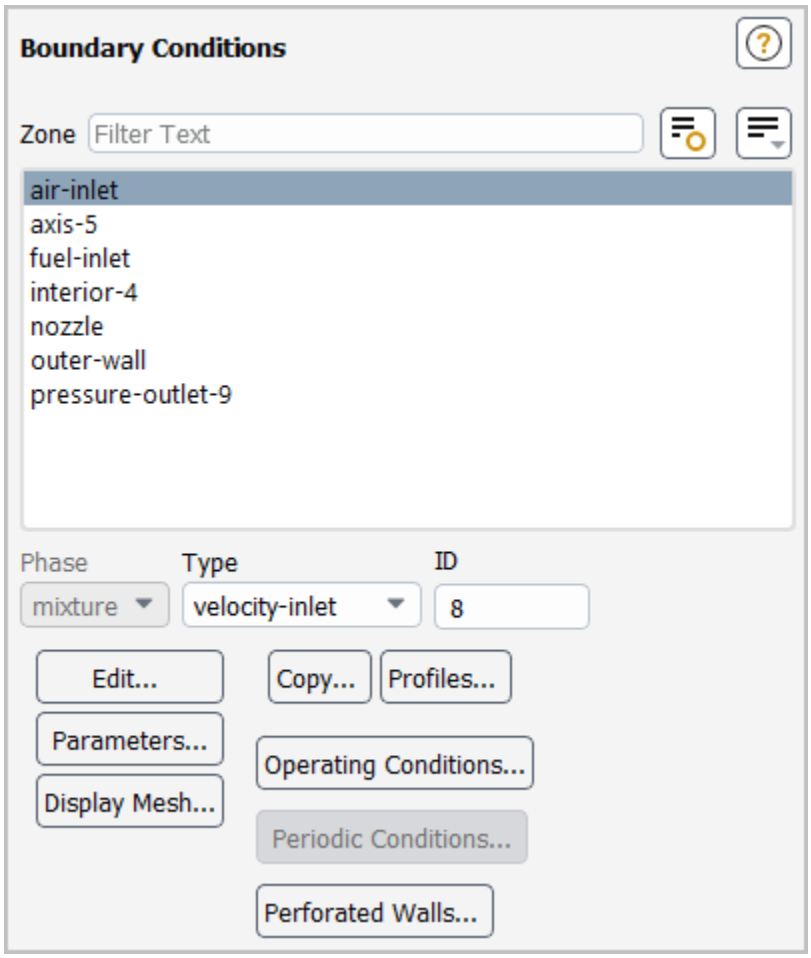
Profile Names

selectable list of profiles associated with the currently-loaded case that allows you to chose which profile(s) is written when you click **Write...** in the Write Profile Dialog Box (p. 4988).

51.7. Boundary Conditions Task Page

The **Boundary Conditions** task page allows you to set the type of a boundary and display other dialog boxes to set the boundary condition parameters for each boundary.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Zone

contains a selectable list of boundary zones from which you can select the zone of interest. You can check a zone type by using the mouse probe (see Controlling the Mouse Button Functions (p. 4020)) on the displayed physical mesh. This feature is particularly handy if you are setting up a problem for the first time, or if you have two or more boundary zones of the same type and you want to determine the zone IDs. To do this you must first display the mesh with the Mesh Display Dialog Box (p. 4688). Then click the boundary zone with the right (select) mouse button. Ansys Fluent will print the zone ID and type of that boundary zone in the console window.

Phase

specifies the phase for which conditions at the selected boundary **Zone** are being set. This item appears if the VOF, mixture, or Eulerian multiphase model is being used. See Defining Multiphase Cell Zone and Boundary Conditions (p. 2957) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Type

contains a drop-down list of boundary condition types for the selected zone. The list contains all possible types to which the zone can be changed.

Important:

Note that you cannot use this method to change zone types to or from the periodic type, since additional restrictions exist for this boundary type. Creating Periodic Zones and Interfaces (p. 1269) explains how to create and uncouple periodic zones.

ID

displays the zone ID number of the selected zone. (This is for informational purposes only; you cannot edit this number.)

Edit...

opens the appropriate dialog box for setting the boundary conditions for that particular boundary type.

Copy...

opens the Copy Conditions Dialog Box (p. 4977), which allows you to copy boundary conditions from one zone to other zones of the same type. See Copying Cell Zone and Boundary Conditions (p. 1305) for details.

Profiles...

opens the Profiles Dialog Box (p. 4981).

Parameters...

opens the Parameters Dialog Box (p. 5338).

Operating Conditions...

opens the Operating Conditions Dialog Box (p. 4978).

Display Mesh...

opens the Mesh Display Dialog Box (p. 4688).

Periodic Conditions...

opens the Periodic Conditions Dialog Box (p. 5104).

Perforated Walls...

opens the **Perforated Walls** Dialog Box (p. 5105).

For additional information, see the following sections:

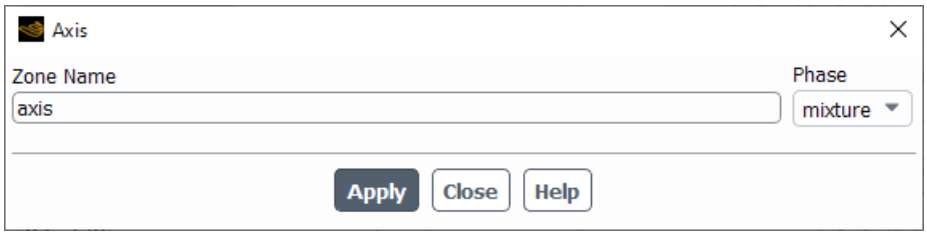
51.7.1. Axis Dialog Box

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

51.7.2. Degassing Dialog Box
51.7.3. Exhaust Fan Dialog Box
51.7.4. Fan Dialog Box
51.7.5. Inlet Vent Dialog Box
51.7.6. Intake Fan Dialog Box
51.7.7. Interface Dialog Box
51.7.8. Interior Dialog Box
51.7.9. Mass-Flow Inlet Dialog Box
51.7.10. Mass-Flow Outlet Dialog Box
51.7.11. Outflow Dialog Box
51.7.12. Outlet Vent Dialog Box
51.7.13. Overset Dialog Box
51.7.14. Periodic Dialog Box
51.7.15. Porous Jump Dialog Box
51.7.16. Pressure Far-Field Dialog Box
51.7.17. Pressure Inlet Dialog Box
51.7.18. Pressure Outlet Dialog Box
51.7.19. Radiator Dialog Box
51.7.20. RANS/LES Interface Dialog Box
51.7.21. Symmetry Dialog Box
51.7.22. Velocity Inlet Dialog Box
51.7.23. Wall Dialog Box
51.7.24. Periodic Conditions Dialog Box
51.7.25. Perforated Walls Dialog Box

51.7.1. Axis Dialog Box

The **Axis** dialog box can be used to modify the name of an axis zone; there are no conditions to be set. It is opened from the Boundary Conditions Task Page (p. 4991). See Axis Boundary Conditions (p. 1504) for information about axis boundaries.



Controls

Zone Name

sets the name of the zone.

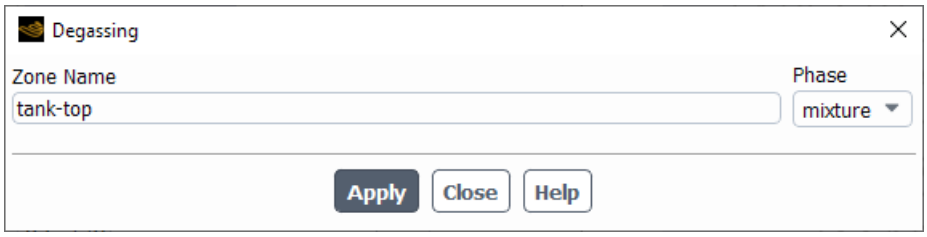
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Phase

displays the name of the phase. This item appears if the VOF, mixture, or Eulerian multiphase model is being used.

51.7.2. Degassing Dialog Box

The **Degassing** dialog box can be used to modify the name of a degassing zone; there are no conditions to be set. It is opened from the Boundary Conditions Task Page (p. 4991). See Degassing Boundary Conditions (p. 1456) for information about axis boundaries.



Controls

Zone Name

sets the name of the zone.

Phase

displays the name of the phase.

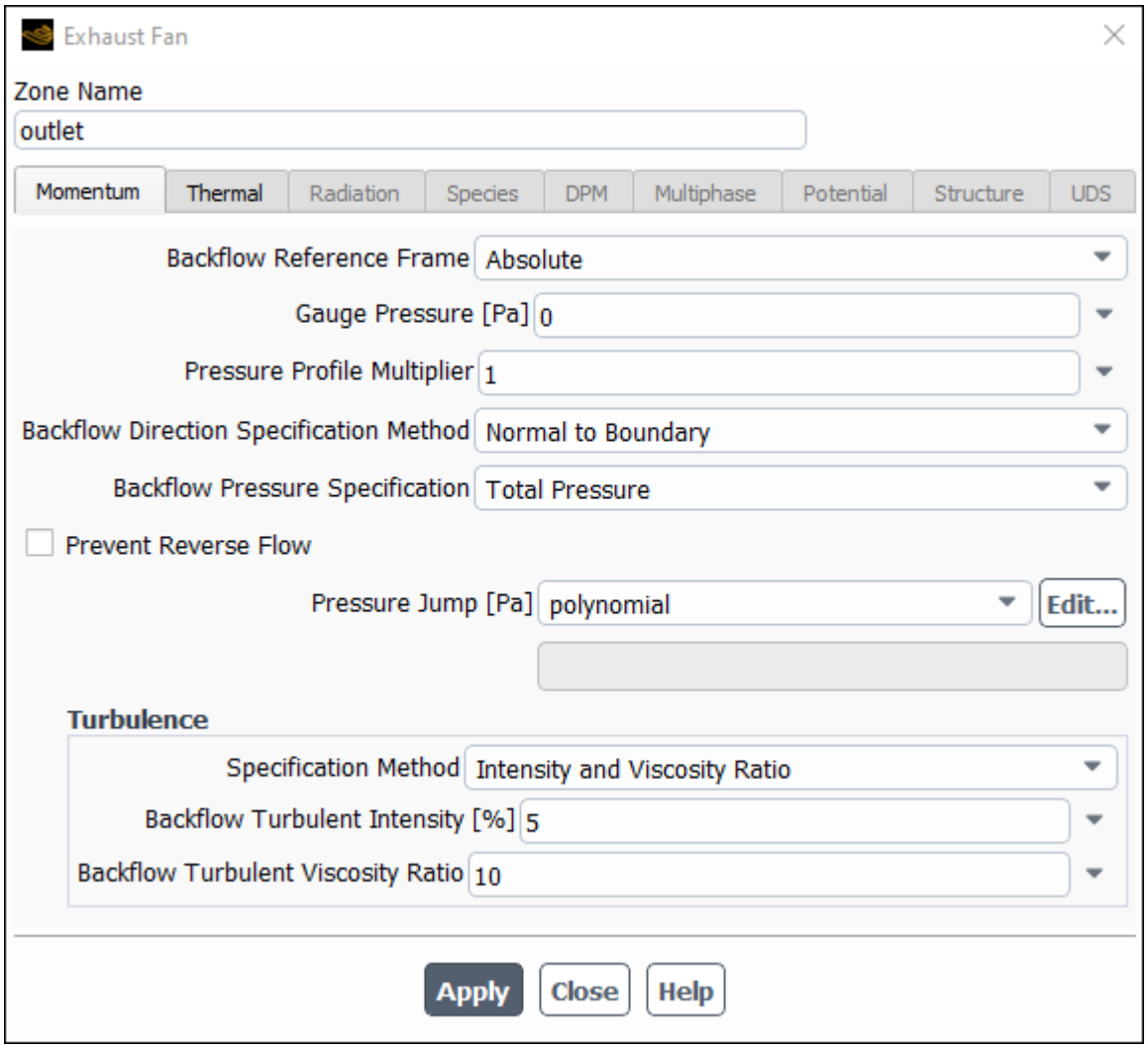
51.7.3. Exhaust Fan Dialog Box

The **Exhaust Fan** dialog box sets the boundary conditions for an exhaust fan zone. It is opened from the Boundary Conditions Task Page (p. 4991). See Inputs at Exhaust Fan Boundaries (p. 1454) for details about defining the items below.

Important:

This feature offers reduced functionality when running Fluent under the **Pro** capability level.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Zone Name

sets the name of the zone.

Phase

displays the name of the phase. It appears only for multiphase flows.

Momentum

contains the momentum parameters.

Backflow Reference Frame

specify whether backflow temperature, pressure, and flow directions are in the **Absolute** or **Relative to the Adjacent Cell Zone** reference frame.

Gauge Pressure

sets the gauge pressure at the outlet boundary.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Backflow Direction Specification Method

sets the direction of the inflow stream should the flow reverse direction. You can choose **Direction Vector**, **Normal to Boundary**,or **From Neighboring Cell**.

Coordinate System

contains a drop-down list for selecting the coordinate system. You can choose **Cartesian**, **Cylindrical**, or **Local Cylindrical**. This option is available only when **Direction Vector** is selected from the **Backflow Direction Specification Method** drop-down list.

X-, Y-, Z-Component of Flow Direction

allows you to specify the velocity components in x, y, and z directions respectively. This option is available when **Cartesian** is selected for the **Coordinate System**.

Radial-, Tangential-, Axial-Component of Flow Direction

set the direction of the flow at the boundary. These items will appear for 2D axisymmetric cases, or for 3D cases for which the selected **Coordinate System** is **Cylindrical** or **Local Cylindrical**.

Backflow Pressure Specification

specifies how the pressure is calculated under backflow conditions. If you select **Static Pressure**, the **Gauge Pressure** is directly imposed as the boundary face pressure; if you select **Total Pressure**, the **Gauge Pressure** will be combined with a dynamic contribution that is based on the velocity in the adjacent cell zone.

Pressure Jump

specifies the rise in pressure across the fan. See Specifying the Pressure Jump (p. 1455) for details.

Axis Origin

sets the **X**, **Y**, and **Z** coordinates of the origin of the local cylindrical coordinate system.

Axis Direction

sets the **X**, **Y**, and **Z** components of the direction of the local cylindrical coordinate system.

Turbulence

displays the turbulence parameters.

Specification Method

specifies which method will be used to define the turbulence parameters. You can choose **K and Epsilon** (k- ε models and RSM only), **K and Omega** (k- ω models only), **Intensity and Length Scale**, **Intensity and Viscosity Ratio**, **Intensity and Hydraulic Diameter**, **Modified Turbulent Viscosity** (Spalart-Allmaras model only), or **Turbulent Viscosity Ratio** (Spalart-Allmaras model only). See Determining Turbulence Parameters (p. 1395) for information about the inputs for each of these methods. (This item will appear only for turbulent flow calculations.)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Backflow Turbulent Kinetic Energy, Backflow Turbulent Dissipation Rate

set values for the turbulence kinetic energy k and its dissipation rate ε. These items will appear if you choose **K and Epsilon** as the **Specification Method**.

Backflow Turbulent Kinetic Energy, Backflow Specification Dissipation Rate

set values for the turbulence kinetic energy k and its specific dissipation rate ω. These items will appear if you choose **K and Omega** as the **Specification Method**.

Backflow Turbulent Intensity, Backflow Turbulent Length Scale

set values for turbulence intensity I and turbulence length scale ℓ. These items will appear if you choose **Intensity and Length Scale** as the **Specification Method**.

Backflow Turbulent Intensity, Backflow Turbulent Viscosity Ratio

set values for turbulence intensity I and turbulent viscosity ratio μ_t / μ. These items will appear if you choose **Intensity and Viscosity Ratio** as the **Specification Method**.

Backflow Turbulent Intensity, Backflow Hydraulic Diameter

set values for turbulence intensity I and hydraulic diameter L. These items will appear if you choose **Intensity and Hydraulic Diameter** as the **Specification Method**.

Backflow Modified Turbulent Viscosity

sets the value of the backflow modified turbulent viscosity $\tilde{\nu}$. This item will appear if you choose **Modified Turbulent Viscosity** as the **Specification Method**.

Backflow Turbulent Viscosity Ratio

sets the value of the backflow turbulent viscosity ratio μ_t / μ. This item will appear if you choose **Turbulent Viscosity Ratio** as the **Specification Method**.

Reynolds-Stress Specification Method

specifies which method will be used to determine the backflow Reynolds stress boundary conditions when the Reynolds stress turbulence model is used. You can choose either **K or Turbulence Intensity** or **Reynolds-Stress Components**. If you choose the former, Ansys Fluent will compute the Reynolds stresses for you. If you choose the latter, you will explicitly specify the Reynolds stresses yourself. See Reynolds Stress Model (p. 2144) for details. (This item will appear only for RSM turbulent flow calculations.)

Backflow UU, VV, WW, UV, VW, UW Reynolds Stresses

specify the backflow Reynolds stress components when **Reynolds-Stress Components** is chosen as the **Reynolds-Stress Specification Method**.

Thermal

contains the thermal parameters. This parameter is available only when the energy equation is turned on.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Backflow Total Temperature

sets the total temperature of the inflow stream should the flow reverse direction.

Radiation

contains the boundary conditions for the radiation model at the exhaust fan.

External Black Body Temperature Method, Internal Emissivity

set the radiation boundary conditions when you are using the P-1, DTRM, DO, S2S, or MC models for radiation heat transfer. See Defining Boundary Conditions for Radiation (p. 2217) for details.

Participates in Solar Ray Tracing

specifies whether or not the fan participates in solar ray tracing.

Solar Transmissivity Factor

specifies a multiplier (ranging from 0 to 1) that is applied to the solar irradiation entering the domain through the fan.

Participates in View Factor Calculation

specifies whether or not the fan participates in the view factor calculation as part of the S2S radiation model. This parameter is available only if you select the **Surface to Surface** radiation model.

Species

contains the species parameters.

Specify Species in Mole Fractions

allows you to specify the species in mole fractions rather than mass fractions.

Mean Mixture Fraction, Mixture Fraction Variance

set inlet values for the PDF mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model.)

Secondary Mean Mixture Fraction, Secondary Mixture Fraction Variance

set inlet values for the secondary mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model with two mixture fractions.)

Species Mass Fractions

contains inputs for the mass fractions of defined species. See Defining Cell Zone and Boundary Conditions for Species (p. 2394) for details about these inputs. These items will appear only if you are modeling non-reacting multi-species flow or you are using the finite-rate reaction formulation.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Backflow Progress Variable

sets the value of the progress variable for premixed turbulent combustion. See Setting Boundary Conditions for the Progress Variable (p. 2541) for details.

This item will appear only if the premixed or partially premixed combustion model is used.

DPM

contains the discrete phase parameters. This tab is available only if you have defined at least one injection.

Discrete Phase BC Type

sets the way that the discrete phase behaves with respect to the boundary. This item appears when one or more injections have been defined.

reflect

rebounds the particle off the boundary with a change in its momentum as defined by the coefficient of restitution (see Particle Reflection at Wall in the *Fluent Theory Guide*).

trap

terminates the trajectory calculations and records the fate of the particle as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See Figure 24.40: "Trap" Boundary Condition for the Discrete Phase (p. 2783).

escape

reports the particle as having "escaped" when it encounters the boundary. Trajectory calculations are terminated. See Figure 24.41: "Escape" Boundary Condition for the Discrete Phase (p. 2783).

reinject

reintroduces the particle into the domain when it reaches a certain domain boundary (for example, outlet). This item appears when one or more injections have been defined and **Unsteady Particle Tracking** is enabled in the **Discrete Phase Model** dialog box. See The **reinject** Boundary Condition (p. 2784) for details.

wall-jet

indicates that the direction and velocity of the droplet particles are given by the resulting momentum flux, which is a function of the impingement angle. See Figure 12.6: "Wall Jet" Boundary Condition for the Discrete Phase in the Theory Guide.

user-defined

specifies a user-defined function to define the discrete phase boundary condition type.

Discrete Phase BC Function

sets the user-defined function from the drop-down list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Multiphase

contains the multiphase parameters.

Backflow Granular Temperature

specifies temperature for the solids phase and is proportional to the kinetic energy of the random motion of the particles.

Volume Fraction Specification Method

sets the method used to specify the volume fraction of the secondary phase selected in the Boundary Conditions Task Page (p. 4991). This section of the dialog box will appear when one of the multiphase models is being used. See Defining Multiphase Cell Zone and Boundary Conditions (p. 2957) for details.

Backflow Volume Fraction

specifies the volume fraction of the secondary phase as a constant, profile, of UDF function.

From Neighboring Cell

calculates the volume fraction from the neighboring cells.

Potential

displays the boundary conditions for the electric potential field. This tab is available only if you have enabled either the **Electric Potential model** or the **Electrochemical** reaction model in the Species Dialog Box (p. 4929).

Potential Boundary Condition

is a drop-down list of available potential boundary condition types for the potential generated by electron current: **Specified Flux** and **Specified Value**. For the **Specified Flux** boundary condition, you will need to specify **Current Density** at the wall. For the **Specified Value** boundary condition, you will need to specify **Potential** at the wall.

Electrolyte Potential Boundary Condition

is a drop-down list of available boundary condition types for the potential generated by ionic current. This item is similar to the **Potential Boundary Condition** drop-down list described above and is available only for the Electrolysis and H2 Pump model.

UDS

contains the UDS parameters.

User-Defined Scalar Boundary Condition

appears only if user-defined scalars are specified.

User Scalar n

specifies whether the scalar is a specified flux or a specified value.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

User-Defined Scalar Boundary Value

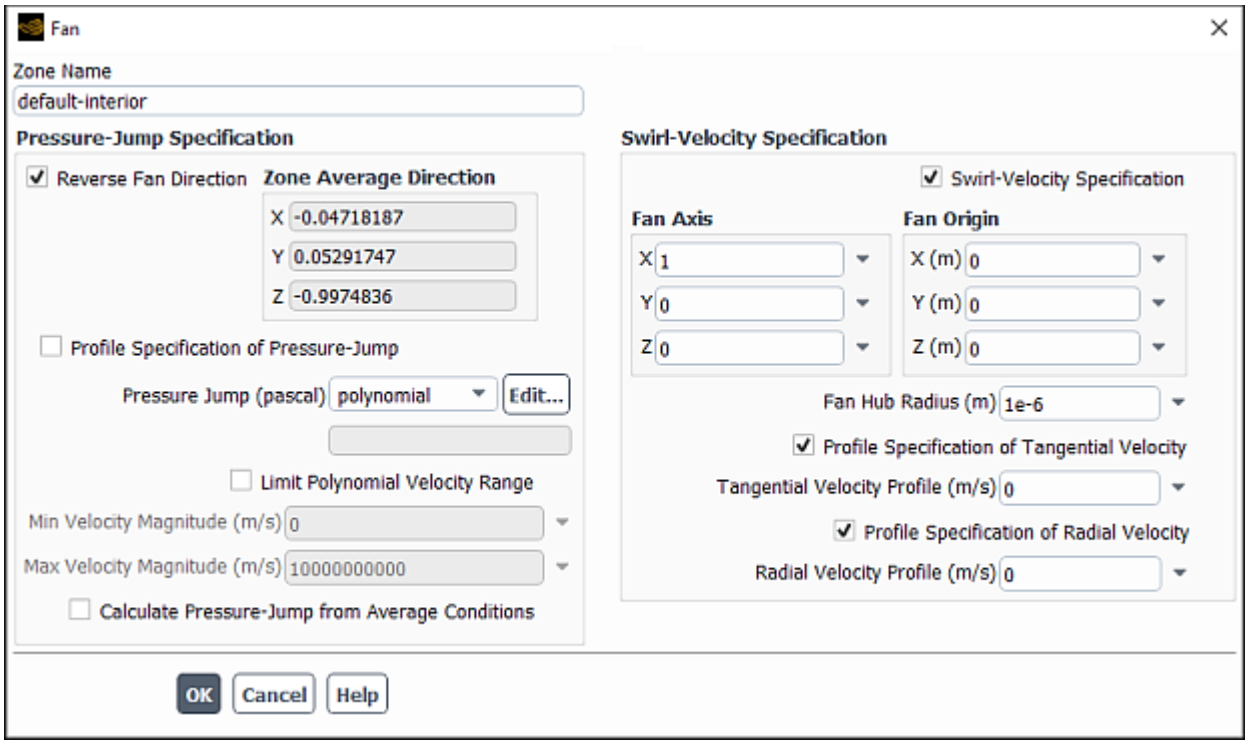
appears only if user-defined scalars are specified.

User Scalar n

specifies the value of the scalar.

51.7.4. Fan Dialog Box

The **Fan** dialog box sets the boundary conditions for a fan zone. It is opened from the Boundary Conditions Task Page (p. 4991). See User Inputs for Fans (p. 1506) for details about the items below.



Controls

Zone Name

sets the name of the zone.

Pressure-Jump Specification

contains inputs that define the pressure jump across the fan.

Reverse Fan Direction

sets the fan flow direction relative to the zone direction. If **Zone Average Direction** is pointing in the direction you want the fan to blow, do *not* select **Reverse Flow**; if it is pointing in the opposite direction, select **Reverse Flow**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Zone Average Direction

displays the (face-averaged) direction vector for the zone as an aid in determining whether or not you want to select **Reverse Flow**.

Profile Specification of Pressure-Jump

enables the use of a boundary profile or user-defined function for the pressure jump specification. See Profiles (p. 1564) or the Fluent Customization Manual for details. When this option is enabled, **Pressure Jump Profile** will appear in the dialog box and the next four items below it will not.

Pressure Jump Profile

contains a drop-down list from which you can select a boundary profile or a user-defined function for the pressure jump definition. This item will appear if you enable **Profile Specification of Pressure-Jump**.

Pressure-Jump

specifies the pressure-jump as a constant value or as a polynomial, piecewise-linear, or piecewise-polynomial function of velocity. See Defining the Pressure Jump (p. 1507) for details.

Limit Polynomial Velocity Range

limits the minimum and maximum velocity magnitudes used to calculate the pressure jump when it is defined as a function of velocity.

Min Velocity Magnitude, Max Velocity Magnitude

specify the minimum and maximum values to which the velocity magnitude is limited (when the **Limit Polynomial Velocity Range** option is enabled).

Calculate Pressure-Jump from Average Conditions

enables the option to use the mass-averaged velocity normal to the fan to determine a single pressure-jump value for all faces in the fan zone.

Discrete Phase BC Type

sets the way that the discrete phase behaves with respect to the boundary. This item appears when one or more injections have been defined.

interior

allows the particles to pass through the boundary.

reflect

rebounds the particle off the boundary with a change in its momentum as defined by the coefficient of restitution. (See Particle Reflection at Wall in the *Fluent Theory Guide*.)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

trap

terminates the trajectory calculations and records the fate of the particle as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See Figure 24.40: “Trap” Boundary Condition for the Discrete Phase (p. 2783).

escape

reports the particle as having "escaped" when it encounters the boundary. Trajectory calculations are terminated. See Figure 24.41: “Escape” Boundary Condition for the Discrete Phase (p. 2783).

reinject

reintroduces the particle into the domain when it reaches a certain domain boundary (for example, outlet). This item appears when one or more injections have been defined and **Unsteady Particle Tracking** is enabled in the **Discrete Phase Model** dialog box. See The **reinject** Boundary Condition (p. 2784) for details.

wall-jet

indicates that the direction and velocity of the droplet particles are given by the resulting momentum flux, which is a function of the impingement angle. See Figure 12.6: "Wall Jet" Boundary Condition for the Discrete Phase in the Theory Guide.

user-defined

specifies a user-defined function to define the discrete phase boundary condition type.

Discrete Phase BC Function

sets the user-defined function from the drop-down list.

Swirl-Velocity Specification

contains inputs for the specification of fan swirl velocity. This section of the dialog box appears only for 3D models.

Swirl-Velocity Specification

enables the specification of a swirl velocity for the fan.

Fan Axis

sets the direction vector for the fan’s axis of rotation.

Fan Origin

sets the origin in the global coordinate system through which the fan rotation axis passes.

Fan Hub Radius

set the radius of the hub. The default is 1e-6 to avoid division by zero in the polynomial.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Profile Specification of Tangential Velocity

enables the use of a boundary profile or user-defined function for the tangential velocity specification. See Profiles (p. 1564) or the Fluent Customization Manual for details. When this option is enabled, **Tangential Velocity Profile** will appear in the dialog box and **Tangential-Velocity Polynomial Coefficients** will not.

Tangential Velocity Profile

contains a drop-down list from which you can select a boundary profile or a user-defined function for the definition of the tangential velocity. This item will appear if you enable **Profile Specification of Tangential Velocity**.

Tangential-Velocity Polynomial Coefficients

sets the coefficients for the tangential velocity polynomial. Separate the coefficients by spaces.

Profile Specification of Radial Velocity

enables the use of a boundary profile or user-defined function for the radial velocity specification. See Profiles (p. 1564) or the Fluent Customization Manual for details. When this option is enabled, **Radial Velocity Profile** will appear in the dialog box and **Radial-Velocity Polynomial Coefficients** will not.

Radial Velocity Profile

contains a drop-down list from which you can select a boundary profile or a user-defined function for the definition of the radial velocity. This item will appear if you enable **Profile Specification of Radial Velocity**.

Radial-Velocity Polynomial Coefficients

sets the coefficients for the radial velocity polynomial. Separate the coefficients by spaces.

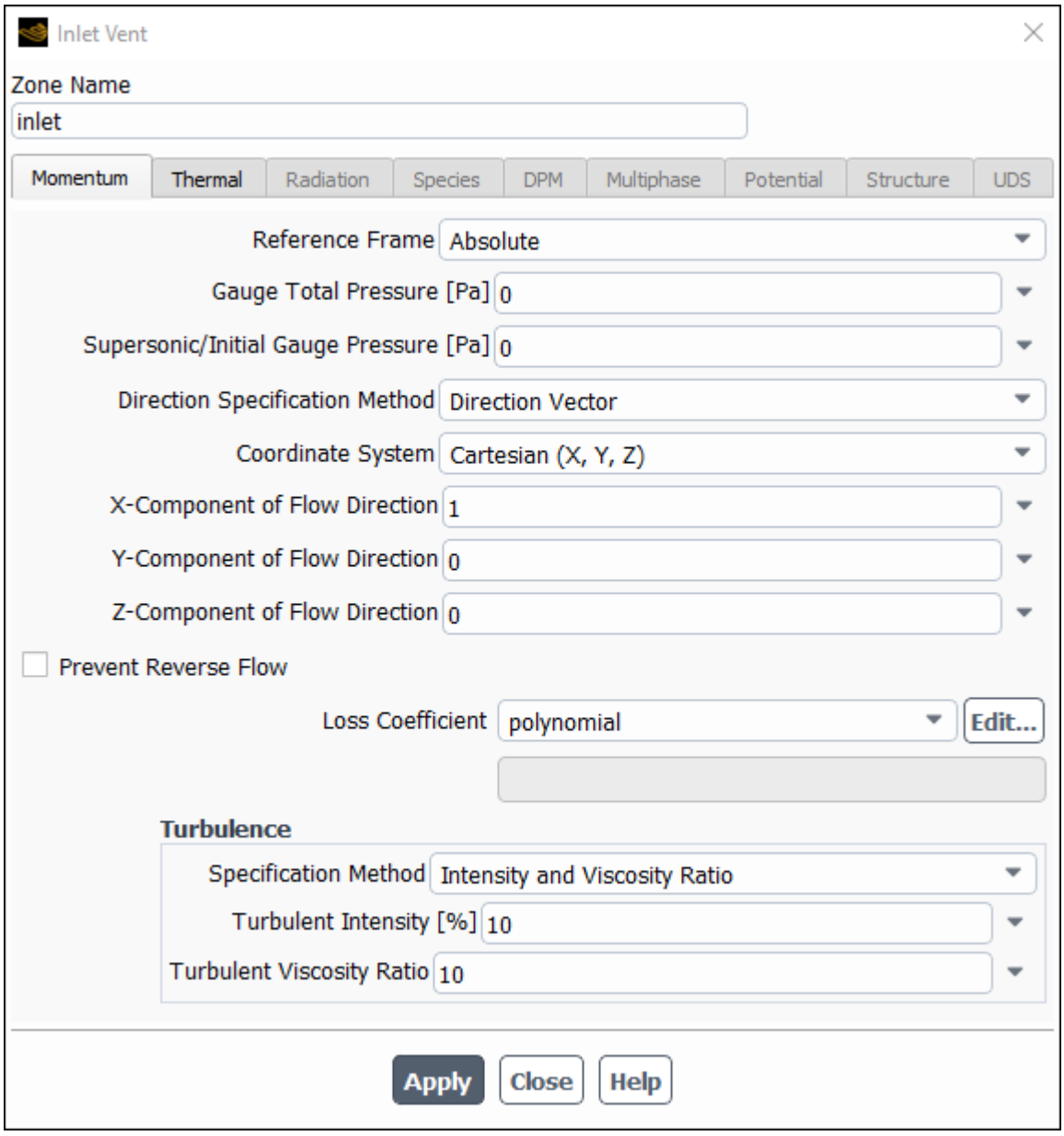
51.7.5. Inlet Vent Dialog Box

The **Inlet Vent** dialog box sets the boundary conditions for an inlet vent zone. It is opened from the Boundary Conditions Task Page (p. 4991). See Inputs at Inlet Vent Boundaries (p. 1429) for details about defining the items below.

Important:

This feature offers reduced functionality when running Fluent under the **Pro** capability level.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Zone Name

sets the name of the zone.

Momentum

contains the momentum parameters.

Reference Frame

specifies the reference frame for the inlet vent. If the cell zone adjacent to the inlet vent is moving, you can choose to specify relative or absolute velocities by selecting **Relative to Adjacent Cell Zone** or **Absolute** in the **Reference Frame** drop-down list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Gauge Total Pressure

sets the gauge total (or stagnation) pressure of the inflow stream. If you are using moving reference frames, see Defining Total Pressure and Temperature (p. 1404) for information about relative and absolute total pressure.

Supersonic/Initial Gauge Pressure

sets the static pressure on the boundary when the flow becomes (locally) supersonic. It is also used to compute initial values for pressure, temperature, and velocity if the inlet vent boundary condition is selected for computing initial values (see Initializing the Entire Flow Field Using Standard Initialization (p. 3628)).

Direction Specification Method

specifies the method you will use to define the flow direction. If you choose **Direction Vector**, you will define the flow direction components, and if you choose **Normal to Boundary** no inputs are required. See Defining the Flow Direction (p. 1405) for information on specifying flow direction.

Coordinate System

specifies whether **Cartesian**, **Cylindrical**, **Local Cylindrical**, **Local Cylindrical Swirl** vector components will be specified. This item will appear only for 3D cases in which you have selected **Direction Vector** as the **Direction Specification Method**.

X,Y,Z-Component of Flow Direction

set the direction of the flow at the inlet boundary. These items will appear if the selected **Coordinate System** is **Cartesian** or the model is 2D non-axisymmetric.

Radial, Tangential, Axial Component of Flow Direction

set the direction of the flow at the inlet boundary. These items will appear for 2D axisymmetric cases, or for 3D cases for which the selected **Coordinate System** is **Cylindrical** or **Local Cylindrical**.

Axis Origin

sets the **X**, **Y**, and **Z** coordinates of the origin of the local cylindrical coordinate system.

Axis Direction

sets the **X**, **Y**, and **Z** components of the direction of the local cylindrical coordinate system.

Loss-Coefficient

sets the non-dimensional loss coefficient used to compute the pressure drop. See Specifying the Loss Coefficient (p. 1431) for details.

Turbulence

lists the turbulence parameters.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Specification Method

specifies which method will be used to define the turbulence parameters. You can choose **K and Epsilon** (k- ε models and RSM only), **K and Omega** (k- ω models only), **Intensity and Length Scale**, **Intensity and Viscosity Ratio**, **Intensity and Hydraulic Diameter**, **Modified Turbulent Viscosity** (Spalart-Allmaras model only), or **Turbulent Viscosity Ratio** (Spalart-Allmaras model only). See Determining Turbulence Parameters (p. 1395) for information about the inputs for each of these methods. (This item will appear only for turbulent flow calculations.)

Turbulent Kinetic Energy, Turbulent Dissipation Rate

set values for the turbulence kinetic energy k and its dissipation rate ε. These items will appear if you choose **K and Epsilon** as the **Specification Method**.

Turbulent Kinetic Energy, Specific Dissipation Rate

set values for the turbulence kinetic energy k and its specific dissipation rate ω. These items will appear if you choose **K and Omega** as the **Specification Method**.

Turbulence Intensity, Turbulence Length Scale

set values for turbulence intensity I and turbulence length scale ℓ. These items will appear if you choose **Intensity and Length Scale** as the **Specification Method**.

Turbulence Intensity, Turbulent Viscosity Ratio

set values for turbulence intensity I and turbulent viscosity ratio μ_t / μ. These items will appear if you choose **Intensity and Viscosity Ratio** as the **Specification Method**.

Turbulence Intensity, Hydraulic Diameter

set values for turbulence intensity I and hydraulic diameter L. These items will appear if you choose **Intensity and Hydraulic Diameter** as the **Specification Method**.

Modified Turbulent Viscosity

sets the value of the modified turbulent viscosity $\tilde{\nu}$. This item will appear if you choose **Modified Turbulent Viscosity** as the **Specification Method**.

Turbulent Viscosity Ratio

sets the value of the turbulent viscosity ratio μ_t / μ. This item will appear if you choose **Turbulent Viscosity Ratio** as the **Specification Method**.

Reynolds-Stress Specification Method

specifies which method will be used to determine the Reynolds stress boundary conditions when the Reynolds stress turbulence model is used. You can choose either **K or Turbulence Intensity** or **Reynolds-Stress Components**. If you choose the former, Ansys Fluent will compute the Reynolds stresses for you. If you choose the latter, you will explicitly specify the Reynolds stresses yourself. See Reynolds Stress Model (p. 2144) for details. (This item will appear only for RSM turbulent flow calculations.)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UU, VV, WW, UV, VW, UW Reynolds Stresses

specify the Reynolds stress components when **Reynolds-Stress Components** is chosen as the **Reynolds-Stress Specification Method**.

Thermal

contains the thermal parameters.

Total Temperature

sets the total temperature of the inflow stream. If you are using moving reference frames, see Defining Total Pressure and Temperature (p. 1404) for information about relative and absolute total temperature.

Radiation

contains the radiation parameters.

External Black Body Temperature Method, Internal Emissivity

set the radiation boundary conditions when you are using the P-1, DTRM, DO, S2S, or MC models for radiation heat transfer. See Defining Boundary Conditions for Radiation (p. 2217) for details.

Participates in Solar Ray Tracing

specifies whether or not the inlet vent participates in solar ray tracing.

Solar Transmissivity Factor

specifies a multiplier (ranging from 0 to 1) that is applied to the solar irradiation entering the domain through the inlet vent.

Participates in View Factor Calculation

specifies whether or not the inlet vent participates in the view factor calculation as part of the S2S radiation model. This parameter is available only if you select the **Surface to Surface** radiation model.

Species

contains the species parameters.

Specify Species in Mole Fractions

allows you to specify the species in mole fractions rather than mass fractions.

Species Mass Fractions

contains inputs for the mass fractions of defined species. See Defining Cell Zone and Boundary Conditions for Species (p. 2394) for details about these inputs. (These items will appear only if you are modeling non-reacting multi-species flow or you are using the finite-rate reaction formulation.)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mean Mixture Fraction, Mixture Fraction Variance

set inlet values for the PDF mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model.)

Secondary Mean Mixture Fraction, Secondary Mixture Fraction Variance

set inlet values for the secondary mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model with two mixture fractions.)

Progress Variable

sets the value of the progress variable for premixed turbulent combustion. See Setting Boundary Conditions for the Progress Variable (p. 2541) for details.

This item will appear only if the premixed or partially premixed combustion model is used.

DPM

contains the discrete phase parameters.

Discrete Phase BC Type

sets the way that the discrete phase behaves with respect to the boundary. This item appears when one or more injections have been defined.

reflect

rebounds the particle off the boundary with a change in its momentum as defined by the coefficient of restitution. (See Particle Reflection at Wall in the *Fluent Theory Guide*.)

trap

terminates the trajectory calculations and records the fate of the particle as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See Figure 24.40: “Trap” Boundary Condition for the Discrete Phase (p. 2783).

escape

reports the particle as having "escaped" when it encounters the boundary. Trajectory calculations are terminated. See Figure 24.41: “Escape” Boundary Condition for the Discrete Phase (p. 2783).

reinject

reintroduces the particle into the domain when it reaches a certain domain boundary (for example, outlet). This item appears when one or more injections have been defined and **Unsteady Particle Tracking** is enabled in the **Discrete Phase Model** dialog box. See The **reinject** Boundary Condition (p. 2784) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

wall-jet

indicates that the direction and velocity of the droplet particles are given by the resulting momentum flux, which is a function of the impingement angle. See Figure 12.6: "Wall Jet" Boundary Condition for the Discrete Phase in the Theory Guide.

user-defined

specifies a user-defined function to define the discrete phase boundary condition type.

Discrete Phase BC Function

sets the user-defined function from the drop-down list.

Multiphase

contains the multiphase parameters.

Granular Temperature

specifies temperature for the solids phase and is proportional to the kinetic energy of the random motion of the particles.

Volume Fraction

specifies the volume fraction of the secondary phase selected in the Boundary Conditions Task Page (p. 4991). This section of the dialog box will appear when one of the multiphase models is being used. See Defining Multiphase Cell Zone and Boundary Conditions (p. 2957) for details.

Open Channel

is available when the VOF model with open channel flow is enabled.

Secondary Phase for Inlet

is where the specified parameters are valid only for one secondary phase. In case of a three-phase flow, select the corresponding secondary phase from this list. This appears when **Open Channel** is enabled.

Flow Specification Method

allows you to select the type of flow. You can choose **Free Surface Level and Velocity**, **Total Height and Velocity**, or **Free Surface Level and Total Height**. This appears when **Open Channel** is enabled.

Free Surface Level

can be determined using the absolute value of height from the free surface to the origin in the direction of gravity, or by applying the correct sign based on whether the free surface level is above or below the origin.

Total Height

is used as an option for describing the flow. It is given by Equation 26.12 (p. 2984).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Bottom Level

is valid only for shallow waves. The bottom level is used for calculating the liquid height.

Velocity Magnitude

sets the magnitude of the velocity vector at the inflow boundary.

Level-Set Function Flux

appears if the **Coupled Level Set + VOF** option is enabled for the VOF model.

Potential

displays the boundary conditions for the electric potential field. This tab is available only if you have enabled either the **Electric Potential model** or the **Electrochemical** reaction model in the Species Dialog Box (p. 4929).

Potential Boundary Condition

is a drop-down list of available potential boundary condition types for the potential generated by electron current: **Specified Flux** and **Specified Value**. For the **Specified Flux** boundary condition, you will need to specify **Current Density** at the wall. For the **Specified Value** boundary condition, you will need to specify **Potential** at the wall.

Electrolyte Potential Boundary Condition

is a drop-down list of available boundary condition types for the potential generated by ionic current. This item is similar to the **Potential Boundary Condition** drop-down list described above and is available only for the Electrolysis and H2 Pump model.

UDS

contains the UDS parameters.

User-Defined Scalar Boundary Condition

appears only if user-defined scalars are specified.

User Scalar n

specifies whether the scalar is a specified flux or a specified value.

User-Defined Scalar Boundary Value

appears only if user-defined scalars are specified.

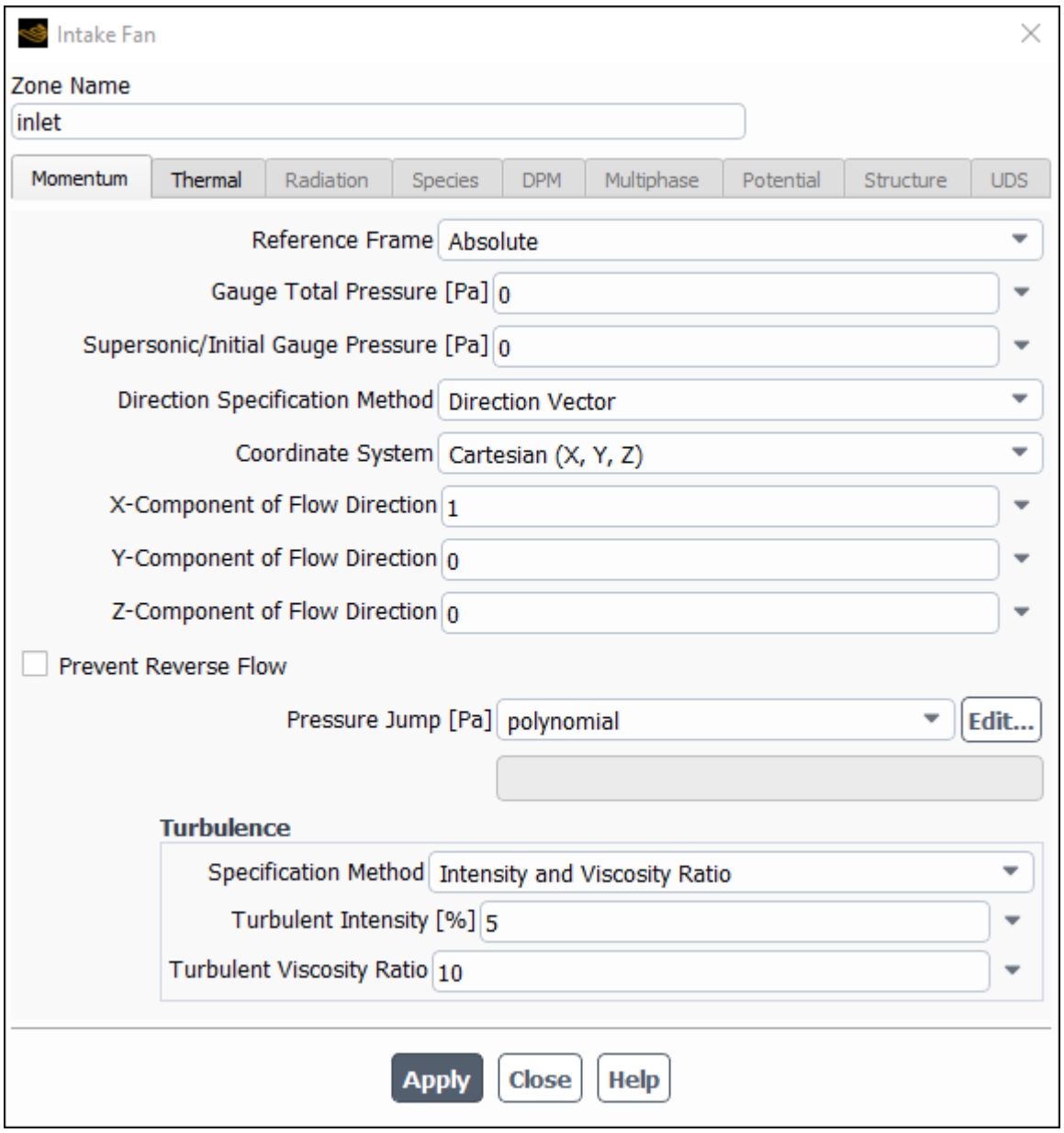
User Scalar n

specifies the value of the scalar.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

51.7.6. Intake Fan Dialog Box

The **Intake Fan** dialog box sets the boundary conditions for an intake fan zone. It is opened from the Boundary Conditions Task Page (p. 4991). See Inputs at Intake Fan Boundaries (p. 1432) for details about defining the items below.



Controls

Zone Name

sets the name of the zone.

Momentum

contains the momentum parameters.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reference Frame

specifies the reference frame for the intake fan. If the cell zone adjacent to the intake fan is moving, you can choose to specify relative or absolute velocities by selecting **Relative to Adjacent Cell Zone** or **Absolute** in the **Reference Frame** drop-down list.

Gauge Total Pressure

sets the gauge total (or stagnation) pressure of the inflow stream. If you are using moving reference frames, see Defining Total Pressure and Temperature (p. 1404) for information about relative and absolute total pressure.

Supersonic/Initial Gauge Pressure

sets the static pressure on the boundary when the flow becomes (locally) supersonic. It is also used to compute initial values for pressure, temperature, and velocity if the intake fan boundary condition is selected for computing initial values (see Initializing the Entire Flow Field Using Standard Initialization (p. 3628)).

Direction Specification Method

specifies the method you will use to define the flow direction. If you choose **Direction Vector**, you will define the flow direction components, and if you choose **Normal to Boundary** no inputs are required. See Defining the Flow Direction (p. 1405) for information on specifying flow direction.

Coordinate System

specifies whether **Cartesian**, **Cylindrical**, **Local Cylindrical**,or **Local Cylindrical Swirl** vector components will be defined. This item will appear only for 3D cases in which you have selected **Direction Vector** as the **Direction Specification Method**.

X-, Y-, Z-Component of Flow Direction

set the direction of the flow at the inlet boundary. For compressible flow, if the inflow becomes supersonic, the velocity is not reoriented. These items will appear if the selected **Coordinate System** is **Cartesian** or the model is 2D non-axisymmetric.

Radial-, Tangential-, Axial-Component of Flow Direction

set the direction of the flow at the inlet boundary. For compressible flow, if the inflow becomes supersonic, the velocity is not reoriented. These items will appear for 2D axisymmetric cases, or for 3D cases for which the selected **Coordinate System** is **Cylindrical** or **Local Cylindrical**.

Pressure Jump

specifies the rise in pressure across the fan. See Specifying the Pressure Jump (p. 1434) for details.

Axis Origin

sets the **X**, **Y**, and **Z** coordinates of the origin of the local cylindrical coordinate system.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Axis Direction

sets the **X**, **Y**, and **Z** components of the direction of the local cylindrical coordinate system.

Turbulence

consists of the turbulence parameters.

Specification Method

specifies which method will be used to define the turbulence parameters. You can choose **K and Epsilon** (k- ε models and RSM only), **K and Omega** (k- ω models only), **Intensity and Length Scale**, **Intensity and Viscosity Ratio**, **Intensity and Hydraulic Diameter**, **Modified Turbulent Viscosity** (Spalart-Allmaras model only), or **Turbulent Viscosity Ratio** (Spalart-Allmaras model only). See Determining Turbulence Parameters (p. 1395) for information about the inputs for each of these methods. (This item will appear only for turbulent flow calculations.)

Turbulent Kinetic Energy, Turbulent Dissipation Rate

set values for the turbulence kinetic energy k and its dissipation rate ε. These items will appear if you choose **K and Epsilon** as the **Specification Method**.

Turbulent Kinetic Energy, Specific Dissipation Rate

set values for the turbulence kinetic energy k and its specific dissipation rate ω. These items will appear if you choose **K and Omega** as the **Specification Method**.

Turbulent Intensity, Turbulent Length Scale

set values for turbulence intensity I and turbulence length scale ℓ. These items will appear if you choose **Intensity and Length Scale** as the **Specification Method**.

Turbulent Intensity, Turbulent Viscosity Ratio

set values for turbulence intensity I and turbulent viscosity ratio μ_t / μ. These items will appear if you choose **Intensity and Viscosity Ratio** as the **Specification Method**.

Turbulent Intensity, Hydraulic Diameter

set values for turbulence intensity I and hydraulic diameter L. These items will appear if you choose **Intensity and Hydraulic Diameter** as the **Specification Method**.

Modified Turbulent Viscosity

sets the value of the modified turbulent viscosity $\tilde{\nu}$. This item will appear if you choose **Modified Turbulent Viscosity** as the **Specification Method**.

Turbulent Viscosity Ratio

sets the value of the turbulent viscosity ratio μ_t / μ. This item will appear if you choose **Turbulent Viscosity Ratio** as the **Specification Method**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reynolds-Stress Specification Method

specifies which method will be used to determine the Reynolds stress boundary conditions when the Reynolds stress turbulence model is used. You can choose either **K or Turbulent Intensity** or **Reynolds-Stress Components**. If you choose the former, Ansys Fluent will compute the Reynolds stresses for you. If you choose the latter, you will explicitly specify the Reynolds stresses yourself. See Reynolds Stress Model (p. 2144) for details. (This item will appear only for RSM turbulent flow calculations.)

UU, VV, WW, UV, VW, UW Reynolds Stresses

specify the Reynolds stress components when **Reynolds-Stress Components** is chosen as the **Reynolds-Stress Specification Method**.

Thermal

contains the thermal parameters.

Total Temperature

sets the total temperature of the inflow stream. If you are using moving reference frames, see Defining Total Pressure and Temperature (p. 1404) for information about relative and absolute total temperature.

Radiation

contains the radiation parameters.

External Black Body Temperature Method, Internal Emissivity

set the radiation boundary conditions when you are using the P-1, DTRM, DO, S2S, or MC models for radiation heat transfer. See Defining Boundary Conditions for Radiation (p. 2217) for details.

Participates in Solar Ray Tracing

specifies whether or not the intake fan participates in solar ray tracing.

Solar Transmissivity Factor

specifies a multiplier (ranging from 0 to 1) that is applied to the solar irradiation entering the domain through the intake fan.

Participates in View Factor Calculation

specifies whether or not the intake fan participates in the view factor calculation as part of the S2S radiation model. This parameter is available only if you select the **Surface to Surface** radiation model.

Species

contains the species parameters.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Specify Species in Mole Fractions

allows you to specify the species in mole fractions rather than mass fractions.

Species Mass Fractions

contains inputs for the mass fractions of defined species. See Defining Cell Zone and Boundary Conditions for Species (p. 2394) for details about these inputs. (These items will appear only if you are modeling non-reacting multi-species flow or you are using the finite-rate reaction formulation.)

Mean Mixture Fraction, Mixture Fraction Variance

set inlet values for the PDF mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model.)

Secondary Mean Mixture Fraction, Secondary Mixture Fraction Variance

set inlet values for the secondary mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model with two mixture fractions.)

Progress Variable

sets the value of the progress variable for premixed turbulent combustion. See Setting Boundary Conditions for the Progress Variable (p. 2541) for details.

This item will appear only if the premixed or partially premixed combustion model is used.

DPM

contains the discrete phase parameters.

Discrete Phase BC Type

sets the way that the discrete phase behaves with respect to the boundary. This item appears when one or more injections have been defined.

reflect

rebounds the particle off the boundary with a change in its momentum as defined by the coefficient of restitution. (See Particle Reflection at Wall in the *Fluent Theory Guide*.)

trap

terminates the trajectory calculations and records the fate of the particle as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See Figure 24.40: "Trap" Boundary Condition for the Discrete Phase (p. 2783).

escape

reports the particle as having "escaped" when it encounters the boundary. Trajectory calculations are terminated. See Figure 24.41: "Escape" Boundary Condition for the Discrete Phase (p. 2783).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

reinject

reintroduces the particle into the domain when it reaches a certain domain boundary (for example, outlet). This item appears when one or more injections have been defined and **Unsteady Particle Tracking** is enabled in the **Discrete Phase Model** dialog box. See The **reinject** Boundary Condition (p. 2784) for details.

wall-jet

indicates that the direction and velocity of the droplet particles are given by the resulting momentum flux, which is a function of the impingement angle. See Figure 12.6: "Wall Jet" Boundary Condition for the Discrete Phase in the Theory Guide.

user-defined

specifies a user-defined function to define the discrete phase boundary condition type.

Discrete Phase BC Function

sets the user-defined function from the drop-down list.

Multiphase

contains the multiphase parameters.

Granular Temperature

specifies temperature for the solids phase and is proportional to the kinetic energy of the random motion of the particles.

Volume Fraction

specifies the volume fraction of the secondary phase selected in the Boundary Conditions Task Page (p. 4991). This section of the dialog box will appear when one of the multiphase models is being used. See Defining Multiphase Cell Zone and Boundary Conditions (p. 2957) for details.

Potential

displays the boundary conditions for the electric potential field. This tab is available only if you have enabled either the **Electric Potential model** or the **Electrochemical** reaction model in the Species Dialog Box (p. 4929).

Potential Boundary Condition

is a drop-down list of available potential boundary condition types for the potential generated by electron current: **Specified Flux** and **Specified Value**. For the **Specified Flux** boundary condition, you will need to specify **Current Density** at the wall. For the **Specified Value** boundary condition, you will need to specify **Potential** at the wall.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Electrolyte Potential Boundary Condition

is a drop-down list of available boundary condition types for the potential generated by ionic current. This item is similar to the **Potential Boundary Condition** drop-down list described above and is available only for the Electrolysis and H2 Pump model.

UDS

contains the UDS parameters.

User-Defined Scalar Boundary Condition

appears only if user-defined scalars are specified.

User Scalar n

specifies whether the scalar is a specified flux or a specified value.

User-Defined Scalar Boundary Value

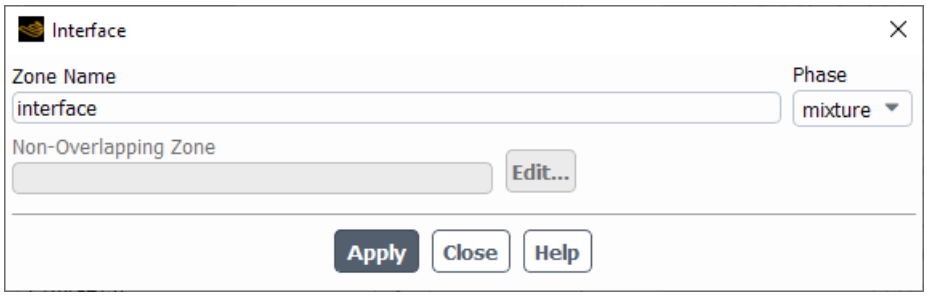
appears only if user-defined scalars are specified.

User Scalar n

specifies the value of the scalar.

51.7.7. Interface Dialog Box

The **Interface** dialog box can be used to modify the name of an interface zone; there are no conditions to be set. It is opened from the Boundary Conditions Task Page (p. 4991). Interface zones are used for multiple reference frame and sliding mesh calculations, and for non-conformal meshes. See The Multiple Reference Frame Model (p. 1779), Setting Up the Sliding Mesh Problem (p. 1805), and Non-Conformal Meshes (p. 1170) for details.



Controls

Zone Name

sets the name of the zone.

Non-Overlapping Zone

displays the name of the non-overlapping zone created for this interface zone as part of the creation of a mesh interface.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Edit...

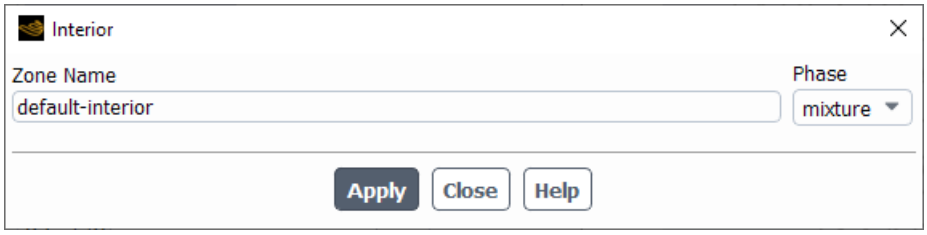
opens the boundary condition dialog box of the non-overlapping zone, so that you can easily review and/or revise the settings.

Phase

displays the name of the phase. This item appears only for multiphase flows.

51.7.8. Interior Dialog Box

The **Interior** dialog box can be used to modify the name of an interior zone; there are no conditions to be set. It is opened from the Boundary Conditions Task Page (p. 4991).



Controls

Zone Name

sets the name of the zone.

Phase

displays the name of the phase. This item appears only for multiphase flows.

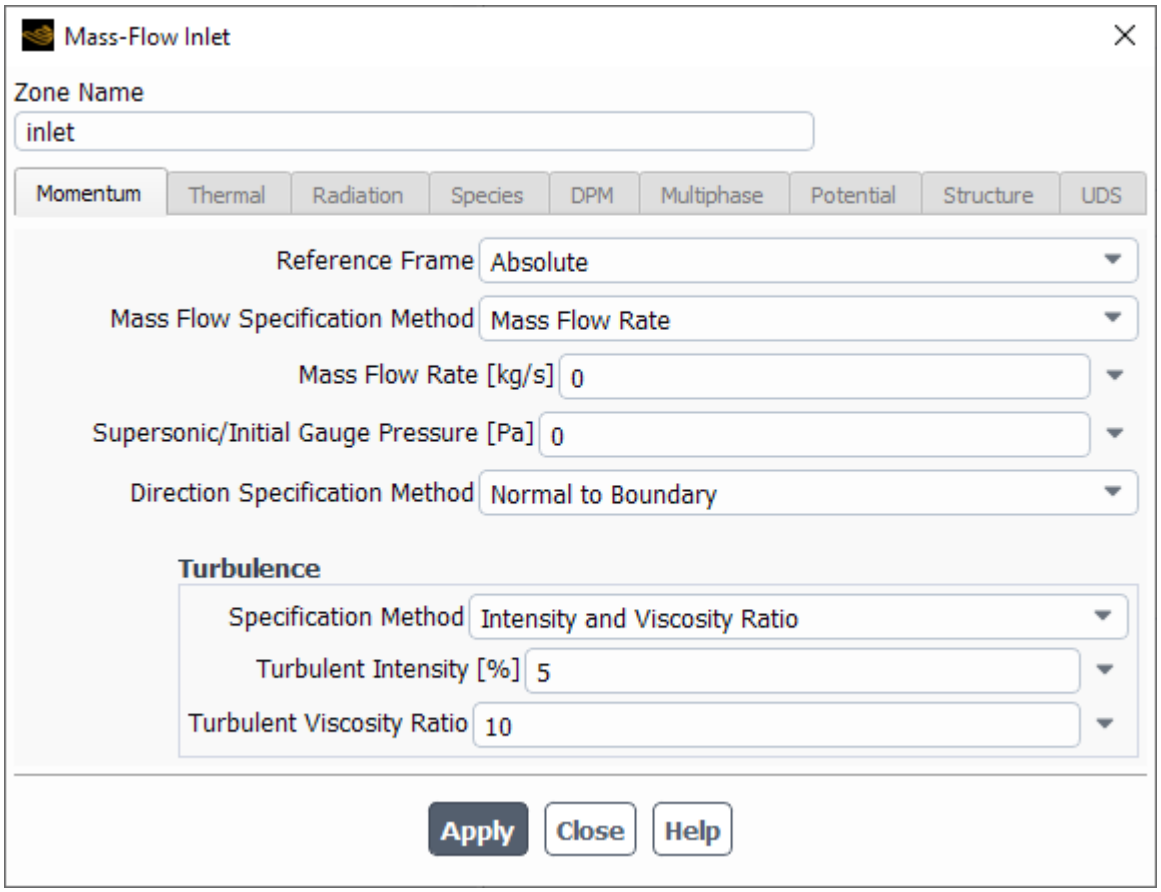
51.7.9. Mass-Flow Inlet Dialog Box

The **Mass-Flow Inlet** dialog box sets the boundary conditions for a mass-flow inlet zone. It is opened from the Boundary Conditions Task Page (p. 4991). See Inputs at Mass-Flow Inlet Boundaries (p. 1418) for details about defining the items below.

Important:

This feature offers reduced functionality when running Fluent under the **Pro** capability level.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Zone Name

sets the name of the zone.

Momentum

displays the momentum boundary conditions.

Reference Frame

specifies the reference frame for the mass flow. If the cell zone adjacent to the mass-flow inlet is moving, you can choose to specify relative or absolute velocities by selecting **Relative to Adjacent Cell Zone** or **Absolute** in the **Reference Frame** drop-down list.

Mass Flow Specification Method

specifies whether you are defining **Mass Flow Rate**, **Mass Flux**, or **Mass Flux with Average Mass Flux**.

Mass Flow Rate

sets the prescribed mass flow rate for the zone. This flow rate is converted internally to a prescribed uniform mass flux over the zone by dividing the flow rate by the flow direction

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

area projection of the zone. This item will appear if you selected **Mass Flow Rate** in the **Mass Flow Specification Method** list.

Important:

Note that for axisymmetric problems, this mass flow rate is the flow rate through the entire (2π-radian) domain, not through a 1-radian slice.

Mass Flux

sets the prescribed mass flux for the zone. This item will appear if you selected **Mass Flux** or **Mass Flux with Average Mass Flux** in the **Mass Flow Specification Method** list.

Important:

Note that for axisymmetric problems, this mass flux is the flux through a 1-radian slice of the domain.

Average Mass Flux

sets the average mass flux through the zone. See More About Mass Flux and Average Mass Flux (p. 1420) for details. This item will appear if you selected **Mass Flux with Average Mass Flux** in the **Mass Flow Specification Method** list.

Important:

Note that for axisymmetric problems, this mass flux is the flux through a 1-radian slice of the domain.

Supersonic/Initialization Gauge Pressure

sets the static pressure that will be used to initialize the flow field if the mass-flow inlet boundary condition is selected for initializing flow properties (see Initializing the Entire Flow Field Using Standard Initialization (p. 3628)).

Direction Specification Method

specifies the method you will use to define the flow direction. If you choose **Direction Vector**, you will define the flow direction components, and if you choose **Normal to Boundary** no inputs are required. See Defining the Flow Direction (p. 1405) for information on specifying flow direction.

Coordinate System

specifies whether **Cartesian**, **Cylindrical**, **Local Cylindrical**,or **Local Cylindrical Swirl** vector components will be defined. This item will appear only for 3D cases in which you have selected **Direction Vector** as the **Direction Specification Method**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

X-, Y-, Z-Component of Flow Direction

set the velocity-direction vector of the inflow stream. This vector does not need to be normalized (for example, you can specify the vector (1 1 1) rather than (0.577 0.577 0.577)). These items will appear if the selected **Coordinate System** is **Cartesian** or the model is 2D non-axisymmetric.

Radial-, Tangential-, Axial-Component of Flow Direction

set the velocity-direction vector of the inflow stream. These items will appear for 2D axisymmetric cases, or for 3D cases for which the selected **Coordinate System** is **Cylindrical** or **Local Cylindrical**.

Axial-, Radial-Component of Flow Direction, Tangential-Velocity

appear for a 3D **Local Cylindrical Swirl** coordinate system.

Axis Origin

sets the **X**, **Y**, and **Z** coordinates of the origin of the local cylindrical (swirl) coordinate system.

Axis Direction

sets the **X**, **Y**, and **Z** components of the direction of the local cylindrical (swirl) coordinate system.

Turbulence

contains the turbulence parameters.

Specification Method

specifies which method will be used to define the turbulence parameters. You can choose **K and Epsilon** (k- ε models and RSM only), **K and Omega** (k- ω models only), **Intensity and Length Scale**, **Intensity and Viscosity Ratio**, **Intensity and Hydraulic Diameter**, **Modified Turbulent Viscosity** (Spalart-Allmaras model only), or **Turbulent Viscosity Ratio** (Spalart-Allmaras model only). See Determining Turbulence Parameters (p. 1395) for information about the inputs for each of these methods. (This item will appear only for turbulent flow calculations.)

Turbulent Kinetic Energy, Turbulent Dissipation Rate

set values for the turbulence kinetic energy k and its dissipation rate ε. These items will appear if you choose **K and Epsilon** as the **Specification Method**.

Turbulent Kinetic Energy, Specific Dissipation Rate

set values for the turbulence kinetic energy k and its specific dissipation rate ω. These items will appear if you choose **K and Omega** as the **Specification Method**.

Turbulent Intensity, Turbulent Length Scale

set values for turbulence intensity I and turbulence length scale ℓ. These items will appear if you choose **Intensity and Length Scale** as the **Specification Method**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Turbulent Intensity, Turbulent Viscosity Ratio

set values for turbulence intensity I and turbulent viscosity ratio μ_t / μ. These items will appear if you choose **Intensity and Viscosity Ratio** as the **Specification Method**.

Turbulent Intensity, Hydraulic Diameter

set values for turbulence intensity I and hydraulic diameter L. These items will appear if you choose **Intensity and Hydraulic Diameter** as the **Specification Method**.

Modified Turbulent Viscosity

sets the value of the modified turbulent viscosity $\tilde{\nu}$. This item will appear if you choose **Modified Turbulent Viscosity** as the **Specification Method**.

Turbulent Viscosity Ratio

sets the value of the turbulent viscosity ratio μ_t / μ. This item will appear if you choose **Turbulent Viscosity Ratio** as the **Specification Method**.

Reynolds-Stress Specification Method

specifies which method will be used to determine the Reynolds stress boundary conditions when the Reynolds stress turbulence model is used. You can choose either **K or Turbulent Intensity** or **Reynolds-Stress Components**. If you choose the former, Ansys Fluent will compute the Reynolds stresses for you. If you choose the latter, you will explicitly specify the Reynolds stresses yourself. See Reynolds Stress Model (p. 2144) for details. (This item will appear only for RSM turbulent flow calculations.)

UU, VV, WW, UV, VW, UW Reynolds Stresses

specify the Reynolds stress components when **Reynolds-Stress Components** is chosen as the **Reynolds-Stress Specification Method**.

Acoustic Wave Model

contains settings for treatment of acoustic pressure waves at the boundary.

Off

disables special treatment of acoustic pressure waves at the boundary.

Non Reflecting

enables the general non-reflecting boundary condition treatment described in General Non-Reflecting Boundary Conditions (p. 1538).

Impedance

enables the impedance boundary condition treatment described in Impedance Boundary Conditions (p. 1545).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Transparent Flow Forcing

enables the transparent flow forcing boundary condition described in Transparent Flow Forcing Boundary Conditions (p. 1554).

Impedance Parameters

contains the parameters for the impedance boundary condition treatment. For details refer to Using the Impedance Boundary Condition (p. 1546).

Transparent Flow Forcing Parameters

contains the parameters for the transparent flow forcing boundary condition treatment. For details refer to Using the Transparent Flow Forcing Boundary Condition (p. 1554).

Thermal

contains the thermal parameters.

Total Temperature

sets the total temperature of the inflow stream.

Radiation

contains the radiation parameters.

External Black Body Temperature Method, Internal Emissivity

set the radiation boundary conditions when you are using the P-1, DTRM, DO, S2S, or MC models for radiation heat transfer. See Defining Boundary Conditions for Radiation (p. 2217) for details.

Participates in Solar Ray Tracing

specifies whether or not the mass-flow inlet participates in solar ray tracing.

Solar Transmissivity Factor

specifies a multiplier (ranging from 0 to 1) that is applied to the solar irradiation entering the domain through the mass-flow inlet.

Participates in View Factor Calculation

specifies whether or not the mass-flow inlet participates in the view factor calculation as part of the S2S radiation model. This parameter is available only if you select the **Surface to Surface** radiation model.

Species

contains the species parameters.

Specify Species in Mole Fractions

allows you to specify the species in mole fractions rather than mass fractions.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Species Mass Fractions

contains inputs for the mass fractions of defined species. See Defining Cell Zone and Boundary Conditions for Species (p. 2394) for details about these inputs. These items will appear only if you are modeling non-reacting multi-species flow or you are using the finite-rate reaction formulation.

Progress Variable

sets the value of the progress variable for premixed turbulent combustion. See Setting Boundary Conditions for the Progress Variable (p. 2541) for details.

This item will appear only if the premixed or partially premixed combustion model is used.

Mean Mixture Fraction, Mixture Fraction Variance

set inlet values for the PDF mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model.)

Secondary Mean Mixture Fraction, Secondary Mixture Fraction Variance

set inlet values for the secondary mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model with two mixture fractions.)

Scalar Mass Fractions

(partially-premixed combustion FGM model only) allows you to specify the scalar species mass fractions for the transported scalars that you selected in the **Select Transported Scalars** dialog box.

DPM

contains the discrete phase parameters.

Discrete Phase BC Type

sets the way that the discrete phase behaves with respect to the boundary. This item appears when one or more injections have been defined.

reflect

rebounds the particle off the boundary with a change in its momentum as defined by the coefficient of restitution. (See Particle Reflection at Wall in the *Fluent Theory Guide*.)

trap

terminates the trajectory calculations and records the fate of the particle as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See Figure 24.40: "Trap" Boundary Condition for the Discrete Phase (p. 2783).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

escape

reports the particle as having "escaped" when it encounters the boundary. Trajectory calculations are terminated. See Figure 24.41: "Escape" Boundary Condition for the Discrete Phase (p. 2783).

reinject

reintroduces the particle into the domain when it reaches a certain domain boundary (for example, outlet). This item appears when one or more injections have been defined and **Unsteady Particle Tracking** is enabled in the **Discrete Phase Model** dialog box. See The **reinject** Boundary Condition (p. 2784) for details.

wall-jet

indicates that the direction and velocity of the droplet particles are given by the resulting momentum flux, which is a function of the impingement angle. See Figure 12.6: "Wall Jet" Boundary Condition for the Discrete Phase in the Theory Guide.

user-defined

specifies a user-defined function to define the discrete phase boundary condition type.

Discrete Phase BC Function

sets the user-defined function from the drop-down list.

Potential

displays the boundary conditions for the electric potential field. This tab is available only if you have enabled either the **Electric Potential model** or the **Electrochemical** reaction model in the Species Dialog Box (p. 4929).

Potential Boundary Condition

is a drop-down list of available potential boundary condition types for the potential generated by electron current: **Specified Flux** and **Specified Value**. For the **Specified Flux** boundary condition, you will need to specify **Current Density** at the wall. For the **Specified Value** boundary condition, you will need to specify **Potential** at the wall.

Electrolyte Potential Boundary Condition

is a drop-down list of available boundary condition types for the potential generated by ionic current. This item is similar to the **Potential Boundary Condition** drop-down list described above and is available only for the Electrolysis and H2 Pump model.

UDS

contains the UDS parameters.

User-Defined Scalar Boundary Condition

appears only if user-defined scalars are specified.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

User Scalar n

specifies whether the scalar is a specified flux or a specified value.

User-Defined Scalar Boundary Value

appears only if user-defined scalars are specified.

User Scalar n

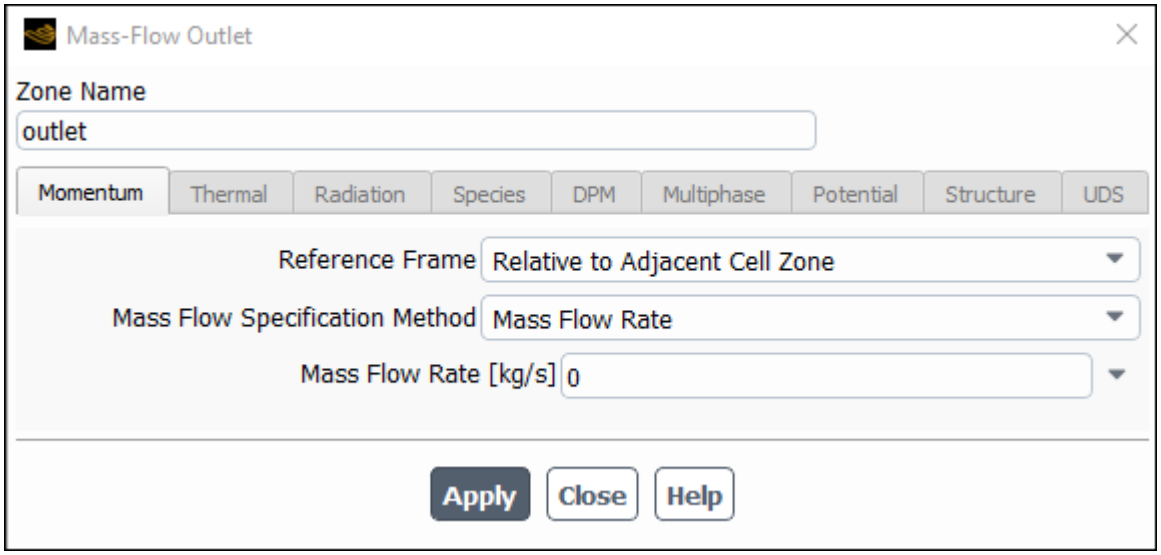
specifies the value of the scalar.

51.7.10. Mass-Flow Outlet Dialog Box

The **Mass-Flow Outlet** dialog box sets the boundary conditions for a mass-flow outlet zone. It is opened from the Boundary Conditions Task Page (p. 4991). See Inputs at Mass-Flow Outlet Boundaries (p. 1426) for details about defining the items below.

Important:

This feature offers reduced functionality when running Fluent under the **Pro** capability level.



Controls

Zone Name

sets the name of the zone.

Momentum

displays the momentum boundary conditions.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reference Frame

specifies the reference frame for the mass flow when the cell zone adjacent to the mass-flow outlet is moving. The only available option is **Relative to Adjacent Cell Zone**, so you must define the mass flow relative to the adjacent cell zone.

Mass Flow Specification Method

specifies whether you are defining a **Mass Flow Rate**, **Mass Flux**, **Mass Flux with Average Mass Flux**, or (if the density of the material is defined either as an ideal gas or using a real gas model) **Exit Corrected Mass Flow Rate**.

Mass Flow Rate

sets the prescribed mass flow rate for the zone. This flow rate is converted internally to a prescribed uniform mass flux over the zone by dividing the flow rate by the flow direction area projection of the zone. This item will appear if you selected **Mass Flow Rate** from the **Mass Flow Specification Method** drop-down list.

Important:

Note that for axisymmetric problems, this mass flow rate is the flow rate through the entire (2π-radian) domain, not through a 1-radian slice.

Mass Flux

sets the prescribed mass flux for the zone. This item will appear if you selected **Mass Flux** or **Mass Flux with Average Mass Flux** from the **Mass Flow Specification Method** drop-down list.

Important:

Note that for axisymmetric problems, this mass flux is the flux through a 1-radian slice of the domain.

Average Mass Flux

sets the average mass flux through the zone. For details, see Defining the Mass Flow Rate or Mass Flux (p. 1426). This item will appear if you selected **Mass Flux with Average Mass Flux** from the **Mass Flow Specification Method** drop-down list.

Important:

Note that for axisymmetric problems, this mass flux is the flux through a 1-radian slice of the domain.

Exit Corrected Mass Flow Rate

sets the value for an exit corrected mass flow rate that is maintained by adjusting the mass flow rate to the total conditions at the outlet. The resulting mass flow rate will be proportional to the exit total pressure and inversely proportional to the square root of the exit total tem-

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

perature (or equivalently, inversely proportional to the stagnation speed of sound). For details, see Exit Corrected Mass Flow Rate (p. 1428). This item will appear if you selected **Exit Corrected Mass Flow Rate** from the **Mass Flow Specification Method** drop-down list.

Important:

Note that for axisymmetric problems, this mass flow rate is the flow rate through the entire (2π-radian) domain, not through a 1-radian slice.

ECMF Reference Temperature

sets the reference temperature for the exit corrected mass flow rate; for details, see Exit Corrected Mass Flow Rate (p. 1428). Typically, it is set to be the same as the inflow total temperature. This item will appear if you selected **Exit Corrected Mass Flow Rate** from the **Mass Flow Specification Method** drop-down list.

ECMF Reference Gauge Pressure

sets the reference gauge pressure for the exit corrected mass flow rate; for details, see Exit Corrected Mass Flow Rate (p. 1428). Typically, it is set to be the same as the inflow total pressure. This item will appear if you selected **Exit Corrected Mass Flow Rate** from the **Mass Flow Specification Method** drop-down list.

Coordinate System

specifies whether **Cartesian**, **Cylindrical**, **Local Cylindrical**,or **Local Cylindrical Swirl** vector components will be defined. This item will appear only for 3D cases in which you have selected **Direction Vector** as the **Direction Specification Method**.

X-, Y-, Z-Component of Flow Direction

set the velocity-direction vector of the outflow stream. This vector does not need to be normalized (for example, you can specify the vector (1 1 1) rather than (0.577 0.577 0.577)). These items will appear if the selected **Coordinate System** is **Cartesian** or the model is 2D non-axisymmetric.

Radial-, Tangential-, Axial-Component of Flow Direction

set the velocity-direction vector of the outflow stream. These items will appear for 2D axisymmetric cases, or for 3D cases for which the selected **Coordinate System** is **Cylindrical** or **Local Cylindrical**.

Axial-, Radial-Component of Flow Direction, Tangential-Velocity

appear for a 3D **Local Cylindrical Swirl** coordinate system.

Axis Origin

sets the **X**, **Y**, and **Z** coordinates of the origin of the local cylindrical (swirl) coordinate system.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Axis Direction

sets the **X**, **Y**, and **Z** components of the direction of the local cylindrical (swirl) coordinate system.

Radiation

contains the radiation parameters.

External Black Body Temperature Method, Black Body Temperature, Internal Emissivity

set the radiation boundary conditions when you are using the P-1, DTRM, DO, S2S, or MC models for radiation heat transfer. See Defining Boundary Conditions for Radiation (p. 2217) for details.

Participates in Solar Ray Tracing

specifies whether or not the mass-flow outlet participates in solar ray tracing.

Solar Transmissivity Factor

specifies a multiplier (ranging from 0 to 1) that is applied to the solar irradiation entering the domain through the mass-flow outlet.

Participates in View Factor Calculation

specifies whether or not the mass-flow outlet participates in the view factor calculation as part of the S2S radiation model. This parameter is available only if you select the **Surface to Surface** radiation model.

DPM

contains the discrete phase parameters.

Discrete Phase BC Type

sets the way that the discrete phase behaves with respect to the boundary. This item appears when one or more injections have been defined.

reflect

rebounds the particle off the boundary with a change in its momentum as defined by the coefficient of restitution. (See Particle Reflection at Wall in the *Fluent Theory Guide*.)

trap

terminates the trajectory calculations and records the fate of the particle as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See Figure 24.40: "Trap" Boundary Condition for the Discrete Phase (p. 2783).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

escape

reports the particle as having "escaped" when it encounters the boundary. Trajectory calculations are terminated. See Figure 24.41: “Escape” Boundary Condition for the Discrete Phase (p. 2783).

reinject

reintroduces the particle into the domain when it reaches a certain domain boundary (for example, outlet). This item appears when one or more injections have been defined and **Unsteady Particle Tracking** is enabled in the **Discrete Phase Model** dialog box. See The **reinject** Boundary Condition (p. 2784) for details.

wall-jet

indicates that the direction and velocity of the droplet particles are given by the resulting momentum flux, which is a function of the impingement angle. See Figure 12.6: "Wall Jet" Boundary Condition for the Discrete Phase in the Theory Guide.

user-defined

specifies a user-defined function to define the discrete phase boundary condition type.

Discrete Phase BC Function

sets the user-defined function from the drop-down list.

UDS

contains the UDS parameters.

User-Defined Scalar Boundary Condition

appears only if user-defined scalars are specified.

User Scalar n

specifies whether the scalar is a specified flux or a specified value.

User-Defined Scalar Boundary Value

appears only if user-defined scalars are specified.

User Scalar n

specifies the value of the scalar.

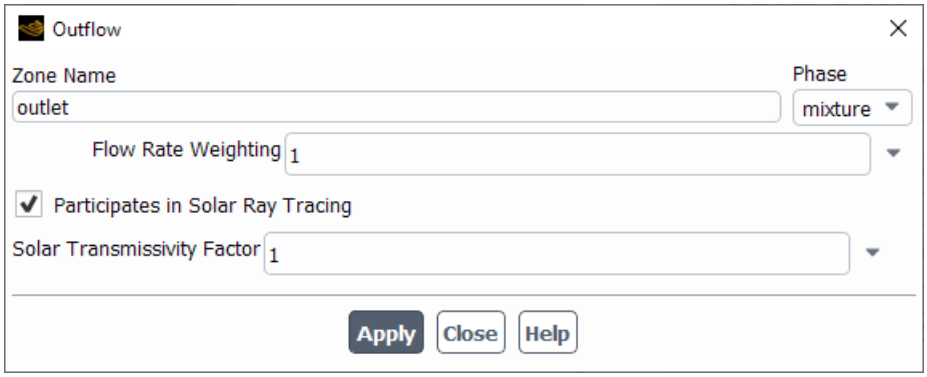
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

51.7.11. Outflow Dialog Box

The **Outflow** dialog box sets the boundary conditions for an outflow zone. It is opened from the Boundary Conditions Task Page (p. 4991). See Using Outflow Boundaries (p. 1449) for details about using outflow boundaries.

Important:

This feature offers reduced functionality when running Fluent under the **Pro** capability level.



Controls

Zone Name

sets the name of the zone.

Flow Rate Weighting

specifies the portion of the outflow that is going through the boundary. See Mass Flow Split Boundary Conditions (p. 1450) for details.

External Black Body Temperature Method, Internal Emissivity

set the radiation boundary conditions when you are using the P-1, DTRM, DO, S2S, or MC models for radiation heat transfer. See Defining Boundary Conditions for Radiation (p. 2217) for details.

Discrete Phase BC Type

sets the way that the discrete phase behaves with respect to the boundary. This item appears when one or more injections have been defined.

reflect

rebounds the particle off the boundary with a change in its momentum as defined by the coefficient of restitution. (See Particle Reflection at Wall in the *Fluent Theory Guide*.)

trap

terminates the trajectory calculations and records the fate of the particle as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and enters the cell adjacent to the boundary. See Figure 24.40: "Trap" Boundary Condition for the Discrete Phase (p. 2783).

escape

reports the particle as having "escaped" when it encounters the boundary. Trajectory calculations are terminated. See Figure 24.41: "Escape" Boundary Condition for the Discrete Phase (p. 2783).

reinject

reintroduces the particle into the domain when it reaches a certain domain boundary (for example, outlet). This item appears when one or more injections have been defined and **Unsteady Particle Tracking** is enabled in the **Discrete Phase Model** dialog box. See The **reinject** Boundary Condition (p. 2784) for details.

wall-jet

indicates that the direction and velocity of the droplet particles are given by the resulting momentum flux, which is a function of the impingement angle. See Figure 12.6: "Wall Jet" Boundary Condition for the Discrete Phase in the Theory Guide.

user-defined

specifies a user-defined function to define the discrete phase boundary condition type.

Participates in Solar Ray Tracing

specifies whether or not outflow participate in solar ray tracing.

Solar Transmissivity Factor

specifies a multiplier (ranging from 0 to 1) that is applied to the solar irradiation entering the domain through the outflow.

Discrete Phase BC Function

sets the user-defined function from the drop-down list.

Participates in View Factor Calculation

specifies whether or not the outflow participates in the view factor calculation as part of the S2S radiation model. This parameter is available only if you select the **Surface to Surface** radiation model.

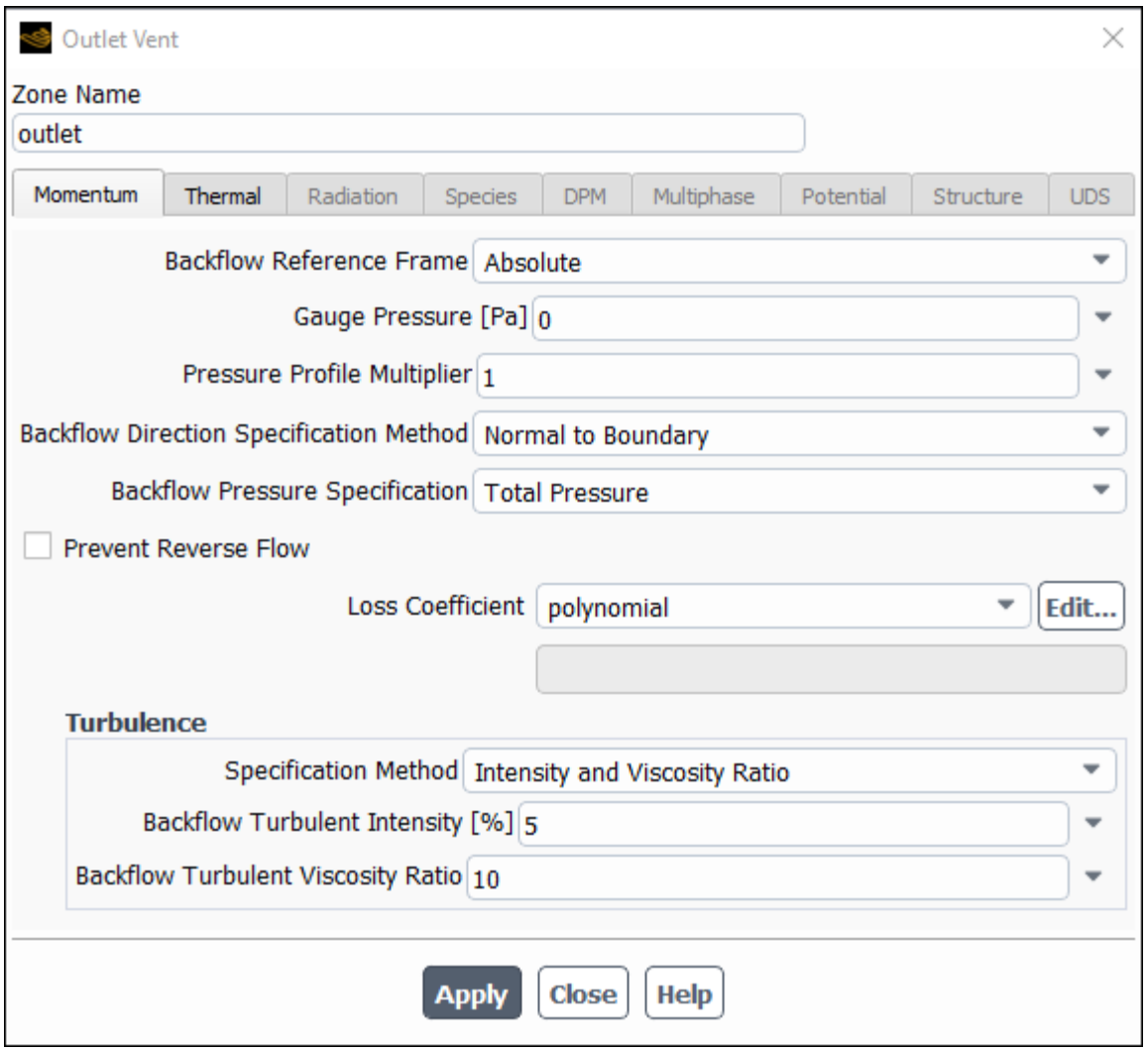
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

51.7.12. Outlet Vent Dialog Box

The **Outlet Vent** dialog box sets the boundary conditions for an outlet vent zone. It is opened from the Boundary Conditions Task Page (p. 4991). See Inputs at Outlet Vent Boundaries (p. 1451) for details about defining the items below.

Important:

This feature offers reduced functionality when running Fluent under the **Pro** capability level..



Controls

Zone Name

sets the name of the zone.

Momentum

contains the momentum parameters.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Backflow Reference Frame

specify whether backflow temperature, pressure, and flow directions are in the **Absolute** or **Relative to the Adjacent Cell Zone** reference frame.

Gauge Pressure

sets the gauge pressure at the outlet boundary.

Backflow Direction Specification Method

specifies the method you will use to define the flow direction. If you choose **Direction Vector**, you will define the flow direction components, and if you choose **Normal to Boundary** or **From Neighboring Cell** no inputs are required. See Defining the Flow Direction (p. 1405) for information on specifying flow direction.

Coordinate System

contains a drop-down list for selecting the coordinate system. You can choose **Cartesian**, **Cylindrical**, or **Local Cylindrical**. This option is available only when **Direction Vector** is selected from the **Backflow Direction Specification Method** drop-down list.

X-, Y-, Z-Component of Flow Direction

allows you to specify the velocity components in x, y, and z directions respectively. This option is available when **Cartesian** is selected for the **Coordinate System**.

Radial-, Tangential-, Axial-Component of Flow Direction

set the direction of the flow at the boundary. These items will appear for 2D axisymmetric cases, or for 3D cases for which the selected **Coordinate System** is **Cylindrical** or **Local Cylindrical**.

Backflow Pressure Specification

specifies how the pressure is calculated under backflow conditions. If you select **Static Pressure**, the **Gauge Pressure** is directly imposed as the boundary face pressure; if you select **Total Pressure**, the **Gauge Pressure** will be combined with a dynamic contribution that is based on the velocity in the adjacent cell zone.

Loss-Coefficient

sets the non-dimensional loss coefficient used to compute the pressure drop. See Specifying the Loss Coefficient (p. 1453) for details.

Axis Origin

sets the **X**, **Y**, and **Z** coordinates of the origin of the local cylindrical coordinate system.

Axis Direction

sets the **X**, **Y**, and **Z** components of the direction of the local cylindrical coordinate system.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Turbulence

contains the turbulence parameters.

Specification Method

specifies which method will be used to define the turbulence parameters. You can choose **K and Epsilon** (k- ε models and RSM only), **K and Omega** (k- ω models only), **Intensity and Length Scale**, **Intensity and Viscosity Ratio**, **Intensity and Hydraulic Diameter**, **Modified Turbulent Viscosity** (Spalart-Allmaras model only), or **Turbulent Viscosity Ratio** (Spalart-Allmaras model only). See Determining Turbulence Parameters (p. 1395) for information about the inputs for each of these methods. (This item will appear only for turbulent flow calculations.)

Backflow Turbulent Kinetic Energy, Backflow Turbulent Dissipation Rate

set values for the turbulence kinetic energy k and its dissipation rate ε. These items will appear if you choose **K and Epsilon** as the **Specification Method**.

Backflow Turbulent Kinetic Energy, Backflow Specific Dissipation Rate

set values for the turbulence kinetic energy k and its specific dissipation rate ω. These items will appear if you choose **K and Omega** as the **Specification Method**.

Backflow Turbulent Intensity, Backflow Turbulent Length Scale

set values for turbulence intensity I and turbulence length scale ℓ. These items will appear if you choose **Intensity and Length Scale** as the **Specification Method**.

Backflow Turbulent Intensity, Backflow Turbulent Viscosity Ratio

set values for turbulence intensity I and turbulent viscosity ratio μ_t / μ. These items will appear if you choose **Intensity and Viscosity Ratio** as the **Specification Method**.

Backflow Turbulent Intensity, Backflow Hydraulic Diameter

set values for turbulence intensity I and hydraulic diameter L. These items will appear if you choose **Intensity and Hydraulic Diameter** as the **Specification Method**.

Backflow Modified Turbulent Viscosity

sets the value of the backflow modified turbulent viscosity $\tilde{\nu}$. This item will appear if you choose **Modified Turbulent Viscosity** as the **Specification Method**.

Backflow Turbulent Viscosity Ratio

sets the value of the backflow turbulent viscosity ratio μ_t / μ. This item will appear if you choose **Turbulent Viscosity Ratio** as the **Specification Method**.

Reynolds-Stress Specification Method

specifies which method will be used to determine the backflow Reynolds stress boundary conditions when the Reynolds stress turbulence model is used. You can choose either **K**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

or Turbulent Intensity or **Reynolds-Stress Components**. If you choose the former, Ansys Fluent will compute the Reynolds stresses for you. If you choose the latter, you will explicitly specify the Reynolds stresses yourself. See Reynolds Stress Model (p. 2144) for details. (This item will appear only for RSM turbulent flow calculations.)

Backflow UU, VV, WW, UV, VW, UW Reynolds Stresses

specify the backflow Reynolds stress components when **Reynolds-Stress Components** is chosen as the **Reynolds-Stress Specification Method**.

Thermal

contains the thermal parameters.

Backflow Total Temperature

sets the total temperature of the inflow stream should the flow reverse direction

Radiation

contains the boundary conditions for the radiation model at the outlet vent.

External Black Body Temperature Method, Internal Emissivity

set the radiation boundary conditions when you are using the P-1, DTRM, DO, S2S, or MC models for radiation heat transfer. See Defining Boundary Conditions for Radiation (p. 2217) for details.

Participates in Solar Ray Tracing

specifies whether or not the outlet vent participates in solar ray tracing.

Solar Transmissivity Factor

specifies a multiplier (ranging from 0 to 1) that is applied to the solar irradiation entering the domain through the outlet vent.

Participates in View Factor Calculation

specifies whether or not the outlet vent participates in the view factor calculation as part of the S2S radiation model. This parameter is available only if you select the **Surface to Surface** radiation model.

Species

contains the species parameters.

Specify Species in Mole Fractions

allows you to specify the species in mole fractions rather than mass fractions.

Mean Mixture Fraction, Mixture Fraction Variance

set inlet values for the PDF mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model.)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Secondary Mean Mixture Fraction, Secondary Mixture Fraction Variance

set inlet values for the secondary mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model with two mixture fractions.)

Species Mass Fractions

contains inputs for the mass fractions of defined species. See Defining Cell Zone and Boundary Conditions for Species (p. 2394) for details about these inputs. These items will appear only if you are modeling non-reacting multi-species flow or you are using the finite-rate reaction formulation.

Backflow Progress Variable

sets the value of the progress variable for premixed turbulent combustion. See Setting Boundary Conditions for the Progress Variable (p. 2541) for details.

This item will appear only if the premixed or partially premixed combustion model is used.

DPM

contains the discrete phase parameters.

Discrete Phase BC Type

sets the way that the discrete phase behaves with respect to the boundary. This item appears when one or more injections have been defined.

reflect

rebounds the particle off the boundary with a change in its momentum as defined by the coefficient of restitution. (See Particle Reflection at Wall in the *Fluent Theory Guide*.)

trap

terminates the trajectory calculations and records the fate of the particle as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See Figure 24.40: "Trap" Boundary Condition for the Discrete Phase (p. 2783).

escape

reports the particle as having "escaped" when it encounters the boundary. Trajectory calculations are terminated. See Figure 24.41: "Escape" Boundary Condition for the Discrete Phase (p. 2783).

reinject

reintroduces the particle into the domain when it reaches a certain domain boundary (for example, outlet). This item appears when one or more injections have been defined and **Unsteady Particle Tracking** is enabled in the **Discrete Phase Model** dialog box. See The **reinject** Boundary Condition (p. 2784) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

wall-jet

indicates that the direction and velocity of the droplet particles are given by the resulting momentum flux, which is a function of the impingement angle. See Figure 12.6: "Wall Jet" Boundary Condition for the Discrete Phase in the Theory Guide.

user-defined

specifies a user-defined function to define the discrete phase boundary condition type.

Discrete Phase BC Function

sets the user-defined function from the drop-down list.

Multiphase

contains the multiphase parameters.

Backflow Granular Temperature

specifies temperature for the solids phase and is proportional to the kinetic energy of the random motion of the particles.

Volume Fraction Specification Method

sets the method used to specify the volume fraction of the secondary phase selected in the Boundary Conditions Task Page (p. 4991). This section of the dialog box will appear when one of the multiphase models is being used. See Defining Multiphase Cell Zone and Boundary Conditions (p. 2957) for details.

Backflow Volume Fraction

specifies the volume fraction of the secondary phase as a constant, profile, of UDF function.

From Neighboring Cell

calculates the volume fraction from the neighboring cells.

Potential

displays the boundary conditions for the electric potential field. This tab is available only if you have enabled either the **Electric Potential model** or the **Electrochemical** reaction model in the Species Dialog Box (p. 4929).

Potential Boundary Condition

is a drop-down list of available potential boundary condition types for the potential generated by electron current: **Specified Flux** and **Specified Value**. For the **Specified Flux** boundary condition, you will need to specify **Current Density** at the wall. For the **Specified Value** boundary condition, you will need to specify **Potential** at the wall.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Electrolyte Potential Boundary Condition

is a drop-down list of available boundary condition types for the potential generated by ionic current. This item is similar to the **Potential Boundary Condition** drop-down list described above and is available only for the Electrolysis and H2 Pump model.

UDS

contains the UDS parameters.

User-Defined Scalar Boundary Condition

appears only if user-defined scalars are specified.

User Scalar n

specifies whether the scalar is a specified flux or a specified value.

User-Defined Scalar Boundary Value

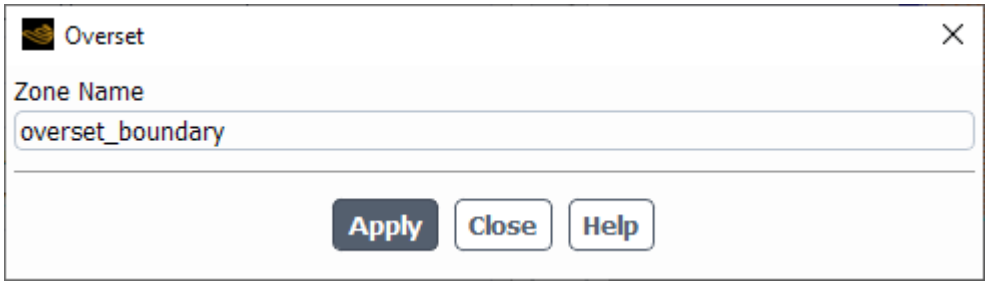
appears only if user-defined scalars are specified.

User Scalar n

specifies the value of the scalar.

51.7.13. Overset Dialog Box

The **Overset** dialog box sets the boundary conditions for an overset zone. It is opened from the Boundary Conditions Task Page (p. 4991). See Setting up an Overset Interface (p. 1227) for further details.



Controls

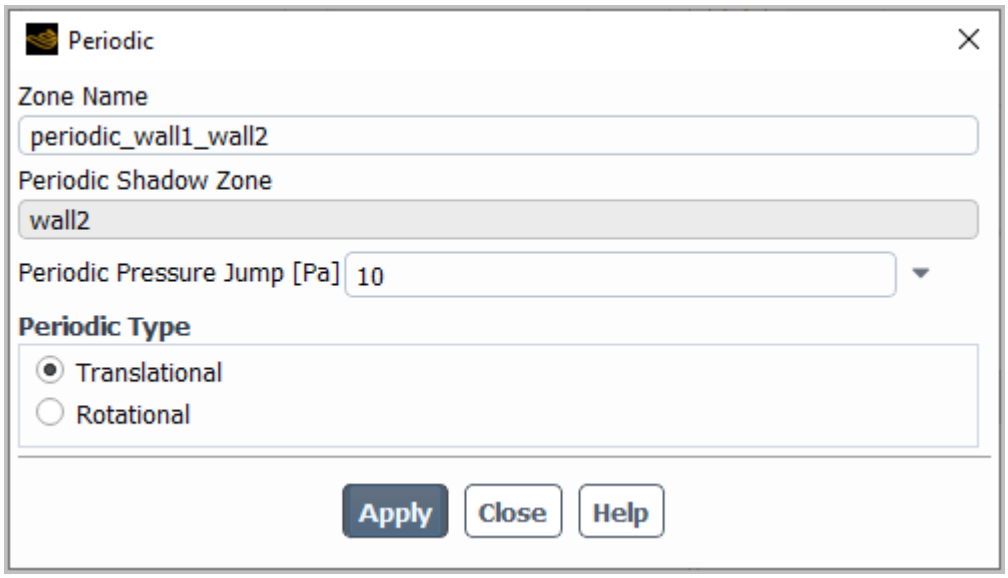
Zone Name

sets the name of the zone.

51.7.14. Periodic Dialog Box

The **Periodic** dialog box sets the boundary conditions for a periodic zone. It is opened from the Boundary Conditions Task Page (p. 4991). See Inputs for Periodic Boundaries (p. 1502) for details about the items below. See Periodic Flows (p. 1747) for information about fully-developed periodic flow.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Zone Name

sets the name of the zone.

Periodic Type

indicates whether the periodicity of the domain is **Translational** or **Rotational**.

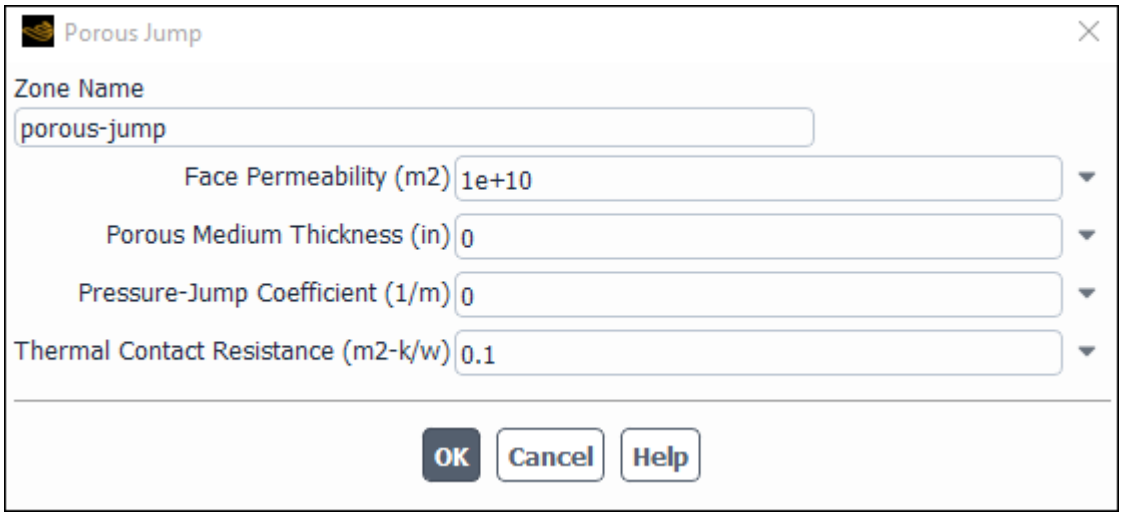
Periodic Pressure Jump

sets the pressure increase/decrease across the periodic boundary. (This item will not appear if the pressure-based (default) solver is used; it is relevant only for the density-based solvers.)

51.7.15. Porous Jump Dialog Box

The **Porous Jump** dialog box sets the boundary conditions for a porous-jump zone. It is opened from the Boundary Conditions Task Page (p. 4991). See Porous Jump Boundary Conditions (p. 1519) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Zone Name

sets the name of the zone.

Face Permeability

sets the face permeability coefficient (α in Equation 7.152 (p. 1519)).

Porous Medium Thickness

sets the thickness of the porous medium (Δm).

Pressure-Jump Coefficient

sets the pressure-jump coefficient (C_2).

Thermal Contact Resistance

sets the thermal contact resistance.

Jump Adhesion

sets the adhesion method and contact angle.

Constrained-Two-Sided Adhesion

constrains the contact angle at the porous jump, When this option is disabled then the forced two-sided adhesion treatment is in effect. See Jump Adhesion for more information.

Contact Angle

is the contact angle at the porous jump.

Discrete Phase BC Type

sets the way that the discrete phase behaves with respect to the boundary. This item appears when one or more injections have been defined.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

interior

allows the particles to pass through the boundary.

reflect

rebounds the particle off the boundary with a change in its momentum as defined by the coefficient of restitution. (See Particle Reflection at Wall in the *Fluent Theory Guide*.)

trap

terminates the trajectory calculations and records the fate of the particle as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See Figure 24.40: "Trap" Boundary Condition for the Discrete Phase (p. 2783).

escape

reports the particle as having "escaped" when it encounters the boundary. Trajectory calculations are terminated. See Figure 24.41: "Escape" Boundary Condition for the Discrete Phase (p. 2783).

reinject

reintroduces the particle into the domain when it reaches a certain domain boundary (for example, outlet). This item appears when one or more injections have been defined and **Unsteady Particle Tracking** is enabled in the **Discrete Phase Model** dialog box. See The **reinject** Boundary Condition (p. 2784) for details.

wall-jet

indicates that the direction and velocity of the droplet particles are given by the resulting momentum flux, which is a function of the impingement angle. See Figure 12.6: "Wall Jet" Boundary Condition for the Discrete Phase in the Theory Guide.

user-defined

specifies a user-defined function to define the discrete phase boundary condition type.

Discrete Phase BC Function

sets the user-defined function from the drop-down list.

Solar Boundary Conditions

contains the settings for solar ray tracing. This group box is available only if you select **Solar Ray Tracing** from the **Model** list in the **Solar Load** group box of the **Radiation Model** dialog box. See Solar Ray Tracing (p. 2261) for details.

Participates in Solar Ray Tracing

specifies whether or not the porous jump participates in solar ray tracing.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Absorptivity

contains the settings that define the absorptivity of the porous jump.

Direct Visible

specifies a multiplier (ranging from 0 to 1) that is applied to the visible portion of the direct solar radiation spectrum to account for the absorption of the porous jump.

Direct IR

specifies a multiplier (ranging from 0 to 1) that is applied to the infrared portion of the direct solar radiation spectrum to account for the absorption of the porous jump.

Transmissivity

contains the settings that define the transmissivity of the porous jump.

Direct Visible

specifies a multiplier (ranging from 0 to 1) that is applied to the visible portion of the direct solar radiation spectrum to account for the transmissivity of the porous jump.

Direct IR

specifies a multiplier (ranging from 0 to 1) that is applied to the infrared portion of the direct solar radiation spectrum to account for the transmissivity of the porous jump.

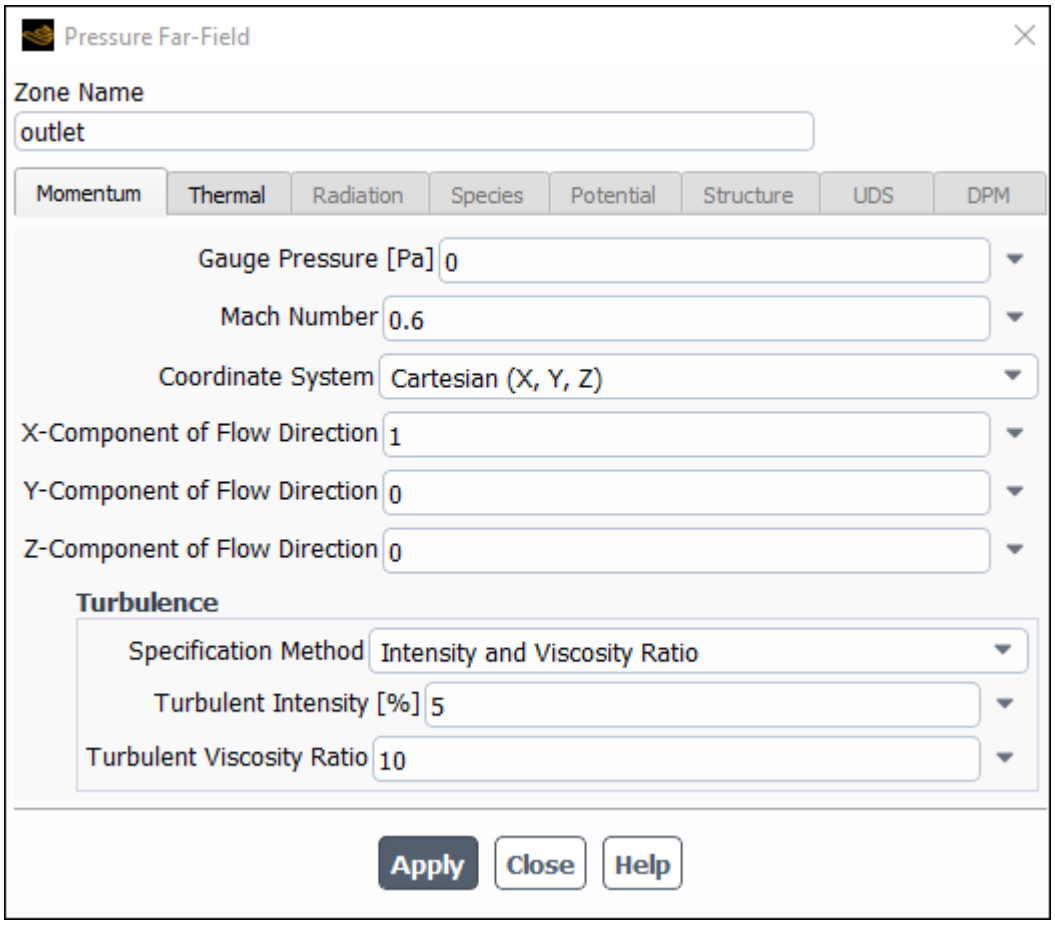
51.7.16. Pressure Far-Field Dialog Box

The **Pressure Far-Field** dialog box sets the boundary conditions for a pressure far-field zone. It is opened from the Boundary Conditions Task Page (p. 4991). See Inputs at Pressure Far-Field Boundaries (p. 1444) for details about defining the items below.

Important:

This feature offers reduced functionality when running Fluent under the **Pro** capability level.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Zone Name

sets the name of the zone.

Momentum

contains the momentum parameters.

Gauge Pressure

sets the far-field gauge static pressure.

Mach Number

sets the far-field Mach number. The Mach number can be subsonic, sonic, or supersonic.

Coordinate System

allows you to select a **Cartesian**, **Cylindrical**, or **Local Cylindrical** coordinate system. This option is available only for 3D geometry.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

X-, Y-, Z-Component of Flow Direction

set the far-field flow direction. These items will appear if the selected **Coordinate System** is **Cartesian** or the model is 2D non-axisymmetric.

Radial-, Tangential-, Axial-Component of Flow Direction

set the far-field flow direction. These items will appear for 2D axisymmetric cases, or for 3D cases for which the selected **Coordinate System** is **Cylindrical** or **Local Cylindrical**.

Axis Origin

sets the **X**, **Y**, and **Z** coordinates of the origin of the local cylindrical coordinate system.

Axis Direction

sets the **X**, **Y**, and **Z** components of the direction of the local cylindrical coordinate system.

Turbulence

contains the turbulence parameters.

Specification Method

specifies which method will be used to define the turbulence parameters. You can choose **K and Epsilon** (k- ε models and RSM only), **K and Omega** (k- ω models only), **Intensity and Length Scale**, **Intensity and Viscosity Ratio**, **Intensity and Hydraulic Diameter**, **Modified Turbulent Viscosity** (Spalart-Allmaras model only), or **Turbulent Viscosity Ratio** (Spalart-Allmaras model only). See Determining Turbulence Parameters (p. 1395) for information about the inputs for each of these methods. (This item will appear only for turbulent flow calculations.)

Turbulent Kinetic Energy, Turbulent Dissipation Rate

set values for the turbulence kinetic energy k and its dissipation rate ε. These items will appear if you choose **K and Epsilon** as the **Specification Method**.

Turbulent Kinetic Energy, Specific Dissipation Rate

set values for the turbulence kinetic energy k and its specific dissipation rate ω. These items will appear if you choose **K and Omega** as the **Specification Method**.

Turbulent Intensity, Turbulent Length Scale

set values for turbulence intensity I and turbulence length scale ℓ. These items will appear if you choose **Intensity and Length Scale** as the **Specification Method**.

Turbulent Intensity, Turbulent Viscosity Ratio

set values for turbulence intensity I and turbulent viscosity ratio μ_t / μ. These items will appear if you choose **Intensity and Viscosity Ratio** as the **Specification Method**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Turbulent Intensity, Hydraulic Diameter

set values for turbulence intensity I and hydraulic diameter L. These items will appear if you choose **Intensity and Hydraulic Diameter** as the **Specification Method**.

Modified Turbulent Viscosity

sets the value of the modified turbulent viscosity $\tilde{\nu}$. This item will appear if you choose **Modified Turbulent Viscosity** as the **Specification Method**.

Turbulent Viscosity Ratio

sets the value of the turbulent viscosity ratio μ_t / μ. This item will appear if you choose **Turbulent Viscosity Ratio** as the **Turbulence Specification Method**.

Reynolds-Stress Specification Method

specifies which method will be used to determine the Reynolds stress boundary conditions when the Reynolds stress turbulence model is used. You can choose either **K or Turbulent Intensity** or **Reynolds-Stress Components**. If you choose the former, Ansys Fluent will compute the Reynolds stresses for you. If you choose the latter, you will explicitly specify the Reynolds stresses yourself. See Reynolds Stress Model (p. 2144) for details. (This item will appear only for RSM turbulent flow calculations.)

UU, VV, WW, UV, VW, UW Reynolds Stresses

specify the Reynolds stress components when **Reynolds-Stress Components** is chosen as the **Reynolds-Stress Specification Method**.

Thermal

contains the thermal parameters.

Temperature

sets the far-field static temperature.

Radiation

contains the boundary conditions for the radiation model at the pressure far-field zone.

External Black Body Temperature Method, Internal Emissivity

set the radiation boundary conditions when you are using the P-1, DTRM, DO, S2S, or MC models for radiation heat transfer. See Defining Boundary Conditions for Radiation (p. 2217) for details.

Participates in Solar Ray Tracing

specifies whether or not the pressure far-field zone participates in solar ray tracing.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solar Transmissivity Factor

specifies a multiplier (ranging from 0 to 1) that is applied to the solar irradiation entering the domain through the pressure far-field zone.

Participates in View Factor Calculation

specifies whether or not the pressure far-field zone participates in the view factor calculation as part of the S2S radiation model. This parameter is available only if you select the **Surface to Surface** radiation model.

Species

contains the species parameters.

Specify Species in Mole Fractions

allows you to specify the species in mole fractions rather than mass fractions.

Species Mass Fractions

contains inputs for the mass fractions of defined species. See Defining Cell Zone and Boundary Conditions for Species (p. 2394) for details about these inputs. (These items will appear only if you are modeling non-reacting multi-species flow or you are using the finite-rate reaction formulation.)

Mean Mixture Fraction, Mixture Fraction Variance

set inlet values for the PDF mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model.)

Secondary Mean Mixture Fraction, Secondary Mixture Fraction Variance

set inlet values for the secondary mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model with two mixture fractions.)

Progress Variable

sets the value of the progress variable for premixed turbulent combustion. See Setting Boundary Conditions for the Progress Variable (p. 2541) for details.

This item will appear only if the premixed or partially premixed combustion model is used.

UDS

contains the UDS parameters.

User-Defined Scalar Boundary Condition

appears only if user-defined scalars are specified.

User Scalar n

specifies whether the scalar is a specified flux or a specified value.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

User-Defined Scalar Boundary Value

appears only if user-defined scalars are specified.

User Scalar n

specifies the value of the scalar.

DPM

contains the discrete phase parameters.

Discrete Phase BC Type

sets the way that the discrete phase behaves with respect to the boundary. This item appears when one or more injections have been defined.

reflect

rebounds the particle off the boundary with a change in its momentum as defined by the coefficient of restitution. (See Particle Reflection at Wall in the *Fluent Theory Guide*.)

trap

terminates the trajectory calculations and records the fate of the particle as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See Figure 24.40: “Trap” Boundary Condition for the Discrete Phase (p. 2783).

escape

reports the particle as having "escaped" when it encounters the boundary. Trajectory calculations are terminated. See Figure 24.41: “Escape” Boundary Condition for the Discrete Phase (p. 2783).

reinject

reintroduces the particle into the domain when it reaches a certain domain boundary (for example, outlet). This item appears when one or more injections have been defined and **Unsteady Particle Tracking** is enabled in the **Discrete Phase Model** dialog box. See The **reinject** Boundary Condition (p. 2784) for details.

wall-jet

indicates that the direction and velocity of the droplet particles are given by the resulting momentum flux, which is a function of the impingement angle. See Figure 12.6: "Wall Jet" Boundary Condition for the Discrete Phase in the Theory Guide.

user-defined

specifies a user-defined function to define the discrete phase boundary condition type.

Discrete Phase BC Function

sets the user-defined function from the drop-down list.

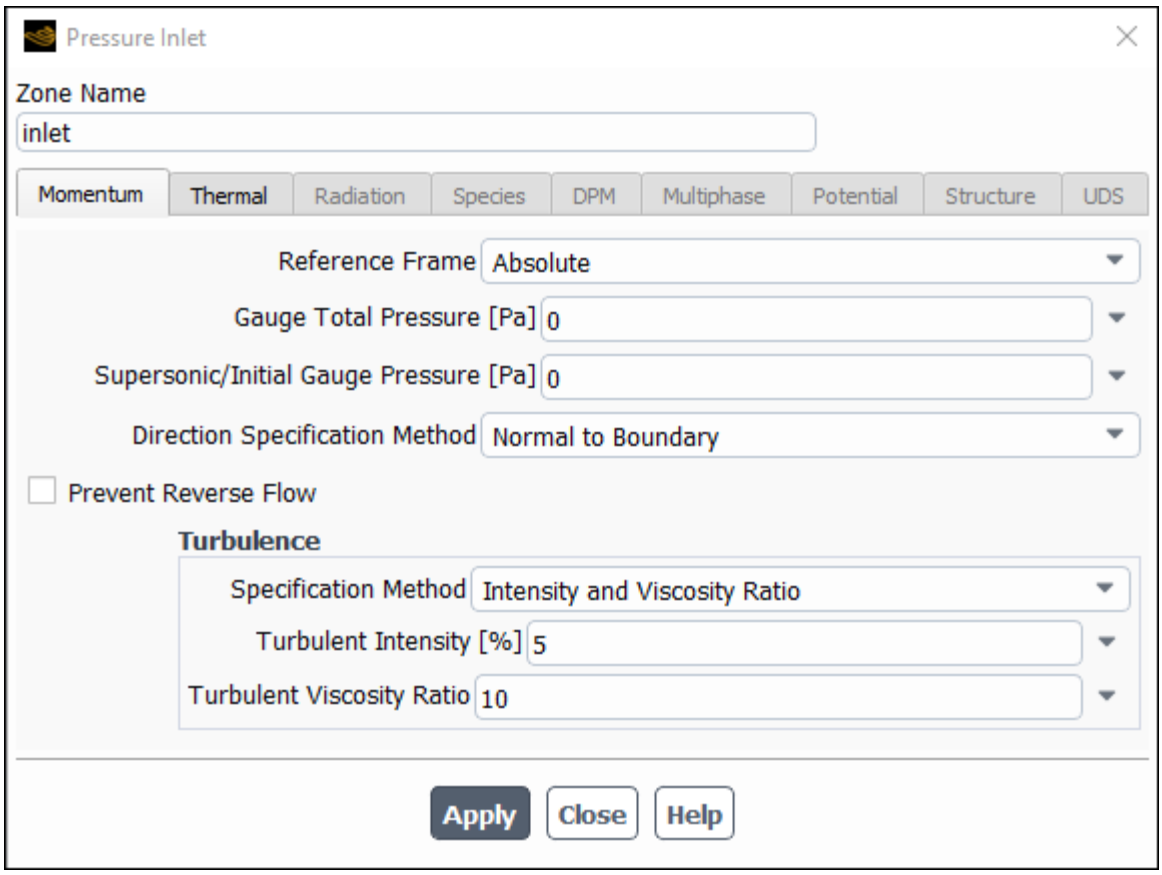
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

51.7.17. Pressure Inlet Dialog Box

The **Pressure Inlet** dialog box sets the boundary conditions for a pressure inlet zone. It is opened from the Boundary Conditions Task Page (p. 4991). See Inputs at Pressure Inlet Boundaries (p. 1402) for details about defining the items below.

Important:

This feature offers reduced functionality when running Fluent under the **Pro** capability level.



Controls

Zone Name

sets the name of the zone.

Momentum

contains the momentum parameters.

Reference Frame

specifies the reference frame for the pressure inlet. If the cell zone adjacent to the pressure inlet is moving, you can choose to specify the total temperature, total pressure, and velocity

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

components as **Relative to Adjacent CellZone** or **Absolute** in the **Reference Frame** drop-down list.

Gauge Total Pressure

sets the gauge total (or stagnation) pressure of the inflow stream. If you are using moving reference frames, see Defining Total Pressure and Temperature (p. 1404) for information about relative and absolute total pressure.

Supersonic/Initial Gauge Pressure

sets the static pressure on the boundary when the flow becomes (locally) supersonic. It is also used to compute initial values for pressure, temperature, and velocity if the pressure inlet boundary condition is selected for computing initial values (see Initializing the Entire Flow Field Using Standard Initialization (p. 3628)).

Direction Specification Method

specifies the method you will use to define the flow direction. If you choose **Direction Vector**, you will define the flow direction components, and if you choose **Normal to Boundary** no inputs are required. You also have the option to enable **Prevent Reverse Flow**. See Defining the Flow Direction (p. 1405) for information on specifying flow direction.

Coordinate System

specifies whether **Cartesian**, **Cylindrical**, **Local Cylindrical**,or **Local Cylindrical Swirl** vector components will be defined. This item will appear only for 3D cases in which you have selected **Direction Vector** as the **Direction Specification Method**.

X,Y,Z-Component of Flow Direction

set the direction of the flow at the inlet boundary. These items will appear if the selected **Coordinate System** is **Cartesian** or the model is 2D non-axisymmetric.

Radial, Tangential, Axial Component of Flow Direction

set the direction of the flow at the inlet boundary. These items will appear for 2D axisymmetric cases, or for 3D cases for which the selected **Coordinate System** is **Cylindrical** or **Local Cylindrical**.

Radial-, Axial-Component of Flow Direction, Tangential-Velocity

appear for a 3D **Local Cylindrical Swirl** coordinate system.

Axis Origin

sets the **X**, **Y**, and **Z** coordinates of the origin of the local cylindrical (swirl) coordinate system.

Axis Direction

sets the **X**, **Y**, and **Z** components of the direction of the local cylindrical (swirl) coordinate system.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Prevent Reverse Flow

Enables Fluent to erect artificial walls on the boundary mesh faces to prevent flow out of the domain. The artificial walls are removed when the flow is no longer leaving the domain and when a favorable pressure gradient is recovered at the boundary mesh faces.

Fluctuating Velocity

contains parameters for Inlet Boundary Conditions for Scale Resolving Simulations (only available for transient cases where scale-resolving turbulence models such as SAS, DES, and LES are enabled).

Fluctuating Velocity Algorithm

Specifies which method will be used for generating fluctuations at the boundary for Scale Resolving Simulations. The following methods are available:

No Perturbations

No fluctuations added.

Vortex Method

For details, see Vortex Method in the *Fluent Theory Guide*.

Spectral Synthesizer

For details, see Spectral Synthesizer in the *Fluent Theory Guide*.

Synthetic Turbulence Generator

For details, see Synthetic Turbulence Generator in the *Fluent Theory Guide*.

Number of Vortices

Specifies number of vortices to be generated with the **Vortex Method**.

Satisfy Mass Conservation?

Enables mass conservation for the **Vortex Method** at the boundary.

Scale Search Limiter

Specifies which method will be used to limit the area from where global scales for the **Synthetic Turbulence Generator** are computed. The following methods are available:

None

Scales are computed using the entire inlet zone.

Turbulent Intensity Threshold

Scales are computed using only part of the inlet zone where Turbulent Intensity is higher than specified threshold value.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Turbulent Viscosity Ratio Threshold

Scales are computed using only part of the inlet zone where Turbulent Viscosity Ratio is higher than specified threshold value.

Wall Distance Threshold

Scales are computed using only part of the inlet zone where Wall Distance is smaller than specified threshold value.

Turbulent Intensity, Turbulent Viscosity Ratio, or Wall Distance

Specifies threshold value for the corresponding **Scale Search Limiter** (only available for the **Synthetic Turbulence Generator** with corresponding **Scale Search Limiter** chosen).

Volumetric Forcing?

Enables generation of fluctuations in the volume adjacent to the inlet boundary instead of introduction of fluctuations directly at the boundary (only available for the **Synthetic Turbulence Generator**).

Forcing Zone Thickness

Specifies which method will be used to set up the streamwise size of the volume where fluctuations are to be generated (only available for the **Synthetic Turbulence Generator** with **Volumetric Forcing?** enabled). The following methods are available:

Automatic

Length is computed based on the turbulent length scale computed from the boundary.

Specified Value

Length is specified by the user.

Thickness

Specifies the streamwise size of the volume where fluctuations are to be generated (only available for the **Synthetic Turbulence Generator** with **Volumetric Forcing?** enabled if **Specified Value** option is chosen for **Forcing Zone Thickness**).

Turbulence

contains the turbulence parameters.

Specification Method

specifies which method will be used to define the turbulence parameters. You can choose **K and Epsilon** (k- ε models and RSM only), **K and Omega** (k- ω models only), **Intensity and Length Scale**, **Intensity and Viscosity Ratio**, **Intensity and Hydraulic Diameter**, **Modified Turbulent Viscosity** (Spalart-Allmaras model only), or **Turbulent Viscosity Ratio** (Spalart-Allmaras model only). See Determining Turbulence Parameters (p. 1395) for information about the inputs for each of these methods. (This item will appear only for turbulent flow calculations.)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Turbulent Kinetic Energy, Turbulent Dissipation Rate

set values for the turbulence kinetic energy k and its dissipation rate ε. These items will appear if you choose **K and Epsilon** as the **Specification Method**.

Turbulent Kinetic Energy, Specific Dissipation Rate

set values for the turbulence kinetic energy k and its specific dissipation rate ω. These items will appear if you choose **K and Omega** as the **Specification Method**.

Turbulent Intensity, Turbulent Length Scale

set values for turbulence intensity I and turbulence length scale ℓ. These items will appear if you choose **Intensity and Length Scale** as the **Specification Method**.

Turbulent Intensity, Turbulent Viscosity Ratio

set values for turbulence intensity I and turbulent viscosity ratio μ_t / μ. These items will appear if you choose **Intensity and Viscosity Ratio** as the **Specification Method**.

Turbulent Intensity, Hydraulic Diameter

set values for turbulence intensity I and hydraulic diameter L. These items will appear if you choose **Intensity and Hydraulic Diameter** as the **Specification Method**.

Modified Turbulent Viscosity

sets the value of the modified turbulent viscosity $\tilde{\nu}$. This item will appear if you choose **Modified Turbulent Viscosity** as the **Specification Method**.

Turbulent Viscosity Ratio

sets the value of the turbulent viscosity ratio μ_t / μ. This item will appear if you choose **Turbulent Viscosity Ratio** as the **Specification Method**.

Reynolds-Stress Specification Method

specifies which method will be used to determine the Reynolds stress boundary conditions when the Reynolds stress turbulence model is used. You can choose either **K or Turbulent Intensity** or **Reynolds-Stress Components**. If you choose the former, Ansys Fluent will compute the Reynolds stresses for you. If you choose the latter, you will explicitly specify the Reynolds stresses yourself. See Reynolds Stress Model (p. 2144) for details. (This item will appear only for RSM turbulent flow calculations.)

UU, VV, WW, UV, VW, UW Reynolds Stresses

specify the Reynolds stress components when **Reynolds-Stress Components** is chosen as the **Reynolds-Stress Specification Method**.

Acoustic Wave Model

contains settings for treatment of acoustic pressure waves at the boundary.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Off

disables special treatment of acoustic pressure waves at the boundary.

Non Reflecting

enables the general non-reflecting boundary condition treatment described in General Non-Reflecting Boundary Conditions (p. 1538).

Impedance

enables the impedance boundary condition treatment described in Impedance Boundary Conditions (p. 1545).

Transparent Flow Forcing

enables the transparent flow forcing boundary condition described in Transparent Flow Forcing Boundary Conditions (p. 1554).

Impedance Parameters

contains the parameters for the impedance boundary condition treatment. For details refer to Using the Impedance Boundary Condition (p. 1546).

Transparent Flow Forcing Parameters

contains the parameters for the transparent flow forcing boundary condition treatment. For details refer to Using the Transparent Flow Forcing Boundary Condition (p. 1554).

Thermal

contains the thermal parameters.

Total Temperature

sets the total temperature of the inflow stream. If you are using moving reference frames, see Defining Total Pressure and Temperature (p. 1404) for information about relative and absolute total temperature.

Radiation

contains the boundary conditions for the radiation model at the pressure inlet.

External Black Body Temperature Method, Internal Emissivity

set the radiation boundary conditions when you are using the P-1, DTRM, DO, S2S, or MC models for radiation heat transfer. See Defining Boundary Conditions for Radiation (p. 2217) for details.

Participates in Solar Ray Tracing

specifies whether or not the pressure inlet participates in solar ray tracing.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solar Transmissivity Factor

specifies a multiplier (ranging from 0 to 1) that is applied to the solar irradiation entering the domain through the pressure inlet.

Participates in View Factor Calculation

specifies whether or not the pressure inlet participates in the view factor calculation as part of the S2S radiation model. This parameter is available only if you select the **Surface to Surface** radiation model.

Species

contains the species parameters.

Specify Species in Mole Fractions

allows you to specify the species in mole fractions rather than mass fractions.

Species Mass Fractions

contains inputs for the mass fractions of defined species. See Defining Cell Zone and Boundary Conditions for Species (p. 2394) for details about these inputs. (These items will appear only if you are modeling non-reacting multi-species flow or you are using the finite-rate reaction formulation.)

Mean Mixture Fraction, Mixture Fraction Variance

set inlet values for the PDF mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model.)

Secondary Mean Mixture Fraction, Secondary Mixture Fraction Variance

set inlet values for the secondary mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model with two mixture fractions.)

Progress Variable

sets the value of the progress variable for premixed turbulent combustion. See Setting Boundary Conditions for the Progress Variable (p. 2541) for details.

This item will appear only if the premixed or partially premixed combustion model is used.

Scalar Mass Fractions

(partially-premixed combustion FGM model only) allows you to specify the scalar species mass fractions for the transported scalars that you selected in the **Select Transported Scalars** dialog box.

DPM

contains the discrete phase parameters.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Discrete Phase BC Type

sets the way that the discrete phase behaves with respect to the boundary. This item appears when one or more injections have been defined.

reflect

rebounds the particle off the boundary with a change in its momentum as defined by the coefficient of restitution. (See Particle Reflection at Wall in the *Fluent Theory Guide*.)

trap

terminates the trajectory calculations and records the fate of the particle as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See Figure 24.40: “Trap” Boundary Condition for the Discrete Phase (p. 2783).

escape

reports the particle as having "escaped" when it encounters the boundary. Trajectory calculations are terminated. See Figure 24.41: “Escape” Boundary Condition for the Discrete Phase (p. 2783).

reinject

reintroduces the particle into the domain when it reaches a certain domain boundary (for example, outlet). This item appears when one or more injections have been defined and **Unsteady Particle Tracking** is enabled in the **Discrete Phase Model** dialog box. See The **reinject** Boundary Condition (p. 2784) for details.

wall-jet

indicates that the direction and velocity of the droplet particles are given by the resulting momentum flux, which is a function of the impingement angle. See Figure 12.6: "Wall Jet" Boundary Condition for the Discrete Phase in the Theory Guide.

user-defined

specifies a user-defined function to define the discrete phase boundary condition type.

Discrete Phase BC Function

sets the user-defined function from the drop-down list.

Multiphase

contains the multiphase parameters.

Granular Temperature

specifies temperature for the solids phase and is proportional to the kinetic energy of the random motion of the particles.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Volume Fraction

specifies the volume fraction of the secondary phase selected in the Boundary Conditions Task Page (p. 4991). This section of the dialog box will appear when one of the multiphase models is being used. See Defining Multiphase Cell Zone and Boundary Conditions (p. 2957) for details.

Potential

displays the boundary conditions for the electric potential field. This tab is available only if you have enabled either the **Electric Potential model** or the **Electrochemical** reaction model in the Species Dialog Box (p. 4929).

Potential Boundary Condition

is a drop-down list of available potential boundary condition types for the potential generated by electron current: **Specified Flux** and **Specified Value**. For the **Specified Flux** boundary condition, you will need to specify **Current Density** at the wall. For the **Specified Value** boundary condition, you will need to specify **Potential** at the wall.

Electrolyte Potential Boundary Condition

is a drop-down list of available boundary condition types for the potential generated by ionic current. This item is similar to the **Potential Boundary Condition** drop-down list described above and is available only for the Electrolysis and H2 Pump model.

UDS

contains the UDS parameters.

User-Defined Scalar Boundary Condition

appears only if user-defined scalars are specified.

User Scalar n

specifies whether the scalar is a specified flux or a specified value.

User-Defined Scalar Boundary Value

appears only if user-defined scalars are specified.

User Scalar n

specifies the value of the scalar.

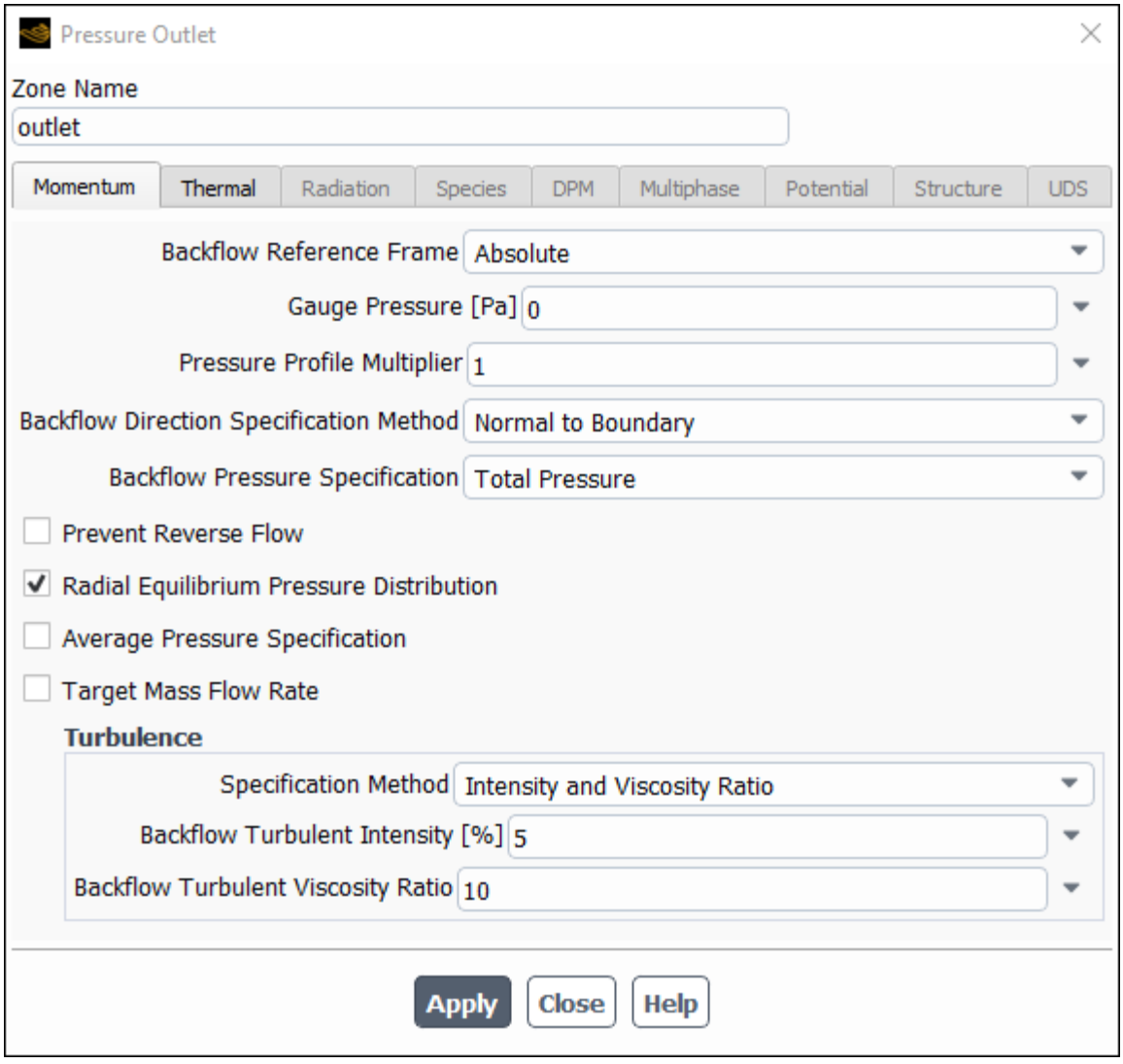
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

51.7.18. Pressure Outlet Dialog Box

The **Pressure Outlet** dialog box sets the boundary conditions for a pressure outlet zone. It is opened from the Boundary Conditions Task Page (p. 4991). See Inputs at Pressure Outlet Boundaries (p. 1434) for details about defining the items below.

Important:

This feature offers reduced functionality when running Fluent under the **Pro** capability level.



Controls

Zone Name

sets the name of the zone.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Momentum

contains the momentum parameters.

Backflow Reference Frame

specify whether backflow temperature, pressure, and flow directions are in the **Absolute** or **Relative to the Adjacent Cell Zone** reference frame.

Gauge Pressure

sets the gauge pressure at the outflow boundary.

Pressure Profile Multiplier

sets a factor by which the **Gauge Pressure** is multiplied. This is provided mainly for cases where a non-uniform distribution of the static pressure at the pressure outlet boundary is specified by means of a profile file or a user-defined function.

Backflow Direction Specification Method

sets the direction of the inflow stream should the flow reverse direction. If you choose **Direction Vector**, you will define the flow direction components, and if you choose **Normal to Boundary** or **From Neighboring Cell**, no inputs are required. You also have the option to enable **Prevent Reverse Flow**. See Inputs at Pressure Outlet Boundaries (p. 1434) for information on specifying flow direction.

Coordinate System

contains a drop-down list for selecting the coordinate system. You can choose **Cartesian**, **Cylindrical**, or **Local Cylindrical**. This option is available only when **Direction Vector** is selected from the **Backflow Direction Specification Method** drop-down list.

X-, Y-, Z-Component of Flow Direction

allows you to specify the velocity components in x, y, and z directions respectively. This option is available when **Cartesian** is selected for the **Coordinate System**.

Radial-, Tangential-, Axial-Component of Flow Direction

set the direction of the flow at the boundary. These items will appear for 2D axisymmetric cases, or for 3D cases for which the selected **Coordinate System** is **Cylindrical** or **Local Cylindrical**.

Backflow Pressure Specification

specifies how the pressure is calculated under backflow conditions. When the NRBC option is not enabled, if you select **Static Pressure**, the **Gauge Pressure** is directly imposed as the boundary face pressure; if you select **Total Pressure**, the **Gauge Pressure** will be combined with a dynamic contribution that is based on the velocity in the adjacent cell zone. For NRBCs, if you select **Static Pressure**, the **Gauge Pressure** is directly imposed as the pressure at infinity; if you select **Total Pressure**, the pressure at infinity is a calculated value that is based on the **Gauge Pressure**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Radial Equilibrium Pressure Distribution

enables the radial equilibrium pressure distribution. See Defining Static Pressure (p. 1436) for details.

This item appears only for 3D and axisymmetric swirl solvers.

Average Pressure Specification

allows the pressure along the outlet boundary to vary, but maintain an average equivalent to the specified value in the **Gauge Pressure** input field. In this boundary implementation, the pressure variation provides a low level of non-reflectivity. For more details, see Calculation Procedure at Pressure Outlet Boundaries (p. 1439).

Note:

The **Average Pressure Specification** option is not available if the **Radial Equilibrium Pressure Distribution** option is enabled.

Target Mass Flow Rate

allows you to set mass flow rate as a boundary condition at the outlet.

Target Mass Flow

allows you to specify the flow as either a constant value or a user-defined function.

Upper Limit of Absolute Pressure, Lower Limit of Absolute Pressure

specifies the range of the pressure limits, which have different pressure variations on different boundaries. The upper and lower pressure limits can be specified as a constant or a profile.

Turbulence

contains the turbulence parameters.

Specification Method

specifies which method will be used to define the turbulence parameters. You can choose **K and Epsilon** (k- ε models and RSM only), **K and Omega** (k- ω models only), **Intensity and Length Scale**, **Intensity and Viscosity Ratio**, **Intensity and Hydraulic Diameter**, **Modified Turbulent Viscosity** (Spalart-Allmaras model only), or **Turbulent Viscosity Ratio** (Spalart-Allmaras model only). See Determining Turbulence Parameters (p. 1395) for information about the inputs for each of these methods. (This item will appear only for turbulent flow calculations.)

Backflow Turbulent Kinetic Energy, Backflow Turbulent Dissipation Rate

set values for the turbulence kinetic energy k and its dissipation rate ε. These items will appear if you choose **K and Epsilon** as the **Specification Method**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Backflow Turbulent Kinetic Energy, Backflow Specific Dissipation Rate

set values for the turbulence kinetic energy k and its specific dissipation rate ω. These items will appear if you choose **K and Omega** as the **Specification Method**.

Backflow Turbulent Intensity, Backflow Turbulent Length Scale

set values for turbulence intensity I and turbulence length scale ℓ. These items will appear if you choose **Intensity and Length Scale** as the **Specification Method**.

Backflow Turbulent Intensity, Backflow Turbulent Viscosity Ratio

set values for turbulence intensity I and turbulent viscosity ratio μ_t / μ. These items will appear if you choose **Intensity and Viscosity Ratio** as the **Specification Method**.

Backflow Turbulent Intensity, Backflow Hydraulic Diameter

set values for turbulence intensity I and hydraulic diameter L. These items will appear if you choose **Intensity and Hydraulic Diameter** as the **Specification Method**.

Backflow Modified Turbulent Viscosity

sets the value of the backflow modified turbulent viscosity $\tilde{\nu}$. This item will appear if you choose **Modified Turbulent Viscosity** as the **Specification Method**.

Backflow Turbulent Viscosity Ratio

sets the value of the backflow turbulent viscosity ratio μ_t / μ. This item will appear if you choose **Turbulent Viscosity Ratio** as the **Specification Method**.

Reynolds-Stress Specification Method

specifies which method will be used to determine the backflow Reynolds stress boundary conditions when the Reynolds stress turbulence model is used. You can choose either **K or Turbulent Intensity** or **Reynolds-Stress Components**. If you choose the former, Ansys Fluent will compute the Reynolds stresses for you. If you choose the latter, you will explicitly specify the Reynolds stresses yourself. See Reynolds Stress Model (p. 2144) for details. (This item will appear only for RSM turbulent flow calculations.)

Backflow UU, VV, WW, UV, VW, UW Reynolds Stresses

specify the backflow Reynolds stress components when **Reynolds-Stress Components** is chosen as the **Reynolds-Stress Specification Method**.

Acoustic Wave Model

contains settings for treatment of acoustic pressure waves at the boundary.

Off

disables special treatment of acoustic pressure waves at the boundary.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Non Reflecting

enables the general non-reflecting boundary condition treatment described in General Non-Reflecting Boundary Conditions (p. 1538).

Impedance

enables the impedance boundary condition treatment described in Impedance Boundary Conditions (p. 1545).

Transparent Flow Forcing

enables the transparent flow forcing boundary condition described in Transparent Flow Forcing Boundary Conditions (p. 1554).

Non Reflecting Parameters

contains settings for the non-reflecting boundary condition treatment.

Exit Pressure Specification

defines how the pressure is calculated. For **Pressure at Infinity**, the pressure at the boundary relaxes toward the **Gauge Pressure** at infinity; for **Average Boundary Pressure**, an effort is made to force the average pressure on the boundary to approach the **Gauge Pressure** value.

Impedance Parameters

contains the parameters for the impedance boundary condition treatment. For details refer to Using the Impedance Boundary Condition (p. 1546).

Transparent Flow Forcing Parameters

contains the parameters for the transparent flow forcing boundary condition treatment. For details refer to Using the Transparent Flow Forcing Boundary Condition (p. 1554).

Thermal

contains the thermal parameters.

Backflow Total Temperature

sets the total temperature of the inflow stream should the flow reverse direction

Radiation

contains the boundary conditions for the radiation model at the pressure outlet.

External Black Body Temperature Method, Internal Emissivity

set the radiation boundary conditions when you are using the P-1, DTRM, DO, S2S, or MC models for radiation heat transfer. See Defining Boundary Conditions for Radiation (p. 2217) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Participates in Solar Ray Tracing

specifies whether or not the pressure outlet participates in solar ray tracing.

Solar Transmissivity Factor

specifies a multiplier (ranging from 0 to 1) that is applied to the solar irradiation entering the domain through the pressure outlet.

Participates in View Factor Calculation

specifies whether or not the pressure outlet participates in the view factor calculation as part of the S2S radiation model. This parameter is available only if you select the **Surface to Surface** radiation model.

Species

contains the species parameters.

Specify Species in Mole Fractions

allows you to specify the species in mole fractions rather than mass fractions.

Mean Mixture Fraction, Mixture Fraction Variance

set inlet values for the PDF mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model.)

Secondary Mean Mixture Fraction, Secondary Mixture Fraction Variance

set inlet values for the secondary mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model with two mixture fractions.)

Species Mass Fractions

contains inputs for the mass fractions of defined species. See Defining Cell Zone and Boundary Conditions for Species (p. 2394) for details about these inputs. These items will appear only if you are modeling non-reacting multi-species flow or you are using the finite-rate reaction formulation.

Backflow Progress Variable

sets the value of the progress variable for premixed turbulent combustion. See Setting Boundary Conditions for the Progress Variable (p. 2541) for details.

This item will appear only if the premixed or partially premixed combustion model is used.

Scalar Mass Fractions

(partially-premixed combustion FGM model only) allows you to specify the scalar species mass fractions for the transported scalars that you selected in the **Select Transported Scalars** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DPM

contains the discrete phase parameters.

Discrete Phase BC Type

sets the way that the discrete phase behaves with respect to the boundary. This item appears when one or more injections have been defined.

reflect

rebounds the particle off the boundary with a change in its momentum as defined by the coefficient of restitution. (See Particle Reflection at Wall in the *Fluent Theory Guide*.)

trap

terminates the trajectory calculations and records the fate of the particle as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See Figure 24.40: “Trap” Boundary Condition for the Discrete Phase (p. 2783).

escape

reports the particle as having "escaped" when it encounters the boundary. Trajectory calculations are terminated. See Figure 24.41: “Escape” Boundary Condition for the Discrete Phase (p. 2783).

reinject

reintroduces the particle into the domain when it reaches a certain domain boundary (for example, outlet). This item appears when one or more injections have been defined and **Unsteady Particle Tracking** is enabled in the **Discrete Phase Model** dialog box. See The **reinject** Boundary Condition (p. 2784) for details.

wall-jet

indicates that the direction and velocity of the droplet particles are given by the resulting momentum flux, which is a function of the impingement angle. See Figure 12.6: "Wall Jet" Boundary Condition for the Discrete Phase in the Theory Guide.

user-defined

specifies a user-defined function to define the discrete phase boundary condition type.

Discrete Phase BC Function

sets the user-defined function from the drop-down list.

Multiphase

contains the multiphase parameters.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Backflow Granular Temperature

specifies temperature for the solids phase and is proportional to the kinetic energy of the random motion of the particles. This option is available only for a granular phase with the Mixture and Eulerian multiphase models.

Volume Fraction Specification Method

sets the method used to specify the volume fraction of the secondary phase selected in the Boundary Conditions Task Page (p. 4991). This section of the dialog box will appear when one of the multiphase models is being used. See Defining Multiphase Cell Zone and Boundary Conditions (p. 2957) for details. This control is not available for the open channel VOF submodel.

Backflow Volume Fraction

specifies the volume fraction of the secondary phase as a constant, profile, of UDF function.

From Neighboring Cell

calculates the volume fraction from the neighboring cells.

Open Channel

(VOF model only) enables the open-channel flow boundary condition. See Modeling Open Channel Flows (p. 2979) for more information. This option is available only when the **Open Channel** VOF submodel is selected in the **Multiphase Model** dialog box. Once you select this option, you can specify the following conditions:

Outlet Group ID

is used to identify the different outlets that are part of the same outlet group. See Setting the Outlet Group (p. 2982) for details.

Pressure Specification Method

specifies which method will be used to define the boundary conditions at the open-channel pressure outlet. You can choose from the following options:

- **Free Surface Level**
- **From Neighboring Cell**
- **Gauge Pressure**

Refer to Pressure Outlet in the *Fluent Theory Guide* for more details.

Secondary Phase for Level Specification

allows you to choose the desired secondary phase. This control is available only for flow applications with more than two phases.

Free Surface Level

can be determined by using the absolute value of height from the free surface to the origin in the direction of gravity and applying the correct sign based on whether the free surface level is above (positive) or below (negative) the origin. See Determining the Free Surface Level (p. 2982) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Bottom Level

is used for calculating the liquid height. It can be determined as described in Determining the Bottom Level (p. 2983).

Density Interpolation Method

allows you to specify whether the mixture density used in the hydrostatic profile will be interpolated using the **From Neighboring Cell**, **From Free Surface Level**, or **Hybrid** method. See Choosing the Density Interpolation Method (p. 2986) for more information. This drop-down list is available for sub-critical two-phase flow applications.

Potential

displays the boundary conditions for the electric potential field. This tab is available only if you have enabled either the **Electric Potential model** or the **Electrochemical** reaction model in the Species Dialog Box (p. 4929).

Potential Boundary Condition

is a drop-down list of available potential boundary condition types for the potential generated by electron current: **Specified Flux** and **Specified Value**. For the **Specified Flux** boundary condition, you will need to specify **Current Density** at the wall. For the **Specified Value** boundary condition, you will need to specify **Potential** at the wall.

Electrolyte Potential Boundary Condition

is a drop-down list of available boundary condition types for the potential generated by ionic current. This item is similar to the **Potential Boundary Condition** drop-down list described above and is available only for the Electrolysis and H2 Pump model.

UDS

contains the UDS parameters.

User-Defined Scalar Boundary Condition

appears only if user-defined scalars are specified.

User Scalar n

specifies whether the scalar is a specified flux or a specified value.

User-Defined Scalar Boundary Value

appears only if user-defined scalars are specified.

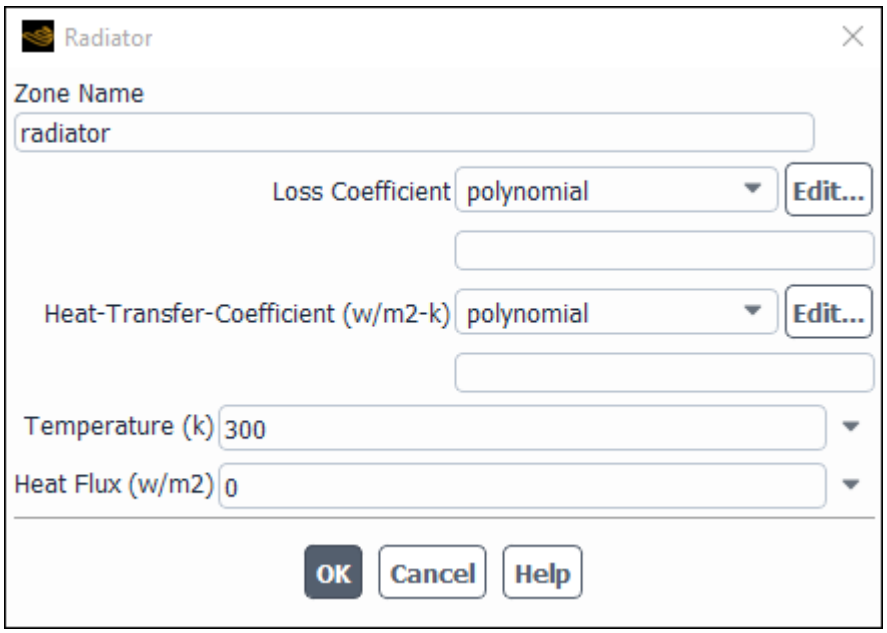
User Scalar n

specifies the value of the scalar.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

51.7.19. Radiator Dialog Box

The **Radiator** dialog box sets the boundary conditions for a radiator model zone. It is opened from the Boundary Conditions Task Page (p. 4991). See User Inputs for Radiators (p. 1513) for details about the items below.



Controls

Zone Name

sets the name of the zone.

Loss Coefficient

specifies the loss coefficient as a constant value or as a polynomial, piecewise-linear, or piecewise-polynomial function of velocity. See Defining the Pressure Loss Coefficient Function (p. 1514) for details.

Heat-Transfer-Coefficient

specifies the heat-transfer coefficient as a constant value or as a polynomial, piecewise-linear, or piecewise-polynomial function of velocity. See Defining the Heat Flux Parameters (p. 1517) for details.

Temperature

sets the temperature used to compute heat flux from the radiator using the **Heat-Transfer-Coefficient**. If **Temperature** is absolute zero, the **Heat Flux** condition is used instead.

Heat Flux

sets the heat flux at the radiator surface (used only when **Temperature** is absolute zero).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Discrete Phase BC Type

sets the way that the discrete phase behaves with respect to the boundary. This item appears when one or more injections have been defined.

interior

allows the particles to pass through the boundary.

reflect

rebounds the particle off the boundary with a change in its momentum as defined by the coefficient of restitution. (See Particle Reflection at Wall in the *Fluent Theory Guide*.)

trap

terminates the trajectory calculations and records the fate of the particle as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See Figure 24.40: "Trap" Boundary Condition for the Discrete Phase (p. 2783).

escape

reports the particle as having "escaped" when it encounters the boundary. Trajectory calculations are terminated. See Figure 24.41: "Escape" Boundary Condition for the Discrete Phase (p. 2783).

reinject

reintroduces the particle into the domain when it reaches a certain domain boundary (for example, outlet). This item appears when one or more injections have been defined and **Unsteady Particle Tracking** is enabled in the **Discrete Phase Model** dialog box. See The **reinject** Boundary Condition (p. 2784) for details.

wall-jet

indicates that the direction and velocity of the droplet particles are given by the resulting momentum flux, which is a function of the impingement angle. See Figure 12.6: "Wall Jet" Boundary Condition for the Discrete Phase in the Theory Guide.

user-defined

specifies a user-defined function to define the discrete phase boundary condition type.

Discrete Phase BC Function

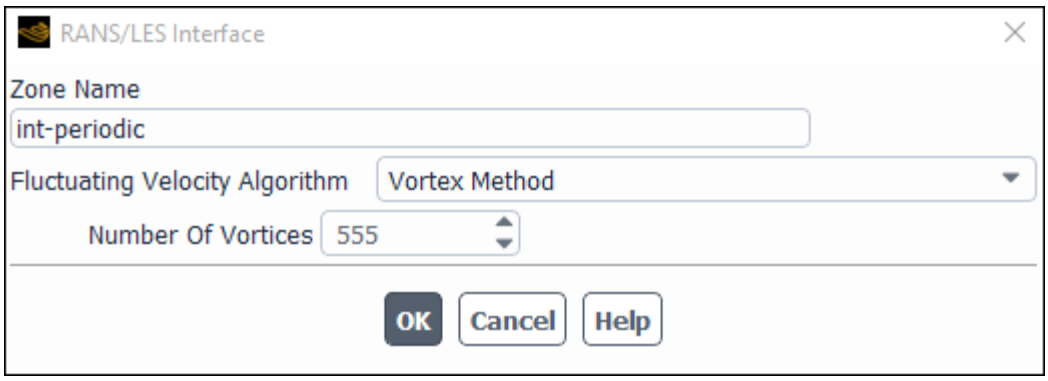
sets the user-defined function from the drop-down list.

51.7.20. RANS/LES Interface Dialog Box

The **RANS/LES Interface** dialog box can be used to create artificial resolved turbulence (fluctuation/perturbations) at the interface where the flow proceeds from the RANS zone into the LES zone of the computational domain for Embedded LES turbulent flows. It is opened from the Boundary Conditions

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Task Page (p. 4991). See Setting Up the Embedded Large Eddy Simulation (ELES) Model (p. 2124) for more information about RANS/LES interfaces.



Controls

Zone Name

sets the name of the RANS/LES interface.

Fluctuating Velocity Algorithm

displays the methods for generating fluctuating velocity components at the RANS/LES interface. Available options include:

- No Perturbations
- Spectral Synthesizer
- Vortex Method

Number of Vortices

displays the amount of vortices that the selected fluctuating velocity method distributes randomly over the face zone and uses to generate turbulent fluctuations (available for the Vortex Method only).

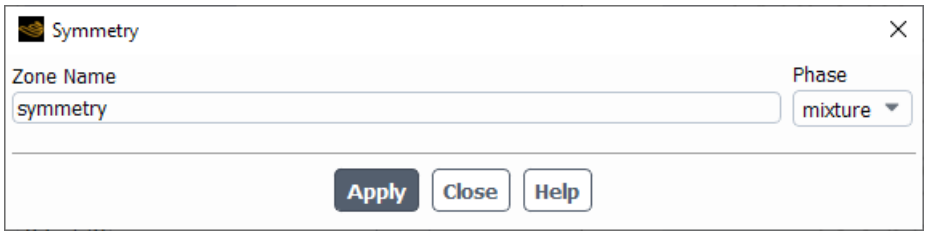
Important:

The RANS/LES 'interface' can either be an interior zone (that has been assigned to be a **rans-les-interface** zone), or it can be a non-conformal interface. If it is a non-conformal interface, then you need to identify the name of the non-conformal interface's "interior" zone (for example, using the **List** button in the Mesh Interfaces Dialog Box (p. 5525)), and then go to that zone in the **Boundary Conditions** task page.

51.7.21. Symmetry Dialog Box

The **Symmetry** dialog box can be used to modify the name of a symmetry zone; there are no conditions to be set. It is opened from the Boundary Conditions Task Page (p. 4991). See Symmetry Boundary Conditions (p. 1499) for information about symmetry boundaries.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Zone Name

sets the name of the zone.

Phase

displays the name of the phase. This item is available only for multiphase flows.

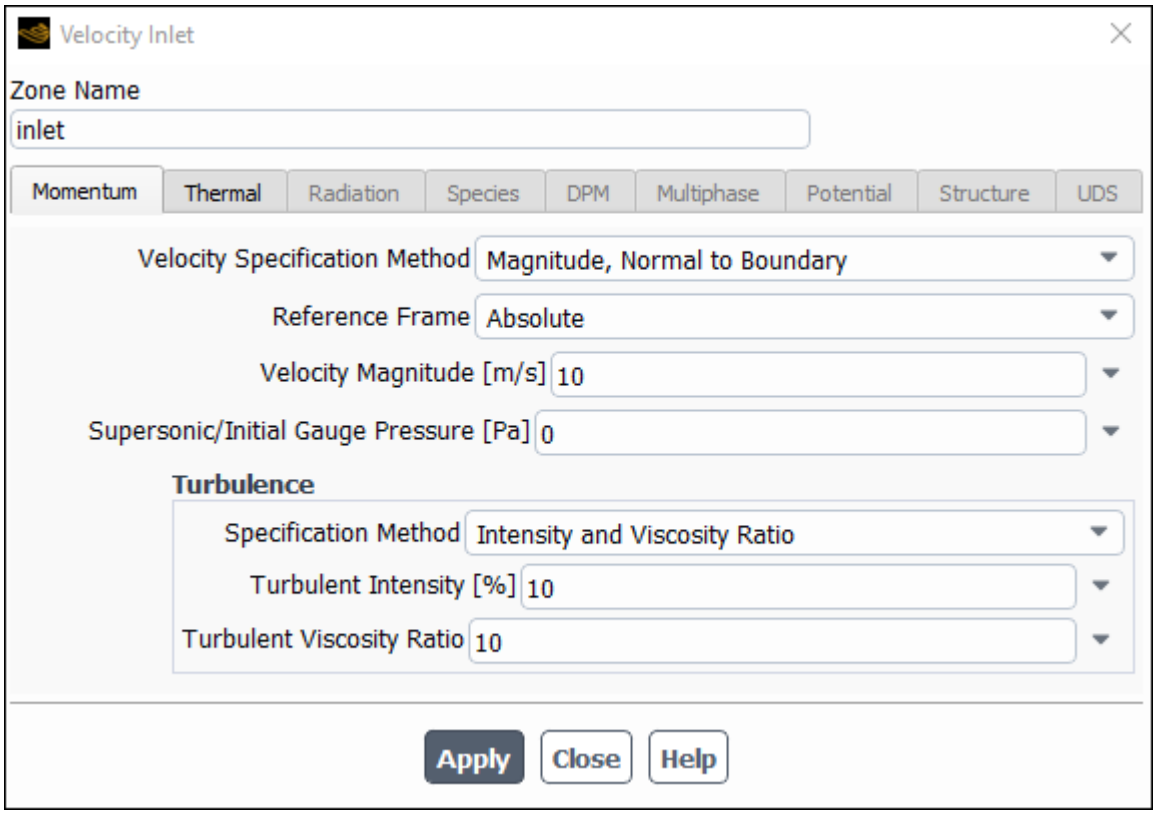
51.7.22. Velocity Inlet Dialog Box

The **Velocity Inlet** dialog box sets the boundary conditions for a velocity inlet zone. It is opened from the Boundary Conditions Task Page (p. 4991). See Inputs at Velocity Inlet Boundaries (p. 1411) for details about defining the items below.

Important:

This feature offers reduced functionality when running Fluent under the **Pro** capability level.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Zone Name

sets the name of the zone.

Phase

displays the name of the phase. This item appears if the VOF, mixture, or Eulerian multiphase model is being used.

Open Channel Wave BC

allows you to set specific parameters for a particular boundary for open channel wave boundaries. This is available when the volume of fluid multiphase model is selected.

Momentum

contains the momentum parameters.

Velocity Specification Method

sets the method used to define the inflow velocity.

Flow Direction Specification Method

sets the method used to define the direction of flow of the wave. This is available when you enable the **Open Channel Wave BC** option.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Magnitude and Direction

allows specification in terms of a **Velocity Magnitude** and **Flow-Direction**.

Components

allows specification in terms of the Cartesian, cylindrical, or local cylindrical velocity components.

Magnitude, Normal to Boundary

allows specification of a **Velocity Magnitude** normal to the boundary.

Reference Frame

specifies relative or absolute velocity inputs. You can choose to enter **Absolute** velocities or velocities **Relative to Adjacent Cell Zone**. If you are not using moving reference frames, both options are equivalent, so you need not choose.

Uniform Flow Velocity Magnitude

is the flow velocity, specified as a constant or a parameter.

Coordinate System

specifies whether **Cartesian**, **Cylindrical**, or **Local Cylindrical** velocities will be defined. This item will appear only for 3D cases in which you have selected **Magnitude and Direction** or **Components** as the **Velocity Specification Method**.

X-, Y-, Z-Velocity

set the components of the velocity vector at the inflow boundary. These items will appear for 2D non-axisymmetric models, or for 3D models if you select the **Components** option as the **Velocity Specification Method** and **Cartesian** as the **Coordinate System**.

Radial-, Tangential-, Axial-Velocity

set the components of the velocity vector at the inflow boundary. These items will appear for 3D models if you select the **Components** option as the **Velocity Specification Method** and **Cylindrical** or **Local Cylindrical** as the **Coordinate System**.

Axial-, Radial-, Swirl-Velocity

set the components of the velocity vector at the inflow boundary. These items will appear for 2D axisymmetric models.

Important:

Swirl-Velocity will appear only for 2D axisymmetric swirl models.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Angular Velocity

specifies the angular velocity Ω for a 3D flow. This item will appear for a 3D model if you select the **Components** option as the **Velocity Specification Method** and **Cylindrical** or **Local Cylindrical** as the **Coordinate System**.

Swirl Angular Velocity

specifies the swirl angular velocity Ω for an axisymmetric swirling flow. This item will appear for an axisymmetric swirl model if you choose **Components** as the **Velocity Specification Method**.

Velocity Magnitude

sets the magnitude of the velocity vector at the inflow boundary. This item will appear if you select the **Magnitude and Direction** or **Magnitude, Normal to Boundary** option as the **Velocity Specification Method**.

X-, Y-, Z-Component of Flow Direction

set the direction of the velocity vector at the inflow boundary. These items will appear for 2D non-axisymmetric models if you select the **Magnitude and Direction** option as the **Velocity Specification Method**, or for 3D models if you select the **Magnitude and Direction** option as the **Velocity Specification Method** and **Cartesian** as the **Coordinate System**.

Radial-, Tangential-, Axial-Component of Flow Direction

set the direction of the velocity vector at the inlet boundary. These items will appear for 3D models if you select the **Magnitude and Direction** option as the **Velocity Specification Method** and **Cylindrical** or **Local Cylindrical** as the **Coordinate System**, or for 2D axisymmetric models.

Important:

Tangential-Velocity will appear only for 2D axisymmetric swirl models.

Axis Origin

sets the **X**, **Y**, and **Z** coordinates of the origin of the local cylindrical coordinate system.

Axis Direction

sets the **X**, **Y**, and **Z** components of the direction of the local cylindrical coordinate system.

Outflow Gauge Pressure

specifies the pressure to be used as the pressure outlet condition if flow exits the domain at any face on the velocity inlet boundary. (Note that this effect is similar to that of the "velocity far-field" boundary that was available in RAMPANT 3.)

This item appears only for the density-based solvers.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fluctuating Velocity

contains parameters for Inlet Boundary Conditions for Scale Resolving Simulations (only available for transient cases where scale-resolving turbulence models such as SAS, DES, and LES are enabled).

Fluctuating Velocity Algorithm

Specifies which method will be used for generating fluctuations at the boundary for Scale Resolving Simulations. The following methods are available:

No Perturbations

No fluctuations added.

Vortex Method

For details, see Vortex Method in the *Fluent Theory Guide*.

Spectral Synthesizer

For details, see Spectral Synthesizer in the *Fluent Theory Guide*.

Synthetic Turbulence Generator

For details, see Synthetic Turbulence Generator in the *Fluent Theory Guide*.

Number of Vortices

Specifies number of vortices to be generated with the **Vortex Method**.

Satisfy Mass Conservation?

Enables mass conservation for the **Vortex Method** at the boundary.

Scale Search Limiter

Specifies which method will be used to limit the area from where global scales for the **Synthetic Turbulence Generator** are computed. The following methods are available:

None

Scales are computed using the entire inlet zone.

Turbulent Intensity Threshold

Scales are computed using only part of the inlet zone where Turbulent Intensity is higher than specified threshold value.

Turbulent Viscosity Ratio Threshold

Scales are computed using only part of the inlet zone where Turbulent Viscosity Ratio is higher than specified threshold value.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Wall Distance Threshold

Scales are computed using only part of the inlet zone where Wall Distance is smaller than specified threshold value.

Turbulent Intensity, Turbulent Viscosity Ratio, or Wall Distance

Specifies threshold value for the corresponding **Scale Search Limiter** (only available for the **Synthetic Turbulence Generator** with corresponding **Scale Search Limiter** chosen).

Volumetric Forcing?

Enables generation of fluctuations in the volume adjacent to the inlet boundary instead of introduction of fluctuations directly at the boundary (only available for the **Synthetic Turbulence Generator**).

Forcing Zone Thickness

Specifies which method will be used to set up the streamwise size of the volume where fluctuations are to be generated (only available for the **Synthetic Turbulence Generator** with **Volumetric Forcing?** enabled). The following methods are available:

Automatic

Length is computed based on the turbulent length scale computed from the boundary.

Specified Value

Length is specified by the user.

Thickness

Specifies the streamwise size of the volume where fluctuations are to be generated (only available for the **Synthetic Turbulence Generator** with **Volumetric Forcing?** enabled if **Specified Value** option is chosen for **Forcing Zone Thickness**).

Turbulence

contains the turbulence parameters.

Specification Method

specifies which method will be used to define the turbulence parameters. You can choose **K and Epsilon** (k- ε models and RSM only), **K and Omega** (k- ω models only), **Intensity and Length Scale**, **Intensity and Viscosity Ratio**, **Intensity and Hydraulic Diameter**, **Modified Turbulent Viscosity** (Spalart-Allmaras model only), or **Turbulent Viscosity Ratio** (Spalart-Allmaras model only). See Determining Turbulence Parameters (p. 1395) for information about the inputs for each of these methods. (This item will appear only for turbulent flow calculations.)

Turbulent Kinetic Energy, Turbulent Dissipation Rate

set values for the turbulence kinetic energy k and its dissipation rate ε. These items will appear if you choose **K and Epsilon** as the **Specification Method**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Turbulent Kinetic Energy, Specific Dissipation Rate

set values for the turbulence kinetic energy k and its specific dissipation rate ω. These items will appear if you choose **K and Omega** as the **Specification Method**.

Turbulent Intensity, Turbulent Length Scale

set values for turbulence intensity I and turbulence length scale ℓ. These items will appear if you choose **Intensity and Length Scale** as the **Specification Method**.

Turbulent Intensity, Turbulent Viscosity Ratio

set values for turbulence intensity I and turbulent viscosity ratio μ_t / μ. These items will appear if you choose **Intensity and Viscosity Ratio** as the **Specification Method**.

Turbulent Intensity, Hydraulic Diameter

set values for turbulence intensity I and hydraulic diameter L. These items will appear if you choose **Intensity and Hydraulic Diameter** as the **Specification Method**.

Modified Turbulent Viscosity

sets the value of the modified turbulent viscosity $\tilde{\nu}$. This item will appear if you choose **Modified Turbulent Viscosity** as the **Specification Method**.

Turbulent Viscosity Ratio

sets the value of the turbulent viscosity ratio μ_t / μ. This item will appear if you choose **Turbulent Viscosity Ratio** as the **Specification Method**.

Turbulent Intensity

sets the value of the turbulence intensity I for the LES model.

Reynolds-Stress Specification Method

specifies which method will be used to determine the Reynolds stress boundary conditions when the Reynolds stress turbulence model is used. You can choose either **K or Turbulent Intensity** or **Reynolds-Stress Components**. If you choose the former, Ansys Fluent will compute the Reynolds stresses for you. If you choose the latter, you will explicitly specify the Reynolds stresses yourself. See Reynolds Stress Model (p. 2144) for details. (This item will appear only for RSM turbulent flow calculations.)

UU, VV, WW, UV, VW, UW Reynolds Stresses

specify the Reynolds stress components when **Reynolds-Stress Components** is chosen as the **Reynolds-Stress Specification Method**.

Acoustic Wave Model

contains settings for treatment of acoustic pressure waves at the boundary.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Off

disables special treatment of acoustic pressure waves at the boundary.

Non Reflecting

enables the general non-reflecting boundary condition treatment described in General Non-Reflecting Boundary Conditions (p. 1538).

Impedance

enables the impedance boundary condition treatment described in Impedance Boundary Conditions (p. 1545).

Transparent Flow Forcing

enables the transparent flow forcing boundary condition described in Transparent Flow Forcing Boundary Conditions (p. 1554).

Impedance Parameters

contains the parameters for the impedance boundary condition treatment. For details refer to Using the Impedance Boundary Condition (p. 1546).

Transparent Flow Forcing Parameters

contains the parameters for the transparent flow forcing boundary condition treatment. For details refer to Using the Transparent Flow Forcing Boundary Condition (p. 1554).

Thermal

contains the thermal parameters.

Temperature

specifies the static temperature of the flow.

Radiation

contains the boundary conditions for the radiation model at the velocity inlet.

External Black Body Temperature Method, Internal Emissivity

set the radiation boundary conditions when you are using the P-1, DTRM, DO, S2S, or MC models for radiation heat transfer. See Defining Boundary Conditions for Radiation (p. 2217) for details.

Participates in Solar Ray Tracing

specifies whether or not the velocity inlet participates in solar ray tracing.

Solar Transmissivity Factor

specifies a multiplier (ranging from 0 to 1) that is applied to the solar irradiation entering the domain through the velocity inlet.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Participates in View Factor Calculation

specifies whether or not the velocity inlet participates in the view factor calculation as part of the S2S radiation model. This parameter is available only if you select the **Surface to Surface** radiation model.

Species

contains the species parameters.

Specify Species in Mole Fractions

allows you to specify the species in mole fractions rather than mass fractions.

Species Mass Fractions

contains inputs for the mass fractions of defined species. See Defining Cell Zone and Boundary Conditions for Species (p. 2394) for details about these inputs. These items will appear only if you are modeling non-reacting multi-species flow or you are using the finite-rate reaction formulation.

Mean Mixture Fraction, Mixture Fraction Variance

set inlet values for the PDF mixture fraction and its variance. These items will appear only if you are using the non-premixed or partially premixed combustion model.

Secondary Mean Mixture Fraction, Secondary Mixture Fraction Variance

set inlet values for the secondary mixture fraction and its variance. (These items will appear only if you are using the non-premixed or partially premixed combustion model with two mixture fractions.)

Progress Variable

sets the value of the progress variable for premixed turbulent combustion. See Setting Boundary Conditions for the Progress Variable (p. 2541) for details.

This item will appear only if the premixed or partially premixed combustion model is used.

Scalar Mass Fractions

(partially-premixed combustion FGM model only) allows you to specify the scalar species mass fractions for the transported scalars that you selected in the **Select Transported Scalars** dialog box.

DPM

contains the discrete phase parameters.

Discrete Phase BC Type

sets the way that the discrete phase behaves with respect to the boundary. This item appears when one or more injections have been defined.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

reflect

rebounds the particle off the boundary with a change in its momentum as defined by the coefficient of restitution. (See Particle Reflection at Wall in the *Fluent Theory Guide*.)

trap

terminates the trajectory calculations and records the fate of the particle as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See Figure 24.40: "Trap" Boundary Condition for the Discrete Phase (p. 2783).

escape

reports the particle as having "escaped" when it encounters the boundary. Trajectory calculations are terminated. See Figure 24.41: "Escape" Boundary Condition for the Discrete Phase (p. 2783).

reinject

reintroduces the particle into the domain when it reaches a certain domain boundary (for example, outlet). This item appears when one or more injections have been defined and **Unsteady Particle Tracking** is enabled in the **Discrete Phase Model** dialog box. See The **reinject** Boundary Condition (p. 2784) for details.

wall-jet

indicates that the direction and velocity of the droplet particles are given by the resulting momentum flux, which is a function of the impingement angle. See Figure 12.6: "Wall Jet" Boundary Condition for the Discrete Phase in the Theory Guide.

user-defined

specifies a user-defined function to define the discrete phase boundary condition type.

Discrete Phase BC Function

sets the user-defined function from the drop-down list.

Multiphase

contains the multiphase parameters.

Volume Fraction

specifies the volume fraction of the secondary phase selected in the Boundary Conditions Task Page (p. 4991). This section of the dialog box will appear when one of the multiphase models is being used. See Defining Multiphase Cell Zone and Boundary Conditions (p. 2957) for details.

Secondary Phase for Inlet

specifies the secondary phase to which the parameters are applied. In case of a three-phase flow, select the corresponding secondary phase from this list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Wave BC Options

allows you to choose between **Shallow Waves**, **Shallow/Intermediate Waves** or **Short Gravity Waves**. Information about the two types of waves is available in Open Channel Wave Boundary Conditions in the Theory Guide.

Free Surface Level

is determined by using the absolute value of height from the free surface to the origin in the direction of gravity and applying the correct sign based on whether the free surface level is above (positive) or below (negative) the origin.

Bottom Level

is valid for shallow and intermediate waves. The bottom level is used for calculating the liquid height.

Reference Wave Direction

is the direction of wave propagation for a zero wave heading angle.

Wave Modeling Options

determines whether to use wave theory or a wave spectrum to model waves in the shallow/intermediate and short gravity wave regimes.

Number of Waves

specifies the number of waves you are defining when modeling waves using wave theory.

Wave Theory

allows you to choose from **First Order Airy** (the default), **Second Order Stokes**, **Third Order Stokes**, **Fourth Order Stokes**, and **Fifth Order Stokes** for shallow/intermediate waves and short gravity waves. For shallow waves, you can choose between **Fifth Order Solitary** and **Fifth Order Cnoidal**. Information about the types of wave theory is available in Open Channel Wave Boundary Conditions in the Theory Guide.

Wave Height

is the height difference between a wave crest to the neighboring trough.

Since a solitary wave does not have troughs, the wave height is the distance between a wave crest to mean free surface level.

Wave Length

is the wave length of the shallow wave or short gravity wave.

Inlet Offset Distance

is the translational distance from the reference point origin in the reference wave propagation direction. This option is used to generate a wave from a location other than the reference frame origin.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Phase Difference

is the phase difference between one wave and another.

Wave Heading Angle

is the angle between the direction of the wavefront and the reference wave direction, in the plane of the flow surface. In 2D, there are only two possibilities, zero degrees when the wave is in the reference wave direction, and 180 degrees when the wave is in the opposite direction.

Frequency Spectrum Method

specifies the spectrum to use. You can select **Pierson-Moskowitz** (appropriate for fully-developed seas), **Jonswap** (appropriate for fetch-limited seas), or **TMA** (appropriate for fetch-limited, finite-depth seas).

Peak Shape Parameter

controls the shape and amplitude of the frequency peak in the **Jonswap** and **TMA** formulations. This corresponds to γ in Equation 14.101 in the *Fluent Theory Guide*.

Significant Wave Height

is the mean wave height of the largest 1/3 of waves. This corresponds to H_s in Equation 14.100 in the *Fluent Theory Guide*.

Peak Wave Frequency

is the wave frequency corresponding to the highest wave energy. This corresponds to ω_p in Equation 14.100 and Equation 14.101 in the *Fluent Theory Guide*. ω_p can be expressed in terms of the peak wave period, T_p, as $\omega_p = {}^{2\pi}/_{T_p}$.

Minimum/Maximum Wave Frequency

specify the frequency range for the spectrum. Note that the **Peak Wave Frequency** must fall in between **Minimum Wave Frequency** and **Maximum Wave Frequency**.

Number of Frequency Components

specifies the number of components into which the frequency spectrum is divided.

Direction Spreading Method

specifies the method to use for specifying the directional spreading characteristics. You can choose **Unidirectional** (for long-crested waves), **Frequency Independent Cosine Function** (for short-crested waves where the directional component does not depend on frequency), and **Frequency Dependant Hyperbolic Function** (for short-crested waves where the directional component depends on frequency).

Frequency Independent Cosine Exponent

specifies the exponent in the Frequency Independent Cosine Function formulation. This corresponds to s in Equation 14.105 in the *Fluent Theory Guide*.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mean Wave Heading Angle

specifies the principal wave heading direction.

Angular Spread

specifies the deviation from the mean wave direction for calculating the angular range.

Number of Angular Components

specifies the number of components into which the angular range is divided.

Potential

displays the boundary conditions for the electric potential field. This tab is available only if you have enabled either the **Electric Potential model** or the **Electrochemical** reaction model in the Species Dialog Box (p. 4929).

Potential Boundary Condition

is a drop-down list of available potential boundary condition types for the potential generated by electron current: **Specified Flux** and **Specified Value**. For the **Specified Flux** boundary condition, you will need to specify **Current Density** at the wall. For the **Specified Value** boundary condition, you will need to specify **Potential** at the wall.

Electrolyte Potential Boundary Condition

is a drop-down list of available boundary condition types for the potential generated by ionic current. This item is similar to the **Potential Boundary Condition** drop-down list described above and is available only for the Electrolysis and H2 Pump model.

UDS

contains the UDS parameters.

User-Defined Scalar Boundary Condition

appears only if user-defined scalars are specified.

User Scalar n

specifies whether the scalar is a specified flux or a specified value.

User-Defined Scalar Boundary Value

appears only if user-defined scalars are specified.

User Scalar n

specifies the value of the scalar.

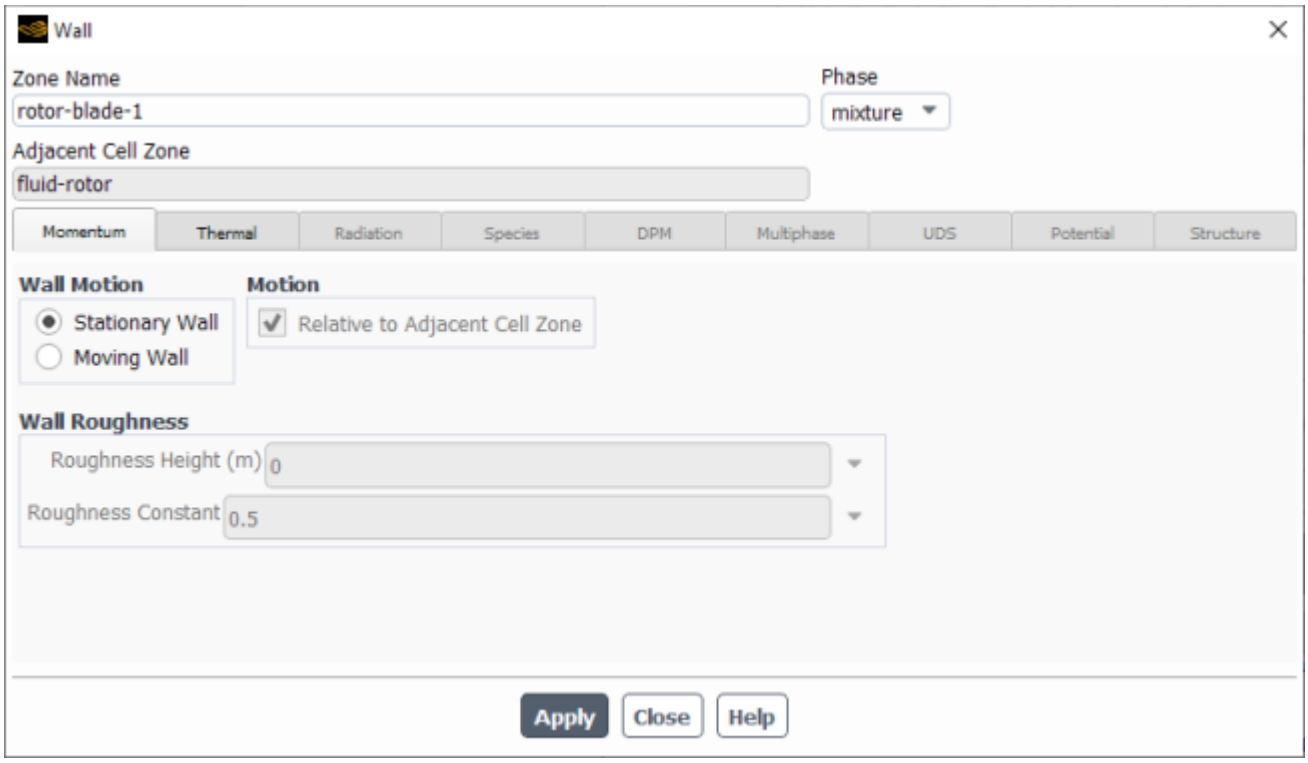
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

51.7.23. Wall Dialog Box

The **Wall** dialog box sets the boundary conditions for a wall zone. It is opened from the Boundary Conditions Task Page (p. 4991). See Inputs at Wall Boundaries (p. 1458) for details about defining the items below.

Important:

This feature offers reduced functionality when running Fluent under the **Pro** capability level.



Controls

Zone Name

sets the name of the zone.

Phase

displays the name of the phase. This item appears if the VOF, mixture, or Eulerian multiphase model is being used.

Adjacent Cell Zone

shows the name of the cell zone adjacent to the wall. (This is for informational use only; you cannot edit this field.)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Momentum

displays the momentum boundary conditions.

Wall Motion

contains options for specifying whether or not the wall is moving.

Stationary Wall

specifies that the wall is not moving relative to the adjacent cell zone.

Moving Wall

enables specification of the tangential wall motion. Tangential wall motion is applicable only to viscous flows. Since the inviscid slip condition decouples the tangential wall velocity from the governing equations, tangential wall motion has no effect on inviscid flow.

Motion

contains inputs related to wall motion. See Velocity Conditions for Moving Walls (p. 1462) for details.

Relative to Adjacent Cell Zone

enables the specification of a wall velocity relative to the velocity of the adjacent cell zone.(If the adjacent cell zone is not moving, this is equivalent to **Absolute**.)

Absolute

enables the specification of an absolute wall velocity,

Translational

enables the specification of a translational wall velocity.

Rotational

enables the specification of a rotational wall velocity.

Components

enables the specification of wall velocity components.

Speed

sets the translational or rotational speed of the wall (depending on whether you selected **Translational** or **Rotational**).

Direction

sets the direction vector of the translational velocity. (This item will appear if you have chosen the **Translational** option.)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Rotation-Axis Origin

sets the coordinates of the origin of the axis of rotation, thereby determining the location of the axis. (This item will appear if you have chosen the **Rotational** option for a non-axisymmetric case.)

Rotation-Axis Direction

sets the direction vector for the axis of rotation. (This item will appear if you have chosen the **Rotational** option for a non-axisymmetric case.)

Velocity Components

sets the **X**, **Y**, and **Z-Velocity** components of the wall motion. (This item will appear if you have chosen the **Components** option.)

Shear Condition

contains options for specifying the shear conditions at the wall.

No Slip

specifies a no-slip condition at the wall. No further inputs are required.

Specified Shear

enables specification of zero or nonzero shear. See Specified Shear (p. 1465) for details. This option is not available for moving walls.

Marangoni Stress

enables the specification of shear stress caused by the variation of surface tension due to temperature. This option is not available for moving walls.

Shear Stress

contains inputs related to wall shear. These items will appear when **Specified Shear** is selected as the **Shear Condition**. See Specified Shear (p. 1465) for details.

X-Component, Y-Component, Z-Component, Swirl Component

specify the x, y, and z or swirl components of shear for a slip wall. **Swirl Component** is available only for axisymmetric swirl cases.

Specularity Coefficient

is used in multiphase granular flow. You can specify the specularity coefficient such that when the value is zero, this condition is equivalent to zero shear at the wall, but when the value is near unity, there is a significant amount of lateral momentum transfer.

Specularity Coefficient

allows you to enter a value between zero and one, which controls the amount of lateral momentum transfer.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Marangoni Stress

contains inputs related to Marangoni stress. This item will appear when **Marangoni Stress** is selected as the **Shear Condition**. See Marangoni Stress (p. 1466) for details.

Surface Tension Gradient

specifies the surface tension gradient with respect to temperature ($d\sigma / dT$ in Equation 7.120 (p. 1466)).

Wall Roughness

contains various models for defining wall roughness size/height in turbulent calculations.

Standard

uses the standard wall roughness methods for most turbulence models. See Wall Roughness Effects in Turbulent Wall-Bounded Flows (p. 1458) for details.

High Roughness (Icing)

contains various models for use only with the Spalart-Allmaras and SST k-ω turbulence models. Specifically applicable to icing simulations, choose from **Specified Roughness**, **NASA Correlation**, **Shin-et-al**, or **ICE3D Roughness File** methods to specify the **Sand-Grain Roughness**. See Wall Roughness Effects in Turbulent Wall-Bounded Flows (p. 1458) for details.

Wall Adhesion

contains inputs related to wall adhesion. This section of the dialog box will appear if you are using the VOF model and have enabled **Wall Adhesion** in the e Multiphase Model Dialog Box (p. 4700) (**Forces** tab).

Contact Angles

specifies the contact angle at the wall for each pair of phases (θ_w in Figure 26.21: Measuring the Contact Angle (p. 2960) in the Theory Guide). See Steps for Setting Boundary Conditions (p. 2957) for details.

Thermal

contains the thermal parameters. This tab is available only when the energy equation is turned on.

Thermal Conditions

contains radio buttons for selecting the thermal boundary condition type. See Thermal Boundary Conditions at Walls (p. 1467) for details about these inputs:

Heat Flux

selects a specified heat flux condition.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Temperature

selects a specified wall temperature condition.

Convection

selects a convective heat transfer boundary condition model.

Radiation

selects an external radiation boundary condition.

Mixed

selects a combined convection/external radiation boundary condition.

Coupled

selects a coupled heat transfer condition. It is applicable only to walls that form the interface between two regions (such as the fluid/solid interface for a conjugate heat transfer problem).

via System Coupling

selects a heat transfer condition where a boundary can receive thermal data (either wall temperature, heat flow, or heat transfer coefficient) from the System Coupling service. It is applicable when Ansys Fluent is coupled with another system in Workbench using System Coupling (see Heat Transfer Boundary Conditions Through System Coupling (p. 1475) for details).

via Mapped Interface

specifies that the thermal data is interpolated across a mapped interface. It is available only for the interface wall boundary zones created as part of a mapped interface (see Heat Transfer Boundary Conditions Across a Mapped Interface (p. 1476) for details).

Once a condition type has been selected, the appropriate conditions can be specified.

Heat Flux

sets the wall heat flux to be used for the **Heat Flux** condition. A specification of zero **Heat Flux** is simply the adiabatic condition (no heat transfer). A positive value of heat flux implies that heat enters *into* the domain.

Temperature

sets the wall temperature to be used for the **Temperature** condition.

Heat Transfer Coefficient

sets the convective heat transfer coefficient to be used for the **Convection** condition (h_{eff} in Equation 7.127 (p. 1482)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Free Stream Temperature

sets the reference or free stream temperature to be used for the **Convection** condition (T_{ext} in Equation 7.127 (p. 1482)).

External Emissivity

sets the emissivity of the external wall to be used for the **Radiation** condition (ε_{ext} in Equation 7.128 (p. 1482)).

External Radiation Temperature

sets the temperature of the external radiation source/sink to be used for the **Radiation** condition (T_{∞} in Equation 7.128 (p. 1482)).

Wall Thickness

sets the thickness of a thin wall for the calculation of thermal resistance. (See Thin-Wall Thermal Resistance Parameters (p. 1470) for details.)

Heat Generation Rate

sets the rate of heat generation in a wall without shell conduction.

Contact Resistance

sets the contact resistance (R_c in Equation 15.23 in the Theory Guide) at the wall. See Modeling Solidification and Melting (p. 3219) for details. This item appears only when the solidification/melting model is used.

Material Name

sets the material type for a thin wall. The conductivity of the material is used for the calculation of thin-wall thermal resistance. (See Thin-Wall Thermal Resistance Parameters (p. 1470) for details.) **Material** is used only when **Wall Thickness** is nonzero. Materials are defined with the Materials Task Page (p. 4883).

Boundary Advection

enables the advection of heat into or out of the domain as a result of motion that is normal to the boundary. This option is only available for a boundary wall that is adjacent to a solid cell zone that undergoes solid motion. For details, see Boundary Advection for Solid Motion (p. 1474).

Shell Conduction

enables shell conduction for the wall. See Shell Conduction (p. 1475) for details.

Edit...

is only available when **Shell Conduction** is enabled, and opens the Conduction Layers Dialog Box (p. 5409) in order to allow you to define the shell conduction settings for the wall.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Radiation

displays the boundary conditions for the S2S model, the DO model, the MC model, and the non-gray P-1 model at the wall. This tab is only available if you are using the surface to surface model, the discrete ordinates model, the Monte Carlo model, or the non-gray P-1 model. See Forming Surface Clusters (p. 2201), Wall Boundary Conditions for the DO Model (p. 2220), and Wall Boundary Conditions for the DTRM, P-1, S2S, and Rosseland Models (p. 2220) for details.

BC Type

contains a drop-down list of available radiation boundary condition types. The available options are **opaque** and **semi-transparent**. This item will appear only if you are using the discrete ordinates or Monte Carlo model.

Boundary Source

enables the wall to be a radiation source. This is only available for external semi-transparent walls with the MC or DO models and for internal or external opaque walls with the MC model.

Polar Distribution Function

enables a polar (rotationally symmetric about a given direction) boundary source which can be used if the irradiation intensity varies with direction such as in the case of LED light sources. The polar distribution function is defined by an **Expression** or by a table of data pairs consisting of an **Angle** and it's corresponding **Relative Intensity**. You have the option to read and write the polar data in the form of `.csv` files or plotting the data in the graphics window. Only available with the MC model.

Reference Direction

Specify the (X,Y,Z) vector that defines the center of the polar distribution function. Only available with the MC model.

Internal Emissivity

sets the internal emissivity of the wall. This item will appear only if you are using the P-1, DO, MC, DTRM, or S2S models for radiation heat transfer.

Diffuse Fraction

specifies the fraction of the irradiation that is to be treated as diffuse. By default, the **Diffuse Fraction** is set to 1, indicating that all of the irradiation is diffuse. If the non-gray DO or MC model is being used, the **Diffuse Fraction** can be specified for each band. This item will appear only if you are using the Monte Carlo or discrete ordinates model.

Beam Width

specifies the beam width for an external semi-transparent wall in terms of the **Theta** and **Phi** extents. This item will appear only if you are using the discrete ordinates radiation model and you have selected **semi-transparent** as the **BC Type**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Beam Direction

specifies the beam direction as an **X,Y,Z** vector. You can specify the **Beam Direction** as a constant, a profile, or a user-defined function. This item will appear only if you are using the Monte Carlo model or the discrete ordinates radiation model and you have selected **semi-transparent** as the **BC Type**.

Direct Irradiation

specifies the value of the irradiation flux. If the non-gray DO or MC models are being used, a constant **Direct Irradiation** can be specified for each band.

This item will appear only if you are using the MC model or the **semi-transparent BC Type** with the DO model.

Apply Direct Irradiation Parallel to the Beam

By default, **Apply Direct Irradiation Parallel to the Beam** is selected, which means Fluent assumes that the value you specify for **Direct Irradiation** is the irradiation flux parallel to the **Beam Direction**. When deselected, Ansys Fluent assumes that the value you specify for **Direct Irradiation** is the irradiation flux normal to the boundary.

Apply Irradiation About the Reference Direction

By default, **Apply Irradiation About the Reference Direction** is selected, which means Fluent assumes that the value you specify for **Direct Irradiation** is the irradiation flux parallel to the **Reference Direction**. When deselected, Ansys Fluent assumes that the value you specify for **Direct Irradiation** is the irradiation flux normal to the boundary. See Figure 5.12: DO Irradiation on External Semi-Transparent Wall in Semi-Transparent Exterior Walls in the Theory Guide for details.

Diffuse Irradiation

specifies the value of the irradiation flux. If the non-gray DO or MC model is being used, a constant **Diffuse Irradiation** can be specified for each band.

This item will appear only if you are using the DO or MC radiation model and you have selected **semi-transparent** as the **BC Type**.

Solar Boundary Conditions

contains the settings for solar ray tracing. This group box is available only if you select **Solar Ray Tracing** from the **Model** list in the **Solar Load** group box of the **Radiation Model** dialog box. For semitransparent coupled walls, the solar ray tracing settings are defined on the original wall and not on the shadow wall. See Solar Ray Tracing (p. 2261) for details.

Participates in Solar Ray Tracing

specifies whether or not the wall participates in solar ray tracing.

Absorptivity

contains the settings that define the absorptivity of wall.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Direct Visible

specifies a multiplier (ranging from 0 to 1) that is applied to the visible portion of the direct solar radiation spectrum to account for the absorption of the wall. The value should be defined for normal incident rays.

Direct IR

specifies a multiplier (ranging from 0 to 1) that is applied to the infrared portion of the direct solar radiation spectrum to account for the absorption of the wall. The value should be defined for normal incident rays.

Diffuse Hemispherical

specifies a multiplier (ranging from 0 to 1) that is applied to the diffuse solar radiation to account for the absorption of the wall. This setting is only available for semi-transparent walls.

Transmissivity

contains the settings that define the transmissivity of wall. This group box is only available when **semi-transparent** is selected for **BC Type**.

Direct Visible

specifies a multiplier (ranging from 0 to 1) that is applied to the visible portion of the direct solar radiation spectrum to account for the transmissivity of the wall. The value should be defined for normal incident rays. This setting is only available for semi-transparent walls.

Direct IR

specifies a multiplier (ranging from 0 to 1) that is applied to the infrared portion of the direct solar radiation spectrum to account for the transmissivity of the wall. The value should be defined for normal incident rays. This setting is only available for semi-transparent walls.

Diffuse Hemispherical

specifies a multiplier (ranging from 0 to 1) that is applied to the diffuse solar irradiation to account for the transmissivity of the wall. This setting is only available for semi-transparent walls.

S2S Parameters

contains the settings for the S2S radiation model. This group box is available only if you select the **Surface to Surface** radiation model. See Forming Surface Clusters (p. 2201) for details.

Faces Per Surface Cluster

sets the number of faces per surface cluster (FPSC) for the wall, and thus controls the number of radiating surfaces and (if you select **Cluster to Cluster** for **Basis** in the **View Factors and Clustering** dialog box) view factor surfaces.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Critical Zone

specifies that the wall is a critical zone. When this option is enabled, the value entered for **Faces Per Surface Cluster** will not be altered when you use **Automatic** clustering in the **View Factors and Clustering** dialog box, and impacts the calculations and actions performed by the buttons in the **Maximum Distance from Critical Zone** group box of the **Participating Boundary Zones** dialog box.

Participates in View Factor Calculation

specifies whether or not the wall participates in the view factor calculation as part of the S2S radiation model. This option is available only if you select the **Surface to Surface** radiation model.

Species

contains the species parameters. This tab is available only if you have enabled the **Species Transport** model in the Species Model Dialog Box (p. 4768).

Reaction

activates reactions at the wall. This item will appear only if you have enabled the **Wall Surface** reactions in the Species Model Dialog Box (p. 4768).

When this option is enabled, you can specify the following settings:

Reaction Mechanisms

allows you to specify a defined group, or mechanism, of available reactions. This item will appear only if the **Reaction** option is enabled. See Defining Zone-Based Reaction Mechanisms (p. 2387) for details about defining reaction mechanisms.

Surface Area Washcoat Factor

allows you to specify a factor, which multiplies the wall area to account for the increased surface area of washcoats. See Species Boundary Conditions for Walls (p. 1477) for details.

Initial Deposition Thickness

is the thickness of the solid species deposition on the wall at the start of the simulation. This item is available is available for transient simulation only and is used only for postprocessing of the **Wall Deposition Thickness** field variable.

Solid Species Density

is the density of the solid species deposited on the wall. This item is available is available for transient simulation only and is used only for postprocessing of the **Wall Deposition Thickness** field variable.

Species Boundary Condition

contains options for the specification of species boundary conditions. You can choose from the following conditions:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Specified Mass Flux** (default): Indicates that the species mass flux will be specified.
- **Specified Mass Fraction**: Indicates that the species mass fraction will be specified.

See Species Boundary Conditions for Walls (p. 1477) for details. This item is available only if the **Reaction** option is disabled.

Species Mass Fraction / Mass Flux

contains inputs for the species mass fractions and mass fluxes (depending on the specification method selected for each species). This item is available only if the **Reaction** option is disabled.

DPM

contains the discrete phase parameters. This tab is available only if you have defined at least one injection.

Discrete Phase Model Boundary Conditions

contains inputs for setting the fate of particle trajectories at the wall. These options will appear when one or more injections have been defined. See Setting Boundary Conditions for the Discrete Phase (p. 2779) for details.

Discrete Phase BC Type

sets the way that the discrete phase behaves with respect to the boundary.

reflect

rebounds the particle off the boundary with a change in its momentum as defined by the coefficients of restitution (see Particle Reflection at Wall in the *Fluent Theory Guide*). When at least one injection accounts for the rough wall model (the **Rough Wall Model** option is selected under the **Physical Models** tab in the **Set Injections Properties** dialog box), you can specify the **DPM Wall Roughness Parameters** (**Ra**, **Rq**, and **RSm** in Equation 12.201 in the *Fluent Theory Guide*). (See Rough Wall Model (p. 2768)).

trap

terminates the trajectory calculations and records the fate of the particle as "trapped". In the case of evaporating droplets, their entire mass instantaneously passes into the vapor phase and enters the cell adjacent to the boundary. See Figure 24.40: "Trap" Boundary Condition for the Discrete Phase (p. 2783).

escape

reports the particle as having "escaped" when it encounters the boundary. Trajectory calculations are terminated. See Figure 24.41: "Escape" Boundary Condition for the Discrete Phase (p. 2783).

reinject

reintroduces the particle into the domain when it reaches a certain domain boundary (for example, outlet). This item appears when one or more injections have been defined

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

and **Unsteady Particle Tracking** is enabled in the **Discrete Phase Model** dialog box. See The **reinject** Boundary Condition (p. 2784) for details.

wall-jet

indicates that the direction and velocity of the droplet particles are given by the resulting momentum flux, which is a function of the impingement angle. See Figure 12.6: "Wall Jet" Boundary Condition for the Discrete Phase in the Theory Guide.

wall-film

allows you to select the following options:

- **Particle-Wall Heat Exchange** (Particle-Wall Impingement Heat Transfer (p. 2786))
- **Film Condensation** (Film Condensation Model (p. 2806))
- **Gas-Side Boundary Layer Model** (default) (Gas-Side Boundary Layer Model (p. 2809))
- **Particle Stripping** (Setting the Lagrangian Wall Film Model (p. 2802))
- **Initialize Wall Film** (Patching the Wall Film (p. 2809))

and specify model parameters in the following group boxes:

- **Impingement/Splashing Parameters** (Setting the Lagrangian Wall Film Model (p. 2802))
- **Separation Model Parameters** (Setting the Lagrangian Wall Film Model (p. 2802))
- **Erosion Models** (Setting Particle Erosion and Accretion Parameters (p. 2787))

Selecting **wall-film** automatically enables the **Consider Children in the Same Tracking Step** option in the **Discrete Phase Model** dialog box (**Physical Models** tab).

See Setting the Lagrangian Wall Film Model (p. 2802) for details about the wall-film models and their parameters.

user-defined

specifies a user-defined function to define the discrete phase boundary condition type.

Discrete Phase BC Function

sets the user-defined function from the drop-down list.

Discrete Phase Reflection Coefficients

determine the behavior of reflecting particles. This item appears when **reflect** is chosen as the **Discrete Phase BC Type**. See Discrete Phase Boundary Condition Types (p. 2780) for details on setting the following items.

Normal

sets the type of function for the normal coefficient of restitution. This function can be **constant**, **piecewise-linear**, **piecewise-polynomial**, or **polynomial**.

Tangent

sets the type of function for the tangential coefficient of restitution. This function can be **constant**, **piecewise-linear**, **piecewise-polynomial**, or **polynomial**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DEM Collision Partner

contains a list of names to designate the collision partner.

Erosion Model

contains a list of erosion models. See Setting Particle Erosion and Accretion Parameters (p. 2787) for details about these items.

Generic Model

enables erosion rates to be computed using the generic erosion model. (See Accretion in the *Fluent Theory Guide* for background information about this model.) When this model is enabled, you must specify the erosion parameters in the **Generic Erosion Model Parameters** dialog box (opened by clicking the corresponding **Edit...** button) as described in Setting Particle Erosion and Accretion Parameters (p. 2787).

Finnie

enables erosion rates to be computed using the Finnie formulation. (See Accretion in the *Fluent Theory Guide* for background information about this model.) When this model is enabled, you must specify the erosion parameters in the **Finnie Model Parameters** dialog box (opened by clicking the corresponding **Edit...** button) as described in Setting Particle Erosion and Accretion Parameters (p. 2787).

McLaury

enables erosion rates to be computed using the McLaury formulation. (See McLaury Erosion Model in the *Fluent Theory Guide* for background information about this model.) When this model is enabled, you must specify the erosion parameters in the **McLaury Model Parameters** dialog box (opened by clicking the corresponding **Edit...** button) as described in Setting Particle Erosion and Accretion Parameters (p. 2787).

Oka

enables erosion rates to be computed using the Oka formulation. (See Oka Erosion Model in the *Fluent Theory Guide* for background information about this model.) When this model is enabled, you must specify the erosion parameters in the **Oka Model Parameters** dialog box (opened by clicking the corresponding **Edit...** button) as described in Setting Particle Erosion and Accretion Parameters (p. 2787).

DNV

enables erosion rates to be computed using the DNV formulation. (See DNV Erosion Model in the *Fluent Theory Guide* for background information about this model.) When this model is enabled, you must specify the erosion parameters in the **DNV Model Parameters** dialog box (opened by clicking the corresponding **Edit...** button) as described in Setting Particle Erosion and Accretion Parameters (p. 2787).

Sheer Stress

enables erosion rates to be computed using the sheer stress model. (See Abrasive Erosion Caused by Solid Particles in the *Fluent Theory Guide* for background information about this model.) When this model is enabled, you must specify the erosion parameters in the

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sheer Stress Model Parameters dialog box (opened by clicking the corresponding **Edit...** button) as described in Setting Particle Erosion and Accretion Parameters (p. 2787). This model is available only for multiphase flow with a dispersed secondary granular phase.

Granular Phase Shielding

includes the shielding effect in other erosion models that are currently enabled. The granular phase shielding is always considered in the sheer stress erosion model. (See Wall Shielding Effect in Dense Flow Regimes in the *Fluent Theory Guide* for background information about this model.) This option is available only when **Shear Stress Model Parameters** is selected.

Friction Coefficient

specifies μ in Equation 12.194 and Equation 12.199 in the *Fluent Theory Guide* as a **constant** or a **piecewise-linear**, **piecewise-polynomial**, or **polynomial** function of relative velocity. This item appears only when **Enable Rotation** is selected under the **Physical Model** tab of the Set Injection Properties Dialog Box (p. 5608).

UDS

displays the boundary conditions for user-defined scalars (UDSs) at the wall. This tab is available only if you have specified a nonzero number of user-defined scalars in the User-Defined Scalars Dialog Box (p. 5654).

User Defined Scalar Boundary Condition

contains options for the specification of UDS boundary conditions. See the separate Fluent Customization Manual for details.

Specified Flux

indicates that the flux of the UDS at the wall will be specified.

Specified Value

indicates that the value for the UDS at the wall will be specified.

User Defined Scalar Boundary Value

contains inputs for the value of the flux of the UDS, or the value of the UDS itself, depending on your selection for that UDS under **User Defined Scalar Boundary Condition**.

Wall Film

displays the boundary conditions for liquid films at the wall. This tab is available only if you have enabled the Eulerian Wall Film model in the Models Task Page (p. 4696).

Eulerian Film Wall

allows you to define a film wall condition for any wall. See Modeling Eulerian Wall Films (p. 3239) for details. Once this option is enabled, you can set the following:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Film Condition Type

allows you to select the desired condition type.

Boundary Condition

allows you to specify settings for boundary **Film Conditions**:

Film Mass Flux

The film mass source in terms of mass flux per unit area (kg/m^2–s).

X-Momentum Flux, Y-Momentum Flux, and Z-Momentum Flux

The film momentum source in terms of momentum flux per unit area (N/m^2).

Incoming Film Temperature

The film temperature (K).

Film Passive Scalar Flux

The film passive scalar source in terms of mass flux per unit area (kg/m^2–s).

Initial Condition

allows you to specify settings for initial **Film Conditions**:

Film Height

The film height at the wall boundary.

X-Velocity, Y-Velocity, and Z-Velocity

The film velocity components at the wall boundary.

Film Temperature

The film temperature (K).

Film Passive Scalar

The film passive scalar at the wall boundary (dimensionless).

Relative Film Momentum Flux

(moving reference frames or moving walls only) enables specification of the film momentum flux relative to the wall on which liquid film is defined (**Boundary Condition** type only). This option is only available when **Moving Wall** is selected in the **Wall** dialog box and/or **Frame Motion** is selected in the **Fluid** dialog box.

Relative Initial Film Velocity

(moving reference frames or moving walls only) enables specification of the initial velocity relative to the wall on which liquid film is defined (**Initial Condition** type only). This option

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

is only available when **Moving Wall** is selected in the **Wall** dialog box and/or **Frame Motion** is selected in the **Fluid** dialog box.

User Source Terms

enables you to specify the additional source terms to the film continuity (**Mass Flux**), momentum (**X-Momentum Flux**, **Y-Momentum Flux**, and **Z-Momentum Flux**), energy (**Heat Flux**), and passive scalar equations (**Scalar Flux**). This option is only available with initial conditions.

Flow Momentum Coupling

When this option is selected, the liquid film and the gas flow will share the same velocity at the interface of the liquid-gas interface using a two-way coupling. When this option is not selected, the coupling between the liquid film and the gas flow is only one-way, namely, while the gas flow impacts the film flow, the film flow does not impact the bulk of the gas flow.

Film Phase Change

enables the phase change for the selected wall. This option is available only when **Phase Change** is selected in the **Model Options and Setup** tab of the Eulerian Wall Film Dialog Box (p. 4870). Once enabled, you can select **Phase Change Model** and specify its parameters (in the **Phase Change Parameters** group box).

The following models are available:

- **diffusion-balance**: You can specify **Condensation Constant** and **Vaporization Constant**.
- **wall-boundary-layer**: No user input is required.
- **user-defined**: You can specify **Condensation Rate** and **Vaporization Rate** as constants, parameters, or profile user-defined functions (UDFs). Note that condensation and vaporization rates must be specified as positive values in kg/s. For more information on profile UDFs, the separate Fluent Customization Manual.

For more information on the phase change models see Coupling of Wall Film with Mixture Species Transport in the *Fluent Theory Guide*.

Film Contact Angle

enables the Partial Wetting model. This option is available only with initial condition when **Surface Tension** is selected in the **Model Options and Setup** tab of the Eulerian Wall Film Dialog Box (p. 4870). Once this option is enabled, you can specify the **Contact Angle Parameters**.

The following parameters are available:

- **Contact Angle**: Corresponds to θ_m described in Partial Wetting Effect in the *Fluent Theory Guide*. The input range for this parameter is 0 to 180 degrees.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Relative Standard Deviation**: Corresponds to α described in Partial Wetting Effect in the *Fluent Theory Guide*. The input for the **Relative Standard Deviation** should be in the range of 0 to 50%.
- **Contact Angle Force Beta**: Corresponds to β in Equation 17.29 in the *Fluent Theory Guide*. The input range for this parameter is 0 to 10.

For more information on modeling film partial wetting effect see Partial Wetting Effect in the *Fluent Theory Guide*.

Impingement Parameters

contains parameters related to particle-wall impingement

Impingement Model

is a drop-down list that contains available impingement models.

The list of models includes:

- **stanton-rutland** (default): consists of four impingement regimes: stick, rebound, spread and splash, which are based on the impact energy and wall temperature. See The Stanton-Rutland Model in the *Fluent Theory Guide* for details.
- **kuhnke**: consists of four impingement regimes: stick, rebound, spread and splash (thermal breakup). See The Kuhnke Model in the *Fluent Theory Guide* for details.

DPM Wall Splash

allows you to enable the wall splashing for the selected wall and specify the **Number of Splashed Particles**. The minimum value is 3. This option is available when **Particle Splashing** is enabled in the **Eulerian Wall Film** dialog box.

Kuhnke Model Parameters

(available when the **kuhnke** model is selected from the **Impingement Model** drop-down list)

Wall Roughness Length

enables you to specify wall roughness values used by the Kuhnke model.

The following options are available:

- **Ra**: the wall mean roughness that is used to compute K_{cold} in Equation 12.249 in the *Fluent Theory Guide*
- **Rz**: the average surface roughness that is used to compute splashed particle reflection angle distribution Equation 12.261 in the *Fluent Theory Guide*.

Critical Temperature Factor

is the dimensionless variable that is used to determine the impingement regime transition temperature in Equation 12.239 in the *Fluent Theory Guide*.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Potential

displays the boundary conditions for the electric potential field at the wall. This tab is available only if you have enabled either the **Electric Potential model** under the **Model** tree branch or the **Electrochemical** Reaction model in the Species Dialog Box (p. 4929).

Electrochemical Reaction

enables electrochemical reactions at the wall. This option is available only if **Electrochemical** reactions are enabled in the **Species Model** dialog box.

When this option is enabled, the following controls are available:

Faradaic Heat

(if enabled) includes heat from the electrochemical reaction in the energy equation. This option is available only if **Electrochemical** reactions are enabled in the **Species Model** dialog box.

Reaction Mechanism

contains a drop-down list of all available electrochemical reaction mechanisms. This option is available only if **Electrochemical** reactions are enabled in the **Species Model** dialog box.

Potential Boundary Condition

contains a drop-down list of available boundary condition types for the potential generated by electron current on walls. You can define the following potential boundary conditions for the :

- For **Specified Flux**, you must specify **Current Density**
- For **Specified Value**, you must specify **Potential** at the wall
- (two-sided internal walls only) For **Coupled**, the boundary condition you specify on a coupled wall will be automatically applied to both walls in the wall / wall-shadow pair.

  If you are using the potential solver alone, you can also use decoupled potential boundary conditions (**Specified Flux** or **Specified Value**) for the wall/wall-shadow pair, that is, you can define different boundary condition for them independently. However, this option is not available for **Lithium-ion Battery Model**, **Electrolysis and H2 Pump Model**, or **Electrochemical** reactions.

- **Contact Resistance**: is the contact resistance at the wall, R in Equation 18.3 in the *Fluent Theory Guide*.

For details about these items, see the following sections:

- Setting Up the Electric Potential Model (p. 3266)
- Setting Up the Lithium-ion Battery Model (p. 3268)
- Setting the Electrolysis and H2 Pump Model (p. 3275)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Electrolyte Potential Boundary Condition

contains a drop-down list of available boundary condition types for the potential generated by ionic current on walls. This item is similar to the **Potential Boundary Condition** drop-down list described above and is available only for the Electrolysis and H2 Pump model.

Ablation

contains the ablation model parameters.

Ablation Model

is a drop-down list that contains available ablation models. This tab is available only when **Ablation Dynamic Mesh Coupling** is enabled in the **Ablation Dynamic Mesh Coupling Model** dialog box. See The Ablation Condition at Wall Boundaries (p. ?) for details.

The available ablation models are:

- **none** (default): Signifies that this wall boundary is not an ablative surface.
- **Vielle's model**: Calculates the ablative surface recessing rate r using Equation 17.11 (p. ?). You need to specify the following Vielle's model parameters:
  - **Parameter A**: is A in Equation 17.11 (p. ?).
  - **Model Parameter n**: is n in Equation 17.11 (p. ?).
  - (optional) **Species Flux**: when enabled, mass and heat transfer of the ablative process are considered using Equation 17.12 (p. ?). For this option, you need to specify:
    - → **Ablative Material Density**: ρ_a in Equation 17.12 (p. ?)
    - → **Species Mass Fraction**: y_i in Equation 17.12 (p. ?)
- **Surface Reaction**: Calculates the ablative surface recessing rate r using Equation 17.13 (p. ?). You need to specify the following parameter:
  - **Ablative Material Density**: is ρ_a in Equation 17.13 (p. ?).

  The **Surface Reaction** model is available only when **Wall Surface** is enabled in the **Species Model** dialog box and **Reaction** is enabled in the **Wall** dialog box (**Species** tab).

Structure

contains the displacement boundary condition settings for walls that are adjacent to solid cell zones. This tab is only available when a model is selected in the Structural Model Dialog Box (p. 4869).

X-Displacement Boundary Condition, Y-Displacement Boundary Condition, Z-Displacement Boundary Condition

contains a drop-down list of methods for calculating the displacement of the nodes in that particular direction:

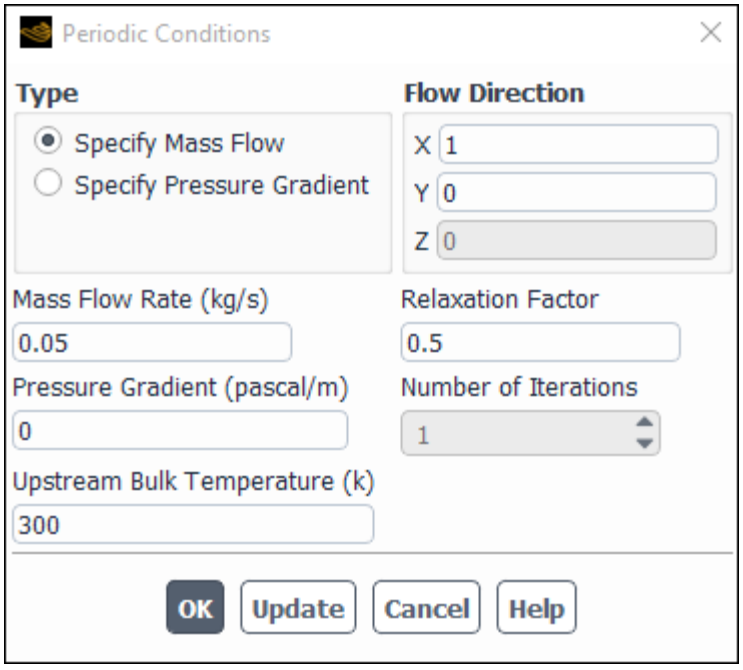
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Stress Free** specifies that the displacement is not affected by stress loads from the fluid flow.
- **Node X-**, **Node Y-**, and (for 3D cases) **Node Z-Force** specifies that the displacement results from a specified force applied on the nodes, which is defined using the **Node X-**, **Node Y-**, and (for 3D cases) **Node Z-Force** field.
- **Node X-**, **Node Y-**, and (for 3D cases) **Node Z-Displacement** applies a specified displacement on the nodes, which is defined using the **X-**, **Y-**, and (for 3D cases) **Z-Displacement** field.
- **Face Pressure** specifies that the displacement results from a specified pressure load on the faces, which is defined using the **Face Pressure** field.
- **Intrinsic FSI** specifies that the displacement results from pressure loads exerted by the fluid flow on the faces. Note that this selection is only available for a zone that is part of a wall / wall-shadow pair.

51.7.24. Periodic Conditions Dialog Box

The **Periodic Conditions** dialog box allows you to set parameters that define fully-developed periodic flow and heat transfer. See User Inputs for the Pressure-Based Solver (p. 1749) and Using Periodic Heat Transfer (p. 2276) for details.

(This dialog box is available only when the pressure-based solver is used; it is not available for the density-based coupled solvers.)



Controls

Specify Mass Flow

enables the specification of the mass flow rate.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Specify Pressure Gradient

enables the specification of the pressure gradient.

Mass Flow Rate

specifies the mass flow rate. This item will not be available if you selected the **Specify Pressure Gradient** option.

Important:

For axisymmetric problems, the mass flow rate is per 2π radians.

Pressure Gradient

specifies the pressure gradient (β in Equation 1.22 in the Theory Guide).

Upstream Bulk Temperature

sets the inlet bulk temperature for periodic heat transfer calculations.

Flow Direction

sets the direction of the periodic flow. The direction vector must be parallel to the periodic translation direction or its opposite.

Relaxation Factor

sets the under-relaxation factor that controls convergence of the iteration process described in Setting Parameters for the Calculation of β (p. 1751) for specified mass flow.

Number of Iterations

sets the number of subiterations done on the correction of β in the pressure correction equation for specified mass flow. See Setting Parameters for the Calculation of β (p. 1751) for details.

Update

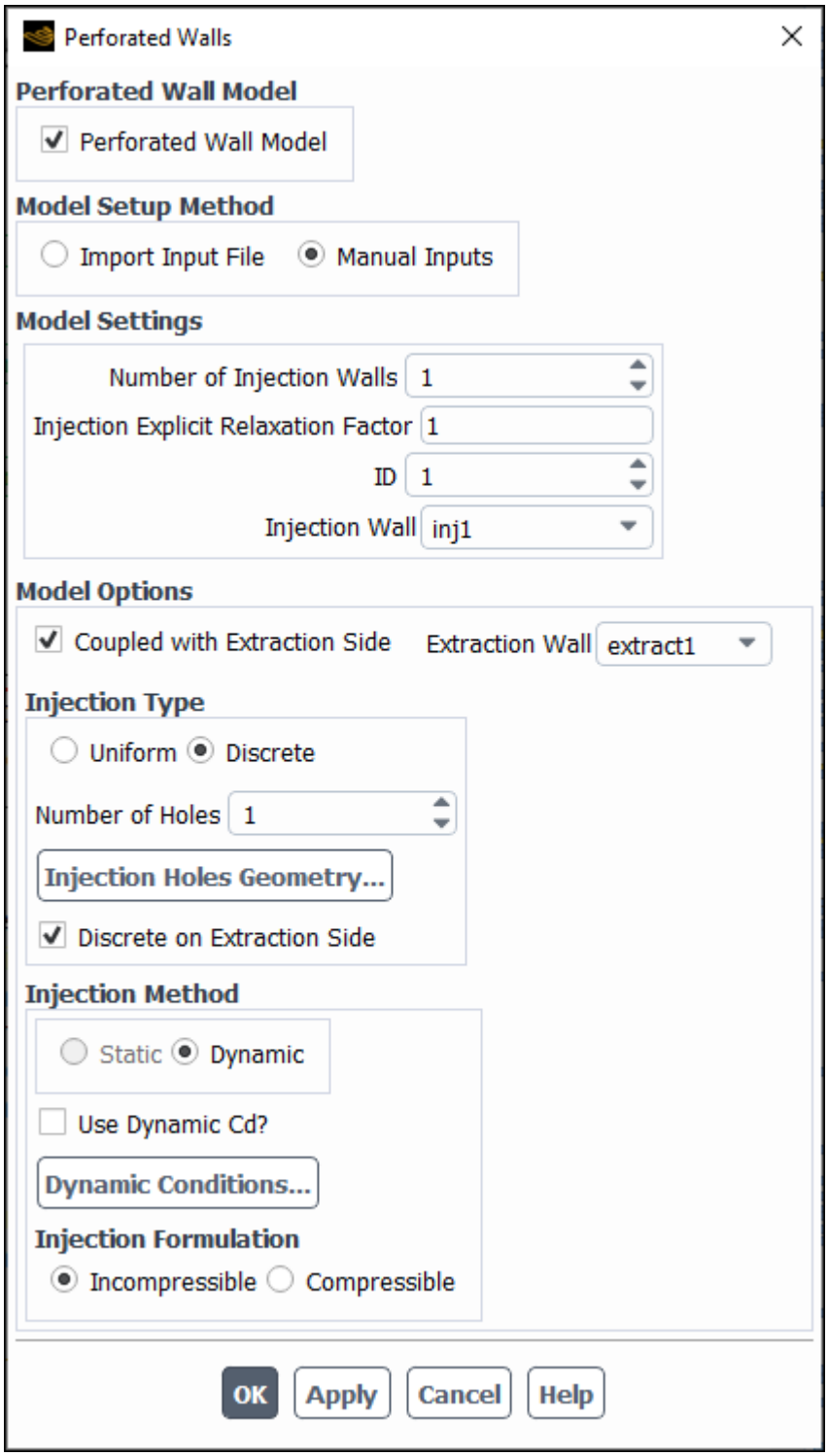
updates the **Pressure Gradient** field with the current value.

51.7.25. Perforated Walls Dialog Box

The **Perforated Walls** dialog box opens when you click the **Perforated Walls...** in the **Boundary Conditions** task page. It allows you to set parameters that define perforated walls boundary conditions. See Perforated Wall Boundary Conditions (p. 1483) for details.

(This dialog box is available only when the pressure-based solver is used; it is not available for the density-based solvers.)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Perforated Wall Model

enables the perforated wall model described in Perforated Wall Boundary Conditions (p. 1483).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Model Setup Method

is where you can select an appropriate method for setting up the perforated wall model. The following parameters and options are available:

Import Input Files

allows you to read an input file that contains the selections of model options and values of the model parameters. This option is useful for cases where hundreds or even thousands of perforated holes need to be defined.

Manual Inputs

allows you to manually specify options and parameters for the perforated wall model. When this option is selected, the **Perforated Walls** dialog box expands to reveal relevant model inputs.

Model Settings

is where you can specify:

Number of Injection Walls

specifies the total number of injection walls. You will need to define boundary conditions for each wall separately.

Injection Explicit Relaxation Factor

controls the update of the injection mass flow rate at each iteration. It is used to blend the injection mass flow rates calculated by Equation 7.131 (p. 1485) (for incompressible flow) or Equation 7.132 (p. 1485) (for compressible flow) at the current and previous iterations.

ID

sets the ID number of the injection wall. This number must be between 1 and the value specified in **Number of Injection Walls**.

Injection Wall

allows you to select the name of the injection wall.

Model Options

is a group box that contains the following settings:

Extraction Wall

allows you to select the extraction wall you want to couple with the injection wall.

Injection Type

is where you can select **Uniform** or **Discrete** modeling approach. See Modeling Concept (p. 1483) for more information about these two approaches.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For the **Discrete** injection type, you also need to specify the following:

Number of Holes

sets the total number of perforated holes in the injection wall.

Injection Holes Geometry...

opens the **Injection Holes** dialog box. See Setting Perforated Walls (p. 1487) for detailed information about this dialog box.

Discrete on Extraction Side

disabling this option allows you to model the uniform condition on the extraction side. This option is enabled by default.

Injection Method

is where you can set the following:

Dynamic Conditions...

opens the **Dynamic Setup** dialog box. See Setting Perforated Walls (p. 1487) for detailed information about this dialog box.

Injection Formulation

allows you to choose the injection formulation from the following options:

- **Incompressible**: uses Equation 7.131 (p. 1485) to calculate the injection mass flow rate at the hole exit.
- **Compressible**: uses Equation 7.132 (p. 1485) to calculate the injection mass flow rate at the hole exit.

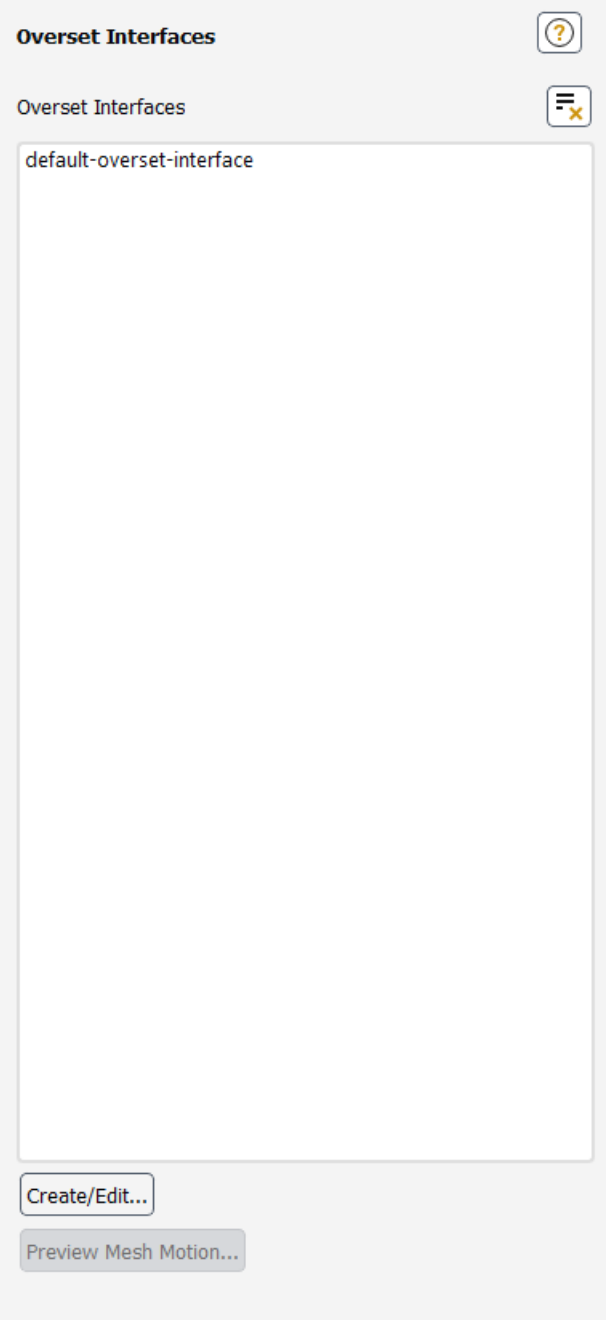
CHT Coupling

(cases with solid zone(s) only) when enabled, includes the effects of conjugate heat transfer between the solid walls and the injection flow. See Procedure for Manual Setup of Perforated Walls (p. 1489) for more information about this option.

51.8. Overset Interfaces Task Page

The **Overset Interfaces** task page allows you to create overset interfaces. For additional information about overset interfaces, see Overset Meshes (p. 1206).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Create/Edit...

displays the Create/Edit Overset Interfaces Dialog Box (p. 5110).

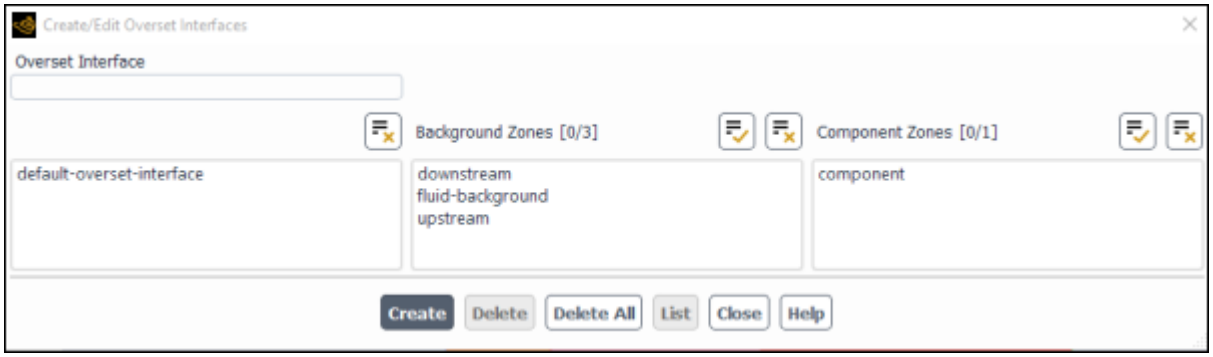
Preview Mesh Motion...

displays the Mesh Motion Dialog Box (p. 5151).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

51.8.1. Create/Edit Overset Interfaces Dialog Box

The **Create/Edit Overset Interfaces** dialog box allows you to create and delete overset interfaces. For additional information, see Overset Meshes (p. 1206).



Overset Interface

contains a text entry box where you must specify the name of the overset interface. Below is a list of all existing overset interfaces.

Background Zones

lists all of the background zones available for selection when creating an overset interface. Multiple background zones can be part of an overset interface; however, a background zone cannot be part of multiple overset interfaces. It is acceptable to have an interface with only component zones and no background zones.

Component Zones

lists all of the component zones available for selection when creating of an overset interface. Multiple components can be part of an overset interface; however, a component zone cannot be part of multiple overset interfaces.

Create

creates an overset interface using the name in the **Overset Interface** text box and the zones selected under **Background Zones** and **Component Zones**. Note that grid priorities, if required, are added to the interface after it is created, using the text command `define/overset-interfaces/grid-priorities`.

Delete

deletes the selected **Overset Interface**.

Delete All

deletes all overset interfaces in the domain.

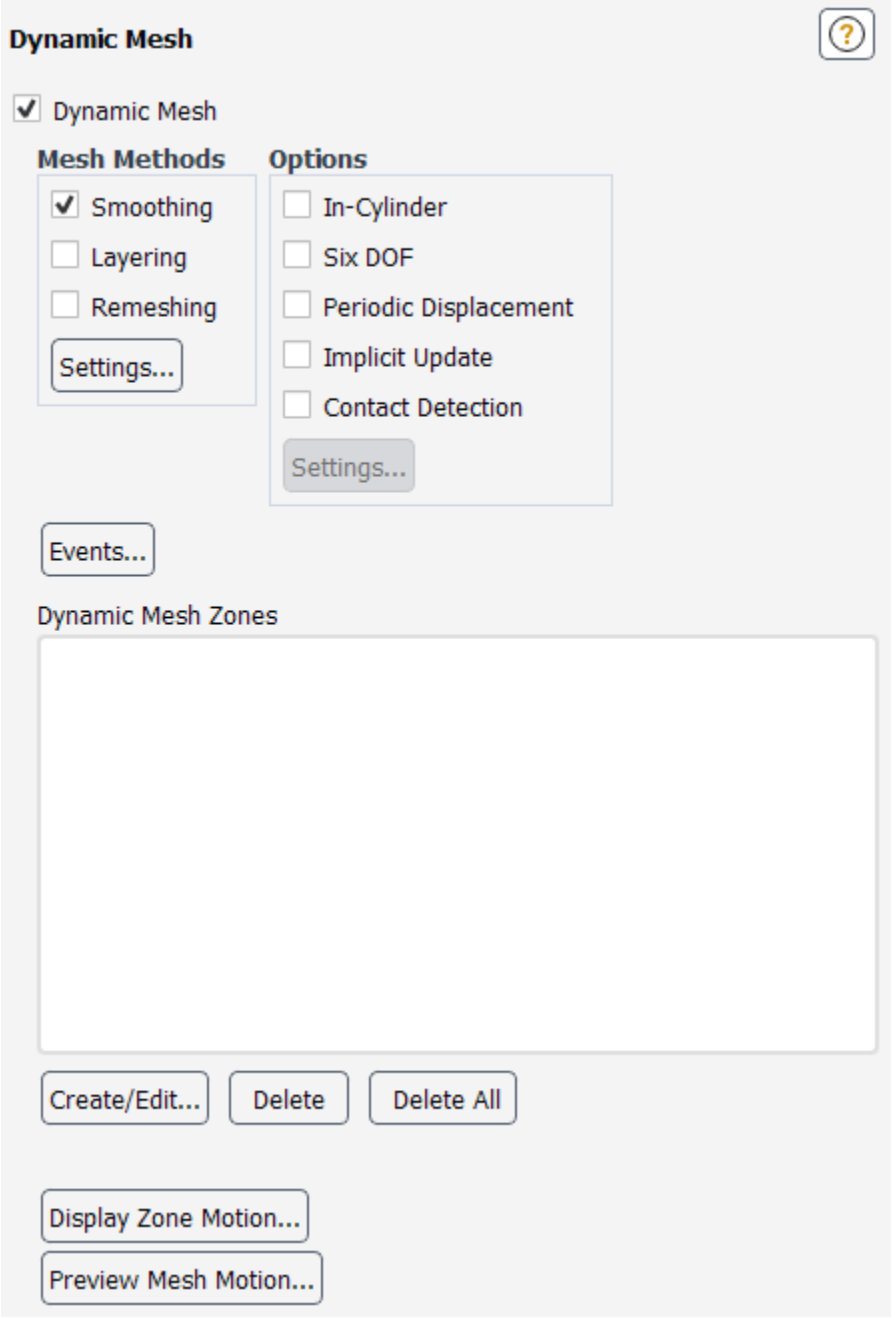
List

prints zonal information and grid priorities for the selected **Overset Interface**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

51.9. Dynamic Mesh Task Page

The **Dynamic Mesh** task page allows you to define the all the parameters for modeling a dynamic mesh model. See Setting Dynamic Mesh Modeling Parameters (p. 1814) for details about using the items below.



Controls

Dynamic Mesh

enables the dynamic mesh model and activates the controls in the task page.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mesh Methods

contains options to specify the mesh update method(s).

Smoothing

enables mesh smoothing. See Smoothing Methods (p. 1816) for details about the smoothing methods available.

Layering

enables the dynamic layering method that can be used to add or remove layers of cells adjacent to a moving boundary based on the height of the layer adjacent to the moving surface in prismatic mesh zones. See Dynamic Layering (p. 1837) for details.

Remeshing

enables remeshing methods. For details, see Remeshing (p. 1841).

Settings...

displays the Mesh Method Settings Dialog Box (p. 5114) in which you can specify settings for the Smoothing, Layering, and Remeshing methods.

Options

contains options to specify specialized dynamic mesh models.

In-Cylinder

enables the in-cylinder model. See In-Cylinder Settings (p. 1868) for more information.

Six DOF

enables the six degrees of freedom (DOF) solver. See Six DOF Solver Settings (p. 1883) for more information.

Periodic Displacement

enables periodic displacement for blade flutter analysis. See Using Dynamic Mesh Zones in a Blade Flutter Simulation (p. 1977) for more information.

Implicit Update

specifies that the mesh is updated during a time step (as opposed to just at the beginning of a time step). See Implicit Update Settings (p. 1888) for more information.

Contact Detection

enables contact detection. See Contact Detection Settings (p. 1890) for more information.

Settings...

opens the Options Dialog Box (p. 5121), where you can set the parameters for the options that are enabled in the **Options** group box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Events...

displays the Dynamic Mesh Events Dialog Box (p. 5135).

Dynamic Mesh Zones

displays a list of dynamic mesh zones.

Create/Edit...

displays the Dynamic Mesh Zones Dialog Box (p. 5140).

Delete

removes the selected dynamic zone(s) from the **Dynamic Mesh Zones** list.

Delete All

removes all dynamic zones from the **Dynamic Mesh Zones** list.

Display Zone Motion...

displays the Zone Motion Dialog Box (p. 5150).

Preview Mesh Motion...

displays the Mesh Motion Dialog Box (p. 5151).

For additional information, see the following sections:

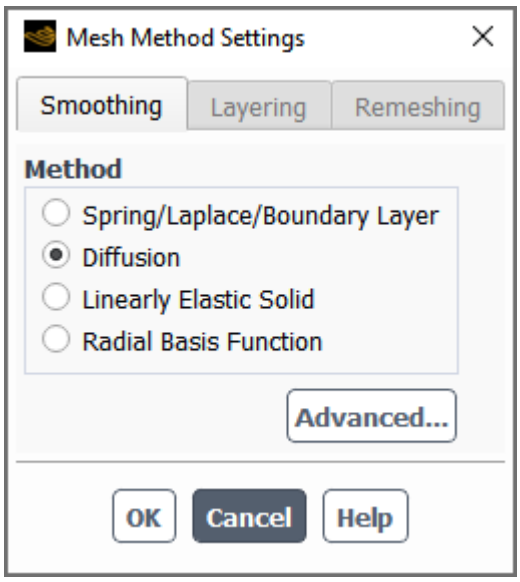
51.9.1. Mesh Method Settings Dialog Box
51.9.2. Mesh Smoothing Parameters Dialog Box
51.9.3. Advanced Remeshing Settings Dialog Box
51.9.4. Mesh Scale Info Dialog Box
51.9.5. Options Dialog Box
51.9.6. In-Cylinder Output Controls Dialog Box
51.9.7. Six DOF Properties Dialog Box
51.9.8. Periodic Displacement Group Dialog Box
51.9.9. Periodic Displacement Properties Dialog Box
51.9.10. Flow Control Settings Dialog Box
51.9.11. Dynamic Mesh Events Dialog Box
51.9.12. Define Event Dialog Box
51.9.13. Events Preview Dialog Box
51.9.14. Dynamic Mesh Zones Dialog Box
51.9.15. Orientation Calculator Dialog Box
51.9.16. Zone Scale Info Dialog Box
51.9.17. Zone Motion Dialog Box
51.9.18. Mesh Motion Dialog Box

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

51.9.19. Autosave Case During Mesh Motion Preview Dialog Box

51.9.1. Mesh Method Settings Dialog Box

The **Mesh Method Settings** dialog box allows you to apply settings for the smoothing, layering, or remeshing methods.



Controls

Smoothing

contains parameters to be specified for the smoothing mesh update method.

Method

allows you to specify the smoothing method.

Spring/Laplace/Boundary Layer

specifies that the smoothing method is spring based, or appropriate for the Laplacian smoothing method (for 2.5D remeshing) or the boundary layer smoothing method.

Diffusion

selects the diffusion-based smoothing method.

Linearly Elastic Solid

selects smoothing based on the equations for a linearly elastic solid.

Radial Basis Function

selects smoothing that is based on a radial basis function interpolation, so that Fluent solves for a displacement field that satisfies the displacements at boundaries with prescribed motion and then interpolates the displacements onto all other nodes in the interior and at the deforming boundaries.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Advanced...

opens the Mesh Smoothing Parameters Dialog Box (p. 5118), in which you can define the settings for the selected smoothing **Method**.

Layering

contains parameters to be specified for the layering mesh update method.

Options

specifies the criteria for splitting or collapsing cell layers.

Height Based

specifies that the cell layers are split or merged based on height.

Ratio Based

specifies that the cell layers are split or merged based on ratios.

Split Factor

specifies the value of α_s in Equation 13.15 (p. 1838). It controls the height or ratio at which the cells are split.

Collapse Factor

specifies the value of α_c in Equation 13.16 (p. 1839). It controls the height or ratio at which the cells are collapsed and merged into the next layer.

Remeshing

contains parameters to be specified for the remeshing mesh update method. For details about using the controls that follow, see Remeshing (p. 1841).

Methods-Based Remeshing

allows you to selectively enable a variety of remeshing options, each of which is suitable for particular kinds of cell types.

Unified Remeshing

specifies that an algorithm is applied that combines aspects of a variety of remeshing methods and by default attempts to maintain the initial mesh size distribution even as the mesh moves. It is applied to triangular or tetrahedral cells and can produce wedge cells in 3D boundary layer meshes. It simplifies the setup and can provide increased robustness, especially for parallel simulations.

Advanced...

opens the Advanced Remeshing Settings Dialog Box (p. 5120), in which you can revise the default settings for the **Unified Remeshing** method.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Methods

contains options that control remeshing. This group box is only available if you have selected **Methods-Based Remeshing**.

Local Cell

allows you to remesh deforming boundary cells.

Local Face

allows you to remesh deforming boundary faces. This option is available for 3D cases.

Region Face

allows you to remesh a region.

2.5D

enables the 2.5D model. This option is only available for 3D cases. See Using the 2.5D Model (p. 1859) for more information.

Parameters

contains parameters that control remeshing. This group box is only available if you have selected **Methods-Based Remeshing**.

Minimum Length Scale

specifies the lower limit of cell size below which the cells are marked for remeshing.

Maximum Length Scale

specifies the upper limit of cell size above which the cells are marked for remeshing.

Maximum Cell Skewness

specifies the desired maximum skewness for the mesh.

Maximum Face Skewness

specifies the desired maximum skewness for the surface mesh. If you selected **Methods-Based Remeshing**, this option is only available when **Local Face** is enabled under **Methods**.

Size Remeshing Interval

specifies the interval in time steps for remeshing based on the above size criteria only. Marking of cells based on skewness occurs automatically at every time step when **Remeshing** is enabled. This field is only available if you have selected **Methods-Based Remeshing**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mesh Scale Info...

opens the Mesh Scale Info Dialog Box (p. 5121), in which you can view the statistics of the mesh, such as the minimum and maximum length scale values and the maximum cell and face skewness values.

Default

sets the fields in the **Parameters** group box to values that represent a reasonable starting point for your particular mesh, as determined by Ansys Fluent. After execution, the **Default** button becomes the **Reset** button.

Reset

resets the fields in the **Parameters** group box to the prior values (that is, those displayed at the moment the **Default** button was clicked). After execution, the **Reset** button becomes the **Default** button.

Sizing Options

contains parameters that control the sizing function. This group box is only available if you have selected **Methods-Based Remeshing**.

Sizing Function

allows you to enable or disable the sizing function.

Resolution

sets the resolution for the sizing function. See Setting Dynamic Mesh Modeling Parameters (p. 1814) for more information.

Variation

specifies the value of α in Equation 13.22 (p. 1850).

Rate

specifies the value of β in Equation 13.23 (p. 1850).

Default

sets the sizing function parameters to the default values, as determined by Ansys Fluent based on the current mesh. After execution, the **Default** button becomes the **Reset** button.

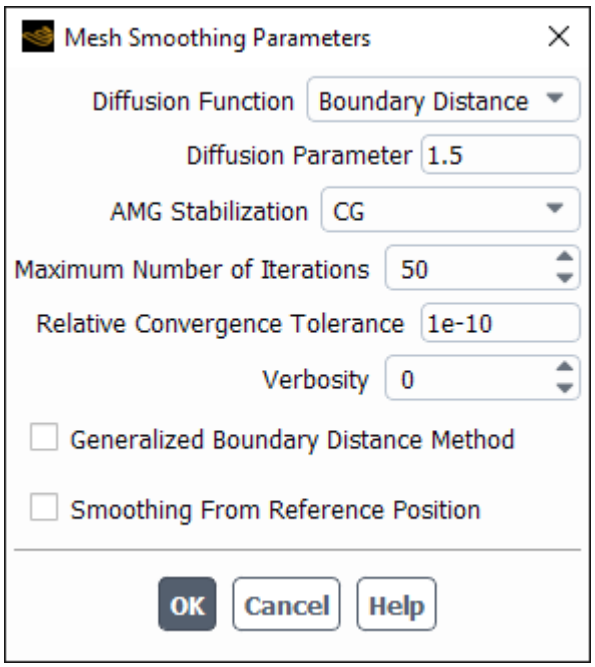
Reset

resets the sizing function parameters to the prior values (that is, those displayed at the moment the **Default** button was clicked). After execution, the **Reset** button becomes the **Default** button.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

51.9.2. Mesh Smoothing Parameters Dialog Box

The **Mesh Smoothing Parameters** dialog box is available by clicking the **Advanced...** button in the **Smoothing** tab of the Mesh Method Settings Dialog Box (p. 5114), and it allows you to define settings for the smoothing method.



Controls

Spring Constant Factor

controls the spring stiffness.

Convergence Tolerance

controls the smoothing convergence.

Maximum Number of Iterations

specifies the maximum number of iterations that will be attempted.

Elements

controls the elements where spring-based smoothing is applied.

Tet in Tet Zones

(3D only) specifies that only cell zones with all tetrahedral elements get smoothed.

Tet in Mixed Zones

(3D only) specifies that tetrahedral elements in mixed element zones get smoothed.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Tri in Tri Zones

(2D only) specifies that only cell zones with all triangular elements get smoothed.

Tri in Mixed Zones

(2D only) specifies that triangular elements in mixed element zones get smoothed.

All

specifies that all element types get smoothed.

Laplace Node Relaxation

specifies how the update of the node positions is relaxed on boundaries where there is Laplacian smoothing.

Verbosity

specifies the level of detail printed to the console during the calculation. Setting this to 1 (for radial basis function smoothing) ensures that the absolute tolerance computed (based on your specified relative tolerance) is printed, or (for all other smoothing methods) ensures that smoothing residuals are printed; alternatively, the default value of 0 suppresses such output.

Diffusion Function

specifies whether the diffusion coefficient is a function of the **Boundary Distance** (Equation 13.3 (p. 1818)) or the **Cell Volume** (Equation 13.4 (p. 1819)). This drop-down list is only available for diffusion-based smoothing.

Diffusion Parameter

specifies α in Equation 13.3 (p. 1818) or Equation 13.4 (p. 1819), depending on the selected **Diffusion Function**. This number-entry box is only available for diffusion-based smoothing.

AMG Stabilization

selects the algebraic multigrid (AMG) stabilization method used by the linear solver to perform mesh smoothing calculations. This drop-down list is only available for smoothing based on diffusion (when using the finite element method) or the linearly elastic solid model.

Relative Convergence Tolerance

specifies the relative residual tolerance for smoothing based on diffusion, the linearly elastic solid model, or a radial basis function interpolation.

Generalized Boundary Distance Method

is an option that is available when you have selected **Boundary Distance** for the **Diffusion Function**, and it allows you to use of a "generalized" boundary distance for the diffusion coefficient calculation. When this option is enabled, the calculation uses the normalized distance to the nearest boundary that is not declared as deforming, regardless of type; when it is disabled, only wall boundaries are considered.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Local Smoothing

limits the amount of the domain that is smoothed to only those cells that are close to the moving boundary zones, to ensure that the mesh away from the moving boundary zones remains undisturbed by the smoothing, and to decrease the computational time required for smoothing. This option is available when the smoothing method is based on a radial basis function interpolation. For details, see Local Smoothing with the Radial Basis Function Smoothing Method (p. 1832).

Smoothing From Reference Position

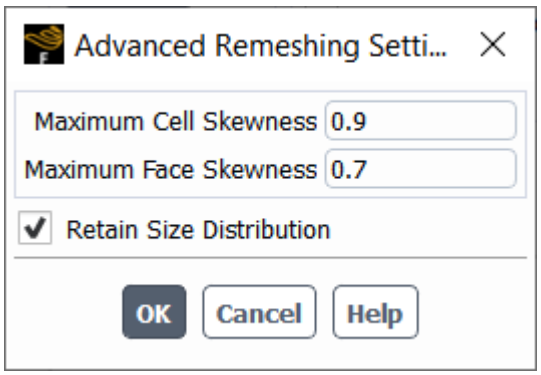
enables smoothing from a reference position. Such smoothing may produce greater mesh quality consistency for stationary or moving meshes with periodic or quasi-periodic motion. It is available when the smoothing method is based on diffusion, the linearly elastic solid model, or a radial basis function interpolation; it is not available with layering, remeshing, and/or automatic adaption.

Poisson's Ratio

specifies the linearly elastic solid material property ν in Equation 13.12 (p. 1829). This number-entry box is only available for smoothing based on the linearly elastic solid model.

51.9.3. Advanced Remeshing Settings Dialog Box

The **Advanced Remeshing Settings** dialog box allows you to revise the default settings for the **Unified Remeshing** method. For further details, see Unified Remeshing (p. 1861).



Controls

Maximum Cell Skewness

defines the maximum cell skewness in the zone.

Maximum Face Skewness

defines the maximum cell skewness in the surface mesh.

Retain Size Distribution

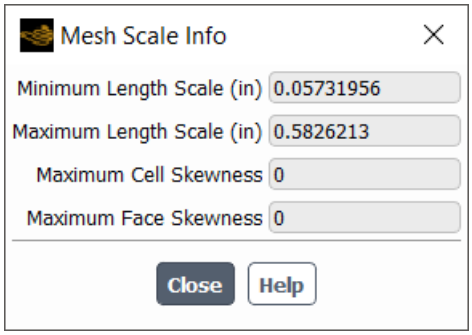
enables the use of local size criteria when marking cells for remeshing, in an attempt to maintain the initial mesh size distribution even as the mesh moves. When disabled, cells are instead marked when their length scales are outside of the minimum and maximum length scale values of the cell zone in the initial mesh. Either marking can be overridden if more restrictive values are specified

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

using the `define/dynamic-mesh/controls/remeshing-parameters/length-min` and `define/dynamic-mesh/controls/remeshing-parameters/length-max` text commands.

51.9.4. Mesh Scale Info Dialog Box

The **Mesh Scale Info** dialog box allows you to inspect the values of minimum and maximum length scale and maximum cell/face skewness in a mesh.



Controls

Minimum Length Scale

displays the minimum length scale in the mesh.

Maximum Length Scale

displays the maximum length scale in the mesh.

Maximum Cell Skewness

displays the maximum cell skewness in the zone.

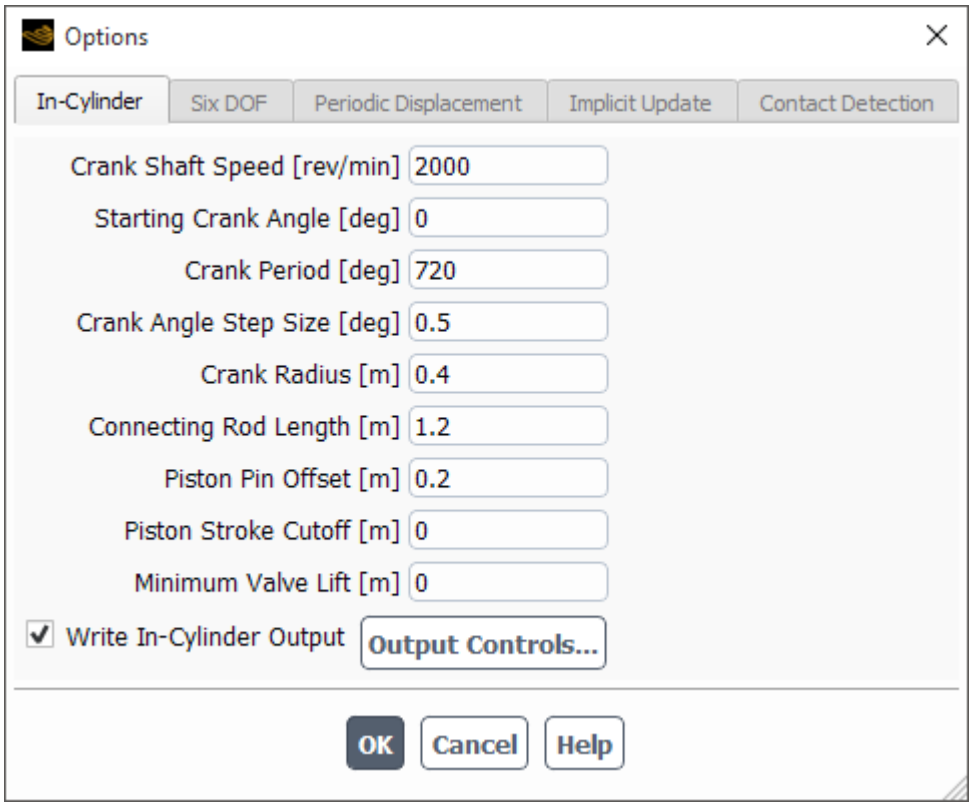
Maximum Face Skewness

displays the maximum cell skewness in the surface mesh.

51.9.5. Options Dialog Box

The **Options** dialog box allows you to set the parameters for the options available in the **Options** group box of the **Dynamic Mesh** task page.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

In-Cylinder

contains parameters to be specified for the in-cylinder model. See In-Cylinder Settings (p. 1868) for more information.

Crank Shaft Speed

specifies the speed of the crank shaft.

Starting Crank Angle

specifies the starting crank angle.

Crank Period

specifies the crank period.

Crank Angle Step Size

specifies the crank angle step size used to determine the time step size to advance the solution.

Crank Radius

specifies the crank radius to calculate the piston location.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Piston Pin Offset

specifies the perpendicular offset of the piston pin from the plane defined by the crank shaft axis and the direction of motion of the piston. The sign of this value is positive if top-dead-center (TDC) occurs prior to a crank angle of 0°.

Connecting Rod Length

specifies the length of the connecting rod.

Piston Stroke Cutoff

specifies the piston stroke cutoff used to control the onset of layering in the cylinder chamber.

Minimum Valve Lift

specifies the minimum valve lift.

Write In-Cylinder Output

enables or disables the writing of in-cylinder specific output parameters. When this option is enabled, the **Output Controls...** button becomes active.

Output Controls...

displays the In-Cylinder Output Controls Dialog Box (p. 5125) and is available only after the **Write In-Cylinder Output** option is enabled.

Six DOF

contains parameters to be specified for the six degrees of freedom (DOF) solver. See Six DOF Solver Settings (p. 1883) for more information.

Six DOF Properties

displays a list of defined sets of six DOF properties.

Create/Edit...

opens the Six DOF Properties Dialog Box (p. 5127).

Delete

removes the selected set of six DOF properties from the **Six DOF Properties** list.

Delete All

removes all sets of six DOF properties from the **Six DOF Properties** list.

Gravitational Acceleration

contains the text entry boxes for gravitational acceleration in X, Y, and Z directions.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

X, Y, Z

specifies the gravitational acceleration in X, Y, and Z directions respectively.

Write Motion History

allows you to keep track of an object's motion history.

File Name

allows you to specify a file name for saving the object's motion history.

Periodic Displacement

contains parameters to be specified for periodic displacement in blade flutter analysis. See Defining the Periodic Displacement of the Blades (p. 1978) for more information.

Periodic Displacement Groups

lists the currently available periodic displacement groups.

Create

creates a new periodic displacement group.

Edit

edits an existing periodic displacement group.

Delete

deletes an existing periodic displacement group.

Delete All

deletes all existing periodic displacement group.

Implicit Update

contains parameters to be specified for implicit mesh updating. See Implicit Update Settings (p. 1888) for more information.

Update Interval

allows you to specify the frequency in iterations at which the mesh will be updated within a time step.

Motion Relaxation

allows you to set a value (within the range of 0 to 1) for ω in Equation 13.29 (p. 1889), which defines the relaxation of the motion (that is, displacement of the nodes) during the mesh update.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Residual Criteria

allows you to set the relative residual threshold that is used to check the motion convergence.

Contact Detection

contains parameters to be specified for contact detection. See Contact Detection Settings (p. 1890) for more information.

Face Zones

allows you to select the face zones that will be involved in contact detection.

Proximity Threshold

allows you to set the threshold below which the user-defined function and/or flow control settings for contact detection will be invoked.

UDF

allows you to specify the user-defined function that will be invoked when contact has been detected.

Contact Region

contains controls that affect the flow in contact regions.

Flow Control

allows you to indicate whether you want the flow conditions to change for cells adjacent to faces zones that fall below the **Proximity Threshold** distance.

Settings...

opens the Flow Control Settings Dialog Box (p. 5134), where you can define flow control settings for the contact regions.

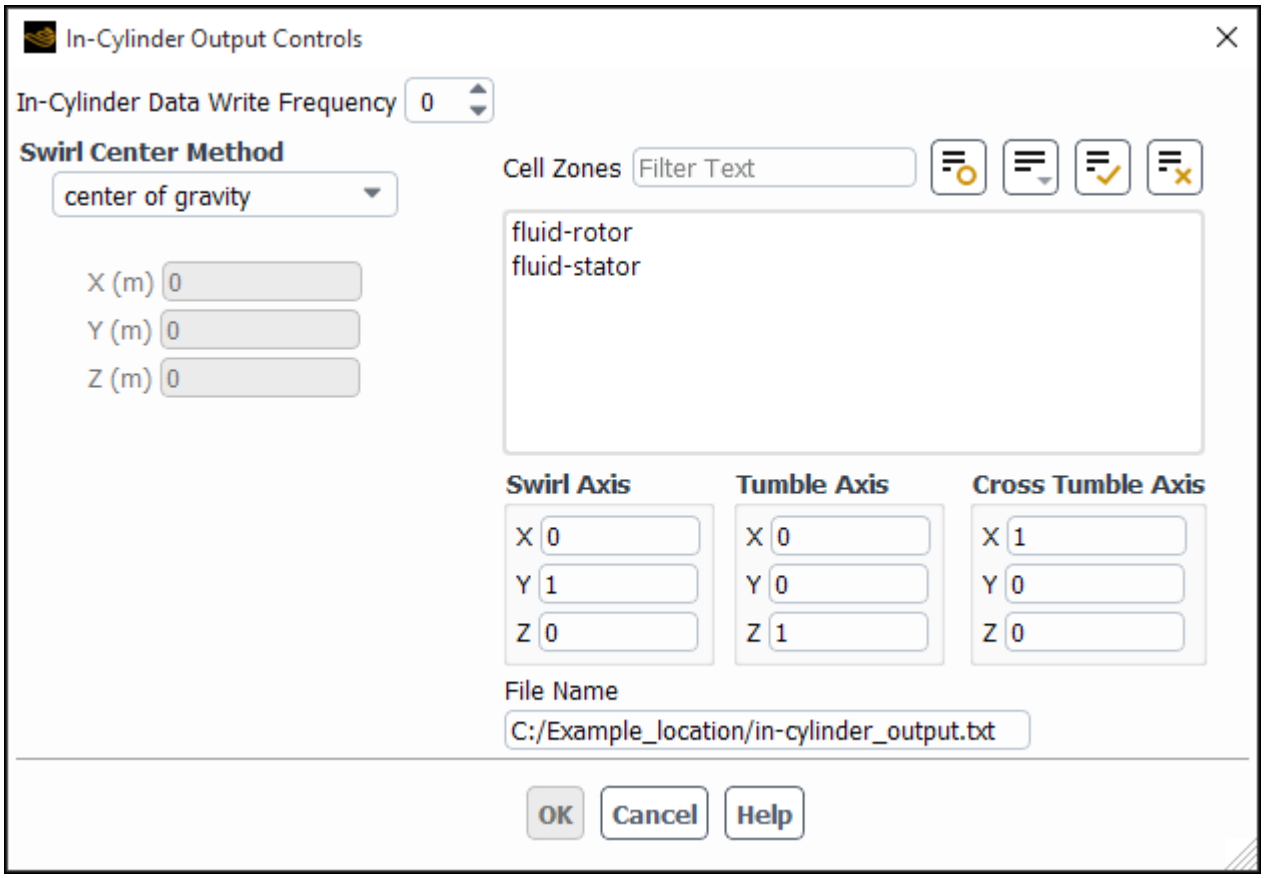
Update Contact Marks

updates which cells are marked in order to block flow in the contact region, using the current **Proximity Threshold** value. This button is only available when the **Contact Marks** method is selected in the Flow Control Settings Dialog Box (p. 5134).

51.9.6. In-Cylinder Output Controls Dialog Box

The **In-Cylinder Output Controls** dialog box contains parameters that control the output for the in-cylinder model. See In-Cylinder Settings (p. 1868) for more information.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

In-Cylinder Data Write Frequency

represents an integer entry specifying the interval in number of time-steps. Make sure that a value other than 0 is used for the frequency, in order to allow you to complete your setup.

Swirl Center Method

contains a drop-down list that allows you to select the method to calculate the swirl center. The list contains **center of gravity** and **fixed**, with **center of gravity** being the default value.

center of gravity

calculates the swirl center inside the code and is used as the center of gravity of the chosen cell zones.

fixed

enables you to specify a swirl center in the **X**, **Y**, and **Z** entry fields below the drop-down list.

In addition to these two options, you can chose to use your own **Compiled** user-defined function to calculate the swirl center. For details on using a dynamic mesh UDF, see the separate Fluent Customization Manual.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cell Zones

is a list that displays the names of all existing cell zones in the case files. You can select only the zones relevant for the swirl and tumble calculations.

Swirl Axis

specifies the swirl axis with three entries for the directional components. By default, **X**, **Y**, **Z**= **0**, **1**, **0**.

Tumble Axis

specifies the directional components of **Tumble Axis** in **X**, **Y**, **Z** directions. By default, **X**, **Y**, **Z** = **0**, **0**, **1**. This applies only in 3D.

Cross Tumble Axis

specifies the directional components of **Cross Tumble Axis** in **X**, **Y**, **Z** directions. By default, **X**, **Y**, **Z** = **1**, **0**, **0**. This applies only in 3D.

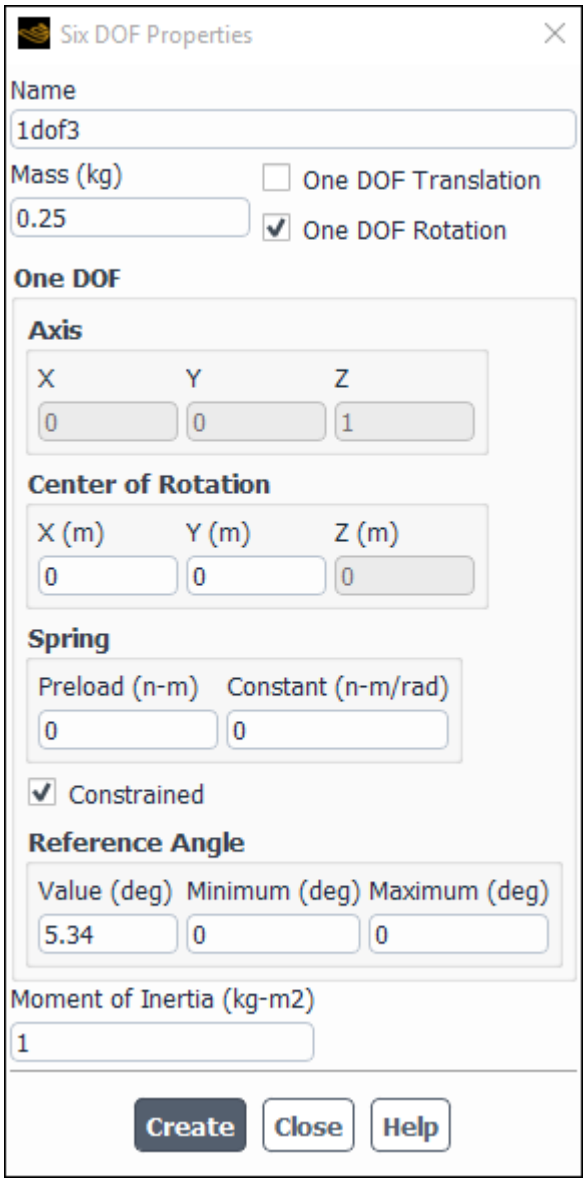
File Name

specifies the name of the **In-Cylinder** output file. By default, the file name contains the name of the case file appended with a `.txt` extension.

51.9.7. Six DOF Properties Dialog Box

The **Six DOF Properties** dialog box allows you to define a set of properties for a rigid body dynamic object that uses the **Six DOF** solver. The set of properties that you create can then be selected in the **Six DOF UDF/Properties** drop-down in the Dynamic Mesh Zones Dialog Box (p. 5140) for the zones that make up the object. See Setting Rigid Body Motion Attributes for the Six DOF Solver (p. 1885) and Rigid Body Motion (p. 1906) for more information.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name

specifies the name of the set of properties, which will be selectable in the **Six DOF UDF/Properties** drop-down in the Dynamic Mesh Zones Dialog Box (p. 5140).

Mass

defines the mass of the rigid body dynamic object.

One DOF Translation

specifies that the dynamic object will only undergo one DOF translation.

One DOF Rotation

specifies that the dynamic object will only undergo one DOF rotation.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

One DOF

contains settings that define the one DOF translation or rotation.

Direction

defines the vector along which the object will be allowed to translate.

Axis

defines the components of the axis of rotation. Note that for 2D cases, the axis will be either the positive or negative *z* direction, depending on the sign of the value you enter for **Z**.

Center of Rotation

defines the point about which the object rotates. Note that this center of rotation can be different than the **Center of Gravity** defined in the Dynamic Mesh Zones Dialog Box (p. 5140).

Spring

contains settings that define an optional Hooke's law spring that can exert forces / torques on the object. It is a compression / extension spring for translations, and it is a torsion spring for rotations.

Preload

defines a spring load that is constantly applied to the object, in addition to the loading resulting from the spring **Constant**. The **Preload** can be a positive or negative value, and is defined relative to the **Direction** (for translation) or the **Axis** (for rotation).

Constant

defines the spring constant / rate.

Constrained

specifies that the range of motion of the object is limited, as defined by the settings in the **Reference Point** group box (for translation) or the **Reference Angle** group box (for rotation).

Reference Point

contains settings that define the limits of the object's translation when the **Constrained** option is enabled.

Location

assigns a coordinate value to a point located on the object in its initial position; the **Minimum** and **Maximum** coordinates are then defined relative to this value along the **Direction** vector, in order to establish the limits of translation.

Minimum

defines the lower extent of the constrained translation along the **Direction** vector, relative to the **Location**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Maximum

defines the upper extent of the constrained translation along the **Direction** vector, relative to the **Location**.

Reference Angle

contains settings that define the limits of the object's rotation when the **Constrained** option is enabled.

Value

assigns an angular coordinate value to the orientation of the object in its initial position; the **Minimum** and **Maximum** angles are then defined relative to this orientation about the axis of rotation, in order to establish the limits of rotation.

Minimum

defines the lower extent of the constrained rotation about the axis of rotation, relative to the **Value**.

Maximum

defines the upper extent of the constrained rotation about the axis of rotation, relative to the **Value**.

Moment of Inertia

defines the moment of inertia for the rigid body dynamic object that undergoes one degree of freedom (DOF) rotation. Note that the values are relative to the **Center of Rotation** and **Axis** settings.

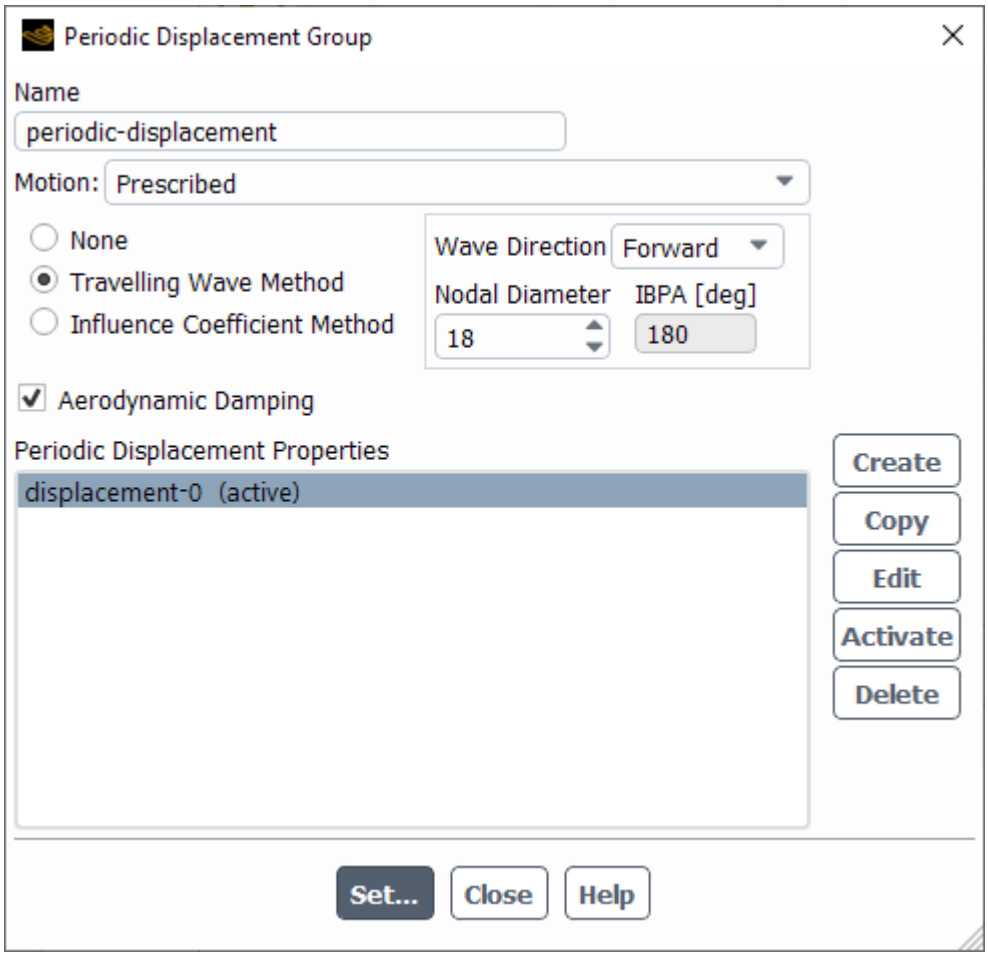
Inertia Tensor

defines the components of the inertia tensor for the object during six DOF motion. Note that the values are relative to the **Center of Gravity** and **Rigid Body Orientation** settings defined in the Dynamic Mesh Zones Dialog Box (p. 5140).

51.9.8. Periodic Displacement Group Dialog Box

The **Periodic Displacement Group** dialog box allows you to quickly run different mode shapes by having different displacements activated under each group. See Defining the Periodic Displacement of the Blades (p. 1978) for more information.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name

specifies the name of the periodic displacement group.

Motion

currently the only option is **Prescribed** with **None** (no wave), **Travelling Wave Method**, or **Influence Coefficient Method**.

Wave Direction

(only with Travelling Wave Method) specifies the direction of the wave as Forward or Backward.

Nodal Diameter

(only with Travelling Wave Method) specifies the vibration of the blades with **Interblade Phase Angle** (IBPA) which results in a traveling wave pattern. For complex mode shapes that are generated with cyclic symmetry, the mode shape profile is already phase-shifted so the IBPA is not needed. For real only mode shapes, you must supply information about the IBPA.

Periodic Displacement Properties

allows the definition of multiple periodic displacements.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Create

creates a new periodic displacement.

Copy

creates a copy of an existing periodic displacement.

Edit

edits an existing periodic displacement.

Activate

activates a periodic displacement for the current run. The `(active)` suffix will appear next to the currently active displacement.

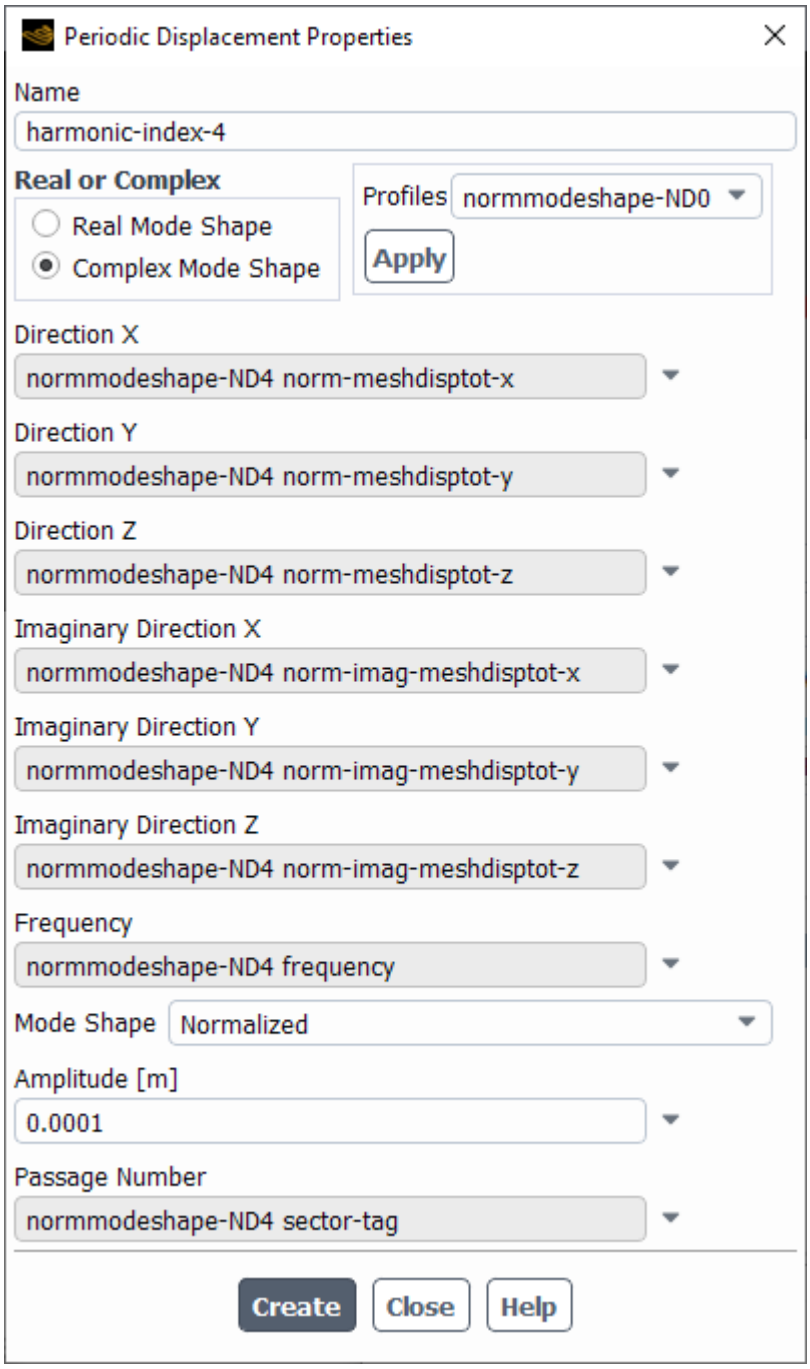
Delete

deletes a periodic displacement.

51.9.9. Periodic Displacement Properties Dialog Box

The **Periodic Displacement Properties** dialog box allows specify periodic displacement for an aerodynamic damping calculation. See Defining the Periodic Displacement of the Blades (p. 1978) for more information.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name

specifies the name of the periodic displacement.

Real or Complex

specifies the mode shape as real or complex.

Profiles

enables you to apply a displacement profiles to the appropriate fields.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Direction X,Y,Z

specifies the real displacement components typically using a mode shape profile.

Imaginary Direction X,Y,Z

(only available for Complex mode shape) specifies the imaginary displacement components typically using a mode shape profile.

Frequency

specifies the mode shape frequency.

Mode Shape

specifies whether the mode shape is **Normalized** or **Non-normalized**.

Amplitude

specifies the amplitude for normalized mode shapes to obtain the maximum modal displacement.

Scaling

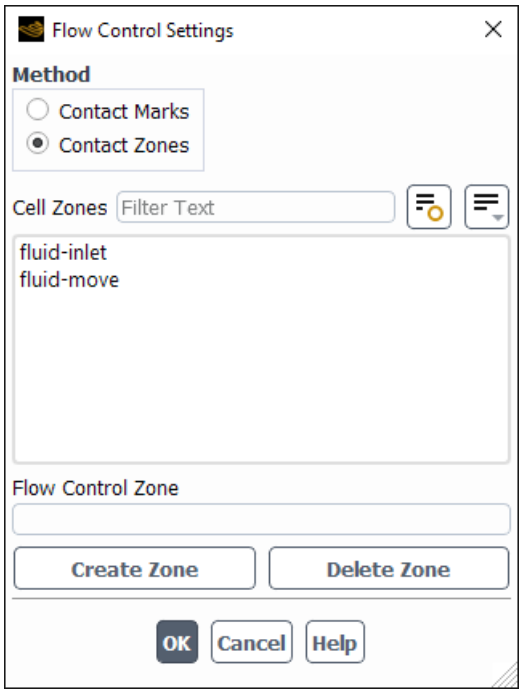
specifies the scaling parameter for non-normalized mode shapes to scale the displacement vectors.

Passage Number

specifies the sector tag, typically from the mode shape profile.

51.9.10. Flow Control Settings Dialog Box

The **Flow Control Settings** dialog box allows you to define flow control settings for the contact regions. See Contact Detection Settings (p. 1890) for more information.



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Controls

Method

allows you to select how the flow is controlled when contact is detected.

Contact Marks

specifies that cells in the contact region are marked and then treated with boundary zone conditions that block the flow.

Contact Zones

specifies that cells in the contact regions are separated into new cell zones that have revised physical properties (for example, inertial and/or viscous porous resistance) that restrict the flow. The physical properties are copied from the respective flow control zones, which are created using the other controls in this dialog box.

Solution Stabilization

enables the performance of additional iterations per time step and the application of solution controls to improve the stability of the solver during contact, as part of the contact marks method.

Cell Zones

selects a cell zone that provides the initial physical properties of a flow control zone.

Flow Control Zone

specifies a new name for or displays the existing name of a flow control zone for the selection in the **Cell Zones** list.

Create Zone

creates a flow control zone (based on the selection from the **Cell Zones** list, with the **Porous Zone** option enabled), and opens the **Fluid** dialog box so you can revise the settings to better represent the restricted flow.

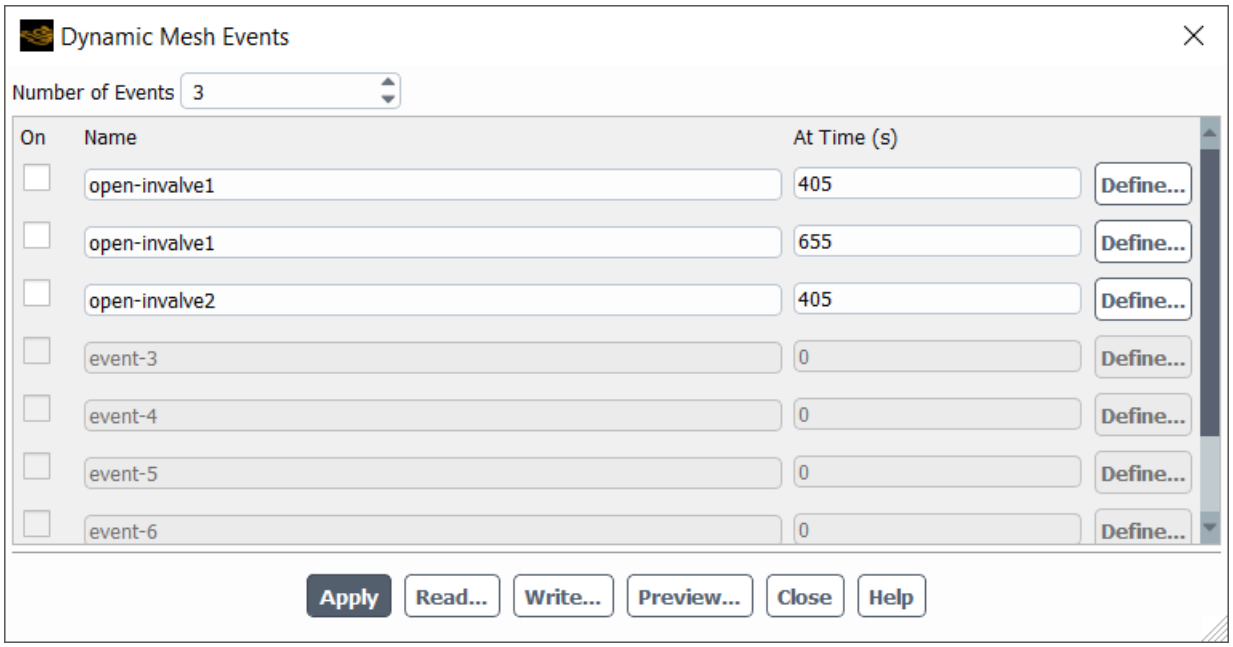
Delete Zone

deletes the current **Flow Control Zone**.

51.9.11. Dynamic Mesh Events Dialog Box

The **Dynamic Mesh Events** dialog box is available to control the timing of specific events during the course of the simulation. See Defining Dynamic Mesh Events (p. 1894) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Number of Events

specifies the number of events to be defined.

On

enables the corresponding event.

Name

specifies the name of the event.

At Crank Angle

specifies the angular location of the crank at which the event should occur. This option appears for in-cylinder flows.

At Time

specifies the time (in seconds) at which you want the event to occur. This option appears for non-in-cylinder flows.

Define...

opens the Define Event Dialog Box (p. 5137).

Read...

opens The Select File Dialog Box (p. 931).

Write...

opens The Select File Dialog Box (p. 931).

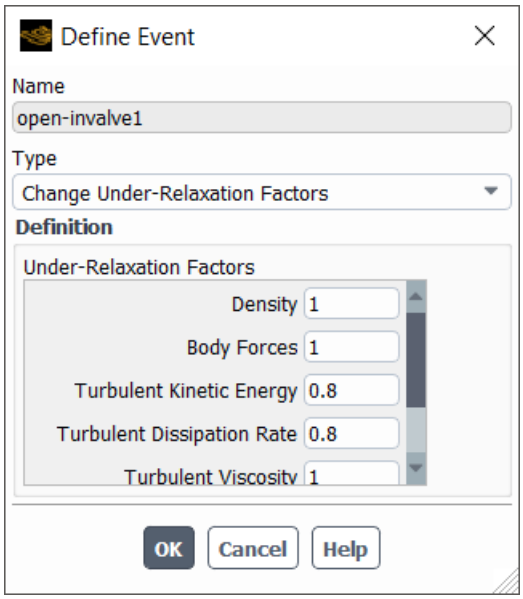
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Preview...

opens the Events Preview Dialog Box (p. 5139).

51.9.12. Define Event Dialog Box

The **Define Event** dialog box allows you to define events.



Controls

Name

contains the name of the event to be defined.

Type

specifies the type of event. You can choose the type of event from the drop-down list. These event types and their definitions are described in Events (p. 1898).

Definition

contains the input parameters corresponding to the type of event selected under **Type**.

Zone

contains a selectable list of the zones. The selection specifies the name of the zone(s) to be changed. This item will appear only for the **Change Zone Type** event.

New Zone Type

specifies the type of zone to which an existing zone must be changed. This item will appear only for the **Change Zone Type** event.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

From Zone

specifies the name of the zone from which the boundary condition is to be copied. This item will appear only for the **Copy Zone BC** event.

To Zone(s)

specifies the name of the zone to which the boundary condition is to be copied. This item will appear only for the **Copy Zone BC** event.

Zone(s)

contains a selectable list of the zones. The selection specifies the name of the zone to be activated/deactivated. This item will appear only for the **Activate Cell Zone** and **Deactivate Cell Zone** events.

Interface Name

contains the name of the interface to be created or deleted. This item will appear only for the **Create Sliding Interface** and **Delete Sliding Interface** events.

Interface Zone 1, Interface Zone 2

specifies the two zones on either side of the interface to be created. This item will appear only for the **Create Sliding Interface** event.

Wall 1 Motion, Wall 2 Motion

specifies the dynamic zones whose motion can be copied for the zones specified under **Interface Zone 1** and **Interface Zone 2**. This item will appear only for the **Create Sliding Interface** event.

Attribute

allows you to select the relevant motion attribute. This item will appear only for the **Change Motion Attribute** event.

Status

allows you to enable or disable the motion attribute. This item will appear only for the **Change Motion Attribute** event.

Dynamic Mesh Zones

contains a list of dynamic zones. This item will appear only for the **Change Motion Attribute** event.

Crank Angle Step Size

specifies the new physical time step size in degrees. This item will appear only for the **Change Time Step Size** event.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Base Dynamic Zone

specifies the zone from which the layer of cells is to be created. This item will appear only for the **Insert Boundary Layer** and **Remove Boundary Layer** event.

Side Dynamic Zone

represents the deforming face zone adjacent to the **Base Dynamic Zone** before the layer is inserted. This item will appear only for the **Insert Boundary Layer** event.

Internal Zone 1 Name, Internal Zone 2 Name

specifies the name of the new internal zones.

Time Step Size

allows you to change the time step size of the event. This item will appear only for the **Change Time Step Size** event.

Under-Relaxation Factors

allows you to specify the under-relaxation factors for the selected event. This item will appear only for the **Change Under-Relaxation Factors** event. See Setting Under-Relaxation Factors (p. 3589) for details.

Adjacent Dynamic Face Zone

allows you to select the dynamic face zone adjacent to the location of the cell layer to be inserted (or deleted). You can select the required zone from the drop-down list.

Direction Parameter

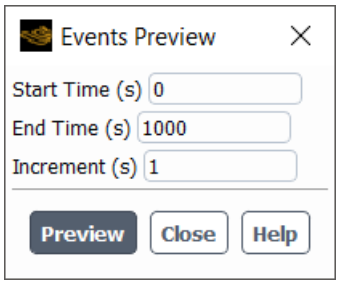
is the direction with respect to the selected dynamic face zone a layer of cells is removed or added.

Command

specifies the series of text / Scheme commands or the macro to be executed during the simulation. This item will appear only for the **Execute Command** event.

51.9.13. Events Preview Dialog Box

The **Events Preview** dialog box allows you to play the events to check that they are defined correctly.



Controls

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Start Crank Angle

specifies the crank angle at which you want to start the preview (for in-cylinder flows).

Start Time

specifies the time at which you want to start the preview (for non-in-cylinder flows).

End Crank Angle

specifies the crank angle at which you want to end the preview (for in-cylinder flows).

End Time

specifies the time at which you want to end the preview (for non-in-cylinder flows).

Increment

specifies the size of the step to take during the preview.

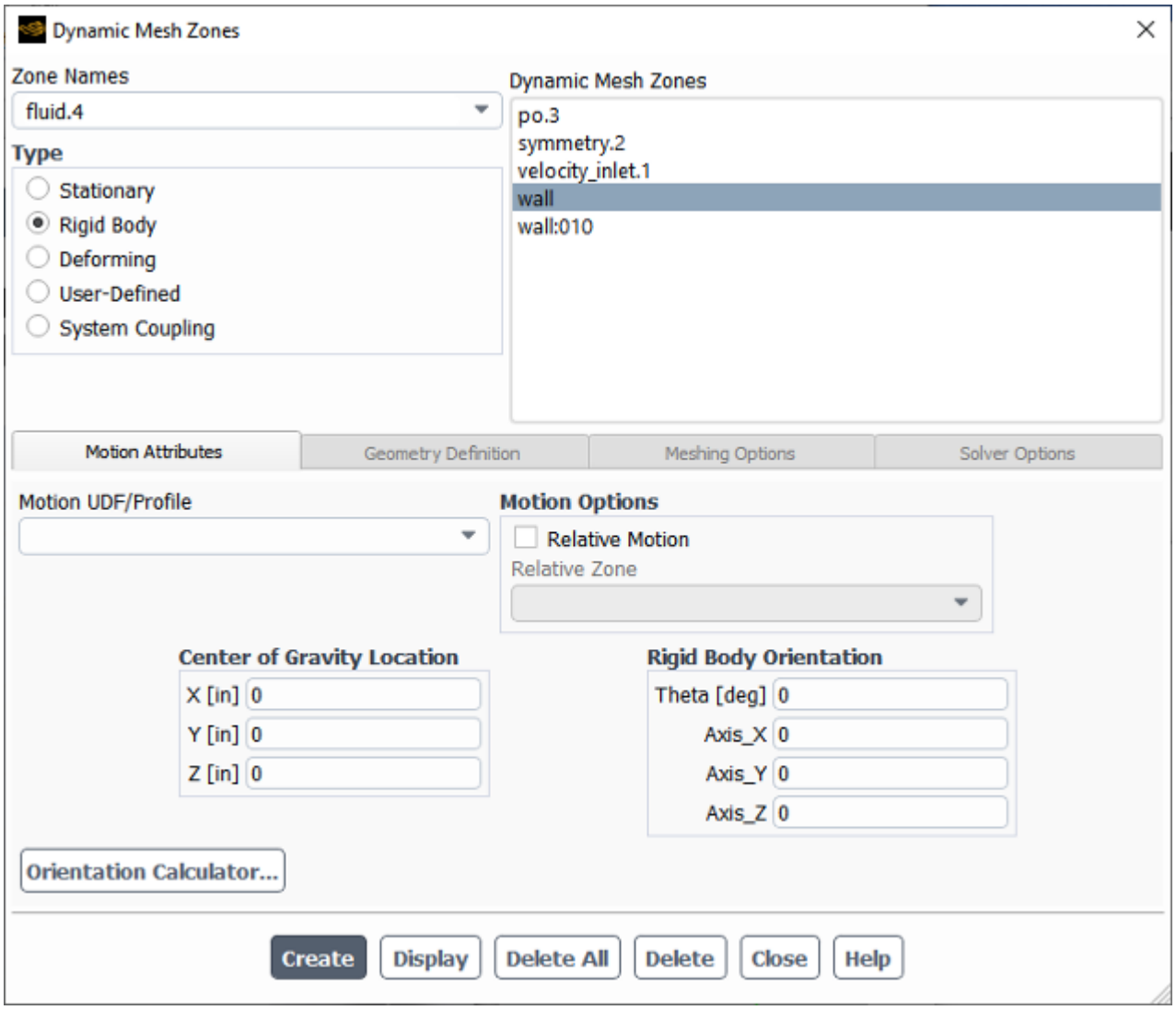
Preview

plays back the events at the crank angle specified for each event and reports when each event occurs in the text (console) window.

51.9.14. Dynamic Mesh Zones Dialog Box

The **Dynamic Mesh Zones** dialog box allows you to specify the motion of the dynamic zones in your model. See Specifying the Motion of Dynamic Zones (p. 1904) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Zone Names

contains the names of the zones in the model.

Type

contains the types of motion that can be specified for a dynamic zone.

Stationary

explicitly declares the zone as stationary, so that the nodes on this zone are excluded when updating the node positions. See Specifying the Motion of Dynamic Zones (p. 1904) for more information.

Rigid Body

specifies the zone as having a rigid-body motion.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Deforming

specifies the zone as deforming.

User-Defined

enables you to specify a user-defined zone motion.

System Coupling

allows the zone to be involved in a system coupling simulation where the motion is defined by the application that Ansys Fluent is coupled with on this zone (see the Fluent in Workbench User's Guide and the System Coupling User's Guide for more details). Once enabled, the **Solver Options** tab can be used to help achieve convergence for system coupling cases where the physics is strongly coupled. If a dynamic mesh zone of the type **System Coupling** exists, and Ansys Fluent is not involved in a system coupling simulation, then this zone type behaves in the same way as a stationary zone.

Intrinsic FSI

specifies that a wall zone between a fluid and solid cell zone deforms according to the deformation of the adjacent solid zone as part of a two-way intrinsic fluid-structure interaction (FSI) simulation. Note that this type can only be used for the side of a two-sided wall (that is, the wall or wall-shadow) that is immediately adjacent to the fluid cell zone (as indicated by the **Adjacent Cell Zone** field in the **Wall** dialog box). This option is only available when a model is selected in the Structural Model Dialog Box (p. 4869).

Dynamic Mesh Zones

lists all of the dynamic zones in the case.

Motion Attributes

contains parameters to specify the motion attributes for a rigid-body-motion zone, a deforming-motion zone, or a user-defined-motion zone.

Motion UDF/Profile

allows you to define the motion of the rigid body zone by selecting a motion definition, profile, or user-defined function from the drop-down list. Note that the names of motion definitions are preceded by **motion::** in this list. See Managing Motion Definitions (p. 1787) for information about motion definitions, see Profiles (p. 1564) and Solid-Body Kinematics (p. 1919) for information on profiles, and see the Fluent Customization Manual for information on user-defined functions.

Six DOF UDF/Properties

allows you to define the motion of a rigid body zone by selecting a profile, user-defined function, or set of six DOF properties (created using the Six DOF Properties Dialog Box (p. 5127)) from the drop-down list. This option is available only when **On** is enabled in the **Six DOF** group box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Six DOF

contains parameters for the **Six DOF** solver.

On

enables the use of the **Six DOF** solver.

Passive

(if enabled) does not take forces and moments on the zone into consideration.

Center of Gravity Location

specifies the initial values of the coordinates for the location of the center of gravity of the selected zone. This item will appear only if the motion **Type** is **Rigid Body** and **In-Cylinder** is not enabled in the **Dynamic Mesh** task page. After calculating, Ansys Fluent updates these fields, allowing you to keep track of the object's current location.

Motion Options

provides a **Relative Motion** option, which allows you to specify that the motion of the dynamic zone is relative to another pre-defined rigid body zone. This group box will only appear if the **Type** is **Rigid Body** or **User-Defined**; for the latter, the **Relative Zone** must be a cell zone.

Rigid Body Orientation

specifies the initial values that define the orientation of the rigid body of the selected zone. This item will appear only if the motion **Type** is **Rigid Body** and **In-Cylinder** is not enabled in the **Dynamic Mesh** task page. After calculating, Ansys Fluent updates these fields, allowing you to keep track of the object's current orientation. The rigid body orientation is used, if needed, by the six DOF solver to evaluate the external load forces and moments (see `DEFINE_SDOF_PROPERTIES` in the *Fluent Customization Manual*), as well as when evaluating the relative motion of a dynamic zone with respect to another (see `DEFINE_CG_MOTION` in the *Fluent Customization Manual*).

Center of Gravity Velocity

specifies the velocity of the center of gravity with respect to inertia coordinate system. This option is available only if the **Six DOF** solver is the selected model.

Rigid Body Angular Velocity

specifies the angular velocity of the rigid body with respect to inertia coordinate system. This option is available only if the **Six DOF** solver is the selected model.

Lift/Stroke

contains the current value of valve lift or piston stroke, which is automatically updated when you click **Create**. This item will appear only if **In-Cylinder** is enabled in the **Dynamic Mesh** task page.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Valve/Position Axis

specifies the direction of the reference axis of the valves or piston for an in-cylinder problem. This item will appear only if **In-Cylinder** is enabled in the **Dynamic Mesh** task page.

Mesh Motion UDF/Profile

allows you to select the user-defined function that defines the geometry and motion of the zone. This item will appear only if the motion **Type** is **User-Defined**.

Exclude Mesh Motion in Boundary Conditions

allows you to specify whether the boundary mesh motion is excluded in the physical boundary conditions of that zone. This option is only available for non-periodic boundary zones.

Geometry Definition

contains parameters to define a deforming zone.

Definition

allows you to select a geometry definition to project nodes of the deforming zone on. Available options are **faceted**, **plane**, **cylinder**, and **user-defined**. When the linearly elastic solid mesh smoothing method or the radial basis function smoothing method is enabled (see Linearly Elastic Solid Based Smoothing Method (p. 1828) or Radial Basis Function Smoothing (p. 1830), respectively), you also have the option of selecting **unspecified**. If you have created auxiliary geometry definitions (as described in Managing Auxiliary Geometry Definitions (p. 1791)), the names of the supported definitions will be available for selection in the list preceded by **geom::**.

Feature Options

allows you to preserve features for 3D cases on boundary zones of type **Deforming** or **User-Defined**. Such features may be at the juncture of different boundary zones, or they may be on a single non-planar boundary zone.

Feature Detection

enables the detection and preservation of features of a particular angular range.

Feature Angle

defines a threshold value in degrees. For details on how this angle is defined, see Feature Detection (p. 1868).

Point on Plane

allows you to specify the position of a point on the plane. This item will appear only when you select **plane** for the **Definition**.

Plane Normal

allows you to specify the direction of the plane normal. This item will appear only when you select **plane** for the **Definition**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cylinder Radius

allows you to specify the radius of the cylinder. This item will appear only when you select **cylinder** for the **Definition**.

Cylinder Origin

allows you to specify the location of the cylinder origin. This item will appear only when you select **cylinder** for the **Definition**.

Cylinder Axis

allows you to specify the direction of the cylinder axis. This item will appear only when you select **cylinder** for the **Definition**.

Geometry UDF

allows you to select a user-defined function for a geometry **Definition**.

Meshing Options

contains parameters for various meshing options.

Adjacent Zone

contains the name of the adjacent zone that is involved in local remeshing or dynamic layering. This item will appear only if the zone type is **Stationary**, **Rigid Body**, **User-Defined**, or **Intrinsic FSI**.

Cell Height

allows you to specify the ideal height of the adjacent cells as either a constant value or a compiled user-defined function. This item will appear only if the motion **Type** is **Stationary**, **Rigid Body**, **User-Defined**, or **Intrinsic FSI**.

Deform Adjacent Boundary Layer with Zone

enables the smoothing of an adjacent boundary layer mesh. This option is not available when **Stationary** or **Deforming** is selected in the **Type** list.

Remeshing

enables remeshing for a deforming zone, and allows you to use the settings in the group box below to specify whether the global settings defined in the **Remeshing** tab of the Mesh Method Settings Dialog Box (p. 5114) are applied to this zone. The settings that are available in the group box are based on whether you are defining a boundary zone or a cell zone, and on the global remeshing settings.

Parameters

provides remeshing settings for boundary zones or cell zones.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Global Settings

allows you to specify that this deforming zone uses the settings you defined globally in the **Remeshing** tab of the **Mesh Method Settings** dialog box. When this option is disabled, you can use the other settings under **Parameters** to define unique local remeshing criteria.

Minimum Length Scale

specifies the lower limit of cell size below which the cells are marked for remeshing for the zone.

Maximum Length Scale

specifies the upper limit of cell size above which the cells are marked for remeshing for the zone.

Maximum Skewness

specifies the desired value for maximum skewness in the zone.

Zone Scale Info...

opens the Zone Scale Info Dialog Box (p. 5150), in which you can view the statistics of the zone, such as minimum, maximum, and average length scale values, as well as the maximum skewness.

Options

provides remeshing options for the zone. This group box is only available if you have selected **Methods-Based Remeshing** in the **Remeshing** tab of the Mesh Method Settings Dialog Box (p. 5114).

Region

enables the region-based remeshing method for boundary zones.

Local

enables the local face remeshing method for boundary zones.

Smoothing

enables smoothing for a deforming zone, and allows you to use the settings in the group box below to specify whether the global settings defined in the **Smoothing** tab of the Mesh Method Settings Dialog Box (p. 5114) dialog box and the Mesh Smoothing Parameters Dialog Box (p. 5118) are applied to this zone. The settings that are available in the group box are based on whether you are defining a boundary zone or a cell zone, and on the global smoothing settings.

Elements

allows you to specify the element types for which spring-based smoothing is used in this zone. This will only be available if you are defining a cell zone and the

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Spring/Laplace/Boundary Layer method has been selected in the **Smoothing** tab of the **Mesh Method Settings** dialog box.

Global Settings

allows you to specify that this deforming zone smooths the element types you specified globally in the Mesh Smoothing Parameters Dialog Box (p. 5118). When this option is disabled, you can make a unique local selection using the radio buttons below.

Tet in Tet Zones

(3D only) specifies that this zone will get smoothed only if it consists entirely of tetrahedral elements.

Tet in Mixed Zones

(3D only) specifies that all tetrahedral elements in this zone should get smoothed. All other elements in a mixed element zone will not undergo smoothing.

Tri in Tri Zones

(2D only) specifies that this zone will get smoothed only if it consists entirely of triangular elements.

Tri in Mixed Zones

(2D only) specifies that all triangular elements in this zone should get smoothed. All other elements in a mixed element zone will not undergo smoothing.

All

specifies that all element types in this zone will get smoothed.

Methods

allows you to specify the smoothing method for the zone. This is only available if you are defining a boundary zone and the **Spring/Laplace/Boundary Layer** method has been selected in the **Smoothing** tab of the **Mesh Method Settings** dialog box.

Spring

enables the spring-based smoothing method for this zone.

Laplace

enables the Laplacian smoothing method for this zone.

Remeshing Method

provides settings for local face remeshing of the dynamic zone. This group box will appear only for boundary zones, when the motion **Type** is **User-Defined**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Local

enables local face remeshing for the dynamic zone. This option is only available when **Unified Remeshing** is selected or **Local Face** is enabled in the **Remeshing** tab of the Mesh Method Settings Dialog Box (p. 5114).

Maximum Skewness

is a field that, if revised (with a reasonable value), sets a maximum skewness for the local face remeshing of the dynamic zone that is different than the global value set in the **Remeshing** tab of the Mesh Method Settings Dialog Box (p. 5114).

Solver Options

contains settings that may help convergence for boundary zones in cases that involve strong fluid-structure interaction (notably zones undergoing rigid body motion with the six degrees of freedom solver, system coupling motion, or two-way intrinsic FSI motion). See Solution Stabilization for Dynamic Mesh Boundary Zones (p. 1918) for more information.

Solution Stabilization

enables/disables solution stabilization settings for boundary zones.

Stabilization Parameters

contains options for solution stabilization settings.

Scale Factor

allows you to set the scale factor used by the selected **Method** for convergence stabilization.

Method

Stabilization is achieved through a boundary source coefficient introduced in the continuity equation. It is designed to improve the diagonal dominance of the matrix system in the cells adjacent to the boundary zone. Two methods for this boundary source coefficient are available:

volume-based

specifies that the diagonal entry of the linear matrix system corresponding to the discretized continuity equation is re-scaled using the cell volume and a scale factor, according to Equation 13.32 (p. 1918).

coefficient-based

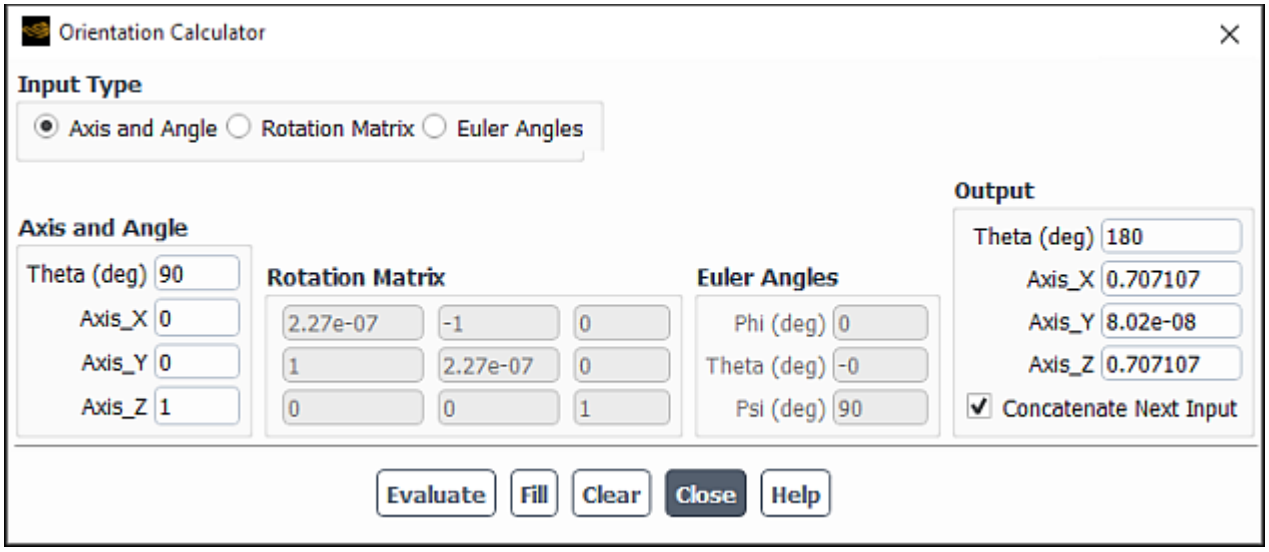
(default) specifies that the diagonal entry of the linear matrix system corresponding to the discretized continuity equation is directly re-scaled by a scale factor, according to Equation 13.33 (p. 1919).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

51.9.15. Orientation Calculator Dialog Box

The **Orientation Calculator** dialog box allows you to convert rotation matrices and Euler angles to axis and angle format. It also allows you to concatenate rotations to determine a rigid body's final orientation. The output of this calculator can then be used to fill in the initial rigid body orientation of your rigid body motion dynamic zone, which can be used to track the orientation during the calculation.

Figure 51.3: Orientation Calculator Dialog Box



Controls

Input Type

allows you to select which method you will use to enter your rigid body orientation values. For additional information about these input types, see Rigid Body Motion (p. 1906).

Axis and Angle

allows you to enter the angle of rotation and the axis about which the rigid body is rotating.

Rotation Matrix

allows you to enter the orientation of the rigid body as a rotation matrix.

Euler Angles

allows you to enter the orientation of the rigid body as a set of Euler Angles. For an explanation of which Euler angles method Fluent uses, see Rigid Body Motion (p. 1906).

Output

displays the angle of rotation and the axis about which the rigid body is rotating after you click **Evaluate**.

Note that if you enter a single orientation in **Axis and Angle** and click **Evaluate**, the values shown in **Output** will be the same as you entered in **Axis and Angle**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Concatenate Next Input

concatenates the next input entered (after clicking **Evaluate**) to the values displayed in **Output**

Evaluate

converts the specified input values into angle and axis format and displays the results in **Output**.

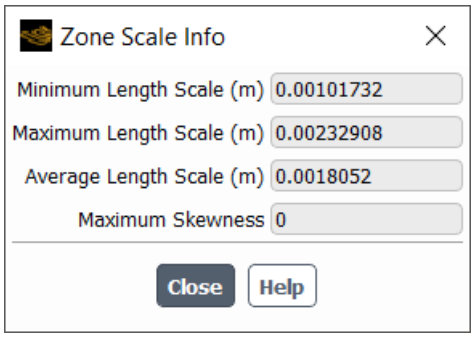
Note that the calculator also converts the entered values into the other two "grayed-out" input formats.

Fill

enters the values displayed in **Output** into the **Rigid Body Orientation** fields of the **Dynamic Mesh Zones** dialog box.

51.9.16. Zone Scale Info Dialog Box

The **Zone Scale Info** dialog box allows you to inspect the values of minimum and maximum cell area or volume, and maximum cell skewness in a zone.



Controls

Maximum Length Scale

displays the maximum cell length in the zone.

Minimum Length Scale

displays the minimum cell length in the zone.

Average Length Scale

displays the average cell length in the zone.

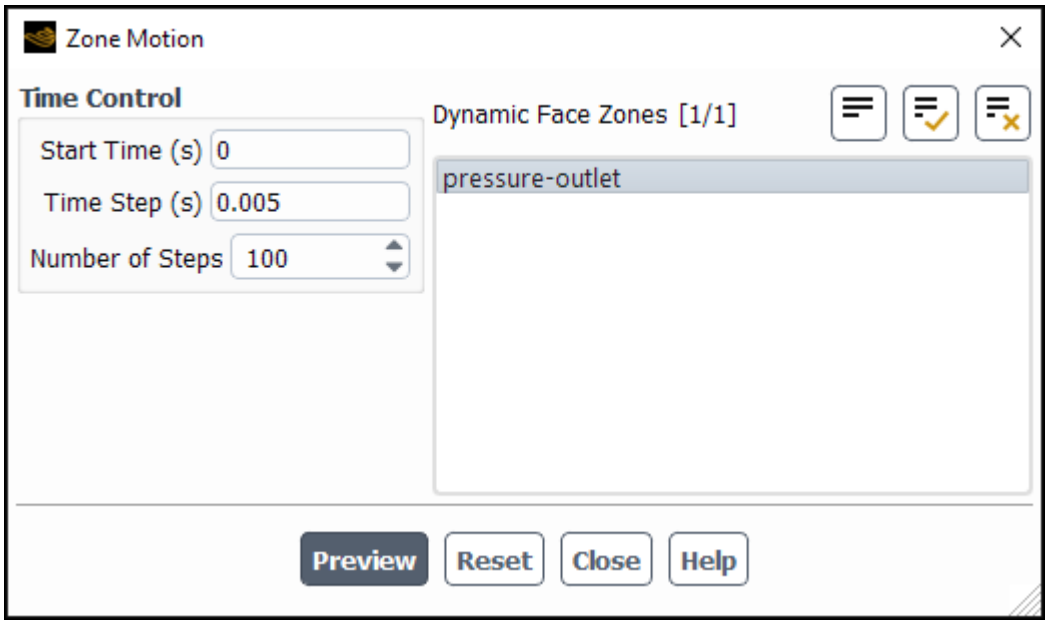
Maximum Skewness

displays the maximum cell skewness in the zone.

51.9.17. Zone Motion Dialog Box

The **Zone Motion** dialog box will display the motion of zones specified with **Rigid Body** or **User-Defined** motion. See Previewing the Dynamic Mesh (p. 1922) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Time Control

contains controls to specify the time intervals at which to display the motion.

Start Time (s)

specifies the time from which to start the zone motion preview.

Time Step (s)

specifies time step size for zone motion preview.

Number of Steps

specifies number of time steps for zone motion preview.

Dynamic Face Zones

allows you to select the dynamic face zones to preview. Only **User-Defined** or **Rigid Body** zones are available.

Preview

previews the zone motion.

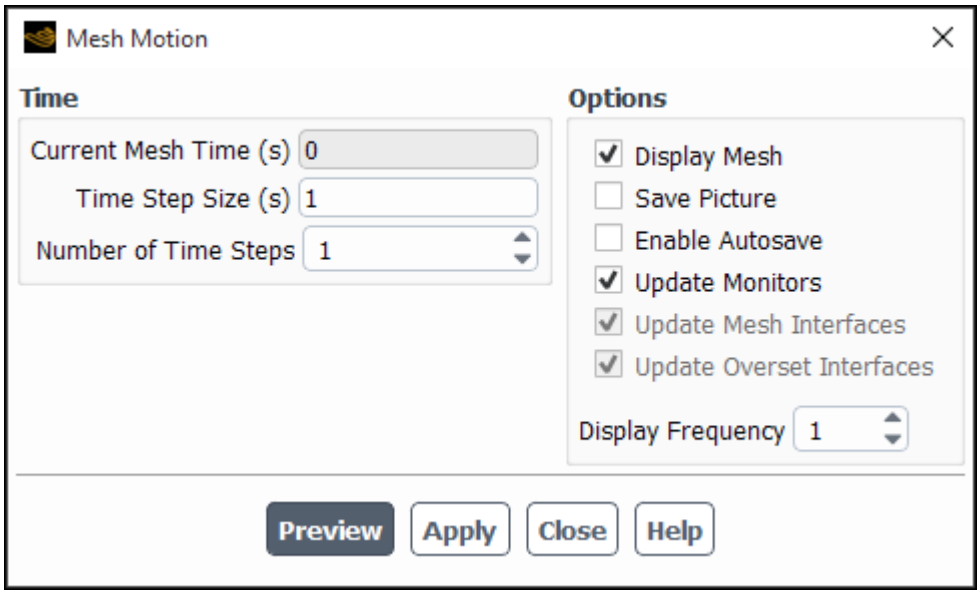
Reset

resets the zone display and the dialog box inputs.

51.9.18. Mesh Motion Dialog Box

The **Mesh Motion** dialog box allows you to preview the dynamic mesh as it changes with time before you start your simulation. See Previewing the Dynamic Mesh (p. 1922) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Time

contains the parameters to specify the time interval at which to update the mesh.

Current Mesh Time

displays the current time after the dynamic mesh has been advanced the specified number of steps.

Time Step Size

specifies the size of each time step.

Number of Time Steps

specifies the number of time steps.

Options

contains options to view the updated mesh.

Display Mesh

displays the mesh.

Save Picture

opens the Save Picture Dialog Box (p. 5254), allowing you to save a picture file of the mesh each time Ansys Fluent updates it during the mesh preview.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Enable Autosave

opens the Autosave Case During Mesh Motion Preview Dialog Box (p. 5153), allowing you to save the case and data files with the specified name and frequency. See Automatic Saving of Case and Data Files (p. 958) for details.

Update Mesh Interfaces

allows you to update the interface at every time step.

Update Monitors

allows you to disable the processing of monitors and computation activities during mesh motion preview.

Display Frequency

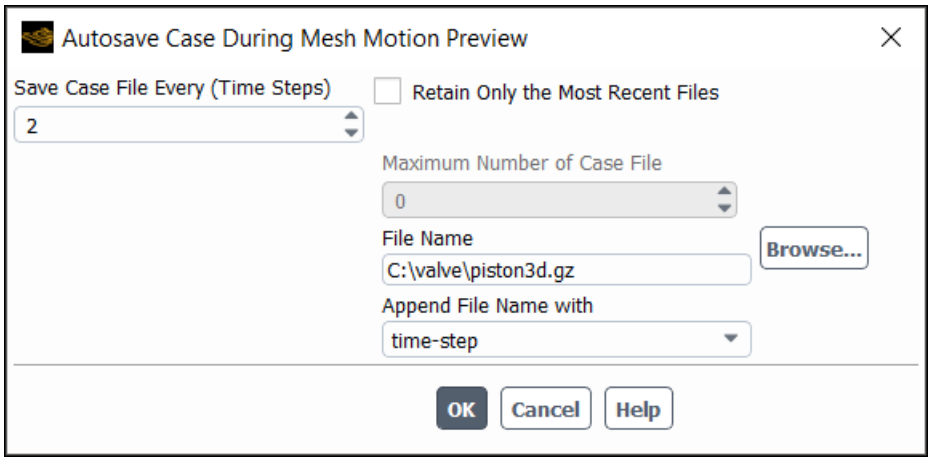
displays the frequency at which Ansys Fluent will update the mesh display.

Preview

allows you to preview the motion of the selected zones in the graphics window.

51.9.19. Autosave Case During Mesh Motion Preview Dialog Box

The **Autosave Case During Mesh Motion Preview** dialog box allows you to automatically save your case during the dynamic mesh preview as it changes with time, while running your simulation. See Automatic Saving of Case and Data Files (p. 958) for details about the inputs.



Controls

Save Case File Every

contains the parameters to specify the time interval at which to update the mesh.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Retain Only the Most Recent Files

allows you to restrict the number of files saved by Ansys Fluent if you have limited disk space. When this option is enabled, you can enter the appropriate value in the **Maximum Number of Data Files** field.

Note:

Only the associated case files are retained when using this option.

Maximum Number of Data Files

specifies the maximum number of data files that can be saved at any instance. If you have constraints on the disk space, you can restrict the number of files to be saved using this field. After saving the specified number of files, Ansys Fluent will overwrite the earliest existing file. The default value for this field is zero, which saves all the files.

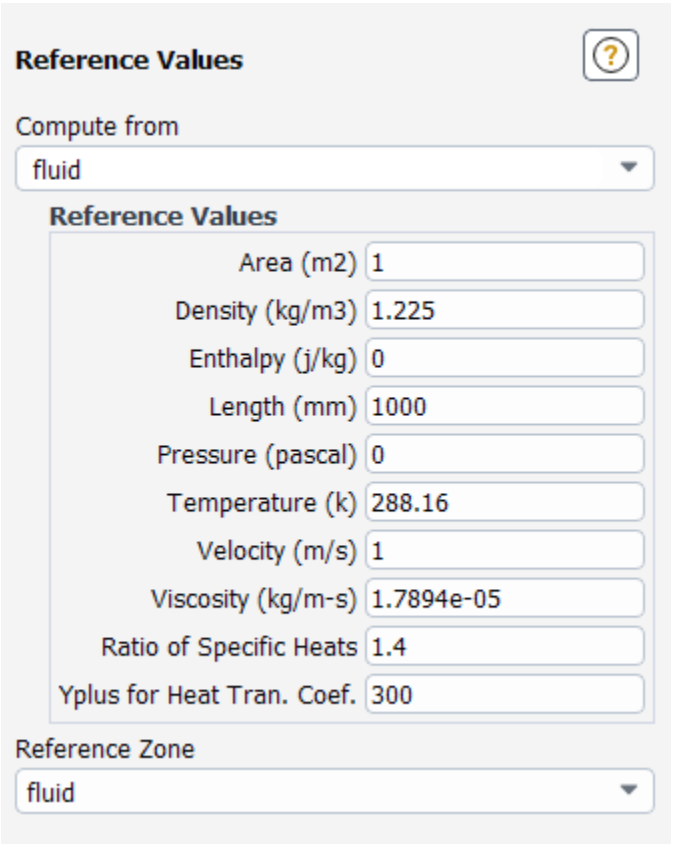
Append File Name with

allows you to select **flow-time** or **time-step** to be appended to the file name. This option is available only for unsteady-state calculations. The default selection is **flow-time**.

51.10. Reference Values Task Page

The **Reference Values** task page allows you to set the reference quantities used for computing normalized flow field variables. It also allows you to specify the reference zone for postprocessing relative velocities in moving-zone problems. See Reference Values (p. 4158) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Compute from

contains a drop-down list of the boundary zones. You may select a zone to be used for automatically defining the reference values, but depending on the boundary condition used, all of the reference values may not be set. For example, the reference length and area will not be set by computing the reference values from a boundary condition; you must set these manually.

Reference Values

contains inputs for the reference values.

Area

sets the reference area, which is used to compute the force and moment coefficients.

Depth

sets the reference depth used for computing cell volumes in 2D.

Density

sets the reference density, which is used to compute the reference dynamic pressure.

Enthalpy

sets the reference enthalpy, which is used to determine the total enthalpy change.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Length

sets the reference length, which is used in the computation of the moment coefficient.

Pressure

sets the reference pressure, which is used to compute the pressure-related forces and moments and the pressure coefficient.

Temperature

sets the reference temperature, which is used to compute entropy for incompressible flows.

Velocity

sets the reference velocity magnitude, which is used to compute the reference dynamic pressure.

Viscosity

sets the reference kinematic viscosity, which is used in the computation of the boundary Reynolds number.

Ratio of Specific Heats

sets the value of the specific heat ratio, which is used in turbomachinery efficiency calculations.

Yplus for Heat Tran. Coef.

sets the heat transfer coefficient based on the temperature of the fluid at a specified Yplus.

Reference Zone

contains a drop-down list of all cell zones in the domain. For flows involving multiple moving zones, you must select the reference zone for postprocessing relative velocities and related quantities. See Setting the Reference Zone (p. 4161) for details.

51.11. Solution Task Page

The **Solution** task page introduces you to the main tasks involved in solving your CFD simulation using Ansys Fluent. For more information about using Ansys Fluent to solve your CFD simulation, see Using the Solver (p. 3571).

Solution

The task pages accessed under Solution allow you to perform solution setup and calculation tasks. The Solving tab in the tabbed UI above provides access the same pages.

Note that solution adaption is available in the Setting Up Domain->Adapt tab above, and creation of surfaces for monitoring during the calculation is available in the Setting Up Domain->Surface tab above.

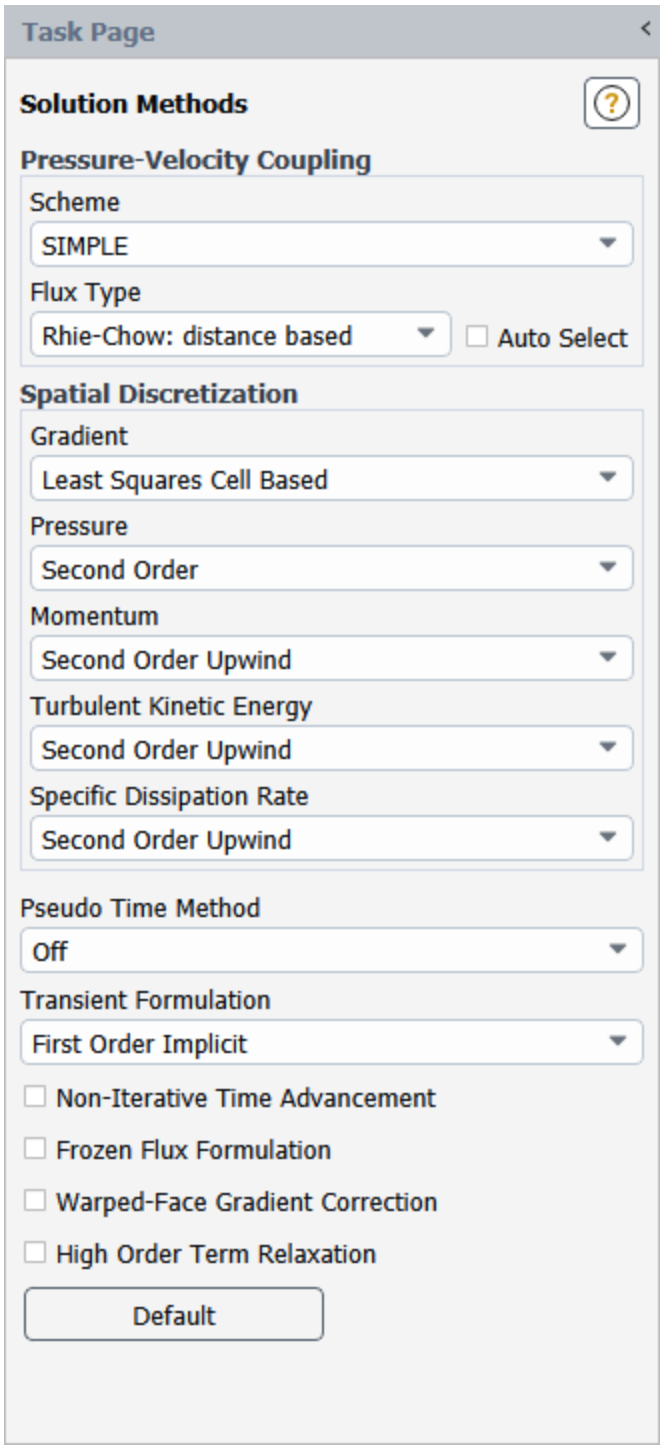
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

51.12. Solution Methods Task Page

The **Solution Methods** task page allows you to specify various parameters associated with the solution method to be used in the calculation.

Important:

This feature offers reduced functionality when running Fluent under the **Pro** capability level.



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Controls

Formulation

provides a drop-down list of the available types of solver formulations: **Implicit** and **Explicit**.

This item appears only when the density-based solver is used.

Flux Type

(for the density-based solver) provides a drop-down list of the convective flux types: **Roe-FDS**, **AUSM**, and **Low Diffusion Roe-FDS**. Details about each of the flux types can be found in Convective Fluxes in the Theory Guide.

Pressure-Velocity Coupling

contains settings for pressure-velocity coupling schemes.

Scheme

(for the pressure-based solver only) provides a drop-down list of the available pressure-velocity coupling schemes: **SIMPLE**, **SIMPLEC**, **PISO**, and **Coupled**. When the non-iterative time advancement (NITA) scheme is enabled in the **Solution Methods** task page, **Fractional Step** is available for single phase flows, and **Phase Coupled SIMPLE** is available for Eulerian multiphase flows. See Pressure-Velocity Coupling in the *Fluent Theory Guide*, and Choosing the Pressure-Velocity Coupling Method (p. 3585), Setting Solution Controls for the Non-Iterative Solver (p. 3592), and Selecting the Pressure-Velocity Coupling Method (p. 3187) for details about these methods.

Skewness Correction

enables the SIMPLEC and PISO skewness correction for highly skewed meshes if the value (number of iterations) is greater than 0. The default value is 0 for SIMPLEC and 1 for PISO.

Neighbor Correction

enables the PISO neighbor correction, which is recommended for transient calculations, if the value (number of iterations) is greater than 0. The default value is 1.

Skewness-Neighbor Coupling

allows for a more economical but a less robust variation of the PISO algorithm.

Coupled with Volume Fraction

couples velocity corrections, shared pressure corrections, and the correction for volume fraction simultaneously. This option is available in the interface after you have selected **Coupled** from the **Scheme** drop-down list for **Pressure-Velocity Coupling**.

Solve N-Phase Volume Fraction Equations

solves all primary and secondary volume fraction equations and scales the resulting volume fractions to satisfy the requirement that they sum to 1. If this is disabled, only the secondary phase volume fractions are solved directly and the primary phase volume fraction is set such that the volume fractions sum to 1.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Flux Type

(for the pressure-based solver) provides a drop-down list of the mass flux types: **Rhie-Chow: distance based** and **Rhie-Chow: momentum based**. Details about each of the flux types can be found in Discretization of the Continuity Equation in the Theory Guide. You cannot change the selection if **Auto Select** is enabled. For multiphase models other than the wet steam model, this list is not available and the **Rhie-Chow: momentum based** flux option is used.

Auto Select

enables the automatic selection of the **Flux Type**, so that the most robust formulation is used for the flow models and mesh attributes of your case (for details, see Mass Flux Types (p. 3588)). This option is enabled by default.

Spatial Discretization

contains settings that control the spatial discretization of the convection terms in the solution equations. See Choosing the Spatial Discretization Scheme (p. 3575) for details.

Gradient

contains a drop-down list of the options for setting the method of computing the gradient in Equation 23.30 in the Theory Guide: **Green-Gauss Cell Based**; **Green-Gauss Node-Based**, and **Least Squares Cell Based**. See Evaluation of Gradients and Derivatives in the Theory Guide for details.

Pressure

(for the pressure-based solver only) contains a drop-down list of the discretization schemes available for the pressure equation: **Standard**, **PRESTO!**, **Linear**, **Second Order**, **Body Force Weighted**, and **Modified Body Force Weighted**.

This item appears only when the pressure-based solver is used.

Flow

(for the density-based solvers only) contains a drop-down list of the discretization schemes available for the pressure, momentum, and (if relevant) energy equations: **First Order Upwind**, **Second Order Upwind**, **Third-Order MUSCL**, and (if the LES, DES, SAS, SDES, or SBES turbulence model is enabled) **Bounded Central Differencing**.

This item appears only when one of the density-based solvers is used.

Momentum, Energy, and so on

are the names of the other convection-diffusion equations being solved. In the drop-down list next to each equation, you can select the **First Order Upwind**, **Second Order Upwind**, **QUICK**, or **Third-Order MUSCL** discretization scheme for that equation.

If the LES, DES, SAS, SDES, or SBES turbulence model is enabled, then you have a choice of selecting **Bounded Central Differencing** or **Central Differencing** for **Momentum**. If **Bounded Central Differencing** is selected, an entry field **BCD Scheme Boundedness** appears under the **Spatial Discretization** group box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

If one of the density-based solvers is used, **Momentum** and **Energy** will not appear. For the density-based solvers, the discretization scheme for these equations is selected in the **Flow** drop-down list (described above).

BCD Scheme Boundedness

specifies the boundedness strength of the BCD scheme as a constant or an expression. In the latter case, the field variable **BCD Scheme Boundedness** is created in the **Derivatives** category and can be visualized.

Volume Fraction

is available when the VOF multiphase model is enabled. The discretization schemes that are used when solving volume fraction equations for the VOF explicit scheme are **Geo-Reconstruct**, **Compressive**, **CICSAM**, **Modified HRIC**, and **QUICK**. The discretization schemes that are used when solving volume fraction equations for the VOF implicit scheme are **First Order Upwind**, **Second Order Upwind**, **Compressive**, **Modified HRIC**, **BGM** (steady state only), and **QUICK**. See Interpolation Near the Interface in the Theory Guide for detailed information about these VOF-specific interpolation schemes.

Pseudo Time Method

allows you to select a form of implicit under-relaxation (see Pseudo Time Method Under-Relaxation): the **Local Time Step** or **Global Time Step** method. This list is available for all calculations when using a pressure-based segregated solver (**Local Time Step**) or for steady-state calculations when using the pressure-based coupled solver or the density-based implicit solver (**Global Time Step**). For details on completing the setup for the pseudo time method, see Performing Calculations with a Pseudo Time Method (p. 3673).

Transient Formulation

contains options for setting different time-dependent solution formulations. This option appears only when **Transient** is enabled under **Time** in the **General** task page. Available options include: **Explicit** (available only for the **Density Based Explicit** solver); **First Order Implicit**; **Second Order Implicit**; and **Bounded Second Order Implicit**. See Performing Time-Dependent Calculations (p. 3643) for details.

Non-Iterative Time Advancement

enables non-iterative time-advancement (NITA) scheme. See Time-Advancement Algorithm in the Theory Guide for details.

Options... (NITA)

Opens the **NITA Options** dialog box where you can enable the **Hybrid Nita** option and adjust the relevant controls. See Hybrid NITA for the VOF and Mixture Multiphase Models (p. 3595) for a description of the available controls. This item is available only for the VOF and Mixture multiphase models when the **Non-Iterative Time Advancement** option is enabled.

Accelerated Time Marching

enables a modified NITA scheme and other setting changes that can speed up the simulation. This option is only available with the Large Eddy Simulation (LES) turbulence model, and is intended for

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

unreacting flow simulations that use a constant-density fluid. For details, see Setting Solution Controls for the Non-Iterative Solver (p. 3592).

Frozen Flux Formulation

enables an option to discretize the convective part of Equation 23.68 in the Theory Guide using the mass flux at the cell faces from the previous time level n. This option is available only for a **Transient** solution. See The Frozen Flux Formulation in the Theory Guide for details.

Warped-Face Gradient Correction

enables an adjustment to the gradient discretization method that improves the gradient accuracy for meshes containing high aspect ratios, cells with non-flat faces, and highly deformed cells, and can help avoid numerical difficulties in simulations that involve a mesh that has large differences in the volumes of neighboring cells. By default, the fast, or more computationally efficient mode of the warped-face gradient correction is used. For additional information about the warped-face gradient correction, see Warped-Face Gradient Correction (p. 3745).

Convergence Acceleration For Stretched Meshes

enable convergence acceleration for stretched meshes to improve the convergence of the implicit density based solver on meshes with high cell stretching (see Convergence Acceleration for Stretched Meshes (CASM) (p. 3605)).

High-Speed Numerics

enables built-in, customized numeric settings that can help to stabilize and accelerate the convergence for high-speed flows. (see Enabling High-Speed Numerics (p. 3608)).

High Order Term Relaxation

enables the relaxation of high order terms to aid in the solution behavior of flow simulations when higher order spatial discretization is used.

Options...

opens the Relaxation Options Dialog Box (p. 5162).

Turbomachinery-Specific Numerics

enables built-in customized numerics settings that can help to achieve and accelerate the convergence for turbomachinery flows. This option is only available when the **Turbo Models** option is enabled in the **Domain** ribbon tab (**Turbomachinery** group) for a steady calculation with the pressure-based solver selected and the **Pseudo Time Method** set to the **Global Time Step** option. For further details, see Turbomachinery-Specific Numerics (p. 2027).

Set All Species Discretizations Together

enables an option to use the same discretization scheme for all the species rather than setting each of the species individually. Notice that you will no longer see your list of individual species, instead a **Species** field will appear with the scheme of your choice. Note that this option is available when species transport is enabled.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Default

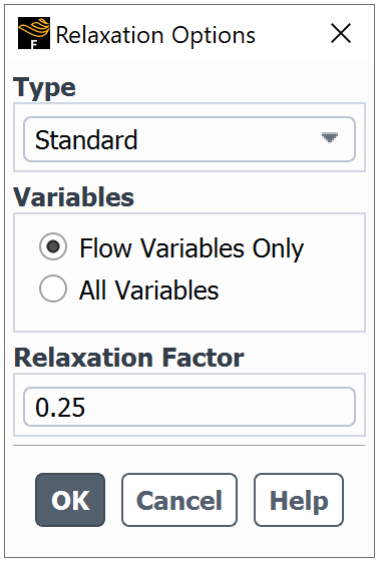
sets the fields to their default values, as assigned by Ansys Fluent.

Report Poor Quality Elements

reports statistics on the number and type of cells that Ansys Fluent has identified as having poor quality.

51.12.1. Relaxation Options Dialog Box

The **Relaxation Options** dialog box allows you to further control the **High Order Term Relaxation** (HOTR), as described in High Order Term Relaxation (HOTR) (p. 3578).



Controls

Type

allows you to select between **Standard** (which is the default selection and applies HOTR to both convection and diffusion terms) or **Convection Only** (which applies HOTR exclusively to the convection terms). The **Type** list is only available with the pressure-based solver.

Variables

allows you to select between under-relaxing **All Variables** or only the default flow variables (**Flow Variables Only**.

Relaxation Factor

is by default 0.25 for steady-state cases and 0.75 for transient cases.

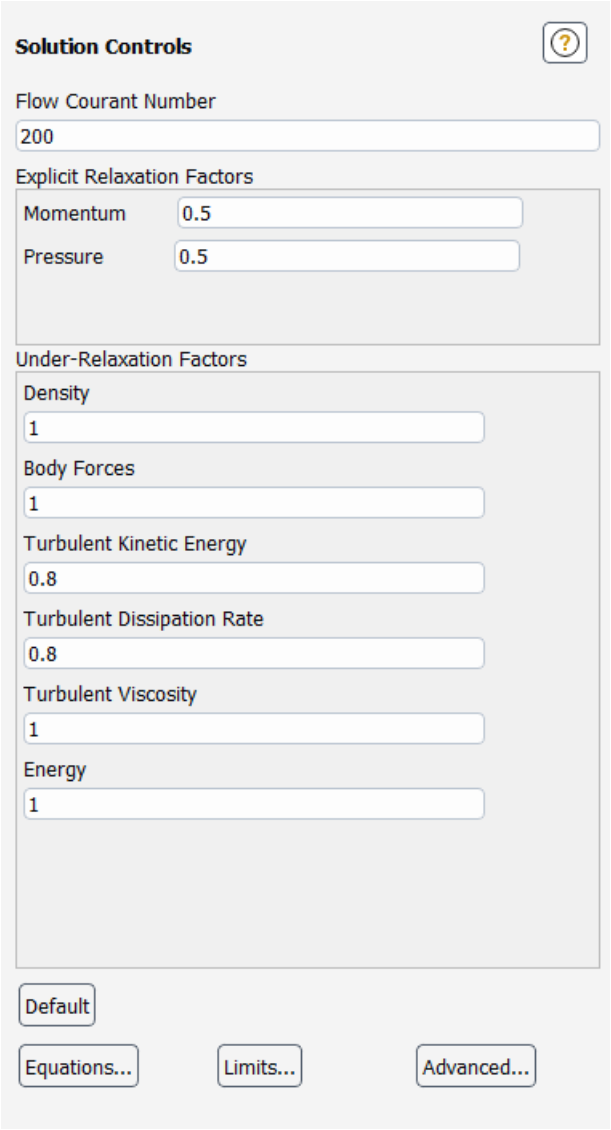
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

51.13. Solution Controls Task Page

The **Solution Controls** task page allows you to set common solution parameters.

Important:

This feature offers reduced functionality when running Fluent under the **Pro** capability level.



Controls

Courant Number

sets the fine-grid Courant number (time step factor) when the density-based solver is used. See Changing the Courant Number (p. 3602) for guidelines on setting the Courant number.

When the pressure-based solver is used for time-independent flows, and the **Coupled** pressure-velocity scheme is used, the **Courant Number** is used to stabilize the convergence behavior. See Under-

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Relaxation of Equations in the Theory Guide for a correlation of the under-relaxation factor and courant number.

Flow Courant Number

sets the Courant number for multiphase flow using the pressure-based coupled solver. The **Flow Courant Number** is used to stabilize the convergence behavior.

Volume Fraction Courant Number

sets the Courant number for multiphase flow using the pressure-based solver is coupled with the volume fraction. The default value is 200 and will appear in the interface when the **Coupled with Volume Fraction** option appears in the **Solution Methods** task page.

Pseudo Time Courant Number

sets the pseudo time Courant number when a **Pseudo Time Method** is selected with a pressure-based segregated solver (see Local Time Step Method Setting (p. 3675)).

Explicit Relaxation Factors

for the **Coupled** scheme defines the explicit relaxation of variables between sub-iterations for momentum and pressure. See Under-Relaxation of Variables in the Theory Guide for information.

Multigrid Levels

specifies the maximum number of coarse levels to be created by the FAS multigrid solver. This item is the same as the **Max Coarse Levels** under **FAS Multigrid Controls** in the **Multigrid** tab of the Advanced Solution Controls Dialog Box (p. 5168), and it appears only when the density-based explicit solver is used.

Residual Smoothing

contains parameters that govern the use of implicit residual smoothing. (See Implicit Residual Smoothing in the Theory Guide and Using Residual Smoothing to Increase the Courant Number (p. 3610) for details.) This section of the task page will appear only when the density-based explicit solver is used.

Iterations

sets the number of iterations of the Jacobi smoother to use. If **Iterations** is 0, then no implicit residual smoothing is performed.

Smoothing Factor

sets the implicit residual smoothing factor. This item will not appear unless **Iterations** is set to a non-zero value.

Under-Relaxation Factors

contains the under-relaxation factors for all equations that are being solved with the pressure-based solver. (See Setting Under-Relaxation Factors (p. 3589) for details.) In the field next to each equation, you can set the under-relaxation factor for that equation.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When one of the density-based solvers is used, **Under-Relaxation Factors** will appear only for the following variables, when they are included in your model: solid (for conjugate heat transfer models), turbulence variables, and viscosity. The solid under-relaxation factor is used to scale the CFL value in the solid region only if the under-relaxation value is set to a value less than 1. When the under-relaxation value is set to 1, then the solver ignores the specified CFL value for the solid and instead uses a very large value of CFL. This is done to speed up the convergence in the solid region. The density-based solvers use a segregated method to solve these equations; all the other equations are solved in a coupled manner, so there are no under-relaxation factors for them.

When **Non-Iterative Time Advancement** is selected for the pressure-based solver, the **Non-Iterative Solver Relaxation Factors** define the explicit relaxation (Under-Relaxation of Variables in the Theory Guide) of variables between sub-iterations and are used to prevent the solution from diverging.

Note the following if a **Pseudo Time Method** is selected: if **Global Time Step** is selected for steady-state cases with the pressure-based coupled solver or the density-based implicit solver, the **Pseudo Time Explicit Relaxation Factors** can be specified (see Global Time Step Method Settings (p. 3676)); if **Local Time Step** is selected for a pressure-based segregated solver, the **Pseudo Time Under-Relaxation Factors** can be specified (see Local Time Step Method Setting (p. 3675)).

Acoustics Wave Equation Solver Controls

contains parameters that govern the acoustics wave equation solver

Relative Convergence Criterion

sets the relative convergence criterion for the acoustics wave equation solver.

Max Iteration/Time Step

sets the maximum number of iterations per time step.

Default

sets the fields to their default values, as assigned by Ansys Fluent.

Equations...

displays the Equations Dialog Box (p. 5166).

Limits...

displays the Solution Limits Dialog Box (p. 5166).

Advanced...

displays the Advanced Solution Controls Dialog Box (p. 5168).

Set All Species URFs Together

enables an option to use the same under-relaxation factors for all the species rather than setting each of the species individually. Notice that you will no longer see your list of individual species, instead a **Species** field will appear with the specified under-relaxation factor. Note that this option is available when species transport is enabled.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

For additional information, see the following sections:

51.13.1. Equations Dialog Box

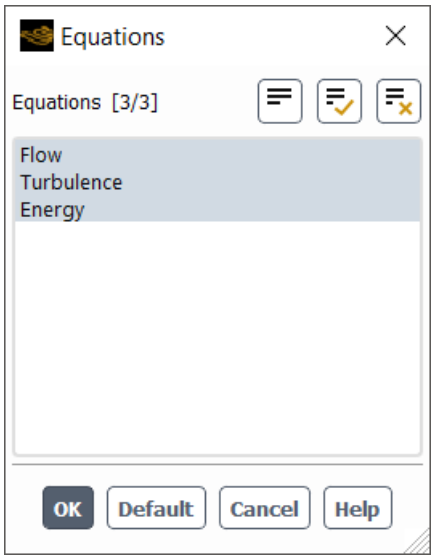
51.13.2. Solution Limits Dialog Box

51.13.3. Advanced Solution Controls Dialog Box

51.13.1. Equations Dialog Box

The **Equations** dialog box contains a list of the equations being solved for the current model. To temporarily disable solution of an equation, deselect it in this list and click **OK**. To re-enable the calculation for an equation, select it and click **OK**. See Step-by-Step Solution Processes (p. 3739) for details about using this feature in a step-by-step solution process.

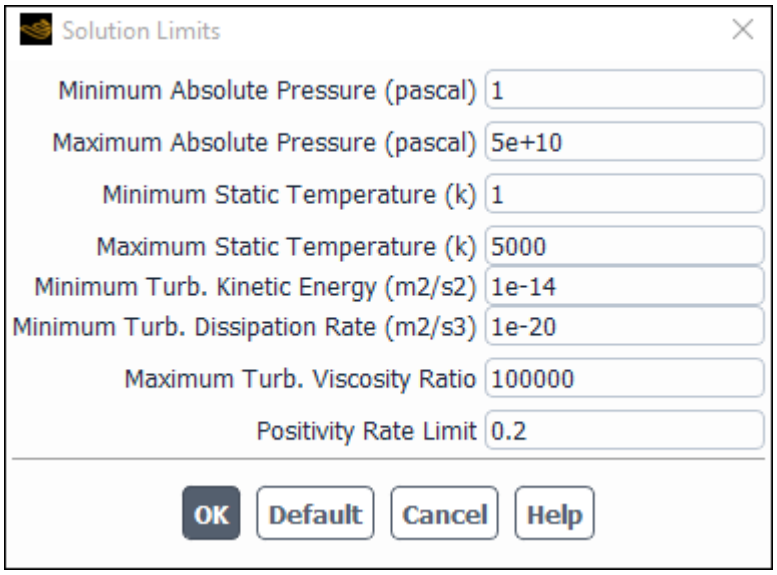
Note that, when one of the density-based solvers is used, **Energy** will not appear as a separate item in the **Equations** list. For the density-based solvers the energy equation is included in the **Flow** category (which also includes the pressure and momentum equations).



51.13.2. Solution Limits Dialog Box

The **Solution Limits** dialog box allows you to improve the stability of the solution. See Setting Solution Limits (p. 3621) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Minimum Absolute Pressure, Maximum Absolute Pressure

set the limiting minimum and maximum allowable values for absolute pressure.

Minimum Static Temperature, Maximum Static Temperature

set the limiting minimum and maximum allowable values for temperature.

Minimum Turb. Kinetic Energy

sets the limiting minimum value of turbulent kinetic energy (k) in the flow field. This parameter appears when one of the k- ε or k- ω models or the RSM is used.

Minimum Turb. Dissipation Rate

sets the limiting minimum value of turbulent dissipation rate (ε) in the flow field. This parameter appears when one of the k- ε models or the RSM is used.

Minimum Spec. Dissipation Rate

sets the limiting minimum value of specific dissipation rate (ω) in the flow field. This parameter appears when one of the k- ω models is used.

Maximum Turb. Viscosity Ratio

sets the limiting maximum allowable value of the ratio of turbulent to laminar viscosity (μ_t / μ). The turbulent viscosity is reduced to the necessary value so as not to exceed the maximum allowable viscosity ratio.

This parameter appears for all turbulent flows.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Minimum Vol. Frac. for Matrix Solution

sets the limiting minimum value for volume fraction in the matrix solution. This field appears for Eulerian multiphase simulations run on a double-precision solver.

Positivity Rate Limit

sets the limiting value for the rate of reduction of temperature. See Adjusting the Positivity Rate Limit (p. 3623) for details.

This item appears only when one of the density-based solvers is used.

Default

sets the fields to their default values, as assigned by Ansys Fluent. After execution, the **Default** button becomes the **Reset** button.

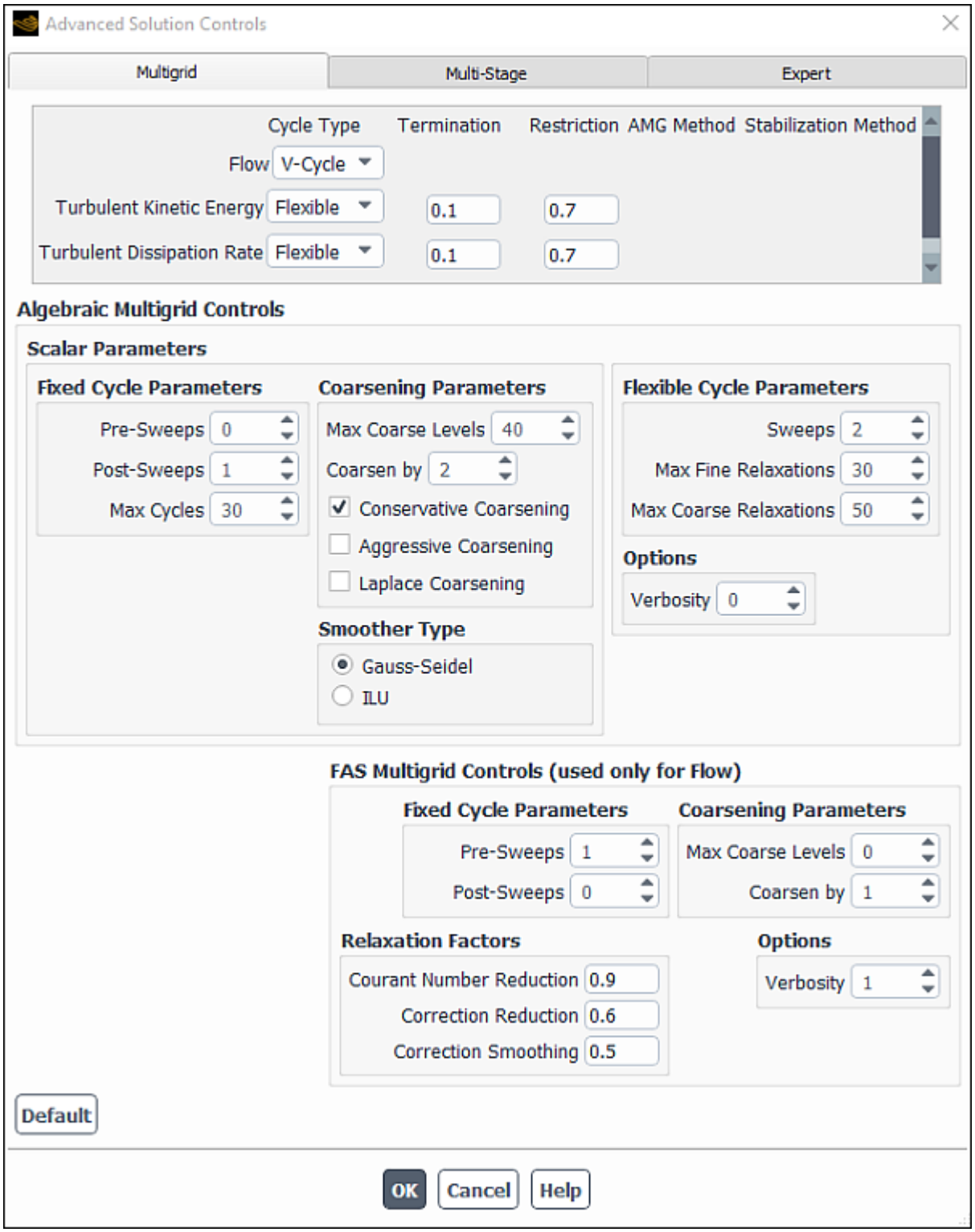
Reset

resets the fields to their most recently saved values (for example, the values before **Default** was selected). After execution, the **Reset** button becomes the **Default** button.

51.13.3. Advanced Solution Controls Dialog Box

The **Advanced Solution Controls** dialog box allows you to set parameters related to the multigrid, multi-stage, and non-iterative solvers.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Multigrid

tab contains parameters related to the multigrid solver. See Modifying Algebraic Multigrid Parameters (p. 3741) and Setting FAS Multigrid Parameters (p. 3618) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cycle Type

contains a drop-down list for each equation that is being solved. From this list you can select the multigrid cycle type (**Flexible**, **V-Cycle**, **W-Cycle**, or **F-Cycle**). See Specifying the Multigrid Cycle Type (p. 3613) for details.

Note that, for the density-based solvers, the **Pressure**, **Momentum**, and **Energy** equations will not appear individually. They will instead be grouped together and called **Flow**.

Furthermore, for the density-based explicit solver, the **Cycle Type** choices for the **Flow** equations will be limited to **V-Cycle** and **W-Cycle**, while the choices for **Flow** for the density-based implicit solver will be limited to **V-Cycle** and **F-Cycle**.

Termination

specifies the termination criterion for each equation that is being solved using algebraic multigrid. See Setting the Termination and Residual Reduction Parameters (p. 3613) for details.

Restriction

specifies the residual reduction criterion for each equation that is being solved using the **Flexible** algebraic multigrid cycle. See Setting the Termination and Residual Reduction Parameters (p. 3613) for details. (This item will not appear for an equation that is using a **V-Cycle**, **W-Cycle**, or **F-Cycle**.)

Stabilization Method

contains the drop-down list to choose the stabilization method. For further details, see Setting the Stabilization Method (p. 3613).

BCGSTAB

enables the bi-conjugate gradient stabilized method.

RPM

enables the recursive projection method. RPM stabilization is mainly used in conjunction with the coupled pressure-based solver.

CG

enables the conjugate gradient method. CG stabilization is mainly used in conjunction with the segregated pressure-based solver.

GMRES

enables the generalized minimal residual method. GMRES stabilization is recommended for cases that fail to converge with the BCGSTAB method.

Algebraic Multigrid Controls

contains parameters related to the algebraic multigrid solver. See Algebraic Multigrid (AMG) in the Theory Guide and Modifying Algebraic Multigrid Parameters (p. 3741) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Scalar and Coupled Parameters

contain parameters that you can set, If you are using the density-based explicit solver or the pressure-based solver with any of the segregated schemes, described in Pressure-Velocity Coupling in the Theory Guide and Choosing the Pressure-Velocity Coupling Method (p. 3585), you will only set **Scalar Parameters**. If you are using the density-based implicit or the pressure-based coupled scheme, described in Coupled Algorithm, then you can set the **Coupled Parameters**.

Fixed Cycle Parameters

contains parameters that control the V, W, and F cycles. You can set the number of **Pre-Sweeps** and **Post-Sweeps**, and the **Max Cycles**. Normally one post-sweep is performed and no pre-sweeps are done. See Fixed Cycle Parameters (p. 3615) for details about using the items below.

Pre-Sweeps

sets the number of sweeps to perform before moving to a coarser level.

Post-Sweeps

sets the number of sweeps to perform after coarser level corrections have been applied.

Max Cycles

sets the maximum number of V, W, or F cycles to be performed. The multigrid solver will continue to solve the set of equations until either the maximum number of cycles has been performed, or the **Termination** criteria are satisfied.

Coarsening Parameters

contains parameters that control the grouping of equations in the algebraic multigrid algorithm. See Coarsening Parameters (p. 3615) for details

Max Coarse Levels

is the maximum number of coarse levels that will be built by the multigrid solver. Sets of coarser simultaneous equations are built until the maximum number of levels has been created, or the coarsest level has only 3 equations. Each level has about half as many unknowns as the previous level, so coarsening until there are only a few nodes left will require about as much total coarse-level coefficient storage as was required on the fine mesh. Reducing the maximum coarse levels will reduce the memory requirements, but may require more iterations to achieve a converged solution. Setting **Max Coarse Levels** to 0 turns off the multigrid solver.

Coarsen by

controls the number of equations on each successively coarser grid level. By default, this parameter is set to 2, indicating that the number of equations on each level will be 1/2 of the number on the previous level. In general, the number of equations

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

on each coarser grid level will be approximately equal to 1/ n of the number on the previous level, where n is the value set for the **Coarsen by** parameter.

Conservative Coarsening

enables the use of conservative coarsening techniques that can improve parallel performance and/or convergence for some difficult cases.

Aggressive Coarsening

specifies the use of a version of the AMG solver that is optimized for high coarsening rates. It is recommended if the AMG solver diverges with the default settings.

Laplace Coarsening

enables the use of Laplace coefficients when grouping cells for coarsening.

Smoother Type

consist of two types:

Gauss-Seidel

is the simplest smoother type and is recommended when using the pressure-based segregated solver.

ILU

is more CPU intensive, but has better smoothing properties for block-coupled systems such as the pressure-based coupled solver and the density-based implicit solver.

Flexible Cycle Parameters

contains parameters that control the flexible multigrid cycle.

Sweeps

specifies the number of times to apply the smoothing method each time a relaxation is performed.

Max Fine Relaxations

sets the maximum number of relaxations to be performed on the Level 1 grid (fine grid level). This parameter eliminates the possibility that the Gauss-Seidel solver will get "stuck" on the fine grid level, unable to reduce the residuals by the fraction (α) required by the **Termination** criteria.

Max Coarse Relaxations

sets the maximum number of relaxations to be performed on each grid level above Level 1 (that is, the coarse grid levels). This parameter eliminates the possibility that the Gauss-Seidel solver will get "stuck" on a coarse grid level, unable to reduce the residuals on that level by the fraction (α) required by the **Termination** criteria. If the

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

iterative solution on a given grid level is unable to meet the accuracy constraint of the **Termination** criteria, the correction equation will be deemed "converged" when this maximum number of relaxations on that grid level has been performed.

Options

contains additional multigrid parameters.

Verbosity

controls the amount of information that is printed out by the multigrid solver for monitoring purposes. See Setting the Verbosity (p. 3617) for details

FAS Multigrid Controls

contains parameters related to the FAS multigrid solver. See Full-Approximation Storage (FAS) Multigrid in the Theory Guide and Setting FAS Multigrid Parameters (p. 3618) for details. As noted in the title of this dialog box section, the FAS multigrid solver is used only for the **Flow** equations (pressure, momentum, and energy).

This section of the dialog box appears only when the density-based explicit solver is used.

Fixed Cycle Parameters

contains parameters that control the V, W, and F cycles of the FAS multigrid solver.

Pre-Sweeps

sets the number of iterations of the multi-stage solver to be performed on a given grid level before proceeding to a coarser grid level (the value of β_1 described in Multigrid Cycles in the Theory Guide). Typically, this is set to 1.

Post-Sweeps

sets the number of multigrid cycles to be performed on a given grid level before proceeding back up to the finer grid level (the value of β_2 described in Multigrid Cycles in the Theory Guide). A value of 1 results in V-cycle multigrid, and a value of 2 results in W-cycle multigrid.

Coarsening Parameters

contains parameters that control the grouping of cells in the FAS multigrid algorithm.

Max Coarse Levels

sets the maximum number of grid levels to be used in the multigrid process. A value of 0 disables multigrid (no coarse grid levels). If the coarse grids do not already exist, they are created automatically when you start iterating; you cannot create them by clicking the **OK** button. See Turning On FAS Multigrid (p. 3609) for details.

Coarsen by

controls the number of cells in each successively coarser grid level. By default, this parameter is set to 2, indicating that the number of cells on each level will be 1/2 of

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

the number on the previous level. In general, the number of cells on each coarser grid level will be equal to 1/ n of the number on the previous level, where n is the value set for the **Coarsen by** parameter.

Relaxation Factors

are provided to moderate and stabilize the multigrid corrections.

Courant Number Reduction

sets the factor by which to reduce the Courant number for coarse grid levels (that is, every level except the finest). Some reduction of time step (such as the default 0.9) is typically required because the stability limit cannot be determined as precisely on the irregularly shaped coarser grid cells.

Correction Reduction

sets the factor by which to reduce the magnitude of the multigrid corrections transferred from one level to the next finer level. A typical value with $\beta_1 = 1$ is 0.6. If two **Pre-Sweeps** and two **Post-Sweeps** are performed, this value can often be increased to 1.0 (that is, full correction transfer).

Species Correction Reduction

sets the factor by which to reduce the magnitude of the species corrections to stabilize the multigrid calculation. This item appears only when species transport is being modeled.

Correction Smoothing

sets the correction smoothing factor used to interpolate corrections from a coarser grid to a finer grid. For multigrid on structured meshes, corrections can be interpolated up to a finer mesh "smoothly" by using, for example, tri-linear interpolation. For unstructured meshes there is no analogous simple, algebraic procedure that can be used to interpolate without introducing substantial high frequency "noise". Instead, the corrections are first interpolated, and then subjected to a smoothing pass. The default **Correction Smoothing** value of 0.5 should be acceptable for all cases; you should not need to change it.

Options

contains additional multigrid parameters.

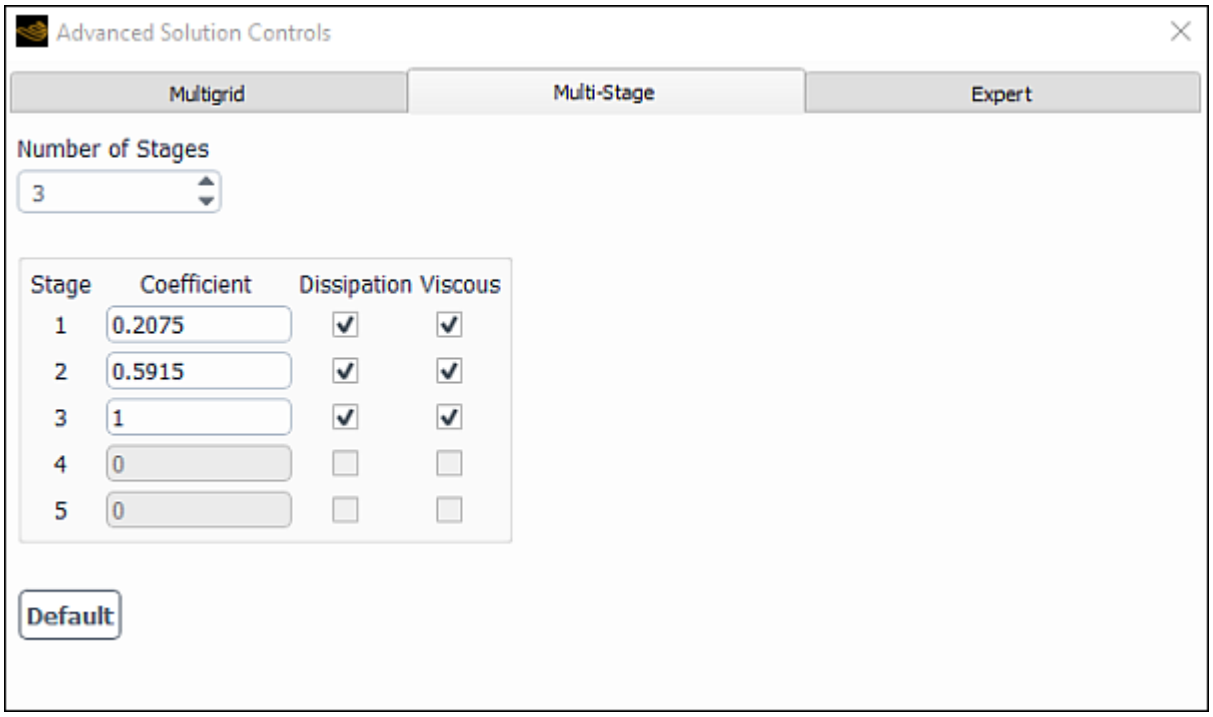
Verbosity

controls the amount of information that is printed out by the multigrid solver for monitoring purposes.

Multi-Stage

tab allows you to set parameters that govern the operation of the multi-stage solver. It is available only when the density-based explicit solver is used. See Changing the Multi-Stage Scheme (p. 3624) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Number of Stages

is the number of stages used in the multi-stage scheme. The default scheme is a 3-stage scheme with coefficients of 0.2075, 0.5915, and 1.0 for the first through third stages, respectively. Although the dialog box limits the maximum number of stages to five, you can define a scheme with an arbitrary number of stages with the `solve/set/multi-stage` text command.

Stage

labels the stage to which the parameters in the other columns apply.

Coefficient

sets the multi-stage coefficient for each stage. Coefficients should be greater than zero and less than one. The final stage should always have a coefficient of 1.

Dissipation

sets the stages for which artificial dissipation is evaluated. If a **Dissipation** box is selected for a particular stage, artificial dissipation will be updated on that stage. If not selected, artificial dissipation will remain "frozen" at the value of the previous stage.

Viscous

sets the stages for which viscous stresses are evaluated. If a **Viscous** box is selected for a particular stage, viscous stresses will be updated on that stage. If not selected, viscous stresses will remain "frozen" at the value of the previous stage. Viscous stresses should always be computed on the first stage, and successive evaluations will increase the "robustness" of the solution process, but will also increase the expense (that is, increase the

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CPU time per iteration). For steady problems, the final solution is independent of the stages on which viscous stresses are updated.

Default

sets the fields to their default values, as assigned by Ansys Fluent. After execution, the **Default** button becomes the **Reset** button.

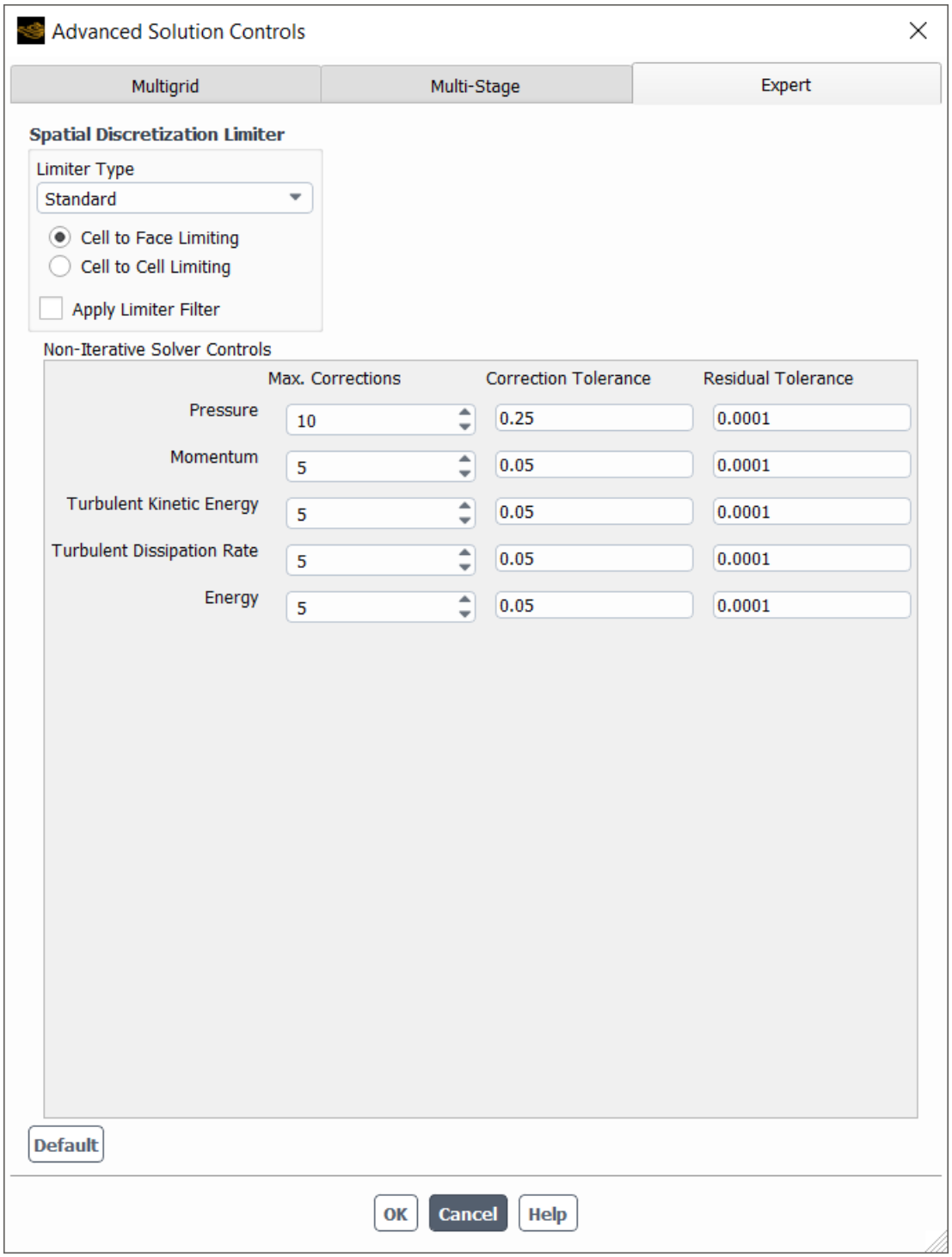
Reset

resets the fields to their most recently saved values (that is, the values before **Default** was selected). After execution, the **Reset** button becomes the **Default** button.

Expert

tab contains specialized parameters for limiting spatial discretization, as well as controls for the non-iterative solver.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Spatial Discretization Limiter

contains controls for limiting the spatial discretization. See Selecting Gradient Limiters (p. 3626) for more information about limiters.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Limiter Type

selects the type of limiter applied to the spatial discretization: **Standard**, **Multidimensional**, or **Differentiable**.

Cell to Face Limiting

is where the limited value of the reconstruction gradient is determined at the cell face centers. This is the default method.

Cell to Cell Limiting

is where the limited value of the reconstruction gradient is determined along a scaled line between two adjacent cell centroids. On an orthogonal mesh (or when the cell-to-cell direction is parallel to face area direction), this method becomes equivalent to the default cell to face method. For smooth field variation, cell to cell limiting may provide less numerical dissipation on meshes with skewed cells.

Apply Limiter Filter

enables the limiter filter. The limiter filter is only available with the **Standard** and **Differentiable** limiter types.

Pseudo Time Method Usage

allows you to select and modify the parameters for each individual equation. See Advanced Solution Controls for the Pseudo Time Method (p. 3679) for details.

On/Off

enables/disables the equation-specific steady-state solution method for a particular equation.

Under-Relaxation Factor

allows you to use the standard steady-state method by turning off the pseudo time method for that particular equation. Specify the corresponding under-relaxation factor to be employed with a particular equation.

Time Scale Factor

allows you to specify a factor that scales the pseudo time step size employed for the flow equations specified in the **Run Calculation** task page. A time scale factor other than 1.0 (default) allows the use of an equation specific time step size in lieu of using a uniform global pseudo time step size.

Non-Iterative Solver Controls

contain parameters that control the sub-iterations for the individual equations. See Time-Advancement Algorithm in the Theory Guide for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Max. Corrections

provide control over the maximum number of sub-iterations for each individual equation.

Correction Tolerance

defines the overall accuracy.

Residual Tolerance

controls the solution of the linear equations.

Default

sets the fields to their default values, as assigned by Ansys Fluent. After execution, the **Default** button becomes the **Reset** button.

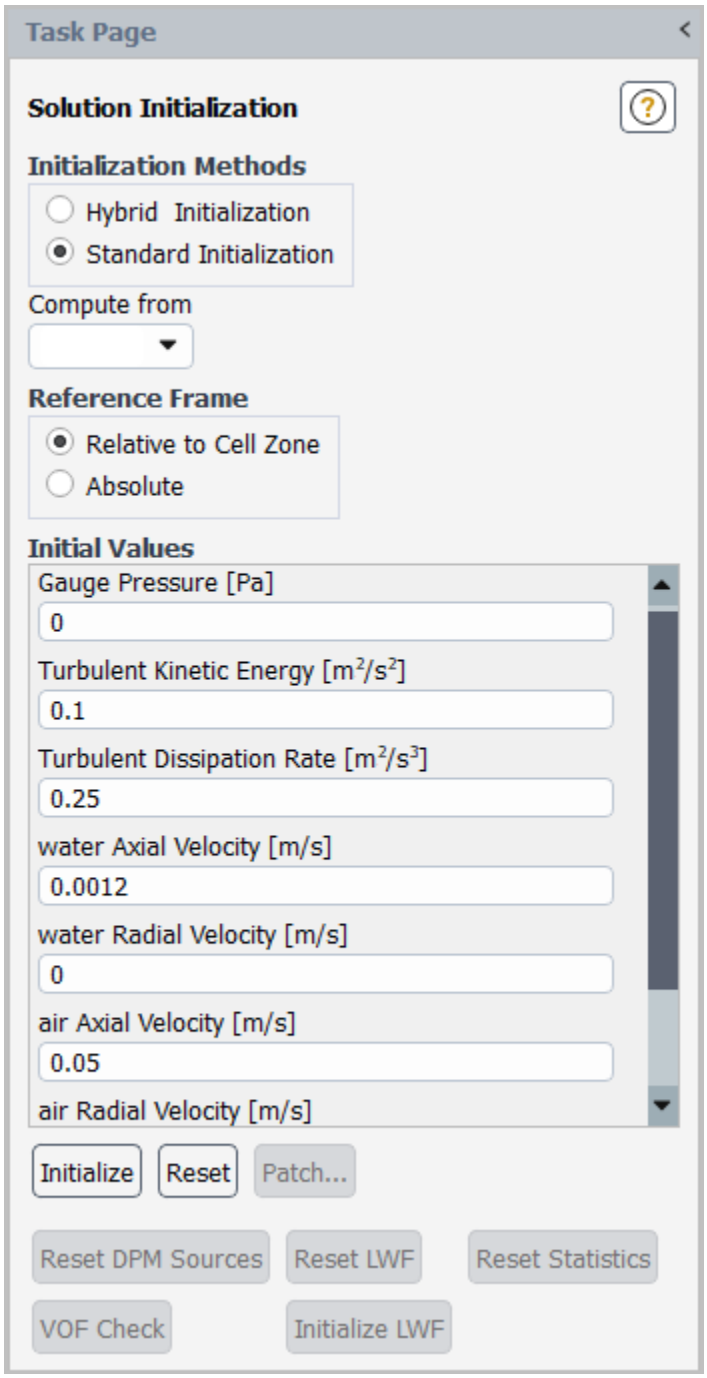
Reset

resets the fields to their most recently saved values (that is, the values before **Default** was selected). After execution, the **Reset** button becomes the **Default** button.

51.14. Solution Initialization Task Page

The **Solution Initialization** task page allows you to define values for flow variables and initialize the flow field to these values. See for details about using this dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Initialization Method

allows you to choose between **Hybrid Initialization** and **Standard Initialization**.

Hybrid Initialization

is a collection of boundary interpolation methods, where variables, such as temperature, turbulence, species fractions, volume fractions, and so on, are automatically patched based on domain averaged values or a particular interpolation recipe (see Hybrid Initialization (p. 3635)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Standard Initialization

allows you to define values for flow variables and initialize the flow field to these values.

Compute from

is a drop-down list of zones; the default values for applicable variables will be computed from information contained in the zone that you select from this list. The computation will occur when you select the required zone, and the variable values will be displayed in **Initial Values**. You can also choose the **all-zones** item in this list to compute average values based on all zones.

Reference Frame

indicates whether the initial velocities are absolute velocities (**Absolute**) or velocities relative to the motion of each cell zone (**Relative to Cell Zone**). This selection is necessary only if your problem involves moving reference frames or sliding meshes. If there is no zone motion, both options are equivalent.

Initial Values

displays the initial values of applicable variables. You can use **Compute from** to compute values from a particular zone, or you can enter values directly. For species transport cases with a single phase flow, only the boundary species that you have selected in the **Select Boundary Species** dialog box (see Figure 19.3: The Select Boundary Species Dialog Box (p. 2367)) appear in the list. You can add or remove a species using the **Select Boundary Species** dialog box that can be accessed by clicking **Species** in the **Initialization** task page.

Initialize

initializes the entire flow field to the values listed.

Initialize Acoustics...

opens the Acoustics Initialization Dialog Box (p. 5182).

Reset

resets the fields to their "saved" values.

Patch...

opens the Patch Dialog Box (p. 5183).

FMG...

opens the Full-Multigrid (FMG) Initialization Dialog Box (p. 5185).

Species

opens the **Select Boundary Species** dialog box (see Figure 19.3: The Select Boundary Species Dialog Box (p. 2367)), where you can select the boundary species to be displayed in the **Initial Values** list and in the **Patch**, **Hybrid Initialization**, and relevant boundary conditions dialog boxes. You can modify the selected boundary species list on the fly during the problem setup. This item appears for species transport cases with a single phase flow only.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reset DPM Sources

allows you to reset the interphase sources/sinks of momentum, heat, and/or mass to zero. This item is available when you perform coupled discrete phase calculations. See Resetting the Interphase Exchange Terms (p. 2832) for details.

Reset LWF

deletes the Lagrangian wall film (LWF) particles and zeroes out the corresponding face variables without affecting the free stream particles. See Patching the Wall Film (p. 2809) for details. This button is available only with the **wall-film** DPM boundary condition.

Reset Statistics

can be used to both initialize the flow statistics and reset the flow statistics after you have gathered some data for time statistics. This item is available when you perform unsteady calculations and have enabled the **Data Sampling for Time Statistics** option in the Run Calculation Task Page (p. 5207). See Inputs for Time-Dependent Problems (p. 3645) for details.

VOF Check

prints a summary report for the VOF case setup. This button is displayed only for VOF multiphase cases. It becomes available after the case is initialized. For details, see Multiphase Case Check (p. 3210).

Initialize LWF

initializes the Lagrangian wall film (LWF) on all walls with the **wall-film** DPM boundary condition. This button is available only when the LWF model is used.

More Settings...

opens the Hybrid Initialization Dialog Box (p. 5186). This button is available only when **Hybrid Initialization** is the selected method.

For additional information, see the following sections:

51.14.1. Acoustics Initialization Dialog Box

51.14.2. Patch Dialog Box

51.14.3. Full-Multigrid (FMG) Initialization Dialog Box

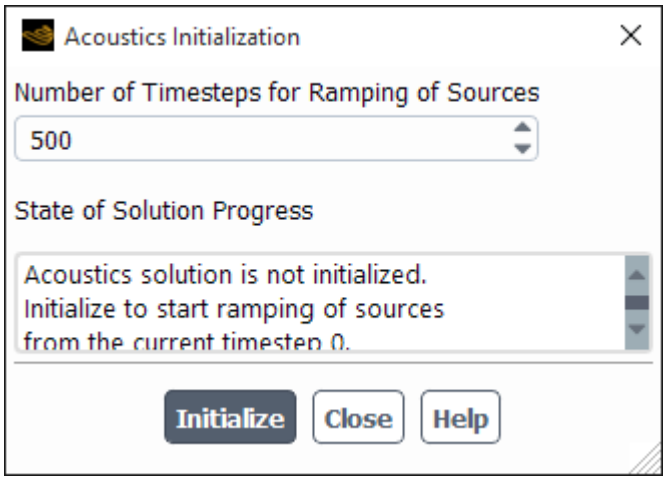
51.14.4. Hybrid Initialization Dialog Box

51.14.1. Acoustics Initialization Dialog Box

The acoustics initialization dialog box allows you to initialize/reinitialize the solution of the acoustics wave equation and to specify how long to ramp acoustic sources.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 51.4: The Acoustics Initialization Dialog Box



Controls

Number of Timesteps for Ramping of Sources

sets how many timesteps the sound sources will ramp to their full values.

51.14.2. Patch Dialog Box

The **Patch** dialog box allows you to patch different values of flow variables into different cells. See Patching Values in Selected Cells (p. 3630) for details about using this feature.

Controls

Reference Frame

indicates whether patched velocities are absolute velocities (**Absolute**) or velocities relative to the motion of each cell zone (**Relative to Cell Zone**). This selection is necessary only if your problem involves moving reference frames or sliding meshes. If there is no zone motion, both options are equivalent. Also, this selection affects only velocities, so the options will not be available unless you have selected a velocity in the **Variable** list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Phase

contains a list of all of the phases in the problem that you have defined. This is available when the VOF, mixture, or Eulerian multiphase model is enabled.

Variable

contains a list of flow variables from which you can choose the variable to be patched. For species transport cases with a single phase flow, only the boundary species that you have selected in the **Select Boundary Species** dialog box (see Figure 19.3: The Select Boundary Species Dialog Box (p. 2367)) appear in the list. You can add or remove a species using the **Select Boundary Species** dialog box that can be accessed by clicking **Species** in the **Initialization** task page.

Volume Fraction Patch Options

contains options that can be used when patching volume fraction for VOF and Eulerian multiphase with Multi-Fluid VOF simulations.

Patch Reconstructed Interface

applies a piecewise-linear reconstruction of the interface when patching a user-defined register and patches cells that intersect the interface with the actual volume fraction.

Volumetric Smoothing

enables smoothing of the volume fraction based on the volumetric average over the cell neighbors. After enabling volumetric smoothing, you can click Smooth to apply smoothing to the volume fraction field for all phases in the entire domain.

Smoothing Relaxation Factor

controls the degree of smoothing applied to the volume fraction field. This can be set between `0` (no smoothing) and `1` (maximum smoothing).

Value

sets a value to be patched for the selected **Variable** (constant, by default). Use the drop-down to the right of the field to select either **Constant** or **Expression** as the method for specifying the **Value**.

Use Field Function

enables the patching of a custom **Field Function**, rather than a constant **Value**, for the selected **Variable**. See Using Field Functions (p. 3632) for details.

Field Function

contains a list of defined custom field functions. If the **Use Field Function** option is enabled, you can patch one of these functions for the selected **Variable**. See Using Field Functions (p. 3632) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Zones to Patch

contains a list of cell zones. The specified **Value** or **Field Function** for the selected **Variable** will be patched into the zones you select from this list.

Registers to Patch

contains a list of cell registers that have been created using the adaption tools. You can patch a different value for a group of cells within a single cell zone by selecting a register containing a subset of the cells in the zone. See Using Registers (p. 3632) for details.

Patch

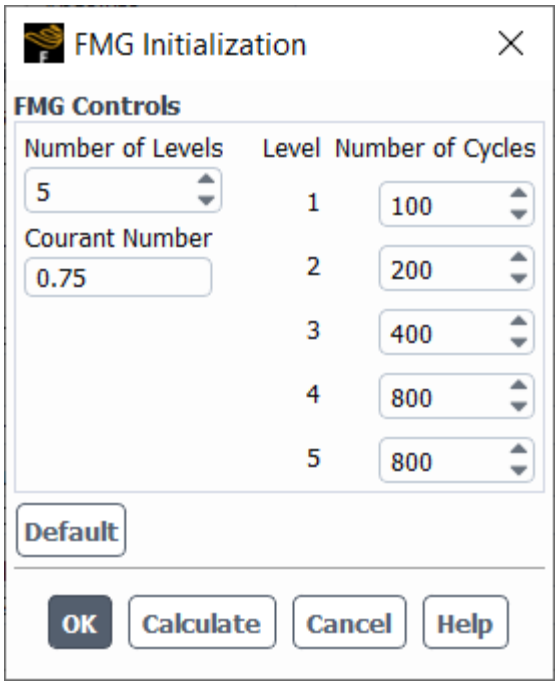
updates the flow-field data based on the inputs above.

Smooth

apply volumetric smoothing to the volume fraction field. This button only appears if **Volumetric Smoothing** is enabled.

51.14.3. Full-Multigrid (FMG) Initialization Dialog Box

The **FMG Initialization** dialog box contains the settings that control the Full-Multigrid Initialization strategy.



FMG Controls

Number of Levels

sets the number of multigrid levels for the FMG iteration (the default is 5).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Number of Cycles

sets the number of cycles per level. The defaults at each level are 100, 200, 400, 800, and 800.

Courant Number

sets the FMG iteration Courant-number (the default is 0.75). This will be the CFL value that the FAS multigrid will use for the FMG initialization.

FMG Options

contains additional FMG customization options supported by the density-based solver (see Additional FMG Initialization Options with the Density-Based Solver (p. 3634)).

Viscous Terms

adds the viscous terms of the Navier-Stokes equation to the FMG initialization procedure.

Species Reactions

adds the species-reactions terms to the FMG initialization procedure.

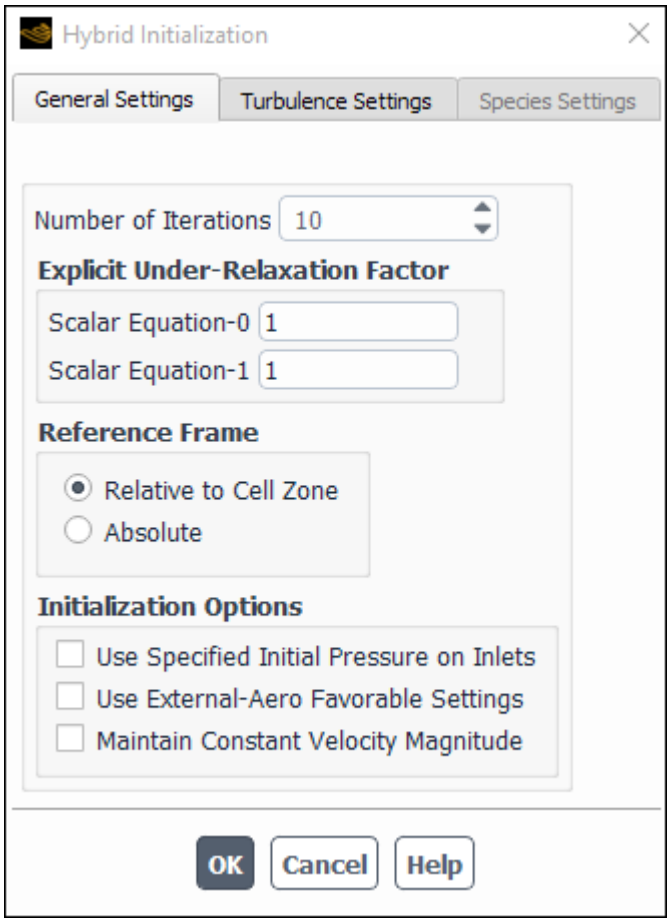
Default

returns the FMG controls to their default values.

51.14.4. Hybrid Initialization Dialog Box

The **Hybrid Initialization** dialog box contains a host of adjustable settings that control the **Hybrid Initialization** strategy.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

General Settings

allow you to adjust such entries as the number of iterations and under-relaxation factors.

Number of Iterations

uses a default value of 10. This is the number of iterations that will be performed while solving the Laplace equations to initialize the velocity and pressure. If the initialized velocity and pressure fields are not to your liking, you may want to increase the number of iterations and re-initialize the solution.

Explicit Under-Relaxation Factor

uses a default value of 1. This value will be used while solving the Laplace equation to initialize the velocity and pressure. If the initialized velocity and pressure fields are not to your liking, you may want to adjust the under-relaxation factor and re-initialize the solution.

Reference Frame

indicates whether the initial velocities are absolute velocities (**Absolute**) or velocities relative to the motion of each cell zone (**Relative to Cell Zone**). This selection is necessary only if your problem involves moving reference frames or sliding meshes. If there is no zone motion, both options are equivalent.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Initialization Options

allows you to include any of the three initialization options.

Use Specified Initial Pressure on Inlets

if you want the specified pressure for **Supersonic/Initialization Gauge Pressure** at the inlet boundaries to be used for solving the Laplace equation for the pressure.

Use External-Aero Favorable Settings

if you want to have the velocity potential patched with a linear value to help accelerate convergence of **Scalar Equation–0** and to obtain a better guess of the velocity field for external-aero problems.

Maintain Constant Velocity Magnitude

if you want to use the flow direction obtained from solving the velocity potential (**Scalar Equation–0**), while maintaining a constant velocity magnitude throughout the computational domain.

Turbulence Settings

uses by default the domain averaged values for the turbulence parameters. If you want to use variable turbulence parameters you can deselect the **Average Turbulent Parameters** check box. When this option is disabled, then it calculates the turbulent parameters, such as kinetic energy, dissipation energy, and so on, using local flow parameters.

Species Settings

will by default initialize secondary species with zero mass or mole fractions. If you want to specify the appropriate value for the species, you must enable **Specify Species Parameters**. Note that only the boundary species that you have selected in he **Select Boundary Species** dialog box (see Figure 19.3: The Select Boundary Species Dialog Box (p. 2367)) appear in the list. You can add or remove a species using the **Select Boundary Species** dialog box that can be accessed by clicking **Species** in the **Initialization** task page.

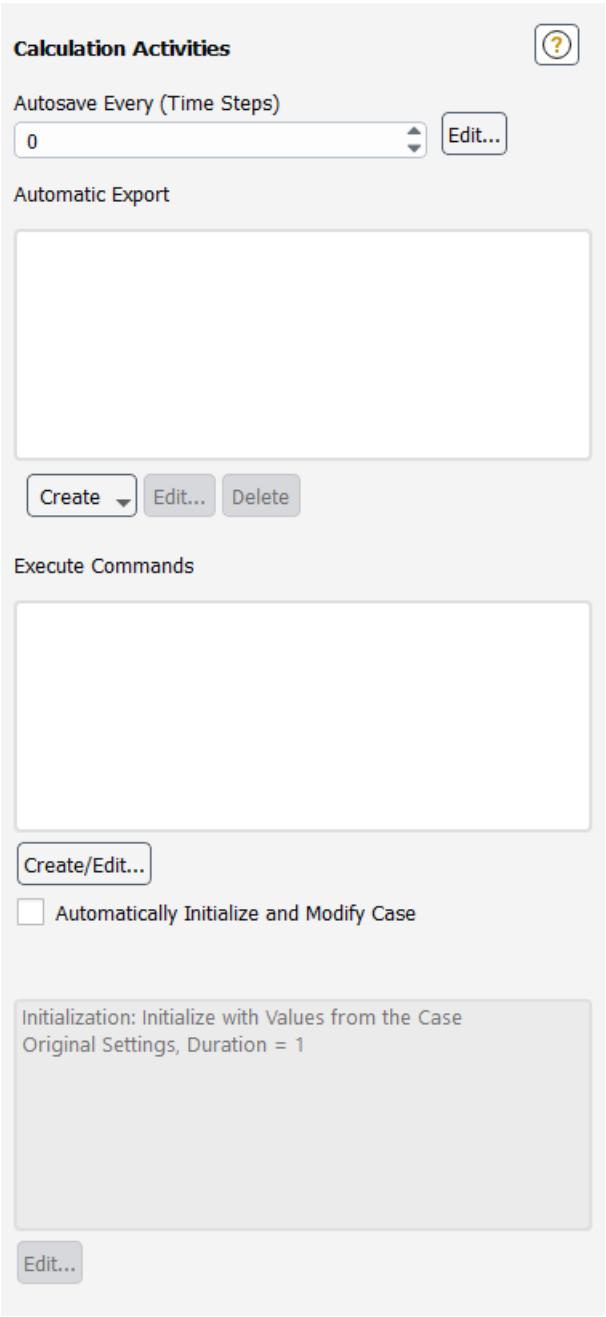
51.15. Calculation Activities Task Page

The **Calculation Activities** task page allows you to set up various tasks that you can perform during the calculation, such as saving files, exporting files, creating solution animations, and command execution.

Important:

This feature offers reduced functionality when running Fluent under the **Pro** capability level.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Autosave Every

allows you to set the frequency of the autosave.

Edit...

opens the Autosave Dialog Box (p. 5190).

Automatic Export

displays a list of available export objects that will be executed during the calculations. This list and its associated buttons are only available for transient flow calculations.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Create

provides a drop-down list that contains options for creating export objects. Two options are available: the **Solution Data Export** option opens the Automatic Export Dialog Box (p. 5193); the **Particle History Data Export** option opens the Automatic Particle History Data Export Dialog Box (p. 5198).

Edit...

opens the appropriate dialog box for the selected item in the **Automatic Export** list.

Delete

removes the selected item from the **Automatic Export** list.

Execute Commands

lists available commands to be executed during the calculations.

Create/Edit...

opens the Execute Commands Dialog Box (p. 5201).

Automatically Initialize Solution and Modify Case

allows you to automatically have your solution initialized and your case file modified.

Edit...

opens the Case Modification Manager Dialog Box (p. 5204). This button is only available when the **Automatically Initialize Solution and Modify Case** option is enabled.

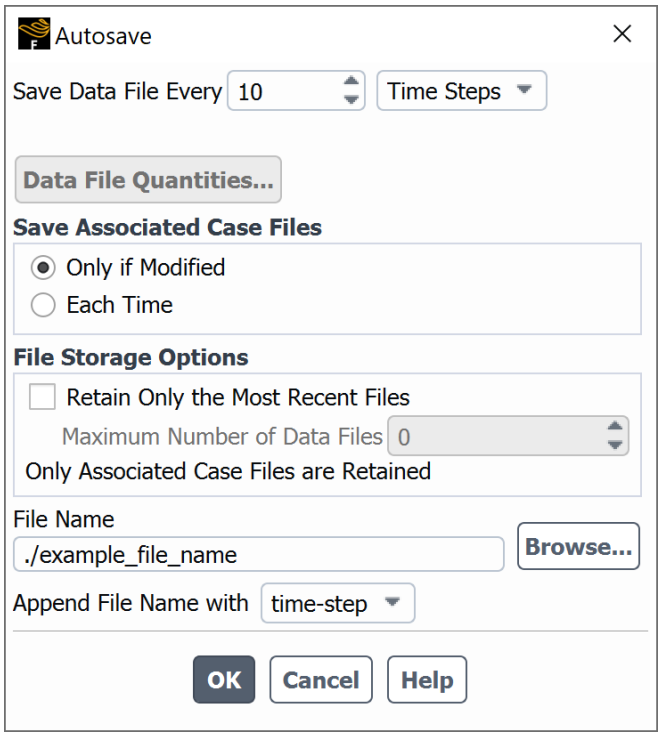
For additional information, see the following sections:

51.15.1. Autosave Dialog Box
51.15.2. Data File Quantities Dialog Box
51.15.3. Automatic Export Dialog Box
51.15.4. Automatic Particle History Data Export Dialog Box
51.15.5. Execute Commands Manager Dialog Box
51.15.6. Execute Commands Dialog Box
51.15.7. Define Macro Dialog Box
51.15.8. Case Modification Manager Dialog Box
51.15.9. Auto Initialization Method Dialog Box
51.15.10. Modify Case Dialog Box

51.15.1. Autosave Dialog Box

The **Autosave** dialog box allows you to specify automatic saving of case and data files at specified intervals during a calculation. See Automatic Saving of Case and Data Files (p. 958) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Save Data File Every

specifies the frequency with which data files are saved. For steady flows you will specify the frequency in iterations, while for unsteady flows you will specify it in either time steps or flow time (unless you are using the explicit time stepping formulation, in which case you will specify the frequency in iterations or if you are running an in-cylinder simulation you can specify the frequency in crank angles). The default value is set to zero, indicating that no automatic saving is performed.

Save Associated Case Files

gives you the choice to save the case file only if it is modified or each time that the data file is saved.

Only if Modified

results in Ansys Fluent saving a case file only if there is a change in the case file settings.

Each Time

allows you to save the case file every time the data file is saved.

Retain Only the Most Recent Files

allows you to restrict the number of files saved by Ansys Fluent if you have limited disk space. When this option is enabled, you can enter the appropriate value in the **Maximum Number of Data Files** field.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Only the associated case files are retained when using this option.

Important:

When the **Retain Only the Most Recent Files** option is selected, the solution history currently in memory will be discarded and the solution history reset.

Data File Quantities

opens the Data File Quantities Dialog Box (p. 5192) where you can specify which quantities you want to automatically save to a data file for postprocessing. This button is only available when you are writing in the legacy format (that is, `.dat` files).

Maximum Number of Data Files

specifies the maximum number of data files that can be saved at any instance. If you have constraints on the disk space, you can restrict the number of files to be saved using this field. After saving the specified number of files, Ansys Fluent will overwrite the earliest existing file. The default value for this field is zero, which saves all the files.

File Name

specifies the root name for the files that are saved. The iteration or time-step number and an appropriate suffix (`.cas.h5` or `.dat.h5`, by default) will be added to the specified root name. If the specified **File Name** ends in `.gz` or `.Z` and you have requested the reading and writing of files in the legacy format, appropriate file compression will be performed. (For details about file compression and the legacy format, see Reading and Writing Compressed Files (p. 950) and Reading and Writing Files in the Legacy Format (p. 958), respectively.)

The default path is `./<filename>` indicating that the autosaved files will be saved in the working directory of the current Fluent session. If you hard code a directory, then the files will always be saved at that location, regardless of the working directory of the current Fluent session.

Append File Name with

allows you to select **flow-time**, **time-step**, or **crank-angle** to be appended to the file name. This option is available only for unsteady-state calculations (only for in-cylinder calculations in the case of **crank-angle**). The default selection is **flow-time**.

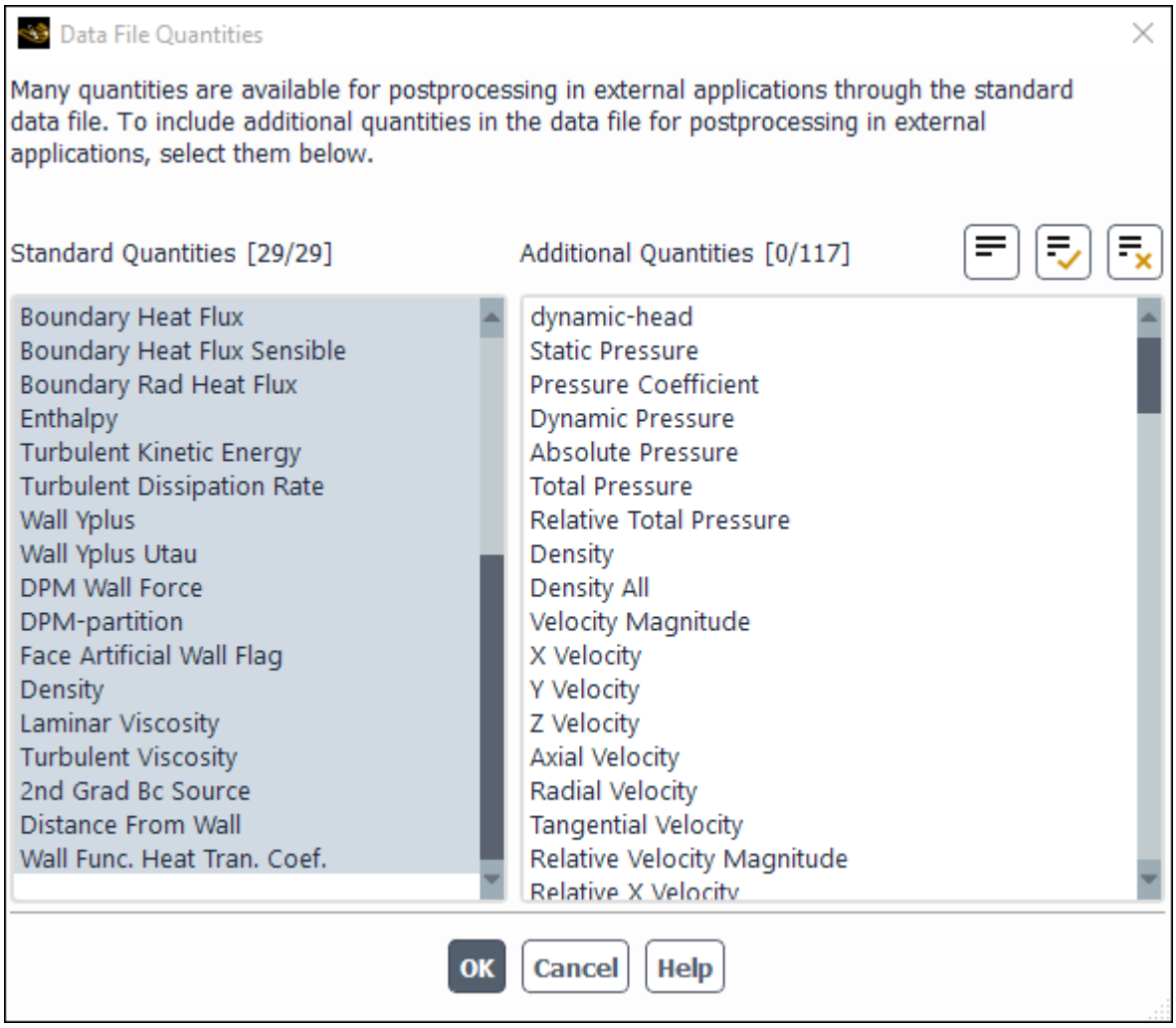
Decimal Places in File Name

allows you to specify the number of decimal digits in the file name. This option is available only when **flow-time** or **crank-angle** is selected in the **Append File Name with** drop-down list. The default value for this field is set to 6.

51.15.2. Data File Quantities Dialog Box

The **Data File Quantities** dialog box allows you to specify various quantities for postprocessing when you are writing in the legacy format (that is, `.dat` files). See Setting Data File Quantities (p. 1026) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Standard Quantities

contains a listing of standard postprocessing quantities (for example, density, Mach number, temperature, and so on).

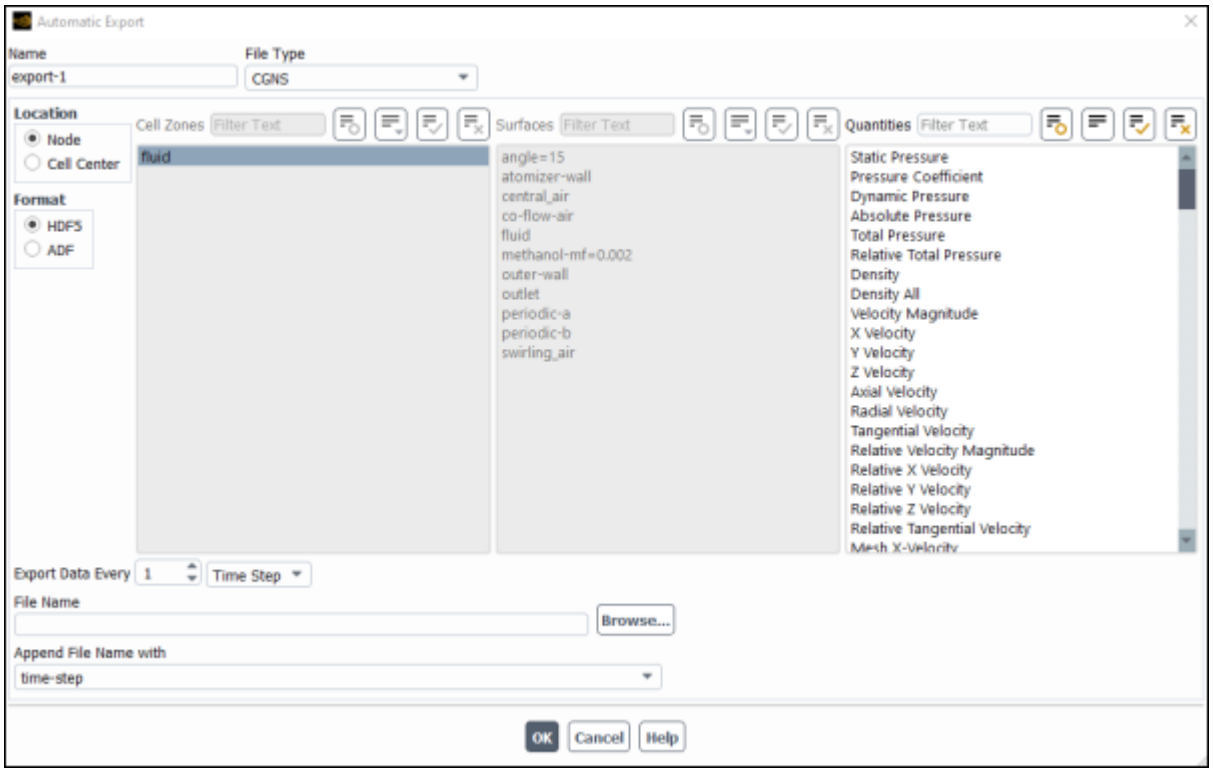
Additional Quantities

contains a listing of additional postprocessing quantities that are derived from the standard quantities (for example, standard pressure, velocity magnitude, and so on).

51.15.3. Automatic Export Dialog Box

The **Automatic Export** dialog box allows you to create an automatic export definition for solution data. See Creating Automatic Export Definitions for Solution Data (p. 1003) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name

specifies the name of the export definition.

File Type

contains a drop-down list of file types, which control the output file format that will be written.

ABAQUS

allows you to specify the surface(s) and optional loads, based on the kind of finite element analysis selected, to be exported to an ABAQUS file (extension `.inp`).

ASCII

allows you to specify the surface(s), scalars, location from which the values of scalar functions are to be taken, and the delimiter separating the fields, to be exported to an ASCII file.

AVS

allows you to specify the scalars you want to write to be exported to an AVS file.

CDAT for CFD-Post & EnSight

allows you to specify the cell zones, surfaces, quantities, and format (**Binary** or **ASCII**) used for data export. The `.cdat` file format is compatible with both CFD-Post and EnSight.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CGNS

allows you to specify the scalars you want to write and the location from which the values of scalar functions are to be taken, to be exported to a CGNS file (extension `.cgns`).

Common Fluids Format - Post

allows you to specify the cell zones, surfaces, and quantities that you want exported *(note that only nodal values are exported)*. The following files are written in the hierarchical data format (HDF) from the **Automatic Export** dialog box: `<filename>.cas.post` and `<filename>.dat.post`. When Fluent is set up to save files in the default Common Fluids Format (CFF), a project file named `<filename>.flprj` is also written as part of the automatic export; it is recommended that you read this file whenever possible when postprocessing in EnSight, as it contains metadata such as the time values for files exported from transient simulations. Note that if the root name of the exported Common Fluids Format - Post files is the same as that used for case and data files automatically saved during the calculation (through the **Autosave** dialog box), then the metadata from both operations will be written to the same project file.

Data Explorer

allows you to specify the surface(s) and the scalars you want to write to be exported to a Data Explorer file (extension `.dx`).

EnSight Case Gold

allows you to specify the scalars you want to write, the cell zones, interior zone surfaces, and location in the cell from which the values of scalar functions are to be taken, and the file format, to be exported to an EnSight file (extension `.geo`, `.vel`, `.scl1`, or `.encas`).

EnSight DVS

allows you to specify the cell zones or surfaces (not both) and the quantities that you want to write, and whether the data is written for nodes or cell centers. Data is exported in `.dvs` format for import to EnSight.

FAST

allows you to specify the scalars you want to write, to be exported as a grid file (Plot3D format), a velocity file, and a scalar file. This option is available only for a triangular or tetrahedral mesh.

FAST Solution

allows you to export a single file containing density, velocity, and total energy data. This option is available only for a triangular or tetrahedral mesh.

Fieldview Unstructured

allows you to specify the scalars you want to write and the cell zones from which the values of scalar functions are to be taken, to be exported to a FIELDVIEW binary file (extension `.fvuns`) and a regions file (extension `.fvuns.fvreg`).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

I-deas Universal

allows you to specify the surface(s), scalars, and optional loads, based on the kind of finite element analysis selected, to be exported to an I-deas Universal file.

Mechanical APDL Input

allows you to specify the surface(s) and optional loads, based on the kind of finite element analysis selected, to be exported to a Mechanical APDL Input file (extension `.cdb`).

NASTRAN

allows you to specify the surface(s), scalars, and optional loads, based on the kind of finite element analysis selected, to be exported to a NASTRAN file (extension `.bdf`).

PATRAN

allows you to specify the surface(s), scalars, and optional loads, based on the kind of finite element analysis selected, to be exported to a PATRAN neutral file (extension `.out`).

TAITherm

allows you to specify the surface(s) for which you want to write data and the method of writing the heat transfer coefficient, to be exported to a PATRAN neutral file (extension `.neu`). This option is available only when the **Energy Equation** is enabled.

Tecplot

allows you to specify the surface(s) and the scalars you want to write, to be exported to a Tecplot file.

Cell Zones

specifies the cell zones for which data is to be written for a CFD-Post compatible, EnSight, or FIELDVIEW file.

Surfaces

specifies the surfaces for which data is to be written for an ABAQUS, ASCII, Data Explorer, I-deas Universal, Mechanical APDL Input, NASTRAN, PATRAN, TAITherm, or Tecplot file.

Quantities

specifies valid quantities for output. The attributes of the list are modified based on the active file type. The list may be a single-selection or a multiple-selection list or it may be disabled, depending on the selected **File Type**.

Analysis (for ABAQUS, I-deas Universal, Mechanical APDL Input, NASTRAN, and PATRAN formats)

specifies the finite element analysis intended.

Structural

specifies structural analysis and allows you to select the **Structural Loads** to be written.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Thermal

specifies thermal analysis and allows you to select the **Thermal Loads** to be written.

Structural Loads

contains optional structural loads that can be written to ABAQUS, I-deas Universal, Mechanical APDL Input, NASTRAN, and PATRAN files. This option is available only when **Structural** analysis is selected.

Force

enables force to be written as a load for a structural analysis.

Pressure

enables pressure to be written as a load for a structural analysis.

Temperature

enables temperature to be written as a load for a structural analysis. This option is available only when the **Energy Equation** is enabled.

Thermal Loads

contains optional thermal loads that can be written to ABAQUS, I-deas Universal, Mechanical APDL Input, NASTRAN, and PATRAN files. This option is available only when **Thermal** analysis is selected.

Temperature

enables temperature to be written as a load for a thermal analysis.

Heat Flux

enables heat flux to be written as a load for a thermal analysis.

Heat Trans Coeff

enables heat transfer coefficient to be written as a load for a thermal analysis.

Location (for ASCII, CGNS, and EnSight Case Gold formats)

specifies the location from which the values of scalar functions are to be taken.

Node

specifies that data values at the node points are to be exported.

Cell Center

specifies that data values from the cell centers are to be exported.

EnSight Parallel

(Available only with Fluent in parallel) creates `.encas` files suitable for use in EnSight Enterprise.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Format (for CDAT for CFD-Post & EnSight and EnSight Case Gold)

specifies the file format.

Binary

specifies the file format as binary.

ASCII

specifies the file format as ASCII.

Heat Transfer Coefficient (for TAITherm format only)

specifies the basis for the heat transfer coefficient exported.

Flux Based

specifies the flux based method for writing the heat transfer coefficient.

Wall Function

specifies the wall function based method for writing the heat transfer coefficient.

Write Case File (for CDAT for CFD-Post & EnSight)

specifies the circumstances under which a case file is written with the `.cdat` file. *It is recommended that you select either **Every Time** or **Only if Case/Mesh Changes** since applications consuming `.cdat` files require a corresponding case/mesh file to consume the data.*

Export Data Every

specifies the frequency for appending the data during the solution process based on either **Time Steps** or **Flow Time**.

File Name

specifies the root name for the files to be saved.

Append File Name with

allows you to select **flow-time** or **time-step** to be appended to the file name.

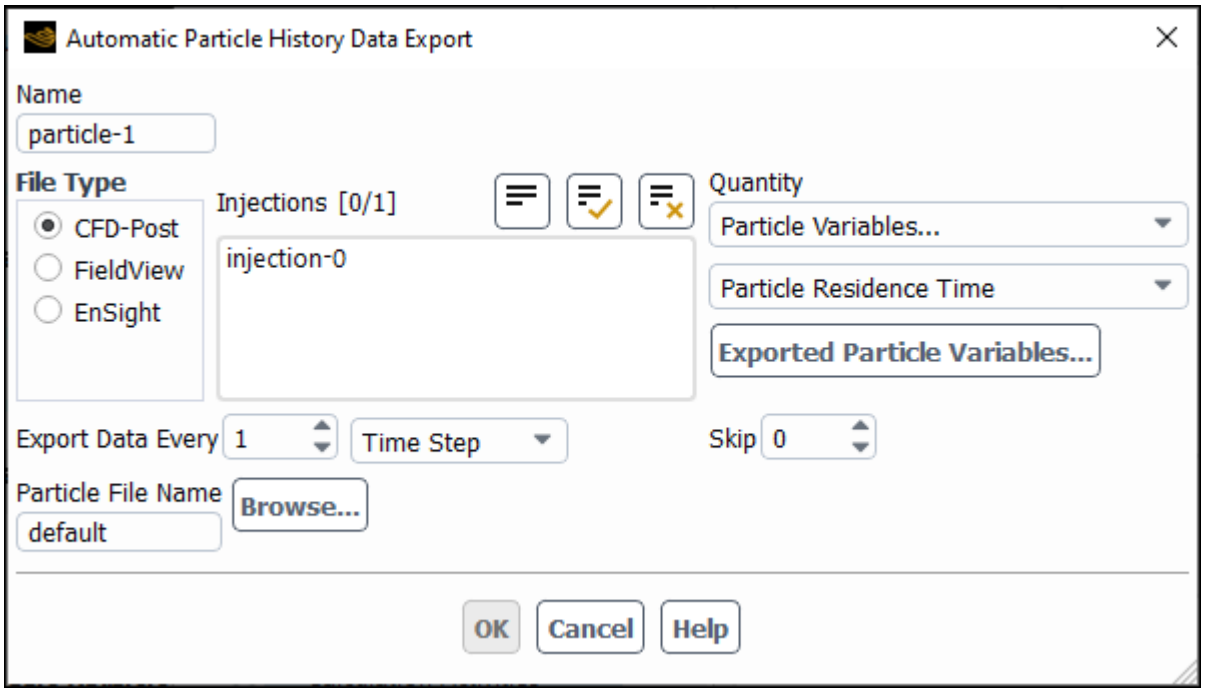
Decimal Places in File Name

allows you to specify the number of decimal digits in the file name. This option is available only when **flow-time** is selected in the **Append File Name with** drop-down list. The default value for this field is set to 6.

51.15.4. Automatic Particle History Data Export Dialog Box

The **Automatic Particle History Data Export** dialog box allows you to create an automatic particle history export definition for solution data. See Creating Automatic Export Definitions for Transient Particle History Data (p. 1004) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name

specifies the name of the particle history export definition.

File Type

specifies the type of the file you want to write.

CFD-Post

allows you to write the file in CFD-Post particle tracks format, which can be read in **CFD-Post**.

FieldView

allows you to write the file in **FIELDVIEW** format, which can be read in **FIELDVIEW**.

EnSight

allows you to write the file in **EnSight** format.

Injections

allows you to select the required injection from the list of predefined injections.

Quantity

contains the list of variables for which you can export the particle data.

Skip

allows you to "thin" or "sample" the number of particles that are exported.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Export Data Every

specifies the frequency for appending the data during the solution process. You can export particle history data frequency based on either **Time Step** or **Flow Time** for unsteady flow cases, or based on **DPM Iteration** for steady flow cases.

Separate Files for Each Time Step

allows you to have separate exported data files for each time step. Available only when **EnSight** is selected as the **File Type**.

Particle File Name

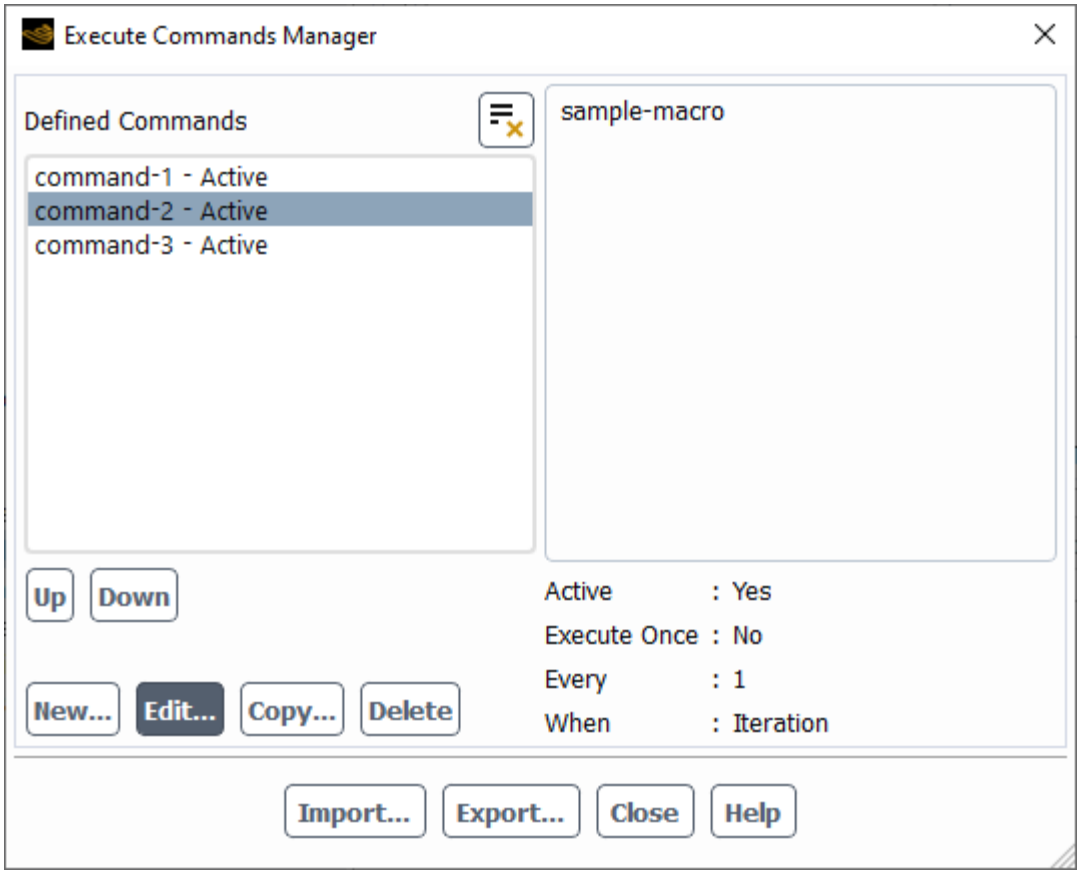
allows you to specify the file name/directory for the exported data, using the **Browse...** button.

Ensight Encas File Name

is the file name you will specify if you selected **EnSight** under **File Type**. Use the **Browse...** button to select the `.encas` file that was created when you exported the file with the **File/Export...** ribbon tab option.

51.15.5. Execute Commands Manager Dialog Box

The **Execute Commands Manager** dialog box allows you to define and manage commands to be executed during the calculation. See Executing Commands During the Calculation (p. 3700) for details about using this feature.



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Controls

Defined Commands

lists the total number of monitor commands that have been defined. Details for a selected command in the list are provided to the right.

Up

moves the selected command up in the list of **Defined Commands** by one.

Down

moves the selected command down in the list of **Defined Commands** by one.

New...

opens the Execute Commands Dialog Box (p. 5201) where you can create a new command.

Edit...

opens the Execute Commands Dialog Box (p. 5201) where you can edit an existing command.

Copy...

opens the Execute Commands Dialog Box (p. 5201) where you can edit an existing command.

Delete...

removes the selected command from the list of **Defined Commands**.

Import...

opens a file selection dialog box where you can designate the name and location for the tab separated value (TSV) file that will contain your imported command(s).

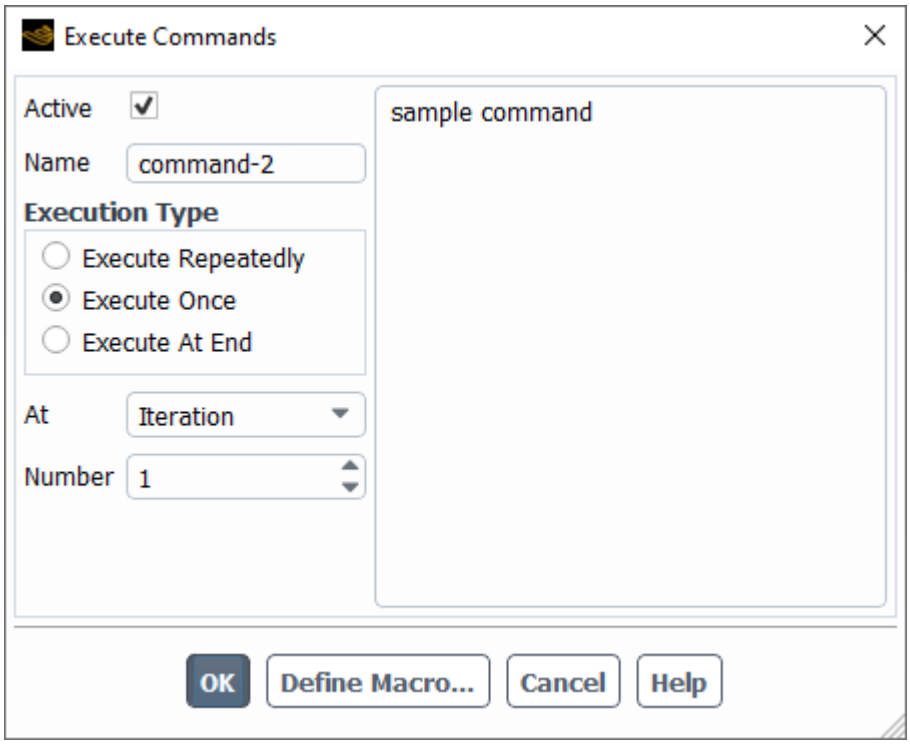
Export...

opens a file selection dialog box where you can designate the name and location for the tab separated value (TSV) file that will contain your exported command(s).

51.15.6. Execute Commands Dialog Box

The **Execute Commands** dialog box allows you to define commands to be executed during the calculation. See Executing Commands During the Calculation (p. 3700) for details about using this feature.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Active

activates/deactivates the execution of each command.

Name

specifies a name for each command.

Execute Repeatedly

Select this option if your command needs to be repeated as part of a sequence of commands.

Every, When

indicate how often the command is to be executed. You can enter the interval under **Every** and select **Iteration**, **Time Step**, **Flow Time**, **Adjoint Iteration**, or **Design Iteration** under **When**. (**Time Step** and **Flow Time** are only valid choices if you are calculating unsteady flow, and **Adjoint Iteration** and **Design Iteration** are only available if you have opened the **Execute Commands** dialog box from the **Gradient-Based Optimizer** or **Run Adjoint Calculation** dialog box.)

Execute Once

Select this option if your command needs to be executed only once.

At/Number

indicate how often the command is to be executed. You can enter the interval under **Number** and select **Iteration**, **Time Step**, **Flow Time**, **Adjoint Iteration**, or **Design Iteration** under

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

At. (**Time Step** and **Flow Time** are only valid choices if you are calculating unsteady flow, and **Adjoint Iteration** and **Design Iteration** are only available if you have opened the **Execute Commands** dialog box from the **Gradient-Based Optimizer** or **Run Adjoint Calculation** dialog box.)

Execute At End

Select this option if your command needs to be executed at the end of the simulation.

Command Field

specifies the command to be executed. You can enter text commands (using multiple lines if required) or the name of a command macro that you have defined in the Define Macro Dialog Box (p. 5203).

Define Macro...

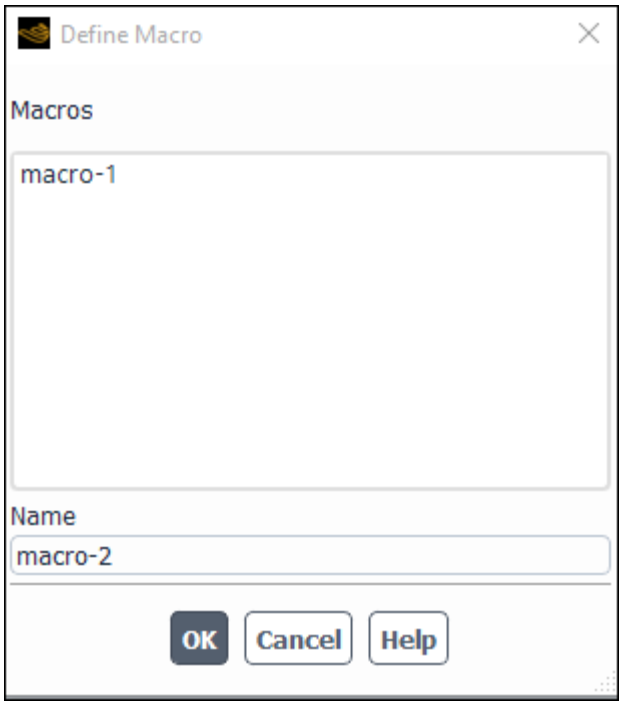
opens the Define Macro Dialog Box (p. 5203), in which you can define command macros.

End Macro

ends the definition of a macro. (This button will replace the **Define Macro...** button when you click **OK** in the **Define Macro** dialog box.)

51.15.7. Define Macro Dialog Box

The **Define Macro** dialog box allows you to define macros for automatic execution by the command monitor, or for interactive use by you. See Defining Macros (p. 3703) for details.



Controls

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Macros

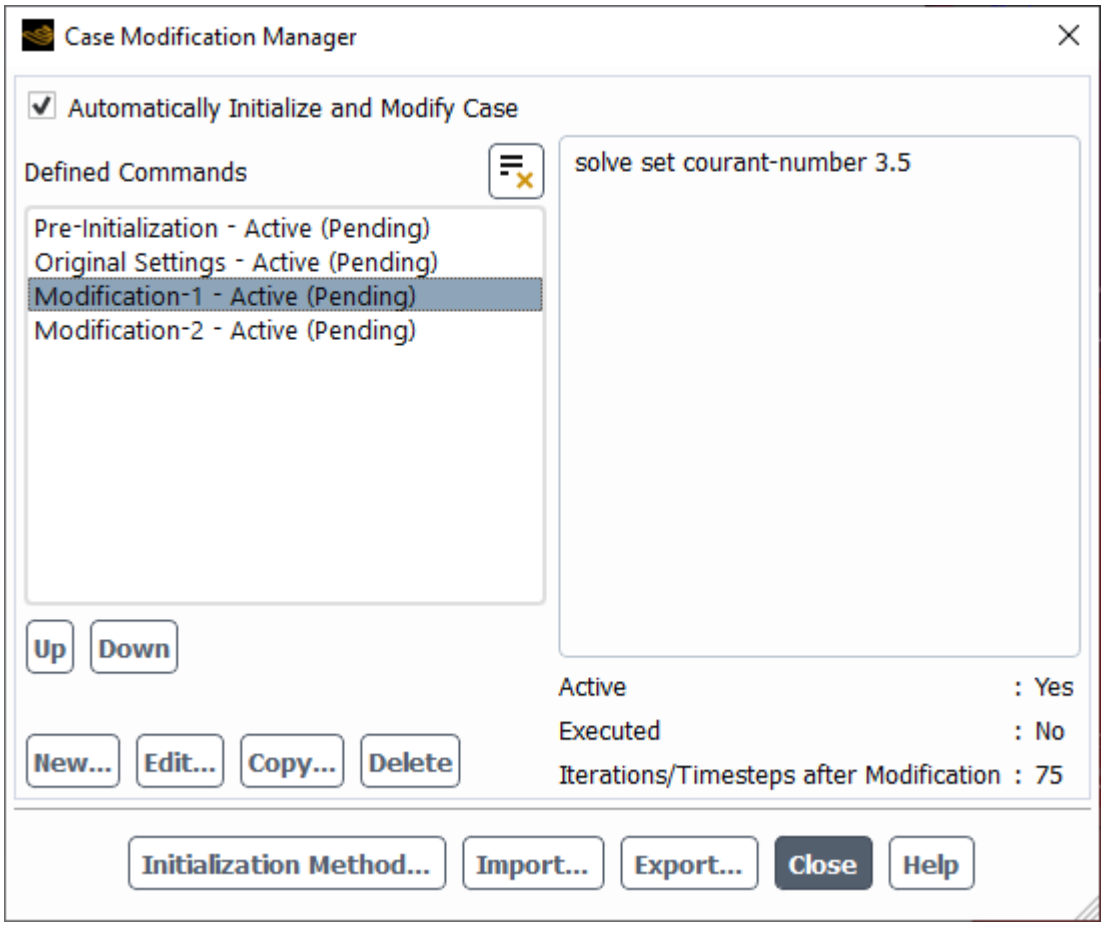
contains a selectable list of the currently-defined macros.

Name

specifies a name for the command macro.

51.15.8. Case Modification Manager Dialog Box

The **Case Modification Manager** dialog box allows you to specify the initialization method and to modify the case. See Automatic Initialization of the Solution and Case Modification (p. 3705) for details.



Controls

Automatically Initialize Solution and Modify Case

allows you to automatically modify the case based on the settings in the dialog box.

Defined Commands

lists your pre-defined commands.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Up

moves the selected command up the list by one increment.

Down

moves the selected command down the list by one.

New...

opens the Modify Case Dialog Box (p. 5206) where you can create a new case modification command.

Edit...

opens the Modify Case Dialog Box (p. 5206) where you can edit an existing case modification command.

Copy

copies the selected command in the list. By default, the copy uses the name of the original command and adds a numerical identifier indicating the number of the copy. For example, making multiple copies of a command `my-new-settings`, gives you copied commands with the default names of `my-new-settings-1`, `my-new-settings-2`, and `my-new-settings-3`, and so on.

Delete

removes the selected command from the list.

Initialization Method...

displays the Auto Initialization Method Dialog Box (p. 5205).

Import...

opens a file selection dialog box where you can locate and select tab-separated value files (`.tsv`) that contain your case modification commands.

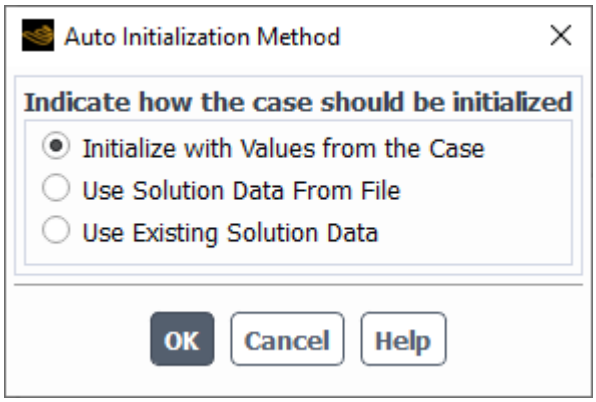
Export...

opens a file selection dialog box where you can specify the name and location of tab-separated value files (`.tsv`) that contain your case modification commands.

51.15.9. Auto Initialization Method Dialog Box

The **Auto Initialization Method** dialog contains several choices for initializing the solution.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Initialize with Values from the Case

uses the values set in the **Solution Initialization** task page.

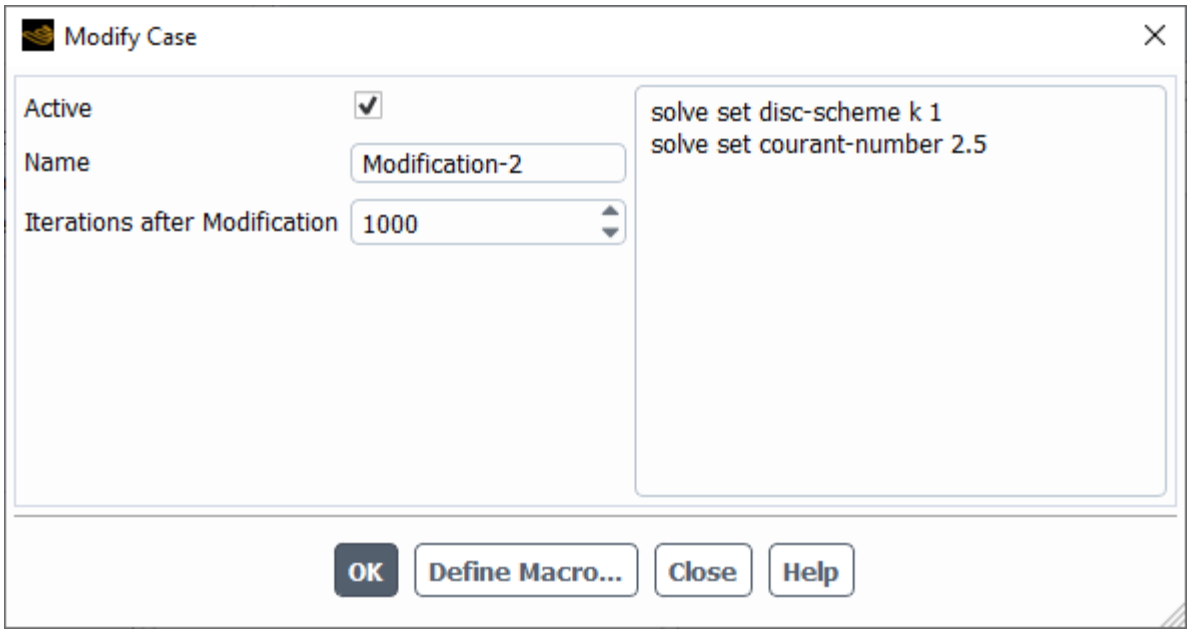
Use Solution Data From File

requires you to read in a data file containing the desired initialization for the case.

Use Existing Solution Data

is analogous to changing the values in a case and continuing the calculation. However, the iteration counter will be reset to 0 so that the modifications can be applied. Use this method when no solution data exists, similar to the first run.

51.15.10. Modify Case Dialog Box



Controls

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Active

allows you to enable or disable a defined case modification.

Name

represents the name of the case modification.

Iterations after Modification

represents the number of iterations that you want to run defined case modification commands.

Quality by which to run with case modification

allows you to choose if the case will be modified using **Time Steps** or by using **Flow Time** (available for transient calculations).

Time Steps after Modification

represents the number of time steps that you want to run defined case modification commands (available for transient calculations when **Time Steps** is selected for the case modification quality).

Flow Time after Modification

represents the number of flow time steps that you want to run defined case modification commands (available for transient calculations when **Flow Time** is selected for the case modification quality).

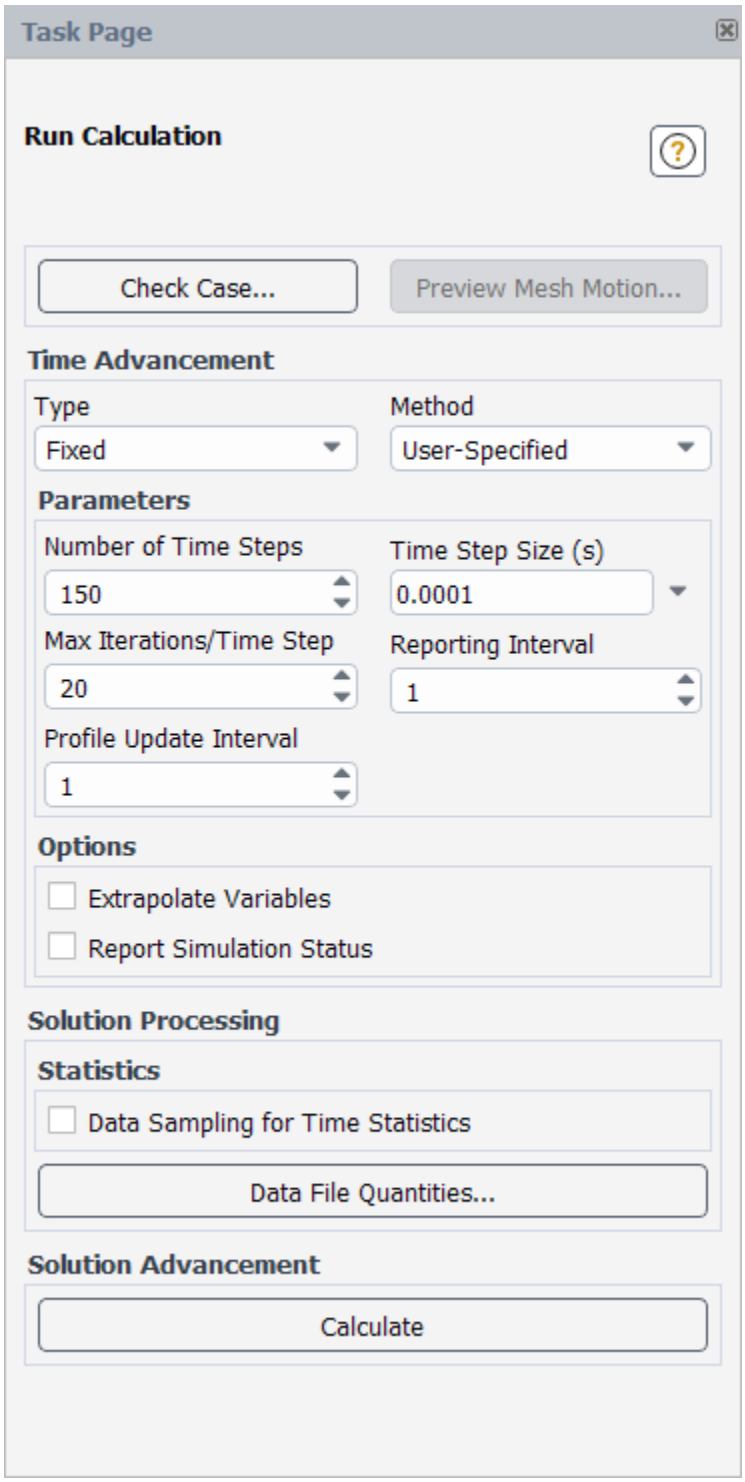
Define Macro...

opens the Define Macro Dialog Box (p. 5203), in which you can define command macros.

51.16. Run Calculation Task Page

The **Run Calculation** task page allows you to start the solver iterations. See Performing Steady-State Calculations (p. 3639) and Performing Time-Dependent Calculations (p. 3643) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Check Case...

opens the Case Check Dialog Box (p. 5219).

Preview Mesh Motion...

opens the Mesh Motion Dialog Box (p. 5151) for transient simulations.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Update Dynamic Mesh...

opens the Mesh Motion Dialog Box (p. 5151) for steady simulations.

Pseudo Time Settings

is available for fluid and solid zones when **Global Time Step** is selected from the **Pseudo Time Method** list in the Solution Methods Task Page (p. 5157) as part of a steady-state case with the pressure-based coupled solver or density-based implicit solver. For details about the available options, see Pseudo Time Settings for the Calculation (p. 3680).

Parameters

(for steady flow calculations) contains settings for running, reporting, and updating the calculation.

Number of Iterations

sets the number of iterations to be performed.

Reporting Interval

sets the number of iterations that will pass before residual monitors will be printed and plotted. The default is 1 (that is, reports will be updated after each iteration).

Note that the **Reporting Interval** also specifies how often Ansys Fluent should check if the solution is converged. For example, if your solution converges after 40 iterations, but your **Reporting Interval** is set to 50, Ansys Fluent will continue the calculation for an extra 10 iterations before checking for (and finding) convergence.

Important:

The **Reporting Interval** only affects convergence checking of residuals, not of report definitions.

Profile Update Interval

sets the number of iterations that will pass before user-defined functions for boundary profiles will be updated. This interval also controls the frequency at which named expression values will be updated.

Time Advancement

(for transient flow calculations) contains settings related to time advancement.

Type

allows you to specify the type of time stepping from the following selections. The type you select will determine what further selections are available in the **Method** drop-down list.

Fixed

specifies that the time step size is fixed to a user-specified value or a value based on a specified period or frequency.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Adaptive

specifies that the time step size changes during the solution, in order to either satisfy the Courant–Friedrichs–Lewy (CFL) condition, maintain a certain truncation error associated with the time integration scheme, or satisfy criteria that is specific for multiphase simulations.

User-Defined Function

specifies that the time step size is defined by a user-defined function that uses the `DEFINE_DELTAT` macro.

Method

allows you to select one of the following, to specify how the time step size and number is determined:

User-Specified

(for the **Fixed** type) specifies the use of a fixed time step size, equal to the specified **Time Step Size**.

Period-Based

(for the **Fixed** type) allows you to specify a period (in seconds) as the basis for determining the time step size and number of time steps. See Inputs for Time-Dependent Problems (p. 3645) for details.

Frequency-Based

(for the **Fixed** type) allows you to specify a frequency (in hertz) as the basis for determining the time step size and number of time steps. See Inputs for Time-Dependent Problems (p. 3645) for details.

CFL-Based

(for the **Adaptive** type) specifies the use of a time step size that gets modified by Ansys Fluent as the calculation proceeds such that a version of the Courant–Friedrichs–Lewy (CFL) condition is satisfied, using the specified **Courant Number**. See CFL-Based Time Stepping (p. 3660) for details.

Error-Based

(for the **Adaptive** type) specifies the use of a time step size that gets modified by Ansys Fluent based on the specified truncation **Error Tolerance**. See Error-Based Time Stepping (p. 3663) for details.

Multiphase-Specific

(for the **Adaptive** type) specifies the use of a time step size that will be modified by Ansys Fluent based on the **Global Courant Number** in the interfacial cells. The time-step-size estimation based on the convective time scale is available for all multiphase models that use the implicit or explicit volume fraction formulation, with the exception of the wet steam model. For the VOF model, the time-step-size estimation includes additional constraints

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

based on the physics and the moving mesh Courant number. See Multiphase-Specific Time Stepping (p. 3665) for details.

Duration Specification Method

(for the **Adaptive** or **User-Defined Function** type) specifies the way in which you will define the calculation duration. The duration can be defined by the total time, the total number of time steps, the incremental time, or the number of incremental time steps. In this context, "total" indicates that Fluent will consider the amount of time / steps that have already been solved and stop appropriately, whereas "incremental" indicates that the solution will proceed for a specified amount of time / steps regardless of what has previously been calculated.

Parameters

(for transient flow calculations) contains settings for running, reporting, and updating the calculation.

Number of Time Steps

sets or reports the number of time steps to be performed.

Period

sets or reports the period to be used in the time step size calculation.

Frequency

sets or reports the frequency to be used in the time step size calculation.

Time Steps per Period

sets the number of time steps to divide each period into.

Total Periods

sets the number of periods for the simulation to run.

Total Time

sets the total time that will be calculated, including the time for which results were generated during previous calculations.

Total Number of Time Steps

sets the total number of time steps to be performed, including the number of time steps performed in previous calculations.

Incremental Time

sets the incremental time (that is, the additional time) for which results are to be calculated.

Time Step Size

sets or reports the size of the (physical) time step Δt.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Courant Number

allows you to specify the Courant number for CFL-based time stepping. The default value for the **Courant number** is 1.

Error Tolerance

specifies the threshold value to which the computed truncation error is compared as part of error-based time stepping. Increasing this value will lead to an increase in the size of the time step and a reduction in the accuracy of the solution. Decreasing it will lead to a reduction in the size of the time step and an increase in the solution accuracy, although the calculation will require more computational time. For most cases, the default value of 0.01 is acceptable.

Global Courant Number

allows you to specify the global Courant number for multiphase-specific time stepping. The default value for the **Global Courant number** is 2.

User-Defined Time Step Size

contains a drop-down list of available user-defined functions (UDFs) that can be used to define the time step size.

Number of Fixed Time Steps

specifies the number of fixed-size time steps that should be performed before the size of the time step starts to change. The size of the fixed time step is the value specified for **Initial Time Step Size**.

Initial Time Step Size

is used for the first time step and then for as many subsequent steps as specified in the **Number of Fixed Time Steps** field. This value must fall between the minimum and maximum time step sizes. For a better startup, it should be chosen such that the Courant number initially remains close to 1 or your specified value (whichever is lower). Note that if you initialize the solution for a previously run case with adaptive time stepping, then Ansys Fluent will use the time step size saved in the case.

Max Iterations/Time Step

(when using an implicit unsteady formulation) sets the maximum number of iterations to be performed per time step. If the convergence criteria are met before this number of iterations is performed, the solution will advance to the next time step. See Inputs for Time-Dependent Problems (p. 3645) for details.

Reporting Interval

sets the number of time steps that will pass before residual monitors will be printed and plotted. The default is 1 (that is, reports will be updated after each time step).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Note that the **Reporting Interval** also specifies how often Ansys Fluent should check if the solution is converged.

Important:

The **Reporting Interval** only affects convergence checking of residuals, not of report definitions.

Profile Update Interval

sets the number of iterations that will pass before user-defined functions for boundary profiles will be updated. This interval also controls the frequency at which named expression values will be updated.

Time Step Size Update Interval

specifies the number of time steps that will pass before the time step size is updated for adaptive time stepping methods.

Settings...

opens the Adaptive Time Stepping Dialog Box (p. 5220), so that you can define further settings for the adaptive time stepping method.

Options (unsteady)

contains options related to unsteady calculations.

Extrapolate Variables

instructs Ansys Fluent to predict the solution variable values for the next time step and then input that predicted value as an initial guess for the inner iterations of the current time step.

Report Simulation Status

opens the Simulation Status Dialog Box (p. 5221), which reports details about the simulation.

Specify Solid Time Step Size

enables a different time step size to be specified for solid zones, so that it is either user specified or automatically calculated. Using this option to specify a larger solid time step size reduces the risk of convergence issues. It is only available for transient cases, when there is a solid zone in the domain and energy is enabled. See Specifying the Solid Time Step Size (p. 2169) for details.

When this option is enabled, the following are available in the **Solid Time Stepping** group box:

Method

specifies how the time step size is determined for solid zones.

Time Step Size

specifies the time step size for solid zones when you have selected **User-Specified** for the **Method**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Loosely Coupled Conjugate Heat Transfer

works in conjunction with the **Specify Solid Time Step Size** option (and thereby enables the use of a user-specified time step size for solid zones that can be larger than that used for the fluid zones) to increase the robustness of the energy equation calculation, and specifies that multi-domain architecture is used within a single Fluent session to enhance the performance of the simulation. It enables a loose coupling of the fluid and solid zone calculations, either at a specified time period or number of fluid time steps. For further details on using this option and its limitations, see Loosely Coupled Conjugate Heat Transfer (p. 2172).

When this option is enabled, the following are available in the **Loosely Coupled Conjugate Heat Transfer** group box for steady flows:

Coupling Method

allows you to select from the following to determine how the coupling is performed.

Instantaneous Implicit

specifies that the fluid-solid interface is solved in a fully coupled manner like conventional coupled thermal boundary condition at each coupling step. For details, see Thermal Conditions for Two-Sided Walls (p. 1472).

Time-Averaged Explicit

specifies that Dirichlet – Neumann explicit thermal coupling is employed at each coupling step. A convective thermal boundary condition is applied at the wall adjacent to the solid and the instantaneous solid-side temperature is applied on the wall adjacent to the fluid. Time-averaged heat flow on the fluid side is used to compute the convective thermal boundary condition parameters (heat transfer coefficient and free-stream temperature) on the solid side.

Coupling Controls

provides the following controls:

Coupling Frequency Basis

specifies how you will define the coupling frequency. With the **Instantaneous Implicit** coupling method, you can select **Time** so that the coupling is performed at a set **Time Period**; otherwise (if you are not using adaptive time stepping), you can select **Fluid Time Steps** so that the coupling is performed at a set **Number of Fluid Time Steps**.

Time Period

sets the time period for when solid calculations are solved and coupled with the fluid calculations. This field is available when **Time** is selected from the **Coupling Frequency Basis** drop-down list.

Number of Fluid Time Steps

sets the number of fluid time steps used to trigger the solution of the solid calculations and coupling with the fluid calculations. This field is available when **Fluid Time Steps** is selected from the **Coupling Frequency Basis** drop-down list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Average Over (Time Steps)

displays the number of time steps used for computing time-averaged thermal variables. This is available when **Time-Averaged Explicit** is selected for the coupling method, and is shown for information only. The input for this field is automatically copied from the specified **Number of Fluid Time Steps**.

Options (steady-state, pressure-based solver),

contains options for steady-state simulations that use the pressure-based solver.

Loosely Coupled Conjugate Heat Transfer

is an option that can increase the robustness of the energy equation calculation, and specifies that multi-domain architecture is used within a single Fluent session to enhance the performance of the simulation. It enables a loose coupling of the fluid and solid zone calculations at a specified number of iterations. For further details on using this option and its limitations, see Loosely Coupled Conjugate Heat Transfer (p. 2172).

When this option is enabled, the following are available in the **Loosely Coupled Conjugate Heat Transfer** group box for steady flows:

Coupling Method

allows you to select from the following to determine how the coupling is performed.

Implicit

specifies that the fluid-solid interface is solved in a fully coupled manner like conventional coupled thermal boundary condition at each coupling iteration. For details, see Thermal Conditions for Two-Sided Walls (p. 1472).

Explicit

specifies that Dirichlet – Neumann explicit thermal coupling is employed at each coupling iteration. A convective thermal boundary condition is applied at the wall adjacent to the solid and the instantaneous solid-side temperature is applied on the wall adjacent to the fluid. Iteration-averaged heat flow on the fluid side is used to compute the convective thermal boundary condition parameters (heat transfer coefficient and free-stream temperature) on the solid side.

Coupling Controls

provides the following controls:

Number of Iterations

sets the number of iterations used to trigger the solution of the solid calculations and coupling with the fluid calculations.

Average Over (Iterations)

displays the number of iterations used for computing iteration-averaged thermal variables. This is available when **Explicit** is selected for the coupling method, and is shown for

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

information only. The input for this field is automatically copied from the specified **Number of Iterations**.

Options (steady-state, density-based solver)

contains options for steady-state simulations that use the density-based implicit solver.

Solution Steering

allows you to set parameters that will help Ansys Fluent guide the calculations to a converged solution. This option is only available for steady-state simulations that use the density-based implicit solver. See Solution Steering (p. 3746) for more information. When enabled, the following options are available:

Flow Type

allows you to select the flow type that best describes the flow in the solution domain. Five choices are available: **incompressible**, **subsonic**, **transonic**, **supersonic**, and **hypersonic**.

Use FMG Initialization

allows for full multigrid initialization.

First- to Higher-Order Blending

allows you to reduce the desired solution accuracy by selecting a blending factor less than 100%. The default setting is 100%. See First- to Higher-Order Blending in the Theory Guide for more information. The blending factor will be grayed out if **Second Order Upwind** discretization for the **Flow** equations is not selected in the **Solution Methods** task page. The solution accuracy may be reduced (typical values are 75% or 50%) if it is not possible to obtain a converged solution with the maximum second-order accuracy (blending = 100%).

More Settings...

opens the Solution Steering Dialog Box (p. 5222).

Courant Number

is a non-adjustable field displaying the current CFL number, which allows you to view it during the calculation.

Solution Processing

contains controls related to the processing of the calculations.

Acoustic

contains controls for simulations that use the FW-H acoustics model.

Time Step Size for Acoustic Data Export

determines the highest frequency that the acoustic analysis reproduces. This is available for the density-based solver when both the explicit formulation and explicit transient formulation are selected.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Acoustic Sources FFT...

opens the Acoustic Sources FFT Dialog Box (p. 5224). This button is only available for three-dimensional transient simulations, when the **Export Acoustic Source Data in ASD Format** option is enabled in the **Acoustics Model** dialog box.

Acoustic Signals...

opens the Acoustic Signals Dialog Box (p. 5230).

Pollutants

contains controls for simulations that model pollutant formation.

Postprocess Pollutants

results in the automatic postprocessing of pollutants during a transient simulation.

Max Post Iterations/Time Step

sets the maximum number of postprocessing iterations to be performed per time step. This field is available when the **Postprocess Pollutants** option is enabled.

PDF Transport Time Averaging

contains solution controls for steady-state simulations that use the composition PDF transport model with the Lagrangian method.

Iterations in Average

specifies the number of iterations included in the PDF transport time averaging. For recommendations on how to use this setting, see Monitoring the Solution (p. 2451).

Iteration Increment

specifies the rate at which the number of iterations included in the PDF transport time averaging will increase in each subsequent iteration. For recommendations on how to use this setting, see Monitoring the Solution (p. 2451).

Statistics

contains controls for sampling statistics.

Data Sampling for Steady Statistics

enables the sampling of data during a steady-state calculation. See Performing Steady-State Calculations (p. 3639) and Postprocessing for Time-Dependent Problems (p. 3669) for details.

Data Sampling for Time Statistics

enables the sampling of data during an unsteady calculation. See Inputs for Time-Dependent Problems (p. 3645) and Postprocessing for Time-Dependent Problems (p. 3669) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sampling Interval

allows you to specify how often data samples are accumulated for computation of mean and RMS values. For example, a sampling interval of 5 means that only every 5th result is added to the statistical sampling. *This is for* ***Data Sampling for Steady Statistics*** *and* ***Data Sampling for Time Statistics****.*

Sampled Iterations

reports the number of iterations over which data has been sampled for the postprocessing of the mean and RMS values.

Sampled Time

reports the time period over which data has been sampled for the postprocessing of the mean and RMS values.

Sampling Options...

opens the Sampling Options Dialog Box (p. 5232) for the **Data Sampling for Steady Statistics** or **Data Sampling for Time Statistics**.

Data File Quantities...

opens the Data File Quantities Dialog Box (p. 5192). This button is only available when you are writing in the legacy format (that is, `.dat` files).

Solution Advancement

contains buttons that advance the solution.

Calculate

starts the calculations. While the calculation is in progress, a progress bar will appear at the bottom of the Fluent window. For steady-state simulations, clicking the **Stop** button next to the progress bar will interrupt the calculation at the earliest safe stopping point after the current iteration; for transient simulations, buttons are available that allow you to stop the calculation at the end of the current iteration or time step. Alternatively, you can type **Ctrl**+**c** in the console; for transient simulations, a single instance will stop the calculation at the end of the current time step, whereas typing it twice will stop it at the end of the current iteration.

Continue Calculation

continues the calculations. This button is available when you have enabled the **Automatically Initialize and Modify Case** option in the **Calculation Activities** task page.

Start Calculation Over

restarts the calculation from the beginning. This button is available when you have enabled the **Automatically Initialize and Modify Case** option in the **Calculation Activities** task page.

For additional information, see the following sections:

51.16.1. Case Check Dialog Box

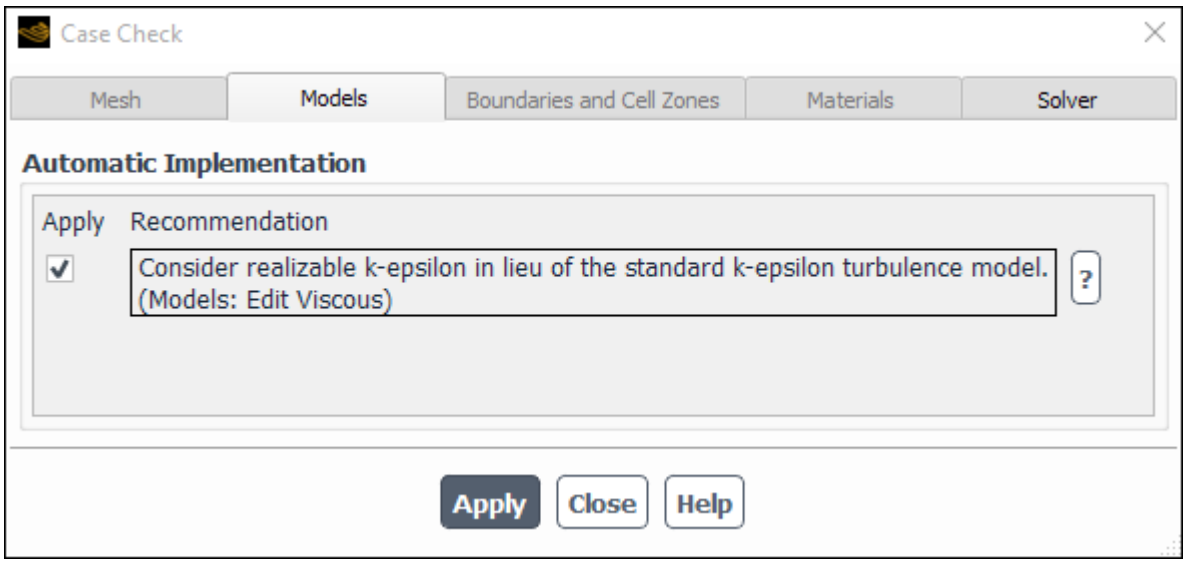
51.16.2. Adaptive Time Stepping Dialog Box

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

51.16.3. Simulation Status Dialog Box
51.16.4. Solution Steering Dialog Box
51.16.5. Acoustic Sources FFT Dialog Box
51.16.6. Acoustic Signals Dialog Box
51.16.7. Sampling Options Dialog Box
51.16.8. Zone-Specific Sampling Options Dialog Box

51.16.1. Case Check Dialog Box

This function provides you with guidance and best practices when choosing case parameters and models. Your case will be checked for compliance in the mesh, models, boundary and cell zone conditions, material properties, and solver categories. Established rules are available for each category, with recommended changes to your current settings. Information about each of the recommendations is available in Checking Your Case Setup (p. 3725).



Controls

Mesh

displays recommendations, if any, relating the to the mesh used in the case. See Checking the Mesh (p. 3727) for details.

Models

displays recommendations, if any, relating the to the models used in the case. See Checking Model Selections (p. 3729) for details.

Boundaries and Cell Zones

displays recommendations, if any, relating the to the cell zones or boundaries defined in the case. See Checking Boundary and Cell Zone Conditions (p. 3731) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Materials

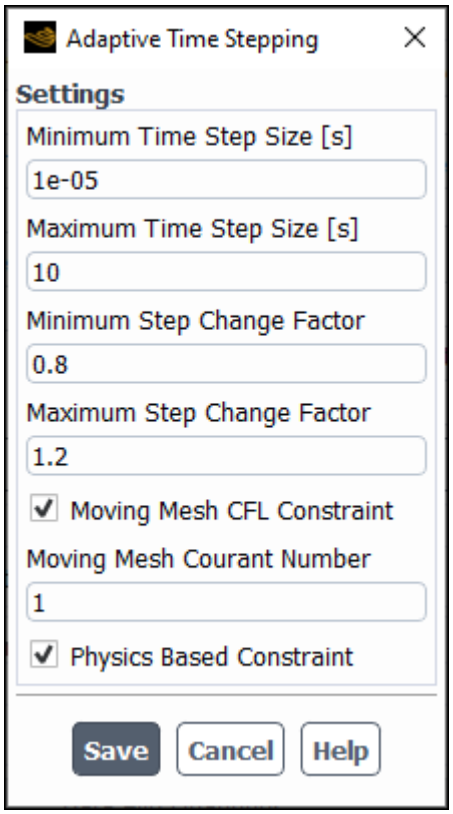
displays recommendations, if any, relating the to the materials defined in the case. See Checking Material Properties (p. 3734) for details.

Solver

displays recommendations, if any, relating the to the solver settings used in the case. See Checking the Solver Settings (p. 3735) for details.

51.16.2. Adaptive Time Stepping Dialog Box

The **Adaptive Time Stepping** dialog box provides additional settings that affect the adaptive time stepping. This dialog box opens when you click **Settings...** in the **Run Calculation** task page. See Performing Time-Dependent Calculations (p. 3643) for details about adaptive time stepping.



Controls

Minimum/Maximum Time Step Size

specify the upper and lower limits for the size of the time step. If the time step size becomes very small, the computational expense may be too high; if the time step size becomes very large, the solution accuracy may not be acceptable to you. You can set the limits that are appropriate for your simulation.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Minimum/Maximum Step Change Factor

limit the degree to which the time step size can change at each time step. Limiting the change results in a smoother calculation of the time step size, especially when high-frequency noise is present in the solution. For the **CFL-Based** and **Multiphase-Specific** methods, the time step size change factor is computed as the ratio between the current time step size and the previous time step size. For the **Error-Based** method, the time step size change factor is computed as the ratio between the specified truncation error tolerance and the computed truncation error.

Moving Mesh CFL Constraint

(VOF model only) provides an option to include moving mesh CFL constraint in the time step size estimation. When this option is enabled, you can adjust **Moving Mesh Courant Number**, which has a default value of 1. For applications with a pure rigid body motion, **Moving Mesh Courant Number** may not be a significant parameter.

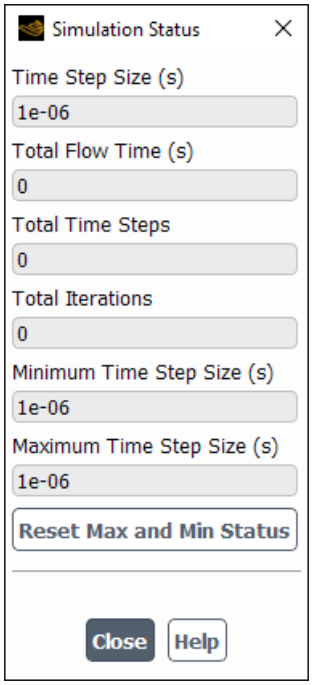
This item is available only for applications that involve either Frame Motion, or Mesh Motion, or Dynamic Mesh. See The Multiphase-Specific Time Stepping Algorithm (p. 3665) for details.

Physics Based Constraint

(VOF model only) provides an option to include physics-based constraints (depending on a specific type of physics involved, such as viscous, acoustic, gravitational, surface-tension-driven, and so on) in the time-step-size estimation. See The Multiphase-Specific Time Stepping Algorithm (p. 3665) for details.

51.16.3. Simulation Status Dialog Box

The **Simulation Status** dialog box reports details about the simulation.



Controls

Time Step Size

(for adaptive time stepping) reports the size of the last time step calculated for the simulation.

Total Flow Time

reports the total time calculated for the simulation.

Total Time Steps

reports the number of time steps calculated for the simulation.

Total Iterations

reports the number of iterations calculated for the simulation.

Minimum Time Step Size

(for adaptive time stepping) reports the size of the smallest time step used since either the start of the simulation or the last time you clicked the **Calculate** button in the **Run Calculation** task page, depending on whether you have clicked the **Reset Min and Max Status** button.

Maximum Time Step Size

(for adaptive time stepping) reports the size of the largest time step used since either the start of the simulation or the last time you clicked the **Calculate** button in the **Run Calculation** task page, depending on whether you have clicked the **Reset Min and Max Status** button.

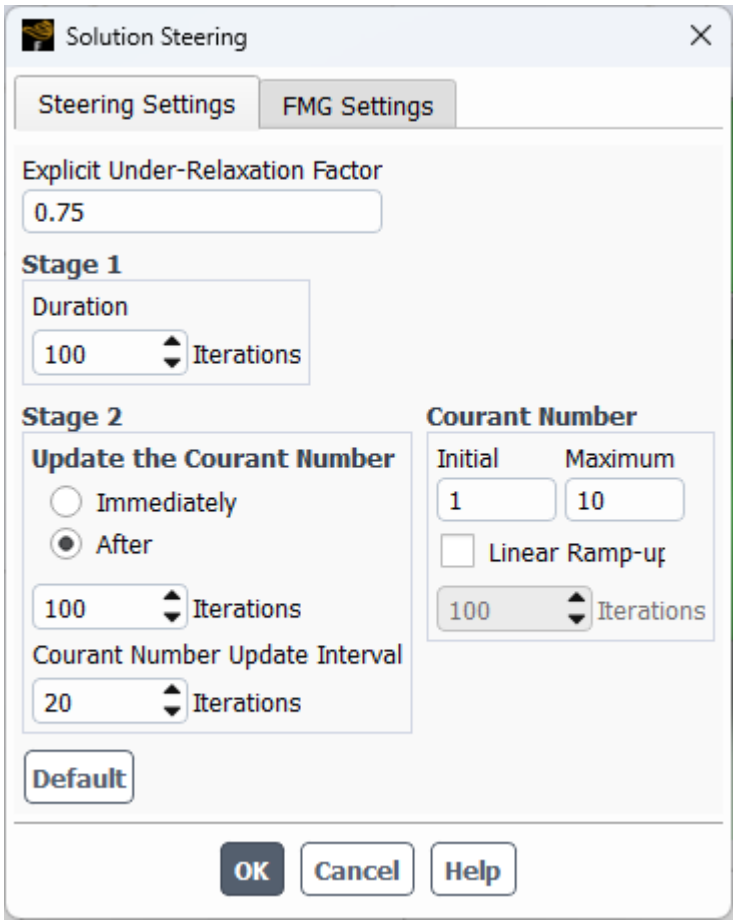
Reset Min and Max Status

(for adaptive time stepping) resets the **Minimum Time Step Size** and **Maximum Time Step Size** fields, so that they consider only the values since the latest time you clicked the **Calculate** button in the **Run Calculation** task page.

51.16.4. Solution Steering Dialog Box

The **Solution Steering** dialog box is used to set the parameters that control the solution steering strategy. Solution steering will typically perform full multigrid (FMG) initialization followed by two iterative stages (Stage 1 and Stage 2). The purpose of Stage 1 is to navigate the solution from the difficult initial phase of the solution toward convergence by insuring maximum stability. During this stage, the solution is advanced gradually from 1st-order accuracy to maximum accuracy (user specified and typically 2nd-order) at a constant low CFL value. In Stage 2, the solution is driven hard towards convergence by regular adjustments of the CFL value to insure fast convergence as well as to prevent possible divergence. In Stage 2, the residual history is monitored and analyzed through regular intervals to determine if an increase or decrease in CFL value is needed to obtain fast convergence or to prevent divergence. See Solution Steering (p. 3746) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Steering Settings

allows you to modify the steering parameters used in Stages 1 and 2.

Stage 1

allows you to set parameters relating to stage 1.

Duration

is the number of iterations in stage 1. The CFL number used during these iterations is set in the **Initial** field, in the **Courant Number** group box.

Stage 2

allows you to set parameters relating to stage 2.

Update the Courant Number

allows you to update the Courant number either **Immediately**, or **After** a specified number of iterations.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Courant Number Update Interval

defines the frequency at which the Courant number is updated.

Courant Number

allows you to set the starting (**Initial**) and maximum allowed (**Maximum**) Courant number values. The solution steering algorithm will not allow the solver to exceed the maximum Courant number, but will allow the solver to use a Courant number less than the initial Courant number if divergence in the solution has occurred.

Explicit Under-Relaxation Factor

allows the solution to be under-relaxed to improve convergence.

Default

resets any changes made to the parameters to their original default values.

FMG Settings

allows you to set FMG parameters. For more information about FMG initialization, refer to Full Multigrid (FMG) Initialization (p. 3633).

Number of Multigrid Levels

allows you to set the number of multigrid levels.

Number of Cycles

allows you to set the number of cycles for a selected level.

FMG Courant Number

allows you to set the FMG Courant number.

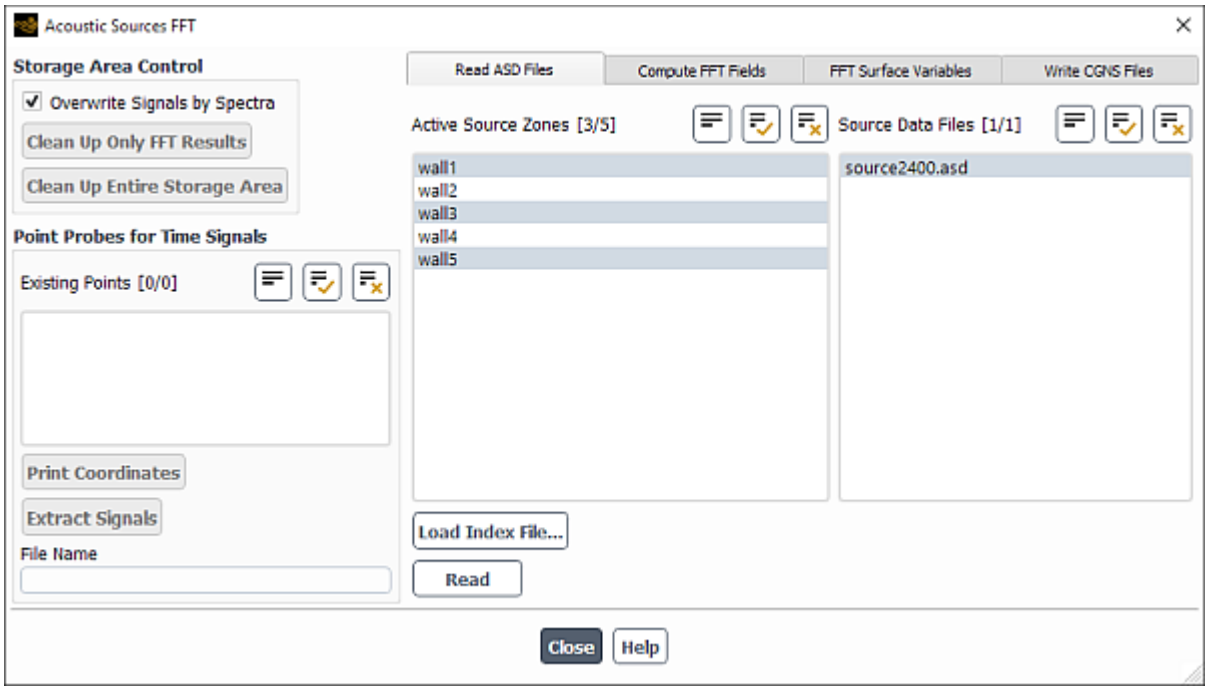
Default

resets any changes to the original default values.

51.16.5. Acoustic Sources FFT Dialog Box

The **Acoustic Sources FFT** dialog box allows you to: read pressure signals from the acoustic source data (ASD) files (see Writing Source Data Files (p. 2632)); compute Fourier spectra of these signals at the selected point probes and over the entire source zones; create surface variables for visualizing the spectral properties of the flow pressure signals; and write CGNS files of the spectrum data. See FFT of Acoustic Sources: Band Analysis and Export of Surface Pressure Spectra (p. 2644) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Storage Area Control

allows you to specify how memory is allocated in the storage area (that is, the large memory array that stores the fields of the pressure histories and the computed fields of the Fourier spectra) when you compute the FFT fields in the **Compute FFT Fields** tab.

Overwrite Signals by Spectra

specifies that the Fourier spectra displace in memory the original time signals, so that they are not kept after the FFT has been computed.

Clean Up Only FFT Results

de-allocates the storage area—so that you can re-compute the FFT without re-reading the pressure histories—and deletes all variables created through the **FFT Surface Variables** tab. This button is only available when the **Overwrite Signals by Spectra** option is disabled.

Clean Up Entire Storage Area

de-allocates the storage area—so that you can re-read ASD files and re-compute the FFT with a clipped time range or different window function—and deletes all variables created through the **FFT Surface Variables** tab.

Point Probes for Time Signals

allows you to get information about existing point probes and create pressure history files.

Existing Points

lists all of the point probes created through the Point Surface Dialog Box (p. 5582) that are available for selection.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Print Coordinates

prints in the console the coordinates of the point probe(s) selected in the **Existing Points** selection list.

Extract Signals

creates ASCII files that contain the pressure histories of the point probe(s) selected in the **Existing Points** selection list.

File Name

specifies a prefix for the names of the files created by the **Extract Signals** button.

Read ASD Files

allows you to read pressure histories from the acoustic source data (ASD) files.

Active Source Zones

is a list of the face zones for which you can read acoustic source data.

Source Data Files

is a list of the ASD files you can read.

Load Index File...

updates the **Active Source Zones** and **Source Data Files** lists by loading an ASCII index file.

Read

reads the pressure signals from the selected items in the **Active Source Zones** and **Source Data Files** lists.

Compute FFT Fields

allows you to compute FFT fields from the pressure signals read through the **Read ASD Files** tab.

Sampling Data

allows you to reduce the time range of the signal data used to compute the FFT fields.

Clip Time to Range

specifies that the time range of the data sampled is reduced to be between the **Min** and **Max**.

Min

specifies the minimum value of the time range of the signal data when the **Clip Time to Range** option is enabled.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Max

specifies the maximum value of the time range of the signal data when the **Clip Time to Range** option is enabled.

Time Step

reports the size of the time step at which data was sampled.

Re-Estimate Spectral Resolution

updates the values that are displayed in the **Spectral Resolution** group box and that are used to compute the FFT fields, based on the settings in the **Sampling Data** group box.

Spectral Resolution

displays the number of samples and the expected spectrum properties for the provided signals.

Number of Samples

reports the total number of samples in the specified time range.

Samples Used for FFT

reports the number of samples that will be used for FFT (see Table 40.2: Numbers of Data Points Supported by the Prime-Factor FFT Algorithm (p. 4080)).

Frequency Min & Step

reports the expected minimum frequency in the spectrum, which is also the expected frequency step.

Frequency Max

reports the expected maximum frequency in the spectrum.

Number of Modes

reports the expected number of Fourier modes in the spectrum.

Window Function

specifies the shape of the window function used to enforce the signal's periodicity. See Windowing (p. 4079) for details. The default is the **Hanning** window.

Compute

computes the FFT fields based on the settings in the **Compute FFT Fields** tab.

FFT Surface Variables

allows you to create new Fluent variables for postprocessing using the spectrum fields computed through the **Compute FFT Fields** tab.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Modes/Frequency Bands

specifies what the created variables will characterize.

Octave Bands

specifies that the variables characterize proportional frequency bands corresponding to the standard technical octaves.

1/3 Octave Bands

specifies that the variables characterize proportional frequency bands corresponding to the standard technical thirds.

Constant Width Bands

specifies that the variables characterize user-defined equidistant frequency bands.

Set of Modes

specifies that the variables characterize the individual Fourier modes.

Octave Central Frequencies

is a list of the octave central frequencies for which you can **Create** Fluent variables for post-processing. This selection list is available when **Octave Bands** is selected from the **Modes/Frequency Bands** drop-down list.

1/3-Octave Central Frequencies

is a list of the 1/3-octave central frequencies for which you can **Create** Fluent variables for postprocessing. This selection list is available when **1/3-Octave Bands** is selected from the **Modes/Frequency Bands** drop-down list.

Frequency Min

specifies the minimum frequency of the set of modes for which you want to **Create** Fluent variables for postprocessing. This field is available when **Set of Modes** or **Constant Width Bands** is selected from the **Modes/Frequency Bands** drop-down list.

Frequency Max

specifies the maximum frequency of the set of modes for which you want to **Create** Fluent variables for postprocessing. This field is available when **Set of Modes** or **Constant Width Bands** is selected from the **Modes/Frequency Bands** drop-down list.

Number of Modes to Skip

allows you to thin or sample the modes for which you want to **Create** the Fluent variables for postprocessing. This field is available when **Set of Modes** is selected from the **Modes/Frequency Bands** drop-down list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Band Width

specifies the width of the constant width bands, for which you want to **Create** the Fluent variables for postprocessing. This field is available when **Constant Width Bands** is selected from the **Modes/Frequency Bands** drop-down list.

Spectrum Property

displays the type of variables created according to the selected choice in the **Modes/Frequency Bands** drop-down list.

PSD of dp/dt

is displayed when frequency bands are selected from the **Modes/Frequency Bands** drop-down list, in order to indicate that the created variables will be the power spectral density (PSD) fields, calculated for the time derivative of flow pressure.

Surface Pressure Level

is displayed when frequency band is selected from the **Modes/Frequency Bands** drop-down list, in order to indicate that the created variables will be the surface pressure level (SPL) fields, in decibels.

Real and Imaginary Amplitude Parts

is displayed when **Set of Modes** is selected from the **Modes/Frequency Bands** drop-down list, in order to indicate that the created variables will be the real and imaginary parts of the complex Fourier amplitudes, with a pair of variables per specified mode.

Existing Variables

lists the processed variables that have been created by the **Create** button, and allows you to make selections for deletion using the **Remove Selected Variables** button.

Remove Selected Variables

deletes the processed variables selected in the **Existing Variables** list from the storage area (as described previously), so that you can analyze more than the maximum of 20 individual modes or constant width bands that are allowed to exist at any one time.

Write Area-Avg Data

When selected, and upon clicking **Create**, Fluent writes the area-averaged pressure data to an ASCII file in the XY plot file format.

Plot Area-Avg Data

When selected, and upon clicking **Create**, Fluent plots the area-averaged pressure data in the form of a bar chart in the graphics window.

Create

creates new Fluent variables for postprocessing (based on your settings in the **FFT Surface Variables** tab), updates the **Existing Variables** selection list, and (when **Set of Modes** or

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Constant Width Bands is selected from the **Modes/Frequency Bands** drop-down list) prints information in the console.

Write CGNS Files

allows you to write CGNS files of the spectrum data.

Processed Source Zones

is a list of the zones you can export.

Frequency Series

allows you to reduce the exported spectrum data.

Reduce Frequency Series

specifies that the exported spectrum data is reduced to be between the **Min** and **Max**.

Min

specifies the minimum of the frequency range to be exported when the **Reduce Frequency Series** option is enabled.

Max

specifies the maximum of the frequency range to be exported when the **Reduce Frequency Series** option is enabled.

Number of Frequencies to Skip

allows you to thin or sample the frequencies to be exported.

Number of Frequencies per File

specifies the number of frequencies exported in each file. A value should be chosen based on the **Number of Modes** field of the **Compute FFT Fields** tab, in order to avoid creating either extremely large output files or a large number of very small files.

File Name

specifies the base file name of the output files.

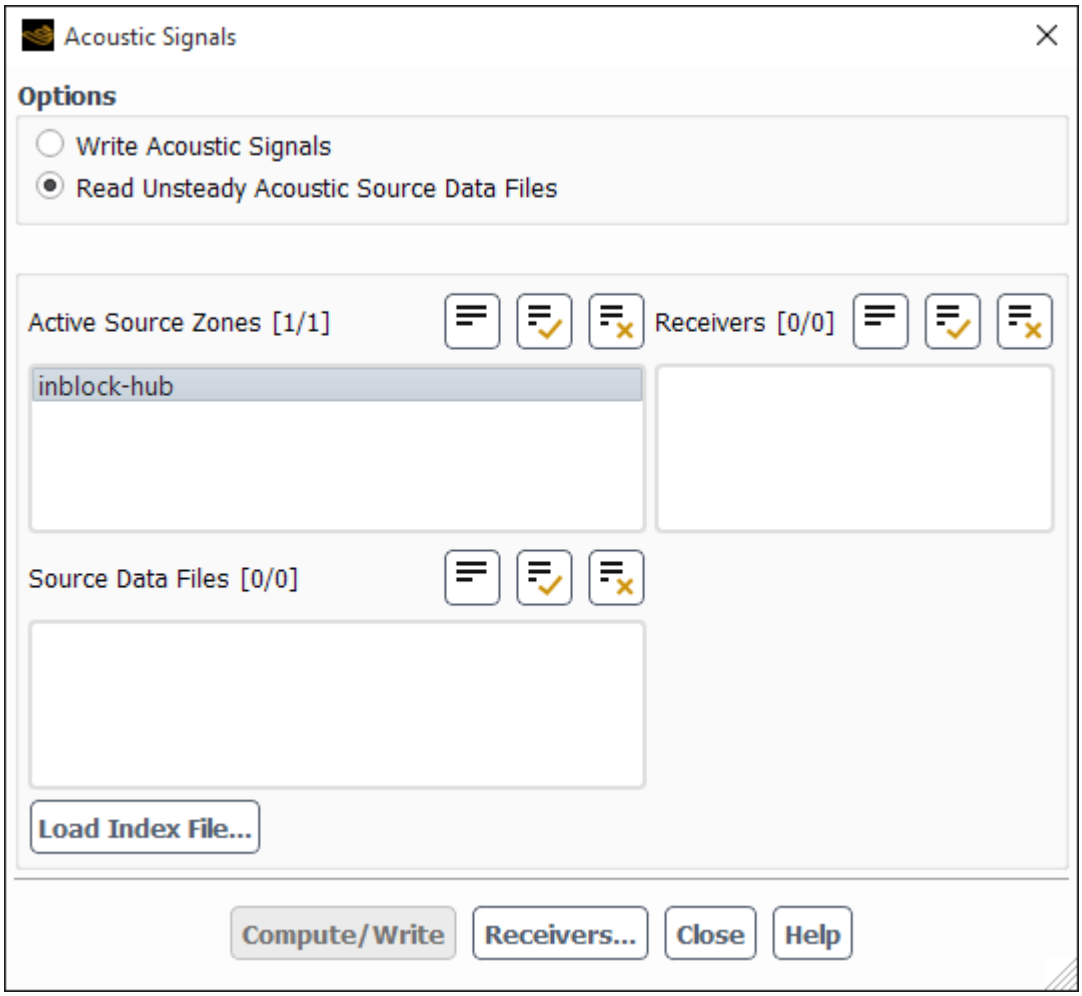
Write

writes CGNS files of the spectrum data.

51.16.6. Acoustic Signals Dialog Box

The **Acoustic Signals** dialog box is used to compute and save the sound pressure signals. See Post-processing the FW-H Acoustics Model Data (p. 2641) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options

contains the options available for acoustic signal postprocessing.

Write Acoustic Signals

enables the parameters needed to write the sound pressure data to files.

Read Unsteady Acoustic Source Data Files

enables the parameters needed to compute the sound pressure signals using the source data saved to files.

Active Source Zones

contains source zones you want to include to compute sound. See Specifying Source Surfaces (p. 2635) for details.

Receivers

contains all the receivers for which you can compute sound.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Source Data Files

contains all the source data files that you can use to compute sound.

Load Index File...

opens the **Select File** dialog box, in which you can select the index file for your computation.

Write

writes the sound pressure data. This button will appear only when **Write Acoustic Signals** is selected under **Options**.

Compute/Write

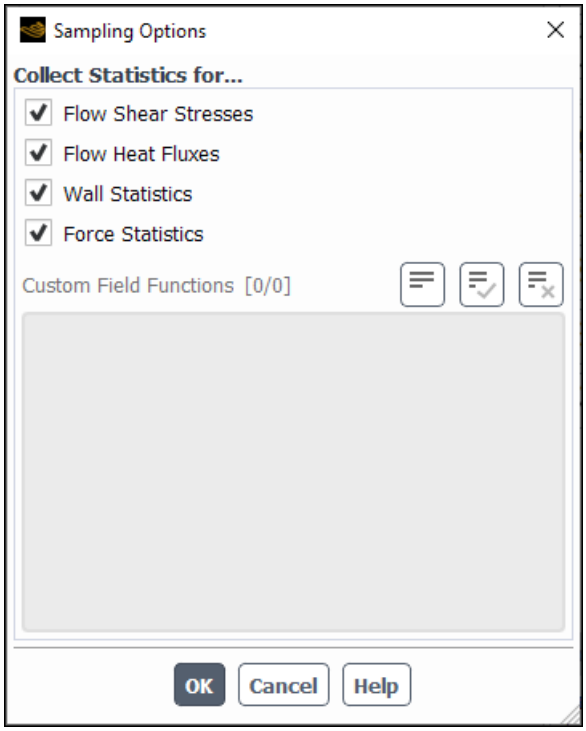
computes and saves the sound pressure data. This button will appear only when **Read Unsteady Acoustic Source Data Files** is selected under **Options**.

Receivers...(button)

opens the Acoustic Receivers Dialog Box (p. 4866) in which you can define additional receivers.

51.16.7. Sampling Options Dialog Box

The **Sampling Options** dialog box allows you to specify a collection of statistics for: **Flow Shear Stresses**, **Flow Heat Fluxes**, **Wall Statistics**, **DPM Variables**, and **Custom Field Functions**.



Controls

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Collect Statistics for...

contains options for the variables you want to be able to postprocess.

Flow Shear Stresses

allows you to enable or disable the flow shear stress statistics for postprocessing.

Flow Heat Fluxes

allows you to enable or disable the flow heat fluxes statistics for postprocessing.

Wall Statistics

allows you to enable or disable the wall statistics for postprocessing.

Force Statistics

allows you to enable or disable the force statistics for postprocessing using cumulative plots (Cumulative Force, Moment, and Coefficients Plots (p. 4094)).

DPM Variables

allows you to enable or disable DPM variable statistics for postprocessing.

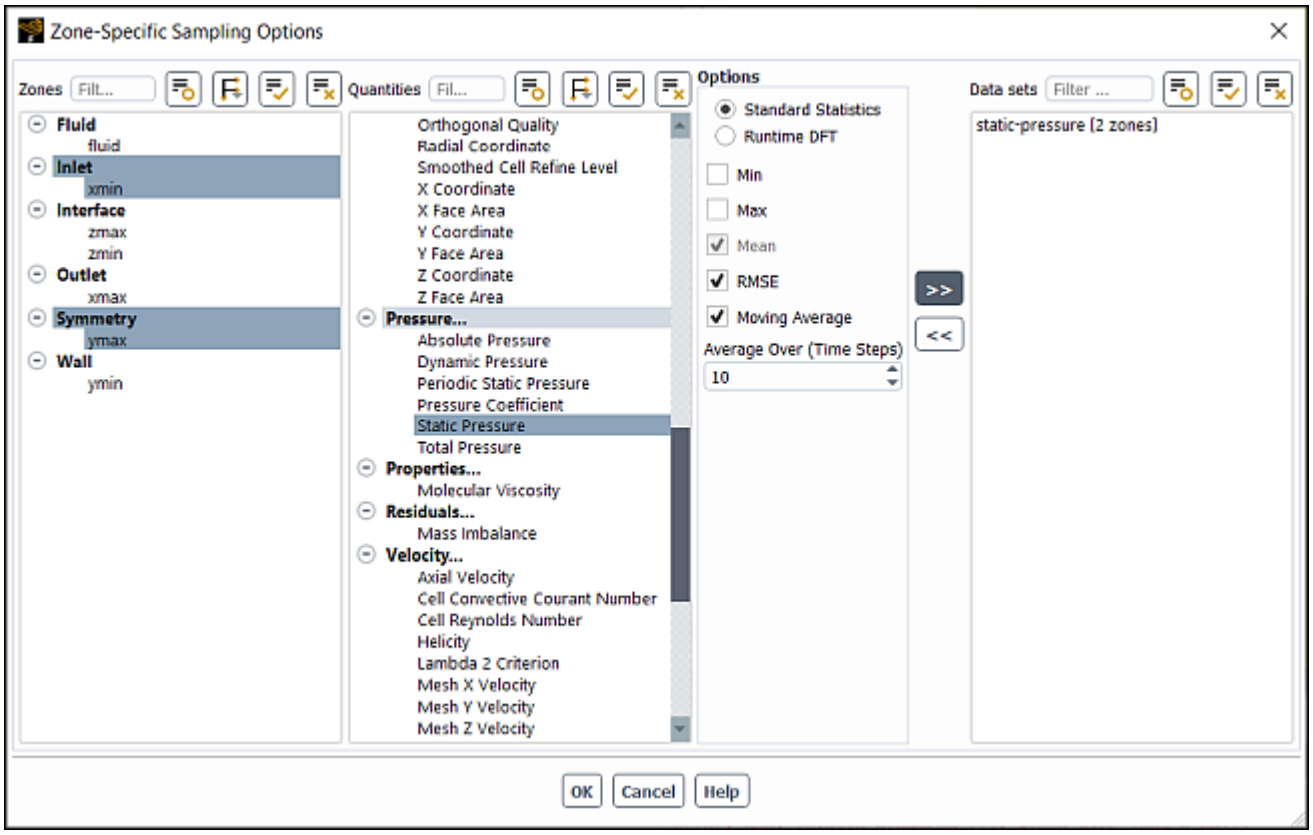
Custom Field Functions

allows you to select the previously defined custom field functions you want to be able to postprocess.

51.16.8. Zone-Specific Sampling Options Dialog Box

The **Zone-Specific Sampling Options** dialog box allows you to specify which statistics are collected and where they are collected. Refer to Inputs for Time-Dependent Problems (p. 3645) and Data Sampling for Steady Statistics (p. 3641) for additional information.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Zones

lists the available locations where statistics can be collected.

Quantities

allows you to specify which statistics are collected on the selected zone(s).

Min

allows you to specify that the minimum value of the selected quantity(s) will be collected.

Max

allows you to specify that the maximum value of the selected quantity(s) will be collected.

Mean

shows that the average value of the selected quantity(s) will be computed and collected (this option is always enabled).

RMSE

allows you to specify that the root mean square error of the selected quantity(s) will be computed and collected.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Moving Average

allows you to specify an interval for averaging of the computed statistics.

Average Over

specifies the number of iterations (steady simulations) or time steps (transient simulations) that will be used for computing the moving average.

Important:

If you want to change the **Average Over** value for a dataset, you must remove it from the **Data sets** list (<<) and re-add the same zones and quantities with the new **Average Over** value.

Runtime DFT

allows you to specify that the runtime discrete Fourier transform of the selected quantity(s) will be computed and collected.

Add

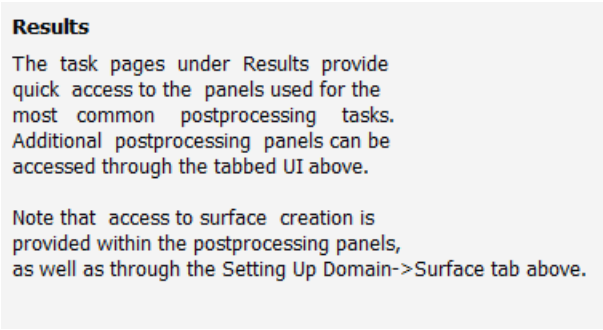
clicking >> allows you to specify which statistics are collected on the selected zone(s).

Remove

clicking << removes the selected **Data Sets**.

51.17. Results Task Page

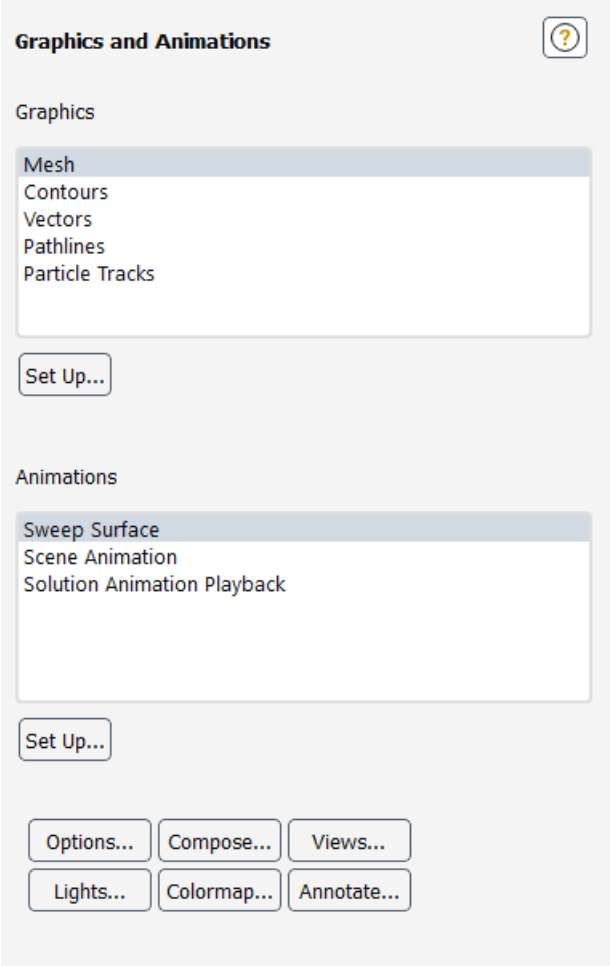
The **Results** task page introduces you to the main tasks involved in setting up and displaying the results of your CFD simulation using Ansys Fluent.



51.18. Graphics and Animations Task Page

The **Graphics and Animation** task page allows you to visualize the results of your CFD simulation by allowing you to set up plots of contours, vectors, pathlines, particle tracks, scene descriptions and animations. See Displaying Graphics (p. 3869) for more information.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Graphics

displays a list of the available graphics objects.

You can double-click an item in the **Graphics** list to open the corresponding dialog box, or you can select the item in the list and click the **Set Up...** button.

Mesh

- selecting this item and clicking the **Set Up...** button opens the Mesh Display Dialog Box (p. 4688).

Contours

- selecting this item and clicking the **Set Up...** button opens the Contours Dialog Box (p. 5412).

Vectors

- selecting this item and clicking the **Set Up...** button opens the Vectors Dialog Box (p. 5655).

Pathlines

- selecting this item and clicking the **Set Up...** button opens the Pathlines Dialog Box (p. 5572).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Particle Tracks

- selecting this item and clicking the **Set Up...** button opens the Particle Tracks Dialog Box (p. 5558).

Set Up...

opens the dialog box corresponding to the selected object in the **Graphics** list.

Animations

displays a list of the available animation objects.

You can double-click an item in the **Animations** list to open the corresponding dialog box, or you can select the item in the list and click the **Set Up...** button.

Sweep Surface

- selecting this item and clicking the **Set Up...** button opens the Sweep Surface Dialog Box (p. 5250).

Scene Animation

- selecting this item and clicking the **Set Up...** button opens the Animate Dialog Box (p. 5251).

Solution Animation

- selecting this item and clicking the **Set Up...** button opens the Playback Dialog Box (p. 5257).

Set Up...

opens the dialog box corresponding to the selected object in the **Animations** list.

Options...

opens the Display Options Dialog Box (p. 5262).

Scene...

opens the Scene Description Dialog Box (p. 5265).

Views...

opens the Views Dialog Box (p. 5272).

Lights...

opens the Lights Dialog Box (p. 5279).

Colormap...

opens the Colormap Dialog Box (p. 5280).

Annotate...

opens the Annotate Dialog Box (p. 5286).

For additional information, see the following sections:

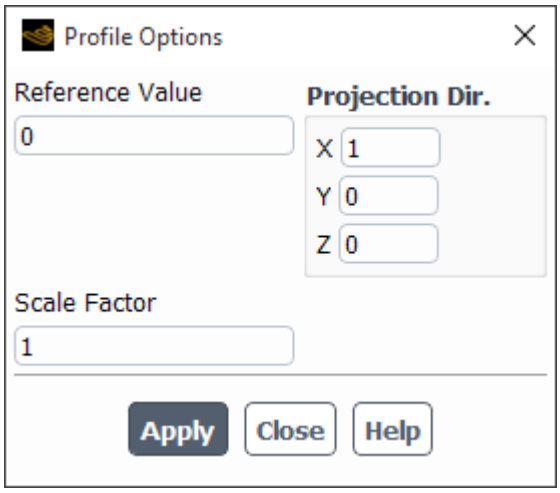
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

51.18.1. Profile Options Dialog Box
51.18.2. Vector Options Dialog Box
51.18.3. Custom Vectors Dialog Box
51.18.4. Vector Definitions Dialog Box
51.18.5. Path Style Attributes Dialog Box
51.18.6. Ribbon Attributes Dialog Box
51.18.7. Particle Filter Attributes Dialog Box
51.18.8. Reporting Variables Dialog Box
51.18.9. Track Style Attributes Dialog Box
51.18.10. Particle Sphere Style Attributes Dialog Box
51.18.11. Particle Vector Style Attributes Dialog Box
51.18.12. Sweep Surface Dialog Box
51.18.13. Create Surface Dialog Box
51.18.14. Animate Dialog Box
51.18.15. Save Picture Dialog Box
51.18.16. Playback Dialog Box
51.18.17. Video Options Dialog Box
51.18.18. Advanced Video Quality Options Dialog Box
51.18.19. Display Options Dialog Box
51.18.20. Scene Description Dialog Box
51.18.21. Display Properties Dialog Box
51.18.22. Transformations Dialog Box
51.18.23. Iso-Value Dialog Box
51.18.24. Pathline Attributes Dialog Box
51.18.25. Bounding Frame Dialog Box
51.18.26. Views Dialog Box
51.18.27. Write Views Dialog Box
51.18.28. Mirror Planes Dialog Box
51.18.29. Periodic Instancing Dialog Box
51.18.30. Camera Parameters Dialog Box
51.18.31. Lights Dialog Box
51.18.32. Colormap Dialog Box
51.18.33. Colormap Editor Dialog Box
51.18.34. Annotate Dialog Box

51.18.1. Profile Options Dialog Box

The **Profile Options** dialog box controls the scaling and projection direction of profiles. It is opened from the Contours Dialog Box (p. 5412), and you will display the profiles using the **Display** button in that dialog box. See Displaying Contours and Profiles (p. 3896) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Reference Value

sets the "zero height" reference value for the profile. Any point on the profile with a value equal to the **Reference Value** will be plotted exactly on the defining surface. Values greater than the **Reference Value** will be projected ahead of the surface (in the direction of **Projection Dir.** and scaled by **Scale Factor**), and values less than the **Reference Value** will be projected behind the surface.

Scale Factor

sets the length scale factor for projection. After subtracting off the **Reference Value**, Ansys Fluent multiplies the resulting solution value by the **Scale Factor** to form a length.

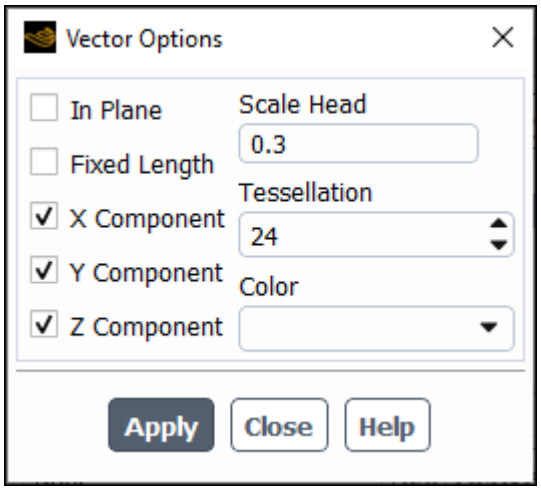
Projection Dir.

sets the direction in which profiles are projected. In 2D, for example, a contour plot of pressure on the entire domain can be projected in the z direction to form a carpet plot, or a contour plot of x velocity on a sequence of x-coordinate slice lines can be projected in the x direction to form a series of velocity profiles.

51.18.2. Vector Options Dialog Box

The **Vector Options** dialog box allows you to set additional parameters for vector displays. It is opened from the Vectors Dialog Box (p. 5655). See Vector Plot Options (p. 3914) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

In Plane

toggles the display of vector components in the plane of the surface selected for display. This feature is useful for visualizing components that are normal to the flow. See Drawing Vectors in the Plane of the Surface (p. 3916) for details.

Fixed Length

enables the display of vectors that are all the same length. See Displaying Fixed-Length Vectors (p. 3916) for details.

X, Y, Z Component

toggle the display of the Cartesian components of the vectors. See Displaying Vector Components (p. 3916) for details.

Scale Head

controls the size of the arrowhead on vector styles that include heads.

Tessellation

controls the tessellation of the **3d arrow** and **3d arrowhead** vector styles. The smaller the tessellation value, the better the performance of the vector plot display. Valid inputs are in the range of `0-74`.

Color

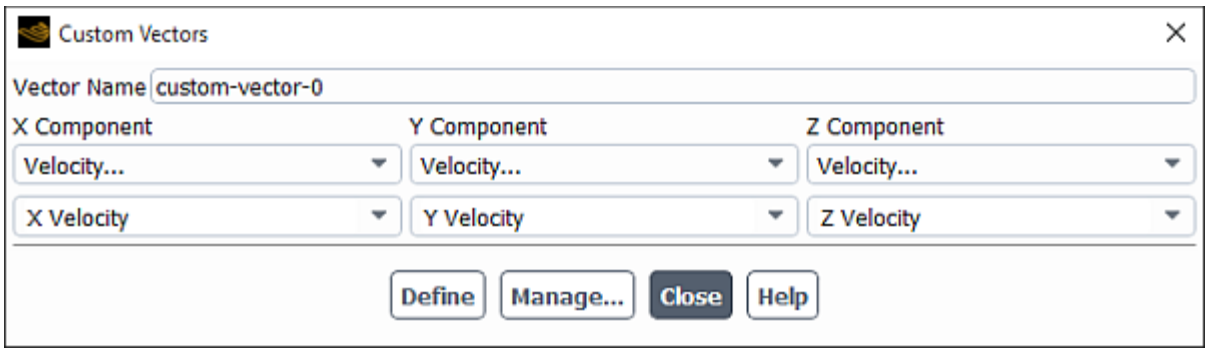
specifies a single color for the display of all vectors. See Displaying Vectors Using a Single Color (p. 3918) for details.

51.18.3. Custom Vectors Dialog Box

The **Custom Vectors** dialog box allows you to define custom vectors based on existing quantities. Any vectors that you define will be added to the **Vectors of** list in the Vectors Dialog Box (p. 5655). To

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

open the **Custom Vectors** dialog box, click **Custom Vectors...** in the **Vectors** dialog box. See Creating and Managing Custom Vectors (p. 3919) for details about custom vectors.



Controls

Vector Name

specifies the name of the vector you are defining. Should you decide to change the name after you have defined the vector, you can do so in the Vector Definitions Dialog Box (p. 5241), which you can open by clicking on the **Manage...** button.

X, Y, Z Component

specify the x, y, and z components of the vector. Each drop-down list contains the available field functions.

Define

creates the vector and adds it to the **Vectors of** list in the Vectors Dialog Box (p. 5655).

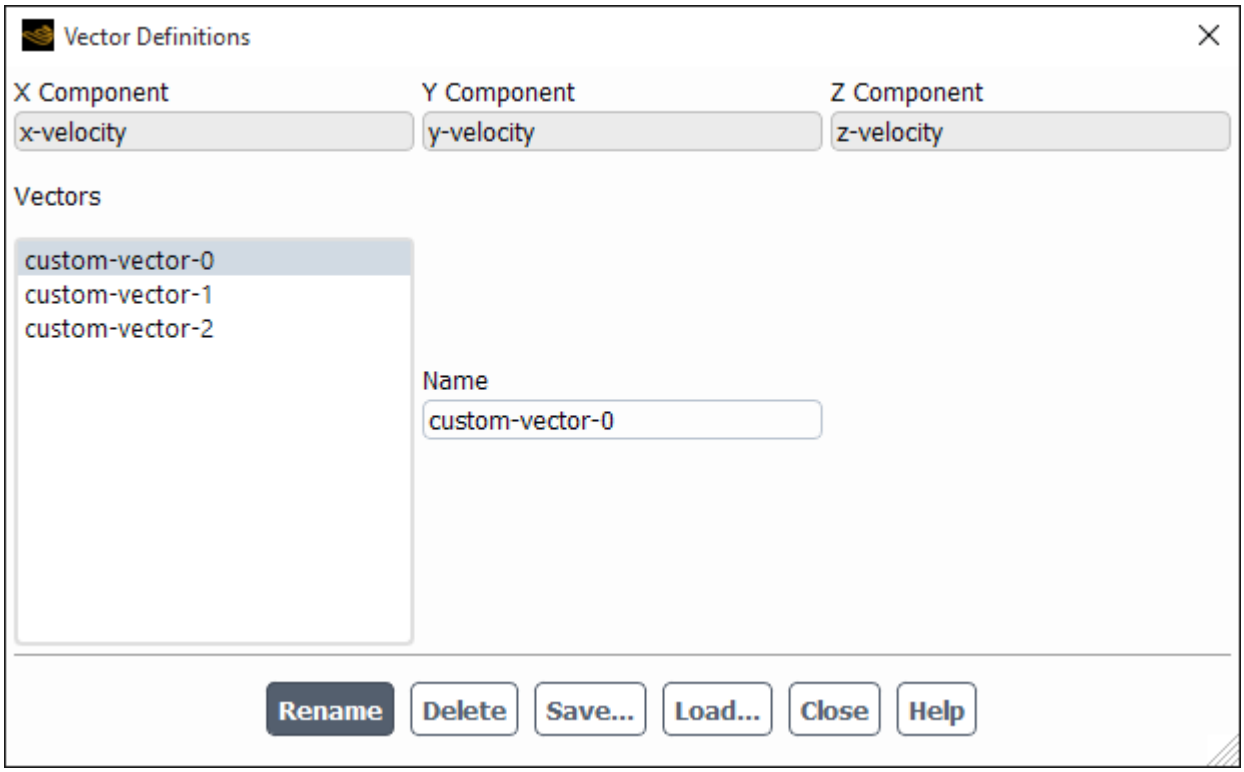
Manage...

opens the Vector Definitions Dialog Box (p. 5241), which enables you to check, rename, save, load, and delete custom vectors.

51.18.4. Vector Definitions Dialog Box

The **Vector Definitions** dialog box allows you to check, rename, save, load, and delete custom vectors that you defined in the Custom Vectors Dialog Box (p. 5240). See Creating and Managing Custom Vectors (p. 3919) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

X, Y, Z Component

display the x, y, and z components of the vector.

Vectors

contains a selectable list of custom vectors. When you select a vector, its components will appear in the **X**, **Y**, and **Z Component** fields, and its name will appear in the **Name** field.

Name

displays the name of the currently selected vector. You can enter a new name in this box if you want to rename the vector.

Rename

changes the name of the selected function to the name specified in the **Name** field.

Delete

deletes the selected vector.

Save...

opens The Select File Dialog Box (p. 931), in which you can specify a file in which to save all of the custom vectors in the **Vectors** list.

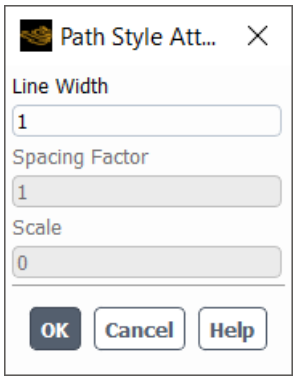
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Load...

opens the **Select File** dialog box, in which you can specify a file from which to read custom vectors (a file that you saved using the **Save...** button above).

51.18.5. Path Style Attributes Dialog Box

To modify the line width, cylinder radius or marker size, use the **Path Style Attributes** dialog box. You can open this dialog box by clicking the **Attributes...** button in the Pathlines Dialog Box (p. 5572). See Controlling the Pathline Style (p. 3925) for details about the items below.



Controls

Line Width/Marker Size/Width

determines the thickness of the pathlines.

Diameter

specifies the diameter of the sphere. This parameter appears only when **sphere** is selected under **Style** in the **Pathlines** dialog box.

Spacing Factor

controls the spacing of arrows when you use the **line-arrows** style.

Scale

controls the size of the arrow heads when you use the **line-arrows** style.

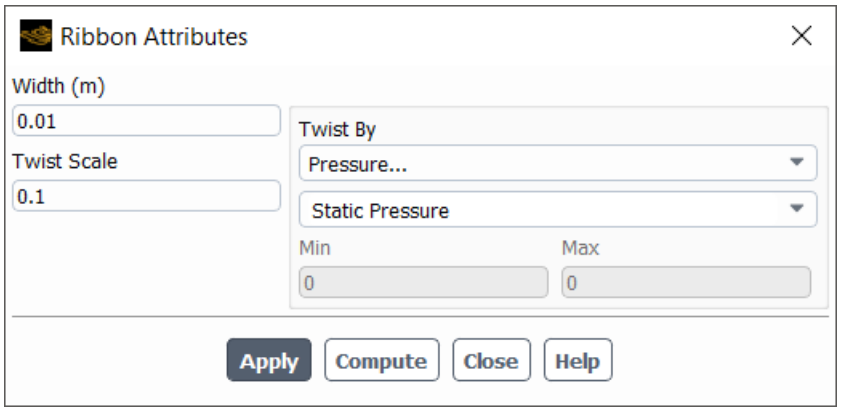
Detail

specifies the detail applied to the graphical rendering of the spheres. This parameter appears only when **sphere** is selected under **Style** in the **Pathlines** dialog box.

51.18.6. Ribbon Attributes Dialog Box

To modify the ribbon width and set the scalar field by which to twist the ribbon, use the **Ribbon Attributes** dialog box. You can open this dialog box by clicking the **Attributes...** button in the Pathlines Dialog Box (p. 5572) when the selected **Style** is ribbon. See Controlling the Pathline Style (p. 3925) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Width

determines the thickness of the ribbon.

Twist Scale

sets the amount of twist for a given field. To magnify the twist for a field with very little change, increase this factor; to display less twist for a field with dramatic changes, decrease this factor.

Twist By

contains a drop-down list from which you can select a scalar field on which pathline twisting is based.

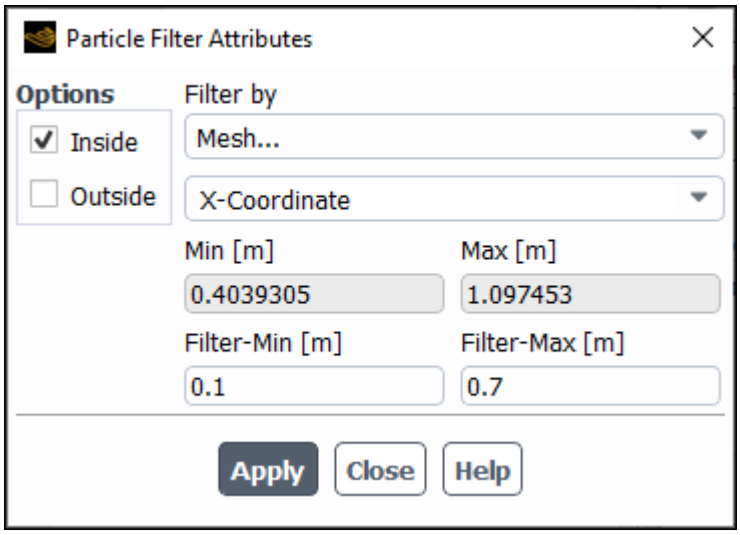
Min/Max

displays the minimum/maximum value of the scalar field selected in **Twist By**.

51.18.7. Particle Filter Attributes Dialog Box

The **Particle Filter Attributes** dialog box allows you to specify how you would like to filter the particles being displayed. This dialog box opens when you click **Filter by...** in Particle Tracks Dialog Box (p. 5558). See Particle Filtering (p. 2845) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options

contains the filtering options.

Inside

enables the filtering of particles with values between **Filter-Min** and **Filter-Max**.

Outside

enables the filtering of particles with values less than **Filter-Min** or greater than **Filter-Max**.

Filter by

contains a list from which you can select any field variable, except for **Custom Field Functions...**, to be used as a filter variable.

Min/Max

displays the minimum and maximum values of the selected field variable. The real number field values are not editable; they are purely informational.

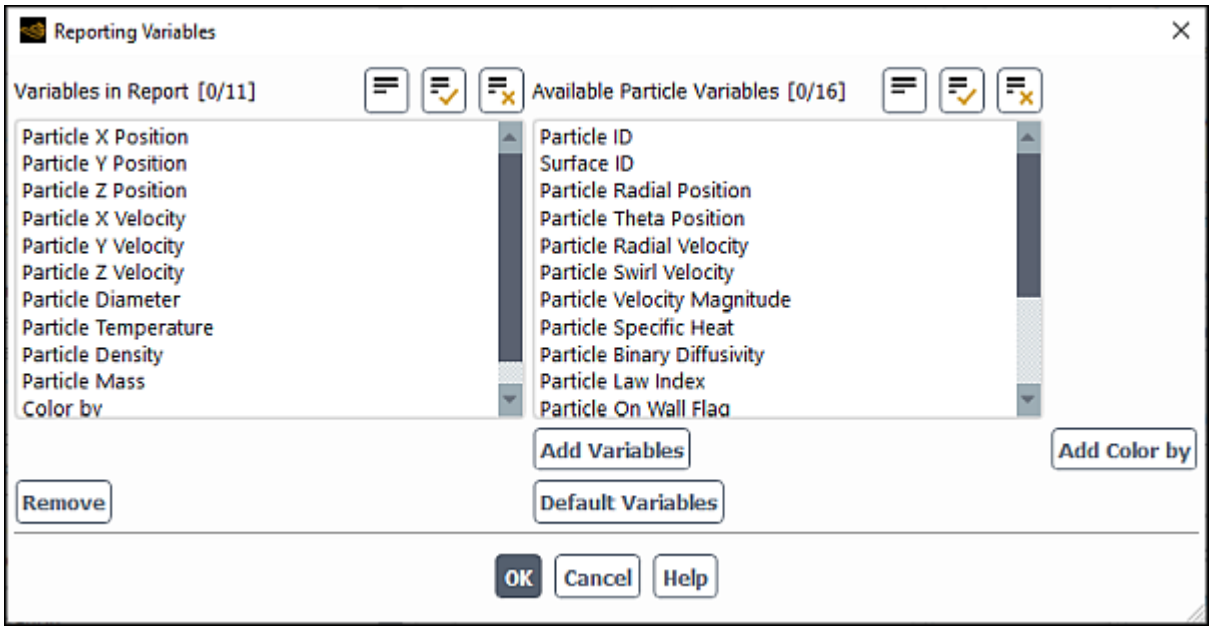
Filter-Min/Filter-Max

defines the minimum/maximum filter threshold.

51.18.8. Reporting Variables Dialog Box

The **Reporting Variables** dialog box allows you to control the particle variables that you include in your reporting. See Step-by-Step Reporting of Trajectories (p. 2856) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Variables in Report

contains all variables currently in the report.

Remove

removes the selected variable from the report.

Default Variables

restores the default list.

Available Particle Variables

contains the particle variables that are available for you to select.

Add Variables

takes the selected variable from the **Available Particle Variables** list and adds it to the **Variables in Report** list.

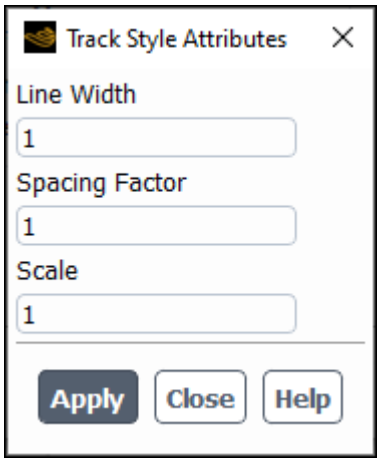
Add Color By

adds the **Color by** variable to the **Variables in Report** list.

51.18.9. Track Style Attributes Dialog Box

To modify the line width, cylinder radius or marker size, use the **Track Style Attributes** dialog box. You can open this dialog box by clicking the **Attributes...** button in the Particle Tracks Dialog Box (p. 5558). See Controlling the Particle Tracking Style (p. 2838) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Line Width/Marker Size/Width

determines the thickness of the particle tracks.

Spacing Factor

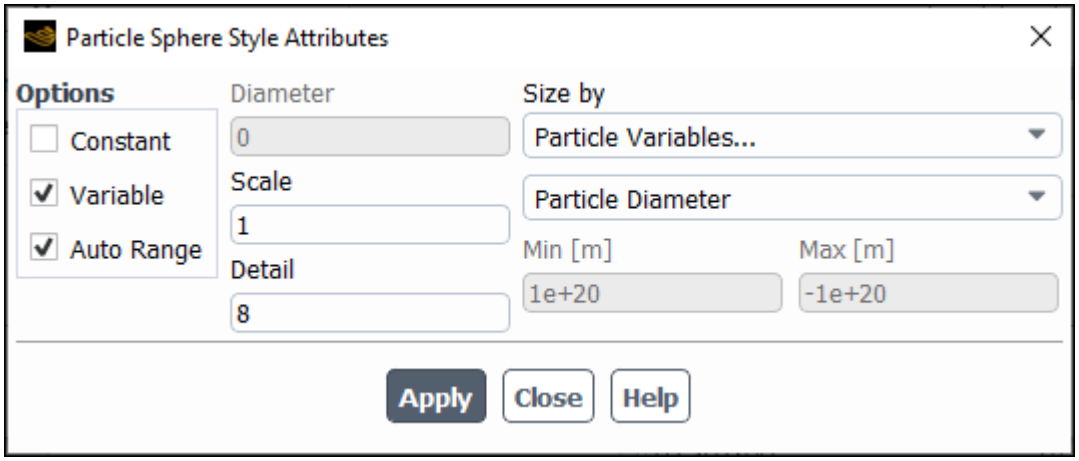
controls the spacing of arrows when you use the **line-arrows** style.

Scale

controls the size of the arrow heads when you use the **line-arrows** style.

51.18.10. Particle Sphere Style Attributes Dialog Box

To modify the attributes of the particle sphere, use the **Particle Sphere Style Attributes** dialog box. Select **sphere** from the **Track Style** drop-down list and click the **Attributes...** button in the Particle Tracks Dialog Box (p. 5558) to open the **Particle Sphere Style Attributes** dialog box. See Controlling the Particle Tracking Style (p. 2838) for details about the items below.



Controls

Options

allows you to choose how you would like to specify the particle diameter.

Constant

allows you to specify the diameter as a constant value.

Variable

allows you to select a particle variable to estimate the size of the spheres.

Auto Range

when disabled clips the displayed particles to the values given in **Min** and **Max**.

Diameter

specifies the diameter of the sphere.

Scale

allows you to scale the spheres by the factor entered in this field.

Detail

specifies the detail applied to the graphical rendering of the spheres.

Size by

contains a list of variables by which you can estimate the size of your particle. This list is selectable only when you are using the **Variable** option.

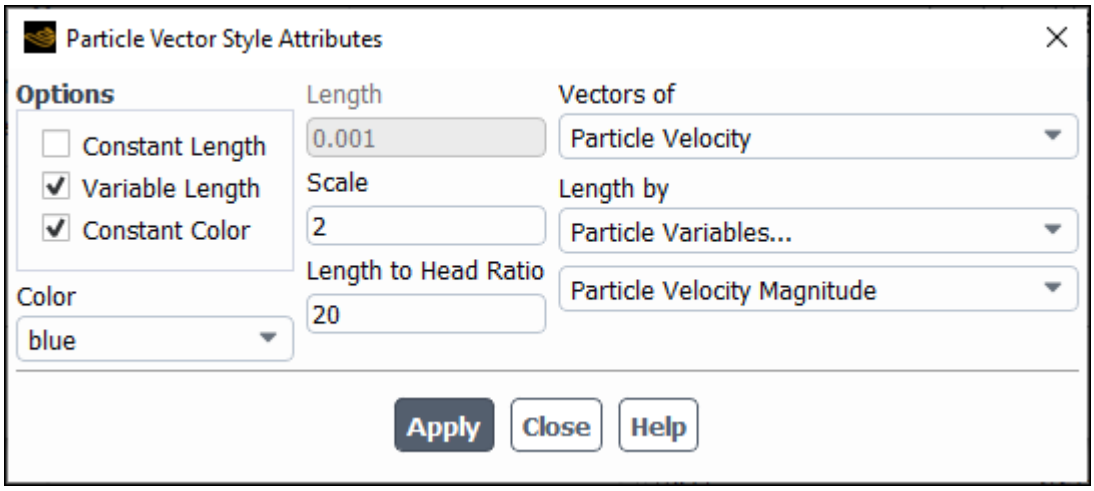
Min/Max

defines the minimum and maximum values of the selected field variable to display.

51.18.11. Particle Vector Style Attributes Dialog Box

To modify the attributes of the vector styles, use the **Particle Vector Style Attributes** dialog box. See Controlling the Vector Style of Particle Tracks (p. 2841) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options

allows you to choose how you would like to specify the particle diameter.

Constant Length

allows you to specify the vector length as a constant value.

Variable Length

results in a vector length that is based on the variable selected under **Length by**.

Constant Color

when enabled allows you to select a vector color from the **Color** drop-down list. Otherwise, the vector is colored based on the variable selected in the **Particle Tracks** dialog box (seen in the **Mesh Colors** dialog box when **Draw Mesh** is enabled).

Length

specifies the length of the vector.

Scale

allows you to scale the vectors by the factor entered in this field.

Length to Head Ratio

is the ratio of vector length to vector head size.

Vectors of

contains the particle vector variable to display.

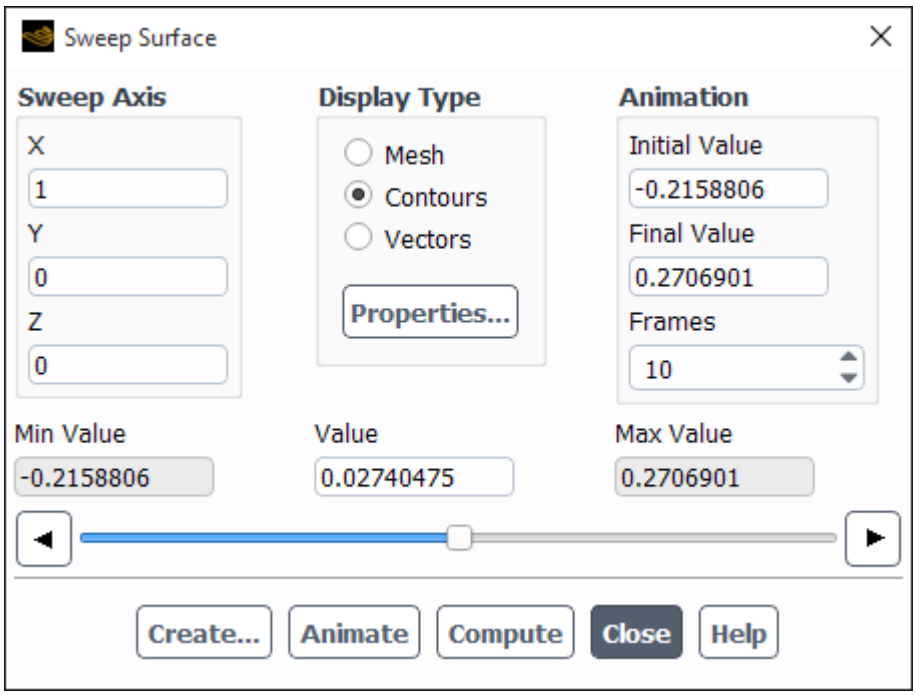
Length by

is used to estimate the length of the vector when the **Variable Length** option is enabled.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

51.18.12. Sweep Surface Dialog Box

The **Sweep Surface** dialog box controls the display and animation of mesh, contour, and vector plots generated on a sweep surface. See Displaying Results on a Sweep Surface (p. 3937) for details about the items below.



Controls

Sweep Axis

specifies the (**X**, **Y**, **Z**) vector representing the axis along which the surface should be swept.

Display Type

specifies the type of display to be swept through the domain (**Mesh**, **Contours**, or **Vectors**).

Properties...

opens the Contours Dialog Box (p. 5412) if **Contours** is the selected **Display Type**, or the Vectors Dialog Box (p. 5655) if **Vectors** is the selected **Display Type**. (This button is not available if **Mesh** is the selected **Display Type**.)

Animation

contains controls for animating the sweep-surface display.

Initial Value, Final Value

specify the initial and final positions for the animation.

Frames

specifies the number of frames in the animation.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Min Value, Max Value

show the minimum and maximum extents of the domain along the specified **Sweep Axis**. These values are updated when you click **Compute**.

Value

shows the current position at which the requested display is plotted. You can change the value by moving the slide bar below it, or by entering a new value and pressing the <RETURN> key.

Create...

opens the Create Surface Dialog Box (p. 5251), where you can create a surface from the currently-displayed sweep surface.

Animate

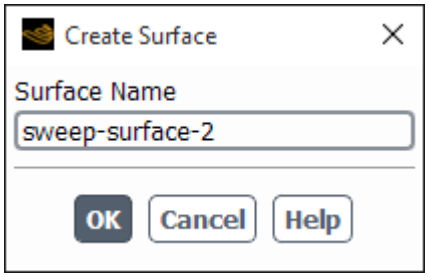
animates the display, sweeping the requested display through the domain along the specified axis.

Compute

updates the **Min Value** and **Max Value** to reflect the minimum and maximum extents of the domain along the specified **Sweep Axis**.

51.18.13. Create Surface Dialog Box

The **Create Surface** dialog box allows you to save a sweep surface for later use. You can open it by clicking **Create...** in the Sweep Surface Dialog Box (p. 5250). See Displaying Results on a Sweep Surface (p. 3937) for details about the items below.



Controls

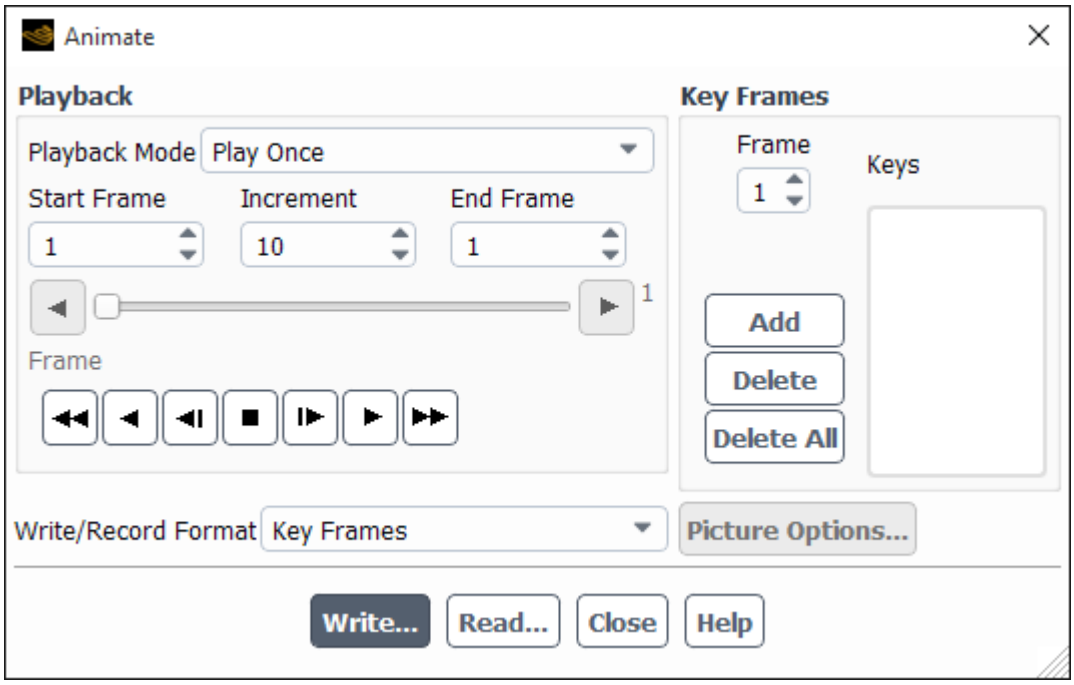
Surface Name

specifies a name for the surface to be created.

51.18.14. Animate Dialog Box

The **Animate** dialog box allows you to specify key frames that define the basic movement of an animated sequence, and then play back the animation. Ansys Fluent interpolates between your specified key frames. See Animating Graphics (p. 4051) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Playback

contains the controls that you use to play back the animation. See Playing an Animation (p. 4054) for details.

Playback Mode

contains a drop-down list of playback options.

Play Once

sets the option to play back frames from **Start Frame** to **End Frame** once.

Auto Repeat

sets the option to continually play back frames from **Start Frame** to **End Frame**.

Auto Reverse

sets the option to continually play back the images while reversing playback direction after each set.

Start Frame, End Frame

set the frames at which the animation should begin and end. By changing these numbers you can view a subset of the frames.

Increment

sets the number of frames to increment the frame-counter by when you use the fast-forward or fast-reverse buttons.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Frame

shows the number of the frame that is currently displayed, as well as its relative position in the entire animation. If you slide the bar to a different location, the frame corresponding to the new frame number will be displayed in the graphics window.

(Tape Player Buttons)

allow you to play the animation forward and backward, fast-reverse and fast-forward the animation, and stop it. The buttons function in a way similar to those on a standard video cassette player.

Key Frames

contains the controls that you use to define the key frames for the animation. See Creating an Animation (p. 4053) for details.

Frame

sets the number to be assigned to the next key frame added to the list of **Keys**.

Keys

contains a list of the key frames that have been defined. If you select a key frame in this list, the associated scene will be displayed in the graphics window.

Add

creates a key frame with the number shown in **Frame** for the scene currently displayed in the graphics window.

Important:

Be sure to change the frame number before you add the new key frame so that you will not overwrite the last key frame that you created.

Delete

deletes the key frame that is selected in the **Keys** list.

Delete All

deletes all key frames in the **Keys** list.

Write/Record Format

specifies **Key Frames**, **Picture Files**, **MPEG**, or **Video** (not available on Windows) as the format in which to save the animation. See Saving an Animation (p. 4055) for details about these options.

Picture Options...

opens the Save Picture Dialog Box (p. 5254), in which you can specify parameters for saving the animation to picture files. This button is available only when **Picture Files** is selected as the **Write/Record Format**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Write...

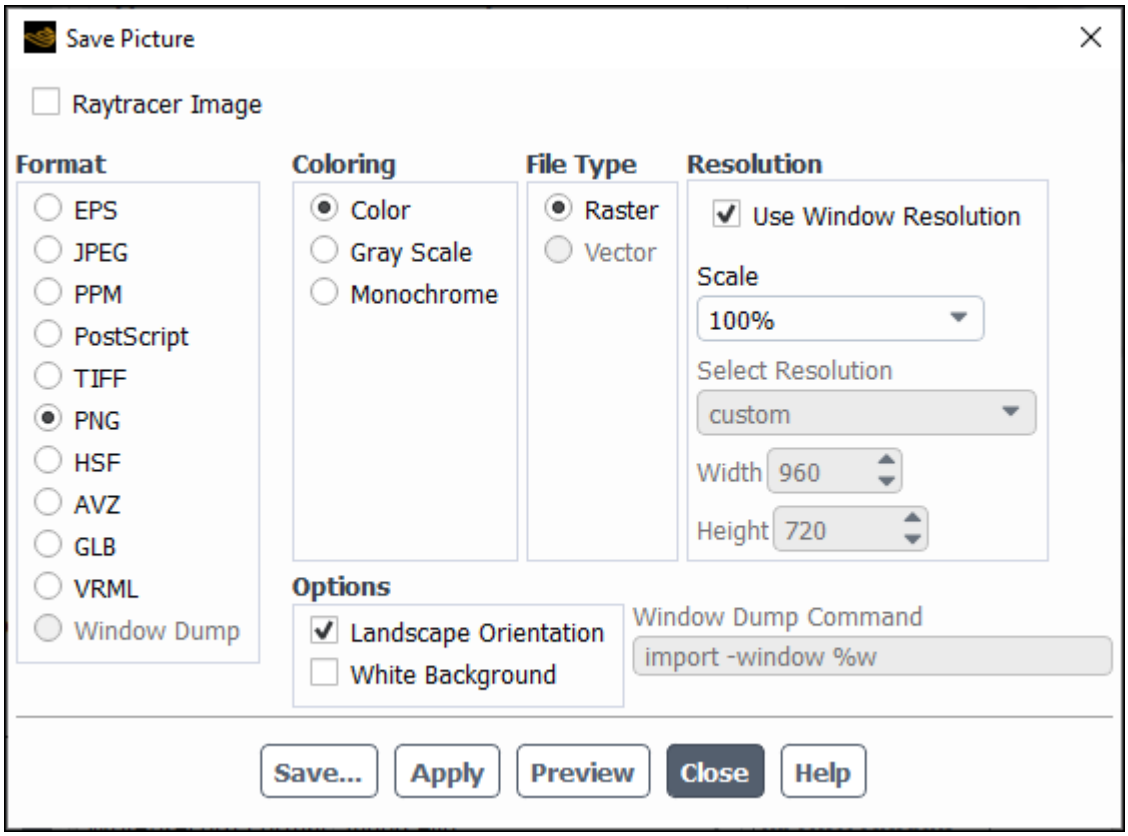
opens The Select File Dialog Box (p. 931), in which you can specify a name for the animation file and save it.

Read...

opens the **Select File** dialog box, in which you can specify the name of the animation file to be read. Note that the current case and data should contain the surfaces and any other information that the key frame description refers to. See Reading an Animation File (p. 4056).

51.18.15. Save Picture Dialog Box

The **Save Picture** dialog box allows you to set save picture parameters and save picture files of graphics windows. See Saving Picture Files (p. 1019) for details on the use of this dialog box.



Controls

Raytracer Image

Saves the current image using a raytracing renderer for enhanced image quality. Refer to Realistic Rendering Using Raytracing (p. 3995) for additional information on raytracing.

Format

allows you to select the format of picture files.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

EPS

(Encapsulated PostScript) output is the same as PostScript output, with the addition of Adobe Document Structuring Conventions (v2) statements. Currently, no preview bitmap is included in EPS output. Often, programs that import EPS files use the preview bitmap to display on-screen, although the actual vector PostScript information is used for printing (on a PostScript device). You can save EPS files in raster or vector format.

JPEG

is a common raster file format.

PPM

output is a common raster file format.

PostScript

is a common vector file format. You can also choose to save a PostScript file in raster format.

TIFF

is a common raster file format.

PNG

is a common raster file format.

HSF

is HOOPS Visualize Stream Format, a highly-compressible and streamable 2D/3D file format.

AVZ

is the free Ansys Viewer Format, which is a 3D file format allowing you to visualize, share, collaborate, and interactively manipulate the displayed object. For additional information on the Ansys Viewer, refer to Ansys Viewer User's Guide.

GLB

is a 3D file format frequently used in virtual reality (VR) and augmented reality (AR) applications. The file size is relatively small as it is a binary format of glTF.

VRML

is a graphics interchange format that allows export of 3D geometrical entities that you can display in the Ansys Fluent graphics window. This format can commonly be used by VR systems and in particular the 3D geometry can be viewed and manipulated in a web-browser graphics window.

Important:

Non-geometric entities such as text, titles, color bars, and orientation axis are not exported. In addition, most display or visibility characteristics set in Ansys Fluent,

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

such as lighting, shading method, transparency, face and edge visibility, and outer face culling, are not explicitly exported but are controlled by the software used to view the VRML file.

Window Dump

(Linux systems only) selects a window dump operation for generating the picture. With this format, you will need to specify the appropriate **Window Dump Command**.

Coloring

(all formats except **Window Dump**) specifies the color mode for the picture file.

Color

specifies a color-scale copy.

Gray Scale

specifies a gray-scale copy.

Monochrome

specifies a black-and-white copy.

Important:

Most monochrome PostScript devices will render **Color** images in shades of gray, but to ensure that the color ramp is rendered as a linearly-increasing gray ramp, you should select **Gray Scale**.

File Type

specifies the type of picture file to be saved. See Choosing the File Type (p. 1024) for details.

Raster

specifies a raster type picture. The supported raster formats are **EPS**, **JPEG**, **PPM**, **PostScript**, **TIFF**, and **PNG**.

Vector

specifies a vector type picture. The supported vector formats are **EPS**, **PostScript**, and **VRML**.

Resolution

specifies the resolution or the size (in pixels) of the picture.

Use Window Resolution

uses the resolution of the current graphics window when the image is saved.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

DPI

specifies the resolution of **EPS** and **PostScript** files in dots per inch (DPI). The default value for **DPI** is set to 75.

Width

specifies the width of the raster picture image.

Height

specifies the height of the raster picture image.

*The default value for **Width** and **Height** is set to zero, so that the default picture is generated at the same resolution as the active graphics window.*

Options

contains additional options for all picture formats except **Window Dump**.

Landscape Orientation

specifies the orientation of the picture. If this option is enabled, the picture is made in landscape mode; otherwise, it is made in portrait mode.

White Background

controls the foreground/background color. If this option is enabled, the picture is saved with a white background and, for 2D plots, the axes and labels are saved in black.

Window Dump Command

(Linux systems only) specifies the command to be used to save the picture file, when you select the **Window Dump** format. See Window Dumps (Linux Systems Only) (p. 1025) for details.

Save...

opens The Select File Dialog Box (p. 931), in which you can specify a name for the picture file to be saved and then save the file. The resulting file will contain a picture of the active graphics window.

Apply

saves the current settings. Ansys Fluent will use these settings when making subsequent pictures.

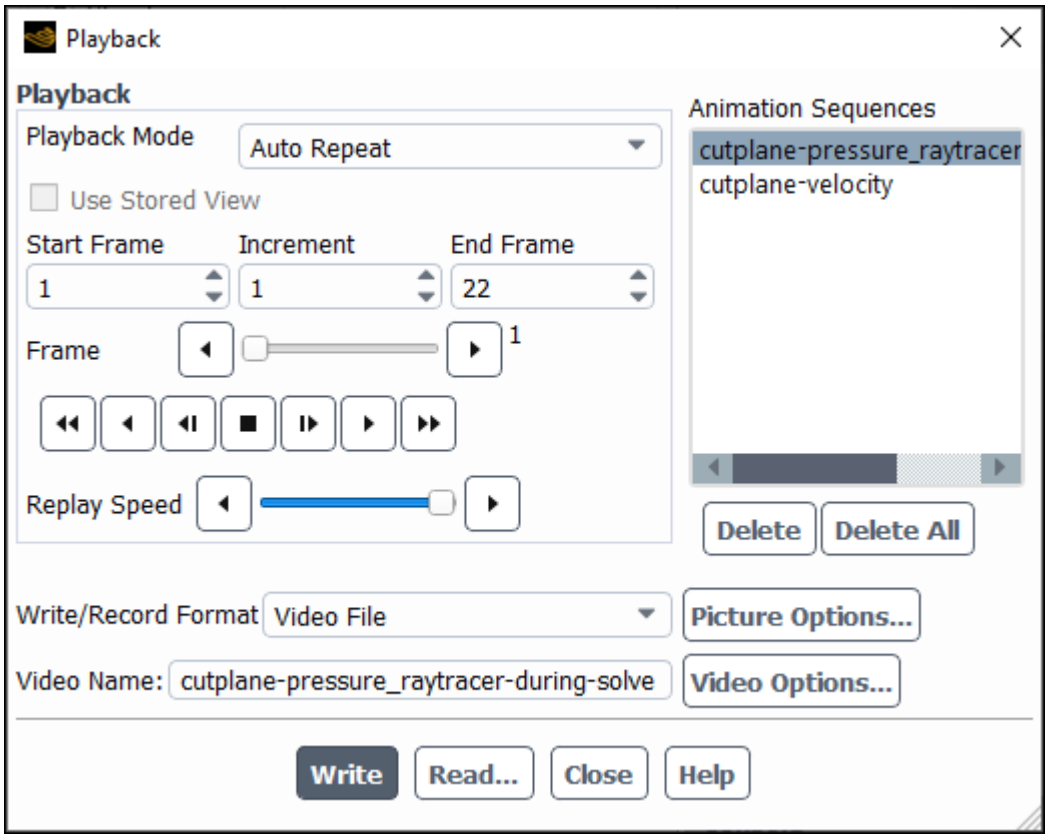
Preview

applies the current settings to the active graphics window so that you can investigate the effects of different options interactively before saving the final picture.

51.18.16. Playback Dialog Box

The **Playback** dialog box allows you to play back an animation sequence. See Playing an Animation Sequence (p. 3717) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Playback

contains the controls that you use to play back the selected animation sequence.

Playback Mode

contains a drop-down list of playback options.

Play Once

sets the option to play back frames from **Start Frame** to **End Frame** once.

Auto Repeat

sets the option to continually play back frames from **Start Frame** to **End Frame**.

Auto Reverse

sets the option to continually play back the images while reversing playback direction after each set.

Use Stored View

disabling allows you to modify the view during animation playback (**In Memory** and **HSF File** 3D animations only).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Start Frame, End Frame

set the frames at which the animation should begin and end. By changing these numbers you can view a subset of the frames.

Increment

sets the number of frames to increment the frame-counter by when you use the fast-forward or fast-reverse buttons.

Frame

shows the number of the frame that is currently displayed, as well as its relative position in the entire animation. If you slide the bar to a different location, the frame corresponding to the new frame number will be displayed in the graphics window.

(Tape Player Buttons)

allow you to play the animation forward and backward, fast-reverse and fast-forward the animation, and stop it. The buttons function in a way similar to those on a standard video cassette player.

Replay Speed

controls the playback speed for the animation. Move the **Replay Speed** slider bar to the left to reduce the playback speed (and to the right to increase it).

Animation Sequences

contains the controls that you use to define the sequence to be played back.

Sequences

contains a list of the animation sequences that have been defined.

Delete

deletes the animation sequence that is selected in the **Sequences** list.

Delete All

deletes all animation sequences in the **Sequences** list.

Write/Record Format

specifies **Animation Frames**, **Picture Files**, or **Video File** as the format in which to save the animation. See Saving an Animation Sequence (p. 3720) for details about these options.

Picture Options...

opens the Save Picture Dialog Box (p. 5254) where you can specify parameters for saving the animation to picture files. This button is available only when **Picture Files** is selected as the **Write/Record Format**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Video Options...

opens the Video Options Dialog Box (p. 5260) where you can specify parameters for saving the animation to video files. This button is available only when **Video File** is selected as the **Write/Record Format**.

Video Name

is the name that the video file will be saved with. This field is available only when **Video File** is selected as the **Write/Record Format**.

Write

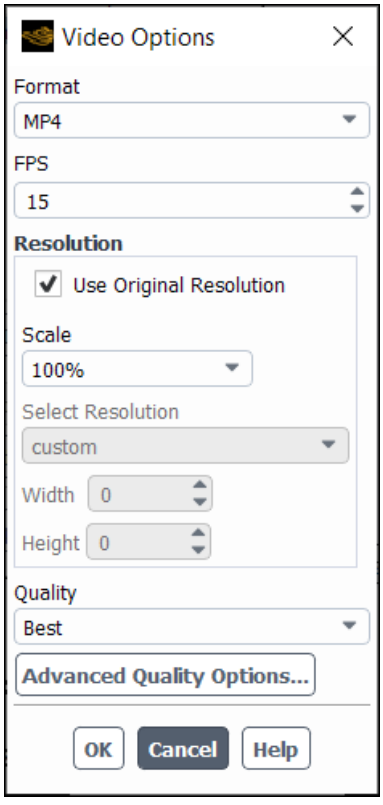
saves the specified file(s) in the current working directory in many cases. However, the animation may be written at the same location as the animation sequence file.

Read...

opens The Select File Dialog Box (p. 931), in which you can specify the name of the solution animation file to be read. See Reading an Animation Sequence (p. 3725) for details.

51.18.17. Video Options Dialog Box

The **Video Options** dialog box allows you control how an animation file is saved in video format. See Video File (p. 3722) for additional information.



Controls

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Format

specifies the format of the saved video file.

FPS

specifies the target frames per second (FPS) for the saved video file.

Resolution

contains the settings that control the resolution of the saved video file.

Use Original Resolution

saves the video file in the same resolution as its comprising images (2D formats—ppm, png, jpeg, tiff) or using the current graphics window size (HSF, in-memory).

Scale

control the scale of the saved video file.

Select Resolution

contains common pre-defined video resolutions. *This option is only available when* ***Use Original Resolution*** *is disabled*.

Width

displays the width, in pixels, of the saved video file. *This option is only available and populated when* ***Use Original Resolution*** *is disabled*.

Height

displays the height, in pixels, of the saved video file. *This option is only available and populated when* ***Use Original Resolution*** *is disabled*.

Quality

specifies the quality of the saved video file *which may affect the final file size in addition to the video picture quality.*.

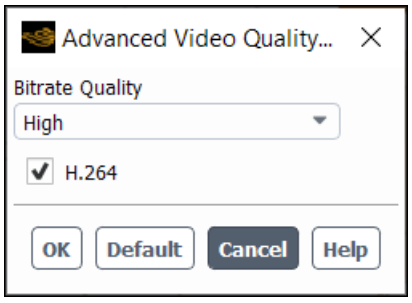
Advanced Quality Options...

opens the Advanced Video Quality Options Dialog Box (p. 5261) where you can access advanced controls for the video file quality.

51.18.18. Advanced Video Quality Options Dialog Box

The **Advanced Video Quality Options** dialog box allows you to control additional options for the quality of a saved animation video file. Refer to Video File (p. 3722) for additional information.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Bitrate Quality

specifies the bitrate quality of the saved video file.

H.264

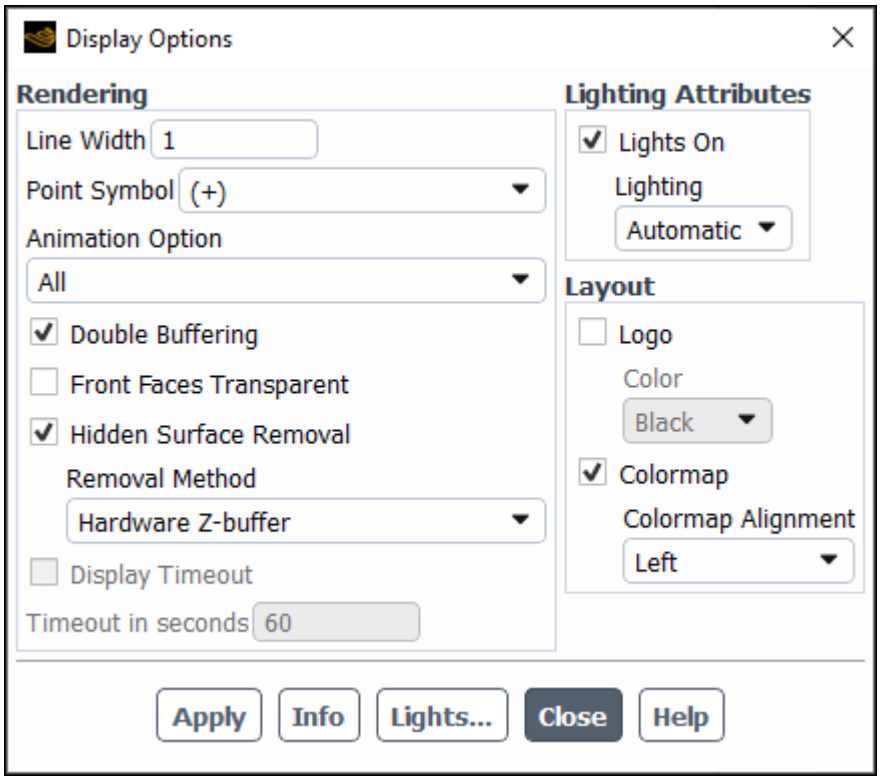
uses the H.264 compression technique when writing the video file.

Default

resets the fields to the default settings.

51.18.19. Display Options Dialog Box

The **Display Options** dialog box provides an interactive mechanism for setting attributes or options that control how and where a scene is rendered.



Controls

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Rendering

allows you to modify characteristics of the display that are related to the way in which scenes are rendered. See Modifying the Rendering Options (p. 3987) for details about these items.

Line Width

controls the thickness of lines. The default is 1.

Point Symbol

sets the symbol used for nodes and data points.

Animation Option

contains a drop-down list of animation options: **All** and **Wireframe**. **Wireframe** uses a wireframe representation of all geometry during mouse manipulation This option is turned on by default. You should turn it off only if your computer has a graphics accelerator; otherwise the mouse manipulation may be very slow.

Double Buffering

turns double buffering on or off, if it is supported by the driver. Double buffering dramatically reduces screen flicker during graphics updates. Note that if your display hardware does not support double buffering and you turn this option on, double buffering will be done in software. Software double buffering uses extra memory.

Front Faces Transparent

allows you to turn off the display of outer faces in wall zones. Making the **Front Faces Transparent** is useful for displaying both sides of a slit wall. By default, when you display a slit wall, one side will "bleed" through to the other. When you turn on the **Front Faces Transparent** option, the display of a slit wall will show each side distinctly as you rotate the display. This option can also be useful for displaying two-sided walls (walls with fluid or solid cells on both sides).

Hidden Surface Removal

turns hidden surface removal on or off. If you do not use hidden surface removal, Ansys Fluent will not try to determine which surfaces in the display are behind others; it will display all of them, and a cluttered display will result. You should turn this option off if you are working with a 2D problem or with geometries that do not overlap.

Removal Method

chooses the method to be used for hidden surface removal. These options vary in speed and quality, depending on the device you are using. The choices are listed below.

Hardware Z-buffer

is the fastest method if your hardware supports it. The accuracy and speed of this method is hardware dependent.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Painters

will show fewer edge-aliasing effects than hardware-z-buffer. This method is often used instead of software-z-buffer when memory is limited.

Software Z-buffer

is the fastest of the accurate software methods available (especially for complex scenes), but it is memory intensive.

Z-sort only

is a fast software method, but it is not as accurate as software-z-buffer.

Timeout in seconds

specifies the value for the timeout.

Lighting Attributes

controls lighting attributes for all lights in the active graphics window. See Adding Lights (p. 3984) for details.

Lights On

turns *all* lights in the active graphics window on or off.

Lighting

specifies the method to be used in lighting interpolation: **Automatic**, **Flat**, **Gouraud**, or **Phong**. **Automatic** automatically selects the best lighting method for the type of display in the graphics window. (**Flat** is the most basic method: there is no interpolation within the individual polygonal facets. **Gouraud** and **Phong** have smoother gradations of color because they interpolate on each facet.)

Layout

controls the display of captions, axes and the colormap in the graphics display window. See Changing the Legend Display (p. 3959) for details.

Logo

allows you to hide or display the ANSYS logo.

Color

contains a drop-down list of colors that can be used for the logo. The choices are **White** or **Black**.

Colormap

enables/disables the display of the color scale.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Colormap Alignment

allows you to adjust the alignment of colormap in the graphics display. Select the side (**Top**, **Bottom**, **Left**, and **Right**) from the drop-down list, where you want to align the colormap.

Apply

updates how new graphics objects will appear. Note that some changes, such as those to line width and the logo are updated in all applicable graphics windows once you click **Apply**.

Info

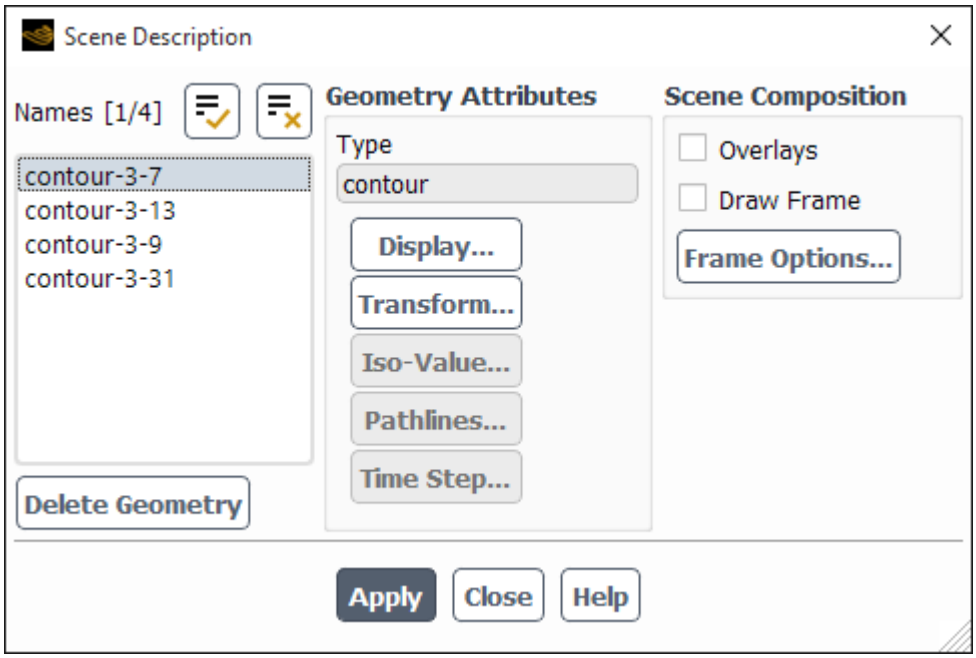
prints out information about your graphics driver in the console.

Lights...

opens the Lights Dialog Box (p. 5279), which allows you to create, delete, and modify directional light sources.

51.18.20. Scene Description Dialog Box

The **Scene Description** dialog box allows you to turn overlays on and off, select geometric objects in the display for modification or deletion, and open dialog boxes that control various characteristics of the selected object(s). (Note that you cannot use the **Scene Description** dialog box to control XY plot and histogram displays.) See Advanced Scene Composition (p. 4042) for details about the items below.



Controls

Names

contains a list of the geometric objects that currently exist in the scene (including those that are presently invisible). You can specify the object or objects to be manipulated by selecting names

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

in this list. If you select more than one object at a time, any operation (transformation, color specification, and so on) will apply to all the selected objects. You can also select objects by clicking on them in the graphics display using the mouse probe button, which is, by default, the right mouse button. (See Controlling the Mouse Button Functions (p. 4020) for information about mouse button functions.) To deselect a selected object, simply click its name in the **Names** list. See Selecting the Object(s) to be Manipulated (p. 4043) for details.

Geometry Attributes

contains information about the type of the selected geometric object and push buttons that open dialog boxes for modifying the object.

Type

reports the type of the selected object. Possible types include `mesh`, `surface`, `contour`, `vector`, and `Group`. This information is especially helpful when you need to distinguish two or more objects with the same name. When more than one object is selected, the type displayed is `Group`.

Display...

opens the Display Properties Dialog Box (p. 5267), which allows you to change the color, visibility, and other properties for the selected object.

Transform...

opens the Transformations Dialog Box (p. 5269), which allows you to translate, rotate, and scale the selected object.

Iso-Value...

opens the Iso-Value Dialog Box (p. 5270), which allows you to change the isovalue of an isosurface. This push button is available only if the geometric object selected in the **Names** list is an isosurface or an object on an isosurface (contour on an isosurface, for example); otherwise it is grayed out.

Pathlines...

opens the Pathline Attributes Dialog Box (p. 5271), which allows you to set the maximum number of steps for the selected pathlines.

Scene Composition

contains controls for enabling overlays and bounding frames.

Overlays

activates the superimposition of a new geometry onto a currently displayed geometry. See Advanced Graphics Overlays (p. 3954) for details.

Draw Frame

activates the display of a bounding frame in the graphics display. See Adding a Bounding Frame (p. 4049) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Frame Options...

opens the Bounding Frame Dialog Box (p. 5271), in which you can define properties of the bounding frame. See Adding a Bounding Frame (p. 4049) for details.

Delete Geometry

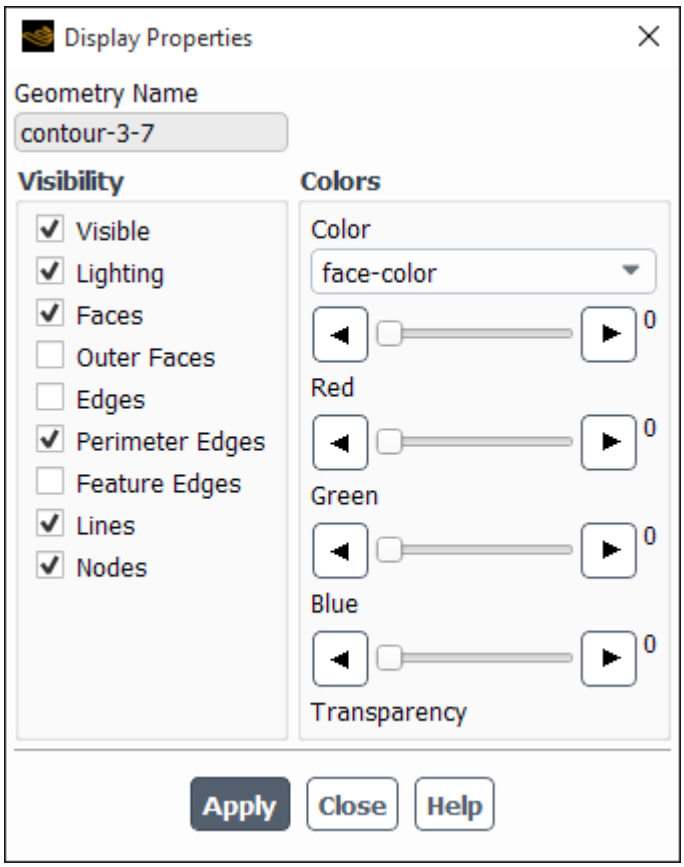
deletes the geometric object that is currently selected in the **Names** list. The ability to delete individual objects is especially useful if you have overlays on and you generate an unwanted object (for example, if you generate contours of the wrong variable). You can simply delete the unwanted object and continue your scene composition, instead of starting over from the beginning.

Apply

saves the status of **Overlays** and **Draw Frame**. When you turn **Overlays** or **Draw Frame** on or off, you must click the **Apply** button to see the effect of the change on subsequent display operations.

51.18.21. Display Properties Dialog Box

To modify the color, visibility, and other display properties for individual geometric objects in the graphics display, use the **Display Properties** dialog box. You can open this dialog box by clicking the **Display...** push button in the Scene Description Dialog Box (p. 5265). See Changing an Object's Display Properties (p. 4043) for details about the items below.



Controls

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Geometry Name

displays the name of the object you selected for modification in the Scene Description Dialog Box (p. 5265).

Visibility

contains check buttons that control options related to the visibility of the selected object. See Controlling Visibility (p. 4044) for details.

Visible

toggles the visibility of the selected object. If it is turned on, the object will be visible in the display, and if it is turned off, the object will be invisible.

Lighting

turns the effect of lighting for the selected object on or off. You can choose to have lighting affect only certain objects instead of all of them. Note that if **Lighting** is turned on for an object such as a contour or vector plot, the colors in the plot will not be exactly the same as those in the colormap at the left of the display.

Faces

toggles the filled display of faces for the selected mesh or surface object. Turning **Faces** on has the same effect as turning on the display of faces in the Mesh Display Dialog Box (p. 4688).

Outer Faces

toggles the display of outer faces.

Edges

toggles the display of interior and exterior edges of the geometric object.

Perimeter Edges

toggles the display of the outline of the geometric object. (This option has no effect on the display of meshes.)

Feature Edges

toggles the display of feature lines (if any) of the geometric object.

Lines

toggles the display of the lines (if any) in the geometric object.

Nodes

toggles the display of the nodes (if any) in the geometric object.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Colors

contains controls for setting face, edge, line, and node colors, and transparency for faces. See Controlling Object Color and Transparency (p. 4045) for details.

Color

specifies the face, edge, line, or node color for modification. When you turn on this button, the color scales below will show the current color specification, which you can modify by moving the sliders on the color scales.

Red, Green, Blue

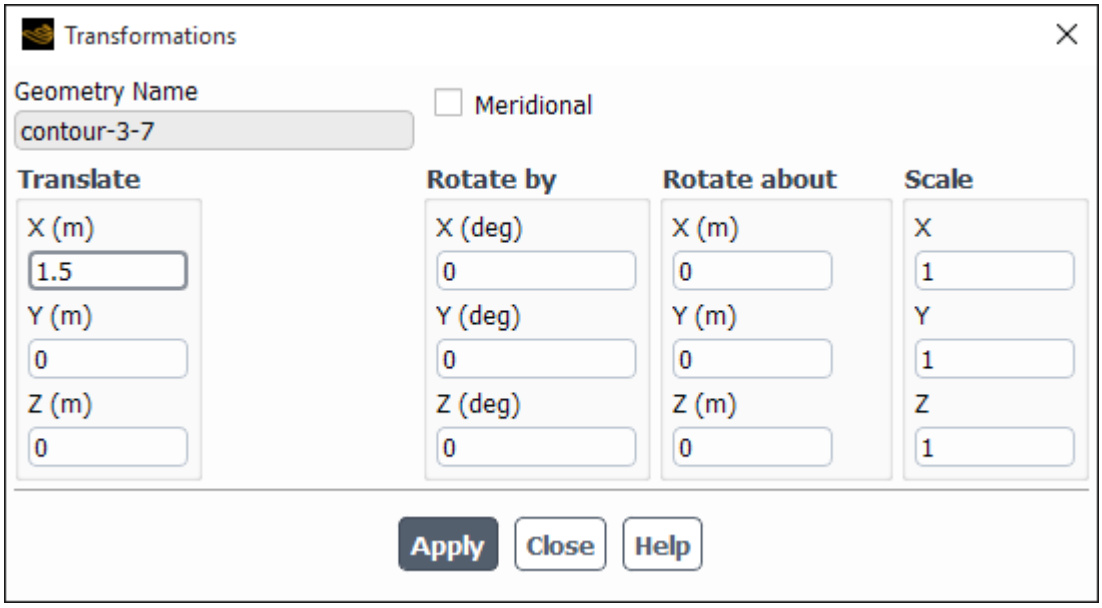
are color scales with which you can specify the RGB components of the face, edge or line color.

Transparency

sets the relative transparency of the selected object. An object with a transparency of 0 is opaque, and an object with a transparency of 100 is transparent.

51.18.22. Transformations Dialog Box

You can use the **Transformations** dialog box to translate, rotate, or scale individual objects in the graphics display. To open this dialog box, click the **Transform...** push button in the Scene Description Dialog Box (p. 5265). See Transforming Geometric Objects in a Scene (p. 4046) for details about the items below.



Controls

Geometry Name

displays the name of the object you selected for modification in the Scene Description Dialog Box (p. 5265).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Meridional

(3D only) enables the display of the meridional view. This option is especially useful in turbomachinery applications.

Translate

contains **X**, **Y**, and **Z** real number fields in which you can enter the distance by which to translate the selected object in each direction.

Rotate by

contains **X**, **Y**,and **Z** integer number fields in which you can enter the number of degrees by which to rotate the selected object about each axis.

Rotate about

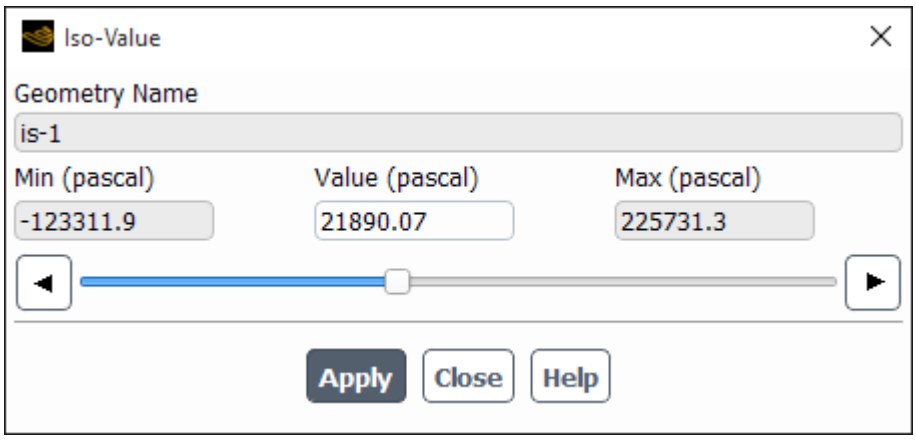
specifies the point about which to rotate the object.

Scale

contains **X**, **Y**, and **Z** real number fields in which you can enter the amount by which to scale the selected object in each direction. To avoid distortion of the object's shape, be sure to specify the same value for all three entries.

51.18.23. Iso-Value Dialog Box

The **Iso-Value** dialog box allows you to change the isovalue of an isosurface. The isosurface can be selected directly in the **Names** list or indirectly by selecting an object displayed on the isosurface. When you change the isovalue, any contours, vectors, and so on that were displayed on the original isosurface will be displayed on the isosurface with the new isovalue. To open this dialog box, click the **Iso-Value...** push button in the Scene Description Dialog Box (p. 5265). See Modifying Iso-Values (p. 4048) for details about using this dialog box.



Controls

Geometry Name

displays the name of the geometric object (isosurface) you selected for modification in the Scene Description Dialog Box (p. 5265).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Min, Max

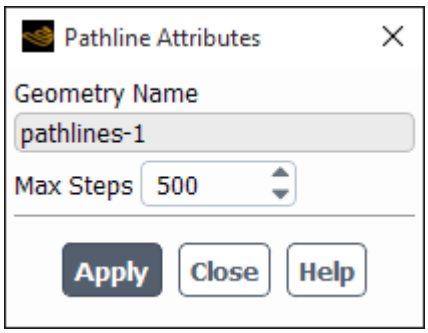
show the minimum and maximum values of the isosurface variable.

Value

sets the new isovalue for isosurfaces. After you change the value and click **Apply**, contours, vectors, or pathlines that were displayed on the original isosurface will be displayed for the new isovalue. You can also use the slide bar to change the **Value**.

51.18.24. Pathline Attributes Dialog Box

The **Pathline Attributes** dialog box allows you to change the number of steps used in the computation of pathlines. This is most useful in creating animations of pathlines. To open this dialog box, click the **Pathlines...** button in the Scene Description Dialog Box (p. 5265). See Modifying Pathline Attributes (p. 4048) for details about using this dialog box.



Controls

Geometry Name

displays the name of the geometric object you selected for modification in the Scene Description Dialog Box (p. 5265).

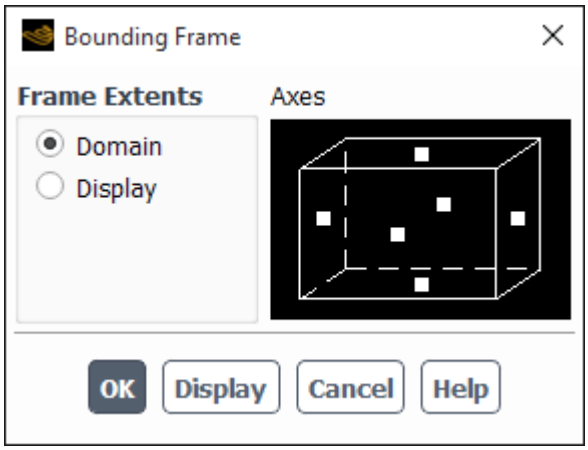
Max Steps

sets the new maximum number of steps for pathline computation. After you change the value and click **Apply**, the selected pathline will be recomputed and redrawn.

51.18.25. Bounding Frame Dialog Box

The **Bounding Frame** dialog box allows you to add a bounding frame with optional measure markings to the display of the domain. See Adding a Bounding Frame (p. 4049) for details about the items in this dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Frame Extents

indicates the extents of the bounding frame.

Domain

specifies that the frame should encompass the domain extents.

Display

specifies that the frame should encompass the portion of the domain that is shown in the display.

Axes

contains controls for specifying the frame boundaries and measurements. See Adding a Bounding Frame (p. 4049) for instructions on using these items.

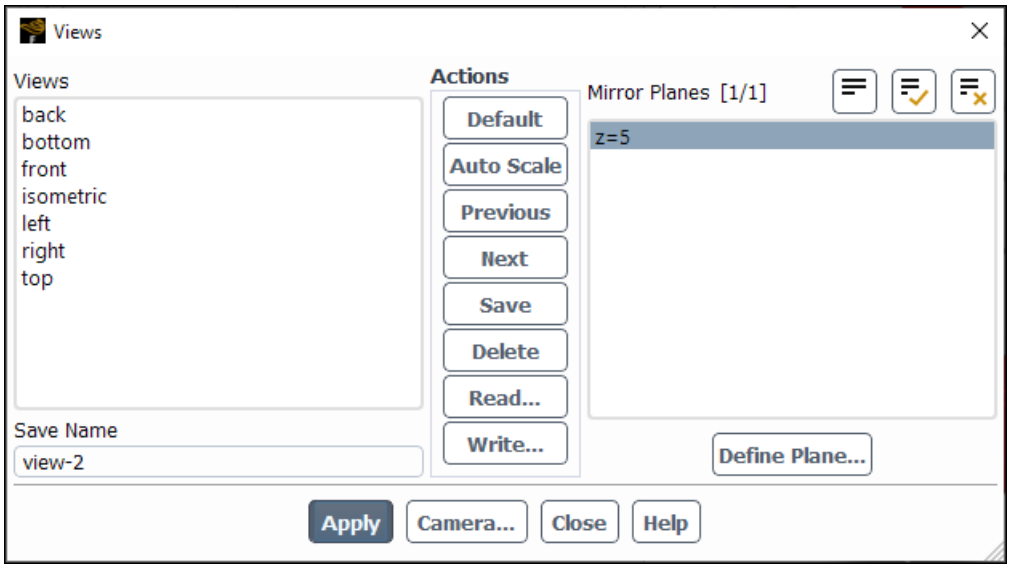
Display

updates the active graphics window with the current frame settings.

51.18.26. Views Dialog Box

With the **Views** dialog box, you can make various modifications to the view displayed in the active graphics window. See Controlling the Display State and Modifying the View (p. 4025) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Views

lists the currently defined views. Clicking on a view name highlights that name and enters it into the **Name** field. Double-clicking on a view name restores that view in the active graphics window.

Save Name

specifies the name to use when saving a view.

Actions

contains buttons for performing various actions related to the **Views** list and the **Save Name**.

Default

restores the "front" view in the active graphics window.

Auto Scale

modifies the view in the active graphics window by scaling and centering the current scene without changing its orientation.

Previous

allows you to return to previous displays.

Save

stores the view in the active graphics window with the name in the **Save Name** box. See Saving Views (p. 4035) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Delete

removes the selected view name from the **Views** list.

> **Important:**
>
> Be careful not to delete any of the pre-defined views.

Read...

opens The Select File Dialog Box (p. 931), in which you can specify the name of a view file to be read. See Reading View Files (p. 4036) for details.

Write...

opens the Write Views Dialog Box (p. 5274), in which you can select the views to be saved to a view file. See Saving Views (p. 4035) for details.

Mirror Planes

displays a list of all symmetry planes in the domain. Mirror images are drawn for all selected symmetry planes. See Mirroring and Periodic Repeats (p. 4036) for details.

Define Plane...

opens the Mirror Planes Dialog Box (p. 5275), in which you can define a mirror plane for a non-symmetric domain.

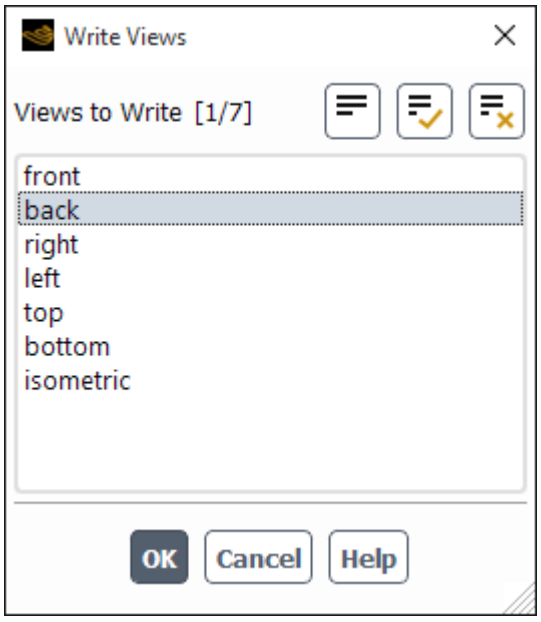
Camera...

opens the Camera Parameters Dialog Box (p. 5277).

51.18.27. Write Views Dialog Box

The **Write Views** dialog box allows you to save selected views to a view file. To open it, click **Write...** in the Views Dialog Box (p. 5272). This feature allows you to transfer views between case files. See Saving Views (p. 4035) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



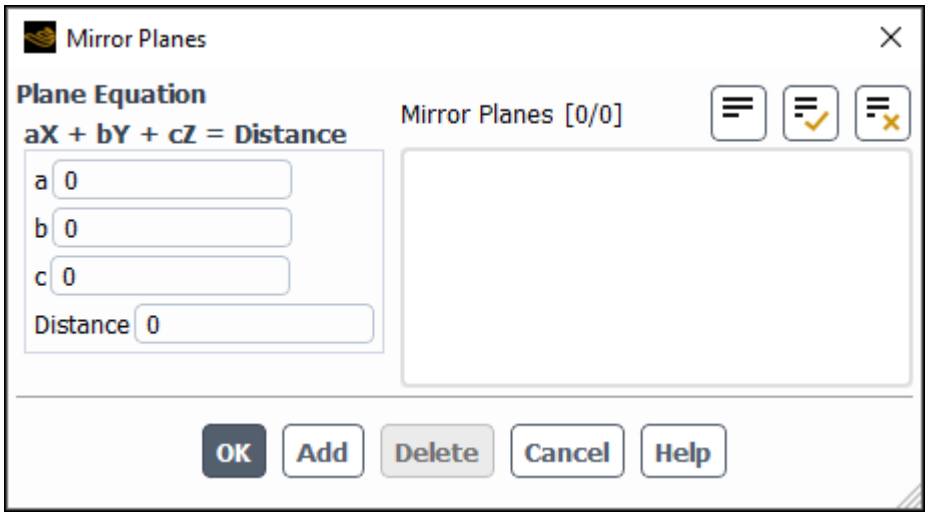
Controls

Views to Write

is a selectable list of the defined views. The selected views will be saved to a view file when you click **OK**.

51.18.28. Mirror Planes Dialog Box

The **Mirror Planes** dialog box allows you to define a symmetry plane for a non-symmetric domain for use with graphics. To open it, click **Define Plane...** in the Views Dialog Box (p. 5272). See Mirroring for Graphics (p. 4041) for details.



Controls

Plane Equation

contains inputs for specifying the equation for the mirror plane: coefficients **a**, **b**, **c**, and **Distance**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mirror Planes

contains a list of all mirror planes you have defined using this dialog box. (Mirror planes that exist in the domain due to symmetry will not appear in this list, since they cannot be modified.)

Add

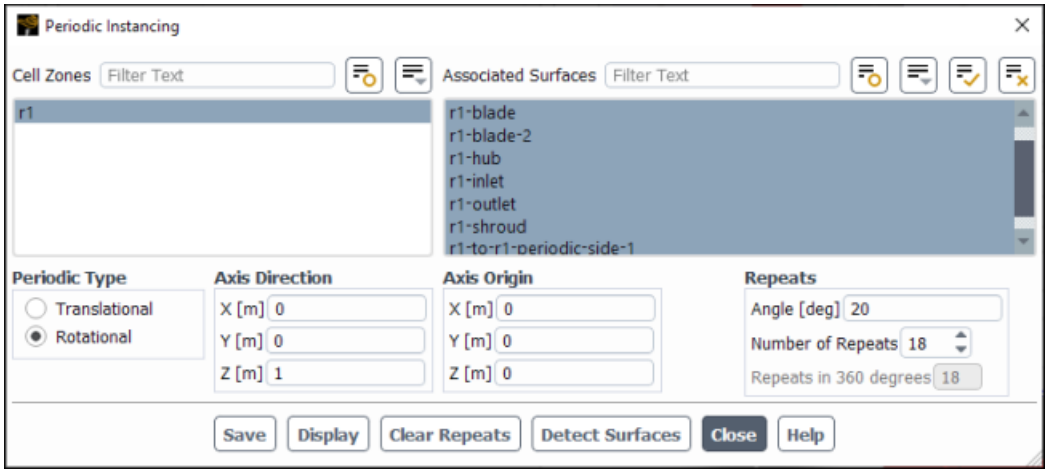
adds the plane defined by the **Plane Equation** to the **Mirror Planes** list.

Delete

deletes the plane(s) selected in the **Mirror Planes** list.

51.18.29. Periodic Instancing Dialog Box

The **Periodic Instancing** dialog box allows you to define a periodic repeats, periodic rotation or translation for a non-periodic domain for use with graphics. To open it, click **Periodic Instancing...** in the **View** ribbon tab (**Display** group box). See Periodic Repeats for Graphics (p. 4039) for details.



Controls

Cell Zones

contains the list of the zones in the mesh. You can select one or more zones in this list and specify different periodicity parameters for each zone separately.

Associated Surfaces

contains the list of the surfaces associated with the selected cell zone.

Periodic Type

specifies **Rotational** or **Translational** periodicity.

Angle

specifies the angle by which the domain is rotated to create the periodic repeat. This item is available when you select **Rotational** as the **Periodic Type**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Translation

specifies the distance in the **X**, **Y**, and **Z** directions by which the domain is translated to create the periodic repeat. This item will appear when you select **Translational** as the **Periodic Type**.

Axis Direction

specifies the direction vector (**X,Y,Z**) for the axis of rotation. This item is available when you select **Rotational** as the **Periodic Type** and the domain is three-dimensional.

Axis Origin

specifies the origin of the axis of rotation. This item is available when you select **Rotational** as the **Periodic Type**.

Number of Repeats

specifies the number of times you want to repeat the periodic domain. Positive values repeat the zone in the direction of rotation. Negative values repeat the zone in the opposite direction of rotation.

Save

saves the periodicity you have defined for the case setup.

Display

saves the current settings and displays the results.

Important:

The surfaces you want to see repeated or translated must already be displayed in the graphics window for you to see any changes in the contents of the graphics window when you click **Display** (for example, mesh display, contours, and so on).

Clear Repeats

Resets the **Number of Repeats** to 1 and saves the settings change.

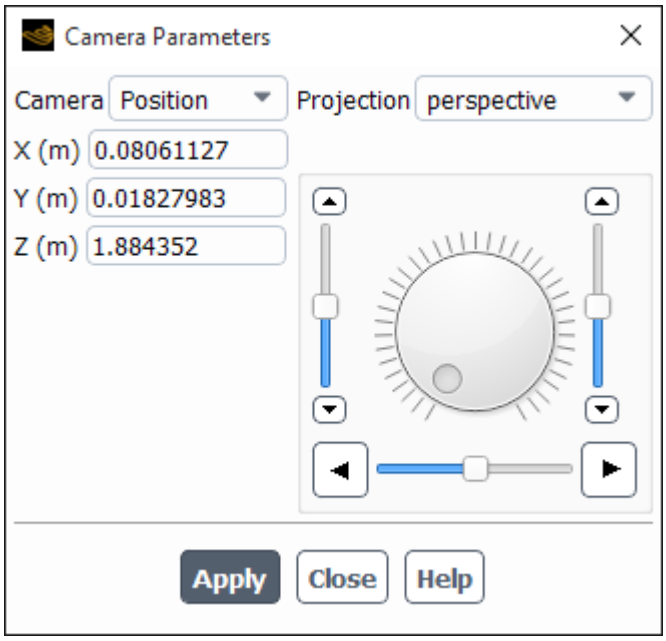
Detect Surfaces

Fluent automatically assesses the loaded case file and selects what it thinks are the appropriate cell zones, surfaces, and other settings (such as angle and number of repeats) for this problem, and saves them.

51.18.30. Camera Parameters Dialog Box

The **Camera Parameters** dialog box allows you to modify the "camera" through which you are viewing the graphics display. See Controlling Perspective and Camera Parameters (p. 4033) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Camera

contains a drop-down list of the parameters that define the camera (**Position**, **Target**, **Up Vector**, and **Field**) and **X**, **Y**, and **Z** fields in which you can define the coordinates or field distances for the parameter selected in the drop-down list. Figure 40.91: Camera Definition (p. 4034) illustrates the definition of the camera by these parameters.

Projection

contains a drop-down list that allows you to select a **Perspective** or **Orthographic** view.

(Dial and Sliders)

allow you to rotate and scale the graphics display. The slider on the scale to the left of the dial rotates the display about the horizontal axis at the center of the screen, the slider on the scale below the dial rotates the display about the vertical axis at the center of the screen, and the dial controls rotation about the axis at the center of and perpendicular to the screen. The slider on the scale to the right of the dial zooms in or out in the display. See Rotating the Display (p. 4030) and Zooming the Display (p. 4032) for details.

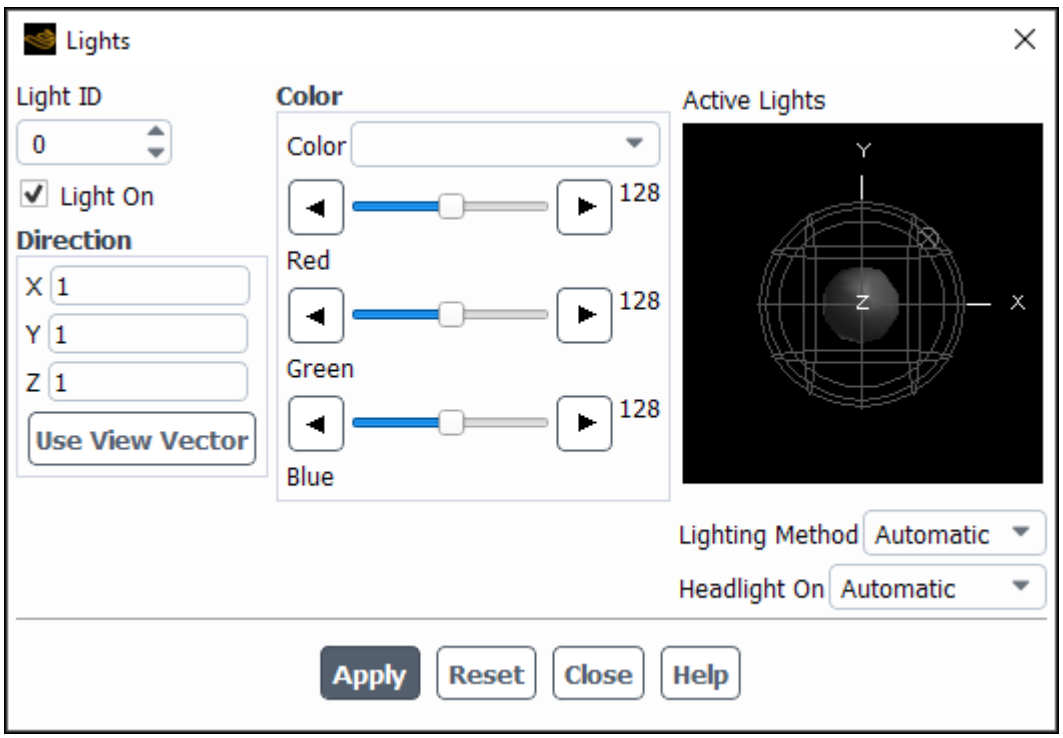
Important:

When you are using the sliders and dial to manipulate the view, you may want to turn off **Wireframe Animation** in the Display Options Dialog Box (p. 5262), so that you can watch the display move interactively while you are moving the slider or the dial indicator.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

51.18.31. Lights Dialog Box

The **Lights** dialog box provides an interactive mechanism for placing colored, directional lights in a scene. See Adding Lights (p. 3984) for details about the items below.



Controls

Light ID

indicates the light that is being added, deleted, or modified. By default, light 1 is defined to be dark gray with a direction of (1,1,1).

Light On

indicates whether or not the light specified in **Light ID** is on or off. By turning off the **Light On** option for a particular light, you can remove this light from the display, while still retaining its definition. To add it to the display again, simply turn on the **Light On** button.

Direction

allows you to specify the direction of the light (the position on the unit sphere from which the light emanates) by entering the **X**, **Y**, and **Z** coordinates or by computing the coordinates based on the current view in the graphics window.

X,Y,Z

specify the direction of the light. For example, the direction (1,1,1) means that the rays from the light will be parallel to the vector from (1,1,1) to the origin.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Use View Vector

updates the **X,Y,Z** fields with the appropriate values for the current view in the active graphics window, and shines a light in that direction. Instead of entering the **X,Y,Z** values for a light's direction vector, you can use your mouse to change the view in the graphics window so that your position in reference to the geometry is the position from which you would like a light to shine. You can then click the **Use View Vector** button to update the **X,Y,Z** fields with the appropriate values for your current position and update the graphics display with the new light direction. This method is convenient if you know where you want a light to be, but you are not sure of the exact direction vector.

Color

allows you to specify the light color with sliders. You can create your desired color by increasing and decreasing the slider values for the colors **Red**, **Green**, and **Blue**. You can also enter a descriptive string (for example, lavender) in the **Color** field.

Active Lights

shows the position and color of all defined directional lights, and allows you to change the position of a light. All directional lights in Ansys Fluent are assumed to be at infinity and pass through the unit sphere at the position shown. All light rays arriving at the scene from one light are parallel. The colored markers on the surface of the sphere represent the color and direction of these *distant* lights. These lights point towards the center of the sphere (the origin, which is usually where the geometry is).

Lighting Method

specifies the method to be used in lighting interpolation: **Automatic**, **Off**, **Flat**, **Gouraud**, or **Phong**. (**Flat** is the most basic method: there is no interpolation within the individual polygonal facets. **Gouraud** and **Phong** have smoother gradations of color because they interpolate on each facet.) When **Off** is selected, lighting effects are disabled.

Headlight On

enables constant lighting effects in the direction of the view. The **Automatic** option automatically enables/disables the headlight depending on the type of display in the graphics window.

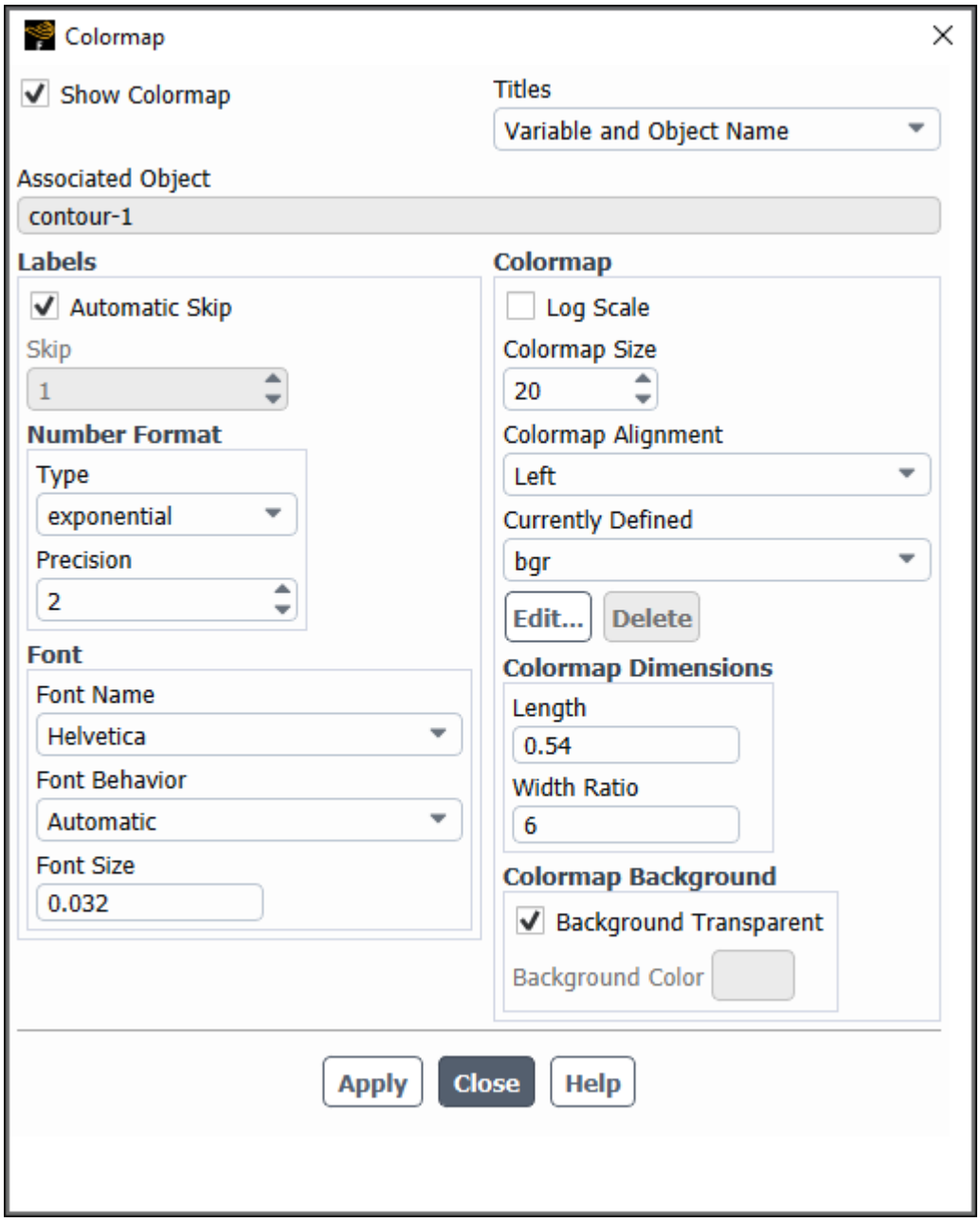
Reset

resets the light definitions to their last saved state (the lighting that was in effect the last time you opened the dialog box or clicked on **Apply**).

51.18.32. Colormap Dialog Box

The **Colormap** dialog box allows you to select and modify existing colormaps. See Selecting a Colormap (p. 3980) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Show Colormap

this option is only visible when accessed through a graphics object dialog box, such as **Contours** or **Vectors**. It allows you to turn the display of the colormap in the graphics window on/off.

Alternatively, you can temporarily hide a colormap by clicking the 'X' that appears in the upper-right of the colormap when you hover over it in the graphics window. There is also a context menu option to temporarily show a hidden colormap. Refer to *Changing the Colormap and Range (p. 3963)* *for additional information.*

Titles

controls what identifying titles are displayed on the colormap. The options include:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Variable and Object Name**—both the variable name and the associated graphics object are listed with the colormap, along with the units.
- **Variable Only**—only the variable name and the units are listed with the colormap.
- **None**—only the units are listed with the colormap.

Labels

allows you to customize the display of your colormap labels.

Automatic Skip

automatically skips enough labels so that they are all legible. Disabling will display the number of labels determined by the **Skip** and **Colormap Size** fields.

Skip

sets the number of labels to be skipped.

Number Format

contains controls for changing the format of the labels on the color scale. These labels are the character strings used to define the color divisions at the left of the graphics window.

Type

sets the form of the labels. You may select from a drop-down list of options, including the following:

general

displays the real value with either float or exponential form based on the size of the number and the defined **Precision**.

float

displays the real value with an integral and fractional part (for example, 1.0000), where the number of digits in the fractional part is determined by **Precision**.

exponential

displays the real value with a mantissa and exponent (for example, 1.0e-02), where the number of digits in the fractional part of the mantissa is determined by **Precision**.

Precision

defines the number of fractional digits displayed in the labels.

Font

allows you to customize the display of your colormap font.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Font Name

lists the available colormap fonts.

Font Behavior

controls whether the font size updates automatically when the colormap is resized or if the font is always a fixed size.

Font Size

when **Font Behavior** is set to **Automatic**, this field controls the font size as a percentage of the colormap size. When the **Font Behavior** is set as **Fixed**, you can set the **Font Size** to a fixed size that is familiar from word processors.

Colormap

contains controls for the colormap size and scale, and for selection of a defined colormap.

Log Scale

enables the use of a logarithmic scale for the color scale (rather than the default decimal scale). See Specifying the Colormap Size and Scale (p. 3980) for details.

Colormap Size

specifies the number of distinct colors in the colormap. You may specify from 2 to 100 colors.

Currently Defined

contains a drop-down list of all pre-defined colormaps and all custom colormaps defined by you. Select the colormap to be used from this list.

Edit...

opens the Colormap Editor Dialog Box (p. 5284), in which you can create a custom colormap.

Delete

Delete the colormap that is selected in the **Currently Defined** drop-down list.

Note:

- You can only delete user-defined colormaps, not the ones that come with Ansys Fluent.
- If you delete a user-defined colormap that is assigned to a graphics object, the graphics object will revert to the colormap selected in the **Colormap** field in Preferences (**Graphics** branch, under **Colormap Settings**).

Colormap Dimensions

allows you to customize the "physical" size of your colormap in the graphics window.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Length

controls the "height" for vertical colormaps, or the "length" for horizontal colormaps, as a percentage of graphics window height or width, respectively.

Width Ratio

controls the "width" of a vertical colormap or the "height" of a horizontal colormap, as a ratio of the **Length** field. Decreasing the **Width Ratio** increases the "width" or "height" of the colormap, respectively.

Colormap Background

allows you to control the appearance of the colormap background.

Background Transparent

enabling this option makes the background of the colormap transparent.

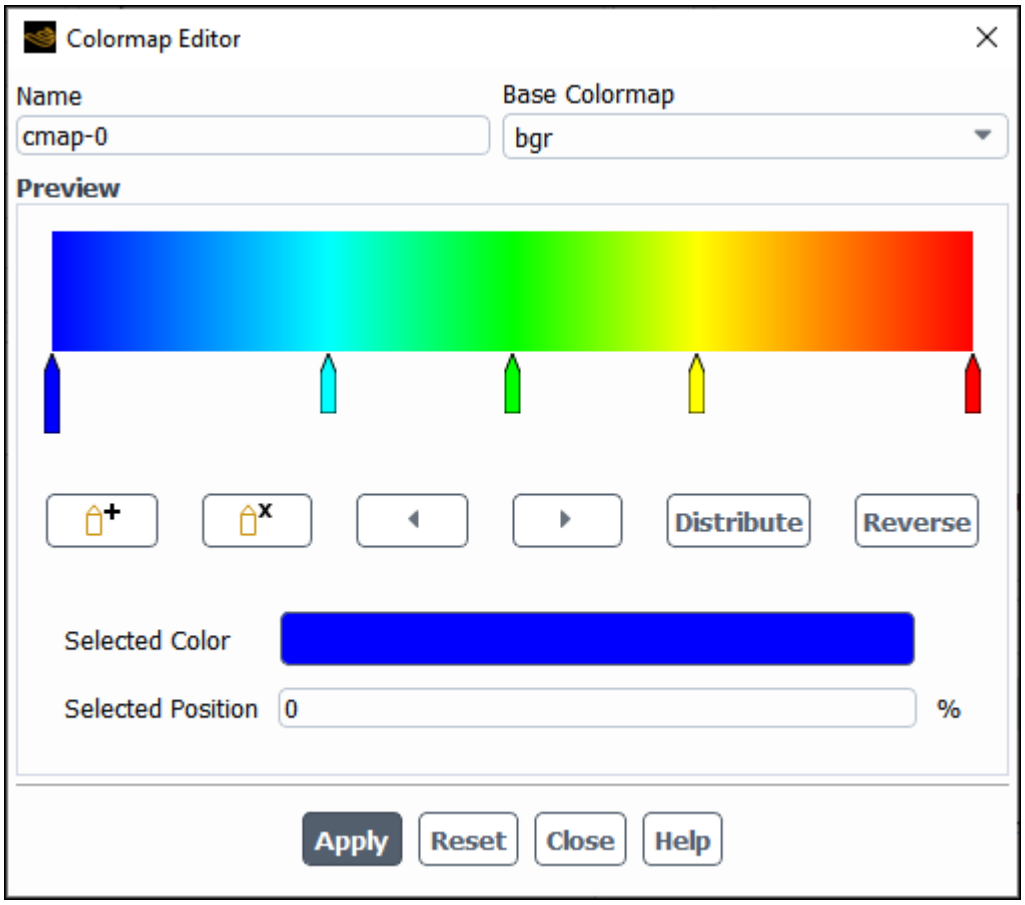
Background Color

specifies which color will be used for the colormap background when the **Background Transparent** option is disabled.

51.18.33. Colormap Editor Dialog Box

The **Colormap Editor** dialog box allows you to create custom colormaps. See Creating a Customized Colormap (p. 3982) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name

is the name that will be applied to the modified colormap.

Base Colormap

is the baseline colormap that you are modifying. You can use this drop-down list to select a different baseline colormap.

Preview

allows you to interactively modify the colormap. Drag the "stops" ( ) to modify the color and color distribution.

Add Stop

clicking "add stop" ( ) adds another color stop next to the selected stop.

Remove Stop

clicking "remove stop" ( ) removes the selected color stop.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Left Arrow

clicking the left arrow ( ) selects the next stop to the left.

Right Arrow

clicking the right arrow ( ) selects the next stop to the left.

Distribute

evenly distributes the stops on the colormap.

Reverse

reverses the colors on the colormap.

Selected Color

shows the color of the selected stop. Clicking the colored field to the right, opens the **Select Color** dialog box, allowing you to choose a different color for the selected stop.

Selected Position

shows the position in the colormap of the selected stop as a percentage, with 0% being the left-most stop and 100% being the right-most stop.

Apply

saves the colormap with the name displayed in the **Name** field. *Note that you cannot overwrite the system colormaps that come with Ansys Fluent.*

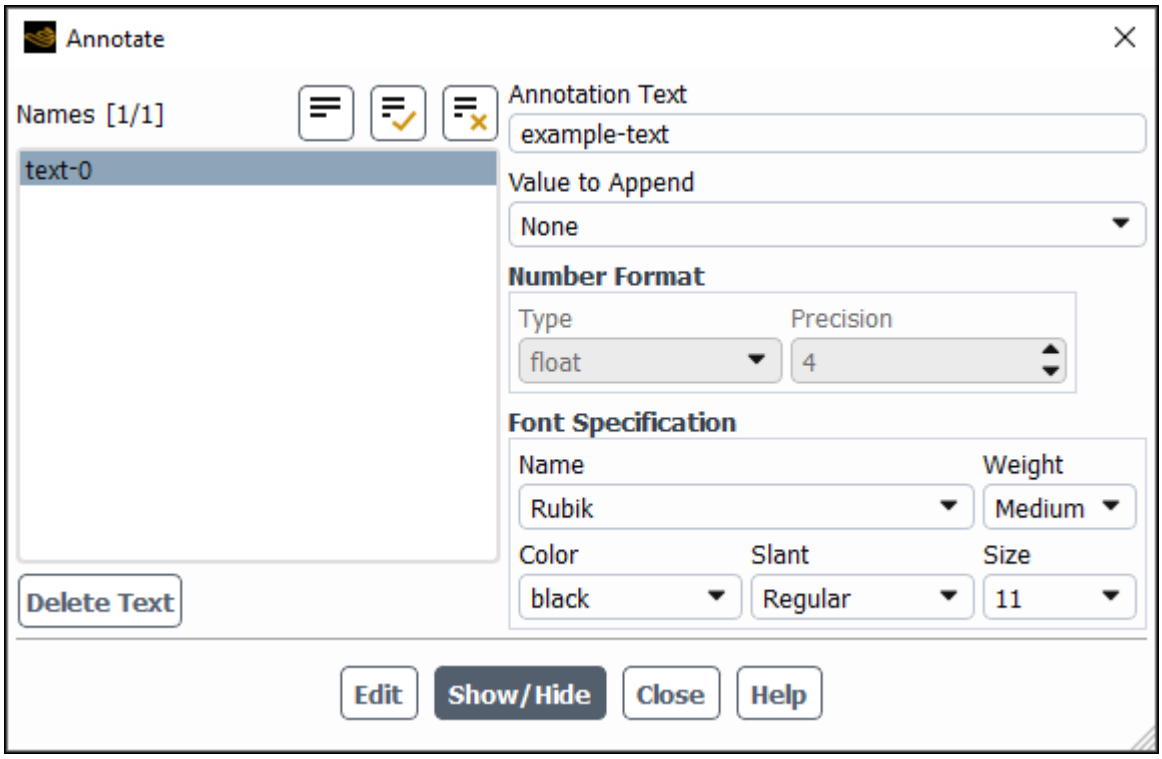
Reset

resets the preview to match the **Base Colormap**.

51.18.34. Annotate Dialog Box

You can use the **Annotate** dialog box to add text with optional attachment lines to the graphics windows, or to modify existing text. See Adding Text Using the Annotate Dialog Box (p. 3961) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Names

contains a selectable list of all annotation text strings that have been defined. You can choose a text string to be deleted or edited.

Delete Text

deletes the text strings selected in the **Names** list from the display.

Annotation Text

contains the annotation text string you want to add, or the annotation text string for the item selected in the **Names** list.

Value to Append

allows you to add the following items to the **Annotation Text**.

- Date
- Time
- Version
- Number of iterations
- Number of timesteps
- Flow time

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- <any defined expressions>

Number Format

contains controls for specifying how appended flow time values are displayed.

Type

controls whether values are displayed as floats.

Precision

specifies how many significant digits are used in the displayed flow time or expressions.

Font Specification

contains controls for defining or modifying the font in the annotation text string.

Name

contains a drop-down list of various font styles.

Weight

contains a drop-down list from which you can select **Medium** or **Bold**.

Color

contains a drop-down list of colors that can be used for the text.

Slant

contains a drop-down list from which you can select **Regular** or **Italic** as the slant type.

Size

contains a drop-down list of font sizes (in points).

Add

adds the current **Annotation Text** to the active graphics window. A dialog box will prompt you to select a screen location using the left-mouse-button.

If you click the mouse button once in the desired location, the text will be placed at that point. Dragging the mouse with the left-mouse-button depressed will draw an attachment line from the point where the mouse was first clicked to the point where it was released. The annotation text will be placed at the point where the mouse button was released.

Show/Hide

shows or removes the defined annotations.

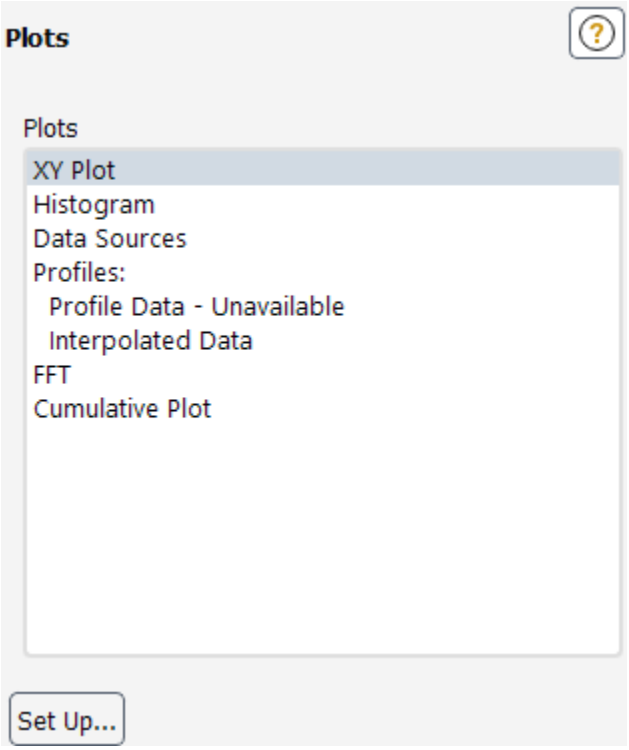
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Edit

updates the edited text in the active graphics window. This button will replace the **Add** button when you are editing an existing text string from the **Names** list.

51.19. Plots Task Page

The **Plots** task page allows you to create plots (XY, histograms, profiles, and so on) of your computational results. See Displaying Graphics (p. 3869) for more information.



Controls

Plots

contains a listing of the various plot types available in Ansys Fluent.

You can double-click an item in the **Plots** list to open the corresponding dialog box, or you can select the item in the list and click the **Set Up...** button.

XY Plot

selecting this item and clicking the **Set Up...** button opens the Solution XY Plot Dialog Box (p. 5290).

Histogram

selecting this item and clicking the **Set Up...** button opens the Histogram Dialog Box (p. 5295).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

File

selecting this item and clicking the **Set Up...** button opens the Plot Data Sources Dialog Box (p. 5297).

Profiles

two types of profile plots are available:

Profile Data

selecting this item and clicking the **Set Up...** button opens the Plot Profile Data Dialog Box (p. 5299).

Interpolated Data

selecting this item and clicking the **Set Up...** button opens the Plot Interpolated Data Dialog Box (p. 5300).

FFT

selecting this item and clicking the **Set Up...** button opens the Fourier Transform Dialog Box (p. 5302).

Set Up...

displays the dialog box corresponding to the selected item in the **Plots** list.

For additional information, see the following sections:

51.19.1. Solution XY Plot Dialog Box
51.19.2. Histogram Dialog Box
51.19.3. Plot Data Sources Dialog Box
51.19.4. Plot Profile Data Dialog Box
51.19.5. Plot Interpolated Data Dialog Box
51.19.6. Fourier Transform Dialog Box
51.19.7. Ansys Sound Analysis Dialog Box
51.19.8. Cumulative Plot Dialog Box
51.19.9. Plot/Modify Input Signal Dialog Box
51.19.10. Axes Dialog Box
51.19.11. Curves Dialog Box

51.19.1. Solution XY Plot Dialog Box

The **Solution XY Plot** dialog box allows you to display zone, surface and file data in an XY plot format. See Steps for Generating Solution XY Plots (p. 4060) for details about the items below.

*You can also access the **Solution XY Plot** dialog box through the **Postprocessing** ribbon tab.*

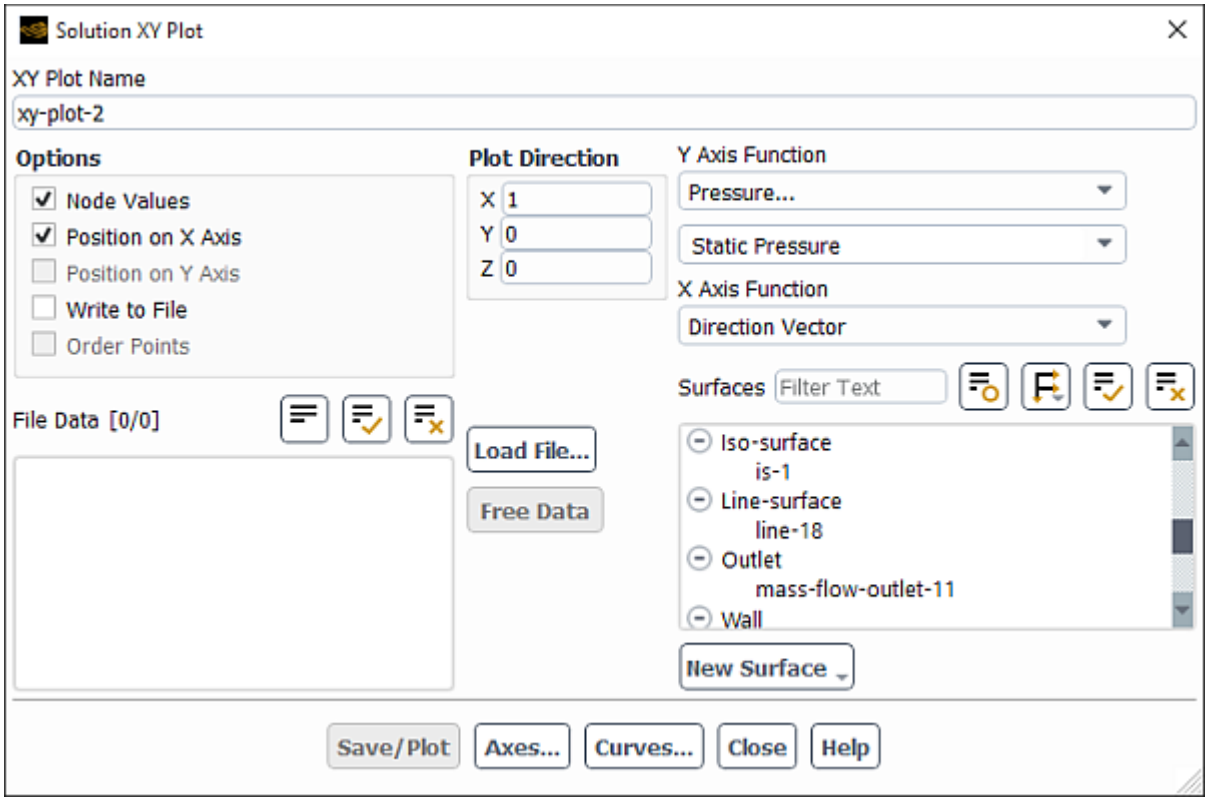
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Postprocessing → Plots → XY Plot → New...

Note:

- The "global"/non-persistent form of graphics objects is hidden by default. If this interrupts your workflow, you can make them available by enabling the **Expose legacy non-persistent graphics** option in the **Graphics** branch of **Preferences** (accessed via **File**>**Preferences...**).
- (When you click **Edit...** instead of **New...**, it opens the "global"/non-persistent version of the **Solution XY Plot** dialog box. This version of the dialog box does not have a **Name** field and it is not saved with the case file.

  You are encouraged to use the persistent version that is accessed by clicking **New...**.



Controls

XY Plot Name

is the name for a solution XY plot definition. You can specify a name or use the default name **xy-plot-*id***. This control appears only for solution XY plot definitions.

Options

contains check buttons that control the presentation of node or cell-averaged values, the selection of axis functions, and the ability to write the plot data to a file.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Node Values

toggles the node averaging of the data presented in the plot. If the option is disabled, boundary face values are presented. See Choosing Node or Cell Values (p. 4064) for details.

Position on X Axis, Position on Y Axis

set the x-axis or y-axis function to be the position. If one of these options is turned on, the other will automatically be turned off. You can turn both options off to generate a plot of one flow-field function vs. another, selecting the function for each axis using the **X Axis Function** and **Y Axis Function** drop-down lists.

Write to File

activates the file-writing option. When this option is selected, the **Plot** push button will change to **Write...**. Clicking on the **Write...** button will open the **Select File** dialog box (The Select File Dialog Box (p. 931)), in which you can specify a name and save a file containing the plot data. The format of this file is described in XY Plot File Format (p. 4071).

Order Points

specifies that plot data being saved to a file should be sorted in order of ascending x axis value. This option is available only when the **Write to File** option is turned on.

Plot Direction

contains inputs for defining the plot direction.

If **Direction Vector** (the default) is selected in the **X Axis Function** or **Y Axis Function** drop-down list (whichever is the position axis), the inputs are the components of the direction vector. The position axis of the plot will have coordinate values that correspond to the dot product of the data coordinate vector with the plot direction vector. See Steps for Generating Solution XY Plots (p. 4060) for details.

X

is the component in the x direction.

Y

is the component in the y direction.

Z

is the component in the z direction.

If **Curve Length** is selected in the **X Axis Function** or **Y Axis Function** drop-down list (whichever is the position axis), the inputs are the direction along the length of the surface selected in the **Surfaces** list. See Steps for Generating Solution XY Plots (p. 4060) for details.

Default

specifies the plot direction as the direction of increasing curve length.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Reverse

specifies the plot direction as the direction of decreasing curve length.

Show

displays the selected surface in the graphics window, marking the start of the surface with a blue dot and the end of the surface with a red dot. Ansys Fluent will also display arrows on the surface showing the direction in which the variable will be plotted.

Y Axis Function, X Axis Function

contain lists of solution variables that can be used for the y or x axis of the plot. If the **Position on X Axis** option is turned on, **X Axis Function** will become a single drop-down list, containing two options: **Direction Vector** (to plot the selected variable as a function of position along a specified direction vector) and **Curve Length** (to plot the selected variable as a function of position along the length of a specified curvilinear surface). See Steps for Generating Solution XY Plots (p. 4060) for details.

Likewise, if **Position on Y Axis** is turned on, **Y Axis Function** will become a single drop-down list containing **Direction Vector** and **Curve Length**. If both **Position on X Axis** and **Position on Y Axis** are turned off, you can select field functions for both axes using the **X Axis Function** and **Y Axis Function** lists.

Note:

When running the parallel version of Ansys Fluent, **Curve Length** is only supported when the curvilinear surface is contained within a single partition.

File Data

is a selectable list of the plot titles associated with the loaded external data files. You may choose any number of files for the data plot. The files are loaded using the **Load File...** push button. The format of these files is presented in XY Plot File Format (p. 4071). See Including External Data in the Solution XY Plot (p. 4063) for details.

Load File...

opens the **Select File** dialog box (The Select File Dialog Box (p. 931)), in which you can select the plot file to be read. See Including External Data in the Solution XY Plot (p. 4063) for details. After the external file is loaded, its plot title will be displayed in the **File Data** list.

Free Data

removes the files selected in the **File Data** list.

Surfaces

is a selectable list of surfaces in the solution domain. You may choose any number of surfaces for the data plot.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

New Surface

is a drop-down list button that contains a list of surface options:

Point

opens the Point Surface Dialog Box (p. 5582).

Line/Rake

opens the Line/Rake Surface Dialog Box (p. 5514).

Plane

opens the Plane Surface Dialog Box (p. 5580).

Quadric

opens the Quadric Surface Dialog Box (p. 5584).

Iso-Surface

opens the Iso-Surface Dialog Box (p. 5501).

Iso-Clip

opens the Iso-Clip Dialog Box (p. 5500).

Structural Point

opens the Structural Point Surface Dialog Box (p. 5623).

Plot

plots the specified surface and/or file data in the active graphics window using the current axis and curve attributes. If the **Write to File** option is turned on, this button becomes the **Write...** button.

Write...

opens the **Select File** dialog box (The Select File Dialog Box (p. 931)), in which you can save the plot data to a file. This button replaces the **Plot** button when the **Write to File** option is turned on.

Axes...

opens the Axes Dialog Box (p. 5313), which allows you to customize the plot axes.

Curves...

opens the Curves Dialog Box (p. 5316), which allows you to customize the curves used in the XY plot.

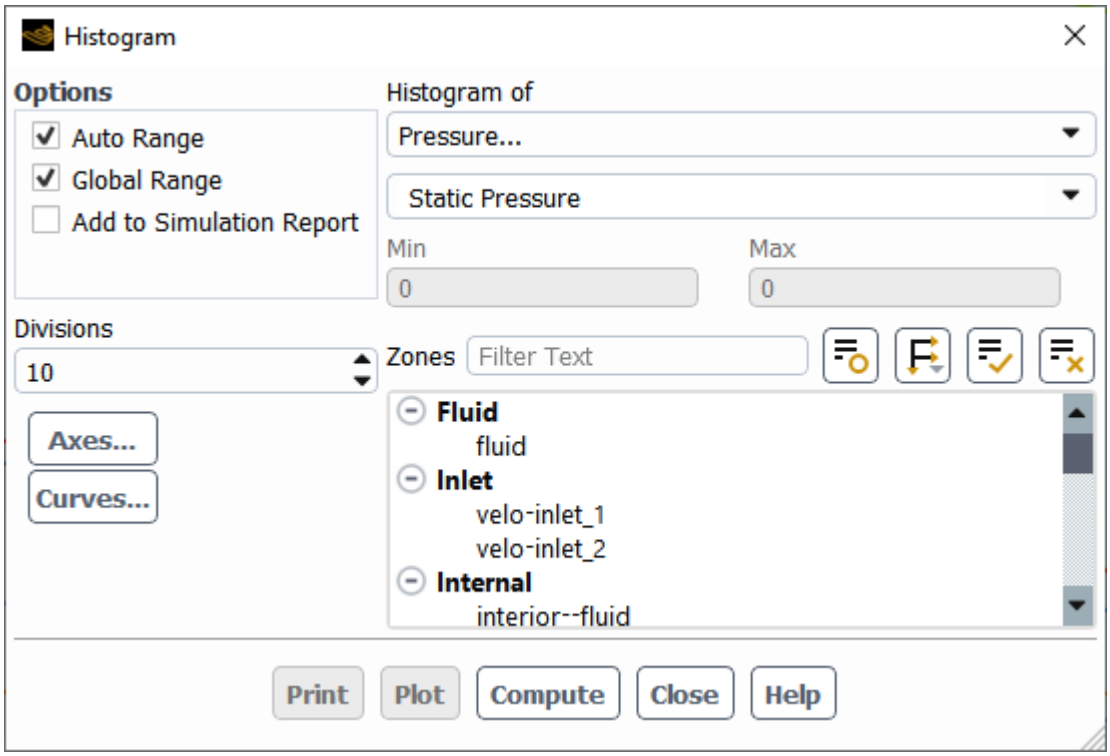
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Save/Plot

plots the specified surface and/or file data in the active graphics window using the current axis and curve attributes and saves the XY plot definition. This button appears only for solution XY plot definitions and replaces the **Plot** button.

51.19.2. Histogram Dialog Box

The **Histogram** dialog box allows you to create histograms of selected geometric or physical data. See Steps for Generating Histogram Plots (p. 4072) for details about the items below.



Controls

Options

contains the check buttons for current histogram options.

Auto Range

toggles the ability to specify the minimum and maximum range of scalar values in the histogram print or plot. If the option is not active, the **Min** and **Max** fields are editable, and you may specify the desired range. If the option is active, the range is defined by the minimum and maximum values in the computational domain.

Global Range

toggles the ability to specify the minimum and maximum values on the range of values on the selected surfaces, rather than in the entire domain.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Add to Simulation Report

toggles the ability to add the histogram and its data to your simulation report.

Divisions

sets the number of data intervals that will exist in the histogram.

Zone Types

contains a list of available face or cell zone types.

Axes...

opens the Axes Dialog Box (p. 5313), which allows you to customize the plot axes.

Curves...

opens the Curves Dialog Box (p. 5316), which allows you to customize the curves used in the histogram plot.

Histogram of

contains a list of scalar quantities that can be used in the histogram.

Min

displays or allows definition of the minimum value of the selected scalar quantity used in the histogram.

Max

displays or allows definition of the maximum value of the selected scalar quantity used in the histogram.

Zones

contains a list of available face or cell zones.

Print

displays the histogram interval.

Plot

displays a plot of the percentage of the total number of cells versus the scalar quantity in the active graphics window.

Add To Report

adds the histogram and its data to the simulation report (when the **Add to Simulation Report** option is enabled).

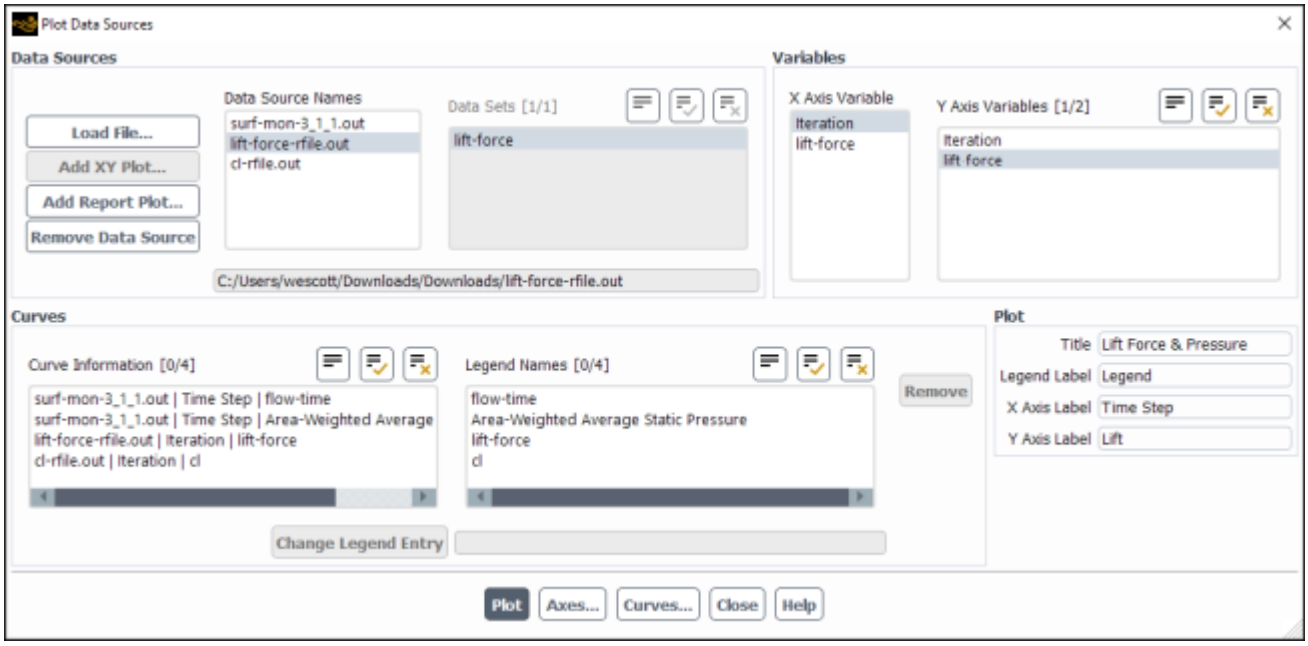
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Compute

computes the minimum and maximum cell values of the selected scalar quantity. The values are displayed in **Min** and **Max**.

51.19.3. Plot Data Sources Dialog Box

The **Plot Data Sources** dialog box allows you display data read from one or more files (in an abscissa/ordinate plot form) and combine them with data from XY plot objects and report plots. The format of the plot file is described in XY Plot File Format (p. 4071). See Steps for Generating XY Plots of Data from Multiple Sources (p. 4065) for details about the items below.



Controls

Data Sources

contains operations and information related to the selected files/plots.

Load file...

opens the **Select File** dialog box, where you can select the file(s) you want to plot.

Add XY Plot...

opens the **Select Plots** dialog box, where you can select the plot(s) you want to include.

Add Report Plot...

opens the **Select Plots** dialog box, where you can select the report plot(s) you want to include.

Remove Data Source

removes the selected data source from the **Data Source Names** list. This also removes any curves associated with that file.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Data Source Names

lists the files and plots that you have loaded.

Data Sets

lists the available data sets for files and plots (typically `.xy`) that contain two columns monitoring the same value at different locations. You can select which data sets you want to plot by selecting/de-selecting the listed items.

Variables

lists X and Y variables for the selected file.

X Axis Variable

lists the variables that you can select for the X axis. The highlighted X variable will be added to the **Curve Information** list in coordination with the selected **Y Axis Variables**. Changing the selected **X Axis Variable** updates all the related curves shown in the **Curve Information** list.

Y Axis Variables

lists the variables that you can select for the Y axis. The highlighted Y variables will be added to the **Curve Information** list in coordination with the selected **X Axis Variable**. Changing the selected **Y Axis Variables** updates how many curves are shown in the **Curve Information** list.

Curves

lists what items will be plotted and how they will be labeled in the legend.

Curve Information

lists the curves that will be plotted.

Information about each curve is shown in the following order:

1. The name of the file where the data originated.
2. [2 column file (typically `.xy`) only] The location on the model where the data originated.
3. X Variable.
4. Y Variable.

Legend Names

lists the name for each curve that will be listed in the legend when the plot is displayed in the graphics window. (see **Change Legend Entry** to learn how to change the legend name for a curve).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Remove

removes the selected curve from the **Curve Information** list.

Change Legend Entry

allows you to change the name that will appear in the plot legend in the graphics window for a particular curve.

To use:

1. Select a name in the **Legend Names** list.
2. Enter the new name in the text entry box to the right of the **Change Legend Entry** button.
3. Click **Change Legend Entry**.

Plot

contains text entry boxes for the various plot labels.

Title

lets you enter a title for the plot.

Legend Label

lets you enter a title for the legend.

X Axis Label

lets you label the X axis.

Y Axis Label

lets you label the Y axis.

Plot (button)

click **Plot** to plot the data listed in the **Curve Information** list.

Axes... (button)

opens the **Axes** dialog box, which allows you to customize the plot axes.

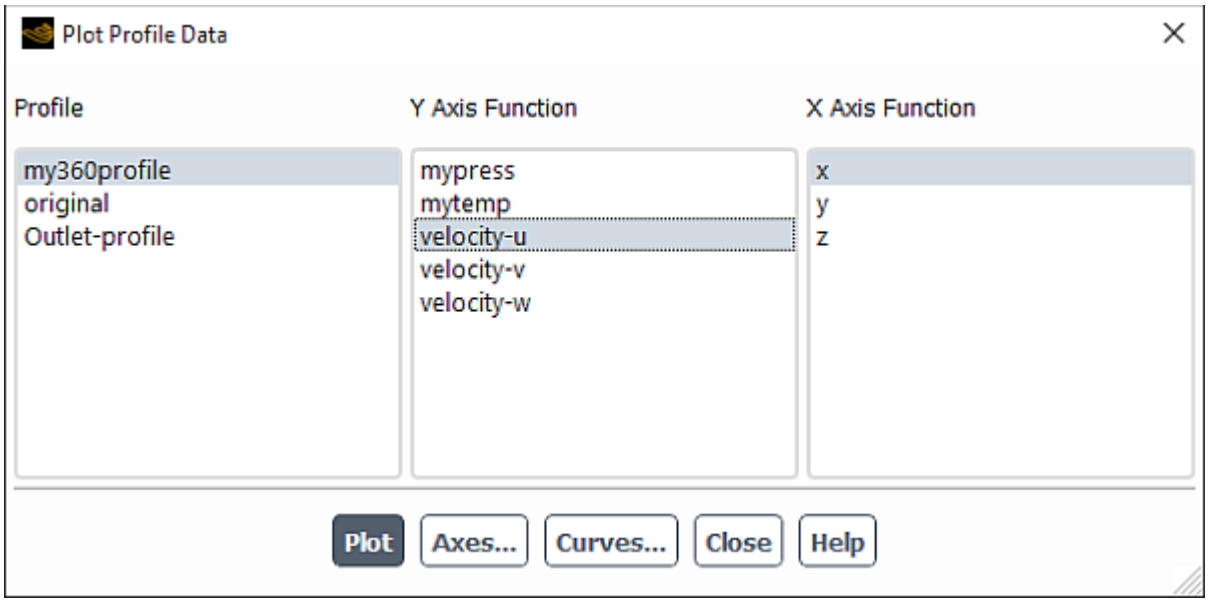
Curves... (button)

opens the **Curves** dialog box, which allows you to customize the curves used in the XY plot.

51.19.4. Plot Profile Data Dialog Box

The **Plot Profile Data** dialog box allows you to display an XY plot of the original data points of a boundary profile before it is interpolated onto the cell faces of a boundary. See Steps for Generating Plots of Profile Data (p. 4066) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Profile

contains a selectable list of available profiles. When a profile is selected its available fields are displayed under **Y Axis Function**.

Y Axis Function

contains a selectable list of the fields available in the selected profile that can be used for the y axis of the plot.

X Axis Function

contains a selectable list of the variables that can be used for the x axis of the plot.

Plot

displays an XY plot of the data points from the selected profile.

Axes...

opens the Axes Dialog Box (p. 5313), which allows you to customize the plot axes.

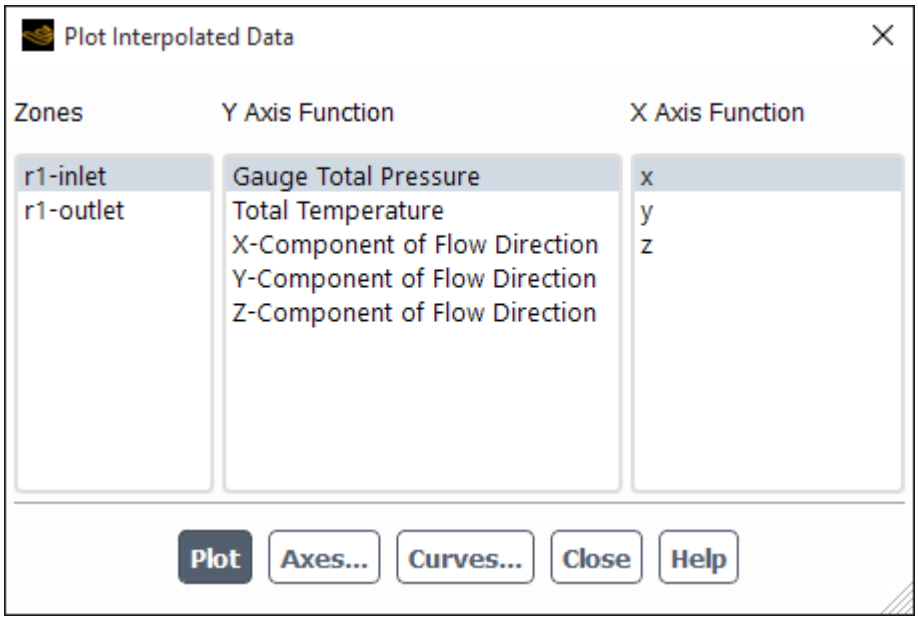
Curves...

opens the Curves Dialog Box (p. 5316), which allows you to customize the curves used in the XY plot.

51.19.5. Plot Interpolated Data Dialog Box

The **Plot Interpolated Data** dialog box allows you to display an XY plot of the values assigned to the cell faces when a profile file has been interpolated on a boundary. See Steps for Generating Plots of Interpolated Profile Data (p. 4067) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Zones

contains a selectable list of the zones for which a profile field has been set as one or more of the parameters are displayed under **Y Axis Function**.

Y Axis Function

contains a selectable list of the profile-related parameters in the selected zone that can be used for the y axis of the plot. The name of the parameter will be the same as that of the drop-down list in the boundary condition dialog box from which the profile field was selected.

X Axis Function

contains a selectable list of the geometry variables that can be used for the x axis of the plot.

Plot

displays an XY plot of the cell face values (as interpolated from the data points of the profile file) of the selected zone.

Axes...

opens the Axes Dialog Box (p. 5313), which allows you to customize the plot axes.

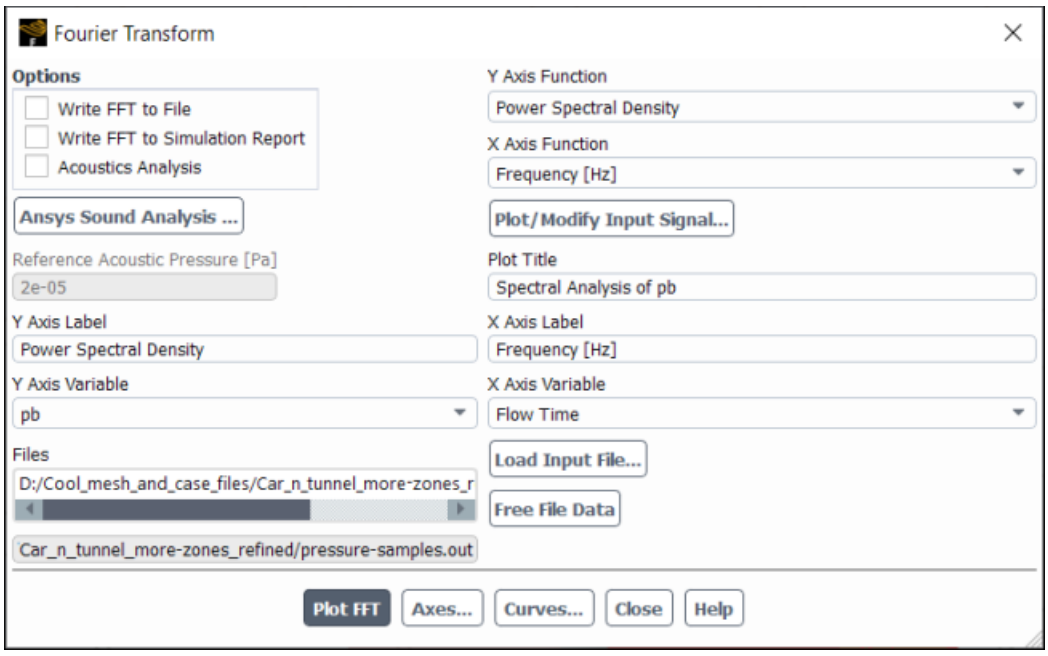
Curves...

opens the Curves Dialog Box (p. 5316), which allows you to customize the curves used in the XY plot.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

51.19.6. Fourier Transform Dialog Box

The **Fourier Transform** dialog box allows you to analyze your time dependent data using the Fast Fourier Transform (FFT) algorithm. See Fast Fourier Transform (FFT) Postprocessing (p. 4078) for details about the items below.



Controls

Options

lets you write the FFT data to a file or display the FFT data in a graphics window.

Write FFT to File

allows you to write out the FFT data directly to a file. When selected, the **Plot FFT...** buttons becomes the **Write FFT...** button.

Acoustics Analysis

enables the acoustics-relevant spectrum options (sound pressure level spectra and frequency band charts), and specifies that the overall sound pressure level is calculated and displayed in the console when you plot or write the FFT data. All dB-measured quantities are based on the **Reference Acoustic Pressure**.

Ansys Sound Analysis...

opens the Ansys Sound Analysis Dialog Box (p. 5304).

Process Options

contains options to analyze signal data.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Process Receiver

allows you to analyze receiver data stored in memory.

Process File Data

allows you to analyze signal data from an existing input file.

Y Axis Function

contains a list from which you can select the function for the y axis.

X Axis Function

contains a list from which you can select the function for the x axis.

Receiver

contains a list of receivers from which you can select when the **Process Receiver** option is enabled.

Plot/Modify Input Signal...

opens the Plot/Modify Input Signal Dialog Box (p. 5309), which you can use to plot the input signal, as well as customize the data set and define spectrum smoothing in preparation for applying the FFT algorithm.

Reference Acoustic Pressure

allows you to specify the reference acoustic pressure (for example, p_{ref} in Equation 40.15 (p. 4086)) used to calculate the dB-measured quantities when the **Acoustics Analysis** option is enabled.

Plot Title

lets you create a new title or edit the original title for the FFT plot. By default, Ansys Fluent adds the string "Spectral Analysis of" to the title originally applied to the input signal plot.

Y Axis Label

allows you to create a new y axis label or edit the original y axis label. By default, the **Y Axis Label** corresponds to the selection in the **Y Axis Function** drop-down list.

X Axis Label

allows you to create a new x axis label or edit the original x axis label. By default, the **X Axis Label** corresponds to the selection in the **X Axis Function** drop-down list.

Y Axis Variable

if your input report file contains multiple report definitions, this allows you to specify the y axis variable.

X Axis Variable

if your input report file contains multiple report definitions, this allows you to specify the x axis variable.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Files

lists the loaded input signal data files.

Load Input File...

loads an input signal data file into Ansys Fluent for FFT analysis. The input file is listed under **Files**.

Free File Data

removes data from FFT analysis once the input signal data file is selected in the **Files** list.

Plot FFT

displays the FFT data in a graphic window and, if **Acoustics Analysis** is enabled, calculates the overall sound pressure level in dB (based on the **Reference Acoustic Pressure**) and displays it in the console. If the **Write FFT to File** option is selected, then this button becomes the **Write FFT** button, which opens a file selection dialog box so that you write the FFT data to a file.

Write FFT

opens The Select File Dialog Box (p. 931) in which you can specify a name and save the FFT data to a file. If **Acoustics Analysis** is enabled, the overall sound pressure level will be calculated in dB (based on the **Reference Acoustic Pressure**) and displayed in the console at this time. If the **Write FFT to File** option is selected, then the **Plot FFT** button changes to the **Write FFT** button.

Axes...

opens the Axes Dialog Box (p. 5313), which allows you to customize the plot axes.

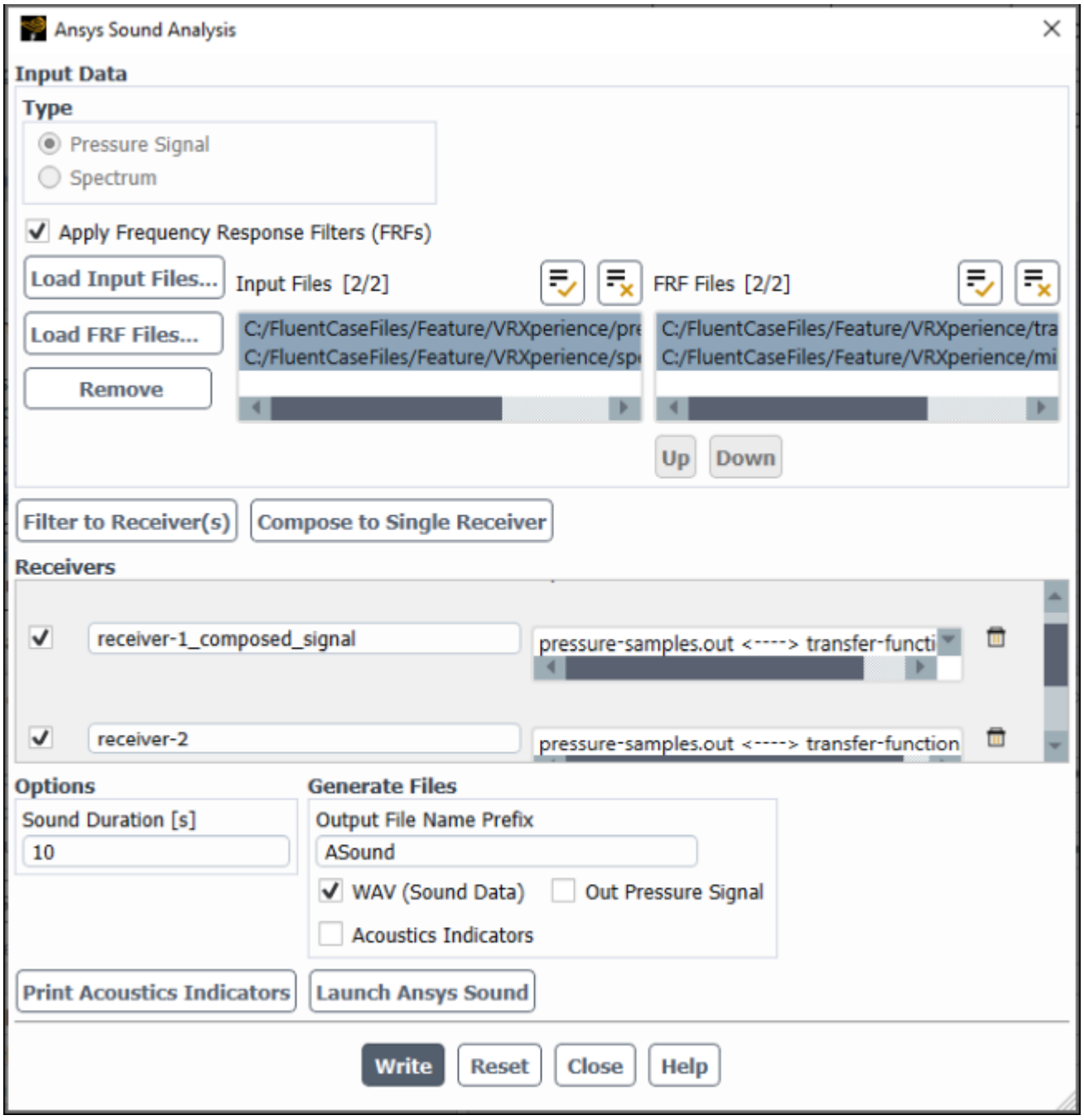
Curves...

opens the Curves Dialog Box (p. 5316), which allows you to customize the curves used in the plot.

51.19.7. Ansys Sound Analysis Dialog Box

Refer to Performing an Ansys Sound Analysis (p. 4089) for additional information on this dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Input Data

Pressure Signal

specifies that the input file you select is an acoustic pressure file (two column file of time vs. pressure).

Spectrum

specifies that the input file you select is an FFT output file (two column file of frequency vs. power spectral density).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Apply Frequency Response Filters (FRFs)

allows you to load a frequency response filter (FRF) file.

Load Input Files...

allows you to read in either a pressure data or spectrum input file.

Load FRF Files...

allows you to read in frequency response filter files.

Remove

removes the selected file from the **Input Files** and **FRF Files** lists.

Input Files

lists the loaded input pressure signal or spectrum files.

FRF Files

lists the loaded frequency response filter (FRF) files.

Up

moves the selected FRF file up in the list.

Down

moves the selected FRF file down in the list.

Filter to Receiver(s)

adds one or more receivers (based on the number of selected files) that are mapped with one input file and one frequency response filter file per receiver to the **Receivers** list. *For example, you could have 3 input files and 3 FRF files selected and this would result in 3 new receivers. You could also have 1 input file selected and 3 FRF files selected, which would also result in 3 new receivers.*

Compose to Single Receiver

combines the selected input files and corresponding FRF files into a single receiver that gets added to the **Receivers** list.

Receivers

lists the defined receivers and their comprising input and FRF files, with arrows indicating how they correspond.

Active

indicates whether the receiver is active. That is, whether or not the receiver will be included or evaluated when you click any of the following buttons: **Print Acoustic Indicators**, **Launch Ansys Sound**, **Write**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name

displays the (editable) name of each defined receiver.

Input Files <---> FRF Files

displays the input and FRF files included within a receiver. The arrow indicates which input file is modified by which FRF file.

Options

Sound Duration

specifies the length of the sound clip for saving a WAV file.

Generate Files

Output File Name Prefix

specifies the name (aside from the extension) for any files saved when you click **Write**.

WAV (Sound Data)

specifies that a WAV sound file will be written when you click **Write**.

Out Pressure Signal

specifies that a pressure signal file will be written when you click **Write**.

Acoustic Indicators

specifies that a file containing the computed acoustic indicators will be written when you click **Write**.

Print Acoustic Indicators

computes the acoustic indicators for the selected files and prints them to the console.

Launch Ansys Sound

launches Ansys Sound and loads the defined receivers.

Write

computes the acoustic indicators for the selected files and prints them to the console.

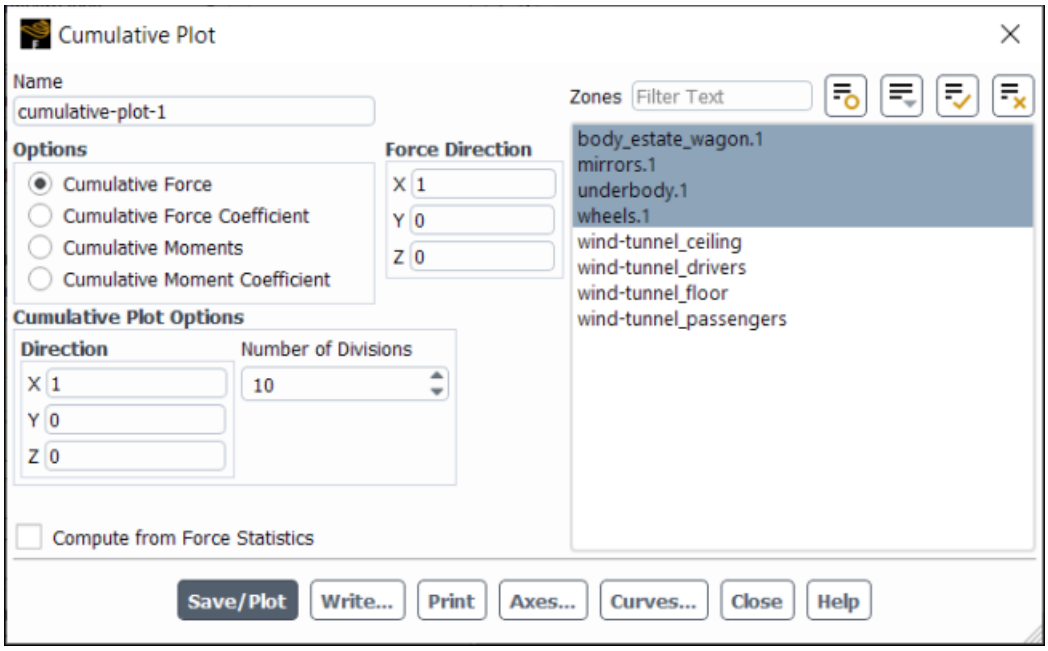
Reset

clears all settings and default changes from the dialog box.

51.19.8. Cumulative Plot Dialog Box

The **Cumulative Plot** dialog box allows you to analyze the development of force, moments, or coefficients for your model in the any desired location or direction. See Cumulative Force, Moment, and Coefficients Plots (p. 4094) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name

name of the cumulative plot object.

Options

lets you select which type of plot to display.

Cumulative Force

displays the force development for the specified direction and walls.

Cumulative Force Coefficient

displays the force coefficient development for the specified direction and walls.

Cumulative Moments

displays the moment development for the specified direction and walls.

Cumulative Moment Coefficient

displays the moment coefficient development for the specified direction and walls.

Cumulative Plot Options

contains options to specify the direction and number of divisions where Fluent will calculate and plot the selected variable.

Direction

allows you to specify the direction along which the cumulative force or moments will be computed.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Number of Divisions

specifies the number of slices across the selected wall zones where the force or moments will be evaluated for the resulting cumulative plot. The planes will be created perpendicular to the **Direction**.

Compute from Force Statistics

use averaged values computed from force statistics instead of the default instantaneous values.

Force Direction

(**Cumulative Force** and **Cumulative Force Coefficient** only) allows you to specify the force component direction.

Moment Center

(**Cumulative Moments** and **Cumulative Moment Coefficient** only) allows you to specify the coordinates of the moment center.

Moment Axis

(**Cumulative Moments** and **Cumulative Moment Coefficient** only) allows you to specify the coordinates of the moment axis.

Zones

lists the zones that you can select for the cumulative plot.

Save/Plot

saves the cumulative plot object and plots the cumulative force, force coefficient, moments, or moment coefficient.

Write...

opens The Select File Dialog Box (p. 931), allowing you to save the plot data to a file.

Print

prints the value of the force or moment at each division to the console.

Axes...

opens the Axes Dialog Box (p. 5313), which allows you to customize the plot axes.

Curves...

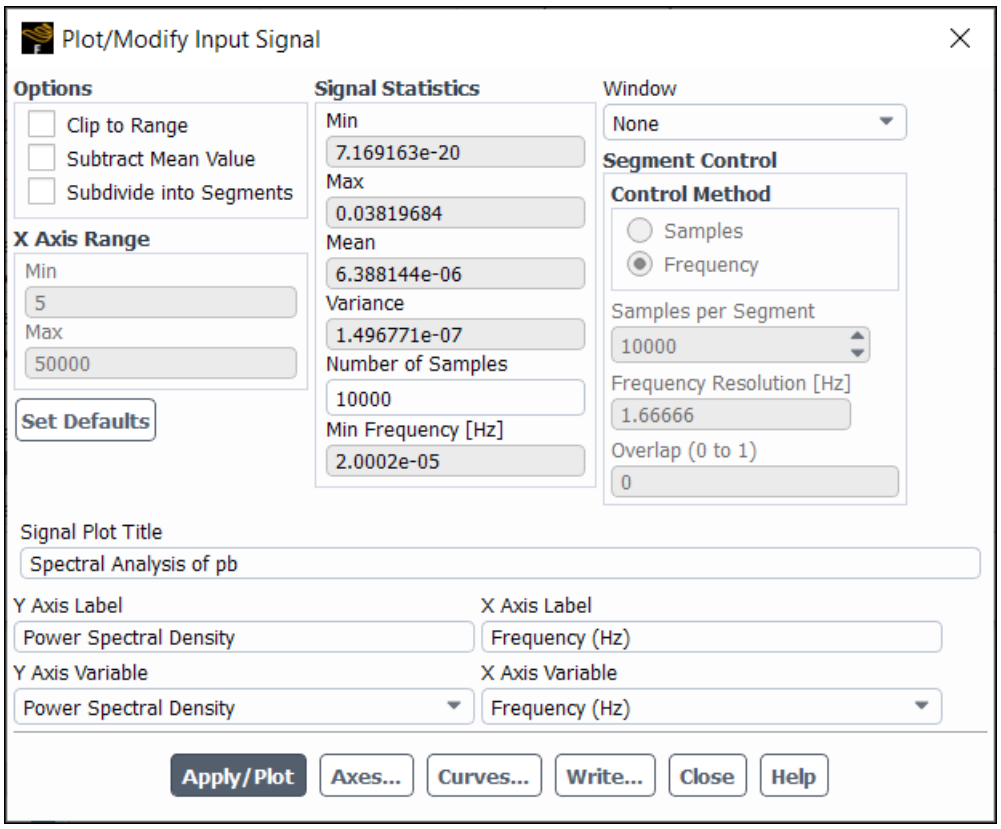
opens the Curves Dialog Box (p. 5316), which allows you to customize the curves used in the plot.

51.19.9. Plot/Modify Input Signal Dialog Box

The **Plot/Modify Input Signal** dialog box allows you to plot the input signal, as well as customize the data set and define spectrum smoothing in preparation for applying the FFT algorithm. It is

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

opened from the Fourier Transform Dialog Box (p. 5302). See Using the FFT Utility (p. 4081) for details about the items below.



Controls

Options

allows you to process some or all of the input signal data.

Clip to Range

allows you to analyze a portion of the input signal by specifying data range.

Subtract Mean Value

reduces y axis quantities by the mean value of the relevant signal property.

Subdivide into Segments

enables / disables spectrum smoothing, which can suppress spurious amplitude fluctuations in the Fourier spectrum by splitting the input signal into multiple overlapping segments, so that Ansys Fluent can apply the FFT algorithm on each segment and then average the resulting spectra. The spectrum smoothing is controlled by the settings in the **Segment Control** group box. See Customizing the Input and Defining the Spectrum Smoothing (p. 4083) for further details.

Signal Statistics

displays signal information.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Min

displays the minimum value for the input signal.

Max

displays the maximum value for the input signal.

Mean

displays the average value for the input signal.

Variance

displays the variance for the input signal.

Number of Samples

displays the total number of samples in the input signal data set.

Min Frequency

displays the finest possible frequency resolution for the input signal.

X Axis Range

allows for a portion of the input signal to be analyzed when the **Process the whole data** option is turned off.

Min

specifies a minimum value for the x axis.

Max

specifies a maximum value for the x axis.

Set Defaults

resets the x axis minimum and maximum values to their original value and turns on the **Process the whole data** option.

Signal Plot Title

lets you create a new title or edit the original title for the input signal plot. By default, the **Signal Plot Title** corresponds with the title originally applied to the input signal plot.

Y Axis Label

allows you to create a new y axis label or edit the original y axis label. By default, the **Y Axis Label** corresponds with the y axis label originally applied to the input signal plot.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

X Axis Label

allows you to create a new x axis label or edit the original x axis label. By default, the **X Axis Label** corresponds with the x axis label originally applied to the input signal plot.

Y Axis Variable

if your input report file contains multiple report definitions, this allows you to specify the y axis variable.

X Axis Variable

if your input report file contains multiple report definitions, this allows you to specify the x axis variable.

Window

lets you specify a windowing technique to remove discontinuities in the FFT calculation.

None

applies no windowing technique. This is the default setting.

Hamming

applies the Hamming technique.

Hanning

applies the Hanning technique.

Bartlett

applies the Bartlett technique.

Blackman

applies the Blackman technique.

Segment Control

provides settings that control the spectrum smoothing when **Subdivide into Segments** is enabled.

Control Method

allows you to specify how you want to define the segment size from the following:

Samples

bases the segment size on the **Samples per Segment**.

Frequency

bases the segment size on the **Frequency Resolution** Δf in Hertz units.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Samples per Segment

defines the number of samples in each segment when **Samples** is selected from the **Control Method** list.

Frequency Resolution

defines the desired frequency resolution Δf in Hertz units when **Frequency** is selected from the **Control Method** list.

Overlap

ranges from 0 to 1 and defines how much adjacent segments overlap. This setting is applicable regardless of which **Control Method** you select.

Apply/Plot

applies any changes you have made in the dialog box and displays the input signal data in a graphics window.

Axes...

opens the Axes Dialog Box (p. 5313), which allows you to customize the plot axes.

Curves...

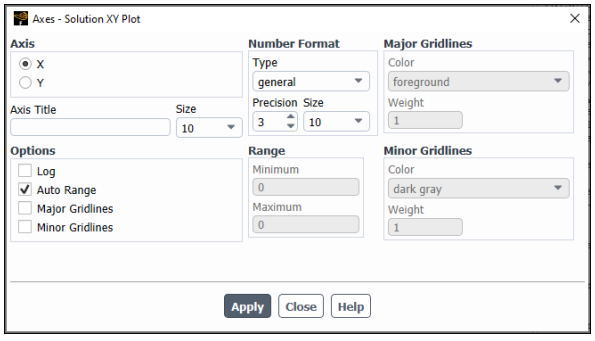
opens the Curves Dialog Box (p. 5316), which allows you to customize the curves used in the plot.

Write...

opens The Select File Dialog Box (p. 931), in which you can save the signals to a file.

51.19.10. Axes Dialog Box

The **Axes** dialog box allows you to independently control the characteristics of the ordinate and abscissa on an XY plot or histogram. You can change the titles, scale, range, number format, and major and minor rules visibility and appearance. Note that the title following **Axes** in the dialog box indicates which plot environment you are changing. You can set different parameters for each type of plot that Ansys Fluent can generate. See Using the Axes Dialog Box (p. 4075) for details about the items below.



Controls

Axis

contains check buttons that allow you to set abscissa (x-axis) or ordinate (y-axis) characteristics.

X

allows you to specify the abscissa characteristics.

Y

allows you to specify the ordinate characteristics.

Axis Title

defines the character string that will label the active axis (the one selected in **Axis**) in the display.

Size

specifies the font size of the **Axis Title**.

Options

contains check buttons that (de)activate scale, range, major rules, and minor rules.

Log

toggles logarithmic scaling of the active axis. By default, decimal scaling is used.

Auto Range

toggles automatic computation of the range of the active axis. If you deactivate this option, you may input the **Minimum** and **Maximum** values in the **Range** box.

Major Rules

toggles the display of major rules on the active axis. Major rules are the horizontal or vertical lines that mark the primary data divisions and span the whole plot window to produce a "grid."

Minor Rules

toggles the display of minor rules on the active axis. Minor rules are the horizontal or vertical lines that mark the secondary data divisions and span the whole plot window to produce a "grid."

Number Format

contains controls for changing the format of the data labels on the active axis. Data labels are the character strings used to define the primary data divisions on the axes.

Type

sets the form of the data labels. You may select from a drop-down list of options, including the following:

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

general

displays the real value with either float or exponential form based on the size of the number and the defined **Precision**.

float

displays the real value with an integral and fractional part (for example, 1.0000), where the number of digits in the fractional part is determined by **Precision**.

exponential

displays the real value with a mantissa and exponent (for example, 1.0e-02), where the number of digits in the fractional part of the mantissa is determined by **Precision**.

Precision

defines the number of fractional digits displayed in the data labels.

Size

specifies the size of the font used for the data labels.

Range

contains the range or extents of the active axis. To set the range manually, you must turn off **Auto Range**. Otherwise the extents are computed automatically.

Minimum

sets the minimum data value for the active axis.

Maximum

sets the maximum data value for the active axis.

Major Rules

contains controls for modifying the appearance of the major rules. To use these controls you must activate **Major Rules** in the **Options** list.

Color

sets the color of the major rules from a drop-down list with numerous color selections.

Weight

sets the line thickness of the major rule. A line of weight 1.0 is normally 1 pixel wide. A weight of 2.0 would make the line twice as thick (2 pixels wide).

Minor Rules

contains controls for modifying the appearance of the minor rules. To use these controls you must activate **Minor Rules** in the **Options** list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color

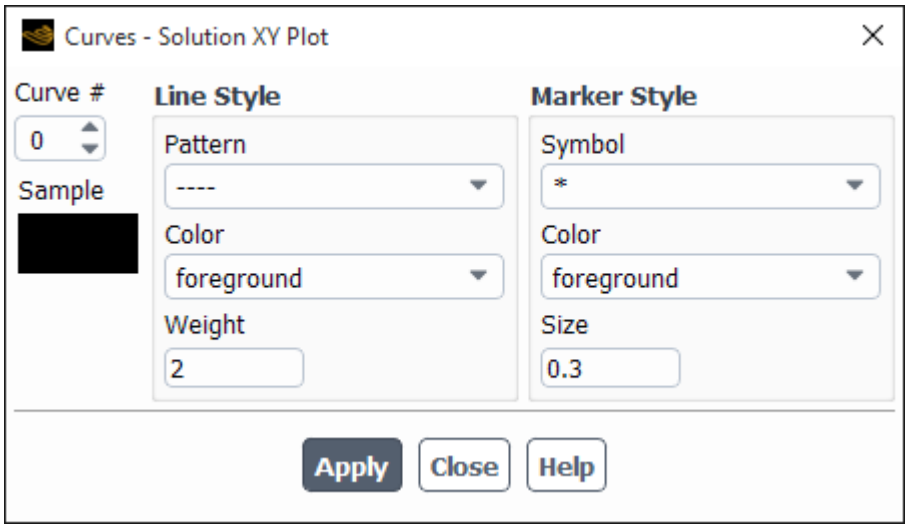
sets the color of the minor rules from a drop-down list with numerous color selections.

Weight

sets the line thickness of the minor rule. A line of weight 1.0 is 1 pixel wide. A weight of 2.0 would make the line twice as thick (2 pixels wide).

51.19.11. Curves Dialog Box

The **Curves** dialog box allows you to modify the appearance of the lines and markers used in XY plots. Note that the title following **Curves** in the dialog box indicates which plot environment you are changing. You can set different parameters for each type of plot that Ansys Fluent can generate. See Using the Curves Dialog Box (p. 4077) for details about the items below.



Controls

Curve #

defines the active curve number. The present and future marker and line styles apply to the defined curve number. The curves are numbered sequentially, starting from 0. For example, if you were plotting flow-field values on two zones, the first zone would be curve 0, and the second, curve 1. If the plot contains only one curve, the **Curve #** is set to 0 and is not editable.

Sample

displays a single marker and line with the current style attributes.

Line Style

contains controls for modifying the appearance of the active curve.

Pattern

sets the pattern of the active curve. A drop-down list allows you to set the line pattern. Except for **center** and **phantom** lines, the list displays examples of the pattern choices. A **center** line

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

alternates a very long dash and a short dash and a **phantom** line alternates a very long dash and a double short dash.

Color

sets the color of the active curve. A drop-down list allows you to select from a list of color names.

Weight

sets the thickness of the active curve. A line weight of 1.0 is normally 1 pixel wide. Therefore, a weight of 2.0 would make the line twice as thick (2 pixels wide).

Marker Style

contains controls for modifying the appearance of the active curve's marker.

Symbol

sets the symbol used to mark data. You can select the symbol from a drop-down list that contains all the symbol choices. The **Sample** box will allow you to experiment with various markers. For example, in plotting pressure-coefficient data on the upper and lower surfaces of an airfoil, the symbol `/*\` (filled-in upward-pointing triangle) could be used for the marker representing the upper surface data, and the symbol `\*/` (filled-in downward-pointing triangle) could be used for the marker representing the lower surface data.

Color

sets the color of the marker on the active line number. A drop-down list allows you to select from a list of color names.

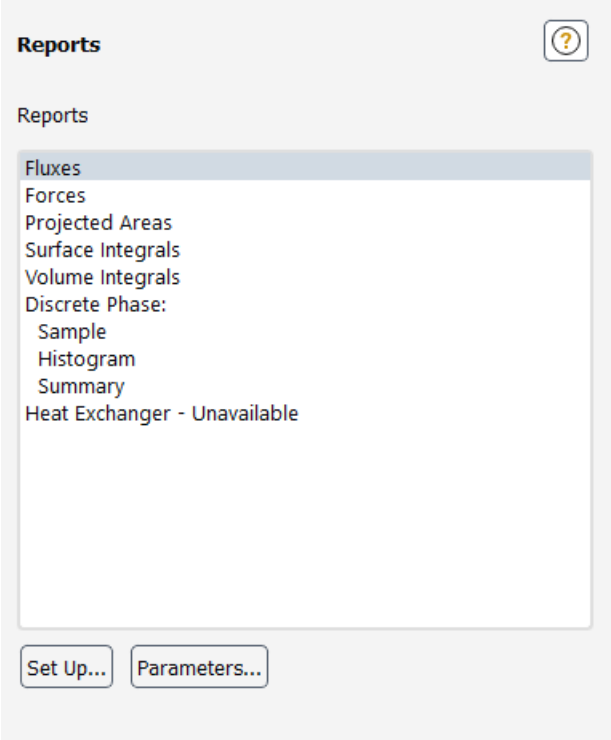
Size

sets the size of the data marker. A symbol of size 1.0 is 3.0% of the height of the display screen, except for the "." symbol, which is always one pixel.

51.20. Reports Task Page

The **Reports** task page allows you to set up reports for your CFD simulation. Reports can be compiled for fluxes, forces, projected areas, surface and volume integrals, among others. See Reporting Alphanumeric Data (p. 4097) for more information.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Reports

displays a list of available report types in Ansys Fluent.

You can double-click an item in the **Reports** list to open the corresponding dialog box, or you can select the item in the list and click the **Set Up...** button.

Fluxes

selecting this item and clicking the **Set Up...** button opens the Flux Reports Dialog Box (p. 5320).

Forces

selecting this item and clicking the **Set Up...** button opens the Force Reports Dialog Box (p. 5322).

Projected Areas

selecting this item and clicking the **Set Up...** button opens the Projected Surface Areas Dialog Box (p. 5324).

Surface Integrals

selecting this item and clicking the **Set Up...** button opens the Surface Integrals Dialog Box (p. 5325).

Volume Integrals

selecting this item and clicking the **Set Up...** button opens the Volume Integrals Dialog Box (p. 5329).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Discrete Phase

allows you to report on one of the following three report types:

Sample

selecting this item and clicking the **Set Up...** button opens the Sample Trajectories Dialog Box (p. 5331).

Histogram

selecting this item and clicking the **Set Up...** button opens the Trajectory Sample Histograms Dialog Box (p. 5333).

Summary

selecting this item and clicking the **Set Up...** button opens the Particle Summary Dialog Box (p. 5336).

Heat Exchanger

selecting this item and clicking the **Set Up...** button opens the Heat Exchanger Report Dialog Box (p. 5337).

Set Up...

opens the dialog box corresponding to the selected item in the **Reports** list.

Parameters...

opens the Parameters Dialog Box (p. 5338).

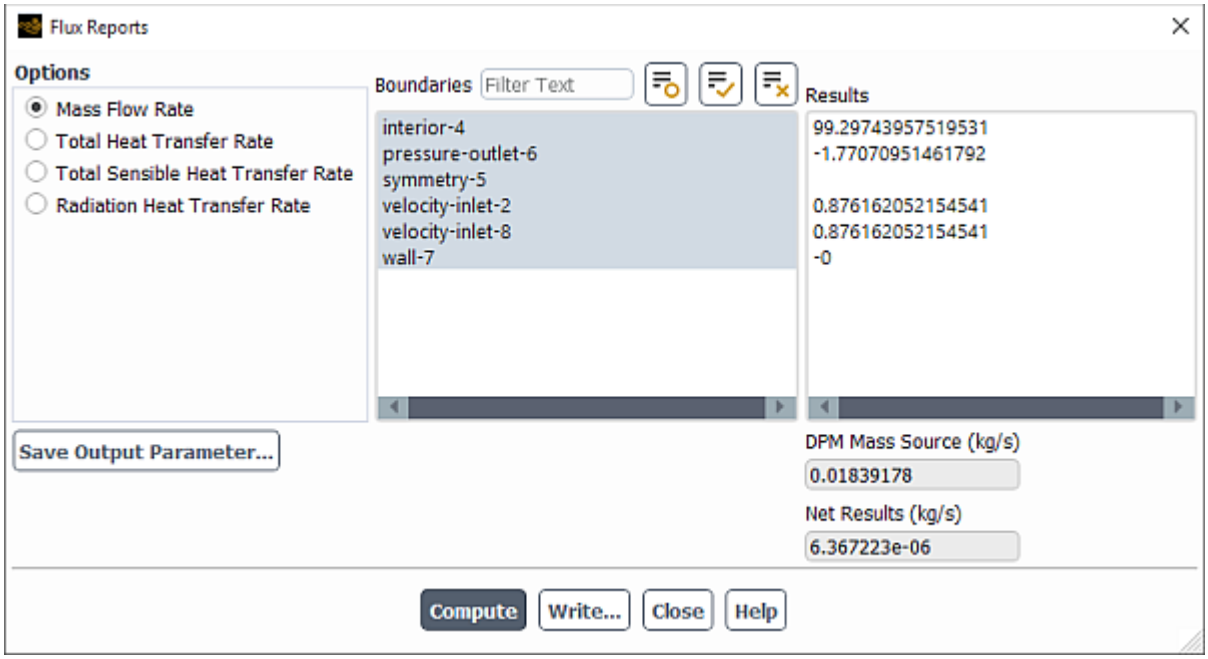
For additional information, see the following sections:

51.20.1. Flux Reports Dialog Box
51.20.2. Force Reports Dialog Box
51.20.3. Projected Surface Areas Dialog Box
51.20.4. Surface Integrals Dialog Box
51.20.5. Volume Integrals Dialog Box
51.20.6. Sample Trajectories Dialog Box
51.20.7. Trajectory Sample Histograms Dialog Box
51.20.8. Particle Summary Dialog Box
51.20.9. Heat Exchanger Report Dialog Box
51.20.10. Parameters Dialog Box
51.20.11. Use Input Parameter in Scheme Procedure Dialog Box
51.20.12. Use Input Parameter for UDF Dialog Box
51.20.13. Rename Dialog Box
51.20.14. Parameter Expression Dialog Box
51.20.15. Save Output Parameter Dialog Box

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

51.20.1. Flux Reports Dialog Box

The **Flux Reports** dialog box allows you to compute the mass flow rate, heat transfer rate, and radiation heat transfer rate on selected boundary zones. See Fluxes Through Boundaries (p. 4135) for details.



Controls

Options

contains the following option buttons:

Mass Flow Rate

specifies that the mass flow rate is computed for the selected boundary zones.

Total Heat Transfer Rate

specifies that the total heat transfer rate is computed for the selected boundary zones.

Total Sensible Heat Transfer Rate

specifies that the total sensible heat transfer rate is computed for the selected boundary zones. It reports the total energy flux as defined in Equation 5.5 in the Theory Guide.

Radiation Heat Transfer Rate

specifies that the radiation heat transfer rate is computed for the selected boundary zones.

Pressure Work Rate

specifies that the pressure work rate is computed for the selected boundary zones. This rate is the energy transfer due to pressure when a fluid zone undergoes normal zone motion relative to an adjacent solid zone (for further details, see Equation 25.1 in the Theory Guide). This pressure work is on the fluid side only, not on the solid side. This option is only available

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

when the energy equation is enabled, the absolute velocity formulation is selected, and the **Frame Motion** and/or **Mesh Motion** option is enabled in the **Fluid** dialog box.

Viscous Work Rate

specifies that the viscous work rate is computed for the selected boundary zones, which is the integral of the product of viscous stress with velocity over the external boundary of the domain with the boundary normal directed inside (for further details, see Equation 25.2 in the Theory Guide). Note that after reading case and data files into Ansys Fluent, the viscous work rate is zero. You need to run at least one iteration or time step to obtain the viscous work rate value. This item is available only when the energy equation is enabled and the pressure-based solver is selected.

Film Mass Flow Rate

specifies that the mass flow rate is computed for the selected boundary zone(s). This option is available only when the Eulerian wall film model is enabled.

Film Heat Transfer Rate

specifies that the film heat transfer rate is computed for the selected boundary zone(s). This option is available only when the Eulerian wall film model is enabled.

Electric Current Rate

specifies that the electric current rate is computed for the selected boundary zone(s). This option is available only if the potential equation is enabled—see Modeling Electric Potential Field and Electrochemistry Models (p. 3265).

Save Output Parameter...

opens the Save Output Parameter Dialog Box (p. 5345).

Boundaries

contains a selectable list of valid boundary zones for flux reporting.

Results

displays the results of the selected flux computation for each boundary zone selected. The summation of the individual zone flux results is displayed in the box below the **Results** list.

Compute

computes the flux for each of the selected boundary zones and updates **Net Results** (for example, in kg/s, W, and so on) and other reports, for example, **Heat of Reaction Source**.

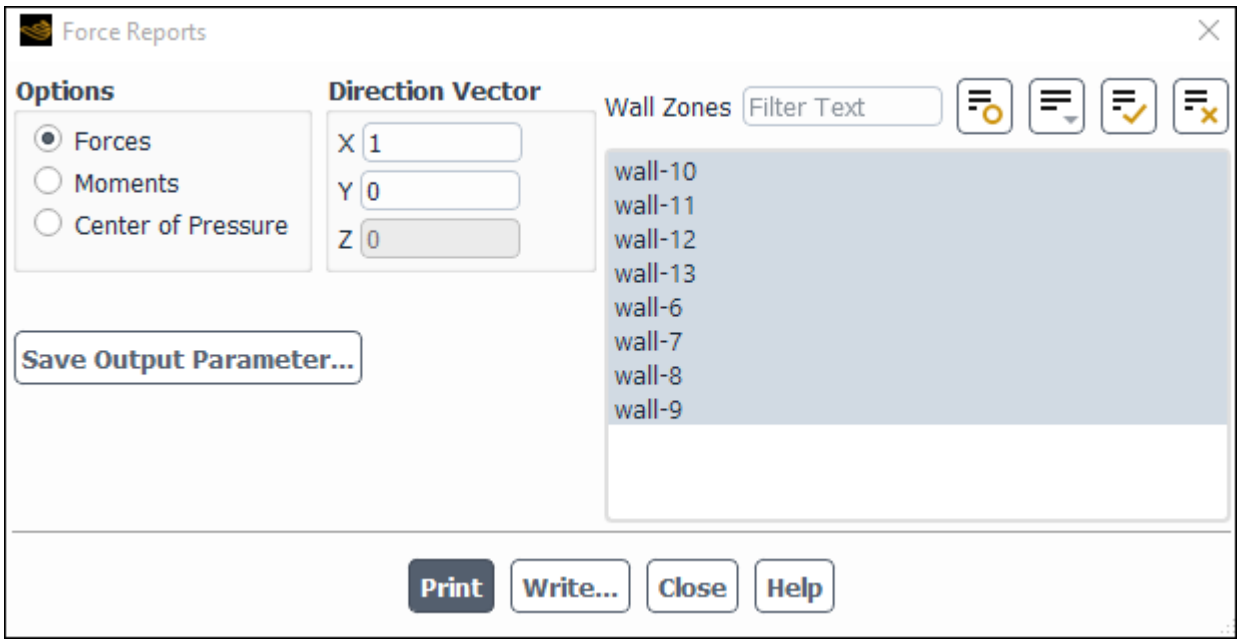
Write...

opens the **Select File** dialog box (The Select File Dialog Box (p. 931)), which you can use to save the reported values to a file.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

51.20.2. Force Reports Dialog Box

The **Force Reports** dialog box allows you to compute the forces along a specified vector, moments about a specified center, and the coordinates of the center of pressure for selected wall zones. See Forces on Boundaries (p. 4145) for details.



Controls

Options

contains option buttons that control computation of the forces, moments, or center of pressure.

Forces

enables the computation of the pressure and viscous forces.

Moments

enables the computation of the pressure and viscous moments.

Center of Pressure

enables the computation of the coordinates about which the moment by pressure forces is zero.

Direction Vector

contains the components of the force vector. This label is visible when the **Forces** option button is active.

X,Y,Z

are the components of the force vector along which the forces will be computed.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Moment Center

contains the Cartesian coordinates of the moment center. This label is visible when the **Moments** option button is active.

X,Y,Z

are the Cartesian coordinates of the moment center about which moments will be computed.

Moment Axis

contains the Cartesian coordinates of the moment axis. This label is visible when the **Moments** option button is active.

X,Y,Z

are the Cartesian coordinates of the moment axis about which moments will be computed.

Coordinate

contains the value of the Cartesian coordinate that is fixed. This label is visible when the **Center of Pressure** option button is active.

X,Y,Z

are the Cartesian coordinates, one of which will be fixed.

Value (n)

is the point where the selected Cartesian coordinate will be fixed.

Wall Zones

contains a selectable list of wall zones and porous zones (when applicable). The force or moment information is printed for each zone, and then a total force or moment for all the zones is presented.

Save Output Parameter...

opens the Save Output Parameter Dialog Box (p. 5345).

Print

displays (in the console) the pressure, viscous (if appropriate), and total forces or moments, and the pressure, viscous, and total force coefficients along the specified force vector or moment center for the selected wall zones. The center of pressure coordinates will print to the console when the **Center of Pressure** option is activated.

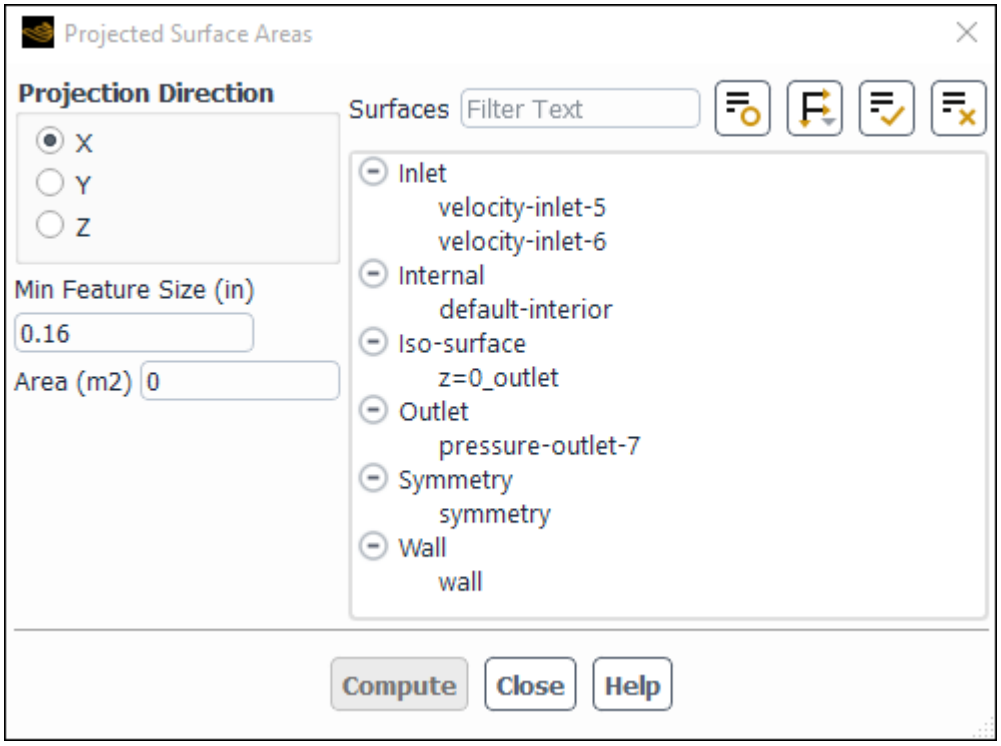
Write...

opens The Select File Dialog Box (p. 931), which you can use to save the reported values to a file.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

51.20.3. Projected Surface Areas Dialog Box

The **Projected Surface Areas** dialog box allows you to compute an estimated area of the projection of selected surfaces along the x, y, or z axis. See Projected Surface Area Calculations (p. 4149) for details.



Controls

Projection Direction

indicates the direction along which to project the surface.

Min Feature Size

specifies the length of the smallest feature in the geometry that you want to resolve in the area calculation.

Area

displays the computed projected area.

Surfaces

contains a list of existing surfaces. You can select the surface(s) for which the projected area is to be calculated in this list.

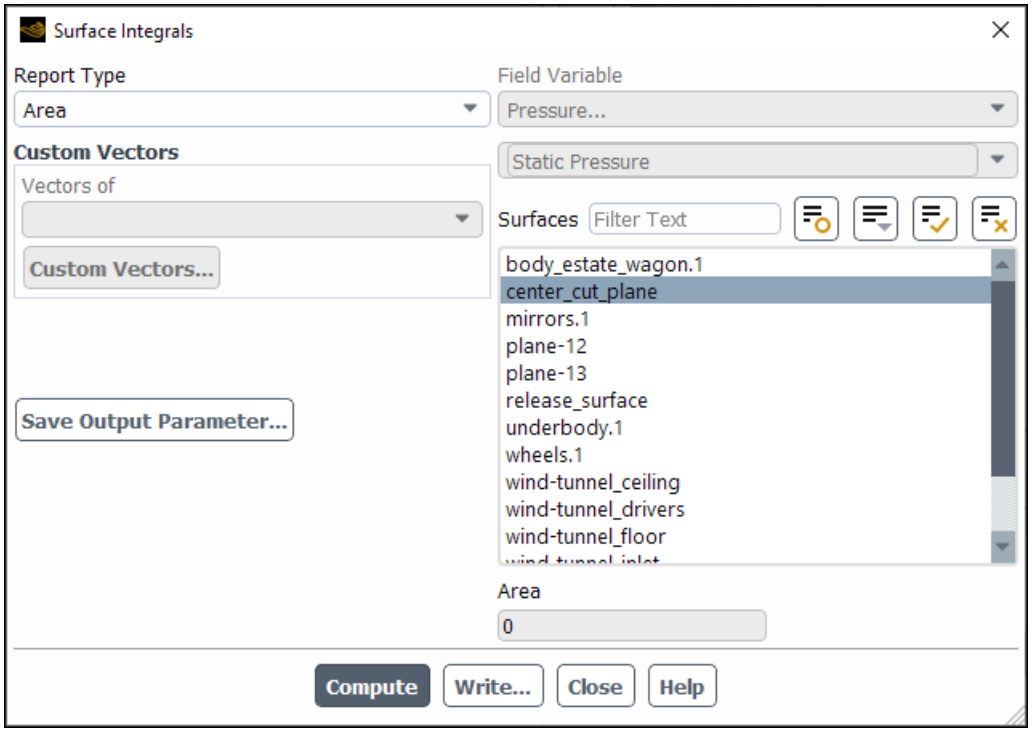
Compute

computes the area of the selected surfaces projected along the selected direction. The area will be printed in the **Area** box and in the console window.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

51.20.4. Surface Integrals Dialog Box

The **Surface Integrals** dialog box allows you to compute the area, mass and volume flow rate, standard deviation, integral, flow rate, area-weighted average, custom vector-based flux, custom vector flux, custom vector-weighted average, mass-weighted average, sum, facet average, facet minimum/maximum, uniformity index (weighted by area or mass), vertex average, or vertex minimum/maximum quantity of a specified field variable on a selected list of surfaces. See Surface Integration (p. 4150) for details.



Controls

Report Type

contains drop-down list that has options that control the method of surface integration.

Area

turns on the computation of the surface area.

Area-Weighted Average

turns on the computation of the area-weighted average on the surface(s).

Custom Vector Based Flux

turns on the computation of the vector-based flow rate of a quantity through the surface(s).

Custom Vector Flux

turns on the computation of the flow rate on a vector through the surface(s).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Custom Vector Weighted Average

turns on the computation of vector-weighted average of a quantity on the surface(s).

Facet Average

turns on the computation of the facet-averaged quantity on the surface(s).

Facet Minimum

turns on the computation of the facet minimum of a quantity on the surface(s).

Facet Maximum

turns on the computation of the facet maximum of a quantity on the surface(s).

Flow Rate

turns on the computation of the flow rate through the surface(s).

Integral

turns on the computation of the integral on the surface(s).

Mass Flow Rate

turns on the computation of the mass flow rate through the surface(s).

Mass-Weighted Average

turns on the computation of the mass-averaged quantity on the surface(s).

Standard Deviation

turns on the computation of the standard deviation of a specified field variable on a surface.

Sum

turns on the computation of the summed quantity on the surface(s).

Uniformity Index - Mass Weighted

turns on the computation of the mass-weighted uniformity index of a quantity on the surface(s).

Uniformity Index - Area Weighted

turns on the computation of the area-weighted uniformity index of a quantity on the surface(s).

Vertex Average

turns on the computation of the vertex-averaged quantity on the surface(s).

Vertex Minimum

turns on the computation of the vertex minimum of a quantity on the surface(s).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Vertex Maximum

turns on the computation of the vertex maximum of a quantity on the surface(s).

Volume Flow Rate

turns on the computation of the volume flow rate through the surface(s).

Custom Vectors

allows you to define custom vectors if you selected **Custom Vector Based Flux**, **Custom Vector Flux**, or **Custom Vector Weighted Average** from the **Report Type** drop-down list.

Field Variable

contains a list of the field variables that can be used in the surface integrations. This option is not active if the **Area**, **Mass Flow Rate**, or **Volume Flow Rate** option is active.

Surfaces

is a selectable list of surfaces.

Area

displays the result of the area summation over all the selected surfaces. This label is visible when **Area** is active.

Area-Weighted Average

displays the result of the area-weighted average computation over all the selected surfaces. This label is visible when **Area-Weighted Average** is active.

Average of Facet Values

displays the result of the facet-averaged computation over all the selected surfaces. This label is visible when **Facet Average** is active.

Minimum of Facet Values

displays the minimum facet value on all the selected surfaces. This label is visible when **Facet Minimum** is active.

Maximum of Facet Values

displays the maximum facet value on all the selected surfaces. This label is visible when **Facet Maximum** is active.

Flow Rate

displays the result of the flow rate computation over all the selected surfaces. This label is visible when **Flow Rate** is active.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Integral

displays the result of the integral computation over all the selected surfaces. This label is visible when **Integral** is active.

Mass Flow Rate

displays the result of the mass flow rate computation over all the selected surfaces. This label is visible when **Mass Flow Rate** is active.

Mass-Weighted Average

displays the result of the mass-averaged computation over all the selected surfaces. This label is visible when **Mass-Weighted Average** is active.

Standard Deviation

displays the result of the standard deviation computation over all the selected surfaces. This label is visible when **Standard Deviation** is active.

Sum of Facet Values

displays the result of the summation over all the selected surfaces. This label is visible when **Sum** is active.

Uniformity Index Mass-Wt.

displays the mass-weighted uniformity index value on all the selected surfaces. This label is visible when **Uniformity Index - Mass Weighted** is active.

Uniformity Index Area-Wt.

displays the area-weighted uniformity index value on all the selected surfaces. This label is visible when **Uniformity Index - Area Weighted** is active.

Average of Surface Vertex Values

displays the result of the vertex-averaged computation over all the selected surfaces. This label is visible when **Vertex Average** is active.

Minimum of Vertex Values

displays the minimum facet value on all the selected surfaces. This label is visible when **Vertex Minimum** is active.

Maximum of Vertex Values

displays the maximum facet value on all the selected surfaces. This label is visible when **Vertex Maximum** is active.

Volumetric Flow Rate

displays the result of the volumetric flow rate computation over all the selected surfaces. This label is visible when **Volume Flow Rate** is active.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Save Output Parameter...

opens the Save Output Parameter Dialog Box (p. 5345).

Compute

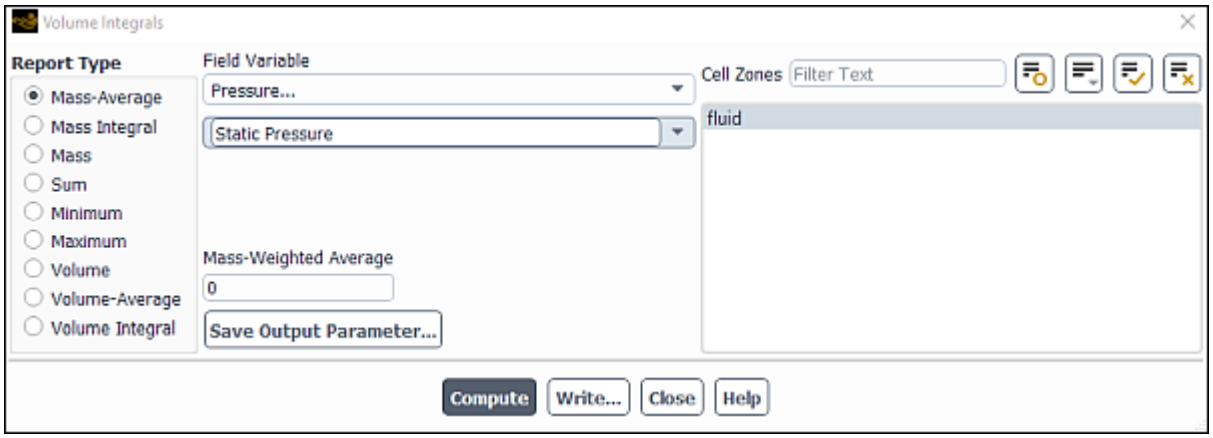
computes the specified integration on the selected surfaces.

Write...

opens The Select File Dialog Box (p. 931), which you can use to save the reported values to a file.

51.20.5. Volume Integrals Dialog Box

The **Volume Integrals** dialog box allows you to compute the volume, sum, maximum, minimum, volume integral, volume-averaged quantity, mass integral, or mass-averaged quantity of a specified field variable on a selected list of cell zones. See Volume Integration (p. 4153) for details.



Controls

Report Type

contains option buttons that control the method of volume integration.

Mass-Average

turns on the computation of the mass-averaged quantity on the cell zone.

Mass Integral

turns on the computation of the mass integral on the cell zone.

Mass

turns on the computation of the total mass in the cell zone for the selected phase.

Sum

turns on the computation of the summation over all cells in the selected zone.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sum*2Pi

computes the summation over all cells in the selected zone of a 2D axisymmetric model and multiplies the result by 2π to give the summation over the revolved domain. This option is only available in 2D axisymmetric cases.

Minimum

computes the minimum value of the selected variable at each cell in the selected zone.

Maximum

computes the maximum value of the selected variable at each cell in the selected zone.

Volume

turns on the computation of the cell zone volume.

Volume Integral

turns on the computation of the volume integral on the cell zone.

Volume-Average

turns on the computation of the volume-weighted average on the cell zone.

Field Variable

contains a list of the field variables that can be used in the sum, volume integral, and average computations. This option is not active if the **Volume** option is active.

Cell Zones

is a selectable list of cell zones.

Total Volume

displays the result of the volume computation over all the selected zones. This label is visible when **Volume** is active.

Sum

displays the result of the summation over all the selected zones. This label is visible when **Sum** is active.

Sum *2Pi

displays the result of the summation over all the selected zones multiplied by 2π. This label is visible when **Sum*2Pi** is active.

Max

displays the result of the maximum value of the selected zone(s). This label is visible when **Maximum** is active.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Min

displays the result of the minimum value of the selected zone(s). This label is visible when **Minimum** is active.

Total Volume Integral

displays the result of the volume-integral computation over all the selected zones. This label is visible when **Volume Integral** is active.

Volume-Weighted Average

displays the result of the volume-averaged computation over all the selected zones. This label is visible when **Volume-Average** is active.

Total Mass-Weighted Integral

displays the result of the mass-integral computation over all the selected zones. This label is visible when **Mass Integral** is active.

Mass-Weighted Average

displays the result of the mass-averaged computation over all the selected zones. This label is visible when **Mass-Average** is active.

Save Output Parameter...

opens the Save Output Parameter Dialog Box (p. 5345).

Compute

computes the specified integration on the selected zones.

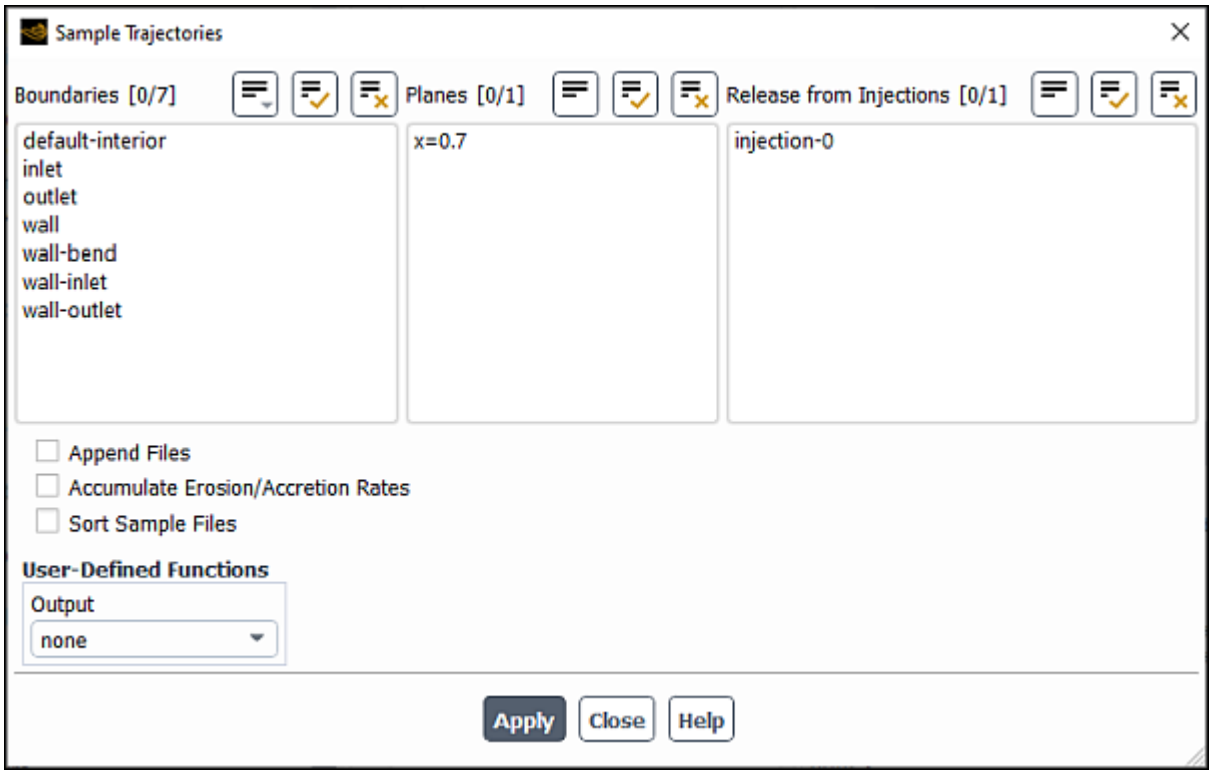
Write...

opens The Select File Dialog Box (p. 931), which you can use to save the reported values to a file.

51.20.6. Sample Trajectories Dialog Box

The **Sample Trajectories** dialog box allows the writing of particle states (position, velocity, diameter, temperature, and mass flow rate) at various boundaries and planes (lines in 2D). See Sampling of Trajectories (p. 2873) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Boundaries

lists boundaries that can be chosen as the surfaces at which samples will be written.

Lines

(in 2D) lists lines that can be chosen as the surfaces at which samples will be written.

Planes

(in 3D) lists planes that can be chosen as the surfaces at which samples will be written.

Release From Injections

lists injections from which the injection to be tracked is chosen.

Append Files

causes the results of multiple calculations to be appended to a single file.

Accumulate Erosion/Accretion Rates

causes erosion and accretion rates to be accumulated for repeated trajectory calculations.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sort Sample Files

causes the sample files to be sorted by injection and particle ID for steady tracking or simulated time (`flow-time` in the 13th column) for unsteady tracking. The latter is required if the sample file is to be used as an unsteady injection file.

User-Defined Functions

allow control of the format and the information written for the sample output.

Output

contains a drop-down list of available user-defined functions.

Compute

causes the particles to be tracked and their status to be written to files when they encounter selected surfaces. This button will not appear for unsteady particle tracking.

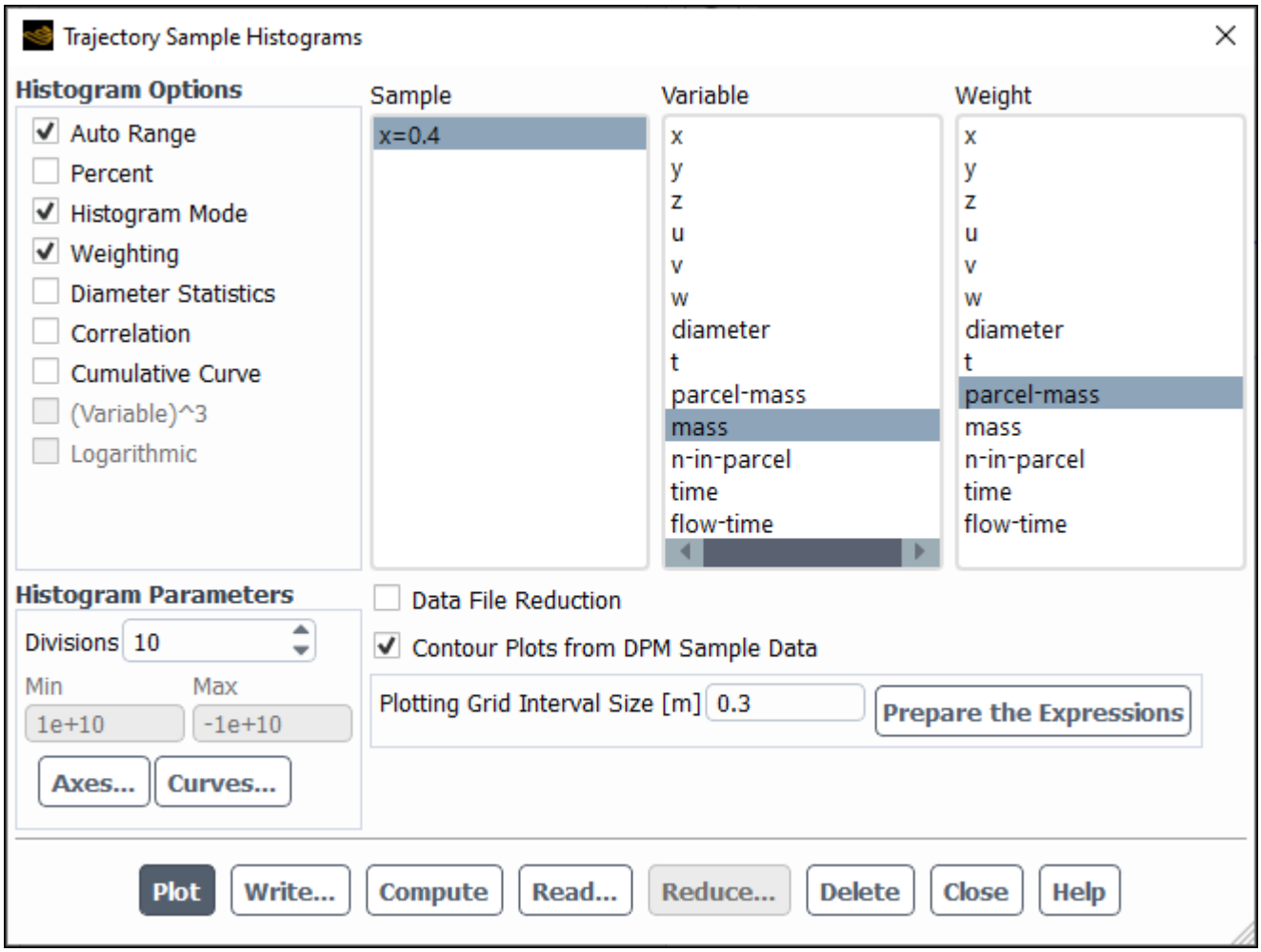
Start

initiates sampling for unsteady particle tracking. This button will replace the **Compute** button if you are performing unsteady particle tracking. After you click it, it will change to the **Stop** button. Click **Stop** to stop the sampling. (The label will change back to **Start**.)

51.20.7. Trajectory Sample Histograms Dialog Box

The **Trajectory Sample Histograms** dialog box allows the plotting of histograms from sample files created in the Sample Trajectories Dialog Box (p. 5331). See Histogram Reporting of Samples (p. 2875) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options

contains check buttons for histogram options.

Auto Range

toggles between automatic and manual settings of the histogram range.

Percent

causes the plot to indicate the percent of particles. Deselecting this will result in the actual number of particles being plotted.

Histogram Mode

allows you to display the histogram with or without bars.

Weighting

allows you to apply a weight to the data sample.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Diameter Statistics

allows you to display a summary of diameter statistics for a selected variable in the console window.

Correlation

allows you to choose correlations between how one particle variable depends on another particle variable.

Cumulative Curve

allows you to compute a cumulative distribution curve for a selected variable.

(Variable^3)

allows you to plot the cumulative mass distribution for a constant particle density using the particle diameter.

Histogram Parameters

contains controls for histogram plotting.

Divisions

sets the number of "bins" in the histogram.

Min

displays the minimum value of the variable selected in the **Variable** list. If **Auto Range** is off, you can set the minimum by typing a new value.

Max

displays the maximum value of the variable selected in the **Fields** list. If **Auto Range** is off, you can set the maximum by typing a new value.

Axes...

opens the Axes Dialog Box (p. 5313), which allows you to customize the histogram axes.

Curves...

opens the Curves Dialog Box (p. 5316), which allows you to customize the curves used in the histogram.

Sample

lists the data samples that have been read in.

Variable

lists the fields variables available in the selected sample.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Scale, No. of Bins

is an informational list that displays the scaling and the number of bins for each variable in the selected sample. This list is displayed only if **Data File Reduction** is selected.

Weight

lists the weighted fields variables available in the selected sample.

Correlation

lists the sampled variables, allowing you to choose the correlation variable.

Data File Reduction

enables the reduction of injection data in the selected sample and expands the dialog box to show inputs for **Data Reduction Settings For Selected Variables** and **Data Reduction Options**. For information about available data reduction settings, see Data Reduction of Samples (p. 2879).

Contour Plots from DPM Sample Data

allows you to generate named expressions from a sample file in order to use them for graphical postprocessing (contour plotting) on the surface where the DPM particles were sampled. For details, see Contour Plots of DPM Particle Sampling Results on a Planar Surface (p. 2880).

Plot

displays the histogram in the active graphics window.

Write...

stores the histograms in an XY-plot file format

Compute

updates the **Min** and **Max** values.

Read...

opens The Select File Dialog Box (p. 931), where you can select a sample file to be read.

Reduce...

reduces sample data in the selected sample. For more information, see Data Reduction of Samples (p. 2879). This button is available only if **Data File Reduction** is selected.

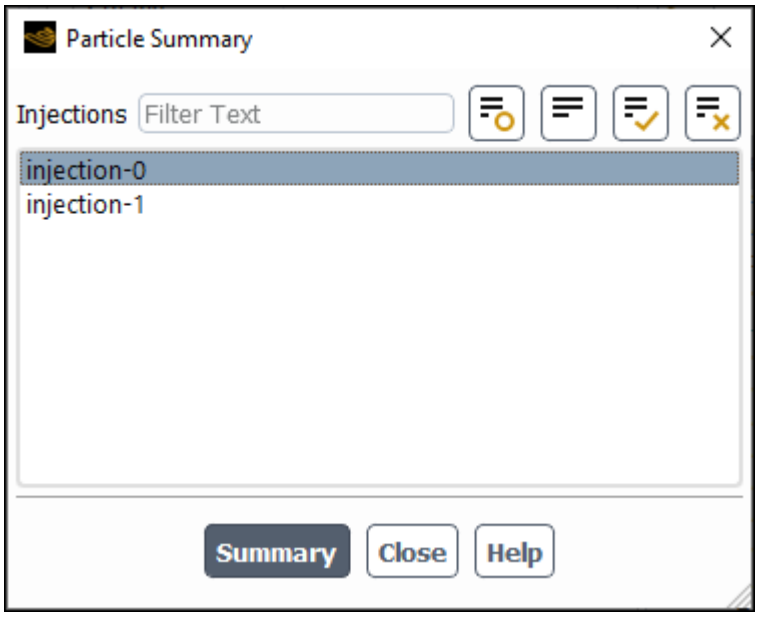
Delete

removes the sample selected in the **Sample** list.

51.20.8. Particle Summary Dialog Box

The **Particle Summary** dialog box allows you to report a summary for particle injections. See Summary Reporting of Current Particles (p. 2883) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Injections

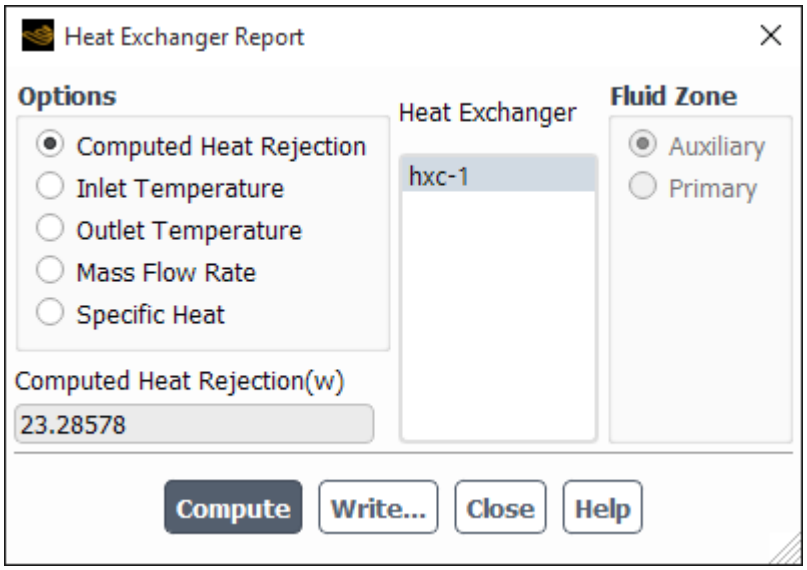
lists the particle injection(s) for which you can generate a summary.

Summary

prints the injection summary in the console window.

51.20.9. Heat Exchanger Report Dialog Box

The **Heat Exchanger Report** dialog box allows you to report a summary for heat exchangers. See Postprocessing for the Heat Exchanger Model (p. 2313) for details about the items below.



Controls

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Options

lists the available reporting options for the heat exchanger.

Computed Heat Rejection

allows you to report the heat rejection calculated from the heat exchanger.

Inlet Temperature

allows you to report the inlet temperature for both the primary and the auxiliary heat exchanger fluid.

Outlet Temperature

allows you to report the outlet temperature for both the primary and the auxiliary heat exchanger fluid.

Mass Flow Rate

allows you to report the mass flow rate for both the primary and the auxiliary heat exchanger fluid.

Specific Heat

allows you to report the specific heat for both the primary and the auxiliary heat exchanger fluid.

Result

displays the results of the calculations once the **Compute** button is selected.

Heat Exchanger

displays a list of heat exchanger fluid zones

Fluid Zone

(not available when **Computed Heat Rejection** option is selected.) allows you to report quantities for either the primary or auxiliary fluid zones.

Compute

computes the specified quantity on the selected zones.

Write...

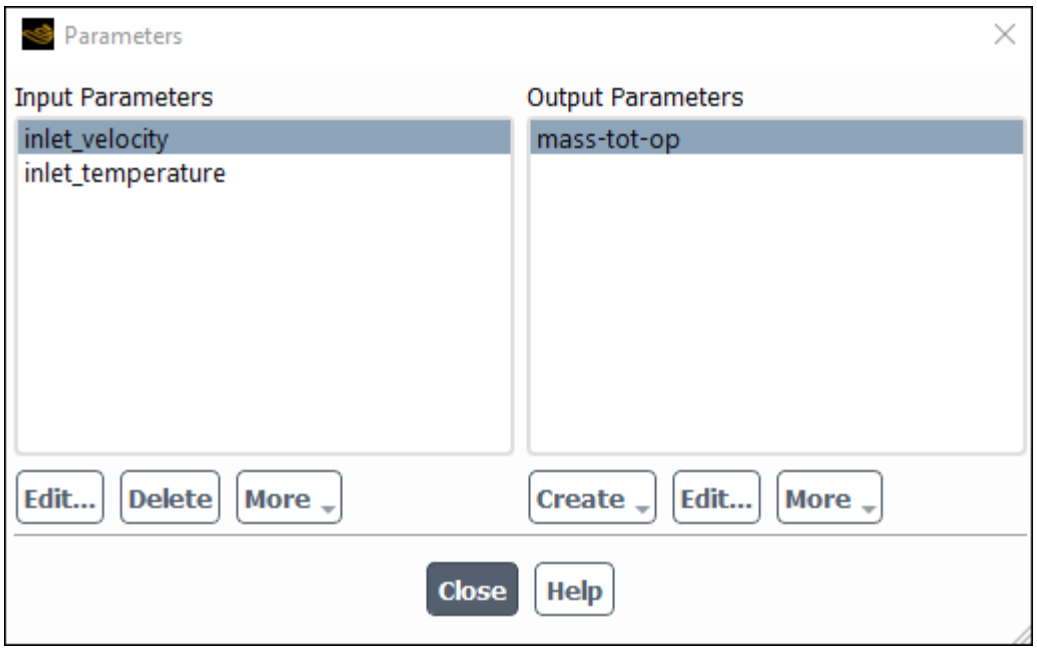
opens The Select File Dialog Box (p. 931), which you can use to save the reported values to a file.

51.20.10. Parameters Dialog Box

The **Parameters** dialog box allows you to create input parameters (in Scheme or user-defined function procedures only) and output parameters, which allow you to compare reporting values for different

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

cases, or include reporting values in the function minimized by the mesh morpher/optimizer. See Creating Output Parameters (p. 4133) for details about the items below.



Controls

Input Parameters

contains a list of existing input parameters. See Defining and Viewing Parameters (p. 1309) for details about creating input parameters.

Edit...

opens the Parameter Expression Dialog Box (p. 5343).

Important:

When using Ansys Fluent in Ansys Workbench, parameters are not editable, so the **Edit...** button becomes the **View...** button. This opens the Parameter Expression Dialog Box (p. 5343) where the parameter properties can only be viewed. For more information, see the separate Ansys Fluent in **Workbench** User's Guide.

Delete

removes the selected input parameter from the list of **Input Parameters**.

More

contains a drop-down list that allows more advanced use of input parameters. Available options include:

Use in Scheme Procedure

opens the Use Input Parameter in Scheme Procedure Dialog Box (p. 5341).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Use in UDF

opens the Use Input Parameter for UDF Dialog Box (p. 5342).

Output Parameters

contains a list of existing output parameters.

Create

opens the Report Definitions Dialog Box (p. 5594), allowing you to create additional output parameters based on report definitions.

Edit...

opens the dialog box corresponding to the selected item in the **Output Parameters** list.

More

contains a drop-down list containing additional tasks that you can perform, including:

Delete

removes the selected output parameter from the list of **Output Parameters**.

Rename

allows you to rename the selected output parameter.

Print to Console

reports values to the console window. If you select multiple output parameters, then the output includes values from multiple output parameters.

Print All to Console

outputs the values from all output parameters to the console window.

Write...

allows you to store the output to a file. A dialog box is displayed allowing you to provide a file name.

Write All...

prompts you for a file name and then writes the values for all of the output parameters to a file.

Note:

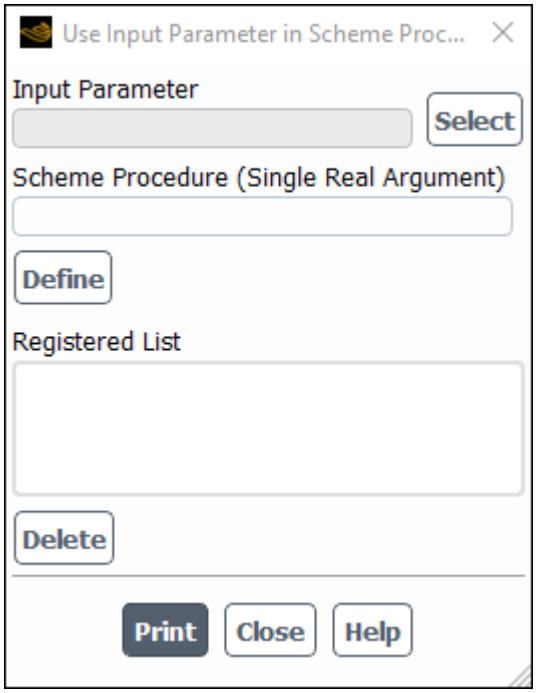
Ansys Fluent automatically creates generic default names for new input and output parameters (for example, `parameter-1`, `parameter-2`, `parameter-3`, and so on) If a parameter is deleted, the default name is not reused. For example, if you have `parameter-`

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

1, `parameter-2`, and `parameter-3`, then delete `parameter-2` and create a new parameter, the default name for the new parameter will be `parameter-4`.

51.20.11. Use Input Parameter in Scheme Procedure Dialog Box

The **Use Input Parameter in Scheme Procedure** dialog box allows you to use the input parameter in a Scheme procedure. In turn, this Scheme procedure can also use text user interface (TUI) commands. For more information, see Working With Advanced Parameter Options (p. 1313).



Controls

Input Parameter

contains the name of an input parameter.

Select

opens the Select Input Parameter Dialog Box (p. 4980) where an input parameter can be chosen.

Scheme Procedure (Single Real Argument)

contains a procedure name (if already defined using a Scheme file) or procedure body written in the Scheme language using `lambda`. This procedure should receive one real argument.

Define

registers the selected input parameter.

Registered List

contains a list of registered input parameters.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Delete

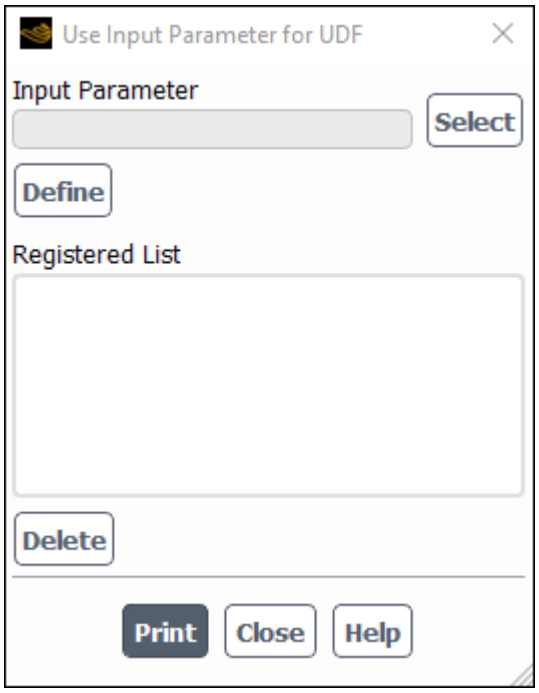
removes a selected registered input parameter from the list.

Print

prints details of the registered input parameters.

51.20.12. Use Input Parameter for UDF Dialog Box

The **Use Input Parameter for UDF** dialog box allows you to select the input parameters that can be used inside a user-defined function (UDF) during calculations.



Controls

Input Parameter

contains the name of an input parameter.

Select

opens the Select Input Parameter Dialog Box (p. 4980) where an input parameter can be chosen.

Define

registers the selected input parameter.

Registered List

contains a list of registered input parameters.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Delete

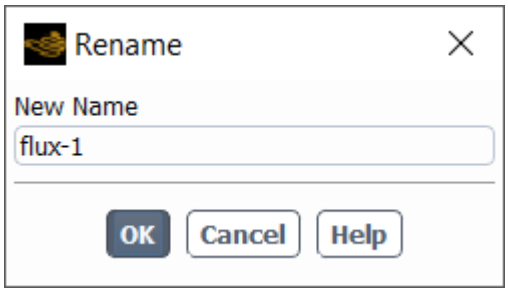
removes a selected registered input parameter from the list.

Print

prints the ID of the registered input parameters. The ID values are used to access the value of the parameter in the user-defined function.

51.20.13. Rename Dialog Box

The **Rename** dialog box allows you to change the name of the output parameter that you created.



Controls

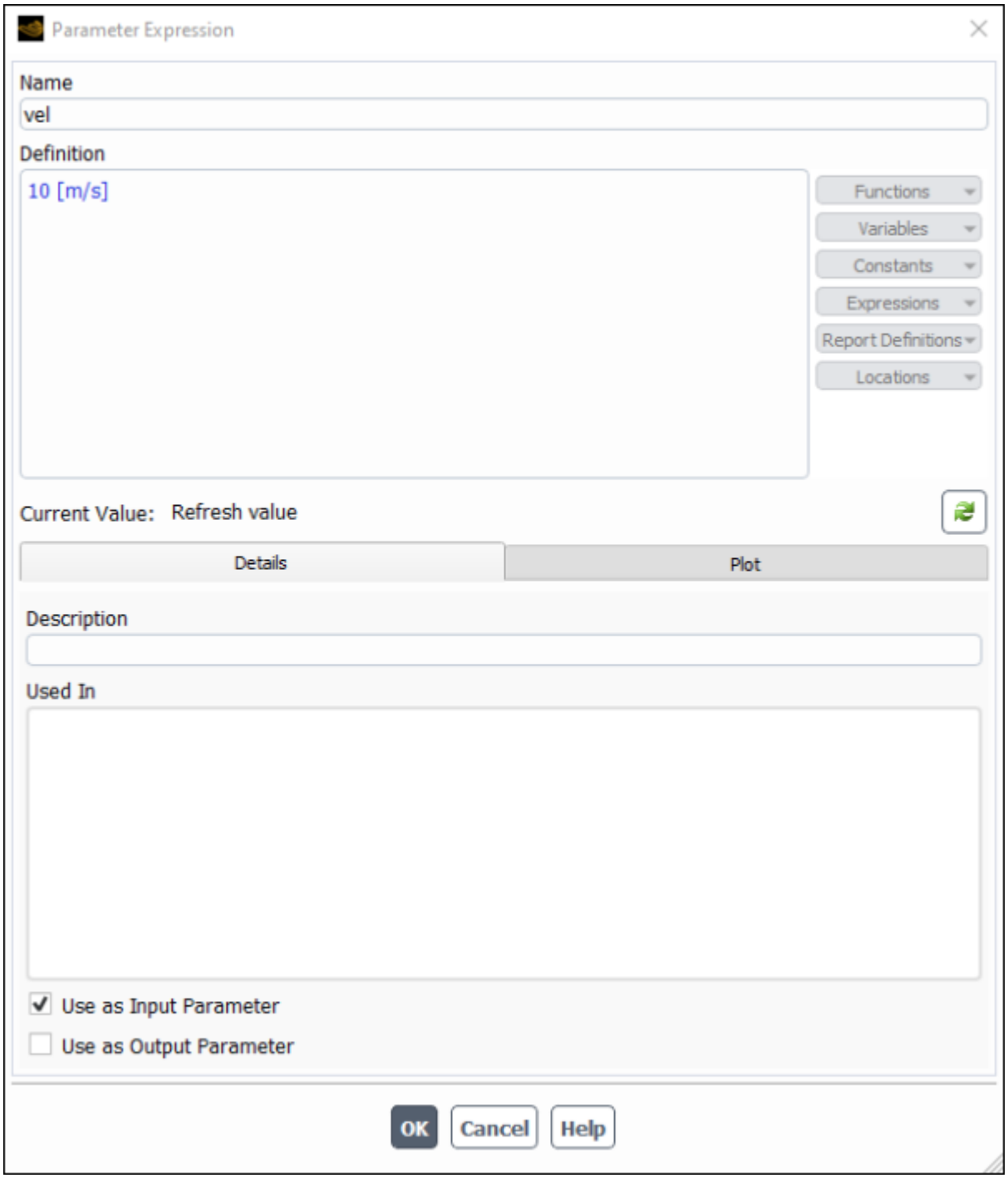
New Name

contains the new name of the output parameter.

51.20.14. Parameter Expression Dialog Box

The **Parameter Expression** dialog box allows you to create input parameters, which can be used to run a case and vary the value of the input to explore how the results change.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name

contains the name of the input parameter expression.

Definition

contains the current value of the input parameter.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Used In

indicates the cell zone or boundary condition where the input parameter is currently used. The relevant zone or boundary condition will be displayed after you create an input parameter.

Important:

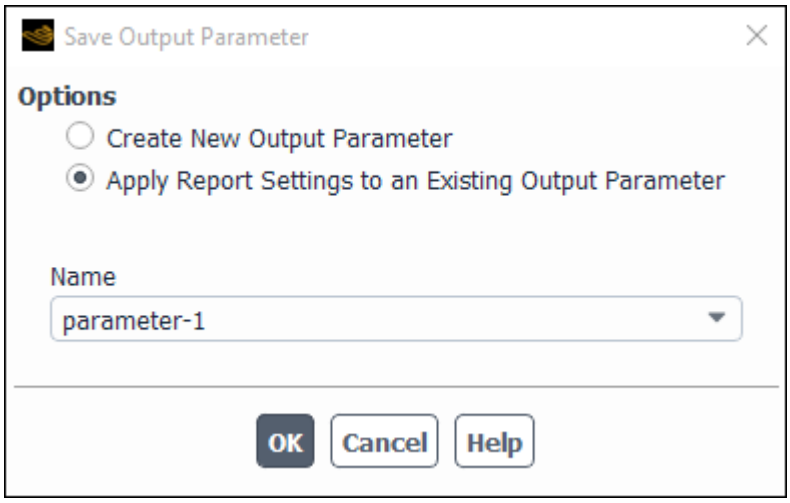
Input parameter expressions must be directly defined as a constant value including appropriate units; it may not reference another expression.

Note:

Ansys Fluent automatically creates generic default names for new input and output parameters (for example, `parameter_1`, `parameter_2`, `parameter_3`, and so on) If a parameter is deleted, the default name is not reused. For example, if you have `parameter_1`, `parameter_2`, and `parameter_3`, then delete `parameter_2` and create a new parameter, the default name for the new parameter will be `parameter_4`.

51.20.15. Save Output Parameter Dialog Box

The **Save Output Parameter** dialog box allows you to save specific output parameters that allow you to compare reporting values for different cases. See Creating Output Parameters (p. 4133) for details about the items below.



Controls

Options

contains options for saving your output parameters.

Create New Output Parameter

allows you to create a new output parameter.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Apply Report Settings to an Existing Output Parameter

allows you to overwrite an existing output parameter of the same type.

Name

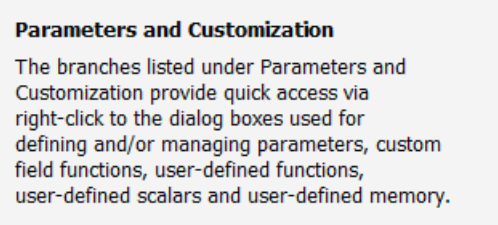
contains the name for the current output parameter.

Note:

Ansys Fluent automatically creates generic default names for new input and output parameters (for example, `parameter-1`, `parameter-2`, `parameter-3`, and so on) If a parameter is deleted, the default name is not reused. For example, if you have `parameter-1`, `parameter-2`, and `parameter-3`, then delete `parameter-2` and create a new parameter, the default name for the new parameter will be `parameter-4`.

51.21. Parameters and Customization Task Page

The **Parameters and Customization** task page introduces you to the parameterization and customization tools available for use in solving your CFD simulation using Ansys Fluent.



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Chapter 52: Ribbon Reference Guide

This reference guide provides information about the ribbon in Ansys Fluent.

52.1. File Ribbon Tab
52.2. Dialog Boxes Available from the Ribbon

52.1. File Ribbon Tab

For additional information, see the following sections:

52.1.1. File/Read/Mesh...
52.1.2. File/Read/Case...
52.1.3. File/Read/Data...
52.1.4. File/Read/Case & Data...
52.1.5. File/Read/Case Settings Only...
52.1.6. File/Read/Case/Mesh Info...
52.1.7. File/Read/PDF...
52.1.8. File/Read/ISAT Table...
52.1.9. File/Read/DTRM Rays...
52.1.10. File/Read/View Factors...
52.1.11. File/Read/Profile...
52.1.12. File/Read/Scheme...
52.1.13. File/Read/Journal...
52.1.14. File/Write/Case...
52.1.15. File/Write/Data...
52.1.16. File/Write/Case & Data...
52.1.17. File/Write/PDF...
52.1.18. File/Write/ISAT Table...
52.1.19. File/Write/Flamelet...
52.1.20. File/Write/Profile...
52.1.21. File/Write/Autosave...
52.1.22. File/Write/Boundary Mesh...
52.1.23. File/Write/Start Journal...
52.1.24. File/Write/Stop Journal
52.1.25. File/Write/Start Transcript...
52.1.26. File/Write/Stop Transcript

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

52.1.27. File/Import/ABAQUS/Input File...
52.1.28. File/Import/ABAQUS/Filbin File...
52.1.29. File/Import/ABAQUS/ODB File...
52.1.30. File/Import/CFX/Definition File...
52.1.31. File/Import/CFX/Result File...
52.1.32. File/Import/CGNS/Mesh...
52.1.33. File/Import/CGNS/Data...
52.1.34. File/Import/CGNS/Mesh & Data...
52.1.35. File/Import/EnSight...
52.1.36. File/Import/FIDAP...
52.1.37. File/Import/GAMBIT...
52.1.38. File/Import/HYPERMESH ASCII...
52.1.39. File/Import/LSTC/Input File...
52.1.40. File/Import/LSTC/State File...
52.1.41. File/Import/Marc POST...
52.1.42. File/Import/NASTRAN/Bulkdata File...
52.1.43. File/Import/NASTRAN/Op2 File...
52.1.44. File/Import/PLOT3D/Grid File...
52.1.45. File/Import/PLOT3D/Result File...
52.1.46. File/Import/Tecplot...
52.1.47. File/Import/Partition/Metis...
52.1.48. File/Import/Partition/Metis Zone...
52.1.49. File/Import/CHEMKIN Mechanism...
52.1.50. File/Import/FMU...
52.1.51. File/Export/Solution Data...
52.1.52. File/Export/Particle History Data...
52.1.53. File/Export/During Calculation/Solution Data...
52.1.54. File/Export/During Calculation/Particle History Data...
52.1.55. File/Export to CFD-Post...
52.1.56. File/Table File Manager...
52.1.57. File/Solution Files...
52.1.58. File/Interpolate...
52.1.59. File/FSI Mapping/Volume...
52.1.60. File/FSI Mapping/Surface...
52.1.61. File/Save Picture...
52.1.62. File/Data File Quantities...
52.1.63. File/Batch Options...
52.1.64. File/Idle Timeout...
52.1.65. File/Exit

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

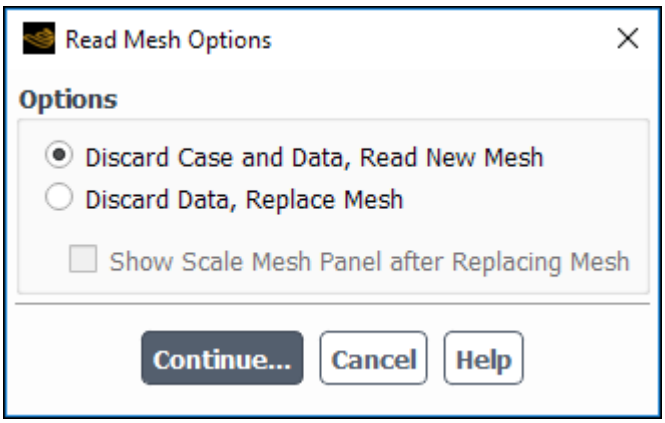
52.1.1. File/Read/Mesh...

The **File/Read/Mesh...** ribbon tab item opens the Read Mesh Options Dialog Box (p. 5349), which allows you to read or replace a mesh.

52.1.1.1. Read Mesh Options Dialog Box

The **Read Mesh Options** dialog box is used to read or replace a mesh. See Reading Mesh Files (p. 953) for more information.

File → Read → Mesh...



Controls

Options

allows you to read in a new mesh or replace the existing mesh.

Discard Case And Data, Read New Mesh

results in both the case and data files being discarded when reading in a new mesh.

Discard Data, Replace Mesh

results in the data file being discarded when replacing an existing mesh.

Show Scale Mesh Panel After Replacing Mesh

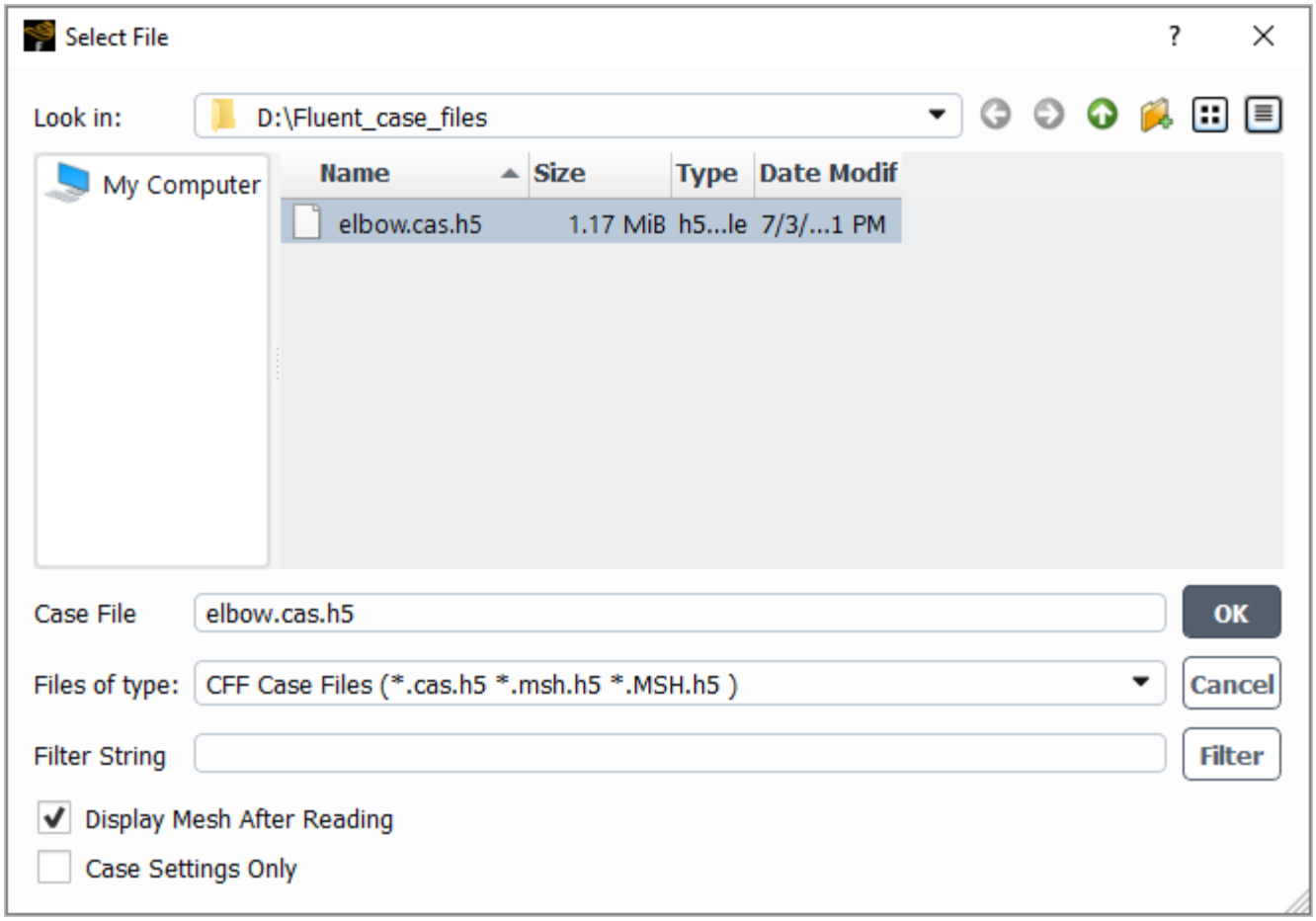
gives you the option to have the Scale Mesh Dialog Box (p. 4686) appear automatically for you to check or scale your mesh.

52.1.2. File/Read/Case...

The **File/Read/Case...** ribbon tab item is used to read an Ansys Fluent case file (extension `.cas.h5` or `.cas`), a mesh file (extension `.msh.h5`, `.msh`, `.grd`, `.MSH`, or `.GRD`), or the case settings only for a file (extension `.cas.h5` or `.msh.h5`) without any of the data associated with the mesh. For details, see Reading and Writing Case Files (p. 956), Reading Mesh Files (p. 953), and Reading Settings Only for Mesh or Case Files with Large Cell Counts (p. 962), respectively.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **File/Read/Case...** ribbon tab item opens The Select File Dialog Box (p. 931), which allows you to select the appropriate file to be read.



52.1.3. File/Read/Data...

The **File/Read/Data...** ribbon tab item is used to read in an Ansys Fluent data file (which has a `.dat.h5` or `.dat` extension). This item will not be available until you read in a case or mesh file. See Reading and Writing Data Files (p. 957) for details.

The **File/Read/Data...** ribbon tab item opens The Select File Dialog Box (p. 931), which allows you to select the appropriate file to be read.

52.1.4. File/Read/Case & Data...

The **File/Read/Case & Data...** ribbon tab item is used to read in an Ansys Fluent case file and the corresponding data file (for example, `myfile.cas.h5` and `myfile.dat.h5`) together. See Reading and Writing Case and Data Files Together (p. 957) for details.

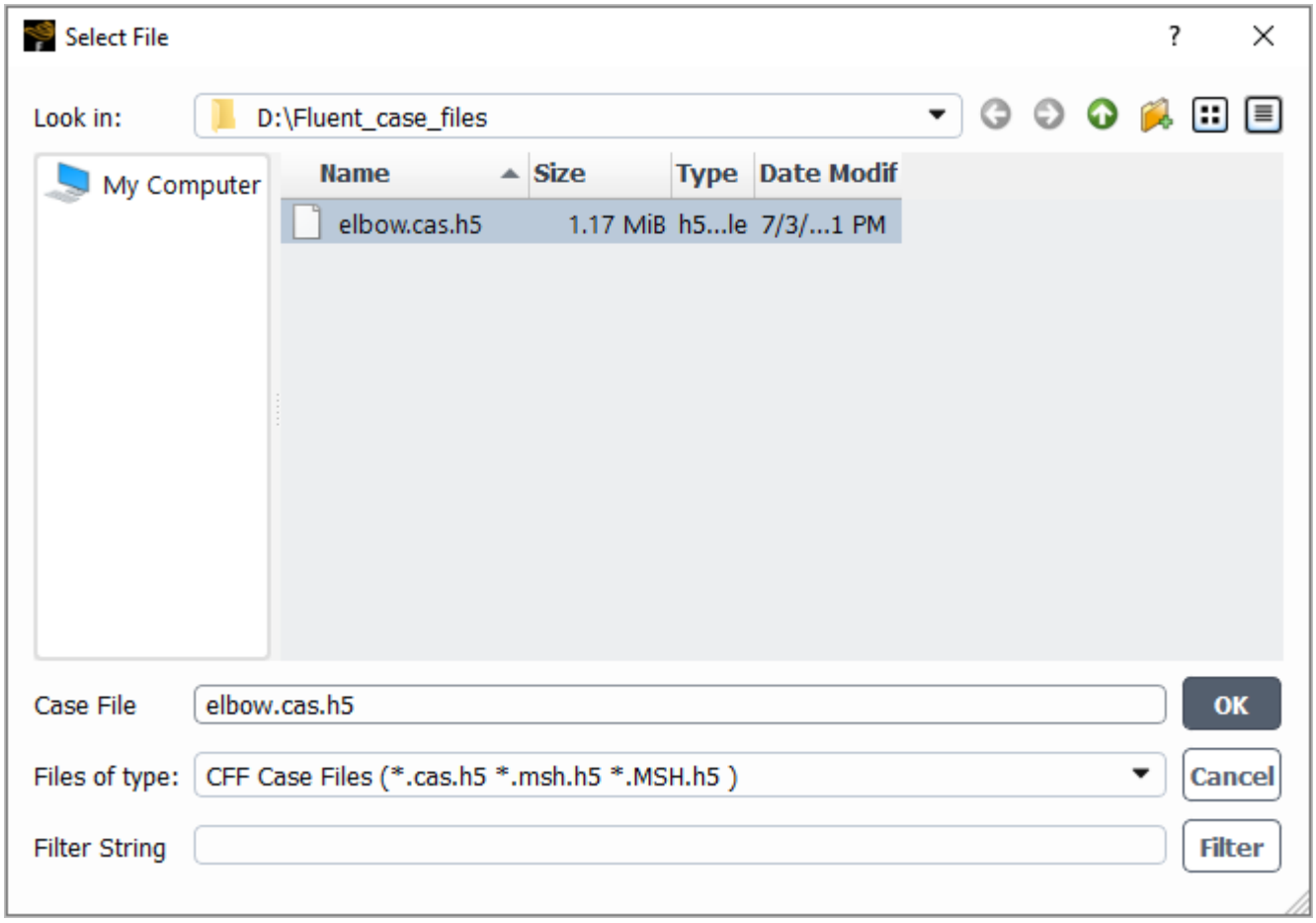
The **File/Read/Case & Data...** ribbon tab item opens The Select File Dialog Box (p. 931), which allows you to select the appropriate files to be read. Select the appropriate case file, and the corresponding data file (that is, the file having the same name with a `.dat.h5` extension) will also be read in.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

52.1.5. File/Read/Case Settings Only...

The **File/Read/Case Settings Only...** ribbon tab item is used to read the case settings only for a file (extension `.cas.h5` or `.msh.h5`), without any of the data associated with the mesh. For details, see Reading Settings Only for Mesh or Case Files with Large Cell Counts (p. 962).

The **File/Read/Case Settings Only...** ribbon tab item opens The Select File Dialog Box (p. 931), which allows you to select the appropriate file for which you want to read the case settings.

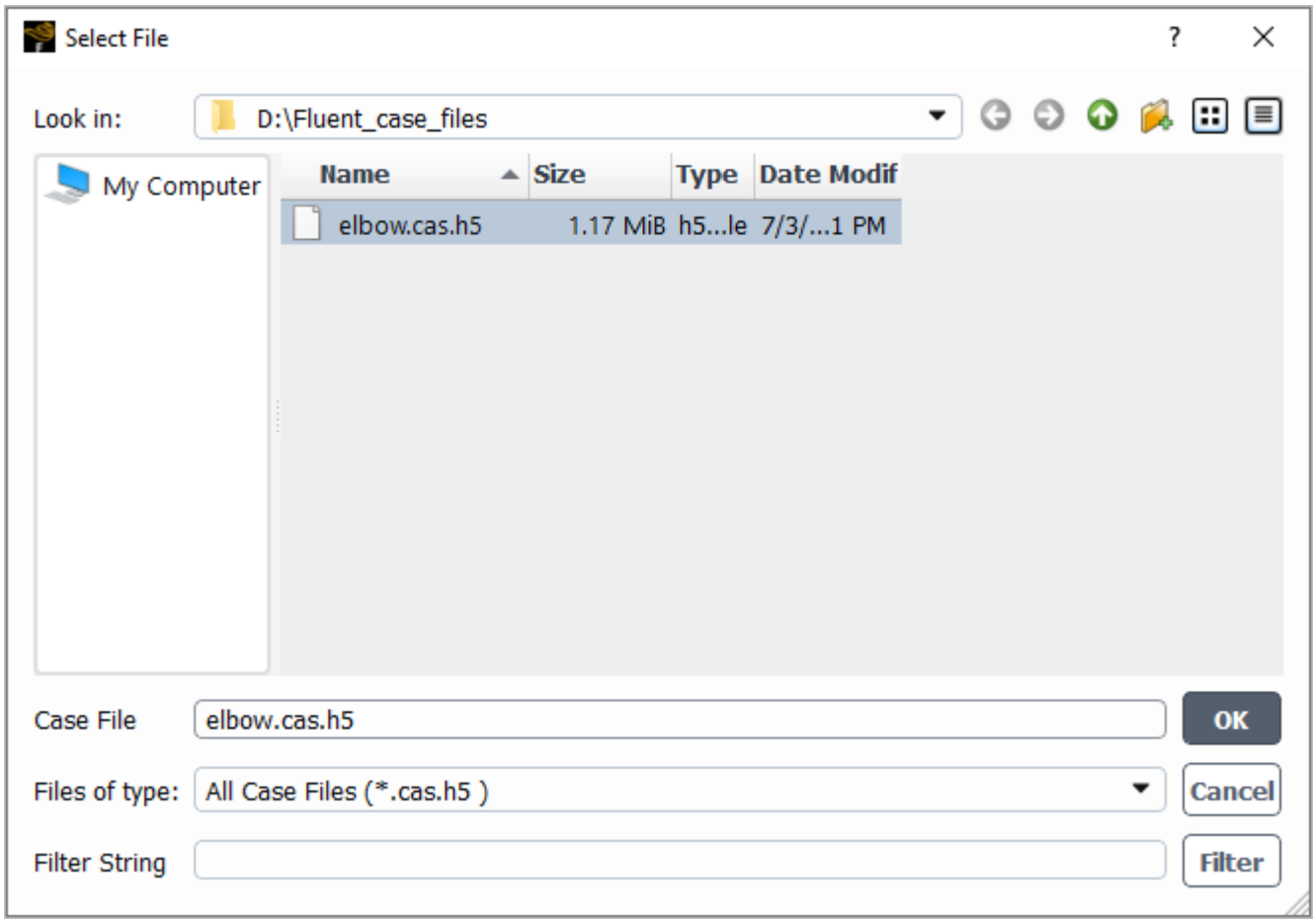


52.1.6. File/Read/Case/Mesh Info...

The **File/Read/Case/Mesh Info...** ribbon tab item is used to read the information about a case or mesh file and print it in the console. The available summary includes the Fluent version that last saved the file, precision and dimension, and the total number of cells, faces, and nodes.

The **File/Read/Case/Mesh Info...** ribbon tab item opens The Select File Dialog Box (p. 931), which allows you to select the appropriate case or mesh file for which you want to read information.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



52.1.7. File/Read/PDF...

The **File/Read/PDF...** ribbon tab item is used to read a PDF file (extension `.pdf`) created by Ansys Fluent for use with the non-premixed or partially premixed combustion model. This item is available only when the non-premixed or partially premixed combustion model has been enabled. See Saving the Look-Up Tables (p. 2514) for details.

52.1.8. File/Read/ISAT Table...

The **File/Read/ISAT Table...** ribbon tab item is used to read an ISAT table (extension `.isat`) for use with the Composition PDF Transport model. This item is available only when the Composition PDF Transport model has been enabled. See Using ISAT Efficiently (p. 2459) for details.

52.1.9. File/Read/DTRM Rays...

The **File/Read/DTRM Rays...** ribbon tab item is used to read a ray file (extension `.ray`) created by Ansys Fluent for use with the DTRM (radiation model). This item is available only when the DTRM has been enabled. See Writing and Reading the DTRM Ray File (p. 2198) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

52.1.10. File/Read/View Factors...

The **File/Read/View Factors...** ribbon tab item is used to read in a view factor file for use with the surface-to-surface (S2S) radiation model. This item is available only when the S2S model has been enabled. See Reading View Factors into Ansys Fluent (p. 2211) for details.

52.1.11. File/Read/Profile...

The **File/Read/Profile...** ribbon tab item opens the **Select File** dialog box for reading profiles (Reading Profile Files (p. 964)), It is used to read cell zone or boundary condition profile files (extensions `.prof`, `.ttab`, `.csv`). A profile file defines profiles that can be used to specify flow conditions for a cell zone or a boundary. See Profiles (p. 1564) for details.

52.1.12. File/Read/Scheme...

The **File/Read/Scheme...** ribbon tab item is used to read in a Scheme source file (extension `.scm`). See Reading Scheme Source Files (p. 972) for details.

52.1.13. File/Read/Journal...

The **File/Read/Journal...** ribbon tab item is used to read in a journal file (extension `.jou`) containing a sequence of Ansys Fluent commands. You can create a journal file using the **File/Write/Journal...** ribbon tab item. See Creating and Reading Journal Files (p. 972) for details.

52.1.14. File/Write/Case...

The **File/Write/Case...** ribbon tab item is used to save an Ansys Fluent case file. See Reading and Writing Case Files (p. 956) for details.

The **File/Write/Case...** ribbon tab item opens The Select File Dialog Box (p. 931), which allows you to save the file with a name of choice. The dialog box is similar to the **Select File** dialog box for reading files, except that it has an additional option for writing binary files.

Important:

- When Ansys Fluent writes a case file, the `.cas.h5` extension is added to the file name specified, unless the name already ends with `.cas.h5`.
- You can also write a compressed file by appending `.gz` or `.Z` to the file name, if you have requested the reading and writing of files in the legacy format. (For details about file compression and the legacy format, see Reading and Writing Compressed Files (p. 950) and Reading and Writing Files in the Legacy Format (p. 958), respectively.)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

52.1.15. File/Write/Data...

The **File/Write/Data...** ribbon tab item is used to save an Ansys Fluent data file (which has a `.dat.h5` or `.dat` extension). See Reading and Writing Data Files (p. 957) for details.

Important:

- When Ansys Fluent writes a data file, the `.dat.h5` extension is added to the file name specified, unless the name already ends with `.dat.h5`.
- You can compress data files by appending `.gz` or `.Z` to the file name, if you have requested the reading and writing of files in the legacy format. (For details about file compression and the legacy format, see Reading and Writing Compressed Files (p. 950) and Reading and Writing Files in the Legacy Format (p. 958), respectively.)

52.1.16. File/Write/Case & Data...

The **File/Write/Case & Data...** ribbon tab item is used to save an Ansys Fluent case file and data file at the same time (for example, `myfile.cas.h5` and `myfile.dat.h5`). See Reading and Writing Case and Data Files Together (p. 957) for details.

Enter the name of the case file in the text entry box in the **Select File** dialog box and the corresponding data file (same file name, but with a `.dat.h5` extension) will also be written.

52.1.17. File/Write/PDF...

The **File/Write/PDF...** ribbon tab item is used to write a PDF file after computing the look-up tables using the non-premixed or partially premixed combustion model in Ansys Fluent. This item is available only when the non-premixed or partially premixed combustion model has been enabled. See Saving the Look-Up Tables (p. 2514) for details.

52.1.18. File/Write/ISAT Table...

The **File/Write/ISAT Table...** ribbon tab item is used to write an ISAT table. See Using ISAT Efficiently (p. 2459) for details.

52.1.19. File/Write/Flamelet...

The **File/Write/Flamelet...** ribbon tab item is used to write a flamelet file generated using the non-premixed or partially premixed combustion model in Ansys Fluent. This item is available only when the non-premixed or partially premixed combustion model has been enabled. See The Diffusion Flamelet Models Theory in the Theory Guide for details.

52.1.20. File/Write/Profile...

The **File/Write/Profile...** ribbon tab item opens the Write Profile Dialog Box (p. 4988).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

52.1.21. File/Write/Autosave...

The **File/Write/Autosave...** ribbon tab item opens the Autosave Dialog Box (p. 5190).

52.1.22. File/Write/Boundary Mesh...

The **File/Write/Boundary Mesh...** ribbon tab item is used to write the boundary zones (surface mesh) of the domain to a file. You can then read this file into the meshing mode of Fluent or GAMBIT to produce an improved volume mesh. See Writing a Boundary Mesh (p. 972) for details.

52.1.23. File/Write/Start Journal...

The **File/Write/Start Journal...** ribbon tab item is used to start the recording of subsequent Ansys Fluent commands to a journal file. You can read this journal file back into Fluent later (using the **File/Read/Journal...** ribbon tab item) to automate the execution of the recorded commands. See Creating and Reading Journal Files (p. 972) for details.

52.1.24. File/Write/Stop Journal

The **File/Write/Stop Journal** ribbon tab item replaces the **File/Write/Start Journal...** ribbon tab item after the recording of a journal file has begun. The **File/Write/Stop Journal** ribbon tab item is used to end the journal recording. See Creating and Reading Journal Files (p. 972) for details.

52.1.25. File/Write/Start Transcript...

The **File/Write/Start Transcript...** ribbon tab item is used to start the recording of a transcript file containing all input to and output from Fluent. (You cannot read a transcript file back into Fluent.) See Creating Transcript Files (p. 976) for details.

52.1.26. File/Write/Stop Transcript

The **File/Write/Stop Transcript** ribbon tab item replaces the **File/Write/Start Transcript...** ribbon tab item after the recording of a transcript file has begun. The **File/Write/Stop Transcript** ribbon tab item is used to end the transcript recording. See Creating Transcript Files (p. 976) for details.

52.1.27. File/Import/ABAQUS/Input File...

The **File/Import/ABAQUS/Input File...** ribbon tab item is used to import an ABAQUS input file (extension `.inp`), which contains the input description of a finite element model for the ABAQUS finite element program. See ABAQUS Files (p. 979) for details.

52.1.28. File/Import/ABAQUS/Filbin File...

The **File/Import/ABAQUS/Filbin File...** ribbon tab item is used to import an ABAQUS filbin file (extension `.fil`), which contains the finite element model and results data. See ABAQUS Files (p. 979) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

52.1.29. File/Import/ABAQUS/ODB File...

The **File/Import/ABAQUS/ODB File...** ribbon tab item is used to import an ABAQUS ODB file with a `.odb` extension. See ABAQUS Files (p. 979) for details.

52.1.30. File/Import/CFX/Definition File...

The **File/Import/CFX/Definition File...** ribbon tab item is used to import a CFX definition file (extension `.def`), which contains mesh information to be read into Fluent. See CFX Files (p. 979) for details.

52.1.31. File/Import/CFX/Result File...

The **File/Import/CFX/Result File...** ribbon tab item is used to import a CFX result file (extension `.res`). See CFX Files (p. 979) for details.

52.1.32. File/Import/CGNS/Mesh...

The **File/Import/CGNS/Mesh...** ribbon tab item is used to read in a CGNS-format mesh file (extension `.cgns`). See Meshes and Data in CGNS Format (p. 980) for details.

52.1.33. File/Import/CGNS/Data...

The **File/Import/CGNS/Data...** ribbon tab item is used to read in a CGNS-format data file. See Meshes and Data in CGNS Format (p. 980) for details.

52.1.34. File/Import/CGNS/Mesh & Data...

The **File/Import/CGNS/Mesh & Data...** ribbon tab item is used to read in a set of CGNS-format mesh and data files. See Meshes and Data in CGNS Format (p. 980) for details.

52.1.35. File/Import/EnSight...

The **File/Import/EnSight...** ribbon tab item is used to import EnSight files (extension `.encas` or `.case`). See EnSight Files (p. 981) for details.

52.1.36. File/Import/FIDAP...

The **File/Import/FIDAP...** ribbon tab item is used to import an ANSYS FIDAP neutral file (extension `.FDNEUT` or `.unv`).

52.1.37. File/Import/GAMBIT...

The **File/Import/GAMBIT...** ribbon tab item is used to read in a neutral file from GAMBIT. See GAMBIT and GeoMesh Mesh Files (p. 981) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

52.1.38. File/Import/HYPERMESH ASCII...

The **File/Import/HYPERMESH ASCII...** ribbon tab item is used to import a **HYPERMESH ASCII** file (extension `.hm`, `.hma`, or `.hmascii`). See HYPERMESH ASCII Files (p. 982) for details.

52.1.39. File/Import/LSTC/Input File...

The **File/Import/LSTC/Input File...** ribbon tab item is used to import an LSTC input file (extension `.k`, `.key`, or `.dyn`), which contains the input description of a finite element model for the LS-DYNA finite element program.

52.1.40. File/Import/LSTC/State File...

The **File/Import/LSTC/State File...** ribbon tab item is used to import an LSTC state file (extension `.d3plot`), which contains control data, geometry data, and state data.

52.1.41. File/Import/Marc POST...

The **File/Import/Marc POST...** ribbon tab item is used to import a Marc POST file generated using the MSC Marc finite element program.

52.1.42. File/Import/NASTRAN/Bulkdata File...

The **File/Import/NASTRAN/Bulkdata File...** ribbon tab item is used to import a NASTRAN Bulkdata file (extension `.nas*`, `.dat*`, or `.bdf*`), which contains mesh information to be read into Fluent. See NASTRAN Files (p. 982) for details.

52.1.43. File/Import/NASTRAN/Op2 File...

The **File/Import/NASTRAN/Op2 File...** ribbon tab item is used to import a NASTRAN Op2 file (extension `.op2`), which is an output binary data file containing data used in the NASTRAN finite element program. See NASTRAN Files (p. 982) for details.

52.1.44. File/Import/PLOT3D/Grid File...

The **File/Import/PLOT3D/Grid File...** ribbon tab item is used to import a PLOT3D grid file (extension `.g`, `.x`, `.xyz`, or `.grd`). See PLOT3D Files (p. 982) for details.

52.1.45. File/Import/PLOT3D/Result File...

The **File/Import/PLOT3D/Result File...** ribbon tab item s used to import a PLOT3D result file (extension `.g`, `.x`, `.xyz`, or `.grd`). See PLOT3D Files (p. 982) for details.

52.1.46. File/Import/Tecplot...

The **File/Import/Tecplot...** ribbon tab item is used to read in a Tecplot binary file. See Tecplot Files (p. 982) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

52.1.47. File/Import/Partition/Metis...

The **File/Import/Partition/Metis...** ribbon tab item is used to partition a mesh and then read it into the parallel version of Ansys Fluent. See Using the Partition Filter (p. 4338) for an explanation of the difference between this item and the **File/Import/Partition/Metis Zone...** ribbon tab item.

52.1.48. File/Import/Partition/Metis Zone...

The **File/Import/Partition/Metis Zone...** ribbon tab item is used to partition each cell zone in a mesh individually and then read the mesh into the parallel version of Ansys Fluent. See Using the Partition Filter (p. 4338) for an explanation of the difference between this item and the **File/Import/Partition/Metis...** ribbon tab item.

52.1.49. File/Import/CHEMKIN Mechanism...

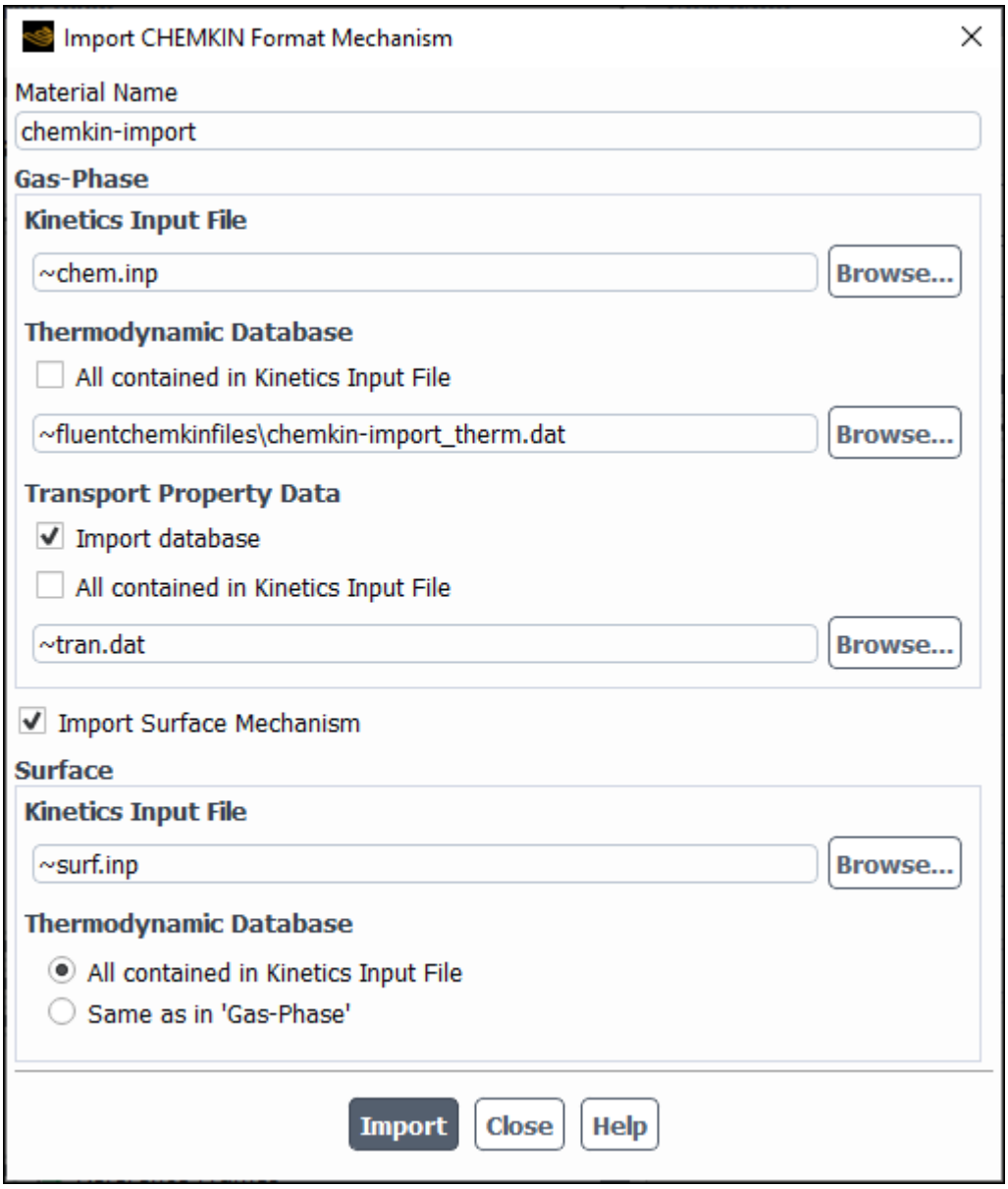
The **File/Import/CHEMKIN Mechanism...** ribbon tab item opens the Import CHEMKIN Format Mechanism Dialog Box (p. 5358).

52.1.49.1. Import CHEMKIN Format Mechanism Dialog Box

The **Import CHEMKIN Format Mechanism** dialog box is used to import a CHEMKIN-format chemical mechanism file into Ansys Fluent. See Importing a Volumetric Kinetic Mechanism in CHEMKIN Format (p. 2368) for details. The dialog box can be accessed either from the Species Model Dialog Box (p. 4768) or from the main menu:

File → Import → CHEMKIN Mechanism...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Material Name

specifies a name for the material.

Gas-Phase

contains inputs for importing the gas-phase mechanism in CHEMKIN format.

Kinetics Input File

specifies the name of the gas-phase file in CHEMKIN-format. If the file is not in the current working folder, include the full path to the folder where it is located.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Thermodynamic Database

specifies thermodynamic property data for species whose thermodynamic data are not specified in the kinetics input file. You can enable **All contained in Kinetics Input Files** to indicate that the kinetics input file contains the thermodynamic data for all the species.

Transport Property Data (Optional)

(if **Import database** is enabled) allows you to specify transport property data for species whose transport data are not specified in the kinetics input file. You can enable **All contained in Kinetics Input Files** to indicate that the kinetics input file contains the transport data for all the species.

Import Surface Mechanism

enables the importing of the surface mechanism.

Surface

contains inputs for importing the surface mechanism. This group box appears only when the **Import Surface Mechanism** option is enabled.

Kinetics Input File

specifies the name of the surface kinetics CHEMKIN-format file.

Thermodynamic Database

specifies whether the thermodynamic data for the surface-chemistry species will be included in the surface kinetics CHEMKIN-format file (if **All contained in Kinetics Input File** is enabled or in the thermodynamic database file specified in the **Gas-Phase** group box (if **Same as 'Gas-Phase'** is enabled). Note that if no thermodynamic database file is specified in the **Gas-Phase** group box, the thermodynamic data for all surface-chemistry species must be included the surface kinetics input file.

Import

imports the specified Chemkin reaction mechanism and thermal files and saves the imported data in the Fluent case file. Note that carrying the original input files along with the Fluent case file is not required. Ansys Fluent will automatically regenerate these files when needed (for example, when using the CHEMKIN-CFD solver).

52.1.50. File/Import/FMU...

The **File/Import/FMU...** ribbon tab item opens the Figure 3.22:**Import FMU File** Dialog Box (p. 1030).

52.1.51. File/Export/Solution Data...

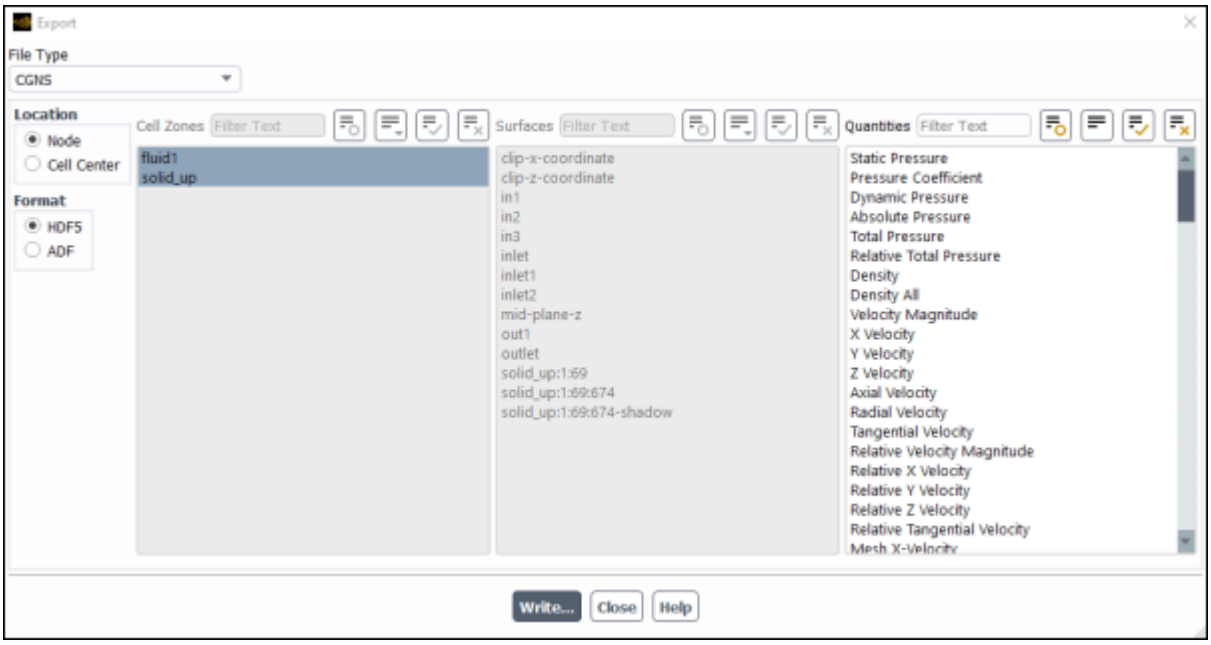
The **File/Export/Solution Data...** ribbon tab item opens the Export Dialog Box (p. 5361).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

52.1.51.1. Export Dialog Box

The **Export** dialog box allows you to write data that can be read by other data visualization and postprocessing tools. See Exporting Solution Data (p. 983) for a complete description of the available data formats and how to use the dialog box.

File → Export → Solution Data...



Controls

File Type

contains a drop-down list that controls the output file format that will be written using the **Write...** button.

ABAQUS

allows you to specify the surface(s) and optional loads, based on the kind of finite element analysis selected, to be exported to an ABAQUS file (extension `.inp`).

ASCII

allows you to specify the surface(s), scalars, location in the cell from which the values of scalar functions are to be taken, and the delimiter separating the fields, to be exported to an ASCII file.

AVS

allows you to specify the scalars you want to write to be exported to an AVS file.

CDAT for CFD-Post & EnSight

allows you to specify the cell zones, surfaces, quantities, and format (**Binary** or **ASCII**) for data export. The `.cdat` file format is compatible with both CFD-Post and EnSight.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

CGNS

allows you to specify the scalars you want to write and the location from which the values of scalar functions are to be taken, to be exported to a CGNS file (extension `.cgns`).

Common Fluids Format - Post

allows you to specify the cell zones, surfaces, and quantities that you want exported *(note that only nodal values are exported)*. The following files are written in the hierarchical data format (HDF) from the **Export** dialog box: `<filename>.cas.post` and `<filename>.dat.post`.

Data Explorer

allows you to specify the surface(s) and the scalars you want to write to be exported to a Data Explorer file (extension `.dx`).

EnSight Case Gold

allows you to specify the scalars you want to write, the cell zones, interior zone surfaces, and location in the cell from which the values of scalar functions are to be taken, and the file format, to be exported to an EnSight file (extension `.geo`, `.vel`, `.scl1`, or `.encas`).

EnSight DVS

allows you to specify the cell zones or surfaces (not both) and the quantities that you want to write, and whether the data is written for nodes or cell centers. Data is exported in `.dvs` format for import to EnSight.

FAST

allows you to specify the scalars you want to write, to be exported as a grid file (Plot3D format), a velocity file, and a scalar file. This option is available only for a triangular or tetrahedral mesh.

FAST Solution

allows you to export a single file containing density, velocity, and total energy data. This option is available only for a triangular or tetrahedral mesh.

Fieldview Unstructured

allows you to specify the scalars you want to write and the cell zones or surfaces from which the values of scalar functions are to be taken, to be exported to a FIELDVIEW binary file (extension `.fvuns`) and a regions file (extension `.fvuns.fvreg`).

Note:

You must deselect all cell zones before you can select any surfaces.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

I-deas Universal

allows you to specify the surface(s), scalars, and optional loads, based on the kind of finite element analysis selected, to be exported to an I-deas Universal file.

Mechanical APDL Input

allows you to specify the surface(s) and optional loads, based on the kind of finite element analysis selected, to be exported to a Mechanical APDL Input file (extension `.cdb`).

NASTRAN

allows you to specify the surface(s), scalars, and optional loads, based on the kind of finite element analysis selected, to be exported to a NASTRAN file (extension `.bdf`).

PATRAN

allows you to specify the surface(s), scalars, and optional loads, based on the kind of finite element analysis selected, to be exported to a PATRAN neutral file (extension `.out`).

TAITherm

allows you to specify the surface(s) for which you want to write data and the method of writing the heat transfer coefficient, to be exported to a PATRAN neutral file (extension `.neu`). This option is available only when the **Energy Equation** is enabled.

Tecplot

allows you to specify the surface(s) and the scalars you want to write, to be exported to a Tecplot file.

Cell Zones

specifies the cell zones for which data is to be written for a CFD-Post compatible, EnSight, or FIELDVIEW file.

Surfaces

specifies the surfaces for which data is to be written for an ABAQUS, ASCII, Data Explorer, I-deas Universal, Mechanical APDL Input, NASTRAN, PATRAN, TAITherm, or Tecplot file.

Quantities

specifies valid functions for output. The attributes of the list are modified based on the active file type. The list may be a single-selection or a multiple-selection list or it may be disabled, depending on the selected **File Type**.

Structural Loads

contains optional structural loads that can be written to ABAQUS, I-deas Universal, Mechanical APDL Input, NASTRAN, and PATRAN files. This option is available only when **Structural** analysis is selected.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Force

enables force to be written as a load for a structural analysis.

Pressure

enables pressure to be written as a load for a structural analysis.

Temperature

enables temperature to be written as a load for a structural analysis. This option is available only when the **Energy Equation** is enabled.

Thermal Loads

contains optional thermal loads that can be written to ABAQUS, I-deas Universal, Mechanical APDL Input, NASTRAN, and PATRAN files. This option is available only when **Thermal** analysis is selected.

Temperature

enables temperature to be written as a load for a thermal analysis.

Heat Flux

enables heat flux to be written as a load for a thermal analysis.

Heat Trans Coeff

enables heat transfer coefficient to be written as a load for a thermal analysis.

Location (for ASCII, CGNS, and EnSight Case Gold formats)

specifies the location in the cell from which the values of scalar functions are to be taken.

Node

specifies that data values at the node points are to be exported.

Cell Center

specifies that data values from the cell centers are to be exported.

EnSight Parallel

(Available only with Fluent in parallel) creates `.encas` files suitable for use in EnSight Enterprise.

Analysis (for ABAQUS, I-deas Universal, Mechanical APDL Input, NASTRAN, and PATRAN formats

specifies the finite element analysis intended.

Structural

specifies structural analysis and allows you to select the **Structural Loads** to be written.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Thermal

specifies thermal analysis and allows you to select the **Thermal Loads** to be written.

Delimiter (for ASCII format only)

specifies the delimiter used to separate the data fields.

Comma

specifies comma as the delimiter separating the data fields.

Space

specifies space as the delimiter separating the data fields.

Transient (for EnSight Case Gold only)

contains options for exporting transient data.

Transient

enables export of transient data.

Separate Files for Each Timestep

enables writing separate files for each timestep.

Append Frequency

specifies the frequency for appending the data during the solution process.

Important:

The **Append Frequency** option is replaced by the **Write Frequency** option when **Separate Files for Each Timestep** is enabled. You can specify the frequency for writing the separate files.

File Name

specifies the root name for the files to be saved.

Format (for EnSight Case Gold and CDAT for CFD-Post & EnSight)

specifies the file format.

Binary

specifies the file format as binary.

ASCII

specifies the file format as ASCII.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Heat Transfer Coefficient (for TAITherm format only)

specifies the basis for the heat transfer coefficient exported.

Flux Based

specifies the flux based method for writing the heat transfer coefficient.

Wall Function

specifies the wall function based method for writing the heat transfer coefficient.

Write Case File (for CDAT for CFD-Post & EnSight only)

specifies that a case file is written with the `.cdat` file.

Write...

opens The Select File Dialog Box (p. 931) for writing the specified function(s) in the specified format.

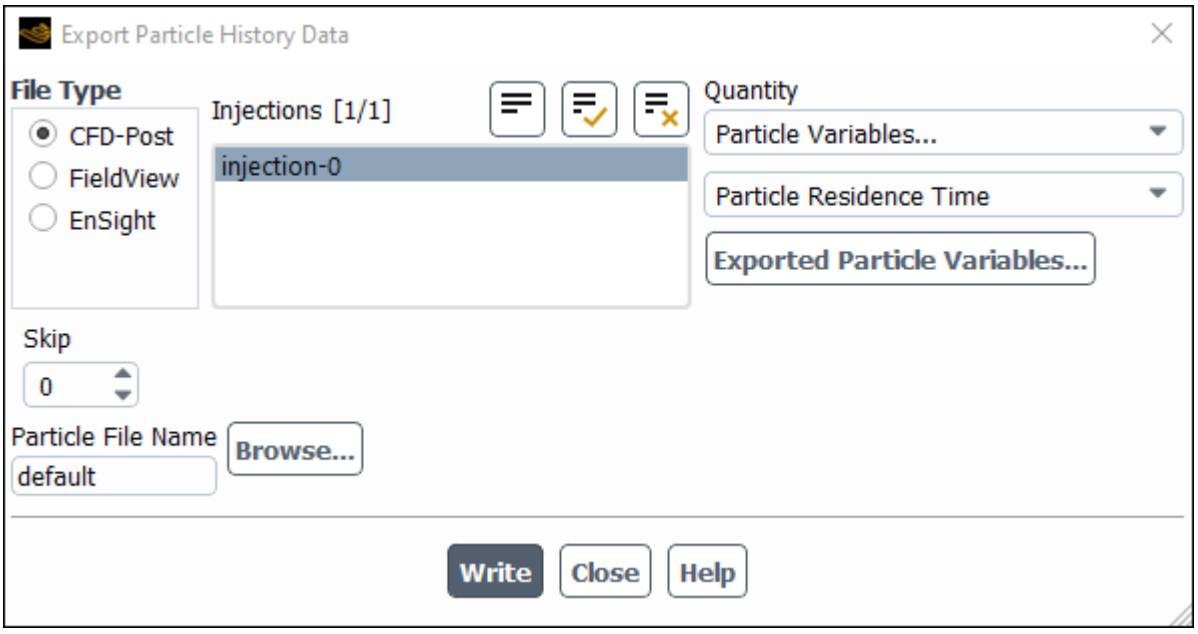
52.1.52. File/Export/Particle History Data...

The **File/Export/Particle History Data...** ribbon tab item opens the Export Particle History Data Dialog Box (p. 5366).

52.1.52.1. Export Particle History Data Dialog Box

The **Export Particle History Data** dialog box allows you to export particle history data as your solution progresses. See Exporting Steady-State Particle History Data (p. 999) for details.

File → Export → Particle History Data...



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Controls

Type

specifies the type of the file you want to write.

CFD-Post

allows you to write the file in CFD-Post particle tracks format, which can be read in **CFD-Post**.

FieldView

allows you to write the file in **FIELDVIEW** format, which can be read in **FIELDVIEW**.

EnSight

allows you to write the file in **EnSight** format.

Geometry

allows you to write the file in `.ibl` format, which can be read in GAMBIT (not available when **Unsteady Particle Tracking** is enabled in the **Discrete Phase Model** dialog box, which is opened by clicking **Discrete Phase...** in the **Physics** ribbon tab).

Injections

allows you to select the required injection from the list of predefined injections.

Quantity

contains the list of variables for which you can export the particle data.

Skip

allows you to "thin" or "sample" the number of particles that are exported.

Coarsen

reduces the exported file size by reducing the number of points that are written for a given trajectory. This is only valid for steady-state cases.

When exporting to an EnSight file, you can use this value to reduce the temporary file size written during tracking with a small effect on accuracy of the trajectory. The number of steps per trajectory finally exported to the EnSight file is the same as the **Number of Particle Time Steps**.

Particle File Name

allows you to specify the file name/directory for the exported data, using the **Browse...** button.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Encas File Name

is the file name you will specify if you selected **EnSight** under **Type**. Use the **Browse...** button to select the `.encas` file that was created when you exported the file with the **File/Export...** ribbon tab option.

Number of Particle Time Steps

appears when you select **EnSight** under **Type**. The specified number is the number of particle time-steps that are saved to the `.encas` file.

52.1.53. File/Export/During Calculation/Solution Data...

The **File/Export/During Calculation/Solution Data...** ribbon tab item opens the Automatic Export Dialog Box (p. 5193) (transient cases only).

52.1.54. File/Export/During Calculation/Particle History Data...

The **File/Export/During Calculation/Particle History Data...** ribbon tab item opens the Automatic Particle History Data Export Dialog Box (p. 5198) (transient cases only).

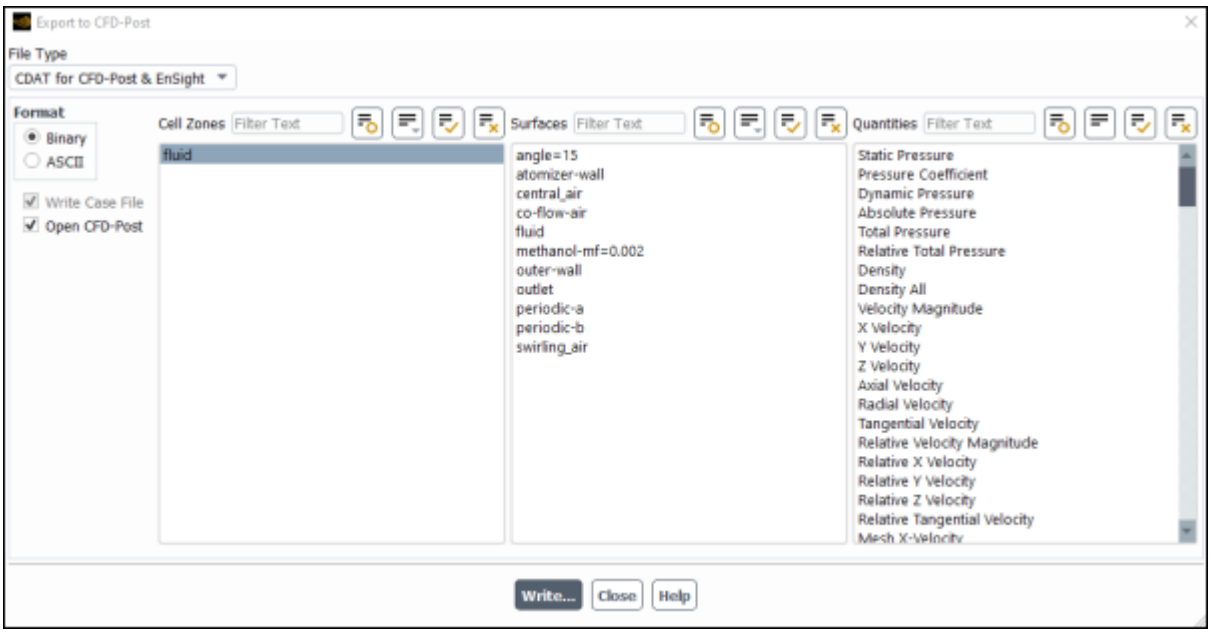
52.1.55. File/Export to CFD-Post...

The **File/Export to CFD-Post...** ribbon tab item opens the Export to CFD-Post Dialog Box (p. 5368).

52.1.55.1. Export to CFD-Post Dialog Box

The **Export to CFD-Post** dialog box allows you to select the quantities that you would like to export to CFD-Post. See Exporting to Ansys CFD-Post (p. 1007) for details.

File → Export to CFD-Post...



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Controls

Select Quantities

contains a selectable list where you can choose the quantities to export.

Format

allows you to export the `.cdat` file in **Binary** or **ASCII** format.

Write Case File

specifies that a case file is written with the `.cdat` file.

Open CFD-Post

specifies that after the files have been written, a CFD-Post session is opened, with the case and `.cdat` files loaded. CFD-Post will display the results.

Write...

opens the **Select File** dialog box, where you can specify the name and location of the exported files.

52.1.56. File/Table File Manager...

The **File/Table File Manager...** ribbon tab item opens the **Table File Manager** dialog box (see Figure 3.7: The Table File Manager Dialog Box (p. 970)).

52.1.57. File/Solution Files...

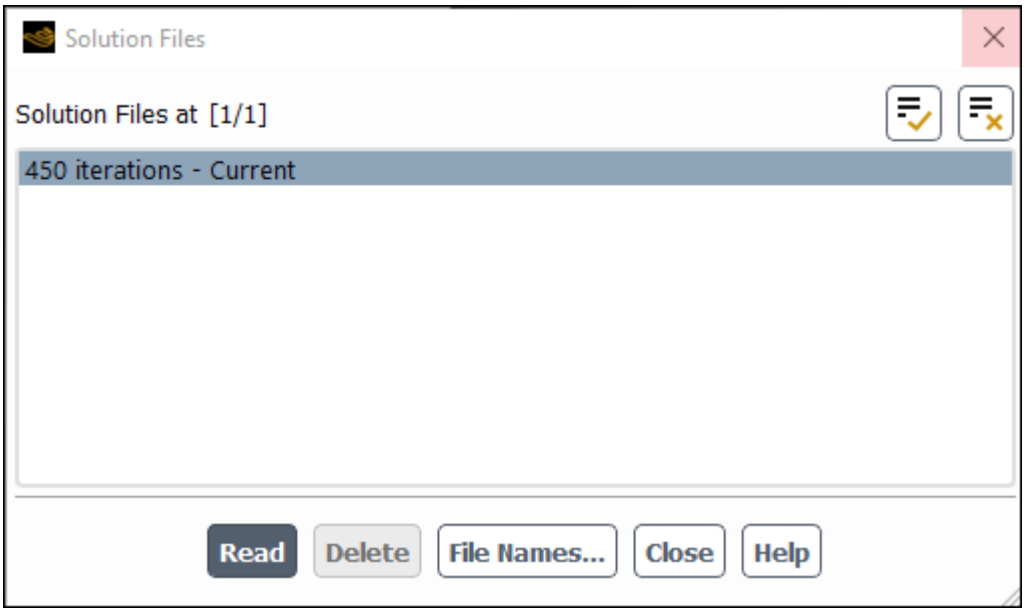
The **File/Solution Files...** ribbon tab item opens the Solution Files Dialog Box (p. 5369).

52.1.57.1. Solution Files Dialog Box

The **Solution Files** dialog box allows you to manage the files that were created through the Autosave Dialog Box (p. 5190). See Managing Solution Files (p. 1009) for details.

File → Solution Files...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Solution Files at

contains a selectable list where you can choose the files to read or delete.

Read

makes the selected file the current file. Note that if more than one file is selected, the **Read** button is disabled. When an earlier solution is made current, the solution files that were generated for a later iteration/time-step will be removed from this list when the calculation continues.

Delete

removes the selected solution files.

File Names...

allows you to obtain information about the solution files and the path of the associated files.

Recover Missing Solution...

Recovers the latest case and data files from the system folder (`<project name>\dp0\FFF\Fluent\` for analysis system and `<project name>\dp0\ FLU \Fluent\` for component system) and list them in the **Solution Files at** list. (This option is available only in Fluent used under Workbench.)

52.1.58. File/Interpolate...

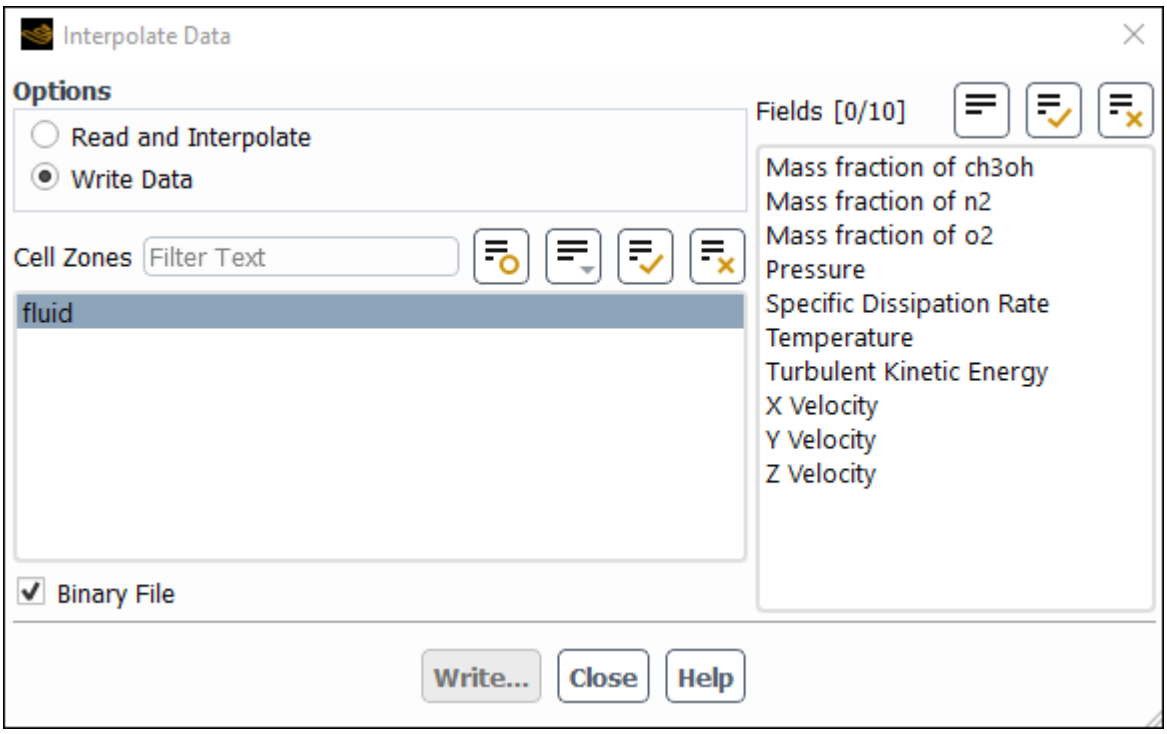
The **File/Interpolate...** ribbon tab item opens the Interpolate Data Dialog Box (p. 5371).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

52.1.58.1. Interpolate Data Dialog Box

The **Interpolate Data** dialog box allows you to interpolate solution data from one mesh to another. See Mesh-to-Mesh Solution Interpolation (p. 1010) for details.

File → Interpolate...



Controls

Options

contains the interpolation options.

Read and Interpolate

allows you to read and interpolate solution data onto the current mesh.

Write Data

allows you to write an interpolation file for the solution data to be interpolated onto another mesh.

Cell Zones

is a list of cell zones that can be selected.

Binary File

allows you to write binary interpolation files. This option is available only when **Write Data** is selected under **Options**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Fields

is a list of all available data fields that can be selected. This list is available only when **Write Data** is selected under **Options**.

Read...

opens The Select File Dialog Box (p. 931), in which you can specify the file to be read. This button is available only when the **Read and Interpolate** option is selected.

Write...

opens The Select File Dialog Box (p. 931), in which you can specify a name for the file to be saved and then save the file. This button replaces the **Read...** button when the **Write Data** option is selected.

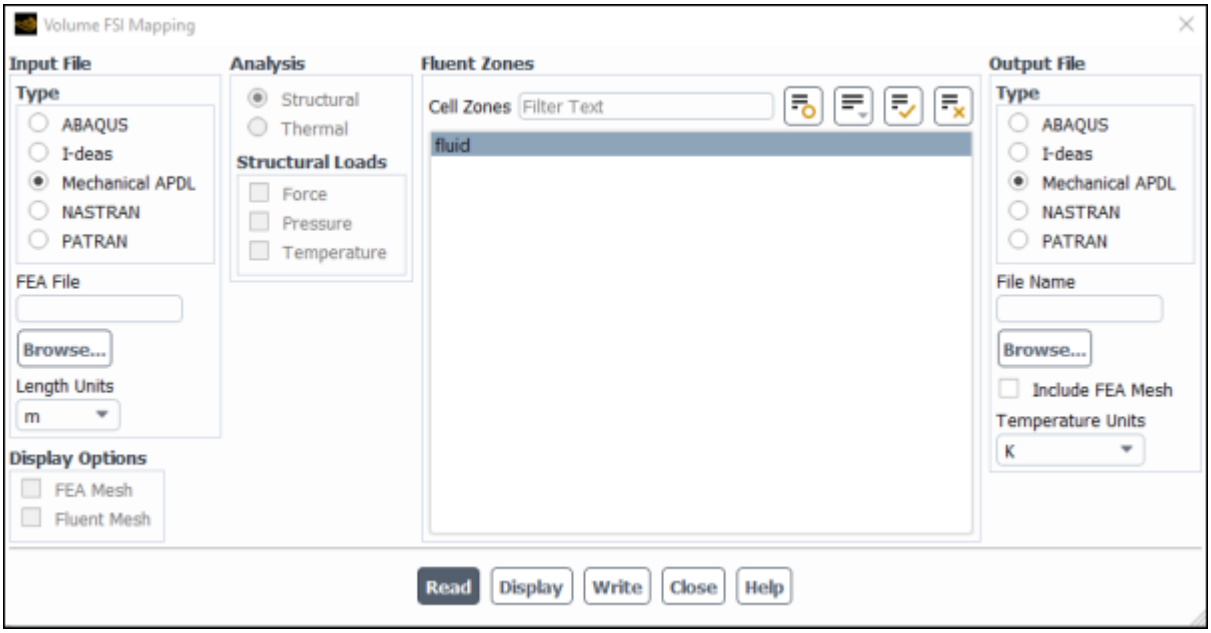
52.1.59. File/FSI Mapping/Volume...

The **File/FSI Mapping/Volume...** ribbon tab item opens the Volume FSI Mapping Dialog Box (p. 5372).

52.1.59.1. Volume FSI Mapping Dialog Box

The **Volume FSI Mapping** dialog box allows you to map cell data for a given geometry from an Ansys Fluent file onto a file with a different mesh and format. See Mapping Data for Fluid-Structure Interaction (FSI) Applications (p. 1013) for a complete description of how to use the dialog box.

File → FSI Mapping → Volume...



Controls

Input File

contains parameters related to the input file.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Type

contains the options for the format of the input mesh file.

ABAQUS

specifies that an ABAQUS file will be used as the input mesh file (extension `.inp`).

I-deas

specifies that an I-deas file will be used as the input mesh file (extension `.unv`).

Mechanical APDL

specifies that a Mechanical APDL file will be used as the input mesh file (extension `.cdb` or `.neu`).

NASTRAN

specifies that a NASTRAN file will be used as the input mesh file (extension `.bdf`).

PATRAN

specifies that a PATRAN file will be used as the input mesh file (extension `.neu`, `.out`, or `.pat`).

FEA File

specifies the name of the input mesh file (see the **Input File Type** descriptions for associated file extensions). If the file is not in the current working folder, include the full path to the folder where it is located.

Browse...

opens The Select File Dialog Box (p. 931), which you can use to specify the input mesh file instead of entering it in the **Input File** text-entry box.

Length Units

specifies the unit of length used in the input file.

Display Options

allows you to choose either the **FEA Mesh** or the **Ansys Fluent Mesh** (or both).

Analysis

contains the options for the kind of data to be mapped.

Structural

specifies that data fields relevant for a structural analysis will be mapped.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Thermal

specifies that data fields relevant for a thermal analysis will be mapped.

Structural Loads

consists of a list of available loads, including **Force**, **Pressure**, and **Temperature**.

Fluent Zones

contains a list of available cell zones from the current Ansys Fluent file from which cell data will be mapped.

Output File

contains the options for the format of the file that will be created from the input mesh file and the mapped data.

Type

contains the options for the format of the output mesh file.

ABAQUS

specifies that an ABAQUS file will be used as the output mesh file (extension `.inp`).

I-deas

specifies that an I-deas file will be used as the output mesh file (extension `.unv`).

Mechanical APDL

specifies that a Mechanical APDL file will be used as the output mesh file (extension `.cdb`).

NASTRAN

specifies that a NASTRAN file will be used as the output mesh file (extension `.bdf`).

PATRAN

specifies that a PATRAN file will be used as the output mesh file (extension `.out`).

File Name

specifies the name of the output mesh file (see the **Output File** descriptions for associated file extensions). If the file is not in the current working folder, include the full path to the folder where it is located.

Browse...

opens The Select File Dialog Box (p. 931), which you can use to specify the output mesh file instead of entering it in the **Output File** text-entry box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Include FEA Mesh

includes additional FEA information like node/element information in the exported output file.

Temperature Units

specifies the unit of temperature when mapping temperature for a structural analysis or any variable for a thermal analysis.

Read

reads the input file into memory.

Display

displays the selected mesh in the graphics window.

Write

writes an output file in which the selected Ansys Fluent cell data is mapped onto the input mesh file.

52.1.60. File/FSI Mapping/Surface...

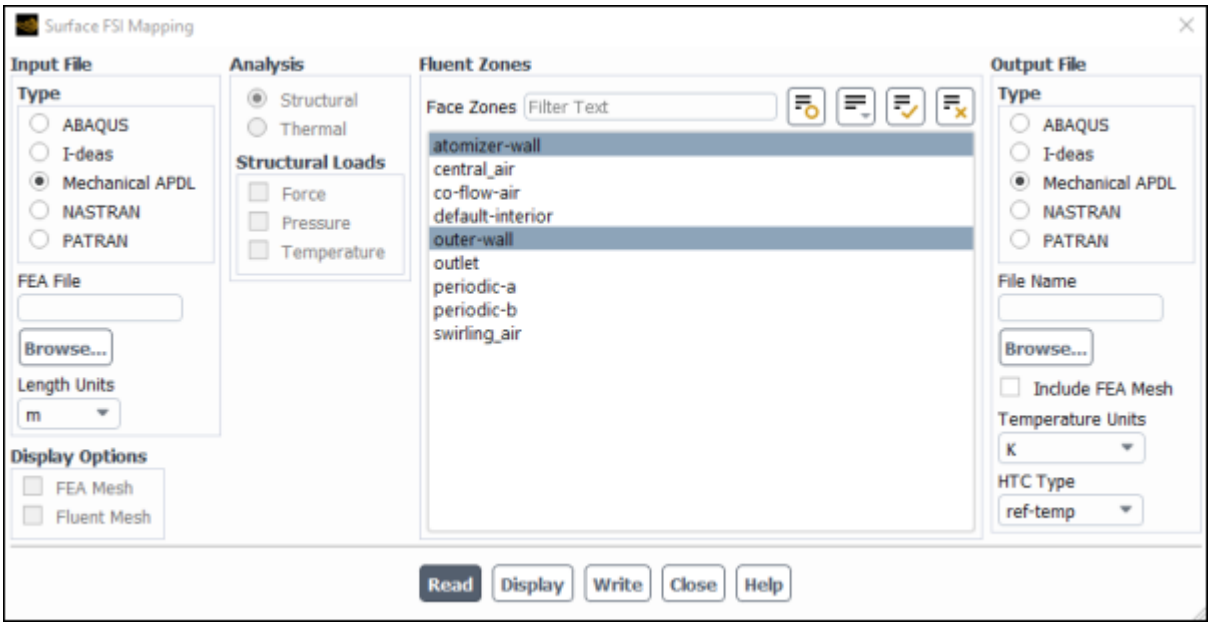
The **File/FSI Mapping/Surface...** ribbon tab item opens the Surface FSI Mapping Dialog Box (p. 5375).

52.1.60.1. Surface FSI Mapping Dialog Box

The **Surface FSI Mapping** dialog box allows you to map face data for a given geometry from an Ansys Fluent file to a file with a different mesh and format. See Mapping Data for Fluid-Structure Interaction (FSI) Applications (p. 1013) for a complete description of how to use the dialog box.

File → **FSI Mapping** → **Surface...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Input File

contains parameters related to the input file.

Type

contains the options for the format of the input mesh file.

ABAQUS

specifies that an ABAQUS file will be used as the input mesh file (extension `.inp`).

I-deas

specifies that an I-deas file will be used as the input mesh file (extension `.unv`).

Mechanical APDL

specifies that a Mechanical APDL file will be used as the input mesh file (extension `.cdb` or `.neu`).

NASTRAN

specifies that a NASTRAN file will be used as the input mesh file (extension `.bdf`).

PATRAN

specifies that a PATRAN file will be used as the input mesh file (extension `.neu`, `.out`, or `.pat`).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

FEA File

specifies the name of the input mesh file (see the **Input File Type** descriptions for associated file extensions). If the file is not in the current working folder, include the full path to the folder where it is located.

Browse...

opens The Select File Dialog Box (p. 931), which you can use to specify the input mesh file instead of entering it in the **Input File** text-entry box.

Length Units

specifies the unit of length used in the input file.

Display Options

allows you to choose either the **FEA Mesh** or the **Ansys Fluent Mesh** (or both).

Analysis

contains the options for the kind of data to be mapped.

Structural

specifies that data fields relevant for a structural analysis will be mapped.

Thermal

specifies that data fields relevant for a thermal analysis will be mapped.

Structural Loads

consists of a list of available loads, including **Force**, **Pressure**, and **Temperature**.

Fluent Zones

contains a list of available face zones from the current Ansys Fluent file from which cell data will be mapped.

Output File

contains the options for the format of the file that will be created from the input mesh file and the mapped data.

Type

contains the options for the format of the output mesh file.

ABAQUS

specifies that an ABAQUS file will be used as the output mesh file (extension `.inp`).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

I-deas

specifies that an I-deas file will be used as the output mesh file (extension `.unv`).

Mechanical APDL

specifies that a Mechanical APDL file will be used as the output mesh file (extension `.cdb`).

NASTRAN

specifies that a NASTRAN file will be used as the output mesh file (extension `.bdf`).

PATRAN

specifies that a PATRAN file will be used as the output mesh file (extension `.out`).

File Name

specifies the name of the input mesh file (see the **Output File** descriptions for associated file extensions). If the file is not in the current working folder, include the full path to the folder where it is located.

Browse...

opens The Select File Dialog Box (p. 931), which you can use to specify the output mesh file instead of entering it in the **Output File** text-entry box.

Include FEA Mesh

includes additional FEA information like node/element information in the exported output file.

Temperature Units

specifies the unit of temperature when mapping temperature for a structural analysis or any variable for a thermal analysis.

HTC Type

specifies the means of calculating the heat transfer coefficient.

ref-temp

uses a temperature equal to the reference temperature to calculate the heat transfer coefficient.

cell-temp

uses a temperature equal to the temperature of the cell adjacent to the face to calculate the heat transfer coefficient.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

wall-func-htc

calculates h_{eff} using Equation 42.69 (p. 4281). Note that this option has the same definition as the field variable **Wall Func. Heat Tran. Coef.**, as described in Alphabetical Listing of Field Variables and Their Definitions (p. 4205).

Read

reads the input file into memory.

Display

displays the selected mesh in the graphics window.

Write

writes an output file in which the Ansys Fluent data has been mapped to the mesh of the input file.

52.1.61. File/Save Picture...

The **File/Save Picture...** ribbon tab item opens the Save Picture Dialog Box (p. 5254).

52.1.62. File/Data File Quantities...

The **File/Data File Quantities...** ribbon tab item opens the Data File Quantities Dialog Box (p. 5192). This is only available when you are writing data files in the legacy format (that is, `.dat` files).

52.1.63. File/Batch Options...

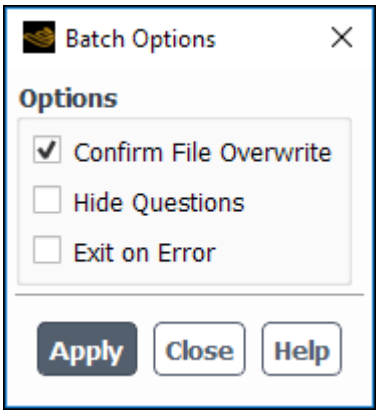
The **File/Batch Options...** ribbon tab item opens the Batch Options Dialog Box (p. 5379).

52.1.63.1. Batch Options Dialog Box

The **Batch Options** dialog box allows you to select options to suppress interactive dialog boxes from Ansys Fluent while running a case in batch mode. For details, see Batch Execution Options (p. 206) in the Getting Started (p. 1) part of this manual.

File → Batch Options...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Confirm File Overwrite

determines whether Ansys Fluent confirms a file overwrite. This option is enabled by default.

Hide Questions

allows you to hide **Question** dialog boxes. This option is disabled by default.

Exit on Error

allows you to automatically exit from batch mode when an error occurs. This option is disabled by default.

When run in batch mode through the command prompt or a journal file with **Exit on Error** enabled, Fluent will exit under the following circumstances:

- Normal run termination upon reaching the end of a journal (return value `0`)
- Error returned during scripted text command execution (return value `1`)
- Unexpected input (wrong type) to text command (return value `1`)
- Licensing error (return value `2`)

If an invalid text command is entered, Fluent will not exit, but proceed to the next text input.

Note that in Windows you must start Fluent with the `-wait` command line option.

52.1.64. File/Idle Timeout...

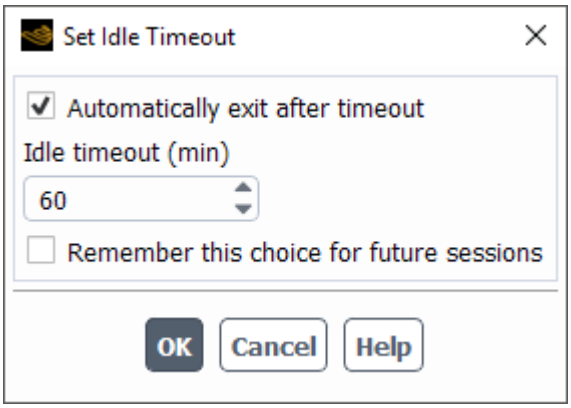
The **File/Idle Timout...** ribbon tab item opens the Set Idle Timeout Dialog Box (p. 5380).

52.1.64.1. Set Idle Timeout Dialog Box

The **Set Idle Timeout** dialog box allows you to specify a time that after which, an idle Ansys Fluent session will close. Refer to Having the Session Close After Sitting Idle (p. 941) for additional information.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

File → Idle Timeout...



Controls

Automatically exit after timeout

enables the idle timer, so that Ansys Fluent will exit after the specified idle time.

Idle timeout (min)

the time after which and idle Ansys Fluent save and exit.

Remember this choice for future sessions

saves the above settings for future sessions. If **Automatically exit after timeout** and **Remember this choice for future sessions** are both enabled when you click **OK**, then next the Ansys Fluent session will have **Automatically exit after timeout** enabled. If **Automatically exit after timeout** is disabled and **Remember this choice for future sessions** is enabled, then the next Ansys Fluent session will not have **Automatically exit after timeout** enabled.

52.1.65. File/Exit

The **File/Exit** ribbon tab item is used to exit from the current solver session.

52.2. Dialog Boxes Available from the Ribbon

The dialog boxes described in this section can be accessed through the ribbon, from the tabs other than the **File** ribbon tab.

52.2.1. 1D Simulation Library Dialog Box
52.2.2. Activate Cell Zones Dialog Box
52.2.3. Adaption Criteria Settings Dialog Box
52.2.4. Adjacency Dialog Box
52.2.5. Advanced Options Dialog Box
52.2.6. Aerodamping (Influence Coefficient Method) Report Definition Dialog Box
52.2.7. Aerodamping (Travelling Wave Method) Report Definition Dialog Box
52.2.8. Animation Definition Dialog Box

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

52.2.9. Application About to Exit Dialog Box
52.2.10. Auto Partition Mesh Dialog Box
52.2.11. Automatic Mesh Adaption Dialog Box
52.2.12. Cell Count Report Definition Dialog Box
52.2.13. Cell Register Display Options Dialog Box
52.2.14. Compiled UDFs Dialog Box
52.2.15. Conduction Layers Dialog Box
52.2.16. Conduction Manager Dialog Box
52.2.17. Contours Dialog Box
52.2.18. Convergence Conditions Dialog Box
52.2.19. Create Mesh Interfaces
52.2.20. Create/Edit Mesh Interfaces Dialog Box
52.2.21. Create/Edit Turbo Interfaces Dialog Box
52.2.22. Curvilinear Coordinate System Dialog Box
52.2.23. Custom Field Function Calculator Dialog Box
52.2.24. Custom Laws Dialog Box
52.2.25. Dashboard Definition Dialog Box
52.2.26. Deactivate Cell Zones Dialog Box
52.2.27. Define Control Points Dialog Box
52.2.28. Delete Cell Zones Dialog Box
52.2.29. Display Options - Adaption Dialog Box
52.2.30. Display States Dialog Box
52.2.31. DPM Report Definition Dialog Box
52.2.32. DPM Source Report Definition Dialog Box
52.2.33. Drag Report Definition Dialog Box
52.2.34. DTRM Graphics Dialog Box
52.2.35. DTRM Rays Dialog Box
52.2.36. Edit Gap Region Dialog Box
52.2.37. Edit Mesh Interfaces Dialog Box
52.2.38. Edit Report File Dialog Box
52.2.39. Edit Report Plot Dialog Box
52.2.40. Execute on Demand Dialog Box
52.2.41. Expression Dialog Box
52.2.42. Expression Editor Dialog Box
52.2.43. Expression Manager Dialog Box
52.2.44. Expression Report Definition Dialog Box
52.2.45. Expression Volume Dialog Box
52.2.46. Face Count Report Definition Dialog Box
52.2.47. Field Function Definitions Dialog Box

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

52.2.48. Flux Report Definition Dialog Box
52.2.49. Force Report Definition Dialog Box
52.2.50. Fuse Face Zones Dialog Box
52.2.51. Gap Model Dialog Box
52.2.52. General Adaption Controls Dialog Box
52.2.53. Generalized/Modal Force Report Definition Dialog Box
52.2.54. Geometry Based Adaption Controls Dialog Box
52.2.55. Geometry Based Adaption Dialog Box
52.2.56. Import Particle Data Dialog Box
52.2.57. Imprint Surface Dialog Box
52.2.58. Improve Mesh Dialog Box
52.2.59. Injections Dialog Box
52.2.60. Input Summary Dialog Box
52.2.61. Interface Creation Options Dialog Box
52.2.62. Interpreted UDFs Dialog Box
52.2.63. Iso-Clip Dialog Box
52.2.64. Iso-Surface Dialog Box
52.2.65. Lift Report Definition Dialog Box
52.2.66. Line Integral Convolutions Dialog Box
52.2.67. Line/Rake Surface Dialog Box
52.2.68. Manual Mesh Adaption Dialog Box
52.2.69. Manage Adaption Criteria Dialog Box
52.2.70. Manage Geometry-Based Adaption Dialog Box
52.2.71. Manage Sponge Layers Dialog Box
52.2.72. Mapped Interface Options Dialog Box
52.2.73. Material Editor Dialog Box
52.2.74. Measure Dialog Box
52.2.75. Merge Zones Dialog Box
52.2.76. Mesh Interfaces Dialog Box
52.2.77. Mesh Morpher/Optimizer Dialog Box
52.2.78. Mixing Planes Dialog Box
52.2.79. Moment Report Definition Dialog Box
52.2.80. Motion Settings Dialog Box
52.2.81. Multi Edit Dialog Box
52.2.82. New Report File Dialog Box
52.2.83. New Report Plot Dialog Box
52.2.84. Objective Function Definition Dialog Box
52.2.85. Optimization History Monitor Dialog Box
52.2.86. Parallel Connectivity Dialog Box

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

52.2.87. Parameter Bounds Dialog Box
52.2.88. Particle Tracks Dialog Box
52.2.89. Partition Surface Dialog Box
52.2.90. Partitioning and Load Balancing Dialog Box
52.2.91. Pathlines Dialog Box
52.2.92. PCB Model Dialog Box
52.2.93. Plane Surface Dialog Box
52.2.94. Point Surface Dialog Box
52.2.95. Quadric Surface Dialog Box
52.2.96. Raytracing Display Dialog Box
52.2.97. Reduced Order Model Dialog Box
52.2.98. Reference Frame Dialog Box
52.2.99. Replace Cell Zone Dialog Box
52.2.100. Report Definitions Dialog Box
52.2.101. Report File Definitions Dialog Box
52.2.102. Report Plot Definitions Dialog Box
52.2.103. Residual Monitors Dialog Box
52.2.104. Rotate Mesh Dialog Box
52.2.105. S2S Information Dialog Box
52.2.106. Select UDM Zones Dialog Box
52.2.107. Select Window Dialog Box
52.2.108. Separate Cell Zones Dialog Box
52.2.109. Separate Face Zones Dialog Box
52.2.110. Set Injection Properties Dialog Box
52.2.111. Set Multiple Injection Properties Dialog Box
52.2.112. Sponge Layer Dialog Box
52.2.113. Structural Point Surface Dialog Box
52.2.114. Surface Meshes Dialog Box
52.2.115. Surface Rendering Properties Dialog Box
52.2.116. Surface Report Definition Dialog Box
52.2.117. Surfaces Dialog Box
52.2.118. Thread Control Dialog Box
52.2.119. Time Averaged Explicit Thermal Coupling Dialog Box
52.2.120. Transform Surface Dialog Box
52.2.121. Translate Mesh Dialog Box
52.2.122. Turbo 2D Contours Dialog Box
52.2.123. Turbo Averaged Contours Dialog Box
52.2.124. Turbo Averaged XY Plot Dialog Box
52.2.125. Turbo Options Dialog Box

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

52.2.126. Turbo Report Dialog Box
52.2.127. Turbo Topology Dialog Box
52.2.128. UDF Library Manager Dialog Box
52.2.129. User-Defined Fan Model Dialog Box
52.2.130. User-Defined Function Hooks Dialog Box
52.2.131. User-Defined Memory Dialog Box
52.2.132. User Defined Report Definition Dialog Box
52.2.133. User-Defined Scalars Dialog Box
52.2.134. Vectors Dialog Box
52.2.135. Volume Report Definition Dialog Box
52.2.136. Warning Dialog Box
52.2.137. Zone Surface Dialog Box
52.2.138. Zone Type Color and Material Assignment Dialog Box

52.2.1. 1D Simulation Library Dialog Box

The **1D Simulation Library** dialog box allows you to set parameters related to coupling between Ansys Fluent and GT-POWER or WAVE. See Coupling Boundary Conditions with GT-POWER (p. 1587) or Coupling Boundary Conditions with WAVE (p. 1592) for details about the items below.

User Defined → **Model Specific** → **1D Coupling...**

Controls

1D Library

specifies the type of library to be used. (Currently only **GTpower** and **WAVE** are available.)

1D Input File Name

specifies the name of the GT-POWER or WAVE input file.

Start

starts up GT-POWER or WAVE and generates Ansys Fluent user-defined functions for each boundary in the input file.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Stop

unlinks the shared library.

52.2.2. Activate Cell Zones Dialog Box

The **Activate Cell Zones** dialog box allows you to activate a single cell zone or multiple zones. See Activating Zones (p. 1278) for details.

Domain → Zones → Activate...

Controls

Cell Zones

contains a list of cell zones from which you can select the zone to be activated.

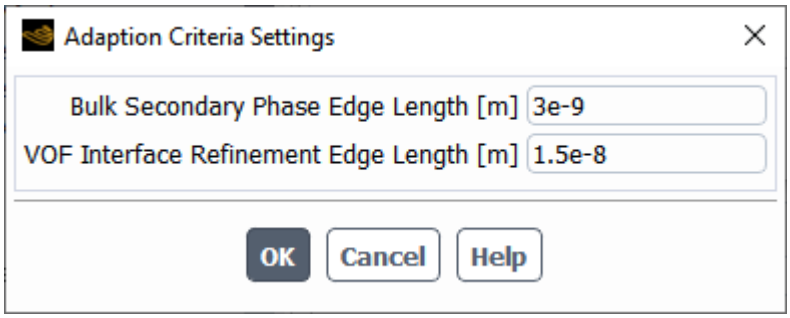
Activate

activates the selected cell zones.

52.2.3. Adaption Criteria Settings Dialog Box

The **Adaption Criteria Settings** dialog box allows you to complete the setup of an adaption criterion when you have made a selection from the **Predefined Criteria** drop-down list in the Manual Mesh Adaption Dialog Box (p. 5515) or Automatic Mesh Adaption Dialog Box (p. 5403); the **Adaption Criteria Settings** dialog box opens automatically, and is used to define necessary settings or parameters in the refinement / coarsening criterion expression or overset adaption settings. See Refining and Coarsening (p. 3777) and Predefined Criteria for Adaption (p. 3784) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

(when setting up the **Cell Distance** predefined criterion under **Boundary Layer...**)

Boundary Zones

specifies the boundary zones where adjacent cell layers will be split for boundary layer adaption. The selections are used in a boundary register named **boundary_cell_distance** that is selected in the **Refinement Criterion** field in the Manual Mesh Adaption Dialog Box (p. 5515).

Cell Layers to Split

specifies the number of cell layers that will be split for boundary layer adaption. The value you enter is used in a boundary register named **boundary_cell_distance** that is selected in the **Refinement Criterion** field in the Manual Mesh Adaption Dialog Box (p. 5515).

(when setting up the **Flame Indicator** predefined criterion under **Combustion...**)

Include Vortex Indicator

specifies that the magnitude of vorticity is included as part of the criteria for refinement and coarsening.

Spark Region Refinement

enables refinement in a temporary spherical spark region at the start of the simulation (which will then later be coarsened back to the original mesh).

Time

sets the maximum time during which refinement is applied to the spark region, as part of the **Spark Region Refinement** option. The value you enter will automatically update an expression named **spark**_<*ID*>, which is a parameter in the **flame_refinement**_<*ID*> expression used in the **Refinement Criterion** field in the Manual Mesh Adaption Dialog Box (p. 5515) or Automatic Mesh Adaption Dialog Box (p. 5403).

Centroid X

sets the x-coordinate for the center of the spark region as part of the **Spark Region Refinement** option. The value you enter will automatically update a cell register named **spark_region**, which is used in an expression named **spark**_<*ID*> that is a parameter in the **flame_refinement**_<*ID*> expression used in the **Refinement Criterion** field in the Manual Mesh Adaption Dialog Box (p. 5515) or Automatic Mesh Adaption Dialog Box (p. 5403).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Centroid Y

sets the y-coordinate for the center of the spark region as part of the **Spark Region Refinement** option. The value you enter will automatically update a cell register named **spark_region**, which is used in an expression named **spark**_<*ID*> that is a parameter in the **flame_refinement**_<*ID*> expression used in the **Refinement Criterion** field in the Manual Mesh Adaption Dialog Box (p. 5515) or Automatic Mesh Adaption Dialog Box (p. 5403).

Centroid Z

sets the z-coordinate for the center of the spark region as part of the **Spark Region Refinement** option. The value you enter will automatically update a cell register named **spark_region**, which is used in an expression named **spark**_<*ID*> that is a parameter in the **flame_refinement**_<*ID*> expression used in the **Refinement Criterion** field in the Manual Mesh Adaption Dialog Box (p. 5515) or Automatic Mesh Adaption Dialog Box (p. 5403).

Radius

sets the radius of the spherical spark region as part of the **Spark Region Refinement** option. The value you enter will automatically update a cell register named **spark_region**, which is used in an expression named **spark**_<*ID*> that is a parameter in the **flame_refinement**_<*ID*> expression used in the **Refinement Criterion** field in the Manual Mesh Adaption Dialog Box (p. 5515) or Automatic Mesh Adaption Dialog Box (p. 5403).

(when setting up the **Volume of Fluid** predefined criterion under **Multiphase...**)

Minimum Cell Length Scale

sets the minimum cell length scale for refinement. By default this is set to the minimum cell edge length in the initial mesh. It is used to automatically update the cell volume in an expression named **vof_refinement**_<*ID*> that is used in the **Refinement Criterion** field in the Manual Mesh Adaption Dialog Box (p. 5515) or Automatic Mesh Adaption Dialog Box (p. 5403).

(when setting up the **VOF-to-DPM [Generic]** predefined criterion under **Multiphase...**)

Number of Cells Across Resolved Droplet

sets the number of cells across the resolved droplet (N). It is used to automatically update the cell volume (as described in the definition that follows) used in the **vof_to_dpm_refinement**_<*ID*> expression that is used in the **Refinement Criterion** field in the Manual Mesh Adaption Dialog Box (p. 5515) or Automatic Mesh Adaption Dialog Box (p. 5403).

Minimum Resolved Droplet Diameter

sets the minimum diameter of droplets that are resolved in the simulation (D). By default this is set to the minimum cell edge length in the initial mesh. It is used (with the **Number of Cells Across Resolved Droplet** setting) to automatically update the cell volume used in the **vof_to_dpm_refinement**_<*ID*> expression, by setting it to $\left(D/N\right)^3$. This expression is then used in the **Refinement Criterion** field in the Manual Mesh Adaption Dialog Box (p. 5515) or Automatic Mesh Adaption Dialog Box (p. 5403).

(when setting up the **VOF-to-DPM [Advanced]** predefined criterion under **Multiphase...**)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Bulk Secondary Phase Edge Length

specifies an edge length that is cubed in order to define the minimum cell volume that initiates refinement in the secondary phase. This volume is used in an expression named **vof_to_dpm_refinement_1** that is used in the **Refinement Criterion** field in the Manual Mesh Adaption Dialog Box (p. 5515) or Automatic Mesh Adaption Dialog Box (p. 5403).

VOF Interface Refinement Edge Length

specifies an edge length that is cubed in order to define minimum / maximum cell volume that initiates refinement / coarsening at the VOF interface. This volume is used in expressions named **vof_to_dpm_refinement_1** and **vof_to_dpm_coarsening_1**, which are then used in the **Refinement Criterion** and **Coarsening Criterion** fields, respectively, in the Manual Mesh Adaption Dialog Box (p. 5515) or Automatic Mesh Adaption Dialog Box (p. 5403).

(when setting up the **Overset** predefined criterion)

Orphan Adaption

enables adaption that attempts to remove orphan cells. For details, see Marking for Orphan Adaption (p. 1219).

Size Adaption

enables adaption that reduces size mismatches between donor and receptor cells. For details, see Marking for Size Adaption (p. 1219).

Maximum Length Scale Ratio

sets the threshold (R_l) used as part of the **Size Adaption** option to determine which cells are marked for adaption based on donor-receptor cell size differences. This is set to 3 by default.

Gap Adaption

enables adaption that increases the mesh resolution in gaps as needed (in order to prevent the creation of orphan cells). For details, see Marking for Gap Adaption (p. 1219).

Gap Resolution

specifies the target (minimum) gap resolution used as part of the **Gap Adaption** option when marking cells for gap adaption. This is set to 4 by default.

Coarsening

enables the coarsening of the mesh when mesh refinement is no longer needed.

(when setting up the **Goal-Based Error Indicator** predefined criterion under **Aerodynamics...**)

Set Up Goal...

Opens the **Adjoint Observables** dialog box where you can select, edit, or create the desired observable.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Skip Initial Adjoint Advancement Before Adaption

When enabled, if the adjoint solution is calculated, the mesh adaption will ignore the computationally expensive initial adjoint iterations which are specified by **Initial Adjoint Iterations before Adaption**.

Initial Adjoint Iterations Before Adaption

This specifies the number of adjoint iterations in the initial adjoint calculation before the first mesh adaption.

Adjoint Iterations Before Adaption

This specifies the number of adjoint iterations during the adjoint calculation before each mesh adaption.

Number of Modes for Adjoint Advancement

This specifies the number of modes in the Krylov subspace used to approximate the solution. A larger value will help to converge the adjoint solution at the expense of memory cost and potentially more computational time.

Smoothing Steps Before Adaption

A larger value will lead to smoother adaption.

Reset Adaption History

Deletes any previous adaption history.

Print Adaption History

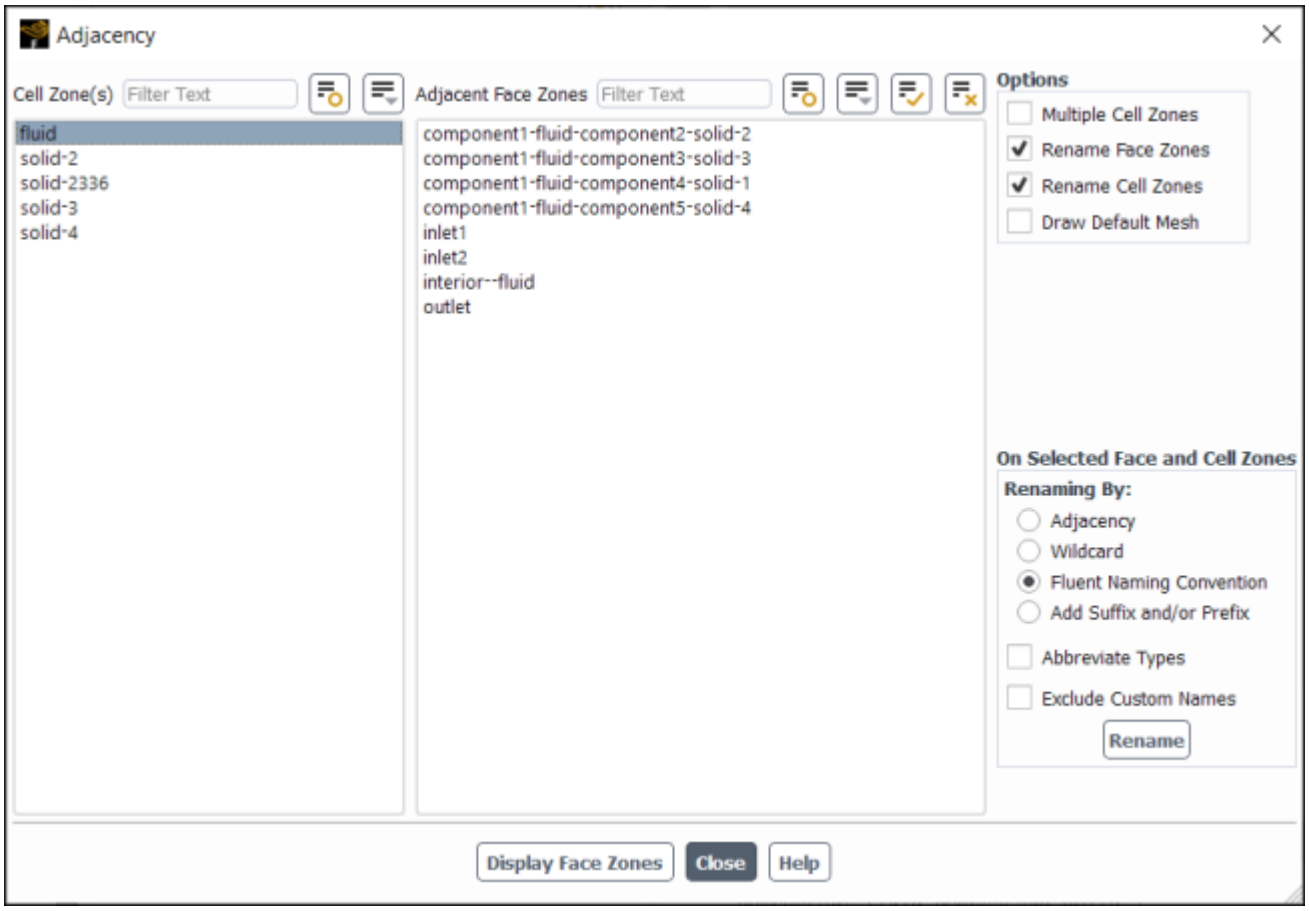
Prints the goal-based adaption history in the console.

52.2.4. Adjacency Dialog Box

The **Adjacency** dialog box allows you to identify, display, and rename face zones based on their adjacency to a selected cell zone. You can also rename cell zones by zone type and ID, by wildcard, and by adding prefixes/suffixes. See Managing Adjacent Zones (p. 1281) for details.

Domain → Zones → Adjacency...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Cell Zone(s)

contains a list of the cell zones for which you can find adjacent face zones.

Adjacent Face Zones

contains a list of the face zones that are adjacent to the selected cell zones.

Options

contains relevant options for adjacency.

Multiple Cell Zones

allows you to select multiple cell zones at once. The zones listed in **Adjacent Face Zones** will be the union of all zones adjacent to the selected cell zones.

Rename Face Zones

allows you to rename selected face zones based on adjacency, zone type, wild card, or add a prefix/suffix.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Rename Cell Zones

allows you to rename selected cell zones based zone type and ID, wild card, or by adding a prefix/suffix.

Draw Default Mesh

brings up the Mesh Display Dialog Box (p. 4688) where you can specify zones of the mesh to be permanently displayed as you display or hide face zones from within the **Adjacency** dialog box. This can be helpful in cases where the mesh is very complex. If this option is not enabled, only the zones selected in **Adjacent Face Zones** will be displayed when you click **Display Face Zones**.

On Selected Face Zones

contains controls for renaming face zones that are selected in **Adjacent Face Zones**. For details on the renaming methods, refer to Renaming Zones Using the Adjacency Dialog Box (p. 1283).

Renaming By:

contains options for how you can rename surfaces.

Adjacency

renames the selected face zones incorporating the name of the adjacent cell zone and the face zone type when you click **Rename**.

Fluent Naming Convention

renames the selected surface to be a combination of the surface type and the surface ID when you click **Rename**. Cell zones are renamed to be a combination of the zone type and the zone ID.

Wildcard

renames the selected face zones or cell zones based on a pattern match string in the **From** field and a replacement string in the **To** field.

Add Suffix and/or Prefix

adds the provided text in the **Suffix or Prefix** field to either the beginning or end of the selected zone name(s). Click **Add Suffix** to add the provided text at the end of the name and click **Add Prefix** to add the provided text at the beginning of the name.

Abbreviate Types

uses abbreviations for the zone types when renaming rather than the full zone type text.

Exclude Custom Names

excludes from renaming any zones that do not match a recognized naming pattern. This can be useful to prevent inadvertently replacing a meaningful name.

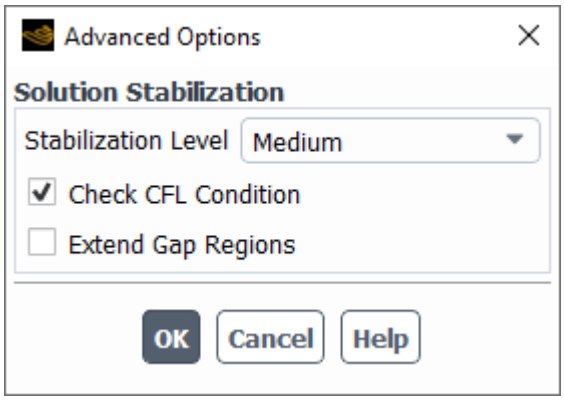
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Displace Face Zones

displays the face zones selected in **Adjacent Face Zones**.

52.2.5. Advanced Options Dialog Box

The **Advanced Options** dialog box provides advanced options for a simulation that uses the gap model to simulate flow through narrow gaps that open or close over time (such as in a valve). It is opened by clicking **Advanced Options...** in the Gap Model Dialog Box (p. 5484). See also Controlling Flow in Narrow Gaps for Valves and Pumps (p. 1234).



Controls

Solution Stabilization

allows you to define solution stabilization settings for the gap model.

Stabilization Level

allows you to define the level of solution stabilization for the gap model simulation. For each level, the solver adjusts the numerical settings, discretization schemes, under-relaxations, AMG settings, and so on, and controls the flow behavior in vicinity of the gap regions. The cost of computation increases with higher levels. Note that these changes are applied globally to all gap regions, and a level other than **None** will override the local stabilization applied by default to flow-modeling regions that use the **Sponge Layer** method.

Check CFL Condition

enables the automatic determination of a desirable acoustic Courant (CFL) number based on the selected **Stabilization Level**, and then provides a warning message in the console if this CFL number would require you to specify a smaller time step for the calculation in order to improve the solver stability.

Extend Gap Regions

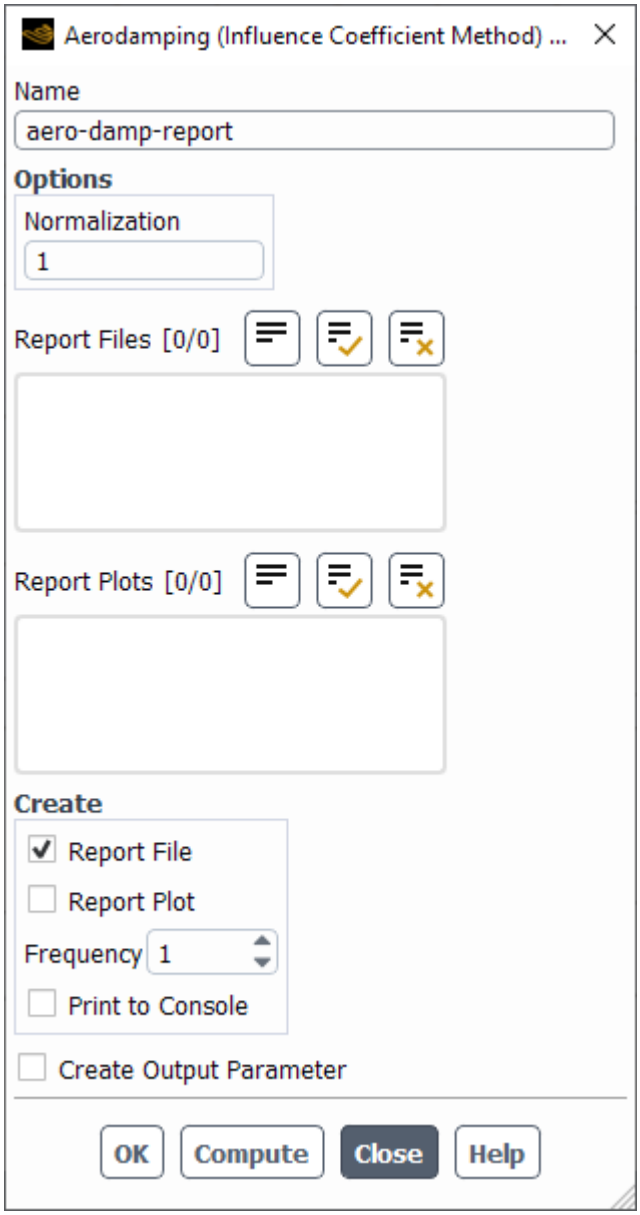
enables the extending of the gap regions by including additional cells in the vicinity of the gap interfaces during marking. This is useful when the default shape of the marked cells is negatively affecting solution stability or convergence behavior.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

52.2.6. Aerodamping (Influence Coefficient Method) Report Definition Dialog Box

For more details, see ICM Method Post-processing (p. 1999).

Solution → Reports → Definitions → New → Aeromechanics Report → Aerodamping (Influence Coefficient Method)...



Controls

Name

specifies the name of the report definition.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Options

contains options for your aerodamping report definition.

Normalization

Sets the normalization factor for the damping value.

Report Files

lists all of the report files where you can write the report definition data.

Report Plots

lists all of the report plots where you can plot the report definition data (assuming this report has the same units as the other report definitions included in the selected report plot).

Create

Report File

creates a new report file that includes this report definition.

Report Plot

creates a new report plot that includes this report definition.

Frequency

specifies how frequently the report definition is written, plotted, or printed to the console.

Print to Console

prints the value of the report definition to the console.

Create Output Parameter

creates an output parameter for this report definition with the name `<report definition name>-op`.

OK

creates the report definition.

Compute

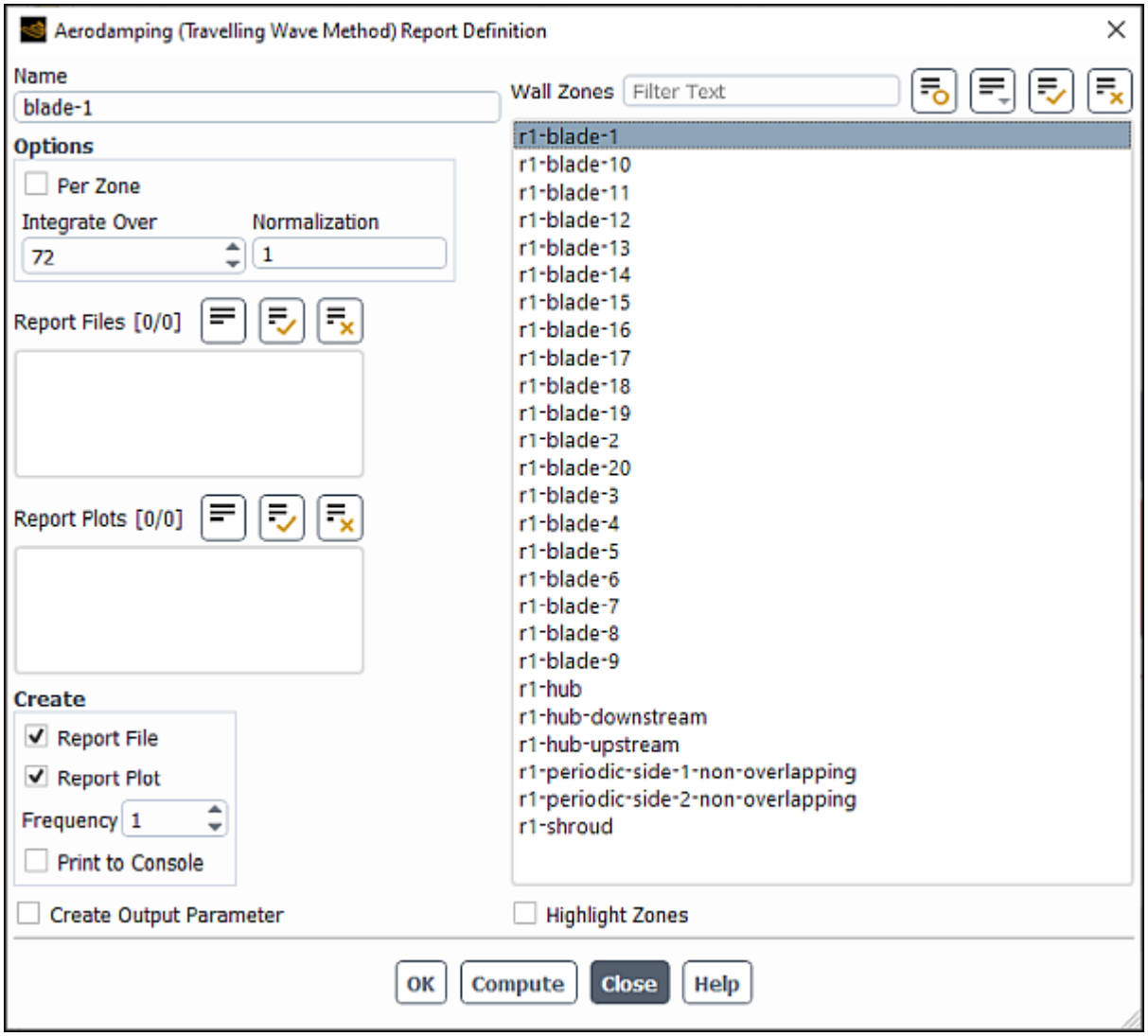
computes the value of the report definition at the selected zone(s) and prints to the console.

52.2.7. Aerodamping (Travelling Wave Method) Report Definition Dialog Box

For more details, see TWM Method Post-processing (p. 1996).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution → Reports → Definitions → New → Aeromechanics Report → Aerodamping (Travelling Wave Method)...



Controls

Name

specifies the name of the report definition.

Options

contains options for your aerodamping report definition.

Per Zone

If you are modeling multiple (main) blades, enables the damping value to be calculated for each dynamic mesh zone (which encompasses one main blade and possibly accompanying splitter blades).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Integrate Over

sets the number of time steps over which to integrate when calculating the damping value; a suggested value is the number of time steps per period..

Normalization

Sets the normalization factor for the damping value.

Report Files

lists all of the report files where you can write the report definition data.

Report Plots

lists all of the report plots where you can plot the report definition data (assuming this report has the same units as the other report definitions included in the selected report plot).

Create

Report File

creates a new report file that includes this report definition.

Report Plot

creates a new report plot that includes this report definition.

Frequency

specifies how frequently the report definition is written, plotted, or printed to the console.

Print to Console

prints the value of the report definition to the console.

Create Output Parameter

creates an output parameter for this report definition with the name `<report definition name>-op`.

Wall Zones

contains a selectable list of valid walls zones for aerodamping report.

OK

creates the report definition.

Compute

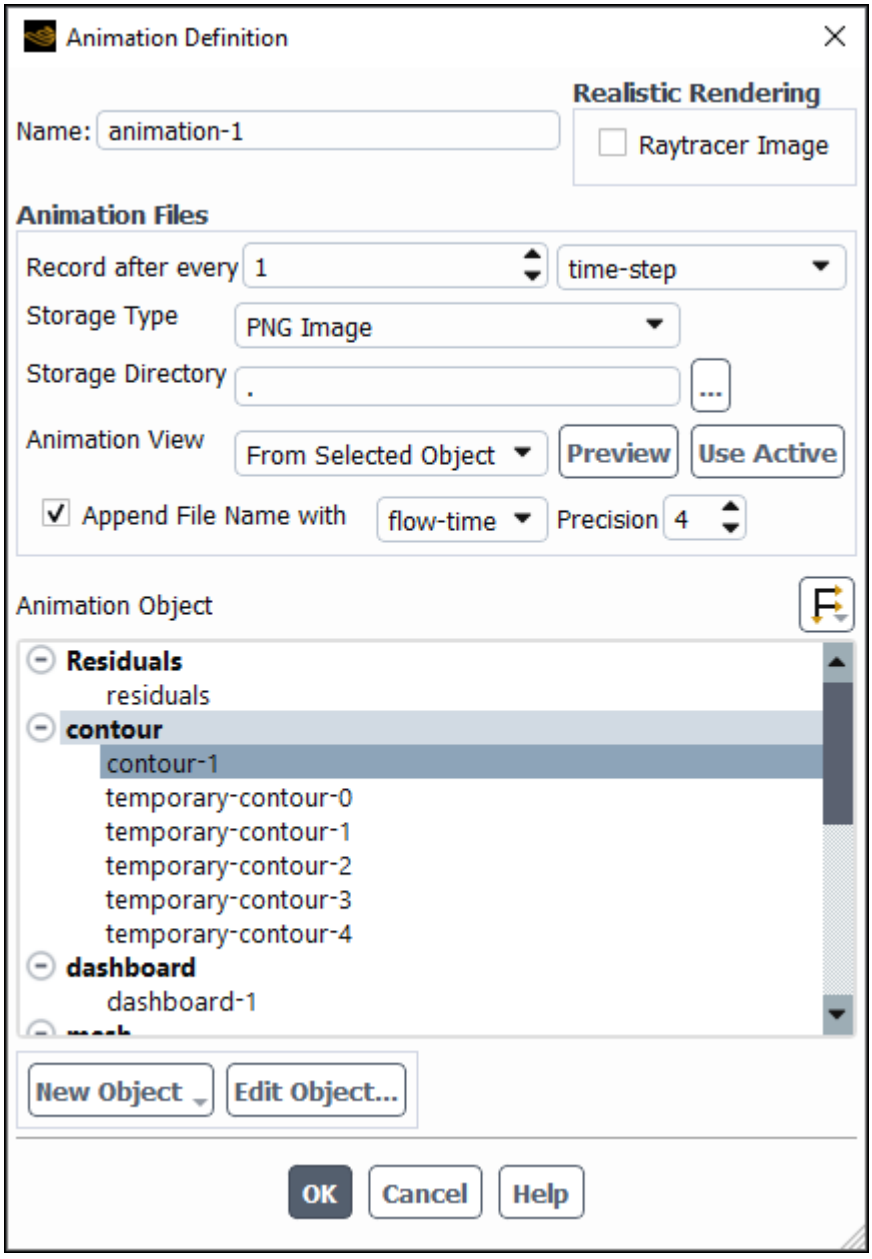
computes the value of the report definition at the selected zone(s) and prints to the console.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

52.2.8. Animation Definition Dialog Box

The **Animation Definition** dialog box allows you to specify a graphics object that will be captured during the solution calculation so that it may be played back as an animation during postprocessing.

Solution → Activities → Create → Solution Animations...



Controls

Name

name of the animation definition.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Realistic Rendering

option for enhancing the appearance of the animation object by using a raytracer for realistic lighting effects.

Raytracer Image

indicates that animation images will use a raytracing renderer for enhanced image quality. Refer to Realistic Rendering Using Raytracing (p. 3995) for additional information on raytracing.

Animation Files

controls for how animation files are stored.

Record after every

specifies the frequency that images are captured during the calculation. You can select whether the frequency is **Iteration**, **Time Step** (transient only) or **Flow Time** (transient only) from the drop-down.

Storage Type

drop-down list with the available formats for storing the animation images.

None

no images are saved. This option is intended for use with embedded window dashboards (Embedded Graphics Window Dashboards (p. 3941)).

In Memory

the image files associated with the animation will lost when you exit Fluent.

PPM Image

Fluent will save the animation images in PPM Image format (`.ppm`), which is 2D.

HSF File

Fluent will save the animation images in HSF file format (`.hsf`), which is 3D.

PNG Image

Fluent will save the animation images in PNG file format (`.png`), which is 2D.

JPEG Image

Fluent will save the animation images in JPEG file format (`.jpg`), which is 2D.

TIFF Image

Fluent will save the animation images in tagged image file format (TIFF—`.tif`), which is 2D.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Storage Directory

shows the location where the animation images will be stored if the **Storage Type** is set to any of the types other than **In Memory**.

Note:

Animation files are written to the same directory as the case file, by default. This is stored as a relative path, represented by a dot (.), so that when a case file is shared, animation files are written to an appropriate location.

If you manually specify a path, then what you are specifying is an absolute path that will not automatically be updated if the case file is shared or run from another directory.

Animation View

specifies the animation object orientation that will be used when the animation images are capture during the calculation.

views drop-down

list with all of the defined views.

Preview

displays the selected animation object in a reserved graphics window.

Use Active

saves the current view in the active graphics window and automatically selects it for use in the animation definition.

Append File Name with

indicates that the **Name** of the animation files will have either the time-step, flow-time, or iteration appended. The drop-down menu allows you to specify which one.

Precision

indicates the number of decimal places included when the **flow-time** is appended.

Animation Object

lists the defined graphics objects that may be selected for animation.

New Object

drop-down list allowing you to create new graphics objects for animating.

- **Mesh...** opens the Mesh Display Dialog Box (p. 4688).
- **Contours...** opens the Contours Dialog Box (p. 5412).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Vectors...** opens the Vectors Dialog Box (p. 5655).
- **Pathlines...** opens the Pathlines Dialog Box (p. 5572).
- **Particle Tracks...** opens the Particle Tracks Dialog Box (p. 5558).
- **Scene...** opens Figure 40.50: The Scene Dialog Box (p. 3936).
- **XY Plot...** opens the Solution XY Plot Dialog Box (p. 5290).
- **Report Plot...** opens the New Report Plot Dialog Box (p. 5552).

Edit Object...

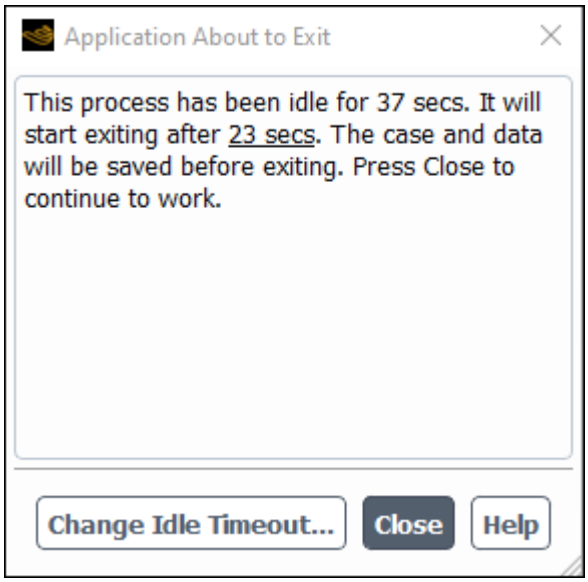
allows you to edit the selected graphics object.

OK

creates the animation definition and/or saves settings changes and closes the dialog box.

52.2.9. Application About to Exit Dialog Box

The **Application About to Exit** dialog box gives you chance to stop the Ansys Fluent session from saving and exiting as the session approaches the idle timeout set in the Set Idle Timeout Dialog Box (p. 5380). Refer to Having the Session Close After Sitting Idle (p. 941) for additional information.



Controls

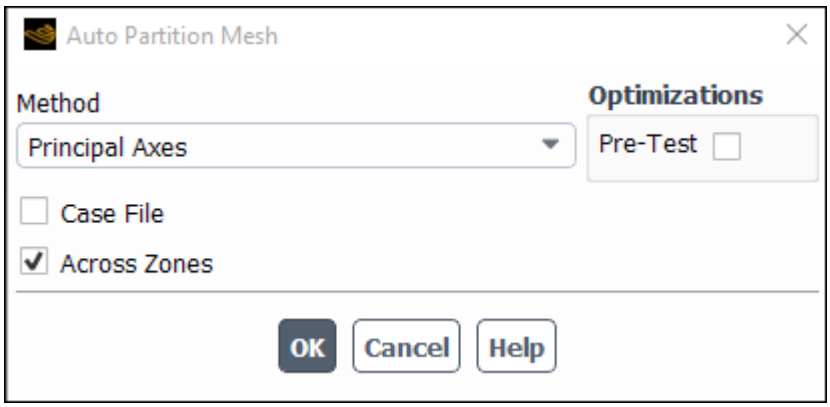
Change Idle Timeout...

opens the Set Idle Timeout Dialog Box (p. 5380).

52.2.10. Auto Partition Mesh Dialog Box

The **Auto Partition Mesh** dialog box allows you to set the parameters for automatic partitioning when reading an unpartitioned mesh into the parallel solver. See Partitioning the Mesh Automatically (p. 4316) for details.

Parallel → General → Auto Partition...



Controls

Method

contains a drop-down list of the recursive partition methods that can be used to create the mesh partitions. The choices include the **Cartesian Axes**, **Cartesian Strip**, **Cartesian X-Coordinate**, **Cartesian Y-Coordinate**, **Cartesian Z-Coordinate**, **Cartesian R Axes**, **Cartesian RX-Coordinate**, **Cartesian RY-Coordinate**, **Cartesian RZ-Coordinate**, **Cylindrical Axes**, **Cylindrical R-Coordinate**, **Cylindrical Theta-Coordinate**, **Cylindrical Z-Coordinate**, **Metis**, **Polar Axes**, **Polar R-Coordinate**, **Polar Theta-Coordinate**, **Principal Axes**, **Principal Strip**, **Principal X-Coordinate**, **Principal Y-Coordinate**, **Principal Z-Coordinate**, **Spherical Axes**, **Spherical Rho-Coordinate**, **Spherical Theta-Coordinate**, and **Spherical Phi-Coordinate** techniques, which are described in Mesh Partitioning Methods (p. 4331).

Case File

allows you to use a valid existing partition section in a case file (that is, one where the number of partitions in the case file divides evenly into the number of compute nodes). You need to turn off the **Case File** option only if you want to change other parameters in the **Auto Partition Mesh** dialog box.

Across Zones

allows partitions to cross zone boundaries (the default). If turned off, it will restrict partitioning to within each cell zone. This is recommended *only* when cells in different zones require significantly different amounts of computation during the solution phase, for example if the domain contains both solid and fluid zones.

Optimizations

contains an option to enable pre-testing.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

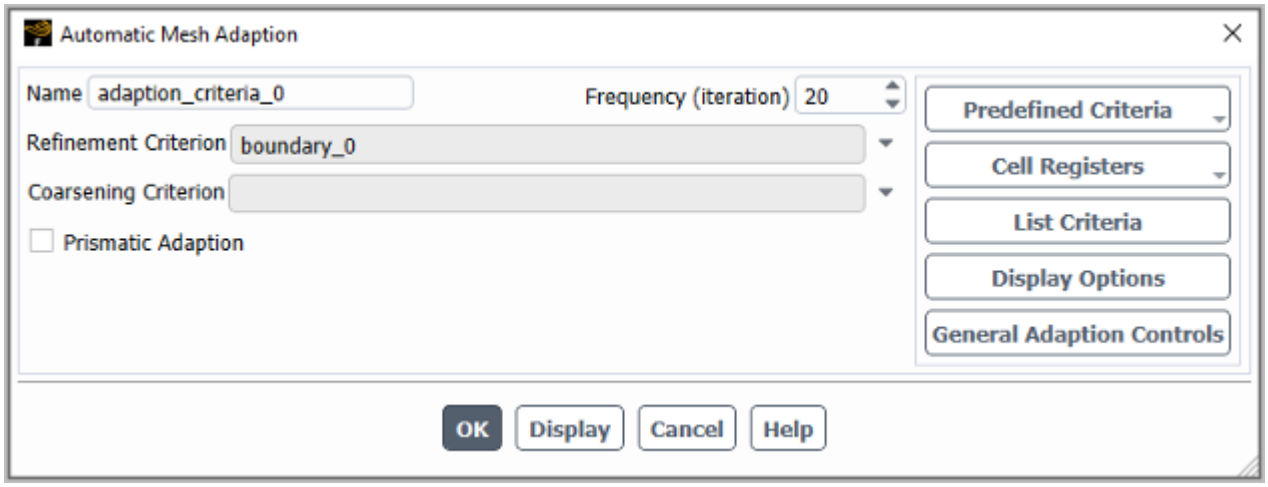
Pre-Test

instructs Fluent to test all coordinate directions and choose the one which yields the fewest partition interfaces for the final bisection. Note that this option is available only when you choose **Principal Axes** or **Cartesian Axes** as the partitioning method.

52.2.11. Automatic Mesh Adaption Dialog Box

The **Automatic Mesh Adaption** dialog box allows you to create and edit criteria that Fluent will use to adapt your mesh during a calculation. See Refining and Coarsening (p. 3777) for details.

Domain → Adapt → Automatic...



Controls

Name

specifies the name of the adaption criteria.

Frequency

defines the frequency of the adaption that will be applied for this criteria during the calculation.

Refinement Criterion

specifies the cell register or expression that Fluent will use during the calculation to determine which cells to refine.

Coarsening Criterion

specifies the cell register or expression that Fluent will use during the calculation to determine which cells to coarsen.

Prismatic Adaption

specifies that prismatic cells are adapted anisotropically. This option is only available with the PUMA adaption method.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Predefined Criteria

allows you to select from a list of commonly used criteria for adapting the mesh (for a description of the list, see Predefined Criteria for Adaption (p. 3784)); if one is selected, Fluent will automatically generate cell registers and/or expressions for refinement and coarsening, and define the settings in the **Automatic Mesh Adaption** dialog box and General Adaption Controls Dialog Box (p. 5486) as needed. Depending on your selection, you may be prompted to complete the setup by defining fields in the Adaption Criteria Settings Dialog Box (p. 5386).

Cell Registers

allows you to create new cell registers and manage existing ones, which may then be selected from the **Refinement Criterion** and/or **Coarsening Criterion** drop-down lists.

List Criteria

prints to the console the number of cells marked for refinement, the number of cells marked for coarsening, and the number marked for both operations.

Display Options

opens the Display Options - Adaption Dialog Box (p. 5435).

General Adaption Controls

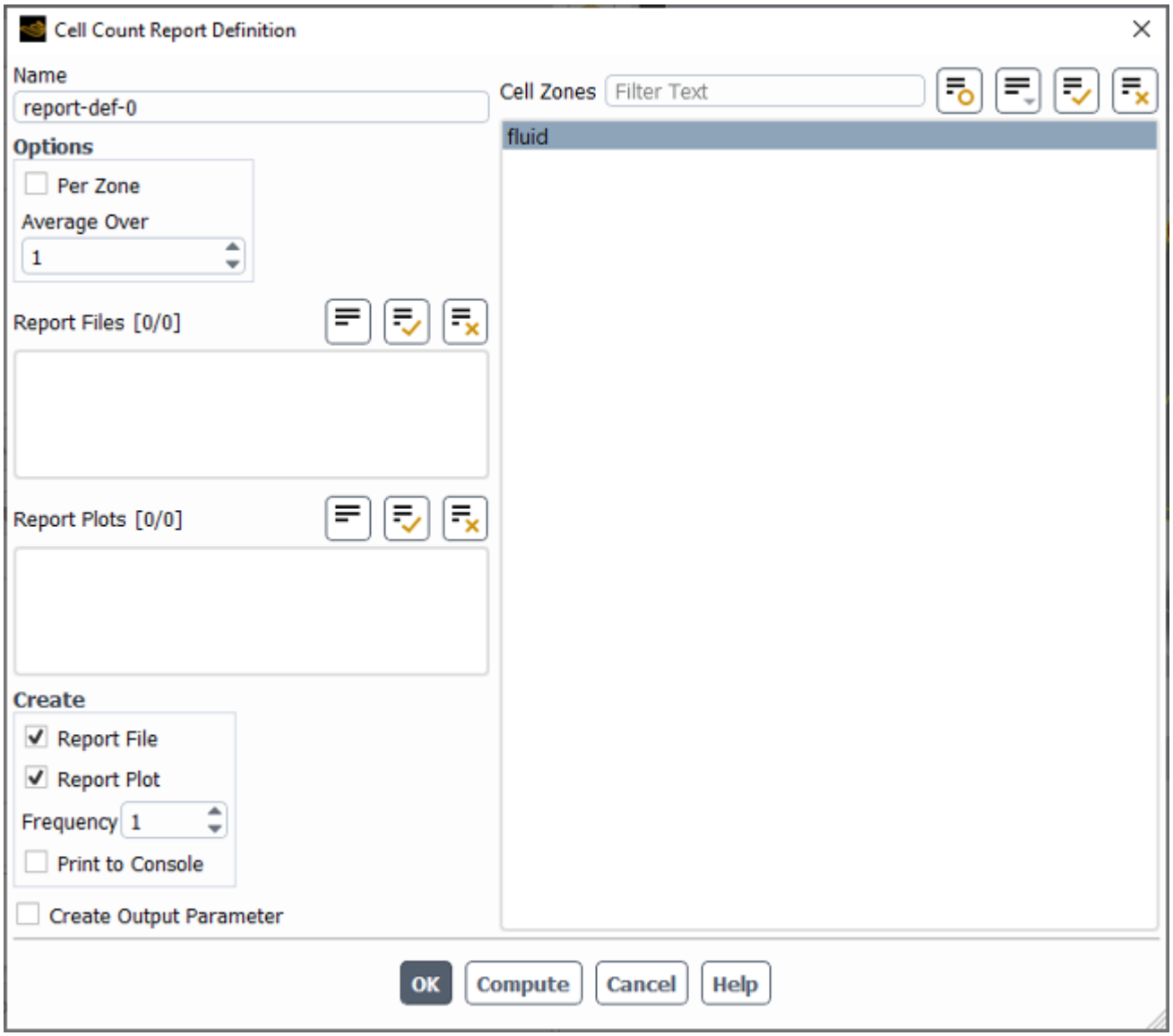
opens the General Adaption Controls Dialog Box (p. 5486), where you can specify the general settings that Fluent uses when adapting the mesh.

52.2.12. Cell Count Report Definition Dialog Box

The **Cell Count Report Definition** dialog box allows you to create a report definition for the reporting of a cell count (that is, the number of cells within one or more cell zones), as described in Mesh Report Definitions (p. 4114).

Solution → **Reports** → **Definitions** → **New** → **Mesh Report** → **Cell Count...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name

specifies the name of the report definition.

Options

Per Surface

enables the reporting of the cell count individually for each of the selected zones (rather than reporting the sum from all of the selected zones).

Average Over

specifies the number of iterations / time steps over which the cell count is averaged when written, plotted, and/or printed. The default value of 1 means that no averaging is performed (that is, only the current cell count is reported), and higher values yields a running average.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When the iteration / time step number is lower than the specified value, Fluent calculates the average of the available cell count values.

Retain Instantaneous Values

enables the retention of instantaneous values in any report file definition or report plot definition that is associated with this report definition. This option is only available if **Average Over** is set to a value greater than 1.

Report Files

allows you to select existing report file definitions to which you want to write cell count data from this report definition.

Report Plots

allows you to select existing report plot definitions to which you want to include cell count data from this report definition. Note that the units used in these report plots may not be suitable for the cell count data.

Create

Report File

creates a new report file definition for the data from this report definition.

Report Plot

creates a new report plot definition for the data from this report definition.

Frequency

specifies how frequently the report definition is written, plotted, and/or printed to the console.

Print to Console

enables the printing of the cell count data from this report definition to the console during the calculation.

Create Output Parameter

creates an output parameter for this report definition with the name <*report_definition_name*>-op.

Cell Zones

allows you to select the cell zones for which you want to report the number of cells.

OK

creates / saves the report definition.

Compute

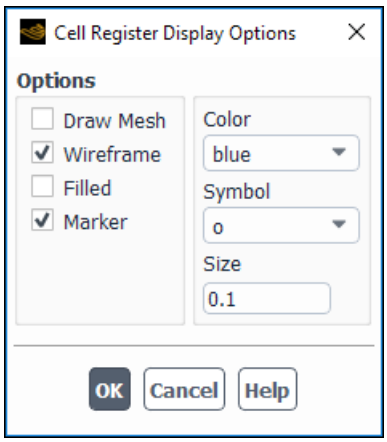
computes the cell count within the selected zone(s) and prints to the console.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

52.2.13. Cell Register Display Options Dialog Box

The **Cell Register Display Options** dialog box allows you to customize the display of cell registers.

You can access this dialog box by clicking **Display Options...** in any of the cell register dialog boxes.



Controls

Options

contains check buttons that control the drawing of the mesh and the type of graphical tool used to display register cells.

Draw Mesh

toggles the ability to draw the mesh with the cell register display. This command opens the Mesh Display Dialog Box (p. 4688), which allows you to select the desired surface or zone meshes to be displayed with the register cells.

Wireframe

toggles the display of the cell wireframe.

Filled

toggles the solid shading of the cell wireframe.

Marker

toggles the display of the cell marker.

Color

is a drop-down list of colors for the wireframe or marker for the cells.

Symbol

is a drop-down list of symbols that can be used cell marker.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

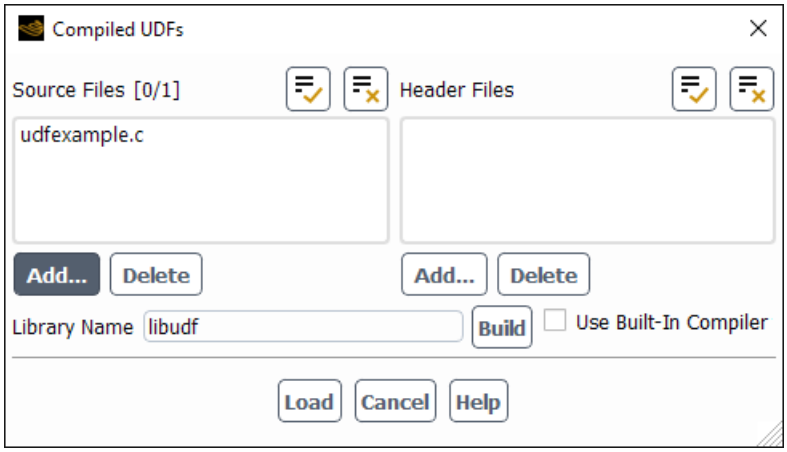
Size

is a real number entry for the size of the cell marker. A symbol of size 1.0 is 3.0% of the height of the display screen.

52.2.14. Compiled UDFs Dialog Box

The **Compiled UDFs** dialog box allows you to open a library of compiled user-defined functions. See the separate Fluent Customization Manual for details.

User Defined → User Defined → Functions → Compiled...



Controls

Source Files

contains a list of source files.

Header Files

contains a list of header files.

Add...

opens the **Select File** dialog box.

Delete

deletes the selected file from the list.

Library Name

specifies the name of the library to be created.

Use Built-In Compiler

enables the use of a built-in compiler (Clang) when the **Build** button is clicked. This option / compiler is available for Windows only, and is provided as part of the Fluent installation. It is recommended that you enable this option when the compiler you installed on your machine is

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

older and no longer supported. Note that the built-in compiler is used automatically if Fluent determines that you have not installed Microsoft Visual Studio or Clang on your computer, whether this option is enabled or not.

Build

builds the library and compiles the user-defined function.

Load

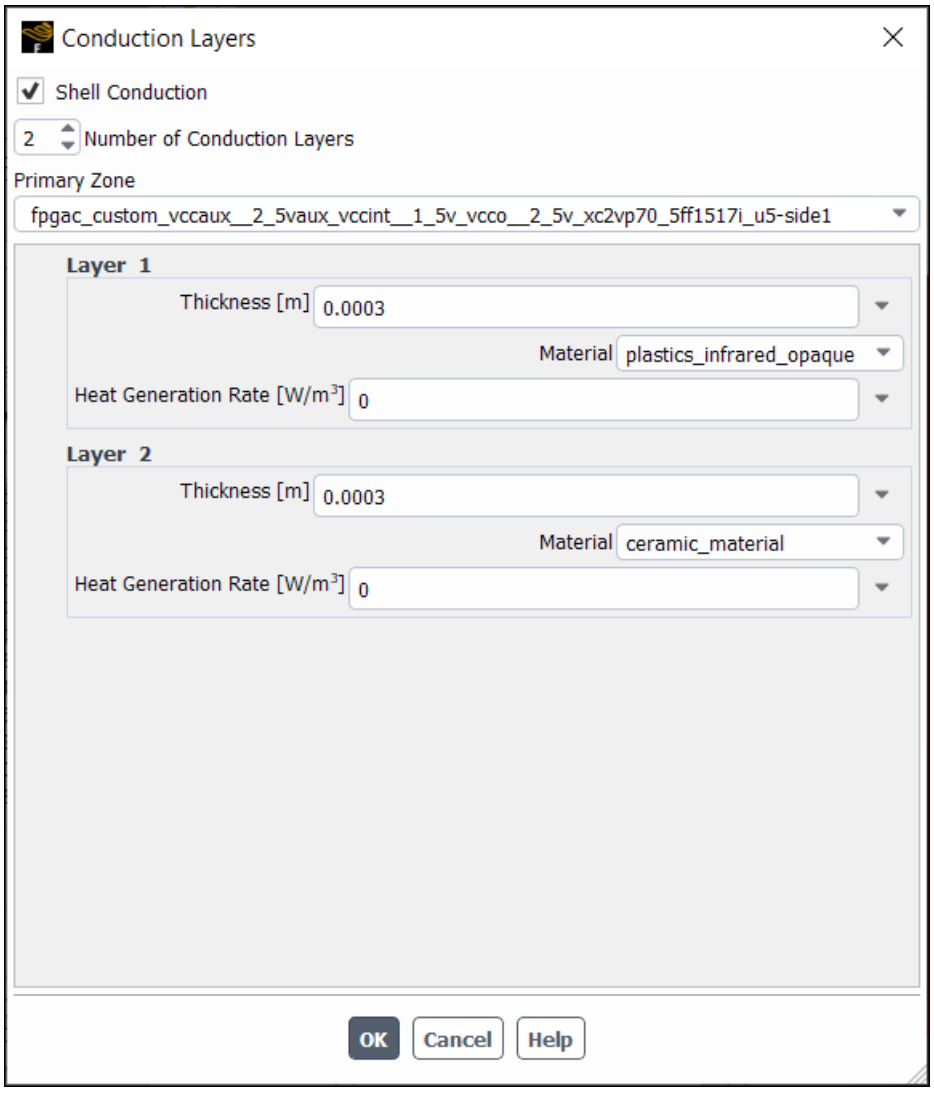
opens the specified library and loads the user-defined function.

52.2.15. Conduction Layers Dialog Box

The **Conduction Layers** dialog box allows you to define the conduction settings for either a single **Wall** dialog box or all of the walls selected in the **Wall Zones** list of the Conduction Manager Dialog Box (p. 5411). See Shell Conduction (p. 1475) and Managing Conduction Walls (p. 2184) for details about the items below.

The **Conduction Layers** dialog box is opened from either the Wall Dialog Box (p. 5085) or the Conduction Manager Dialog Box (p. 5411).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Shell Conduction

enables/disables shell conduction for the selected wall(s).

Number of Shell Conduction Layers

allows you to define the number of layers that make up the wall(s).

Primary Zone

sets a wall to copy conduction settings from to the other selected walls.

Layer 1, Layer 2...

allows you to define the settings for each layer. Note that **Layer 1** refers to the layer closest to the fluid / solid adjacent to the wall zone, and layers with higher numbers are further away.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Thickness

sets the thickness of the layer for the calculation of the thermal resistance. Each layer must have a nonzero thickness. Note that non-shell walls with a specified **Thickness** will be treated as thin walls.

Material Name

sets the material type for the layer/wall. The conductivity of the material is used for the calculation of thermal resistance. Materials are defined using the Materials Task Page (p. 4883).

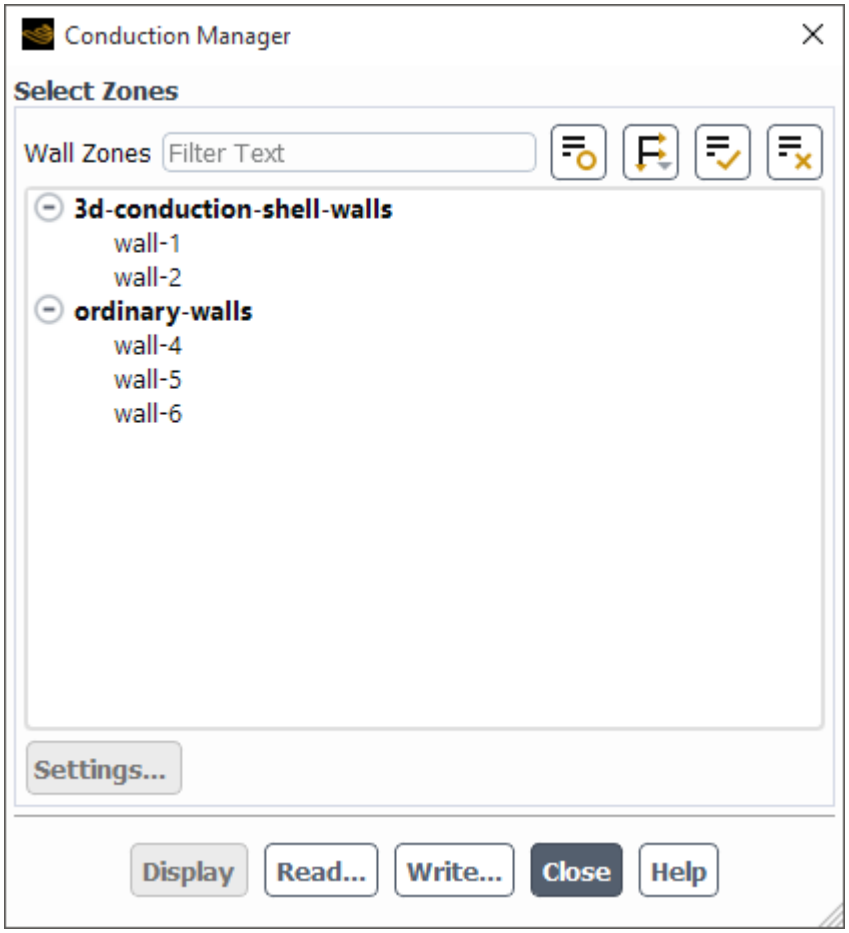
Heat Generation Rate

sets the rate of heat generation in the layer. Note that you have the option of defining the heat generation rate using a user-defined function (UDF) that utilizes the `DEFINE_PROFILE` macro; for more information on creating and using user-defined functions, see the Fluent Customization Manual.

52.2.16. Conduction Manager Dialog Box

The **Conduction Manager** dialog box allows you to manage, define, and display conduction zones all in one location.

Physics → **Model Specific** → **Conduction Manager...**



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Controls

Select Zones

contains the list of conduction zones (1d thin walls and 3d shell walls) and ordinary walls.

Wall Zones

contains a list of wall zones.

Settings...

opens the Conduction Layers Dialog Box (p. 5409), where you can define the conduction properties of the walls selected in the **Wall Zones** list.

Display

displays the selected walls in the graphics window.

Read...

allows you to define your conduction settings by reading a `.csv` file.

Write...

allows you to write your saved conduction settings to a `.csv` file.

52.2.17. Contours Dialog Box

The **Contours** dialog box controls the display of contour and profile plots. See Displaying Contours and Profiles (p. 3896) for details about the items below.

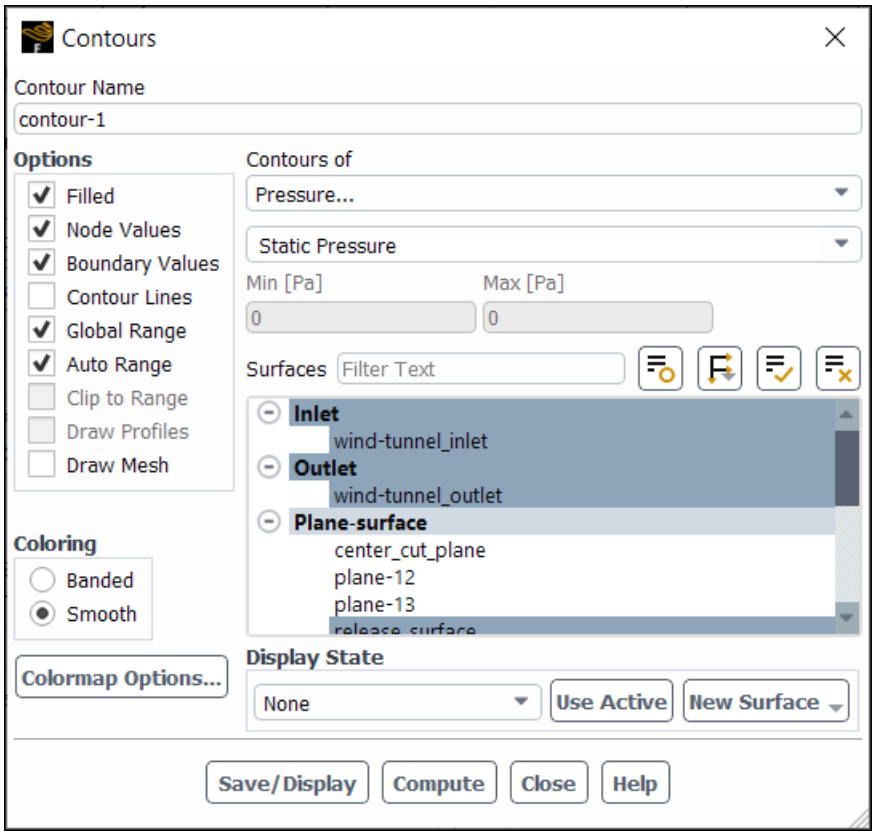
Results → **Graphics** → **Contours** → **New...**

Note:

- The "global"/non-persistent form of graphics objects is hidden by default. If this interrupts your workflow, you can make them available by enabling the **Expose legacy non-persistent graphics** option in the **Graphics** branch of **Preferences** (accessed via **File**>**Preferences...**).
- (When you click **Edit...** instead of **New...**, it opens the "global"/non-persistent version of the **Contours** dialog box. This version of the dialog box does not have a **Name** field and it is not saved with the case file.

  You are encouraged to use the persistent version that is accessed by clicking **New...**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Contour Name

is the name for a contour plot definition. You can specify a name or use the default name **`contour-id`** . This control appears only for contour plot definitions.

Options

contains the check buttons that set various contour display options.

Filled

toggles between filled contours and line contours.

Node Values

toggles between using scalar field values at nodes and at cell centers for computing the contours. When the **Filled** option is off, **Node Values** is always on. See Choosing Node or Cell Values and Node or Boundary Values (p. 3905) and Node Values (p. 4168) for details.

Boundary Values

enabling overwrites the node values (on boundaries) with a simple average of the boundary face values. Refer to Choosing Node or Cell Values and Node or Boundary Values (p. 3905) for more information.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Contour Lines

combines filled contours with line contours. This option is only available when both the **Filled** and **Node Values** options are enabled.

Global Range

toggles between basing the minimum and maximum values on the range of values on the selected surfaces (off), and basing them on the range of values in the entire domain (on, the default).

Auto Range

toggles between automatic and manual setting of the contour range. Any time you change the **Contours of** selection, **Auto Range** is reset to on.

Clip to Range

determines whether or not values outside the prescribed **Min**/ **Max** range are contoured when using **Filled** contours. If selected, values outside the range will not be contoured. If not selected, values below the **Min** value will be colored with the lowest color on the color scale, and values above the **Max** value will be colored with the highest color on the color scale. See Specifying the Range of Magnitudes Displayed (p. 3902) for details.

Draw Profiles

causes the addition of a profile plot to the contour plot. The Profile Options Dialog Box (p. 5238) is opened when **Draw Profiles** is selected.

Note:

Draw Profiles is only available for the non-persistent/"global" version of the **Contours** dialog box that does not have a **Name** field. This version of the dialog box is opened by:

- Right-clicking **Contours** in the Outline View tree and selecting **Edit...** (located under the **Results** branch).
- Clicking **Contours** and selecting **Edit...** under **Graphics** in the **Results** ribbon tab.

Draw Mesh

toggles between displaying and not displaying the mesh. The Mesh Display Dialog Box (p. 4688) is opened when **Draw Mesh** is selected.

Coloring

specifies how the contours appear.

Banded

the contour coloring features distinct color bands corresponding to the colormap.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Smooth

the contours feature a smooth transition between colors.

Colormap Options...

opens the Colormap Dialog Box (p. 5280), allowing you to customize the colormap for this graphics object.

Contours of

contains a list from which you can select the scalar field to be contoured.

Min

shows the minimum value of the scalar field. If **Auto Range** is off, you can set the minimum by typing a new value.

Max

shows the maximum value of the scalar field. If **Auto Range** is off, you can set the maximum by typing a new value.

Surfaces

contains a list from which you can select the surfaces on which to draw contours. For 2D cases, if no surface is selected, contouring is done on the entire domain. For 3D cases, you must always select at least one surface.

Display State

specifies the display state that is associated with this graphics object.

drop-down list of display states

selectable list of all the existing display states associated with this case.

Use Active

creates a display state that matches the settings of the active graphics window and assigns it to this graphics object (this association is only saved after you click **Save/Display**).

New Surface

is a drop-down list button that contains a list of surface options:

Point

opens the Point Surface Dialog Box (p. 5582).

Line/Rake

opens the Line/Rake Surface Dialog Box (p. 5514).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Plane

opens the Plane Surface Dialog Box (p. 5580).

Quadric

opens the Quadric Surface Dialog Box (p. 5584).

Iso-Surface

opens the Iso-Surface Dialog Box (p. 5501).

Iso-Clip

opens the Iso-Clip Dialog Box (p. 5500).

Structural Point

opens the Structural Point Surface Dialog Box (p. 5623).

Display

draws the contours in the active graphics window.

Compute

calculates the scalar field and updates the **Min** and **Max** values (even when **Auto Range** is off).

Save/Display

plots the contour in the active graphics window and saves the contour plot definition. This button appears only for contour plot definitions and replaces the **Display** button.

52.2.18. Convergence Conditions Dialog Box

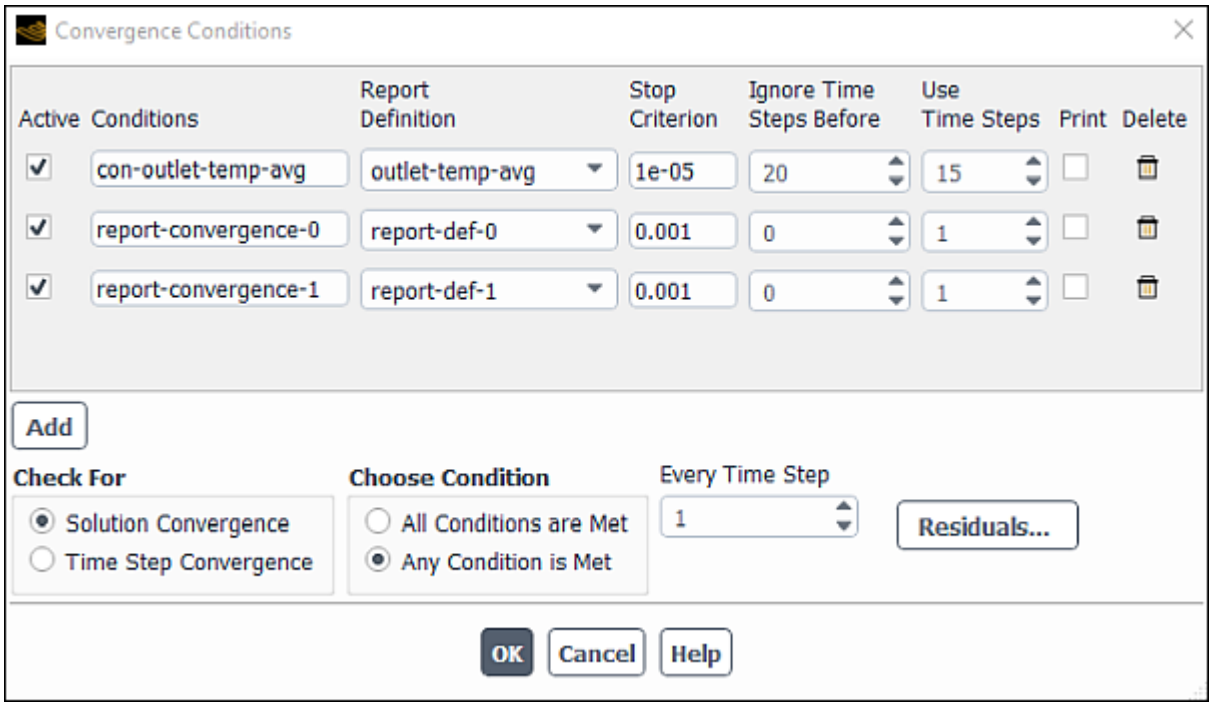
The convergence conditions facility allows you to set convergence conditions on the solution based on the values from report definitions (surface, volume, lift, drag, and so on). See Convergence Conditions (p. 3696) for details on setting up the **Convergence Conditions** dialog box.

Note:

If you are solving a transient case, the **Convergence Conditions** dialog box will relabel some fields since the transient case uses time-steps rather than iterations. These alternate labels are indicated below.

Solution → Reports → Conditions...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Active

check box for deactivating/activating individual convergence conditions.

Conditions

name of the convergence condition.

Report Definition

name of the report definition used in judging convergence.

Stop Criterion

specifies the criterion below which the solution is considered to be converged.

Ignore Iterations Before | Ignore Time Steps Before

ignores the first few iterations/time-steps if you expect the solution to fluctuate initially.

Use Iterations | Use Time Steps

specifies the number of previous iterations/time-steps to be included in the report definition convergence check.

Check For

specify whether Fluent checks for convergence at every time step or every iteration.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Solution Convergence

Fluent checks for solution convergence at every time step.

Time Step Convergence

Fluent checks for solution convergence at every iteration.

Choose Condition

to select the convergence conditions.

All Conditions are Met

The solution is considered to be converged if all of the convergence conditions' criteria are satisfied, including those in the Residual Monitors Dialog Box (p. 5599).

Any Condition is Met

The solution is considered to be converged if any of the convergence conditions' criteria is satisfied, including those in the Residual Monitors Dialog Box (p. 5599).

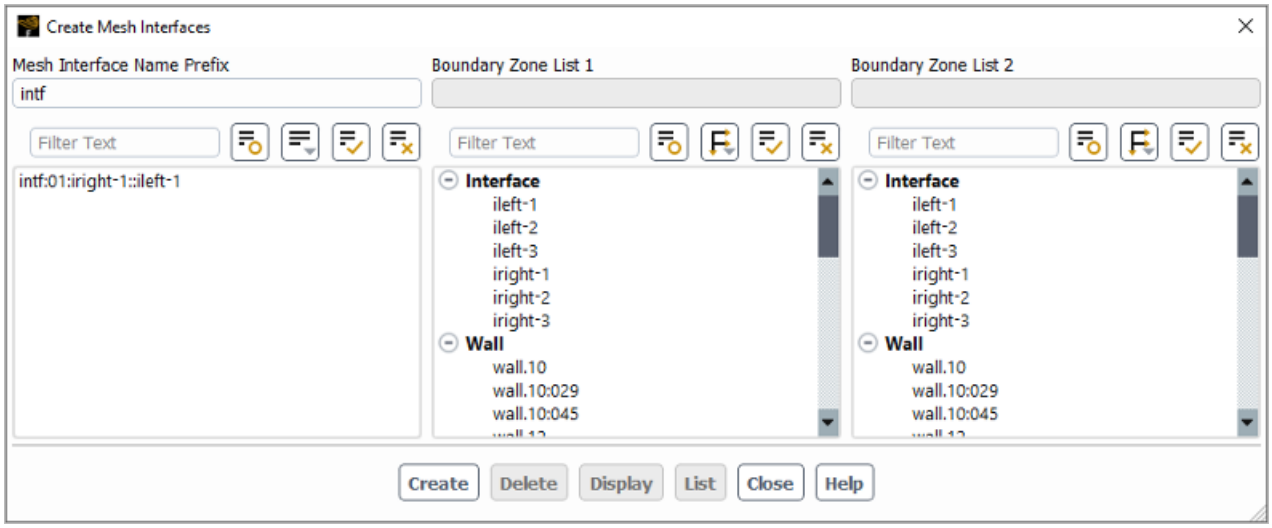
Every Iteration | Every Time-Step

to select how often convergence checks are done.

52.2.19. Create Mesh Interfaces

The **Create Mesh Interfaces** dialog box allows you to easily create many mesh interfaces between boundary zones that do not currently overlap (for example, for a sliding mesh simulation), as described in Manually Creating Many Non-Overlapping Mesh Interfaces (p. 1204). Note that if you only want to create mesh interfaces between boundary zones that currently overlap, you should instead use the Mesh Interfaces Dialog Box (p. 5525). For details, see Using a Non-Conformal Mesh in Ansys Fluent (p. 1185).

The **Create Mesh Interfaces** dialog box is opened by ensuring that the default one-to-one interface creation method is enabled (using the `define/mesh-interfaces/one-to-one-pairing?` text command) and clicking the **Manual Create...** button in the Mesh Interfaces Dialog Box (p. 5525).



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Controls

Mesh Interface Name Prefix

contains a text entry box in which you can specify the prefix used in the names of all the mesh interfaces created when you click the **Create** button, and a list from which you can select existing mesh interfaces.

Boundary Zone List 1, Boundary Zone List 2

allows you to select the zones that you want to pair up to make mesh interfaces. Note that mesh interfaces will be created for every possible combination between these two lists, regardless of whether the zones in question currently overlap. (You cannot edit the top fields; the names in these fields will be the name of the last zone you selected in the list below it.)

Create

creates mesh interfaces for every possible combination between the zones selected under **Boundary Zone List 1** and **Boundary Zone List 2**, regardless of whether the zones in question currently overlap.

Delete

deletes the mesh interfaces selected from the list under **Mesh Interface Name Prefix**.

Display

allows you to display selected interface zones or mesh interfaces in the graphics window. Note that after you have selected a mesh interface, you can modify what zones are displayed by selecting / deselecting zones under **Boundary Zone List 1** and/or **Boundary Zone List 2**.

List

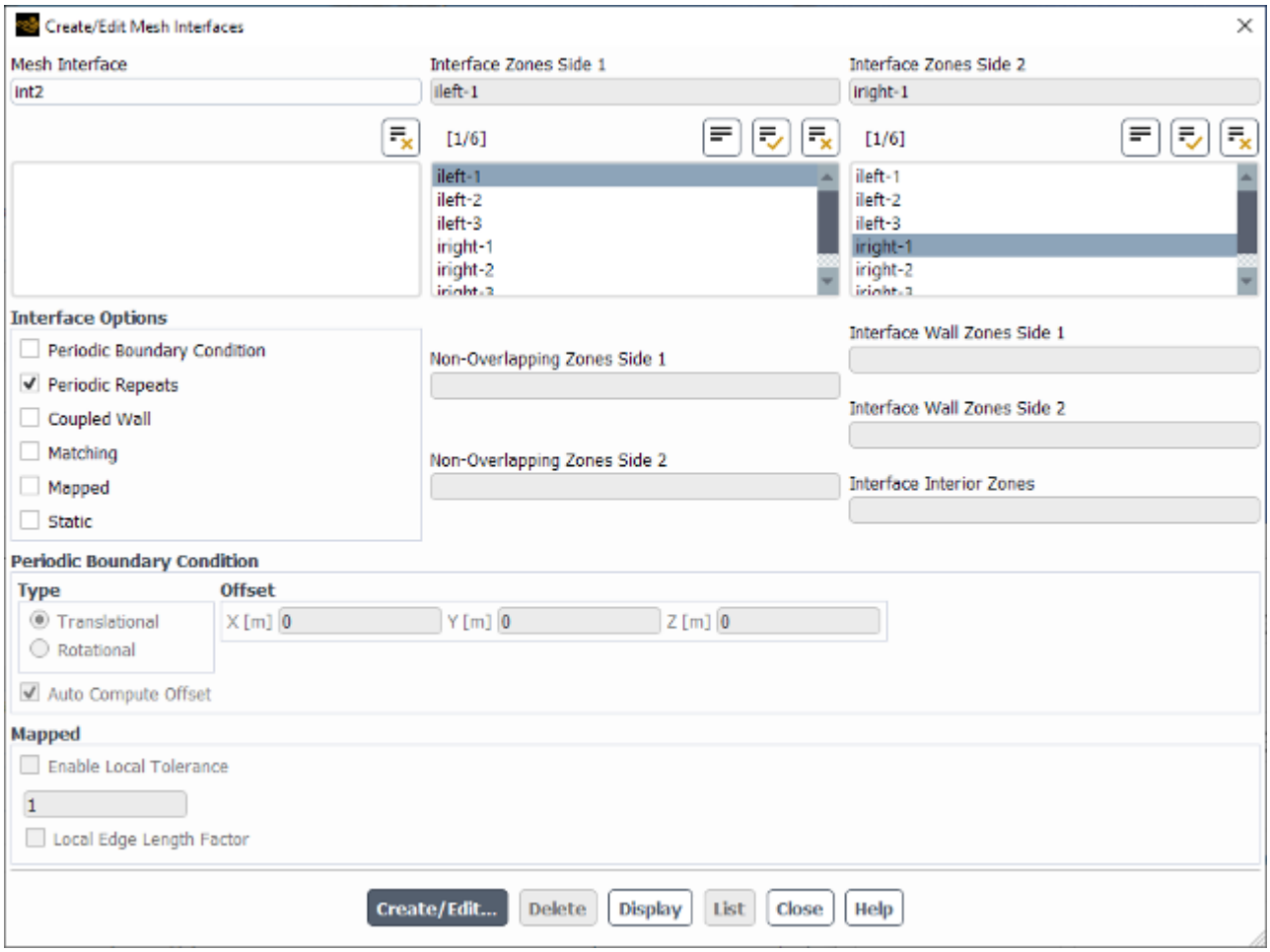
prints information about the mesh interfaces selected in the list under **Mesh Interface Name Prefix** in the console.

52.2.20. Create/Edit Mesh Interfaces Dialog Box

The **Create/Edit Mesh Interfaces** dialog box allows you to manually create mesh interfaces for use with sliding meshes (see Using Sliding Meshes (p. 1804)) or multiple reference frames (see Mesh Setup for a Multiple Moving Reference Frame (p. 1781)), or for meshes with non-conformal boundaries (see Non-Conformal Meshes (p. 1170)). This dialog box requires you to decide which interface zones make up both sides of each mesh interface. While it is possible to create every type of many-to-many mesh interface using this dialog box, it is only necessary when you want your interface to use the periodic or periodic repeats option; for all other types it is more convenient to automatically create interfaces using the Mesh Interfaces Dialog Box (p. 5525), especially when you have many interface zones and/or are unfamiliar with their names / locations. For details, see Using a Non-Conformal Mesh in Ansys Fluent (p. 1185).

The **Create/Edit Mesh Interfaces** dialog box is opened by disabling the default one-to-one interface creation method (using the `define/mesh-interfaces/one-to-one-pairing?` text command) and clicking the **Manual Create...** button in the Mesh Interfaces Dialog Box (p. 5525).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Mesh Interface

contains a text entry box in which you can set the name of the mesh interface, and a list from which you can select an existing mesh interface.

Interface Zones Side 1, Interface Zones Side 2

contain selectable lists for the interface zones that make up the mesh interface and informational fields that show the names of the zones you selected in each list. (You cannot edit the top fields; the names in these fields will be the names of the zones you selected in the list below it.)

Interface Options

contains options related to the interface type.

Periodic Boundary Condition

allows you to create a non-conformal periodic boundary condition interface. Note that the **Matching** option is enabled by default with such periodic mesh interfaces.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Periodic Repeats

should be enabled when each of the two cell zones has a single pair of conformal or non-conformal periodics adjacent to the interface. This option is typically used in conjunction with the sliding mesh model, when simulating the interface between a rotor and stator; it allows Ansys Fluent to treat the interface between the sliding and non-sliding zones as periodic where the two zones do not overlap. For details, see The Periodic Repeats Option (p. 1174).

Coupled Wall

allows you to specify that the interface acts as a thermally coupled wall.

Matching

is relevant if only interface internal zones should be created (that is, interior or wall / shadow pairs), since the interface zones on both sides are aligned. With the **Matching** option, even interface zones that are not perfectly aligned are treated as if they are; however, if the discrepancy between the interface zones on both sides exceeds default thresholds, then warning messages will be displayed. Note that the **Matching** option is compatible with the **Periodic Boundary Condition**, **Coupled Wall**, and **Static** options. For more information about the recommended uses of this option, see Matching Option (p. 1177).

Mapped

enables an alternative approach for modeling coupled walls between zones. This approach is more robust than the standard non-conformal interface formulations when the interface zones penetrate each other or have gaps between them. It requires that at least one side of the interface consists of only solid zones.

Note:

The **Mapped** option is not compatible with shell conduction.

Static

reduces the memory usage and processing time (for interface creation and solution), especially when there are many zones on both sides of the interface. This option will only produce correct results if the interface zones do not move or deform relative to each other at the interface, and it is not compatible with the **Periodic Boundary Condition**, **Periodic Repeats**, or **Mapped** options.

Non-Overlapping Zones Side 1, Non-Overlapping Zones Side 2

display the names of any wall boundary zones created by Ansys Fluent during the process of creating the selected mesh interface. If the two interface zones overlap each other completely, then the wall boundaries are created but with zero faces.

Interface Wall Zones Side 1, Interface Wall Zones Side 2

display the names of any wall interface zones created by Ansys Fluent during the process of creating the selected mesh interface.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Interface Interior Zones

displays the names of any interface interior zones created by Ansys Fluent during the process of creating the selected mesh interface. This is used for embedded LES, when there is a need to be able to convert an interior zone into a RANS-LES interface.

Periodic Boundary Condition

Type

allows you to select a periodicity that is either **Translational** or **Rotational**.

Offset

is the offset coordinates or angle, depending on whether **Translational** or **Rotational** periodicity is selected. Note that when **Auto Compute Offset** is enabled, the **Offset** fields are not editable.

Auto Compute Offset

will result in Ansys Fluent finding the offset. If this option is disabled, then you will have to provide the offset coordinates or angle in the required fields, depending on whether **Translational** or **Rotational** periodicity is selected.

Mapped

Enable Local Tolerance

(when enabled) allows you to overwrite any value specified in the **Tolerance** group box in the Mapped Interface Options Dialog Box (p. 5520), and is used on the selected interface only.

Local Edge Length Factor

(when enabled) allows you to specify the multiplier that Fluent uses with the smallest edge length of the interface zones to calculate the tolerance for mapping. Disabling **Local Edge Length Factor** allows you to specify an absolute value for the tolerance.

Create/Edit...

performs one of the following:

- If you have entered a new **Mesh Interface** name and selected unassigned zones from the **Interface Zones Side 1** and **Interface Zones Side 2** selection lists, a new mesh interface is created.
- If you have made a selection from the **Mesh Interface** list, the Edit Mesh Interfaces Dialog Box (p. 5454) will open so that you can revise the settings of existing mesh interfaces.

Delete

deletes the mesh interface selected under **Mesh Interface**.

Display

allows you to display interface zones or mesh interfaces in the graphics window. Note that you can only select and display an interface zone from **Interface Zones Side 1** or **Interface Zones**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

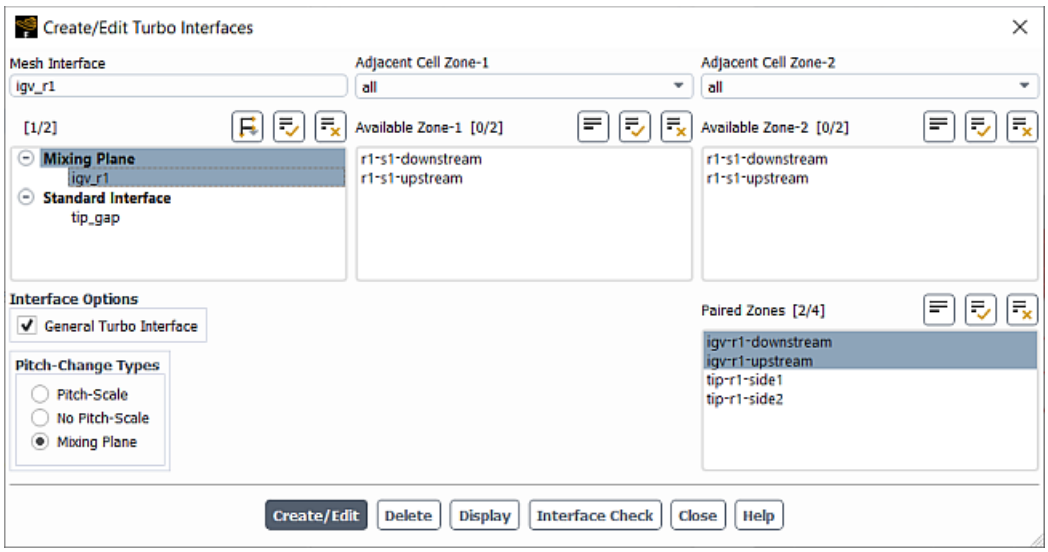
Side 2 if it is not yet assigned to an existing mesh interface. After a **Mesh Interface** is created, you can select the appropriate mesh interface and click the **Display** button to display all of the zones under **Interface Zones Side 1** and **Interface Zones Side 2**.

List

prints information about the selected **Mesh Interface** in the console. When you click this button, Ansys Fluent will list the two interface boundaries and (if you have initialized the solution) all new zones that were created (that is, wall and/or interior zones).

52.2.21. Create/Edit Turbo Interfaces Dialog Box

The **Create/Edit Turbo Interfaces** dialog box allows you to manually create many-to-many mesh interfaces to connect blades to one another. This dialog box requires you to decide which interface zones make up both sides of each turbo interface and specify the blade row interaction model. For details, see Creating and Editing General Turbo Interfaces (p. 1962).



Controls

Mesh Interface

contains a text entry box in which you can set the name of the turbo interface, and a list from which you can select an existing mesh interface.

Interface Zones Side 1, Interface Zones Side 2

contain selectable lists for the interface zones that make up the turbo interface and informational fields that show the names of the zones you selected in each list. (You cannot edit the top fields; the names in these fields will be the names of the zones you selected in the list below it.)

Interface Options

contains options related to the interface type.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Periodic Boundary Condition

allows you to create a non-conformal periodic boundary condition interface. Note that the **Matching** option is enabled by default with such periodic mesh interfaces; for details about the **Matching** option, see Matching Option (p. 1177).

General Turbo Interface

(only available when the Turbo Model is enabled) connects two turbo zones. This option is typically used in conjunction with the sliding mesh model, when simulating the interface between a rotor and stator. These models permit for a pitch-change between the modeled blade rows.

Pitch-Change Types

Pitch-Scale

stretches or compresses the flow profiles between the cell zones to maintain the interaction between blade row passages. See Blade Row Interaction Modeling (p. 1956) for details.

No Pitch-Scale

maintains the blade row interaction between blade row passages by creating virtual copies of the smaller pitch blade passage. See Blade Row Interaction Modeling (p. 1956) for details.

Mixing Plane

uses a pitchwise averaging procedure to mix out the flow variation between the blade rows. See Blade Row Interaction Modeling (p. 1956) for details.

Create/Edit...

performs one of the following:

- If you have entered a new **Mesh Interface** name and selected unassigned zones from the **Interface Zones Side 1** and **Interface Zones Side 2** selection lists, a new mesh interface is created.
- If you have made a selection from the **Mesh Interface** list, the Edit Mesh Interfaces Dialog Box (p. 5454) will open so that you can revise the settings of existing mesh interfaces.

Delete

deletes the mesh interface selected under **Mesh Interface**.

Display

allows you to display interface zones or mesh interfaces in the graphics window. Note that you can only select and display an interface zone from **Interface Zones Side 1** or **Interface Zones Side 2** if it is not yet assigned to an existing mesh interface. After a **Mesh Interface** is created, you can select the appropriate mesh interface and click the **Display** button to display all of the zones under **Interface Zones Side 1** and **Interface Zones Side 2**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

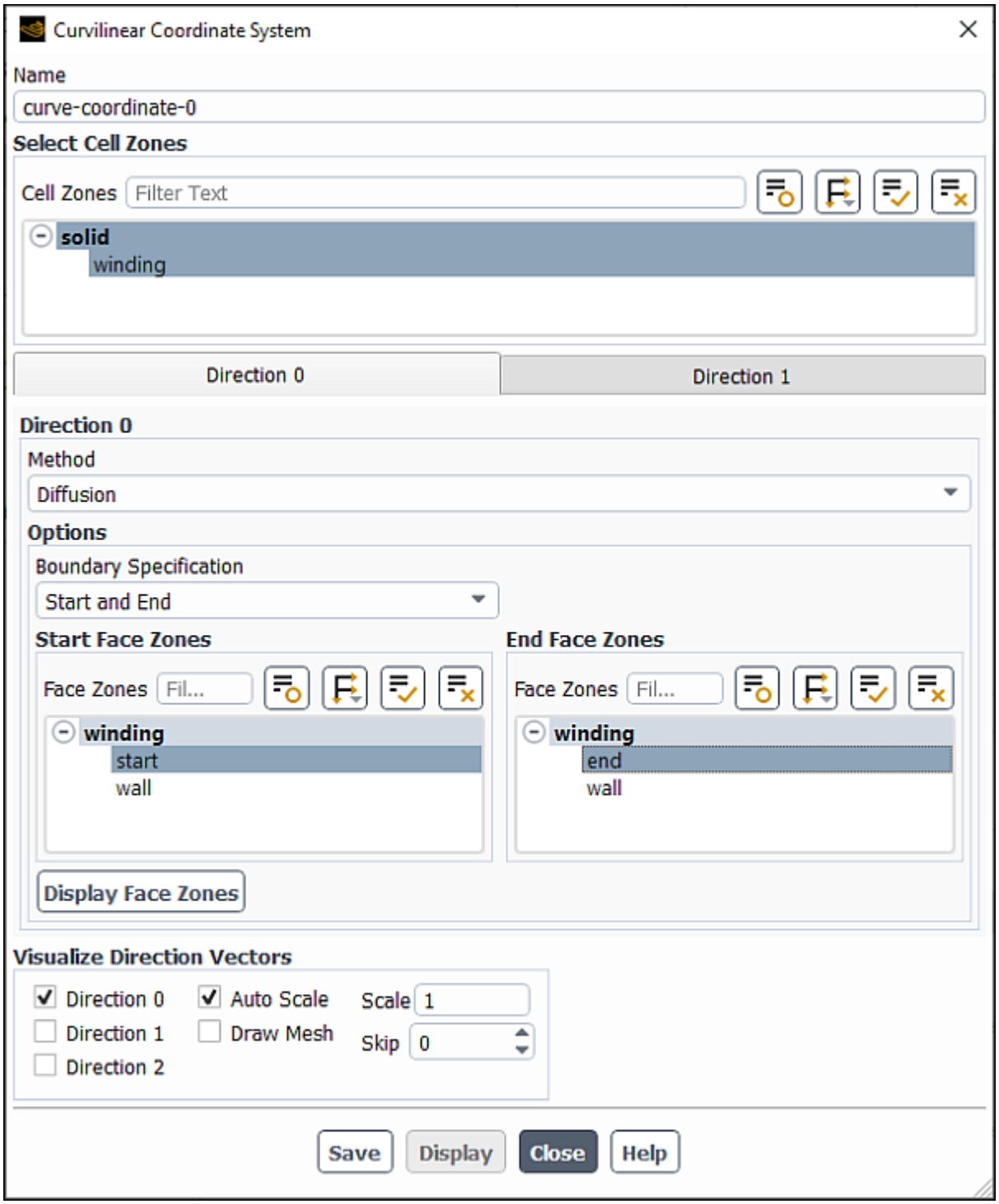
List

prints information about the selected **Mesh Interface** in the console. When you click this button, Ansys Fluent will list the two interface boundaries and (if you have initialized the solution) all new zones that were created (that is, wall and/or interior zones).

52.2.22. Curvilinear Coordinate System Dialog Box

The **Curvilinear Coordinate System** dialog box allows you to create local coordinate systems that follow the geometry of the model. Refer to Curvilinear Coordinate Systems (p. 1164) for additional information.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name

specifies the name of the curvilinear coordinate system.

Cell Zones

specifies the cell zone whose geometry will be used to create the curvilinear coordinate system.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Direction 0 (1)

tabs to specify Direction 0 and 1 of the curvilinear coordinate system. Note that Direction 2 is calculated from the cross product of Direction 0 and 1.

Method

There are 2 options for specifying a direction.

- **Diffusion** - utilizes geometry to specify direction as determined by **Boundary Specification**.

  Start and End

  specifies the direction using a **Start Face Zone** and an **End Face Zone**.

  Jump

  for closed loop geometries, specifies the direction from **Interior Zones** (utilizing a coupled wall as a cross section, which is then split into start and end faces).

- **Base Vector** - specifies the direction using an **X**, **Y**, **Z** vector

Display Face Zones

displays currently selected face zones.

Visualize Direction Vectors

options to visualize the curvilinear coordinate system.

Direction 0

selects Direction 0 to display in the graphics viewer.

Direction 1

selects Direction 1 to display in the graphics viewer.

Direction 2

selects Direction 2 to display in the graphics viewer.

Auto Scale

scales the vectors to an appropriate size.

Draw Mesh

opens the Mesh Display dialog box.

Scale

specifies the size of the vectors for display.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Skip

Thins the number of vectors displayed to improve visibility.

Save

Saves the curvilinear coordinate system.

Display

Displays the curvilinear coordinate system according the settings under **Visualize Direction Vectors**.

52.2.23. Custom Field Function Calculator Dialog Box

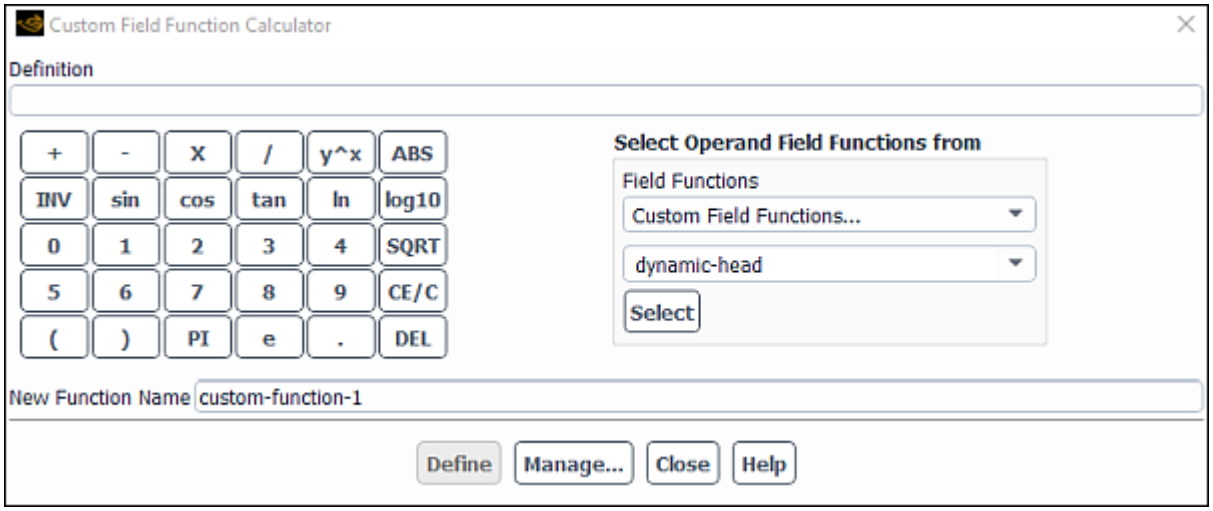
The **Custom Field Function Calculator** dialog box allows you to define custom field functions based on existing functions, using simple calculator operators. Any functions that you define will be added to the list of default flow variables and other field functions provided by Ansys Fluent.

Important:

Recall that you must enter all constants in the function definition in SI units.

See Creating a Custom Field Function (p. 4286) for details about the items below.

User Defined → Field Functions → Custom...



Controls

Definition

displays the function that you are currently defining. As you select each item from the **Field Functions** list or the calculator keypad, it will appear in the **Definition** text entry box. You *cannot* edit the contents of this box directly; if you want to delete part of a function, use the **Delete** button on the keypad.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(Calculator Buttons)

are buttons that perform calculator operations. When you select a calculator button (by clicking on it), the appropriate symbol will appear in the **Definition** text entry box.

Select Operand Field Functions from

contains the available field functions and the means for selecting them.

Field Functions

contains a list from which you can select a variable to be used in the definition of a new function.

Select

enters the variable that is currently selected in the **Field Functions** list in the **Definition** field.

New Function Name

specifies the name of the function you are defining. Should you decide to change the function name after you have defined the function, you can do so in the Field Function Definitions Dialog Box (p. 5476), which you can open by clicking on the **Manage...** button.

Define

creates the function and adds it to the list of **Custom Field Functions** within the drop-down list of available field functions. The **Define** button is grayed out after you create a new function or if the **Definition** field is empty.

Manage...

opens the Field Function Definitions Dialog Box (p. 5476), which enables you to check, rename, save, load, and delete custom field functions.

52.2.24. Custom Laws Dialog Box

The **Custom Laws** dialog box is used to incorporate user-defined functions (see the separate Fluent Customization Manual for details) in place of the default physical laws used in the heat/mass transfer calculations.

The **Custom Laws** dialog box is opened from the Set Injection Properties Dialog Box (p. 5608).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Controls

First Law, Second Law, Third Law,...

contain drop-down lists in which you can choose a user-defined particle law to replace the standard law.

Switching

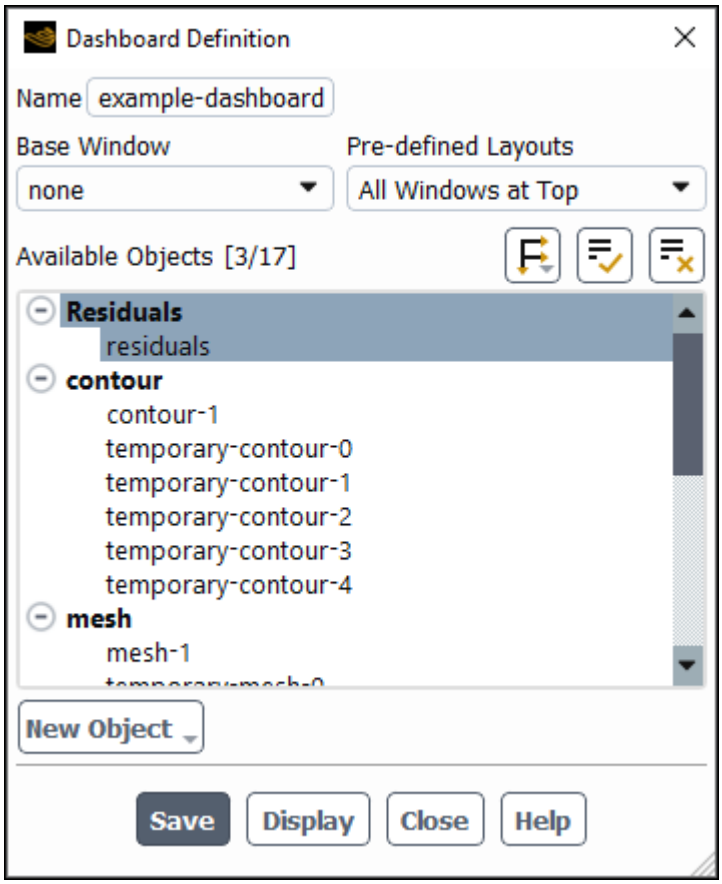
contains a drop-down list in which you can select a user-defined function that customizes the way Ansys Fluent switches between particle laws.

52.2.25. Dashboard Definition Dialog Box

The **Dashboard Definition** dialog box allows you create custom embedded window dashboards that are saved with the case file and added to the Outline View tree. See Geometry-Based Adaption with the Hanging Node Method (p. 3810) for details.

Results → Graphics → Dashboard...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name

is the name for a dashboard definition. You can specify a name or use the default name **dashboard-*id***.

Base Window

specifies which window will be the background, upon which all other graphics and plot objects are embedded. *You can also leave it set to **none**, allowing you to arrange the child windows without any overlap.*

Pre-defined Layouts

controls how the embedded windows are arranged when they are displayed first displayed. You can manually move and resize the windows once they are displayed.

Available Objects

lists the available graphics and plot objects that can be included in the dashboard.

New Object

drop-down list allowing you to create additional graphics objects that can be included in the dashboard.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Save

saves the dashboard to the case file.

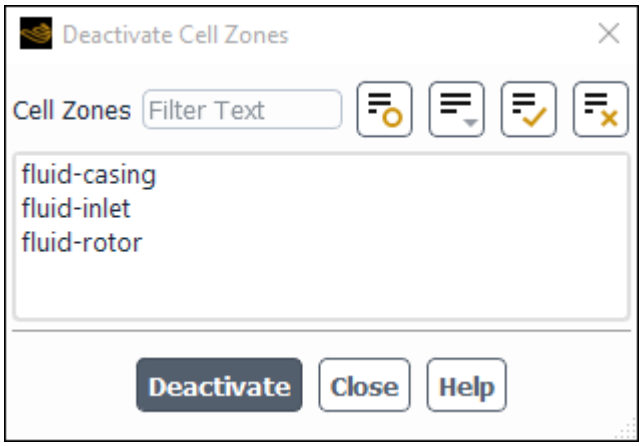
Display

saves the dashboard and displays it in the graphics window.

52.2.26. Deactivate Cell Zones Dialog Box

The **Deactivate Cell Zones** dialog box allows you to deactivate a single cell zone or multiple zones. See Deactivating Zones (p. 1277) for details.

Domain → Zones → Deactivate...



Controls

Cell Zones

contains a list of cell zones from which you can select the zone to be deactivated.

Deactivate

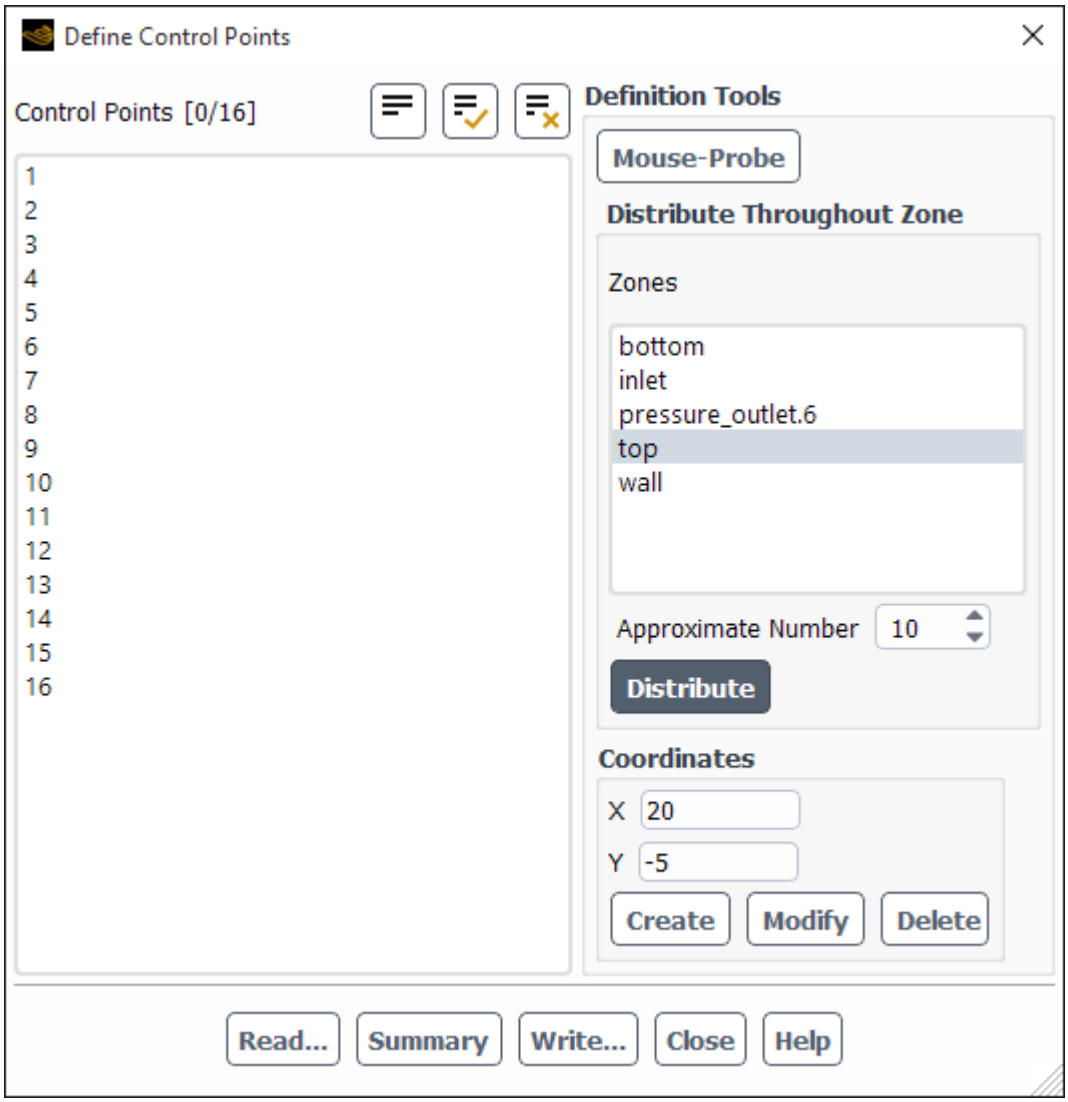
deactivates the selected cell zones.

52.2.27. Define Control Points Dialog Box

The **Define Control Points** dialog box allows you to create, modify, and delete control points for the mesh morpher/optimizer when the unstructured control point distribution is selected. See Using the Mesh Morpher/Optimizer (p. 4620) for details about using this dialog box.

The **Define Control Points** dialog box is opened from the Mesh Morpher/Optimizer Dialog Box (p. 5528).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Control Points

lists the control points you have created and allows you to modify and delete them.

Definition Tools

provides tools for defining the control points.

Mouse-Probe

allows you to create control points by probing (using the right mouse button, by default) in the graphics window.

Distribute Throughout Zone

allows you to quickly create a number of control points and distribute them on mesh nodes throughout a specific boundary zone.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Zones

allows you to select the boundary zone throughout which you want to distribute control points.

Approximate Number

allows you to specify the approximate number of control points you want to distribute throughout a zone. Note that due to the method by which the control points are distributed, the actual number may exceed the value you enter.

Distribute

creates a number of control points on mesh nodes throughout the specified zone (within the bounding box) with a distribution that is based on the distribution of the cell faces in that zone.

Coordinates

allows you to view and modify the coordinates of control points, as well as to create new control points and to delete existing control points.

X, Y, Z

displays the coordinates of the selected **Control Point**, and allows you to define new ones and edit existing ones.

Create

creates a new control point at the currently displayed **X**, **Y**, and **Z** coordinates.

Modify

revises the **X**, **Y**, and **Z** coordinates of the selected **Control Point**.

Delete

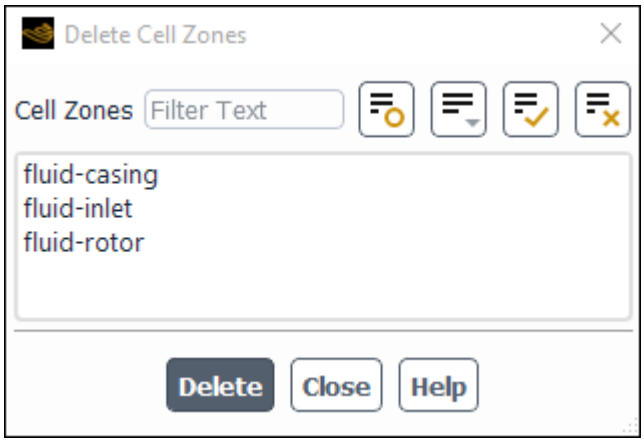
deletes the selected **Control Point**.

52.2.28. Delete Cell Zones Dialog Box

The **Delete Cell Zones** dialog box allows you to delete a single cell zone or multiple zones. See Deleting Zones (p. 1277) for details.

Domain → Zones → Delete...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Cell Zones

contains a list of cell zones from which you can select the zone to be deleted.

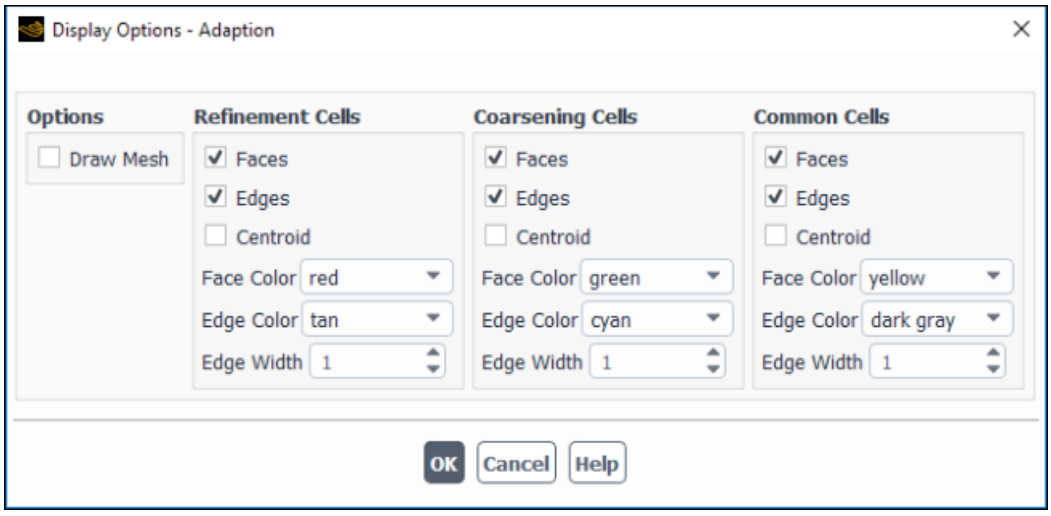
Delete

deletes the selected cell zones.

52.2.29. Display Options - Adaption Dialog Box

The **Display Options - Adaption** dialog box allows you to customize how the cells marked for adaption are displayed.

This dialog box is accessed by clicking **Display Options** in the Manual Mesh Adaption Dialog Box (p. 5515) or Automatic Mesh Adaption Dialog Box (p. 5403).



Controls

Options

contains check boxes that controls the drawing of the mesh.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Draw Mesh

toggles the ability to draw the mesh with the adaption display. This command opens the Mesh Display Dialog Box (p. 4688), which allows you to select the desired surface or zone meshes to be displayed with the markings.

Refinement Cells

contains options related to the display of cells marked for refinement.

Faces

controls whether cell faces are displayed on the cells marked for refinement.

Edges

controls whether cell edges are displayed on the cells marked for refinement.

Centroid

controls whether the cell centroid is displayed on the cells marked for refinement.

Face Color

controls the color of the cell faces on the cells marked for refinement.

Edge Color

controls the color of the cell edges on the cells marked for refinement.

Edge Width

controls the thickness of the cell edges on the cells marked for refinement.

Coarsening Cells

contains options related to the display of cells marked for coarsening.

Faces

controls whether cell faces are displayed on the cells marked for coarsening.

Edges

controls whether cell edges are displayed on the cells marked for coarsening.

Centroid

controls whether the cell centroid is displayed on the cells marked for coarsening.

Face Color

controls the color of the cell faces on the cells marked for coarsening.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Edge Color

controls the color of the cell edges on the cells marked for coarsening.

Edge Width

controls the thickness of the cell edges on the cells marked for coarsening.

Common Cells

contains options related to the display of cells marked for both refinement and coarsening.

Faces

controls whether cell faces are displayed on the cells marked for both refinement and coarsening.

Edges

controls whether cell edges are displayed on the cells marked for both refinement and coarsening.

Centroid

controls whether the cell centroid is displayed on the cells marked for both refinement and coarsening.

Face Color

controls the color of the cell faces on the cells marked for both refinement and coarsening.

Edge Color

controls the color of the cell edges on the cells marked for both refinement and coarsening.

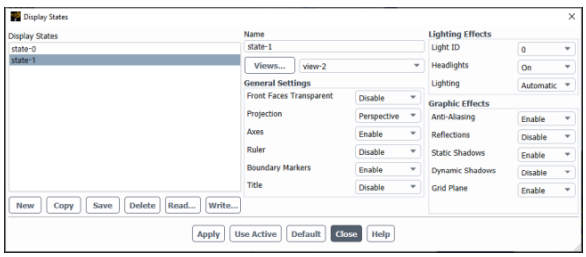
Edge Width

controls the thickness of the cell edges on the cells marked for both refinement and coarsening.

52.2.30. Display States Dialog Box

This dialog box is accessed by clicking **Display States...** in the **View** ribbon tab.

View → Display → Display States...



Controls

Display States

single selection listing of all defined display states.

New

create a new display state with the same settings as the active graphics window (or the last active user window if the current active window is displaying a 2D plot).

Copy

create a new display state the copies the properties of the selected display state.

Save

save the properties of the selected display state.

Delete

delete the selected state.

Read...

read in display states from a file.

Write...

write display states to a file.

Name

name of the selected display state.

Views...

opens the Views Dialog Box (p. 5272).

Views Drop-down List

list of views that can be associated with this display state.

General Settings

lists general graphics window settings.

Front Faces Transparency

controls whether the front faces are transparent.

Projection

specifies whether the display is in a perspective or orthographic view.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Axes

controls whether or not the axes triad is displayed.

Ruler

controls whether or not the ruler is displayed.

Boundary Markers

controls whether or not boundary markers are displayed. *Note that boundary markers are only displayed on mesh, not other graphics objects like contours or vectors.*

Title

controls whether or not the titles bar is displayed.

Edge Color

controls the color of the cell edges on the cells marked for refinement.

Lighting Effects

lists lighting settings.

Light ID

controls which light ID is saved. If more than one light ID is active when you click **Use Active** from a graphics object dialog box, this option is set to **Don't Save**.

Headlights

controls whether the headlight is on or off.

Lighting

specifies the lighting method that is used for the display.

Graphic Effects

lists the graphics effects.

Anti-Aliasing

controls whether or not lines and text are smooth.

Reflections

controls whether or not a reflection of the model is shown (with the axis of reflection being controlled in Preferences).

Static Shadows

controls whether or not static shadows are displayed.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Dynamic Shadows

controls whether or not dynamic shadows are displayed. *These shadows move as you rotate your model and are computationally expensive to render.*

Grid Plane

controls whether or not the grid plane is shown (with the axis for the grid being controlled in Preferences).

Apply

save the settings of the selected display state and apply these settings to the active graphics window display.

Use Active

update the currently-selected display state to inherit the display state settings of the active graphics window. *This button is not available if the active graphics window is displaying a 2D plot.*

Default

resets the selected display state to match the default settings.

Table 52.1: Default Display State Settings

Setting	State
View	isometric
Front Faces Transparent	Disable
Projection	Perspective
Axes	Enable
Ruler	Disable
Boundary Markers	Enable
Title	Disable
Light ID	0
Headlights	On
Lighting	Automatic
Anti-Aliasing	Enable
Reflections	Enable
Static Shadows	Enable
Dynamic Shadows	Disable

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

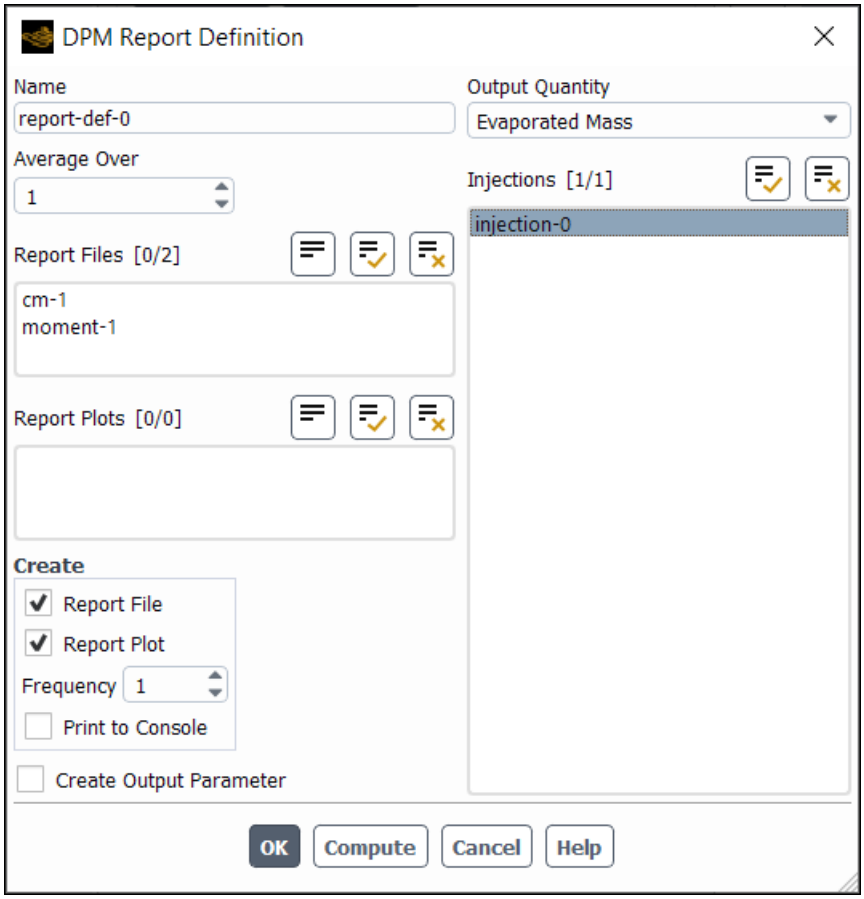
Setting	State
Grid Plane	Enable

Note:

The default graphics effects settings are dependent on the available graphics driver. If no modern graphics driver is available, graphics effects will be disabled.

52.2.31. DPM Report Definition Dialog Box

Solution → Reports → Definitions → New → DPM Report



Controls

Name

specifies the name of the report definition.

Average Over

(optional) To have Fluent calculate a running average for the **DPM Report Definition** you can enter a positive integer greater than 1 (the default) for **Average Over**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Specifying a number greater than 1 means that Fluent will print, plot, and write the running average value of the selected variable instead of the current value of the same variable.

The value reported is averaged over the last N *iterations/time steps, where* N *is your specified* ***Average Over*** *value. When the iteration/time step number is lower than* N*, Fluent calculates the average of the available variable values.*

Retain Instantaneous Values

appears when **Average Over** is set to a value greater than 1. If enabled when you are creating a new report definition, it ensures that all instantaneous values are saved in the new report file that is being created for this report definition. It also results in the **Plot Instantaneous Values** option being enabled in the newly created report plot.

Report Files

lists all of the report files where you can write the report definition data.

Report Plots

lists all of the report plots where you can plot the report definition data (assuming this report has the same units as the other report definitions included in the selected report plot).

Create

options you can choose when creating your report.

Report File

creates a new report file that includes this report definition.

Report Plot

creates a new report plot that includes this report definition.

Frequency

specifies how frequently the report definition is written, plotted, or printed to the console.

Print to Console

prints the value of the report definition to the console.

Create Output Parameter

creates an output parameter for this report definition with the name `<report definition name>-op`.

Output Quantity

quantities that can be applied to the output parameter.

Injected Mass

(unsteady particle tracking only) reports the total DPM mass injected into the domain.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mass in Domain

(unsteady particle tracking only) reports the total DPM mass present in the domain.

Mass in Fluid

(unsteady particle tracking only) reports all DPM mass currently residing as free-stream particles in the domain. This option is available only with the **wall-film** DPM boundary condition.

Mass in Film

(unsteady particle tracking only) reports all DPM mass currently residing in Lagrangian wall-film. This option is available only with the **wall-film** DPM boundary condition.

Escaped Mass

reports the DPM mass that has left the domain through a certain boundary or boundaries.

Evaporated Mass

reports the total evaporated DPM mass for the selected injection(s). The quantity accounts for fully evaporated particles as well as all mass that has evaporated from any other particles. For unsteady tracking, this includes particles that are still present in the domain.

Penetration Length

(unsteady particle tracking only) reports the total penetration length in meters.

Injections

lists the injections that you can select for discrete phase reporting. For the **Escaped Mass** output quantity, this item is available only if you have enabled the `report/dpm-zone-summaries-per-injection` text command. Note that for unsteady particle tracking, if you want to report the mass of escaped particles per injection, this text command must be enabled before any particles are injected into the domain.

Boundaries

contains a selectable list of valid boundary zones for escaped mass reporting.

Show Mass Flow/Change Rate

(unsteady particle tracking only) if enabled, specifies that the mass flow rate will be reported (default). Otherwise the total mass will be reported. This item is not available for the **Penetration Length** report.

Particles Mass Fraction

the mass fraction expected between the injection nozzle and the penetration length. This item is available only for the **Penetration Length** reports.

User-Specified Origin and Direction

once enabled, it allows you to specify the **Origin** and **Direction** of the selected injection(s). This item is available only for the **Penetration Length** reports.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

OK

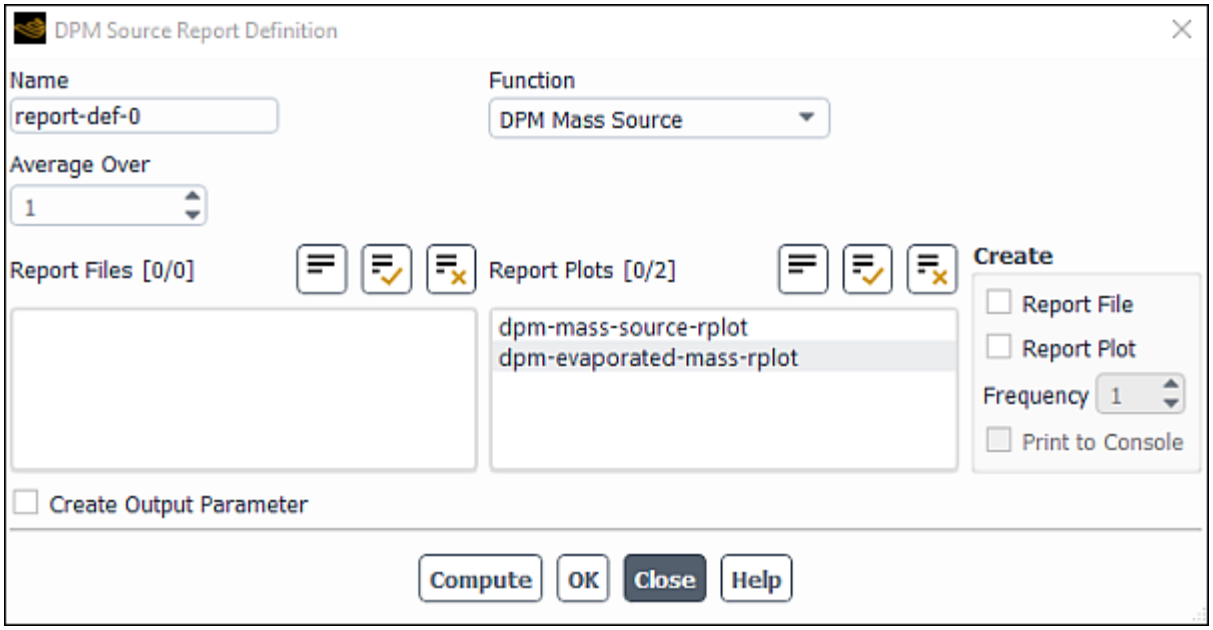
creates the report definition.

Compute

computes the value of the report definition and prints to the console.

52.2.32. DPM Source Report Definition Dialog Box

Solution → Reports → Definitions → New → Flux Report → DPM Mass Source | DPM Enthalpy Source | DPM Sensible Enthapy Source



Controls

Name

specifies the name of the report definition.

Function

functions available for the source report definition.

DPM Mass Source

reports the mass contributions from the particle injections.

DPM Enthalpy Source

reports the heat contributions from the particle injections.

DPM Sensible Enthalpy Source

reports the sensible heat contributions from the particle injections.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Average Over

(optional) To have Fluent calculate a running average for the **DPM Source Report Definition** you can enter a positive integer greater than 1 (the default) for **Average Over**.

Specifying a number greater than 1 means that Fluent will print, plot, and write the running average value of the selected variable instead of the current value of the same variable.

The value reported is averaged over the last N *iterations/time steps, where* N *is your specified* ***Average Over*** *value. When the iteration/time step number is lower than* N*, Fluent calculates the average of the available variable values.*

Retain Instantaneous Values

appears when **Average Over** is set to a value greater than 1. If enabled when you are creating a new report definition, it ensures that all instantaneous values are saved in the new report file that is being created for this report definition. It also results in the **Plot Instantaneous Values** option being enabled in the newly created report plot.

Report Files

lists all of the report files where you can write the report definition data.

Report Plots

lists all of the report plots where you can plot the report definition data (assuming this report has the same units as the other report definitions included in the selected report plot).

Create Output Parameter

creates an output parameter for this report definition with the name `<report definition name>-op`.

Create

options you can choose while generating a report.

Report File

creates a new report file that includes this report definition.

Report Plot

creates a new report plot that includes this report definition.

Frequency

specifies how frequently the report definition is written, plotted, or printed to the console.

Print to Console

prints the value of the report definition to the console.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Compute

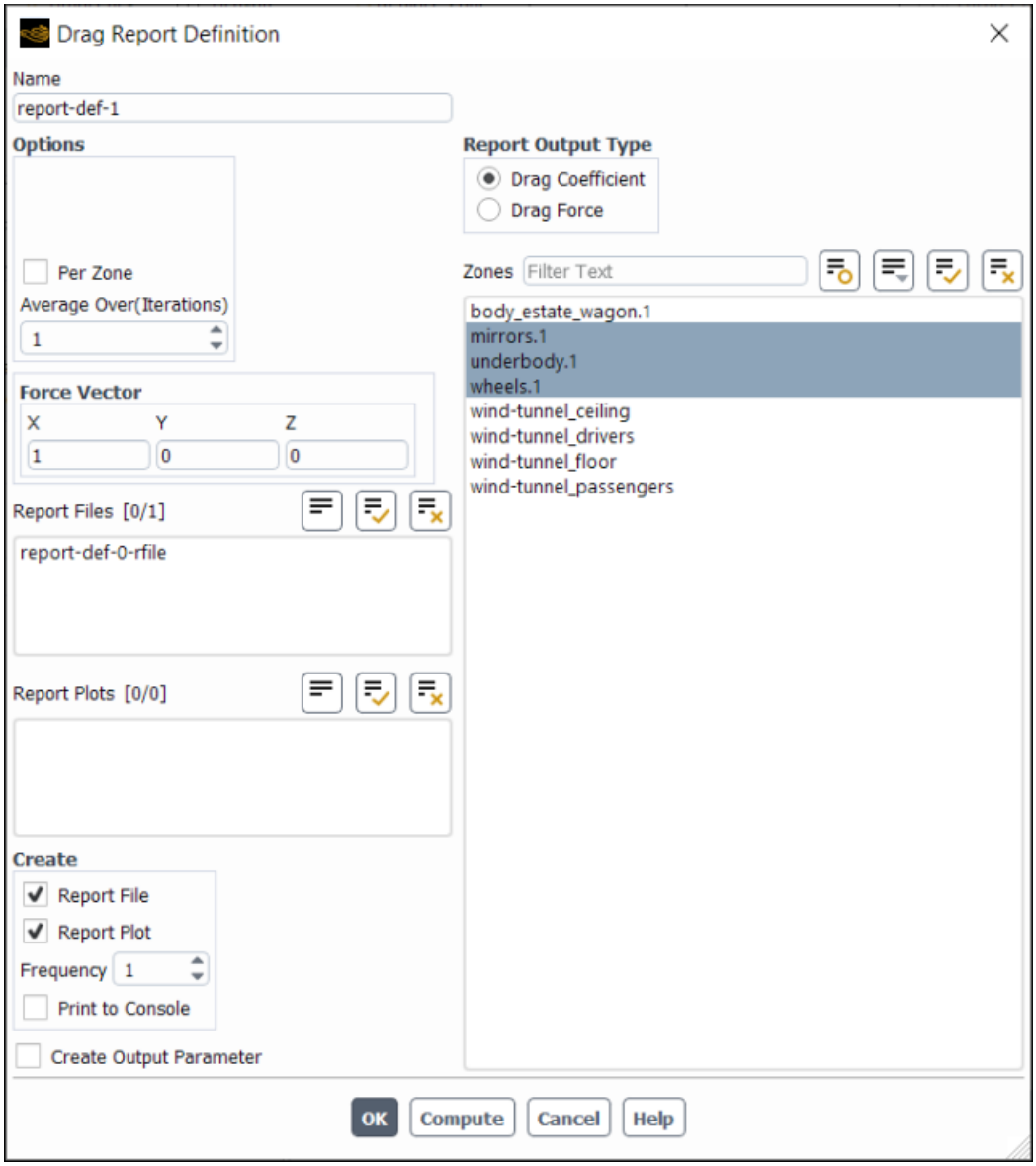
computes the value of the report definition and prints to the console.

OK

creates the report definition.

52.2.33. Drag Report Definition Dialog Box

Solution → Reports → Definitions → New → Force Report → Drag...



Controls

Name

specifies the name of the report definition.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Options

options you can choose for the drag report.

Per Zone

specifies whether or not the drag for multiple walls zones should be monitored on each wall zone separately. When this option is turned off (the default), Fluent computes and monitors the total drag for all of the selected walls zones combined together.

Average Over (Time-Steps), Average Over (Iterations)

(optional) To have Fluent calculate a running average for the **Drag Report Definition** you can enter a positive integer greater than 1 (the default) for **Average Over**.

Specifying a number greater than 1 means that Fluent will print, plot, and write the running average value of the selected variable instead of the current value of the same variable.

The value reported is averaged over the last N *iterations/time steps, where* N *is your specified* ***Average Over*** *value. When the iteration/time step number is lower than* N*, Fluent calculates the average of the available variable values.*

Retain Instantaneous Values

appears when **Average Over** is set to a value greater than 1. If enabled when you are creating a new report definition, it ensures that all instantaneous values are saved in the new report file that is being created for this report definition. It also results in the **Plot Instantaneous Values** option being enabled in the newly created report plot.

Force Vector

contains the components of the force vector.

X,Y,Z

are the components of the force vector along which the forces will be computed.

Report Files

lists all of the report files where you can write the report definition data.

Report Plots

lists all of the report plots where you can plot the report definition data (assuming this report has the same units as the other report definitions included in the selected report plot).

Create

options you can choose while generating a report.

Report File

creates a new report file that includes this report definition.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Report Plot

creates a new report plot that includes this report definition.

Frequency

specifies how frequently the report definition is written, plotted, or printed to the console.

Print to Console

prints the value of the report definition to the console.

Create Output Parameter

creates an output parameter for this report definition with the name `<report definition name>-op`.

Report Output Type

options you can choose relevant to report output.

Drag Coefficient

specifies the report definition as a unitless quantity.

Drag Force

specifies that the report definition has units of force.

Zones

lists the zones you can select for drag coefficient or drag force computation.

OK

creates the report definition.

Compute

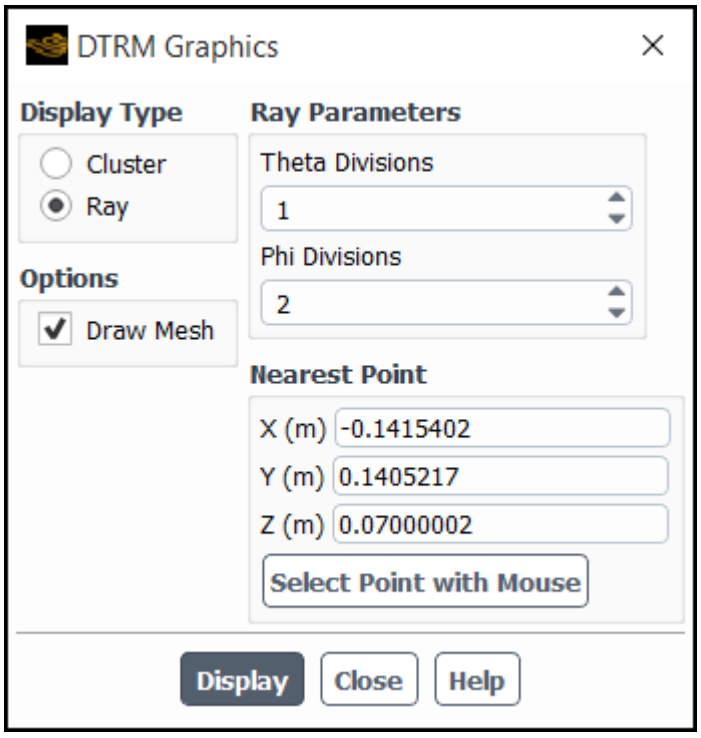
computes the value of the report definition at the selected zone(s) and prints to the console.

52.2.34. DTRM Graphics Dialog Box

The **DTRM Graphics** dialog box allows you to display rays and clusters used by the DTRM. See Displaying Rays and Clusters for the DTRM (p. 2245) for details.

Results → **Model Specific** → **DTRM Graphics...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Display Type

contains options for the different items you can display.

Cluster

specifies the display of clusters.

Ray

specifies the display of rays.

Options

contains check buttons that control display options.

Draw Mesh

toggles between displaying and not displaying the mesh. The Mesh Display Dialog Box (p. 4688) is opened when **Draw Mesh** is selected.

Cluster Type

specifies whether **Surface** clusters or **Volume** clusters are to be displayed. This section of the dialog box appears when **Cluster** is selected as the **Display Type**.

Cluster Selection

contains options for cluster displays. This section of the dialog box appears when **Cluster** is selected as the **Display Type**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Display All Clusters

enables the display of all surface or volume clusters in the domain.

Ray Parameters

contains controls for displaying rays. This section of the dialog box appears when **Ray** is selected as the **Display Type**.

Theta Divisions, Phi Divisions

control the number of rays being traced. (See Controlling the Rays (p. 2197).)

Nearest Point

specifies a point (**X,Y,Z**) near the cluster to be displayed (or the cluster from which the rays should start).

The **Nearest Point** controls are not available when **Display All Clusters** is selected under **Cluster Selection**.

Select Point With Mouse

is an alternative method for specifying the **Nearest Point** using your mouse, by clicking the button.

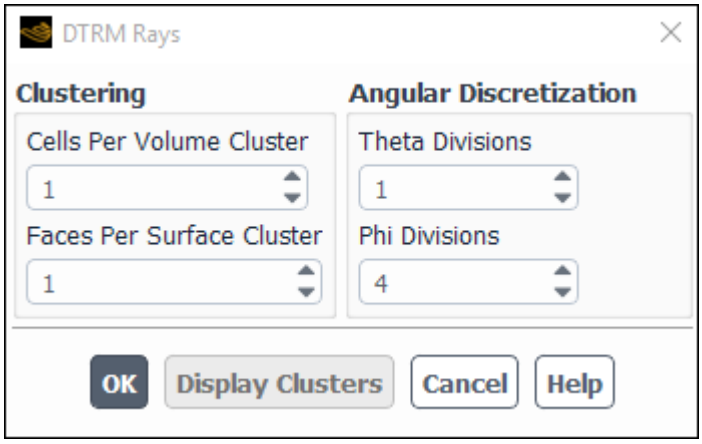
Display

displays the specified cluster(s) or rays.

52.2.35. DTRM Rays Dialog Box

The **DTRM Rays** dialog box allows you to define the rays used by the discrete transfer radiation model (DTRM). It opens automatically when you click **OK** after selecting the **Discrete Transfer** model in the Radiation Model Dialog Box (p. 4736). See Setting Up the DTRM (p. 2195) for details about the items below.

Physics → Model Specific → DTRM Rays...



Controls

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Clustering

contains parameters for the clusters (see Clustering in the Theory Guide).

Cells Per Volume Cluster, Faces Per Surface Cluster

control the number of radiating surfaces and absorbing cells. (See the explanation in Controlling the Clusters (p. 2197).)

Angular Discretization

contains parameters for the ray traces (see Ray Tracing in the Theory Guide).

Theta Divisions, Phi Divisions

control the number of rays being traced. (Guidelines are provided in Controlling the Rays (p. 2197).)

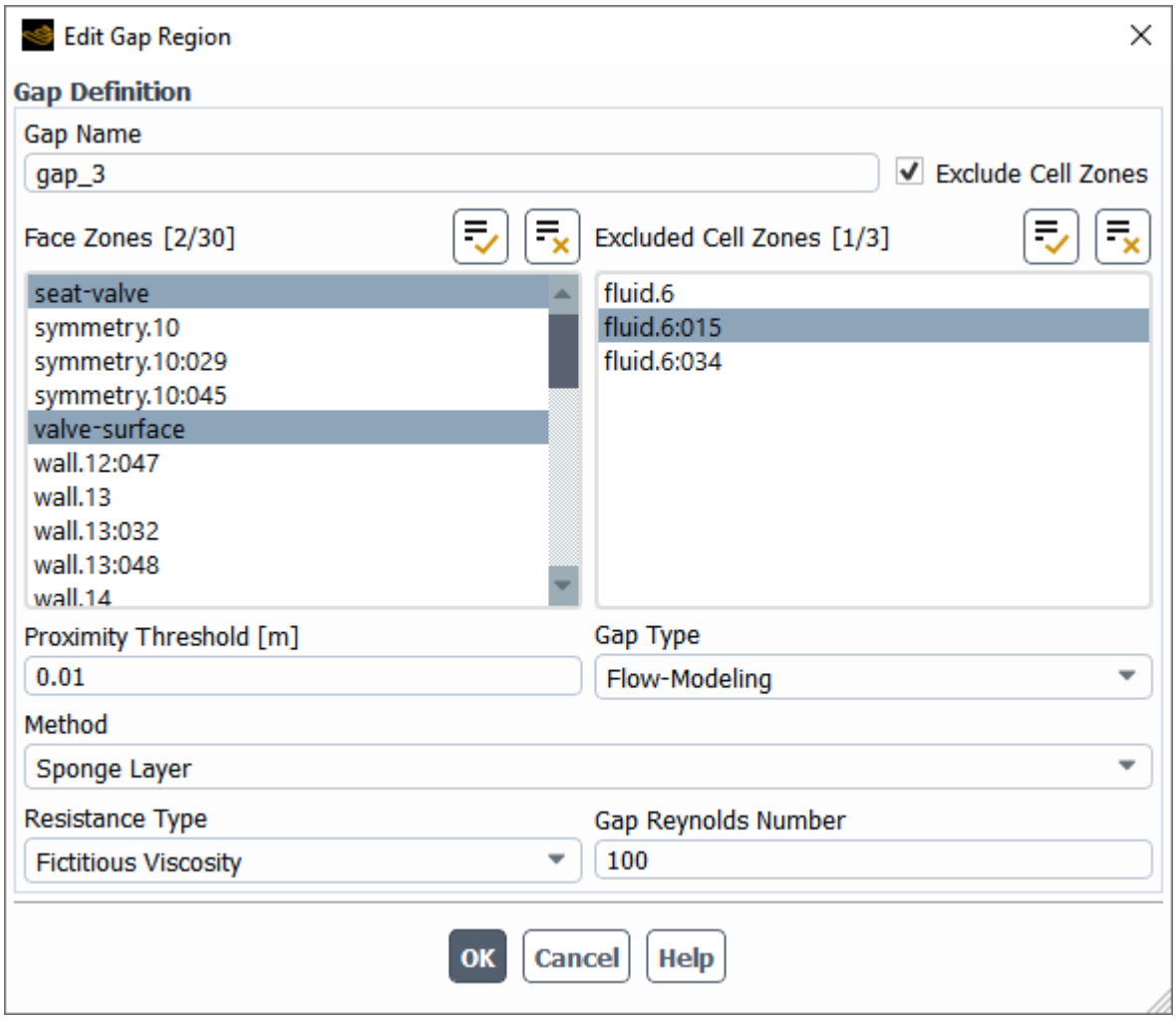
Display Clusters

generates a graphical display of the clusters in the domain. (This item is available only after you have created or read a ray file.)

52.2.36. Edit Gap Region Dialog Box

The **Edit Gap Region** dialog box allows you to edit an existing gap region as part of a gap model simulation. It is opened by making a selection from the **Gap Regions** list in the Gap Model Dialog Box (p. 5484) and clicking **Edit...**. See also Controlling Flow in Narrow Gaps for Valves and Pumps (p. 1234).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Gap Definition

contains input controls for editing an existing gap region.

Gap Name

specifies the name of the gap region. It cannot consist solely of numbers.

Face Zones

allows you to select the two or more face zones that constitute the gap region.

Proximity Threshold

defines the maximum distance between the face zones when flow blockage / deceleration occurs in the gap.

Gap Type

defines the type of the gap region. A **Flow-Blocking** type completely blocks the flow in the gap region, as a numerical zero-mass-flux boundary condition is applied at the boundary faces

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

of the marked region so that the marked cells no longer participate in the solution. A **Flow-Modeling** type decelerates the flow in the cells of the marked region by manipulating the viscosity or source terms.

Method

defines the method used to scale the viscosity or to define source terms in the gap region. The **Sponge Layer** method specifies that the viscosity inside the gap regions is scaled based on the specified **Gap Reynolds Number**. The **User-Defined Source** method specifies the use of a compiled user-defined function (UDF) to define the source terms for the momentum and energy equations in the gap region. This drop-down list is only available when **Flow-Modeling** is selected for the **Gap Type**.

Resistance Type

specifies how the viscosity is scaled in the gap region. The **Fictitious Viscosity** approach scales the laminar viscosity inside the gap region during discretization of the viscous forces for the momentum equation; thus, its impact is explicitly seen in the velocity and pressure computation, and it has an implicit effect on the other solution fields through the changes in the velocity and pressure field. The **Real Viscosity** approach changes the actual laminar viscosity inside the gap region based on your specified **Gap Reynolds Number**; therefore, the effect of modeling will be seen not only in flow equation but also in other equations. This drop-down list is only available when **Flow-Modeling** is selected for the **Gap Type** and **Sponge Layer** is selected for the **Method**.

Gap Reynolds Number

defines the intended Reynolds number in the gap region, and thus determines the level of sealing. A lower **Gap Reynolds Number** results in better sealing. It can be estimated according to Equation 6.4 (p. 1238). This drop-down list is only available when **Flow-Modeling** is selected for the **Gap Type** and **Sponge Layer** is selected for the **Method**.

Gap Source Terms

selects a compiled user-defined function (UDF) that specifies the source terms for the momentum and energy equations in the gap region using the `DEFINE_GAP_MODEL_SOURCE` macro. This drop-down list is only available when **Flow-Modeling** is selected for the **Gap Type** and **User-Defined Source** is selected for the **Method**.

Exclude Cell Zones

enables the ability to specify that cells in particular cell zones are not marked for flow blockage / deceleration as part of the gap region, in order to have greater control over the shape of the gap region.

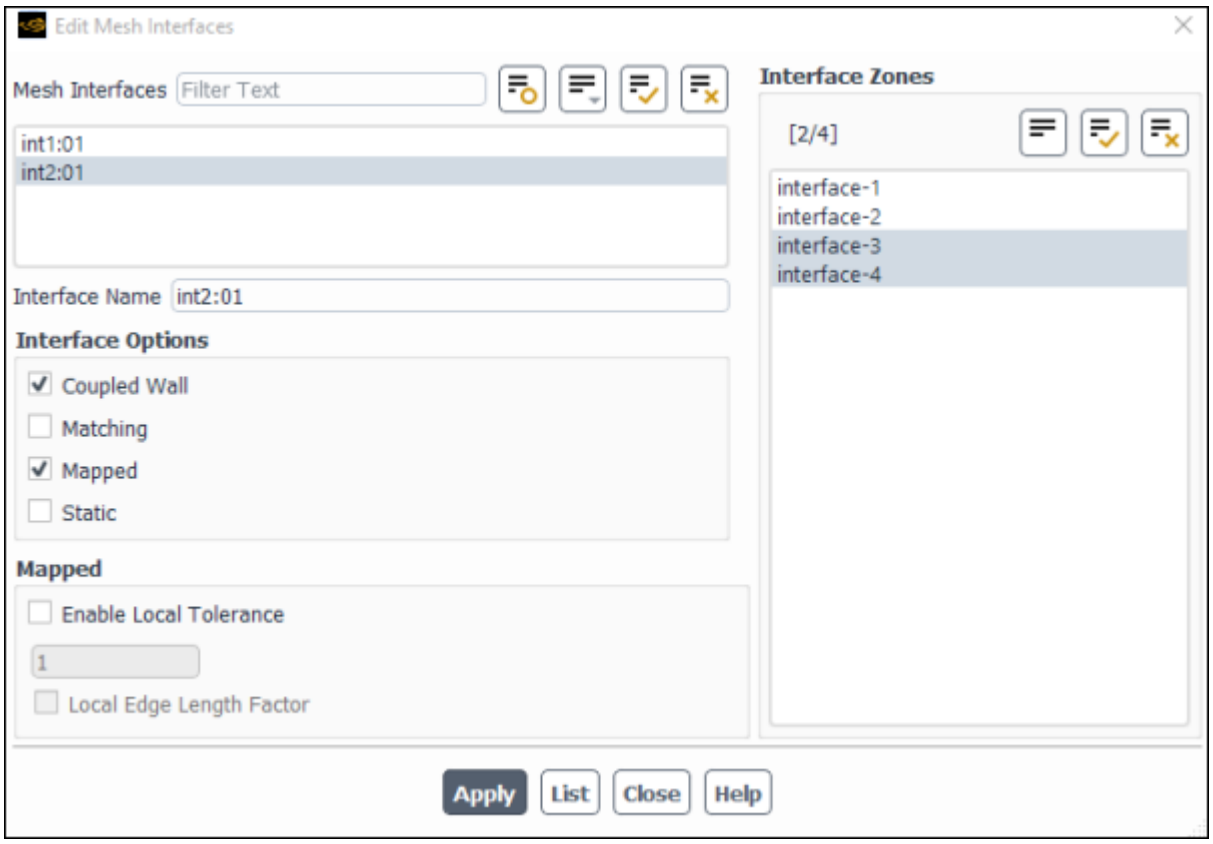
Excluded Cell Zones

selects the cell zones in which cells are not marked for flow blockage / deceleration as part of the gap region.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

52.2.37. Edit Mesh Interfaces Dialog Box

The **Edit Mesh Interfaces** dialog box allows you to edit existing many-to-many mesh interfaces. It is opened by making a selection from the **Mesh Interfaces** list in the Mesh Interfaces Dialog Box (p. 5525) and clicking **Edit...**. See also Using a Non-Conformal Mesh in Ansys Fluent (p. 1185).



Controls

Mesh Interfaces

allows you to select the mesh interfaces you would like to edit. Note that you can click the button to group the interfaces in the list by their currently enabled interface option(s), which allows you to select / deselect all those of a particular type by simply clicking the top-level branch.

Interface Name

edits the name of the mesh interface. This is only available when a single interface is selected.

Interface Options

contains options related to the interface type. These options can be applied when a single or multiple interfaces are selected.

Periodic Boundary Condition

allows you to create a non-conformal periodic boundary condition interface.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Periodic Repeats

should be enabled when each of the two cell zones has a single pair of conformal or non-conformal periodics adjacent to the interface. This option is typically used in conjunction with the sliding mesh model, when simulating the interface between a rotor and stator; it allows Ansys Fluent to treat the interface between the sliding and non-sliding zones as periodic where the two zones do not overlap. For details, see The Periodic Repeats Option (p. 1174).

Coupled Wall

allows you to specify that the interface acts as a thermally coupled wall.

Matching

is relevant if only interface internal zones should be created (that is, interior or wall / shadow pairs), since the interface zones on both sides are aligned. With the **Matching** option, even interface zones that are not perfectly aligned are treated as if they are; however, if the discrepancy between the interface zones on both sides exceeds default thresholds, then warning messages will be displayed. Note that the **Matching** option is compatible with the **Periodic Boundary Condition**, **Coupled Wall**, and **Static** options. For more information about the recommended uses of this option, see Matching Option (p. 1177).

Mapped

enables an alternative approach for modeling coupled walls between zones. This approach is more robust than the standard non-conformal interface formulations when the interface zones penetrate each other or have gaps between them. It requires that at least one side of the interface consists of only solid zones.

Note:

The **Mapped** option is not compatible with shell conduction.

Static

reduces the memory usage and processing time (for interface creation and solution), especially when there are many zones on both sides of the interface. This option will only produce correct results if the interface zones do not move or deform relative to each other at the interface, and it is not compatible with the **Periodic Boundary Condition**, **Periodic Repeats**, or **Mapped** options.

Periodic Boundary Conditions

Type

allows you to select a periodicity that is either **Translational** or **Rotational**.

Offset

is the offset coordinates or angle, depending on whether **Translational** or **Rotational** periodicity is selected. Note that when **Auto Compute Offset** is enabled, the **Offset** fields are not editable.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Auto Compute Offset

will result in Ansys Fluent finding the offset. If this option is disabled, then you will have to provide the offset coordinates or angle in the required fields, depending on whether **Translational** or **Rotational** periodicity is selected.

Mapped

Enable Local Tolerance

(when enabled) allows you to overwrite any value specified in the **Tolerance** group box in the Mapped Interface Options Dialog Box (p. 5520), and is applied to the selected interface only.

Local Edge Length Factor

(when enabled) allows you to specify the multiplier that Fluent uses with the smallest edge length of the interface zones to calculate the tolerance for mapping. Disabling **Local Edge Length Factor** allows you to specify an absolute value for the tolerance.

Interface Zones

highlights the list of interface zones that make up the selected mesh interface and allows you to edit the list (provided that the zones you select are unassigned and located appropriately). This functionality is only available when a single interface is selected.

List

prints information in the console about the first mesh interface you selected. When you click this button, Ansys Fluent will list the interface boundaries and (if you have initialized the solution) all new zones that were created (that is, wall and interior zones).

Apply

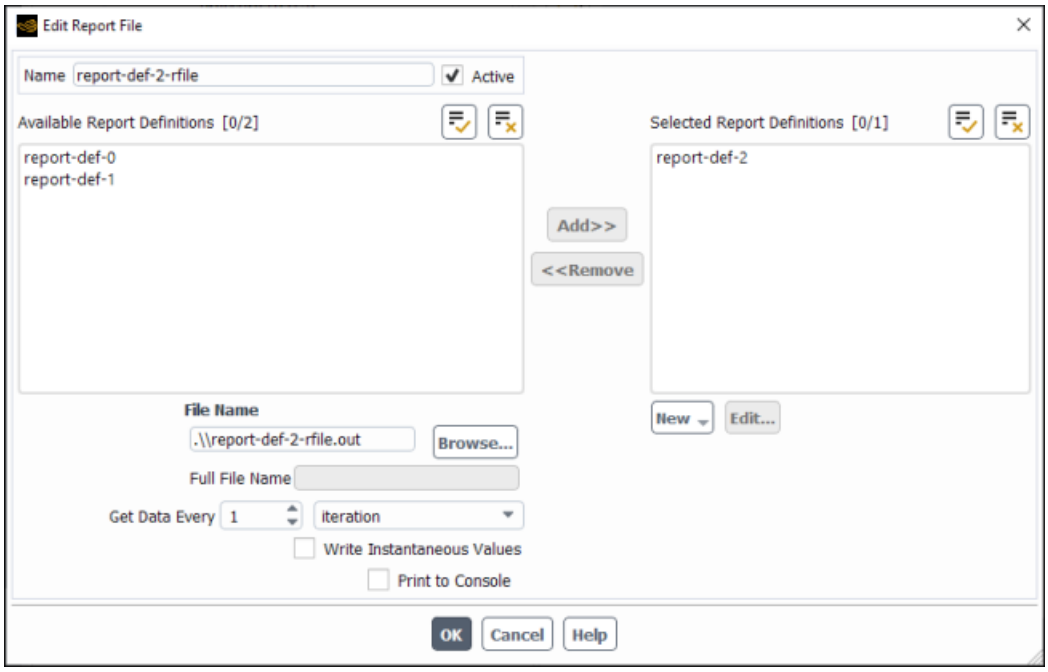
saves the revised settings for the selected mesh interfaces.

52.2.38. Edit Report File Dialog Box

Solution → **Reports** → **File...**

Select a report file in the **Report File Definition** dialog box and click **Edit...** to open the **Edit Report File** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name

editable field displaying the name of the report file.

Active

check box (enabled by default) that determines if the file is written during the solution calculation.

Available Report Definitions

lists all of the existing report definitions not currently part of this report file.

Add>>

moves the selected report definition(s) to the **Selected Report Definitions** list.

<<Remove

moves the selected report definition(s) to the **Available Report Definitions** list.

File Name

allows you to specify the name used when the file is saved.

Browse...

opens the **Select File** dialog box, allowing you to specify the location where the report file will be saved.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Full File Name

displays the name and location of the report file once it has been saved. *You can use the keyboard arrow keys to scroll within this field (<, >).*

Get Data Every

indicates the frequency that the report definition data is written to the file. The default value of 1 allows you to write to the file at every iteration or time-step. If you select flow-time, then you must specify the frequency for writing the file in units of time. Time-step and flow-time are only a valid choices when you are calculating unsteady flow.

Print to Console

indicates whether or not the contents of the report file are printed to the console at the frequency specified in the **Get Data Every** field.

Selected Report Definitions

lists all of the existing report definitions currently included in this report file.

New

drop-down list allowing you to create new report definitions (surface, volume, force, flux, mesh, aeromechanics, dpm, expression, and user-defined).

Edit...

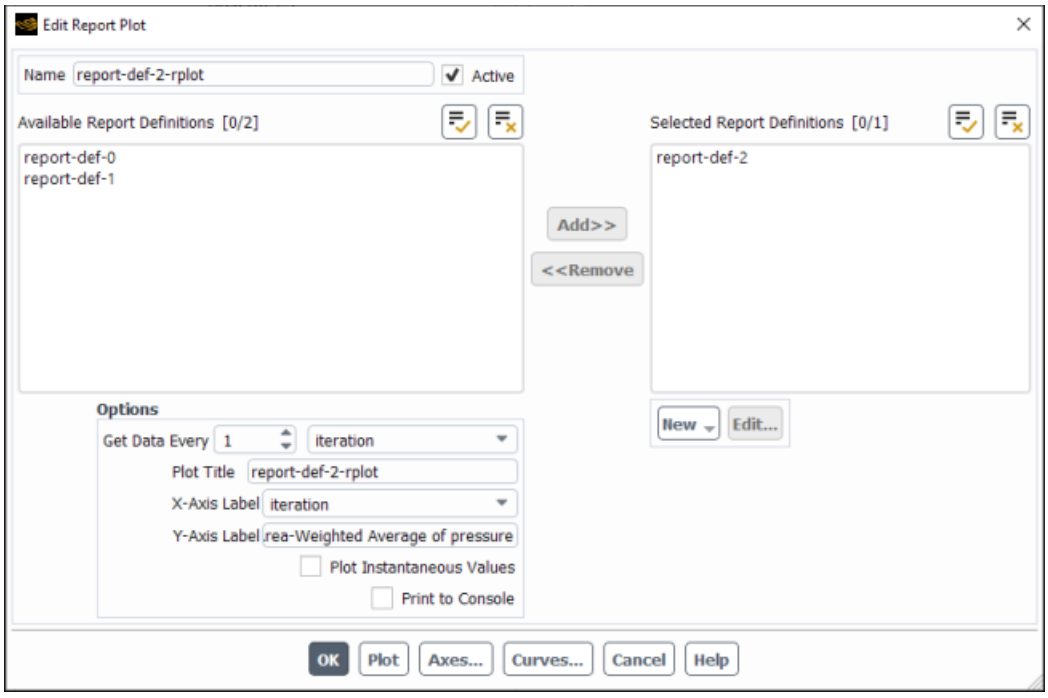
allows you to modify the highlighted report definition in the **Selected Report Definition** list.

52.2.39. Edit Report Plot Dialog Box

Solution → Reports → Plot...

Select a report plot in the **Report Plot Definition** dialog box and click **Edit...** to open the **Edit Report Plot** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name

editable field displaying the name of the report plot.

Active

check box (enabled by default) that determines if the plot is plotted during the solution calculation.

Available Report Definitions

lists all of the existing report definitions not currently part of this report plot.

Add>>

moves the selected report definition(s) to the **Selected Report Definitions** list.

<<Remove

moves the selected report definition(s) to the **Available Report Definitions** list.

Options

Get Data Every

indicates the frequency that the report definition data is plotted. The default value of 1 allows you to write to the file at every iteration or time-step. If you select flow-time, then you must specify the frequency for plotting the data in units of time. Time-step and flow-time are only a valid choices when you are calculating unsteady flow.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Plot Title

provide a title for the plot.

X-Axis Label

choose whether the X-axis is flow-time or time-step.

Y-Axis Label

enter a label for the Y-axis.

Print to Console

indicates whether or not the contents of the report plot are printed to the console at the frequency specified in the **Get Data Every** field.

Selected Report Definitions

lists all of the existing report definitions currently included in this report plot.

New

drop-down list allowing you to create new report definitions (surface, volume, force, flux, mesh, aeromechanics, dpm, expression, and user-defined).

Edit...

allows you to modify the highlighted report definition in the **Selected Report Definition** list.

Curves...

opens the Curves Dialog Box (p. 5316).

Axes...

opens the Axes Dialog Box (p. 5313).

52.2.40. Execute on Demand Dialog Box

The **Execute on Demand** dialog box allows you to run a specified user-defined function immediately. See the separate Fluent Customization Manual for details.

User Defined → User Defined → Execute on Demand...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Controls

Execute on Demand

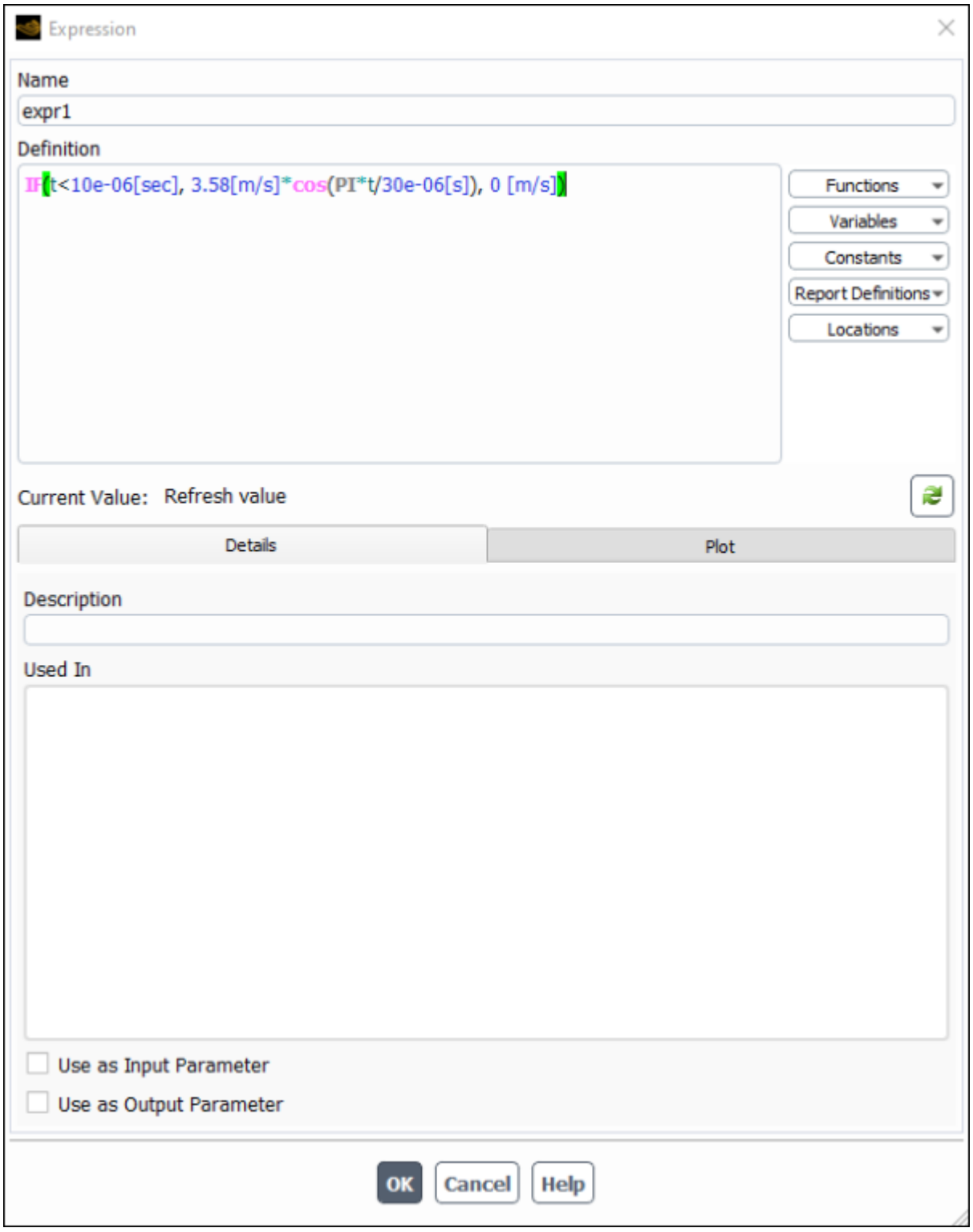
selects the user-defined function to be run.

Execute

runs the selected function.

52.2.41. Expression Dialog Box

Setup → Named Expressions New...



Controls

Name

an editable field allowing you to name the expression. *The **Name** field is not editable when the expression is in use.*

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Definition

displays the formula of the expression as you create it.

Current Value

Displays the current value of the expression.

Refresh Current Value

Evaluates the expression and updates the **Current Value** field.

Details

This tab contains the optional description of the expression, as well as where the expression is used, and the options to set the expression as an input or output parameter.

Description

(Optional) allows you to provide more of a description of this expression beyond the name.

Used In

lists the locations where the expression is used.

Use as Input Parameter

allows you to define the expression as an input parameter.

Use as Output Parameter

allows you to define the expression as an output parameter.

Plot

This tab allows you to plot your expression to visualize and confirm its behavior.

Primary Independent Variable

allows you to select which variable is the independent variable.

Count

allows you to specify the number of plot points.

Min

sets the minimum value for the independent variable.

Max

sets the maximum value for the independent variable.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Secondary Independent Variable

allows you to select the secondary independent variable.

Curves

allows you to specify the number of lines or curves to include in the plot.

Min

sets the minimum value for the secondary independent variable.

Max

sets the maximum value for the secondary independent variable.

Copy to Clipboard

copies the current plot onto your clipboard.

Reset Zoom

resets to the default zoom. *You can click and drag in the plot to change the zoom level.*

Drop-downs

The items in these drop-downs may be used in creating named expressions.

Functions

lists the available functions (conditional, mathematical, and so on) that are available for constructing your expression.

Variables

lists all of the supported variables.

Cell Registers

lists the defined cell registers.

Constants

lists the available pre-defined constants.

Expressions

lists the defined expressions.

Report Definitions

lists the defined report definitions.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Locations

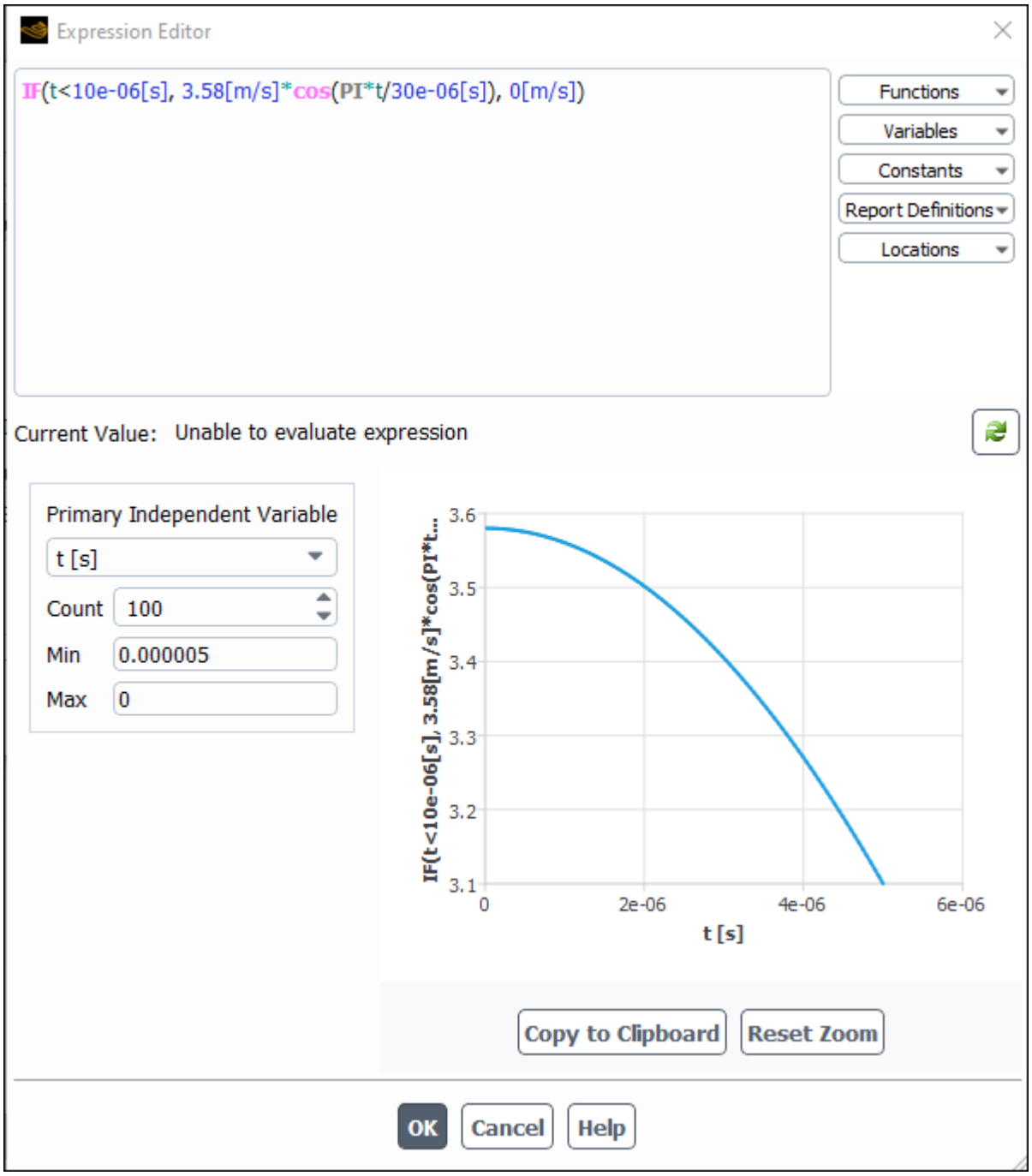
lists the available locations that you can include in your expression volume definition.

For additional information on expressions, refer to Fluent Expressions Language (p. 1037).

52.2.42. Expression Editor Dialog Box

The **Expression Editor** dialog box is used for setting expressions for fields, such as velocity, material density, and so on. It is opened by selecting **Expression** in the drop-down list for that field, and clicking the button that appears. Refer to Directly Applied Expressions (p. 1050) for additional information.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Current Value

Displays the current value of the expression.

Refresh Current Value

Evaluates the expression and updates the **Current Value** field.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Primary Independent Variable

allows you to select which variable is the independent variable.

Count

allows you to specify the number of plot points.

Min

sets the minimum value for the independent variable.

Max

sets the maximum value for the independent variable.

Secondary Independent Variable

allows you to select the secondary independent variable.

Curves

allows you to specify the number of lines or curves to include in the plot.

Min

sets the minimum value for the secondary independent variable.

Max

sets the maximum value for the secondary independent variable.

Copy to Clipboard

copies the current plot onto your clipboard.

Reset Zoom

resets to the default zoom. *You can click and drag in the plot to change the zoom level.*

Drop-downs

The items in these drop-downs may be used in creating named expressions.

Functions

lists the available functions (conditional, mathematical, and so on) that are available for constructing your expression.

Variables

lists all of the supported variables.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cell Registers

lists the defined cell registers.

Constants

lists the available pre-defined constants.

Expressions

lists the defined expressions.

Report Definitions

lists the defined report definitions.

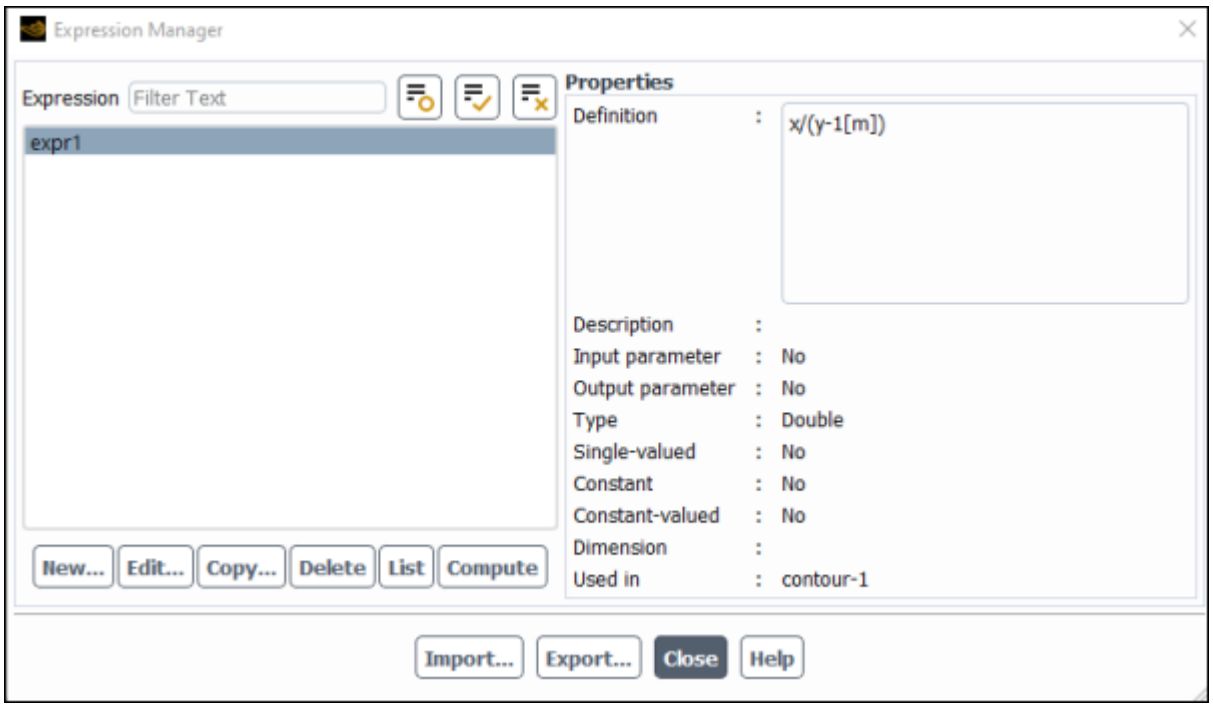
Locations

lists the available locations that you can include in your expression volume definition.

For additional information on expressions, refer to Fluent Expressions Language (p. 1037).

52.2.43. Expression Manager Dialog Box

Setup → Named Expressions → Manage...



Controls

Expression

Lists the defined named expressions.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Properties

Lists the properties of the selected expression in the **Expression** list. *This list is blank when no expressions are selected or when more than one expression is selected.*

New...

Opens the Expression Dialog Box (p. 5461), allowing you to create a new named expression.

Edit...

(Only available when a single expression is selected) Opens the Expression Dialog Box (p. 5461), allowing you to edit the selected expression.

Copy...

(Only available when a single expression is selected) Opens the Expression Dialog Box (p. 5461) with the same definition as the selected expression. The **Name** field is automatically appended with a digit, but you may modify the name, if desired.

Delete

Deletes the selected expression(s). Note that some expressions cannot be deleted if they are included in the definitions of other expressions or if they are in use elsewhere in Fluent.

List

Prints the definition and description of all named expressions to the console.

Compute

Computes the value of the selected expression(s) and prints the value(s) to the console.

Import...

Opens the **Select File** dialog box so that you can read-in one or more expressions from a file.

Export...

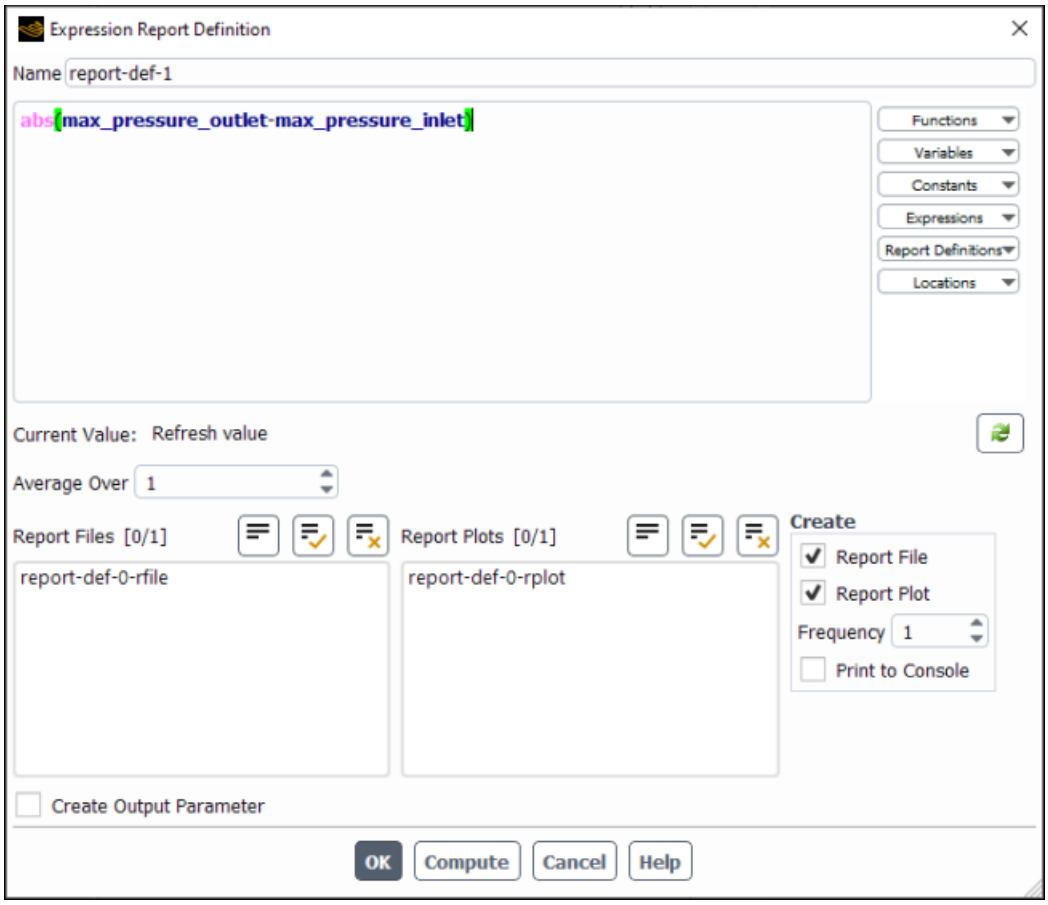
Opens the **Select File** dialog box so that you can export the selected expression(s) to a file.

For additional information on expressions, refer to Expression Manager (p. 1063).

52.2.44. Expression Report Definition Dialog Box

Solution → **Reports** → **Definitions** → **New** → **Expression...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name

an editable field allowing you to name the expression. *The **Name** field is not editable when the expression is in use.*

Definition

displays the formula of the expression as you create it.

Current Value

Displays the current value of the expression.

Refresh Current Value

Evaluates the expression and updates the **Current Value** field.

Average Over

(optional) To have Fluent calculate a running average for the **Expression Report Definition** you can enter a positive integer greater than 1 (the default) for **Average Over**.

Specifying a number greater than 1 means that Fluent will print, plot, and write the running average value of the selected variable instead of the current value of the same variable.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The value reported is averaged over the last N *iterations/time steps, where* N *is your specified* ***Average Over*** *value. When the iteration/time step number is lower than* N*, Fluent calculates the average of the available variable values.*

Retain Instantaneous Values

appears when **Average Over** is set to a value greater than 1. If enabled when you are creating a new report definition, it ensures that all instantaneous values are saved in the new report file that is being created for this report definition. It also results in the **Plot Instantaneous Values** option being enabled in the newly created report plot.

Report Files

lists all of the report files where you can write the report definition data.

Report Plots

lists all of the report plots where you can plot the report definition data (assuming this report has the same units as the other report definitions included in the selected report plot).

Create

Report File

creates a new report file that includes this report definition.

Report Plot

creates a new report plot that includes this report definition.

Frequency

specifies how frequently the report definition is written, plotted, or printed to the console.

Print to Console

prints the value of the report definition to the console.

Create Output Parameter

creates an output parameter for this report definition with the name `<report definition name>-op`.

Compute

prints the value of the expression report definition to the Console.

Drop-downs

The items in these drop-downs may be used in creating expression report definitions.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Functions

lists the available functions (conditional, mathematical, and so on) that are available for constructing your expression.

Variables

lists all of the supported variables.

Cell Registers

lists the defined cell registers.

Constants

lists the available pre-defined constants.

Expressions

lists the defined expressions.

Report Definitions

lists the defined report definitions.

Locations

lists the available locations that you can include in your expression report definition.

For additional information on expression report definitions, refer to Expression Report Definition (p. 4121).

52.2.45. Expression Volume Dialog Box

Results → Surface → Create → Expression Volume...

Controls

Name

an editable field allowing you to name the expression volume.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Definition

displays the formula of the expression volume as you create it.

Create

creates the expression volume.

Save

saves the changes you made to an existing expression volume.

Drop-downs

The items in these drop-downs may be used in creating expression report definitions.

Functions

lists the available functions (conditional, mathematical, and so on) that are available for constructing your expression.

Variables

lists all of the supported variables.

Cell Registers

lists the defined cell registers.

Constants

lists the available pre-defined constants.

Expressions

lists the defined expressions.

Report Definitions

lists the defined report definitions.

Locations

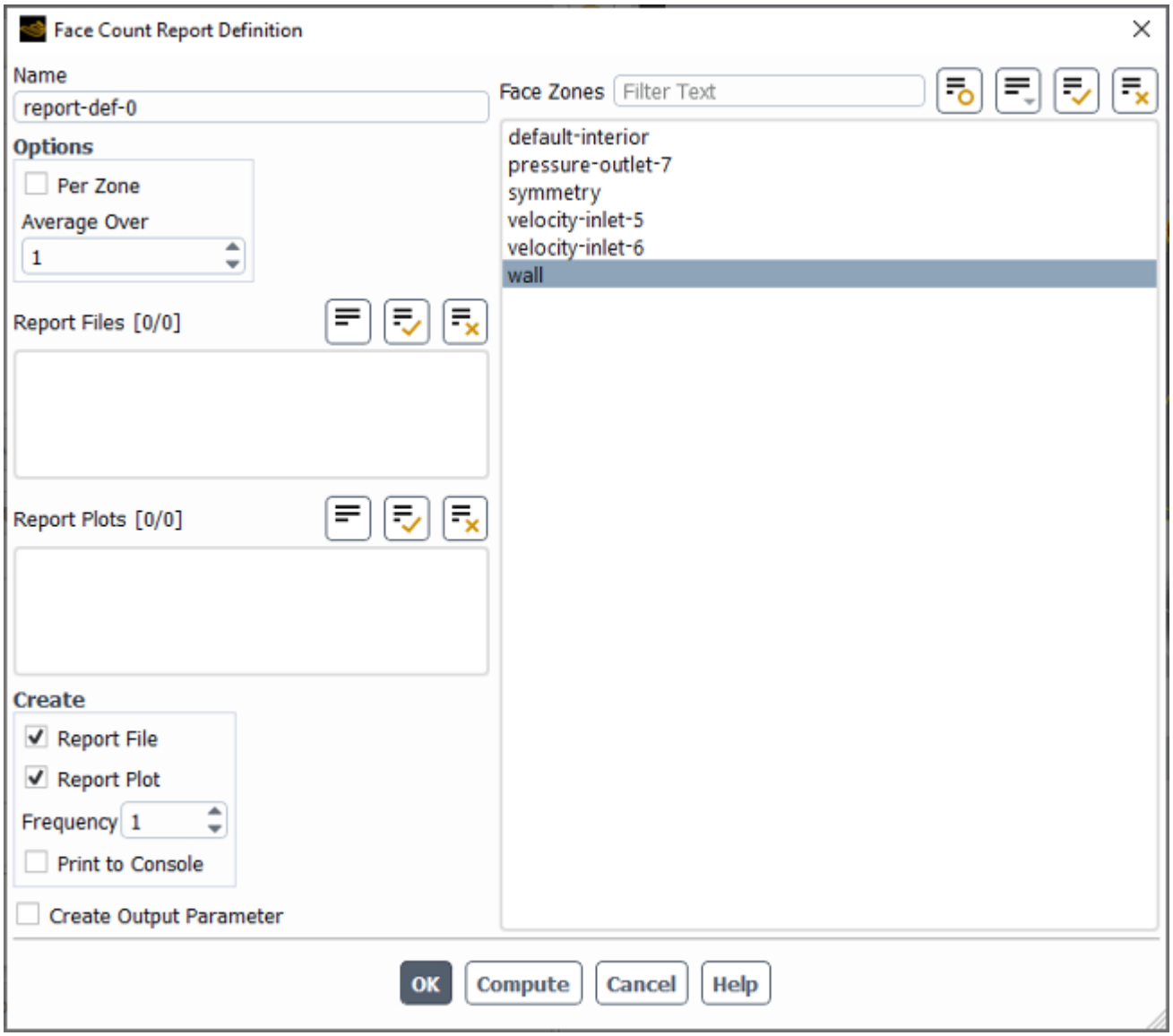
lists the available locations that you can include in your expression volume definition.

For additional information on expression report definitions, refer to Expression Report Definition (p. 4121).

52.2.46. Face Count Report Definition Dialog Box

The **Face Count Report Definition** dialog box allows you to create a report definition for the reporting of a face count (that is, the number of faces within one or more boundary zones), as described in Mesh Report Definitions (p. 4114).

Solution → **Reports** → **Definitions** → **New** → **Mesh Report** → **Face Count...**



Controls

Name

specifies the name of the report definition.

Options

Per Surface

enables the reporting of the face count individually for each of the selected zones (rather than reporting the sum from all of the selected zones).

Average Over

specifies the number of iterations / time steps over which the face count is averaged when written, plotted, and/or printed. The default value of 1 means that no averaging is performed (that is, only the current face count is reported), and higher values yields a running average.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

When the iteration / time step number is lower than the specified value, Fluent calculates the average of the available face count values.

Retain Instantaneous Values

enables the retention of instantaneous values in any report file definition or report plot definition that is associated with this report definition. This option is only available if **Average Over** is set to a value greater than 1.

Report Files

allows you to select existing report file definitions to which you want to write face count data from this report definition.

Report Plots

allows you to select existing report plot definitions to which you want to include face count data from this report definition. Note that the units used in these report plots may not be suitable for the face count data.

Create

Report File

creates a new report file definition for the data from this report definition.

Report Plot

creates a new report plot definition for the data from this report definition.

Frequency

specifies how frequently the report definition is written, plotted, and/or printed to the console.

Print to Console

enables the printing of the face count data from this report definition to the console during the calculation.

Create Output Parameter

creates an output parameter for this report definition with the name *<report_definition_name>*-op.

Face Zones

allows you to select the face zones for which you want to report the number of faces.

OK

creates / saves the report definition.

Compute

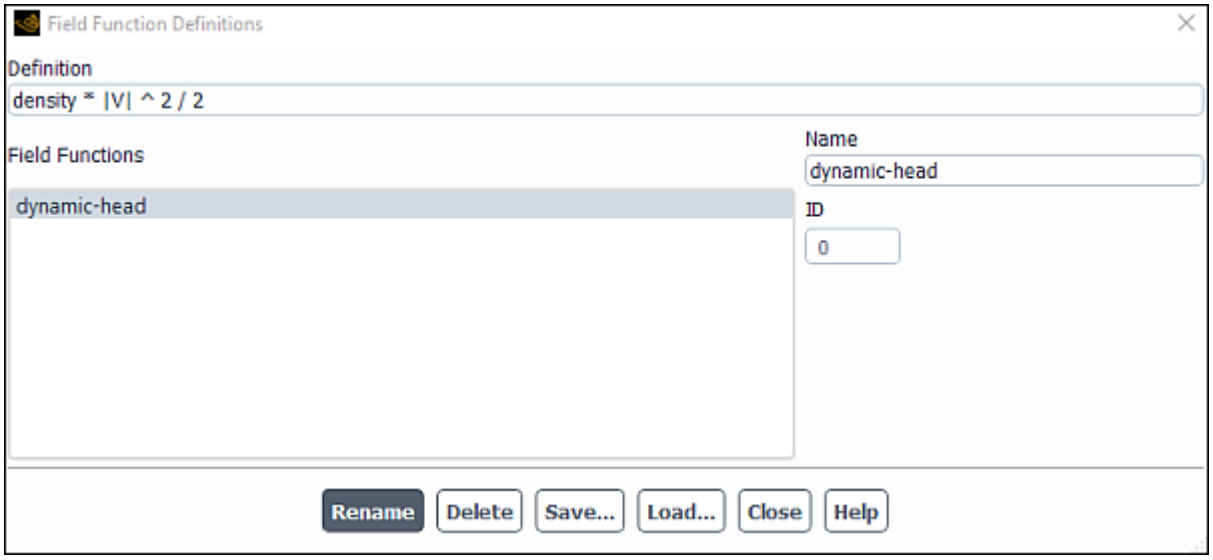
computes the face count within the selected zone(s) and prints to the console.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

52.2.47. Field Function Definitions Dialog Box

The **Field Function Definitions** dialog box allows you to check, rename, save, load, and delete custom field functions that you defined in the Custom Field Function Calculator Dialog Box (p. 5428). See Manipulating, Saving, and Loading Custom Field Functions (p. 4289) for details about the items below.

The **Field Function Definitions** dialog box is opened from the Custom Field Function Calculator Dialog Box (p. 5428).



Controls

Definition

displays the function selected in the **Field Functions** list. This display is for informational purposes only; you cannot edit it.

Field Functions

contains a selectable list of custom field functions. When you select a function, its definition will appear in the **Definition** box and its name will appear in the **Name** text entry box.

Name

displays the name of the currently selected field function. You can enter a new name in this box if you want to rename the function.

ID

reports the ID number of the selected function. The field function at the top of the list has an ID of 0, the second function has an ID of 1, and so on.

Rename

changes the name of the selected function to the name specified in the **Name** text entry box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Delete

deletes the selected field function.

Save...

opens The Select File Dialog Box (p. 931), where you can specify a file in which to save all of the functions in the **Field Functions** list.

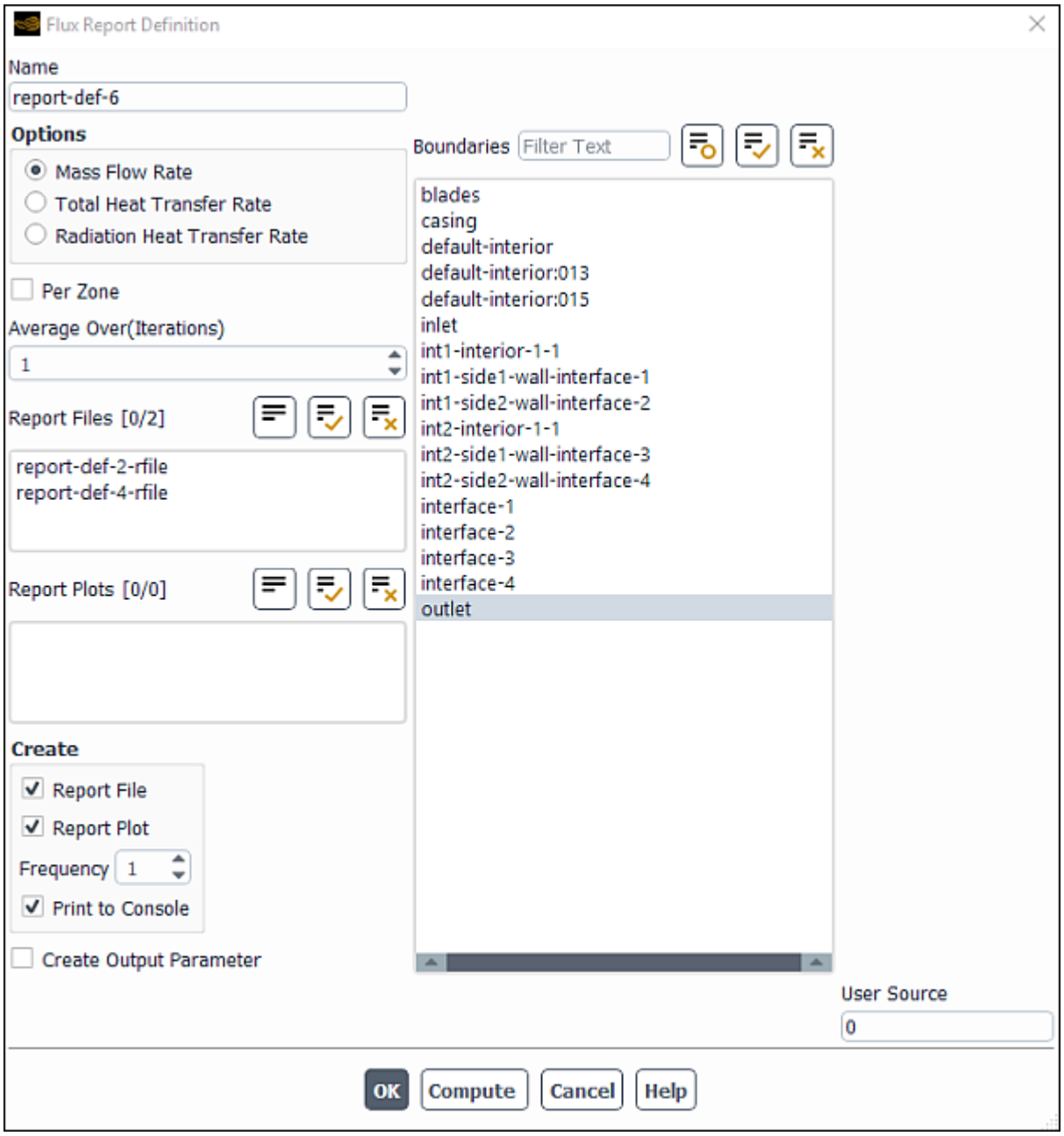
Load...

opens the **Select File** dialog box, where you can specify a file from which to read custom field functions (that is, a file that you saved using the **Save...** button above).

52.2.48. Flux Report Definition Dialog Box

Solution → **Reports** → **Definitions** → **New** → **Flux Report**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name

specifies the name of the report definition.

Options

Mass Flow Rate

specifies that the mass flow rate is computed for the selected boundary zones.

Total Heat Transfer Rate

specifies that the total heat transfer rate is computed for the selected boundary zones.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Total Sensible Heat Transfer Rate

specifies that the total sensible heat transfer rate is computed for the selected boundary zones. It reports the total energy flux as defined in Equation 5.5 in the Theory Guide.

Radiation Heat Transfer Rate

specifies that the radiation heat transfer rate is computed for the selected boundary zones.

Pressure Work Rate

specifies that the pressure work rate is computed for the selected boundary zones. This rate is the energy transfer due to pressure when a fluid zone undergoes normal zone motion relative to an adjacent solid zone (for further details, see Equation 25.1 in the Theory Guide). This pressure work is on the fluid side only, not on the solid side. This option is only available when the energy equation is enabled, the absolute velocity formulation is selected, and the **Frame Motion** and/or **Mesh Motion** option is enabled in the **Fluid** dialog box.

Viscous Work Rate

specifies that the viscous work rate is computed for the selected boundary zones, which is the integral of the product of viscous stress with velocity over the external boundary of the domain with the boundary normal directed inside (for further details, see Equation 25.2 in the Theory Guide). Note that after reading case and data files into Ansys Fluent, the viscous work rate is zero. You need to run at least one iteration or time step to obtain the viscous work rate value. This item is available only when the energy equation is enabled and the pressure-based solver is selected.

Film Mass Flow Rate

(available only when the Eulerian wall film model is enabled) specifies that the mass flow rate is computed for the selected boundary zone(s).

Film Heat Transfer Rate

(available only when the Eulerian wall film model is enabled) specifies that the film heat transfer rate is computed for the selected boundary zone(s).

Phase

contains a list of all of the phases in the problem that you have defined. This is available when the VOF, mixture, or Eulerian multiphase model is enabled.

Per Zone

specifies whether or not the chosen field variable is calculated from all of the selected boundary zones combined (default) or individually on each of the selected boundary zones.

Average Over (Time-Steps), Average Over (Iterations)

(optional) To have Fluent calculate a running average for the **Flux Report Definition** you can enter a positive integer greater than 1 (the default) for **Average Over**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Specifying a number greater than 1 means that Fluent will print, plot, and write the running average value of the selected variable instead of the current value of the same variable.

The value reported is averaged over the last N *iterations/time steps, where* N *is your specified* ***Average Over*** *value. When the iteration/time step number is lower than* N*, Fluent calculates the average of the available variable values.*

Retain Instantaneous Values

appears when **Average Over** is set to a value greater than 1. If enabled when you are creating a new report definition, it ensures that all instantaneous values are saved in the new report file that is being created for this report definition. It also results in the **Plot Instantaneous Values** option being enabled in the newly created report plot.

Report Files

lists all of the report files where you can write the report definition data.

Report Plots

lists all of the report plots where you can plot the report definition data (assuming this report has the same units as the other report definitions included in the selected report plot).

Create

Report File

creates a new report file that includes this report definition.

Report Plot

creates a new report plot that includes this report definition.

Frequency

specifies how frequently the report definition is written, plotted, or printed to the console.

Print to Console

prints the value of the report definition to the console.

Create Output Parameter

creates an output parameter for this report definition with the name `<report definition name>-op`.

Report Type

selects the integration method used on the selected cell zones. The available report types are the same as those in the Volume Integrals Dialog Box (p. 5329). See Volume Integration (p. 4153) for details.

Boundaries

contains a selectable list of valid boundary zones for flux reporting.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

OK

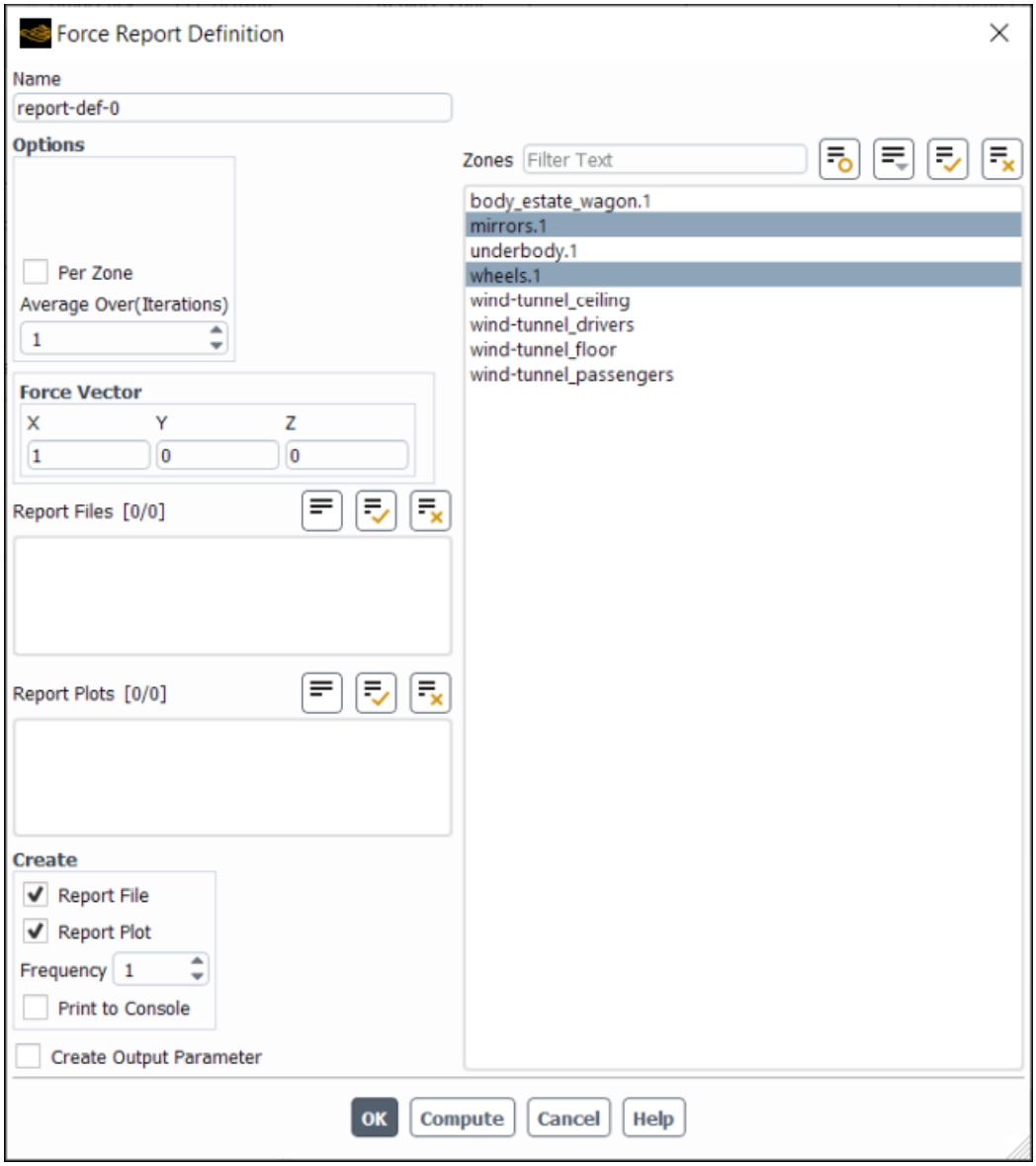
creates the report definition.

Compute

computes the value of the report definition at the selected zone(s) and prints to the console.

52.2.49. Force Report Definition Dialog Box

Solution → Reports → Definitions → New → Force Report → Force...



Controls

Name

specifies the name of the report definition.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Options

Per Zone

specifies whether or not the force for multiple walls zones should be monitored on each wall zone separately. When this option is turned off (the default), Fluent computes and monitors the total force for all of the selected walls zones combined together.

Average Over (Time-Steps), Average Over (Iterations)

(optional) To have Fluent calculate a running average for the **Force Report Definition** you can enter a positive integer greater than 1 (the default) for **Average Over**.

Specifying a number greater than 1 means that Fluent will print, plot, and write the running average value of the selected variable instead of the current value of the same variable.

The value reported is averaged over the last N *iterations/time steps, where* N *is your specified* ***Average Over*** *value. When the iteration/time step number is lower than* N*, Fluent calculates the average of the available variable values.*

Retain Instantaneous Values

appears when **Average Over** is set to a value greater than 1. If enabled when you are creating a new report definition, it ensures that all instantaneous values are saved in the new report file that is being created for this report definition. It also results in the **Plot Instantaneous Values** option being enabled in the newly created report plot.

Force Vector

contains the components of the force vector.

X,Y,Z

are the components of the force vector along which the forces will be computed.

Report Files

lists all of the report files where you can write the report definition data.

Report Plots

lists all of the report plots where you can plot the report definition data (assuming this report has the same units as the other report definitions included in the selected report plot).

Create

Report File

creates a new report file that includes this report definition.

Report Plot

creates a new report plot that includes this report definition.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Frequency

specifies how frequently the report definition is written, plotted, or printed to the console.

Print to Console

prints the value of the report definition to the console.

Create Output Parameter

creates an output parameter for this report definition with the name `<report definition name>-op`.

Zones

lists the zones you can select for force computation.

OK

creates the report definition.

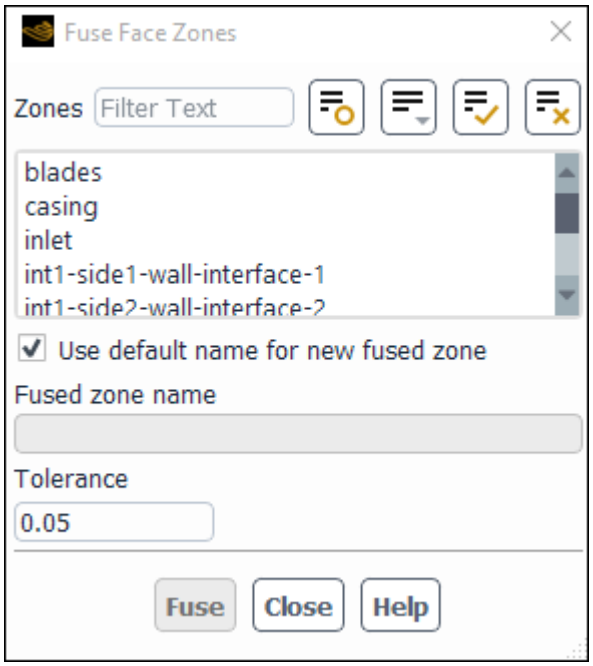
Compute

computes the value of the report definition at the selected zone(s) and prints to the console.

52.2.50. Fuse Face Zones Dialog Box

The **Fuse Face Zones** dialog box allows you to fuse boundaries (that is, remove duplicate nodes and faces and delete artificial internal boundaries) created by assembling multiple mesh regions. (See Reading Multiple Mesh/Case/Data Files (p. 1156) for details on importing such meshes.) See Fusing Face Zones (p. 1267) for information about using this dialog box.

Domain → Zones → Combine... → Fuse...



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Controls

Zones

contains a list of face zones from which you can select the boundaries to be fused.

Use default name for new fused zone

specifies whether to use an automatically generated default name for the fused zone.

Fused zone name

specifies the desired name for the fused zone if **Use default name for new fused zone** is disabled.

Tolerance

is a fraction of the minimum edge length of the face, used to determine whether or not nodes are coincident. If all of the appropriate faces do not get fused using the default **Tolerance**, you should increase it and attempt to fuse the zones again. The **Tolerance** should not exceed 0.5, or you may fuse the wrong nodes.

Fuse

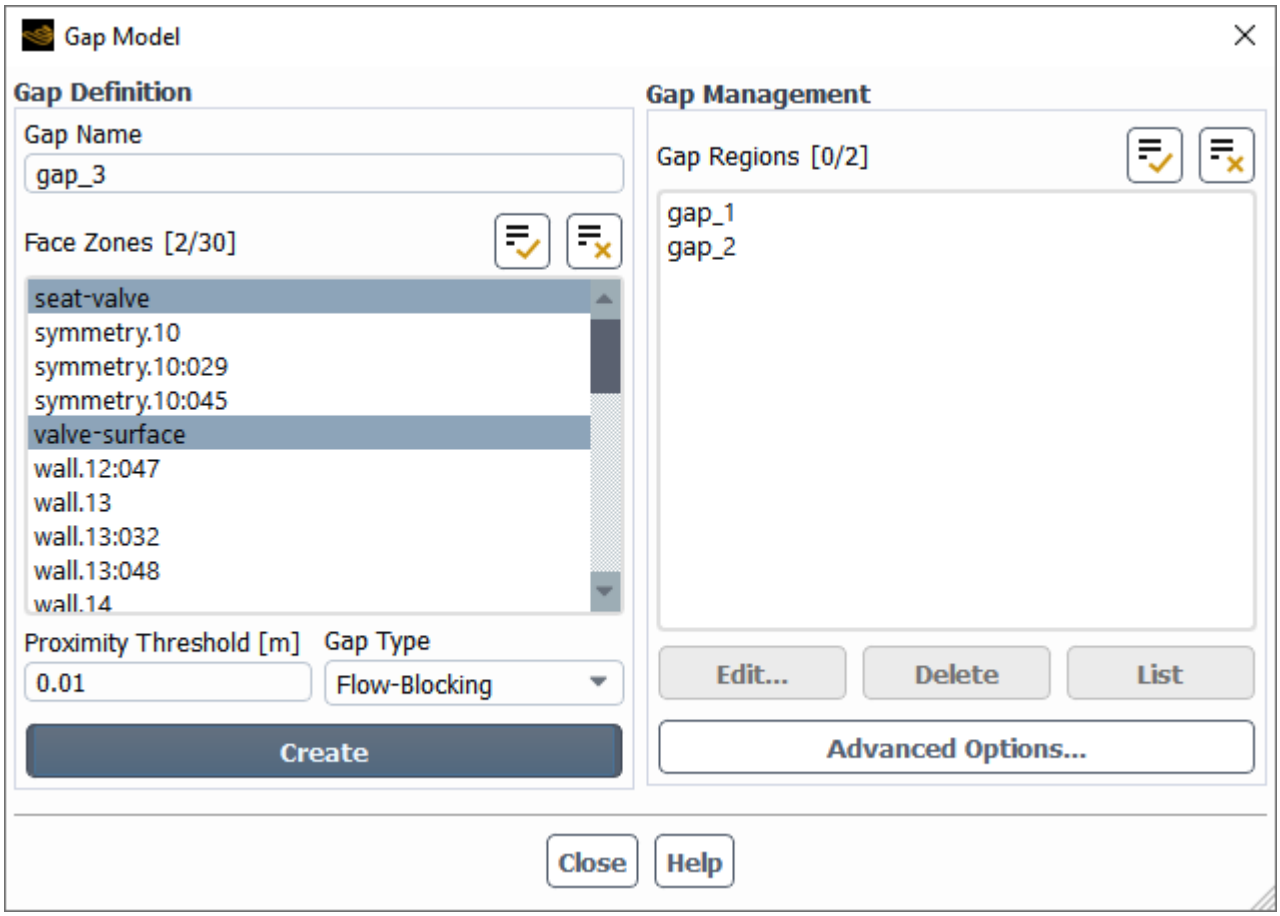
fuses the selected zones. It is enabled only when you have selected a minimum of two zones.

52.2.51. Gap Model Dialog Box

The **Gap Model** dialog box allows you to create gap regions for use with dynamic meshes (see Using Dynamic Meshes (p. 1812)), overset meshes (see Overset Meshes (p. 1206)), and/or sliding meshes (see Using Sliding Meshes (p. 1804)) to simulate flow through narrow gaps that open or close over time (such as in a valve or rotary pump). For information about using this dialog box, see Controlling Flow in Narrow Gaps for Valves and Pumps (p. 1234).

Domain → **Mesh Models** → **Gap Model...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Gap Definition

allows you to create one or more gap regions. A gap region is made up of two or more face zones, so that when one moves within a specified proximity threshold distance of another, the fluid cells within the threshold of the zones are marked, and then the flow in the cells is blocked or decelerated.

Gap Name

specifies the name of the gap region created by the **Create** button. It cannot consist solely of numbers.

Face Zones

allows you to select the two or more face zones that make up the gap region created by the **Create** button.

Proximity Threshold

defines the maximum distance between the face zones when flow blockage / deceleration occurs in the gap region created by the **Create** button.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Gap Type

defines the type of the gap region. A **Flow-Blocking** type completely blocks the flow in the gap region, as a numerical zero-mass-flux boundary condition is applied at the boundary faces of the marked region so that the marked cells no longer participate in the solution. A **Flow-Modeling** type decelerates the flow in the cells of the marked region by manipulating the viscosity or source terms.

Gap Reynolds Number

defines the intended Reynolds number in the gap region, and thus determines the level of sealing. A lower **Gap Reynolds Number** results in better sealing. It can be estimated according to Equation 6.4 (p. 1238). This drop-down list is only available when **Flow-Modeling** is selected for the **Gap Type**.

Create

creates a gap region.

Gap Management

allows you to manage existing gap regions and set advanced controls for the simulation.

Gap Regions

allows you to select gap regions that you want to manage using the other buttons in the group box.

Edit...

opens the Edit Gap Region Dialog Box (p. 5451), which you can use to edit or complete the definition of a single selected gap region.

Delete

deletes the selected gap regions.

List

prints information in the console about the selected gap regions.

Advanced Options...

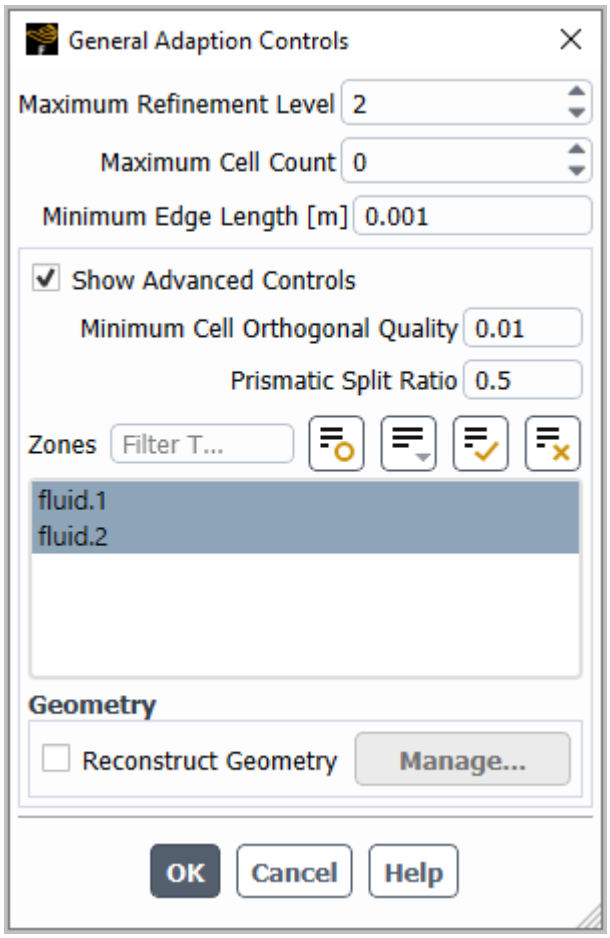
opens the Advanced Options Dialog Box (p. 5393), where you can define additional options (such as the level of solution stabilization for the simulation).

52.2.52. General Adaption Controls Dialog Box

The **General Adaption Controls** dialog box allows you to specify the general settings that Fluent uses when adapting the mesh. See Refining and Coarsening (p. 3777) for details.

Domain → **Adapt** → **Controls...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Maximum Refinement Level

controls the number of levels of refinement used to split cells during the adaption. A value of zero places no limits on the number of levels.

Maximum Cell Count

sets an approximate limit to the total cell count of the mesh. Fluent uses this value to determine when to stop marking cells for refinement. A value of zero places no limits on the number of cells. This field is only available with the PUMA adaption method.

Minimum Edge Length

sets an approximate limit to the edge length for cells that are considered for refinement. Even if a cell is marked for refinement, it will not be refined if (for 3D) its volume is less than the cube of this field or (for 2D) its area is less than the square of this field. The default value of zero places no limits on the size of cells that are refined.

Show Advanced Controls

reveals additional advanced options for adaption.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Additional Refinement Layers

allows you to specify additional cell layers for refining the mesh. This field is only available for transient simulations.

Prismatic Split Ratio

sets the split ratio for the cells as part of the **Prismatic Adaption** option in the Manual Mesh Adaption Dialog Box (p. 5515) or Automatic Mesh Adaption Dialog Box (p. 5403).

Minimum Cell Orthogonal Quality

specifies the minimum orthogonal quality that you would like Fluent to attempt to maintain in the cells that result from adaption. Note that this value is not guaranteed to be maintained, as to do so would negatively affect the solution speed. The worse quality cells will have an orthogonal quality closer to 0 and better quality cells will have an orthogonal quality closer to 1. This field is not available for 2D cases or if the hanging node adaption method is selected.

Zones

contains a list of cell zones from which you can select the zones in which to perform adaption (or marking). By default, all cell zones are selected.

Geometry

contains controls for geometry-based adaption when using the PUMA adaption method.

Reconstruct Geometry

enables geometry-based adaption, using the definitions created through the Manage Geometry-Based Adaption Dialog Box (p. 5518) to produce refined wall zones that better conform to specified auxiliary geometry definitions.

Manage...

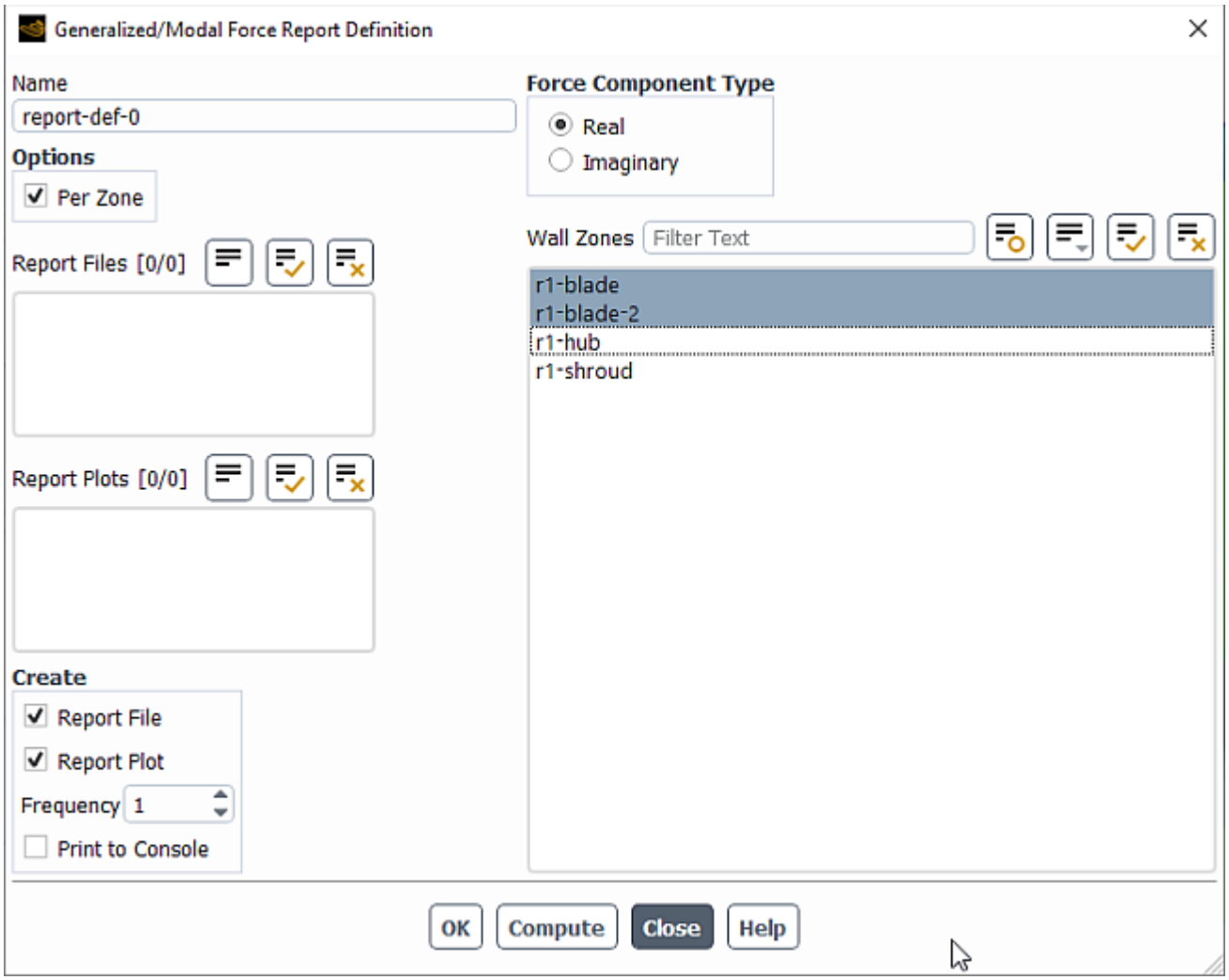
opens the Manage Geometry-Based Adaption Dialog Box (p. 5518), which you can use to specify which wall zones conform to which auxiliary geometry definitions.

52.2.53. Generalized/Modal Force Report Definition Dialog Box

For more details, see ICM Method Post-processing (p. 1999).

Solution → Reports → Definitions → New → Aeromechanics Report → Generalized/Modal Force...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name

specifies the name of the report definition.

Options

contains options for your aerodamping report definition.

Per Zone

If you are modeling multiple (main) blades, enables the force value to be calculated for each dynamic mesh zone (which encompasses one main blade and possibly accompanying splitter blades).

Report Files

lists all of the report files where you can write the report definition data.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Report Plots

lists all of the report plots where you can plot the report definition data (assuming this report has the same units as the other report definitions included in the selected report plot).

Create

Report File

creates a new report file that includes this report definition.

Report Plot

creates a new report plot that includes this report definition.

Frequency

specifies how frequently the report definition is written, plotted, or printed to the console.

Print to Console

prints the value of the report definition to the console.

Force Component Type

specifies the output **Real** or **Imaginary** force on the blade.

Wall Zones

contains a selectable list of valid walls zones for force report.

OK

creates the report definition.

Compute

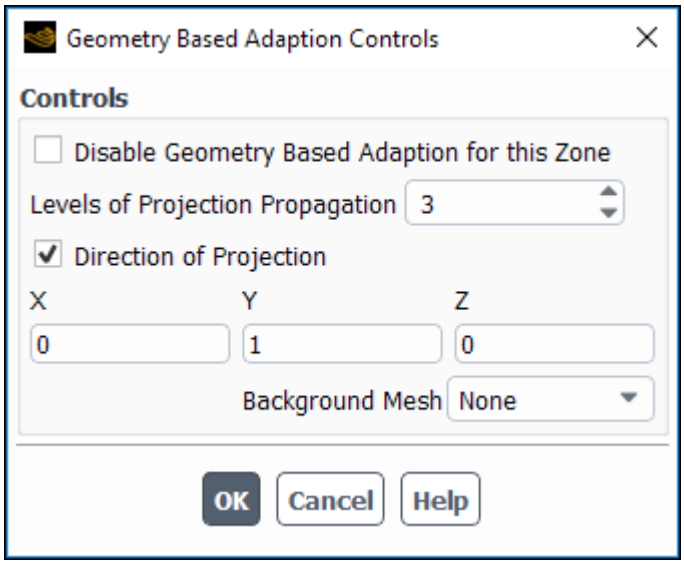
computes the value of the report definition at the selected zone(s) and prints to the console.

52.2.54. Geometry Based Adaption Controls Dialog Box

The **Geometry Based Adaption Controls** dialog box allows you to set projections. See Performing Geometry-Based Adaption with the Hanging Node Method (p. 3811) for details about the items below.

The **Geometry Based Adaption Controls** dialog box is opened from the Geometry Based Adaption Dialog Box (p. 5491).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Controls

contains parameters to define geometry based adaption.

Disable Geometry Based Adaption for this Zone

disables the geometry reconstruction for selected zones in the domain.

Levels of Projection Propagation

is the number of layers of the nodes you want to project.

Direction of Projection

allows you to specify the direction in which you want to project the nodes.

Background Mesh

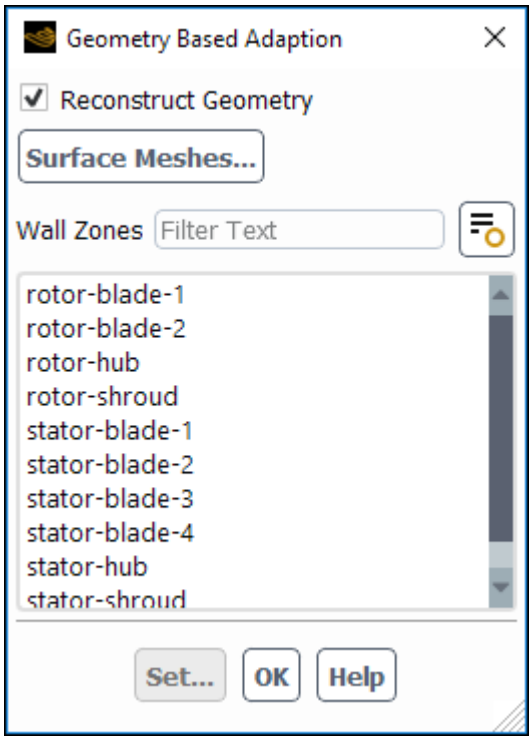
allows you to load the surface mesh as a background mesh for adaption.

52.2.55. Geometry Based Adaption Dialog Box

The **Geometry Based Adaption** dialog box allows you to reconstruct the geometry while performing boundary adaption. See Geometry-Based Adaption with the Hanging Node Method (p. 3810) for details. It is only available for 2D cases or when the hanging node adaption method is selected (through the `mesh/adapt/set/method` text command).

Domain → Adapt → More → Geometry...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Reconstruct Geometry

enables geometry based adaption parameters.

Surface Meshes...

opens the Surface Meshes Dialog Box (p. 5624).

Wall Zones

allows you to select the wall zones you want to adapt.

Set...

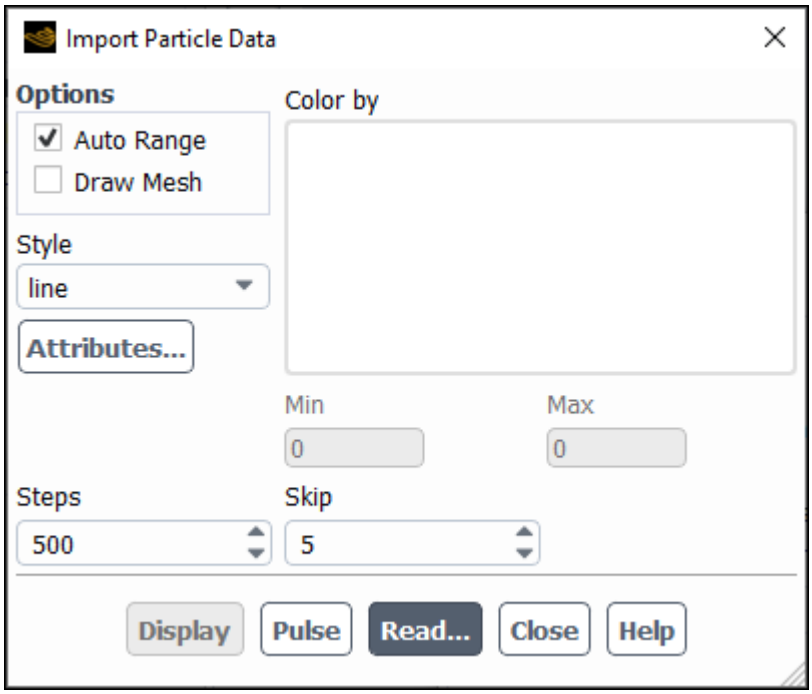
opens the Geometry Based Adaption Controls Dialog Box (p. 5490).

52.2.56. Import Particle Data Dialog Box

The **Import Particle Data** dialog box allows you to import particle history data for display purposes in transient cases. See Importing Particle Data (p. 2844) for details.

Results → Model Specific → Discrete Phase → Import Particle Data...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options

contains the check buttons that set various import particle data options.

Auto Range

toggles between automatic and manual setting of the particle data range.

Draw Mesh

toggles between displaying and not displaying the mesh. The Mesh Display Dialog Box (p. 4688) is opened when **Draw Mesh** is selected.

Style

allows you to select pathline style.

Attributes...

opens the Path Style Attributes Dialog Box (p. 5243). This allows you to modify the line width, cylinder radius or marker size.

Color by

contains a list from which you can select the scalar field to be used to color the particle data.

Min,Max

allows you to set the range when **Auto Range** is disabled.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Steps

sets the maximum number of steps a particle can advance.

Skip

allows you to “thin out” the pathlines.

Display

displays pathlines.

Pulse

animates the particle positions. The **Pulse** button will become the **Stop !** button during the animation, and you must click **Stop !** to stop the pulsing.

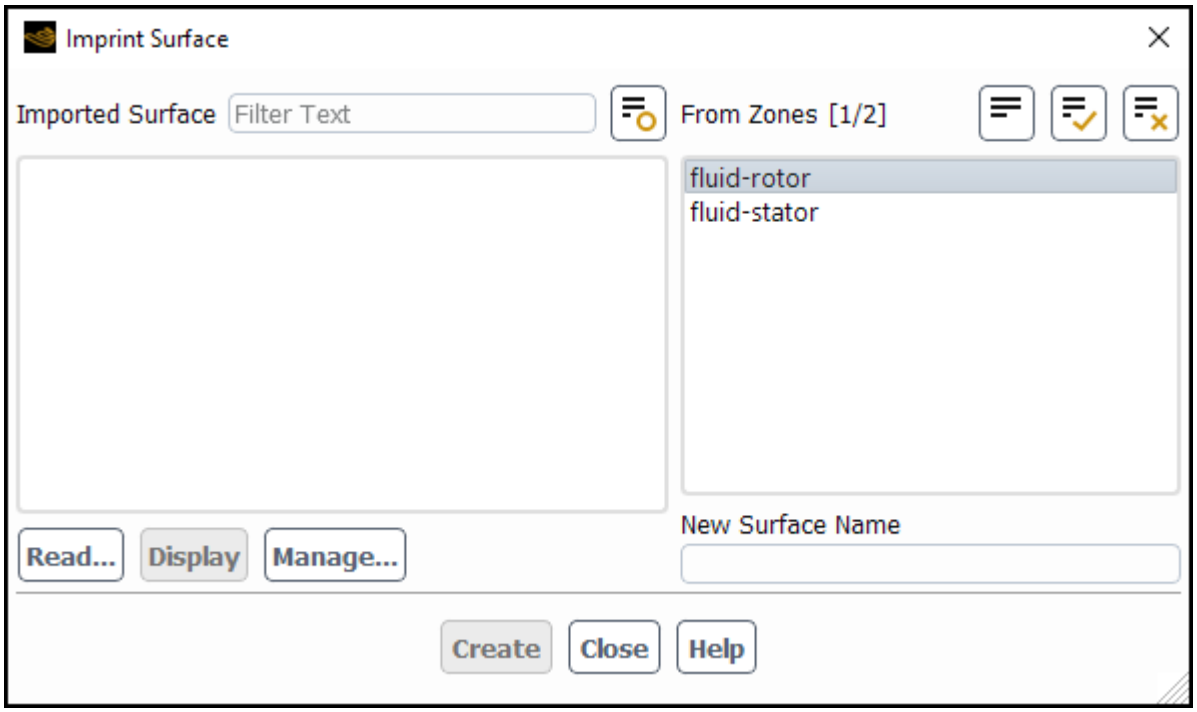
Read...

opens a file selection dialog box where you can enter a file name and a directory for the imported data.

52.2.57. Imprint Surface Dialog Box

The **Imprint Surface** dialog box allows you to define a surface on top of an existing cell zone for postprocessing purposes. See Imprint Surfaces (p. 3819) for details about the following items.

Domain → Surface → Create → Imprint...



Controls

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Imported Surface

lists the surfaces that you have imported.

Read...

opens The Select File Dialog Box (p. 931), so that you can select surface files for import.

Display

allows you to display the selected surfaces.

Manage...

opens the Surfaces Dialog Box (p. 5629), where you can rename and delete surfaces and determine their sizes.

New Surface Name

designates the name of the new surface. The default name format is `<original-name>(imprint)-<id>`.

From Zones

allows you to select the zones where you want to import/imprint the surface mesh.

Note:

If no zone is selected, then the entire domain is considered.

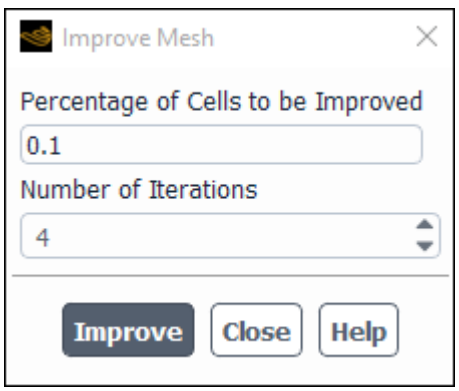
Create

creates the new surface, imprinted with the mesh of the selected cell zone.

52.2.58. Improve Mesh Dialog Box

The **Improve Mesh** dialog box controls the quality-based smoothing of the numerical mesh. See Improving the Mesh by Smoothing and Swapping (p. 1291) for details about the items below.

Domain → **Mesh** → **Improve...**



Controls

Percentage of Cells to be Improved

sets the percentage of the total cells on which improvements are attempted. Note that the cells selected for improvement are in the areas of the mesh that exhibit the lowest orthogonal quality.

Number of Iterations

defines the number of successive smoothing sweeps performed on the mesh.

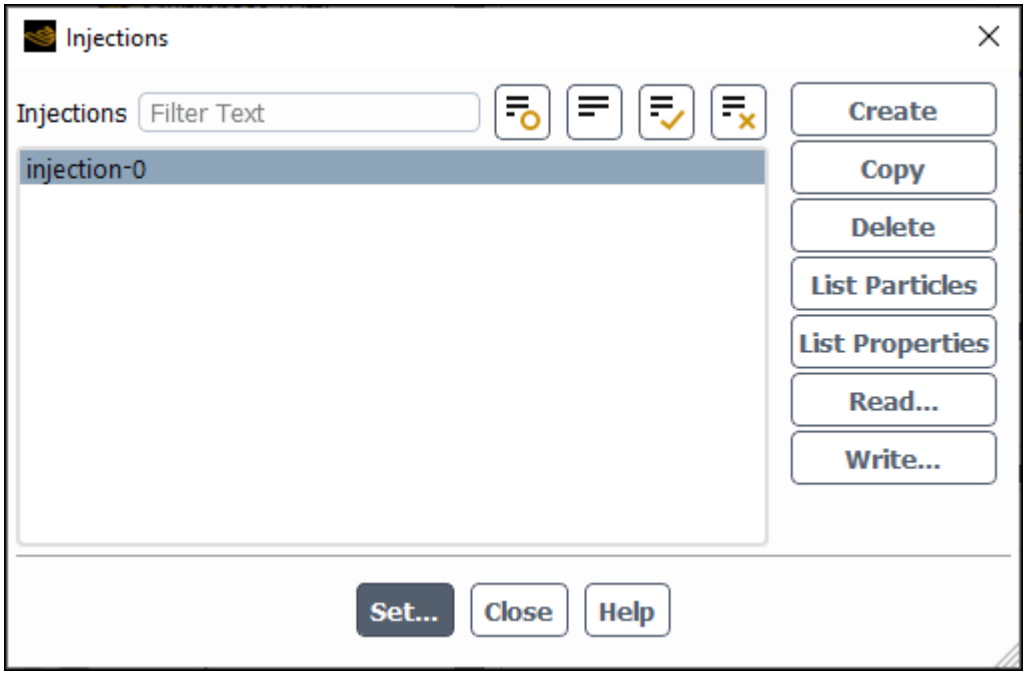
Improve

initiates the desired number of smoothing iterations.

52.2.59. Injections Dialog Box

The **Injections** dialog box allows you to create, delete, and list discrete phase injections, and access the Set Injection Properties Dialog Box (p. 5608) and the Set Multiple Injection Properties Dialog Box (p. 5621), in which you can set the properties for the injections. See Creating and Modifying Injections (p. 2753) for details.

Physics → Model Specific → Discrete Phase → Injections...



Controls

Injections

contains a list from which you can select one or more injections in order to set, copy, or modify properties, or delete or list injections.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Create

creates a new injection and opens the Set Injection Properties Dialog Box (p. 5608), in which you can set its properties.

Copy

creates a new injection with the same properties as the selected injection and opens the Set Injection Properties Dialog Box (p. 5608) where the new injection's properties can be modified.

Delete

deletes the injection(s) selected in the **Injections** list.

List Particles

lists the initial conditions for the particle streams in the injection(s) selected in the **Injections** list.

List Properties

lists the properties of the injection(s) selected in the **Injections** list.

Read...

opens The Select File Dialog Box (p. 931) where you will select the injection file to read in.

Write...

allows you to select the injection from the list and write it to a file.

Set...

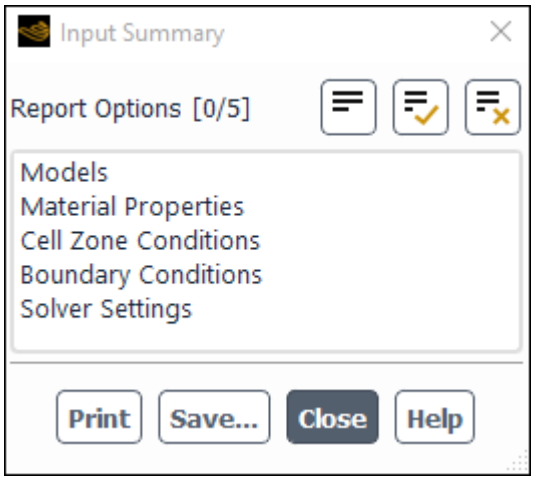
opens the Set Injection Properties Dialog Box (p. 5608) for the injection selected in the **Injections** list or the Set Multiple Injection Properties Dialog Box (p. 5621) if more than one injection is selected in the **Injections** list.

52.2.60. Input Summary Dialog Box

The **Input Summary** dialog box allows you to report the current settings for physical models, boundary conditions, material properties, and solution parameters. See Generating a Summary Report (p. 4162) for details about the items below.

Solution → **Run Calculation** → **Input Summary...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Report Options

contains a selectable list of the information that is available for the report.

Print

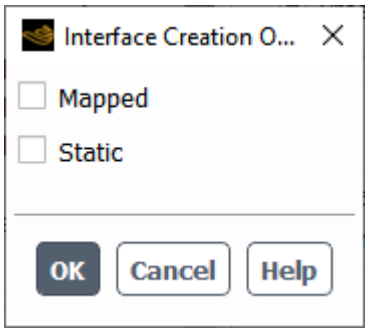
prints the selected information to the console.

Save...

opens The Select File Dialog Box (p. 931), in which you can specify the filename under which to save the output of the summary report.

52.2.61. Interface Creation Options Dialog Box

The **Interface Creation Options** dialog box allows you to define settings that you want applied to the many-to-many mesh interfaces that are created through the automatic method (for additional information, see Using a Non-Conformal Mesh in Ansys Fluent (p. 1185)). It is opened by clicking the **Options...** button in the **Boundary Zones** group box of the Mesh Interfaces Dialog Box (p. 5525). Note that this button is only available if you have disabled the default one-to-one interface creation method by using the `define/mesh-interfaces/one-to-one-pairing?` text command.



Controls

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mapped

specifies that the mapped option is applied to all of the resulting interfaces for which at least one side of the interface consists of only solid zones. Note that the mapped option will be applied regardless of the mesh quality, and the tolerance used for mapping will be that specified in the Mapped Interface Options Dialog Box (p. 5520).

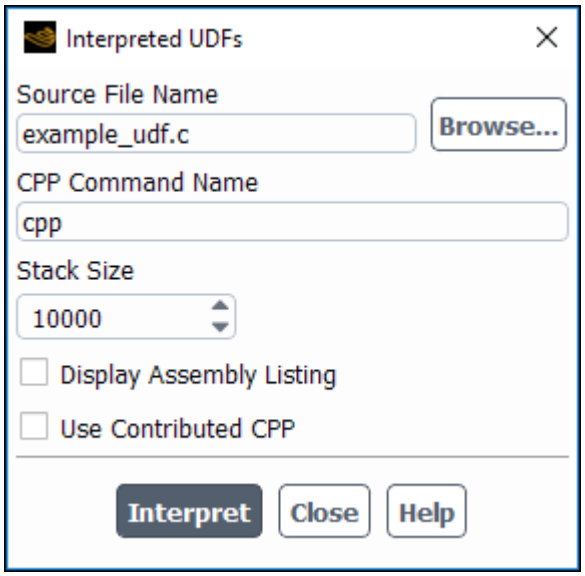
Static

reduces the memory usage and processing time for interface creation (as well as for solution), especially when you are creating interfaces using many unassigned interface zones. This option will only produce correct results if the interface zones do not move or deform relative to each other at the interfaces.

52.2.62. Interpreted UDFs Dialog Box

The **Interpreted UDFs** dialog box allows you to compile user-defined functions. See the separate Fluent Customization Manual for details.

User Defined → User Defined → Functions → Interpreted...



Controls

Source File Name

sets the name of your user-defined function.

CPP Command Name

sets the name of your C preprocessor.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Stack Size

sets the size of the stack. Keep the default **Stack Size** setting of 10000, unless the number of local variables in your function will cause the stack to overflow. In this case, set the **Stack Size** to a number that is greater than the number of local variables used.

Display Assembly Listing

indicates whether or not to display a listing of assembly language code in your console window as the function compiles.

Use Contributed CPP

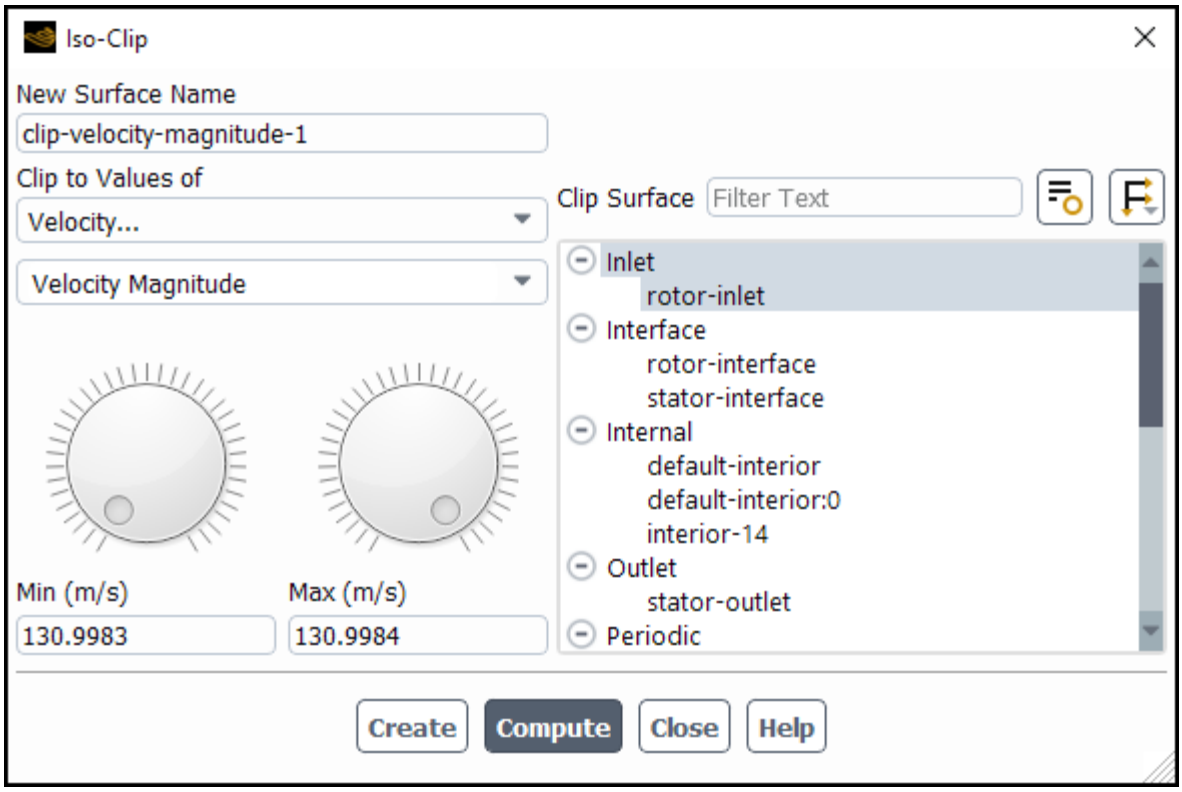
specifies the use of the C preprocessor that Ansys Fluent has supplied, instead of your own.

Interpret

interprets the specified function.

52.2.63. Iso-Clip Dialog Box

The **Iso-Clip** dialog box allows you to interactively clip surfaces. The clipped surface consists of those points on the selected surface where the scalar field values are within the specified range. See Clipping Surfaces (p. 3840) for details about the items below.



Controls

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Clip to Values of

contains a list from which you can select the scalar field to be used for clipping.

Phase

allows you to select the phase when one of the multiphase models is selected in the Multiphase Model Dialog Box (p. 4700).

Min, Max

set the minimum and maximum values in the clipping range. There are two ways you can set the **Min** and **Max**:

- You can set a value interactively by moving the indicator in the dial above the **Min** or **Max** field with the left mouse button. This will also create a temporary surface in the graphics window. Using the dial allows you to preview the clipped surfaces before defining them. Note: Even though the clipped surface is displayed, it is only a temporary surface. To clip the surface, use the **Clip** button after deciding on the minimum and maximum values.
- You can type in values in the **Min** and **Max** fields directly.

Clip Surface

contains a list of surfaces from which you can select the surface to be clipped. You must select a surface to make the **Clip** button available.

New Surface Name

designates the name of the surface to be created. The default is the concatenation of the surface type and an integer which is the new surface ID.

Clip

creates the clipped surface. (The original surface will remain unchanged.)

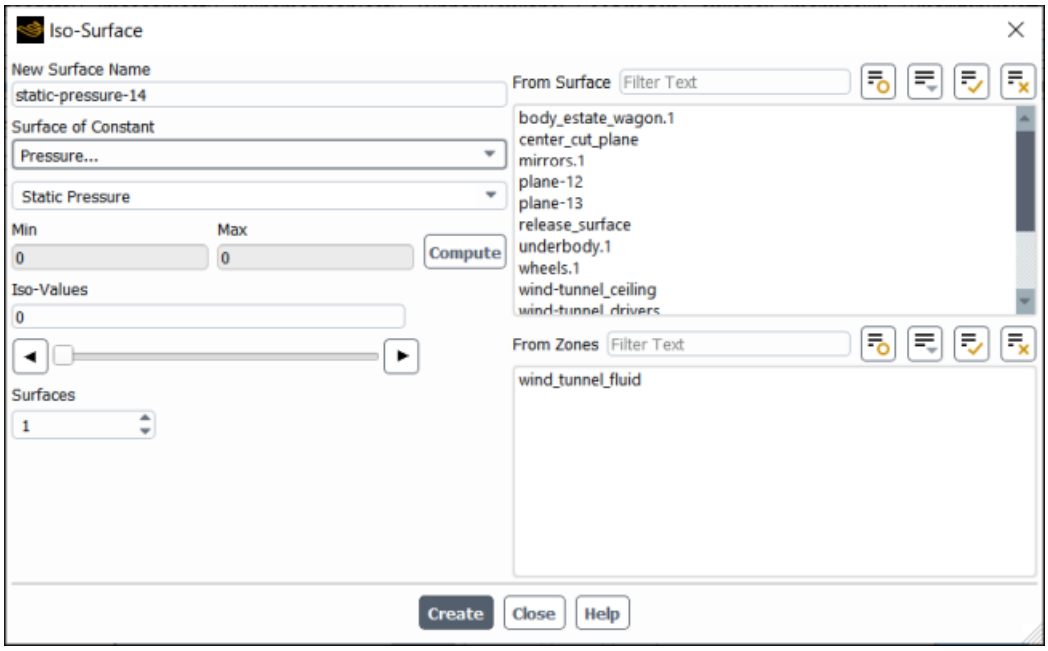
Compute

computes the minimum and maximum values of the scalar field across the domain, and displays them in the **Min** and **Max** boxes.

52.2.64. Iso-Surface Dialog Box

The **Iso-Surface** dialog box allows you to interactively create iso-surfaces. These surfaces can be iso-valued sections of an existing surface or of the entire domain. For more effective use of the slider bar, press the **Compute** button before using it. See Iso-surfaces (p. 3838) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

New Surface Name

designates the name of the surface to be created. The default is the concatenation of the scalar field name and an integer which is the new surface ID.

Surface of Constant

contains a list from which you can select the scalar field which will be used for isosurfacing.

Min

displays the minimum field value, which is computed when you click **Compute**.

Max

displays the maximum field value, which is computed when you click **Compute**.

Compute

computes the minimum and maximum of the scalar field across the domain and displays them in the **Min** and **Max** boxes.

Iso-Values

sets user-specified isovalues. There are two ways you can set the **Iso-Values**:

- You can set an isovalue interactively by moving the slider with the left mouse button. This will also create a temporary isosurface in the graphics window. Using the slider allows you to preview the isosurfaces before defining them. Note: Even though the isosurface is displayed, it is only a temporary surface. To create an isosurface, use the **Create** button after deciding on a particular isovalue.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- You can type in isovalues in the **Iso-Values** field directly, separating multiple values by white space. Multiple isovalues will be contained in a single isosurface; that is, you cannot select subsurfaces within the resulting isosurface.

From Surface

contains a list of existing surfaces from which you can select the surface to be used for isosurfacing. If you do not select a surface from the list, the isosurfacing will be performed on the entire domain.

From Zones

contains a list of cell zones from which you can select the zone for creating an isosurface.

Surfaces

lists the number of surfaces that will be created at once.

Spacing

specifies the distance between surfaces.

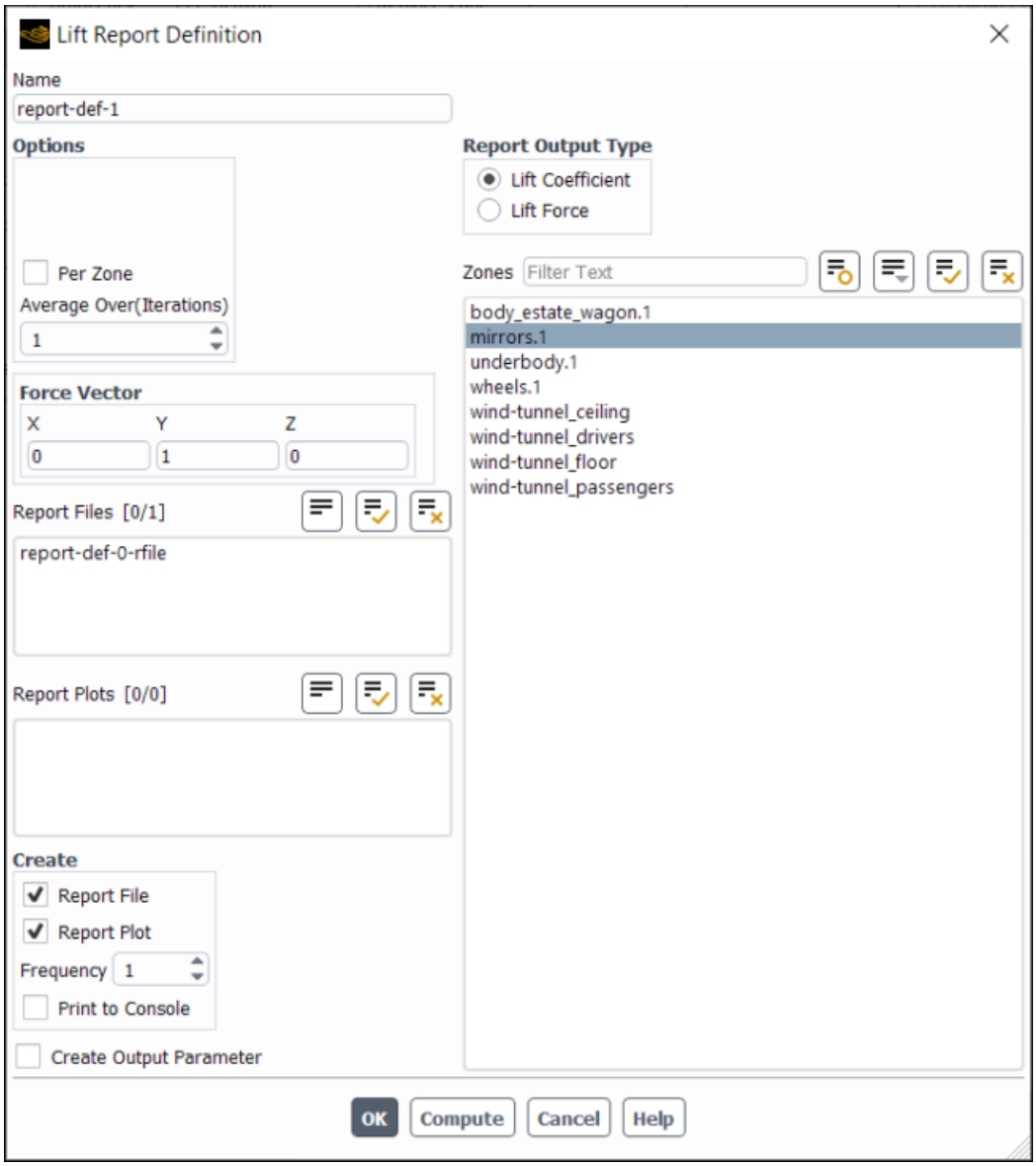
Create

creates the surface(s).

52.2.65. Lift Report Definition Dialog Box

Solution → Reports → Definitions → New → Force Report → Lift...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name

specifies the name of the report definition.

Options

Per Zone

specifies whether or not the lift for multiple walls zones should be monitored on each wall zone separately. When this option is turned off (the default), Fluent computes and monitors the total lift for all of the selected walls zones combined together.

Average Over (Time-Steps), Average Over (Iterations)

(optional) To have Fluent calculate a running average for the **Lift Report Definition** you can enter a positive integer greater than 1 (the default) for **Average Over**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Specifying a number greater than 1 means that Fluent will print, plot, and write the running average value of the selected variable instead of the current value of the same variable.

The value reported is averaged over the last N *iterations/time steps, where* N *is your specified* ***Average Over*** *value. When the iteration/time step number is lower than* N*, Fluent calculates the average of the available variable values.*

Retain Instantaneous Values

appears when **Average Over** is set to a value greater than 1. If enabled when you are creating a new report definition, it ensures that all instantaneous values are saved in the new report file that is being created for this report definition. It also results in the **Plot Instantaneous Values** option being enabled in the newly created report plot.

Force Vector

contains the components of the force vector.

X,Y,Z

are the components of the force vector along which the forces will be computed.

Report Files

lists all of the report files where you can write the report definition data.

Report Plots

lists all of the report plots where you can plot the report definition data (assuming this report has the same units as the other report definitions included in the selected report plot).

Create

Report File

creates a new report file that includes this report definition.

Report Plot

creates a new report plot that includes this report definition.

Frequency

specifies how frequently the report definition is written, plotted, or printed to the console.

Print to Console

prints the value of the report definition to the console.

Create Output Parameter

creates an output parameter for this report definition with the name `<report definition name>-op`.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Report Output Type

Lift Coefficient

specifies the report definition as a unitless quantity.

Lift Force

specifies that the report definition has units of force.

Zones

lists the zones you can select for lift coefficient or drag lift computation.

OK

creates the report definition.

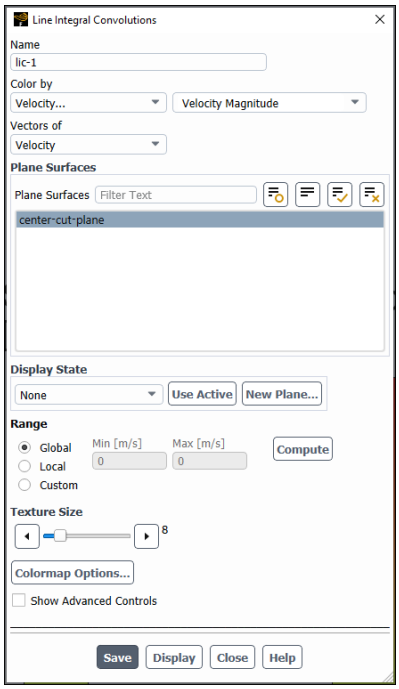
Compute

computes the value of the report definition at the selected zone(s) and prints to the console.

52.2.66. Line Integral Convolutions Dialog Box

Results → Graphics → LICs → New...

The **Line Integral Convolution** dialog box allows you to create a line integral convolutions (LIC) plot. Refer to Displaying Line Integral Convolutions (LICs) (p. 3932) for additional information.



Controls

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Name

designates the name of the LICs plot.

Color by

drop-down lists for the category and field variable used to color the plot.

Vectors of

drop-down list specifying whether the vectors represent velocity or relative velocity.

Phase

(Multiphase only) specifies the phase that the colors apply to.

Vectors of

contains a list from which you can select the vector field to be plotted.

Phase

(Multiphase only) specifies the phase for the velocity or relative velocity vectors.

Plane Surfaces

list of planar surfaces available for displaying the LICs plot.

Display State

specifies the display state that is associated with this graphics object.

drop-down list of display states

selectable list of all the existing display states associated with this case.

Use Active

creates a display state that matches the settings of the active graphics window and assigns it to this graphics object (this association is only saved after you click **Save/Display**).

New Plane...

opens the Plane Surface Dialog Box in the *Fluent User's Guide* (p. 5580).

Range

specifies how the range is computed.

Global

indicates that the range is computed based on the entire domain.

Local

indicates that the range is computed based on the selected surface(s).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Custom

allows you to manually specify the **Min** and **Max** fields.

Compute

computes the minimum and maximum values of the field variable coloring the LICs. If **Global** is selected, the range is computed based on the entire domain, if **Local** is selected, the range is computed based on the selected surface(s).

Texture Size

controls the size (in 100's of pixels) of the texture image used in the convolution.

Colormap Options...

opens the Colormap Dialog Box in the *Fluent User's Guide* (p. 5280), allowing you to customize the colormap for this graphics object.

Show Advanced Options

exposes additional advanced controls.

Intensity Factor

scales the intensity levels of the convolution results up or down. You can use this to control the brightness or contrast of the resulting image.

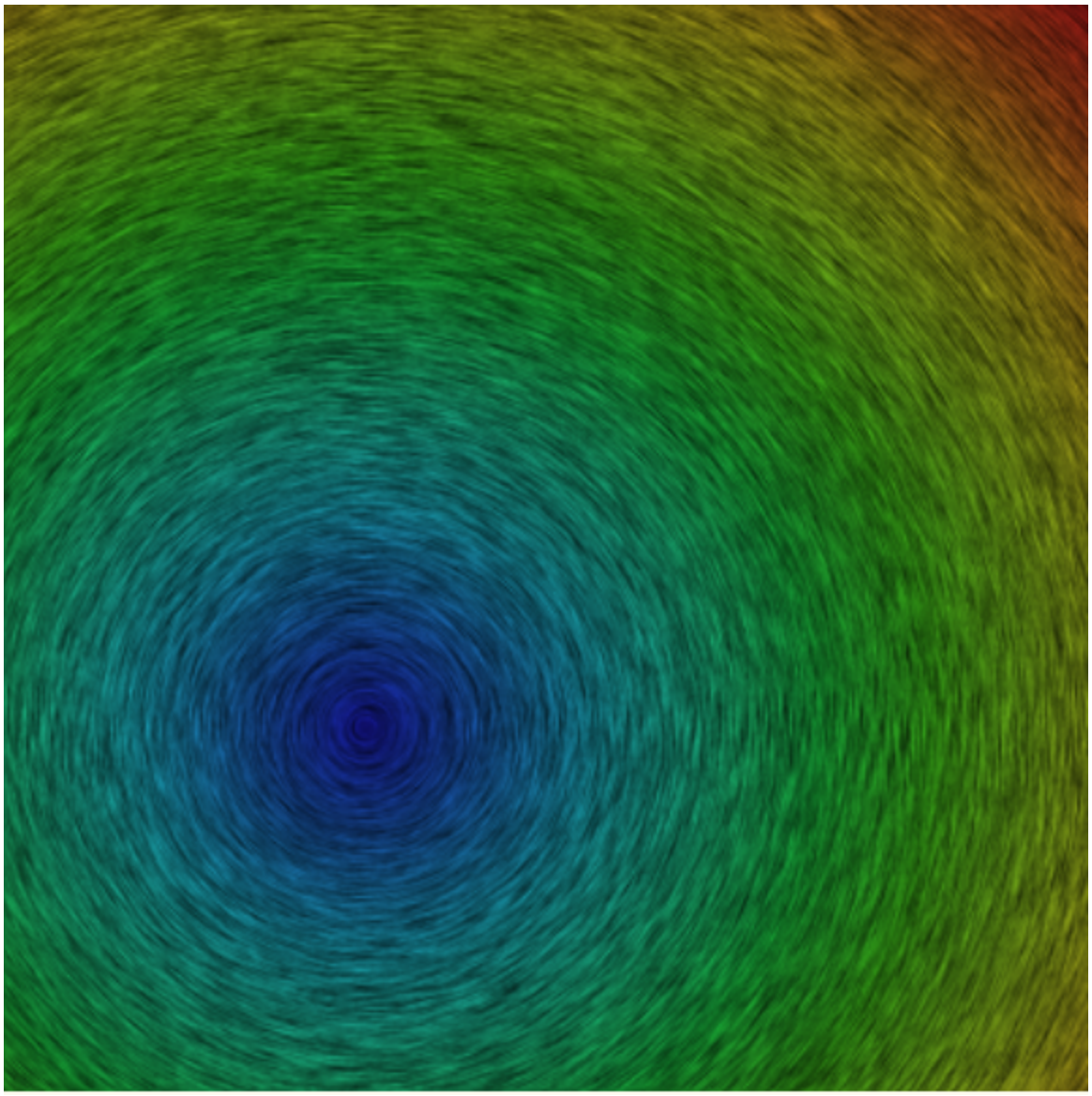
Image Filter

applies an image filter to the convolutions.

- **None**—no filter applied.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

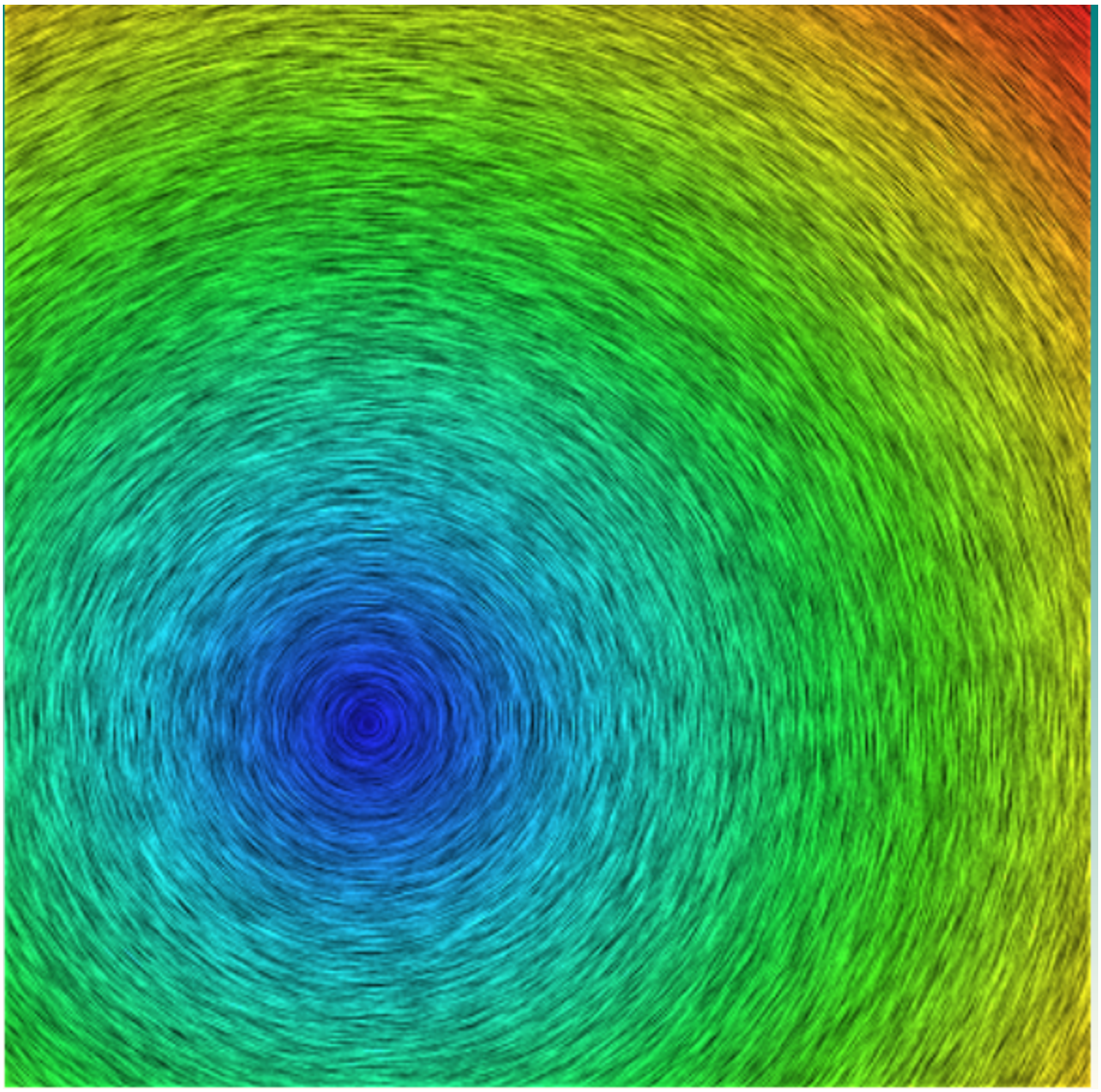
Figure 52.1: Example with the Image Filter Set to None



- **Mild Sharpen**—provides a mild increase in the focus or contrast of the texture image.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

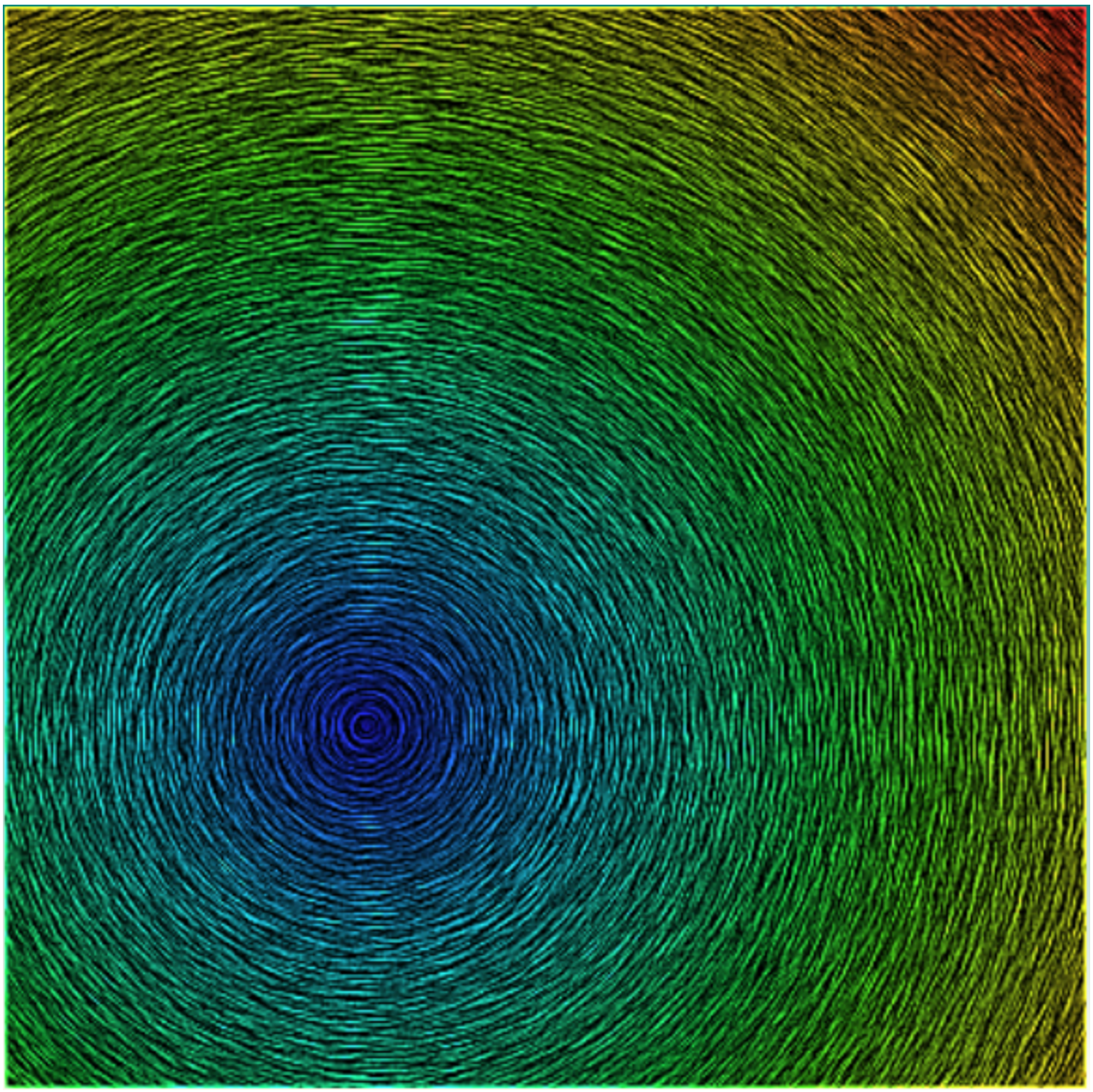
Figure 52.2: Example with the Image Filter Set to Mild Sharpen



- **Strong Sharpen**—provides a stronger increase in the focus or contrast of the texture image.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

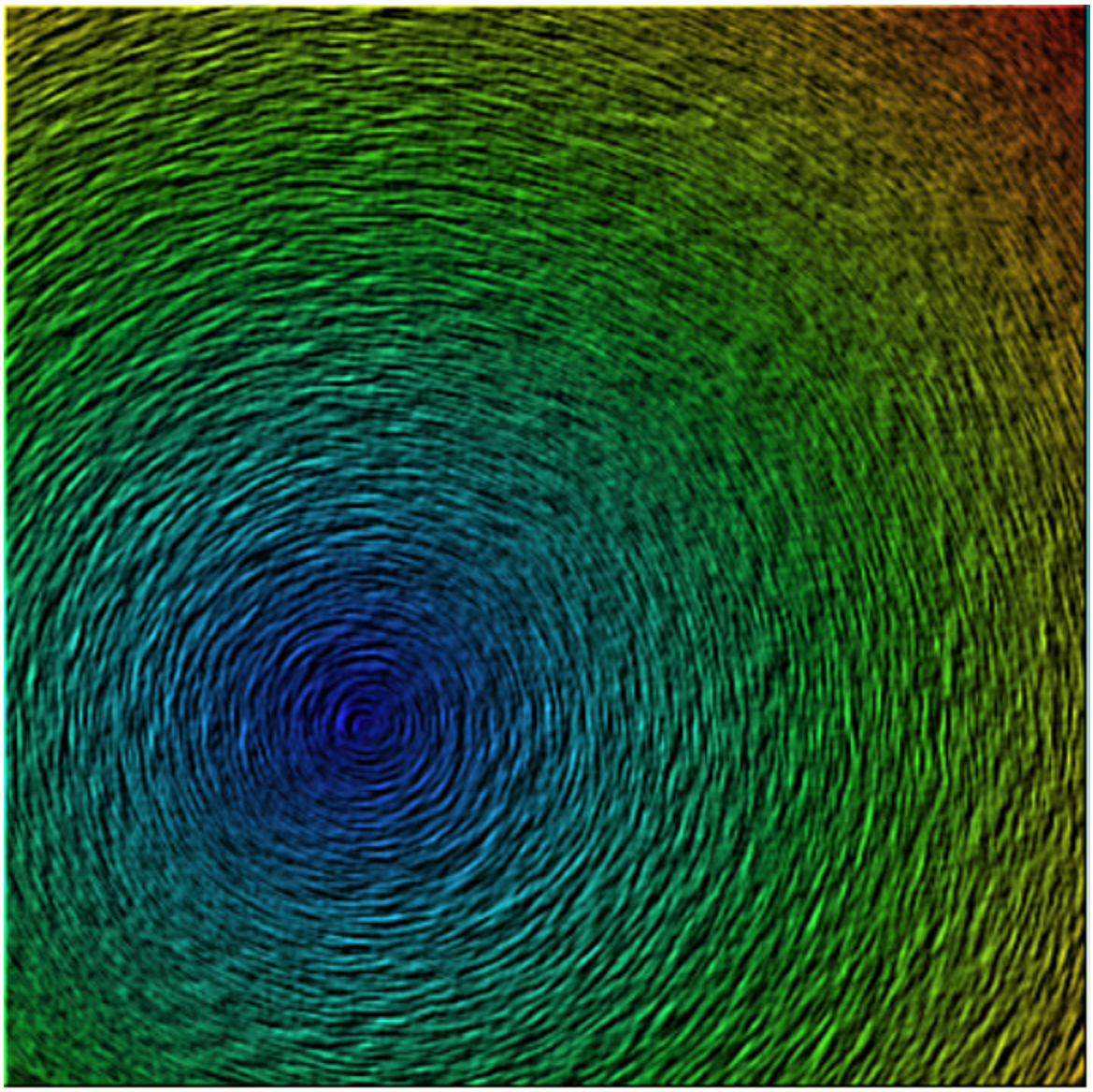
Figure 52.3: Example with the Image Filter Set to Strong Sharpen



- **Mild Emboss**—creates shadowing and highlights to give edges a raised effect on features in the image.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

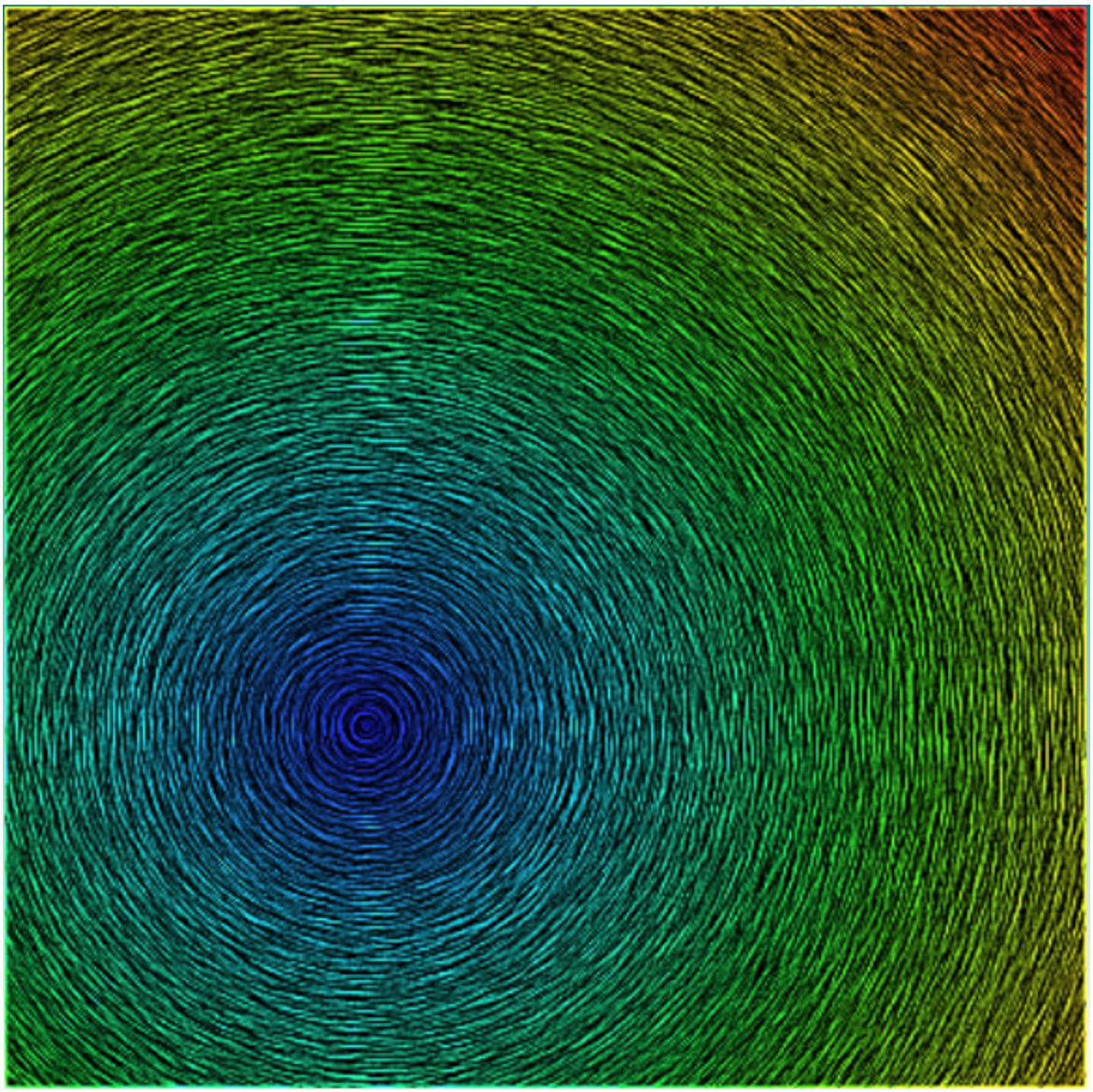
Figure 52.4: Example with the Image Filter Set to Mild Emboss



- **Strong Emboss**—a stronger version of the **Mild Emboss** option. This will also result in the loss of color in areas with little definition before the filter was applied.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 52.5: Example with the Image Filter Set to Strong Emboss



Normalize

normalizes the vector field magnitude during the integration. The effectiveness of the convolution can be very sensitive to the vector field magnitude, so normalization is typically done to maintain a clear definition of the flow field and in particular, the location of singularities.

Save

saves the settings of the plot as they currently appear in the dialog box.

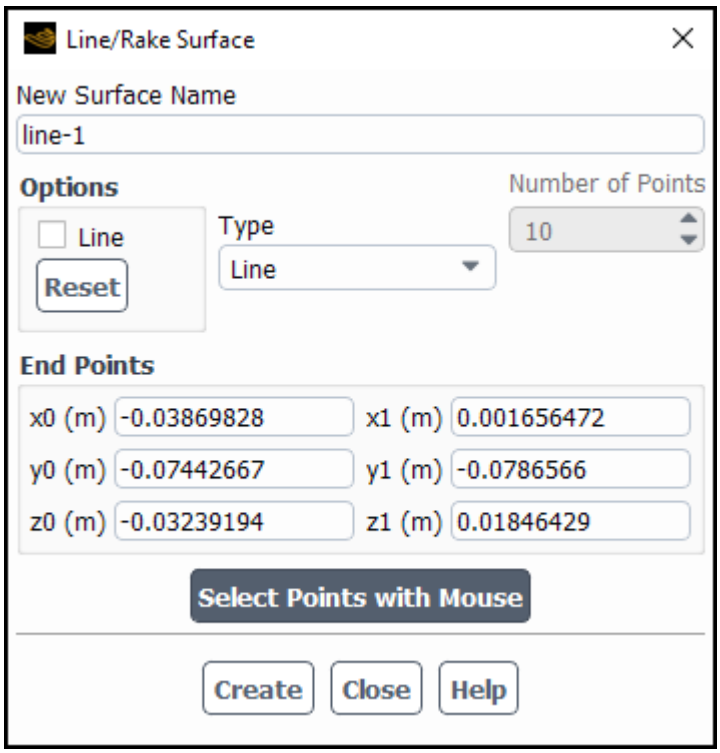
Display

displays a plot with the settings shown in the dialog box. *Note that this button does not save the settings, so if you modify settings and display the updated plot, you must click* ***Save*** *to preserve these settings before closing the dialog box.*

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

52.2.67. Line/Rake Surface Dialog Box

The **Line/Rake Surface** dialog box allows you to interactively create line and rake surfaces. A rake is a linear, uniform distribution of points between two endpoints. See Line and Rake Surfaces (p. 3826) for details about the items below.



Controls

Options

contains options related to the line tool. See Using the Line Tool (p. 3828) for details about using this feature.

Line Tool

enables the line tool.

Reset

resets the line tool to its default position.

Type

selects line or rake as the surface to be created.

Number of Points

defines the number of points in the rake surface (inactive for line surfaces).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

End Points

designates the coordinates of the first point (**x0**, **y0**, **z0**) and the last point (**x1**, **y1**, **z1**).

Select Points with Mouse

allows you to select endpoints with the mouse. You can select endpoints by clicking on locations in the active window with the mouse-probe button. (See Controlling the Mouse Button Functions (p. 4020) for information about setting mouse button functions.)

New Surface Name

designates the name of the new surface. The default is the concatenation of the surface type and an integer which is the new surface ID.

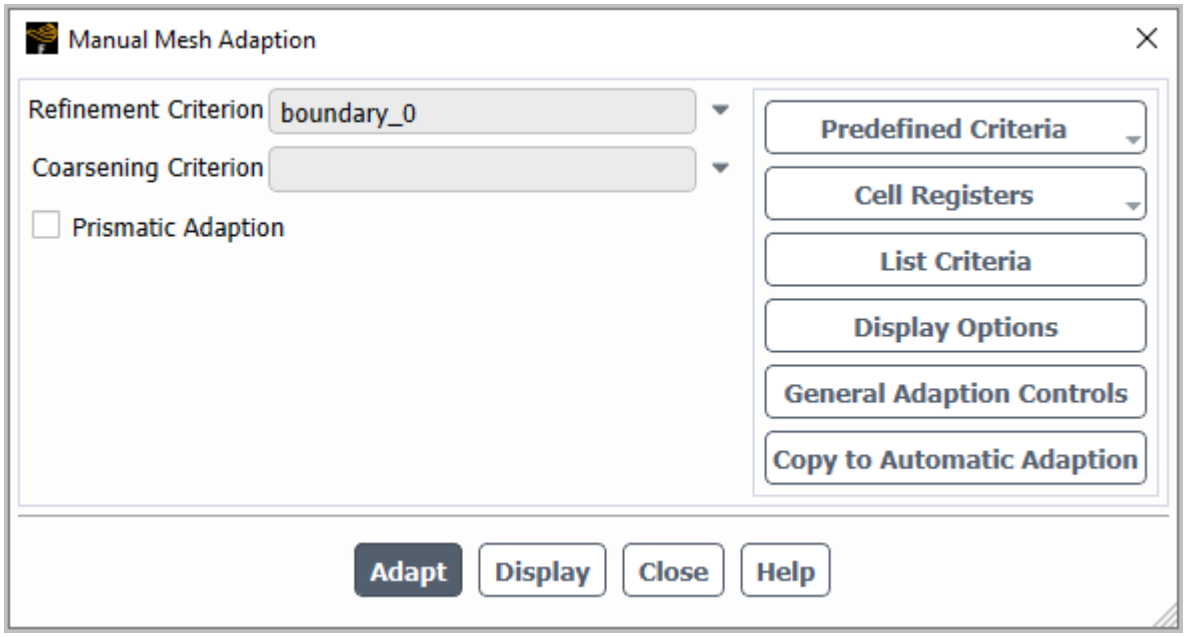
Create

creates the surface.

52.2.68. Manual Mesh Adaption Dialog Box

The **Manual Mesh Adaption** dialog box allows you to manually adapt your mesh. See Refining and Coarsening (p. 3777) for details.

Domain → **Adapt** → **Manual...**



Controls

Refinement Criterion

specifies the cell register or expression that Fluent will use during the calculation to determine which cells to refine.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Coarsening Criterion

specifies the cell register or expression that Fluent will use during the calculation to determine which cells to coarsen.

Prismatic Adaption

specifies that prismatic cells are adapted anisotropically. This option is only available with the PUMA adaption method.

Predefined Criteria

allows you to select from a list of commonly used criteria for adapting the mesh (for a description of the list, see Predefined Criteria for Adaption (p. 3784)); if one is selected, Fluent will automatically generate the necessary cell registers and/or expressions for refinement and coarsening, and define the settings in the **Manual Mesh Adaption** dialog box and General Adaption Controls Dialog Box (p. 5486) accordingly. Depending on your selection, you may be prompted to complete the setup by defining fields in the Adaption Criteria Settings Dialog Box (p. 5386).

Cell Registers

allows you to create new cell registers and manage existing ones, which may then be selected from the **Refinement Criterion** and/or **Coarsening Criterion** drop-down lists.

List Criteria

prints to the console the number of cells marked for refinement, the number of cells marked for coarsening, and the number marked for both operations.

Display Options

opens the Display Options - Adaption Dialog Box (p. 5435).

General Adaption Controls

opens the General Adaption Controls Dialog Box (p. 5486), where you can specify the general settings that Fluent uses when adapting the mesh.

Copy to Automatic Adaption

copies the settings to the Automatic Mesh Adaption Dialog Box (p. 5403), where you can create adaption criteria which can be applied during the solution.

Adapt

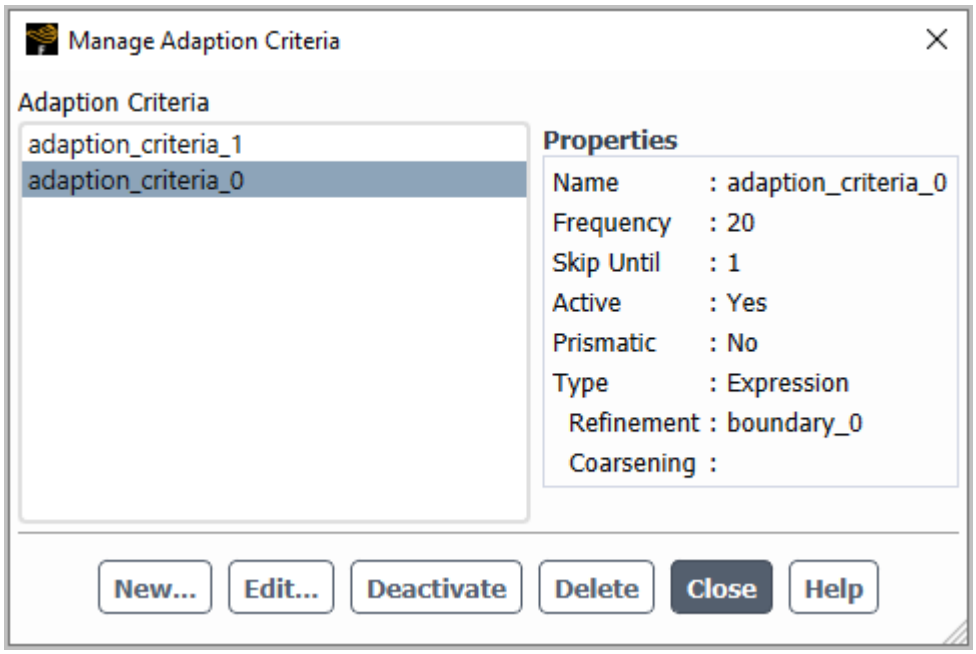
performs adaption based on the settings in this dialog box.

52.2.69. Manage Adaption Criteria Dialog Box

The **Manage Adaption Criteria** dialog box allows you to manage criteria for automatic mesh adaption. See Refining and Coarsening (p. 3777) for details.

Domain → **Adapt** → **Manage...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Adaption Criteria

lists the defined criteria for automatic adaption during a calculation.

Properties

displays information about the criterion selected in the **Adaption Criteria** list.

New...

opens the Automatic Mesh Adaption Dialog Box (p. 5403), where you can create new automatic adaption criteria.

Edit...

opens the Automatic Mesh Adaption Dialog Box (p. 5403), where you can edit the criterion selected in the **Adaption Criteria** list.

Deactivate

deactivates the criterion selected in the **Adaption Criteria** list, so that it is not applied during the calculation.

Activate

activates the criterion selected in the **Adaption Criteria** list, so that it is applied during the calculation.

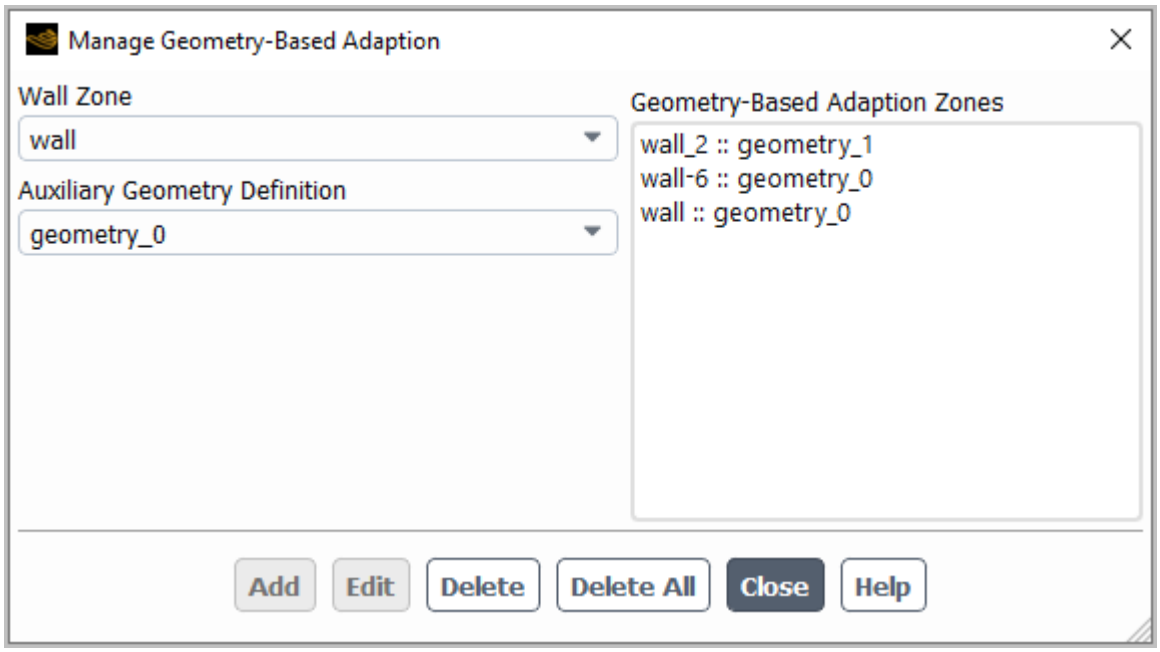
Delete

deletes the criterion selected in the **Adaption Criteria** list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

52.2.70. Manage Geometry-Based Adaption Dialog Box

The **Manage Geometry-Based Adaption** dialog box allows you to manage definitions for geometry-based adaption with the PUMA adaption method. It is opened by clicking the **Manage...** button under the **Show Advanced Controls** option in the General Adaption Controls Dialog Box (p. 5486). See Refining and Coarsening (p. 3777) for details.



Controls

Wall Zone

allows you to select the wall zone to which you want geometry-based adaption applied.

Auxiliary Geometry Definition

allows you to select the auxiliary geometry definition to which you want the selected **Wall Zone** to conform. For details on creating auxiliary geometry definitions, see Managing Auxiliary Geometry Definitions (p. 1791).

Geometry-Based Adaption Zones

lists the pairings of wall zones and auxiliary geometry definitions that you have created through the **Add** button. You can make a single selection from this list and click the **Delete** button if you want delete the pairing.

Add

creates a pairing (listed under **Geometry-Based Adaption Zones**) of the selected **Wall Zone** and **Auxiliary Geometry Definition**.

Edit

allows you to revise which **Auxiliary Geometry Definition** is paired with a given **Wall Zone**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Delete

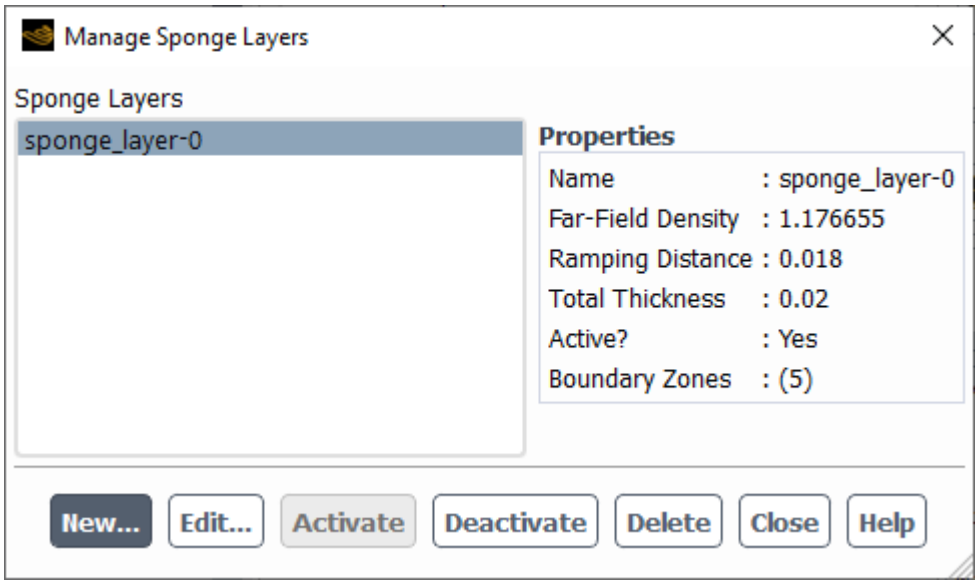
deletes the selected pairing under **Geometry-Based Adaption Zones**.

Delete All

deletes all of the pairings under **Geometry-Based Adaption Zones**.

52.2.71. Manage Sponge Layers Dialog Box

The **Manage Sponge Layers** dialog box allows you to define sponge layers at boundary zones (so that the density is blended from the value calculated by the solver to user-specified far-field values), in order to eliminate non-physical pressure wave reflections from boundary zones in transient simulations of compressible flows with the pressure-based solver. For further details, see Sponge Layers (p. 2667).



Controls

Sponge Layers

lists the existing sponge layer definitions.

Properties

lists the properties of the selected sponge layer in the **Sponge Layers** list. *This list is blank when a sponge layer is not selected.*

New...

opens the Sponge Layer Dialog Box (p. 5622), allowing you to create a new sponge layer definition, which by default is active.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Edit...

opens the Sponge Layer Dialog Box (p. 5622), allowing you to edit the definition selected in the **Sponge Layers** list.

Activate

activates the selection in the **Sponge Layers** list, so that it is used when the calculation is run.

Deactivate

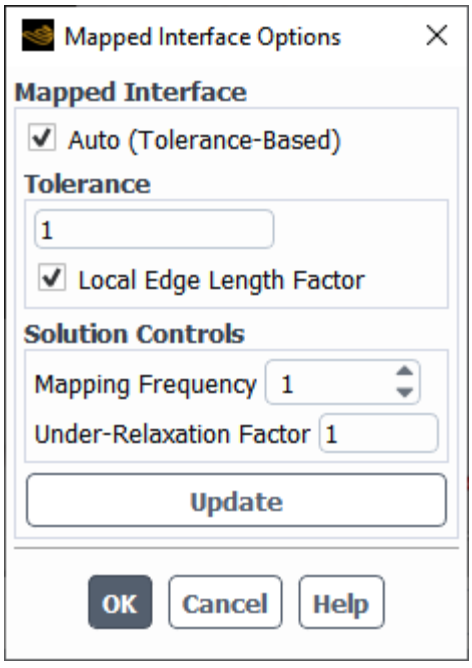
deactivates the selection in the **Sponge Layers** list, so that it is not used when the calculation is run.

Delete

deletes the definition that is selected in the **Sponge Layers** list.

52.2.72. Mapped Interface Options Dialog Box

The **Mapped Interface Options** dialog box allows you to update which mesh interfaces use the mapped option and the related settings (see The Mapped Option (p. 1178) for additional information). It is opened by selecting a many-to-many mesh interface and clicking the **Options...** button in the **Mesh Interfaces** group box of the Mesh Interfaces Dialog Box (p. 5525).



Controls

Mapped Interface

contains options related to mapped interface specification.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Auto (Tolerance-Based)

specifies that all mesh interfaces that have interface zones that penetrate each other are automatically defined as **Mapped** interfaces (provided that at least one side of each interface consists only of solid zones).

Tolerance

Local Edge Length Factor

(when enabled) allows you to specify the multiplier that Fluent uses with the smallest edge length of the interface zones to calculate the tolerance for mapping. Disabling **Local Edge Length Factor** allows you to specify an absolute value for the tolerance. Note that this does not overwrite any locally specified tolerance in the **Edit Mesh Interfaces** dialog box.

Solution Controls

Mapping Frequency

specifies how often calculation values are interpolated across mapped interfaces.

Under-Relaxation Factor

allows you to reduce the rate of change per iteration of physical quantities across mapped interfaces to aid with convergence. For additional information on setting under-relaxation factors, see Setting Under-Relaxation Factors (p. 3589).

Update

applies the selection(s) made in the **Mapped Interface Options** dialog box. This button is grayed out until you make enable / disable the **Auto (Tolerance-Based)** option; it also returns to being grayed out once it is clicked.

Note:

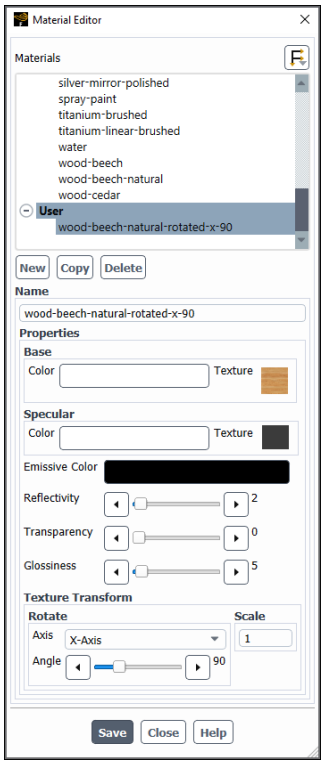
Enabling the **Auto (Tolerance-Based)** option and clicking **Update** applies these selections globally for all relevant interfaces. Once you click **Update** you can return to the **Mesh Interfaces** dialog box to modify local settings for individual interfaces.

52.2.73. Material Editor Dialog Box

The **Material Editor** dialog box allows you create new materials for realistic rendering. *Refer to Creating Custom Realistic Materials (p. 3889) for additional information on using the **Material Editor** dialog box.*

View → **Graphics** → **Material Editor...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Materials

lists the available realistic materials.

New

creates a new material with default properties.

Copy

creates a new material with the properties of the selected **Material**.

Delete

removes the material from the **Materials** list and from any mesh objects referencing that material. *You can only delete user-defined materials; the default materials cannot be deleted.*

Properties

lists the editable properties that define the material's appearance.

Base Color

specifies the bulk color of the material. **Texture** is a non-editable field that indicates the base texture associated with the material, if there is one. *You can hover over the texture image to see an enlarged picture of the texture.*

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Specular Color

specifies the highlight color seen due to lighting effects. **Texture** is a non-editable field that indicates the specular texture associated with the material, if there is one. *You can hover over the texture image to see an enlarged picture of the texture.*

Emissive Color

specifies the color emitted from the material.

Reflectivity

specifies how reflective the material appears.

Transparency

specifies how transparent the material appears.

Glossiness

specifies how glossy the material appears.

Texture Transform

provides options for modifying the appearance of the material texture.

Rotate

controls the orientation of the texture.

Axis

specifies the axis the texture will be rotated about.

Angle

controls the angle (in degrees) that the texture is rotated about the selected axis.

Scale

controls the scaling of the texture. A larger value increases the texture scaling, similarly to zooming in on the texture. *Valid inputs are in the range 0.0004 to 4.*

Save

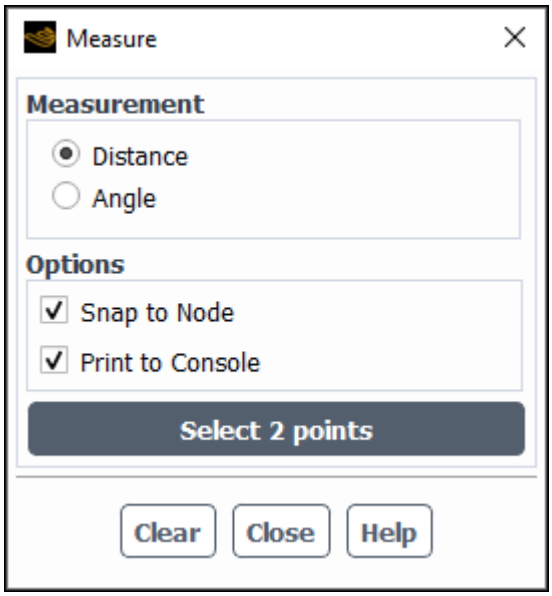
saves the current settings of any custom material.

52.2.74. Measure Dialog Box

The **Measure** dialog box allows you measure objects displayed in the graphics window. You can measure the distance between 2 points or the angle formed by 3 points. See Merging Zones (p. 1261) for details.

View → Graphics → Measure...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Measurement

lists the two types of measurements that you can make:

Distance

indicates that you will be measuring the distance between two points in the graphics window when you click **Select 2 Points**. *You can use the left-mouse-button (LMB) or right-mouse-button (RMB) to select the points.*

Angle

indicates that you will be measuring the angle between three points in the graphics window when you click **Select 3 Points**. *You can use the left-mouse-button (LMB) or right-mouse-button (RMB) to select the points.*

Options

contains controls for whether the points clicked in the interface automatically snap to the node, and whether the measurement details are printed to the console.

Snap to Node

indicates that measurement clicks in the graphics window are snapped to the nearest node, when enabled. When disabled, the measurements are taken at the location where the mouse is clicked.

Print to Console

indicates that the measurement details are printed to the console, when enabled.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

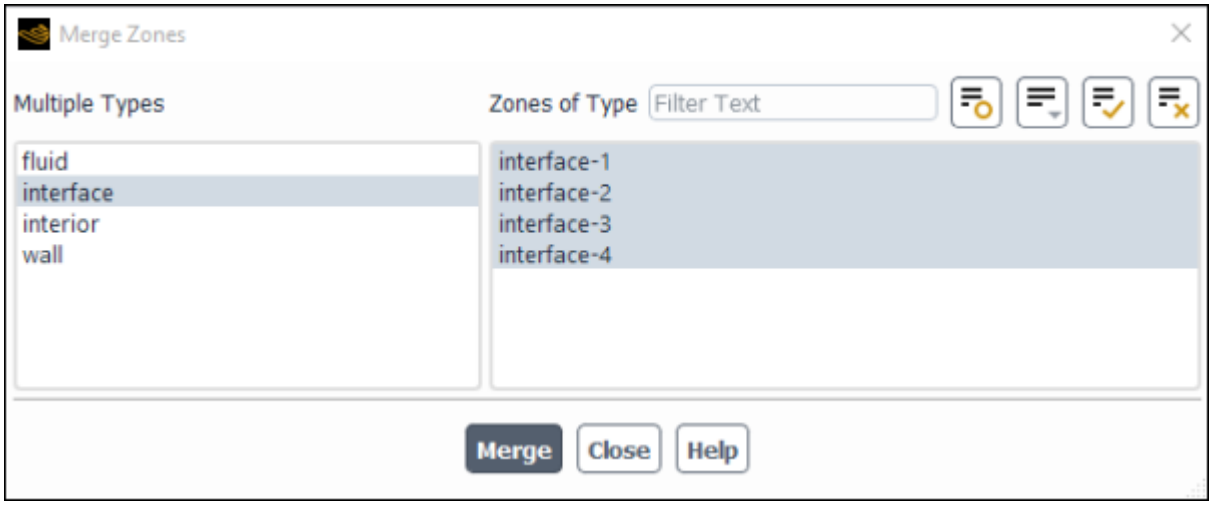
Select 2 Points | Select 3 Points

activates the measurement tool so that you can select the points for measuring distance (**Select 2 Points**) or angle (**Select 3 Points**).

52.2.75. Merge Zones Dialog Box

The **Merge Zones** dialog box allows you to merge multiple zones of the same type into a single zone. See Merging Zones (p. 1261) for details.

Domain → Zones → Combine → Merge...



Controls

Multiple Types

contains a selectable list of the zone types for which you can merge multiple zones. You select a type and the corresponding zones of that type will be displayed in the **Zones of Type** list.

Zones of Type

contains a list from which you can select two or more zones to be merged into a single zone. The zones are of the type selected in the **Multiple Types** list.

Merge

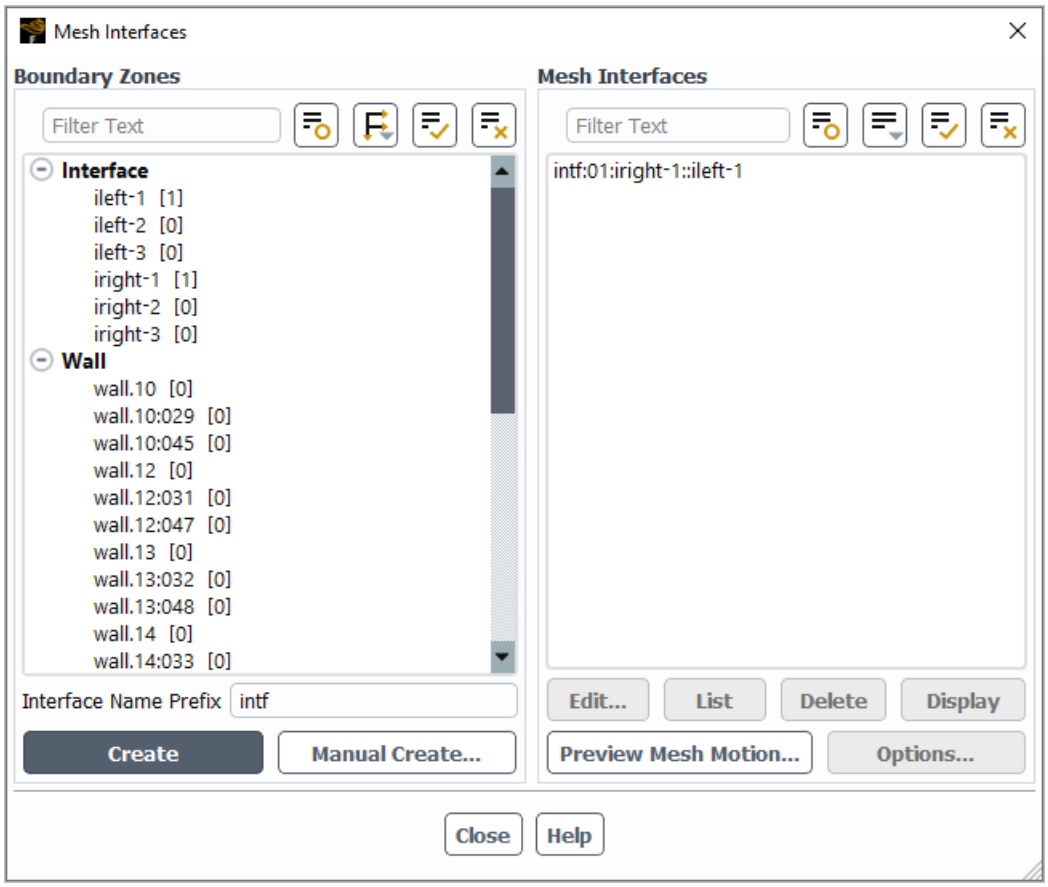
merges the selected zones into a single zone. If your case file has dynamic zones or mesh interfaces, the Warning Dialog Box (p. 5662) will open and require your input prior to the merge.

52.2.76. Mesh Interfaces Dialog Box

The **Mesh Interfaces** dialog box allows you to automatically or manually create mesh interfaces for use with sliding meshes (see Using Sliding Meshes (p. 1804)) or multiple reference frames (see Mesh Setup for a Multiple Moving Reference Frame (p. 1781)), or for meshes with non-conformal boundaries (see Non-Conformal Meshes (p. 1170)). For information about using this dialog box, see Using a Non-Conformal Mesh in Ansys Fluent (p. 1185).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Domain → Interfaces → Mesh...



Controls

Boundary Zones

allows you to create new mesh interfaces using the automatic or manual method. Note that the selection list at the top of the group box (which lists the boundary zones available for selection, along with a number that indicates the number of mesh interfaces that zone is a part of) is only used as part of the automatic method; when a selection is made, the **Mesh Interfaces** of which it is a part are also highlighted.

Interface Name Prefix

specifies the prefix used in the names of all the mesh interfaces created automatically through the **Create** button.

Options...

opens the Interface Creation Options Dialog Box (p. 5498), so you can define settings that you want applied to the mesh interfaces that are created automatically through the **Create** button. This button is only available if you have disabled the default one-to-one interface creation method by using the `define/mesh-interfaces/one-to-one-pairing?` text command.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Create

creates mesh interfaces using the automatic method, so that Fluent is responsible for determining which of the selected **Boundary Zones** can be grouped together to form the two sides of one or more mesh interfaces. Note that mesh interfaces that use the periodic or periodic repeats option cannot be created using this button, but must instead be created by disabling the one-to-one interface creation method and using the **Manual Create...** button.

Manual Create...

opens additional dialog boxes that allow you to manually create mesh interfaces with methods or options that are not available in the **Mesh Interfaces** dialog box. When the default one-to-one interface creation method is enabled, this button opens the Create Mesh Interfaces (p. 5418), so that you can easily create many mesh interfaces between boundary zones that do not currently overlap (for example, for a sliding mesh simulation). When the one-to-one interface creation method is disabled (through the `define/mesh-interfaces/one-to-one-pairing?` text command), this button opens the Create/Edit Mesh Interfaces Dialog Box (p. 5419), so you can create mesh interfaces that use the periodic or periodic repeats option.

Turbo Create...

opens the Create/Edit Turbo Interfaces Dialog Box (p. 5423), so you can create turbo interfaces. This button is only available if you have disabled the default one-to-one interface creation method by using the `define/mesh-interfaces/one-to-one-pairing?` text command or, before creating any one-to-one mesh interfaces, by enabling the **Enable** option in the **Turbo Model** group of the **Domain** ribbon tab.

Mesh Interfaces

allows you to manage existing mesh interfaces. Note that many of the buttons in this group box are not accessible unless you make a selection from the list at the top. This list enumerates the mesh interfaces that have been created; when a selection is made, the **Boundary Zones** that comprise it are also highlighted.

Edit...

opens the **Edit Mesh Interface** dialog box or (for a many-to-many mesh interface) the Edit Mesh Interfaces Dialog Box (p. 5454), so you can edit the name or (for a many-to-many mesh interface) the name, list of interface zones assigned to the interface, and/or interface options.

List

prints information about the selected mesh interface(s) in the console.

Delete

deletes the selected mesh interfaces.

Display

allows you to display the selected mesh interface in the graphics window. Note that after you have selected a mesh interface, you can modify what zones are displayed by selecting / deselecting zones in the **Boundary Zones** list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Preview Mesh Motion...

opens the Mesh Motion Dialog Box (p. 5151).

Options...

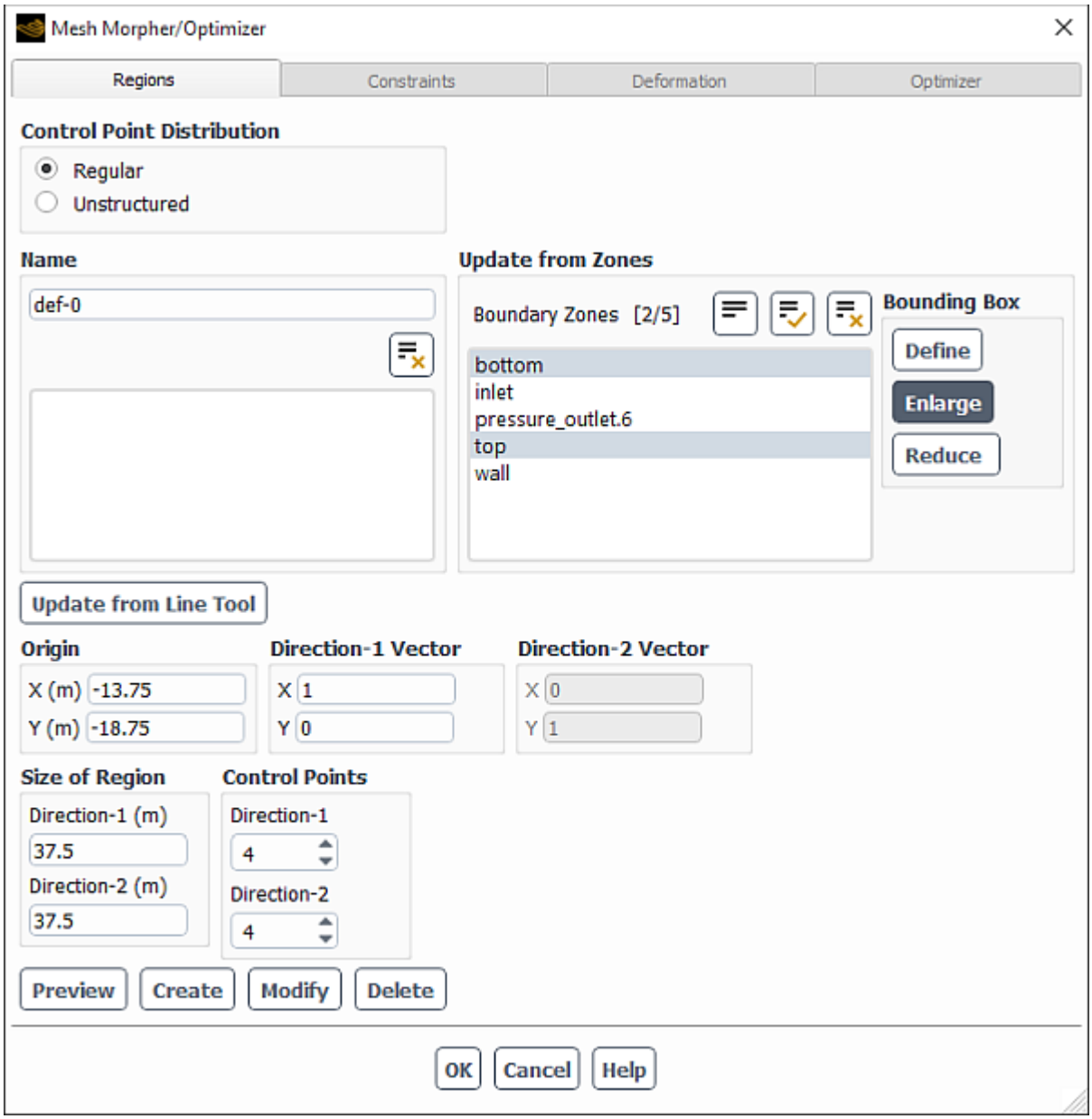
opens the Mapped Interface Options Dialog Box (p. 5520), where you can update which mesh interfaces use the mapped option and set global parameters for the mapped mesh interfaces. This button is only available if you selected a many-to-many mesh interface from the selection list in the **Mesh Interfaces** group box.

52.2.77. Mesh Morpher/Optimizer Dialog Box

The **Mesh Morpher/Optimizer** dialog box allows you to use the mesh morpher/optimizer to deform the mesh in order to solve shape optimization problems. You can deform the mesh manually, allow a built-in optimizer to define the deformation in order to minimize an objective function, or use Design Exploration in Ansys Workbench to easily explore a variety of deformation scenarios. See Using the Mesh Morpher/Optimizer (p. 4620) for further details.

Design → **Parameter-Based** → **Optimizer...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Regions

tab allows you to define the regions of the domain where the mesh will be deformed in order to optimize the shape.

Control Point Distribution

allows you to select the way in which you will define the locations of the control points, as well as the kinds of motions you can apply.

Regular

specifies that the control points will be spread in a regular distribution throughout the entire deformation region. The number of control points along each of the defining direc-

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

tion vectors will be specified in the **Control Points** group box. The motion of the control points will be limited to translations, as the ability to define rotational or radial motions will not be available.

Unstructured

specifies that control point locations will be specified by right-clicking with the mouse on boundary zones, distributing an approximate number throughout a zone, or by entering coordinates, using the Define Control Points Dialog Box (p. 5432). The motion of the control points will be able to be defined as translations, as rotations about a point / axis, or radially about a point / axis.

Name

provides controls for the deformation regions you are defining. The upper text-entry box defines the name of the region that is created when you click the **Create** button. The lower selection list allows you to select a region that has already been created, in order to display it in the graphics window, modify the settings via the **Modify** button, or delete the region via the **Delete** button.

Update from Zones

allows you to create a bounding box that defines the deformation region.

Boundary Zones

specifies the boundary zones that will be used to generate a bounding box for the deformation region when you click the **Define** button.

Define

updates the values in the **Origin**, **Direction-1 Vector**, **Direction-2 Vector**, and **Size of Region** group boxes based on a bounding box that encompasses the selections in the **Boundary Zones** list, and previews the bounding box in green in the graphics window.

Enlarge

increases the size of the bounding box created by the **Define** button, and previews the revised bounding box in green in the graphics window.

Reduce

decreases the size of the bounding box created by the **Define** button, and previews the revised bounding box in green in the graphics window.

Update from Plane Tool

updates the values in the **Direction-1 Vector** and **Direction-2 Vector** group boxes based on the settings in the plane tool. See Using the Plane Tool (p. 3833) for information about the plane tool. This button is only available for 3D cases.

Open the Plane Surface Dialog Box (p. 5580) to make the plane tool appear.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Update from Line Tool

updates the values in the **Direction-1 Vector** and **Direction-2 Vector** group boxes based on the settings in the line tool. See Using the Line Tool (p. 3828) for information about the line tool. This button is only available for 2D cases.

Open the Line/Rake Surface Dialog Box (p. 5514) to make the line tool appear.

Origin

allows you to define the Cartesian coordinates of the origin of the deformation region.

X

defines the *x* coordinate of the origin of the deformation region.

Y

defines the *y* coordinate of the origin of the deformation region.

Z

defines the *z* coordinate of the origin of the deformation region. This number-entry box is only available for 3D cases.

Direction-1 Vector

allows you to define a vector that acts as the first direction of the deformation region relative to the **Origin**.

X

defines the *x* component of the first direction vector of the deformation region.

Y

defines the *y* component of the first direction vector of the deformation region.

Z

defines the *z* component of the first direction vector of the deformation region. This number-entry box is only available for 3D cases.

Direction-2 Vector

defines a vector that is used to calculate the second direction of the deformation region relative to the **Origin**. For 2D cases, the values in the **Direction-2 Vector** group box are automatically defined by Ansys Fluent to be perpendicular to the **Direction-1 Vector**. For 3D cases, the vector defined by the **Direction-2 Vector** group box is projected onto a plane that intersects the **Origin** and is perpendicular to the vector defined in the **Direction-1 Vector** group box, in order to calculate the second direction.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

X

defines the *x* component of the vector used to calculate the second direction of the deformation region. This number-entry box is only editable for 3D cases.

Y

defines the *y* component of the vector used to calculate the second direction of the deformation region. This number-entry box is only editable for 3D cases.

Z

defines the *z* component of the vector used to calculate the second direction of the deformation region. This number-entry box is only available for 3D cases.

Size of Region

allows you to define the overall dimensions of the deformation region.

Direction-1

defines the length of the deformation region along the direction specified by the **Direction-1 Vector** group box.

Direction-2

defines the length of the deformation region along the direction specified by the **Direction-2 Vector** group box.

Direction-3

defines the length of the deformation region along the third axis (that is, the cross product of the vectors defined by the **Direction-1 Vector** and **Direction-2 Vector** group boxes). This number-entry box is only available for 3D cases.

Boundary Continuity

allows you to specify whether the mesh transitions are smooth or abrupt wherever the mesh intersects with the limits of the deformation region. This group box is only available when **Unstructured** is selected from the **Control Point Distribution** list.

Smooth Transitions

enables smooth mesh transitions wherever the mesh intersects with the limits of the deformation region, based on the creation of stationary control points at those intersections.

Display Control Points

displays the stationary control points created when the **Smooth Transitions** option is enabled.

Control Points

allows you to define the control points for the deformation regions.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Direction-1

defines the number of control points for the deformation region selected in the **Name** group box along the direction specified by the **Direction-1 Vector** group box. This number-entry box is only available when **Regular** is selected from the **Control Point Distribution** list.

Direction-2

defines the number of control points for the deformation region selected in the **Name** group box along the direction specified by the **Direction-2 Vector** group box. This number-entry box is only available when **Regular** is selected from the **Control Point Distribution** list.

Direction-3

defines the number of control points for the deformation region selected in the **Name** group box along the third axis (that is, the cross product of the vectors defined by the **Direction-1 Vector** and **Direction-2 Vector** group boxes). This number-entry box is only available for 3D cases when **Regular** is selected from the **Control Point Distribution** list.

Define...

opens the Define Control Points Dialog Box (p. 5432), which you can use to define the control points. This button is only available when **Unstructured** is selected from the **Control Point Distribution** list.

Preview

displays (in green) the bounding box that results from the current settings in the **Origin**, **Direction-1 Vector**, **Direction-2 Vector**, and **Size of Region** group boxes.

Create

creates a new deformation region with the name specified in the upper text-entry box of the **Name** group box and with the settings defined using the **Origin**, **Direction-1 Vector**, **Direction-2 Vector**, **Size of Region**, and (for regular control point distributions) **Control Points** group boxes. The newly created deformation region will be added to and selected in the lower **Name** selection list, and will be displayed (in blue) in the graphics window.

Modify

modifies the saved settings for the deformation region selected in the lower **Name** selection list and displays the modified region in the graphics window.

Delete

deletes the deformation region selected in the lower **Name** selection list.

Constraints

tab allows you to define the constraints on the boundary zones, in order to limit the freedom of particular zones that fall within the deformation region(s) during the morphing of the mesh.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Zones

allows you to select the boundary zones for which you are defining a constraint.

Option

specifies the constraint on the boundary zones selected from the **Zones** list.

Unconstrained

specifies that the boundary zones are completely free to be deformed according to the assigned parameters.

Passive

specifies that the nodes of the boundary zones are partially constrained to varying degrees, based on their proximity to adjacent boundary zones that are fixed. The nodes in a passive boundary zone behave in a similar manner to the interior mesh nodes, in order to ensure that there is a smooth transition between fixed and unconstrained boundary zones.

Fixed

specifies that the boundary zones are fixed and will not be deformed.

Display

displays the selected **Zones** in the graphics window.

Summary

prints a list in the console that summarizes the constraint definitions for all of the boundary zones.

Deformation

tab allows you to define the deformation.

Number of Parameters

defines the number of parameters available to define the motion of the control points.

Parameter Values

allows you to define either the magnitude of deformation associated with each parameter (when you are manually specifying the deformation or using Design Exploration in Ansys Workbench to explore multiple deformation scenarios) or the bounds for each parameter (when you are using a built-in optimizer).

par1, par2...

specify the magnitude of deformation associated with each parameter. You can enter a numeric value in the field or use the icon to create or assign an input parameter (see Select Input Parameter Dialog Box (p. 4980) for details). These fields and icons are only available when **none** or **workbench** is selected from the **Optimizer** drop-down list in the

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Optimizer tab. Note that the value entered here will specify a length in meters for translations and radial motions, and will specify an angle in degrees for rotations; these units will be used regardless of what units you are using in the case.

Apply

saves definitions for the parameter fields in the **Parameter Values** group box. This button is only available when **none** or **workbench** is selected from the **Optimizer** drop-down list in the **Optimizer** tab.

Set Bounds...

opens the Parameter Bounds Dialog Box (p. 5557), where you can define strict minimum and maximum values for the parameters. Note that this button is not available when **workbench** is selected from the **Optimizer** drop-down list in the **Optimizer** tab.

Motion Settings...

opens the Motion Settings Dialog Box (p. 5543), where you can define the motion of the control points.

Deform

applies the current settings in the **Parameter Values** group box, modifies the mesh, and updates the mesh display in the graphics window. This button is only available when **none** is selected from the **Optimizer** drop-down list in the **Optimizer** tab.

Check

displays a mesh check report in the console for the mesh displayed in the graphics window. This button is only available when **none** is selected from the **Optimizer** drop-down list in the **Optimizer** tab. The mesh check report provides volume statistics, mesh topology and periodic boundary information, verification of simplex counters, and verification of node position with reference to the x axis for axisymmetric cases, in the same manner as the **Check** button in the **General** task page (see Checking the Mesh (p. 1245) for details).

Reset

undoes any modifications made to the mesh as a result of the **Deform** button and updates the mesh display in the graphics window.

Optimizer

tab allows you to define the settings for the optimizer.

Optimizer

specifies which optimizer is used. The choices include **none** (for manual deformation); **compass**, **newuoa**, **powell**, **rosenbrock**, **simplex**, and **torczon** (for optimization in Ansys Fluent); and **workbench** (for optimizing using Design Exploration in Ansys Workbench). Note that **workbench** is only available if you have launched your Ansys Fluent session from Ansys Workbench. For information about how the built-in optimizers function or how to use Design Exploration, see Design Analysis and Optimization (p. 4491) in the User's Guide or Working With Input and Output Parameters in Workbench in the Ansys Fluent in Workbench User's Guide, respectively.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Objective Function Definition...

opens the Objective Function Definition Dialog Box (p. 5553), where you can specify the format of the objective function that will be minimized during the optimization process. This button is not available if **none** or **workbench** is selected from the **Optimizer** drop-down list.

Optimizer Settings

allows you to specify the optimizer settings. This group box is not available if **none** or **workbench** is selected from the **Optimizer** drop-down list.

Maximum Number of Designs

defines the maximum number of design stages the optimizer will undergo to reach the specified objective function.

Maximum Iterations per Design

defines the maximum number of iterations Ansys Fluent will perform for each design change.

Optimizer Convergence Criteria

defines the convergence criteria for the optimizer.

Initial Parameter Variation

defines how much the parameters will be allowed to vary during the initial calculations when **newuoa** is selected from the **Optimizer** drop-down list.

Mesh Quality

allows you to enable and define a mesh quality check: if the orthogonal quality (as defined in Mesh Quality (p. 1146)) for any cell of a mesh is less than a specified value, no solution is calculated for the mesh and Fluent proceeds to the next design stage. This group box is not available if **none** is selected from the **Optimizer** drop-down list.

Reject Poor Quality Meshes

enables the consideration of orthogonal quality when determining whether a solution should be calculated for a mesh.

Minimum Orthogonal Quality

defines the minimum orthogonal quality value every cell must have in order for a solution to be calculated for a mesh. Values may range from `0`–`1` (where `0` represents the worst quality).

Case and Data File Sets

allows you to enable the automatic saving of intermediate case and data files at specified intervals during the optimization run, so that you can restart an interrupted solution in the same or a different Fluent session without increasing the overall number of design iterations

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

needed to reach convergence. This group box is not available if **none** or **workbench** is selected from the **Optimizer** drop-down list.

Save Every

specifies the frequency (in number of design iterations) that you want intermediate case and data file sets saved. Note that the default value is **0**, which specifies that no intermediate files will be saved.

Maximum Number Retained

specifies the maximum number of intermediate case and data files sets you want to retain. After the maximum limit of file sets has been saved, Ansys Fluent begins overwriting the earliest existing intermediate file set.

File Name

specifies a root name for the intermediate case and data file sets, which will be appended with a number that represents the design iteration during which it was saved. Note that you can include a folder path in the file name if you want the files saved outside of the working folder.

Initialization

provides settings that determine how the solution variables are treated after the mesh is deformed during the optimization process. This group box is not available if **none** or **workbench** is selected from the **Optimizer** drop-down list.

Method

allows you to select one of the following:

Initialize Data After Morphing

specifies that the solution variables should be initialized to the values defined in the **Solution Initialization** task page after deformation.

Continue with Current Data

specifies that the solution variables should remain the values obtained in the previous design iteration.

Read Data File After Morphing

specifies that the solution variables are set to the values obtained from the data file specified in the **Data File Name** text-entry box after deformation. This selection is not available when **newuoa** is selected from the **Optimizer** drop-down list.

Data File Name

specifies the data file from which the solution variables are obtained when **Read Data File After Morphing** is selected.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Execute Commands

allows you to specify commands (text commands or command macros) that will be run during the optimization runs of the mesh morpher/optimizer. This group box is not available if **none**, **newuoa**, or **workbench** is selected from the **Optimizer** drop-down list.

Initial Commands

specifies the commands that will be run after the design has been modified, but before Ansys Fluent has started to run the calculation for that design stage.

End Commands

specifies the commands that will be run after the solution has run and converged for a design stage.

Monitor...

opens the Optimization History Monitor Dialog Box (p. 5556), which allows you to plot and/or record how the value of the objective function varies with each design stage. This button is not available if **none**, **newuoa**, or **workbench** is selected from the **Optimizer** drop-down list.

Apply

saves the settings in the **Optimizer** tab.

Summary

displays a summary of the mesh morpher/optimizer settings in the console.

Optimize

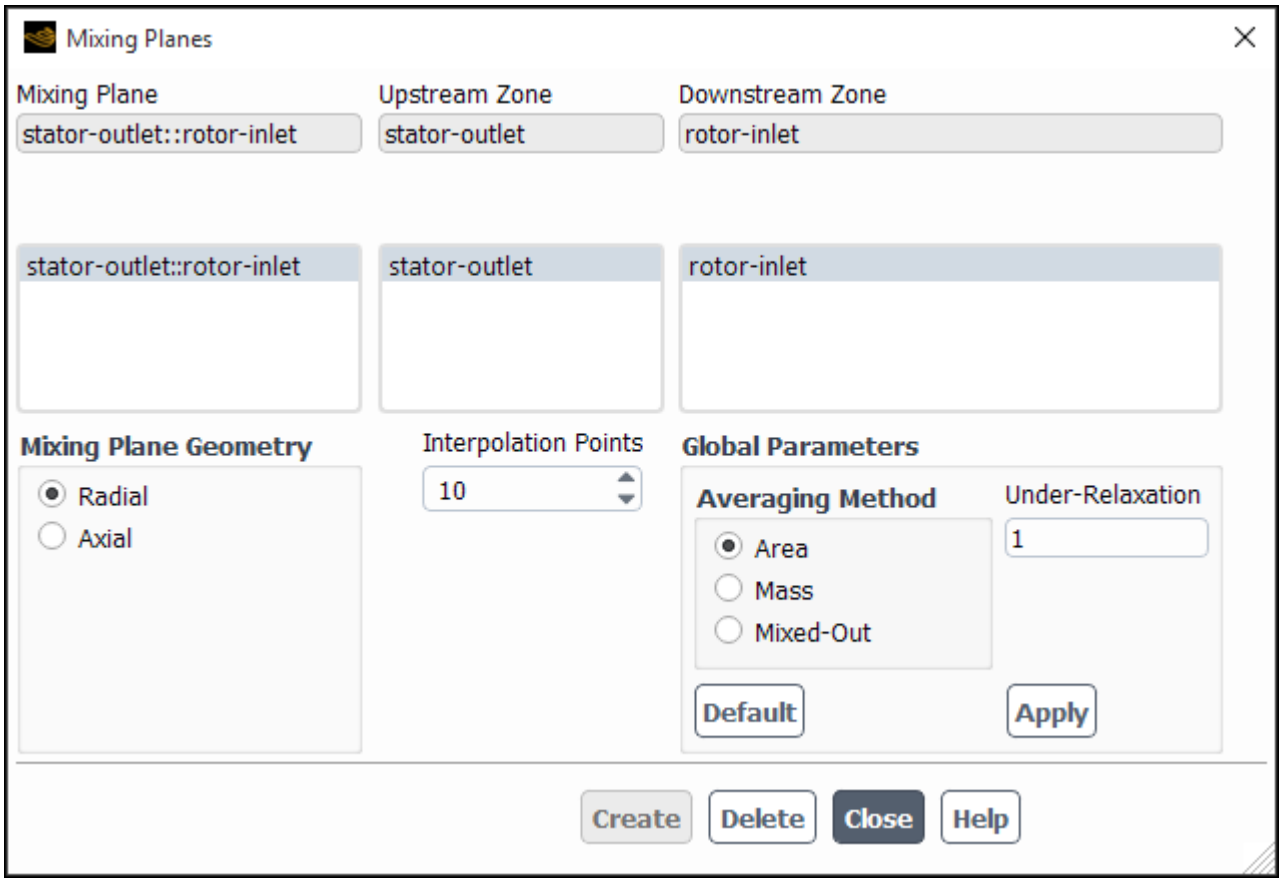
initiates the optimization process using the settings saved in all of the tabs of the **Mesh Morpher/Optimizer** dialog box. This button is not available if **none** or **workbench** is selected from the **Optimizer** drop-down list.

52.2.78. Mixing Planes Dialog Box

The **Mixing Planes** dialog box allows you to define the mixing planes for a mixing plane model. See Setting Up the Legacy Mixing Plane Model (p. 1967) for details about using the items below.

Domain → **Mesh Models** → **Mixing Planes...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Mixing Plane

contains a list from which you can select an existing mixing plane, and an informational field in which Ansys Fluent displays the name of the currently selected (or most recently created) mixing plane.

Upstream Zone, Downstream Zone

contain lists from which you can select the boundaries on the upstream and downstream sides of the mixing plane, and informational fields that show the names of the zone you selected in each list. (You cannot edit these fields; the name in each field will be the name of the zone you selected in the list below it.)

Interpolation Points

specifies the number of radial or axial locations used in constructing the boundary profiles for circumferential averaging. This item appears only in 3D.

Mixing Plane Geometry

defines the geometry of the mixing plane interface. This item appears only in 3D.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Radial

specifies that information at the mixing plane interface is to be circumferentially averaged into profiles that vary in the radial direction, for example, $p(r)$, $T(r)$.

Axial

specifies that circumferentially averaged profiles are to be constructed that vary in the axial direction, for example, $p(x)$, $T(x)$.

Global Parameters

contains parameters related to the mixing plane calculation.

Averaging Method

consists of three profile averaging methods.

Area

is the default method and is expressed using Equation 2.17 in the Theory Guide.

Mass

provides better representation of the total quantities than the area averaging method. It is defined by Equation 2.18 of the Theory Guide.

Mixed-Out

is most representative of non-uniform flow profiles and is expressed using Equation 2.20 in the Theory Guide.

Under-Relaxation

specifies the under-relaxation factor for updating the boundary values at mixing planes.

Apply

sets the specified **Under-Relaxation**.

Default

sets the **Under-Relaxation** to its default value, as assigned by Ansys Fluent. After execution, the **Default** button becomes the **Reset** button.

Reset

resets the **Under-Relaxation** to its most recently saved value (that is, the value before **Default** was selected). After execution, the **Reset** button becomes the **Default** button.

Create

creates the specified mixing plane (and assigns it a name in the **Mixing Plane** field).

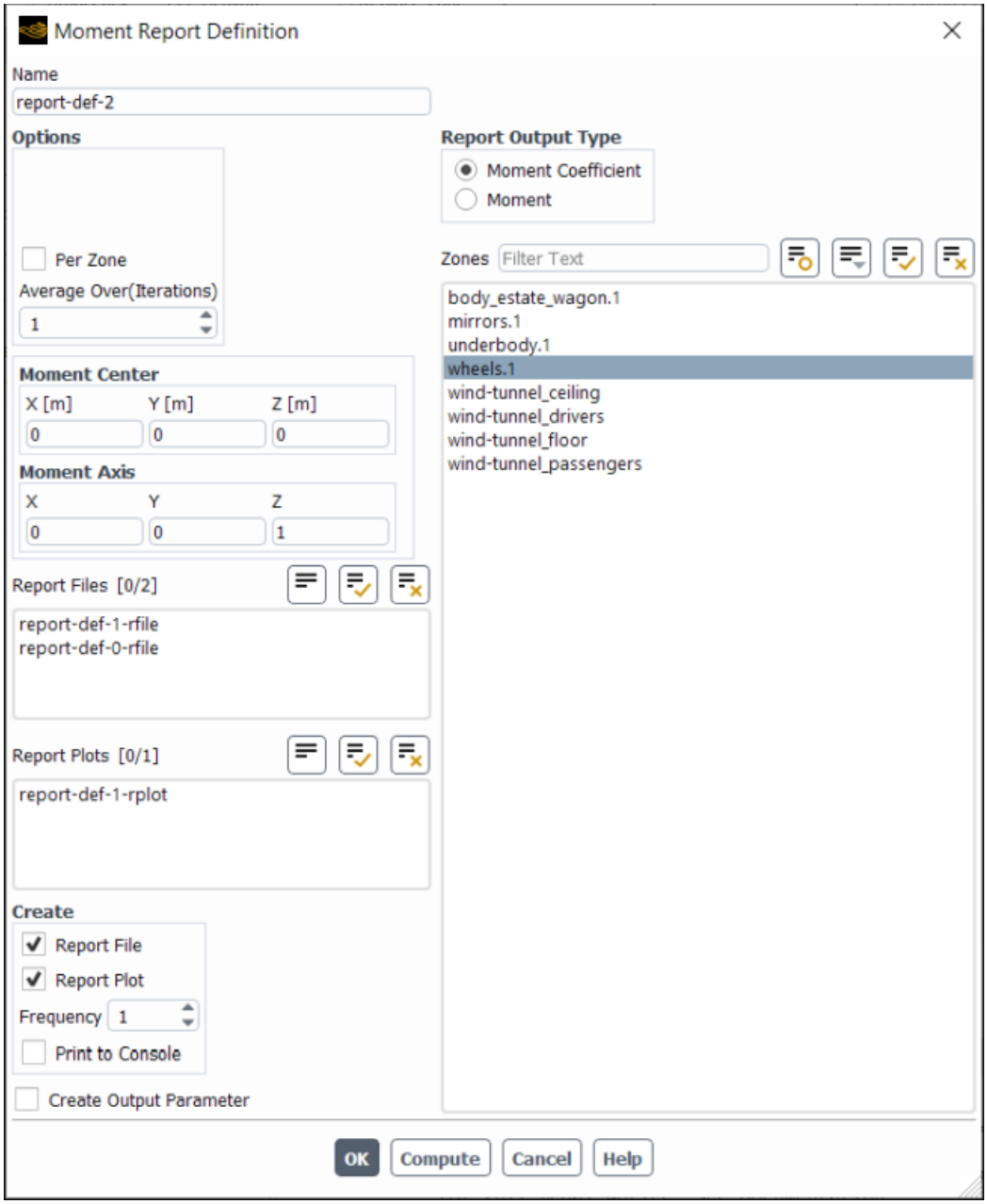
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Delete

deletes the mixing plane selected under **Mixing Plane**.

52.2.79. Moment Report Definition Dialog Box

Solution → Reports → Definitions → New → Force Report → Moment...



Controls

Name

specifies the name of the report definition.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Options

Per Zone

specifies whether or not moments for multiple walls should be monitored on each wall separately. When this option is turned off (the default), Fluent computes and monitors the total force for all of the selected walls combined together.

Average Over (Time-Steps), Average Over (Iterations)

(optional) To have Fluent calculate a running average for the **Moment Report Definition** you can enter a positive integer greater than 1 (the default) for **Average Over**.

Specifying a number greater than 1 means that Fluent will print, plot, and write the running average value of the selected variable instead of the current value of the same variable.

The value reported is averaged over the last N *iterations/time steps, where* N *is your specified* ***Average Over*** *value. When the iteration/time step number is lower than* N*, Fluent calculates the average of the available variable values.*

Retain Instantaneous Values

appears when **Average Over** is set to a value greater than 1. If enabled when you are creating a new report definition, it ensures that all instantaneous values are saved in the new report file that is being created for this report definition. It also results in the **Plot Instantaneous Values** option being enabled in the newly created report plot.

Moment Center

contains the Cartesian coordinates of the moment center.

Moment Axis

contains the Cartesian coordinates of the moment vector to be monitored.

Report Files

lists all of the report files where you can write the report definition data.

Report Plots

lists all of the report plots where you can plot the report definition data (assuming this report has the same units as the other report definitions included in the selected report plot).

Create

Report File

creates a new report file that includes this report definition.

Report Plot

creates a new report plot that includes this report definition.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Frequency

specifies how frequently the report definition is written, plotted, or printed to the console.

Print to Console

prints the value of the report definition to the console.

Create Output Parameter

creates an output parameter for this report definition with the name `<report definition name>-op`.

Report Output Type

Moment Coefficient

specifies the report definition as a unitless quantity.

Moment

specifies that the report definition has units of force.

Zones

lists the zones you can select for moment computation.

OK

creates the report definition.

Compute

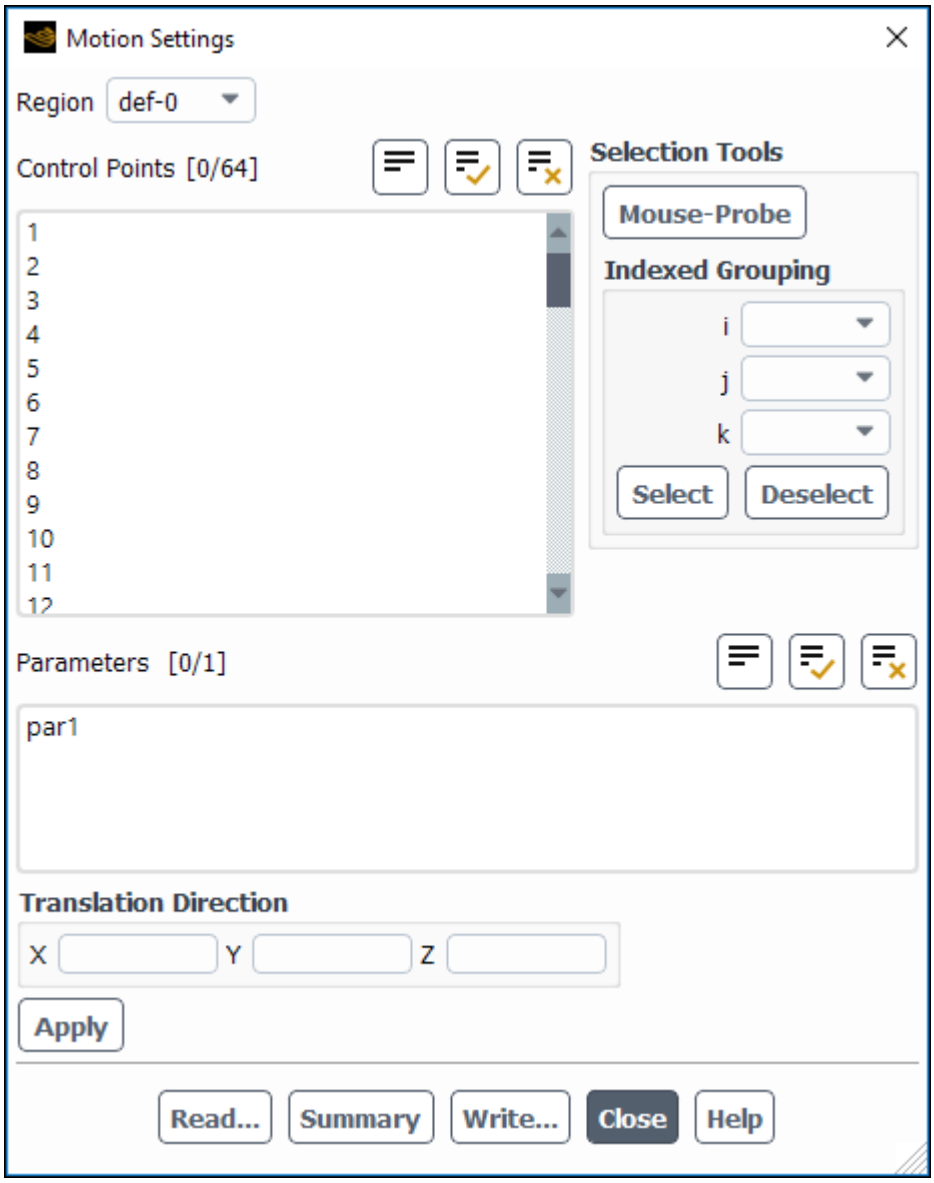
computes the value of the report definition at the selected zone(s) and prints to the console.

52.2.80. Motion Settings Dialog Box

The **Motion Settings** dialog box allows you to define the motion of the control points, as part of the definition of the deformation produced by the mesh morpher/optimizer. Note that the controls and workflow of this dialog box are very much dependent on the selection in the **Control Point Distribution** list in the **Regions** tab of the **Mesh Morpher/Optimizer** dialog box. See Using the Mesh Morpher/Optimizer (p. 4620) for details about using this dialog box.

The **Motion Settings** dialog box is opened from the Mesh Morpher/Optimizer Dialog Box (p. 5528).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls (when **Regular** is selected from the **Control Point Distribution** list in the **Regions** tab of the **Mesh Morpher/Optimizer** dialog box)

Region

selects the deformation region for which you are assigning parameters and translation directions to control points.

Control Points

defines the control points to which you are assigning parameters and translation directions.

Selection Tools

provides tools for selecting the control points to which you are assigning parameters and translation directions.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mouse-Probe

allows you to specify the **Control Points** by clicking in the graphics window with the mouse-probe button (which is the right mouse button, by default).

Indexed Grouping

allows you to select the control points based on their assigned indices.

i

allows you to select from a list of control point index numbers, which identify the location in a sequence (starting at the origin of the region) of control points along the direction-1 vector (as defined in the **Regions** tab of the **Mesh Morpher/Optimizer** dialog box).

j

allows you to select from a list of control point index numbers, which identify the location in a sequence (starting at the origin of the region) of control points along the direction-2 vector (as defined in the **Regions** tab of the **Mesh Morpher/Optimizer** dialog box).

k

allows you to select from a list of control point index numbers, which identify the location in a sequence (starting at the origin of the region) of control points along the direction-3 vector (as defined—though not displayed—in the **Regions** tab of the **Mesh Morpher/Optimizer** dialog box).

Select

adds the control points that satisfy the selected index criteria (that is, the selections from the **i**, **j**, and **k** drop-down lists) to the list of selected **Control Points**.

Deselect

removes the control points that satisfy the selected index criteria (that is, the selections from the **i**, **j**, and **k** drop-down lists) from the list of selected **Control Points**.

Parameters

specifies the names of the parameters (whose values are displayed in the **Parameter Values** group box in the **Deformation** tab of the **Mesh Morpher/Optimizer** dialog box) that you are assigning to the **Control Points**. The products of the values associated with these parameters and the settings specified in the **Translation Direction** group box define the displacement of the control points during manual deformation.

Translation Direction

allows you to define coefficients in the *x*, *y*, and (for 3D cases) *z* directions to scale the magnitude of deformation for the selected parameters. If you use a built-in optimizer, these coefficients provide the direction of the displacement of the control points and the optimizer determines the overall magnitude of displacement. Alternatively, if you manually specify the deformation or use Design Exploration in Ansys Workbench to explore multiple deformation scenarios, the values you enter in the **Translation Direction** group box are multiplied with the values of the **Parameters**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

(as displayed in the **Parameter Values** group box in the **Deformation** tab of the **Mesh Morpher/Optimizer** dialog box) to define the displacement of the control points.

X

defines the scaling coefficient applied to the deformation parameter in the *x* direction.

Y

defines the scaling coefficient applied to the deformation parameter in the *y* direction.

Z

defines the scaling coefficient applied to the deformation parameter in the *z* direction. This number-entry box is only available for 3D cases.

Apply

saves the selections made in the **Parameters** selection list and the values entered in the **Translation Direction** group box, and assigns these parameter settings to the control points selected in the **Control Points** selection list.

Read...

allows you to define your motion settings by reading an ASCII text file. See Mesh Morpher/Optimizer File Formats (p. 5693) for further details about the text file format.

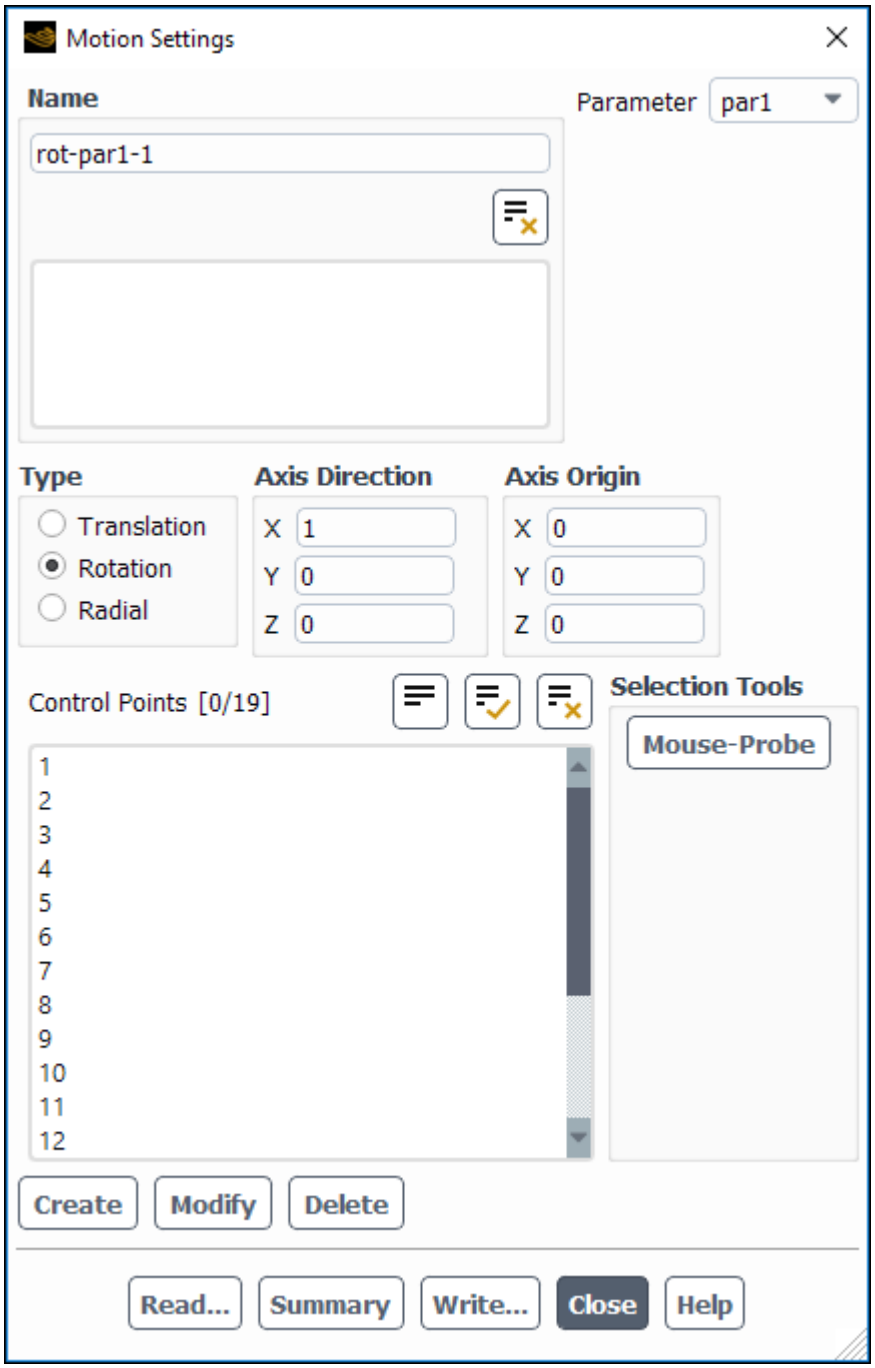
Summary

displays a summary of the saved motion settings in the console.

Write...

allows you to write your saved motion settings to an ASCII text file. See Mesh Morpher/Optimizer File Formats (p. 5693) for further details about the text file format.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls (denoted by * - when **Unstructured** is selected from the **Control Point Distribution** list in the **Regions** tab of the **Mesh Morpher/Optimizer** dialog box)

Name

provides controls for the motions you are defining. The upper text-entry box defines the name of the motion that is created when you click the **Create** button (which by default is based on the type of motion and the related parameter). The lower selection list allows you to select a motion that has already been created, in order to display the selections associated with it, or to modify or delete the motion using the **Modify** or **Delete** buttons, respectively.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parameter

allows you to select the parameter (whose value is displayed in the **Parameter Values** group box in the **Deformation** tab of the **Mesh Morpher/Optimizer** dialog box) for the motion.

Type

allows you to specify the type of motion you want to define.

Translation

specifies that the motion is a translation, based on the settings in the **Direction** group box.

Rotation

specifies that the motion is a rotation about a point (for 2D) or an axis (for 3D), based on the settings in the **Axis Origin** group box and (for 3D cases) **Axis Direction** group box.

Radial

specifies that the motion is defined radially about a point (for 2D) or an axis (for 3D), based on the settings in the **Axis Origin** group box and (for 3D cases) **Axis Direction** group box.

Direction

allows you to define coefficients in the **X**, **Y**, and (for 3D cases) **Z** directions to scale the magnitude of deformation for the selected parameter. If you use a built-in optimizer, these coefficients provide the direction of the displacement of the control points and the optimizer determines the overall magnitude of displacement. Alternatively, if you manually specify the deformation or use Design Exploration in Ansys Workbench to explore multiple deformation scenarios, the values you enter in the **Direction** group box are multiplied with the values of the **Parameter** (as displayed in the **Parameter Values** group box in the **Deformation** tab of the **Mesh Morpher/Optimizer** dialog box) to define the displacement of the control points.

This group box is only available when **Translation** is selected from the **Type** list.

Axis Direction

allows you to define the **X**, **Y**, and **Z** components of the vector about which the control points will rotate or move radially, when the selection in the **Type** list is **Rotation** or **Radial**, respectively. This group box is only available for 3D cases.

Axis Origin

allows you to define the **X**, **Y**, and (for 3D cases) **Z** coordinates of the origin about which the control points will rotate or move radially, when the selection in the **Type** list is **Rotation** or **Radial**, respectively.

Control Points*

allows you to select the control points that will undergo the motion.

Selection Tools*

provides tools for selecting the control points that will undergo the motion.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Mouse-Probe*

allows you to specify the **Control Points** by clicking in the graphics window with the mouse-probe button (which is the right mouse button, by default).

Create

creates a new motion based on the currently displayed settings.

Modify

modifies the saved settings for the motion selected in the lower **Name** selection list.

Delete

deletes the motion selected in the lower **Name** selection list.

Read...*

allows you to define your motion definitions by reading an ASCII text file. See Mesh Morpher/Optimizer File Formats (p. 5693) for further details about the text file format.

Summary*

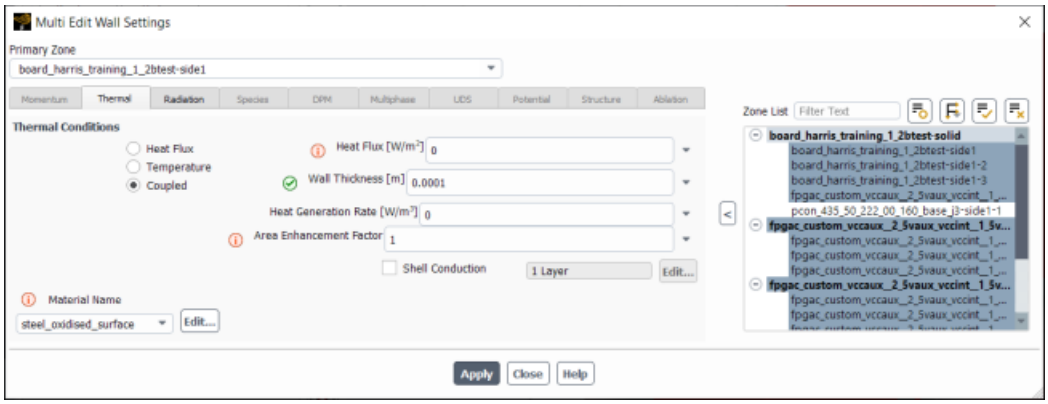
displays a summary of the saved motion definitions in the console.

Write...*

allows you to write your saved motion definitions to an ASCII text file. See Mesh Morpher/Optimizer File Formats (p. 5693) for further details about the text file format.

52.2.81. Multi Edit Dialog Box

Setup → Boundary Conditions → <selected boundaries> Multi-Edit...



Controls

Primary Zone

drop-down list displaying the name of the selected "primary" boundary. *When the dialog box is first opened, the values of the primary zone are shown in all fields.*

Zone List

lists all of the zones that are compatible for multi-editing with the primary zone. The selected zones are the ones that will be modified on **Apply**.

Apply

applies the settings marked with the icon to the selected boundaries in the **Zone List**.

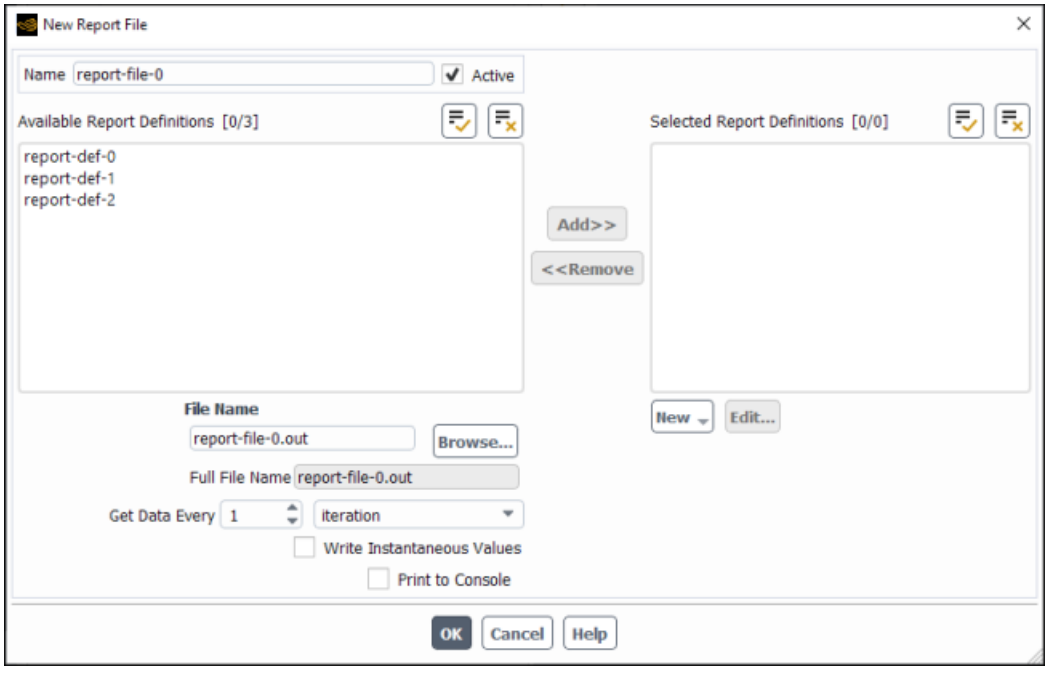
- indicates that the value of this setting varies between the selected boundaries in the **Zone List**.
- indicates that the displayed value will be applied to the selected boundaries in the **Zone List** when you click **Apply**.

Refer to Editing Multiple Boundary Conditions at Once (p. 1521) for additional information about this functionality. Note that while this image shows the **Multi Edit Velocity Wall Settings** dialog box, the functionality that is specific to multi-editing is consistent across the different multi-edit dialog boxes. For information on specific settings of boundary conditions, refer to Boundary Conditions Task Page (p. 4991) (which contains descriptions of each boundary's dialog box), and Boundary Conditions (p. 1392).

52.2.82. New Report File Dialog Box

Solution → Reports → File...

Click **New...** in the **Report File Definitions** dialog box to open the **New Report File** dialog box.



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Controls

Name

editable field displaying the name of the report file.

Active

check box (enabled by default) that determines if the file is written during the solution calculation.

Available Report Definitions

lists all of the existing report definitions not currently part of this report file.

Add>>

moves the selected report definition(s) to the **Selected Report Definitions** list.

<<Remove

moves the selected report definition(s) to the **Available Report Definitions** list.

File Name

allows you to specify the name used when the file is saved.

Browse...

opens the **Select File** dialog box, allowing you to specify the location where the report file will be saved.

Full File Name

displays the name and location of the report file once it has been saved.

Get Data Every

indicates the frequency that the report definition data is written to the file. The default value of 1 allows you to write to the file at every iteration or time-step. If you select flow-time, then you must specify the frequency for writing the file in units of time. Time-step and flow-time are only a valid choices when you are calculating unsteady flow.

Print to Console

indicates whether or not the contents of the report file are printed to the console at the frequency specified in the **Get Data Every** field.

Selected Report Definitions

lists all of the existing report definitions currently included in this report file.

New

drop-down list allowing you to create new report definitions (surface, volume, force, flux, mesh, aeromechanics, dpm, expression, and user-defined).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

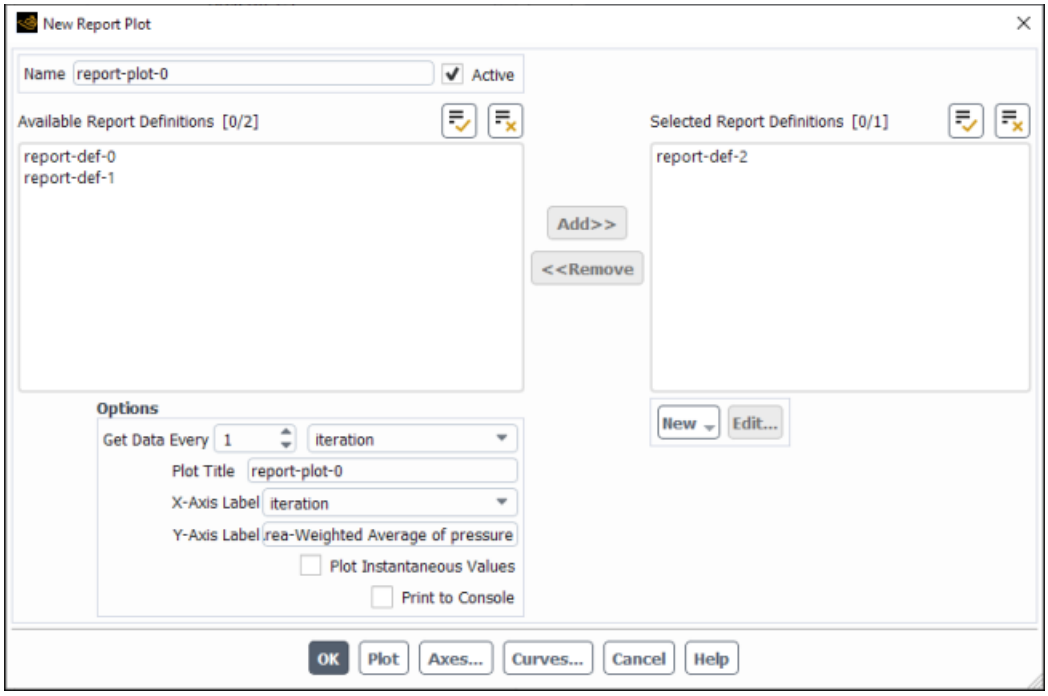
Edit...

allows you to modify the highlighted report definition in the **Selected Report Definition** list.

52.2.83. New Report Plot Dialog Box

Solution → Reports → Plot...

Click **New...** in the **Report Plot Definitions** dialog box to open the **New Report Plot** dialog box.



Controls

Name

editable field displaying the name of the report plot.

Active

check box (enabled by default) that determines if the plot is plotted during the solution calculation.

Available Report Definitions

lists all of the existing report definitions not currently part of this report plot.

Add>>

moves the selected report definition(s) to the **Selected Report Definitions** list.

<<Remove

moves the selected report definition(s) to the **Available Report Definitions** list.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Options

Get Data Every

indicates the frequency that the report definition data is plotted. The default value of 1 allows you to write to the file at every iteration or time-step. If you select flow-time, then you must specify the frequency for plotting the data in units of time. Time-step and flow-time are only a valid choices when you are calculating unsteady flow.

Plot Title

provide a title for the plot.

X-Axis Label

choose whether the X-axis is flow-time or time-step.

Y-Axis Label

enter a label for the Y-axis.

Print to Console

indicates whether or not the contents of the report plot are printed to the console at the frequency specified in the **Get Data Every** field.

Selected Report Definitions

lists all of the existing report definitions currently included in this report plot.

New

drop-down list allowing you to create new report definitions (surface, volume, force, flux, mesh, aeromechanics, dpm, expression, and user-defined).

Edit...

allows you to modify the highlighted report definition in the **Selected Report Definition** list.

Curves...

opens the Curves Dialog Box (p. 5316).

Axes...

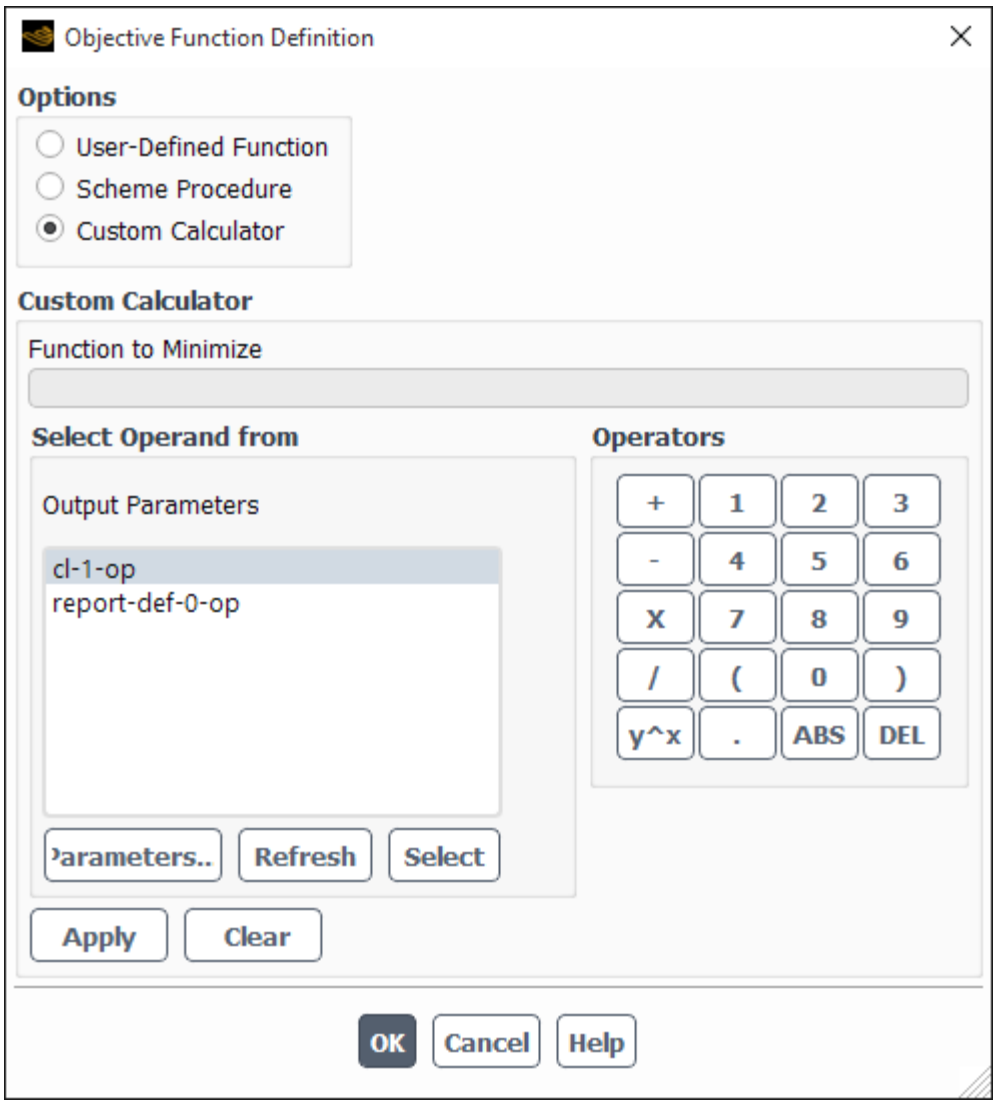
opens the Axes Dialog Box (p. 5313).

52.2.84. Objective Function Definition Dialog Box

The **Objective Function Definition** dialog box allows you to specify the format of the objective function that will be minimized by the mesh morpher/optimizer and (when it is a customized function of output parameters) to define the objective function. See Using the Mesh Morpher/Optimizer (p. 4620) for details about using this dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The **Objective Function Definition** dialog box is opened from the Mesh Morpher/Optimizer Dialog Box (p. 5528).



Controls

Options

contains options for the format of the objective function.

User-Defined Function

specifies that the objective function is provided via a user-defined function.

Scheme Procedure

specifies that the objective function is provided via a Scheme source file. This option is not available if **newuoa** is selected from the **Optimizer** drop-down list in the **Optimizer** tab of the **Mesh Morpher/Optimizer** dialog box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Custom Calculator

specifies that the objective function is based on output parameters, and is defined by the controls in the **Custom Calculator** group box. This option is not available if **newuoa** is selected from the **Optimizer** drop-down list in the **Optimizer** tab of the **Mesh Morpher/Optimizer** dialog box.

Custom Calculator

allows you to define the objective function. This group box is only available when **Custom Calculator** is selected from the **Options** list.

Function to Minimize

displays the objective function defined by the controls in the **Custom Calculator** group box.

Select Operand from

allows you to include output parameters in the definition of the objective function.

Output Parameters

provides a list of available output parameters, which can be included in the definition of the objective function.

Parameters...

opens the Parameters Dialog Box (p. 5338), which you can use to create additional output parameters.

Refresh

updates the **Output Parameters** list, so that all available output parameters are displayed.

Select

enters the output parameter that is currently selected in the **Output Parameters** list in the **Function to Minimize** field.

Operators

allows you to include calculator operators in the definition of the objective function. Clicking a button in this group box causes the appropriate symbol to appear in the **Function to Minimize** field.

Apply

saves the custom objective function (displayed in the **Function to Minimize** field) as part of the case file.

Clear

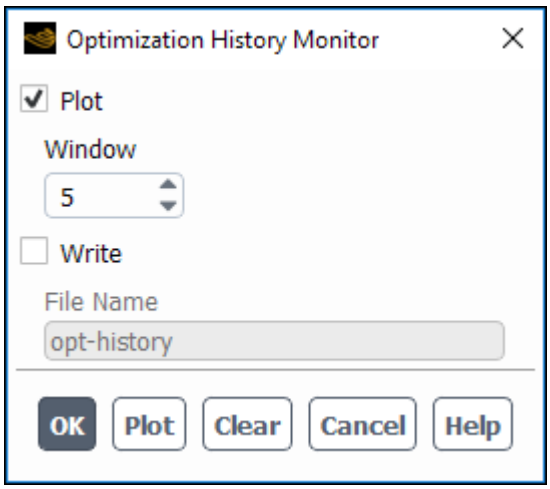
deletes all of the text in the **Function to Minimize** field.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

52.2.85. Optimization History Monitor Dialog Box

The **Optimization History Monitor** dialog box allows you to plot and/or record optimization history data, that is, how the value of the objective function (defined using the Mesh Morpher/Optimizer Dialog Box (p. 5528)) varies with each design stage produced by the mesh morpher/optimizer. See Using the Mesh Morpher/Optimizer (p. 4620) for details about using this dialog box.

The **Optimization History Monitor** dialog box is opened from the Mesh Morpher/Optimizer Dialog Box (p. 5528).



Controls

Plot

enables the plotting of the optimization history data in the graphics window.

Window

optimization history data are automatically displayed in the appropriate window, so changing the number in the **Window** field will not modify the display location.

Write

enables the saving of the optimization history data to a file (with the name specified in **File Name**).

File Name

specifies the name of the file to which the optimization history data is written. This text-entry box is only available when the **Write** option is enabled.

Plot (button)

displays an XY plot of the optimization history data generated during the last calculation. Note that no plot will be displayed if the data was discarded using the **Clear** button.

Clear

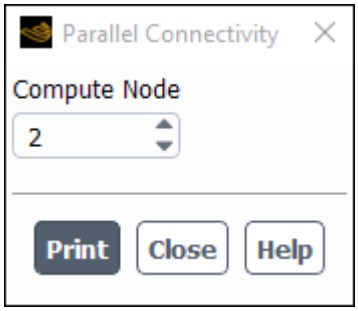
discards the optimization history data, including the associated files. Each of these actions must be confirmed in a Question Dialog Box (p. 931).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

52.2.86. Parallel Connectivity Dialog Box

The **Parallel Connectivity** dialog box prints the connectivity of the selected compute node. See Checking Network Connectivity (p. 4345) for details.

Parallel → Network → Connectivity...



Controls

Compute Node

indicates the compute node ID for which connectivity information is desired.

Print

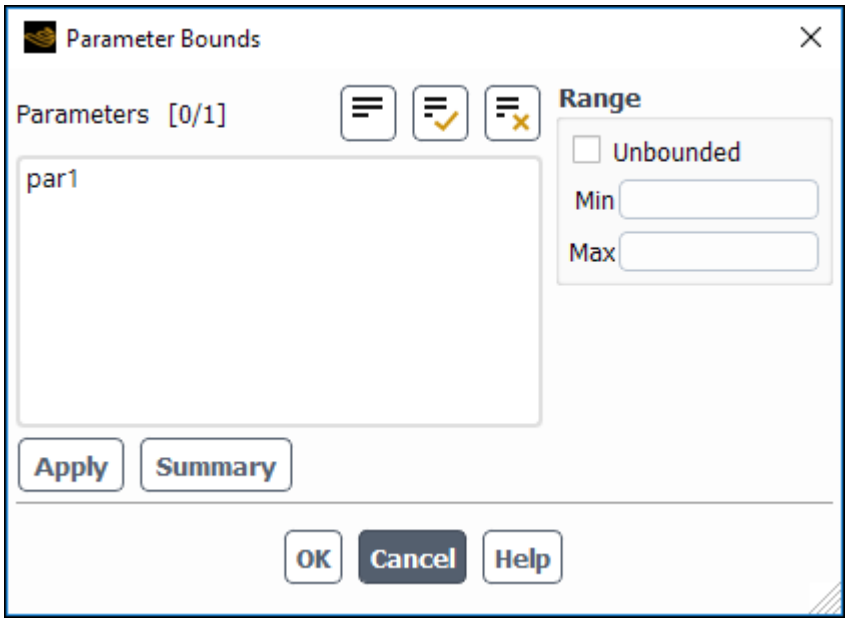
prints information (in the console window) about the network connectivity for the selected compute node.

52.2.87. Parameter Bounds Dialog Box

The **Parameter Bounds** dialog box allows you to limit how much each parameter is allowed to deform when using the mesh morpher/optimizer. See Using the Mesh Morpher/Optimizer (p. 4620) for details about using this dialog box.

The **Parameter Bounds** dialog box is opened from the Mesh Morpher/Optimizer Dialog Box (p. 5528).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Parameters

selects the parameters (created using the **Deformation** tab of the Mesh Morpher/Optimizer Dialog Box (p. 5528)) for which you are defining bounds.

Range

allows you to specify the bounds.

Unbounded

allows you to specify whether the selected **Parameters** are unbounded.

Min, Max

allows you to set the minimum and maximum values allowed for the selected **Parameters** when the **Unbounded** option is disabled.

Apply

saves the bounds for the currently selected **Parameters**.

Summary

displays a summary of all of the saved parameter bounds in the console.

52.2.88. Particle Tracks Dialog Box

The **Particle Tracks** dialog box controls the generation and display of discrete phase particles. See Displaying of Trajectories (p. 2834) for details about the items below.

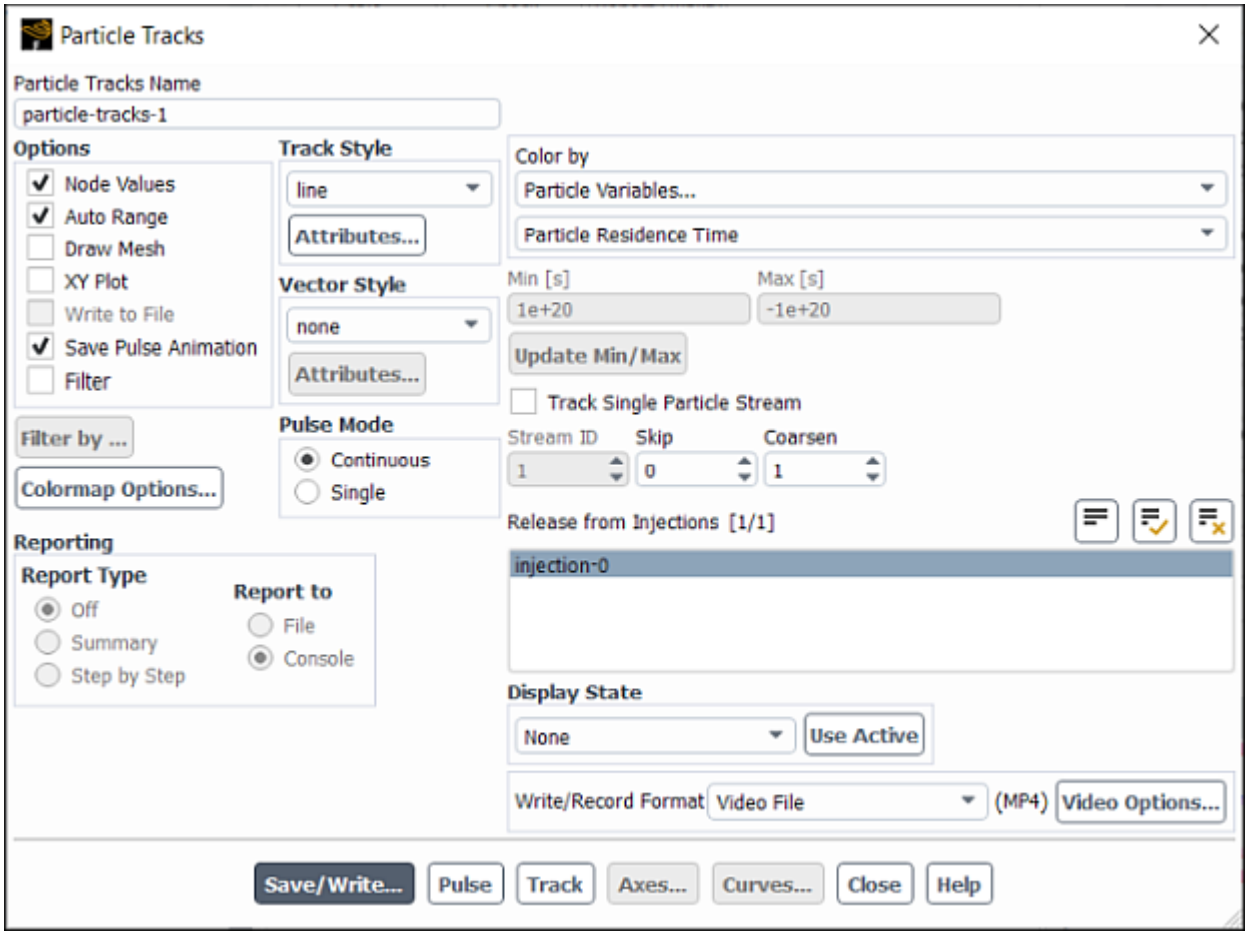
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Results → Graphics → Particle Tracks → New...

Note:

- The "global"/non-persistent form of graphics objects is hidden by default. If this interrupts your workflow, you can make them available by enabling the **Expose legacy non-persistent graphics** option in the **Graphics** branch of **Preferences** (accessed via **File**>**Preferences...**).
- (When you click **Edit...** instead of **New...**, it opens the "global"/non-persistent version of the **Particle Tracks** dialog box. This version of the dialog box does not have a **Name** field and it is not saved with the case file.

  You are encouraged to use the persistent version that is accessed by clicking **New...**.



Controls

Particle Tracks Name

is the name for a particle tracks definition. You can specify a name or use the default name **`particle-tracks-`*`id`*** . This control appears only for particle tracks definitions.

Options

contains the options described below:

Node Values

specifies that node values should be interpolated to compute the scalar field at a particle location. This option is turned on by default. See Choosing Node or Cell Values (p. 3932) for details.

Auto Range

specifies that the minimum and maximum values of the scalar field will be the limits of that field. If this option is not selected, you can enter **Min** and **Max** values manually. These values determine the range of the color scale.

Draw Mesh

toggles between displaying and not displaying the mesh. The Mesh Display Dialog Box (p. 4688) is opened when **Draw Mesh** is selected.

XY Plot

enables the display of an XY plot along the particle trajectories for steady particle tracking and at the current particle locations for unsteady tracking. When this option is selected, the **Display** push button will change to **Plot**.

Write to File

(available only when **XY Plot** is on) activates the file-writing option. When this option is selected, the **Plot** push button will change to **Write...**. Clicking on the **Write...** button will open The Select File Dialog Box (p. 931), in which you can specify a name and save a file containing the XY plot data. The format of this file is described in XY Plot File Format (p. 4071).

Save Pulse Animation

allows you to save a video or picture files of the pulse animation. When this option is enabled, an additional field for **Write/Record Format** appears, allowing you to specify how the pulse animation will be saved. Clicking **Save/Write...** opens The Select File Dialog Box (p. 931) where you can specify a name and save the video or picture files.

A **Continuous** pulse video file will be 5 seconds long, while a **Single** pulse video will be one full cycle.

Filter

when selected, the **Filter by...** button becomes available.

Filter by...

opens the Particle Filter Attributes Dialog Box (p. 5244) where you can specify how you would like to filter the particles being displayed. This button is available when **Filter** is selected in the **Options** group box.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Colormap Options...

opens the Colormap Dialog Box (p. 5280), allowing you to customize the colormap for this graphics object.

Reporting

allows you to specify how you would like to report particle tracks.

Report Type

controls the type of trajectory-fate reports to be displayed.

Off

disables reporting of trajectory fates.

Summary

enables summary reports of trajectory fates. See Particle Tracking Statistics (p. 2846) for details.

Step by Step

enables the step-by-step reporting of trajectories. This item will not appear if you have requested unsteady tracking in the Discrete Phase Model Dialog Box (p. 4845). See Step-by-Step Reporting of Trajectories (p. 2856) for details.

Current Positions

enables the reporting of the positions and velocities of all particles that are in the domain at the current time. This item appears only when unsteady tracking has been requested in the Discrete Phase Model Dialog Box (p. 4845). See Reporting of Current Positions for Unsteady Tracking (p. 2858) for details.

Report to

indicates the destination of the trajectory report.

File

enables the writing of the trajectory report to a file. When this option is enabled, the **Track** button will become the **Write...** button.

Console

enables the display of the trajectory report information in the console window.

Significant Figures

controls the number of significant figures used when a **Step by Step** or **Current Positions** report is selected.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Sample File Format

allows you to generate a file containing particle current positions in the same file format that is used for particle sampling. This item is available only for unsteady particle tracking when **Current Positions** and **File** are selected in the **Reporting** group box. See Reporting of Current Positions for Unsteady Tracking (p. 2858) for details.

Reporting Variables...

opens the Reporting Variables Dialog Box (p. 5245) that contains a list from which you can select the scalar field to be used to color the particle tracks.

Track Style

sets the trajectory style. Trajectories can be displayed as lines, ribbons, spheres, cylinders, or a set of points. Pulsing can be done only on **point** or **line** styles.

Attributes...

opens the Track Style Attributes Dialog Box (p. 5246), Particle Sphere Style Attributes Dialog Box (p. 5247), or the Ribbon Attributes Dialog Box (p. 5243), depending on the selected **Track Style**. These dialog boxes allow you to set the attributes for the chosen trajectory style.

Vector Style

sets the trajectory vector style. You can display the trajectories vectors in three different ways: **vector**, **centered-vector**, and **centered-cylinder**. All three styles are demonstrated in Controlling the Vector Style of Particle Tracks (p. 2841).

Attributes...

opens the Particle Vector Style Attributes Dialog Box (p. 5248).

Pulse Mode

specifies either a single or continuous release of particles that follow the trajectories, starting at the release surface, when you use the **Pulse** button to animate the trajectories.

Continuous

sets the pulse mode to continuously release particles from the initial positions.

Single

sets the pulse mode to release a single wave of particles.

The **Pulse Mode** is used only when particles are being **Pulsed**.

Y Axis Function

contains a list of solution variables that can be used for the y axis of the plot.

This item appears when the **XY Plot** option is turned on.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

X Axis Function

contains a list of functions for the X-axis of the plot. You can plot the quantity selected in the **Y Axis Function** drop-down list as a function of the **Time** elapsed along the trajectory, or the **Path Length** along the trajectory.

This item appears when the **XY Plot** option is turned on.

Phase

(Multiphase only) specifies the phase where the postprocessing operations are being applied.

Min/Max

display the values to which the lower and upper ends of the color scale map. If you are using the **Auto Range** option, these values will be automatically computed when you click **Compute**. If you are not using **Auto Range**, you can enter values manually.

Update Min/Max

computes the **Min** and **Max** for the field chosen in **Color by**.

Track PDF Transport Particles

enables the tracking of PDF transport particles. This item is available only when the composition PDF transport model is enabled.

Free Stream Particles

enables the tracking of free stream particles. Note that this option is available only when the Lagrangian wallfilm model is enabled in the wall boundary conditions dialog box, under the **DPM** tab.

Wall Film Particles

enables the tracking of wall film particles. Note that this option is available only when the Lagrangian wallfilm model is enabled in the wall boundary conditions dialog box, under the **DPM** tab.

Track Single Particle Stream

enables the tracking of a single particle stream in the selected injection, instead of all the streams in that injection. See Displaying of Trajectories (p. 2834) for details.

Stream ID

specifies the particle stream to be tracked when the **Track Single Particle Stream** option is enabled.

Skip

allows you to "thin" or "sample" the particles that are displayed.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Coarsen

reduces the plotting time by reducing the number of points that are plotted for a given trajectory in any cell. This is valid only for steady-state cases.

Release from Injections

selects the injections from which to release particles. You must select at least one injection before any particles can be released.

Display State

specifies the display state that is associated with this graphics object.

drop-down list of display states

selectable list of all the existing display states associated with this case.

Use Active

creates a display state that matches the settings of the active graphics window and assigns it to this graphics object (this association is only saved after you click **Save/Display**).

Write/Record Format

(only present when **Save Pulse Animation** is enabled) drop-down list specifying whether you are saving the pulse animation as a video file or as picture files. The default formats are shown in parenthesis next to the field.

Video Options...

(only present when **Save Pulse Animation** is enabled and **Video File** is selected for **Write/Record Format**) opens the Video Options Dialog Box (p. 5260).

Picture Options...

(only present when **Save Pulse Animation** is enabled and **Picture Files** is selected for **Write/Record Format**) opens the Save Picture Dialog Box (p. 5254).

Display

displays the particle trajectories and generates a trajectory report in the console window (if requested) for steady particle tracking. For unsteady particle tracking, only the current particle locations will be shown.

Save/Display

displays the particle tracks (steady tracking) or the current particle locations (unsteady tracking) in the active graphics window and saves the particle track definition. This button appears only for particle track definitions and replaces the **Display** button.

Plot

displays an XY plot along the particle trajectories. This button replaces the **Display** button when the **XY Plot** option is turned on.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Write...

opens The Select File Dialog Box (p. 931), in which you can save the XY plot data or trajectory report to a file. This button replaces the **Plot** button when the **Write to File** option is turned on, or the **Track** button when the **File** option is selected under **Report to**.

Pulse

animates the position of particles. If **Pulse Mode** is set to **Continuous**, particles are reintroduced at the seed positions repeatedly. If the mode is **Single**, a single wave of particles moves through the domain. **Pulse** is not available for unsteady tracking.

The label of this button changes to **Stop !** when the animation is in progress. Click **Stop !** to stop pulsing (and the label goes back to **Pulse**).

Track

computes the particle trajectories and generates any reports without displaying the trajectories in the graphics window.

Axes...

opens the Axes Dialog Box (p. 5313), which allows you to customize the XY plot axes. This item appears when the **XY Plot** option is turned on.

Curves...

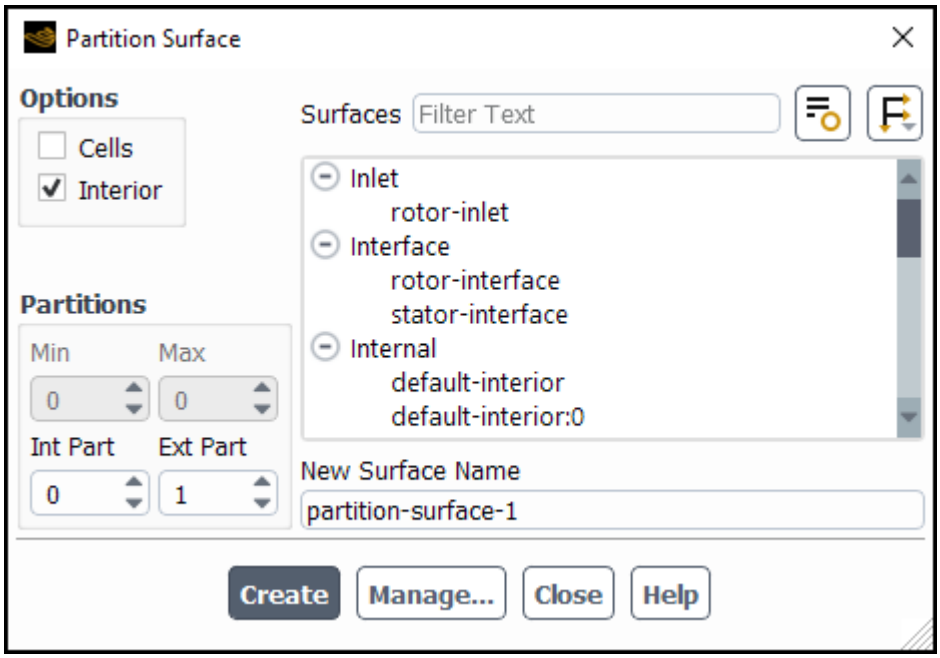
opens the Curves Dialog Box (p. 5316), which allows you to customize the curves used in the XY plot. This item appears when the **XY Plot** option is turned on.

52.2.89. Partition Surface Dialog Box

The **Partition Surface** dialog box allows you to create a surface defined by the boundary of two adjacent mesh partitions. See Partition Surfaces (p. 3817) for details about the items below.

Domain → **Surface** → **Create** → **Partition...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options

contains check buttons that allow you to choose interior or exterior faces or cells to be contained in the partition surface.

Cells

specifies, when enabled, that the partition surface will consist of cells (either interior or exterior, depending on the status of the **Interior** check button) that lie on the partition boundary. When this option is disabled, the faces on the boundary between partitions will make up the partition surface. Depending on the status of the **Interior** check button, the faces will reflect data values for the interior or exterior cells.

Interior

toggles between interior and exterior cells. If the partition surface consists of faces instead of cells, **Interior** specifies for which cells data will be displayed on the faces. When this option is enabled, interior cells (cells that are on the **Int Part** side of the partition boundary) will make up the partition surface. When it is disabled, the partition surface will consist of exterior cells (cells that are on the **Ext Part** side of the partition boundary).

Partitions

contains information about which partitions are under consideration. The boundary that defines the partition surface is the boundary between the **Int Part** and the **Ext Part**. If there are more than two mesh partitions, each interior partition will share boundaries with several exterior partitions. By setting the appropriate values for **Int Part** and **Ext Part**, you can create surfaces for any of these boundaries.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Min/Max

indicates the minimum and maximum ID numbers of the mesh partitions. The minimum is always zero, and the maximum is one less than the number of processors.

Int Part

indicates the ID number of the interior partition (that is, the partition under consideration).

Ext Part

indicates the ID number of the bordering partition.

Surfaces

displays an informational list of existing surfaces.

New Surface Name

designates the name of the new surface. The default is the concatenation of the surface type and an integer which is the new surface ID.

Create

creates the surface.

Manage...

opens the Surfaces Dialog Box (p. 5629) in which you can rename and delete surfaces and determine their sizes.

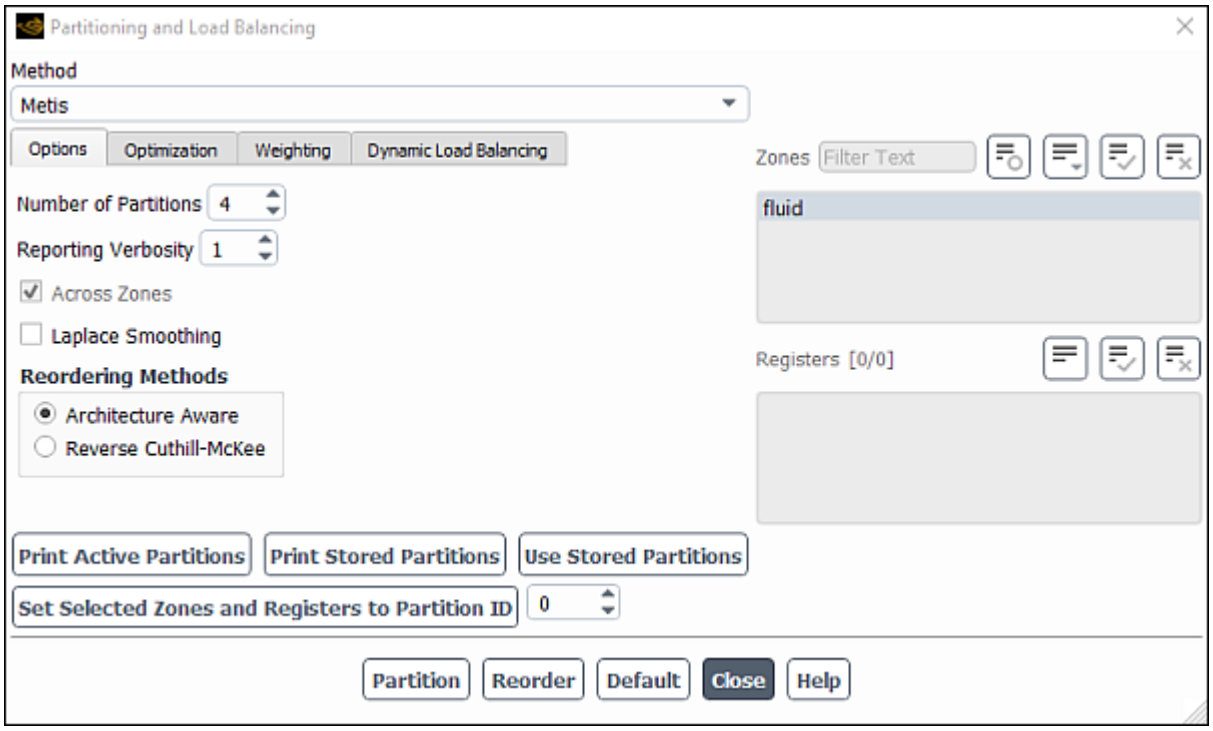
52.2.90. Partitioning and Load Balancing Dialog Box

The **Partitioning and Load Balancing** dialog box allows you to partition the mesh into separate clusters of cells for separate processors on a parallel computer. See Partitioning the Mesh Manually and Balancing the Load (p. 4319) for details.

The **Partitioning and Load Balancing** dialog box also allows you to enable and control Ansys Fluent's load balancing feature. Load balancing will automatically detect and analyze parallel performance, and redistribute cells between the existing compute nodes to optimize it. See Load Balancing (p. 4329) for more information about load balancing.

Parallel → General → Partitioning/Load Balancing...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Method

contains a drop-down list of the recursive partition methods that can be used to create the mesh partitions. The choices include the **Cartesian Axes**, **Cartesian Strip**, **Cartesian X-Coordinate**, **Cartesian Y-Coordinate**, **Cartesian Z-Coordinate**, **Cartesian R Axes**, **Cartesian RX-Coordinate**, **Cartesian RY-Coordinate**, **Cartesian RZ-Coordinate**, **Cylindrical Axes**, **Cylindrical R-Coordinate**, **Cylindrical Theta-Coordinate**, **Cylindrical Z-Coordinate**, **Metis**, **Polar Axes**, **Polar R-Coordinate**, **Polar Theta-Coordinate**, **Principal Axes**, **Principal Strip**, **Principal X-Coordinate**, **Principal Y-Coordinate**, **Principal Z-Coordinate**, **Spherical Axes**, **Spherical Rho-Coordinate**, **Spherical Theta-Coordinate**, and **Spherical Phi-Coordinate** techniques, which are described in Mesh Partitioning Methods (p. 4331).

Options

tab

Number of Partitions

defines the desired number of mesh partitions. This usually matches the number of processors available for parallel computing.

Reporting Verbosity

specifies the amount of information to be reported in the text (console) window during the partitioning. With the default value of 1, Fluent will print the number of partitions created, the number of bisections performed, the time required for the partitioning, and the minimum and maximum cell, face, interface, and face-ratio variations. (See Interpreting Partition Statistics (p. 4339) for details.) If you increase the **Reporting Verbosity** to 2, the partition method used, the partition ID, number of cells, faces, and interfaces, and the ratio of interfaces to

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

faces for each partition will also be printed in the console window. If you decrease the **Reporting Verbosity** to 0, only the number of partitions created and the time required for the partitioning will be reported.

Across Zones

allows partitions to cross zone boundaries (the default). If turned off, it will restrict partitioning to within each cell zone. This is recommended *only* when cells in different zones require significantly different amounts of computation during the solution phase, for example if the domain contains both solid and fluid zones.

Laplace Smoothing

enables the Laplace smoothing method, which can be used to prevent partition boundaries from passing through regions of high aspect ratio cells. This can improve the convergence and scalability of the flow and conjugate heat transfer computations for cases with regions of highly stretched cells.

Cutoff Aspect Ratio

controls the threshold for the Laplace smoothing method.

Reordering Method

is used to optimize parallel performance

Architecture Aware

is the default option and it accounts for the system architecture and network topology in remapping the partitions to the processors.

Reverse Cuthill-McKee

minimizes the bandwidth of the compute-node connectivity matrix (the maximum distance between two connected processes) without incorporating the system architecture.

Optimization

tab contains options for enabling schemes to optimize the partitions created by the selected partition method. In addition, the optimization scheme will be applied until appropriate criteria are met, or the maximum number of iterations have been run. If the **Iterations** counter is set to 0, the optimization scheme will be applied until completion, without limit on the maximum number of iterations.

Merge

attempts to decrease the number of interfaces by eliminating orphan cell clusters (an orphan cluster is a group of connected cells whose members each have at least one face coincident with an interface boundary).

Smooth

attempts to minimize the number of interfaces by sacrificing cells on the partition boundary to the neighboring partition to reduce the partition boundary surface area.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Pre-Test

instructs Fluent to test all coordinate directions and choose the one which yields the fewest partition interfaces for the final bisection. Note that this option is available only when you choose **Principal Axes** or **Cartesian Axes** as the partitioning method.

Weighting

tab allows you to set the appropriate weights, prior to partitioning the mesh.

Weight Types

allows you to turn on/off a specific weight. Weight types include: **Faces per Cell** (enabled by default, with additional weighting of **2**), **Solid Zones** (available and enabled by default only if solid cell zones are defined); **VOF** (available only if the volume of fluid multiphase model is turned on); **DPM** (available only for discrete phase simulations with injections defined), where you can specify **Hybrid Optimization**; and **ISAT**.

User Specified

allows you to specify that Ansys Fluent timers set the weight scaling (by not enabling the check box), or to modify the value yourself (by enabling the check box and entering a numerical value in the **Value** field.)

Value

allows you to enter a user-specified numerical value to modify the weighting set by the Ansys Fluent timers. Note that for **VOF**, **DPM**, and **ISAT**, this value is not relevant when using model-weighted partitioning.

Dynamic Load Balancing

tab allows you to set load balancing thresholds and intervals.

Physical Models

allows you to set thresholds and intervals for the simulation's physical models.

Threshold

is a percentage and depending on the value, load balancing will occur if the load imbalance is greater than the value entered.

Interval

allows you to specify the desired interval for load balancing cycles, in terms of number of iterations. When a value of 0 is specified, Ansys Fluent will internally determine the best value to use, initially using an interval of 25 iterations. You can override this behavior by specifying a nonzero value. Ansys Fluent will then attempt to perform load balancing after every N iterations, where N is the specified **Balance Interval**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Dynamic Mesh

allows you to set load balancing threshold and interval values for your dynamic mesh. This option is only available when the dynamic mesh model is enabled.

Auto

allows a percentage of interface faces and loads to be automatically traced.

Threshold

is a percentage and depending on the value, load balancing will occur if the load imbalance is greater than the value entered.

Interval

allows you to specify the desired interval for load balancing cycles, in terms of number of iterations. When a value of 0 is specified, Ansys Fluent will internally determine the best value to use, initially using an interval of 25 iterations. You can override this behavior by specifying a nonzero value. Ansys Fluent will then attempt to perform load balancing after every N iterations, where N is the specified **Balance Interval**.

Mesh Adaption

allows you to set threshold values for mesh adaption.

Threshold

is a percentage and depending on the value, load balancing will occur if the load imbalance is greater than the value entered.

Zones

contains a list of cell zones. Partitioning will be applied to cells in zones selected from this list.

Registers

contains a list of cell registers that have been created using the adaption tools. You can restrict partitioning to a group of cells by selecting a register containing the cells. See Partitioning Within Zones or Registers (p. 4328) for details.

Print Active Partitions

prints the partition ID, number of cells, faces, and interfaces, and the ratio of interfaces to faces for each active partition (see Checking the Partitions (p. 4338) for information about active partitions) in the console. In addition, it prints the minimum and maximum cell, face, interface, and face-ratio variations.

Print Stored Partitions

prints the partition ID, number of cells, faces, and interfaces, and the ratio of interfaces to faces for each stored partition (see Checking the Partitions (p. 4338) for information about stored partitions) in the console. In addition, it prints the minimum and maximum cell, face, interface, and face-ratio variations.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Set Selected Zones and Registers to Partition ID

allows you to set a value that assigns selected **Zones** and/or **Registers** to a specific partition ID. A region or zone is marked before setting the marked cells to one of the partition IDs.

Use Stored Partitions

allows you to make the stored cell partitions the active cell partitions in a parallel session. The active cell partition is used for the current calculation, while the stored cell partition (the last partition performed) is used when you save a case file.

Partition

subdivides the mesh into the selected number of partitions using the prescribed method and optimization(s).

Default

sets all controls to their default values, as assigned by Ansys Fluent. After execution, the **Default** button becomes the **Reset** button.

Reset

resets the fields to their most recently saved values (that is, the values before **Default** was selected). After execution, the **Reset** button becomes the **Default** button.

52.2.91. Pathlines Dialog Box

The **Pathlines** dialog box controls the generation and display of pathlines. See Displaying Pathlines (p. 3922) for details about the items below.

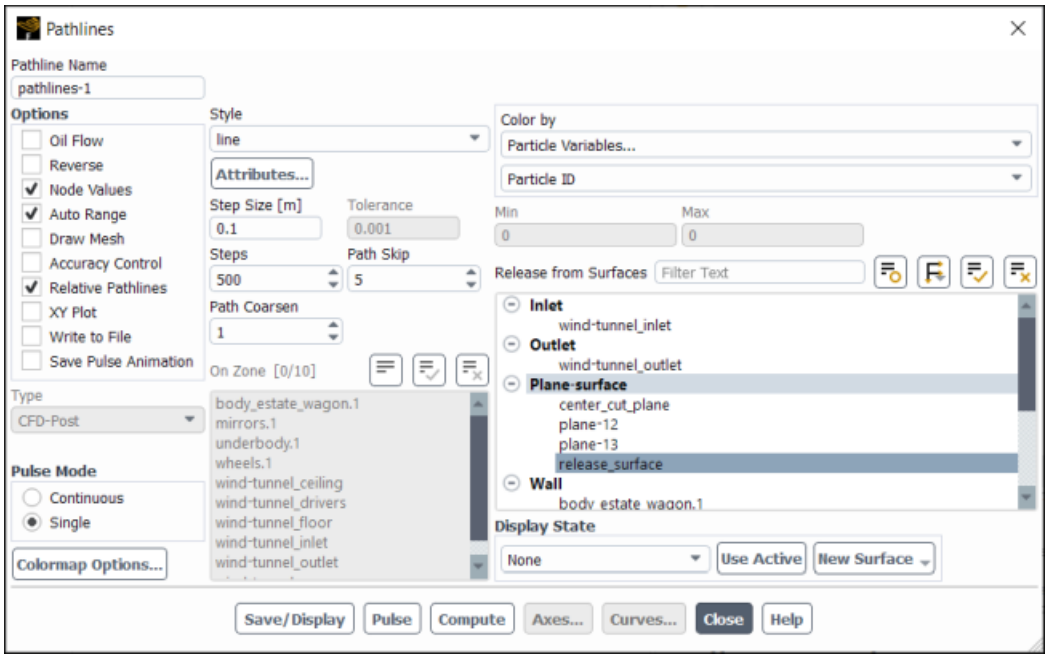
Results → **Graphics** → **Pathlines** → **New...**

Note:

- The "global"/non-persistent form of graphics objects is hidden by default. If this interrupts your workflow, you can make them available by enabling the **Expose legacy non-persistent graphics** option in the **Graphics** branch of **Preferences** (accessed via **File**>**Preferences...**).
- (When you click **Edit...** instead of **New...**, it opens the "global"/non-persistent version of the **Pathlines** dialog box. This version of the dialog box does not have a **Name** field and it is not saved with the case file.

  You are encouraged to use the persistent version that is accessed by clicking **New...**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Pathline Name

is the name for a pathline definition. You can specify a name or use the default name **`path-line-`*`id`*** . This control appears only for pathline definitions.

Options

contains the options described below:

Oil Flow

toggles between regular pathlines and oil-flow pathlines. When this option is selected, pathlines are constrained to lie on the zone(s) selected in the **On Zone** list.

Reverse

specifies that each particle's path is traced back in time. This option is turned off by default.

Node Values

specifies that node values should be interpolated to compute the scalar field at a particle location. This option is turned on by default. See Choosing Node or Cell Values (p. 3932) for details.

Auto Range

specifies that the minimum and maximum values of the scalar field will be the limits of that field. If this option is not selected, you can enter **Min** and **Max** values manually. These values determine the range of the color scale.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Draw Mesh

when enabled opens the Mesh Display Dialog Box (p. 4688) where you can specify how to display your mesh.

Accuracy Control

allows you to specify tolerance to control the pathlines accuracy.

Relative Pathlines

allows you to display the pathlines relative to the rotating reference frame.

XY Plot

enables the display of an XY plot along the pathline trajectories. When this option is selected, the **Display** push button will change to **Plot**.

Write to File

activates the file-writing option. When this option is enabled, the **Type** option becomes active and the **Save/Plot** button changes to **Save/Write...**. Clicking **Save/Write...** opens The Select File Dialog Box (p. 931) where you can specify a name and save a file containing the XY plot data. The format of this file is described in XY Plot File Format (p. 4071).

Save Pulse Animation

allows you to save a video or picture files of the pulse animation. When this option is enabled, an additional field for **Write/Record Format** appears, allowing you to specify how the pulse animation will be saved. Clicking **Save/Write...** opens The Select File Dialog Box (p. 931) where you can specify a name and save the video or picture files.

A **Continuous** pulse video file will be 5 seconds long, while a **Single** pulse video will be one full cycle.

Type

specifies the type of the file you want to write.

CFD-Post

allows you to write the file in CFD-Post compatible format, which can be read in **CFD-Post**.

Fieldview

allows you to write the file in **FIELDVIEW** format, which can be read in **FIELDVIEW**.

Geometry

allows you to write the file in `.ibl` format, which can be read in GAMBIT.

EnSight

allows you to write the file in `.encas` format, which can be read in EnSight.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Pulse Mode

specifies either a single or continuous release of particles that follow the pathlines, starting at the release surface, when you use the **Pulse** button to animate the pathlines.

Continuous

sets the pulse mode to continuously release particles from the initial positions.

Single

sets the pulse mode to release a single wave of particles.

The **Pulse Mode** is used only when particles are being pulsed.

Colormap Options...

opens the Colormap Dialog Box (p. 5280), allowing you to customize the colormap for this graphics object.

Style

sets the pathline style. Pathlines can be displayed as lines, ribbons, spheres, cylinders, or a set of points. The **ribbon** style also uses the **Twist By** field and the **TwistFactor** value (specified in the Ribbon Attributes Dialog Box (p. 5243)). Pulsing can be done only on **point** or **line** styles.

Attributes...

opens the Path Style Attributes Dialog Box (p. 5243) or the Ribbon Attributes Dialog Box (p. 5243), depending on the selected **Style**. These dialog boxes allow you to set the attributes (ribbon width, marker size, cylinder radius, twist-by field, and so on) for the chosen pathline style.

Step Size

sets the interval used for computing the next position of a particle. This is in units of length. Particle positions are always computed when they enter/leave a cell. If you specify a very large size, the particle positions at entry/exit of each cell will still be computed (and displayed).

Tolerance

allows you to control the error when using large time step sizes for the calculation. This is available when **Accuracy Control** is selected under **Options**.

Steps

sets the maximum number of steps a particle can advance. A particle's path may also end if it leaves the domain.

Path Skip

allows you to "thin" or "sample" the pathlines that are displayed. See "Thinning" Pathlines (p. 3926) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Path Coarsen

reduces the time and the number of points plotted in a pathline. Coarsening factor of 'n' will result in plotting of each 'n'th point for a given pathline in each cell.

On Zone

contains a list from which you can select the zone(s) on which the particles are constrained to lie. This list is activated only when the **Oil Flow** option is selected.

Track in Phase

(for the Eulerian Multiphase model only) allows you to select the phase or the mixture of all phases for tracking particle pathlines.

Color by

contains a list from which you can select the scalar field to be used to color the pathlines. The default is **Particle ID**, a unique ID for each particle.

Y Axis Function

contains a list of solution variables that can be used for the y axis of the plot.

This item appears when the **XY Plot** option is turned on.

X Axis Function

contains a list of functions for the x axis of the plot. You can plot the quantity selected in the **Y Axis Function** drop-down list as a function of the **Time** elapsed along the trajectory, or the **Path Length** along the trajectory.

This item appears when the **XY Plot** option is turned on.

Phase

(Multiphase only) specifies the phase where the postprocessing operations are being applied.

Min/Max

display the values to which the lower and upper ends of the color scale map. If you are using the **Auto Range** option, these values will be automatically computed when you click **Compute**. If you are not using **Auto Range**, you can enter values manually.

Release from Surfaces

selects the surfaces from which to release particles. You must select at least one surface before any particles can be released.

Display State

specifies the display state that is associated with this graphics object.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

drop-down list of display states

selectable list of all the existing display states associated with this case.

Use Active

creates a display state that matches the settings of the active graphics window and assigns it to this graphics object (this association is only saved after you click **Save/Display**).

New Surface

is a drop-down list button that contains a list of surface options:

Point

opens the Point Surface Dialog Box (p. 5582).

Line/Rake

opens the Line/Rake Surface Dialog Box (p. 5514).

Plane

opens the Plane Surface Dialog Box (p. 5580).

Quadric

opens the Quadric Surface Dialog Box (p. 5584).

Iso-Surface

opens the Iso-Surface Dialog Box (p. 5501).

Iso-Clip

opens the Iso-Clip Dialog Box (p. 5500).

Structural Point

opens the Structural Point Surface Dialog Box (p. 5623).

Write/Record Format

(only present when **Save Pulse Animation** is enabled) drop-down list specifying whether you are saving the pulse animation as a video file or as picture files. The default formats are shown in parenthesis next to the field.

Video Options...

(only present when **Save Pulse Animation** is enabled and **Video File** is selected for **Write/Record Format**) opens the Video Options Dialog Box (p. 5260).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Picture Options...

(only present when **Save Pulse Animation** is enabled and **Picture Files** is selected for **Write/Record Format**) opens the Save Picture Dialog Box (p. 5254).

EnSight Encas File Name

allows you to specify a.encas file name if you selected **EnSight** under **Type**.

Browse...

opens The Select File Dialog Box (p. 931) where you can select the `.encas` file.

Time Steps For EnSight Export

allows you to set the number of time levels that are available for animation in EnSight.

Display

displays the pathlines in the active graphics window.

Save/Display

plots the pathlines in the active graphics window and saves the pathline definition. This button appears only for pathline definitions and replaces the **Display** button.

Plot

displays an XY plot along the pathline trajectories. This button replaces the **Display** button when the **XY Plot** option is turned on.

Write...

opens The Select File Dialog Box (p. 931), in which you can save the XY plot data to a file. This button replaces the **Plot** button when the **Write to File** option is turned on.

Pulse

animates the position of particles. If **Pulse Mode** is set to **Continuous**, particles are reintroduced at the seed positions repeatedly. If the mode is **Single**, a single wave of particles moves through the domain.

The label of this button changes to **Stop !** when the animation is in progress. Click **Stop !** to stop pulsing (and the label goes back to **Pulse**).

Compute

computes the **Min** and **Max** for the field chosen in **Color by**.

Axes...

opens the Axes Dialog Box (p. 5313), which allows you to customize the XY plot axes. This item appears when the **XY Plot** option is turned on.

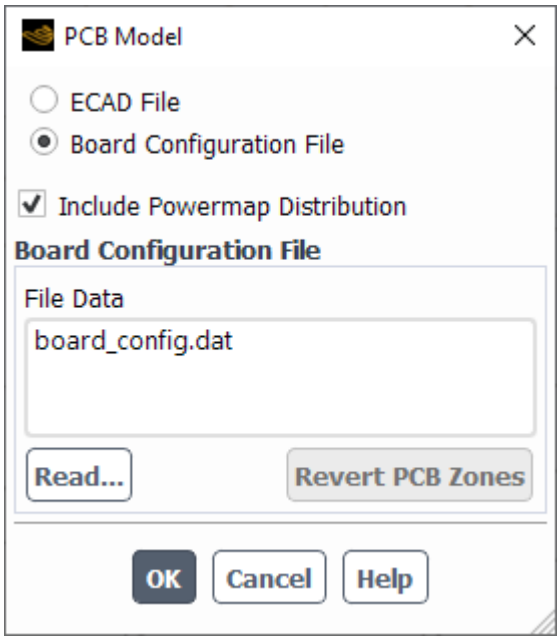
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Curves...

opens the Curves Dialog Box (p. 5316), which allows you to customize the curves used in the XY plot. This item appears when the **XY Plot** option is turned on.

52.2.92. PCB Model Dialog Box

The **PCB Model** dialog box allows you to model printed circuit boards (PCB) that vary considerably due to their layered construction and highly variable metal fraction at any particular location. See Thermal Analysis of Printed Circuit Boards (p. 2319) for details about the items below.



Controls

Board Configuration File

contains the options for the PCB model described below:

Read..

reads the board configuration file produced by Ansys Icepak.

File Data

displays the name of the board configuration file.

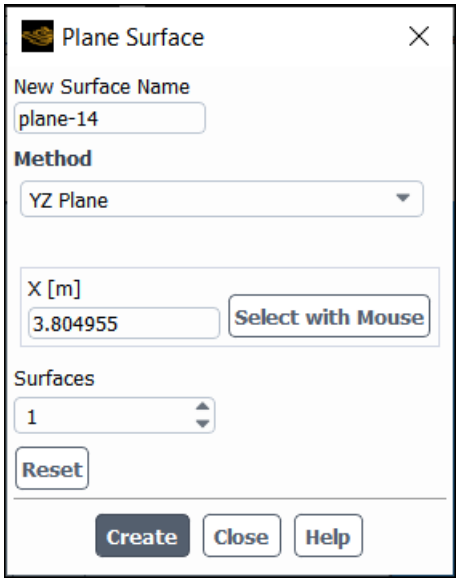
Include Powermap Distribution

select to include powermap distribution as part of the analysis from additional files generated by Ansys Icepak.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

52.2.93. Plane Surface Dialog Box

The **Plane Surface** dialog box allows you to interactively create one or more planar surfaces that cut through the domain. See Plane Surfaces (p. 3830) for details about the items below.



Controls

New Surface Name

designates the name of the new surface. The default is the concatenation of the surface type and an integer which is the new surface ID.

Name Format

[Creating multiple YZ, ZX, XY planes only] indicates how Fluent will name the surfaces.

The field will look something like `plane-x={x:+.6f}`. This naming format results in plane names that reflect the coordinates of the plane. For example, if you create three YZ planes, with a spacing of 0.05m, starting from X=2.325m, the planes will be named `plane-x=+2.325000`, `plane-x=+2.375000`, and `plane-x=+2.425000`. You can control how many significant digits are included in the plane name by varying the value represented by 6 in the example name format. The `f` in the example name indicates that the output is a float. Changing the `f` to an `e` results in the coordinate being represented as an exponential.

Method

displays the selected plane creation method.

YZ Plane

create a plane surface in the YZ plane.

ZX Plane

create a plane surface in the ZX plane.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

XY Plane

create a plane surface in the XY plane.

Point and Normal

create a plane surface based on a point and its normal. Normal can optionally be computed from a surface or the current view.

Three Points

create a plane surface based on three points.

Normal

settings related to specifying the normal direction for a **Point and Normal** plane surface.

iX, iY, iZ

designates the coordinates of the end point of the normal vector (the start point being 0, 0, 0).

Compute from Surface

computes values for the **Normal** based on the surface selected from the drop-down above.

Compute from View Plane

computes values for the **Normal** based on the current view orientation in the graphics window.

Options

contains options for defining the plane surface.

Bounded

enables the creation of a bounded parallelepiped, 3 of whose 4 corners are the 3 points that define the plane equation (or the 4 corners of the **Plane Tool**).

Sample Points

enables the specification of a point density along the 2 directions (of the parallelepiped). This creates a uniformly distributed set of points on the plane. (This item is available only when the **Bounded** option is enabled.)

Sample Density

specifies the density of points when the **Sample Points** option is enabled.

Edge 1, Edge 2

sets the point density along the two directions of the plane. (Edge 1 extends from point 0 to point 1, and edge 2 extends from point 1 to point 2.)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Select with Mouse

allows you to select points with the mouse. You can select endpoints by clicking on locations in the active window with the mouse-probe button. (See Controlling the Mouse Button Functions (p. 4020) for information about setting mouse button functions.)

Point 1

designates the coordinates of the first point.

Point 2

designates the coordinates of the second point.

Point 3

designates the coordinates of the third point.

Surfaces

lists the number of surfaces that will be created at once.

Spacing

specifies the distance between surfaces.

Reset

resets the plane tool to its default position.

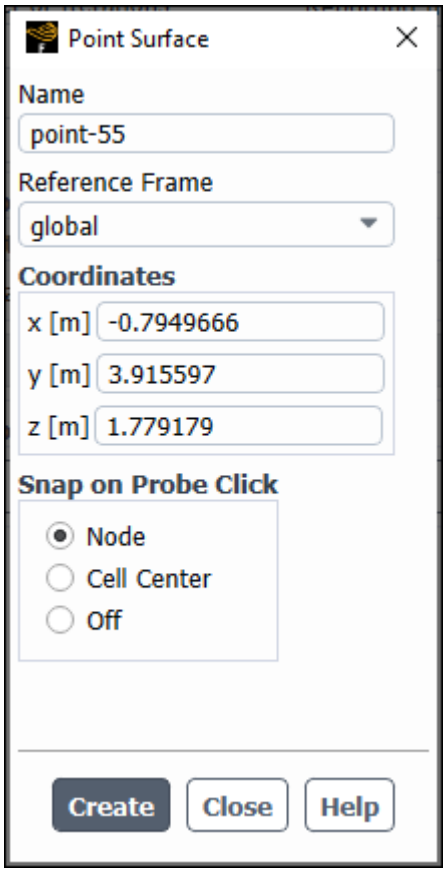
Create

creates the surface.

52.2.94. Point Surface Dialog Box

The **Point Surface** dialog box allows you to interactively create point surfaces (surfaces containing a single data point). See Point Surfaces (p. 3821) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name

designates the name of the new surface. The default is the concatenation of the surface type and an integer which is the new surface ID.

Reference Frame

allows you to specify a local reference frame (if defined) for the point surface.

Coordinates

designates the coordinates of the point in the surface (**x0**, **y0**, **z0**).

Snap on Probe Click

updates the coordinates of the point to the closest **Node** or **Cell Center** when you click on the model in the graphics window using the probe tool—*that is, either by right-clicking or setting the left-mouse-button to probe (* *)*.

You can disable automatic snapping by selecting **Off**.

Create

creates the surface.

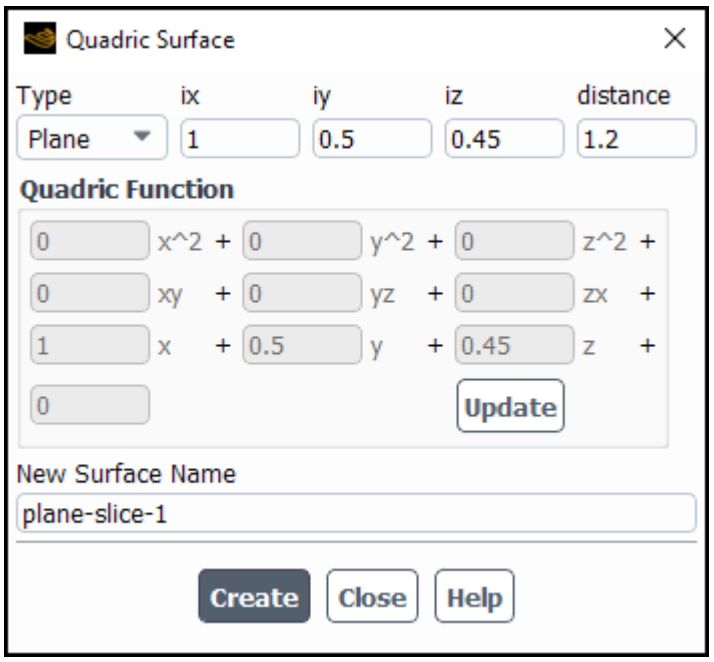
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

52.2.95. Quadric Surface Dialog Box

The **Quadric Surface** dialog box allows you to interactively define quadric functions and create surfaces from them. Lines and circles are commonly used types of quadric functions, so additional support is provided for defining them. See Quadric Surfaces (p. 3836) for details about using the items below.

Important:

Note that your inputs for the quadric function must be in SI units.



Controls

Type

selects the type of quadric function. Presently, you may select **Line/Plane**, **Circle/Sphere** or a general **Quadric**. Some of the control buttons take on different meanings for each of the three quadric function types.

Line/Plane

specifies that the surface will consist of all points on the domain that satisfy the equation $ix*x+iy*y+iz*z=distance$, where

ix

is the coefficient of x in the quadric function.

iy

is the coefficient of y in the quadric function.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

iz

is the coefficient of z in the quadric function.

distance

is the distance of the line/plane from the origin.

When you click the **Update** button in **Quadric Function**, the coefficients in the quadric function will change to reflect your inputs.

Circle/Sphere

specifies that the surface will consist of all points in the domain that satisfy the equation $(x-x0)^2+(y-y0)^2+(z-z0)^2=r^2$, where

x0,y0,z0

are the x,y,z coordinates of the center.

r

is the radius.

When you click the **Update** button in **Quadric Function**, the coefficients in the quadric function will change to reflect your inputs.

Quadric

specifies that the surface will consist of all points in the domain that satisfy the quadric function $Q=value$, where Q is the quadric function you define as shown below. You will enter **value** to the right of the **Type** entry, but the coefficients in the quadric function are entered directly in the **Quadric Function** box. Note that in 2D problems the z entries are ignored.

x^2, y^2, z^2, xy, yz, zx, x, y, z

are the coefficients of the corresponding terms in the quadric function.

Quadric Function

contains the display of the quadric function and the **Update** button. If you select **Line/Plane** or **Circle/Sphere** as the **Type**, the coefficients of the quadric function will be updated here when you click the **Update** button. You will not be able to edit the values in the **Quadric Function** box directly. If you select **Quadric** as the **Type**, you will enter the coefficients directly in this box, and the **Update** button will be inactive.

Update

updates the coefficients in **Quadric Function** to reflect the current input.

New Surface Name

designates the name of the new surface. The default is the concatenation of the surface type and an integer which is the new surface ID.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Create

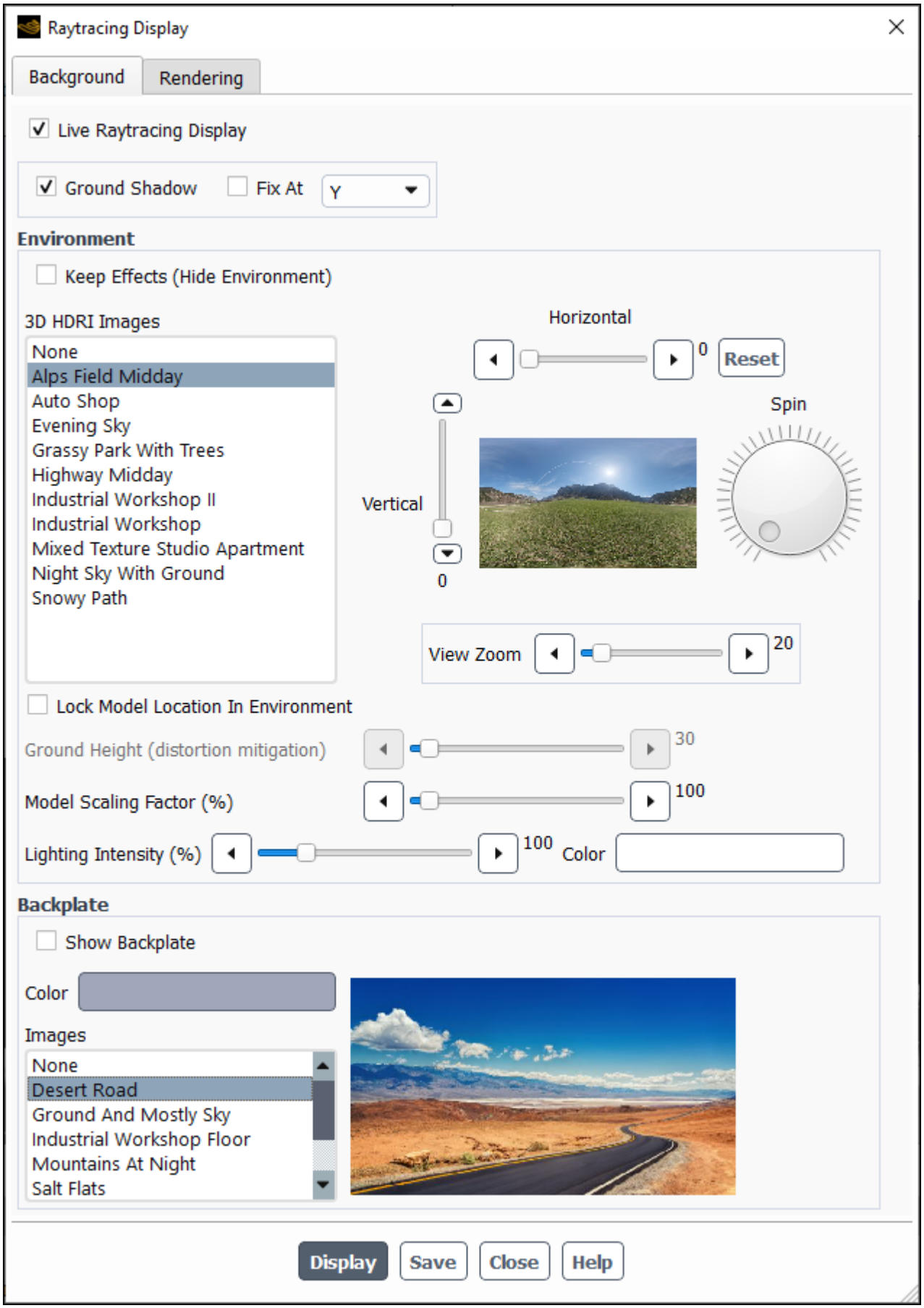
creates the surface.

52.2.96. Raytracing Display Dialog Box

The **Raytracing Display** dialog box allows you to create a realistic display that can incorporate lighting effects from an environment or lights defined in the case, and show your model in an environment with a backplate image or color. For information on how to use this dialog box, refer to Realistic Rendering Using Raytracing (p. 3995).

View → Graphics → Raytracing Display...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Controls

Background

tab that contains settings for the environment and backplate, which can be used to provide lighting effects and a background to the displayed model. *Changes made to settings on the **Background** tab while this option is enabled are automatically saved without the need to click **Save**.*

Live Raytracing Display

allows for changes made to controls in the **Background** tab to be immediately reflected in the raytracing window (which is only seen after you click **Display**).

Ground Shadow

displays shadows below the displayed object based on the lighting defined in the session as well as the lighting in the environment.

Fix At

locks the plane location below the model so that the shadows are shown appropriately for the orientation of the displayed model. The axis that you specify should be the plane that is opposite to the pull of gravity. For example, if gravity is in the -Z direction, then select **Z** from the drop-down.

Environment

group box containing options that control the environment that can be used to affect the appearance of the model.

Keep Effects (Hide Environment)

this option keeps the effects of the environment on the model without showing the model in the environment. The background will be the same as the standard graphics display, unless you enable the **Show Backplate** option (in the **Backplate** group box).

3D HDRI Images

lists the available environments that you can use for your raytracing display. The light sources in these environments affect the appearance of the displayed model. A preview of each environment is shown in the image window to the right of the list.

Vertical

slider that controls the orientation of the environment. Adjusting this slider moves the environment vertically.

Horizontal

slider that controls the orientation of the environment. Adjusting this slider moves the environment horizontally.

Reset

resets the environment orientation.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Spin

rotation control that controls the orientation of the environment. Adjusting the spin control rotates the environment either clockwise or counterclockwise.

View Zoom

slider that controls the overall zoom level of the raytracing window (sometimes referred to as "field view"). Zooming in/out with this control changes the zoom level of the entire raytracing display window, including the environment and the model. You can then re-adjust the zoom level of the model by zooming in/out on the model in the original active window containing the interactive model display that is being raytraced.

Lock Model Location In Environment

converts the environment from "limitless bounds" to "finite bounds", thus making it so that the displayed object always appears in the same location, even when rotated. *Note that your raytracing display may require additional adjustment of the **Ground Height** to compensate for the distortion that can appear on vertical objects in the environment. The distortion comes because 3D HDRI images are spherical.*

Ground Height (distortion mitigation)

(only available when **Lock Model Location in Environment** is enabled) controls where the ground height appears in within the sphere that is the environment. *Adjusting this slider can help address some of the distortion that may appear on vertical objects due to the rounded nature of the spherical environment.*

Model Scaling Factor

controls how large or small the displayed object appears in relationship to the environment.

Lighting Intensity

slider controlling the intensity of the lighting for the selected environment. Moving the slider to the right increases the intensity of the light and moving it to the left decreases it.

Color

specifies the color that will be used to color the environment lighting.

Backplate

group box containing options that control the backplate that can be displayed behind the model.

Show Backplate

controls whether or not a backplate is displayed behind the model in the raytracing window. *When this option is enabled, the **Color** will be used unless you select an image from the **Images** list. The backplate will always be displayed "over top" of any selected environment.*

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Color

specifies the color that will be displayed as the background behind the model when **Show Backplate** is enabled and **Images** is set to **None**.

Images

lists the available 2D backplates that you can use for your raytracing display. The selected image will be displayed as the background behind the model when **Show Backplate** is enabled. A preview of each image is shown in the window to the right of the list.

Rendering

tab that contains settings controlling how the raytracer functions and how pictures or hardcopies are saved.

CPU Threads

specifies how many processors are used by the raytracer in rendering the image. *If the number of **CPU Threads** specified is greater than the number of cores on the machine, the raytracer will use the maximum number available.*

Quality

specifies how "good" an image looks when rendered and saved as hardcopy picture. Better quality renderings are more expensive, that is they require more machine resources and take longer to render.

Save Picture...

opens the **Save Picture** dialog box (Save Picture Dialog Box (p. 5254)), allowing you to capture a hardcopy of the raytracer rendering.

Max. Rendering Time Before Image Capture [s]

specifies how long Fluent waits for the raytracer to render the display to the specified **Quality** before capturing the image.

Display

saves the current settings and displays the raytracer rendering of the active graphics window according to the specified settings.

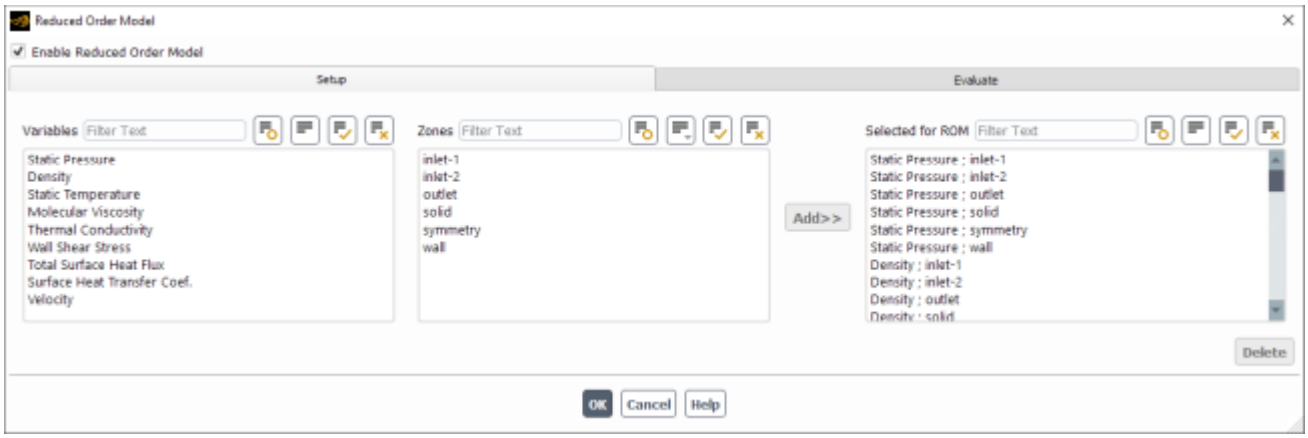
Save

saves the raytracer settings. *This allows you to define a raytracer rendering and save a picture without displaying it in the graphics window first.*

52.2.97. Reduced Order Model Dialog Box

The **Reduced Order Model** dialog box allows you to enable the reduced order model (ROM) and select which variables and zones to include in the ROM—refer to Creating Reduced Order Models (ROMs) (p. 3561) additional information. You can also use it to evaluate a produced ROM as discussed in Reduced Order Model (ROM) Evaluation in Fluent (p. 3563).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Setup tab

Variables

lists the variables that are available for creating the ROM.

Zones

lists all of the zones in your case. You can select from this list to tell the ROM where the selected variables in the **Variables** list will be evaluated.

Add>>

adds the selected variables and zones to the ROM.

Selected for ROM

lists all of the selected variables and their associated locations for the ROM.

Delete

removes the selected variables and zones from the **Selected for ROM** list.

Evaluate tab

Parameters used for ROM creation:

lists all of the variables that were used in creation of the reduced order model (ROM). The **Min-Max Range** on the right of the fields lists the accepted input ranges for the number entry fields.

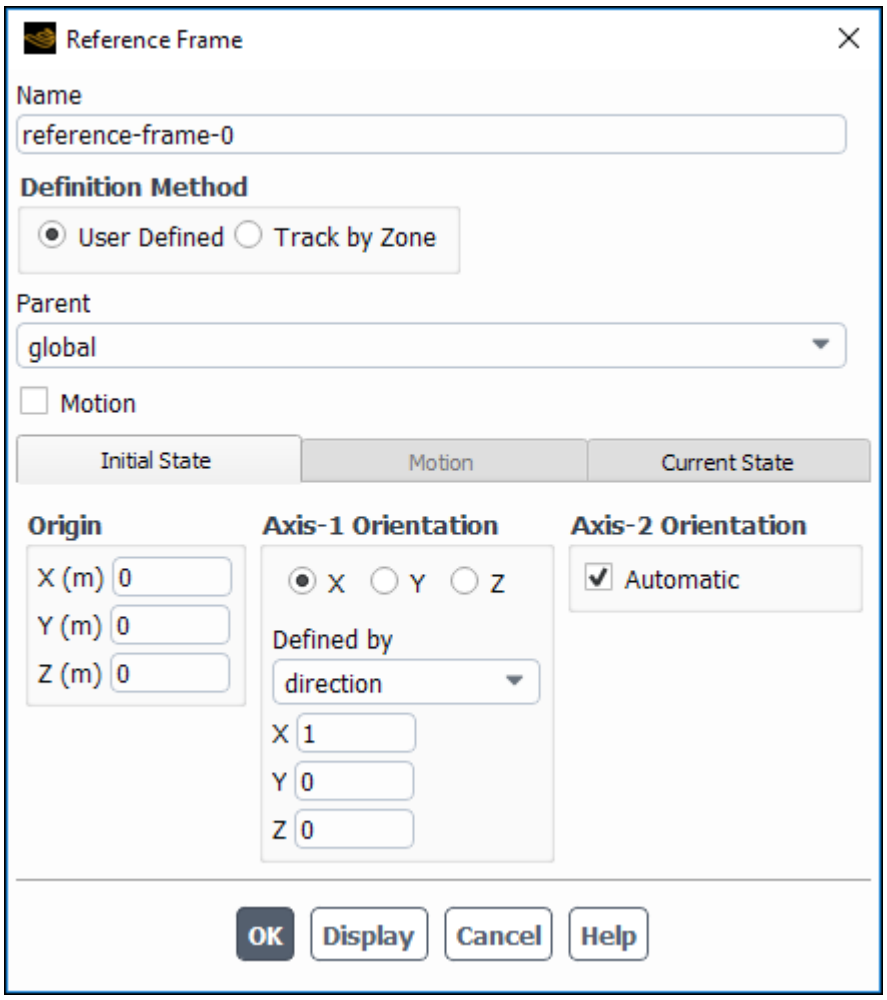
Evaluate

evaluates the case using the values you specify for the listed parameters.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

52.2.98. Reference Frame Dialog Box

The **Reference Frame** dialog box allows you to create local reference frames either with the coordinates and orientation that you specify or by tracking a cell zone. Refer to Reference Frames (p. 1160) additional information.



Controls

Definition Method

specifies how you want to define the reference frame.

Parent (User Defined only)

specifies the parent reference frame.

Zone (Track by Zone only)

specifies which cell zone this reference frame will track.

Motion (User Defined Only)

allows you to specify a motion for this reference frame (accomplished in the **Motion** tab.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Linear Velocity (Motion tab)

is the linear velocity with respect to the parent reference frame orientation.

Rotational Velocity (Motion tab)

is the rotational velocity with respect to the parent reference frame orientation.

Rotation Axis

is the axis about which the rotational velocity will be applied.

Origin

specifies the origin of the reference frame.

Axis-1 Orientation

specifies the orientation of the selected axis.

Axis-2 Orientation

specifies the orientation of the second axis. By default Fluent does this automatically, however you can disable **Automatic** and specify the orientation manually.

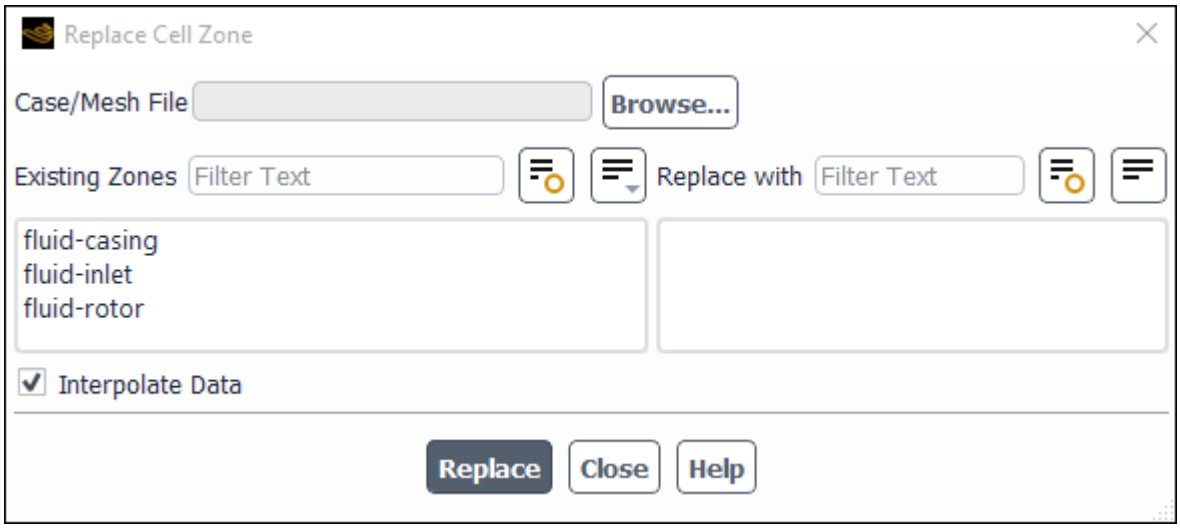
Current State (tab)

shows you the current origin and axes directions of the reference frame with respect to the global reference frame.

52.2.99. Replace Cell Zone Dialog Box

The **Replace Cell Zone** dialog box allows you to replace a single cell zone or multiple zones. See Replacing Zones (p. 1275) for details.

Domain → Zones → Replace Zone...



Controls

Case/Mesh File

allows you to specify a mesh file from which you want to replace the zone.

Existing Zones

contains a list of cell zones from which you can select the zone to be replaced.

Replace with

contains a list of cell zones from which you can select the zone to replace the zone selected in **Existing Zones** list.

Interpolate Data

allows you to enable/disable data interpolation if data already exists.

Replace

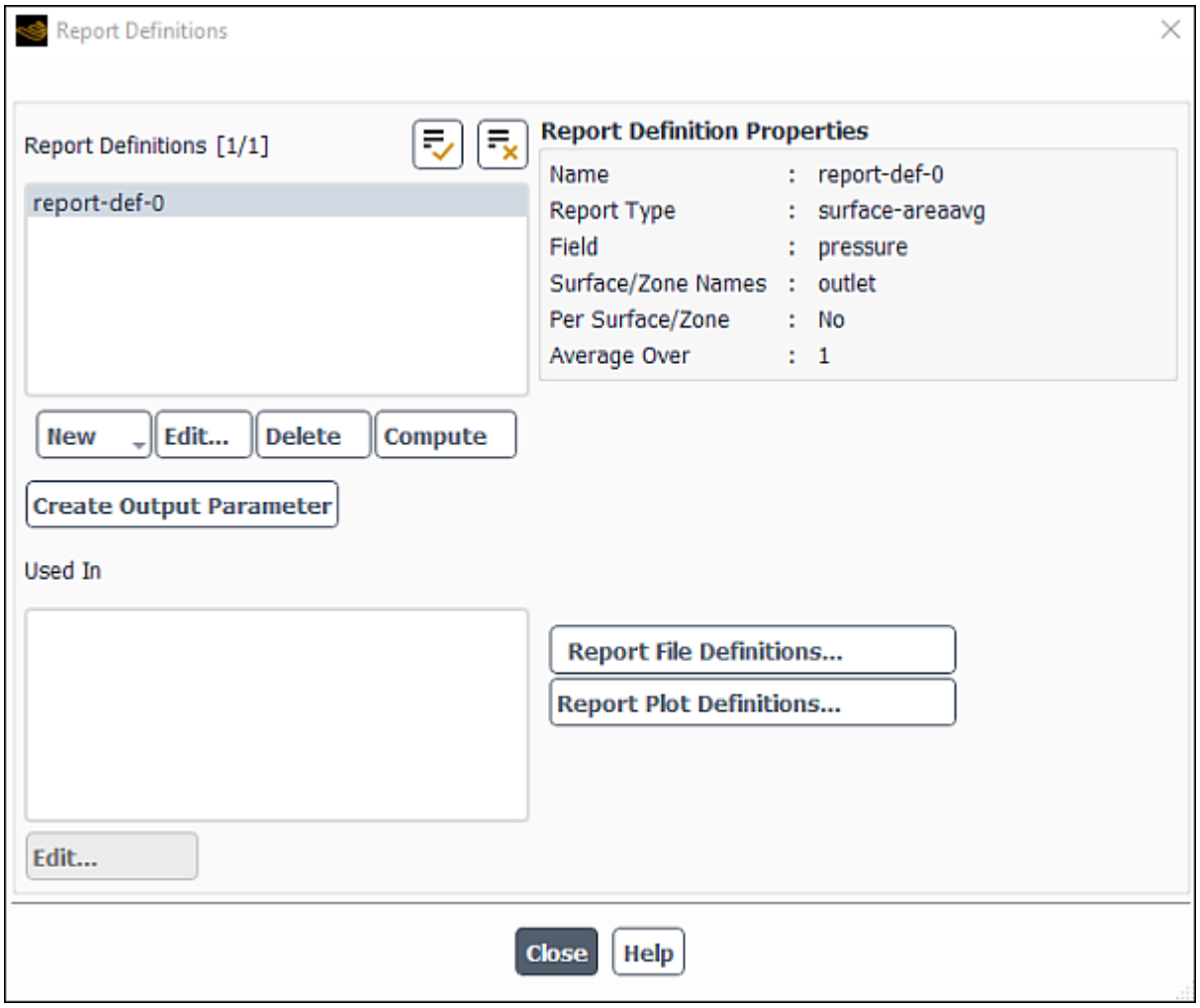
replaces the selected cell zone.

52.2.100. Report Definitions Dialog Box

Solution → **Reports** → **Definitions** → **Edit...**

For information on creating report definitions, see Monitoring and Reporting Solution Data (p. 4098).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Report Definitions

lists all of the existing report definitions.

New

drop-down list allowing you to create new report definitions (surface, volume, force, flux, mesh, aeromechanics, dpm, expression, and user-defined).

Edit...

allows you to edit the properties of the selected report definition.

Delete

deletes the selected report definition.

Compute

computes the value of the selected report definition and prints to the console.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Create Output Parameter

creates an output parameter for the selected report definition(s) with the name `<report definition name>-op`.

Used In

lists the report files and report plots that contain the selected report definition.

Edit... (button)

opens the report file/plot dialog box for the report file/plot selected in the **Used In** list.

Report Definition Properties

displays the properties of the selected report definition in the **Report Definition** list.

Report File Definitions...

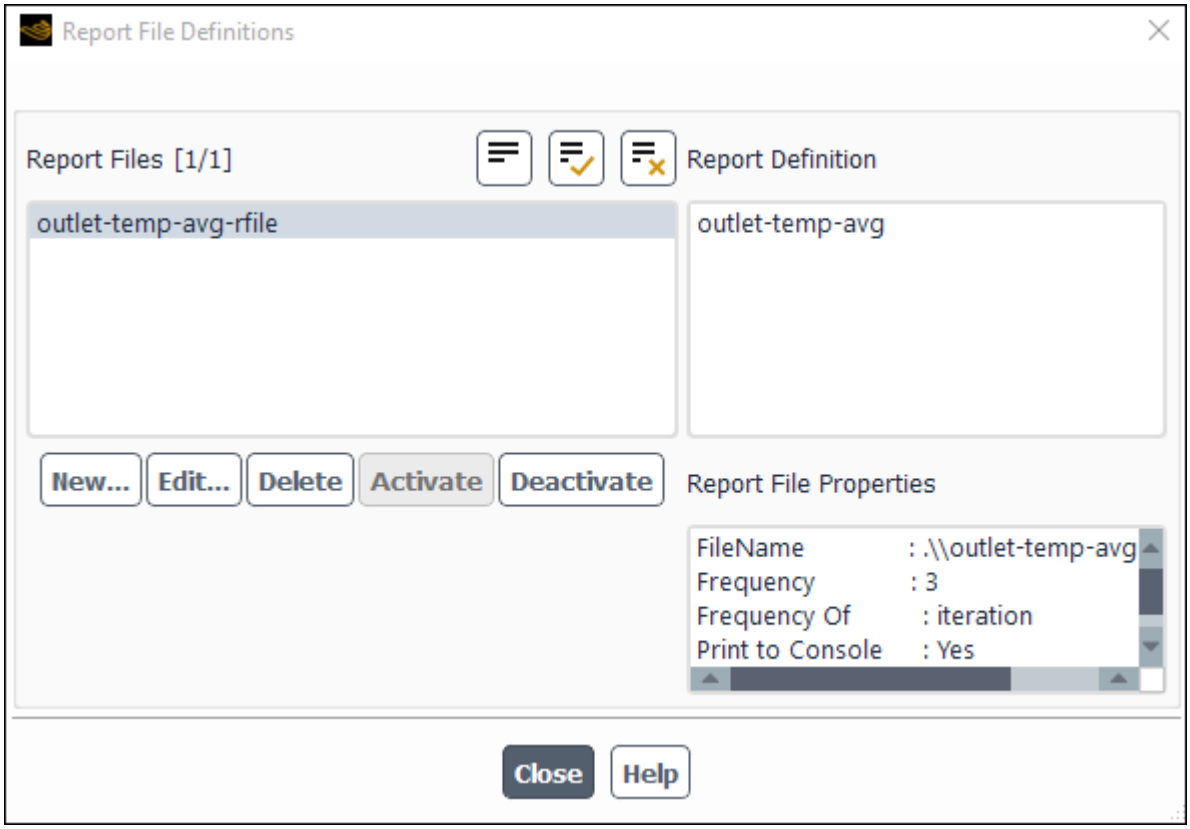
opens the Report File Definitions Dialog Box (p. 5596).

Report Plot Definitions

opens the Report Plot Definitions Dialog Box (p. 5597).

52.2.101. Report File Definitions Dialog Box

Solution → Reports → File...



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Controls

Report Files

lists all of the existing report files.

New...

opens the New Report File Dialog Box (p. 5550).

Edit...

allows you to edit the properties of the selected report file in the Edit Report File Dialog Box (p. 5456).

Delete

deletes the selected report file.

Activate

activates the selected (previously deactivated) report file for writing during the solution calculation.

Deactivate

deactivates the writing of the selected report file during the solution calculation.

Report Definition

lists the report definitions included in the selected report file.

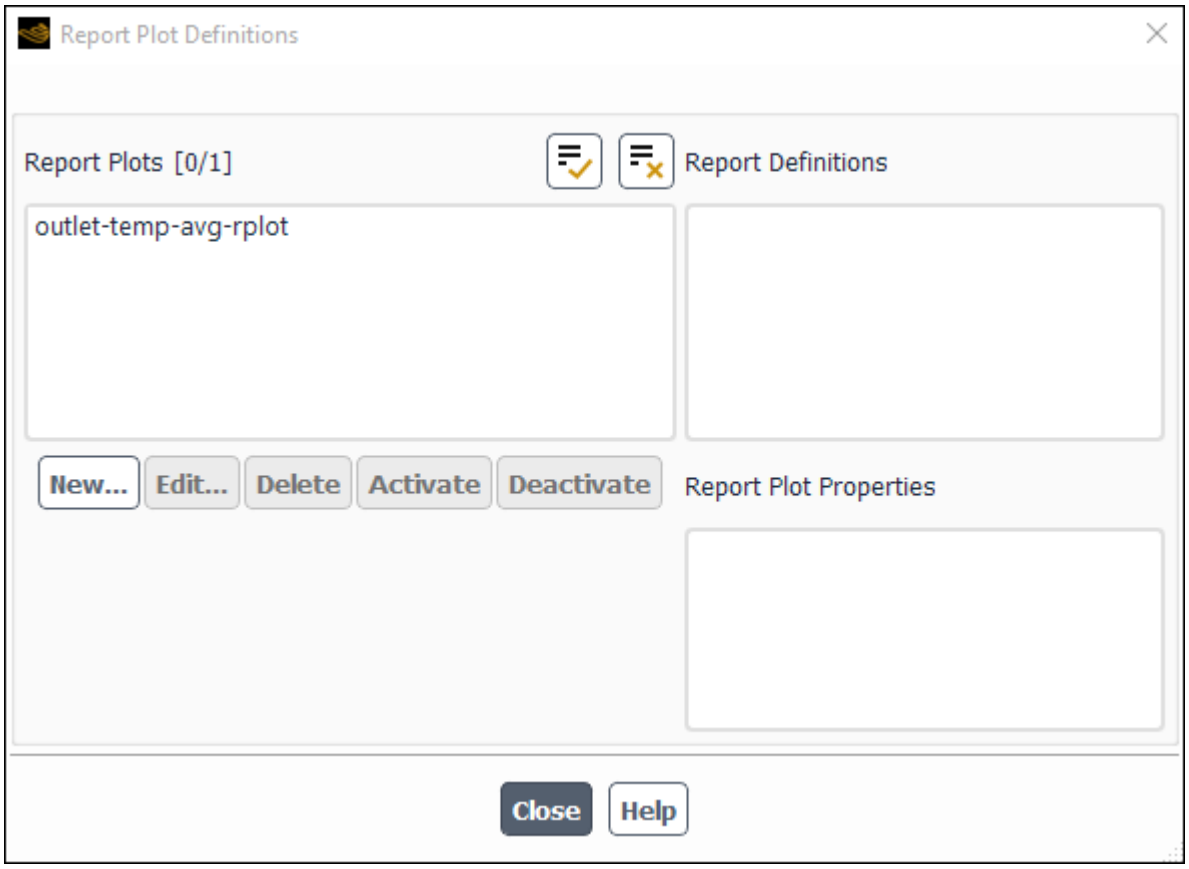
Report File Properties

lists the properties of the selected report file.

52.2.102. Report Plot Definitions Dialog Box

Solution → Reports → Plot...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Report Plots

lists all of the existing report plots.

New...

opens the New Report Plot Dialog Box (p. 5552).

Edit...

allows you to edit the properties of the selected report plot in the Edit Report Plot Dialog Box (p. 5458).

Delete

deletes the selected report plot.

Activate

activates the selected (previously deactivated) report plot for plotting during the solution calculation.

Deactivate

deactivates the plotting of the selected report plot during the solution calculation.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Report Definition

lists the report definitions included in the selected report plot.

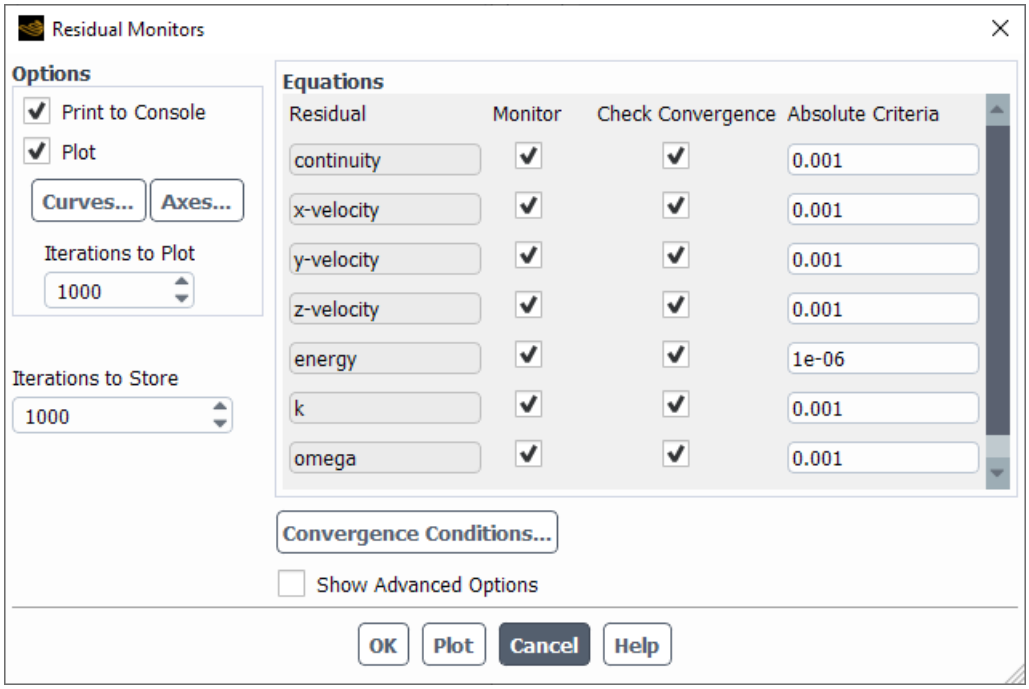
Report Plot Properties

lists the properties of the selected report plot.

52.2.103. Residual Monitors Dialog Box

You can use the **Residual Monitors** dialog box to control the residual information that Fluent reports. See Monitoring Residuals (p. 3684) for details about the items below.

Solution → Reports → Residuals...



Controls

Options

selects any combination of the following methods for reporting residuals. See Printing and Plotting Residuals (p. 3689) for details.

Print to Console

specifies whether or not to print residuals in the text window after each iteration.

Plot

specifies whether or not to plot residuals in the graphics window after each iteration. See Plot Parameters (p. 3694) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Iterations to Plot

is the number of history points to display on the residual plot.

Axes...

opens the Axes Dialog Box (p. 5313), which allows you to modify the attributes of the axes.

Curves...

opens the Curves Dialog Box (p. 5316), which allows you to modify the attributes of the residual curves.

Iteration to Store

sets the number of residual history points to be stored in the data file. Due to the compaction algorithm used, saving 1000 points does not result in just the last 1000 iterations being saved; the history reaches back quite a bit further than that, but does not save a point at every iteration. Further back in the iteration history, the spacing between saved iterations grows larger. See Storing Residual History Points (p. 3690) for details.

Convergence Conditions...

opens the Convergence Conditions Dialog Box (p. 5416).

Residual

indicates the name of each variable for which residual information is available.

Monitor

indicates whether or not the residuals for each variable are to be monitored. You can toggle monitoring on and off for each variable by turning the corresponding check box in the **Monitor** list on or off.

Normalization Factor

shows the normalization factor for each variable. The default is the maximum residual value after the first 5 iterations. To set this value manually, enter a new value in the corresponding **Normalization Factor** field. This list will not appear if the **Normalize** option is turned off.

Check Convergence

indicates whether or not the convergence of each variable is to be monitored. If convergence is being monitored, the solution will stop automatically when each variable meets its specified convergence criterion. You can check convergence only for variables for which you are monitoring residuals. You can toggle convergence checking on and off for each variable by turning the corresponding check box in the **Check Convergence** list on or off.

Absolute Criteria, Relative Criteria

shows the residual value for which the solution of each variable will be considered to be converged. To set this value manually, enter a new value in the corresponding **Absolute Criteria**/**Relative Criteria** field.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Plot

displays the current residual history plot.

Show Advanced Options

Residual Values

controls the normalization and scaling of residuals. See Controlling Normalization and Scaling (p. 3690) for details.

Normalize

specifies whether or not to normalize the printed or plotted residual for each variable by the value indicated as the **Normalization Factor** for that variable. The default **Normalization Factor** is the maximum residual value after the first 5 iterations.

This option is off by default.

Iterations

sets the number of iterations for which Ansys Fluent will search for the largest residual to normalize by. (If the **Normalize** option is turned off, this item will not be editable.)

Scale

specifies whether or not to print or plot scaled residuals for each variable. This option is on by default.

Compute Local Scale

computes and stores both the locally and globally scaled residuals from subsequent iterations, for the purpose of reporting. You will select the type of residual scaling form the **Reporting Option** drop-down list. See Definition of Residuals for the Pressure-Based Solver (p. 3684) and Definition of Residuals for the Density-Based Solver (p. 3686) for more information.

Reporting Option

gives you the choice of plotting or printing to the console the **global scaling** or **local scaling** of residuals.

Renormalize

sets the normalization factors to the maximum values in the residual histories. **Renormalize** should be used to renormalize the residual plot in cases where the maximum residuals occur sometime after the first five iterations.

Convergence Criterion

consists of four options that are available for checking an equation for convergence.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

absolute

is the default and is available for steady-state cases. The residual (scaled and/or normalized) of an equation at an iteration is compared with a user-specified value. If the residual is less than the user-specified value, that equation is deemed to have converged for a time step.

relative

is where the residual of an equation at an iteration of a time step is compared with the residual at the start of the time step. If the ratio of the two residuals is less than a user-specified value, that equation is deemed to have converged for a time step.

relative or absolute

is where either the **absolute** convergence criterion or the **relative** convergence criterion is met. At that point, the equation is considered converged.

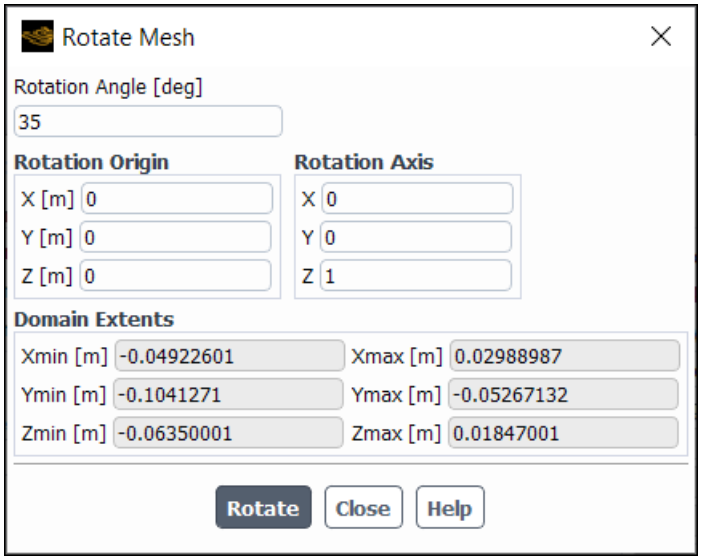
none

is used to disable convergence checking.

52.2.104. Rotate Mesh Dialog Box

The **Rotate Mesh** dialog box allows you to rotate the mesh about the required axis and rotation origin by specifying the angle of rotation. See Rotating the Mesh (p. 1289) for details.

Domain → Mesh → Transform → Rotate...



Controls

Rotation Angle

is the angle with which you want to rotate the mesh. You can enter a positive or negative real number.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Rotation Origin

defines the new origin for the mesh rotation.

Rotation Axis

defines the axis about which you want to rotate the mesh.

Domain Extents

displays the Cartesian coordinate extremes of the nodes in the mesh. (These values are not editable; they are purely informational.)

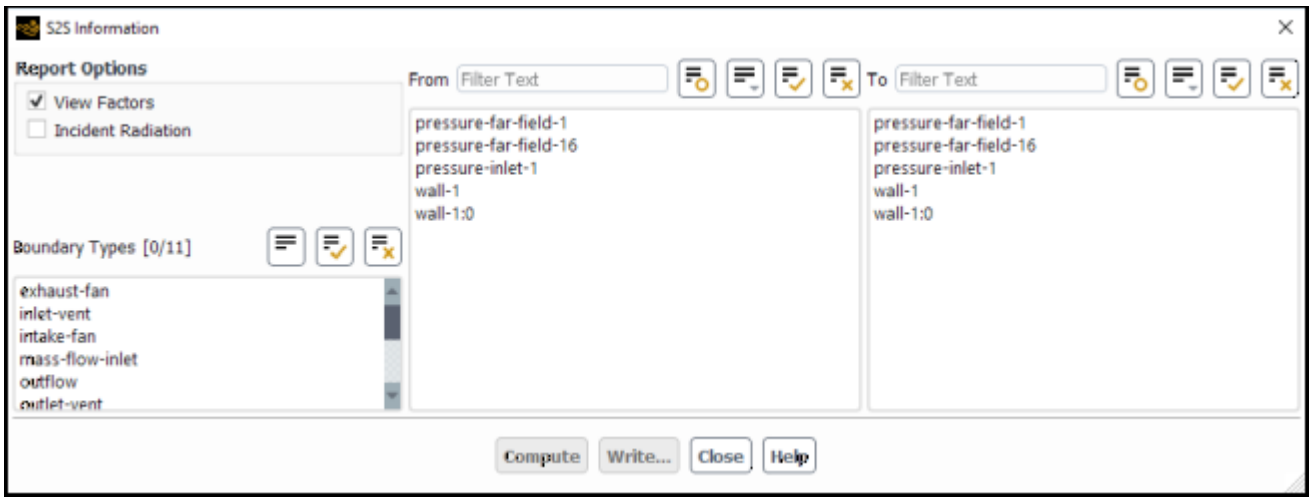
Rotate

adds the specified rotation parameters to the appropriate Cartesian coordinate of every node in the mesh.

52.2.105. S2S Information Dialog Box

The **S2S Information** dialog box allows you to report values of the view factor and radiation emitted from one zone to any other zone. See Reporting Radiation in the S2S Model (p. 2246) for details about the items below.

Results → Model Specific → S2S Information...



Controls

Report Options

contains items for which information is available for reporting.

View Factors

turns on the computation of view factors from one zone to the other.

Incident Radiation

turns on the computation of the incident radiation from one zone to the other.

Boundary Types

contains a selectable list of types of boundary zones. If you select (or deselect) an item in this list, all zones of that type will be selected (or deselected) automatically in the **From** and **To** lists.

From

contains a selectable list of boundary zones for which you would like data reported from the selected zone.

To

contains a selectable list of boundary zones for which you would like data reported to the selected zone.

Compute

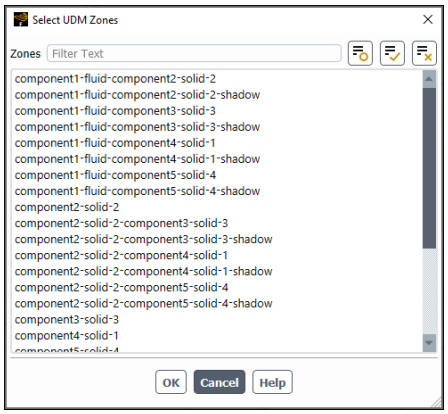
computes the view factors and/or incident radiation on the selected zones.

Write...

opens The Select File Dialog Box (p. 931), which you can use to save the data as an S2S Info File (`.sif` format).

52.2.106. Select UDM Zones Dialog Box

The **Select UDM Zones** dialog box allows you allocate user-defined memory on specific zones. This dialog box is opened by clicking **Edit...** in the User-Defined Memory Dialog Box (p. 5652).



Controls

Zones

lists the zones available for user-defined memory allocation.

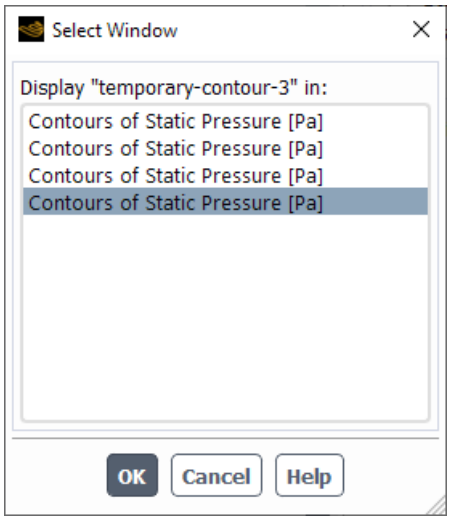
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

OK

confirms the selection of the highlighted zone(s) for user-defined memory allocation.

52.2.107. Select Window Dialog Box

The **Select Window** dialog box allows you specify the window where the specified object (accessed via right-clicking the object or boundary in the tree and selecting **Display in** → **More...** or **Add To** → **More...**) will be displayed. See Managing Multiple Graphics Windows (p. 3955) for additional information.



Controls

Display <object/boundary name> in: | Add <object/boundary name> to:

lists the available graphics windows for display. Selecting a graphics window from the list makes the window active.

OK

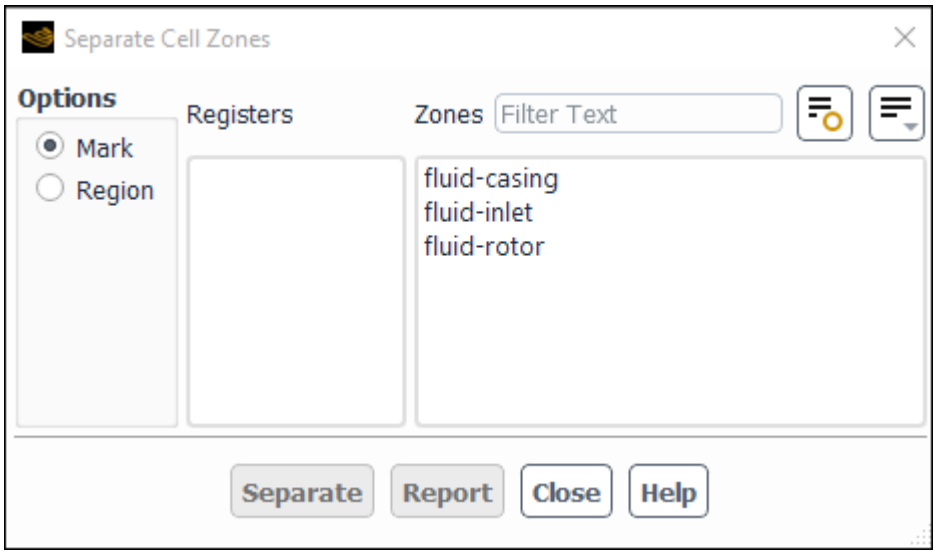
displays the specified object or boundary in the selected graphics window.

52.2.108. Separate Cell Zones Dialog Box

The **Separate Cell Zones** dialog box allows you to separate a single cell zone into multiple zones of the same type. See Separating Zones (p. 1262) for details.

Domain → **Zones** → **Separate** → **Cells...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Options

specifies the method on which the cell-zone separation is to be based.

Mark

indicates that the cell zone is to be separated based on the marks stored in cell registers.

Region

indicates that the cell zone is to be separated into two or more contiguous regions based on an internal boundary within the original zone.

Registers

contains a list of defined cell registers. If you are separating the cell zone by mark, select the cell register to be used in the **Registers** list. When the separation is performed, cells that are marked will be placed into a new zone.

Zones

contains a list of cell zones from which you can select the zone to be separated.

Separate

separates the selected cell zone based on the specified parameters.

Report

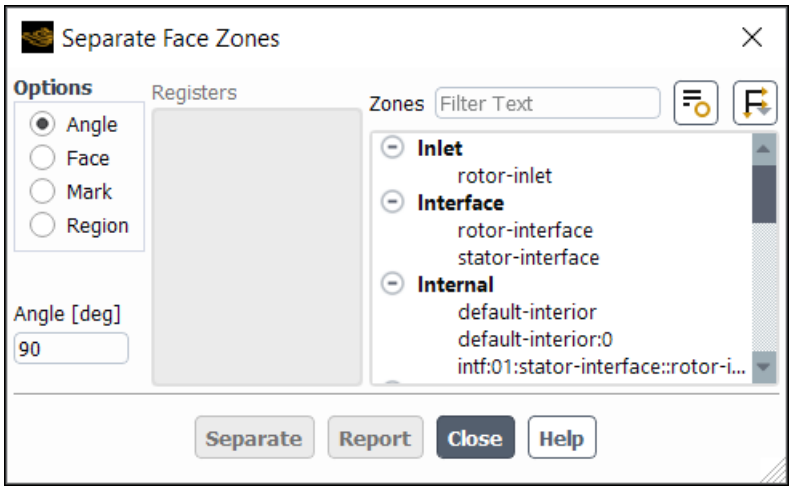
reports what the result of the separation will be without actually separating the cell zone.

52.2.109. Separate Face Zones Dialog Box

The **Separate Face Zones** dialog box allows you to separate a single face zone into multiple zones of the same type. See Separating Zones (p. 1262) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Domain → Zones → Separate → Faces...



Controls

Options

specifies the method on which the face zone separation is to be based.

Angle

indicates that the face zone is to be separated based on significant angle (specified in the **Angle** field).

Face

indicates that the face zone is to be separated by putting each face in the zone into its own zone.

Mark

indicates that the face zone is to be separated based on the marks stored in cell registers.

Region

indicates that the face zone is to be separated based on contiguous regions.

Registers

contains a list of defined cell registers. If you are separating faces by mark, select the cell register to be used in the **Registers** list. When the separation is performed, all faces of cells that are marked will be placed into a new face zone.

Zones

contains a list of face zones from which you can select the zone to be separated.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Angle

specifies the significant angle to be used when you separate a face zone based on angle. Faces with normal vectors that differ by an angle greater than or equal to the specified significant angle will be placed in different zones when the separation occurs.

Separate

separates the selected face zone based on the specified parameters.

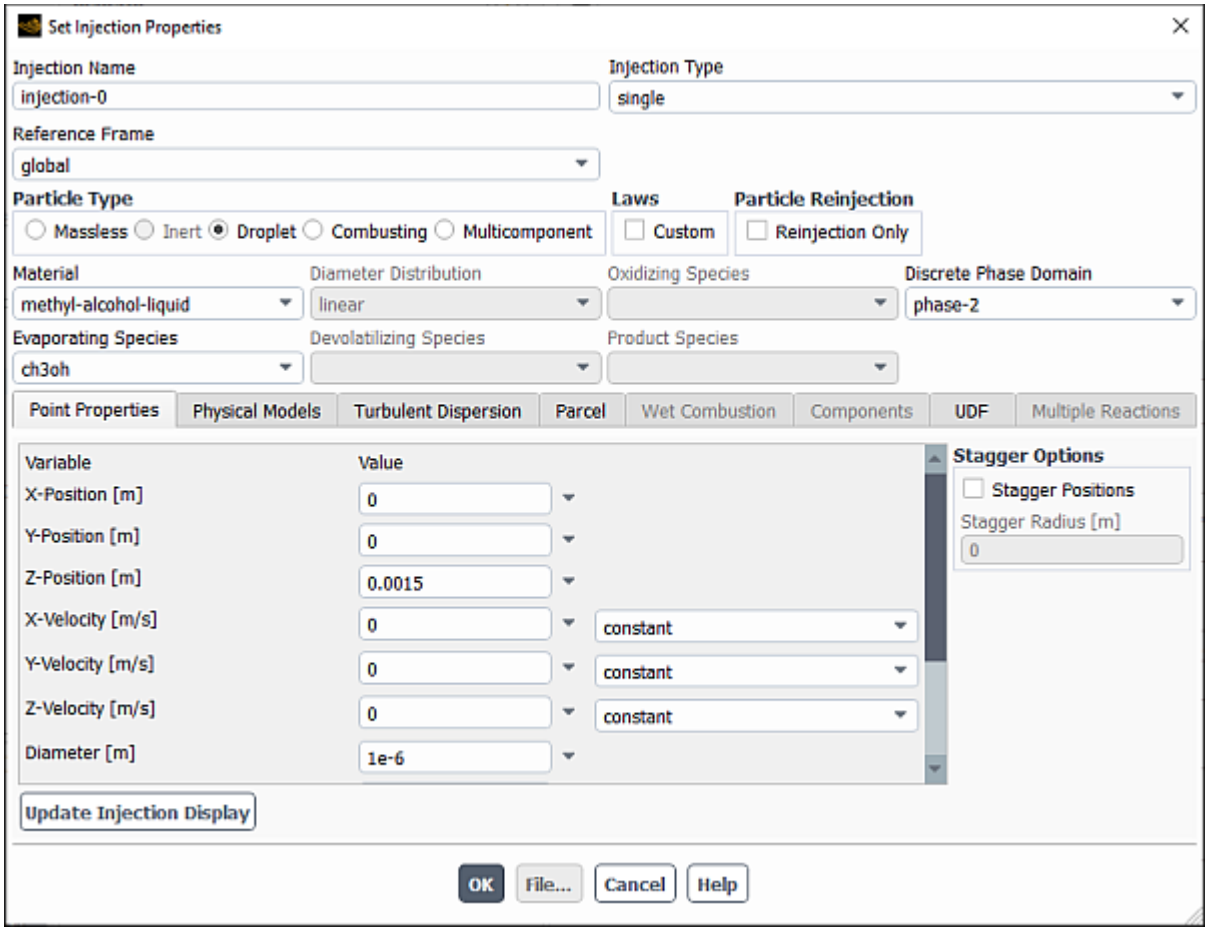
Report

reports what the result of the separation will be without actually separating the face zone.

52.2.110. Set Injection Properties Dialog Box

The **Set Injection Properties** dialog box allows you to define the properties of an existing discrete-phase injection (which was created in the Injections Dialog Box (p. 5496)). See Defining Injection Properties (p. 2756) for details about the items listed in this section.

The **Set Injection Properties** dialog box is opened from the Injections Dialog Box (p. 5496).



Controls

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Injection Name

sets the name of the injection.

Injection Type

contains a drop-down list of the available injection types: **single**, **group**, **cone**, **surface**, **volume**, **plain-orifice-atomizer**, **pressure-swirl-atomizer**, **air-blast-atomizer**, **flat-fan-atomizer**, **effer-vescent-atomizer**, and **file**. (**cone** is not available in 2D.) These choices are described in Injection Types (p. 2720).

Number of Streams

indicates the number of particle streams in a group, cone, or atomizer injections, or surface injections with the **Randomize Starting Points** option. (This item will not appear for single, volume, or file injections, or surface injections without the **Randomize Starting Points** option.)

Reference Frame

is a drop-down list where you can select the reference frame for the injection point properties. By default, the global reference frame is used. This item is not available for surface, volume, and condensate injections.

Injection Surfaces

is where you can select the surface(s) from which the particles in a **surface** injection will be released. This item will appear only for a **surface** injection.

Release From

is where you can select either the **zone** or **bounding-geometry** particle release region. This item will appear only for a **volume** injection.

Zones

is where you can select the zone(s) from which the particles in a **volume** injection will be released. This item will appear only when **zone** is selected from the **Release From** drop-down list.

Bounding Shape

is a drop-down list where you can select **sphere**, **cylinder**, **cone**, or **hexahedron** to model the particle release region. This item will appear only when **bounding-geometry** is selected from the **Release From** drop-down list.

Shape Coordinates

designates the coordinates to define the geometry of the selected bounding shape. See Defining Injection Properties (p. 2756) for more information. This item will appear only when **bounding-geometry** is selected from the **Release From** drop-down list.

Parcel Specification

specifies the number of particle parcels to be injected. You can choose either the **number-of-starting-points** or **parcels-per-cell** specification method. See Defining Injection Properties (p. 2756)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

for details. This item appears only for a **volume** injection and is not available for DDPM cases with the **Discrete Phase Domain** setting other than **none**.

Injection Packing Limit per Cell

specifies the highest particle volume fraction that is possible in a computational cell. See Defining Injection Properties (p. 2756) for details. This item appears only for a **volume** injection when the DDPM with the **Discrete Phase Domain** setting other than **none** is used.

Particle Type

specifies the particle type as **Massless**, **Inert**, **Droplet**, **Combusting**, or **Multicomponent**. These types are described in Particle Types (p. 2722).

Laws

(not for massless particles) contains inputs for customized particle laws.

Custom

enables the specification of customized particle laws and opens the Custom Laws Dialog Box (p. 5429).

Reinjection Only

(unsteady particle tracking only) specifies whether the injection is used only for reinjecting particles as described in The **reinject** Boundary Condition (p. 2784).

Material

(not for massless particles) indicates the material for the particles. If this is the first time you have created a particle of this type, you can choose from all of the materials by copying them from the database or creating them from scratch, as discussed in Setting Discrete-Phase Physical Properties (p. 2814) and described in detail in Using the **Create/Edit Materials** Dialog Box (p. 1599).

Diameter Distribution

(not for massless particles) allows you to select one of the following interpolation methods for determining the size of the particles:

- **linear** (default, **group** injections only)
- **uniform** (default, **surface** and **volume** injections only)
- **rosin-rammler**
- **rosin-rammler-logarithmic**
- **tabulated** (**cone**, **surface** or **volume** injections only)

  Selecting **tabulated** opens the **Association of Particle Size Distribution Data** dialog box.

See Using the Rosin-Rammler Diameter Distribution Method (p. 2745) and Using the Tabulated (Discrete) Diameter Distribution (p. 2748) for more information on the Rosin-Rammler method and the tabulated method, respectively.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Evaporating Species

(for **droplet** particles) specifies the gas-phase species created by the vaporization and boiling laws (laws 2 and 3).

Devolatilizing Species

(for **combusting** particles) specifies the gas-phase species created by the devolatilization law (law 4).

This item will not appear for two-mixture-fraction non-premixed combustion calculations.

Devolatilizing Stream

(for **combusting** particles) specifies the destination stream for the gas-phase species created by the devolatilization law (law 4).

This item will appear only for two-mixture-fraction non-premixed combustion calculations.

Oxidizing Species

(for **combusting** particles) specifies the gas phase species that participates in the surface char combustion reaction (law 5).

Product Species

(for **combusting** particles) specifies the gas-phase species created by the surface char combustion reaction (law 5).

This item will not appear for two-mixture-fraction non-premixed combustion calculations.

Product Stream

(for **combusting** particles) specifies the destination stream for the gas-phase species created by the surface char combustion reaction (law 5).

This item will appear only for two-mixture-fraction non-premixed combustion calculations.

Discrete Phase Domain

is available when using the **Dense Discrete Phase Model**, described in Including the Dense Discrete Phase Model (p. 3107).

DEM Collision Partner

is available when using the **DEM Collision** model, described in Modeling Collision Using the DEM Model (p. 2701).

Point Properties

displays the inputs for the point properties for the injection (for example, position, velocity, diameter, temperature, mass flow rate, and stagger options). These inputs are described for each injection type in Point Properties for Single Injections (p. 2724) – Point Properties for Effervescent Atomizer Injections (p. 2741).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Stagger Options

allows you to set the stagger options:

Stagger Positions

enables injection-specific spatial staggering of the particles.

Stagger Radius

specifies the region from which particles are released. This item is available only for non-atomizer injections.

Stagger in Injection Plane Only

when enabled, restricts the spatial staggering of the initial particle positions to purely the radial direction of the spray. The spatial staggering clusters the initial particle positions about the outer radius of the hollow cone using a Gaussian distribution with 0 mean and +/- 2 standard deviation. This item is available only for **hollow-cone** injections.

Surface Options

(surface injections only) allows you to select the following options:

Scale Flow Rate by Face Area

scales the individual mass flow rates of the particles by the ratio of the area of the face they are released from to the total area of the surface.

Inject Using Face Normal Direction

uses the surface normal for the injection direction. When this option is selected, you need to specify only the velocity magnitude of the injection, not the individual components of the velocity magnitude.

Randomize Starting Points

when this option is selected, particles are injected from positions distributed randomly over the selected boundary surfaces.

Wallfilm

(surface injections only) is where you can select the **Inject Wallfilm Particles** option if you want to inject wall film particles from surfaces. This item is available for unsteady particle tracking only.

First Point

specifies the first point properties for the injection. This item appears only for group injections.

Last Point

specifies the last point properties for the injection. This item appears only for group injections.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Update Injection Display

applies the current point property settings to the injection graphics windows so that you can preview the effect of modified settings before saving the injection. This item is not available for **surface**, **volume**, **file**, and **condensate** injection types.

Use Mass Instead of Flow Rate

When selected, you can specify the **Total Mass** instead of the default **Total Flow Rate** of the parcel to be injected into the particle release region. This item is available only for the **volume** injection type.

Physical Models

displays the inputs for injection-specific physics models.

Drag Parameters

allows the setting of the drag law used in calculating the force balance on the particles. See Particle Force Balance in the Theory Guide for details on the items below.

Drag Law

is a drop-down list containing the available choices for the drag law:

spherical

assumes that the particles are smooth spheres.

nonspherical

assumes that the particles are not spheres, but are all identically shaped. The shape is specified by the **Shape Factor**.

Stokes-Cunningham

is for use with sub-micron particles. A **Cunningham Correction** is added to Stokes' drag law to determine the drag.

high-Mach-number

is similar to the spherical law with corrections to account for a particle Mach number greater than 0.4 or a particle Reynolds number greater than 20.

Ishii-Zuber

is used in gas-liquid flows. This drag law is only available if the magnitude and direction of the gravity vector are specified in the **Operating Conditions** dialog box. See Ishii-Zuber Drag Model in the *Fluent Theory Guide* for details.

Grace

is used in gas-liquid flows. This drag law is only available if the magnitude and direction of the gravity vector are specified in the **Operating Conditions** dialog box. See Grace Drag Model in the *Fluent Theory Guide* for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

dynamic-drag

accounts for the effects of droplet distortion. This drag law is available only when one of the droplet breakup models is used in conjunction with unsteady tracking. See Dynamic Drag Model Theory in the Theory Guide for details.

Shape Factor

specifies the shape of the particles when **nonspherical** is selected as the **Drag Law** (φ in Equation 12.52 in the Theory Guide). It is the ratio of the surface area of a sphere having the same volume as the particle to the actual surface area of the particle. The shape factor value cannot be greater than 1.

Cunningham Correction

(C_c in Equation 12.55 in the Theory Guide) is used with Stokes' drag law to determine the force acting on the particles when the particles are sub-micron size. It appears when **Stokes-Cunningham** is selected as the **Drag Law**.

Wen-Yu

uses the fluid-solid drag function of Wen and Yu described in Fluid-Solid Exchange Coefficient in the *Fluent Theory Guide*. See Specifying the Drag Function (p. 3075) for recommendations on the usage of this option. This drag law is available only when the dense discrete phase model (DDPM) is enabled.

Gidaspow

uses the Gidaspow formulation described in Fluid-Solid Exchange Coefficient in the *Fluent Theory Guide*. See Specifying the Drag Function (p. 3075) for recommendations on the usage of this drag law. This drag law is available only when the dense discrete phase model (DDPM) is enabled.

Huilin-Gidaspow

uses the Huilin-Gidaspow function described in Fluid-Solid Exchange Coefficient in the *Fluent Theory Guide*. See Specifying the Drag Function (p. 3075) for recommendations on the usage of this drag law. This drag law is available only when the dense discrete phase model (DDPM) is enabled.

Syamlal-OBrien

uses the Syamlal-O'Brien model described in Fluid-Solid Exchange Coefficient in the *Fluent Theory Guide*. See Specifying the Drag Function (p. 3075) for recommendations on the usage of this drag law. This drag law is available only when the dense discrete phase model (DDPM) is enabled.

Gibilaro

uses the Gibilaro model described in Fluid-Solid Exchange Coefficient in the *Fluent Theory Guide*. See Specifying the Drag Function (p. 3075) for recommendations on the usage of this drag law. This drag law is available only when the dense discrete phase model (DDPM) is enabled.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

EMMS

uses the energy-minimization multi-scale (EMMS) drag model described in Fluid-Solid Exchange Coefficient in the *Fluent Theory Guide*. See Specifying the Drag Function (p. 3075) for recommendations on the usage of this drag law. This drag law is available only when the dense discrete phase model (DDPM) is enabled.

Filtered

uses the filtered Two-Fluid Model (TFM) described in The Filtered Two-Fluid Model in the *Fluent Theory Guide*. See Specifying the Drag Function (p. 3075) for recommendations on the usage of this drag law. This drag law is available only when the dense discrete phase model (DDPM) is enabled.

Heat Transfer Coefficient

is a drop-down list from which you can select the heat transfer model for your simulation. See Heat Transfer Coefficient (p. 2766) for a full description of available models and their usage.

Particle Rotation

contains settings for the modeling of particle rotation when the **Enable Rotation** check box is selected (see Wall-Particle Reflection Model Theory in the *Fluent Theory Guide*).

Rotational Drag Law

is a drop-down list of available rotational drag law models:

Dennis-et-al

uses Equation 12.67 in the *Fluent Theory Guide* to calculate the rotational drag coefficient C_ω.

none

implies zero torque, that is, it excludes the influence of the fluid on the rotational particle motion.

Magnus Lift Law

is a drop-down list of available Magnus lift law models:

Oesterle-Bui-Dinh

uses Oesterle and Bui Dinh's formulation (Equation 12.18 in the *Fluent Theory Guide*).

Tsuji-et-al

is based on the particle spin rate (Equation 12.19 in the *Fluent Theory Guide*).

Rubinow-Keller

uses a linear dependence of the rotational lift coefficient C_{RL} on the spin rate (Equation 12.21 in the *Fluent Theory Guide*).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

none

implies that the Magnus lift will not be included in your simulation.

Brownian Motion

enables the incorporation of the effects of Brownian motion. This option is available only when the **Energy** equation is enabled (in the **Models** group) and the **Stokes-Cunningham** drag law is selected from the **Drag Law** drop-down list. See Brownian Force in the Theory Guide for details.

Breakup

contains parameters that control droplet breakup and collision. (This section of the dialog box appears only if **Unsteady Tracking** is enabled in the Discrete Phase Model Dialog Box (p. 4845).) See Breakup (p. 2769) for more information.

Enable Breakup

enables breakup for this injection.

Breakup Model

is a drop-down list containing parameters that control droplet breakup. (This item appears only if **Enable Breakup** is selected.)

TAB

enables the Taylor Analogy Breakup (TAB) model, which is applicable to many engineering sprays. This method is based upon Taylor's analogy between an oscillating and distorting droplet and a spring mass system. See Taylor Analogy Breakup (TAB) Model in the Theory Guide for details.

For the **TAB** model, you must specify the following constants:

y0

is the constant y_0 in Equation 12.408 in the Theory Guide.

Breakup Parcels

is the number of child parcels the droplet is split into, as described in Velocity of Child Droplets in the Theory Guide.

Wave

enables the Wave breakup model, which considers the breakup of the injected liquid to be induced by the relative velocity between the gas and liquid phases. See Wave Breakup Model for details.

For the **Wave** model, you must specify the following constants:

B0

is the constant B_0 in Equation 12.435 in the Theory Guide.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

B1

is the constant B_1 in Equation 12.437 in the Theory Guide.

KHRT

enables the Kelvin-Helmholtz Rayleigh-Taylor breakup model, which considers the two competing effects of aerodynamic breakup and instabilities due to droplet acceleration. See KHRT Breakup Model in the Theory Guide for details.

For the **KHRT** model, you must specify the following constants:

B0

is the constant B_0 in Equation 12.435 in the Theory Guide.

B1

is the constant B_1 in Equation 12.437 in the Theory Guide.

Ctau

is the constant C_τ in Equation 12.442 in the Theory Guide.

CRT

is the constant C_{RT} in Equation 12.443 in the Theory Guide.

CL

is the constant C_L in Equation 12.438 in the Theory Guide.

SSD

enables the stochastic secondary droplet breakup model, where the probability of breakup is independent of the parent droplet size and the secondary droplet size is sampled from an analytical solution of the Fokker-Planck equation for the probability distribution. See Stochastic Secondary Droplet (SSD) Model in the Theory Guide for details.

For the **SSD** model, you must specify the following constants:

Critical We

is the critical Weber number in Equation 12.444 in the Theory Guide.

Core B1

is B in Equation 12.445 in the Theory Guide.

Target Np

is the average number in parcels for daughter parcels.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Xi

is $\langle \xi \rangle$ in Equation 12.446 in the Theory Guide.

Madabhushi

Enables the Madabhushi breakup model. This model is available only with single, group, or cone (solid cone) injections. Solid cone injection is recommended. For more information about the Madabhushi breakup model, see Madabhushi Breakup Model in the *Fluent Theory Guide*.

For the **Madabhushi** model, you must specify the following constants:

B0

is the constant B_0 in Equation 12.435 in the Theory Guide. The recommended value is 2.44.

B1

is the constant B_1 in Equation 12.437 in the Theory Guide. The recommended value is 10.

Jet Diameter

is the nozzle orifice exit diameter.

Ligament Factor

Adjusts the diameter of the child droplets arising from the column breakup mechanism

C0

is the column breakup time constant. The value recommended for the Madabhushi breakup model is 3.44.

Column Drag Cd

is the constant drag coefficient for calculating the drag force acting on droplets within the liquid column.

Breakup Max Generation

is the maximum number of child parcel generations.

Subsequent Statistical Breakup

when enabled, the droplet breakup mechanism switches to the subsequent statistical breakup model after the **Breakup Max Generation** is reached. No child parcels are created during the breakup mechanism. Instead, the particle number in the parcel is increased, and the particle diameter is adjusted in such a way as to conserve the full filling mass of the parent parcel.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Schmehl

Enables the Schmehl breakup model. For more information about the Schmehl breakup model, see Schmehl Breakup Model in the *Fluent Theory Guide*.

For this model, you can specify the following settings:

Breakup Parcels

is the number of child parcels generated after splitting the droplet as described in Schmehl Breakup Model in the *Fluent Theory Guide*.

Breakup Max Generation

is the maximum number of child parcel generations.

Subsequent Statistical Breakup

when enabled, the droplet breakup mechanism switches to the subsequent statistical breakup model after the **Breakup Max Generation** is reached.

Turbulent Dispersion

displays the inputs for stochastic tracking.

Stochastic Tracking

controls the stochastic tracking for turbulent flows. Stochastic tracking includes the effect of turbulent velocity fluctuations on the particle trajectories using the DRW model described in Stochastic Tracking in the Theory Guide. See Stochastic Tracking (p. 2772) for details about the items below.

Discrete Random Walk Model

includes the effect of instantaneous turbulent velocity formulations on the particle trajectories through stochastic method.

Random Eddy Lifetime

specifies that the characteristic lifetime of the eddy is to be random.

Number of Tries

controls the inclusion of turbulent velocity fluctuations.

An input of 1 or greater tells Ansys Fluent to include turbulent velocity fluctuations in the particle force balance.

Time Scale Constant

is C_L in Equation 12.24 in the Theory Guide. The default is 0.15; if you use the RSM, a value of 0.3 is recommended.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parcel

displays the inputs for controlling the discrete phase parcels. This tab is not available for volume injections when the DDPM with the **Discrete Phase Domain** option is used.

Parcel Definitions

contains settings for how parcels are defined

Parcel Release Method

specifies the method for releasing parcels. Available settings are **standard**, **constant-number**, **constant-mass**, and **constant-diameter**. See Steady/Transient Treatment of Particles (p. 2679) for details about these settings.

Wet Combustion

displays the inputs for the wet combustion model.

Wet Combustion Model

allows the combusting particles to include an evaporating/boiling material.

Liquid Material

contains a drop-down list of liquid materials that can be chosen as the evaporating/boiling material to be included with the combusting particles.

Liquid Fraction

sets the volume fraction of the liquid present in the particle.

Components

contains the inputs for the **Multicomponent** particle type. For details on the following items, see Defining Injection Properties (p. 2756). This item is available only when **Multicomponent** is selected as **Particle Type**.

Multicomponent Settings

contains the multicomponent injections.

Component

specifies the component that is a part of the multicomponent species.

Mass Fraction

specifies the mass fraction of the component in a multicomponent species.

Evaporating Species

specifies the gas-phase species to be evaporated.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Evaporating Stream

specifies the source stream from which the species will be evaporated. This item appears only when the non/partially-premixed model is used in the case setup.

UDF

displays the inputs for **User-Defined Functions** for use in the definition of the particle injection. For details about user-defined functions, see the separate Fluent Customization Manual.

Initialization

contains a drop-down list of available user-defined functions. The UDF that you choose will be used to modify the injection properties at the time the particles are injected into the domain.

Heat/Mass Transfer

allows you to select the user-defined function that defines the heat or mass transfer.

Multiple Reactions

displays the inputs for **Multiple Surface Reactions**. See Combusting Particle Surface Reactions (p. 2406) for details about this model.

Species Mass Fractions

specify the combustible fraction of the combusting particle if you have defined more than one particle surface species. See Using the Multiple Surface Reactions Model for Discrete-Phase Particle Combustion (p. 2408) for details.

File...

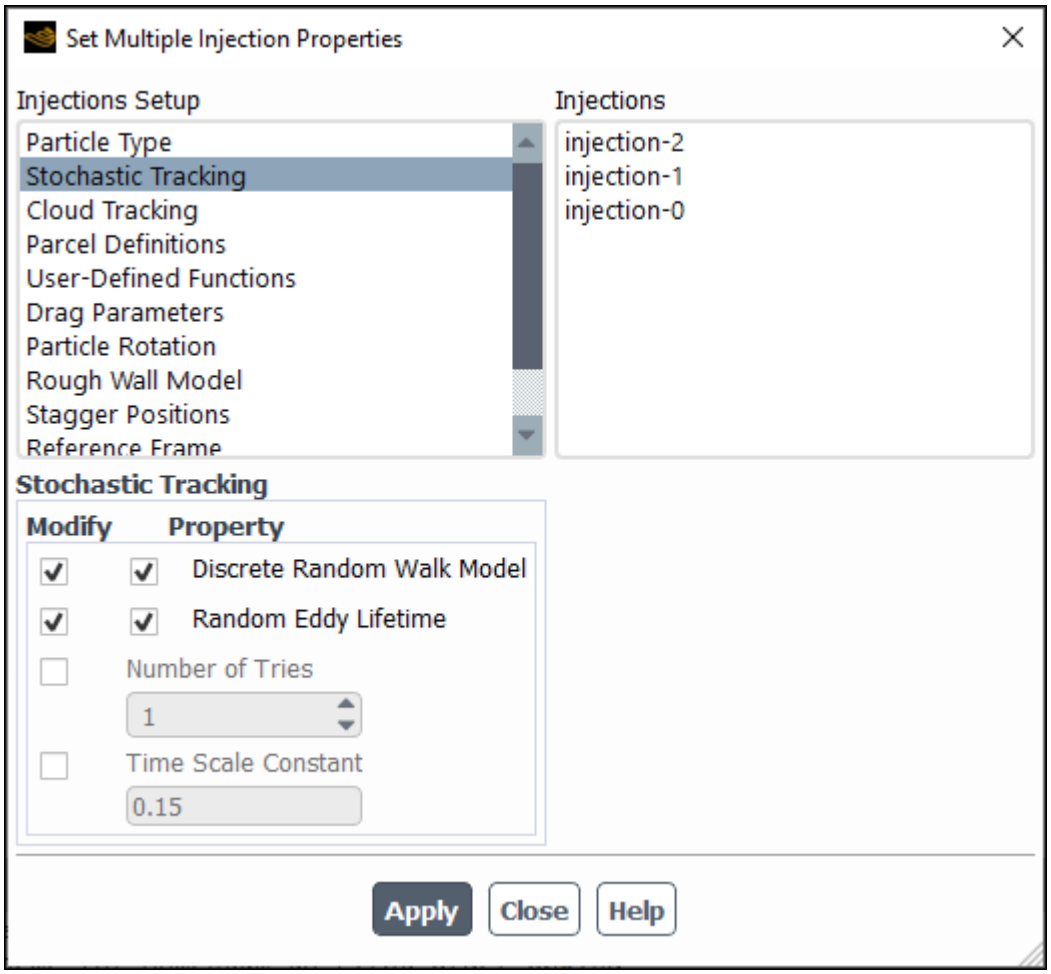
opens The Select File Dialog Box (p. 931), in which you can select a file containing the injection definition (when **file** is selected as the **Injection Type**).

52.2.111. Set Multiple Injection Properties Dialog Box

The **Set Multiple Injection Properties** dialog box allows you to set properties that are common to multiple injections. See Defining Properties Common to More than One Injection (p. 2775) for details about the items below.

The **Set Multiple Injection Properties** dialog box is opened from the Injections Dialog Box (p. 5496).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Injections Setup

contains a list of the categories of injection properties that you can set for the injections in the **Injections** list. These categories correspond to the categories of inputs in the Set Injection Properties Dialog Box (p. 5608). When you select an item in the **Injections Setup** list, the dialog box will expand to show the relevant inputs, which are the same as those in the **Set Injection Properties** dialog box.

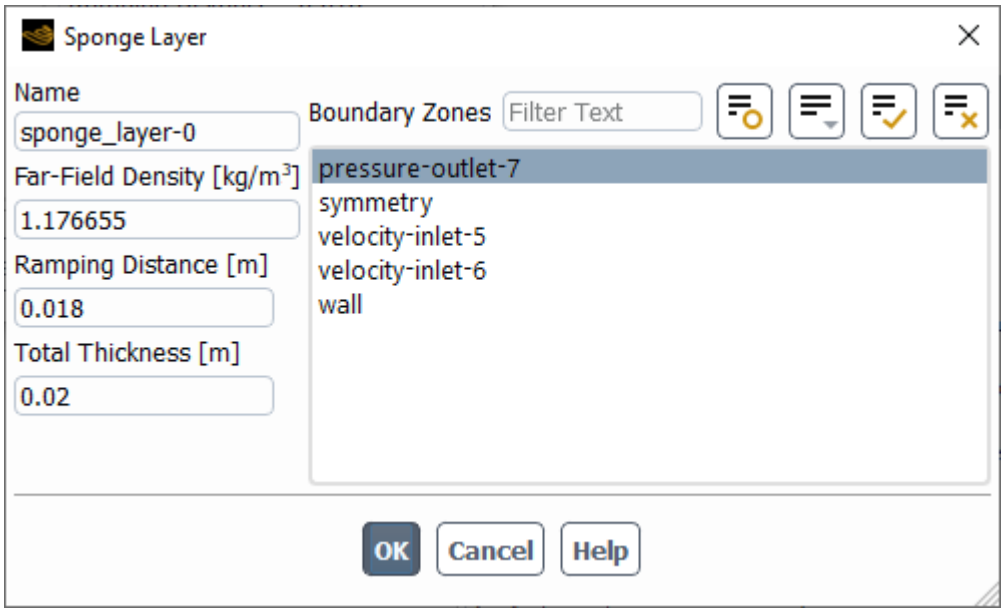
Injections

displays an informational list of the injections for which you are setting common properties. These are the injections that you selected in the Injections Dialog Box (p. 5496).

52.2.112. Sponge Layer Dialog Box

The **Sponge Layer** dialog box creates or edits a sponge layer definition, which can be used to eliminate non-physical pressure wave reflections from boundary zones in transient simulations of compressible flows with the pressure-based solver, by blending the density from the value calculated by the solver to a user-specified far-field value in the domain near one or more specified boundary zones. It is opened by clicking **New...** or by making a selection from the **Sponge Layers** list and clicking **Edit...** in the Manage Sponge Layers Dialog Box (p. 5519). For further details, see Sponge Layers (p. 2667).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name

specifies the name of the sponge layer definition.

Far-Field Density

specifies a value of density that is used for blending with the solver value within the sponge layer. It is recommended that you use the average density at the boundaries throughout the simulation.

Ramping Distance

specifies a distance within the sponge layer over which the density changes from the solver value to the far-field value. It is recommended that this distance is 80–100% of the **Total Thickness**.

Total Thickness

specifies the total thickness of the sponge layer, a distance from the selected boundaries to the interior boundary of the sponge layer. It is recommended that this thickness is at least twice the wavelength of the expected pressure waves.

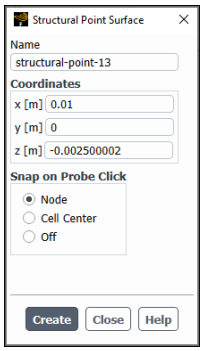
Boundary Zones

contains a list of available boundary zones on which the sponge layer can be applied.

52.2.113. Structural Point Surface Dialog Box

The **Structural Point Surface** dialog box allows you to interactively create structural point surfaces (surfaces containing a single data point). See Structural Point Surfaces (p. 3825) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name

designates the name of the new surface. The default is the concatenation of the surface type and an integer which is the new surface ID.

Reference Frame

allows you to specify a local reference frame (if defined) for the point surface.

Coordinates

designates the coordinates of the point in the surface (**x0**, **y0**, **z0**).

Snap on Probe Click

updates the coordinates of the point to the closest **Node** or **Cell Center** when you click on the model in the graphics window using the probe tool—*that is, either by right-clicking or setting the left-mouse-button to probe (* *)*.

You can disable automatic snapping by selecting **Off**.

Create

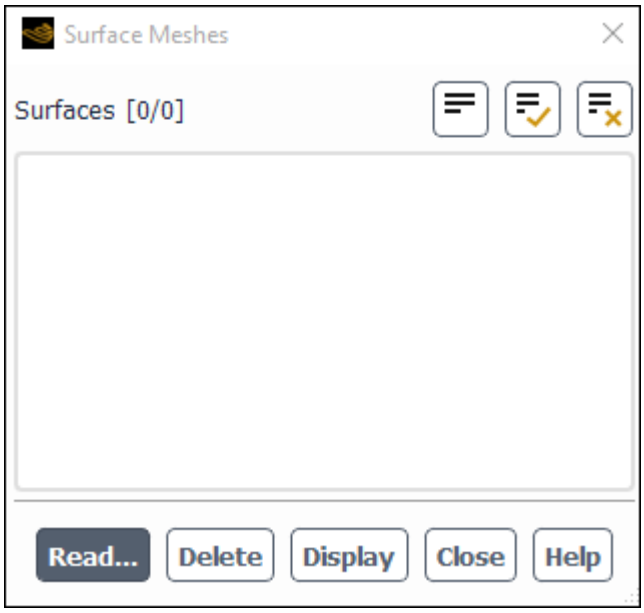
creates the surface.

52.2.114. Surface Meshes Dialog Box

The **Surface Meshes** dialog box allows you to read the surface meshes in Ansys Fluent (See Reading Surface Mesh Files (p. 1159) for details on reading surface meshes.)

The **Surface Meshes** dialog box is opened from the Geometry Based Adaption Dialog Box (p. 5491).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Surfaces

contains a list of the surfaces available in the surface mesh you read.

*You can select/deselect the surfaces listed under **Surfaces**.*

Read...

opens the **Select File** dialog box in which you can select the surface mesh you want to read.

Delete

allows you to delete the selected surfaces under **Surfaces** area.

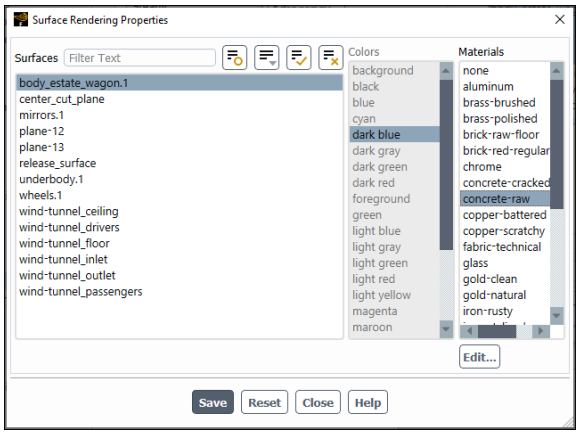
Display

allows you to display the selected surfaces under **Surfaces**.

52.2.115. Surface Rendering Properties Dialog Box

The **Surface Rendering Properties** dialog box is opened by clicking **Edit...** when **By Surface** is selected for **Coloring** in the Mesh Display Dialog Box (p. 4688).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Surfaces

contains a list of the surfaces where you can specify a color and/or material for rendering purposes.

Colors

contains a list of the colors that you can apply to the surfaces selected in the **Surfaces** list.

Materials

lists the available materials that may be used to color the surfaces selected in the **Surfaces** list. Click **Edit** if you want to modify a material's color or other properties. *See Creating Custom Realistic Materials (p. 3889) for additional information.*

Edit

opens the Material Editor Dialog Box (p. 5521).

Save

Saves all specified surface and color/material assignments.

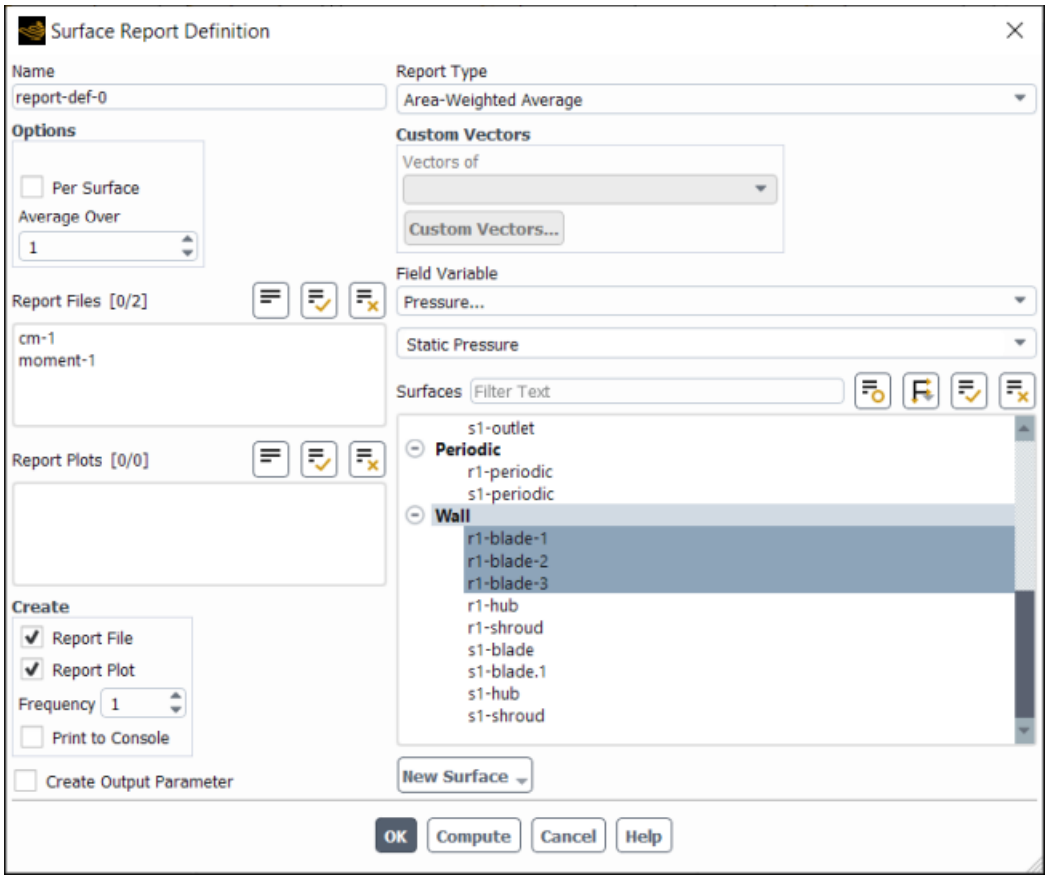
Reset

Resets all color and/or material assignments back to the default colors.

52.2.116. Surface Report Definition Dialog Box

Solution → Reports → Definitions → New → Surface Report

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name

specifies the name of the report definition.

Options

Per Surface

specifies whether or not the chosen field variable is calculated from all of the selected surfaces combined (default) or individually on each of the selected surfaces.

Average Over

(optional) To have Fluent calculate a running average for the **Surface Report Definition** you can enter a positive integer greater than 1 (the default) for **Average Over**.

Specifying a number greater than 1 means that Fluent will print, plot, and write the running average value of the selected variable instead of the current value of the same variable.

The value reported is averaged over the last N *iterations/time steps, where* N *is your specified* ***Average Over*** *value. When the iteration/time step number is lower than* N*, Fluent calculates the average of the available variable values.*

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Retain Instantaneous Values

appears when **Average Over** is set to a value greater than `1`. If enabled when you are creating a new report definition, it ensures that all instantaneous values are saved in the new report file that is being created for this report definition. It also results in the **Plot Instantaneous Values** option being enabled in the newly created report plot.

Report Files

lists all of the report files where you can write the report definition data.

Report Plots

lists all of the report plots where you can plot the report definition data (assuming this report has the same units as the other report definitions included in the selected report plot).

Create

Report File

creates a new report file that includes this report definition.

Report Plot

creates a new report plot that includes this report definition.

Frequency

specifies how frequently the report definition is written, plotted, or printed to the console.

Print to Console

prints the value of the report definition to the console.

Create Output Parameter

creates an output parameter for this report definition with the name `<report definition name>-op`.

Report Type

selects the integration method used on the selected surfaces. The available report types are the same as those in the Surface Integrals Dialog Box (p. 5325). See Surface Integration (p. 4150) for details.

Custom Vectors

allows you to specify a vector if you selected **Custom Vector Based Flux**, **Custom Vector Flux** or **Custom Vector Weighted Average** from the **Report Type** drop-down list.

Field Variable

contains a list of solution variables that can be monitored on the selected surfaces. This list will not be available if you select **Mass Flow Rate** or **Volume Flow Rate** as the **Report Type**.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Phase

contains a list of all of the phases in the problem that you have defined. This is available when the VOF, mixture, or Eulerian multiphase model is enabled.

Surfaces

contains a selectable list of the current surfaces.

New Surface

is a drop-down list button that contains a list of surface options:

Point

opens the Point Surface Dialog Box (p. 5582).

Line/Rake

opens the Line/Rake Surface Dialog Box (p. 5514).

Plane

opens the Plane Surface Dialog Box (p. 5580).

Quadric

opens the Quadric Surface Dialog Box (p. 5584).

Iso-Surface

opens the Iso-Surface Dialog Box (p. 5501).

Iso-Clip

opens the Iso-Clip Dialog Box (p. 5500).

Structural Point

opens the Structural Point Surface Dialog Box (p. 5623).

OK

creates the report definition.

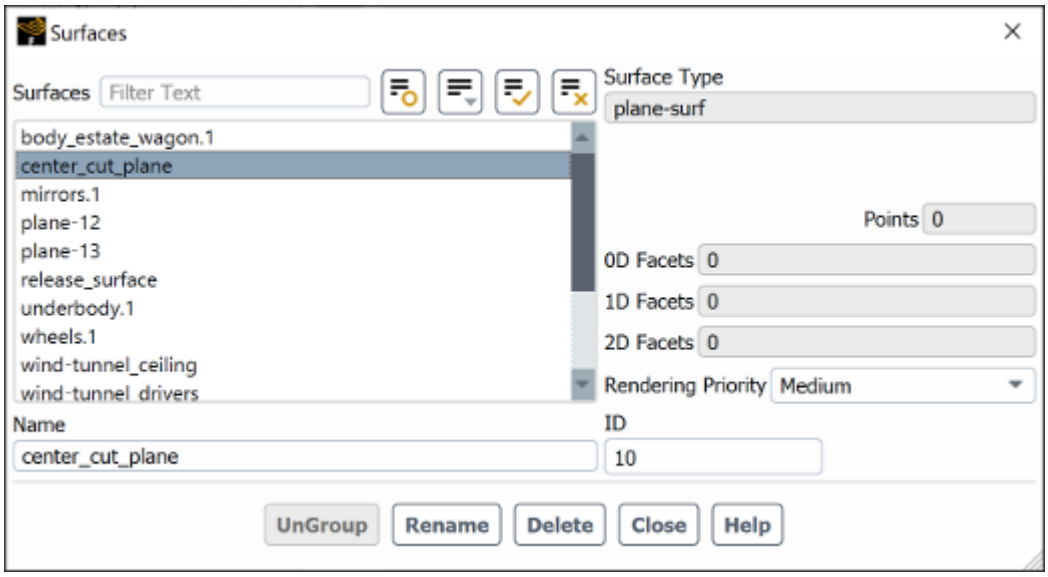
Compute

computes the value of the report definition at the selected surface(s) and prints to the console.

52.2.117. Surfaces Dialog Box

The **Surfaces** dialog box allows you to interactively group, rename, and delete surfaces and obtain information about their components. See Grouping, Prioritizing, Editing, Renaming, and Deleting Surfaces (p. 3847) for details about the items below.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Surfaces

contains a list of existing surfaces from which you can select the surface(s) of interest.

Name

displays the name of the selected surface. You can edit the text field to modify the surface name. (If more than one surface is selected, the name of the first one you selected will be displayed.)

Surface Type

displays the type of surface that is selected (for example, `zone-surf` if one surface is selected, or `Multiple Surfaces` if more than one surface is selected).

Points, 0D Facets, 1D Facets,

and **2D Facets** display the number of points and facets in the selected surface. If more than one surface is selected, the sum over all selected surfaces is displayed for each quantity.

Note that if you want to check these statistics for a surface that was read from a case file, you will need to first display it.

Rendering Priority

specifies the rendering priority for the selected surface, which determines which surface will be seen when there are coincident surfaces within a display.

ID

displays the ID of the selected surface. You cannot change this value.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

UnGroup

ungroups the selected surface. This button is available only if the selected surface was created by **Group**ing two or more surfaces together.

Rename

renames the selected surface in **Surfaces** with the name specified in **Name**. This button is available when just one surface is selected. (If two or more surfaces are selected, it becomes the **Group** button.)

Group

groups two or more selected surfaces and gives the group the name entered in **Name**. This button replaces the **Rename** button when two or more surfaces are selected.

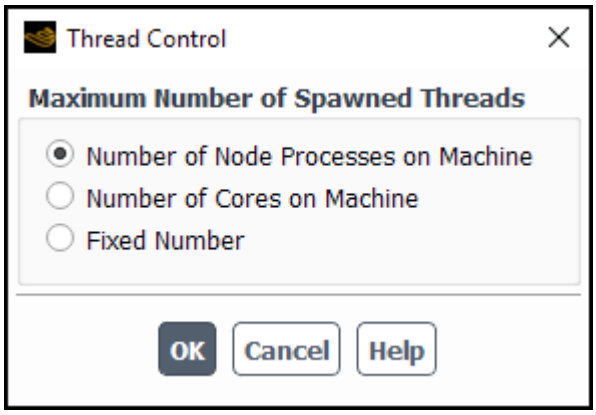
Delete

deletes the selected surface(s).

52.2.118. Thread Control Dialog Box

The **Thread Control** dialog box allows you to control the maximum number of threads on each machine, as described in Controlling the Threads (p. 4345).

Parallel → General → Thread Control...



Controls

Maximum Number of Spawned Threads

contains a list of options for defining the maximum number of threads on each machine.

Number of Node Processes on Machine

specifies that the maximum number of threads on each machine is equal to the number of Ansys Fluent node processes on each machine.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Number of Cores on Machine

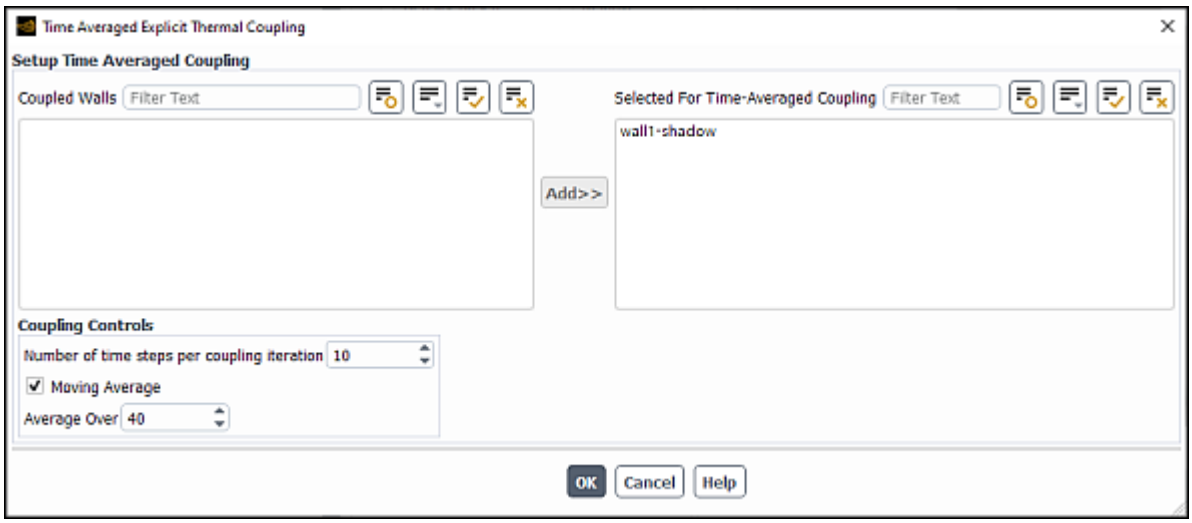
specifies that the maximum number of threads on each machine is equal to the number of cores on the machine. Ansys Fluent obtains the number of cores from the OS.

Fixed Number

specifies that the maximum number of threads that can be spawned on each machine is equal to the number you provide in the number-entry box below **Fixed Number**.

52.2.119. Time Averaged Explicit Thermal Coupling Dialog Box

The **Time Averaged Explicit Thermal Coupling** dialog box specifies transient conjugate heat transfer. Each coupling cycle is specified for a number of time steps at which the exchange of thermal boundary condition quantities occurs. See Time Averaged Explicit Thermal Coupling (p. 2177) for details about the items below.



Controls

Coupled Walls

list of coupled walls available for CHT.

Selected for Time-Averaged Coupling

list of walls selected for CHT.

Coupling Controls

contains options for the thermal coupling.

Number of time steps per coupling iteration

specifies the number of time steps between coupling iterations.

Moving Average

enables a moving average for the explicit coupling.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

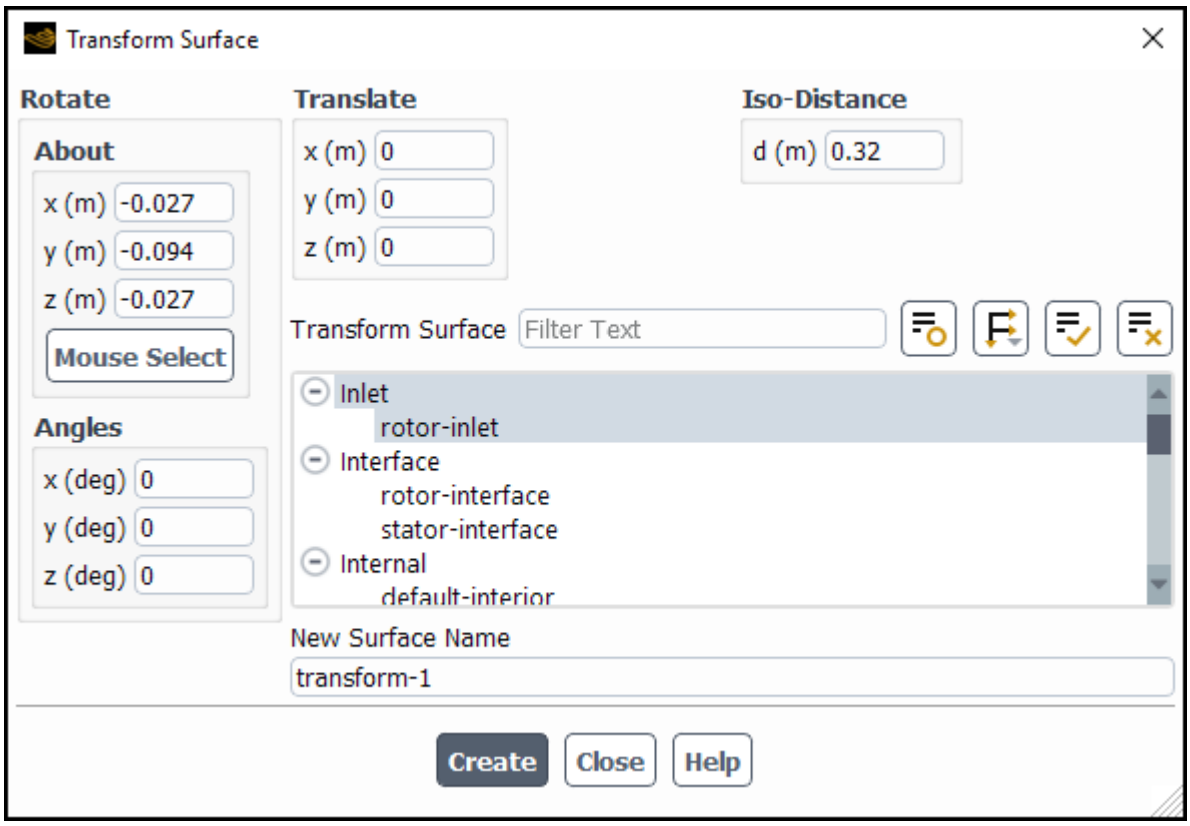
Average Over

specifies the number of times steps over which the data is averaged for explicit coupling.

52.2.120. Transform Surface Dialog Box

The **Transform Surface** dialog box allows you to create a new data surface by rotating and/or translating an existing surface, and/or by specifying a constant normal distance from it. See Transforming Surfaces (p. 3843) for details about the items below.

Domain → Surface → Create → Transform...



Controls

Rotate

contains the transformation parameters for rotation.

About

defines the origin about which the surface is rotated. You will specify a point, and the origin of the coordinate system for the rotation will be set to the specified point. For example, if you specified the point (1,0) in 2D, rotation would be about the z axis anchored at (1,0). You can either enter the point's coordinates in the **x,y,z** fields or click the **Mouse Select** button and select a point in the graphics window using the mouse.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Angles

define the angles about the **x**, **y**, and **z** axes (that is, the axes of the coordinate system with the origin defined under **About**) by which the surface is rotated. For 2D problems, you can specify rotation about the **z** axis only.

Translate

contains the transformation parameters for translation.

x,y,z

define the distance by which the surface is translated in each direction.

Iso-Distance

contains the transformation parameters for "isodistancing."

d

sets the normal distance between the original surface and the transformed surface.

Transform Surface

contains a list of existing surfaces from which you can select the surface to be transformed. The selected surface will remain unchanged; the transformation will create a new surface.

New Surface Name

designates the name of the new surface. The default is the concatenation of the transformation type (that is, iso-distance, rotate, or translate) and an integer which is the new surface ID.

Create

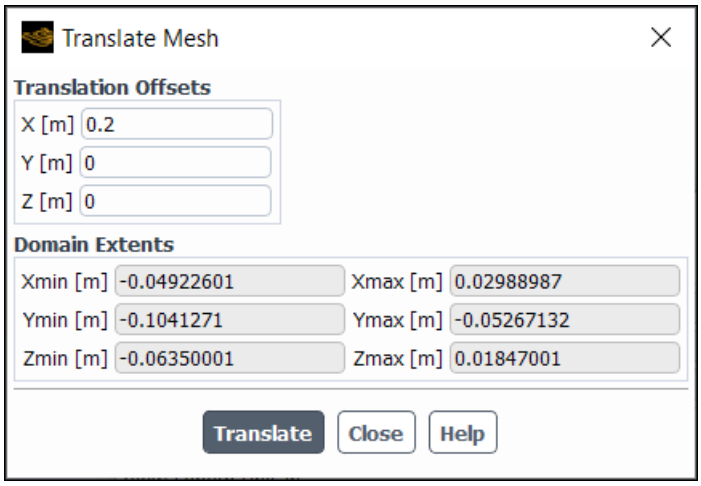
creates the surface.

52.2.121. Translate Mesh Dialog Box

The **Translate Mesh** dialog box allows you to change the origin of the mesh. See Translating the Mesh (p. 1288) for details.

Domain → Mesh → Transform → Translate...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Translation Offsets

contains the desired changes in the mesh coordinates (that is, the desired delta in the axes origin). You can enter a positive or negative real number.

X,Y,Z

defines the deltas in the x, y, and z directions, in the current units of length.

Domain Extents

displays the Cartesian coordinate extremes of the nodes in the mesh. (These values are not editable; they are purely informational.)

Xmin, Ymin, Zmin

shows the minimum values of Cartesian coordinates in the mesh.

Xmax, Ymax, Zmax

shows the maximum values of Cartesian coordinates in the mesh.

Translate

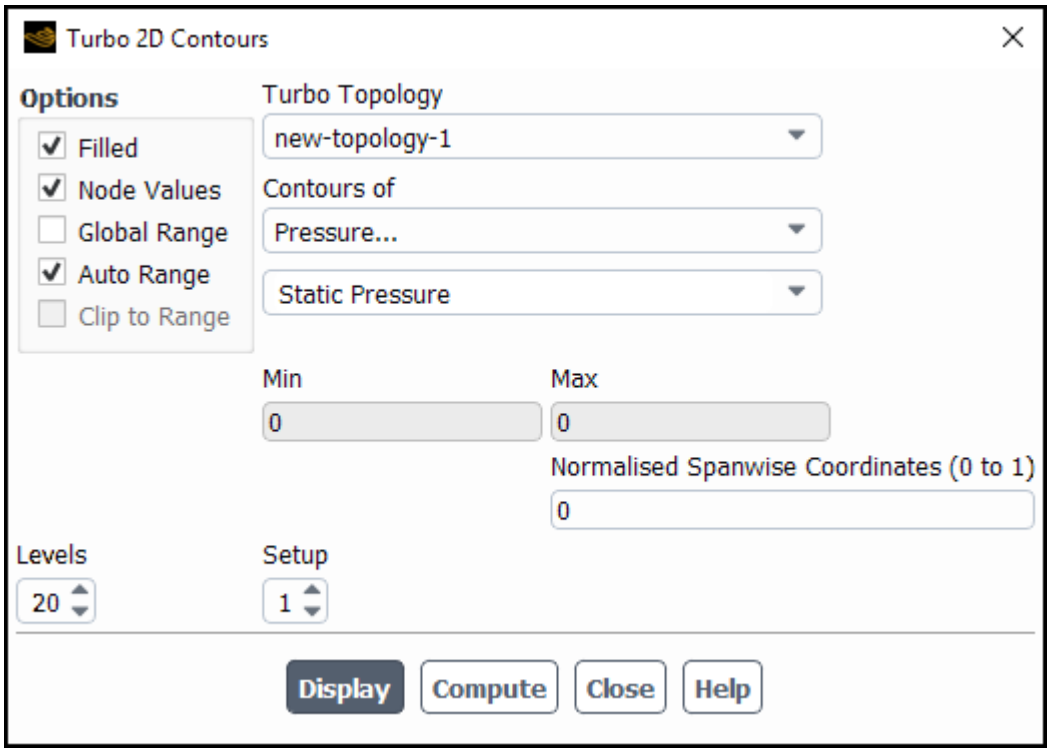
adds the specified translation offsets to the appropriate Cartesian coordinate of every node in the mesh.

52.2.122. Turbo 2D Contours Dialog Box

The **Turbo 2D Contours** dialog box enables you to display turbomachinery-specific contours of variables on surfaces of constant pitchwise, spanwise, or meridional coordinates, projected onto a plane. See Displaying Turbomachinery 2D Contours (p. 2051) for details.

Results → Model Specific → Turbo Topology → 2D Contours...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Turbo Topology

contains a list of defined topologies. Select from the list to display the values for the selected topology.

Options

contains the check buttons that set various contour display options.

Filled

toggles between filled contours and line contours.

Node Values

toggles between using scalar field values at nodes and at cell centers for computing the contours.

Global Range

toggles between basing the minimum and maximum values on the range of values on the selected surfaces (off), and basing them on the range of values in the entire domain (on, the default).

Auto Range

toggles between automatic and manual setting of the contour range. Any time you change the **Contours of** selection, **Auto Range** is reset to on.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Clip to Range

determines whether or not values outside the prescribed **Min**/ **Max** range are contoured when using **Filled** contours. If selected, values outside the range will not be contoured. If not selected, values below the **Min** value will be colored with the lowest color on the color scale, and values above the **Max** value will be colored with the highest color on the color scale. See Specifying the Range of Magnitudes Displayed (p. 3902) for details.

Levels

sets the number of contour levels that are displayed.

Setup

indicates the ID number of the contour setup. For frequently used combinations of contour fields and options, you can store the information needed to generate the contour plot by specifying a **Setup** number and setting up the desired information in the dialog box. See Storing Contour Plot Settings (p. 3910) for details.

Contours of

contains a list from which you can select the scalar field to be contoured.

Min

shows the minimum value of the scalar field. If **Auto Range** is off, you can set the minimum by typing a new value.

Max

shows the maximum value of the scalar field. If **Auto Range** is off, you can set the maximum by typing a new value.

Normalised Spanwise Coordinates (0 to 1)

enables you to specify the coordinate for the spanwise surface you want to create.

Display

displays the contours in the active graphics window.

Compute

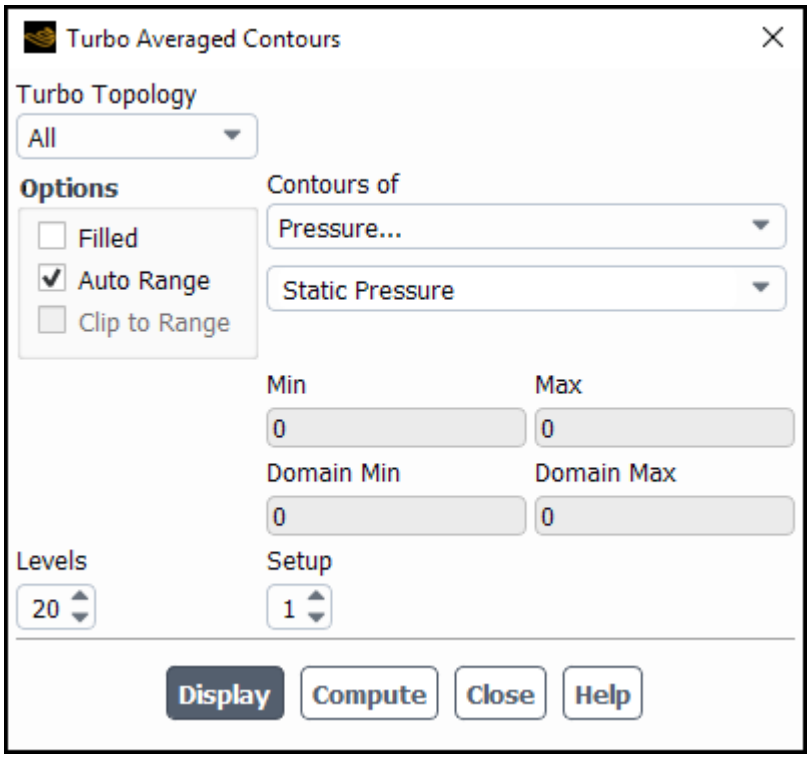
calculates the scalar field and updates the **Min** and **Max** values (even when **Auto Range** is off).

52.2.123. Turbo Averaged Contours Dialog Box

The **Turbo Averaged Contours** dialog box enables you to display turbomachinery-specific circumferentially averaged contours of variables projected on an r- z plane. See Displaying Turbomachinery Averaged Contours (p. 2049) for details.

Results → **Model Specific** → **Turbo Topology** → **Averaged Contours...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Turbo Topology

contains a list of defined topologies. Select from the list to display the values for the selected topology.

Options

contains the check buttons that set various contour display options.

Filled

toggles between filled contours and line contours.

Auto Range

toggles between automatic and manual setting of the contour range. Any time you change the **Contours of** selection, **Auto Range** is reset to on.

Clip to Range

determines whether or not values outside the prescribed **Min**/ **Max** range are contoured when using **Filled** contours. If selected, values outside the range will not be contoured. If not selected, values below the **Min** value will be colored with the lowest color on the color scale, and values above the **Max** value will be colored with the highest color on the color scale. See Specifying the Range of Magnitudes Displayed (p. 3902) for details.

Levels

sets the number of contour levels that are displayed.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setup

indicates the ID number of the contour setup. For frequently used combinations of contour fields and options, you can store the information needed to generate the contour plot by specifying a **Setup** number and setting up the desired information in the dialog box. See Storing Contour Plot Settings (p. 3910) for details.

Contours of

contains a list from which you can select the scalar field to be contoured.

Min

shows the minimum value of the scalar field. If **Auto Range** is off, you can set the minimum by typing a new value.

Max

shows the maximum value of the scalar field. If **Auto Range** is off, you can set the maximum by typing a new value.

Domain Min

shows the global minimum value of the scalar field for the entire domain.

Domain Max

shows the global maximum value of the scalar field for the entire domain.

Display

displays the contours in the active graphics window.

Compute

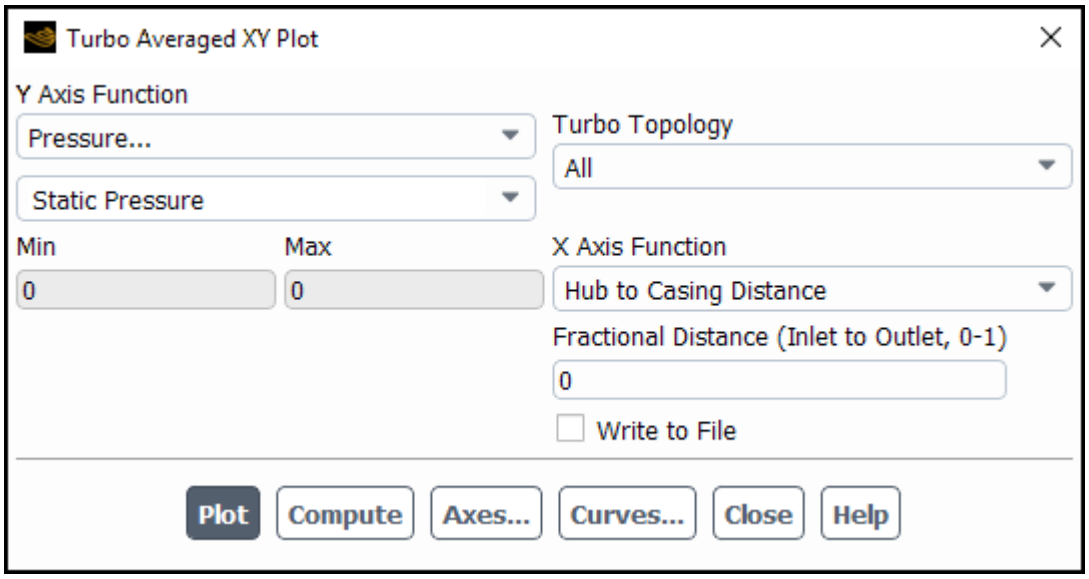
calculates the scalar field and updates the **Min** and **Max** values (even when **Auto Range** is off).

52.2.124. Turbo Averaged XY Plot Dialog Box

The **Turbo Averaged XY Plot** dialog box enables you to display data in an XY plot format as a function of either the meridional or the spanwise coordinate. See Generating Averaged XY Plots of Turbomachinery Solution Data (p. 2053) for details.

Results → Model Specific → Turbo Topology → Averaged XY Plot...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Y Axis Function

contains a list of solution variables that can be used for the y axis of the plot.

Min

shows the minimum value of the scalar field.

Max

shows the maximum value of the scalar field.

Turbo Topology

contains a list of defined topologies. Select from the list to display the values for the selected topology.

X Axis Function

enables you to select the coordinate to be used for the x axis of the plot. The choices are **Hub to Casing Distance** (spanwise coordinate), and **Meridional Distance** (meridional coordinate).

Fractional Distance

sets a fractional value (0 to 1) for either the spanwise **Hub to Casing** distance or the meridional **Inlet to Outlet** distance, depending on your selection for **X Axis Function**.

Write to File

enables the file-writing option. When this option is selected, the **Plot** button will change to **Write...**. Clicking on the **Write...** button will open The Select File Dialog Box (p. 931), in which you can specify a name and save a file containing the plot data. The format of this file is described in XY Plot File Format (p. 4071).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Plot

plots the specified surface and/or file data in the active graphics window using the current axis and curve attributes. If the **Write to File** option is turned on, this button becomes the **Write...** button.

Write...

opens The Select File Dialog Box (p. 931), in which you can save the plot data to a file. This button replaces the **Plot** button when the **Write to File** option is turned on.

Compute

calculates the scalar field and updates the **Min** and **Max** values.

Axes...

opens the Axes Dialog Box (p. 5313), which enables you to customize the plot axes.

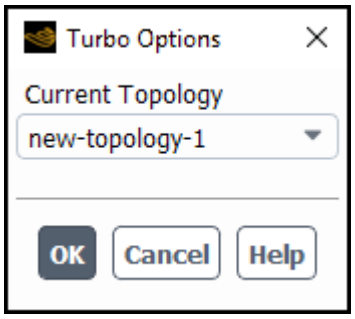
Curves...

opens the Curves Dialog Box (p. 5316), which enables you to customize the curves used in the XY plot.

52.2.125. Turbo Options Dialog Box

The **Turbo Options** dialog box enables you to globally set the turbomachinery topology for your model. See Globally Setting the Turbomachinery Topology (p. 2055) for details.

Results → **Model Specific** → **Turbo Topology** → **Options...**



Controls

Current Topology

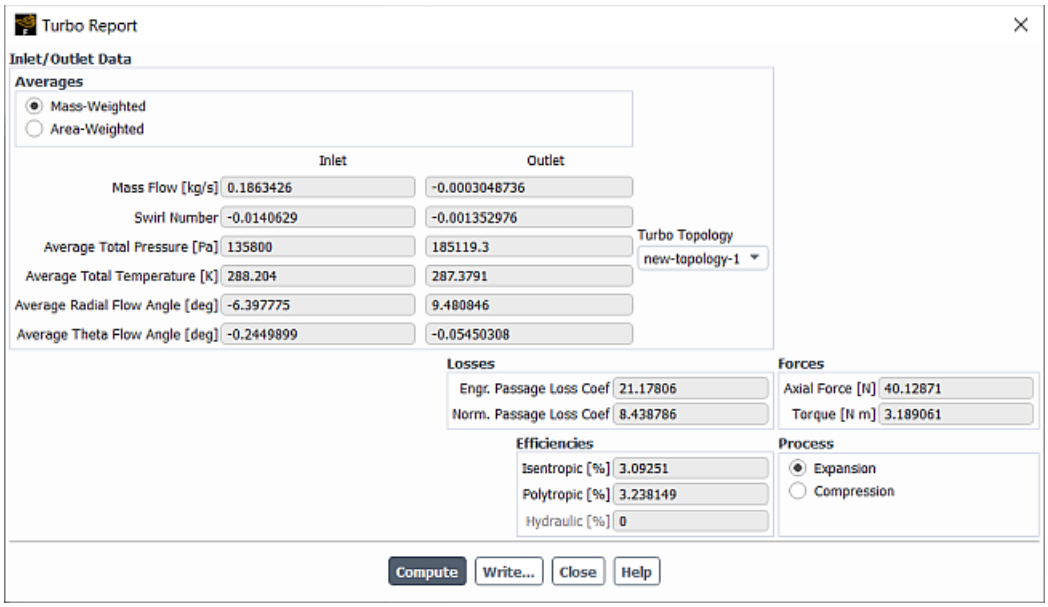
contains a list of predefined turbo topologies.

52.2.126. Turbo Report Dialog Box

The **Turbo Report** dialog box allows you to calculate turbomachinery-specific quantities and integrals. See Generating Reports of Turbomachinery Data (p. 2041) for details.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Results → Model Specific → Turbo Topology → Report...



Controls

Inlet/Outlet Data

contains quantities that can be calculated at inlets and outlets.

Averages

allows you to choose between **Mass-Weighted** and **Area-Weighted** averages for all applicable computed quantities. These quantities are calculated for the **Inlet** and **Outlet** topologies where applicable.

Turbo Topology

contains a list of defined topologies. Select from the list to display the values for the selected topology.

Mass Flow

is the mass flow rate through a surface as defined in Equation 14.19 (p. 2042).

Swirl Number

is the swirl number as defined in Equation 14.20 (p. 2042).

Average Total Pressure

is the area-averaged or mass-averaged total pressure as defined in Equation 14.21 (p. 2043) or Equation 14.22 (p. 2043).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Average Total Temperature

is the area-averaged or mass-averaged total temperature as defined in Equation 14.23 (p. 2043) or Equation 14.24 (p. 2043).

Average Radial Flow Angle

is the area-averaged or mass-averaged radial flow angle as defined in Equation 14.25 (p. 2044) or Equation 14.27 (p. 2044).

Average Theta Flow Angle

is the area-averaged or mass-averaged tangential flow angle as defined in Equation 14.26 (p. 2044) or Equation 14.28 (p. 2044).

Losses

contains the values of loss-related coefficients.

Engr. Passage Loss Coef

is the engineering loss coefficient as defined in Equation 14.29 (p. 2044).

Norm. Passage Loss Coef

is the normalized loss coefficient as defined in Equation 14.30 (p. 2044).

Forces

contains the axial force and the torque on the rotating parts.

Axial Force

is the axial force on the rotating parts as defined in Equation 14.31 (p. 2045).

Torque

is the torque on the rotating parts as defined in Equation 14.32 (p. 2045).

Efficiencies

contains the values of isentropic, polytropic and hydraulic efficiencies.

Isentropic

is the isentropic efficiency for a compressor or a turbine (motor) calculated in the presence of a compressible working fluid as defined in Equation 14.37 (p. 2047) or Equation 14.43 (p. 2049).

Polytropic

is the polytropic efficiency for a compressor or a turbine (motor) calculated in the presence of a compressible working fluid as defined in Equation 14.38 (p. 2047) or Equation 14.44 (p. 2049).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Hydraulic

is the hydraulic efficiency for a pump or a hydraulic turbine (motor) calculated in the presence of an incompressible working fluid as defined in Equation 14.33 (p. 2046) or Equation 14.39 (p. 2048).

Process

specifies whether to compute the efficiency for the flow **Expansion** (for example, in a turbine) or **Compression** (for example, in a pump or compressor).

Compute

starts the calculation of the quantities in all the fields in the **Turbo Report** dialog box. Note that this process may take some time for a large problem.

Write...

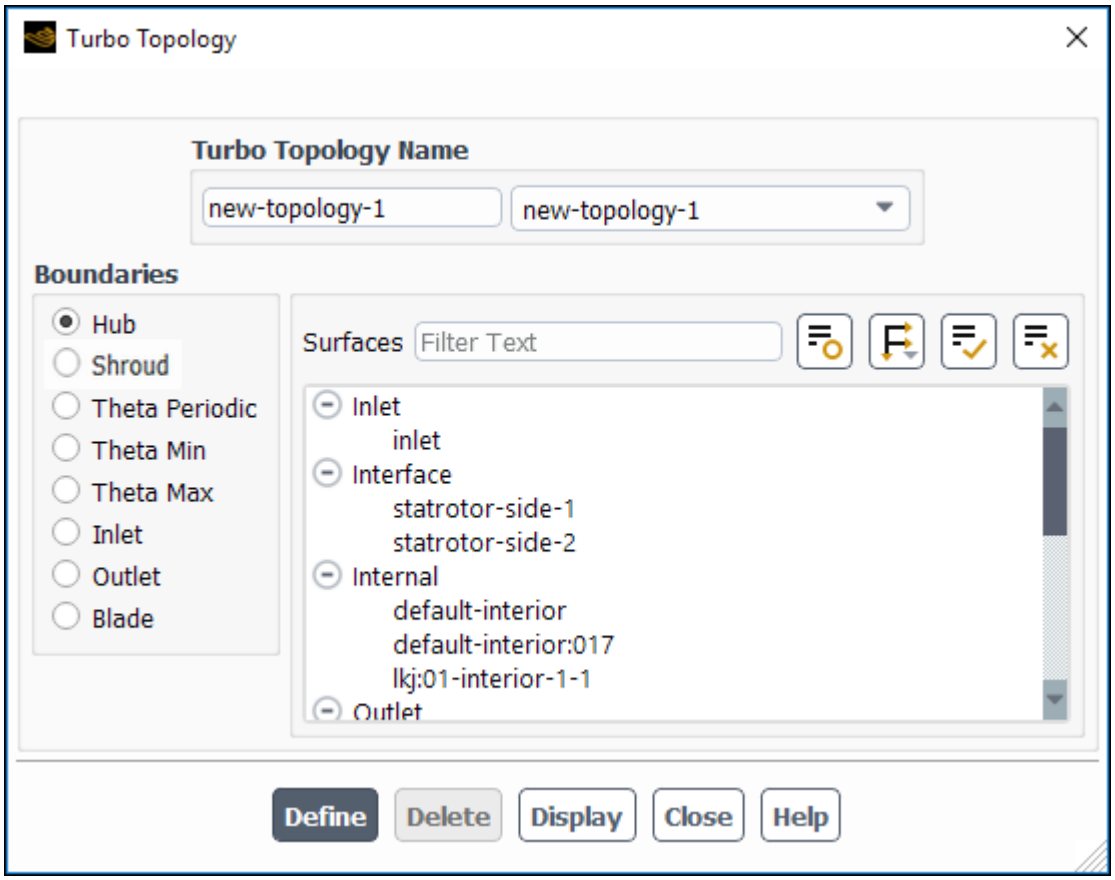
opens the **Select File** dialog box, which you can use to save the reported values to a file.

52.2.127. Turbo Topology Dialog Box

The **Turbo Topology** dialog box allows you to define the topology for a turbomachinery application, so that you can use the turbomachinery-specific postprocessing features described in Turbomachinery Postprocessing (p. 2029). See Defining the Turbomachinery Topology (p. 2029) for details about the items below.

Domain → **Mesh Models** → **Turbo Topology...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Turbo Topology Name

specifies the name of the new topology.

Boundaries

contains radio buttons for the topology boundaries to be defined.

Hub

specifies the definition for the wall zone(s) forming the lower boundary of the flow passage (generally toward the axis of rotation of the machine).

, Shroud

specifies the definition of the wall zone(s) forming the upper boundary of the flow passage (away from the axis of rotation of the machine).

Theta Periodic

specifies the definition of the periodic boundary zone(s) on the circumferential boundaries of the flow passage.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Theta Min, Theta Max

specify the definition of the wall zones at the minimum and maximum angular (θ) positions on a circumferential boundary.

Inlet

specifies the definition of the inlet zone(s) through which the flow enters the passage.

Outlet

specifies the definition of the outlet zone(s) through which the flow exits the passage.

Blade

specifies the definition of the wall zone(s) that defines the blade(s) (if any). Note that these zones cannot be attached to the circumferential boundaries. For this situation, use **Theta Min** and **Theta Max** to define the blade.

Surfaces

contains a selectable list of the available surfaces, from which you can select the surface(s) that represent the boundary selected under **Boundaries**.

Define

defines the new topology. If you have selected an existing topology the **Define** button is replaced by the **Modify** button.

Display

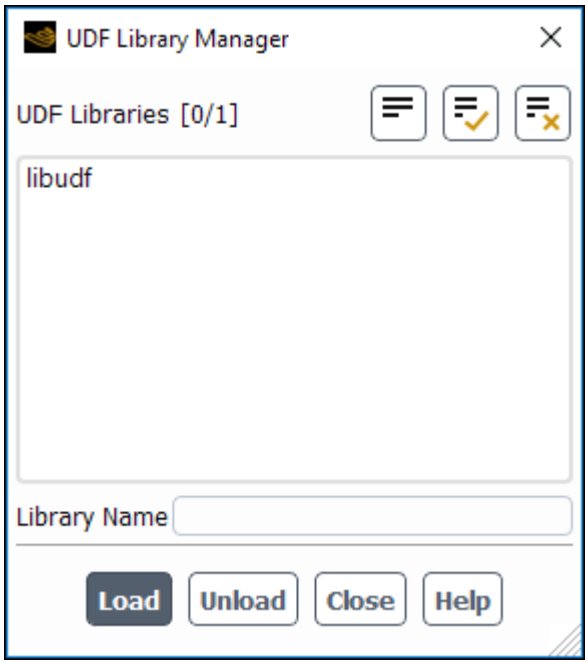
displays the defined topology in the active graphics window.

52.2.128. UDF Library Manager Dialog Box

The **UDF Library Manager** dialog box allows you to load/unload the user-defined function libraries. See the separate Fluent Customization Manual for details.

User Defined → **User Defined** → **Functions** → **Manage...**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

UDF Libraries

lists the user-defined function libraries that are loaded in Ansys Fluent.

Library Name

specifies the name of the library to be loaded/unloaded.

Load

opens the specified library and loads the user-defined function.

Unload

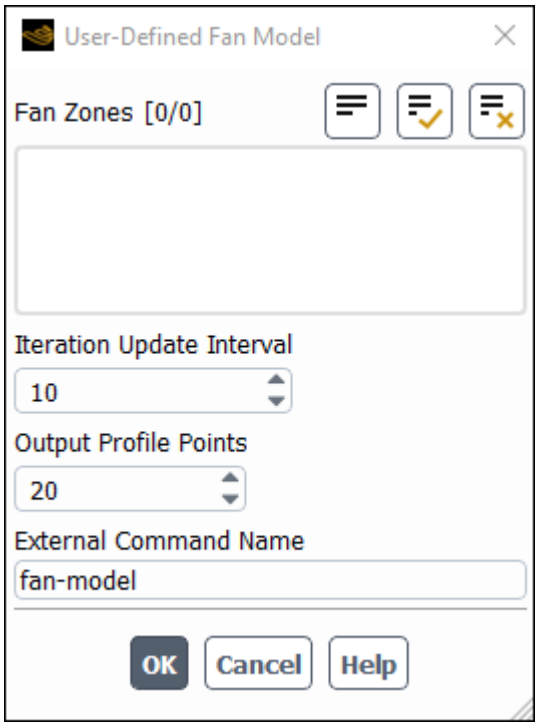
unloads the specified library.

52.2.129. User-Defined Fan Model Dialog Box

The **User-Defined Fan Model** dialog box allows you to periodically regenerate a profile file that can be used to specify the characteristics of a fan, including pressure jump across the fan, and radial and swirling components of velocity generated by the fan. See User-Defined Fan Model (p. 1556) for details about this feature and how to use this dialog box.

User Defined → Model Specific → Fan Model...

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Fan Zones

contains a list from which you can select the fan zone(s) on which your executable will operate.

Iteration Update Interval

specifies how often the executable will be called on to update the fan profile file.

Output Profile Points

specifies the number of points in the profile file to be written by Ansys Fluent.

External Command Name

specifies the name of the executable.

52.2.130. User-Defined Function Hooks Dialog Box

The **User-Defined Function Hooks** dialog box allows you to specify user-defined functions (UDFs) connected to a number of models and procedures in Ansys Fluent. See the separate Fluent Customization Manual for details.

Important:

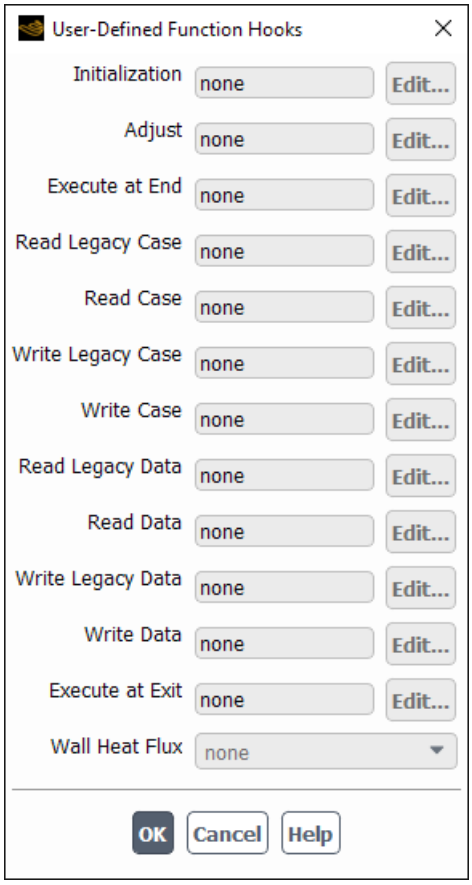
You can hook multiple UDFs to the following functions:

- **Initialization**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- **Adjust**
- **Execute At End**
- **Read Legacy Case**
- **Read Case**
- **Write Legacy Case**
- **Write Case**
- **Read Legacy Data**
- **Read Data**
- **Write Legacy Data**
- **Write Data**
- **Execute at Exit**

User Defined → User Defined → Function Hooks...



Controls

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Initialization

selects a UDF that is called immediately after you initialize your flow field.

Adjust

selects a UDF that is called at the beginning of an iteration before solution of velocities, pressure, and other quantities begins.

Execute At End

selects a UDF that is called at the end of an iteration or time-step.

Read Legacy Case

selects a UDF that defines a customized section that is to be read from the legacy case file.

Read Case

selects a UDF that defines a customized section that is to be read from the case file.

Write Legacy Case

selects a UDF that defines a customized section that is to be written to the legacy case file.

Write Case

selects a UDF that defines a customized section that is to be written to the case file.

Read Legacy Data

selects a UDF that defines a customized section that is to be read from the legacy data file.

Read Data

selects a UDF that defines a customized section that is to be read from the data file.

Write Legacy Data

selects a UDF that defines a customized section that is to be written to the legacy data file.

Write Data

selects a UDF that defines a customized section that is to be written to the data file.

Execute at Exit

selects a UDF that is called when exiting an Ansys Fluent session.

Wall Heat Flux

selects a UDF that modifies the way that the solver computes the heat flux between a wall and the neighboring fluid cells.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Net Reaction Rate

selects a UDF that defines the net reaction rate.

Volume Reaction Rate

selects a UDF that defines a volumetric reaction rate.

Surface Reaction Rate

selects a UDF that defines a surface reaction rate.

Particle Reaction Rate

selects a UDF that defines a particle reaction rate.

Turbulent Premixed Source

selects a UDF that defines the turbulent flame speed and source term for the premixed or partially premixed combustion model.

Chemistry Step

selects a UDF that defines the chemistry step function.

Spray Collide

selects a UDF that defines the spray collide function.

Cavitation Mass Rate

selects a UDF that defines the cavitation rate.

DO Source

selects a UDF that defines the discrete ordinate source function.

DO Diffuse Reflectivity

selects a UDF that defines the diffuse reflectivity function for the DO radiation model.

DO Specular Reflectivity

selects a UDF that defines the specular reflectivity function for the DO radiation model.

Emissivity Weighting Factor

selects a UDF that defines the emissivity weighting factor for the non-gray DO radiation model or the non-gray P-1 radiation model.

Thickened Flame Model Parameters

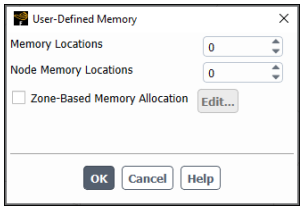
selects a UDF that defines the parameters for the Thickened Flame Model.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

52.2.131. User-Defined Memory Dialog Box

The **User-Defined Memory** dialog box allows you to allocate memory for user-defined storage variables. See the separate Fluent Customization Manual for details.

User-Defined → User Defined → Memory...



Controls

Memory Locations

specifies the number of memory locations to be allocated.

Node Memory Locations

specifies the number of node memory locations to be allocated.

Zone-Based Memory Allocation

specifies that the user-defined memory will only be allocated to the specified zones, rather than the whole domain.

Edit...

opens the Select UDM Zones Dialog Box (p. 5604).

Important:

When writing data files in the legacy format (`.dat` files), you must select the UDM quantities using the **Data File Quantities** dialog box in order to postprocess them in CFD-Post. Alternatively, you have the option of exporting the desired quantities to a `.cdat` file. This will ensure that all the UDM quantities are available for postprocessing in CFD-Post. Refer to the Ansys CFD-Post manual for more information.

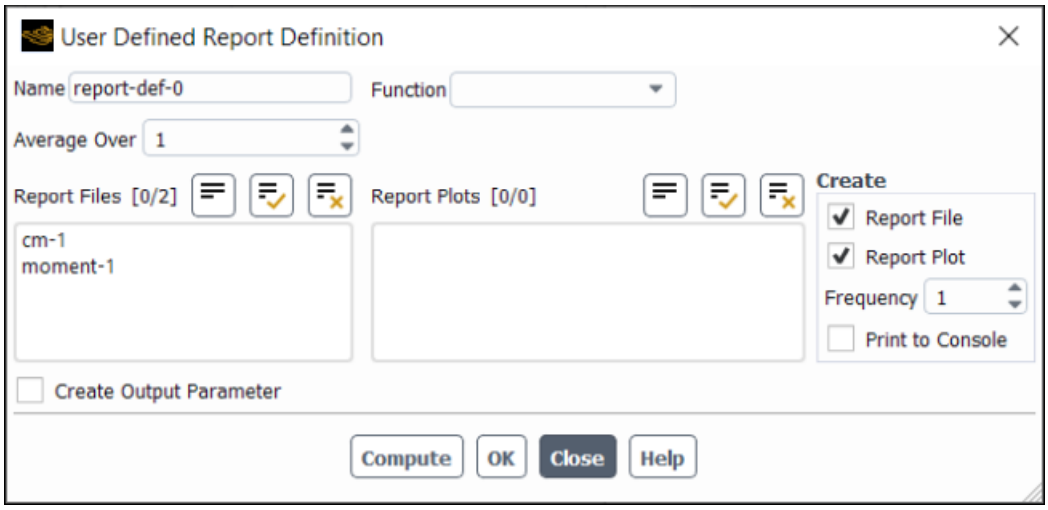
52.2.132. User Defined Report Definition Dialog Box

Solution → Reports → Definitions → New → User Defined...

or

Solution → Report Definitions → New → User Defined

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name

specifies the name of the report definition.

Function

drop-down list where you can select the function that you loaded for this report definition.

Average Over

(optional) To have Fluent calculate a running average for the **User Defined Report Definition** you can enter a positive integer greater than 1 (the default) for **Average Over**.

Specifying a number greater than 1 means that Fluent will print, plot, and write the running average value of the selected variable instead of the current value of the same variable.

The value reported is averaged over the last N *iterations/time steps, where* N *is your specified* ***Average Over*** *value. When the iteration/time step number is lower than* N*, Fluent calculates the average of the available variable values.*

Retain Instantaneous Values

appears when **Average Over** is set to a value greater than 1. If enabled when you are creating a new report definition, it ensures that all instantaneous values are saved in the new report file that is being created for this report definition. It also results in the **Plot Instantaneous Values** option being enabled in the newly created report plot.

Report Files

lists all of the report files where you can write the report definition data.

Report Plots

lists all of the report plots where you can plot the report definition data (assuming this report has the same units as the other report definitions included in the selected report plot).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Create

Report File

creates a new report file that includes this report definition.

Report Plot

creates a new report plot that includes this report definition.

Frequency

specifies how frequently the report definition is written, plotted, or printed to the console.

Print to Console

prints the value of the report definition to the console.

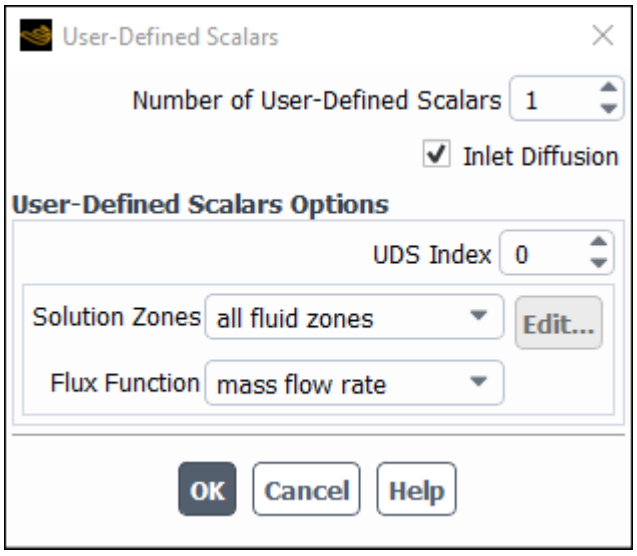
Create Output Parameter

creates an output parameter for this report definition with the name `<report definition name>-op`.

52.2.133. User-Defined Scalars Dialog Box

The **User-Defined Scalars** dialog box allows you to include user-defined scalar transport equations in your calculation. See User-Defined Scalar (UDS) Transport Equations (p. 1737) for details.

User Defined → User Defined → Scalars...



Controls

Number of User-Defined Scalars

specifies how many additional scalar transport equations you would like to include in the calculation.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Inlet Diffusion

when enabled allows you to include the diffusion term in the UDS transport equation for all inflow and outflow boundaries.

User-Defined Scalars Options

contains settings that define the scalar transport equation.

UDS Index

when set to 0 marks the first user-defined scalar equation.

Solution Zones

specifies in which zone the scalar equation will be solved: **all fluid zones**, **all solid zones**, **all zones** (fluid and solid) or **selected zones**.

Flux Function

is a drop-down list containing available functions for the convection term of the user-defined scalar transport equation(s).

none

(the default) indicates that there is no convection term included in the scalar transport equation(s); that is, you want to solve a Poisson equation instead of a convection/diffusion equation.

mass flow rate

indicates that the convection term in the scalar transport equation(s) is equal to the mass flow rate $\rho\vec{v}\cdot\vec{A}$.

Note that the **none** and **mass flow rate** options will apply to all solved user-defined scalars. A user-defined flux function must be supplied if a different convective flux is desired for each user-defined scalar.

Unsteady Function

is a drop-down list containing available functions for the unsteady term of the user-defined scalar transport equation(s).

52.2.134. Vectors Dialog Box

The **Vectors** dialog box controls the display of vector plots. See Displaying Vectors (p. 3912) for details about the items below.

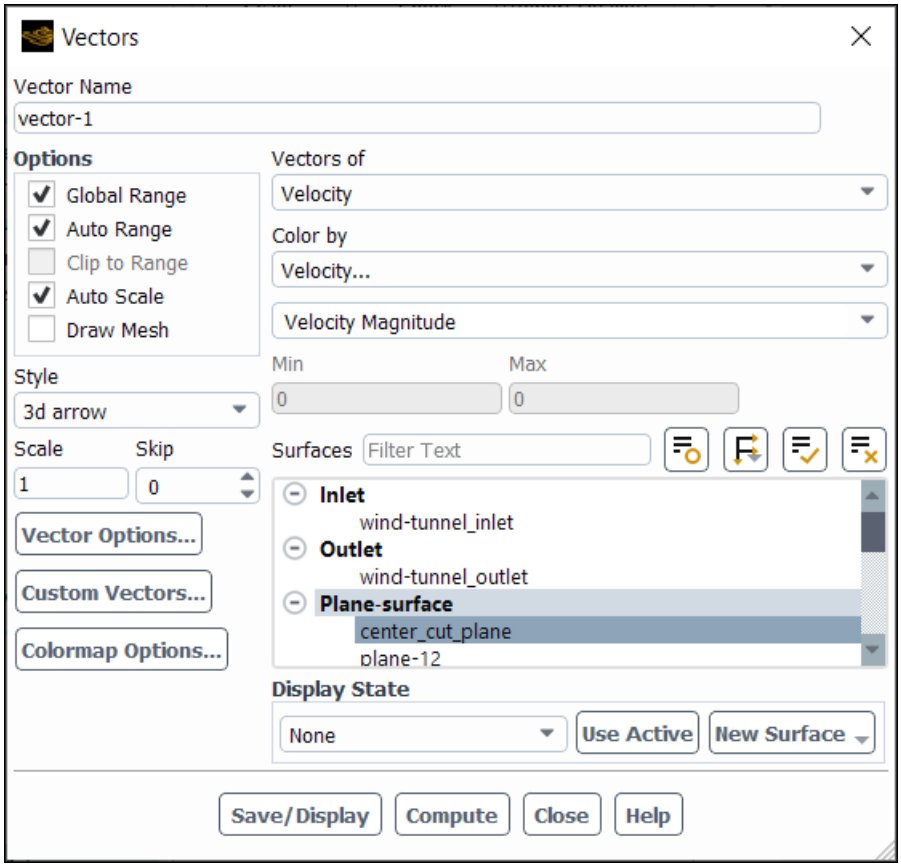
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Results → Graphics → Vectors → New...

Note:

- The "global"/non-persistent form of graphics objects is hidden by default. If this interrupts your workflow, you can make them available by enabling the **Expose legacy non-persistent graphics** option in the **Graphics** branch of **Preferences** (accessed via **File**>**Preferences...**).
- (When you click **Edit...** instead of **New...**, it opens the "global"/non-persistent version of the **Vectors** dialog box. This version of the dialog box does not have a **Name** field and it is not saved with the case file.

  You are encouraged to use the persistent version that is accessed by clicking **New...**.



Controls

Vector Name

is the name for a vector plot definition. You can specify a name or use the default name **`vector-id`**. This control appears only for vector plot definitions.

Options

contains check buttons that set various display options.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Global Range

toggles between basing the minimum and maximum values on the range of values on the selected surfaces (off), and basing them on the range of values in the entire domain (on, the default).

Auto Range

toggles between automatic and manual setting of the range of scalar field values.

Clip to Range

controls the display of vectors that have a value outside the range specified by **Min** and **Max**. When on, no vectors are displayed outside the range. When off, vectors are displayed outside the range using the colors at the top and bottom of the color scale. This option is applicable only when **Auto Range** is off. See Specifying the Range of Magnitudes Displayed (p. 3917) for details.

Auto Scale

enables the scaling of all vectors in the domain such that when the **Scale** is 1, there will be minimal overlap of vectors.

Draw Mesh

toggles between displaying and not displaying the mesh. The Mesh Display Dialog Box (p. 4688) is opened when **Draw Mesh** is selected.

Style

selects the style in which the vectors are drawn. Available styles are **3d arrow**, **3d arrowhead**, **cone**, **filled-arrow**, **arrow**, **harpoon**, and **headless**.

Scale

sets the factor by which the vectors should be scaled. See Scaling the Vectors (p. 3915) for details.

Skip

allows you to "thin" or "sample" the vectors that are displayed. See Skipping Vectors (p. 3915) for details.

Vector Options...

opens the Vector Options Dialog Box (p. 5239), in which you can set additional options for vector displays.

Custom Vectors...

opens the Custom Vectors Dialog Box (p. 5240), in which you can define your own vector fields.

Colormap Options...

opens the Colormap Dialog Box (p. 5280), allowing you to customize the colormap for this graphics object.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Vectors of

contains a list from which you can select the vector field to be plotted.

Color by

contains a list from which you can select the scalar field by which the vectors are colored.

Phase

(Multiphase only) specifies the phase where the postprocessing operations are being applied.

Min

shows the value to which the lower end of the color scale is mapped. You can set this value manually if **Auto Range** is off.

Max

shows the value to which the upper end of the color scale is mapped. You can set this value manually if **Auto Range** is off.

Surfaces

contains a list from which you can select the surfaces on which to display vectors. In 2D, vectors are displayed on the entire domain if no surface is selected.

Display State

specifies the display state that is associated with this graphics object.

drop-down list of display states

selectable list of all the existing display states associated with this case.

Use Active

creates a display state that matches the settings of the active graphics window and assigns it to this graphics object (this association is only saved after you click **Save/Display**).

New Surface

is a drop-down list button that contains a list of surface options:

Point

opens the Point Surface Dialog Box (p. 5582).

Line/Rake

opens the Line/Rake Surface Dialog Box (p. 5514).

Plane

opens the Plane Surface Dialog Box (p. 5580).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Quadric

opens the Quadric Surface Dialog Box (p. 5584).

Iso-Surface

opens the Iso-Surface Dialog Box (p. 5501).

Iso-Clip

opens the Iso-Clip Dialog Box (p. 5500).

Structural Point

opens the Structural Point Surface Dialog Box (p. 5623).

Display

displays the vectors in the active graphics window.

Compute

calculates the scalar field and updates the **Min** and **Max** values (even when **Auto Range** is off).

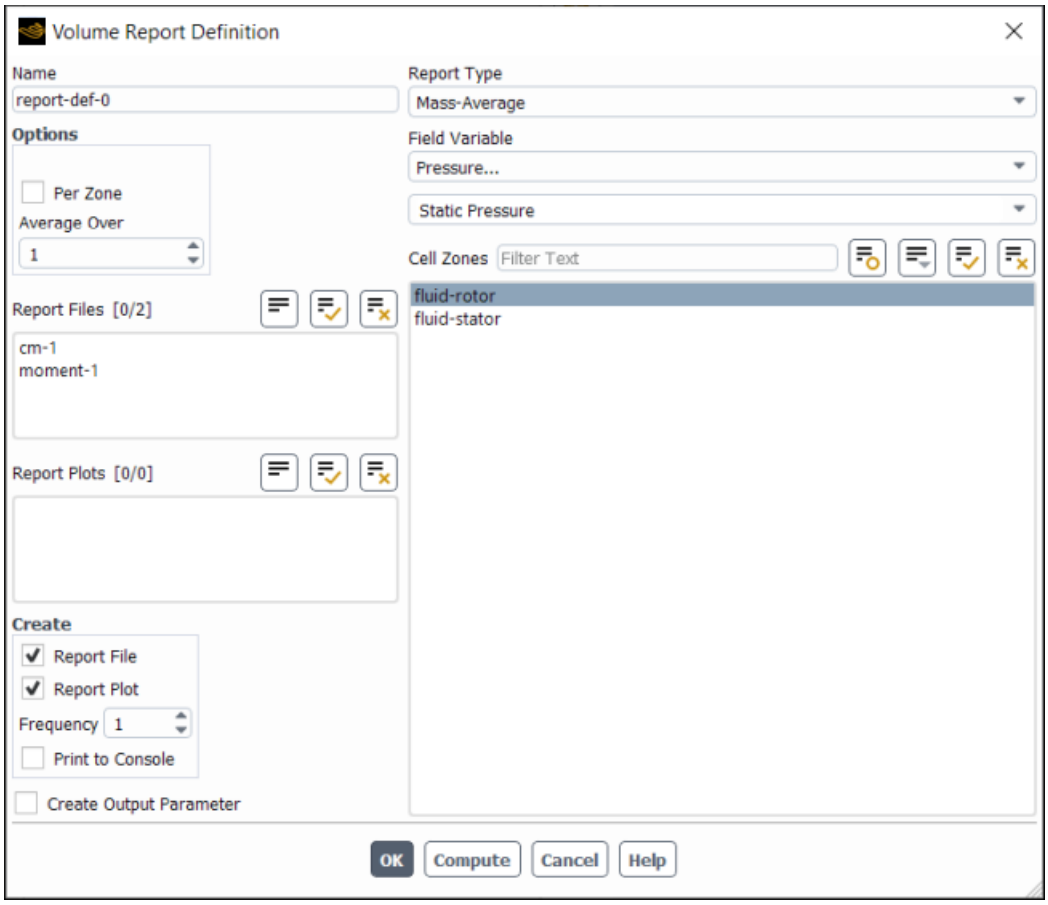
Save/Display

plots the vectors in the active graphics window and saves the vector plot definition. This button appears only for vector plot definitions and replaces the **Display** button.

52.2.135. Volume Report Definition Dialog Box

Solution → **Reports** → **Definitions** → **New** → **Volume Report**

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.



Controls

Name

specifies the name of the report definition.

Options

Per Zone

specifies whether or not the chosen field variable is calculated from all of the selected cell zones combined (default) or individually on each of the selected cell zones.

Average Over

(optional) To have Fluent calculate a running average for the **Volume Report Definition** you can enter a positive integer greater than 1 (the default) for **Average Over**.

Specifying a number greater than 1 means that Fluent will print, plot, and write the running average value of the selected variable instead of the current value of the same variable.

The value reported is averaged over the last N *iterations/time steps, where* N *is your specified* ***Average Over*** *value. When the iteration/time step number is lower than* N*, Fluent calculates the average of the available variable values.*

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Retain Instantaneous Values

appears when **Average Over** is set to a value greater than 1. If enabled when you are creating a new report definition, it ensures that all instantaneous values are saved in the new report file that is being created for this report definition. It also results in the **Plot Instantaneous Values** option being enabled in the newly created report plot.

Report Files

lists all of the report files where you can write the report definition data.

Report Plots

lists all of the report plots where you can plot the report definition data (assuming this report has the same units as the other report definitions included in the selected report plot).

Create

Report File

creates a new report file that includes this report definition.

Report Plot

creates a new report plot that includes this report definition.

Frequency

specifies how frequently the report definition is written, plotted, or printed to the console.

Print to Console

prints the value of the report definition to the console.

Create Output Parameter

creates an output parameter for this report definition with the name `<report definition name>-op`.

Report Type

selects the integration method used on the selected cell zones. The available report types are the same as those in the Volume Integrals Dialog Box (p. 5329). See Volume Integration (p. 4153) for details.

Field Variable

contains a list of solution variables that can be monitored on the selected cell zones. This list will not be available if you select **Volume** as the **Report Type**.

Phase

contains a list of all of the phases in the problem that you have defined. This is available when the VOF, mixture, or Eulerian multiphase model is enabled.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cell Zones

contains a selectable list of the current cell zones.

OK

creates the report definition.

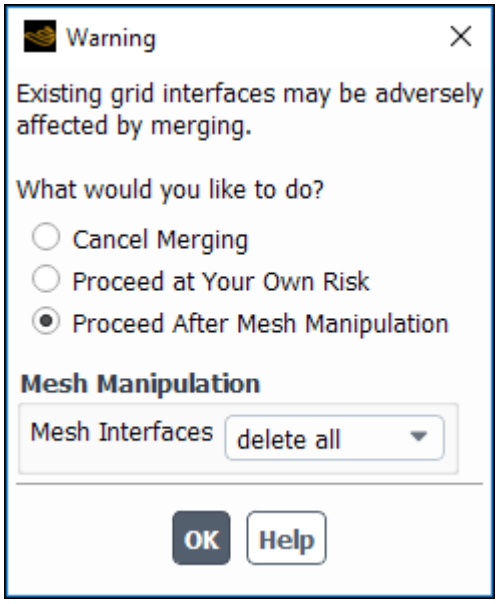
Compute

computes the value of the report definition at the selected zone(s) and prints to the console.

52.2.136. Warning Dialog Box

The **Warning** dialog box allows you to specify whether existing dynamic zones and/or mesh interfaces are deleted when you are using the **Merge Zones** dialog box to merge multiple zones of the same type into a single zone (see Merge Zones Dialog Box (p. 5525) for details). Note that dynamic zones and mesh interfaces that exist at the time of the merge may be adversely affected by the merge.

The **Warning** dialog box opens automatically when you click **Merge** in the Merge Zones Dialog Box (p. 5525) if your case has dynamic zones or mesh interfaces.



Controls

Cancel Merging

returns you to the **Mesh Zones** dialog box when you click **OK**, without initiating the merge.

Proceed at Your Own Risk

allows the merge to proceed when you click **OK**, without deleting the existing dynamic zones and mesh interfaces beforehand.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Proceed After Mesh Manipulation

deletes the existing dynamic zones and/or mesh interfaces (based on the settings in the **Mesh Manipulation** group box) and then merges the zones when you click **OK**.

Mesh Manipulation

allows you to specify whether the existing dynamic zones and/or mesh interfaces are deleted prior to the initiation of the merge. This group box is only available when **Proceed After Mesh Manipulation** is selected.

Dynamic Zones

specifies whether all existing dynamic zones are deleted prior to the merge.

Mesh Interfaces

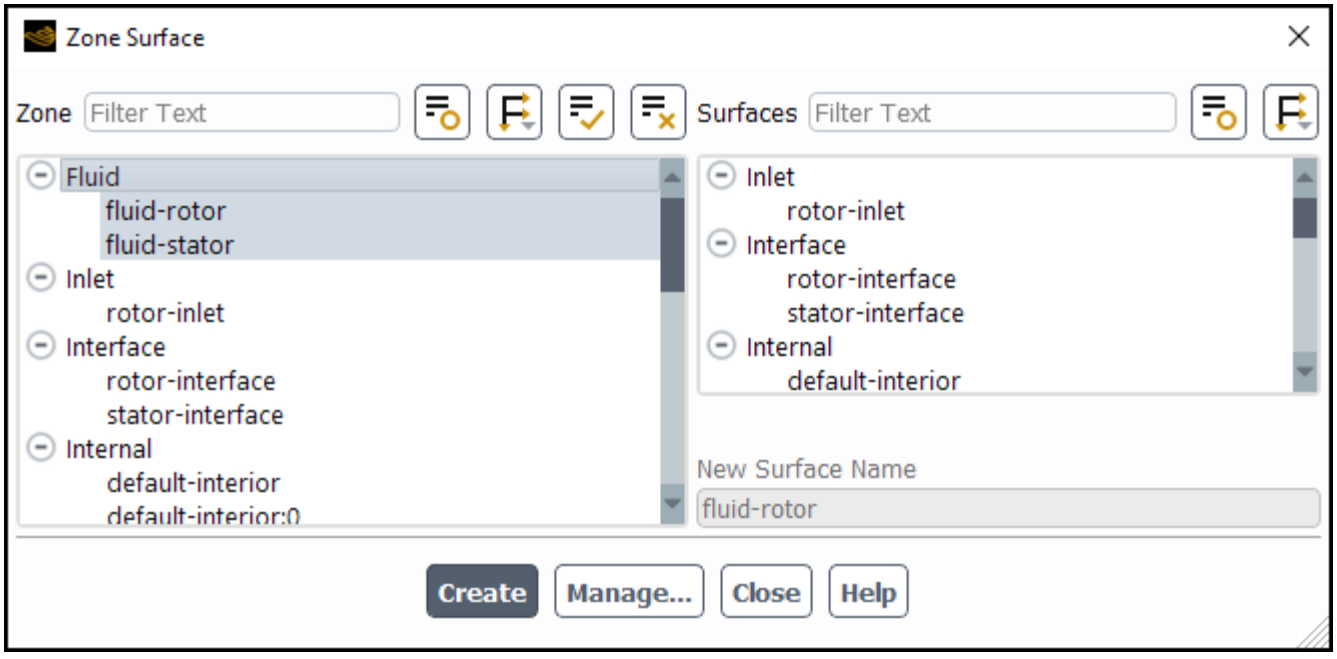
specifies whether all existing mesh interfaces are deleted prior to the merge.

Note that selecting **keep all** from both drop-down lists in the **Mesh Manipulation** group box is the equivalent of selecting **Proceed at Your Own Risk**.

52.2.137. Zone Surface Dialog Box

The **Zone Surface** dialog box allows you to interactively create surfaces from face and cell zones in the domain. See Zone Surfaces (p. 3815) for details about the items below.

Domain → Surface → Create → Zone...



Controls

Zone

lists zones that you can select for creating zone surfaces.

Surfaces

displays an informational list of existing surfaces.

New Surface Name

designates the name of the surface to be created. The default is the zone name.

Create

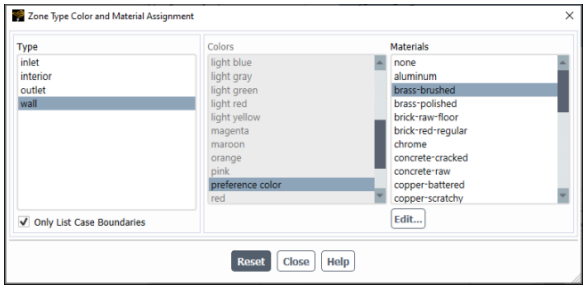
creates the surface.

Manage...

opens the Surfaces Dialog Box (p. 5629) in which you can rename and delete surfaces and determine their sizes.

52.2.138. Zone Type Color and Material Assignment Dialog Box

The **Zone Type Color and Material Assignment** dialog box allows you to control the mesh coloring for meshes displayed using the Displaying the Mesh (p. 3872) with **By Type** selected for **Coloring**. It is opened by clicking **Edit...** next to **By Type** in the Mesh Display Dialog Box (p. 4688) See Zone Surfaces (p. 3815) for details about the items below.



Controls

Type

contains a selectable list of zone types. Select a **Type** to set its color and/or material.

Only List Case Boundaries

limits the **Type** list to only include zone types that are included in the current case file.

Colors

lists the available colors that may be used to color the selected zone type.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Materials

lists the available materials that may be used to color the selected zone type. Click **Edit** if you want to modify a material's color. *See Creating Custom Realistic Materials (p. 3889) for additional information.*

Edit

opens the Material Editor Dialog Box (p. 5521).

Reset

resets the mesh colors and material assignments to the default selections for all zone types.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Appendix A. Ansys Fluent Model Compatibility

The following tables summarize the compatibility of several Ansys Fluent model categories:

- Multiphase Models (see Modeling Multiphase Flows (p. 2915))
- Moving Domain Models (See Modeling Flows with Moving Reference Frames (p. 1769))
- Turbulence Models (See Modeling Turbulence (p. 2061))
- Combustion Models (See Chapters Modeling Species Transport and Finite-Rate Chemistry (p. 2355) – Modeling Engine Ignition (p. 2571))

Note that a ✓ indicates that two models are compatible with each other, while the absence of a ✓ indicates that two models are not compatible with each other.

Table 1: Moving Domain Models vs. Multiphase Models

	Eulerian	VOF	Mixture	Discrete Phase
Sliding Mesh	✓	✓	✓	✓
Mixing Plane	✓	✓	✓	
Dynamic Mesh	✓	✓	✓	✓
Multiple Reference Frame	✓	✓	✓	✓
Single Reference Frame	✓	✓	✓	✓

Table 2: Multiphase Models vs. Turbulence Models

	Spalart–Allmaras	k-epsilon*	k-omega**	Reynolds Stress	LES
Eulerian		✓	✓	✓	
VOF	✓	✓	✓	✓	✓
Mixture	✓	✓	✓	✓	✓
Discrete Phase	✓	✓	✓	✓	✓

Table 3: Combustion Models vs. Multiphase Models

	Eulerian	VOF	Mixture	Discrete Phase
Laminar Finite Rate	✓	✓	✓	✓
Eddy Dissipation	✓	✓	✓	✓

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Eddy Dissipation Concept				✓
Non-Premixed				✓
Premixed				✓
Partially Premixed				✓
Composition PDF Transport				Eulerian PDF only
Pollutants				✓

Table 4: Moving Domain Models vs. Turbulence Models

	Spalart–Allmaras	k-epsilon*	k-omega**	Reynolds Stress	LES
Sliding Mesh	✓	✓	✓	✓	✓
Mixing Plane	✓	✓	✓	✓	
Dynamic Mesh	✓	✓	✓	✓	✓
Multiple Reference Frame	✓	✓	✓	✓	✓
Single Reference Frame	✓	✓	✓	✓	✓

Table 5: Combustion Models vs. Moving Domain Models

	Sliding Mesh	Mixing Plane	Dynamic Mesh	Multiple Reference Frame	Single Reference Frame
Laminar Finite Rate	✓		✓	✓	✓
Eddy Dissipation	✓		✓	✓	✓
Eddy Dissipation Concept	✓		✓	✓	✓
Non-Premixed	✓		✓	✓	✓
Premixed	✓		✓	✓	✓
Partially Premixed	✓		✓	✓	✓
Composition PDF Transport	✓		✓	✓	✓
Pollutants	✓		✓	✓	✓

Table 6: Combustion Models vs. Turbulence Models

	Spalart-Allmaras	k-epsilon*	k-omega**	Reynolds Stress	LES
Laminar Finite Rate	✓	✓	✓	✓	✓

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Eddy Dissipation		✓	✓	✓	✓
Eddy Dissipation Concept		✓	✓	✓	✓
Non-Premixed		✓	✓	✓	✓
Premixed		✓	✓	✓	✓
Partially Premixed		✓	✓	✓	✓
Composition PDF Transport		✓	✓	✓	✓
Pollutants		✓	✓	✓	✓

Key:

✓ - compatible

* - Includes Standard, RNG, and Realizable k-epsilon models

** - Includes Standard, BSL, and SST k-omega models

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Appendix B. Ansys Fluent File Formats

This appendix provides information about the following:

B.1. CFF File Format
B.2. Legacy Case and Data File Formats
B.3. Mesh Morpher/Optimizer File Formats
B.4. Conduction Settings File Format
B.5. 3D Fan Curve File Format

B.1. CFF File Format

By default, Ansys Fluent writes case and data files in the Common Fluids Format (CFF). CFF files offer greatly improved read and write performance compared to legacy files and can be postprocessed in CFD-Post and EnSight, as well as in third-party products. A case file contains a mesh and settings for your specific problem. Solution data are typically saved in a separate data file related to the case file where the mesh and settings are stored.

The CFF format is based on the HDF5 format, which is an open-source hierarchical data format and a library of APIs that provides a near-random access to any dataset in both serial and parallel processing environments. All settings are saved as formatted strings in multiple groups of CFF files. Heavy-weight data are stored as a 1D or 2D array in the HDF5 dataset. Additional light-weight data (such as minimum and maximum IDs, total number of elements/subgroups, and so on) are added to the groups and dataset as attributes.

Information is presented in the following sections:

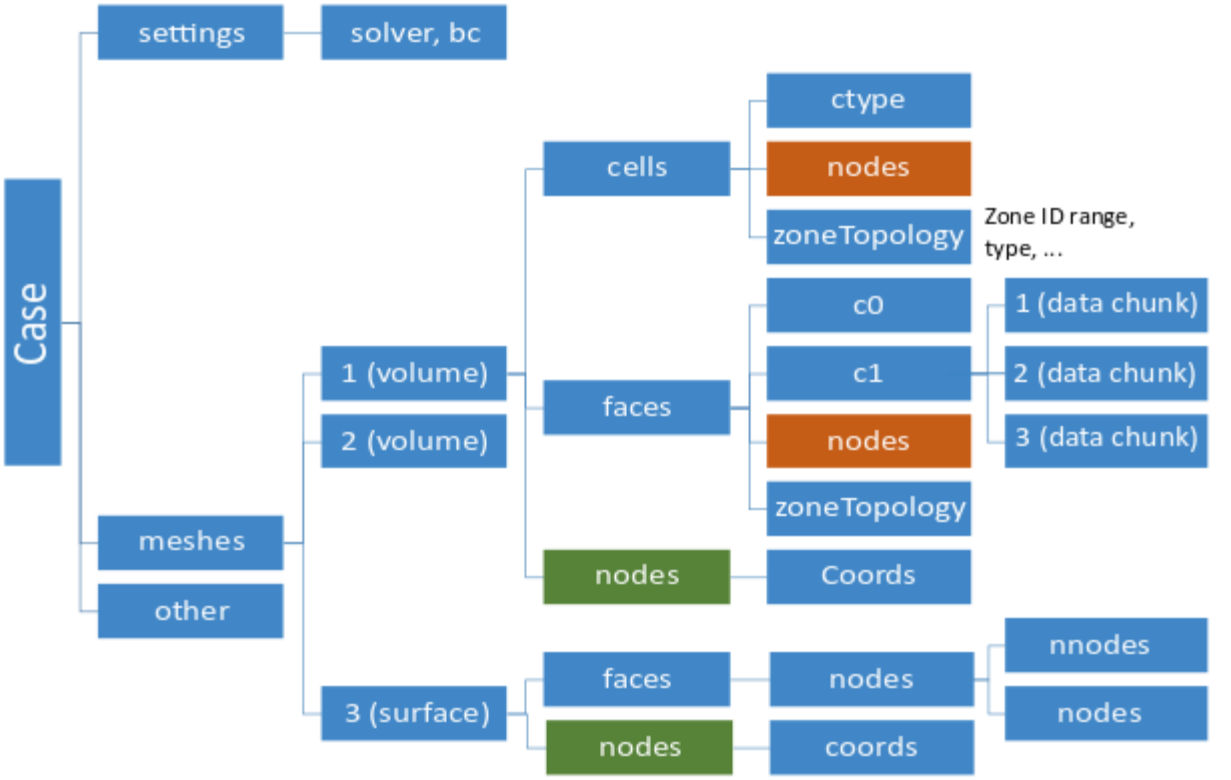
B.1.1. CFF Case File Layout
B.1.2. CFF Solution Data File Layout
B.1.3. Variable Sized Data on Collected Element / Node Sets

B.1.1. CFF Case File Layout

Figure 1: The CFF Case File Layout (p. 5672) schematically shows a structure of a CFF case file.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 1: The CFF Case File Layout



A case file contains the following groups:

- `meshes`

  The `meshes` group stores mesh data and can host multiple meshes. Each mesh is stored in a separate subgroup with an integer ID as the section name. The ID is currently not used, and only one section is expected (ID = 1). The connectivities of faces, cells and nodes are each stored in separate sections called `faces`, `cells`, and `nodes`, respectively. The connectivity is local to each mesh. All these settings are saved as formatted strings in multiple file sections. Any lightweight settings are stored within a specific section.

- `settings`

  The `settings` group stores information about case settings, which include:

  – Physical models

  – Solver settings

  – Boundary conditions

  – Materials

  Each solver can write its settings in its native format as strings that can be read back natively.

- `other` (optional)

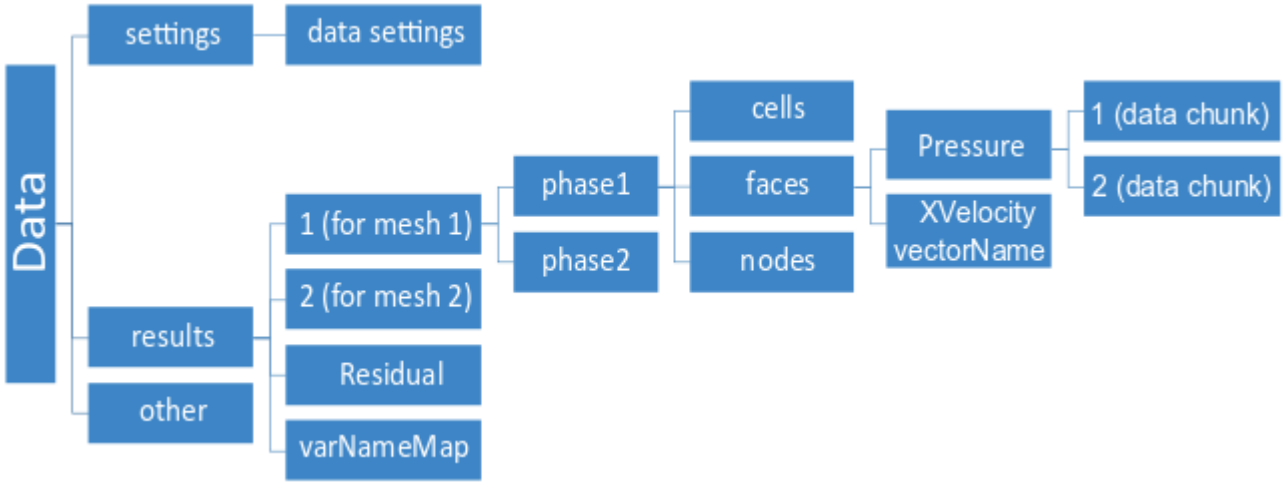
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The `other` group is used by specific solvers to save any mesh-related data other than those stored in the `meshes` group. These data will normally be consumed by the original solver that wrote it. If it is consumed by other parties, a certain agreement will be used between the parties involved.

B.1.2. CFF Solution Data File Layout

Figure 2: The CFF Data File Layout (p. 5673) schematically shows a structure of a CFF data file.

Figure 2: The CFF Data File Layout



A CFF data file contains the following groups:

- `results`

  The `results` group stores solution data. This group hosts results for multiple meshes, each related to the same mesh ID used in the `meshes` group in the case file. For a multiphase problem, each `results` group has a `phase` subgroup under each mesh (for example `results/1/phases-1/`) that stores the results for this specific phase. The name of the node (for example `phase-1`, `phase-2`, and so on) is synchronized with the phase ID attribute. Under each `phase` subgroup, one or more zone type sections are defined; each zone type contains a list of relevant variables. Lists of variables for different zone types can be different. A data section `fields` lists all available variables as a string for each zone type.

- `settings`

  The `settings` group stores solution-specific settings (such as time step, solver time, and so on). The group contains lightweight data. Each solver writes its settings in its native format as strings that can be read back natively.

- `other` (optional)

  The `other` group stores any solution data associated with meshes in the corresponding case file. These data will normally be consumed by the original solver (usually by Ansys Fluent).

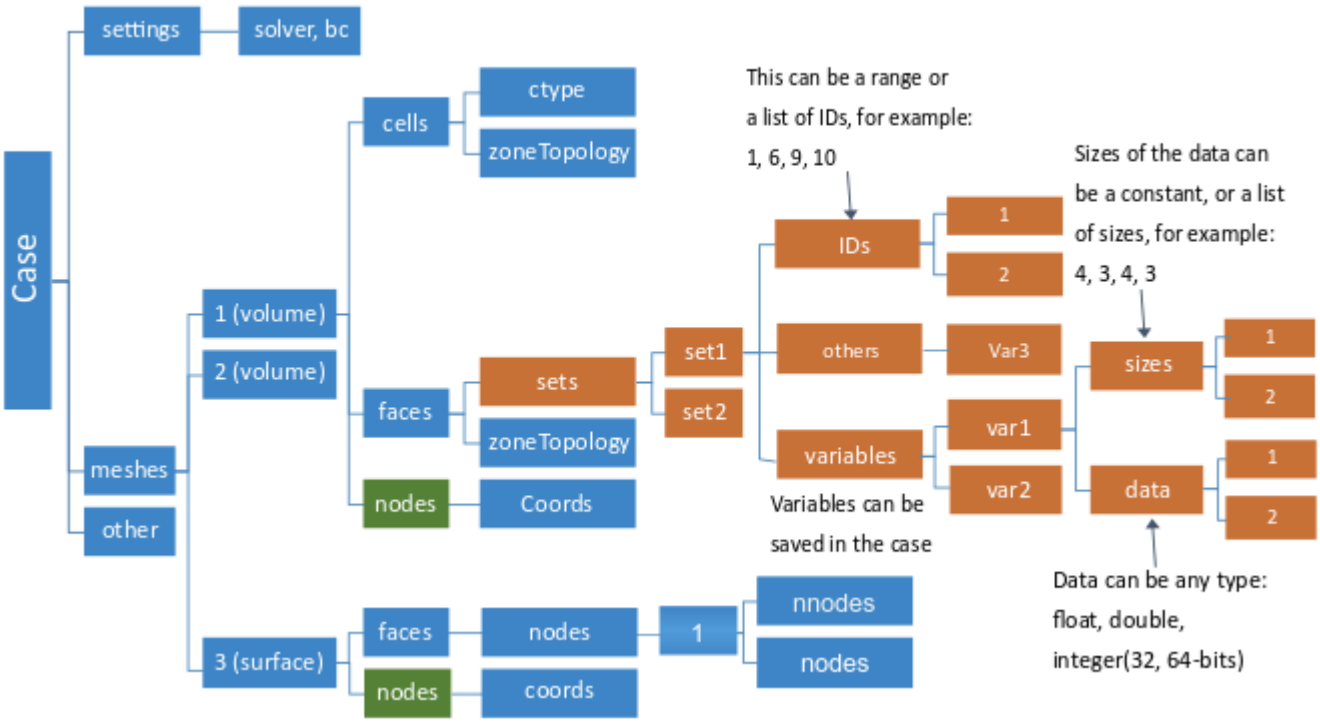
Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

B.1.3. Variable Sized Data on Collected Element / Node Sets

Other special data only store information related to specific models (such as the radiation model). They are organized in sets that hold data on a collection of cells/faces/nodes, where their IDs may not be contiguous. With these sets, any data can be saved as needed. The set data will be appended to the case or data files under corresponding zone group.

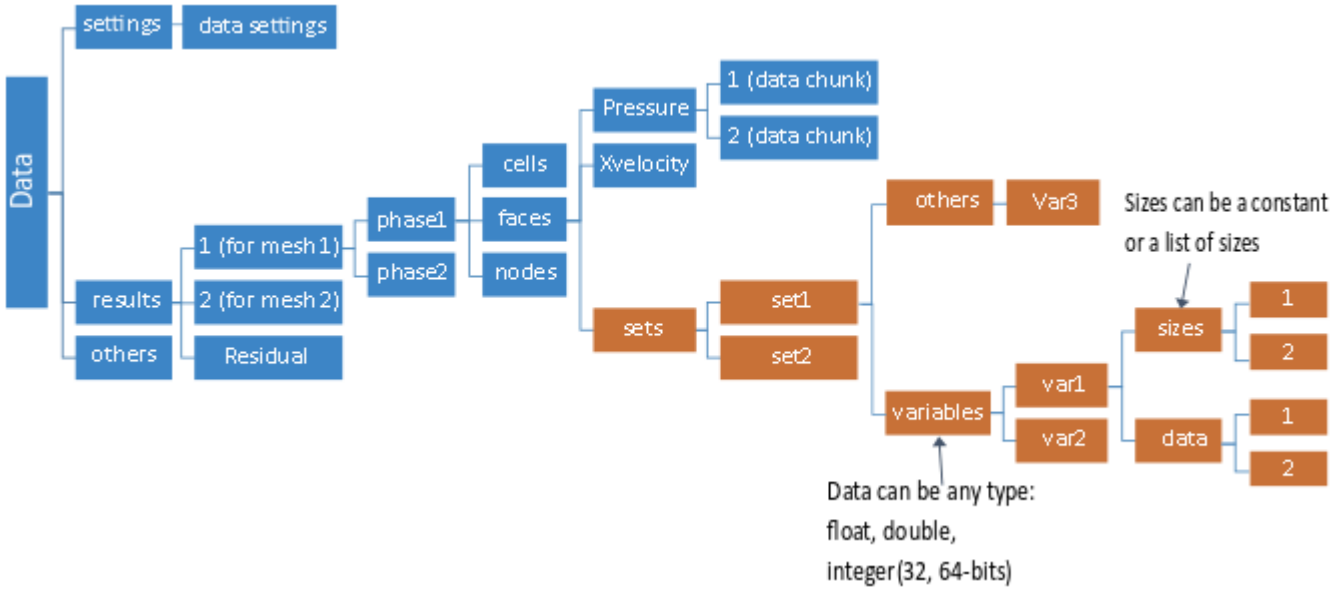
The data associated with elements can be of either a constant or variable length. For the variable length data, the sizes and data are saved in two separate 1D arrays (see Figure 3: Variable Sized Data Appended to the Case File (p. 5674) and Figure 4: Variable Sized Data Appended to the Data File (p. 5675)).

Figure 3: Variable Sized Data Appended to the Case File



Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 4: Variable Sized Data Appended to the Data File



B.2. Legacy Case and Data File Formats

This section describes the contents and formats of Ansys Fluent legacy case and data files. After discussing the Guidelines (p. 5675) and Formatting Conventions in Binary and Formatted Files (p. 5676), the section descriptions are grouped according to function:

- Grid Sections (p. 5676) : Creating grids for Ansys Fluent.
- Other (Non-Grid) Legacy Case Sections (p. 5688)
- Data Sections (p. 5691) : Importing solutions into another postprocessor.

The legacy case and data files may contain other sections that are intended for internal use only.

B.2.1. Guidelines

The Ansys Fluent legacy case and data files are broken into several sections according to the following guidelines:

- Each section is enclosed in parentheses and begins with a decimal integer indicating its type. This integer is different for formatted and binary files (Formatting Conventions in Binary and Formatted Files (p. 5676)).
- All groups of items are enclosed in parentheses. This makes skipping to ends of (sub)sections and parsing them very easy. It also allows for easy and compatible addition of new items in future releases.
- Header information for lists of items is enclosed in separate sets of parentheses preceding the items, and the items are enclosed in their own parentheses.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

B.2.2. Formatting Conventions in Binary and Formatted Files

For formatted files, examples of file sections are given in Grid Sections (p. 5676) and Other (Non-Grid) Legacy Case Sections (p. 5688). For binary files, the header indices described in this section (for example, `10` for the node section) are preceded by `20` for single-precision binary data, or by `30` for double-precision binary data (for example, `2010` or `3010` instead of `10`). The end of the binary data is indicated by `End of Binary Section 2010` or `End of Binary Section 3010` before the closing parenthesis of the section.

An example with the binary data represented by periods is as follows:

```
(2010 (2 1 2aad 2 3)(
 .
 .
 .
 )
 End of Binary Section 2010)
```

B.2.3. Grid Sections

Grid sections are stored in the legacy case file. A grid file is a subset of a legacy case file, containing only those sections pertaining to the grid. The currently defined grid sections are:

- Comment (See Comment (p. 5676))
- Header (See Header (p. 5677))
- Dimensions (See Dimensions (p. 5677))
- Nodes (See Nodes (p. 5677))
- Periodic Shadow Faces (See Periodic Shadow Faces (p. 5678))
- Cells (See Cells (p. 5679))
- Faces (See Faces (p. 5681))
- Face Tree (See Face Tree (p. 5683))
- Cell Tree (See Cell Tree (p. 5683))
- Interface Face Parents (See Interface Face Parents (p. 5684))

The section ID numbers are indicated in both symbolic and numeric forms. The symbolic representations are available as symbols in a Scheme source file (`xfile.scm`), which is available from Ansys, Inc., or as macros in a C header file (`xfile.h`), which is located in your installation area.

B.2.3.1. Comment

Index:	0
Scheme symbol:	`xf-comment`
C macro:	`XF_COMMENT`
Status:	optional

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Comment sections can appear anywhere in the file (except within other sections) as:

```
(0 "comment text")
```

You should precede each long section, or group of related sections, by a comment section explaining what is to follow.

Example:

```
(0 "Variables:")
 (37 (
 (relax-mass-flow 1)
 (default-coefficient ())
 (default-method 0)
 ))
```

B.2.3.2. Header

Index:	1
Scheme symbol:	`xf-header`
C macro:	`XF_HEADER`
Status:	optional

Header sections can appear anywhere in the file (except within other sections). The following is an example:

```
(0 "fluent24.2.0 build-id: 0")
```

The purpose of this section is to identify the program that wrote the file. Although it can appear anywhere, it is one of the first sections in the file. Additional header sections indicate other programs that may have been used in generating the file. It provides a history mechanism showing where the file came from and how it was processed.

B.2.3.3. Dimensions

Index:	2
Scheme symbol:	`xf-dimension`
C macro:	`XF_DIMENSION`
Status:	optional

The dimensions of the grid appear as:

```
(2 ND)
```

where ND is 2 or 3. This section is supported as a check that the grid has the appropriate dimension.

B.2.3.4. Nodes

Index:	10
Scheme symbol:	`xf-node`

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

C macro:	XF_NODE
Status:	required

Format:

```
(10 (zone-id first-index last-index type ND)(
 x1 y1 z1
 x2 y2 z2
 .
 .
 .
 ))
```

- If zone-id is zero, this provides the total number of nodes in the grid. first-index will then be one, last-index will be the total number of nodes *in hexadecimal*, type is equal to 1, ND is the dimensionality of the grid, and there are no coordinates following (the parentheses for the coordinates are omitted as well).

  For example: (10 (0 1 2d5 1 2))

- If zone-id is greater than zero, it indicates the zone to which the nodes belong. first-index and last-index are the indices of the nodes in the zone, *in hexadecimal*. The values of last-index in each zone must be less than or equal to the value in the declaration section. **Type** is always equal to 1.

  ND is an optional argument that indicates the dimensionality of the node data, where ND is 2 or 3.

  If the number of dimensions in the grid is two, as specified by the node header, then only x and y coordinates are present on each line.

The following is an example of a 2D grid:

```
(10 (1 1 2d5 1 2)(
 1.500000e-01 2.500000e-02
 1.625000e-01 1.250000e-02
   .
   .
   .
 1.750000e-01 0.000000e+00
 2.000000e-01 2.500000e-02
 1.875000e-01 1.250000e-02
 ))
```

Because the grid connectivity is composed of integers representing pointers (see Cells and Faces), using hexadecimal conserves space in the file and provides for faster file input and output. The header indices are in hexadecimal so that they match the indices in the bodies of the grid connectivity sections. The zone-id and type are also in hexadecimal for consistency.

B.2.3.5. Periodic Shadow Faces

Index:	18
Scheme symbol:	xf-periodic-face
C macro:	XF_PERIODIC_FACE
Status:	required only for grids with periodic boundaries

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

This section indicates the pairings of periodic faces on periodic boundaries. Grids without periodic boundaries do not have sections of this type. The format of the section is as follows:

```
(18 (first-index last-index periodic-zone shadow-zone)(
 f00 f01
 f10 f11
 f20 f21
 .
 .
 .
 ))
```

where

`first-index` = index of the first periodic face pair in the list

`last-index` = index of the last periodic face pair in the list

`periodic-zone` = zone ID of the periodic face zone

`shadow-zone` = zone ID of the corresponding shadow face zone

These are in hexadecimal format. The indices in the section body (`f*`) refer to the faces on each of the periodic boundaries (in hexadecimal), the indices being offsets into the list of faces for the grid.

Note:

In this case, `first-index` and `last-index` do *not* refer to face indices. They refer to indices in the list of periodic pairs.

Example:

```
(18 (1 2b a c) (
 12 1f
 13 21
 ad 1c2
 .
 .
 .
 ))
```

B.2.3.6. Cells

Index:	12
Scheme symbol:	`xf-cell`
C macro:	`XF_CELL`
Status:	required

The declaration section for cells is similar to that for nodes.

```
(12 (zone-id first-index last-index type element-type))
```

Again, `zone-id` is zero to indicate that it is a declaration of the total number of cells. If `last-index` is zero, then there are no cells in the grid. This is useful when the file contains only a surface

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

mesh to alert Ansys Fluent that it cannot be used. In a declaration section, the `type` has a value of zero and the `element-type` is not present.

For example,

```
(12 (0 1 3e3 0))
```

It states that there are 3e3 (hexadecimal) = 995 cells in the grid. This declaration section is required and must precede the regular cell sections.

The `element-type` in a regular cell section header indicates the type of cells in the section, as follows:

`element-type`	***description***	***nodes/cell***	***faces/cell***
0	mixed		
1	triangular	3	3
2	tetrahedral	4	4
3	quadrilateral	4	4
4	hexahedral	8	6
5	pyramid	5	5
6	wedge	6	5
7	polyhedral	NN	NF

where NN and NF will vary, depending on the specific polyhedral cell.

Regular cell sections have no body, but they have a header of the same format where `first-index` and `last-index` indicate the range for the particular zone, `type` indicates whether the cell zone is an active zone (solid or fluid), or inactive zone (currently only parent cells resulting from hanging node adaption). Active zones are represented with `type`=1, while inactive zones are represented with `type`=32.

In the earlier versions of Ansys Fluent, a distinction was made between solid and fluid zones. This is now determined by properties (that is, material type).

A `type` of zero indicates a dead zone and will be skipped by Ansys Fluent. If a zone is of mixed type (`element-type`=0), it will have a body that lists the `element-type` of each cell.

Example:

```
(12 (9 1 3d 0 0)(
 1 1 1 3 3 1 1 3 1
 .
 .
 .
 ))
```

Here, there are 3D (hexadecimal) = 61 cells in cell zone 9, of which the first 3 are triangles, the next 2 are quadrilaterals, and so on.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

B.2.3.7. Faces

Index:	13
Scheme symbol:	`xf-face`
C macro:	XF_FACE
Status:	required

The format for face sections is as follows:

```
(13 (zone-id first-index last-index bc-type face-type))
```

where

`zone-id` = zone ID of the face section
`first-index` = index of the first face in the list
`last-index` = index of the last face in the list
`bc-type` = ID of the boundary condition represented by the face section
`face-type` = ID of the type(s) of face(s) in the section

The current valid boundary condition types are defined in the following table:

`bc-type`	*description*
2	interior
3	wall
4	pressure-inlet, inlet-vent, intake-fan
5	pressure-outlet, exhaust-fan, outlet-vent
7	symmetry
8	periodic-shadow
9	pressure-far-field
10	velocity-inlet
12	periodic
14	fan, porous-jump, radiator
20	mass-flow-inlet, mass-flow-outlet
24	interface
31	parent (hanging node)
36	outflow
37	axis

The faces resulting from the intersection of non-conformal grids are placed in a separate face zone, where a factor of 1000 is added to the `bc-type` (for example, 1003 is a wall zone).

The current valid face types are defined in the following table:

`face-type`	*description*	*nodes/face*

0	mixed	
2	linear	2
3	triangular	3
4	quadrilateral	4
5	polygonal	NN

where NN will vary, depending on the specific polygonal face.

A `zone-id` of zero indicates a declaration section, which provides a count of the total number of faces in the file. Such a section omits the `bc-type` and is not followed by a body with further information.

A nonzero `zone-id` indicates a regular face section, and will be followed by a body that contains information about the grid connectivity. Each line of the body will describe one face and will have the following format:

```
n0 n1 n2 c0 c1
```

where,

`n*` = defining nodes (vertices) of the face

`c*` = adjacent cells

This is the format for a 3D grid with a triangular face format. The actual number of nodes depends on the `face-type`. The order of the cell indices is important, and is determined by the right-hand rule: if you curl the fingers of your right hand in the order of the nodes, your thumb will point toward `c0`.

For 2D grids, `n2` is omitted. `c1` is determined by the cross product of two vectors, $\hat{r}$ and $\hat{k}$. The $\hat{r}$ vector extends from `n0` to `n1`, whereas the $\hat{k}$ vector has its origin at `n0` and points out of the grid plane toward the viewer. If you extend your right hand along $\hat{r}$ and curl your fingers in the direction of the angle between $\hat{r}$ and $\hat{k}$, your thumb will point along $\hat{r}\times\hat{k}$ toward `c1`.

If the face zone is of mixed type (`face-type`= 0) or of polygonal type (`face-type`= 5), each line of the section body will begin with a reference to the number of nodes that make up that particular face, and has the following format:

```
x n0 n1 ... nf c0 c1
```

where,

`x` = the number of nodes (vertices) of the face

`nf` = the final node of the face

All cells, faces, and nodes have positive indices. If a face has a cell only on one side, then either `c0` or `c1` is zero. For files containing only a surface mesh, both these values are zero.

For information on face-node connectivity for various cell types in Ansys Fluent, refer to Face-Node Connectivity in Ansys Fluent (p. 1136).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

B.2.3.8. Face Tree

Index:	59
Scheme symbol:	`xf-face-tree`
C macro:	XF_FACE_TREE
Status:	only for grids with hanging node adaption

This section indicates the face hierarchy of the grid containing hanging nodes. The format of the section is as follows:

```
 (59 (face-id0 face-id1 parent-zone-id child-zone-id)
 (
 number-of-kids kid-id-0 kid-id-1 ... kid-id-n
.
.
.
 ))
```

where,

`face-id0` = index of the first parent face in the section

`face-id1` = index of the last parent face in the section

`parent-zone-id` = ID of the zone containing parent faces

`child-zone-id` = ID of the zone containing children faces

`number-of-kids` = the number of children of the parent face

`kid-id-n` = the face IDs of the children

These are in hexadecimal format.

B.2.3.9. Cell Tree

Index:	58
Scheme symbol:	`xf-cell-tree`
C macro:	XF_CELL_TREE
Status:	only for grids with hanging node adaption

This section indicates the cell hierarchy of the grid containing hanging nodes. The format of the section is as follows:

```
 (58 (cell-id0 cell-id1 parent-zone-id child-zone-id)
 (
 number-of-kids kid-id-0 kid-id-1 ... kid-id-n
.
.
.
 ))
```

where,

`cell-id0` = index of the first parent cell in the section

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

`cell-id1` = index of the last parent cell in the section

`parent-zone-id` = ID of the zone containing parent cells

`child-zone-id` = ID of the zone containing children cells

`number-of-kids` = the number of children of the parent cell

`kid-id-n` = the cell IDs of the children

These are in hexadecimal format.

B.2.3.10. Interface Face Parents

Index:	61
Scheme symbol:	`xf-face-parents`
C macro:	`XF_FACE_PARENTS`
Status:	only for grids with non-conformal interfaces

This section indicates the relationship between the intersection faces and original faces. The intersection faces (children) are produced from intersecting two non-conformal surfaces (parents) and are some fraction of the original face. Each child will refer to at least one parent. The format of the section is as follows:

```
 (61 (face-id0 face-id1)
 (
 parent-id-0 parent-id-1
.
.
.
 ))
```

where,

`face-id0` = index of the first child face in the section

`face-id1` = index of the last child face in the section

`parent-id-*` = index of parent faces

These are in hexadecimal format.

If you set up and save a non-conformal mesh in the solution mode of Fluent and then read it using the meshing mode of Fluent, this section will be skipped; consequently, all the information necessary to preserve the non-conformal interface will not be maintained. When you switch to or read the mesh back into the solution mode, you will need to recreate the interface.

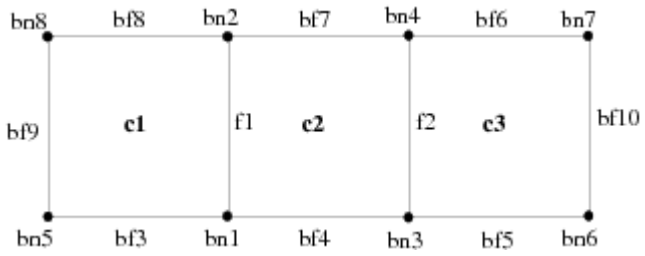
B.2.3.11. Example Files

B.2.3.11.1. Example 1

Figure 5: Quadrilateral Mesh (p. 5685) illustrates a simple quadrilateral mesh with no periodic boundaries or hanging nodes.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 5: Quadrilateral Mesh



The following describes this mesh:

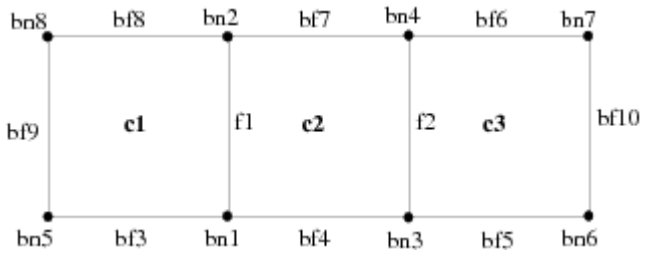
```
(0 "Grid:")

(0 "Dimensions:")
(2 2)

(12 (0 1 3 0))
(13 (0 1 a 0))
(10 (0 1 8 0 2))

(12 (7 1 3 1 3))

(13 (2 1 2 2 2)(
1 2 1 2 3 4 2 3))

(13 (3 3 5 3 2)(
5 1 1 0 1 3 2 0
3 6 3 0))

(13 (4 6 8 3 2)(
7 4 3 0 4 2 2 0
2 8 1 0))

(13 (5 9 9 a 2)(
8 5 1 0))

(13 (6 a a 24 2)(
6 7 3 0))

(10 (1 1 8 1 2)
(
1.00000000e+00 0.00000000e+00
1.00000000e+00 1.00000000e+00
2.00000000e+00 0.00000000e+00
2.00000000e+00 1.00000000e+00
0.00000000e+00 0.00000000e+00
3.00000000e+00 0.00000000e+00
3.00000000e+00 1.00000000e+00
0.00000000e+00 1.00000000e+00))
```

B.2.3.11.2. Example 2

Figure 6: Quadrilateral Mesh with Periodic Boundaries (p. 5686) illustrates a simple quadrilateral mesh with periodic boundaries but no hanging nodes. In this example, bf9 and bf10 are faces on the periodic zones.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 6: Quadrilateral Mesh with Periodic Boundaries



The following describes this mesh:

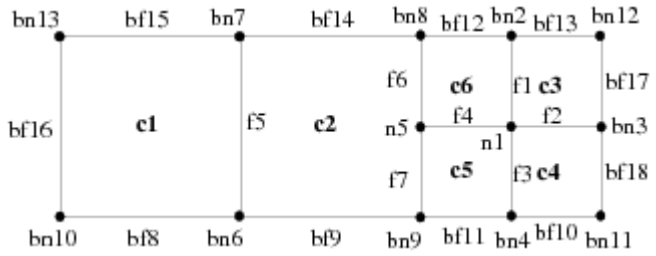
```
(0 "Dimensions:")
(2 2)

(0 "Grid:")

(12 (0 1 3 0))
(13 (0 1 a 0))
(10 (0 1 8 0 2))

(12 (7 1 3 1 3))

(13 (2 1 2 2 2)(
1 2 1 2 3 4 2 3))

(13 (3 3 5 3 2)(
5 1 1 0 1 3 2 0
3 6 3 0))

(13 (4 6 8 3 2)(
7 4 3 0 4 2 2 0
2 8 1 0))

(13 (5 9 9 c 2)(
8 5 1 0))

(13 (1 a a 8 2)(
6 7 3 0))

(18 (1 1 5 1)(
9 a))

(10 (1 1 8 1 2)(
1.00000000e+00 0.00000000e+00
1.00000000e+00 1.00000000e+00
2.00000000e+00 0.00000000e+00
2.00000000e+00 1.00000000e+00
0.00000000e+00 0.00000000e+00
3.00000000e+00 0.00000000e+00
3.00000000e+00 1.00000000e+00
0.00000000e+00 1.00000000e+00))
```

B.2.3.11.3. Example 3

Figure 7: Quadrilateral Mesh with Hanging Nodes (p. 5687) illustrates a simple quadrilateral mesh with hanging nodes.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Figure 7: Quadrilateral Mesh with Hanging Nodes



The following describes this mesh:

```
(0 "Grid:")

(0 "Dimensions:")
(2 2)

(12 (0 1 7 0))
(13 (0 1 16 0))
(10 (0 1 d 0 2))

(12 (7 1 6 1 3))
(12 (1 7 7 20 3))

(58 (7 7 1 7)(
4 6 5 4 3))

(13 (2 1 7 2 2)(
1 2 6 3
1 3 3 4
1 4 4 5
1 5 5 6
6 7 1 2
5 8 2 6
9 5 2 5))

(13 (3 8 b 3 2)(
a 6 1 0
6 9 2 0  4 b 4 0
9 4 5 0))

(13 (4 c f 3 2)(
2 8 6 0
c 2 3 0
8 7 2 0
7 d 1 0))

(13 (5 10 10 a 2)(
d a 1 0))

(13 (6 11 12 24 2)(
3 c 3 0
b 3 4 0))

(13 (b 13 13 1f 2)(
c 8 7 0))

(13 (a 14 14 1f 2)(
b c 7 0))

(13 (9 15 15 1f 2)(
9 b 7 0))

(13 (8 16 16 1f 2)(
9 8 2 7))

(59 (13 13 b 4)(
```

```
2 d c))

(59 (14 14 a 6)(
2 12 11))

(59 (15 15 9 3)(
2 b a))

(59 (16 16 8 2)(
2 7 6))

(10 (1 1 d 1 2)
(
2.50000000e+00 5.00000000e-01
2.50000000e+00 1.00000000e+00
3.00000000e+00 5.00000000e-01
2.50000000e+00 0.00000000e+00
2.00000000e+00 5.00000000e-01
1.00000000e+00 0.00000000e+00
1.00000000e+00 1.00000000e+00
2.00000000e+00 1.00000000e+00
2.00000000e+00 0.00000000e+00
0.00000000e+00 0.00000000e+00
3.00000000e+00 0.00000000e+00
3.00000000e+00 1.00000000e+00
0.00000000e+00 1.00000000e+00))
```

B.2.4. Other (Non-Grid) Legacy Case Sections

The following sections store boundary conditions, material properties, and solver control settings.

B.2.4.1. Zone

B.2.4.2. Partitions

B.2.4.1. Zone

Index:	39 or 45
Scheme symbol:	`xf-rp-tv`
C macro:	XF_RP_TV
Status:	required

There is typically one zone section for each zone referenced by the grid. Although some grid zones may not have corresponding zone sections, there cannot be more than one zone section for each zone.

A zone section has the following form:

```
(39 (zone-id zone-type zone-name domain-id)(
(condition1 . value1)
(condition2 . value2)
(condition3 . value3)
.
.
.
 ))
```

Grid generators and other preprocessors need only provide the section header and leave the list of conditions empty, as in

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
(39 (zone-id zone-type zone-name domain-id)())
```

The empty parentheses at the end are required. The solver adds conditions as appropriate, depending on the zone type. When only `zone-id`, `zone-type`, `zone-name`, and `domain-id` are specified, the index 45 is preferred for a zone section. However, the index 39 must be used if boundary conditions are present, because any and all remaining information in a section of index 45 after `zone-id`, `zone-type`, `zone-name`, and `domain-id` will be ignored.

Here the `zone-id` is in *decimal* format. This is in contrast to the use of hexadecimal in the grid sections.

The `zone-type` is one of the following:

```
axis
exhaust fan
fan
fluid
inlet vent
intake fan
interface
interior
mass-flow-inlet
mass-flow-outlet
outlet vent
outflow
periodic
porous-jump
pressure-far-field
pressure-inlet
pressure-outlet
radiator
shadow
solid
symmetry
velocity-inlet
wall
```

The `interior`, `fan`, `porous-jump`, and `radiator` types can be assigned only to zones of faces inside the domain. The `interior` type is used for the faces within a cell zone; the others are for interior faces that form infinitely thin surfaces within the domain. Ansys Fluent allows the `wall` type to be assigned to face zones both on the inside and on the boundaries of the domain. Some zone types are valid only for certain types of grid components. For example, cell (element) zones can be assigned only one of the following types:

```
fluid
 solid
```

All of the other types listed above can be used only for boundary (face) zones.

The `zone-name` is a user-specified label for the zone. It must be a valid Scheme symbol [1] and is written without quotes. The rules for a valid `zone-name` (Scheme symbol) are as follows:

- The first character must be a lowercase letter [2] or a special-initial.

[1] See Revised (4) Report on the Algorithmic Language Scheme, William Clinger and Jonathan Rees (Editors), 2 November 1991, Section 7.1.1.

[2] The Standard actually only requires that case be insignificant; the Ansys Fluent implementation accomplishes this by converting all uppercase input to lowercase.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- Each subsequent character must be a lowercase letter, a special-initial, a digit, or a special-subsequent.

where a special-initial character is one of the following:

```
! $ % & * / : < = > ? ~ _ ^
```

and a special-subsequent is one of the following:

```
. + -
```

Examples of valid zone names are `inlet-port/cold!—`, `eggs/easy`, and `e=m*cˆ2`.

Some examples of zone sections produced by grid generators and preprocessors are as follows:

```
(39 (1 fluid fuel 1)())
 (39 (8 pressure-inlet pressure-inlet-8 2)())
 (39 (2 wall wing-skin 3)())
 (39 (3 symmetry mid-plane 1)())
```

The `domain-id` is an integer that appears after the zone name, associating the boundary condition with a particular phase or mixture (sometimes referred to as phase-domains and mixture-domains).

B.2.4.2. Partitions

Index:	40
Scheme symbol:	`xf-partition`
C macro:	XF_PARTITION
Status:	only for partitioned grids

This section indicates each cell's partition. The format of the section is as follows:

```
(40 (zone-id first-index last-index partition-count)(
p1
p2
p3
.
.
.
pn
))
```

where,

`p1` = the partition of the cell whose ID is `first-index`

`p2` = the partition of the cell whose ID is `first-index` +1, and so on

`pn` = the partition of the cell whose ID is `last-index`

`partition-count` = the total number of partitions

Partition IDs must be between 0 and one less than `partition-count`.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

B.2.5. Data Sections

The following sections store iterations, residuals, and data field values.

B.2.5.1. Grid Size
B.2.5.2. Data Field
B.2.5.3. Residuals

B.2.5.1. Grid Size

Index:	33
Scheme symbol:	`xf-grid-size`
C macro:	`XF_GRID_SIZE`
Status:	optional

This section indicates the number of cells, faces, and nodes in the grid that corresponds to the data in the file. This information is used to check that the data and grid match. The format is

```
(33 (n-elements n-faces n-nodes))
```

where the integers are written in decimal.

B.2.5.2. Data Field

Index:	300
Scheme symbol:	`xf-rf-seg-data`
C macro:	`XF_RF_SEG_DATA`
Status:	required

This section lists a flow field solution variable for a cell or face zone. The data are stored in the same order as the cells or faces in the legacy case file. Separate sections are written out for each variable for each face or cell zone on which the variable is stored. The format is

```
(300 (sub-section-id zone-id size n-time-levels
 n-phases first-id last-id)
 ( data for cell or face with id = first-id
 data-for-cell-or-face with id = first-id+1
 ..
 data-for-cell-or-face with id = last-id
 ))
```

where `sub-section-id` is a (decimal) integer that identifies the variable field (for example, 1 for pressure, 2 for velocity). The complete list of these is available in the header file (`xfile.h`), which is located in your installation area.

where,

`zone-id` = the ID number of the cell or face zone

`size` = the length of the variable vector

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

zone-id matches the ID used in legacy case file. size is 1 for a scalar, 2 or 3 for a vector, equal to the number of species for variables defined for each species). n-time-levels currently are not used.

A sample legacy data file section for the velocity field in a cell zone for a steady-state, single-phase, 2D problem is shown below:

```
(300 (2 16 2 0 0 17 100)
(8.08462024e-01 8.11823010e-02
 8.78750622e-01 3.15509699e-02
 1.06139672e+00 -3.74040119e-02
...
 1.33301604e+00 -5.04243895e-02
 6.21703446e-01 -2.46118382e-02
 4.41687912e-01 -1.27046436e-01
 1.03528820e-01 -1.01711005e-01
 ))
```

The variables that are listed in the legacy data file depend on the models active at the time the file is written. Variables that are required by the solver based on the current model settings but are missing from the legacy data file are set to their default values when the legacy data file is read. Any extra variables that are present in the legacy data file but are not relevant according to current model settings are ignored.

B.2.5.3. Residuals

Index:	302
Scheme symbol:	xf-rf-scaled-residuals
C macro:	XF_RF_SCALED_RESIDUALS
Status:	optional

This section lists the values of the residuals for a particular data field variable at each iteration:

```
(302 (n residual-section-id size domain-id)
(
iteration_number unscaled_residual scaling_factor
 .
 .
 .
 ))
```

where,

n = the number of residuals

size = the length of the variable vector

residual-section-id = an integer (decimal) indicating the equation

domain-id = domain ID

size is 1 for a scalar, 2 or 3 for a vector, equal to the number of species for variables defined for each species. The residual-section-id indicates the equation for which the residual is stored in the section, according to the C constants defined in a header file (xfile.h) available in your installation area, as noted in Grid Sections (p. 5676).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

The equations for which residuals are listed in the legacy data file depend on the models active at the time the file is written. If the residual history is missing from the legacy data file for a currently active equation, it is initialized with zeros.

B.3. Mesh Morpher/Optimizer File Formats

This section describes the format of the ASCII text files that can be used as part of the setup for the mesh morpher/optimizer, as described in Using the Mesh Morpher/Optimizer (p. 4620).

The following is an example of a file that can be read or written from the Define Control Points Dialog Box (p. 5432):

```
Control Point Positions
CP No.        X            Y             Z
1             0.73398191   -0.048181638  5.0344162
2             0.51003051   0.52833155    5.100265
3             0.10806149   0.72560567    5.1306067
4             -0.37284759  0.62890923    5.1368165
5             -0.69803506  0.21491928    5.144598
```

The following is an example of a file that can be read or written from the Motion Settings Dialog Box (p. 5543) when **Regular** is selected from the **Control Point Distribution** list in the **Regions** tab of the Mesh Morpher/Optimizer Dialog Box (p. 5528):

```
Reg     CP      Par      X       Y       Z
def-0   2       par1     -1      0       0
def-0   4       par1     2       -1      3
def-0   2       par2     0       1       0
def-1   2       par3     0       0       1
def-1   3       par1     2       0       2
```

The following is an example of a file that can be read or written from the Motion Settings Dialog Box (p. 5543) when **Unstructured** is selected from the **Control Point Distribution** list in the **Regions** tab of the Mesh Morpher/Optimizer Dialog Box (p. 5528):

```
Motion Definition
Name        Type        Par         Dir-x       Dir-y       Dir-z       Orig-x      Orig-y      Orig-z
rad-par1-1  Rad         par-1       0           0           1           0           0           0
tra-par2-1  Trans       par-2       1           1           1
rot-par3-1  Rot         par-3       1           0           0           2           1           1

Assignment to Control Points
Name        Control Points...
rad-par1-1  1           2           3           4           5           6           7
```

When creating or editing a text file used to set up the mesh morpher/optimizer, note the following:

- The file can be created from nothing using a text editor or spreadsheet program, as long as it adheres to the formats described in this section. An easier alternative is to apply some sample definitions using the appropriate dialog box, write a file, and then edit the file using a text editor or spreadsheet program.
- The file must be a tab-delimited ASCII text file.
- The names of the deformation regions (for example, `def-0`), deformation parameters (for example, `par1`), and motions (for example, `tra-par2-1`) must correspond to those that exist in the intended case file.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

- When defining control points, the coordinates must lie within a deformation region in the intended case file.
- For the motion settings, the control point numbers must range between 1 and the maximum number of available control points in the intended case file.
- The direction components and coordinates can be any real numbers.

B.4. Conduction Settings File Format

This section describes the format of the `.csv` files that can define conduction settings. The following is an example of such a file:

```
Zone ID,Zone Name,Enable-Shell? (1/0),Layer Number,Material,Layer Thickness (m),Heat Generation Rate (w/m3)
6,wall-1,1,1,aluminum,0.02,0
6,wall-1,1,2,aluminum,0.02,0
7,wall-2,1,1,aluminum,0.05,0
10,wall-6,1,1,aluminum,0.02,0
10,wall-6,1,2,aluminum,0.02,0
8,wall-4,0,1,aluminum,0.02,0
9,wall-5,0,1,aluminum,0,0
```

When creating or editing a file that defines the conduction settings, note the following:

- The file can be created from nothing using a text editor or spreadsheet program, as long as it adheres to the format described in this section. An easier alternative is to apply some sample conduction definitions using the Conduction Manager Dialog Box (p. 5411), generate a file using the **Write...** button, and then edit the file using a text editor or spreadsheet program.
- The file must be a comma-delimited ASCII file.
- The first line contains the column headings shown in the previous example, and then each of the subsequent lines define the settings for a conduction zone/shell layer.
- The zone IDs, zone names, layer numbers, and materials must correspond to those that exist in the intended case file.
- The layer thicknesses must be positive, nonzero values.
- Information will not be written for any wall that defines the heat generation rate using a UDF.
- Fluent always writes conduction files in SI units. If you are using non-SI units, Fluent will do the necessary conversions when the file in re-read.

B.5. 3D Fan Curve File Format

This section describes the format of the ASCII text files that can define the relationship between pressure and flow rate for all of the 3D fan cell zones in a case file, as described in 3D Fan Zones (p. 1372). The following is an example of such a file:

```
cell zone id
17 !CELL_ID
Pressure(Pa) Flowrate(m3/s)
-324.0 1.255
-215.0 1.162
```


Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

```
-109.0 1.032
0.0 0.997
63.0 0.913
105.0 0.862
302.0 0.584
394.0 0.492
406.0 0.432
443.0 0.365
cell zone id
18 !CELL_ID
Pressure(Pa) Flowrate(m3/s)
-333.0 1.130
-215.0 1.122
-165.0 1.064
0.0 0.981
42.0 0.912
90.0 0.855
112.0 0.786
303.0 0.546
344.0 0.492
395.0 0.451
411.0 0.399
442.0 0.324
```

When creating or editing a file that defines the 3D fan zone curve, note the following:

- The file can be created from nothing using a text editor or spreadsheet program, as long as it adheres to the format described in this section.
- The file must be a tab-delimited ASCII file.
- A single file can define multiple cell zones, but it must be read in each of the applicable **Fluid** dialog boxes.
- For each section that defines the curve for a cell zone, the first three lines must follow the convention shown in the previous example, and then each of the subsequent lines must list the experimental results.
- The zone IDs (for example, 12) must correspond to those that exist in the intended case file.
- The units must match those in the case file.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Appendix C. Controlling CHEMKIN-CFD Solver Parameters Using Text Commands

You have access to many parameter settings that can be used to control or improve the performance and convergence behavior of the CHEMKIN-CFD solver within Ansys Fluent. These parameters are accessed through the Ansys Fluent Text User Interface (TUI), through typed text commands. This appendix describes the available parameters, their default values, and their recommended use.

In general, default settings for the CHEMKIN-CFD parameters should be sufficient for most problems. However, it may be necessary to change these settings to improve performance for specific types of problems. In particular, the most appropriate solver parameters may vary depending on the nature of the chemistry set.

For example, for chemistries with many trace species that have a large impact on solution results (that is, the chemistry is "stiff"), solution tolerances should be relatively small (for example, 1.E-9 to 1.E-12 for absolute tolerances and 1.E-4 to 1.E-5 for relative tolerances). For chemistry descriptions where all of the species included are within a few orders of magnitude in terms of mass fraction, however, much looser tolerances are more appropriate. Also, depending on the characteristic time scales of the dominant chemical reactions, you may need to adjust the maximum time-step used by the CHEMKIN-CFD solver.

```
/define/models/species/CHEMKIN-CFD-parameters> basic-options
Print Level [0]
Absolute Tolerance For Gas Species [1e-08]
Relative Tolerance For Gas Species [0.0001]
Absolute Tolerance For Surface Species [1e-08]
Relative Tolerance For Surface Species [0.0001]


/define/models/species/CHEMKIN-CFD-parameters> advanced-options
Pseudo Time Stepping Solver Option (0:Basic   1:Robust) [1]
Pseudo Time Stepping Size [0.0001]
Maximum Pseudo Time Step Attempts [1]
Maximum Time Step Allowed [0.01]
Minimum Time Step Allowed [1e-10]
Initial Time Step For Solver [0]
Upper Bound For Gas Species Mass Fractions [1.01]
Lower Bound For Gas Species Mass Fractions [-1e-12]
Upper Bound For Surface Species Mass Fractions [1.1]
Lower Bound For Surface Species Mass Fractions [-0.0001]
Small Positive Value To Replace Negative Mass Fractions [1e-12]
Enter Auxiliary Input Filename [""]
Enter Arbitrary String [""]
Enter Arbitrary Value [0]
```

Table 1: Summary of Basic CHEMKIN-CFD Parameters (p. 5698) and Table 2: Summary of Advanced CHEMKIN-CFD Parameters (p. 5699) summarize basic and advanced parameters, respectively. These tables include

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

descriptions of the default values and physical units, where applicable, for each of the CHEMKIN-CFD parameters, as well as a discussion of the recommended parameter usage.

Table 1: Summary of Basic CHEMKIN-CFD Parameters

Parameter	Usage	Default Value
`Print Level`	Controls the level of diagnostic printing to the CHEMKIN-CFD diagnostic output file, where 0 indicates no printing, 1 indicates standard printing, and 2 enables a verbose level of printing that provides details about the solver operations. Note that diagnostic output can significantly contribute to compute time such that high levels of printing should be avoided unless problems are encountered that require diagnosis.	0
`Absolute Tolerance for Gas-phase Species`	Controls the smallest value of a species mass fraction that will be tracked by the CHEMKIN-CFD solver convergence algorithm. This should represent either the smallest mass fraction that is of importance in the problem or, for stiff chemistry, the smallest fraction pertaining to a species that can have a significant effect through chemical reaction on other species of interest. Typical values range from 1.E-6 to 1.E-12.	1.E-8
`Relative Tolerance for Gas-phase Species`	Controls the accuracy to which gas species mass fractions are resolved. Roughly this parameter controls the number of significant digits in the solution. However, this parameter can also be important for convergence of problems with stiff chemistry, as it is important to sufficiently resolve species that have a major impact on trace species of interest. Typical values range from 1.E-3 to 1.E-5.	1.E-4
`Absolute Tolerance for Surface Species`	Controls the smallest value of a surface species site fraction that will be tracked by the CHEMKIN-CFD solver convergence algorithm. This should represent either the smallest mass fraction that is of importance in the problem or, for stiff chemistry, the smallest fraction pertaining to a species that can have a significant effect through chemical reaction on other species of interest. Typical values range from 1.E-6 to 1.E-15.	1.E-8

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parameter	Usage	Default Value
Relative Tolerance for Surface Species	Controls the accuracy to which surface species site fractions are resolved. Roughly this parameter controls the number of significant digits in the solution. However, this parameter can also be important for convergence of problems with stiff chemistry, as it is important to sufficiently resolve species that have a major impact on trace species of interest. Typical values range from 1.E-3 to 1.E-5.	1.E-4

Table 2: Summary of Advanced CHEMKIN-CFD Parameters

Parameter	Usage	Default Value
Pseudo Time-stepping Solver Option	Determines which solver to use during pseudo time-stepping used to improve the initial guess in a cell for steady-state simulations. By default, it is set to 1 and a robust solver with strict error control will be used when the steady-state algorithm requires time-stepping assistance. You may optionally revert to a less accurate time-stepping technique, which may be computationally faster but more prone to failure, by setting it to zero. See Advanced Parameters Used in the Steady-State Solution Algorithm (p. 5701) for more information. This parameter is ignored for transient simulations.	1
Pseudo Time-stepping Size	This option is only relevant if the Pseudo Time-stepping Solver Option is set to 1 (the default setting). This controls the time interval for time-stepping when time-stepping is used to assist a steady-state calculation. See Advanced Parameters Used in the Steady-State Solution Algorithm (p. 5701) for more information. A value of 1.E-4 seconds is recommended for combustion problems.	1.E-6 [seconds]
Maximum Pseudo Time-stepping Attempts	This parameter controls how many iterations between time-stepping and steady-state solution attempts can be tried before the solver gives up and registers a failure for the cell. Usually only one attempt is needed, but the maximum can be increased from this default value in difficult cases. Note that in some cases it is most efficient to ignore a small number of cell failures early on in the simulation as long as these cells recover on subsequent iterations. See Advanced Parameters Used in the Steady-State Solution Algorithm (p. 5701) for more information. A value of 10 to 100 is recommended to minimize error messages.	1
Maximum Time Step Allowed	Maximum pseudo time-step value for time integration during steady-state solution. This value is relevant to steady-state simulations only.	1.E-2 [seconds]

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Parameter	Usage	Default Value
`Minimum Time Step Allowed`	Minimum pseudo time-step value for time integration during steady-state solution. This value is relevant to steady-state simulations only.	1.E-10 [seconds]
`Initial Time Step for Solver`	Initial time-step value for use in time integration. By default, the solver will automatically choose an appropriate initial time step, which is the recommended approach. In rare cases, however, control of initial time step value may improve convergence behavior.	0 (that is, not specified) [seconds]
`Upper Bound for Gas-species Mass Fraction`	Sets the upper bounds for the gas-phase species mass fractions during steady-state solution. In some cases, it may help convergence to allow species to go slightly over the physical bounds during interim steady-state iterations rather than forcing the bounds to be one.	1.01
`Lower Bound for Gas-species Mass Fraction`	Sets the lower bounds for the gas-phase species mass fractions during steady-state solution. In some cases, it may help convergence to allow species to go slightly negative during interim steady-state iterations rather than forcing the bounds to be zero. A value as negative as about -1.E-4 may be used.	-1.E-12
`Upper Bound for Surface Site Fraction`	Sets the upper bounds for the surface species site fractions during steady-state solution. In some cases, it may help convergence to allow species to go slightly over the physical bounds during interim steady-state iterations rather than forcing the bounds to be one.	1.1
`Lower Bound for Surface Site Fraction`	Sets the lower bounds for the surface species site fractions during steady-state solution. In some cases, it may help convergence to allow species to go slightly negative during interim steady-state iterations rather than forcing the bounds to be zero.	-1.E-4
`Small Positive Value to Replace Negative Species Fractions`	This value will be used to substitute any non-positive species fractions during interim steady-state iterations. In some cases, setting this to a very small positive number may aid convergence for the steady-state solver.	1.E-12
`Enter Auxiliary Input Filename`	Currently not used. Provided for future use.	""
`Enter Arbitrary String`	Currently not used. Provided for future use.	""
`Enter Arbitrary Value`	Currently not used. Provided for future use.	0

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

C.1. Advanced Parameters Used in the Steady-State Solution Algorithm

Many of the parameters listed in Table 2: Summary of Advanced CHEMKIN-CFD Parameters (p. 5699) are relevant only for steady-state solution. Here a brief description of the steady-state solution algorithm is provided in order to clarify the role of these parameters and their impact on solution convergence.

For steady-state problems, the Ansys CHEMKIN-CFD solver is called by Ansys Fluent for each cell during each Fluent iteration towards a steady state. Within each cell, the chemistry solver will attempt to determine an appropriate steady-state using a combination of a modified Newton-iteration procedure and a "pseudo" time-stepping procedure. The Newton-iteration method is used to attempt to directly iterate towards the desired solution. However, at early times in the simulation, this procedure may not succeed, especially when the initial guess is far from the actual steady solution. In such cases, the CHEMKIN-CFD module will automatically switch to a time-stepping mode to advance the initial guess closer to the physical solution. After taking a certain number of time steps, the Newton-iteration algorithm tries again. In this way, the steady-state solution is achieved with a minimum amount of time-stepping. Since time-stepping is computationally expensive, this results in a computationally efficient but robust method for achieving the steady-state solution.

You can control the number of times the CHEMKIN-CFD module is allowed to use time-stepping to advance a solution towards steady-state before reporting a failure for any individual cell (see `Maximum Pseudo Time-stepping Attempts` in Table 2: Summary of Advanced CHEMKIN-CFD Parameters (p. 5699)).

In performing the pseudo time-stepping procedure, there are two solver options - a "basic" time-stepping procedure and a "robust" time-stepping procedure (see `Pseudo Time-stepping Solver Option` in Table 2: Summary of Advanced CHEMKIN-CFD Parameters (p. 5699)). The "robust" procedure (default) uses an integration method that is very accurate with error control enforced on all solution variables. This can sometimes be more time consuming but is generally more reliable in tracking transient states towards an eventual steady condition. The "basic" solver option is much more approximate in the time-tracking technique, but is often faster and sufficiently robust for some cases.

When the "robust" time-stepping procedure is selected, you can specify the time interval used for each attempt to advance the solution using time integration (See `Pseudo Time-stepping Size` in Table 2: Summary of Advanced CHEMKIN-CFD Parameters (p. 5699)). The actual time steps used during the time integration will be automatically determined based on the rate of change of the solution variables. The time interval, however, controls the total time advanced before returning the solution to be used as the next initial guess in the Newton-iteration.method.

In addition to the pseudo time-stepping controls, you may also control certain parameters that affect the convergence behavior of the Newton-iteration algorithm itself. In particular, you can modify the solution bounds imposed during the iterations (see `Upper Bounds and Lower Bounds parameters` in Table 2: Summary of Advanced CHEMKIN-CFD Parameters (p. 5699)). By extending the bounds slightly outside of the physical limits, the Newton iteration can more easily resolve species fractions that are very near to those limits. This can be especially helpful in early iterations where variable values may be changing rapidly. It is also helpful to try to "reset" any negative species fractions to very small values between iterations, so that these values do not have unphysical effects on reaction rates. For this purpose, you can specify a small positive value, which should not exceed the specified absolute tolerance, as a replacement value for negative fractions (see `Small Positive Value to Replace Negative Species Fractions` in Table 2: Summary of Advanced CHEMKIN-CFD Parameters (p. 5699)).

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

C.2. Setting Up Monitor Cells for the Ansys CHEMKIN-CFD Chemistry Solver

At the same level of the basic-options and advanced-options in the Text User Interface, that is, under `define/models/species/CHEMKIN-CFD-parameters`, you can add, delete, and list `monitor cells`.

By specifying a `monitor cells` in the CHEMKIN-CFD options, you are requesting additional printout diagnostics for every iteration or time-step at this cell location. Such printout can add significantly to computational times and should be avoided unless there is a problem has been encountered and more diagnosis is needed.

Important:

Using monitor cells may add significantly to the job run time. It is recommended that monitor cells only be included if extra diagnosis is required to resolve a convergence problem or local mesh issue, for example.

Monitor cells are added using the `add-cell-monitor` option and providing the physical coordinates that correspond as closely as possible to the centroid location of the desired cell in the Fluent mesh. You can create more than one cell monitor and you can be delete all of them using the `delete-cell-monitor option`. You may also list the currently activated monitor cells using the `list-cell-monitor option`.

C.3. Diagnostic Files and Error Messages

In addition to the standard Ansys Fluent files produced during a run using the CHEMKIN-CFD solver, the CHEMKIN-CFD module creates special files to hold diagnostic output and error messages that may be helpful in diagnosing a problem with the pre-processing or run steps. In addition, error messages may be written in the Ansys Fluent Graphical User Interface window during run-time processing.

Table 3: Diagnostic Output Files Created During a CHEMKIN-CFD Run (p. 5702) lists additional files that may be created by the chemistry solver, the data printed to them, and an indication of where they are created in the Ansys Fluent run process. All files are created within the working directory of the simulation run. The actual names of many of the files will depend on variables, such as the chemistry-set file names, the case-file name, the cell number, iteration or time-step number, and the processor id. Such variables are indicated by < > in Table 3: Diagnostic Output Files Created During a CHEMKIN-CFD Run (p. 5702).

Table 3: Diagnostic Output Files Created During a CHEMKIN-CFD Run

Filename	When File Is Created	Contents Description
<gas_chemistry>.out	Always created during pre-processing, upon saving the CHEMKIN mechanism definition.	This file echoes back the gas-phase chemistry reaction set, species names, and elements in formatted text. In addition, detailed error messages are contained in this file in case an error is encountered while processing the gas-phase chemistry input file and gas-phase thermodynamic data.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Filename	When File Is Created	Contents Description
`chem.asc`	Always created during pre-processing, upon saving the CHEMKIN mechanism definition.	A "Linking File" used to transfer the pre-processed information to the CHEMKIN-CFD module at run-time. This file is typically not viewed by the user, but may occasionally be requested by Technical Support during problem diagnosis.
`<surface_chemistry>.out`	Created during pre-processing, only if surface chemistry box is checked in CHEMKIN mechanism definition.	This file echoes back the surface chemistry reaction set and the surface phase and species names. In addition, detailed error messages are contained in this file in case an error is encountered while processing the surface chemistry input file and surface-species thermodynamic data.
`surf.asc`	Created during pre-processing, only if surface chemistry box is checked in CHEMKIN mechanism definition.	A "Linking File" used to transfer the pre-processed information to the CHEMKIN-CFD module at run-time. This file is typically not viewed by the user, but may occasionally be requested by Technical Support during problem diagnosis.
`<transport_data>.out`	Created during pre-processing, only if transport-property box is checked in CHEMKIN mechanism definition.	This file echoes back the transport-data fitting coefficients in formatted text. In addition, detailed error messages are contained in this file in case an error is encountered while processing the gas-phase transport data.
tran.asc	Created during pre-processing, only if transport-property box is checked in CHEMKIN mechanism definition.	A "Linking File" used to transfer the pre-processed information to the CHEMKIN-CFD module at run-time. This file is typically not viewed by the user, but may occasionally be requested by Ansys Technical Support during problem diagnosis.
`KINetics-log.txt`	Created when the simulation run is initiated in the subdirectory `fluentKINlog`.	Messages are written to this file that record the progress of chemistry-set pre-processor and CHEMKIN-CFD solver. The printing level during the simulation run can be controlled by `Print Level` as indicated in Table 1: Summary of Basic CHEMKIN-CFD Parameters (p. 5698).
`KINCELL_<cell_id>_<iteration_number>.DBG`	Created for Monitor Cells (see Setting Up Monitor Cells for the Ansys CHEMKIN-CFD Chemistry Solver (p. 5702)) at every iteration or time-step, any time that the `Print Level`	Includes debug information, including the actual input sent into the CHEMKIN-CFD module by Ansys Fluent for the particular cell and iteration. This information may be requested by Ansys Technical Support when diagnosing cell failures.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Filename	When File Is Created	Contents Description
	(Table 1: Summary of Basic CHEMKIN-CFD Parameters (p. 5698)) is >0. Also created any time that a cell fails in the CHEMKIN-CFD solver and the `Print Level` is >0.	

C.4. Error Messages Printed in the Ansys Fluent Graphical User Interface

Advanced Parameters Used in the Steady-State Solution Algorithm (p. 5701) lists possible error messages that may appear in the Ansys Fluent Graphical User Interface during run using the CHEMKIN-CFD solver. The table includes recommended actions that may help resolve or narrow down the problem. In some cases, you may be referred to some of the additional diagnostic output files generated by the CHEMKIN-CFD module and listed in Table 3: Diagnostic Output Files Created During a CHEMKIN-CFD Run (p. 5702).

Table 4: Error Messages that May Be Printed to the Fluent GUI

Setting	Value
`Cannot read gas-phase chemistry input file.`	Verify the filename and file path for the gas-phase chemistry input file in the **Import CHEMKIN Format Mechanism** dialog box and re-import.
`Cannot read surface chemistry input file.`	Verify the filename and file path for the surface chemistry input file in the **Import CHEMKIN Format Mechanism** dialog box and re-import.
`Cannot read thermodynamic data file.`	Verify the filename and file path for the thermodynamic data file in the **Import CHEMKIN Format Mechanism** dialog box and re-import.
`Cannot read transport properties data file.`	Verify the filename and file path for the transport-property data file in the **Import CHEMKIN Format Mechanism** dialog box and re-import.
`Cannot read gas-phase chemistry linking data file during gas-phase pre-processing.`	Check the `<gas_chemistry>.out` file in the working directory for error messages.
`Cannot read gas-phase chemistry linking data file during surface pre-processing.`	Check the `<gas_chemistry>.out` file in the working directory for error messages.
`Cannot read gas-phase chemistry linking data file during transport pre-processing.`	Check the `<gas_chemistry>.out` file in the working directory for error messages.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
Cannot read surface chemistry linking data file during surface pre-processing.	Check the <surface_chemistry>.out file in the working directory for error messages.
Cannot read transport properties linking data file during transport pre-processing.	Check the <transport_data>.out file in the working directory for error messages.
Cannot write gas-phase chemistry output file.	Check file permissions on working directory and make sure that there is only one CHEMKIN-CFD process using these files at a time.
Cannot write surface chemistry output file.	Check file permissions on working directory and make sure that there is only one CHEMKIN-CFD process using these files at a time.
Cannot write transport properties output file.	Check file permissions on working directory and make sure that there is only one CHEMKIN-CFD process using these files at a time.
Cannot write gas-phase chemistry linking data file.	Check file permissions on working directory and make sure that there is only one CHEMKIN-CFD process using these files at a time.
Cannot write surface chemistry linking data file.	Check file permissions on working directory and make sure that there is only one CHEMKIN-CFD process using these files at a time.
Cannot write transport properties linking data file.	Check file permissions on working directory and make sure that there is only one CHEMKIN-CFD process using these files at a time.
Gas-phase chemistry file is NULL.	Verify gas-phase chemistry file path in the CHEMKIN Mechanism definition and re-import the chemistry files.
Surface chemistry file is NULL.	Verify surface-chemistry file path in the CHEMKIN Mechanism definition and re-import the chemistry files.
Thermodynamic data file is NULL.	Verify thermodynamic-data file path in the CHEMKIN Mechanism definition and re-import the chemistry files
Transport properties data file is NULL.	Verify transport-data file path in the CHEMKIN Mechanism definition and re-import the chemistry files.
Failure in pre-processing of gas-phase chemistry mechanism.	Check the <gas_chemistry>.out file in the working directory for error messages.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Setting	Value
`Failure in pre-processing of surface chemistry mechanism.`	Check the `<surface_chemistry>.out` file in the working directory for error messages.
`Failure in pre-processing of transport properties.`	Check the `<transport_data>.out` file in the working directory for error messages.
`Failure in CHEMKIN Gas-phase library call.`	Check to see if the Ansys Fluent solution has diverged or has unphysical values for the variables. Try turning `Print Level` > 0 and setting a monitor cell to obtain more information.
`Failure in CHEMKIN Surface library call.`	Check to see if the Ansys Fluent solution has diverged or has unphysical values for the variables. Try turning `Print Level` > 0 and setting a monitor cell to obtain more information.
`Failure in CHEMKIN Transport library call.`	Check to see if the Ansys Fluent solution has diverged or has unphysical values for the variables. Try turning `Print Level` > 0 and setting a monitor cell to obtain more information.
`Need to run CHEMKIN-CFD PrePro-cess first.`	Re-import the CHEMKIN chemistry set and make sure it is selected in the **Species Model** dialog box (under **Setup**/**Models**).
`No surface data for the chemistry set.`	Either de-select the Surface Chemistry option in the **Import CHEMKIN Format Mechanism** dialog box, or make sure that a valid surface chemistry file is indicated.
`No transport data for the chem-istry set.`	Either de-select the Transport Data option in the **Import CHEMKIN Format Mechanism** dialog box, or make sure that a valid transport data file is indicated.
`Failure in writing data into the diagnostics file.`	Check file permissions on the working directory and make sure that there is only one CHEMKIN-CFD process using these files at a time.
`Error in CHEMKIN-CFD model, see diagnostic output file for de-tails.`	Check the `KINetics-log.txt` file in the subdirectory `fluentKINlog` for error messages. See Error Messages Printed in the Ansys Fluent Graphical User Interface (p. 5704) for more information.

C.5. Diagnostic Messages in the `KINetics-log.txt` File

The `KINetics-log.txt` file is a "catch-all" for many diagnostic messages that are generated by the CHEMKIN-CFD module. The most common types of messages are error messages related to setup or cell convergence.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Cell-convergence error messages can take a variety of forms. The most common are summarized in Table 5: Other Error Messages in KINetics-log.txt (p. 5707). In general, if there are just a few failed cells at the beginning of a simulation it is safe to ignore them. When the number of failed cells continues to grow with each iteration, there is likely an issue with the problem set-up or with the chemistry data. Table 5: Other Error Messages in KINetics-log.txt (p. 5707) contains some broad suggestions of ways to improve cell-convergence behavior based on the type of cell failure message received.

Table 5: Other Error Messages in KINetics-log.txt

Error Message	Recommended Action
FAILURE INITIALIZING SETTINGS FOR THE KINetics STEADY-STATE GAS-PHASE CHEMISTRY SOLVER	Most likely suggests an error in setting the convergence criteria (that is absolute and relative tolerance) and the pseudo-steady-state time stepping parameters. Check that the user-provided values are reasonable.
WARNING: Exceeded allowed time-stepping intervals in cell	Indicates that a convergence problem was encountered during steady-state solution, where time-stepping is used to help advance the steady-state iterations. In this case, you can either ignore the message (see if the cell convergence recovers in subsequent iterations) or try to increase either the time interval or the number of allowed time-stepping intervals (See Table 2: Summary of Advanced CHEMKIN-CFD Parameters (p. 5699) for more information).
KINETICS ERROR: SOLUTION FAILED	Indicates that the cell failed to converge (either steady-state or transient).In this case, you can either ignore the message (see if the cell convergence recovers in subsequent iterations) or try to modify some of the user-controlled parameters to improve convergence (See Table 2: Summary of Advanced CHEMKIN-CFD Parameters (p. 5699) for more information).
Warning - The transient chemistry solver tried 500 steps without convergence in this cell; 500 more steps will be allowed	Indicates that the problem is difficult to converge so that the transient solver is taking extra steps. However, it is only a warning and the cell has not actually failed. If you receive an excessive number of these messages, try reducing the absolute or relative solution tolerances that control the CHEMKIN-CFD solver.
KINETICS ERROR: Problem getting species names; Check that the CHEMKIN files have been processed correctly	Indicates a possible error in the pre-processing of the gas-phase chemistry input file. Check the <gas_chemistry>.out file for error messages.
KINETICS ERROR: Problem getting surface chemistry info; Check	Indicates a possible error in the pre-processing of the surface chemistry input file. Check the <surface_chemistry>.out file for error messages.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Error Message	Recommended Action
`that the CHEMKIN files have been processed correctly`	
`DASPK:...`	These messages are from the CHEMKIN-CFD transient solver, DASPK. The exact message usually contains more information about the problem.

Note:

During steady-state simulations, once an iteration occurs where there are no failed cells, the effect of previously failed cells in earlier iterations is completely erased.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Appendix D. Nomenclature

A	Area (m^2, ft^2)
$\vec{a}$	Acceleration (m/s^2, ft/s^2)
a	Local speed of sound (m/s, ft/s)
c	Concentration (mass/volume, moles/volume)
C_D	Drag coefficient, defined different ways (dimensionless)
c_p, c_v	Heat capacity at constant pressure, volume (J/kg-K, Btu/lb_m- °F)
d	Diameter; $d_p$$D_p$, particle diameter (m, ft)
D_H	Hydraulic diameter (m, ft)
$_{ij}$,D	Mass diffusion coefficient (m^2/s, ft^2/s)
E	Total energy, activation energy (J, kJ, cal, Btu)
f	Mixture fraction (dimensionless)
$\vec{F}$	Force vector (N, lb_f)
F_D	Drag force (N, lb_f)
$\vec{g}$	Gravitational acceleration (m/s^2, ft/s^2); standard values = 9.80665 m/s^2, 32.1740 ft/s^2
Gr	Grashof number ≡ ratio of buoyancy forces to viscous forces (dimensionless)
H	Total enthalpy (energy/mass, energy/mole)
h	Heat transfer coefficient (W/m^2-K, Btu/ft^2-h- °F)
h	Species enthalpy; h^0, standard state enthalpy of formation (energy/mass, energy/mole)
I	Radiation intensity (energy per area of emitting surface per unit solid angle)
J	Mass flux; diffusion flux (kg/m^2-s, lb_m/ft^2-s)
K	Equilibrium constant = forward rate constant/backward rate constant (units vary)
k	Kinetic energy per unit mass (J/kg, Btu/lb_m)
k	Reaction rate constant, for example, k_1, k_{-1}, $k_{f,r}$, $k_{b,r}$ (units vary)
k	Thermal conductivity (W/m-K, Btu/ft-h- °F)
k_B	Boltzmann constant (1.38×10^{-23} J/molecule-K)
k, k_c	Mass transfer coefficient (units vary); also K, K_c
ℓ,l,L	Length scale (m, cm, ft, in)
Le	Lewis number ≡ ratio of thermal diffusivity to mass diffusivity (dimensionless)
m	Mass (g, kg, lb_m)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

$\dot{m}$ Mass flow rate (kg/s, lb_ms)

M_w Molecular weight (kg/kmol, lb_m/lb_mmol)

M Mach number ≡ ratio of fluid velocity magnitude to local speed of sound (dimensionless)

Nu Nusselt number ≡ dimensionless heat transfer or mass transfer coefficient (dimensionless); usually a function of other dimensionless groups

p Pressure (Pa, atm, mm Hg, lb_f/ft^2)

Pe Peclet number $\equiv Re \times Pr$ for heat transfer, and $\equiv Re \times Sc$ for mass transfer (dimensionless)

Pr Prandtl number ≡ ratio of momentum diffusivity to thermal diffusivity (dimensionless)

Q Flow rate of enthalpy (W, Btu/h)

q Heat flux (W/m^2, Btu/ft^2-h)

R Gas-law constant (8.31447 $\times 10^3$ J/kmol-K, 1.98588 Btu/lb_mmol- °F)

r Radius (m, ft)

$\mathcal{R}$ Reaction rate (units vary)

Ra Rayleigh number $\equiv Gr \times Pr$; measure of the strength of buoyancy-induced flow in natural (free) convection (dimensionless)

Re Reynolds number ≡ ratio of inertial forces to viscous forces (dimensionless)

S Total entropy (J/K, J/kmol-K, Btu/lb_mmol- °F)

s Species entropy; s^0, standard state entropy (J/kmol-K, Btu/lb_mmol- °F)

Sc Schmidt number ≡ ratio of momentum diffusivity to mass diffusivity (dimensionless)

S_{ij} Mean rate-of-strain tensor (s^{-1})

T Temperature (K, °C, °R, °F)

t Time (s)

U Free-stream velocity (m/s, ft/s)

u, v, w Velocity magnitude (m/s, ft/s); also written with directional subscripts (for example, v_x, v_y, v_z, v_r)

V Volume (m^3, ft^3)

$\vec{v}$ Overall velocity vector (m/s, ft/s)

We Weber number ≡ ratio of aerodynamic forces to surface tension forces (dimensionless)

X Mole fraction (dimensionless)

Y Mass fraction (dimensionless)

α Permeability, or flux per unit pressure difference (L/m^2-h-atm, ft $_3$/ft^2-h-(lb_f/ft^2))

α Thermal diffusivity (m^2/s, ft^2/s)

α Volume fraction (dimensionless)

β Coefficient of thermal expansion (K^{-1})

γ Porosity (dimensionless)

γ Ratio of specific heats, c_p/c_v (dimensionless)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Δ	Change in variable, final − initial (for example, Δp, Δt, ΔH, ΔS, ΔT)
δ	Delta function (units vary)
ε	Emissivity (dimensionless)
ε	Lennard-Jones energy parameter (J/molecule)
ε	Turbulent dissipation rate (m^2/s^3, ft^2/s^3)
ε	Void fraction (dimensionless)
η	Effectiveness factor (dimensionless)
η', η''	Rate exponents for reactants, products (dimensionless)
θ_r	Radiation temperature (K)
λ	Molecular mean free path (m, nm, ft)
λ	Wavelength (m, nm, Å, ft)
μ	Dynamic viscosity (cP, Pa-s, lb_m/ft-s)
ν	Kinematic viscosity (m^2/s, ft^2/s)
ν', ν''	Stoichiometric coefficients for reactants, products (dimensionless)
ρ	Density (kg/m^3, lb_m/ft^3)
σ	Stefan-Boltzmann constant ($5.67x10^{-8}$ W/m^2-K^4)
σ	Surface tension (kg/m, dyn/cm, lb_f/ft)
σ_s	Scattering coefficient (m^{-1})
$\bar{\bar{\tau}}$	Stress tensor (Pa, lb_f/ft^2)
τ	Shear stress (Pa, lb_f/ft^2)
τ	Time scale, for example, τ_c, τ_p, τ_c (s)
τ	Tortuosity, characteristic of pore structure (dimensionless)
ϕ	Equivalence ratio (dimensionless)
ϕ	Thiele modulus (dimensionless)
Ω	Angular velocity; Ω_{ij}, Mean rate of rotation tensor (s^{-1})
ω	Specific dissipation rate (s^{-1})
Ω,Ω'	Solid angle (degrees, radians, gradians)
Ω_D	Diffusion collision integral (dimensionless)

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.

Bibliography

Important:

Under U.S. and international copyright law, Ansys, Inc. is unable to distribute copies of the papers listed in the bibliography, other than those published internally by Ansys, Inc. Use your library or a document delivery service to obtain copies of copyrighted papers.

[1] *CRANIUM, Version 1.0*. Molecular Knowledge Systems, Inc., Bedford, NH. 2007.

[2] *FIELDVIEW Reference Manual, Software Release Version 10*. Intelligent Light. 2004.

[3] *KINetics for Fluent, Version 2.4*. Reaction Design, Inc., San Diego, CA. 2009.

[4] *VdmTools Programmer Manual, Version 3.2.0*. Visual Kinematics, Inc. 2005.

[5] T. Ahmad, S. L. Plee, and J. P. Myers. "Computation of Nitric Oxide and Soot Emissions from Turbulent Diffusion Flames". *J. of Engineering for Gas Turbines and Power*. 107. 48–53. 1985.

[6] C.O. Alanyalioglu and Y. Özyörük. "Fully Transient Conjugate Analysis of Silica-Phenolic Charring Ablation Coupled with Interior Ballistics". *AIAA Propulsion and Energy 2019 Forum*. 2019.

[7] F. Alauzet and A. Loseille. "A decade of progress on anisotropic mesh adaptation for computational fluid dynamics". *Computer-Aided Design*. Volume 72. 13–39. March 2016.

[8] F. Alauzet, A. Loseille, D. Marcum, and T. Michal. "Assessment of anisotropic mesh adaptation for high-lift prediction of the HL-CRM configuration". *23rd AIAA Computational Fluid Dynamics Conference*. 1–15. June 2017.

[9] A. A. Amsden, P. J. O'Rourke, and T. D. Butler. "KIVA-2: A Computer Program for Chemically Reactive Flows with Sprays". *Technical Report LA-11560-MS, UC-96*. Los Alamos National Laboratory, Los Alamos, New Mexico. May 1989.

[10] J. D. Anderson. Hypersonics and High Temperature Gas Dynamics. *Second edition*. AIAA. 2006.

[11] A. Andreini, R. Da Soghe, B. Facchini, L. Mazzei, S. Colantuoni, and F. Turrini . "Local source based CFD modeling of effusion cooling holes: Validation and application to an actual combustor test case". *J. Eng. Gas Turbines Power*. 136. 1. 011506. 2014.

[12] S. Asgari, X. Hu, M. Tsuk, and S. Kaushik. "Application of POD plus LTI ROM to Battery Thermal Modeling: SISO Case". *SAE International Journal of Commercial Vehicles*. 7. 1. 278–285. 2014.

[13] M. J. Assael, J. P. M. Trusler, and T. F. Tsolakis. "Thermophysical Properties of Fluids". *Technical report*. Imperial College Press, London. 1996.

[14] T. Aumeier and T. Behrendt . "Application of an Aerothermal Model for the Effusion Cooling Systems and Finite Rate Chemistry in Aero-Engine Combustors". *Turbulence Heat and Mass Transfer*. 8. 2015.

[15] R. H. Aungier. "A Fast, Accurate Real Gas Equation of State for Fluid Dynamic Analysis Applications". *Journal of Fluids Engineering*. 117. 277–281. 1995.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

[16] R. Barrett, M. Berry, T. F. Chan, J. Demmel, J. Donato, J. Dongarra, V. Eijkhout, R. Pozo, C. Romine, and H. Van der Vorst. "Templates for the Solution of Linear Systems: Building Blocks for Iterative Methods". *SIAM, 2nd edition*. Philadelphia, Pennsylvania. 1994.

[17] A. Bejan. *Convection Heat Transfer*. John Wiley and Sons, New York. 1984.

[18] J. H. Bell, R. D. Mehta. "Development of a two-stream mixing layer from tripped and untripped boundary layers". *SIAM, 2nd edition*. AIAA Journal. vol. 28, no.12. 2034-2042. 1990.

[19] D. T. Blackstock. *Fundamentals of Physical Acoustics*. John Wiley & Sons. 2000.

[20] C. T. Bowman. "Chemistry of Gaseous Pollutant Formation and Destruction". *Fossil Fuel Combustion*. W. Bartok and A. F. Sarofim, editors. J. Wiley and Sons, Canada. 1991.

[21] R. N. Bracewell. *The Fourier transform and its applications*. McGraw Hill. 1986.

[22] W. Brack, B. Heine, F. Birkhold, M. Kruse, O. Deutschmann. "Formation of Urea-Based Deposits in an Exhaust System: Numerical Predictions and Experimental Observations on a Hot Gas Test Bench". *Emiss. Control Sci. Technol.*. 2. 3. 115–123. 2016.

[23] L.J.S. Bradbury,. "The Structure of a Self-Preserving Plane Jet". *Journal of Fluid Mechanics*. 23. 31-64. 1965.

[24] W.-P. Breugem. "The Effective Viscosity of a Channel-type Porous Medium". *Phys. of Fluids*. 19. 10. 2007.

[25] H. Brinkman. "A Calculation of the Viscous Force Exerted by a Flowing Fluid on a Dense Swarm of Particles". *Applied Scientific Research*. A1. 27–34. 1949.

[26] H. Brinkman. "The Viscosity of Concentrated Suspensions and Solutions". *J. Chem. Phys*. 20. 4. 571–584. 1952.

[27] R. J. Brooks and A. T. Corey. "Hydraulic properties of porous media". *Hydrology Paper 3*. Colorado State University. Fort Collins, Colorado, USA: 1964.

[28] S. J. Brookes and J. B. Moss. "Prediction of Soot and Thermal Radiation in Confined Turbulent Jet Diffusion Flames". *Combustion and Flame*. 116. 486–503. 1999.

[29] R. Carsel and R. Parrish. "Developing joint probability distributions of soil water retention characteristics". *Water Resources Research*. 24. 5. 755–769. 1988.

[30] J. R. Cash and A. H. Karp. "A variable order Runge-Kutta method for initial value problems with rapidly varying right-hand sides". *ACM Transactions on Mathematical Software*. 16. 201–222. 1990.

[31] T. Cebeci and P. Bradshaw. *Momentum Transfer in Boundary Layers*. Hemisphere Publishing Corporation, New York. 1977.

[32] P. J. Coelho and M. G. Carvalho. "Modeling of Soot Formation and Oxidation in Turbulent Diffusion Flames". *J. of Thermophysics and Heat Transfer*. 9(4). 644–652. 1995.

[33] M. F. Cohen and D. P. Greenberg. "The Hemi-Cube: A Radiosity Solution for Complex Environments". *Computer Graphics*. 19(3). 31–40. 1985.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

[34] H. W. Coleman, B. K. Hodge, and R. P. Taylor. "A Re-Evaluation of Schlichting's Surface Roughness Experiment". *Journal of Fluids Engineering*. 106. 1984.

[35] E. H. Cuthill and J. McKee. "Reducing Bandwidth of Sparse Symmetric Matrices". *In Proc. ACM 24th National Conf. New York*. 157–172. 1969.

[36] J. F. Daunenhofer and J. R. Baron. "Grid Adaption for the 2D Euler Equations". *Technical Report AIAA-85-0484*. American Institute of Aeronautics and Astronautics. 1985.

[37] J. E. Dec. "A Conceptual Model of DI Diesel Combustion Based on Laser Sheet Imaging". *SAE Technical Paper 970873*. SAE. 1997.

[38] J. Ding and D. Gidaspow. "A Bubbling Fluidization Model Using Kinetic Theory of Granular Flow". *AIChE J*. 36(4). 523–538. 1990.

[39] D.M. Driver. "Reynolds Shear Stress Measurements in a Separated Boundary Layer Flow". *AIAA 22nd Fluid Dynamics, Plasma Dynamics and Laser Conference*. AIAA-91-1787. 1991.

[40] Y. Egorov and F. Menter. "Experimental Implementation of the RPI Wall Boiling Model in CFX-5.6". *Technical Report*. ANSYS/TR-04-10, ANSYS GmbH. 2004.

[41] J. I. Erdos, E. Alzner, W. McNally. "Numerical Solution of Periodic Transonic Flow through a Fan Stage". *AIAA Journal*. 15. 10. 115. 1977.

[42] S. Ergun. "Fluid Flow through Packed Columns". *Chem. Eng. Prog*. 48(2). 89–94. 1952.

[43] G. M. Faeth. "Spray Atomization and Combustion". *Technical Report 1986-136*. AIAA. 1986.

[44] J. E. Ffowcs-Williams and D. L. Hawkings. "Sound Generation by Turbulence and Surfaces in Arbitrary Motion". *Proc. Roy. Soc. London*. A264. 321–342. 1969.

[45] E. U. Finlayson, D. K. Arasteh, C. Huizenga, M. D. Rubin, and M. S. Reilly. "WINDOW 4.0: Documentation of Calculation Procedures". *Technical Report Publication LBL-33943/ TA-309*. Lawrence Berkeley Laboratory, Energy and Environmental Division, Berkeley, CA 94720. 1993.

[46] K.–Y. Fung and H. Ju. "Broadband Time-Domain Impedance Models". *AIAA Journal*. 39(8). January 2001.

[47] K.–Y. Fung, H. Ju, and B. Tallapragada. "Impedance and Its Time-Domain Extensions". *AIAA Journal*. 38(1). 30–38. January 2000.

[48] M. T. van Genuchten. "A closed form equation for predicting the hydraulic conductivity of unsaturated soils". *Soil Science Society of American Journal*. 44. 5. 892–898. 1980.

[49] T. Gessner. "Dynamic Mesh Adaption for Supersonic Combustion Waves Modeled with Detailed Reaction Mechanisms". *PhD thesis*. University of Freiburg, Freiburg, Germany. 2001.

[50] A. Ghobadian and S. A. Vasquez. "A General Purpose Implicit Coupled Algorithm for the Solution of Eulerian Multiphase Transport Equation". *International Conference on Multiphase Flow*. Leipzig, Germany. 2007.

[51] N. E. Gibbs, W. G. Poole, Jr., and P. K. Stockmeyer. "An algorithm for reducing the bandwidth and profile of a sparse matrix". *SIAM J. Numer. Anal*. 13. 236–250. 1976.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

[52] M. B. Giles. "Non-Reflecting Boundary Conditions for the Euler Equations". *Technical Report 88-1*. Computational Fluid Dynamics Laboratory, Massachusetts Institute of Technology, Cambridge, MA. 1988.

[53] T. Gokcen. "N2-CH4-Ar chemical kinetic model for simulations of atmospheric entry to Titan.". *Journal of Thermophysics and Heat Transfer 21*. 1. 9–18. 2007.

[54] R.A. Gore and C.T. Crowe. “Effect of particle size on modulating turbulent intensity”. *Int. J. Multiphase Flow*. 15. 279–285. 1989.

[55] J. Gottgens, F. Mauss, and N. Peters. "Analytic Approximations of Burning Velocities and Flame Thicknesses of Lean Hydrogen, Methane, Ethylene, Ethane, Acetylene and Propane Flames". *In Twenty-Fourth Symposium (Int.) on Combustion, Pittsburgh*. 129–135. 1992.

[56] P. M. Gresho, R. L. Lee, and R. L. Sani. "On the Time-Dependent Solution of the Incompressible Navier-Stokes Equations in Two and Three Dimensions". *In Recent Advances in Numerical Methods in Fluids*. Pineridge Press, Swansea, U. K. 1980.

[57] R.N. Gupta, M.Y. Jerrold, R.A. Thompson, and K.P. Lee.. A review of reaction rates and thermodynamic and transport properties for an 11-species air model for chemical and thermal nonequilibrium calculations to 30000 K.". 1990.

[58] B. Gustavsen and A. Semlyen. "Rational approximation of frequency domain responses by vector fitting". *IEEE Transactions on Power Delivery*. IEEE. Volume 14, Issue 3. 1052-1061. July, 1999.

[59] W. G. Habashi, J. Dompierre, Y. Bourgault, D. Ait-Ali-Yahia, M. Fortin, and M.-G. Vallet. "Anisotropic mesh adaptation: towards user-independent, mesh-independent and solver-independent CFD. Part I: general principles". *Int. J. Numer. Meth. Fluids*. Volume 32, Issue 6. 725–744. March 14, 2000.

[60] R. J. Hall, M. D. Smooke, and M. B. Colket. *Physical and Chemical Aspects of Combustion*. Gordon and Breach. 1997.

[61] L. He and H.D. Li. "Single-Passage Analysis of Unsteady Flows around Vibrating Blades of a Transonic Fan under Inlet Distortion". *ASME Journal of Turbomachinery*. 124. 2. 285–295. 2002.

[62] R. A. W. M. Henkes and C. J. Hoogendoorn. "Scaling of the Turbulent Natural Convection Flow in a Heated Square Cavity". *Trans. of the ASME*. 116. 400–408. May 1994.

[63] J. B. Heywood. *Internal Combustion Engine Fundamentals*. McGraw-Hill, New York, rev. edition. 1988.

[64] C. Hirsch. *Numerical Computation of Internal and External Flows: Computational Methods for Inviscid and Viscous Flows, volume 2*. John Wiley & Sons, New York. 1984.

[65] J. O. Hirschfelder, C. F. Curtiss, and R. B. Bird. *Molecular Theory of Gases and Liquids*. John Wiley & Sons, New York. 1954.

[66] K. Hsu and A. Jemcov. "Numerical investigation of detonation in premixed hydrogen-air mixture - assessment of simplified chemical mechanisms". *Technical Report AIAA-2000-2478*. American Institute of Aeronautics and Astronautics, Fluids 2000 Conference and Exhibit, Denver, CO. June 2000.

[67] Y. Y. Hsu. "On the size range of active nucleation cavities on a heating surface". *J. Heat Transfer*. 84(3). 207–213. August 1962.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

[68] X. Hu et al.. "A transient reduced order model for battery thermal management based on singular value decomposition". *2014 IEEE Energy Conversion Congress and Exposition (ECCE)*. 3971–3976. 2014.

[69] X. Hu, L. Collins, S. Stanton, and S. Jiang. "A Model Parameter Identification Method for Battery Applications". *SAE Technical Paper*. 2013–01–1529. 2013.

[70] X. Hu, S. Lin, S. Stanton, and W. Lian . "A Foster Network Thermal Model for HEV/EV Battery Modeling". *IEEE Transactions on Industry Applications*. 47. 4. 1692–1699. 2011.

[71] G. W. Jackson and D. F. James. "The Permeability of Fibrous Porous Media". *Canadian Journal of Chemical Engineering*. 64(3). 364–374. June 1986.

[72] A. Jemcov and J. P. Maruszewski. "Shape Optimization Based on Downhill Simplex Optimizer and Free-Form Deformation in General Purpose CFD Code". *17th Annual Conference of CFD Society of Canada*. Ottawa, Ontario, Canada. 2009.

[73] C.O. Johnson and A. M. Brandis. "Modeling of nonequilibrium CO Fourth-Positive and CN Violet emission in CO2–N2 gases.". *Journal of Quantitative Spectroscopy and Radiative Transfer 149*. 303–317. 2014.

[74] P. C. Johnson and R. Jackson. "Frictional-Collisional Constitutive Relations for Granular Materials, with Application to Plane Shearing". *J. Fluid Mech*. 176. 67–93. 1987.

[75] G. Karypis and V. Kumar. *METIS - A Software Package for Partitioning Unstructured Graphs, Partitioning Meshes, and Computing Fill-Reducing Orderings of Sparse Matrices, Version 3.0. Manual*. University of Minnesota and Army HPC Research Center. 1997.

[76] G. Karypis and K. Schloegel. *PARMETIS - Parallel Graph Partitioning and Sparse Matrix Ordering Library, Version 4.0. Manual*. University of Minnesota. 2013.

[77] A. Kaufmann, F. Nicoud, and T. Poinsot. Flow Forcing Techniques for Numerical Simulation of Combustion Instabilities. *Combustion and Flame*. 131. 371–385. 2002.

[78] R. J. Kee, F. M. Rupley, and J. A. Miller. "SURFACE CHEMKIN: A Software Package for the Analysis of Heterogeneous Chemical Kinetics at a Solid-Surface - Gas-Phase Interface". *Technical Report SAND 96-8217*. Sandia National Labs. 2000.

[79] R. J. Kee, F. M. Rupley, J. A. Miller, M. E. Coltrin, J. F. Grcar, E. Meeks, H. K. Moffat, A. E. Lutz, G. Dixon-Lewis, M. D. Smooke, J. Warnatz, G. H. Evans, R. S. Larson, R. E. Mitchell, L. R. Petzold, W. C. Reynolds, M. Caracotsios, W. E. Stewart, P. Glarborg, C. Wang, O. Adigun, W. G. Houf, C. P. Chou, S. F. Miller, P. Ho, and D. J. Young. CHEMKIN v. 4.0. Technical Report. San Diego, CA. Reaction Design, Inc. 2004.

[80] T. G. Kolda, R. M. Lewis, and V. Torczon. "Optimization by Direct Search: New Perspectives on Some Classical and Modern Methods". *SIAM Review*. 45. 3. 385–482. 2003.

[81] P. Kolodziej, and D.Stewart. . "Nitrogen Recombination on High-temperature Reusable Surface Insulation and the Analysis of its Effect on Surface Catalysis" . *22nd Thermophysics Conference*. 1637. 1987.

[82] B. Koncar, I. Kljenak, and R. Mavko. "Modeling of Local Two-Phase Flow Parameters in Upward Subcooled Flow Boiling at Low Pressure". *International Journal of Heat and Mass Transfer*. 47. 1499–1513. 2004.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

[83] K. K. Y. Kuo. *Principles of Combustion*. John Wiley and Sons, New York. 1986.

[84] B. E. Launder and D. B. Spalding. "The Numerical Computation of Turbulent Flow". *Computer Methods in Applied Mechanics and Engineering*. 3. 269–289. 1974.

[85] M. C. Leverett. "Capillary behavior in porous solids". *Transactions of the AIME*. 142. 152–169. 1941.

[86] H. Li, S. A. Vasquez, H. Punekar, and R. Muralikrishnan. "Prediction of Boiling and Critical Heat Flux Using an Eulerian Multiphase Boiling Model". *Paper No: IMECE2011-65539*. 463–476. 2012.

[87] J. Li, Z. Zhao, et al. "On Sound Generated Aerodynamically". *Proc. Roy. Soc. London*. A211. 564–587. 1952.

[88] M. J. Lighthill. "A Comprehensive Kinetic Mechanism for CO, CH2O, and CH3OH Combustion". *Int. J. Chem. Kinet*. 39. 109-136. 2007.

[89] P. Lin. *Numerical Modeling of Water Waves*. Taylor & Francis. 2008.

[90] R. P. Lindsted, F. C. Lockwood, and M. A. Selim. "Rate Constants Based on Partial Equilibrium Assumptions". *Internal Report No. TF/95/3, Technical report*. Imperial College, London, UK. 1995.

[91] M. S. Liou. "A sequel to AUSM: AUSM+". *Journal of Computational Physics*. 129. 364–382. 1996.

[92] F. C. Lockwood, S. M. A. Rizvi, and N. G. Shah. "Comparative Predictive Experience of Coal Firing". *In Proceedings Inst. Mechanical Engineers*. 200. 79–87. 1986.

[93] J. Y. Luo, R. I. Issa, and A. D. Gosman. "Prediction of Impeller-Induced Flows in Mixing Vessels Using Multiple Frames of Reference". *In I ChemE Symposium Series*. 136. 549–556. 1994.

[94] B. F. Magnussen and B. H. Hjertager. "On mathematical models of turbulent combustion with special emphasis on soot formation and combustion". *In 16th Symp. (Int'l.) on Combustion*. The Combustion Institute. 1976.

[95] M. Manninen, V. Taivassalo, and S. Kallio. "On the mixture model for multiphase flow". *VTT Publications 288*. Technical Research Centre of Finland. 1996.

[96] J. Maxwell. "On Stresses in Rarified Gases Arising from Inequalities of Temperature". *Philosophical Transactions of the Royal Society of London*. vol. 170. p. 231–256. 1879.

[97] B. J. McBride, S. Gordon, and M. A. Reno. "Coefficients for Calculating Thermodynamic and Transport Properties of Individual Species". *NASA TM-4513*. October 1993.

[98] K. I. M. McKinnon. "Convergence of the Nelder-Mead Simplex Method a Nonstationary Point ". *SIAM J. Optimization*. 9. 1. 148–158. 1998.

[99] F.R. Menter. "Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications". *AIAA Journal*. 32(8). 1598–1605. August 1994.

[100] F. R. Menter and Y. Egorov. "Re-visiting the Turbulent Scale Equation". *Proc. IUTAM Symp. One Hundred Years of Boundary Layer Research, Göttingen*. Springer. 2004.

[101] F. R. Menter and Y. Egorov. "A Scale-Adaptive Simulation Model Using Two-Equation Models". *AIAA Paper-2005-1095*. Reno NV. 2005.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

[102] R. Merz, J. Kruckels, J. Mayer, and H. Stetter. "Computation of Three Dimensional Viscous Transonic Turbine Stage Flow Including Tip Clearance Effects". *ASME 95-GT-76*. 1995.

[103] J. H. Miller, J. C. Tennehill, G. Wadawadigi, T. A. Edwards, S. L. Lawrence. "Computation of Hypersonic Flows with Finite Catalytic Walls." . *Journal of Thermophysics and Heat Transfer*. 9. 3. 486–493. 1995.

[104] C. V. Naik, K. V. Puduppakkam, et al. "Modeling Combustion Characteristics of Components of Model-Biodiesel at High Temperatures". *WSS/CI Spring 2008 Meeting*. Los Angeles, CA: 2008.

[105] J. A. Nelder and R. Mead. "A Simplex Method for Function Minimization". *Computer Journal*. 7. 4. 308–313. 1965.

[106] S. Nesic, A. Kahyarian, and Y. S. Choi. "Implementation of a comprehensive mechanistic prediction model of mild steel corrosion in multiphase oil and gas pipelines". *Corrosion*. 75 (3). 274–291. 2019.

[107] I. Nompelis, G. V. Candler and M. S. Holden. "Effect of Vibrational Nonequilibrium on Hypersonic Double-Cone Experiments". *AIAA JOURNAL*. 41. 11. pp. 2162–2169. 2003.

[108] Y.I. Oka et al.. "Practical estimation of erosion damage caused by solid particle impact, Part 1: Effects of impact parameters on a predictive equation". *15th International Conference on Wear of Materials*. 259. 1-6. 95–101. 2005.

[109] P. J. O'Rourke. "Collective Drop Effects on Vaporizing Liquid Sprays". *PhD thesis*. Princeton University, Princeton, New Jersey. 1981.

[110] M. Østberg, P. Glarborg, A. Jensen, J. E. Johnsson, L. S. Pedersen, and K. Dam-Johansen. "A model of the coal reburning process". *In Proc. Combust Inst*. 27. 3027–3035. 1998.

[111] I. Owczarek and K. Blazej. "Recommended Critical Temperatures. Part I. Aliphatic Hydrocarbons". *J. Phys. Chem*. 32. 1411–1427. 2003.

[112] I. Owczarek and K. Blazej. "Recommended Critical Pressures. Part I. Aliphatic Hydrocarbons". *J. Phys. Chem*. 35. 1461–1474. 2006.

[113] C. Park. *Review of chemical-kinetic problems of future NASA missions. I-Earth entries*. *Journal of Thermophysics and Heat transfer*. 7(3). 385–398. 1993.

[114] S. V. Patankar. *Numerical Heat Transfer and Fluid Flow*. Hemisphere, Washington, DC. 1980.

[115] S. V. Patankar. *Numerical Heat Transfer and Fluid Flow*. McGraw-Hill. Washington, New York, London 1980.

[116] S. V. Patankar, C. H. Liu, and E. M. Sparrow. "Fully Developed Flow and Heat Transfer in Ducts Having Streamwise-Periodic Variations of Cross-Sectional Area". *ASME J. of Heat Transfer*. 99. 180–186. 1977.

[117] D. Y. Peng and D. B. Robinson. "A New Two-Constant Equation of State". *Industrial and Engineering Chemistry: Fundamentals*. 15. 59–64. 1976.

[118] R. H. Perry, D. W. Gree, and J. O. Maloney. *Perry's Chemical Engineers' Handbook*. McGraw-Hill, New York, 6th edition. 1984.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

[119] A. A. F. Peters and R. Weber. "Mathematical Modeling of a 2.25 MWt Swirling Natural Gas Flame. Part 1: Eddy Break-up Concept for Turbulent Combustion; Probability Density Function Approach for Nitric Oxide Formation". *Combustion Science and Technology*. 110. 67–101. 1995.

[120] T. J. Poinsot and S. K. Lele. "Boundary Conditions for Direct Simulation of Compressible Viscous Flows". *Journal of Computational Physics*. 101. 104–129. 1992.

[121] T. J. Poinsot and L. Selle. "Actual Impedance of Nonreflecting Boundary Conditions: Implications for Computation of Resonators". *AIAA Journal*. 42(5). 958–964. May 2004.

[122] B. E. Poling, J. M. Prausnitz, and J. P. O'Connell. *The properties of Gases and Liquids*. McGraw-Hill, International Edition, 5th edition. 2007.

[123] S. B. Pope. *ISAT-CK (Version 3.0) User's Guide and Reference Manual*. 2000.

[124] M. J. D. Powell. "The NEWUOA software for unconstrained optimization without derivatives". *Large-Scale Nonlinear Optimization (Nonconvex Optimization and Its Applications)*. 83. 255–297. G. Di Pillo and M. Roma, editors. Springer. 2006.

[125] W. H. Press, S. A. Teukolsky, W. T. Vetterling, and B. P. Flannery. *Numerical Recipes: The Art of Scientific Computing, Third Edition*. Cambridge University Press, Cambridge, England. 2007.

[126] J. C. Ragusa. "A simple Hessian-based 3D mesh adaptation technique with applications to the multigroup diffusion equations". *Annals of Nuclear Energy*. Volume 35, Issue 11. 2006–2018. November 2008.

[127] C. L. Rasmussen and A. E. Rasmussen. "Sensitizing Effects of NOx on CH4 Oxidation at High Pressure". *Comb. Flame*. 154. 233. 1949.

[128] O. Redlich and J. N. S. Kwong. "On the Thermodynamics of Solutions. An Equation of State. Fugacities of Gaseous Solutions". *Chem. Rev*. 44. 3. 529-545. 2008.

[129] W. C. Reynolds. *Thermodynamic Properties in SI: Graphs, Tables, and Computational Equations for 40 Substances*. Department of Mechanical Engineering, Stanford University. 1979.

[130] J. W. Rose and J. R. Cooper. *Technical Data on Fuel*. Scottish Academic Press, Edinburgh. 1977.

[131] J. W. Rose and J. R. Cooper, editors. *Technical Data on Fuels*. Wiley, 7th edition. 1977.

[132] H. H. Rosenbrock. "An Automatic Method for Finding the Greatest or Least Value of a Function". *Computer Journal*. 3. 3. 175–184. 1960.

[133] F. Qian, L. Lu, T.Feng, and D.Yang. "Experimental and 3D Embedded Modelling for Diesel Engine SCR Deposit". *International Journal of Automotive Technology*. 18. 2. 219–227. 2017.

[134] Y. Saad and M. H. Schultz. "GMRES: A Generalized Minimal Residual Algorithm for Solving Nonsymmetric Linear Systems". *SIAM J. Sci. and Stat. Comput*. 7(3). 856–869. July 1986.

[135] A. Saxer. "A Numerical Analysis of a 3D Inviscid Stator/Rotor Interaction Using Non-Reflecting Boundary Conditions". *PhD thesis*. Massachusetts Institute of Technology, Cambridge, Massachusetts. March 1992.

[136] S. Sarkar and L. Balakrishnan. "Application of a Reynolds-Stress Turbulence Model to the Compressible Shear Layer". ICASE Report 90-18*NASA CR 182002*. 1990.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

[137] H. Schlichting. *Boundary-Layer Theory*. 7th Edition, McGraw-Hill, New York. 1979.

[138] H. Schlichting and K. Gersten. *Grenzschicht-Theorie*. Springer-Verlag, Berlin, Heidelberg, New York. 1997.

[139] C. Scott. "Catalytic Recombination of Nitrogen and Oxygen on High-temperature Reusable Surface Insulation." . *15th Thermophysics Conference*. 1477. 1981.

[140] A. A. Shabana. *Computational Dynamics*. John Wiley and Sons, New York. 1994.

[141] J. Shin and T. H. Bond. "Experimental and Computational Ice Shapes and Resulting Drag Increase for a NACA 0012 Airfoil". *NASA Technical Memorandum 105743*. 1992.

[142] G. M. Shroff and H. B. Keller. "Stabilization of unstable procedures: The recursive projection method". *SIAM J. Numer. Anal*. 30(4). 1099–1120. 1993.

[143] M. L. Shur, P. R. Spalart, M. K. Strelets, and A. K. Travin. "A Hybrid RANS-LES Approach With Delayed-DES and Wall-Modelled LES Capabilities". *International Journal of Heat and Fluid Flow*. 29:6. December 2008. 1638-1649.

[144] Y. R. Sivathanu and G. M. Faeth. "Generalized State Relationships for Scalar Properties in Non-Premixed Hydrocarbon/Air Flames". *Combustion and Flame*. 82. 211–230. 1990.

[145] Skjaeveland, S.M., Siqveland, L.M., Kjosavik, A., Hammervold Thomas, W.L., Virnovsky, G.A. . "Capillary Pressure Correlation for Mixed-Wet Reservoirs". *SPE Reservoir Eval. & Eng.*. 3. (1). February, 2000.

[146] I. B. Sladkov. "Physicochemical Properties of Methyl- and Ethylhalosilanes". *Russian Journal of Applied Chemistry*. 74. 1801–1805. 2001.

[147] A. L. Smith. "Family Plots for Evaluating Physical Properties of Organosilicon Compounds". *AIChE Journal*. 40. 373–377. 1994.

[148] G. P. Smith, D. M. Golden, et al. "GRI-Mech 3.0". 2008. http://www.me.berkeley.edu/gri_mech.

[149] H. Smith, T. Lauer, M. Mayer, and S. Pierson . "Optical and Numerical Investigations on the Mechanisms of Deposit Formation in SCR Systems". *SAE Int. J. Fuels Lubr.*. 7. 2. 525–542. 2014. DOI: 10.4271/2014-01-1563.

[150] P. L. Smith, Jr. and M. Van Winkle. "Discharge coefficients through perforated plates at reynolds numbers of 400 to 3,000". *AIChE Journal*. 4(3). 266–268. 1958.

[151] D. O. Snyder, E. K. Koutsavdis, and J. S. R. Anttonen. "Transonic store separation using unstructured cfd with dynamic meshing". *Technical Report AIAA-2003-3913*. American Institute of Aeronautics and Astronautics, 33th AIAA Fluid Dynamics Conference and Exhibit. 2003.

[152] G. Soave. "Equilibrium Constants from a modified Redlich-Kwong equation of State". *Chemical Engineering Science*. 27. 1197. 1972.

[153] G. R. Somayajulu. "Estimation Procedures for Critical Constants". *J. Chem. Eng. Data*. 34. 106–120. 1989.

[154] P. R. Spalart, S. Deck, M. L. Shur, K. D. Squires, M. Kh. Strelets, and A. Travin. "A New Version of Detached-eddy Simulation, Resistant to Ambiguous Grid Densities ". *Theoretical and Computational Fluid Dynamics*. Springer Berlin / Heidelberg. Volume 20, Number 3. July, 2006.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

[155] K. Stueben. "Introduction to Algebraic Multigrid". *Multigrid*. C. W. Oosterlee, U. Trottenberg, and A. Schuller, editors. Academic Press, New York. 413–532. 2001.

[156] Y. Sun, B. Vernham and S. A. Drenna. "Demonstration and Validation of Urea Deposit Predictions on a Practical Mid/Heavy Duty Vehicle Aftertreatment System". *ILASS-Americas 29th Annual Conference on Liquid Atomization and Spray Systems*. Atlanta, GA2017.

[157] M. Syamlal, W. Rogers, and T. J. O'Brien. *MFIX Documentation: Volume 1, Theory Guide*. DOE/METC-9411004, NTIS/DE9400087 National Technical Information Service, Springfield, VA. 1993.

[158] L. Talbot et al. "Thermophoresis of Particles in a Heated Boundary Layer". *J. Fluid Mech*. 101(4). 737–758. 1980.

[159] W. Tang and K. Brezinsky. "Chemical kinetic simulations behind reflected shock waves". *International Journal of Chemical Kinetics*. 38(2). 75–97. February 2006.

[160] R. I. Tanner. *Engineering Rheology*. Clarendon Press, Oxford, rev. edition. 1988.

[161] C. Temperton. "Implementation of a Self-Sorting In-Place Prime Factor FFT Algorithm". *Journal of Computational Physics*. 58. 283–299. 1985.

[162] P. A. Tesner, T. D. Snegiriova, and V. G. Knorre. "Kinetics of Dispersed Carbon Formation". *Combustion and Flame*. 17. 253–260. 1971.

[163] K. W. Thompson. "Time Dependent Boundary Conditions for Hyperbolic Systems". *Journal of Computational Physics*. 68. 1–24. 1987.

[164] K. W. Thompson. "Time Dependent Boundary Conditions for Hyperbolic Systems II". *Journal of Computational Physics*. 89. 439–461. 1990.

[165] V. Torczon. "On the Convergence of the Multidirectional Search Algorithm". *SIAM J. Optimization*. 1. 1. 123–145. 1991.

[166] J.C. Vogel, J.K. Eaton. "Combined Heat Transfer and Fluid Dynamic Measurements Downstream of a Backward-Facing Step". *Journal of Heat Transfer*. 107. 922-929.. 1985.

[167] H. Wagner. "Soot Formation in Combustion". *In 17th Symp. (Int'l.) on Combustion*. The Combustion Institute. 3–19. 1979.

[168] G. P. Warren, W. K. Anderson, J. L. Thomas, and S. L. Krist. "Grid convergence for adaptive methods". *Technical Report AIAA-91-1592*. AIAA10th Computational Fluid Dynamics Conference, Honolulu, HawaiiAmerican Institute of Aeronautics and Astronautics. June 1991.

[169] E. W. Washburn. "Note on a Method of Determining the Distribution of Pore Sizes in a Porous Material". Proc. National Academy of Science. 7. 4. 115–116. 1921.

[170] J. M. Weiss. "Calculations of Reacting Flowfield Involving Stiff Chemical Kinetics". *AIAA-99-3369*. January 1999.

[171] J. M. Weiss, J. P. Maruszewski, and W. A. Smith. "Implicit Solution of the Navier-Stokes Equations on Unstructured Meshes". *Technical Report AIAA-97-2103, 13th AIAA CFD Conference*. Snowmass, CO. July 1997.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

[172] C. Westbrook and F. Dryer. "Simplified Reaction Mechanisms for Oxidation of Hydrocarbon Fuels in Flames". *Comb. Sci. Tech*. 27. 31–43. 1981.

[173] D. C. Wilcox. *Turbulence Modeling for CFD*. DCW Industries, Inc., La Canada, California. 1998.

[174] I. Wygnanski, H.E. Fiedler. "Some Measurements in the Self-Preserving Jet". *Boeing Scientific Research Labs Document D1-82-0712*. . 1968.

[175] E. Zoby, A. Simmonds, and R. Gupta. "Temperature-dependent Reaction-rate Expression for Oxygen Recombination at Shuttle Entry Conditions" . *22nd Aerospace Sciences Meeting*. 224. 1984.

[176] G. Zheng. "CFD Modeling of Urea Spray and Deposits for SCR Systems". *SAE Technical Paper 2016-01-8077*. 2016.

[177] V. Zimont, W. Polifke, M. Bettelini, and W. Weisenstein. "An Efficient Computational Model for Premixed Turbulent Combustion at High Reynolds Numbers Based on a Turbulent Flame Speed Closure". *J. of Gas Turbines Power*. 120. 526–532. 1998.

[178] R. Ewert and W. Schroeder. "Acoustic perturbation equations based on flow decomposition via source filtering". *Journal of Computational Physics*. Volume 188, Issue 2. 365–398. 2003.

[179] Blasius, P.R.H.. "Das Aehnlichkeitsgesetz bei Reibungsvorgangen in Flüssigkeiten". Forschungsheft 131. –41. 1913.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information of ANSYS, Inc. and its subsidiaries and affiliates.

Release 2024 R2 - © ANSYS, Inc. All rights reserved. - Contains proprietary and confidential information
of ANSYS, Inc. and its subsidiaries and affiliates.